ANILAM

6000i CNC User's Manual

January 2008
Ve 02
627785-21 · 1/2008 · VPS · Printed in USA · Subject to change without notice

www.anilam.com

Warranty

ANILAM® warrants its products to be free from defects in material and workmanship for one (1) year from date of installation. At our option, we will repair or replace any defective product upon prepaid return to our factory.

This warranty applies to all products when used in a normal industrial environment. Any unauthorized tampering, misuse or neglect will make this warranty null and void.

Under no circumstances will ANILAM, any affiliate, or related company assume any liability for loss of use or for any direct or consequential damages.

The foregoing warranties are in lieu of all other warranties expressed or implied, including, but not limited to, the implied warranties of merchantability and fitness for a particular purpose.

The information in this manual has been thoroughly reviewed and is believed to be accurate. ACU-RITE® Companies Inc. reserves the right to make changes to improve reliability, function, or design without notice. ACU-RITE Companies Inc. assumes no liability arising out of the application or use of the product described herein. All rights reserved. Subject to change without notice.

ANILAM® and ACU-RITE® are registered trademarks of ACU-RITE Companies Inc.

© Copyright 2008 ACU-RITE Companies Inc.

Section 1 - Introduction

Section 2 - CNC Console and Software Basics

Section 3 - Manual Operation and Machine Setup

All rights reserved. Subject to change without notice.

Section 4 - Preparatory Functions: G-Codes

Section 5 - Canned Cycles and Subprograms

All rights reserved. Subject to change without notice.

Section 6 - Program Editor

All rights reserved. Subject to change without notice.

All rights reserved. Subject to change without notice.

Section 10 - Program Management

Section 11 - Running Programs

All rights reserved. Subject to change without notice.

All rights reserved. Subject to change without notice.

All rights reserved. Subject to change without notice.

Section 1 - Introduction

This manual describes the concepts, programming commands, and CNC programming formats used to program ANILAM CNC products. Use the Contents and Index to locate topics of interest. In general, topics are presented in order of complexity. For example, "Section 1" describes basic CNC topics while later sections describe Drawing Exchange Format (DXF) converter programming and special programming features that require a firm grasp of CNC programming.

The following topics are described in this section:

- **Effectivity Notation**
- Getting Started
- Programming Concepts
- Programs
- Axis Descriptions
- Defining Positions
- Angle Measurement
- Plane Selection
- Arc Direction

Effectivity Notation

Some sections of this manual apply only to specific ANILAM CNC product(s). In these sections, icons in the left margin identify the product(s) to which the information applies. **Table 1-1** lists the icons for each CNC product and the number of axes supported by each product.

Table 1-1, CNC Effectivity Icon Description

Icon	Product	Axes Supported
6000i-3X	6000i-3X Systems	3
6000i-4X	6000i-4X Systems	4

NOTE: All systems also support one spindle axis.

The main difference between the products is the number of axes supported. Generally, this manual describes the 6000i-3X systems. The 6000i-4X operates exactly as the 6000i-3X system except for features that include the additional axes.

All rights reserved. Subject to change without notice.

Getting Started

Before you start to write a program, determine the work holding device and the location of Part Zero (the point to which all movement is referenced). Since absolute positions are defined from Part Zero, try to select a location that directly corresponds to dimensions provided on the part print, such as the lower left corner of the work. Then, you can develop a program using a procedure similar to the one that follows:

1. To enter the Program Manager from the Manual screen, press **Program** (**F2**). Create a program name for the part.
2. Enter the Program Editor (**Edit F8**) to open the new program and start writing blocks.
3. The first block of any program is usually a safe start position and tool-change position (a position away from the work where the axes can return for safe tool changing). The first block is normally also used to specify the units of measurement (Inch/MM), mode of operation (Absolute), move type (Rapid), and to cancel all auxiliary functions (Tool Offsets, Spindle, and Coolant).

   Typical first blocks:

   ```
   G70  G90  G0  G28  Z0
   M5
   ```

4. Subsequent blocks in the program set Spindle information, call Tool number, turn on Coolant, and make the initial move toward the work.
5. The remaining blocks in the program describe the required moves, Canned Cycles, and Tool changes to complete the machining.
6. The next to the last block in the program returns the axes to the Tool change position, turning off any auxiliary functions (Tool Offsets, Spindle, and Coolant). The last block (M2) ends the program.

