ANILAM

Conversational Programming for 6000i CNC

November 2009

www.anilam.com

Section 1 - Introduction

Section 2 - Conversational Mode Programming Hot Keys

Programming Hot Keys....2-1
Editing Keys....2-2

Section 3 - Writing Conversational Programs

Program Basics 3-1
Developing Part Programs....3-1
Writing Program Blocks 3-3
Using Graphic Menus....3-3
No Move Blocks....3-4
Programming an Absolute/Incremental Mode Change....3-5
Programming an Inch/MM Mode Change....3-5
Programming a Tool Change 3-5
Activating a Tool....3-6
Activating Tool-Diameter Compensation....3-7
Programming a Dwell 3-8
Programming a Return to Machine Zero 3-9
Programming Fixture Offsets....3-10
Resetting Absolute Zero (Part Zero) 3-14
Programming a Plane Change 3-16
Programming a Feedrate Change....3-17
Programming a Spindle RPM....3-17
Straight Moves....3-18
Programming a Rapid Move....3-18
Programming a Line Move 3-19
Programming a Modal Move 3-19
Line or Rapid Moves....3-20
Programming a Move Using XY Location, Radii, or Angles 3-21
Arcs 3-22
Selecting the Plane for an Arc....3-22
Programming an Arc Using an Endpoint and Radius....3-23
Programming an Arc Using the Center and Endpoint 3-25
Programming an Arc Using the Center and the Included Angle....3-27
Programming M-Code Blocks....3-29
Dry Run M-Codes....3-30

Section 4 - Programming Canned Cycles

Drilling Cycles....4-1
Basic Drill Cycle 4-2
Pecking Drill Cycle 4-3
Boring Cycle 4-5
Chip Break Cycle....4-6
Tapping Cycle 4-8
Pattern Cycle....4-10

Bolt Hole Cycle 4-11
Thread Milling Cycle 4-13
Pocket Cycles 4-16
Face Mill Cycle 4-17
Rectangular Profile Cycle 4-19
Circular Profile Cycle 4-21
Rectangular Pocket Cycle 4-23
Circular Pocket Cycle 4-25
Circular Slot Cycle 4-27
Frame Pocket Cycle 4-29
Hole Mill Cycle 4-31
Irregular Pocket Cycle 4-33
Slot Cycle 4-35
Pockets with Islands 4-38
Subprograms 4-42
Situation: 1 (Repetitive Drilling Cycle) 4-42
Situation: 2 (Rough and Finish Cycles) 4-42
Subprogram Structure 4-43
Subprogram Example 4-43
Organizing Programs Containing Subprograms 4-43
Calling Subprograms from the Main Program 4-44
Ending Main Programs 4-44
Starting Subprograms 4-44
Ending Subprograms 4-44
Looping Subprograms 4-45
Rotate, Mirror, and Scale Subprograms (RMS) 4-46
Engraving, Repeat, and Mill Cycles 4-47
Engraving Cycle 4-47
Repeat Cycle 4-49
Mill Cycle 4-51
Probing Cycles 4-53
Probing Canned Cycle Parameter Settings 4-53
Tool Probe Cycles 4-54
Spindle Probe Cycles 4-71
Using the Z Work Offset Update Feature 4-91

Section 5 - Editing Programs

Activating the Conversational Program Editor 5-1
Saving Edits 5-5
Canceling Unsaved Edits 5-5
Deleting a Block 5-5
Inserting a Block 5-5
Editing Blocks 5-6
Searching Blocks for Words or Numbers 5-6
Scrolling the Program Listing 5-6
Paging Through the Program Listing 5-7
Jumping to First or Last Block in the Program 5-7

Using Comments 5-7
Writing a Comment Block 5-7
Commenting Out Existing Blocks 5-7
Canceling a Comment 5-8
Using Block Operations to Edit a Program 5-8

Section 6 - Four-Axis Programming

Axis Types 6-1
Rotary Axis Programming Conventions 6-2
Programming Examples 6-2
Example 1: Drill 6-3
Example 2: Mill 6-5
Example 3: Mill 6-6

Index Index-1

Section 1 - Introduction

The 6000i CNCs support a conversational programming feature. This feature is standard on 6000i. The feature allows these CNCs to be programmed in conversational or G-Code. The conversational programming language in these CNCs is compatible with the conversational programming in the 3000M 3-Axis Kit CNC. The program type (conversational or G-Code) is determined when you create the program. Creating a program with extension of .M makes it a conversational program. Creating a program with extension of .G (or no extension) makes it a G-Code program. If no extension is assigned, the default extension .G is assigned.

The Program Management screen normally displays programs with .G extension. To use conversational programs, the Program Management screen must display the .M programs also. To always display conversational programs, set the "Program directory pattern" parameter under Control Software in the Setup Utility to *.G+*.M. For more information on this, refer to *6000i CNC Technical Manual*, P/N 627787-21. Alternatively, in the Program Management screen you can press **SHIFT + F9** until the conversational programs are visible. Refer to **Figure 1-1**.

CONVERSATIONAL

Figure 1-1, Conversational Screen

NOTE:	Review the differences between the Conversational screen above and the G-code screen.

Conversational programs are used the same way as G-Code programs. They can be edited, drawn, and executed in Auto or Single Step. The only feature that will operate differently is the editor. Conversational programs are edited with the conversational editor (similar to the editor in the 3000M 3-Axis Kit) while G-Code programs are edited with the standard G-Code editor. The program extension determines the editor that is used.

For more information on the Program Management, Draw, Auto/S.Step, etc. refer to *6000i CNC User's Manual*, P/N 627785-21.

Section 2 - Conversational Mode Programming Hot Keys

The following topics are described in this section:

- **Programming Hot Keys**
- **Editing Keys**

Programming Hot Keys

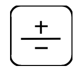
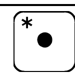
Programming hot keys allow you to enter position coordinates and provide quick access to functions that speed up programming. They are active in the Edit Mode. Refer to **Table 2-1**.

Table 2-1, Programming - Hot Keys

Label or Name	Key Face	Purpose
Letter **X**	X	Selects X-axis for position inputs.
Letter **Y**	Y	Selects Y-axis for position inputs.
Letter **Z**	Z	Selects Z-axis for position inputs.
Letter **E**	E	Switches CNC between Absolute and Incremental Modes.
Number **0** (Comment)	= 0	Zero / Switches comment asterisk in edit mode.
1/RAPID	1 RAPID	One / Hot key for programming a Rapid move.
2/LINE	2 LINE	Two / Hot key for programming a Line move.
3/ARC	3 ARC	Three / Hot key for programming an Arc.
4/FEED	4 FEED	Four / Hot key for changing feedrate.
5/TOOL	5 TOOL	Five / Hot key for programming a tool.
6/MCODE	6 MCODE	Six / Hot key for programming an M-Code.
7/UNIT	7 UNIT	Seven / Hot key for switching between inches (Inch) and millimeters (mm).
8/DWELL	8 DWELL	Eight / Hot key for programming a Dwell.
9/PLANE	9 PLANE	Nine / Hot key for selecting a plane.

(Continued…)

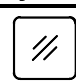
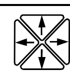
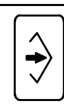
Table 2-2, Programming - Hot Keys (Continued)

Label or Name	Key Face	Purpose
+/-		Sign change / Toggle hot key.
Period/Decimal Point (Spindle RPM)		Decimal point / Hot key for programming the spindle RPM.

Editing Keys

Editing keys allow you to edit program blocks. These keys are located below the Programming Hot Keys. Refer to **Table 2-3**.

Table 2-3, Editing Keys

Label or Name	Key Face	Purpose
CLEAR		Clears the selected messages values, commands, and program blocks.
ARROW		Allows you to move highlight bars and cursor around the screen.
ENTER		Selects blocks for editing, activates menu selections, and activates number entry.

Section 3 - Writing Conversational Programs

The following topics are described in this section:

- **Program Basics**
- **Developing Part Programs**
- **Writing Program Blocks**
- **No Move Blocks**
- **Straight Moves**
- **Line or Rapid Moves**
- **Arcs**
- **Programming M-Code Blocks**

Program Basics

Each program consists of blocks of instructions that direct machine movements. Give each program a unique name.

Many settings remain active until changed or turned off. These are modal settings. For example, move type (Rapid/Feed), feedrate (IPM), units (Inch/MM), or ABS/INCR.

Write programs with combinations of moves, mode changes, and canned cycles. The CNC has a built-in library of canned cycles stored in its permanent memory.

Developing Part Programs

First, decide how to clamp the part and where to set Part Zero (Absolute Zero). Locate Part Zero at a point on the work positively positioned by the clamping fixture. This allows consistent machining of subsequent parts. Since Absolute positions are measured from Part Zero, locate Part Zero at a convenient location.

Determine the required tools and set the length offset for each tool.

Refer to the blueprint to select a Part Zero. Note the moves, positions, and tools needed to cut the part.

To develop a part program:

1. Enter the Program Directory (the **PROGRAM** screen) and create the program for the part. Use the extension **.M**.
2. Enter the Program Editor (the **Edit** screen) to open the new program and begin to write blocks (refer to “Section 5 - Editing Programs”).
3. The first block in a program is usually an Absolute Mode block. Put the CNC in the Absolute Mode at the start of a program to enable absolute positioning. (Use Incremental Mode only when specifically needed.)

4. Put the CNC in the appropriate Inch/MM Mode in the second block.
5. In the first move of the program, rapid to Tool #0, Z0 to retract the quill fully for the next move.
6. In the second move, rapid to a convenient part-change position.
7. Execute moves toward a part in two steps: A Rapid X, Y move at a clear height, followed by a Z move to 0.1 inch (2mm) above the surface of the cut (standard starting height). If necessary, activate the first tool mount at this time.
8. Subsequent blocks in the program are the moves, cycles, and tool changes required to machine the part.
9. Make the last three blocks of the program as follows:
   a) a Home Z0
   b) a Rapid XY move to the same part change position used at the start of the program, and
   c) an **EndMain** block.
10. To verify and troubleshoot finished programs, run them in Draw Graphics Mode.
11. Secure the work on the table with the appropriate work-holding device.
12. Go to the Manual screen and set Part Zero at a convenient point on the part.
13. Go to the Tool Page and organize the tooling. Assign each tool a number (in the order of use). Assign length offset and tool diameter as appropriate.
14. If Fixture Offsets are used, define them in the Fixture Offsets Table. Refer to “Programming Fixture Offsets” in this section.
15. Before you cut a part, perform a dry run. There are several ways to get a close look at the programmed moves. Run the program in Single-Step Mode to hold between each block. Run the program with no tool installed.
16. After a successful dry run, the program is ready for production. Back up the program for safekeeping.

Writing Program Blocks

You can program a block for a move type, mode, or cycle using one of the following: hot keys, soft keys, or pop-up menus.

To program a block, activate its graphic menu and fill in the appropriate values. To save a program block, press **Save** (**F10**) or press **ENTER** on the last entry field in the graphic menu. The CNC adds the new block to the Program Listing.

The last block of the Main Program must be **EndMain**. If this block is omitted, a warning displays stating “Missing M2 or M30!”

The **<End Of Program>** block is the last line of a program. The CNC automatically numbers new blocks and inserts them in front of the **<End Of Program>** block.

The following topic is described:

- **Using Graphic Menus**

Using Graphic Menus

The Program Editor displays full screen graphic menus to write and edit program blocks. Refer to **Figure 3-1**.

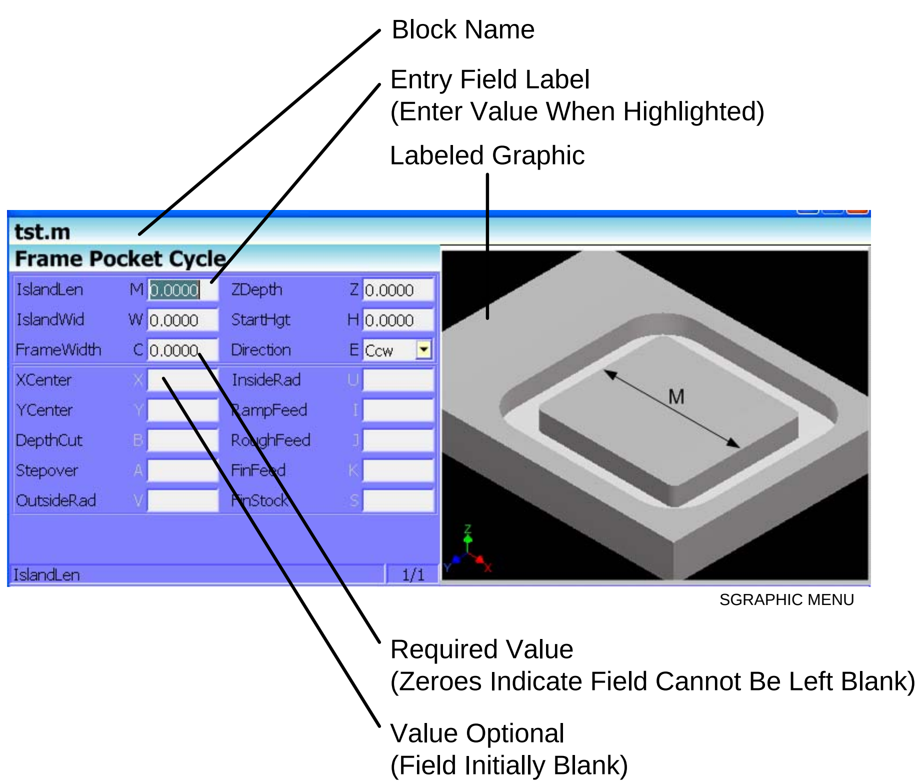


Figure 3-1, Sample Graphic Menu

Graphic menus activate with the first entry field highlighted. To type values, highlight the appropriate entry field. Press **ENTER** to advance the highlight to the next entry field. With the last entry field highlighted, press **ENTER** to close the menu and add the block to the program.

Press **Save** (**F10**) from any entry field to close the graphic menu and add the block to the program. Move the highlight from field to field using the **ARROW** keys. Fill out entry fields in any order.

Press **CLEAR** to remove values in the highlighted field.

There are two types of entry fields in a graphic menu:

Optional entry fields	are blank when the graphic menu activates.
Required entry fields	contain **0.000** when the graphic menu activates.

Required entry fields contain a 0.0000 default value. Change the value as required. Optional entry fields do not require a value. When left blank, the CNC usually assumes a default value or position. If the optional field is a position, the value defaults to the current position. If the optional field is a mode or tool change, the current mode and tool remain active. If the optional field is an angle, the value defaults to 0.0 degrees.

Type decimal points and negative signs where needed. Otherwise, the CNC assumes a positive whole number.

Press **+/-** to insert a negative sign or toggle selections in some entry fields (for example, **Cw/Ccw** fields).

No Move Blocks

No Move Blocks does not initiate machine moves. Use No Move Blocks to set modes (Incremental/Absolute, etc.), activate tools (**Tool#**), and set feedrates (**Feed**).

The following topics are described:

- **Programming an Absolute/Incremental Mode Change**
- **Programming an Inch/MM Mode Change**
- **Programming a Tool Change**
- **Activating a Tool**
- **Activating Tool-Diameter Compensation**
- **Programming a Dwell**
- **Programming a Return to Machine Home**
- **Programming Fixture Offsets**
- **Resetting Absolute Zero (Part Zero)**
- **Programming a Plane Change**
- **Programming a Feedrate Change**
- **Programming a Spindle RPM**

Programming an Absolute/Incremental Mode Change

A **Dim** (dimension) block sets the Absolute (**Abs.**) or Incremental (**Incr.**) Mode.

To program a **Dim** block:

1. In Edit Mode, press the letter **E**. The **SET ABS/INCR DIMENSION** graphic menu prompts you to select **Abs** or **Incr**.
2. Press **+/-** to toggle the mode.
3. Press **Save** (**F10**) or **ENTER** to add the block to the Program Listing.

Programming an Inch/MM Mode Change

A Unit block sets the Inch (**Inch**) or Millimeter (**MM**) Mode.

To program a Unit block:

1. In Edit Mode, press **7/UNIT**. The **SET INCH/MM UNIT** graphic menu prompts you to select **Inch**/**MM**.
2. Press **+/-** to toggle the selection.
3. Press **Save** (**F10**) or **ENTER** to add block to the Program Listing.

Programming a Tool Change

Identify tools with tool numbers. When you activate a tool, its tool length and diameter offsets activate. List these values on the corresponding row of the Tool Page.

Tool-length offset remains in effect until a different tool activates. Always turn off tool-diameter compensation and ramp off before activating a new tool.

NOTE: Each time a tool activates, the CNC holds the program to permit installation of the new tool. Programming unnecessary tool changes slows down production.

Activate Tool #0 to set the tool-length offset and diameter to 0.0.

To change a tool:

NOTE: An absolute move to Tool #0, Z0 fully retracts the quill. An incremental command to Z0 maintains the current position.

1. In Absolute Mode, program a Tool #0, Rapid Z0 to cancel length offsets and retract the quill to a safe position.
2. Program a Rapid move to the tool change XY position (usually Machine Zero).
3. Program a block to activate the required tool (example: **Tool#1**). When the CNC encounters the **Tool#** command, it holds program execution. The operator can now change the tool.

4. Press **START** to resume operation. The CNC activates applicable tool compensation.

Activating a Tool

To activate a tool:

1. In Edit Mode, press **5/TOOL**. The **TOOL MOUNT** graphic menu prompts for **Tool #**.
2. Type tool number and press **ENTER**.
3. The cursor advances to the M-Code field. If you have Automatic Tool Changer, type the appropriate activation M-Code. (For example, type **6**)
4. Press **ENTER** to add the **Tool#** block to the Program Listing.

Activating Tool-Diameter Compensation

Turn compensation on or off in (Rapid or Line) ramp moves. Ramp moves offset the tool on the programmed path by half the tool diameter. Tool compensation affects all subsequent moves until canceled.

The **ToolComp** command, available in Line or Rapid graphic menus, sets the required tool compensation. Settings include:

- **Left** (of the path)
- **Right** (of the path)
- **Off** (cancel compensation)

When the field is left blank, the current compensation, if any, remains in effect.

Many canned cycles include automatic tool compensation. Activate the correct tool diameter to ensure accuracy in these cycles. The required tool activates within the cycle.

Refer to **Table 3-1** for a list of move and cycle compensation requirements.

Table 3-1, Move and Cycle Compensation Requirements

Move or Cycle	Program a Rapid or Line move to activate tool comp before you program the move or cycle. Tool diameter must be active.	Activate/deactivate compensation automatically when you program the move or cycle. Tool diameter must be active.
Rapid	X	---
Line	X	---
Modal	X	---
Arc	X	---
Face (Affects only step-over.)	---	X
Rectangular Profile Cycle	---	X
Circular Profile Cycle	---	X
Rectangular Pocket Cycle	---	X
Circular Pocket Cycle	---	X
Frame Pocket Cycle	---	X
Irregular Pocket Cycle	---	X

Programming a Dwell

Dwell pauses a running program for a specified length of time, in seconds. Dwell resolution is 0.1 sec. When the operator types 0.0 seconds (infinite dwell), the CNC will hold indefinitely. Press **START** to restart the CNC after an infinite dwell.

To program a Dwell using hot keys:

1. In Edit Mode, press **8/DWELL**. The **DWELL** graphic menu prompts for length of time in seconds.
2. Type the time and press **ENTER** to add Dwell block to the Program Listing.

To program a Dwell using soft keys:

1. In Edit Mode, press **Sub** (**F8**) to display the Secondary soft key functions.
2. Press **Dwell** (**F7**) to activate the **DWELL** graphic menu.
3. Type the time and press **ENTER** to add Dwell block to the Program Listing.

Programming a Return to Machine Zero

NOTE:	The CNC measures all entered coordinates in the Machine Home graphic menu from Machine Zero. The CNC homes axes one at a time, in the order indicated in the Setup Utility.

A **Home** block re-establishes a permanent reference position located on the machine. The position is called Machine Zero. Program a Home block using one of the two methods described in **Table 3-2**.

Table 3-2, Homing Methods

Homing Method	Required Action
1. Indicate axes	Activate Machine Home graphic menu. Press required **X**, **Y**, **Z**-axis keys. On each axis selected, machine feeds from the current position to the limit switch, reverses direction, and travels to the first detected zero crossing and sets Machine Zero at that point.
2. Enter coordinates	Activate Machine Home graphic menu. For each required axis, highlight the axis entry field and type a coordinate (example: X0, Y-1, Z-4). The machine rapids to the typed coordinate, then feeds to the limit switch, reverses direction, and travels to the first detected zero crossing and sets Machine Zero at that point.

Use Homing Method 1 to execute a homing sequence in feed. Use Homing Method 2 to execute a homing sequence that rapids to the entered coordinate, then initiates the homing sequence.

To activate the Machine Home graphic menu:

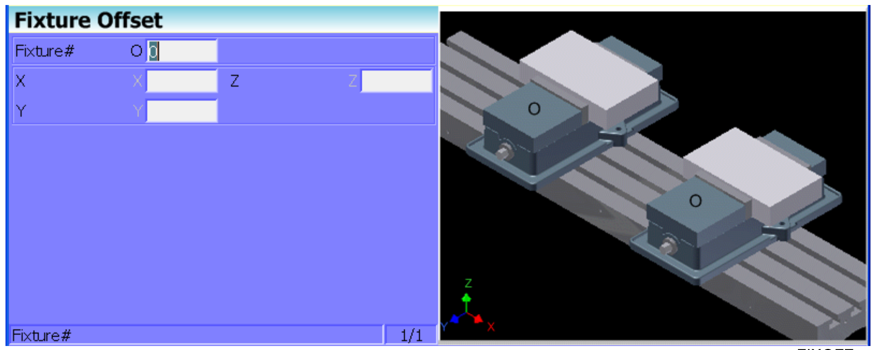
1. In the Edit Mode, press **Milling** (**F5**) to display the Mill soft keys.
2. Press **More** (**F7**) to display the More pop-up menu.
3. Highlight **Home** and press **ENTER** to display the Machine Home graphic menu.

The method used to set Machine Zero depends on which options the builder installs. Check with the machine builder for detailed information.

Programming Fixture Offsets

Refer to **Figure 3-2**.

NOTE: Presets and SetZero will work with Fixture Offsets.

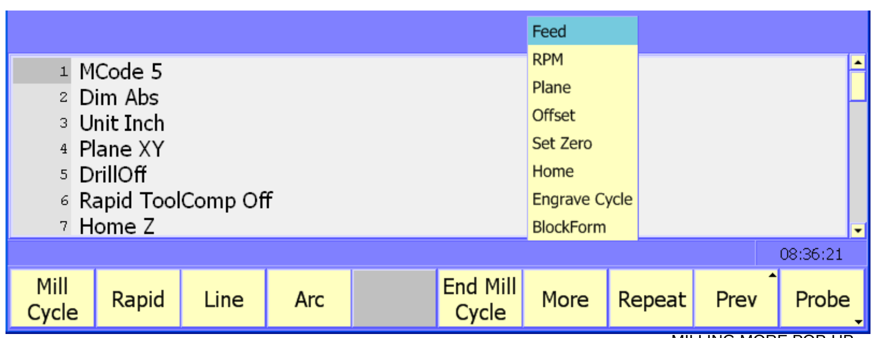


FIXOFF

Figure 3-2, Fixture Offset Graphic Menu

To program:

1. In Program Mode, press **Edit** (**F7**) to display the Edit soft keys.
2. Press **Milling** (**F5**) to display the Milling soft keys.
3. Press **More** (**F7**) to display the More pop-up menu. Refer to **Figure 3-3**.



MILLING MORE POP-UP

Figure 3-3, Milling (F5)>More (F7) Pop-up Menu

4. Select **Offset** and press **ENTER** to display the Fixture Offset graphic menu. Refer to **Figure 3-2**.
5. Fill in the labeled entry fields. Refer to **Table 3-3, Fixture Offsets Address Words**.

Table 3-3, Fixture Offsets Address Words

Label	Address Word	Description
Fixture#	**O**	The Fixture-Offset number. Indicates which set of values from the Fixture Offsets Table will be activated or changed. Type a number **1** through **99**, corresponding to the Fixture Offsets Table, to activate or change an offset. Type **0** to cancel fixture offsets. (Required)
X	**X**	X-offset coordinate. If you do not type a value, the CNC activates the offsets listed in the Fixture Offsets Table for the entered **Fixture#**. If you do type a value, the CNC applies the entered offset. When the program runs, the CNC updates the Fixture Offsets Table with the specified X offset and clears the previous value. (Optional)
Y	**Y**	Y-offset coordinate. If you do not type a value, the CNC activates the offsets listed in the Fixture Offsets Table for the entered **Fixture#**. If you do type a value, the CNC applies the entered offset. When the program runs, the CNC updates the Fixture Offsets Table with the specified Y offset and clears the previous value. (Optional)
Z	**Z**	Z-Offset coordinate. If you do not type a value, the CNC activates the offsets listed in the Fixture Offsets Table for the entered **Fixture#**. If you do type a value, the CNC applies the entered offset. When the program runs, the CNC updates the Fixture Offsets Table with the specified Z offset and clears the previous value. (Optional)

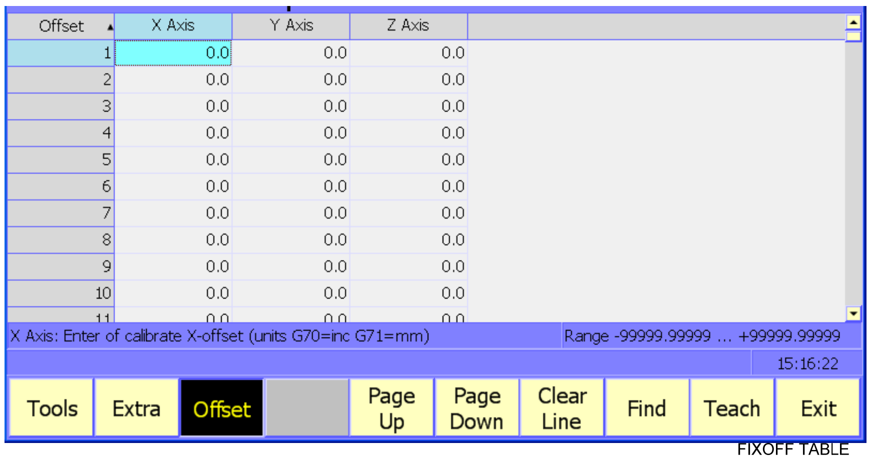
To cancel Fixture Offsets:

1. In Edit Mode, press **Milling** (**F5**) to display the Milling soft keys.
2. Press **More** (**F7**) to display the More pop-up menu. Refer to **Figure 3-3, Milling (F5)>More (F7) Pop-up Menu**.
3. Select **Offset** and press **ENTER** to display the Fixture Offset graphic menu.
4. Select **Fixture#**. In the highlighted entry field, type **0** to cancel Fixture Offsets. (Do not fill in the other entry fields.)

Fixture Offsets Table

The Fixture Offsets Table, accessed using the Tool Page, contains the entered values for Fixture Offsets 1 through 99. Refer to **Figure 3-4**.

NOTE: Handwheel and Jog features are available while the Fixture Offsets Table is active.



FIXOFF TABLE

Figure 3-4, Fixture Offsets Table Screen

Activating the Fixture Offsets Table

To activate the Fixture Offsets Table:

1. In the Tools Page, press **Offset** (**F3**).

Changing Fixture Offsets in the Table

There are two ways to change the values in the table, manually tape a value or calibrate the fixture offset table entry to the machine's current location (shown on the axis display).

To change a fixture offset to a manually entered coordinate:

1. Highlight a Fixture Offset (row 1 to 99) in the Fixture Offset Table.
2. Highlight axis column (**X**, **Y**, or **Z**) using the **ARROW** keys.
3. Type a value. Press **ENTER**. The CNC stores the value in the table.

To calibrate the fixture offset table entry to the machine's current location:

1. Highlight a Fixture Offset (row 1 to 99) in the Fixture Offset Table.
2. Highlight axis column (**X**, **Y**, or **Z**) using the **ARROW** keys.
3. Press **Teach** (**F9**) to store the current machine position for the selected axis in the table.

Adjusting Fixture Offsets in the Table

To adjust an existing fixture offset:

1. Highlight a Fixture Offset (row 1 to 99) in the Fixture Offset Table.
2. Highlight axis column (**X**, **Y**, or **Z**) using the **ARROW** keys.
3. Press the letter **A** key to display the message, “Add value:”.
4. Type the adjustment value. The adjustment value may be positive or negative.
5. Press **ENTER** to adjust the value, and display the adjusted value in the table.

Resetting Absolute Zero (Part Zero)

Absolute Zero is the X0, Y0 position for absolute dimensions. A **SetZero** block sets the Absolute Zero Reference of one or more axes to a new position. Use **SetZero** in one of two ways: to reset X0 Y0 or to preset the current location to entered coordinates.

In axis presetting, non-zero XY values set the current machine position to the entered coordinates. In axis resetting, X0 and Y0 values set the current machine position as the new Absolute Zero Reference.

When the CNC executes the block, the X and Y values (zero or non-zero) in the graphic menu redefine Absolute Zero.

In **Figure 3-5**, diagram A shows Part Zero and tool position prior to a **SetZero** block. In this example, the operator programs a **SetZero** block with the following coordinates: **X2**, **Y-1**.

Diagram B shows Part Zero and tool position following the **SetZero** block. The coordinates at the tool position become **X2**, **Y-1**. This, in effect, moves Part Zero, as indicated.

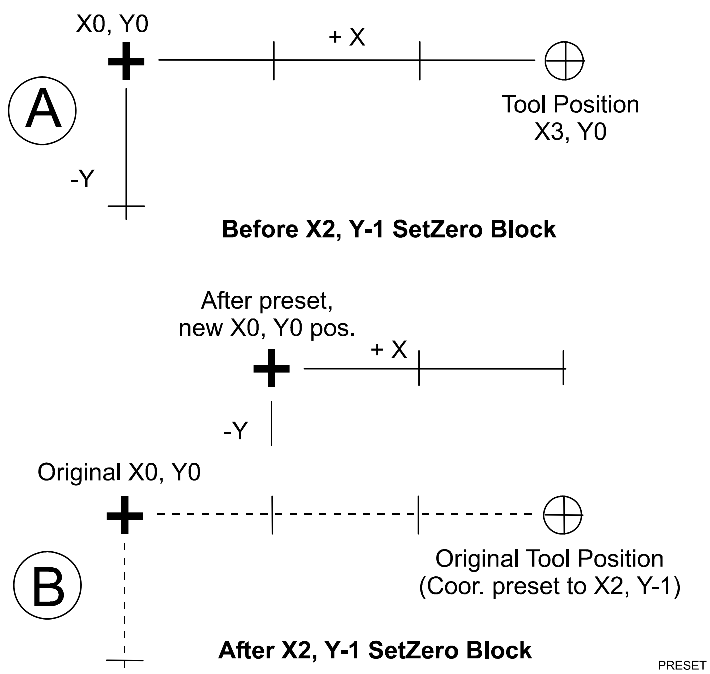


Figure 3-5, Executing a SetZero Block

Change Absolute Zero to cut more than one part with the same moves. Restore the location of the original X0, Y0 reference at the end of the program so that programmed part change positions do not move each time the program runs. Refer to **Figure 3-6, Using SetZero in a Program**.

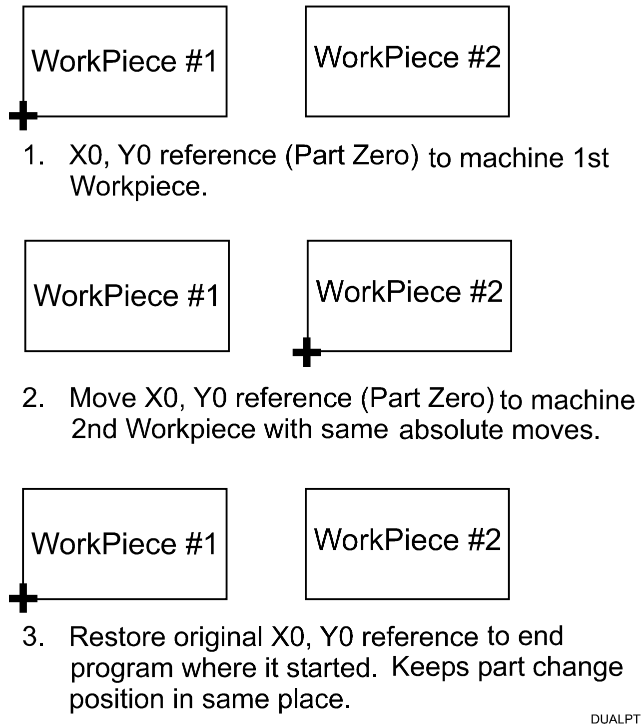


Figure 3-6, Using SetZero in a Program

When an axis entry field (**X**, **Y**, **Z**, or **U**) remains blank in a graphic menu, the CNC does not change the position of that axis. Refer to **Figure 3-7**.

NOTE:	In most programs, the Z-axis position does not change. Changing the Z-axis position changes the Tool #0, Z0 position, which alters all existing tool-length offsets.

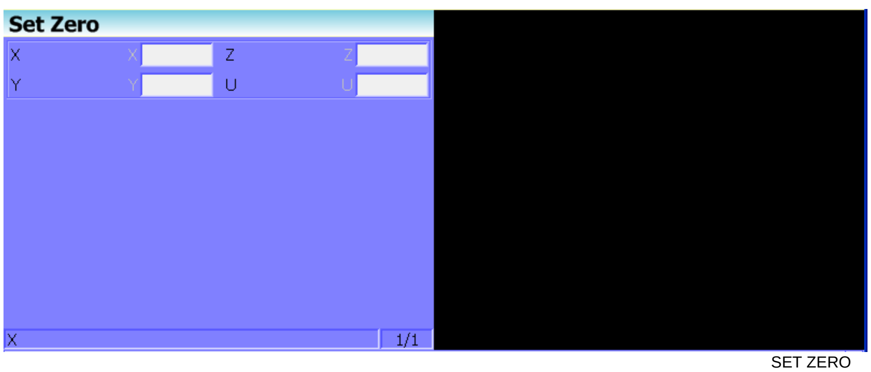


Figure 3-7, Set Zero Graphic Menu

To program a **Set Zero** block:

1. In Edit Mode, press **Milling** (**F5**) to change the soft key labels.
2. Press **More** (**F7**) to display the pop-up menu. Refer to **Figure 3-3, Milling (F5)>More (F7) Pop-up Menu**.
3. Position the highlight to select **Set Zero**, and then press **ENTER**. The Set Zero graphic menu prompts for the absolute coordinates of the machine's current position.
4. Type the appropriate **X**, **Y**, **Z**, and **U** coordinates and press **ENTER** to add the block to the Program Listing.

Programming a Plane Change

The CNC executes Arc moves and compensates for tool diameters in three different planes (**XY**, **YZ**, and **XZ**). By default, the CNC operates in the XY plane. Program a Plane block to change the CNC's active plane. Following moves in the XZ or YZ plane, program a second Plane block to return to the XY plane.

A Plane block also changes the active plane of the Program Editor. The Program Editor customizes Arc graphic menus for the active plane.

When a plane block is deleted from the Program Listing, the active plane of the Editor does not change.

To program a Plane block using hot keys:

1. In Edit Mode, press **9/PLANE**. The **PLANE** graphic menu prompts for plane selection.
2. Press the **+/-** key to change the selection to the desired plane. Press **ENTER** to add the block to the Program Listing.

To program a Plane block using soft keys:

1. In Edit Mode press **Milling** (**F5**) to display the Mill soft keys.
2. Press **More** (**F7**) to display the **More** pop-up menu. Refer to **Figure 3-3, Milling (F5)>More (F7) Pop-up Menu**.
3. Highlight **Plane** and press **ENTER**. The Plane graphic menu prompts for plane selection.
4. Select the desired plane and press **ENTER** to add the block to the Program Listing.