   Typical final blocks:

   ```
   M9
   M5
   G0  G28  Z0
   X0  Y0
   M2
   ```

7. After you write a program, verify it. Run it in Draw Graphics Mode to troubleshoot for errors. Verify that all programmed moves are safe and accurate to the part print dimensions.
8. Now, load the stock material into the selected work-holding device.
9. Set the Tool Offsets for each tool in the Tool Page.
10. Before running the part in the Auto Mode, run it in Single-Step Mode to verify that both the program and the setting of Tool Offsets have been correctly completed. Single-Step Mode allows you to execute the program block-by-block.
11. After you test the program, make any necessary corrections.
12. When the finished program is ready for production, back it up on a USB Memory Stick.

All rights reserved. Subject to change without notice.

Programming Concepts

This section contains programming concepts for the beginning programmer. You must master these concepts and be familiar with the terminology in order to write programs.

Programs

A program is the set of instructions that the CNC uses to direct the machine movements. Each line of instructions is called a block. Each block runs independently, thus allowing the program to be stepped along, one block at a time.

Axis Descriptions

The machine moves along its axes of motion. All movements along an axis are either in a positive or negative direction. Not all machines use the same system to identify axes. The descriptions used in this manual are commonly used to identify 3-axis mills.

NOTE:	To visualize machine movements correctly, imagine tool motion rather than table motion.

The following topics are described:

- X-Axis
- Y-Axis
- Z-Axis

All rights reserved. Subject to change without notice.

X-Axis

Table movement along the X-axis is to the left and right. Positive motion is table movement to the left; negative motion is table movement to the right. Refer to **Figure 1-1**.

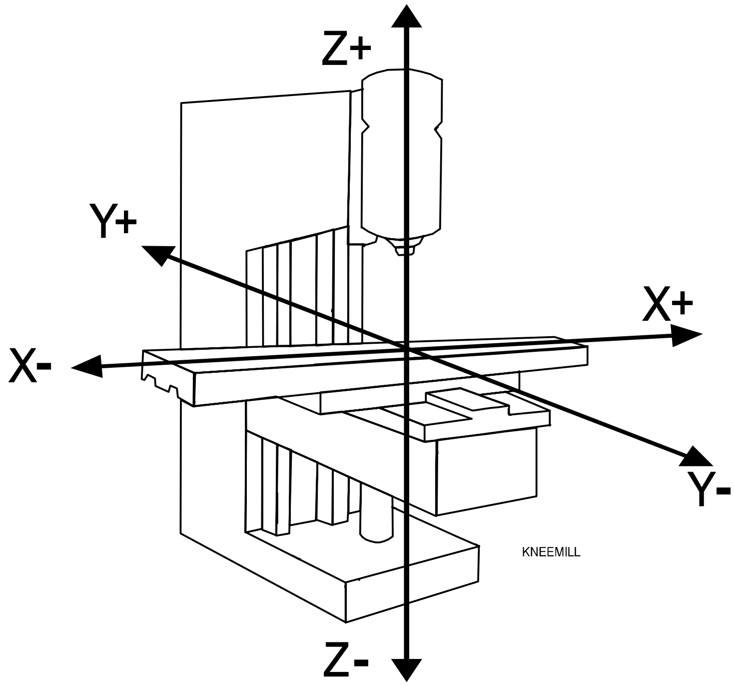


Figure 1-1, Mill Axes of Motion

Y-Axis

Table movement along the Y-axis is inward and outward. Positive motion is table movement outward; negative motion is table movement inward.

Z-Axis

Spindle movement along the Z-axis is upward and downward. Positive motion is tool movement upward (away from the workpiece); negative motion is tool movement downward (into the workpiece).

All rights reserved. Subject to change without notice.

Defining Positions

The intersection of the X-, Y-, and Z-axes is the reference point from which to define most positions. Refer to **Figure 1-2**. This point is the X0, Y0, Z0 position.

Most positions are identified by there X, Y, and Z coordinates. A position two inches left, three inches back, and four inches up has an X coordinate of X -2.0, a Y coordinate of Y3.0, and a Z coordinate of Z4.0.