Programming a Feedrate Change

A Feed block sets the feedrate for Line moves, arcs, and cycles that do not contain specifically programmed feedrates. Feed blocks also set the feedrate for modal moves. Add Feed blocks whenever necessary.

NOTE: A Feed block does not activate the Feed Mode.

To program a Feed block from the hot keys:

1. In Edit Mode, press **4/FEED** to display the Feedrate graphic menu.
2. Type the required feedrate and press **ENTER** to add the block to the Program Listing.

To program a Feed block from the soft keys:

1. In Edit Mode, press **Milling** (**F5**) to display the Mill soft keys.
2. Press **More** (**F7**) to display the **More** pop-up menu. Refer to **Figure 3-3, Milling (F5)>More (F7) Pop-up Menu**.
3. Highlight **Feed** and press **ENTER** to activate the Feedrate graphic menu.
4. Press **Use** (**F10**) or **ENTER** to add Feed block to the Program Listing.

Programming a Spindle RPM

If your CNC has a programmable spindle RPM, you can set the RPM as follows:

To program an RPM block from the hot keys:

1. In Edit Mode, press the **.** (Decimal/RPM) key to display the Spindle RPM graphic menu.
2. Type the required spindle RPM and press **ENTER** to add the block to the Program Listing.

To program an RPM block from the soft keys:

1. In Edit Mode, press **Milling** (**F5**) to display the Mill soft keys.
2. Press **More** (**F7**) to display the **More** pop-up menu. Refer to **Figure 3-3, Milling (F5)>More (F7) Pop-up Menu**.
3. Highlight **RPM** and press **ENTER** to activate the Spindle RPM graphic menu.
4. Type the required RPM.
5. Press **Use** (**F10**) or **ENTER** to add RPM block to the Program Listing.

Straight Moves

The following topics are described:

- **Programming a Rapid Move**
- **Programming a Line Move**
- **Programming a Modal Move**

Programming a Rapid Move

Rapid moves run at the CNC's Rapid rate and save time when positioning for a cut or a canned cycle.

Use Rapid moves to activate/deactivate tool diameter compensation and cutter compensation. Refer to **Figure 3-8**.

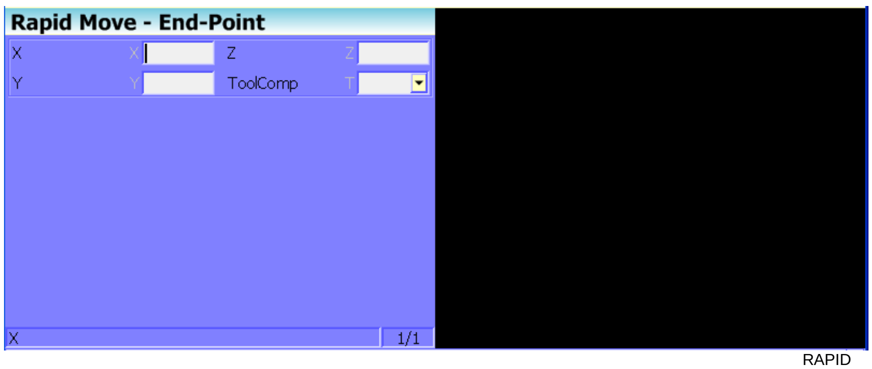


Figure 3-8, Rapid Move Graphic Menu

To program a Rapid move using hot keys:

1. In Edit Mode, press **1/RAPID** to activate the Rapid Move graphic menu.
2. Type the **X**, **Y**, and **Z** coordinates in the appropriate entry fields.
3. Press **+/-** to set **ToolComp** (optional) and press **ENTER**.

To program a Rapid move using soft keys:

1. In Edit Mode, press **Milling** (**F5**) to display the Mill soft keys.
2. Press **Rapid** (**F2**) to activate the Rapid Move graphic menu.
3. Type the appropriate values and settings in the labeled entry fields.

Programming a Line Move

Straight-line moves run in Feed. Refer to **Figure 3-9**.

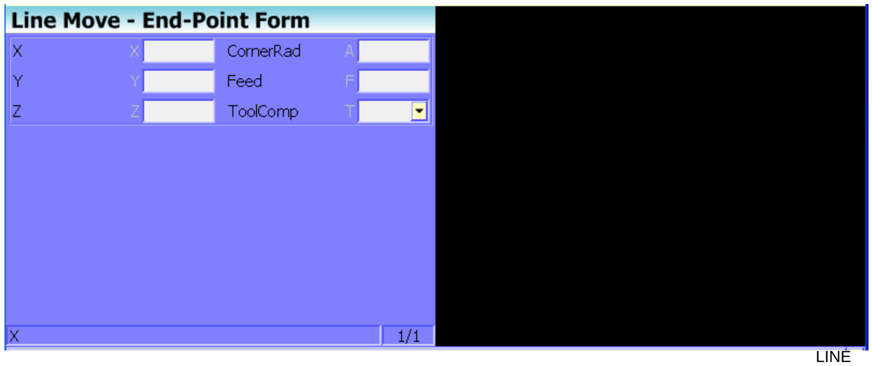


Figure 3-9, Line Move Graphic Menu

To program a Line move using hot keys:

1. In Edit Mode, press **2/LINE** to activate the Line Move graphic menu.
2. Type the appropriate values and settings in the labeled entry fields.

To program a Line move using soft keys:

1. In Edit Mode, press **Milling** (**F5**) to change the soft key labels.
2. Press **Line** (**F3**) to display the Line Move graphic menu.
3. Type the appropriate values and settings in the labeled entry fields.
4. With the last entry field highlighted, press **ENTER** to add the block to the Program Listing.

Programming a Modal Move

A modal move is a straight move executed in the active Rapid or Feed Mode.

To program a Modal move:

1. In Edit Mode, press **X**, **Y**, or **Z**. The Modal XYZ graphic menu prompts for the X, Y, and Z positions.
2. Type the required positions.
3. From the last field on the graphic menu, press **ENTER** or **Use** (**F10**) to add the modal move block (Xn Yn Zn) to the program.

NOTE:	When using modal moves, be sure the CNC is in the required Rapid or Line Mode. The CNC executes Line Mode moves in Feed Mode.

Line or Rapid Moves

Using the X, Y, or XY endpoints, the CNC can write Line or Rapid moves. The CNC calculates the missing endpoint(s). Define the move as part of a right triangle with the components identified as in **Figure 3-10**.

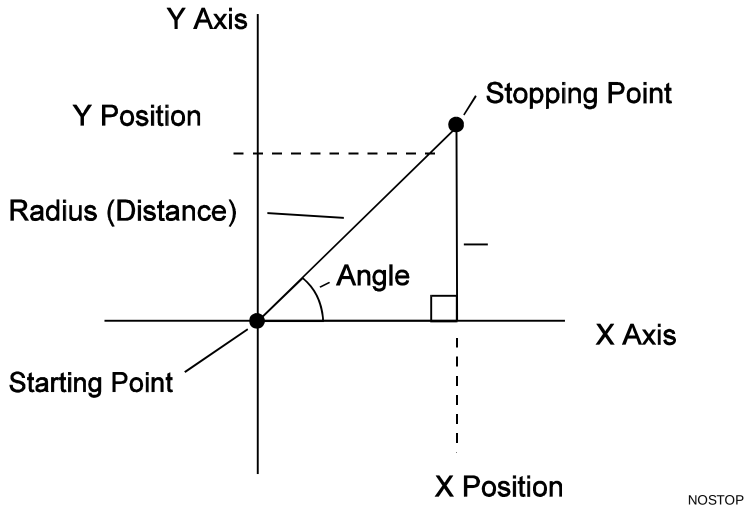


Figure 3-10, Move Orientation

The CNC can calculate move endpoints, given:

- Angle and radius
- X position and angle
- Y position and angle
- X position and radius
- Y position and radius

The **Rapid** and **Line** graphic menus are similar. However, the **Rapid** graphic menus do not contain **CornerRad** or **Feed** entry fields. Use either the Absolute or Incremental Mode.

The following topic is described:

- ❑ **Programming a Move Using XY Location, Path, or Angles**

Programming a Move Using XY Location, Radii, or Angles

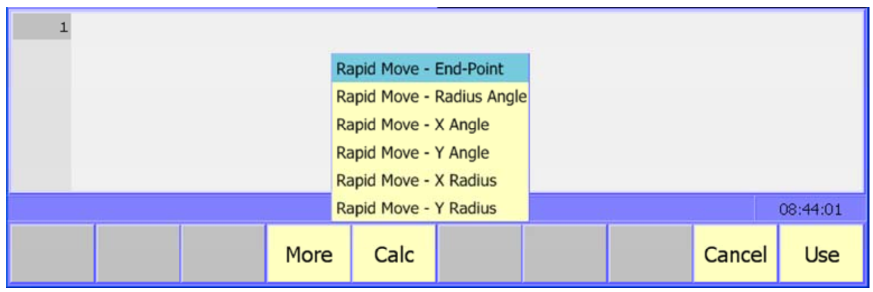
To program a move using a Line or Rapid block:

1. In Edit Mode, press **Milling** (**F5**) and select either **Rapid** (**F2**) or **Line** (**F3**).

   – or –

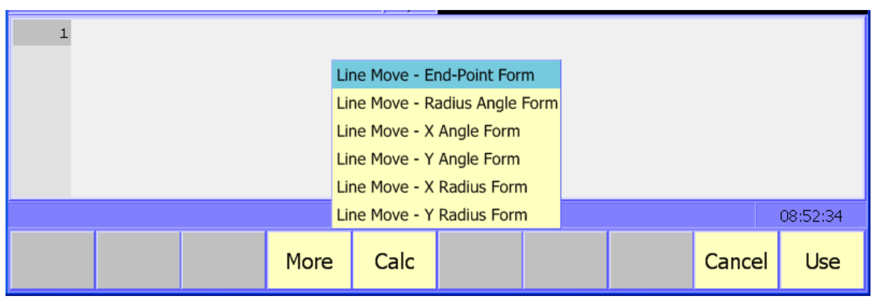
   In Edit Mode, press **1/RAPID** or **2/LINE** to display the Rapid Move or Line Move graphic menu.

2. Press **More** (**F4**) to display the More pop-up menu. Refer to **Figure 3-11** and **Figure 3-12**.



RAPID MORE POPUP

Figure 3-11, Rapid More (F4) Pop-up Menu



LINE MORE POPUP

Figure 3-12, Line More (F4) Pop-up Menu

3. Highlight the appropriate selection and press **ENTER** to display the graphic menu.

4. Type the required values/settings in the entry fields.

Arcs

The following topics are described:

- **Selecting the Plane for an Arc**
- **Programming an Arc Using the Endpoint and Radius**
- **Programming an Arc Using the Center and Endpoint**
- **Programming an Arc Using the Center and Included Angle**

Selecting the Plane for an Arc

The CNC executes Arcs in the XY plane by default. For an Arc in the XZ or YZ plane, program the plane change before the Arc move. The plane change customizes the Arc graphic menus for the required plane.

The graphic menus for moves in the XY, XZ, and YZ planes contain the same entry fields. Entry fields for selected plane positions require a value.

After a move in the XZ or YZ plane, return the CNC to the XY plane.

> **NOTE:** To activate a new plane in the Program Editor, program a plane change block.

Program Arc moves:

- Using the endpoint and radius
- Using the center and endpoint
- Using the center and angle

Programming an Arc Using an Endpoint and Radius

To define the **Endpoint - Radius Arc**, type the direction of the Arc, the endpoint, and the radius. The CNC cuts an Arc of the specified radius from the current position to the endpoint. You must correctly define the modal endpoint coordinates in the **Absolute** or **Incremental** Mode.

In the XY plane, if the Z-axis starting and end points differ, the arc is a helix.

Two Arcs can intersect any two points: an Arc with an included angle less than 180 degrees and an Arc with an included angle greater than 180 degrees. Refer to **Figure 3-13**.

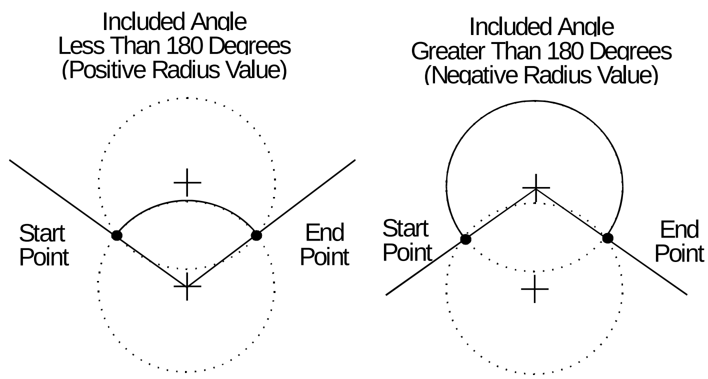


Figure 3-13, Endpoint Radius Arc Types

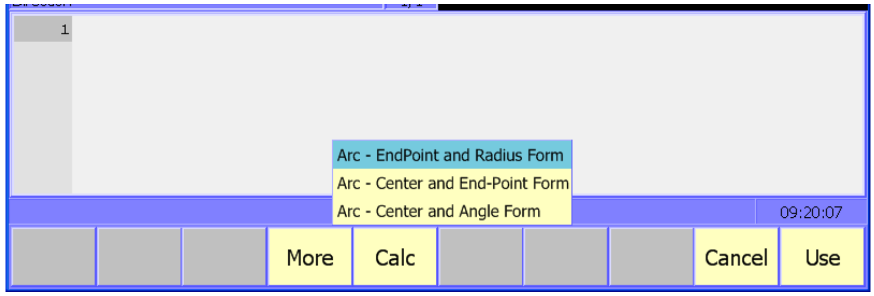
To program an Arc with an included angle less than 180 degrees, type a positive radius value. To program an Arc with an included angle greater than 180 degrees, type a negative radius value. The CNC selects which Arc center to use, based on the sign of the typed value.

To program an Arc using an endpoint and radius, using hot keys:

1. In Edit Mode, press **3/ARC**. The **Arc – EndPoint and Radius Form** graphic menu prompts for labeled values.
2. Fill in the entry fields as labeled.

To program an Arc using an endpoint and radius, using soft keys:

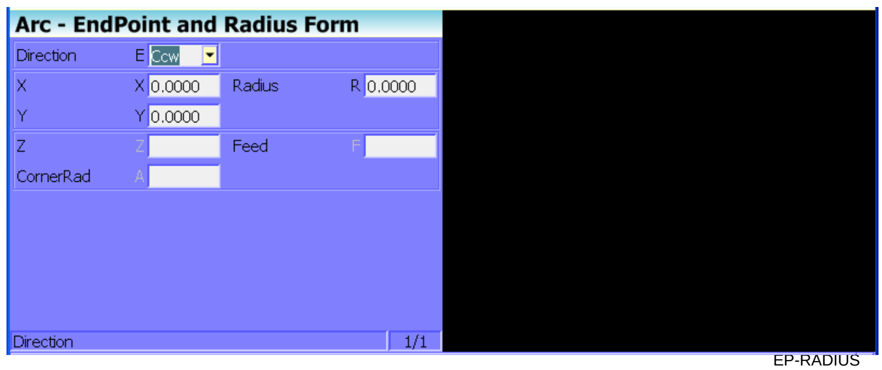
1. In Edit Mode, press **Milling** (**F5**) to display the Mill secondary soft keys.
2. Press **Arc** (**F4**) to display the Arc soft keys.
3. Press More (**F4**) to display the More pop-up menu. Refer to **Figure 3-14, Arc More (F4) Pop-up Menu**.



ARC MORE POPUP

Figure 3-14, Arc More (F4) Pop-up Menu

4. Highlight **Arc – EndPoint and Radius Form** and press **ENTER** to display the graphic menu. Refer to **Figure 3-15**.



EP-RADIUS

Figure 3-15, Arc – EndPoint and Radius Graphic Menu

5. Fill in the Arc – EndPoint and Radius entry fields. Refer to **Table 3-4**.

Table 3-4, Arc – EndPoint and Radius Address Words

Label	Address Word	Description
Direction	**E**	Specifies a clockwise (**Cw**) or counterclockwise (**Ccw**) direction. Press **+/-** to toggle the setting. (Required)
XCenter	**X**	The X coordinate of the Arc endpoint (Required)
YCenter	**Y**	The Y coordinate of the Arc endpoint (Required)
Radius	**R**	The radius of the Arc (Required - positive or negative)
Z	**Z**	The Z coordinate of the endpoint (Optional)
CornerRad	**A**	Corner radius setting (Optional)
Feed	**F**	Feedrate (Optional)

Programming an Arc Using the Center and Endpoint

NOTE: Use Center and Endpoint Arcs to cut helical threads.

To define the **Center - Endpoint Arc**, type the endpoint, arc center, and direction. The CNC cuts an Arc from the current position to the end point.

In Absolute Mode, the CNC measures the Arc center and endpoint from Absolute Zero. In Incremental Mode, the CNC measures the Arc center and end point from the starting position of the arc.

NOTE: Ensure that the required Absolute or Incremental Mode is active.

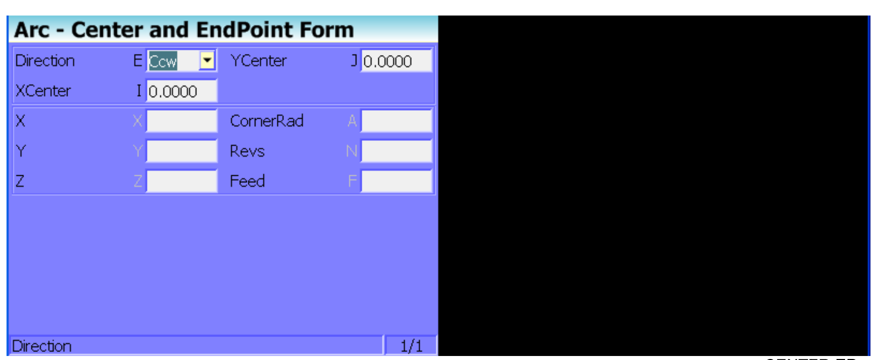
When the Z-axis start and end points differ in the XY plane, the Arc is a helix. The **Revs** value determines the number of rotations used to machine the helix.

To program a **Center - EndPoint Arc** using hot keys:

1. In Edit Mode, press **3/ARC**.
2. Press **More** (**F4**) to display the More pop-up menu. Refer to **Figure 3-14, Arc More (F4) Pop-up Menu**.
3. Highlight **Arc – Center and Endpoint Form** and press **ENTER** to display the graphic menu. Refer to **Figure 3-16**.
4. Fill in the entry values as labeled.

To program an **Arc – Center and EndPoint** using soft keys:

1. In Edit Mode, press **Milling** (**F5**) to display the Mill secondary soft keys.
2. Press **Arc** (**F4**) to display the Arc soft keys.
3. Press **More** (**F4**) to display the More pop-up menu. Refer to **Figure 3-14, Arc More (F4) Pop-up Menu**.
4. Highlight **Arc – Center and EndPoint Form** and press **ENTER** to display the graphic menu. Refer to **Figure 3-16**.



CENTER-EP

Figure 3-16, Arc – Center and EndPoint Graphic Menu

5. Fill in the Arc – Center and EndPoint entry fields. Refer to **Table 3-5**.

Table 3-5, Arc – Center and EndPoint Address Words

Label	Address Word	Description
Direction	**E**	Specifies a clockwise (**Cw**) or counterclockwise (**Ccw**) direction. Press **+/-** to toggle the setting. (Required)
XCenter	**I**	The X coordinate of the Arc center (Required)
YCenter	**J**	The Y coordinate of the Arc center (Required)
X	**X**	The X coordinate of the endpoint (Optional)
Y	**Y**	The Y coordinate of the endpoint (Optional)
Z	**Z**	The Z coordinate of the endpoint (Optional)
CornerRad	**A**	Corner radius setting (Optional)
Revs	**N**	The number of complete plus partial revolutions, referenced from the start point.
Feed	**F**	Feedrate (Optional)

Programming an Arc Using the Center and the Included Angle

To define the **Center - Angle Arc**, type the arc center and the included angle. The CNC cuts the Arc from the present position until the Arc travels the specified number of degrees. The CNC calculates the radius, which is the distance between the start position and the center point.

Specify the appropriate Absolute or Incremental Mode for the angle and center point. Refer to **Figure 3-17** and **Figure 3-18**.

The direction (Cw/Ccw) of the Arc and the sign (+/-) of the angle control the path of the tool.

If the Z-axis starting and end points differ, the Arc will be a helix.

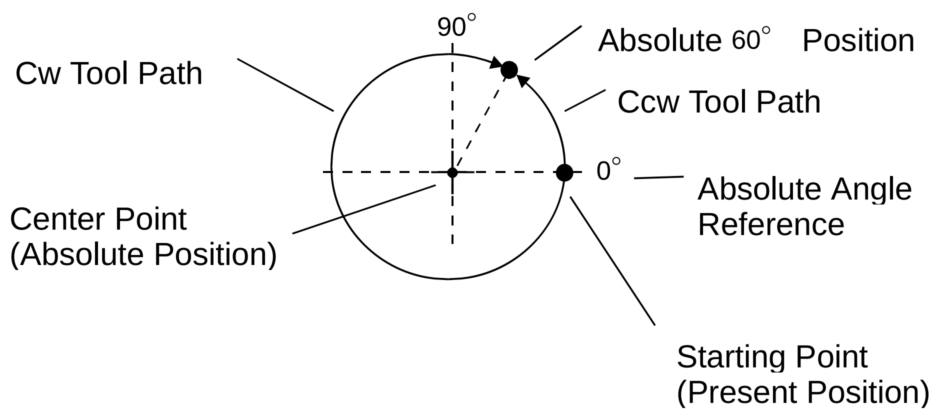


Figure 3-17, Absolute Mode, Center - Angle Arc

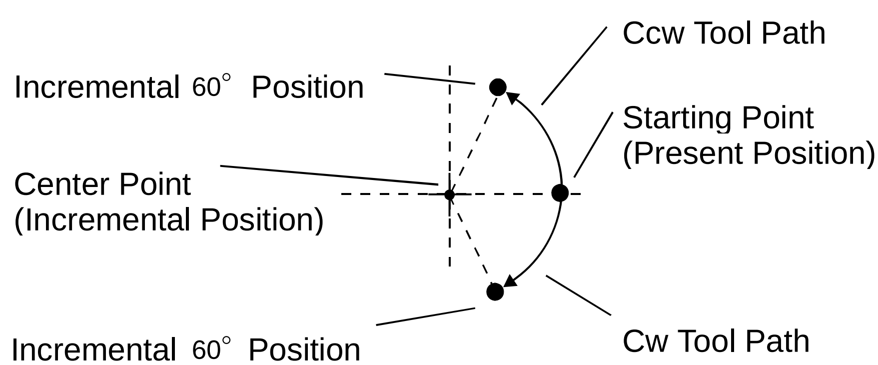
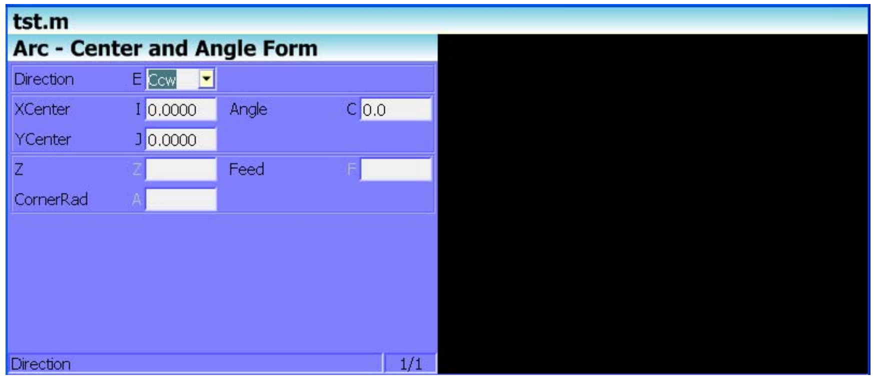


Figure 3-18, Incremental Mode, Center - Angle Arc

Refer to **Figure 3-19**.



CENTER_ANGLE

Figure 3-19, Arc – Center and Angle Graphic Menu

To program an Arc using the center and the included angle using hot keys:

1. In Edit Mode, press **3/ARC**.
2. Press **More** (**F4**) to display the More pop-up menu. Refer to **Figure 3-14, Arc More (F4) Pop-up Menu**.
3. Highlight **Arc – Center and Angle Form** and press **ENTER** to display the graphic menu. Refer to **Figure 3-19**.
4. Type the required values and settings in the entry fields.

To program an Arc using the center and the included angle using soft keys:

1. In Edit Mode, press **Milling** (**F5**) to activate the Mill secondary soft keys.
2. Press **Arc** (**F4**) to display the Arc soft keys.
3. Press **More** (**F4**) to display the More pop-up menu. Refer to **Figure 3-14, Arc More (F4) Pop-up Menu**.
4. Highlight the **Arc – Center and Angle Form** and press **ENTER** to display the Arc – Center and Angle graphic menu.

6. Type the required values or setting in the Arc – Center and Angle entry fields. Refer to **Table 3-6, Arc – Center and Angle Address Words**.

Table 3-6, Arc – Center and Angle Address Words

Label	Address Word	Description
Direction	**E**	Specifies a clockwise (**Cw**) or counterclockwise (**Ccw**) direction. Press **+/-** to toggle the setting. (Required)
XCenter	**I**	The X coordinate of the Arc's center (Required)
YCenter	**J**	The Y coordinate of the Arc's center (Required)
Angle	**C**	**I**ncluded angle of the Arc (Required)
Z	**Z**	The Z coordinate of the Arc endpoint (Optional)
CornerRad	**A**	Corner radius setting (Optional)
Feed	**F**	Feedrate (Optional)

Programming M-Code Blocks

The CNC supports M-Code functions. Enable available M-Codes at installation. Refer to the machine builder's technical data to determine which M-Codes are available.

Some programmed events initiate the same functions activated using M-Codes. Refer to **Table 3-7** for a list of the most commonly used M-Code functions.

NOTE: Leading zeros are ignored (for example, **M01** is read as **M1**).

Table 3-7, Common M-Code Functions

M Code	Programmed Event	Description
M0 or **M00**	Program stop	Program stop mode
M1 or **M01**	Program stop	Optional program stop
M3 or **M03**	MCode 3	Start **CW** spindle rotation; spindle forward
M4 or **M04**	MCode 4	Start **CCW** spindle rotation; spindle reverse
M5 or **M05**	MCode 5	Stop spindle in normal manner; spindle Off
M8 or **M08**	MCode 8	Turn coolant pump **ON**
M9 or **M09**	MCode 9	Turn coolant pump **OFF**
M19	Orients the spindle	Spindle orientation

To program an **MCode** block:

1. In Edit Mode, press **Sub Progs** (**F8**) and then press **MCode** (**F8**). The Graphic menu prompts for the **MCode** number and **X**, **Y**, **Z** values.
2. Type the values and press **Save** (**F10**) or **ENTER** to add **MCode** block to the program.

The following topic is described:

- **Dry Run M-Codes**

Dry Run M-Codes

In Dry Run Mode, the machine axes (X, Y, and Z) move through the program without cutting into the work. The CNC disables coolant operation and the work may or may not be placed on the table.

Activate Dry Run Mode with M-Codes 105 and 106. Deactivate it with M107. Refer to **Table 3-8**. Dry Run feedrates are set in the Setup Utility. They are often set at greater speeds than conventional feedrates. You can set them at any desired rate.

Table 3-8, Dry Run Mode M-Codes

M-Code	Function	Description
M105	Dry Run Mode **ON**, all axes	Enables machine Z Dry Run Mode. Program runs at dry run feedrates specified in the Setup Utility.
M106	Dry Run Mode **ON**, no Z-axis	Enables machine Dry Run Mode. Program runs at dry run feedrates specified in the Setup Utility, without moving the Z-axis.
M107	Cancel Dry Run Mode	Cancels active Dry Run Mode (Cancel M105 and M106)

Section 4 - Programming Canned Cycles

The following topics are described in this section:

- **Drill Cycles**
- **Pocket Cycles**
- **Subprograms**
- **Engraving, Repeat, and Mill Cycles**
- **Probing Cycles**

Drilling Cycles

NOTE: Program all blocks by filling in the entry fields of a Graphic Menu.

Drill cycles simplify the programming required for repetitive drilling, boring, and tapping operations. Select specific drill cycles from the Program Editor's **Drill Cycles** (**F3**) pop-up menu:

- Basic (drill cycle)
- Pecking
- Boring
- ChipBreak
- Tapping
- DrillOff
- Pattern
- BoltHole
- ThreadMill

Drill cycles are modal. When the CNC encounters a block for any type of Drill cycle, it executes that cycle at the endpoint of each subsequent move until it encounters a **DrillOff** block. To change drill cycle parameters between moves, program a new drill block.

The following topics are described:

- **Basic Drill Cycle**
- **Pecking Drill Cycle**
- **Boring Cycle**
- **Chip Break Cycle**
- **Tapping Cycle**
- **Pattern Cycle**
- **Bolt Hole Cycle**
- **Thread Milling Cycle**

Basic Drill Cycle

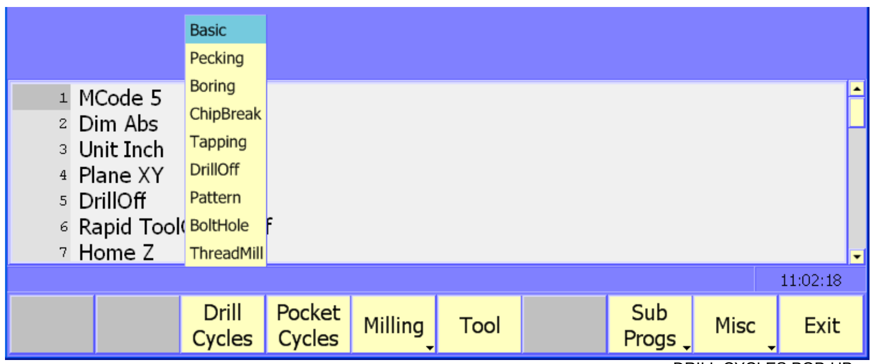
The Basic Drill Cycle is a modal operation. When the CNC receives a **BasicDrill** command, it performs the drilling operation at the endpoint of every subsequent block until it receives a **DrillOff** block. To change Basic Drilling dimensions cancel the current cycle and program a new cycle.

During the cycle, the tool rapids to the **StartHgt**, then Z feeds to **ZDepth**. To provide clearance for the next move, at the end of the cycle, the tool rapids to **ReturnHgt**.

Program a **DrillOff** block to deactivate the cycle. You can program any number of patterns and moves before turning off the cycle.

To program a **BasicDrill** block:

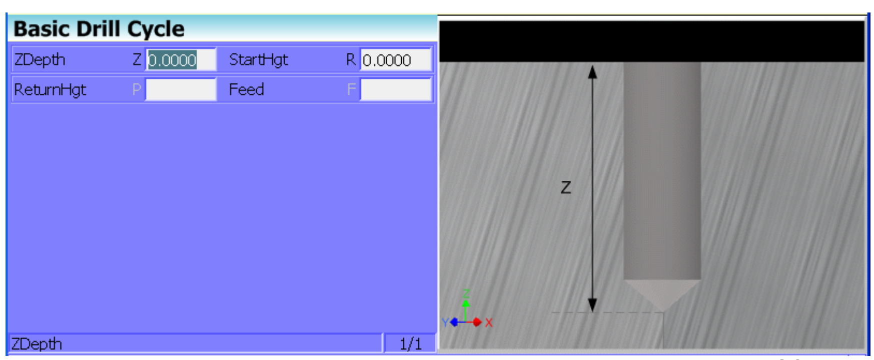
1. In Edit mode, press **Drill Cycles** (**F3**) to display the Drill Cycles pop-up menu. Refer to **Figure 4-1**.



DRILL CYCLES POP-UP

Figure 4-1, Drill Cycles (F3) Pop-up Menu

2. Highlight **Basic** and press **ENTER** to display the Basic Drill Cycle Graphic Menu. Refer to **Figure 4-2**.



BASIC DRILL

Figure 4-2, Basic Drill Cycle Graphic Menu

3. Type the required values and settings in the entry fields. With the last entry field highlighted, press **ENTER**. The display clears and the CNC adds the **BasicDrill** block to the program listing.

4. Program subsequent moves to position the tool at the required drilling location(s). The CNC will drill a hole at the endpoint of every move.
5. After programming the last drill move, press **Drill Cycles** (**F3**) to display the Drill Cycles pop-up menu. Fill in the Basic Drill Cycle entry fields. Refer to **Table 4-1**.
6. Highlight **DrillOff** and press **ENTER** to cancel the Drilling Mode.

Table 4-1, Basic Drill Cycle Address Words

Label	Address Word	Description
ZDepth	**Z**	The absolute depth of the finished hole. (Required) **NOTE: ZDepth** must be lower than **StartHgt**. **StartHgt** is 0.100 inches (2.0 mm) above the work surface.
StartHgt	**R**	The absolute Z position to which the CNC rapids to before feeding into the work. (Required)
ReturnHgt	**P**	The absolute position to which the tool returns at the end of the cycle. (Optional)
Feed	**F**	Feedrate (Optional)

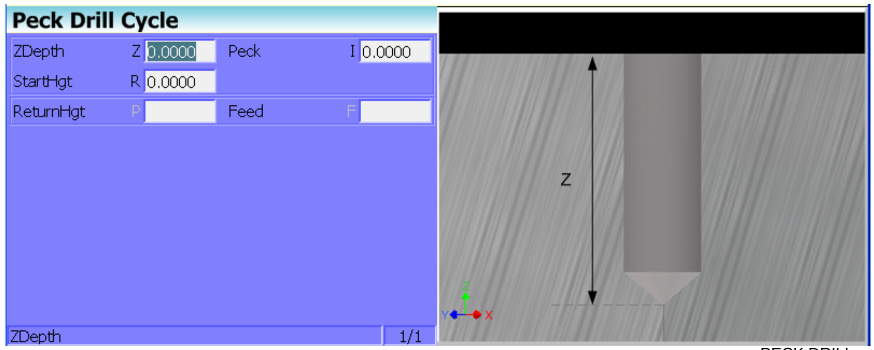
Pecking Drill Cycle

Peck drilling is a modal operation. When the CNC receives a **PeckDrill** command, it peck-drills at the endpoint of every subsequent block until it receives a **DrillOff** block. To change Peck Drilling dimensions, cancel the current **PeckDrill** cycle and program a new cycle.

The cycle starts when the CNC is in position. The tool rapids to the Z start height (**StartHgt**); feeds to the **Peck** depth; then rapids back to the **StartHgt**. This cycle repeats until the tool reaches **ZDepth**. At the end of the cycle, the tool rapids to the **ReturnHgt** to provide clearance for the next move.

To program a Peck Drilling cycle:

1. In Edit mode, press **Drill Cycles** (**F3**) to display the Drill Cycles pop-up menu. Refer to **Figure 4-1, Drill Cycles (F3) Pop-up Menu**.
2. Highlight **Pecking** and press **ENTER** to display the Peck Drill Cycle Graphic Menu. Refer to **Figure 4-3, Peck Drill Cycle Graphic Menu**.



PECK DRILL

Figure 4-3, Peck Drill Cycle Graphic Menu

3. Type the required values and settings in the entry fields. Refer to **Table 4-2**. With the last entry field highlighted, press **ENTER**. The display clears and the CNC adds the **PeckDrill** block to the program listing.
4. Program subsequent moves to position the tool at the required drilling location(s). The CNC will drill a hole at the endpoint of every move.
5. After programming the last drill move, press **Drill Cycles** (**F3**) to display the Drill Cycles pop-up menu. Refer to **Figure 4-1, Drill Cycles (F3) Pop-up Menu**.
6. Highlight **DrillOff** and press **ENTER** to cancel the Drilling Mode.