The following topics are described:

- Polar Coordinates
- Absolute Positioning
- Incremental Positioning

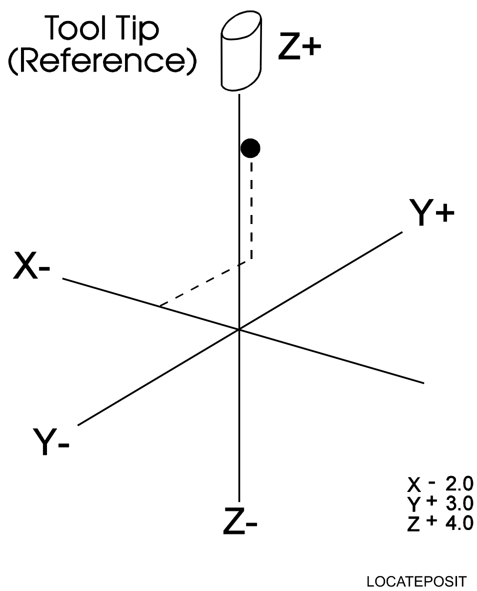


Figure 1-2, Locating Positions

All rights reserved. Subject to change without notice.

Polar Coordinates

Polar Coordinates define points that lie only on a single plane. Polar coordinates use the distance from the origin and an angle to locate points. Refer to **Figure 1-3**.

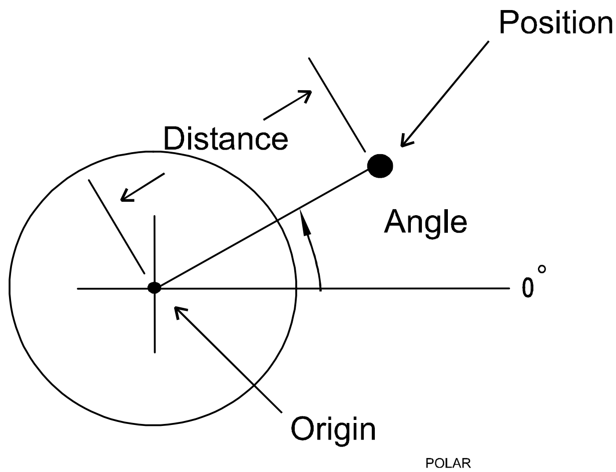


Figure 1-3, Polar Coordinate System

Absolute Positioning

In Absolute Mode, all positions are measured from Absolute Zero. Absolute Zero is not a fixed position on the machine. It is a selected point. Refer to **Figure 1-4**.

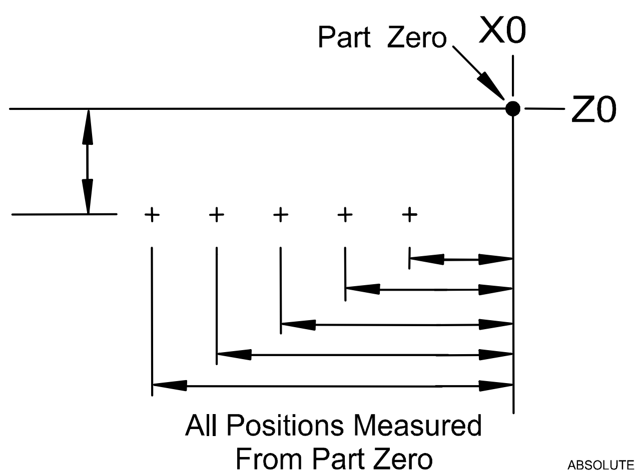


Figure 1-4, Absolute Positioning

You can set Absolute Zero (X0, Y0) anywhere. Usually, it is set at a position that enables you to use the dimensions specified on the blueprint. This is also called setting the Part Zero.

The Absolute Zero (Part Zero) can be moved as often as necessary, either manually or in a program.

All rights reserved. Subject to change without notice.
25-January-08

Incremental Positioning

Incremental positions are measured from one point to another, or from the machines present position. This is convenient for performing an operation at regular intervals. Incremental positions are measured from the tool's present position. Refer to **Figure 1-5**.

NOTE:	An incremental 0 inch (0 mm) move will not make a position change because you are located at the 0 reference point (current position).

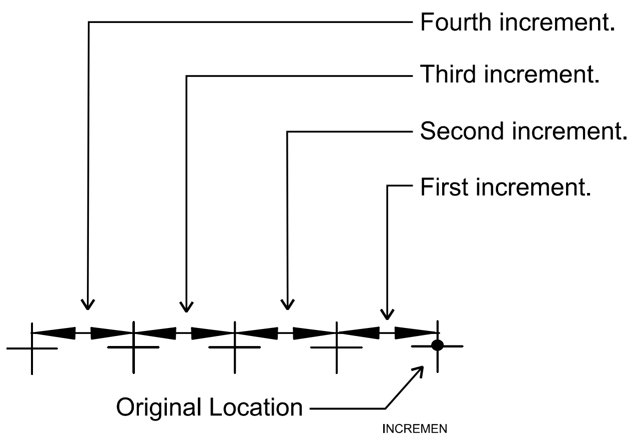


Figure 1-5, Incremental Positioning

Angle Measurement

Angles are measured with the 3 o'clock position as the Zero Degree Reference. Positive angles rotate counter-clockwise; negative angles rotate clockwise. Refer to **Figure 1-6**.

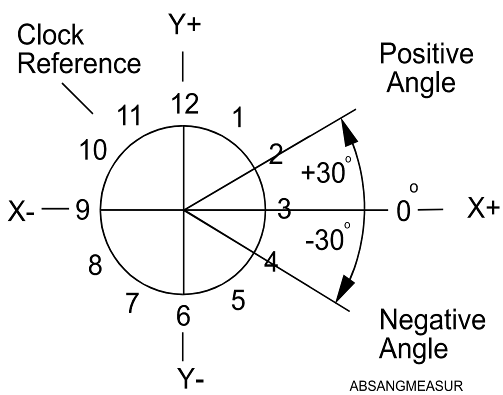


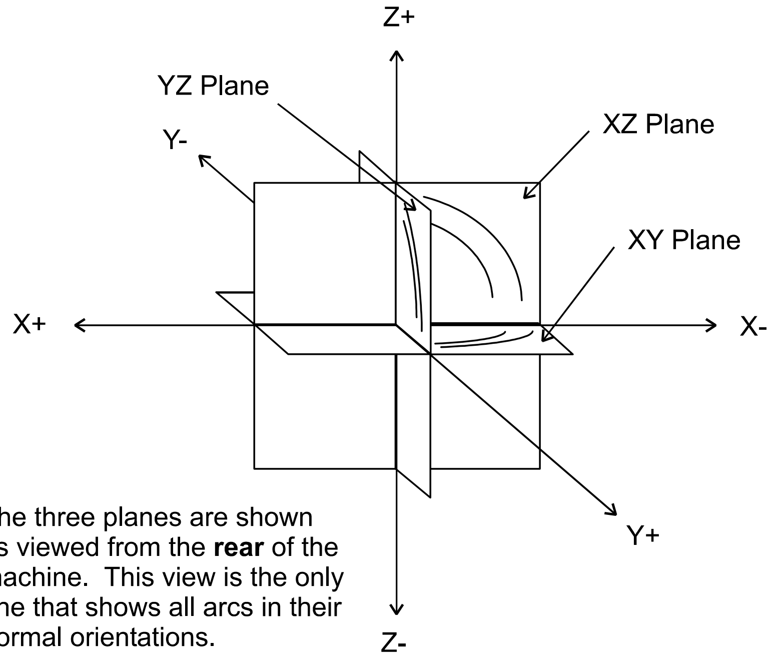
Figure 1-6, Absolute Angle Measurement

All rights reserved. Subject to change without notice.

Plane Selection

Circular moves and tool diameter compensation are confined to the plane you select. Three planes are available: the XY plane (G17), the XZ plane (G18), and the YZ plane (G19). It is important to view a plane correctly when you plan a circular move. If a plane is viewed from the wrong side, arc directions, angle references, and axis signs to appear reversed.

The standard rule is to view a plane looking in the negative direction along the unused axis. Refer to **Figure 1-7**.



The three planes are shown as viewed from the **rear** of the machine. This view is the only one that shows all arcs in their normal orientations.

Figure 1-7, Plane Identification

All rights reserved. Subject to change without notice.

Arc Direction

The standard rule is to view arc direction for a plane from the positive towards the negative direction along the unused axis. From this viewpoint clockwise (Cw) and counterclockwise (Ccw) arc directions can be determined. For example, in the XY plane, you view along the Z-axis, from Z+ toward Z-, to determine Cw/Ccw directions. The Cw/Ccw arc directions for each plane are shown in **Figure 1-8**.