Table 4-2, Peck Drill Cycle Address Words

Label	Address Word	Description
ZDepth	**Z**	The absolute depth of the finished hole. (Required) **NOTE: ZDepth** must be lower than **StartHgt**. **StartHgt** is 0.100 inches (2.0 mm) above the work surface.
StartHgt	**R**	The absolute Z position to which the CNC rapids to before feeding into the work. (Required)
Peck	**I**	Depth drilled in each peck (Required)
ReturnHgt	**P**	The absolute position to which the tool returns at the end of the cycle. (Optional)
Feed	**F**	Feedrate (Optional)

Boring Cycle

Boring is a modal operation. When the CNC encounters a **Boring** block it executes a Boring Cycle at the endpoint of every subsequent move until it sees a **DrillOff** block. To change Boring Cycle dimensions between moves, deactivate the cycle and program a new boring block.

The cycle starts when the CNC is in position. The tool rapids to the **StartHgt**, feeds to **ZDepth**, and then feeds back to **StartHgt**. At the end of the cycle, the tool moves to **ReturnHgt** to provide clearance for the next move.

When running a **Dwell** block, the CNC pauses at **ZDepth** for the indicated time period (in seconds). **Dwell** resolution is 0.1 sec. When you type 0.0 sec., the CNC dwells until manually restarted.

To program a Boring cycle:

1. In Edit mode, press **Drill Cycles** (**F3**) to display the Drill Cycles pop-up menu. Refer to **Figure 4-1, Drill Cycles (F3) Pop-up Menu**.
2. Highlight **Boring** and press **ENTER** to display the Boring-Bidirectional Cycle Graphic Menu. Refer to **Figure 4-4**.

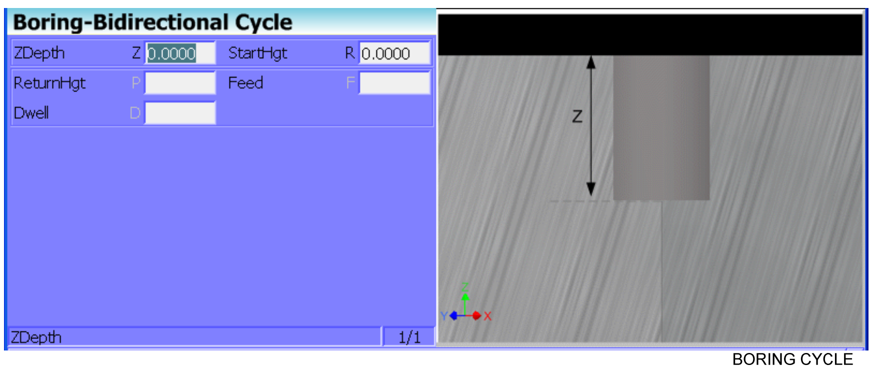


Figure 4-4, Boring Cycle Graphic Menu

3. Type the required values and settings in the entry fields. Refer to **Table 4-3, Boring Cycle Address Words**. With the last entry field highlighted, press **ENTER**. The display clears and the CNC adds **Boring** block to the program listing.
4. Program subsequent moves to position the work at the required boring locations. The CNC executes the Boring Cycle at the endpoint of every move.
5. After programming the last boring position, press **Drill Cycles** (**F3**) to display the Drill Cycles pop-up menu.
6. Highlight **DrillOff** and press **ENTER** to cancel the cycle.

Table 4-3, Boring Cycle Address Words

Label	Address Word	Description
ZDepth	**Z**	The absolute depth of the finished hole (Required) **NOTE: ZDepth** must be lower than **StartHgt**. **StartHgt** is 0.100 inches (2.0 mm) above the work surface.
StartHgt	**R**	The absolute Z position to which the CNC rapids to before feeding into the work. (Required)
ReturnHgt	**P**	Absolute position to which the tool returns at the end of the cycle. (Optional)
Dwell	**D**	Length of time for pause at **ZDepth** (Optional)
Feed	**F**	Feedrate (Optional)

Chip Break Cycle

The Chip Break Cycle is modal. Once the CNC encounters a **ChipBreak** block, it executes the Chip Break Cycle at the endpoint of each block until it sees a **DrillOff** block. To change Chip Break values between moves, deactivate the cycle and program a new one.

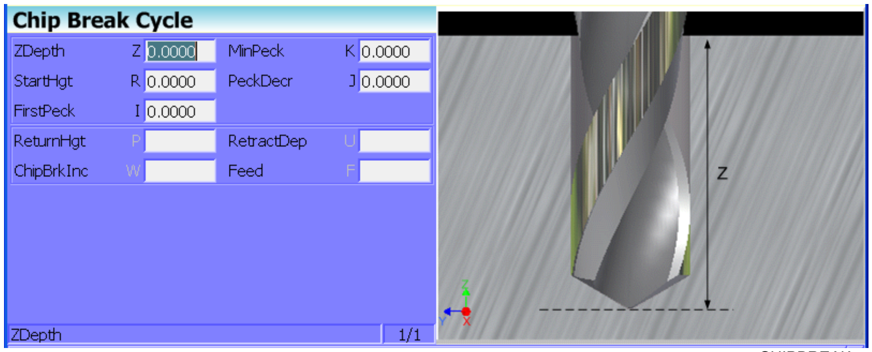
The cycle starts when the CNC is in position. The tool rapids to the **StartHgt**, feeds to the **FirstPeck**, retracts 0.02 inches [0.4 mm (default value)], then feeds to the next peck. Retract moves occur at the end of each peck in order to break the chip. This cycle repeats until the tool reaches **ZDepth**. At the end of the cycle, the tool moves to **ReturnHgt** to provide clearance for the next move.

Type a **PeckDecr** value to decrement the depth of each peck by the specified amount. The **MinPeck** sets the minimum peck the cycle can decrement. A **ChipBrkInc** is the size of the retract move that breaks the chip.

Peck to the **RetractDep**, retract to the **StartHgt**, and then peck to the next **RetractDep** increment. The first full retract occurs one **RetractDep** increment after the first peck.

To program a Chip Break cycle:

1. In Edit Mode, press **Drill Cycles** (**F3**) to display the Drill Cycles pop-up menu. Refer to **Figure 4-1, Drill Cycles (F3) Pop-up Menu**.
2. Highlight **Chip Break** and press **ENTER** to display the Chip Break Cycle Graphic Menu. Refer to **Figure 4-5, Chip Break Cycle Graphic Menu**.



CHIPBREAK

Figure 4-5, Chip Break Cycle Graphic Menu

3. Type the required values and settings in the entry fields. Refer to **Table 4-4**. With the last entry field highlighted, press **ENTER**. The display clears and the CNC adds the **Chip Break** block to the program listing.

Table 4-4, Chip Break Cycle Address Words

Label	Address Word	Description
ZDepth	**Z**	The absolute depth of the finished hole (Required) **NOTE: ZDepth** must be lower than **StartHgt**. **StartHgt** is 0.100 inches (2.0 mm) above the work surface.
StartHgt	**R**	The absolute Z position to which the CNC rapids to before feeding into the work. (Required)
FirstPeck	**I**	Absolute depth drilled in each peck (Required)
MinPeck	**K**	Smallest peck allowed (Required)
PeckDecr	**J**	Amount to subtract from previous peck (positive dimension) (Required)
ReturnHgt	**P**	Absolute position to which the tool returns at the end of the cycle. (Optional)
ChipBrkInc	**W**	Size of chip break retract (Optional)
RetracDep	**U**	Z increment before full retract (Optional)
Feed	**F**	Feedrate (Optional)

4. Program the drilling location(s). The CNC will drill a hole at the endpoint of every move until it receives the **DrillOff** command.
5. Program the **DrillOff** command. After programming the last drill move, press **Drill Cycles** (**F3**) to display the Drill Cycles pop-up menu.
6. Highlight **DrillOff** and press **ENTER** to add the **DrillOff** block to the program Listing.

Tapping Cycle

The Tapping Cycle is available only on machines equipped with spindle RPM control and M-Codes (M3, M4, and M5).

In order for the cycle to operate, you must program a **Spindle RPM** block. During execution, the CNC uses the **Spindle RPM** programmed value and the programmed threads per inch (or pitch) value from the block to calculate the proper feedrate for tapping.

When the cycle, runs the CNC rapids to the **StartHgt** and feeds to the **ZDepth**. The spindle stops and reverses direction to retract the tool from the hole. At **ReturnHgt**, the spindle stops and changes back to the original direction in preparation for the next programmed move.

Use the Tapping Cycle with any available pattern. A **DrillOff** block cancels the cycle.

NOTE:	The system supports spindle FWD (M3), REV (M$), OFF (M5), and spindle RPM control. At machine setup, the machine builder determines which M-Codes to install.

To program a **Tapping** block:

1. In Edit mode, press **Drill Cycles** (**F3**) to display the Drill Cycles pop-up menu. Refer to **Figure 4-1, Drill Cycles (F3) Pop-up Menu**.
2. Highlight **Tapping** and press **ENTER** to display the Tapping Cycle Graphic Menu. Refer to **Figure 4-6**.

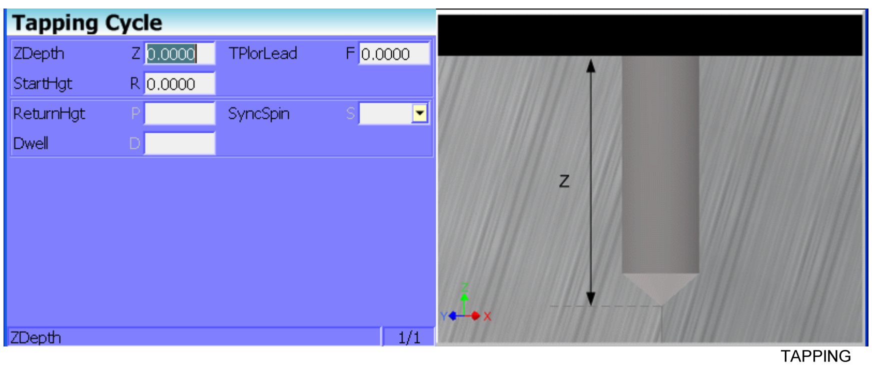


Figure 4-6, Tapping Cycle Graphic Menu

3. Type the required settings and values in the entry fields. Refer to **Table 4-5, Tapping Cycle Address Words**. With the last entry field highlighted, press **ENTER** to clear the display and add the new tapping cycle block to the program listing.

Table 4-5, Tapping Cycle Address Words

Label	Address Word	Description
ZDepth	**Z**	The absolute depth of the tapped threads. (Required) **NOTE: ZDepth** must be lower than **StartHgt**. **StartHgt** is 0.100 inches (2.0 mm) above the work surface.
StartHgt	**R**	The absolute Z position to which the CNC rapids to before feeding into the work. (Required)
TPIorLead	**F**	TPI in Inch or Lead in millimeters. (Required)
ReturnHgt	**P**	Absolute position to which the tool returns at the end of the cycle. (Optional)
Dwell	**D**	Length of time for pause at **ZDepth** and **StartHgt**. (Optional)
SyncSpin	**S**	Select **Yes** for Rigid Tapping (synchronization On) Defaults to **No** for Floating Tapping Head (No synchronization) (Optional)

4. Program subsequent moves to position the location of the tapped holes. The CNC will tap a hole at the endpoint of every move until it receives a **DrillOff** command.
5. To program a **DrillOff** block, press **Drill Cycles** (**F3**) to display the Drill Cycles pop-up menu. Refer to **Figure 4-1, Drill Cycles (F3) Pop-up Menu**.
6. Highlight **DrillOff** and press **ENTER** to display the **DrillOff** block in the program listing.

Pattern Cycle

The Pattern Cycle instructs the CNC to execute a pattern of regularly spaced moves. Locate a Pattern Cycle between a Drill Cycle and a **DrillOff** block. The CNC executes the Drill Cycle at every endpoint in the pattern.

In a Pattern Cycle, type a size, location, spacing, Tool #, and rotation angle of the pattern.

To program a Drill Pattern cycle:

1. In Edit Mode, press **Drill Cycles** (**F3**) to display the Drill Cycles pop-up menu. Refer to **Figure 4-1, Drill Cycles (F3) Pop-up Menu**.
2. Select **Pattern** from the **Drill** pop-up menu and press **ENTER** to display the Drill Pattern Cycle Graphic Menu. Refer to **Figure 4-7**.

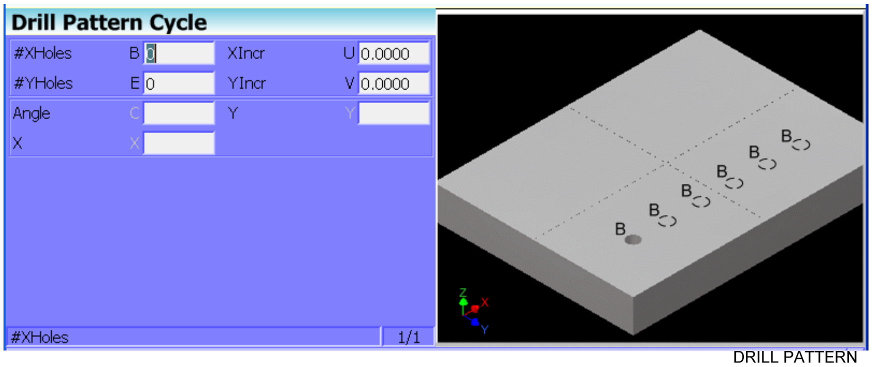


Figure 4-7, Drill Pattern Cycle Graphic Menu

3. Type the required values and settings in the entry fields. Refer to **Table 4-6**. With the last entry field highlighted, press **ENTER** to clear the display and add the new drill pattern cycle block to the program listing.

NOTE: Use a **DrillOff** block to cancel cycle.

Table 4-6, Drill Pattern Cycle Address Words

Label	Address Word	Description
#XHoles	**B**	Number of rows that lie along the X-axis. Must type value greater than 0. (Required) **NOTE:** Type **1** in either the **#XHoles** or the **#YHoles** field to drill a single row or column.
#YHoles	**E**	Number of rows that lie along the Y-axis. Must type value greater than 0. (Required) **NOTE:** Type **1** in either the **#XHoles** or the **#YHoles** field to drill a single row or column.

(Continued…)

Table 4-6, Drill Pattern Cycle Address Words (Continued)

Label	Address Word	Description
XIncr	**U**	X-axis increment (spacing) of holes (Required)
YIncr	**V**	Y-axis increment (spacing) of holes (Required)
Angle	**C**	This value rotates the pattern. The XY corner hole is the pivot, the rotation angle is the number of degrees counterclockwise from the X-axis or 3 o'clock position. (Optional)
X	**X**	X coordinate of corner hole. If no entry made, CNC puts corner hole at present location. (Optional)
Y	**Y**	Y coordinate of corner hole. If no entry made, CNC puts corner hole at present location. (Optional)

Bolt Hole Cycle

The Bolt Hole Cycle instructs the CNC to run a series of moves with endpoints that form a circular pattern. At each of these endpoints, you can run a previously programmed Drill Cycle.

You should first program a Drill Cycle to describe the hole being drilled. Then follow the Drill Cycle by one or more moves, patterns, or Bolt Hole cycles to position the CNC for the Drill Cycle. A **DrillOff** block cancels the cycle.

To program a Bolt Hole Cycle:

1. In Edit Mode, press **Drill Cycles** (**F3**) to display the Drill Cycles pop-up menu. Refer to **Figure 4-1, Drill Cycles (F3) Pop-up Menu**.
2. Select **BoltHole** from the pop-up menu and press **ENTER** to display the Drill Bolt Hole Cycle Graphic Menu. Refer to **Figure 4-8**.

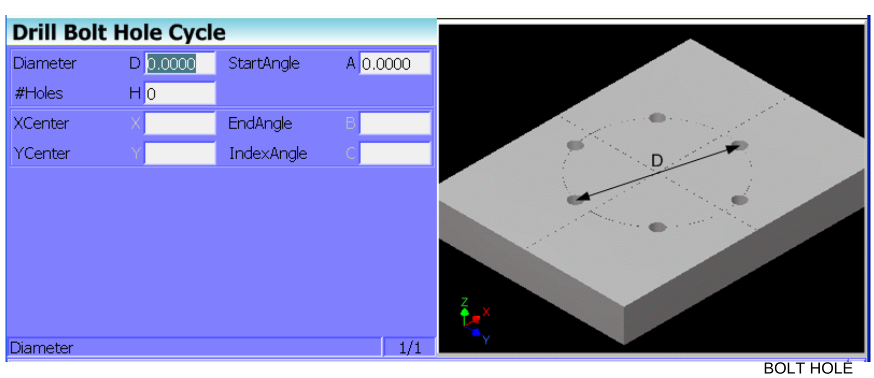


Figure 4-8, Drill Bolt Hole Cycle Graphic Menu

3. Type the following required values and settings in the entry fields. Refer to **Table 4-7**. With the last entry field highlighted, press **ENTER** to clear the display and add the new drill bolt hole cycle block to the program listing.

NOTE: Use a **DrillOff** block to cancel the cycle.

Table 4-7, Drill Bolt Hole Cycle Address Words

Label	Address Word	Description
Diameter	**D**	Diameter of the circular pattern. (Required)
#Holes	**H**	Number of equally spaced holes in the circular pattern. (Required)
StartAngle	**A**	The number of degrees (from the 3 o'clock position) to the first hole. (Required)
XCenter	**X**	Absolute X center of the bolt hole pattern. If no entry is made, the CNC puts the center of the Bolt Hole pattern at X0. (Optional) **NOTE:** Use absolute center point coordinates whenever possible.
YCenter	**Y**	Absolute Y center of the bolt hole pattern. If no entry is made, the CNC puts the center of the Bolt Hole pattern at Y0. (Optional) **NOTE:** Use absolute center point coordinates whenever possible.
EndAngle	**B**	The number of degrees (from the 3 o'clock position) to the last hole. (Optional)
IndexAngle	**C**	The number of degrees that the 3 o'clock reference position rotates around the center (rotates entire pattern). (Optional)

Thread Milling Cycle

Thread Milling Cycle simplifies the programming required to mill a thread. It will cut inside or outside, up or down, straight or tapered right or left hand, and inch or metric. Tool must be position at center of hole or boss. This can be done either by positioning before putting cycle in the program or in the cycle.

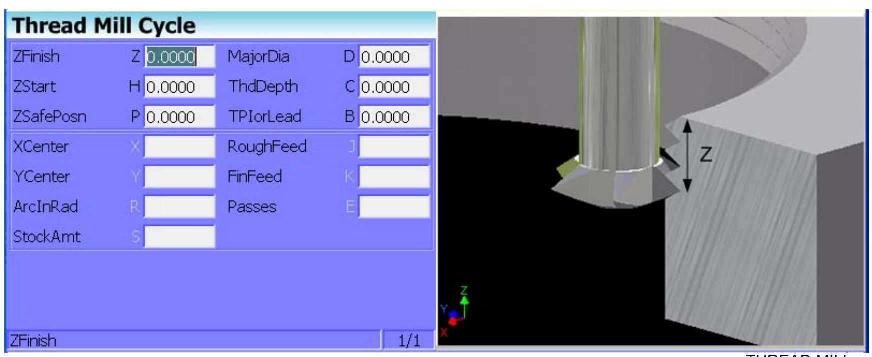
Cutter compensation is built into cycle, so cutter diameter must be entered into tool table correctly.

XCenter, **YCenter, RoughFeed, FinFeed**, and **TaperAng** are all optional Input; all other parameters must be programmed. If the feed rates are not programmed, the CNC will use last feed rate used.

Cycle will always take the final pass twice. **Diameter** is always the major diameter of thread. Inside diameter is at finished depth and outside diameter is the diameter of boss.

To program a Thread Milling Cycle:

1. In Edit Mode, press **Drill Cycles** (**F3**) to display the Drill Cycles pop-up menu. Refer to **Figure 4-1, Drill Cycles (F3) Pop-up Menu**.
2. Select **ThreadMill** from the pop-up menu, and press **ENTER** to display the Thread Mill Cycle Graphic Menu. Refer to **Figure 4-9**.



THREAD MILL

Figure 4-9, Thread Mill Cycle Graphic Menu

3. Type the following required values and settings in the entry fields. Refer to **Table 4-8**. With the last entry field highlighted, press **ENTER** to clear the display and add the new drill thread mill cycle block to the program listing.

NOTE: Use a **DrillOff** block to cancel the cycle.

Table 4-8, Thread Mill Cycle Address Words

Label	Address Word	Description
ZFinish	**Z**	Absolute Z position where the thread cut will finish. This can be above or below the start position depending on the direction of the thread cut: up or down. (Required)
ZStart	**H**	Absolute Z position where the thread cut starts. This can be above or below the finish position depending on the direction of the thread cut, up or down. If not set, cycle will use the current Z tool position. (Required)
ZSafePosn	**P**	An Absolute safe Z position above the part for rapid moves in X and/or Y. (Required) **Warning: P must be above the part to avoid a crash while positioning.**
MajorDia	**D**	Major thread Diameter. If this is a tapered thread, it is the major diameter at the Z start position. Hence, if you have a tapered hole and you start at the top and cut down, you would have a different major diameter than if you started at the bottom and cut up. A plus (+) value cuts in the CW direction and a minus (-) value cuts in the CCW direction. (Required)
ThdDepth	**C**	Depth of thread. The incremental depth of thread on one side. A plus (+) value is inside thread, a minus (-) value is outside thread. (Required)
TPIorLead	**B**	Threads per inch (TPI) or lead of thread in MM. (Required) **NOTE:** The minimum number of threads per inch is “1”.
XCenter	**X**	Absolute X coordinate of the center of the thread. If no coordinate is entered, the CNC puts the center of thread at the current tool position. (Optional)
YCenter	**Y**	Absolute Y coordinate of the center of the thread. If no coordinate is entered, the CNC puts the center of thread at the current tool position. (Optional)
ArcInRad	**R**	Size of radius arcing into start of thread. (Optional) **NOTE:** If **R** is a positive value or not set and the thread is “inside”, the cycle will always return to the center between passes. If **R** is a negative value, the cutter will move to the start or end point that is closest to the center if inside thread, and farthest away from center if outside thread. If **R** is not specified at all and the thread is outside, the cutter will back away from the largest diameter by an amount equal to the thread depth.
StockAmt	**C**	Amount to leave for a finish pass after the roughing passes. (Optional)

(Continued…)

Table 4-8, Thread Mill Cycle Address Words (Continued)

Label	Address Word	Description
RoughFeed	**J**	Feedrate for roughing. (If not set (blank), the cycle will use the current active feedrate.) (Optional)
FinFeed	**K**	Feedrate for the finish pass. (If not set (blank), the cycle will use the current active feedrate.) (Optional)
Passes	**E**	Number of roughing cuts to be taken. **NOTE:** If **Stock** is not set or set to zero and **E** is 1 or 0, the cycle will make just one pass at the full depth. If **Stock** is set to greater than zero and **E** is 1 or 0, the cycle will make one pass at the stock depth and one pass at full thread depth. **NOTE:** If you would like all non-cutting positioning moves to be rapid, set **E** to a negative number. (Optional)

Sample Thread Program

This program will cut a 10 TPI thread starting at the bottom of the hole with a single pitched toothed cutter, 1.5 diameter taking 4 cuts plus a finish cut. Cutter will cut counter clockwise with a ramp in of 0.25 inch.

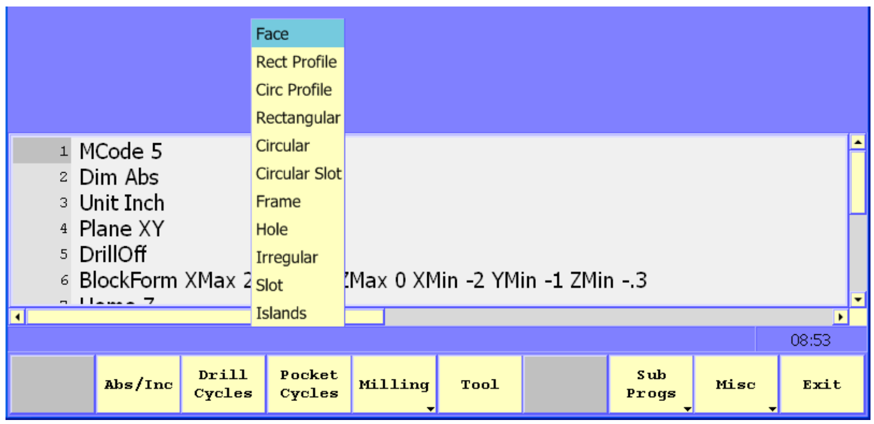
```
Dim Abs
Tool# 1
Rapid        X 1.0000 Y 1.0000
Rapid        Z 0.1000
ThreadMill StartHgt 0.1000 Diameter 1.5000 ZDepth -1.0000 Side In
Ramp 0.2500 TPIorLead 0.1000 DownUp 11 DepthPass 0.0100 Turns
11 RoughFeed 20.0 FinFeed 10.0 Passes 4
Rapid        Z 3.0000
EndMain
```

Pocket Cycles

NOTE: Program all blocks by filling in the entry fields of a Graphic Menu.

Pocket canned cycles simplify the programming of repetitive moves required to mill out pockets. Select the pocket-canned cycles from the Program Editor **Pocket Cycles** (**F4**) pop-up menu. Refer to **Figure 4-10**.

NOTE: Programming a **Tool#** in a pocket cycle automatically activates the necessary tool compensation for that cycle.



POCKET CYCLES POP-UP

Figure 4-10, Pocket Cycles (F4) Pop-up Menu

Select specific pocket cycles from the Program Editor's Pocket Cycles (**F4**) pop-up menu:

- **Face Mill Cycle**
- **Rectangular Profile Cycle**
- **Circular Profile Cycle**
- **Rectangular Pocket Cycle**
- **Circular Pocket Cycle**
- **Circular Slot Cycle**
- **Frame Pocket Cycle**
- **Hole Mill Cycle**
- **Irregular Pocket Cycle**
- **Slot Cycle**
- **Pockets with Islands**

Face Mill Cycle

Face cycles simplify the programming required to face the surface of a part.

Execution begins one tool radius from the start point. The selected step-over determines the approach axes.

NOTE: A **ZDepth** entry is not necessary if you program only one level plus finish stock.

Facing cycles can start in any corner of the surface and cut in any direction, depending on the sign (+/-) of the **Length** and **Width** values. Program a slightly oversize **Length** and **Width** to ensure complete facing of the surface.

At the end of the cycle, the tool rapids to **StartHgt**, then rapids back to the start position. Refer to **Figure 4-11**.

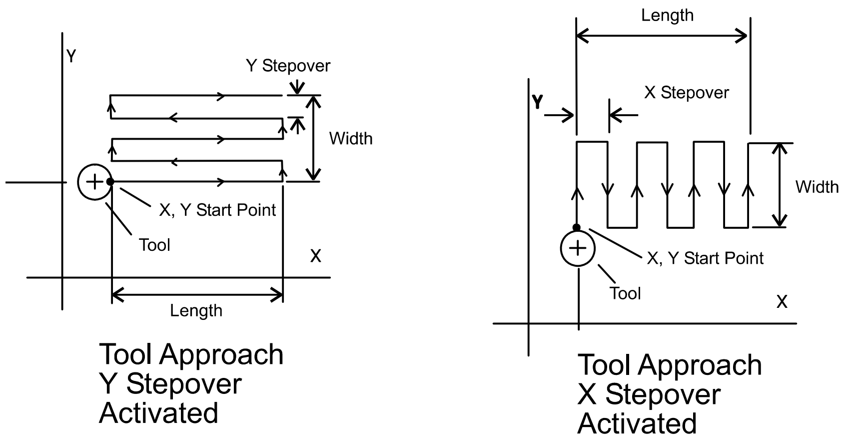


Figure 4-11, Face Mill Cycle Tool Approach

To program a Face Mill cycle:

1. In Edit Mode, press **Pocket Cycles** (**F4**) to display the pop-up menu. Refer to **Figure 4-10, Pocket Cycles (F4) Pop-up Menu**.
2. Move the highlight to select **Face** and press **ENTER** to display the Face Mill Cycle Graphic Menu. Refer to **Figure 4-12**.

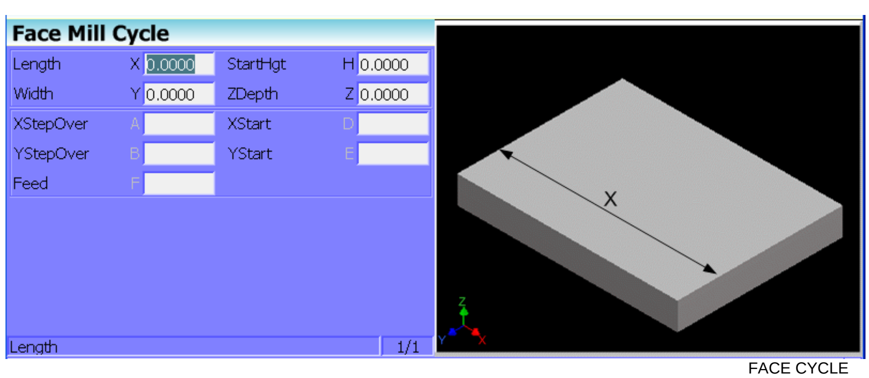


Figure 4-12, Face Mill Cycle Graphic Menu

3. Type the required values and settings in the entry fields. Refer to **Table 4-9**.

Table 4-9, Face Mill Cycle Address Words

Label	Address Word	Description
Length	**X**	X-axis length to be faced. (Required)
Width	**Y**	Y-axis length to be faced. (Required)
StartHgt	**H**	The Absolute Z position to which the CNC rapids before feeding into the work. This must be 0.1 inch (or 2 mm) above the surface. Executed in rapid. (Required)
ZDepth	**Z**	Absolute depth of the finished surface. (Required) **NOTE: ZDepth** must be lower than **StartHgt**. **StartHgt** is 0.1 Inch (2.0 mm) above the work surface.
XStepOver	**A**	Width of cut in the X-axis direction. When you do not enter a value, the CNC defaults to 70% of the active tool radius. Maximum step-over permitted is 70% of the active tool radius. (Optional)
YStepOver	**B**	Width of cut in the Y-axis direction. When you do not enter a value, the CNC defaults to 70% of the active tool radius. Maximum stepover permitted is 70% of the active tool radius. (Optional)
Feed	**F**	Feedrate used in cycle. (Optional)
XStart	**D**	X coordinate of the starting point. Defaults to current position. (Optional) **NOTE:** Type the required absolute **XStart** and **YStart** coordinates when possible.
YStart	**E**	Y coordinate of the starting point. Defaults to current position. (Optional) **NOTE:** Type the required absolute **XStart** and **YStart** coordinates when possible.

NOTE: Enter either an **XStepover** or **YStepover**. Do not enter both.

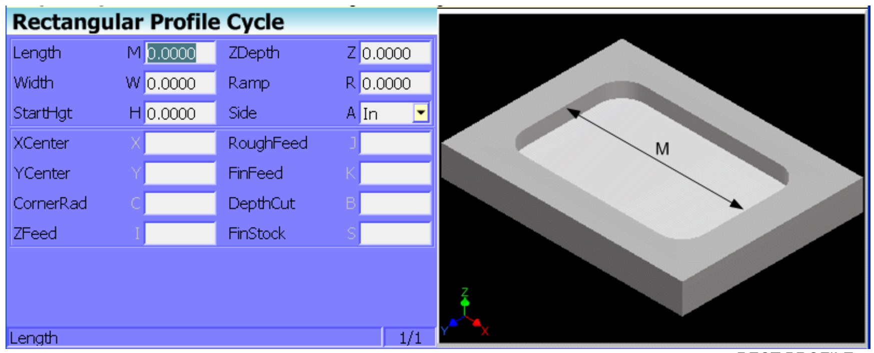
NOTE: The Program Editor will allow you to inadvertently write a block containing a stepover value greater than 70% of the active tool radius. Test a program in the Draw Graphics Mode to reveal this type of error.

Rectangular Profile Cycle

The Rectangular Profile Cycle cleans up the inside or outside profile of a rectangle. When this cycle runs, the CNC rapids to the Ramp #1 starting position, rapids to **StartHgt**, then feeds to the depth of the first cut.

The machine feeds into the profile along Ramp #1, cuts the rectangle to the **Length** and **Width** specified, and then ramps away from the work along Ramp #2.

When cutting an inside profile, the Rectangular Profile Cycle Graphic Menu displays ramp moves. Refer to **Figure 4-13**.



RECT PROFILE

Figure 4-13, Rectangular Profile Cycle Graphic Menu

When cutting an outside profile, the tool ramps into the profile along Ramp #1 and away from the profile along Ramp #2, as shown in **Figure 4-14**.

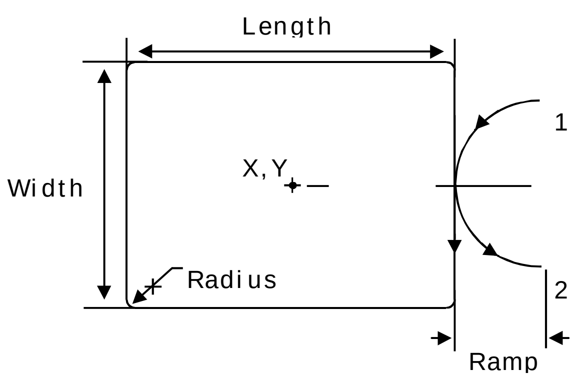


Figure 4-14, Outside Profile Ramp Moves

The Rectangular Profile Cycle automatically compensates for tool diameter. Activate the correct tool diameter before or within the **ProfRect** block.

When you type a **DepthCut** value, the CNC executes the number of passes required to get from the **StartHgt** to the **ZDepth**, cutting the **DepthCut** on each pass.

When you type a **FinStock** value, the CNC leaves the specified stock on the profile and depth for a finish pass. The CNC cuts the rectangle to the **Length**, **Width**, and **ZDepth** dimensions on the finish pass. Type a negative **FinStock** to leave the finish stock without making a finish pass.

When you do not type a **RoughFeed** or **FinFeed**, the CNC executes feed moves at the current feedrate. **RoughFeed** controls the feedrate of the roughing cycle. **FinFeed** controls the feedrate of the finishing cycle.

To program a Rectangular Profile cycle:

In Edit Mode, press **Pocket Cycles** (**F4**) to display the Pocket Cycles pop-up menu. Refer to **Figure 4-10, Pocket Cycles (F4) Pop-up Menu**.

1. Highlight **Rect.Profile** and press **ENTER**. The Rectangular Profile Cycle Graphic Menu prompts for labeled values.
2. Type the required Rectangular Profile Cycle values and settings in the entry fields. Refer to **Table 4-10**.

Table 4-10, Rectangular Profile Cycle Address Words

Label	Address Word	Description
Length	**X**	X-axis length to be faced (Required)
Width	**Y**	Y-axis length to be faced (Required)
StartHgt	**H**	The Absolute Z position before beginning the facing cycle. This must be 0.1 inch (or 2 mm) above the surface. Executed in rapid. (Required)
ZDepth	**Z**	Absolute depth of the finished profile. (Required) **NOTE: ZDepth** must be lower than **StartHgt**. **StartHgt** is 0.1 Inch (2.0 mm) above the work surface.
Ramp	**R**	Radius of the ramping moves. (Required)
Side	**A**	Toggles cutting mode between the inside (In) or outside (Out) of the profile. Press **+/-** to toggle the selection. (Required)
XCenter	**X**	X coordinate of the center. If no coordinate is typed, the CNC centers the pocket at its present position. (Optional)
YCenter	**Y**	Y coordinate of the center. If no coordinate is typed, the CNC centers the pocket at its present position. (Optional)
CornerRad	**C**	Corner radius setting. If you type a negative value, both direction of cut and the starting and endpoints reverse. (Optional)
ZFeed	**I**	Z-axis feedrate (Optional)
RoughFeed	**J**	Rough pass feedrate (Optional)
FinFeed	**K**	Finish pass feedrate (Optional)
DepthCut	**B**	Z-axis increment used for each pass. (Optional)
FinStock	**S**	Amount of stock left by the machine before the finish pass. Default: **0**. If you type a negative value, the CNC leaves the stock without making a finish pass. (Optional)

Circular Profile Cycle

The Circular Profile Cycle cleans up the inside or outside profile of an existing circle. Refer to **Figure 4-15**.

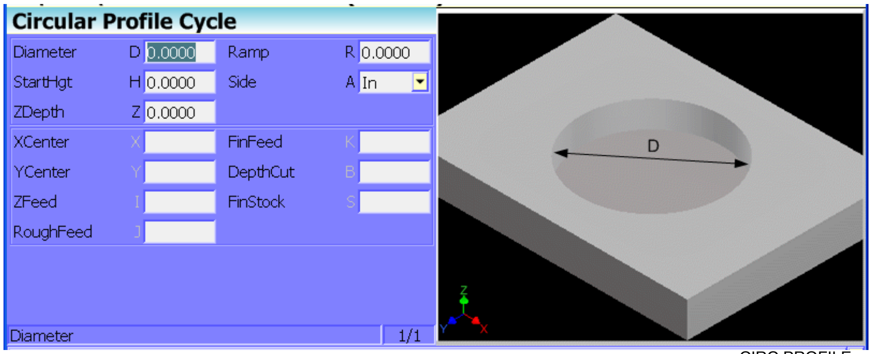


Figure 4-15, Circular Profile Cycle Graphic Menu

When executed, the CNC rapids to Ramp#1 starting position, rapids to **StartHgt**, then feeds to the depth of the first cut.

The machine feeds into the profile along Ramp #1, cuts the circle to the **Diameter** specified, and then ramps away from the work along Ramp #2.

When cutting an outside profile, the tool ramps into the work along Ramp #1 and away from the work along Ramp #2 as shown in **Figure 4-16**.

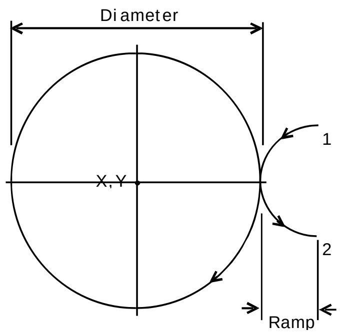


Figure 4-16, Ramp Position for Outside Profile

The Circular Profile Cycle automatically compensates for tool diameter. Activate the correct tool diameter before or within the **ProfCirc** block.

If you type a **DepthCut**, the CNC executes the number of passes required to get from the **StartHgt** to the **ZDepth**, cutting to the **DepthCut** on each pass.

When you type a **FinStock** value, the CNC leaves the specified stock on the profile and depth for a finish pass. The CNC finishes to the typed diameter on the finish pass. Type a negative **FinStock** to leave the finish stock without making a finish pass.

If you do not type a **RoughFeed** or **FinFeed**, the CNC executes feed moves at the current feedrate. **RoughFeed** controls feedrate of the roughing cycle. **FinFeed** controls the feedrate of the finishing cycle.

To program a Circular Profile cycle:

1. In Edit Mode, press **Pocket Cycles** (**F4**) to display the Pocket Cycles pop-up menu. Refer to **Figure 4-10, Pocket Cycles (F4) Pop-up Menu**.
2. Highlight **Circ Profile** and press **ENTER** to display the Circular Profile Cycle Graphic Menu. Refer to **Figure 4-15, Circular Profile Cycle Graphic Menu**.
3. Type the required Circular Profile Cycle values and settings in the entry fields. Refer to **Table 4-11**.

Table 4-11, Circular Profile Cycle Address Words

Label	Address Word	Description
Diameter	**X**	Finished diameter of circle. If you type a negative value, both the direction of cut and the starting and endpoints reverse. (Required)
StartHgt	**H**	Absolute Z position to which the CNC rapids before feeding into work. This must be 0.1 inch (or 2 mm) above the surface. Executed in rapid. (Required)
ZDepth	**Z**	Absolute depth of the finished profile. (Required) **NOTE: ZDepth** must be lower than **StartHgt**. **StartHgt** is 0.1 Inch (2.0 mm) above the work surface.
Ramp	**R**	Radius of the ramp into and away from the cut. (0 value allowed). (Required)
Side	**A**	Selection for cutting on the inside of the profile (In) or the outside (Out). Press **+/-** to toggle the selection. (Required)
XCenter	**X**	X coordinate of the center. If no coordinate is typed, the CNC centers the pocket at its present position. Default: Present position (Optional)
YCenter	**Y**	Y coordinate of the center. If no coordinate is typed, the CNC centers the pocket at its present position. Default: Present position (Optional)
ZFeed	**I**	Z-axis feedrate (Optional)
RoughFeed	**J**	Rough pass feedrate (Optional)
FinFeed	**K**	Finish pass feedrate (Optional)
DepthCut	**B**	Z-axis increment used for each pass (Optional)

(Continued…)

Table 4-11, Circular Profile Cycle Address Words (Continued)

Label	Address Word	Description
FinStock	**S**	Amount of stock left by the machine before the finish pass. Default: **0**. If you type a negative value, the CNC leaves the stock without making a finish pass. (Optional)

Rectangular Pocket Cycle

Rectangular Pocket cycles simplify the programming required to mill out rectangular pockets. When executed, the CNC rapids to the center of the lower left radius, rapids to the **StartHgt**, then ramps into the work toward the narrow center of the pocket. From the pocket center, the CNC mills increasingly larger rectangles until it reaches the specified **Length** and **Width**.

The Rectangular Pocket Cycle automatically compensates for tool diameter. Activate the correct tool diameter before the **RectPock** block.

If you type **DepthCut**, the CNC executes the number of passes required to get from the **StartHgt** to the **ZDepth**, cutting the **DepthCut** on each pass.

Use **FinStock** to leave the specified stock on the profile and depth for a finish pass. The CNC cuts the rectangle to the **Length**, **Width**, and **ZDepth** dimensions on the finish pass. Type a negative **FinStock** to leave the finish stock without adding a finish pass.

If you do not type a **RoughFeed** or **FinFeed** value, the CNC executes feed moves at the current feedrate. **RoughFeed** controls the feedrate of the roughing cycle. **FinFeed** controls the feedrate of the finishing cycle.

To program a Rectangular Pocket cycle:

1. In Edit Mode, press **Pocket Cycles** (**F4**) to display the Pocket Cycles pop-up menu. Refer to **Figure 4-10, Pocket Cycles (F4) Pop-up Menu**.
2. Highlight **Rectangular** and press **ENTER** to display the Rectangular Pocket Cycle Graphic Menu. Refer to **Figure 4-17**.

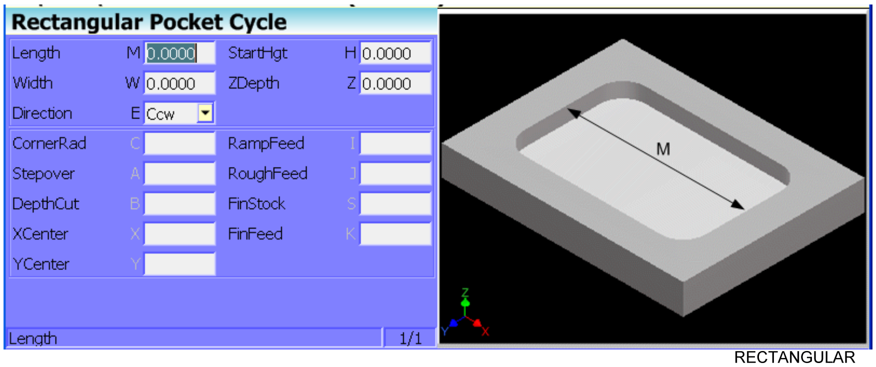


Figure 4-17, Rectangular Pocket Cycle Graphic Menu

3. Type the required Rectangular Pocket Cycle values and settings in the entry fields. Refer to **Table 4-12**.

Table 4-12, Rectangular Pocket Cycle Address Words

Label	Address Word	Description
Length	**M**	Inside Y length of the finished pocket (Required)
Width	**W**	Inside X length of the finished pocket (Required)
Direction	**E**	Allows you to select a clockwise (**Cw**) or counterclockwise (**Ccw**) direction. Press **+/-** to toggle the selection. (Required)
StartHgt	**H**	Absolute Z position to which the CNC rapids before feeding into work. This must be 0.1 inch (or 2 mm) above the surface. Executed in rapid. (Required)
ZDepth	**Z**	Absolute depth of the finished pocket. (Required) **NOTE: ZDepth** must be lower than **StartHgt**. **StartHgt** is 0.1 Inch (2.0 mm) above the work surface.
CornerRad	**C**	Actual corner radius of pocket (all four corners will be same). Must be equal to or greater than tool radius. Defaults to tool radius. (Optional)
Stepover	**A**	Width of cut. If you do not type a value, the CNC defaults to 70% of the active tool radius. The maximum step-over permitted is 70% of the active tool diameter. (Optional)
DepthCut	**B**	Depth the machine takes in a single pass. Defaults to a single **ZDepth** cut minus the finish stock. (Optional)
XCenter	**X**	X coordinate of the center. If no coordinate is typed, the CNC centers the pocket at its present position. Default: Present position (Optional)
YCenter	**Y**	Y coordinate of the center. If no coordinate is typed, the CNC centers the pocket at its present position. Default: Present position (Optional)
RampFeed	**I**	Ramp in feed: The tool will ramp into the first depth of cut with an XYZ move from the centerline of the lower-left radius toward the center of the pocket. The feedrate for this move is programmed as I. Defaults to last programmed feedrate. (Optional)
RoughFeed	**J**	Rough pass feedrate (Optional)
FinStock	**S**	Amount of stock left by the machine before the finish pass. Default: **0**. If you type a negative value, the CNC leaves the stock without making a finish pass. (Optional)
FinFeed	**K**	Finish pass feedrate (Optional)

Circular Pocket Cycle

Circular Pocket cycles simplify the programming of circular pockets. When executed, the CNC rapids to the center, rapids to the **StartHgt**, and then ramps into the work. The tool will circle outward from the center starting position, until it reaches the pocket **Diameter**. The tool circles back toward the center until the pass is complete.

The Circular Pocket Cycle automatically compensates for tool diameter. Activate the correct tool diameter before the **CircPock** block.

Use **DepthCut** to specify the number of passes required to get from the **StartHgt** to the **ZDepth**, cutting the **DepthCut** on each pass.

Use **FinStock** to leave the specified stock on the profile and depth for a finish pass. The cycle cuts the profile to the **Diameter** and **ZDepth** dimensions on the finish pass. Type a negative **FinStock** to leave the finish stock without making a finish pass.

Leave **RoughFeed** and **FinFeed** blank to execute feed moves at the current feedrate. **RoughFeed** controls the feedrate of the roughing cycle. **FinFeed** controls the feedrate of the finishing cycle.

To program a Circular Pocket cycle:

1. In Edit Mode, press **Pocket Cycles** (**F4**) to display the Pocket Cycles pop-up menu. Refer to **Figure 4-10, Pocket Cycles (F4) Pop-up Menu**.
2. Move the highlight to select **Circular** and press **ENTER** to display the Circular Pocket Cycle Graphic Menu. Refer to **Figure 4-18**.

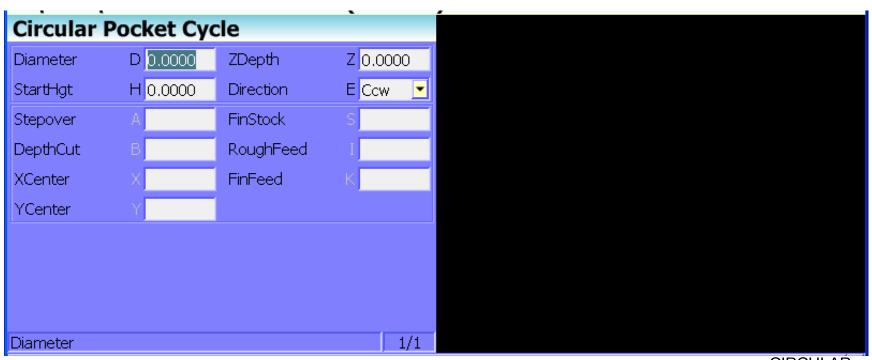


Figure 4-18, Circular Pocket Cycle Graphic Menu

3. Type the required Circular Pocket Cycle values and settings in the entry fields. Refer to **Table 4-13, Circular Pocket Cycle Address Words**.

Table 4-13, Circular Pocket Cycle Address Words

Label	Address Word	Description
Diameter	**D**	Diameter of pocket. The direction CCW (climb milling) is reversible (Required): +D dimension = climb (CCW) -D dimension = conventional (CW)
StartHgt	**H**	Absolute Z position to which the CNC rapids before feeding into work. This must be 0.1 inch (or 2 mm) above the surface. Executed in rapid. (Required)
ZDepth	**Z**	Absolute depth of the finished hole. (Required) **NOTE: ZDepth** must be lower than **StartHgt**. **StartHgt** is 0.1 Inch (2.0 mm) above the work surface.
Direction	**E**	Allows you to select a clockwise (**Cw**) or counterclockwise (**Ccw**) direction. Press **+/-** to toggle the selection. (Required)
Stepover	**A**	Width of cut. If you do not type a value, the CNC defaults to 70% of the active tool radius. The maximum step-over permitted is 70% of the active tool diameter. (Optional)
DepthCut	**B**	Depth the machine takes in a single pass. Defaults to a single **ZDepth** cut minus the finish stock. (Optional)
XCenter	**X**	X coordinate of the center. If no coordinate is typed, the CNC centers the pocket at its present position. Default: Present position (Optional)
YCenter	**Y**	Y coordinate of the center. If no coordinate is typed, the CNC centers the pocket at its present position. Default: Present position (Optional)
FinStock	**S**	Amount of stock left by the machine before the finish pass. Default: **0**. If you type a negative value, the CNC leaves the stock without making a finish pass. (Optional)
RampFeed	**I**	Ramp in feed: The tool will ramp into the first depth of cut with an XYZ move from the centerline of the lower-left radius toward the center of the pocket. The feedrate for this move is programmed as I. Defaults to last programmed feedrate. (Optional)
RoughFeed	**J**	Rough pass feedrate (Optional)
FinFeed	**K**	Finish pass feedrate (Optional)

Circular Slot Cycle

Circular Slot cycles simplify the programming required to mill out a circular slot. When executed, the CNC rapids to a location above the slot, rapids to **StartHgt**, then plunges into the work piece. The CNC mills out the interior of the slot to the specified **Width** and **SweepAngle**.

The Circular Slot Cycle automatically compensates for tool diameter. Activate the correct tool diameter before the **CircSlot** block.

If you type **DepthCut**, the CNC executes the number of passes required to get from the **StartHgt** to the **ZDepth**, cutting the **DepthCut** on each pass.

Use **FinStock** to leave the specified stock on the profile and depth for a finish pass. The CNC cuts the slot to the **Width** and **SweepAngle** dimensions on the finish pass.

If you do not type a **RoughFeed** or **FinFeed** value, the CNC executes feed moves at the current feedrate. **RoughFeed** controls the feedrate of the roughing cycle. **FinFeed** controls the feedrate of the finishing cycle.

Roughing will always climb mill and finish will always conventional mill unless a negative **K** (**FinFeed**) value is used. If a negative **K** (**FinFeed**) is used, the finish pass will also climb mill. A **K** (**FinFeed**) value of 0 will leave the finish stock without adding a finish pass.

When the cycle completes, the CNC rapids to the **StartHgt**.

To program an Circular Slot cycle:

1. In Edit Mode, press **Pocket Cycles** (**F4**) to display the Slot Cycle pop-up menu. Refer to **Figure 4-10, Pocket Cycles (F4) Pop-up Menu**.
2. Highlight **Circular Slot** and press **ENTER** to display the Circular Slot Cycle Graphic Menu. Refer to **Figure 4-19**.

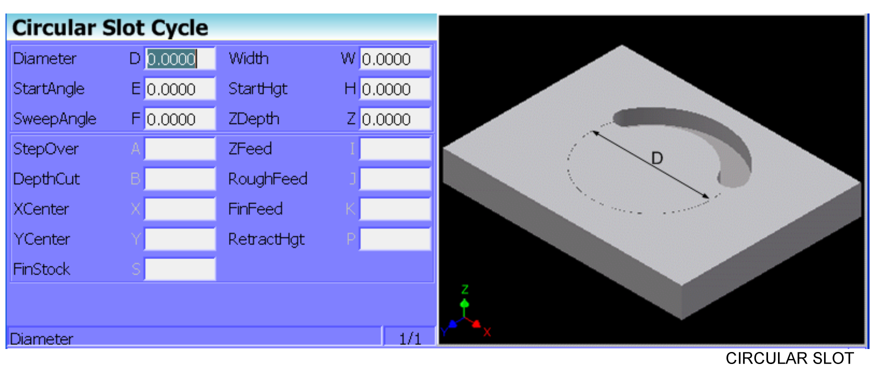


Figure 4-19, Circular Slot Graphic Menu

3. Type the required Circular Slot Cycle values and settings in the entry fields. Refer to **Table 4-14, Circular Slot Address Words**.

Table 4-14, Circular Slot Address Words

Label	Address Word	Description
Diameter	**D**	Diameter of the slot circle. The diameter must be larger than the slot width. (Required)
StartAngle	**E**	The angle in degrees to the slot's first end. (Required)
SweepAngle	**F**	Sweep angle of the slot measured in degrees between the two ends. **SweepAngle** (**F**) is applied CCW from **StartAngle** (**E**) regardless of the sign of its value. Must be non-zero. (Required)
Width	**W**	Width of the slot. (Required)
StartHgt	**H**	The absolute Z position before beginning to mill the slot. This must be 0.1 inch (or 2 mm) above the surface. (Required)
ZDepth	**Z**	The absolute depth of the slot. Must be below the **StartHgt** (**H**). (Required)
StepOver	**A**	Maximum tool step over in the XY plane. Must be 50% or less of the tool diameter. The distance the tool will step over (width of cut) as it mills out the slot. Default: 50% of tool diameter (Optional)
DepthCut	**B**	Maximum Z depth per pass the CNC will cut while roughing. Default: **ZDepth** less finish stock (Optional)
XCenter	**X**	Center of the slot circle. Default: Current X position (Optional)
YCenter	**Y**	Center of the slot circle. Default: Current Y position (Optional)
FinStock	**S**	Finish–stock amount per side and bottom. Default: No finish stock (Optional)
ZFeed	**I**	Z-axis feed rate while plunging. Default: Current Z-axis feed rate (Optional)
RoughFeed	**J**	Rough-pass feed rate. Default: Current feed rate (Optional)
FinFeed	**K**	Finish-pass feed rate. If negative, the finish pass will also climb mill (CW). If zero, the finish stock will not be removed. (Optional)

Frame Pocket Cycle

A Frame Pocket Cycle simplifies the programming required to mill out a Frame. When executed, the CNC rapids to a starting position near the island, rapids to **StartHgt**, then ramps into the work while moving across the **Frame**. The CNC cuts from the outside edge to the island in rectangles of decreasing size to complete the pass.

The Frame Pocket Cycle automatically compensates for tool diameter. Activate the correct tool diameter before the **FramePock** block.

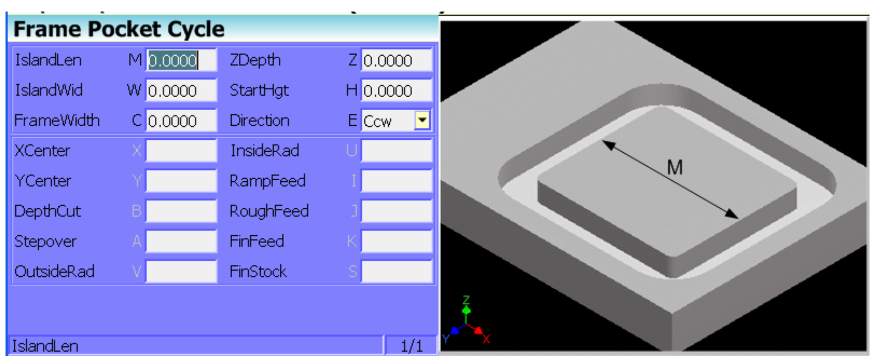
If you type a **DepthCut** value, the CNC executes the number of passes required to get from the **StartHgt** to the **ZDepth**, cutting the **DepthCut** on each pass.

Use **FinStock** to leave the specified stock on the profile and depth for a finish pass. The CNC cuts the frame to the **IslandLen**, **IslandWid**, and **FrameWidth** dimensions on the finish pass. Type a negative **FinStock** value to leave the finish stock without adding a finish pass.

Leave **RoughFeed** and **FinFeed** blank to execute feed moves at the current feedrate. **RoughFeed** controls the feedrate of the roughing cycle. **FinFeed** controls the feedrate of the finishing cycle.

To program a Frame Pocket cycle:

1. In Edit Mode, press **Pocket Cycles** (**F4**) to display the Pocket Cycles pop-up menu. Refer to **Figure 4-10, Pocket Cycles (F4) Pop-up Menu**.
2. Highlight **Frame** and press **ENTER** to display the Frame Pocket Cycle Graphic Menu. Refer to **Figure 4-20**.



FRAME

Figure 4-20, Frame Pocket Cycle Graphic Menu

3. Type the required Frame Pocket Cycle values and settings in the entry fields. Refer to **Table 4-15, Frame Pocket Cycle Address Words**.

Table 4-15, Frame Pocket Cycle Address Words

Label	Address Word	Description
IslandLen	**M**	Outside length (X-axis) of finished island. (Required)
IslandWid	**W**	Outside width (Y-axis) of finished island. (Required)
FrameWidth	**C**	Width of the finished frame. (Required)
ZDepth	**Z**	Absolute depth of the finished pocket. (Required) **NOTE: ZDepth** must be lower than **StartHgt**. **StartHgt** is 0.1 Inch (2.0 mm) above the work surface.
StartHgt	**H**	Absolute Z position to which the CNC rapids before feeding into work. This must be 0.1 inch (or 2 mm) above the surface. Executed in rapid. (Required)
Direction	**E**	Allows you to select conventional or climb milling for the pocket. The selections are clockwise (**Cw**) and counterclockwise (**Ccw**). Press **+/-** to toggle the selection. (Required)
XCenter	**X**	X coordinate of the center. If no coordinate is typed, the CNC centers the pocket at its present position. Default: Pocket centers at present position (Optional)
YCenter	**Y**	Y coordinate of the center. If no coordinate is typed, the CNC centers the pocket at its present position. Default: Pocket centers at present position (Optional)
DepthCut	**B**	Z-axis increment used for each pass. Depth the machine takes in a single pass. Defaults to a single **ZDepth** cut minus the finish stock. (Optional)
Stepover	**A**	Width of cut. If you do not type a value, the CNC defaults to 70% of the active tool radius. The maximum step-over permitted is 70% of the active tool diameter. (Optional)
OutsideRad	**V**	Outside radius of the frame corners. (Optional)
InsideRad	**U**	Radius of the island corners. (Optional)
RampFeed	**I**	Z-axis feedrate. Ramp in feed: The tool will ramp into the first depth of cut with an XYZ move from the centerline of the lower-left radius toward the center of the pocket. The feedrate for this move is programmed as I. Defaults to last programmed feedrate. (Optional)
RoughFeed	**J**	Rough pass feedrate (Optional)
FinFeed	**K**	Finish pass feedrate (Optional)
FinStock	**S**	Amount of stock left by the machine before the finish pass. Default: **0**. If you type a negative value, the CNC leaves the stock without making a finish pass. (Optional)

Hole Mill Cycle

Use Hole Mill cycles to cut through holes, clean up the inside diameter of existing holes, or counter-bore existing holes. When executed the CNC rapids to the ramp, feeds into the circumference along the ramp, and cuts to the **Diameter**. After it completes the hole, the CNC ramps away from the circumference and rapids back to the center.

The Hole Mill Cycle automatically compensates tool diameter. Activate the correct tool diameter before the **HolePock** block.

Use **StartHgt** and **ZDepth** together, if at all. Type a **DepthCut** to execute the number of passes required to get from the **StartHgt** to the **ZDepth**, cutting the **DepthCut** amount on each pass.

Use **FinStock** to leave the specified amount on the profile and make an additional pass cutting to the **Diameter**. Type a negative **FinStock** value to leave finish stock, without executing a finish pass.

Leave **RoughFeed** and **FinFeed** blank to execute feed moves at the current feedrate. **RoughFeed** controls the feedrate of the roughing cycle. **FinFeed** controls the feedrate of the finishing cycle.

To program a Hole Mill Cycle:

1. In Edit Mode, press **Pocket Cycles** (**F4**) to display the Pocket Cycles pop-up menu. Refer to **Figure 4-10, Pocket Cycles (F4) Pop-up Menu**.
2. Highlight **Hole** and press **ENTER** to display the Hole Mill Cycle Graphic Menu. Refer to **Figure 4-21**.

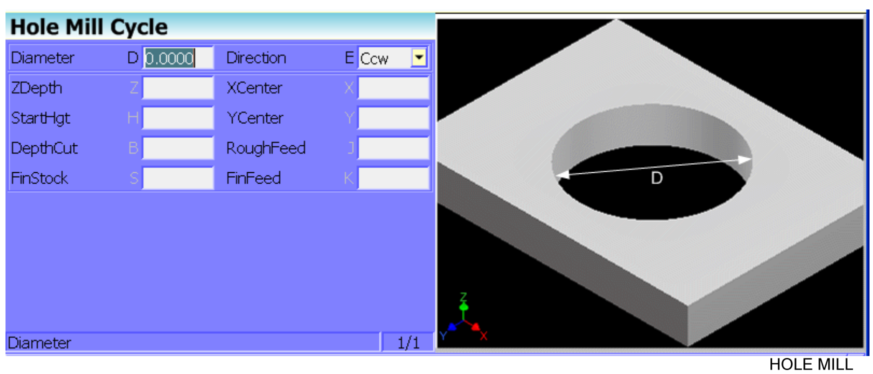


Figure 4-21, Hole Mill Cycle Graphic Menu

3. Fill in the **HOLE-MILL POCKET** entry fields:
4. Type the required Hole Mill Cycle values and settings in the entry fields. Refer to **Table 4-16, Hole Mill Cycle Address Words**.

Table 4-16, Hole Mill Cycle Address Words

Label	Address Word	Description
Diameter	**D**	Diameter of the pocket (Required)
Direction	**E**	Allows you to select a clockwise (**Cw**) or counterclockwise (**Ccw**) direction. Press **+/-** to toggle the selection. (Required)
ZDepth	**Z**	Absolute depth of the finished pocket. (Optional) **NOTE: ZDepth** must be lower than **StartHgt**. **StartHgt** is 0.1 Inch (2.0 mm) above the work surface. Use **StartHgt** and **ZDepth** together, if at all.
StartHgt	**H**	Absolute Z position to which the CNC rapids before feeding into work. This must be 0.1 inch (or 2 mm) above the surface. Executed in rapid. (Optional) **NOTE:** Use **StartHgt** and **ZDepth** together, if at all.
DepthCut	**B**	Z-axis increment used for each pass. Depth the machine takes in a single pass. Defaults to a single **ZDepth** cut minus the finish stock. (Optional)
FinStock	**S**	Amount of stock left by the machine before the finish pass. Default: **0**. If you type a negative value, the CNC leaves the stock without making a finish pass. (Optional)
XCenter	**X**	X coordinate of the center. If no coordinate is typed, the CNC centers the pocket at its present position. Default: Present position (Optional)
YCenter	**Y**	Y coordinate of the center. If no coordinate is typed, the CNC centers the pocket at its present position. Default: Present position (Optional)
RoughFeed	**J**	Rough pass feedrate (Optional)
FinFeed	**K**	Finish pass feedrate (Optional)

Irregular Pocket Cycle

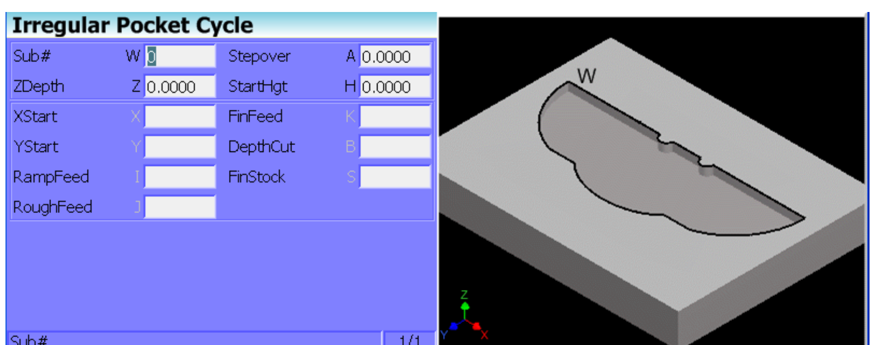
Use **Irregular Pocket Cycle** to mill irregular pockets. You must enter the perimeter of the shape into a subprogram. The main irregular pocket needs to be a closed contiguous line and arc movements starting and ending at the same point. The first line in the input subroutine for outside shape or islands needs a **Line ToolComp Left** [use **Milling** (**F5**)>**Line** (**F3**)] or **Line ToolComp Right** to indicate which side of the contour the cutter needs to be, as viewed from the direction of travel. No ramp on or off movement is allowed. The cycle will calculate these moves on and off the defined shape. Do not include feedrates in the subprogram, only the exact perimeter of the pocket. In a closed shape, the start point of the first (rapid) move and the endpoint of the last move (line or arc) are the same. The CNC will automatically calculate the moves necessary to clear out the shape. Refer to **Table 4-17, Irregular Pocket Cycle Address Words**.

Roughing will always climb mill and finish will always conventional mill unless a negative **K** (**FinFeed**) value is used. If a negative **K** (**FinFeed**) is used, the finish pass will also climb mill.

If there are islands to be avoided, they must be defined in the line preceding **Irregular Pocket Cycle** line using **Islands**.

To program an Irregular Pocket cycle:

5. In Edit Mode, press **Pocket Cycles** (**F4**) to display the Pocket Cycles pop-up menu. Refer to **Figure 4-10, Pocket Cycles (F4) Pop-up Menu**.
6. Highlight **Irregular** and press **ENTER** to display the Irregular Pocket Cycle Graphic Menu. Refer to **Figure 4-22**.



IRREGULAR

Figure 4-22, Irregular Pocket Cycle Graphic Menu

7. Type the required Irregular Pocket Cycle values and settings in the entry fields. Refer to **Table 4-17, Irregular Pocket Cycle Address Words**.

Table 4-17, Irregular Pocket Cycle Address Words

Label	Address Word	Description
Sub#	**W**	The number of the subprogram that contains the perimeter of the pocket. Must be a closed shape. (Required)
ZDepth	**Z**	The Absolute depth of the finished pocket. (Required) **NOTE: ZDepth** must be lower than **StartHgt**. **StartHgt** is 0.1 Inch (2.0 mm) above the work surface.
Stepover	**A**	The distance the tool will step over (width of cut) as it mills out the pocket. The step over selected may need to be adjusted to ensure that excessive stock is not left on any of the pocket sides. (Required) **NOTE:** The CNC will default to 0.5 of the cutter diameter if **StepOver** = 0.000)
StartHgt	**H**	The Absolute Z position before beginning to mill the pocket. This must be 0.1 inch (or 2 mm) above the surface. (Required)
XStart	**X**	X coordinate of the ramp move to the starting position. (Optional) **NOTE:** Use **XStart** and **YStart** values together, if at all. If not given, the cycle will use the current position.
YStart	**Y**	Y coordinate of a ramp move to the starting position. (Optional) **NOTE:** Use **XStart** and **YStart** values together, if at all. If not given, the cycle will use the current position.
RampFeed	**I**	The feedrate at which the tool will "ramp" into the pocket in all three axes. (Optional)
RoughFeed	**J**	Rough-cycle feedrate (Optional)
FinFeed	**K**	Finish-cycle feedrate (Optional)
DepthCut	**B**	The depth per pass. If a deep pocket is necessary, it might not be feasible to take all the stock in one cut, so the Depth of Cut can be programmed to allow two or more passes. (Optional)
FinStock	**S**	Finish stock. If **K** (**FinFeed**) is set, the CNC automatically executes a finish pass after it roughs out the pocket at **K** (**FinFeed**) feedrate. If you do not specify a value, finish stock is not left. (Optional)

Slot Cycle

The **Slot** cycle simplifies programming of a slot. When executed, the CNC rapids to the **RetractHgt**, rapids to a location above the workpiece, rapids to the **StartHgt** and then feeds into the workpiece to a depth specified by **DepthCut**. A slot is machined from the inside towards the outside leaving the finish allowance on the side and bottom. It will repeat this until it reaches the rough depth. Once the roughing passes are complete, if a finish allowance is specified, it will then perform a finish pass. The material at the bottom of the part is removed first and then the material on the outside of the slot is removed. The roughing pass climb mills (counter clockwise direction) and the finish pass defaults to conventional milling (clockwise direction). If a negative **FinFeed** value is used, the finish pass will climb mill.

A slot is specified by **Length**, **Width**, **StartHgt**, and **ZDepth**. If any of these are not displayed, an error message is displayed.

The Slot Cycle is performed in the XY plane. If the XY plane is not activated prior to the **Slot** block, an error message is displayed and the Slot Cycle will not be performed.

The Slot Cycle automatically compensates for the tool diameter. Activate the correct tool diameter before the **Slot** block.

Use **DepthCut** to specify the number of passes required to get from the **StartHgt** to the **ZDepth**, cutting the **DepthCut** on each pass.

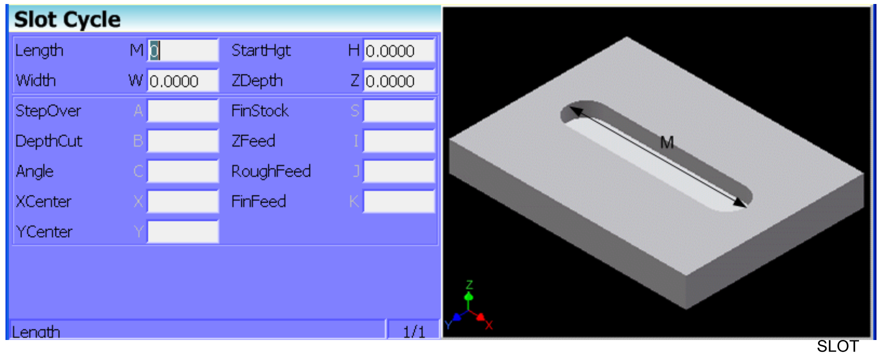
If the **XCenter** or **YCenter** parameters are not specified then they default to the current X or Y position.

Use **FinStock** to leave the specified stock on the outside and depth for a finish pass. The cycle cuts the profile of the slot and **ZDepth** on the finish pass.

If a **ZFeed**, **RoughFeed**, or **FinFeed** value is not specified, the feed moves are executed at the current feed rate. **ZFeed** specifies the feed rate for the Z plunge cuts, **RoughFeed** specifies the feed rate of the roughing cycle, and **FinFeed** specifies the feed rate of the finishing cycle.

To program an Slot cycle:

8. In Edit Mode, press **Pocket Cycles** (**F4**) to display the Slot Cycle pop-up menu. Refer to **Figure 4-10, Pocket Cycles (F4) Pop-up Menu**.
9. Highlight **Slot** and press **ENTER** to display the Slot Cycle Graphic Menu. Refer to **Figure 4-23, Slot Graphic Menu**.



SLOT

Figure 4-23, Slot Graphic Menu

10. Type the required Slot Cycle values and settings in the entry fields and press **USE** to add the Slot Cycle to the program. Refer to **Table 4-18**.