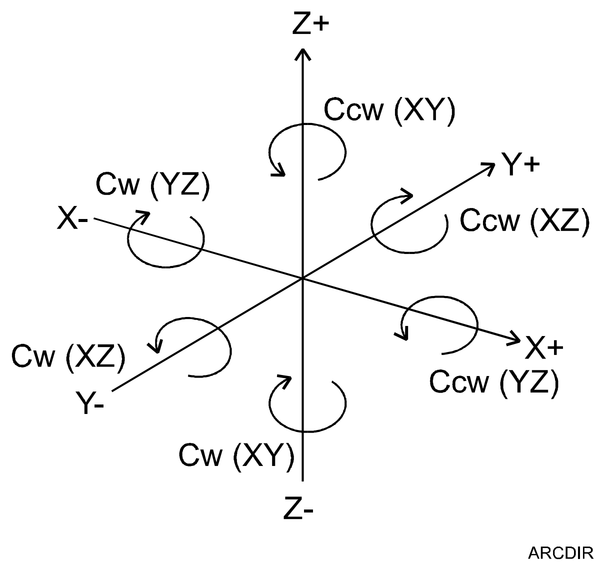


Figure 1-8, Clockwise and Counterclockwise Arc Directions

All rights reserved. Subject to change without notice.

Section 2 - CNC Console and Software Basics

The following topics are described in this section:

The Console

The CNC console consists of a 12.1-inch color, flat-panel liquid crystal display (LCD), keypad, soft keys, and manual panel (MP 6000M or MP 6001M Manual Panel). Refer to **Figure 2-1**.

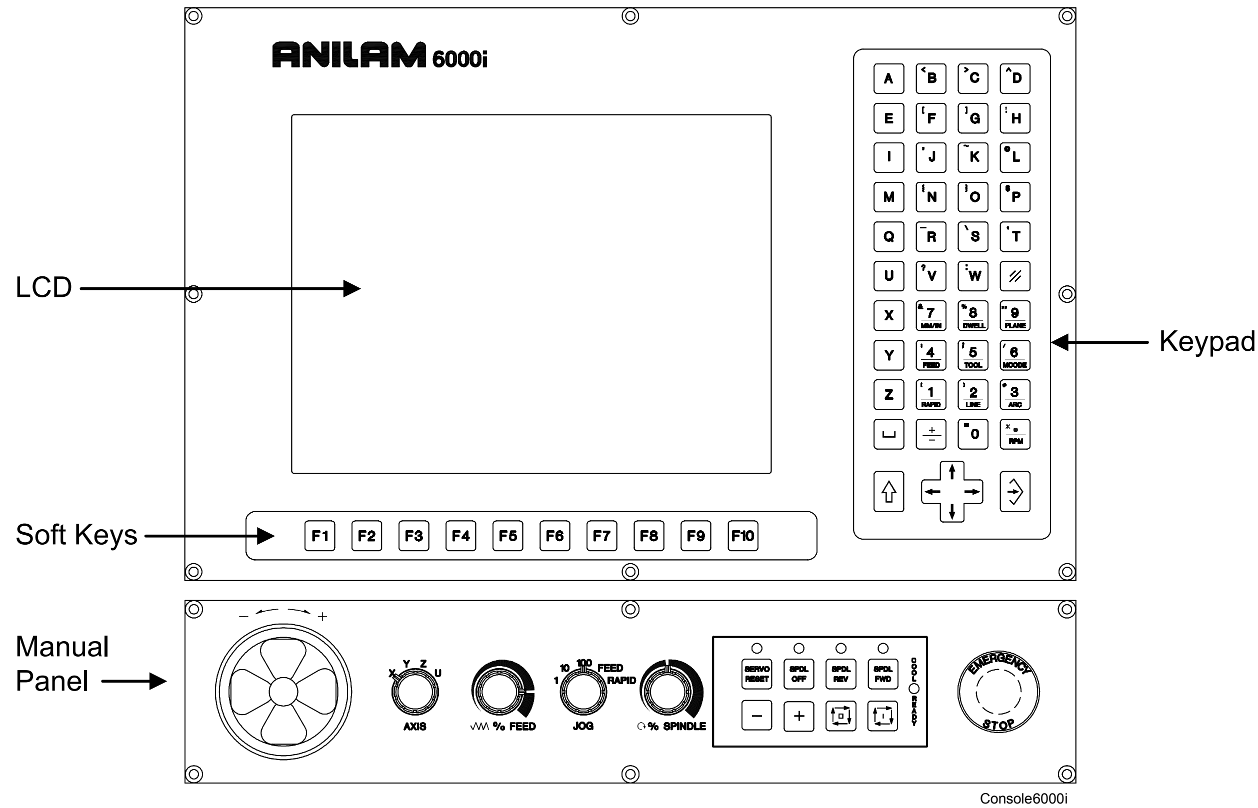


Figure 2-1, CNC Console

All rights reserved. Subject to change without notice.