Table 4-18, Slot Address Words

Label	Address Word	Description
Length	**M**	Length of slot in X-axis (Required)
Width	**W**	Width of slot in Y-axis (Required)
StartHgt	**H**	The Absolute Z position before beginning to mill the slot. This must be 0.1 inch (or 2 mm) above the surface. (Required)
ZDepth	**Z**	The Absolute depth of the slot. Must be below **StartHgt** (**H**). (Required)
StepOver	**A**	Maximum tool step over (must be 50% or less of tool diameter). The distance the tool will step over (width of cut) as it mills out the slot. The step over selected may need to be adjusted to ensure that excessive stock is not left in the middle of the slot. (Optional) **NOTE:** The CNC will default to 0.5 of the cutter diameter if **StepOver** = 0.000 or is not specified.
DepthCut	**B**	Z-axis increment used for each pass. **B** is programmed as a positive dimension. Defaults to **ZDepth** less finish stock. (Optional)
Angle	**C**	The angle in degrees by which the slot is rotated. The center of rotation lies in the center of the slot. Default is 0 degrees. (Optional)
XCenter	**X**	Center of slot in X-axis. Default: Current X position (Optional)
YCenter	**Y**	Center of slot in Y-axis. Default: Current Y position (Optional)
FinStock	**S**	Finish-stock amount per side and bottom of slot. If not programmed, no finish stock is left. Defaults to no finish pass. (Optional)

(Continued…)

Table 4-18, Slot Address Words (Continued)

Label	Address Word	Description
ZFeed	**I**	Z-axis feed rate (plunging federate). Defaults to current Z-axis feed rate. (Optional)
RoughFeed	**J**	Rough-pass feed rate. Defaults to current feed rate. (Optional)
FinFeed	**K**	Finish-pass feed rate. If negative, the finish pass will climb mill (CW). If 0, material will be left, but no finish pass will occur. Defaults to last programmed feed rate. (Optional)

Pockets with Islands

This cycle allows islands in irregular pockets. Pockets with Islands must be programmed using subroutines.

More than one **Islands cycle** can be programmed at a time. They may be strung together, or on separate lines. Islands can be programmed inside of islands. Five islands can be put on a line. The subroutine number is used as inputs. Refer to **Table 4-19, Islands Address Words.**

Islands that are defined to be avoided on the inside of an irregular pocket are done so by using the Islands cycle followed by a list of up to 5 subprogram label names. If more than 5 islands need to be defined, the Islands cycle can be used to define as many subsequent islands as desired in multiples of 5 up to as many as needed. As in the following example:

Islands FirstIsl 2 SecondIsl 3 ThirdIsl 4 FourthIsl 5 FifthIsl 6

Islands FirstIsl 7 SecondIsl 8 ThirdIsl 9 FourthIsl 10 FifthIsl 11

Islands FirstIsl 12 SecondIsl 13 ThirdIsl 14 FourthIsl 15 FifthIsl 16

Islands FirstIsl 17 SecondIsl 18

and so forth … prior to calling the Irregular Pocket Cycle area clearance or irregular pocket command.

The islands need to be a closed contiguous line and/or arc movements starting and ending at the same point and starting with a **Line ToolComp Left** [use **Milling** (**F5**)>**Line** (**F3**)] or **Line ToolComp Right** as the first line to indicate which side of the contour the cutter needs to be, as viewed from the direction of travel. (No ramp on or off movement is allowed. The cycle will calculate these moves on and off the islands).

Activate a tool prior to programming Islands cycle and Irregular Pocket cycle, so cutter diameter is known. Islands cycle is for use with Irregular Pocket cycle only. Program Islands cycle before Irregular Pocket cycle.

To program an Islands cycle:

11. In Edit Mode, press **Pocket Cycles** (**F4**) to display the Pocket Cycles pop-up menu. Refer to **Figure 4-10, Pocket Cycles (F4) Pop-up Menu**.
12. Highlight **Islands** and press **ENTER** to display the Islands Graphic Menu. Refer to **Figure 4-24, Islands Graphic Menu**.

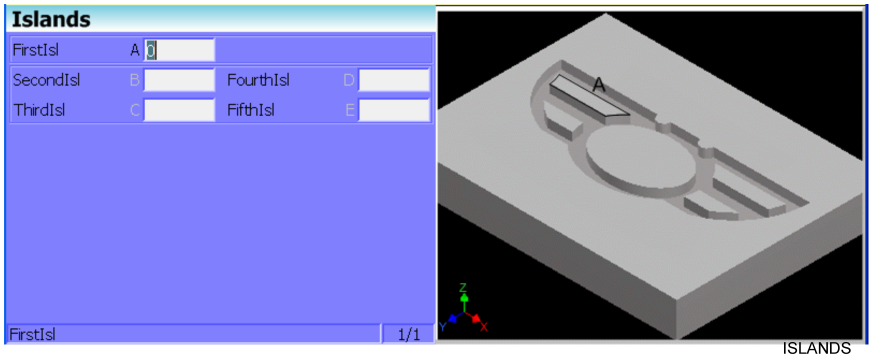


Figure 4-24, Islands Graphic Menu

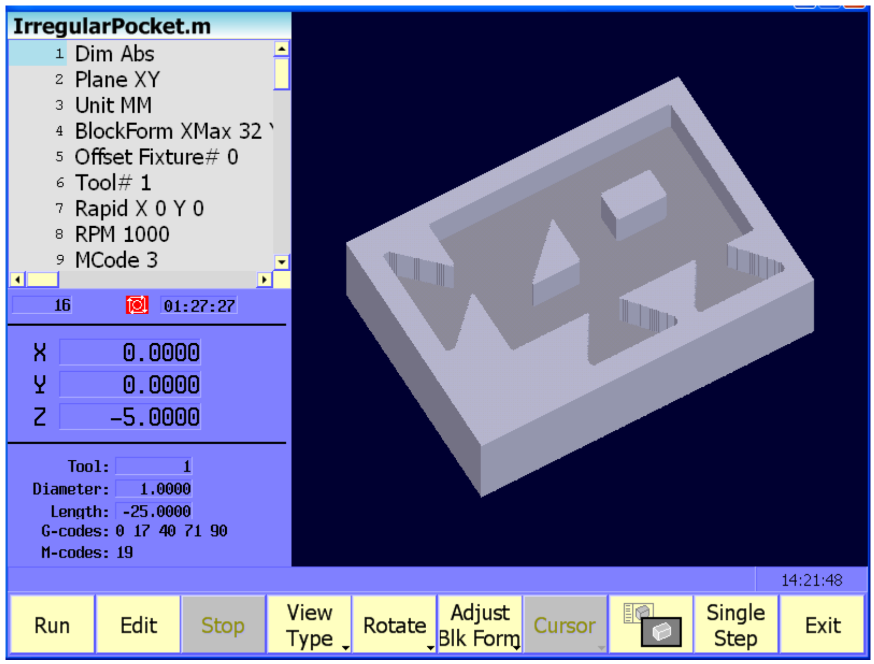
13. Type the required Irregular Pocket Cycle values and settings in the entry fields. Refer to **Table 4-19**.

Table 4-19, Islands Address Words

Label	Address Word	Description
FirstIsl	**A**	First island (Required)
SecondIsl	**B**	Second island
ThirdIsl	**C**	Third island
FourthIsl	**D**	Fourth island
FifthIsl	**E**	Fifth island

Using Subroutines for Pockets with Islands

The program below is the same one used in the DXF portion with subroutines added for the letters. See **Figure 4-25** and **Table 4-20**.



SUBROUTINES ISLANDS

Figure 4-25, Subroutines Pockets with Islands Example Workpiece

Table 4-20, Pockets with Islands Subroutines Programming Example

1	Dim Abs
2	Plane XY
3	Unit MM
4	BlockForm XMax 32 YMax 22 ZMax -6 XMin -2 YMin -2 ZMin -15
5	Offset Fixture# 0
6	Tool# 1
7	Rapid X 0 Y 0
8	RPM 1000
9	MCode 3
10	Islands FirstIsl 10 SecondIsl 20
11	Pocket Sub# 1 ZDepth -10 StepOver 0.69 StartHgt 2.0 XStart 5 YStart 5 RampFeed 500 RoughFeed 2000 FinFeed 1500 DepthCut 0.1 FinStock 0.2
12	Dim Abs
13	Rapid Z 25
14	MCode 5
15	Rapid X 0 Z -5
16	EndMain
17	Sub 1

```
18  Line ToolComp Left
19  Rapid X 5 Y 5
20  Line X 13
21  Line X 10 Y 0
22  Line X 20
23  Line X 16 Y 5
24  Line X 24
25  Line X 21 Y 0
26  Line X 31
27  Line X 27 Y 5
28  Line X 30
29  Line Y 20
30  Line X 5
31  Line Y 15
32  Line X 0 Y 20
33  Line Y 7
34  Line X 5 Y 12
35  Line Y 5
36  EndSub
37  Sub 10
38  Line ToolComp Right
39  Dim Abs
40  Rapid X 10 Y 10
41  Line X 15
42  Line Y 15
43  Line X 10 Y 10
44  EndSub
45  Sub 20
46  Line ToolComp Left
47  Dim Abs
48  Rapid X 20 Y 12
49  Line Y 15
50  Line X 25
51  Line Y 12
52  Line X 20 Y 12
53  EndSub
```

Subprograms

Program repetitive operations in a subprogram called from the main program.

- Call (or nest) subprograms within other subprograms. The CNC supports up to ten levels of nesting.
- Repeat or loop subprograms moving along any axis in increments each time the loop runs.
- Rotate, scale, or mirror subprograms.

The following examples describe two situations where subprograms save time.

The following topics are described:

- **Situation: 1 (Repetitive Drilling Cycle)**
- **Situation: 2 (Rough and Finish Cycles)**
- **Subprogram Structure**
- **Subprogram Example**
- **Organizing Programs from the Main Program**
- **Calling Subprograms from the Main Program**
- **Ending Main Programs**
- **Starting Subprograms**
- **Ending Subprograms**
- **Looping Subprograms**
- **Rotate, Mirror, and Scale Subprograms (RMS)**

Situation: 1 (Repetitive Drilling Cycle)

When a workpiece must be center-drilled, drilled, then counterbored, each of the three tools must go to the same hole positions consecutively. Ten hole positions would require thirty programmed hole locations (ten for each tool). Program the ten hole locations in a subprogram called three times from the main program (once for each tool).

Situation: 2 (Rough and Finish Cycles)

Use subprograms for jobs that require both roughing and finishing cycles. Rough out the outside of a workpiece with a roughing mill, and then finish it with a finishing mill. Program the profile in a subprogram. The main program calls the subprogram twice, once for each tool. You can set the tool diameter to 0.5300 inches for the .5-inch roughing mill and to 0.5000 inches for the .5-inch finishing mill. Tool #1 will leave 0.0150 inch excess stock per side. Tool #2 finishes the work to size.

Subprogram Structure

When using subprograms, define the end of the main program and the start and end of each subprogram

Subprogram Example

1	Dim Abs		
2	Rapid	X 5.0000 Y -5.0000	
3	Call 1		
4	Rapid	X 6.0000 Y -6.0000	
5	Call 1		
6	Rapid	X 7.0000 Y -5.0000	
7	Call 1		
8	**EndMain**		**End of Main Program**
9	**Sub 1**		**Start of Subprogram 1**
10		Z -0.0625	
11	Dim Incr		
12	Line	X 0.375	
13	Line	Y 0.375	
14	Line	X -0.375	
15	Line	Y -0.375	
16	Dim Abs		
17		Z 0.1000	
18	**EndSub**		**End of Subprogram 1**
19	**<End Of Program>**		**End of Program**

The main program begins at Block 1 and runs through Block 8. The subprogram begins at Block 9 and runs through Block 18.

When the main program reaches Block 3, the CNC jumps to Block 9, runs the subprogram through Block 18, and then returns to the main program, Block 4.

Blocks 5 and 7 call the subprogram again.

Organizing Programs Containing Subprograms

To write a program that includes a subprogram:

1. Write the main program and include the subprogram **Call** blocks.
2. Insert an **EndMain** block at the end of the main
3. Insert a **Sub** block, followed by a unique subprogram call number (1 to 9999), on the first block of the subprogram. Example: Sub1.
4. Write the subprogram blocks.
5. Finish the subprogram with an **EndSub** block.
6. End the program with an **<End of Program>** block.

Calling Subprograms from the Main Program

To call a subprogram from the main program:

1. In Edit Mode, press **Sub Progs** (**F8**) to display the Subprogram soft key labels. Refer to **Figure 4-26**.

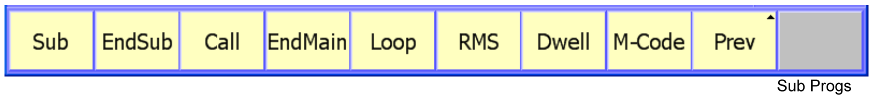


Figure 4-26, Sub Progs (F8) Soft Keys

2. Press **Call** (**F3**). The Graphic Menu prompts for the subprogram number.
3. Type a subprogram number and press **ENTER** to add **Call** block to program.

Ending Main Programs

To program an **EndMain** block:

1. In Edit Mode, press **Sub Progs** (**F8**) to display the Soft Key subprogram labels. Refer to **Figure 4-26**.
2. Press **EndMain** (**F4**) to display the **EndMain** block in the program listing.

Starting Subprograms

Start subprograms with a Sub block.

To program a Sub block:

1. In Edit Mode, press **Sub Progs** (**F8**) to display the Sub Prog soft keys. Refer to **Figure 4-26**.
2. Press **Sub** (**F1**). The CNC prompts for subprogram number.
3. Type Sub number and press **ENTER** to add a Sub block to the program. The Sub number must agree with the matching Call number.

Ending Subprograms

End subprograms with an **EndSub** block.

To program an **EndSub** block:

1. In Edit Mode, press **Sub Progs** (**F8**) to display the **Sub** soft keys. Refer to **Figure 4-26**.
2. Press **EndSub** (**F2**) to add an **EndSub** block to the program.

Looping Subprograms

Looping subprograms repeat a set number of times before they return to the main program. The CNC tracks the number of loops.

NOTE: Only subprograms can loop.

To call a Loop subprogram:

1. In Edit Mode, press **Sub Progs** (**F8**) to display the **Sub** soft keys. Refer to **Figure 4-26, Sub Progs (F8) Soft Keys**.
2. Press **Loop** (**F5**) to activate the **LOOP SUB** Graphic Menu.
3. Type the required Loop values and settings in the entry fields. Refer to **Table 4-21**.

Table 4-21, Loop Address Words

Label	Address Word	Description
Sub#	**P**	Subprogram identification number. (Required)
#Loops	**M**	Number of times loop repeats before it returns to the main program. (Required)
XIncr	**X**	Distance X-axis increments every cycle. (Optional)
YIncr	**Y**	Distance Y-axis increments every cycle.
ZIncr	**Z**	Distance Z-axis increments every cycle. Cannot be used with **XIncr** or **YIncr**. (Optional)
ZFeed	**I**	Feedrate used with **ZIncr**. (Optional)

Rotate, Mirror, and Scale Subprograms (RMS)

Use **RMS** blocks to scale, rotate, and/or mirror subprograms. These functions turn off when the subprogram ends.

To call an RMS subprogram:

1. In Edit Mode, press **Sub Progs** (**F8**). Soft Key labels display subprogram functions. Refer to **Figure 4-26, Sub Progs (F8) Soft Keys**.
2. Press **RMS** (**F6**). The Rotate, Mirror and Scale Cycle Graphic Menu prompts for labeled values.
3. Type the required Rotate, Mirror and Scale Cycle values and settings in the entry fields. Refer to **Table 4-22**.

Table 4-22, Rotate, Mirror, and Scale Cycle Address Words

Label	Address Word	Description
Sub#	**P**	Subprogram number. (Required)
#Loops	**M**	Number of times subprogram will loop before it returns to the main program. (Optional) **NOTE: RMS** subprograms loop only when rotating.
StartAngle	**F**	Number of degrees the pattern rotates for the first loop. (Optional) **NOTE:** Sometimes, it is easier to program a part from the 3 o' clock position, and then rotate it to desired angle.
Angle	**C**	Number of degrees the pattern rotates per loop. (Optional)
XCenter	**I**	Point of rotation X coordinate. (Optional)
YCenter	**J**	Point of rotation Y coordinate. (Optional)
MirrorX	**U**	Press **+/-** to toggle between **Yes** and **No**. If **Yes**, CNC mirrors the X-axis values. (Optional)
MirrorY	**V**	Press **+/-** to toggle between **Yes** or **No**. If Yes, mirrors the Y-axis values. (Optional)
XScale	**X**	X-axis scale factor. Multiplies all X-axis positions by the number typed. (Optional)
YScale	**Y**	Y-axis scale factor. Multiplies all Y-axis positions by the number typed. (Optional)

Engraving, Repeat, and Mill Cycles

The following topics are described:

- **Engraving Cycle**
- **Repeat Cycle**
- **Mill Cycle**

Engraving Cycle

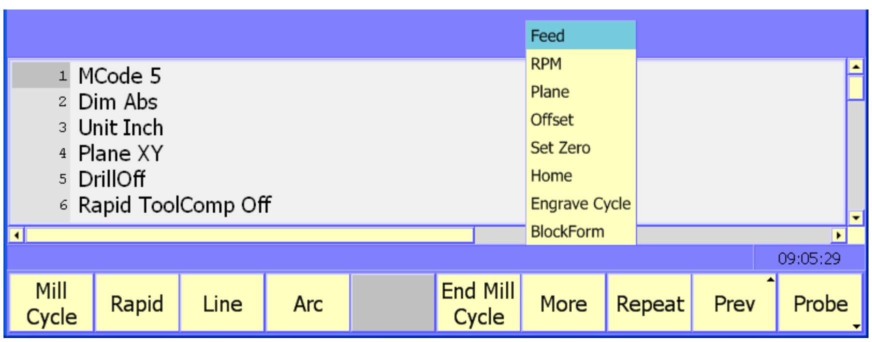
The Engraving cycle provides a quick and easy way to engrave part numbers, legends, or any alpha/numeric inscription. The usual type of cutter is a sharp point or center-drill type tool. Options are given for engraving on an angle and mirror is supported for engraving molds. When executed, the CNC rapids to the start point, then to the StartHgt. It then feeds to the ZDepth specified and begins cutting the Text selected.

Programming the Engraving Cycle

To program the Engraving Cycle:

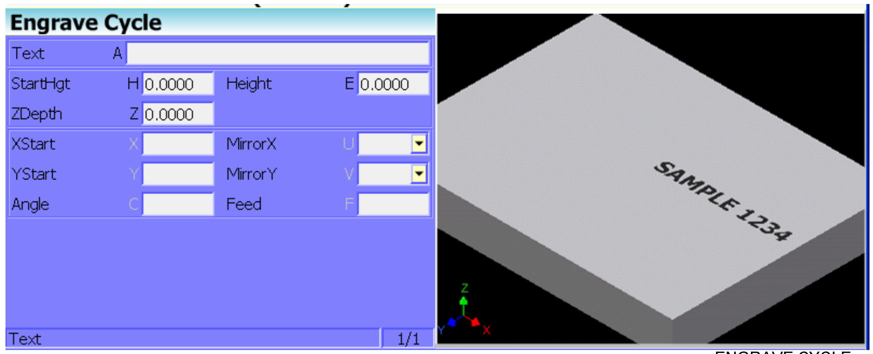
1. In Edit mode, press **Milling** (**F5**) and **More** (**F7**) to display the More pop-up menu, **Figure 4-27**. Highlight **Engrave** and press **ENTER** to display the Engraving Cycle Graphic Menu, **Figure 4-28, Engrave Cycle Graphic Menu**.

2. Complete the entry fields (refer to **Table 4-23, Engrave Cycle Address Words**), and press **ENTER**.



MILLING_MORE POP-UP

Figure 4-27, Milling (F5)>More (F7) Pop-up Menu



ENGRAVE CYCLE

Figure 4-28, Engrave Cycle Graphic Menu

Table 4-23, Engrave Cycle Address Words

Label	Address Word	Description
Text	**A**	When the cursor is on Text, it displays an entry field for the letters to be engraved. Letters A – Z, numbers 0 – 9, and: space, ampersand, plus, minus, comma, period, and slash right are supported. No lower case letters are allowed. Press **ENTER** to accept the text. (Required)
StartHgt	**H**	Z absolute start height. Must be higher than **ZDepth**. (Required)
ZDepth	**Z**	Z absolute depth of engraving. Must be below **StartHgt**. (Required)
Height	**E**	Letter height. Width will be proportional to height. Height is measured at the centerline of the cutter. (Required)
XStart	**X**	X coordinate for lower-left corner of the text. Defaults to current position if not given. (Optional)
YStart	**Y**	Y coordinate for lower-left corner of the text. Defaults to current position if not given. (Optional)
Angle	**C**	Angle in degrees. Default is 0 degrees. (Optional)
MirrorX	**U**	Mirrors all X moves. Set by using minus key (-) while in this field. (Optional)
MirrorY	**V**	Mirrors all Y moves. Set by using minus key (-) while in this field. (Optional)
Feed	**F**	Feedrate used while engraving. Default is current feedrate. (Optional)

Sample Engraving Cycle Program

```
1     Dim Abs
2     Unit Inch
3     Rapid     X 0.00000 Y 0.00000
4     Tool# 1
5     Rapid     X 1.00000 Y 1.00000
6     Rapid     Z 0.10000
7     Engrave   Text "ABCD" StartHgt 0.0100 ZDepth -0.0100 Height 0.5000
8     Rapid     Z 1.00000
9     Rapid     X 0.00000 Y 0.00000
10    EndMain
```

This program will rapid to X1.0 Y1.0. Z will rapid to 0.1 and the letters ABCD will be engraved 0.0100” deep and 0.500” high.

Repeat Cycle

The Repeat cycle allows a series of previously programmed blocks to be repeated. Some examples are going over the same contour while lowering the Z-axis, or drilling over a series of holes with a different drill cycle, or moving an operation to a different location using fixture offsets. Wherever it is used, the repeated blocks will be processed, just as if they were written in the program at that point.

Programming the Repeat Cycle

To program the Repeat Cycle:

1. In Edit mode, press **Milling** (**F5**) to display the Milling soft keys. Refer to **Figure 5-5, Milling (F5) Soft Keys**. Press **Repeat** (**F8**) to display the Repeat screen.
2. Complete the entry fields (refer to **Table 4-24**), and press **ENTER**.

Table 4-24, Repeat Cycle Address Words

Label	Address Word	Description
Repeat	**R**	Type the block number you want to begin repeating. (Required)
Thru	**T**	Type the block number you want to end the repeat. (Required)

3. When using a Modal Drilling Cycle with the Repeat feature, a **DrillOff** or non-move command must be included as the final block. For example, see “Sample Repeat Cycle Program” block 7–12 and block 15.

Sample Repeat Cycle Program

```
1     Dim Abs
2     Unit Inch
3     Offset    Fixture# 0
4     Rapid     X 0.0000 Y 0.0000
5     Tool# 1
6     Rapid     Z 0.1000
7     BasicDrill ZDepth –0.50000 StartHgt 0.10000 Feed 15.0
8     Rapid     X 1.00000
9               Y 1.0000
10              X 0.0000
11              Y 0.0000
12    DrillOff
13    Offset    Fixture# 1 X 3.0000 Y 0.0000
14    Offset    Fixture# 1
15    Repeat 7   Thru 12
16    Rapid     Z 0.5000
17    EndMain
```

This program will drill four holes. A Fixture Offset is used to relocate X Y zero. When the Repeat Cycle is encountered, it will drill four more holes at the offset location.

Mill Cycle

The Mill cycle is intended for contour milling operations. Cutter compensation, Z pecking, Z finish stock, RoughFeed, and FinishFeed are supported. The cycle will rapid to the XY start point (compensated, if comp is on) rapid to the start height and then feed to the **ZDepth** or **DepthCut** using the **ZFeed**. Subsequent milling blocks are then executed using the **ToolComp** parameter and **Feed** specified. The feedrate can be changed in the blocks that are being milled, but any change of feedrate must accompany an axis move and overrides the feedrate specified in the canned cycle (**RoughFeed** or **FinFeed**) from the point of the new feedrate forward in the cycle. Cutter Compensation cannot be changed from within the cycle. The cycle is terminated with the **EndMill** block; at which point, it rapids up to the StartHgt and returns to the un-comped **XStart YStart** location.

Programming the Mill Cycle

To program the Mill Cycle:

1. In Edit mode, press **Milling** (**F5**) and **Mill Cycle** (**F1**) to display the Mill Cycle screen, **Figure 4-29**.

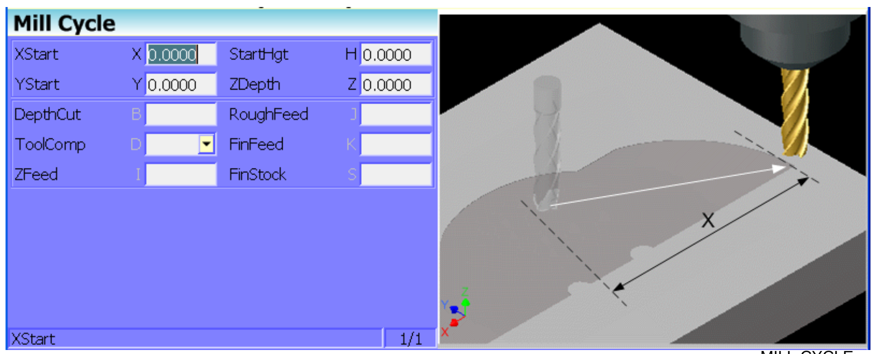


Figure 4-29, Mill Cycle Graphic Menu

2. Complete the entry fields (refer to **Table 4-25, Mill Cycle Address Words**), and press **ENTER**.

NOTE:	If the "**D**" parameter is used for cutter compensation, the lines of code in the mill cycle must start with an uncompensated ramp-on move and end with an uncompensated ramp-off move as the first and last lines in the mill cycle will not be automatically compensated by the cycle.

Programming the EndMill Block

To program the EndMill Block:

1. In Edit mode, press **End Mill Cycle** (**F6**) to end the cycle.

Table 4-25, Mill Cycle Address Words

Label	Address Word	Description
XStart	**X**	X coordinate for start of Mill cycle. Defaults to current position if not given. (Required)
YStart	**Y**	Y coordinate for start of Mill cycle. Defaults to current position if not given. (Required)
StartHgt	**H**	Z absolute start height. Must be 0.100" above work surface (0.2mm). (Required)
ZDepth	**Z**	Absolute depth of finished contour (Required)
DepthCut	**B**	Depth of cut taken in a single pass. Cuts will be adjusted so that all are equal pecks. (Optional)
ToolComp	**D**	Tool radius compensation Left or Right of programmed path. Set by using minus key (-) while in this field. (Optional)
ZFeed	**I**	Feedrate for Z-axis. Defaults to current feedrate. (Optional)
RoughtFeed	**J**	Feedrate for X and Y-axis. Defaults to current feedrate. (Optional)
FinFeed	**K**	Feedrate used for **FinStock** (Optional)
FinStock	**S**	Amount of stock to take for last Z peck (Optional)

Sample Mill Cycle Program

```
1    Dim Abs
2    Unit Inch
3    BlockForm XMax .1 YMax 1.1 ZMax 0 XMin -1.1 YMin -.1 ZMin -.25
4    Rapid  X -.5 Y .5
5    Tool# 1    MCode 6
6    Mill XStart -.5 YStart .5 StartHgt .1 ZDepth -.25 DepthCut .125
     ToolComp Left ZFeed 25 RoughFeed 35 FinFeed 45 FinStock .01
7    Dim Incr
8    Line X -.5
9    Y -.5
10   X 1
11   Y 1
12   X -1
13   Y -.5
14   EndMill X -.5 Y .5
15   Dim Abs
16   Rapid Z 1
17   X 0 Y 0
18   EndMain
```

This program will contour a square, in two Z pecks of 0.120" each and one finish pass on .01". The blocks 7 thru 13 are the contour moves that will be comped to the left of tool path direction (in this case inside). Block 14, EndMill is required to show the end of the contour. The cutter will be returned to the start point, X-.5 Y.5 at the start height of 0.100".

Probing Cycles

Probing cycles have the following features:

- Tool probe cycles
- Spindle probe cycles

This topic describes operation and an overview of the tool and spindle probe canned cycles in conversational format. Probing is a standard in the 6000i CNC system.

The cycles provided perform the most common tool and spindle probing functions. If Probing has been added post-sale, besides Setup Utility changes, there will be Programmable Logic Controller (PLC) program modifications required.

The tool probe cycles are only supported on machines with automatic spindle forward/reverse and spindle speed, and homing with a permanent X, Y, and Z machine position. The method described assumes the use of negative tool-length offsets. In this method, the Tool-Length Offset (TLO) in the length column for each tool represents the distance from the tool tip at machine home to top of work piece and is a negative number. This method does not require the use of any Z work coordinate offset to be active. This procedure will find the effective tool diameter by turning the spindle on in reverse and touching two sides of the probe stylus, then storing the tool's diameter in the tool's diameter offset table.

The spindle probing cycles are designed to assist in part setup. Using these cycles, one or more features (edges) of a part can be measured. Using the data obtained with these measurements, calculations are made that can be used to set a given fixture offset. It is also possible to find the orientation angle of a part so as to not always have to align the part exactly.

Tool and spindle probing do not allow rotation, scaling, and mirroring. Plane will be set to XY when these cycles are complete.

The following topics are described:

- **Probing Canned Cycle Parameter Settings**
- **Tool Probe Cycles**
- **Spindle Probe Cycle**
- **Using the Z Work Offset Update Feature**

Probing Canned Cycle Parameter Settings

Before you set the cycle parameters for the probe you must note that when entering values in the probing machine parameters, keep in mind that all values are entered in metric.

Set the following 6000i Machine Probing Parameters (refer to the *6000i CNC Technical Manual*, P/N 627787-21).

Tool Probe Cycles

The tool probe will update the tool registers only. If you are going to use the tool being measured after the probing cycle, you must recall that tool for the new offsets to be active.

For tool probing, Tool-Length Offset (TLO) is the distance from machine home to top of work piece or wherever you wish to set your part Z zero or if used in conjunction with a Z axis work offset, a fixed surface on the machine.

Before starting to set your tools, you must calibrate the probe. Once the probe has been calibrated, calibration does not have to be done again unless you remove the probe or replace the stylus.

Recalibration may also be required if the Z location of the top of the part changes, and is not compensated by a Z work offset shift.

The following topics are described:

- **Tool Probe Cycle Designations**
- **Description of Tool Probe Cycles**

Tool Probe Cycle Designations

The following summarizes the cycles available:

Probe Calibration (CalibTlPrb)

Tool Probe Calibration Cycle

This is used to set the Z datum for length preset, the effective probe stylus diameter for setting tool diameter registers, and establishes the center of the probe stylus.

> **NOTE:** Calibration must be done at least once before using the tool probe. Once the probe has been calibrated, calibration does not need to be done again unless the probe is moved or a new part is being setup. The cycle must always know the relationship between the top of the part and the top of the probe to set the TLO.

Length and Diameter (LenDiamMea)

Tool Length and Diameter Offset Preset

Updates length and diameter tool registers.

> **NOTE:** If the tool has a hole on the bottom so that the probe would fall between the tool teeth, do not use this cycle. Damage to the probe could result. In this case, use **Length Special** for manual length preset or **Diameter Special** for manual diameter preset.

Length Special (LenSpecMea)

Manual Tool-Length Offset Preset

Updates tool-length register. To be used for large face mill style tools or shell mill tools that have a hole in the center of the bottom of the tool.

Diameter Special (DiaSpecMea)	**Manual Tool Diameter Preset** Updates tool diameter register for irregular shaped tools or tools with a hole in the center of the bottom.
Break and Wear (BrkWearDet)	**Tool Breakage, Length and Diameter Wear Detection** **Breakage** Checks the tool and gives an alarm if not within tolerance. Length and Diameter Wear – Check the Length and/or Diameter and updates the Length and/or Diameter wear registers up to a user-defined limit. Once the user-defined limit has been reached, the cycle will give an alarm and stop the program.

Description of Tool Probe Cycles

- For tool probing or tool length presetting, Tool-Length Offset (TLO) is the distance from machine home to top of work piece or wherever you set your part "**Z**" zero.
- Before starting to set your tools, you must calibrate the probe. Once the probe has been calibrated, calibration does not have to be done again unless you remove the probe or replace the stylus.
- Recalibration may also be required if the Z location of the top of the part changes, and is not compensated for by a Z work offset.
- The probing cycles can be found in the conversational side of 6000i by pressing **Mill** (**F5**), then **Probe** (**F10**) and **ToolPro** (**F1**) or **SpinPro** (**F3**). You can also access the probe cycles on the 6000i from the main edit screen by pressing the **SHIFT** key and then **F1** or **F3**.

The following tool probe cycles are described:

- Tool Probe Calibration Cycle (**CalibTlPrb**)
- Tool Length and Diameter Offset Preset (**LenDiamMea**)
- Manual Tool Length Measure for Special Tools (**LenSpecMea**)
- Manual Tool Diameter Measure for Special Tools (**DiaSpecMea**)
- Tool Breakage, Length, and Diameter Wear Detection (**BrkWearDet**)

Tool Probe Calibration Cycle (CalibTlPrb)

Format: CalibTlPrb DiamOfStd(n) DistDown(n)

This cycle is used to calibrate the probe. This sets the Z datum for length preset to the top of the part, establishing the center of the probe stylus, and the effective probe stylus diameter for setting tool diameter registers. Refer to **Table 4-26**.

Table 4-26, CalibTlPrb Entry Fields

Entry Fields	Description
DiamOfStd	The diameter of the part of the calibration standard that comes in contact with the probe stylus during calibration. This should be an exact measurement. (Optional override for **Diameter of tool probe gauge**)
DistDown	The distance to go down along the side of the probe stylus with the probe calibration standard when touching the side of the stylus for diameter calibration. The maximum **DistDown** value is 0.55” (13.97 mm). Without any **DistDown** value, the cycle will bring the calibration standard down past the top of the probe stylus the default 0.1” (2.54mm). If you put a number higher than 0.55” (13.97 mm), the control displays an error. (Optional) [Default: 0.1”]

To calibrate the tool probe:

1. Jog the calibration standard (the calibration standard should be in the spindle) to the top of your work piece or a common surface where all your tools will be calibrated to, and set its tool-length offset to the top of the work piece or to wherever you would like your Z zero to be. To calibrate the tool:
   a) Jog the tip of the calibration standard to the proper spot
   b) Press the Teach (F9) function key
2. Manually jog the calibration standard over the probe stylus center and less then 0.1” (2.54 mm) above the probe stylus. It should be no more then 0.1” (2.54 mm) from the center of the stylus.
3. From the MDI mode, pick
   F5 (**Mill**) > **F10** (**Probe**) > **F1** (**ToolPro**) > **Probe Calibration**
   For example: **CalibTlPrb** exit by pressing **F9** twice and **F10** to exit.
4. The Z-axis will initially go down and touch the top of the probe stylus at the feedrate specified in **Z first pick, MEDIUM feedrate** machine setup parameter. Then retouch at the slow feedrate, **Z final pick, SLOW feedrate** machine setup parameter, establishing the zero probe stylus top.
5. Then incrementally rapid up whatever value that is in **Z retract distance** machine setup parameter.

6. The spindle will come on at the RPM specified at the **RPM for calibration and tool measurement** machine setup parameter and then the calibration standard will move over an incremental amount that is equal to (Half the value entered in the **DiamOfStd** cycle parameter (or machine setup parameter Diameter of probe gauge) + Half the value entered in **Nominal probe stylus diameter** machine setup parameter + The value in the **XY retract distance** machine setup parameter). The direction the probe will move over depends on what is placed in the **Probe orientation** machine setup parameter:

–1	Go first to the left
1	Go first to the right
–2	Go first to the front
2	Go first to the back

7. The Z-axis will then do a guarded Z move down 0.1” (2.54 mm) or whatever amount was placed in the **DistDown** cycle parameter and then move over toward the probe stylus 0.3” (7.62 mm) or until it touches the probe stylus. If contact is not made with the probe or if contact is made during a guarded move, then an alarm will be generated and the canned cycle will terminate.

8. After the probe stylus is touched on the first side, the machine will then rapid up and over the stylus then down on the opposite side then over to the other two sides until it has touched the probe stylus on all four quadrants. This will establish the center of the probe stylus.

9. The spindle will then turn off and the machine will touch off on two sides of the probe with the spindle off finding the effective probe stylus diameter. Then, will rapid up above the probe stylus and over to the center.

10. Remove the calibration standard. You are now ready to start running the **Length and Diameter** cycle (or one of the other cycles for setting or checking length and diameter of the tool) to set your tool-length offsets or tool diameter registers.

Tool Length and Diameter Offset Preset (LenDiamMea)

Format: LenDiamMea Tool#(tool#) EstDiam(n) MeasType (Length, Diameter, or Both) DistDown(n) OvrFstFeed(n) OvrMedFeed(n) OvrSlwFeed(n) OvrRPM(n)

Each tool must have the length set once before trying to set the diameter. Call this cycle up the first time using **Both** because it will automatically set the length first then the diameter.

- Calibrate the tool probe at least once before trying to automatically preset a tool. This is done initially, but if the stylus is ever changed or the probe is moved, then you must again calibrate the tool probe.
- This tool preset (**LenDiamMea**) can be run from within a program or from the MDI mode. Refer to **Table 4-27**.

Table 4-27, LenDiamMea Entry Fields

Entry Fields	Description
Tool#	Tool number. (Required) With only the **Tool#** cycle parameter present, the canned cycle will not step over half the tool's diameter but come straight down measuring the tool length and storing it in the tool register.
EstDiam	This is the rough diameter of the tool. This should be within 0.04" (1.0 mm). (Optional) If the **EstDiam** cycle parameter is present, the tool will step over half of its diameter, the spindle will turn on in reverse and then the canned cycle will measure the tool's length. A negative **EstDiam** value is for a left-handed tool and will cause the spindle to come on forward instead of reverse. For on center length measurement, do not give a **EstDiam** cycle parameter.
MeasType	The option specifies to measure length, diameter, or both and the appropriate tool registers are updated. (Optional) [Default: **Length**] **Diameter** Measure the diameter only **Length** Measure the length only **Both** Measure both length and diameter If **Length** is not set, the cycle will measure the tool length only. If **Diameter** or **Both** are programmed, you must also have an **EstDiam** cycle parameter or the control will display an error message.

(Continued…)

Table 4-27, LenDiamMea Entry Fields (Continued)

Entry Fields	Description
DistDown	The distance to go down along the side of the probe stylus when doing a diameter pick. The maximum **DistDown** value is 0.55” (13.97 mm) or the tool may crash into the probe or table. If you enter a value larger than 0.55” (13.97 mm), the control will issue an error message. If **DistDown** is not set, the cycle will use a default value of 0.1” (2.54 mm). (Optional) [Default: 0.1”] Ball nose cutters and special cutters that require a move down more than 0.55” (13.97 mm) are not supported.
OvrFstFeed	This is the override for the fast Z feedrate that was set in the machine setup parameter **Z first pick, FAST feedrate**. Sometimes there may be a tool that has a large diameter making it necessary to slow it down to prevent the touch probe from being hit too hard. This can only be set slower. Trying to set this higher will only result in the software using the original feedrate. (Optional)
OvrMedFeed	This is the override for the medium feedrate that was set in the machine setup parameter **Z first pick, MEDIUM feedrate**. This is used for the same reason as the **OvrFstFeed** cycle parameter. This can only be set slower. Trying to set this higher will only result in the software using the original feedrate. (Optional)
OvrSlwFeed	This is the override for the slow feedrate that was set in the machine setup parameter **Z final pick, SLOW feedrate**. This is used for the same reason as the **OvrFstFeed** cycle parameter. This can only be set slower. Trying to set this higher will only result in the software using the original feedrate. (Optional)
OvrRPM	This is the override for the RPM that was set in the machine setup parameter **RPM for calibration and tool measurement**. This is used for the same reason as the **OvrFstFeed** cycle parameter. This can only be set slower. Trying to set this higher will only result in the software using the original RPM. (Optional)

To use the tool preset probing cycle:

1. Install all the tools you wish to set, in the tool changer.
2. Input:
   LenDiamMea Tool#(tool#) EstDiam(tool rough diameter) MeasType

   If run from the inside of a program, this line needs to be repeated for every tool that you want to set.

3. Execute that line if you are in MDI mode, or run the program if you have set all the tools up in a program.
4. If you have done a single tool in MDI mode, that tool is now measured and you are ready to measure the next tool. If you have placed multiple lines in a program, one for each tool, all your tools are measured and ready for use.

Shell mill style tools that have a hole in the center of the bottom will not work with this canned cycle; in this case, you must use the manual canned cycles **LenSpecMea (Length Special Manual Tool Length Measure for Special Tools)** for length and **DiaSpecMea (Diameter Special Manual Tool Diameter Measure for Special Tools)** for diameter. See **Table 4-27, LenDiamMea Entry Fields**. This cycle is only good for drills, taps, reamers, ball nosed endmills, and standard endmills with a flat bottom, the cycle updates length and diameter tool registers clearing anything in the wear registers.

Format: LenDiamMea Tool#(tool#)

With **Tool#** parameter only set:

1. The machine will rapid the Z-axis up and pick up the tool designated in the **Tool#** cycle parameter and rapid directly over the center of the probe stylus.
2. The Z-axis will rapid down the distance placed in the **Z rapid to start position from home** machine setup parameter, then start feeding down toward the probe for the initial touch at the feedrate that was placed in the **Z first pick, FAST feedrate** machine setup parameter, then will back up and retouch the probe at the feedrate that is in the **Z final pick, SLOW feedrate** machine setup parameter.
3. The tool-length register for that tool is now updated, and that tool's length-wear register is set to zero.
4. Then the Z-axis will rapid up to home position.
5. If you have done a single tool in MDI mode, that tool is now measured and you are ready to measure the next tool. If you have placed multiple lines in a program, one for each tool, the machine will then grab the next tool and repeat steps 1 through 4 until all the tools have been measured.

Format: LenDiaMea Tool#(tool#) EstDiam(tool rough diameter)

With **Tool#** and **EstDiam** parameters only set:

1. The machine will rapid the Z-axis up and pick up the tool designated in the **Tool#** cycle parameter and rapid directly over the center of the probe stylus.
2. The Z-axis will rapid down the distance placed in the **Z rapid to start position from home** machine setup parameter then start feeding down toward the probe for the initial touch at the feedrate that was placed in the **Z first pick, FAST feedrate** machine setup parameter then will back up.
3. The machine will rapid over half the diameter of the cutter from the probe stylus center in the direction related to the **Probe orientation** machine setup parameter.
4. The spindle will then come on in reverse at the RPM specified in the **RPM for calibration and tool measurement** machine setup parameter and retouch the probe at the feedrate that is in the **Z final pick, SLOW feedrate** machine setup parameter.
5. The tool-length register for that tool is now updated, and that tool's length-wear register is set to zero.
6. Then the Z-axis will rapid up to the home position.
7. If you have done a single tool in MDI mode, that tool is now measured and you are ready to measure the next tool. If you have placed multiple lines in a program, one for each tool, the machine will then grab the next tool and repeat steps 1 through 6 until all the tools have been measured.

Format: LenDiaMea Tool#(tool#) EstDiam(tool rough diameter) MeasType (Both)

With **Tool#**, **EstDiam**, and **MeasType** parameters set:

1. The machine will rapid the Z-axis up and pick up the tool designated in the **Tool#** cycle parameter and rapid directly over the center of the probe stylus.
2. The Z-axis will rapid down the distance placed in the **Z rapid to start position from home** machine setup parameter then start feeding down toward the probe for the initial touch at the feedrate that was placed in the **Z first pick, FAST feedrate** machine setup parameter then will back up.
3. The machine will rapid over half the diameter of the cutter from the probe stylus center in the direction related to the **Probe orientation** machine setup parameter.

4. The spindle will then come on counter clockwise at the RPM specified in the **RPM for calibration and tool measurement** machine setup parameter and retouch the probe at the feedrate that is in the **Z final pick, SLOW feedrate** machine setup parameter.

5. The tool-length register for that tool is now updated, and any value in the length wear register will be reset to zero.

6. Then the Z-axis will rapid up above the probe stylus the distance specified in the **Z retract distance** machine setup parameter and then rapid the X & Y axes over the center of the probe and turn the spindle on in reverse.

7. The machine will move the tool's edge off to one side of the probe stylus in the direction indicated in the **Probe orientation** machine setup parameter before making a guarded move down 0.1" (2.54 mm) or whatever value has been placed in the **DistDown** cycle parameter.

8. The machine will then touch the tool to the probe stylus on two opposite sides at the feedrate specified in the **Z first pick, MEDIUM feedrate** machine setup parameter with the spindle running at the RPM specified in the **RPM for calibration and tool measurement** machine setup parameter, backing up 0.02" (0.508 mm) after each first touch then retouching and the feedrate specified in the **Z final pick, SLOW feedrate** machine setup parameter, calculating the diameter of the tool and placing the calculated diameter value in the diameter register for the tool being preset and any value in the diameter wear register will be reset to zero.

9. Then the Z-axis will rapid up to the home position.

10. If you have done a single tool in MDI mode, that tool is now measured and you are ready to measure the next tool. If you have placed multiple lines in a program, one for each tool, the machine will then grab the next tool and repeat steps 1 through 9 until all the tools have been measured.

Manual Tool Length Measure for Special Tools (LenSpecMea)

Format: LenSpecMea Tool#(tool#) DiamOfStd(n) OvrMedFeed(n) OvrSlwFeed(n) OvrRPM(n)

This cycle is used to measure the length of large face mill style tools that have a hole in the center of the bottom of the tool. Refer to **Table 4-28**.

Table 4-28, LenSpecMea Entry Fields

Entry Fields	Description
Tool#	Tool number. (Required) With only the **Tool#** cycle parameter present, the spindle will turn on in reverse and the canned cycle will come straight down measuring the tool length and storing it in the tool-length register. The **Tool#** cycle parameter must the same as the current tool in the spindle.
EstDiam	This is the rough diameter of the tool and is only used in this cycle to determine if the spindle should be turned on in reverse or forward. If you have a left-handed tool you would give a negative value to the diameter. If this parameter is left off the control will always turn on in reverse by default. (Optional)
OvrMedFeed	This is the override for the medium feedrate that was set in the machine setup parameter **Z first pick, MEDIUM feedrate**. Sometimes there may be a tool that has a large diameter making it necessary to slow it down to prevent the touch probe from being hit too hard. This can only be set slower. Trying to set this higher will only result in the software using the original feedrate. (Optional)
OvrSlwFeed	This is the override for the slow feedrate that was set in the machine setup parameter **Z final pick, SLOW feedrate**. This is used for the same reason as the **OvrMedFeed** cycle parameter. This can only be set slower. Trying to set this higher will only result in the software using the original feedrate. (Optional)
OvrRPM	This is the override for the RPM that was set in the machine setup parameter **RPM for calibration and tool measurement**. This is used for the same reason as the **OvrMedFeed** cycle parameter. This can only be set slower. Trying to set this higher will only result in the software using the original RPM. (Optional)

Warning: Large tools can result in probe damage if the touch feedrate is set too fast. For this reason, the parameters: **OvrMedFeed**, **OvrSlwFeed**, and **OvrRPM** have been added to enable the programmer/operator to override the values in the parameters for the specific tool being checked or set.

You must have the tool positioned over the probe stylus so that the tooth that sticks down the furthest is directly over the center of the probe stylus and above the stylus less then 0.200” (5.08 mm).

NOTE: If the spindle is locked, you may have to unlock it to manually orient the tool tooth over the probe stylus.

To measure the tool length:

1. Jog the tool to the top of the probe stylus so that the tooth that sticks down the furthest is directly over the center of the probe stylus.
2. From the MDI mode, input:

   LenSpecMea Tool#(tool#) EstDiam(n), Exit and press the **START** button. Where **Tool#** is the tool number and **EstDiam** is roughly the diameter of the special tool.
   For example: LenSpecMea Tool# 3 EstDiam 3.5
3. The spindle will turn on in reverse and the Z-axis should go down and touch the top of the probe stylus keeping the X and Y position the same then rapid up 0.02” (0.508 mm) and then retouch using the slow feedrate programmed in the machine variables. The cycle will then update the tool-length offset register clearing any value in the length wear register, turn the spindle off and return the tool to the Z height where it started.
4. The Tool Length has been set and you can now change to another tool, and repeat steps 1 through 3.

Manual Tool Diameter Measure for Special Tools (DiaSpecMea)

Format: DiaSpecMea Tool#(tool#) EstDiam(n) DistDown(n) OvrMedFeed(n) OvrSlwFeed(n) OvrRPM(n)

This cycle is used to measure the diameter of irregularly shaped tools or tools with a hole in the center of the bottom. Refer to **Table 4-29**.

Table 4-29, DiaSpecMea Entry Fields

Entry Fields	Description
Tool#	Tool number. (Required) The **Tool#** cycle parameter must be the same as the current tool in the spindle.
EstDiam	This is the rough diameter of the tool. (Required) The diameter specified in this parameter should be larger then the actual diameter of the tool being measured but no more then 0.04” (1.0 mm) over. If you have a left-handed tool, you would give a negative value to the diameter so the spindle will turn on in forward direction.
DistDown	The distance to go down along the side of the probe stylus when doing a diameter pick. The maximum **DistDown** value is 0.55” (13.97 mm) or the tool may crash into the probe or table. If you enter a value larger than 0.55” (13.97 mm), the control will issue an error message. If **DistDown** is not set, the cycle will use a default value of 0.1” (2.54 mm). (Optional) [Default: 0.1”] Ball nose cutters and special cutters that require a move down more than 0.55” (13.97 mm) are not supported.
OvrMedFeed	This is the override for the medium feedrate that was set in the machine setup parameter **Z first pick, MEDIUM feedrate**. Sometimes there may be a tool that has a large diameter making it necessary to slow it down to prevent the touch probe from being hit too hard. This can only be set slower. Trying to set this higher will only result in the software using the original feedrate. (Optional)
OvrSlwFeed	This is the override for the slow feedrate that was set in the machine setup parameter **Z final pick, SLOW feedrate**. This is used for the same reason as the **OvrMedFeed** cycle parameter. This can only be set slower. Trying to set this higher will only result in the software using the original feedrate. (Optional)

(Continued…)

Table 4-29, DiaSpecMea Entry Fields (Continued)

Entry Fields	Description
OvrRPM	This is the override for the RPM that was set in the machine setup parameter **RPM for calibration and tool measurement**. This is used for the same reason as the **OvrMedFeed** cycle parameter. This can only be set slower. Trying to set this higher will only result in the software using the original RPM. (Optional)

Warning: Large tools can result in probe damage if the touch feedrate is set too fast. For this reason, the parameters: **OvrMedFeed**, **OvrSlwFeed**, and **OvrRPM** have been added to enable the programmer/operator to override the values in the parameters for the specific tool being checked or set.

You must:

- Load the tool in the spindle and call up that tools offset.
- Know the distance from the top of the probe stylus down you will have to move so that the largest part of the tool diameter is even with the side of the probe stylus for diameter measurement. That value will be placed in **DistDown** if different then the default 0.1” (2.54 mm).
- Position the tool over the probe stylus so that the tooth that sticks down the furthest is directly over the center of the probe stylus and above the stylus less then 0.200 (5.08 mm).

To measure the tool diameter:

1. Jog the tool to the top of the probe stylus so that the tooth that sticks down the furthest is directly over the center of the probe stylus.
2. From the MDI mode and the spindle off, input:

   DiaSpecMea Tool#(n) EstDiam(n) DistDown(n), exit and press the **START** button. Where **Tool#**(n) is the tool number, **EstDiam**(n) is roughly the diameter of the special tool (this should be larger but not more then 0.100” (2.54 mm) larger), and **DistDown**(n) is the Z-axis move down needed if different then the default 0.100” (2.54mm) so that the largest part of the tool diameter comes in contact with the edge of the probe stylus.
   For example:
   DiaSpecMea Tool# 3 EstDiam 3.5 DistDown .25
   exit and press the **START** button.

3. The Z-axis will feed down with the spindle on, touching the top of the probe stylus. Once the top of the probe is found, the Z-axis will rapid back up above the probe and move over to one side of the probe stylus. The tool will then move down the distance in **DistDown** or 0.1" (2.54 mm) if **DistDown** is not programmed. Then, with the spindle turning in reverse, the canned cycle will touch the side of the tool to the probe stylus twice on opposite sides establishing the tool's diameter. The new diameter will then be stored in that tool's diameter register and clear any value in the diameter wear register. The Z-axis will rapid up to machine home.

4. The Tool Diameter has now been set and you can change to another tool and repeat steps 1 through 3.

Tool Breakage, Length, and Diameter Wear Detection (BrkWearDet)

Format: BrkWearDet Tool#(tool#) EstDiam(n) MaxLenAdj(n) MaxDiaAdj(n) DistDown(n) Update(n) OvrMedFeed(n) OvrSlwFeed(n) OvrRPM(n)

Refer to **Table 4-30**.

Table 4-30, BrkWearDet Entry Fields

Entry Fields	Description
Tool#	Tool number. (Required) The **Tool#** cycle parameter will be the tool number you want checked.
EstDiam	This is the rough diameter on the bottom of the tool. (Optional) The diameter specified in this parameter should be roughly the diameter on the bottom of the tool that you want to be over the center of the probe stylus. If you have a left-handed tool, you would give a negative value to this diameter so the spindle will turn on forward verses reverse. When stepping over for checking the diameter of the tool, this cycle will use the diameter in the tool table for the tool being checked.
MaxLenAdj	The maximum length wear value limit. The cycle will check to see if the cutter length has changed by more then this amount and will alarm, stopping the program if exceeded. If not set, the cycle will not check the tool length. (Optional) **NOTE:** At least one, **MaxLenAdj** or **MaxDiaAdj** must be set or the cycle will alarm.
MaxDiaAdj	The maximum diameter wear value limit. The cycle will check to see if the cutter diameter has changed by more then this amount and will alarm, stopping the program if exceeded. If not set, the cycle will not check the tool diameter. (Optional) **NOTE:** At least one, **MaxLenAdj** or **MaxDiaAdj** must be set or the cycle will alarm.
DistDown	The distance to go down along the side of the probe stylus when doing a diameter check. The maximum **DistDown** value is 0.55” (13.97 mm) or the tool may crash into the probe or table. If you enter a value larger than 0.55” (13.97 mm), the control will issue an error message. If **DistDown** is not set, the cycle will use a default value of 0.1” (2.54 mm). (Optional) [Default: 0.1”] Ball nose cutters and special cutters that require a move down more than 0.55” (13.97 mm) are not supported.

(Continued…)

Table 4-30, BrkWearDet Entry Fields (Continued)

Entry Fields	Description
Update	If this is undefined or set to **No**, the **Break and Wear** cycle will not update the diameter or length wear register each time it checks a tool. If set to **Yes**, the cycle will update the wear registers. In both cases the control will alarm when the maximum limit set in **MaxLenAdj** or **MaxDiaAdj** has been exceeded. (Optional)
OvrMedFeed	This is the override for the medium feedrate that was set in the machine setup parameter **Z first pick, MEDIUM feedrate**. Sometimes there may be a tool that has a large diameter making it necessary to slow it down to prevent the touch probe from being hit too hard. This can only be set slower. Trying to set this higher will only result in the software using the original feedrate. (Optional)
OvrSlwFeed	This is the override for the slow feedrate that was set in the machine setup parameter **Z final pick, SLOW feedrate**. This is used for the same reason as the **OvrMedFeed** cycle parameter. This can only be set slower. Trying to set this higher will only result in the software using the original feedrate. (Optional)
OvrRPM	This is the override for the RPM that was set in the machine setup parameter **RPM for calibration and tool measurement**. This is used for the same reason as the **OvrMedFeed** cycle parameter. This can only be set slower. Trying to set this higher will only result in the software using the original RPM. (Optional)

Warning: Large tools can result in probe damage if the touch feedrate is set too fast. For this reason, the parameters: **OvrMedFeed**, **OvrSlwFeed**, and **OvrRPM** have been added to enable the programmer/operator to override the values in the parameters for the specific tool being checked or set.

Warning: Running this cycle without first initially setting the length and diameter offset could result in damage to the probe and/or the machine tool. **CalTlPrb** (**Probe Calibration**) and **LenSpecMea** (**Length and Diameter**) Automatic Tool Length and Diameter set, or **LenSpecMea** (**Length Special Manual Tool Length Measure for Special Tools**) and **DiaSpecMea (Diameter Special Manual Tool Diameter Measure for Special Tools**) must be run first before using the **BrkWearDet** (**Break and Wear**) cycle.

The **Break and Wear** cycle loads the tool, checks, and updates length and diameter wear registers, if specified, until a maximum value is exceeded, then it will alarm out stopping the program.

This cycle can be used in place of calling up a tool before running it.

You must know the distance from the top of the probe stylus down that you will have to move so that the largest part of the tool diameter is even with the side of the probe stylus for diameter measurement. That value will be placed in **DistDown** if different then the default 0.1” (2.54 mm).

To check the tool length and/or tool diameter for wear or breakage:

In place of the usual **Tool#(tool#) MCode6** command, use:

BrkWearDet Tool#(tool#) EstDiam(n) MJaxLenAdy(n) MaxDiaAdj(n) DistDown(n) Update(n)

at a tool change according to the instructions above and the control will check the tool prior to using it. To activate the new offset wear values you must call that tool with “Tool#(Tool#) MCode6” after this cycle has been run.

Spindle Probe Cycles

This topic describes operation and an overview of the conversational programming spindle probing cycles available in 6000i.

Before using your spindle probe for part setup, you must set the probe up according to the probe manufacturer’s specification so it is set to turn on with a signal (if cordless) from the optical module sending unit and to automatically time out after approximately 120 seconds.

Also, before using the spindle probe or spindle probe cycles, you must have the tool number of the spindle probe active with its tool attribute “Type” set to “Touch Probe” verses “Milling Cutter” as shown below with tool #1.

Tool Number	Diameter	Length	Diameter Wear	Length Wear	Type
1	0.5	-11	0	0	Touch Probe
2	0	0	0	0	Milling Cutter

Rotation, Mirroring, and Scaling with (RMS) is not allowed while running these cycles. If any of these cycles are in a subprogram, you cannot call them using RMS. Plane will be set to XY when these cycles are complete.

The following topics are described:

- **Spindle Probe Cycle Designations**
- **Description of Spindle Probe Cycles**

Spindle Probe Cycle Designations

The following summarizes the cycles available:

CalibPtPrb **Spindle Probe Calibration Cycle**
This is used to set the effective probe stylus diameter and set the compensation factor for any run-out of the probe stylus.

You will also need to calibrate the probe using the **CalibPtPrb** cycle.

> **NOTE:** On machines that do not have spindle orientation or if you are using a corded probe or cordless UD probe and cannot orient the spindle 180 degrees during calibration, the spindle probe stylus needs to be indicated true to the spindle centerline. Also, before calibrating the probe with a wired type probe, the center of spindle rotation must be indicated exactly over the probe gauge center. In this case the accuracy of the spindle probe is only as good as the stylus concentricity to the spindle and the closeness to the probe gauge center. Calibration must be done at least once before using the spindle probe. Once calibrated, calibration does not have to be done again unless you replace the probe stylus.

EdgeFind **Single Surface Measure/Edge Find**
This cycle will find a single surface and store that surface in a work or fixture offset register if programmed. If the **SearchDir** option **TLO** is selected, the result will calibrate the Tool Length Offset of the Spindle Probe.

CornerOut **Outside Part Corner Find**
This cycle will find the X & Y surface on an outside corner of a part and store that location in a work or fixture offset register if programmed.

CornerIn **Inside Part Corner Find**
This cycle will find the X & Y surface in an inside corner of a part and store that location in a work or fixture offset register if programmed.

InOutBoss **Inside or Outside Hole or Boss Center Find**
This cycle will find the X & Y center of an inside hole or outside standing boss on a part and store that location in a work or fixture offset register if programmed.

InOutWeb **Inside or Outside Web or Slot Center Find**
This cycle will find the X or Y center of an inside or outside web or slot on a part and store that location in a work or fixture offset register if programmed. The slot or standing web must be parallel to either the X or Y axes.

ProbeMove **Protected Positioning Move**
This cycle allows for safe positioning of the probe around the part and will generate an alarm if an obstruction is encountered.

SkewComp **Skew Error or Angle Find**
This cycle will make two touches on a surface in the X- or Y-axes and stores the angle relative to the 3 O'clock position. This cycle can also activate **SkewComp** at the same time as it is measured or in a subsequent call at another place in the program without measuring again.

Description of Spindle Probe Cycles

The following spindle probe cycles are described:

- Spindle Probe Calibration (**CalibPtPrb**)
- Edge Finding (**EdgeFind**)
- Outside Corner Finding (**CornerOut**)
- Inside Corner Finding (**CornerIn**)
- Inside/Outside Boss/Hole Finding (**InOutBoss**)
- Inside/Outside Web Finding (**InOutWeb**)
- Protected Probe Positioning (**ProbeMove**)
- Skew Error Find (**SkewComp**)

Spindle Probe Calibration (CalibPtPrb)

Format: CalibPtPrbn) Top(n) DistDown(n) DistBack(n) GaugeDiam(n) DistInX(n) DistInY(n)

Refer to **Table 4-31**.

Table 4-31, CalibPtPrb Entry Fields

Entry Fields	Description
Boss	Set **Boss** to **Yes** if you are calibrating to a boss verses a ring gauge; otherwise, do not set or set to **No**. Default is **No**. (Optional)
Top	If set to **Yes**, the cycle will find the top of the part before calibrating the probe. If **Boss** cycle parameter is set to **Yes**, **Top** is forced to **Yes** as well; otherwise, the default is **No**. (Optional)
DistDown	The distance to go down from the top of the ring gauge or standing boss for calibration. This is only used if the **Boss** cycle parameter is set to **Yes**. Without any **DistDown** value, the cycle will bring the probe down past the top of the ring gauge after finding the top, 0.1” (2.54 mm). Note: If the stylus ball is greater than .2” (5.08 mm), **DistDown** must be set to at least half the ball diameter. (Optional)
DistBack	The **DistBack** parameter specifies the distance to back away from the edge for the probe to fast feed to before trying to find it. Default is 0.1” (2.54 mm) if not set. (Optional)
GaugeDiam	The diameter of the ring gauge hole the probe stylus will come in contact with. This is only to override the value in the machine setup parameter **Diameter of spindle probe Gauge** if needed and <u>should be an exact measurement</u>. (Optional)
DistInX	The distance from the starting point to move in the X-axis to find the top of the gauge. The default, if **Boss** is not set or set to **No**, is 0.1” (2.54 mm) beyond the edge of the ring gauge hole. If **Boss** is set to **Yes**, the default is the current probe position. (Optional)
DistInY	The distance from the starting point to move in the Y-axis to find the top of the gauge. The default is the current probe position. (Optional)

You must have:

1. The probe in the spindle with its tool number active and the tool type set to “Touch Probe”.
2. The Ring Gauge mounted on the machine table.

To calibrate the probe:

1. Using a “Wireless Probe ONLY”, jog the probe to the approximate center of the ring gauge by eye and into the hole of the ring gauge at the depth that you wish the probe stylus to come in contact with the inside of the ring gauge hole.

NOTE:	If you are using a wired probe as opposed to a wireless probe, you must indicate the probe stylus true to the spindle rotation center and you also must be “exactly” over the center of the gauge hole by indicating it in, because a wired type probe is not able to orient the spindle.

2. From the MDI mode, go to the **F5** (**Mill**), then **F10** (**Probe**), then **F3** (**SpinPro**), then select **Probe Calibration**. Type the appropriate information, press **exit**, and then press **START**.

3. The probe will touch four sides of the inside of the hole. The spindle will rotate (if the machine has spindle orientation) 180 degrees and touch the same four sides again establishing the center of the ring gauge. The spindle will then orient and touch four sides one more time calibrating the probe.

4. Remove the ring gauge from the machine and you are now ready to start spindle probing.

NOTE:	On machines that allow the spindle probe to be installed in the spindle with more than one orientation, the probe stylus must be indicated true to the spindle centerline or the probe will not be accurate once removed and replaced into the spindle again.

Edge Finding (EdgeFind)

Format: EdgeFind SearchDir(XPlus, XMinus, YPlus, YMinus, ZPlus, or ZMinus) Offset(0–9)

- Calibrate the work probe at least once before trying to use this cycle.
- A preliminary tool-length offset must be set by eye for the work probe and that tool offset active before using this cycle in a program. See the operations manual for setting and activating tool-length offsets.
- A preliminary work offset must be set by eye and that work coordinate active before using this cycle in a program. See the operations manual for setting and activating work coordinate offsets.
- The EdgeFind Edge Finding Cycle can be run from within a program or from the MDI mode. Refer to **Table 4-32**.

Table 4-32, EdgeFind Entry Fields

Entry Fields	Description
SearchDir	Axis and direction to find edge. **XPlus**, **XMinus**, **YPlus**, **YMinus**, **ZPlus**, **ZMinus**, or **TLO** (Required)
Offset	Work Coordinate to update with edge location in X- or Y-axes. If set, work coordinate will be updated if **XPlus**, **XMinus**, **YPlus**, and **YMinus** are specified for **SearchDir**, or Z work offset (or TLO if **updateTloOrWorkOffsetZAxis** is set to TLO) if **SearchDir** is set to **ZPlus** or **ZMinus** and Z TLO if **SearchDir** is set to **TLO**. **NOTE:** Before any tool-length offset is active, you must re-call that tool. Work coordinate register or Tool-length register is not updated if **W** is not set and a warning message tells the operator no update has taken place except when **SearchDir** is set to **TLO,** in which case the Spindle Probe TLO will always be reset. [Default: **0**] Range (0–255) (Optional)

To use the Edge Finding Cycle:

1. Place the probe in the spindle with its tool number active and the tool type set to “Touch Probe”.
2. Manually jog the probe stylus less then 0.1” (2.54 mm) away from the surface to be found.
3. Input EdgeFind SearchDir(?) Offset(n). If this is run from inside a program, this line needs to be repeated for every surface you wish to find.

> **Caution:** When positioning the probe from within the program you should always use the **ProbeMove** cycle. Refer to “Protected Probe Positioning (ProbeMove).”

4. Execute that line in MDI by exiting and pressing **START**.

Outside Corner Finding (CornerOut)

Format: CornerOut SearchQuad(XPlusYPlus, XMinusYPlus, XMinusYMinus, XPlusYMinus) Top(Yes/NO) DistDown(n) DistSide(n) DistBack(n) DistInX(n) DistInY(n) X(n) Y(n) Z(n) Offset (0–9))

- Calibrate the work probe at least once before trying to use this cycle.
- A preliminary tool-length offset must be set by eye for the work probe and that tool offset must be active before using this cycle in a program. See the operations manual for setting and activating tool-length offsets.
- A preliminary work offset must be set by eye and that work coordinate active before using this cycle in a program. See the operations manual for setting and activating work coordinate offsets.
- The CornerOut Outside Corner Finding Cycle can be run from within a program or from the MDI mode. Refer to **Table 4-33**.

Table 4-33, CornerOut Entry Fields

Entry Fields	Description
SearchQuad	Quadrant of corner to find. **XPlusYPlus** = +,+ (upper right) **XMinusYPlus** = -,+ (upper left) **XMinusYMinus** = -,- (lower left) **XPlusYMinus** = +,- (lower right) (Required)
Top	If set to **Yes**, the cycle will find the top of the part before finding the X & Y corner coordinate. Default is **No**. If **Top** is not set or is set to **No**, the Z-axis must be at the picking depth. If **Top** = **Yes**, then the Z-axis must be within 0.1” (2.54 mm) above the part. The probe stylus must be positioned within 0.1” (2.54 mm) from the outside of the corner in X & Y. (Optional)
DistDown	The distance to go down from the top of part to find X & Y coordinate of the corner. This is only used if **Top** parameter is set to **Yes**. Without any **DistDown** value, the cycle will bring the probe stylus center down past the top of the part after finding the top, 0.1” (2.54 mm). (Optional)
DistSide	The distance over from the corner to find X & Y edge. This will allow for a part corner that has a large chamfer or radius where you cannot pick the edge close to the theoretical corner or has an obstruction interfering with the default move. Default is 0.4” (10.16 mm). (Optional)

(Continued…)

Table 4-33, CornerOut Entry Fields (Continued)

Entry Fields	Description
DistBack	Specifies the distance away from the edge for the probe to fast feed to before trying to find it. Default is, 0.1” (2.54 mm) if not set. (Optional)
DistInX	The distance from the starting point to move in the X-axis to find the top of the part. The default is toward the corner being found 0.4” (10.16 mm). (Optional)
DistInY	The distance from the starting point to move in the Y-axis to find the top of the part. The default is toward the corner being found 0.4” (10.16 mm). (Optional)
X	This causes the cycle to make a protected X move to the coordinate entered relative to the current active work coordinate before finding the corner. (Optional)
Y	Same as **X** only for the Y-axis. (Optional)
Z	Same as **X** only for the Z-axis. (Optional)
Offset	Work Coordinate to update with edge location in X- and Y-axes. If set, work coordinate will be updated. Work coordinate register will not be updated if not set and a warning message will tell the operator no update has taken place if **Offset** is not set. [Default: **0**] Range (0–9) (Optional)

To use the Outside Corner Finding Cycle:

1. Place the probe in the spindle with its tool number active and the tool type set to “Touch Probe”.
2. Manually jog the probe stylus less then 0.1” (2.54 mm) away from the outside of the corner you wish to find in X & Y. If **Top** = **Yes**, the Z-axis should be within 0.1” (2.54 mm) above the part otherwise the Z-axis should be at the side picking depth.
3. Input CornerOut SearchQuad(XPlusYPlus) Offset(n)
   If this is run from inside a program, this line needs to be repeated for every corner you wish to find or whose position you want to reestablish.

   Caution: When positioning the probe from within the program you should always use the **ProbeMove** (**Protected Probe Positioning**) cycle (refer to “Protected Probe Positioning (ProbeMove)”) or use the **X**, **Y**, or **Z** parameters for the same purpose.

4. Execute that line in MDI by exiting and pressing **START**.

Inside Corner Finding (CornerIn)

Format: CornerIn SearchQuad(XPlusYPlus, XMinusYPlus, XMinusYMinus, XPlusYMinus) Top(Yes/No) DistDown(n) DistSide(n) DistBack(n) DistInX(n) DistInY(n) X(n) Y(n) Z(n) Offset(0-9)

- Calibrate the work probe at least once before trying to use this cycle.
- A preliminary tool-length offset must be set by eye for the work probe and that tool offset active before using this cycle in a program. See the operations manual for setting and activating tool-length offsets.
- A preliminary work offset must be set by eye and that work coordinate active before using this cycle in a program. See the operations manual for setting and activating work coordinate offsets.
- The CornerIn Inside Corner Finding Cycle can be run from within a program or from the MDI mode. Refer to **Table 4-34**.