Keypad

The following topics are described:

- Alphanumeric Keys
- Editing Keys

Refer to **Figure 2-2**. The keypad to the right of the LCD has the following areas:

Alphanumeric Keys: This area consists of the letters of the alphabet listed sequentially from **A** to **W**, and also includes the **CLEAR** key (lower right), the numerical keypad (**0** through **9**) and the **SPACE** key (lower-left).

Edit Keys: This area contains the **SHIFT** (left), **ENTER** (right) and the cursor control keys (**ARROWS**).

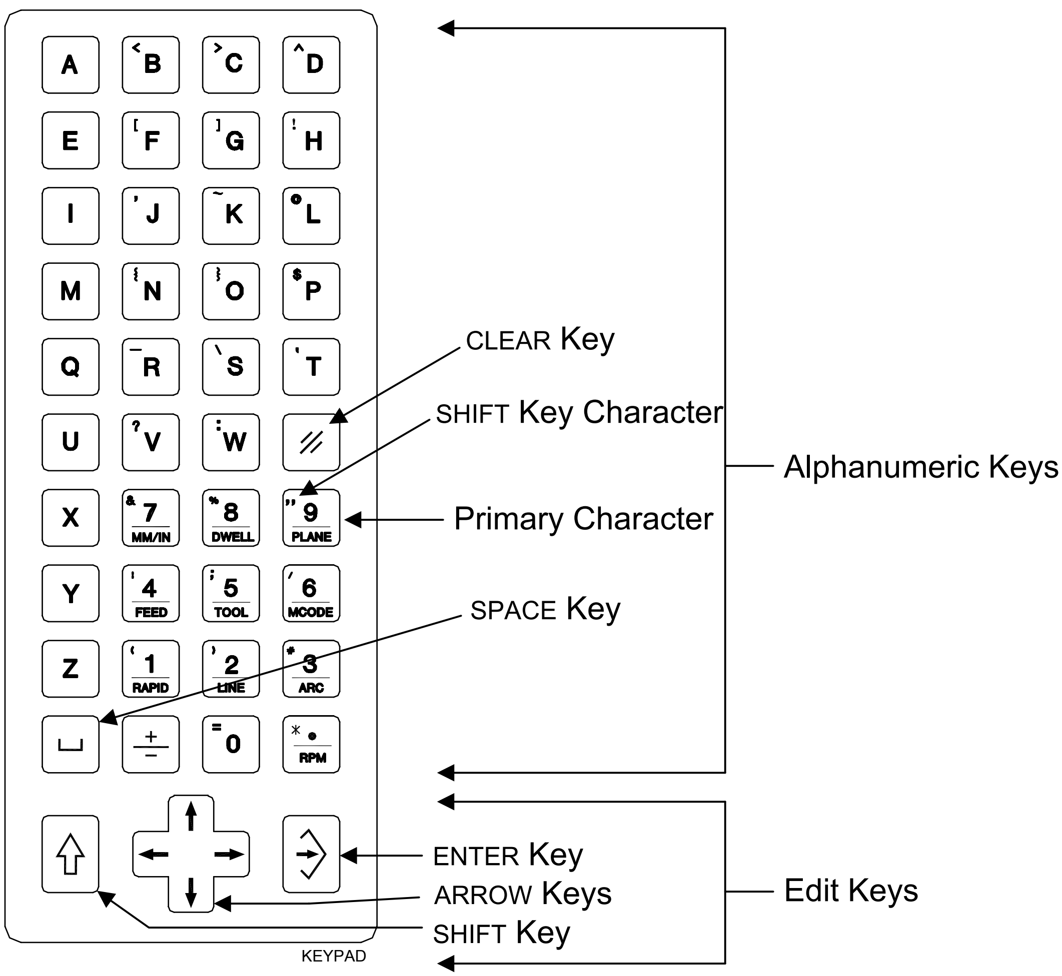


Figure 2-2, Keypad

All rights reserved. Subject to change without notice.