Table 4-34, CornerIn Entry Fields

Entry Fields	Description
SearchQuad	Quadrant of corner to find. **XPlusYPlus** = +,+ (upper right) **XMinusYPlus** = -,+ (upper left) **XMinusYMinus** = -,- (lower left) **XPlusYMinus** = +,- (lower right) (Required)
Top	If set to **Yes**, the cycle will find the top of the part before finding the X & Y corner coordinate. Default is **No**. If **Top** is not set or is set to **No**, the Z-axis must be at the picking depth. If **Top** = **Yes**, then the Z-axis must be within 0.1” (2.54 mm) above the part. The probe stylus must be positioned within 0.1” (2.54 mm) from the outside of the corner in X & Y. (Optional)
DistDown	The distance to go down from the top of part to find X & Y coordinate of the corner. This is only used if **Top** parameter is set to **Yes**. Without any **DistDown** value, the cycle will bring the probe stylus center down past the top of the part after finding the top, 0.1” (2.54 mm). (Optional)
DistSide	The distance over from the corner to find X & Y edge. This will allow for a part corner that has a large chamfer or radius where you cannot pick the edge close to the theoretical corner or has an obstruction interfering with the default move. Default is 0.4” (10.16 mm). (Optional)

(Continued…)

Table 4-34, CornerIn Entry Fields (Continued)

Entry Fields	Description
DistBack	Specifies the distance away from the edge for the probe to fast feed to before trying to find it. Default is, 0.1" (2.54 mm) if not set. (Optional)
DistInX	The distance from the starting point to move in the "X" axis to find the top of the part. The default is toward the corner being found 0.4" (10.16 mm). (Optional)
DistInY	The distance from the starting point to move in the "Y" axis to find the top of the part. The default is toward the corner being found 0.4" (10.16 mm). (Optional)
X	This causes the cycle to make a protected X move to the coordinate entered relative to the current active work coordinate before finding the corner. (Optional)
Y	Same as **X** only for the Y-axis. (Optional)
Z	Same as **X** only for the Z-axis. (Optional)
Offset	Work Coordinate to update with edge location in X- and Y-axes. If set, work coordinate will be updated. Work coordinate register will not be updated if not set and a warning message will tell the operator no update has taken place if **Offset** is not set. [Default: **0**] Range (0–9) (Optional)

To use the Inside Corner Finding Cycle:

1. Place the probe in the spindle with its tool number active and the tool type set to "Touch Probe".
2. Manually jog the probe stylus less then 0.1" (2.54 mm) away from the outside of the corner you wish to find in X & Y. If **Top** = **Yes**, the Z-axis should be within 0.1" (2.54 mm) above the part; otherwise, the Z-axis should be at the side picking depth.
3. Input CornerIn SearchQuad(XPlusYPlus) Offset(0–9)
   If this is run from inside a program, this line needs to be repeated for every corner you wish to find or whose position you want to reestablish.

   > **Caution:** When positioning the probe from within the program you should always use the **ProbeMove** (**Protected Probe Positioning**) cycle (see "Protected Probe Positioning (ProbeMove)") or use the **X**, **Y**, or **Z** parameters for the same purpose.

4. Execute that line in MDI by exiting and pressing **START**.

Inside/Outside Boss/Hole Finding (InOutBoss)

Format: InOutBoss Side(In/Out) Length(n) Width(n) Top(Yes/No) DistDown(n) DistBack(n) DistInX(n) DistInY(n) X(n) Y(n) Z(n) Offset(0–9) RepeatMeas(Yes/No)

- Calibrate the work probe at least once before trying to use this cycle.
- A preliminary tool-length offset must be set by eye for the work probe and that tool offset active before using this cycle in a program. See the operations manual for setting and activating tool-length offsets.
- A preliminary work offset must be set by eye and that work coordinate active before using this cycle in a program. See the operations manual for setting and activating work coordinate offsets.
- The InOutBoss Inside or Outside Boss/Hole Finding Cycle can be run from within a program or from the MDI mode. Refer to **Table 4-35**.

Table 4-35, InOutBoss Entry Fields

Entry Fields	Description
Side	Inside or Outside. **In** = Inside Hole **Out** = Outside Boss (Required)
Length	Estimated length in the X-axis of boss/hole if rectangular or the Diameter if round. (Required)
Width	Estimated width in the Y-axis of boss/hole. **Width** is only specified if boss or hole is rectangular in shape. (Optional)
Top	If set to "Yes" the cycle will find the top of the part before finding center of hole or boss. If **Side** parameter is set to **Out**, **Top** is forced to **Yes** as well; otherwise, the default is **No**. (Optional)
DistDown	The distance to go down from the top of part to find X & Y coordinate of the center. This is only used if **Top** parameter is set to **Yes**. Without any **DistDown** value, the cycle will bring the probe stylus center down past the top of the part after finding the top, 0.1" (2.54 mm). (Optional)
DistBack	Specifies the distance away from the edge for the probe to fast feed to before trying to find it. Default is, 0.1" (2.54 mm) if not set. (Optional)

(Continued…)

Table 4-35, InOutBoss Entry Fields (Continued)

Entry Fields	Description
DistInX	The distance from the starting point to move in the X-axis to find the top of the part. The default, if **Side** is not set or set to **In**, is 0.1” beyond the edge of the boss/hole. If **Side** is set to **Out**, the default is the current probe position. (Optional)
DistInY	The distance from the starting point to move in the Y-axis to find the top of the part. The default is the current probe position. (Optional)
X	This causes the cycle to make a protected X move to the coordinate entered relative to the current active work coordinate before finding the boss/hole center. (Optional)
Y	Same as **X** only for the Y-axis. (Optional)
Z	Same as **X** only for the Z-axis. (Optional)
Offset	Work Coordinate to update with the center location in X- and Y-axes. If set, work coordinate will be updated. Work coordinate register will not be updated if not set and a warning message will tell the operator no update has taken place if **Offset** is not set. [Default: **0**] Range (0–9) (Optional)
RepeatMeas	If set to **Yes**, the cycle will do a preliminary measure in the “X” axis to get on center before measuring the Y-axis, making a total of 6 touches. If set to **No**, the cycle will only measure “X” once for a total of 4 touches. Default is **No**. (Optional)

To use the Inside/Outside Boss/Hole Finding Cycle:

1. Place the probe in the spindle with its tool number active and the tool type set to “Touch Probe”.
2. Manually jog the probe stylus the approximate center in X & Y within 0.1” (2.54 mm). If **Top** = **Yes**, the Z-axis should be within 0.1” (2.54 mm) above the part otherwise the Z-axis should be at the side picking depth.
3. Input InOutBoss Side(In/Out) Length(n) Width(n) Offset(0–9)
   If this is run from inside a program, this line needs to be repeated for every boss/hole you wish to find or whose position you want to reestablish.

   > **Caution:** When positioning the probe from within the program, you should always use the **ProbeMove** (**Protected Probe Positioning**) cycle (see “Protected Probe Positioning (ProbeMove)”) or use the **X**, **Y**, or **Z** parameters for the same purpose.

4. Execute that line in MDI by exiting and pressing **START**.

Inside/Outside Web Finding (InOutWeb)

Format: InOutWeb Side(In/Out) Length(n) Width(n) Top(Yes/No) DistDown(n) DistBack(n) DistInX(n) DistInY(n) X(n) Y(n) Z(n) Offset(0–9)

- An inside Web is a slot. An outside Web is a standing rib.
- Webs can only be measured in the X- or Y-axis.
- Calibrate the work probe at least once before trying to use this cycle.
- A preliminary tool-length offset must be set by eye for the work probe and that tool offset active before using this cycle in a program. See the operations manual for setting and activating tool-length offsets.
- A preliminary work offset must be set by eye and that work coordinate active before using this cycle in a program. See the operations manual for setting and activating work coordinate offsets.
- The InOutWeb Inside or Outside Web Finding Cycle can be run from within a program or from the MDI mode. Refer to **Table 4-36**.

Table 4-36, InOutWeb Entry Fields

Entry Fields	Description
Side	Inside or Outside. **In** = Inside Hole **Out** = Outside Boss (Required)
Length	Estimated X width of Web if measuring in the X-axis. **Length** or **Width** must be specified but only one, not both.
Width	Estimated Y width of Web if measuring in the Y-axis. **Length** or **Width** must be specified but only one, not both.
Top	If set to **Yes**, the cycle will find the top of the part before finding center of Web. If **Side** parameter is set to **Out**, **Top** is forced to **Yes** as well; otherwise, the default is **No**. (Optional)
DistDown	The distance to go down from the top of part to find X or Y coordinate of the center. This is only used if **Top** parameter is set to **Yes**. Without any **DistDown** value, the cycle will bring the probe stylus center down past the top of the part after finding the top, 0.1” (2.54 mm). (Optional)
DistBack	Specifies the distance away from the edge for the probe to fast feed to before trying to find it. Default is 0.1” (2.54 mm), if not set. (Optional)

(Continued…)

Table 4-36, InOutWeb Entry Fields (Continued)

Entry Fields	Description
DistInX	The distance from the starting point to move in the X-axis to find the top of the part. The default, if **Side** is not set or set to **In**, is 0.1” beyond the edge of the web. If **Side** is set to **Out**, the default is the current probe position. (Optional)
DistInY	The distance from the starting point to move in the Y-axis to find the top of the part. The default is the current probe position. (Optional)
X	This causes the cycle to make a protected X move to the coordinate entered relative to the current active work coordinate before finding the web center. (Optional)
Y	Same as **X** only for the Y-axis. (Optional)
Z	Same as **X** only for the Z-axis. (Optional)
Offset	Work Coordinate to update with the center location in X- or Y-axes. If set, work coordinate will be updated. Work coordinate register will not be updated if not set and a warning message will tell the operator no update has taken place if **Offset** is not set. [Default: **0**] Range (0–9) (Optional)

To use the Inside/Outside Web Finding Cycle:

1. Place the probe in the spindle with its tool number active and the tool type set to “Touch Probe”.
2. Manually jog the probe stylus the approximate center in X or Y within 0.1” (2.54 mm). If **Top** = **Yes**, the Z-axis should be within 0.1” (2.54 mm) above the part otherwise the Z-axis should be at the side picking depth.
3. Input InOutWeb Side(In/Out) Length(n) Offset(0–9)
   If this is run from inside a program, this line needs to be repeated for every web you wish to find or whose position you want to reestablish.

   > **Caution:** When positioning the probe from within the program you should always use the **ProbeMove** (**Protected Probe Positioning**) cycle (see “Protected Probe Positioning (ProbeMove)”) or use the **X**, **Y**, or **Z** parameters for the same purpose.

4. Execute that line in MDI by exiting and pressing **START**.

Protected Probe Positioning (ProbeMove)

Format: ProbeMove X(n) Y(n) Z(n) Feed(n)

- When an X, Y, and/or Z move is programmed using the **ProbeMove** (Protected Positioning Cycle), the control will stop and alarm if the probe stylus is triggered before reaching the target set in the X, Y, and/or Z parameters.
- This cycle is intended to offer some degree of safety when moving the probe around the part; however, it is not fool proof and will not protect against gross bad programming where the probe body would encounter an obstruction before the probe stylus is triggered. Extreme care should be taken to avoid this condition as probe damage may result.
- Calibrate the work probe at least once before trying to use this cycle.
- A preliminary tool-length offset must be set by eye for the work probe and that tool offset active before using this cycle. See the operations manual for setting and activating tool-length offsets.
- A preliminary work offset must be set by eye and that work coordinate active before using this cycle. See the operations manual for setting and activating work coordinate offsets.
- The ProbeMove Protected Probe Positioning Cycle can be run from within a program or from the MDI mode. Refer to **Table 4-37**.

Table 4-37, ProbeMove Entry Fields

Entry Fields	Description
X	X Target position relative to current active work coordinate.
Y	Y Target position relative to current active work coordinate.
Z	Z Target position relative to current active work coordinate combined with the current active tool-length offset.
Feed	Feedrate at which to travel to target. **Feed** is only active for the current move so it must be restated every time or the default will take precedence. The default is set in the machine setup parameter **Positioning feedrate normally**. (Optional)

To use the ProbeMove Protected Probe Positioning Cycle:

1. Place the probe in the spindle with its tool number active and the tool type set to “Touch Probe”.
2. Input: “ProbeMove X(n) Y(n) Z(n) Feed(n)”.
   If this is run from inside a program, this line needs to be repeated for every move you wish to make.
3. Execute that line in MDI by exiting and pressing **START**.

Skew Error Find (SkewComp)

Format: SkewComp Action(Find/FindActive/Activate) EstAngle(n) DistPicks(n) Top(Yes/No) DistDown(n) DistBack(n) DistInX(n) DistInY(n) X(n) Y(n) Z(n)

- RMS cannot be used with **SkewComp**, skew error find.
- Skew error is only supported for along the side edge of a part relative to the X,Y plane.
- Calibrate the work probe at least once before trying to use this cycle.
- A preliminary tool-length offset must be set by eye for the work probe and that tool offset active before using this cycle in a program. See the operations manual for setting and activating tool-length offsets.
- A preliminary work offset must be set by eye and that work coordinate active before using this cycle in a program. See the operations manual for setting and activating work coordinate offsets.
- The probe must be pre-positioned to the proper spot in relation to the part in accordance with the specified **EstAngle** cycle parameter as described below or an X, Y, and/or Z should be included for pre-positioning.
- The SkewComp Skew Error Finding Cycle can be run from within a program or from the MDI mode. Refer to **Table 4-38**, **SkewComp Entry Fields**.

Table 4-38, SkewComp Entry Fields

Entry Fields	Description
Action	**Find** Finds the skew angle, but does not activate skew compensation. **FindActive** Finds the skew angle, and activates skew compensation. **Activate** Activates skew compensation with the current skew value, but will not rerun the cycle on the part. [Default: **Find**] **NOTE:** If **Activate** is used, all other **SkewComp** parameters are ignored. **NOTE:** Before using **SkewComp Activate**, you must have called **SkewComp** at least once with **Find** or **FindActive**, or the error message “Skew error has not been found!” is displayed. Skew compensation will be activated around the current active work coordinate and will only work from within the program being run. Skew compensation cannot be activated directly or indirectly using **SkewComp** from the MDI mode. The operator can run the **SkewComp** from MDI but must place **SkewComp Activate** inside the program for skew compensation to take effect. An **Offset** work coordinate call will deactivate skew compensation, necessitating a re-issuance of **SkewComp Activate** to activate skew compensation. Using **FindActive** or **Activate** will default the control to Absolute. If you are in Incremental, you will need to switch back after the cycle has been run. (Optional)
EstAngle	Estimated amount of angle from 3 O’clock. Default is “0” which will cause the cycle to find the angle of the back edge of the part starting its first pick in the upper-left corner and making the second pick to the left of that, as you are facing the surface being picked. Examples: **EstAngle**=90 Would start in the lower-left side, picking in the X positive direction, finding the skew of the left side of the part. **EstAngle**=-90 Would start in the upper-right side, picking in the X negative direction, finding the skew of the right side of the part. **EstAngle**=180 Would start in the lower-right front edge of the part, picking in the Y positive direction, finding the skew of the front edge of the part. [Default: 0] (Optional)
DistPicks	The distance from the first pick to the second pick. Default is 2.0” (50.8 mm) (Optional)
Top	If set to **Yes**, the cycle will find the top of the part before finding part skew angle. Default is **No**. If **Top** is set to **Yes**, the probe stylus should be pre-positioned within 0.1” (2.54 mm) above the part. If **Top** is set to **No**, the probe stylus should be positioned at the Z-axis depth from which you want to make side picks. (Optional)

Table 4-38, SkewComp Entry Fields (Continued)

Entry Fields	Description
DistDown	The distance to go down from the top of part to find part skew angle. This is only used if **Top** parameter is set to **Yes**. Without any **DistSide** value, the cycle will bring the probe stylus center down past the top of the part after finding the top, 0.1" (2.54 mm). (Optional)
DistBack	Specifies the distance away from the edge for the probe to fast feed to before trying to find it. Default is, 0.1" (2.54 mm) if not set. This would be used to make sure that the cycle is picking from far enough away from the edge so that it will not trigger the probe prematurely when stepping over to make the second pick. **Hint:** If the **EstAngle** parameter is relatively accurate, this parameter will not be needed because the default will be good enough. (Optional)
DistInX	The distance from the starting point to move in the "X" axis to find the top of the part. The default is 1.0" (25.4 mm) toward the part at the angle specified in the **EstAngle** parameter. (Optional)
DistInY	The distance from the starting point to move in the "Y" axis to find the top of the part. The default is 1.0" (25.4 mm) toward the part at the angle specified in the **EstAngle** parameter. (Optional)
X	This causes the cycle to make a protected X move to the coordinate entered relative to the current active work coordinate before finding the skew angle. (Optional)
Y	Same as **X** only for the Y-axis. (Optional)
Z	Same as **X** only for the Z-axis. (Optional)

To use the Skew Error Finding Cycle:

1. Place the probe in the spindle with its tool number active and the tool type set to "Touch Probe".
2. Manually jog the probe stylus to the appropriate start position relative to the part as specified by the **EstAngle** cycle parameter in **Table 4-38**. X or Y should be within 0.1" (2.54 mm) of the part edge. If **Top** = **Yes**, the Z-axis should be within 0.1" (2.54 mm) above the part; otherwise, the Z-axis should be at the side picking depth. If run from within a program, probe must be pre-positioned.
3. Input SkewComp Action(Find/FindActive/Activate) EstAngle(n)
   If this is run from inside a program, this line needs to be repeated every time you wish to find a skew angle.

> **Caution:** When positioning the probe from within the program you should always use the **ProbeMove** (**Protected Probe Positioning**) cycle (see "Protected Probe Positioning (ProbeMove)") or use the **X**, **Y**, or **Z** parameters for the same purpose.

4. Execute that line in MDI by exiting and pressing **START**.

Using the Z Work Offset Update Feature

If you would like to calibrate all your tools to a fixed Z axis location on the machine then use the Z Axis Work Offset to shift all the tools to the top of a part, you must use the Edge Finding (**EdgeFind**) cycle with Spindle Probing parameter **updateTloOrWorkOffsetZAxis** set to **WorkOffset**. Only **SearchDir Z+**, **Z-**, and **TLO** cycle parameters will affect the Z axis.

1. First, use **TLO** to set the spindle probe tool length offset to the fixed surface on the machine where all the tools are calibrated.
2. Next, use **Z+** or **Z-** on the top of the work piece or to where you want the Z zero to be located, to set the Z-axis work offset shift to that Z position.

> **Warning:** . Both, the tool length offset and the "Offset" work offset must be active before the Z part zero point will be correct. If one of the other is not active at the same time a collision could occur.

Section 5 - Editing Programs

Write and edit conversational program blocks using the CNC's Conversational Program Editor (the Edit screen). Activate the Conversational Program Editor to put the CNC in the Edit Mode.

The following topics are described in this section:

- **Activating the Conversational Program Editor**
- **Saving Edits**
- **Canceling Unsaved Edits**
- **Deleting a Block**
- **Inserting a Block**
- **Editing Blocks**
- **Using Comments**
- **Using Block Operations to Edit a Program**

Activating the Conversational Program Editor

You can activate the Conversational Program Editor screen either from the Program Directory or from the Manual screen. When you activate the Program Editor from the Program Directory, the highlighted program opens for editing. When you activate the Program Editor from the Manual screen, the selected program opens for editing.

To activate the Program Editor from the Program Directory:

1. In the Program Directory, highlight a program.
2. Press **Edit** (**F8**). The Program Editor opens the selected program for editing.

To activate the Program Editor from the Manual screen:

1. In the Manual screen, press **Edit** (**F3**). The Editor opens the loaded program.
2. Press **Program** (**F2**) to activate the Program Directory.
3. Highlight a program with a .M extension.
4. Press **Edit** (**F3**). The Program Editor opens the selected program for editing.

5. The Program Editor Screen

The Program Editor monitors mode changes written to a program. The mode indicators displayed in the Program Editor indicate the CNC's active modes. Refer to **Figure 5-1**.

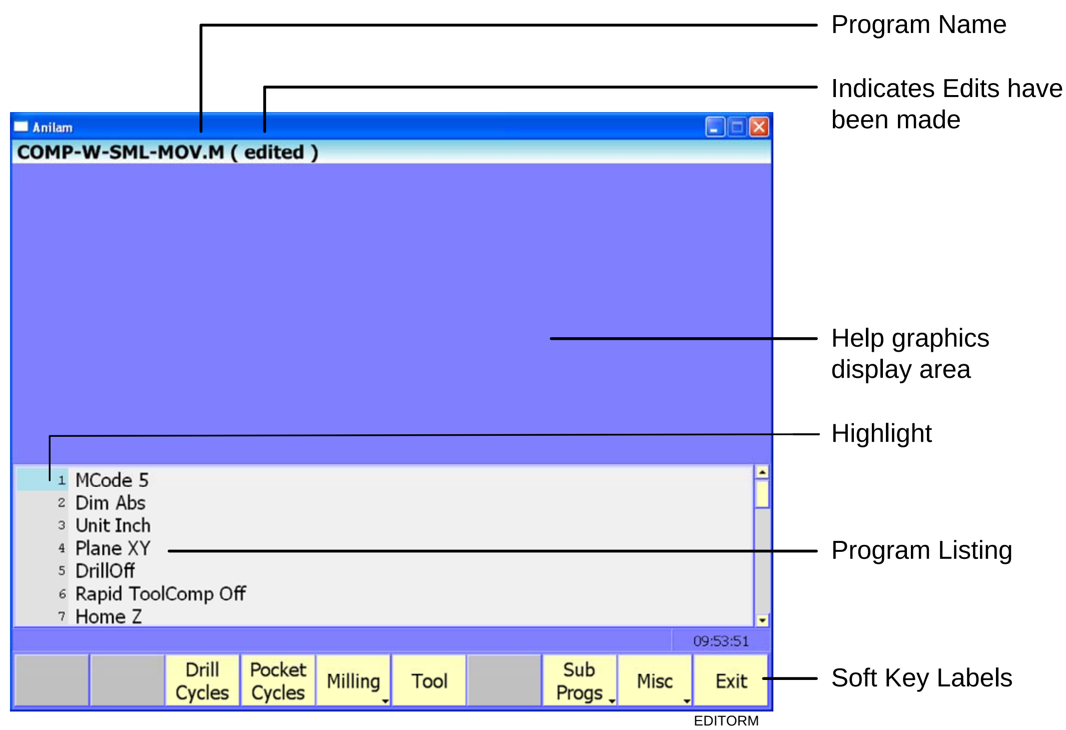


Figure 5-1, Program Editor

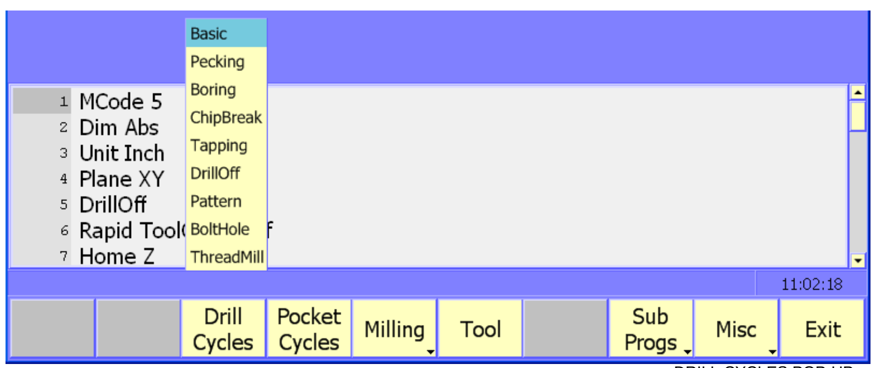
Program Name	Name of the program opened for editing.
(edited) marker	Indicates that you have edited the program, but the edits have not been saved.
Help graphics display area	Area for displaying the Help graphics display.
Program Listing	Current listing of the blocks in the open program.
Highlight	Selects a block for editing and acts as an insertion marker for adding new blocks. The CNC tracks program mode changes up to this point in the Program Listing.
Soft Key Labels	These labels define the soft key functions.

The following sets of soft keys are available.

- Default set, normally visible. Refer to **Figure 5-1, Program Editor**.
- Program Editor **SHIFT** screen soft keys. Refer to **Figure 5-2**.
- For pop-up menus, refer to:**Figure 5-3** and
  Figure 5-4, Pocket Cycles (F4) Pop-up Menu
- Milling soft keys, activated by pressing **Milling** (**F5**). Refer to
  Figure 5-5, Milling (F5) Soft Keys.
- Tool Page soft keys, activated by pressing **Tool** (**F6**). Refer to
  Figure 5-6, Tool (F6) Soft Keys.
- Sub Progs soft keys, activated by pressing **Sub Progs** (**F8**). Refer to
  Figure 5-7, Sub Progs (F8) Soft Keys.
- Misc soft keys, activated by pressing **Misc** (**F9**). Refer to
  Figure 5-8, Misc (F9) Soft Keys.

SHIFT EDIT SKEYS

Figure 5-2, Program Editor SHIFT Screen Soft Keys



DRILL CYCLES POP-UP

Figure 5-3, Drill Cycles (F3) Pop-up Menu

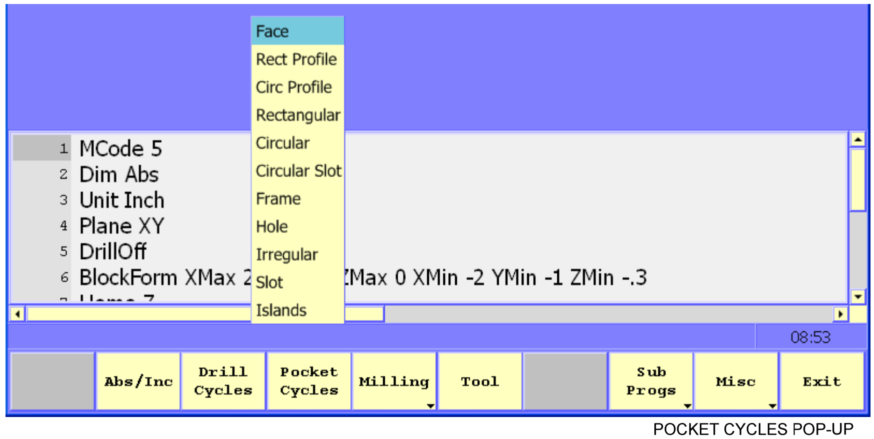


Figure 5-4, Pocket Cycles (F4) Pop-up Menu

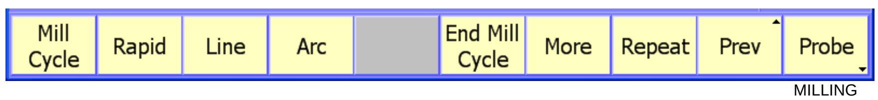


Figure 5-5, Milling (F5) Soft Keys

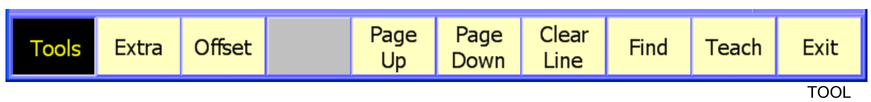


Figure 5-6, Tool (F6) Soft Keys

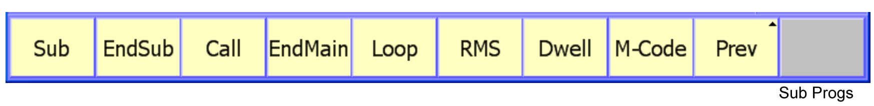


Figure 5-7, Sub Progs (F8) Soft Keys

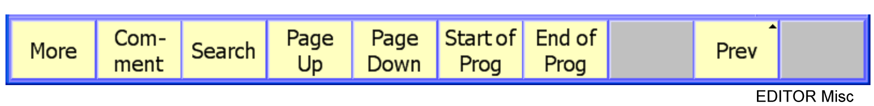


Figure 5-8, Misc (F9) Soft Keys

For more information on these soft keys, refer to *6000i CNC User's Manual*, P/N 627785-21, "Section 6 – Program Editor."

Saving Edits

The Program Listing displays text entered by the programmer. The CNC does not save edits until you exit the Editor. If the **(edited)** marker is visible at the top of the Program Editor, the open program contains unsaved edits.

To save edits:

1. In Edit Mode, press **Exit** (**F10**). The CNC saves all edits and returns to the Program Directory.

Canceling Unsaved Edits

To cancel unsaved edits:

1. In Edit Mode, press **SHIFT** to display the Program Editor **SHIFT** soft key labels. Refer to **Figure 5-2, Program Editor SHIFT Screen Soft Keys**.
2. Press **Quit** (**F10**). The CNC displays the message **ProgramFilename has changes. Would you like to quit without saving?** and changes the soft key labels.
3. Press **Yes** (**F1**). The CNC returns to the Program Directory without saving the edits. Press **No** (**F3**) to cancel.

Deleting a Block

To delete a program block:

1. In Edit Mode, highlight a block.
2. Press **CLEAR**.

Inserting a Block

To insert a program block:

1. In Edit Mode, highlight the block that will follow the inserted block.
2. Program the new block from the appropriate Graphic Menu. When you save the new block, it is displayed in front of the highlighted block. Blocks are automatically renumbered.

Editing Blocks

To edit a program block:

1. In Edit Mode, highlight a block.
2. Press **ENTER** if the existing block is a move or cycle. The appropriate Graphic Menu opens.
3. Highlight the entry fields that require changes. Press **CLEAR** to erase the existing values.
4. Make the appropriate changes. Press **Use** (**F10**) to close the block.

NOTE:	When the program block's Graphic Menu offers two modes (for example, Cw/Ccw), highlight the block and press **+/-** to change the selection.

The following topics are described:

- **Searching Blocks for Words or Numbers**
- **Scrolling the Program Listing**
- **Paging Through the Program Listing**
- **Jumping to First or Last Block in the Program**

Searching Blocks for Words or Numbers

Use **Search** to find a block number or word. **Search** looks only from the cursor position forward. To search an entire program, place the cursor at the beginning of the program, and then activate **Search**.

To search for a block number or word:

1. In Edit Mode, Press **Misc** (**F9**). The soft key menu changes. Refer to **Figure 5-8, Misc (F9) Soft Keys**.
2. Press **Search** (**F3**). The CNC prompts for the block number or word.
3. Use the ASCII chart to type the block number or word. (You can also use the keypad to type numbers.)
4. Press **ENTER**. The CNC searches and highlights the next block that contains the specified word or block number.

Scrolling the Program Listing

In Edit Mode, use the up and down **ARROWS** to scroll through the Program Listing.

Paging Through the Program Listing

To scroll through the Program Listing one page at a time:

1. In Edit Mode, Press **Misc** (**F9**). The CNC displays the soft key secondary functions. Refer to **Figure 5-8, Misc (F9) Soft Keys**.
2. Press the **Page Up** (**F4**) or **Page Down** (**F5**) keys to page forward or backward.
3. Press **Prev** (**F9**). The CNC redisplays the Program Editor default soft keys.

Jumping to First or Last Block in the Program

To jump to the first or last block of the Program Listing:

1. In Edit Mode, press **Misc** (**F9**). The CNC displays the soft key secondary functions. Refer to **Figure 5-8, Misc (F9) Soft Keys**.
2. Press **Start of Prog** (**F6**). The Program Listing displays the first block of the program.

   – or –

   Press **End of Prog** (**F7**). The Program Listing displays last block of the program.

Using Comments

The CNC will ignore comment blocks. You can add a new comment block to a program or convert an existing block into a comment. Comment blocks typically contain program setup or tool information, or are used to comment out existing blocks.

The following topics are described:

- **Writing a Comment Block**
- **Commenting Out Existing Blocks**
- **Canceling a Comment**

Writing a Comment Block

To write a comment block:

1. In Edit Mode, press **Misc.** (**F9**). The CNC displays the soft key secondary functions. Refer to **Figure 5-8, Misc (F9) Soft Keys**.
2. Press **Comment** (**F2**). The CNC prompts for a comment.
3. Use your keyboard to type comments.

Commenting Out Existing Blocks

To comment out an existing block:

1. In Edit Mode, highlight the block being commented out.
2. Press **0** on the keypad. The CNC displays an asterisk after the block number.

NOTE:	Off-line keyboard users, use the **0** key (not the asterisk key) to produce a comment block.

Canceling a Comment

To cancel a comment:

1. In Edit Mode, highlight the comment block to be canceled.
2. Press **0**. The CNC deletes the asterisk and will no longer ignore the block during program execution.

NOTE:	Off-line keyboard users, use the **0** key to switch the asterisk on or off.

Using Block Operations to Edit a Program

In conversational editor, use the **Misc** (**F9**)>**More** (**F1**) soft key to display the More pop-up menu. See **Table 5-1** for a description of the features. See **Figure 5-9**. To display the More pop-up menu:

1. In the conversational editor, select **Misc** (**F9**).
2. Press **F1** (**More**) to display the More pop-up menu. Refer to **Figure 5-9, Misc (F9)>More (F1) Pop-up Menu**.

Table 5-1, Misc (F9)>More (F1) Pop-up Menu

Feature	Description
Copy	Copies marked blocks into scrap buffer for a subsequent Paste operation. Marking is turned off. Selecting Copy with no blocks marked copies the current block into the scrap buffer.
Paste	Paste contents of scrap buffer in current location (i.e., above current block).
Cut	Copies marked blocks into scrap buffer and deletes them. Marking is turned off. Selecting Cut with no blocks marked cuts the current block into the scrap buffer.
Delete	Deletes marked blocks.
Open	Allows the user to open another program for editing without leaving the editor. The scrap buffer is preserved which allows blocks to be copied or moved from one program to another program.

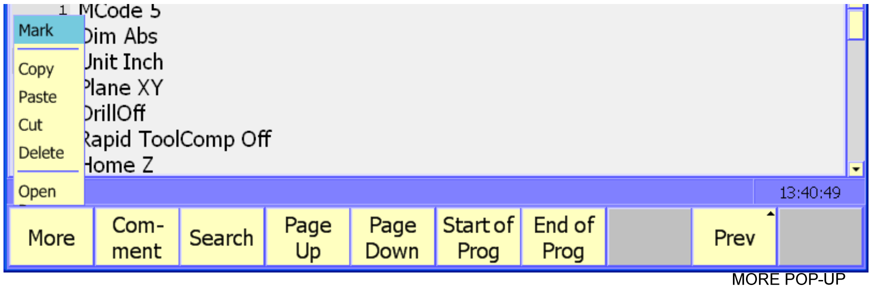


Figure 5-9, Misc (F9)>More (F1) Pop-up Menu

Section 6 - Four-Axis Programming

The following topics are described in this section:

- **Axis Types**
- **Rotary Axis Programming Conventions**
- **Programming Examples**

Axis Types

6000i-4X

The machine builder sets up the fourth-axis as linear or rotary axes. The three basic axes are X, Y, and Z. The additional axis is designated as U (6000i-4X). This section will discuss the rotary axis option in detail.

Below are the programming formats for linear or rotary additional axes:

Linear: Program as Feed Mode (**G1**) or Rapid (**G0**) moves. Only rapid and linear feed moves can be programmed. U can be programmed along with X, Y, and Z-axis in rapid, linear, and circular moves. The U-axis is always synchronous to the XYZ moves.

Rotary: Program rotary moves in degrees. The typical resolution is 0.001 degrees. Minutes and seconds cannot be programmed. Therefore, you must convert minutes and seconds to a decimal value.

Conversion formula for minutes, seconds to decimal degrees:

Minutes to decimal: min/60 = decimal degrees.

Example: 15 min/60 = 0.25 degrees

Seconds to decimal: sec/3600 = decimal degrees

Example: 30 sec/3600 = 0.008 degrees

Example:

5 deg. 30 min. 15 sec. =
5 + (30/60) + (15/3600) =
5 + 0.5 + 0.004 =
5.504 degrees

When the U-axis is programmed alone without an X, Y, or Z linear move, you must program a feedrate for the U-axis in degrees per minute (dpm).

Format: FeedU 500.0 = 500 dpm for the U axis.

FeedU is also allowed when the U-axis is linear. A federate is programmed in in/min or mm/min.

Rotary Axis Programming Conventions

A rotary axis (typically U) will program differently based on the setting of the (Axes->PhysicalAxis->U->CfgRollOver>shortestDistance) parameter, which is determined by the builder. The default for this parameter is **off**; in which case, the U-axis behaves like a linear axis. If set to **on**, the behavior of the rotary axis (U) is described below.

If programming the U-axis in Absolute:

The rotary axis will never rotate more than 180 degrees in one move. So, if a move of greater than 180 degrees is programmed, the control will resolve the number to a positive value less than 360 degrees and move to that target, taking the shortest distance (always less than 180 degrees). A move of exactly 180 degrees will always move positive and a move of exactly 360 degrees will not move at all.

If programming the U-axis in Incremental:

The rotary axis will move the exact amount of degrees programmed and in the direction indicated with the plus or minus sign. The display will reset to zero every time 360 degrees is crossed so that the highest value in the U-axis display will be 359.999 degrees depending on the displayed resolution.

Feedrate display is always vectored.

Programming Examples

All programming examples are for 4-axis machining with the rotary table mounted on the left end of the mill table, with the centerline of the rotary axis parallel to the X-axis. The face of the rotary table faces X+.

The examples contain both milling and drilling applications. Modal cycles **G81** to **G89** and **G66** can be executed at rotary locations as in XYZ locations. Non-modal canned cycles can be executed at rotary locations. Position the rotary axis before you execute a non-modal canned cycle.

The following topics are described:

- **Example 1: Drill**
- **Example 2: Mill**
- **Example 3: Mill**

Example 1: Drill

Mount the fourth axis as described above. Mount a part 6-inches wide and 8-inches long on the face of the rotary table. **shortestDistance** is set to **off**.

Table 6-1 shows a drilling example. You must drill ten 0.375-inch holes 36-degrees apart, 1-inch deep, 0.75-inches in from the end of the cylinder. Then, starting at X-2 U0, drill a spiral series of holes 36-degrees and X-0.500 inches apart each. Set X0 at the right end, Y0 at the cylinder's centerline, U0 at a pre-milled keyway on the cylinder. Measure tool offsets from the top of the cylinder, with Y-axis at 0.

Table 6-1, Four-Axis Example 1

```
* 4-AX-DRL
* SET shortestDistance to “off”
Dim Abs
Unit Inch
Plane XY
MCode 5
Rapid
Home Z
Offset Fixture# 1
Rapid X 0 Y 0 U 0
*#3 CENTER DRILL
Tool# 1    MCode 6
RPM 2400
MCode 3
BasicDrill ZDepth -.22 StartHgt .1 Feed 12
Call 1
*3/8" DRILL
Tool# 2 MCode 6
RPM 1850
MCode 3
*RE-ACTIVATE OFFSET CANCELED BY G28 IN SUBR #1
Offset Fixture# 1
ChipBreak ZDepth -1 StartHgt .1 FirstPeck .18 MinPeck .1 PeckDecr
.012 RetractDep .3334 Feed 14
Call 1
EndMain
*ROTARY HOLE LOCATIONS
Sub 1
Dim Abs
Rapid X -.75 Y 0 U 0
Loop Sub# 2 #Loops 9
Dim Abs
Rapid X -2 U 0
Loop Sub# 3 #Loops 9
DrillOff
MCode 5
Dim Abs
Rapid
```

```
Home Z
Rapid X 0 Y 0 U 0
EndSub
Sub 2
Dim Incr
Rapid U 36.0
EndSub
Sub 3
Dim Incr
Rapid X -.5 U -36.0
EndSub
```

Example 2: Mill

Mount the fourth axis as described above. Mount a part 3 inches in diameter and 5 inches long on the face of the rotary table. The part has a 0.25-inch radius turned on the end. **shortestDistance** is set to **off**.

Table 6-2 shows a milling example only. Assume that a series of six 0.25-inch wide grooves must be milled 60-degrees apart, 0.25-inch deep at the start, tapering up to 0.125-inch deep and rotating 15 degrees at the far end. The groove must follow the end contour of the part (radius). Set X0 at the right end, Y0 at the cylinder centerline, U0 at a pre-milled keyway on the cylinder. Set the tool offset so that the centerline of the 0.25-inch ball-end mill is at the centerline of the 3-inch diameter part (with Y axis at 0).

Table 6-2, Four-Axis Example 2

```
* 4-AX-MILL
* SET shortestDistance TO “off”
Dim Abs
Unit Inch
Plane XY
Rapid
Home Z
Offset     Fixture# 1
Rapid X 0 Y 0 U 0
*.25 BALL-END-MILL
Tool# 1    MCode 6
RPM 2400
MCode 3
Loop Sub# 1 #Loops 6
Dim Abs
Rapid
MCode 5
Home Z
Rapid X 0 Y 0 U 0
EndMain
*GROOVE
Sub 1
Dim Abs
Rapid X .225
Rapid Z 2.625
Line X .125 Feed 5
Dim Incr
Plane XZ
Arc CW XCenter -.25 ZCenter 0.0000 X -.25 Z .25 U -2.0
Plane XY
Line X -3.25 Z .125 U -13
Dim Abs
Rapid Z 3.225
Rapid X .225
Dim Incr
Rapid U -45.0
EndSub
```

Example 3: Mill

Mount a fourth axis as described above. Mount a part 4-inches in diameter and 8-inches long on the face of the rotary table. Support the part on the X+ end by a live center. The part has a 0.25-inch, 45-degree chamfer on one end. **shortestDistance** is set to **on**. This will prevent the need to unwind the U-axis, saving operation time.

Table 6-3 shows a thread-milling example. Assume that a 4-8 UN 2A thread must be milled from the right end, 6-inches long. The tool is tapered to conform to the thread. Set X0 at the right end, Y0 at the cylinder's centerline, U0 at a pre-milled keyway on the cylinder. Measure the tool offset from the top of the part (with Y axis at 0).

The X start position will be one pitch (0.125 in.) to the right of X0, so that the tool enters the work smoothly.

Table 6-3, Four-Axis Example 3

```
* 4-AX-THD
* SET shortestDistance TO “on"
*THIS IS TO PREVENT THE NEED TO UNWIND THE U AXIS
Dim Abs
Unit Inch
Plane XY
Rapid
Home Z
Offset     Fixture# 1
Rapid X 0 Y 0 U 0
*SPECIAL THREAD TOOL
Tool# 1    MCode 6
RPM 3500
MCode 3
Rapid X .125 Y 0 U 0
Rapid Z .1
Line Z -.075 Feed 20
* U AXIS MOVE IS (360 X 8 PITCH X 6" LONG)
* + 360 FOR 1 TURN X.125 LEAD-IN
* U MOVE WILL BE 17,640.00 DEGREES OR 49 TURNS
Dim Incr
Line X -6.125 U 17640.00
Dim Abs
Rapid
MCode 5
Home Z
Rapid X 0 Y 0 U 0
EndMain
```

.G, extension, 1-1
.M, extension, 1-1
.M, extension, program editor, 5-1
4-axis
  programming conventions, 6-2
  programming, description, 6-1
6000i CNC Technical Manual, P/N 627787-21, referenced, 1-1, 4-53
6000i CNC User's Manual, P/N 627785-21, referenced, 1-2, 5-4
6000i-4X, description, 6-1

A

absolute mode
  block, 3-1
  change, 3-5
  hot key (E), 2-1
absolute move, 3-5
absolute zero, resetting, 3-14
absolute/incremental key (E), hot key, 2-1
Action, SkewComp entry field, 4-87
Activate, SkewComp entry field, 4-87
adding, blocks to programs, 3-4
arc
  hot key (3), 2-1
  More (F4), pop-up menu, illustration, 3-24
  move, compensation, 3-7
  select the plane, 3-22
  to program
    using center-endpoint
      hot keys, 3-25
      soft keys, 3-25
    using center-included angle
      hot keys, 3-28
      soft keys, 3-28
    using endpoint-radius
      hot keys, 3-23
      soft keys, 3-23
arrow keys, editing keys, 2-2
asterisk, displayed, 5-7
axis
  four-axis, types
    linear, description, 6-1
    rotary, description, 6-1

B

basic drill cycle
  description, 4-2
  graphic menu, illustration, 4-2
BasicDrill block, to program, 4-2
block number, search for, 5-6
bolt hole cycle
  description, 4-11
  to program, 4-11
Boring block, 4-5
boring cycle
  description, 4-5
  graphic menu, bidirectional, illustration, 4-5
  to program, 4-5
Boss, 4-74
break and wear, BrkWearDet, 4-55, 4-69
BrkWearDet, break and wear, 4-55, 4-69

C

CalibPtPrb, spindle probe calibration cycle
  defined, 4-72
  description, 4-74
CalibTlPrb, probe calibration, 4-54, 4-57
call
  a loop subprogram, 4-45
  RMS subprogram, 4-46
  subprogram, 4-43
  subprograms from the main program, 4-44
canned cycles
  library, 3-1
  probing cycles, conventional programming, 4-53
  programming, description, 4-1
  spindle probe, description, 4-53
  spindle probing cycles, conversational programming, 4-71
change, hole dimensions, 4-1, 4-5, 4-6
chip break cycle
  description, 4-6
  graphic menu, illustration, 4-6
  to program, 4-6

circular pocket cycle
compensation, 3-7
description, 4-25
graphic menu, illustration, 4-25
to program, 4-25
circular profile cycle
compensation, 3-7
description, 4-21
graphic menu, illustration, 4-21
to program, 4-22
circular slot
description, 4-27
graphic menu, illustration, 4-27
to program, 4-27
cleaning up, inside diameter, 4-31
CLEAR key, editing keys, 2-2
comment
asterisk, hot key (0), 2-1
block
comment out, 5-7
description, 5-7
to cancel, 5-8
to write, 5-7
Comment (F2), from Misc (F9), 5-7
conversational
editor, Misc (F9)>More (F1), pop-up menu, 5-8
program editor
from manual screen, activate, 5-1
from program directory, activate, 5-1
programs
M, extension, 3-1
writing, 3-1
screen, illustration, 1-1
conversational programming
probe cycles, description, 4-53
spindle probe cycles, description, 4-53
spindle probing
description, 4-71, 4-73
designations, 4-72
tool probe cycle
description, 4-54, 4-56
designations, listed, 4-54
listed, 4-56
conversion formula
minutes to decimal, 6-1
seconds to degrees, 6-1
coolant
Off (M9), 3-29
On (M8), 3-29
CornerIn, inside part corner find
defined, 4-72
description, 4-79
CornerOut, outside part corner find
defined, 4-72
description, 4-77
counter bore, existing holes, 4-31
curved moves, programming, 3-22
cut through holes, 4-31
cutting direction, 4-17
cycle compensation, requirements, listed, 3-7

D

decimal point, hot key, 2-2
diameter special, DiaSpecMea, 4-55, 4-66
diameter, inside, cleaning up, 4-31
DiamOfStd, 4-57
DiaSpecMea, diameter special, 4-55, 4-66
dim(ension) block, to program, 3-5
disclaimer, iii
DistBack, 4-74, 4-78, 4-80, 4-81, 4-83, 4-88
DistDown, 4-57, 4-60, 4-66, 4-69, 4-74, 4-77, 4-79, 4-81, 4-83, 4-88
DistInX, 4-74, 4-78, 4-80, 4-82, 4-84, 4-88
DistInY, 4-74, 4-78, 4-80, 4-82, 4-84, 4-88
DistPicks, SkewComp entry field, 4-87
DistSide, 4-77, 4-79
dpm, degrees per minute, defined, 6-1
drill
4-axis, programming examples, 6-3
cycles
basic, 4-2
description, 4-1
listed, 4-1
pattern
description, 4-10

to program, 4-10
Drill Cycles (F3)
Basic, graphic menu, 4-3
DrillOff, description, 4-3
pop-up menu, illustration, 4-2, 5-3
drilling cycles, listed, 4-1
DrillOff
block, 4-3, 4-6, 4-8
block, description, 4-1
block, to program, 4-2
description, 4-3
dry run
all axes (M105), 3-30
mode
description, 3-30
M-Codes, listed, 3-30
No Z-axis (M106), 3-30
Off - cancels M105 and M106 (M107), 3-30
successful, 3-2
dwell
block, to add, 3-8
graphic menu, 3-8
hot key (8), 2-1
programming
description, 3-8
using hot keys, 3-8
using soft keys, 3-8
resolution, 3-8, 4-5

E

edge finding, EdgeFind, 4-76
EdgeFind, single surface measure/edge find
defined, 4-72
description, 4-76
Edit (F3), manual screen, 5-1
Edit (F8), program editor, 5-1
edit mode, hot keys, listed, 2-1
editing
keys
arrow keys, 2-2
CLEAR key, 2-2
ENTER key, 2-2
listed, 2-2
program blocks, F9 (Misc)>More (F1), 5-8
programs, conversational, 5-1
to search, 5-6
editor, conversational, Misc (F9)>More (F1), pop-up menu, 5-8
edits
to exit, 5-5
to save, 5-5
unsaved, to cancel, 5-5
End Mill Cycle (F6), description, 4-51
end of main, block, 4-43
End of Prog (F7), from Misc (F9), 5-7
end of program, 3-3
ending
main programs, 4-44
subprograms, 4-44
EndMain (F4), description, 4-44
EndMain, last program block, 3-3
EndMill block, to program, 4-51
EndSub (F2), description, 4-44
engrave cycle
description, 4-47
graphic menu, illustration, 4-48
sample program, 4-49
to program, 4-47
ENTER key, editing keys, 2-2
entry fields
optional, graphic menu, 3-4
required, graphic menu, 3-4
EstAngle, SkewComp entry field, 4-87
EstDiam, 4-59, 4-64, 4-66, 4-69
Exit (F10), edit mode, 5-5
extension, M, 3-1

F

F1 (Mill Cycle), description, 4-51
F1 (More), pop-up menu
from Misc (F9), description, 5-8
from Misc (F9), illustration, 5-9
F1 (Sub), description, 4-44
F1 (ToolPro), access probe cycles, 4-56
F10 (Exit), edit mode, 5-5
F10 (Probe), access probe cycles, 4-56
F2 (Comment), from Misc (F9), 5-7
F2 (EndSub), description, 4-44
F2 (Program), manual screen, 5-1
F3 (Drill Cycles)
Basic, graphic menu, 4-3

DrillOff, description, 4-3
pop-up menu, illustration, 4-2, 5-3
F3 (Edit), manual screen, 5-1
F3 (Offset), description, 3-12
F3 (SpinPro), access probe cycles, 4-56
F4 (EndMain), description, 4-44
F4 (More), pop-up menu
arc, illustration, 3-24
line, illustration, 3-21
F4 (Page Up), from Misc (F9), 5-7
F4 (Pocket Cycles), pop-up menu, illustration, 4-16, 5-4
F5 (Loop), description, 4-45
F5 (Mill), access probe cycles, 4-56
F5 (Milling)
description, 3-9
soft keys, illustration, 5-4
F5 (Milling)>F7 (More), pop-up menu, illustration, 4-47
F5 (Page Down), from Misc (F9), 5-7
F6 (End Mill Cycle), description, 4-51
F6 (RMS), rotate, mirror, scale description, 4-46
F6 (Start of Prog), from Misc (F9), 5-7
F6 (Tool), soft keys, illustration, 5-4
F7 (End of Prog), from Misc (F9), 5-7
F7 (More), pop-up menu
from Milling (F5), description, 3-9
from Milling (F5), illustration, 3-10, 4-47
F8 (Edit), program editor, 5-1
F8 (MCode), M-Code block, to program, 3-29
F8 (Sub Progs)
M-Code block, to program, 3-29
soft keys, illustration, 4-44, 5-4
F9 (Misc), soft keys, illustration, 5-4
F9 (Misc)>F1 (More)
description, 5-8
pop-up menu, illustration, 5-9
F9 (Prev), from Misc (F9), 5-7
F9 (Teach), description, 3-12
face mill cycle
description, 4-17
graphic menu, illustration, 4-17
to program, 4-17
tool approach, illustration, 4-17
face, the surface of a part, 4-17
feed
block
description, 3-17
to program
hot keys, 3-17
soft keys, 3-17
hot key (4), 2-1
moves, programming, 3-19
Feed, 4-85
feedrate change, programming, 3-17
Find, SkewComp entry field, 4-87
FindActive, SkewComp entry field, 4-87
fixture offsets
define, 3-2
entry fields, 3-10
table
illustration, 3-12
to activate, 3-12
to adjust, 3-13
to calibrate, machine current location, 3-12
to change, 3-12
to cancel, 3-11
to program, 3-10
four-axis programming, description, 6-1
frame pocket cycle
compensation, 3-7
description, 4-29
graphic menu, illustration, 4-29
to program, 4-29

G

GaugeDiam, 4-74
graphic menu
basic drill cycle, illustration, 4-2
bolt hole drill, illustration, 4-11
boring-bidirectional, illustration, 4-5
chip breaking cycle, illustration, 4-6
circular pocket, illustration, 4-25
circular profile, illustration, 4-21
circular slot, illustration, 4-27
drill pattern cycle, illustration, 4-10

engrave cycle, illustration, 4-48
entry fields, types, listed, 3-4
face mill cycle, illustration, 4-17
frame pocket, illustration, 4-29
hole mill, illustration, 4-31
irregular pocket, illustration, 4-33
mill cycle, illustration, 4-51
peck drilling, illustration, 4-4
plane, description, 3-16
pocket with islands, illustration, 4-38
rectangular pocket, illustration, 4-23
rectangular profile, illustration, 4-19
slot, illustration, 4-35
thread mill, illustration, 4-13
using, 3-3

H

height, standard starting, 3-2
helical threads, 3-25
hole mill cycle
description, 4-31
graphic menu, illustration, 4-31
to program, 4-31
home, 3-9
homing method, 3-9
hot key
absolute mode (E), 2-1
absolute/incremental key (E), 2-1
arc key(3), 2-1
comment asterisk (0), 2-1
decimal point key, 2-2
dwell key (8), 2-1
feed key (4), 2-1
inch mode (7), 2-1
incremental mode (E), 2-1
line move (2), 2-1
M-CODE key (6), 2-1
millimeter mode (7), 2-1
plane key (9), 2-1
rapid move (1), 2-1
sign change key (+/-), 2-2
spindle speed, 2-2
tool key (5), 2-1
unit key (7), 2-1
X-axis, 2-1
Y-axis, 2-1
Z-axis, 2-1
hot keys
arc, to program
using center-endpoint, 3-25
using center-included angle, 3-28
using endpoint-radius, 3-23
feed block, to program, 3-17
line move, to program, 3-19
listed, 2-1
plane block, to program, 3-16
rapid move, to program, 3-18
RPM, to program, 3-17

I

inch mode, 3-5
inch mode, hot key (7), 2-1
incremental command, 3-5
incremental mode
change, 3-5
hot key (E), 2-1
InOutBoss, inside or outside hole or boss center find
defined, 4-72
description, 4-81
InOutWeb, inside or outside web or slot center find
defined, 4-73
description, 4-83
inside corner finding, CornerIn, 4-79
inside or outside hole or boss center find, InOutBoss, 4-72, 4-81
inside or outside web or slot center find, InOutWeb, 4-73, 4-83
inside part corner find, CornerIn, 4-72, 4-79
inside profile, 4-19
inside/outside hole/boss finding, InOutBoss, 4-81
inside/outside web finding, InOutWeb, 4-83
introduction, 1-1
irregular pocket
cycle
compensation, 3-7
description, 4-33
to program, 4-33
graphic menu, illustration, 4-33
island, in rectangles, 4-29
islands
description, 4-38
example, 4-40
graphic menu, illustration, 4-38
using subroutines, with, 4-40

K

keys, editing, listed, 2-2

L

last, program block, EndMain, 3-3
LenDiamMea, length and diameter, 4-54, 4-59
Length, 4-81, 4-83
length and diameter, LenDiamMea, 4-54, 4-59
length offset, 3-2
length special, LenSpecMea, 4-54, 4-64
LenSpecMea, length special, 4-54, 4-64
line
- (feed) moves, 3-19
- block, move, programming, 3-21
- More (F4) pop-up menu, illustration, 3-21
- move
  - compensation, 3-7
  - hot key (2), 2-1
  - programming, 3-19
  - to program
    - hot keys, 3-19
    - soft keys, 3-19

Loop (F5), description, 4-45
loop, subprogram, description, 4-45
looping, subprograms, 4-42, 4-45

M

M, extension, 3-1
M0, program stop mode, 3-29
M00. *See* M0
M01. *See* M1
M03. *See* M3
M04. *See* M4
M05. *See* M5
M08. *See* M8
M09. *See* M9
M1, optional, program stop mode, 3-29
M105, dry run, all axes, 3-30
M106, dry run, No Z-axis, 3-30
M107, dry run Off - cancels M105 and M106, 3-30
M19, spindle orientation, 3-29
M3, spindle forward, 3-29
M4, spindle reverse, 3-29
M5, spindle Off, 3-29
M8, coolant On, 3-29
M9, coolant Off, 3-29
machine home
- graphic menu, to activate, 3-9
- return to, 3-9

machine zero, 3-9
manual tool diameter
- measure for special tools, DiaSpecMea, 4-66
- preset, DiaSpecMea, 4-55, 4-66

manual tool length
- measure for special tools, LenSpecMea, 4-64
- offset preset, LenSpecMea, 4-54, 4-64

MaxDiaAdj, 4-69
MaxLenAdj, 4-69
M-Code
- block, to program, 3-29
- dry run mode, listed, 3-30
- functions, listed, 3-29
- hot key (6), 2-1

MCode (F8), M-Code block, to program, 3-29
MeasType, 4-59
menus
- graphic
  - basic drill cycle, illustration, 4-2
  - bolt hole drill, illustration, 4-11
  - boring-bidirectiona, illustration, 4-5
  - chip breaking cycle, illustration, 4-6
  - circular pocket, illustration, 4-25
  - circular profile, illustration, 4-21
  - circular slot, illustration, 4-27
  - drill pattern cycle, illustration, 4-10
  - engrave cycle, illustration, 4-48
  - face mill cycle, illustration, 4-17
  - frame pocket, illustration, 4-29
  - hole mill, illustration, 4-31
  - irregular pocket, illustration, 4-33
  - mill cycle, illustration, 4-51
  - peck drilling, illustration, 4-4
  - pocket with islands, illustration, 4-38
  - rectangular pocket, illustration, 4-23
  - rectangular profile, illustration, 4-19
  - slot, illustration, 4-35
  - thread mill, illustration, 4-13
- pop-up
  - arc, More (F4), illustration, 3-24

Drill Cycles (F3), illustration, 4-2, 5-3
editing, program blocks, Misc (F9)>More (F4), 5-8
line, More (F4), illustration, 3-21
Milling (F5)>More (F7), illustration, 4-47
Misc (F9)>More (F1), illustration, 5-9
Pocket Cycles (F4), illustration, 4-16, 5-4
rapid, More (F4), illustration, 3-21
Mill (F5), access probe cycles, 4-56
mill cycle
description, 4-51
graphic menu, illustration, 4-51
sample program, 4-52
to program, 4-51
Mill Cycle (F1), description, 4-51
mill, 4-axis, programming example, 6-5, 6-6
millimeter mode, hot key (7), 2-1
Milling (F5)
description, 3-9
soft keys, illustration, 5-4
Milling (F5)>More (F7), pop-up menu, illustration, 4-47
minutes to decimal, conversion formula, 6-1
mirroring and scaling, subprograms, 4-46
mirroring, subprograms, 4-42
Misc (F9), soft keys, illustration, 5-4
Misc (F9)>More (F1)
description, 5-8
pop-up menu, illustration, 5-9
mm mode, 3-5
modal
move
block, 3-19
compensation, 3-7
programming, 3-19
setting, description, 3-1
More (F1), pop-up menu
from Misc (F9), description, 5-8
from Misc (F9), illustration, 5-9
More (F4), pop-up menu
arc, illustration, 3-24
line, illustration, 3-21
rapid, illustration, 3-21
More (F7), pop-up menu
from Milling (F5), description, 3-9
from Milling (F5), illustration, 3-10, 4-47
move compensation, requirements, listed, 3-7
moves with unknown endpoints, programming, 3-20

N

negative radius value, 3-23
nesting subprograms, 4-42
no move blocks, description, 3-4
Nominal probe stylus diameter, 4-58

O

off-line
comment block, to cancel, 5-8
program block, comment out, 5-7
Offset, 4-76, 4-78, 4-80, 4-82, 4-84
Offset (F3), description, 3-12
organizing programs, containing subprograms, 4-43
outside corner finding, CornerOut, 4-77
outside part corner find, CornerOut, 4-72, 4-77
outside profile, 4-21
outside profile ramp moves, illustration, 4-19
OvrFstFeed, 4-60
OvrMedFeed, 4-60, 4-64, 4-66, 4-70
OvrRPM, 4-60, 4-64, 4-67, 4-70
OvrSlwFeed, 4-60, 4-64, 4-66, 4-70

P

P/N 627785-21, *6000i CNC User's Manual*, referenced, 1-2, 5-4
P/N 627787-21, *6000i CNC Technical Manual*, referenced, 1-1, 4-53
Page Down (F5), from Misc (F9), 5-7
Page Up (F4), from Misc (F9), 5-7
part programs, to develop, 3-1
part zero location, 3-1
part, moves toward, 3-2
pattern drill. *See* drill pattern

pattern drill cycle
description, 4-10
graphic menu, illustration, 4-10
peck drilling cycle
description, 4-3
graphic menu, illustration, 4-4
to program, 4-3
plane block, to program
hot keys, 3-16
soft keys, 3-16
plane, graphic menu,
description, 3-16
plane, hot key (9), 2-1
pocket
cycle, tool compensation, 4-16
pop-up menu, illustration, 4-16
with islands
description, 4-38
example, 4-40
graphic menu, illustration, 4-38
to program, 4-38
using subroutines, for, 4-40
pocket cycles
circular profile cycle, description, 4-21
description, 4-16
face mill cycle, description, 4-17
hole mill cycle, description, 4-31
listed, 4-16
Pocket Cycles (F4), pop-up
menu, illustration, 4-16, 5-4
point of rotation, 4-46
pop-up menus
arc, More (F4), illustration, 3-24
Drill Cycles (F3), illustration, 4-2, 5-3
editing, program blocks, Misc (F9)>More
(F1), 5-8
line, More (F4), illustration, 3-21
Milling (F5)>More (F7), illustration, 4-47
Misc (F9)>More (F1), illustration, 5-9
Pocket Cycles (F4), illustration, 4-16, 5-4
rapid, More (F4), illustration, 3-21
positive radius value, 3-23
Prev (F9), from Misc (F9), 5-7
probe
calibration, CalibTIPrb, 4-54, 4-57
canned cycle, settings, 4-53
cycles, conversational programming,
description, 4-53
orientation, description, 4-58
spindle cycles
conversational programming,
description, 4-53
description, 4-71
listed, 4-72
tool cycles
description, 4-54, 4-56
designations, 4-54
listed, 4-56
tool-length offset, description, 4-53, 4-54
Probe (F10), access probe
cycles, 4-56
ProbeMove, protected
positioning move
defined, 4-73
description, 4-85
progamming
feed block
hot keys, 3-17
soft keys, 3-17
line move
hot keys, 3-19
soft keys, 3-19
rapid move
hot keys, 3-18
soft keys, 3-18
RPM block
hot keys, 3-17
soft keys, 3-17
Set Zero block, 3-16
program
arc
using center-endpoint
hot keys, 3-25
soft keys, 3-25
using center-included angle
hot keys, 3-28
soft keys, 3-28
using endpoint-radius
hot keys, 3-23
soft keys, 3-23
back up, 3-2
basics, description, 3-1
editing, conversational, 5-1
end of, 3-3
ending, main, 4-44
optional, stop mode (M1), 3-29
stop mode (M0), 3-29
Program (F2), manual screen,
5-1
program block
adding to program, 3-4

comment out, 5-7
comments, description, 5-7
editing, F9 (Misc)>More (F1), 5-8
EndMain, last, 3-3
saving, 3-3
scroll, 5-6
to delete, 5-5
to edit, 5-6
to insert, 5-5
to jump, 5-7
to page, 5-7
writing, 3-3
program editor, 3-3
conversational
from manual screen, activate, 5-1
from program directory, activate, 5-1
screen, illustration, 5-2
SHIFT screen, soft keys, illustration, 5-3
program type
conversational, description, 1-1
G-Code, description, 1-1
programming
absolute mode, change, 3-5
arcs, 3-22
basic drill cycles, 4-2
canned cycles, description, 4-1
circular pocket cycle, 4-25
circular profile cycle, 4-21
circular slot cycle, 4-27
conventions, rotary/U-axis, 6-2
curved moves, 3-22
drill patterns, 4-10
dwell, 3-8
using hot keys, 3-8
using soft keys, 3-8
examples
4-axis, description, 6-2
4-axis, drill, 6-3
4-axis, mill, 6-5, 6-6
face mill cycle, 4-17
feed moves, 3-19
feedrate change, 3-17
frame pocket cycles, 4-29
hole mill cycle, 4-31
hot keys, listed, 2-1
inch/MM mode changes, 3-5
incremental mode, change, 3-5
irregular pocket cycle, 4-33
line (feed) moves, 3-19
M-Code block, 3-29
modal moves, 3-19
move, using line block, 3-21
move, using rapid block, 3-21
moves with unknown endpoints, 3-20
part programs, 3-1
plane block
hot keys, 3-16
soft keys, 3-16
pocket with islands cycle, 4-38
rapid move, 3-18
rectangular pocket cycles, 4-23
rectangular profile cycle, 4-19, 4-20
slot cycle, 4-35
spindle RPM, 3-17
straight moves, 3-18
tool change, 3-5
protected positioning move, ProbeMove, 4-73, 4-85
protected probe positioning, ProbeMove, 4-85

R

ramp moves, 3-7
rapid block, move, programming, 3-21
rapid More (F4) pop-up menu, illustration, 3-21
rapid move
compensation, 3-7
hot key (1), 2-1
hot keys, 3-18
programming, 3-18
soft keys, 3-18
to program
hot keys, 3-18
soft keys, 3-18
rectangular pocket cycle
compensation, 3-7
description, 4-23
graphic menu, illustration, 4-23
to program, 4-23
rectangular profile cycle
compensation, 3-7
description, 4-19
graphic menu, illustration, 4-19
to program, 4-20
repeat cycle
description, 4-49
sample program, 4-49
to program, 4-49
RepeatMeas, 4-82

repetitive
drilling cycle, subprograms, 4-42
operations, 4-16
operations, subprograms, 4-42
return to machine home, 3-9
RMS
description, 4-46
subprogram, to call, 4-46
subprograms, 4-46
RMS (F6), rotate, mirror, scale description, 4-46
RMS (rotation, mirroring, and scaling), defined, 4-71
rotary axis
programming conventions, 6-2
programming, description, 6-1
programming, in absolute, 6-2
programming, in incremental, 6-2
rotate, mirror, and scale. *See* RMS
rotating, subprograms, 4-42
rotation, mirroring, and scaling (RMS), defined, 4-71
rough and finish cycles, subprograms, 4-42
RPM block, to program
hot keys, 3-17
soft keys, 3-17
RPM for calibration and tool measurement, 4-58, 4-60, 4-64, 4-67, 4-70

S

saving, program block, 3-3
scaling, subprograms, 4-42, 4-46
screens
conversational, illustration, 1-1
pocket
with islands, subroutines, example, illustration, 4-40
scroll, program, 5-6
search, blocks, 5-6
SearchDir, 4-76
SearchQuad, 4-77, 4-79
seconds to degrees, conversion formula, 6-1
Set Zero block, to program, 3-16
SetZero block
description, 3-14
to set, 3-14
SHIFT program editor screen, soft keys, illustration, 5-3
shortestDistance, parameter, 6-2
Side, 4-81, 4-83
sign change, hot key (+/-), 2-2
single surface measure/edge find, EdgeFind, 4-72, 4-76
skew error find, SkewComp, 4-86
skew error or angle find, SkewComp, 4-73, 4-86
SkewComp, skew error or angle find
defined, 4-73
description, 4-86
slot
description, 4-35
graphic menu, illustration, 4-35
to program, 4-35
soft keys
feed block, to program, 3-17
line move, to program, 3-19
Milling (F5), illustration, 5-4
Misc (F9), illustration, 5-4
Misc (F9)>More (F1), editing, program blocks, 5-8
plane block, to program, 3-16
program editor screen, 5-2
program, arc
using center-endpoint, 3-25
using center-included angle, 3-28
using endpoint-radius, 3-23
rapid move, to program, 3-18
RPM, to program, 3-17
SHIFT program editor screen, illustration, 5-3
Sub Progs (F8), illustration, 4-44, 5-4
Tool (F6), illustration, 5-4
spindle
forward (M3), 3-29
Off (M5), 3-29
orientation (M19), 3-29
probe calibration
CalibPtPrb, 4-74
cycle, CalibPtPrb, 4-72, 4-74
wired probe, description, 4-75
wireless probe, description, 4-75
probe cycles
conversational programming, description, 4-53
description, 4-71
listed, 4-72
reverse (M4), 3-29

RPM, programmable, 3-17
speed, hot key, 2-2
SpinPro (F3), access probe cycles, 4-56
Start of Prog (F6), from Misc (F9), 5-7
starting, subprograms, 4-44
stepover
face cycle, 4-17
value, 4-18
straight moves, programming, 3-18
Sub (F1), description, 4-44
sub block, to program, 4-44
Sub Progs (F8)
M-Code block, to program, 3-29
soft keys, illustration, 4-44, 5-4
subprogram
call, 4-43
call a loop, 4-45
calling, 4-44
description, 4-42
ending, 4-44
example, 4-43
looping, 4-45
program structure, 4-43
programs, containing, 4-43
repetitive drilling cycle, 4-42
RMS, to call, 4-46
rotate, mirror, and scale, 4-46
rough and finish cycles, 4-42
to program, 4-44
to start, 4-44
to write, 4-43
subroutines, using for, pockets with islands, 4-40

T

tapping cycle
description, 4-8
to program, 4-8
Teach (F9), description, 3-12
thread milling cycle
description, 4-13
graphic menu, illustration, 4-13
sample program, 4-15
to program, 4-13
TLO. See tool-length offset, *See* tool-length offset
TLO, axis and direction, 4-76
TLO, defined, 4-53, 4-54
tool
approach, face mill cycle, illustration, 4-17
change, progamming, 3-5
compensation, automatic, 3-7
diameter, 3-2
diameter compensation, activating, 3-7
hot key (5), 2-1
to activate, 3-6
to change, 3-5
Tool (F6), soft keys, illustration, 5-4
tool breakage, length and diameter wear detection, BrkWearDet, 4-55, 4-69
tool length and diameter offset preset, LenDiamMea, 4-54, 4-59
tool probe calibration cycle, CalibTIPrb, 4-54, 4-57
tool probe cycles
description, 4-54, 4-56
designations, listed, 4-54
listed, 4-56
tool# block, 3-6
ToolComp command, 3-7
tooling, organize, 3-2
tool-length offset
for probe, description, 4-53
probe, description, 4-54
ToolPro (F1), access probe cycles, 4-56
Top, 4-74, 4-77, 4-79, 4-81, 4-83, 4-87
troubleshoot, finished programs, 3-2

U

U-axis
programming, in absolute, 6-2
programming, in incremental, 6-2
unit block, to program, 3-5
unit, hot key (7), 2-1
Update, 4-70

V

vectored, feedrate display, 6-2

W

Width, 4-81, 4-83
wired probe, spindle, description, 4-75

wireless probe, spindle,
description, 4-75
word, search for, 5-6
writing, conversational programs,
3-1
writing, program blocks, 3-3

X

X-axis, hot key, 2-1
XMinus, axis and dirrection, 4-76
XMinusYMinus, 4-77, 4-79
XMinusYPlus, 4-77, 4-79
XPlus, axis and direction, 4-76
XPlusYMinus, 4-77, 4-79
XPlusYPlus, 4-77, 4-79

Y

Y-axis, hot key, 2-1
YMinus, axis and direction, 4-76
YPlus, axis and direction, 4-76

Z

Z final pick, SLOW feedrate, 4-
60, 4-64, 4-66, 4-70
Z first pick
FAST feedrate, 4-60
MEDIUM feedrate, 4-57, 4-60, 4-64, 4-66,
4-70
Z-axis, hot key, 2-1
ZMinus, axis and direction, 4-76
ZPlus, axis and direction, 4-76

HEIDENHAIN CORPORATION

333 East State Parkway
Schaumburg, IL 60173-5337 USA
☏ +1 (847) 490-1191
FAX +1 (847) 490-3931
E-Mail: info@heidenhain.com

www.heidenhain.com

634 755-22 · Ver00 · 1 · 11/2009 · Printed in USA