Alphanumeric Keys

Alphanumeric keys allow you to enter position coordinates (XYZ moves) and program G, M, S, and T codes. Some keyfaces have two characters, a large one in the middle of the key and a smaller one in the upper-left corner. The large characters are Primary characters. The smaller characters are **SHIFT** key characters.

To type a primary character, press the key that contains that character. To type a **SHIFT** key character:

1. Press **SHIFT** and release. You do not need to hold down the key, the **SHIFT** condition remains On until you press the next key.
2. Press the key that displays the required character in the upper-left corner. Refer to **Table 2-1**.

Table 2-1, Alphanumeric Keys

Key Face	Primary Function	SHIFT Function
A	Letter A	None
< B	Letter B	Less Than Symbol
> C	Letter C	Greater Than Symbol
^ D	Letter D	Caret
E	Letter E	None
[F	Letter F/Feedrate	Left Bracket
] G	Letter G/G Codes	Right Bracket
! H	Letter H	Exclamation Point
I	Letter I	None
' J	Letter J	Apostrophe
~ K	Letter K	Tilde Symbol
@ L	Letter L	"At" Symbol
M	Letter M Miscellaneous Functions	None

(Continued…)

All rights reserved. Subject to change without notice.

Table 2-1, Alphanumeric Keys (Continued)

Key Face	Primary Function	SHIFT Function
{ N	Letter N	Left Curly Bracket
} O	Letter O Program Number Designator	Right Curly Bracket
$ P	Letter P	Dollar Sign
Q	Letter Q	None
_ R	Letter R	Underscore
\ S	Letter S/Spindle Speed Designator	Backslash
` T	Letter T/Tool words	Single Quote
U	Letter U	None
? V	Letter V	Question Mark
: W	Letter W	Colon
X	Letter X X Axis Coordinate	None
Y	Letter Y Y Axis Coordinate	None
Z	Letter Z Z Axis Coordinate	None
(1 RAPID	Number One RAPID	Left Parenthesis
) 2 LINE	Number Two LINE	Right Parenthesis
# 3 ARC	Number Three ARC	Pound or Number Sign
| 4 FEED	Number Four FEED	Vertical Bar: used to separate parts of a blueprint-programming block for angles/chamfers/radii.

(Continued…)

All rights reserved. Subject to change without notice.

Table 2-1, Alphanumeric Keys *(Continued)*

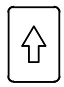
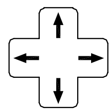
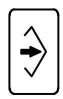
Key Face	Primary Function	SHIFT Function
; 5 TOOL	Number Five TOOL (shortcut key not enabled)	Semi-Colon
/ 6 MCODE	Number Six MCODE (shortcut key not enabled)	Slash (Right)
& 7 MM/IN	Number Seven MM/IN (shortcut key not enabled)	Ampersand
% 8 DWELL	Number Eight DWELL (shortcut key not enabled)	Percent Symbol
" 9 PLANE	Number Nine PLANE (shortcut key not enabled)	Inch Symbol
= 0	Number 0	Equal Sign
+ −	Minus Sign/Dash	Plus Sign
* • RPM	Period/Decimal Sign RPM (shortcut key not enabled)	Asterisk: used to "comment out" all or part of a block (characters to the right of the asterisk are ignored). The CNC ignores these blocks.
␣	Space Key	Blank Space

All rights reserved. Subject to change without notice.

Editing Keys

Use the Editing Keys to edit programs and move around the screen. Refer to **Table 2-2**.

Table 2-2, Editing Keys

Label or Name	Key Face	Purpose
SHIFT		Displays additional options on the soft key menu. Allows access to additional soft keys.
CLEAR		Clears selected messages, values, commands, and program blocks.
ARROW		Allows you to move highlight bars and cursor around the screen.
ENTER		Activates menu selections, activates alphanumeric entry, or creates new line.

Use Editing Keys to control machine movements manually. Refer to "Section 3 - Manual Operation and Machine Setup" for a detailed description of the Manual Panel.

CNC Keyboard (Option)

The CNC supports most standard USB PC keyboards. Refer to "Section 13 - Machine Software and Peripherals Installation." All keypad inputs except **E-STOP** and **SERVO RESET** have assigned keyboard equivalents.

Soft Keys (F1) to (F10)

Labeled soft keys **F1** to **F10**, also called function keys, are located just below the monitor. Soft key functions are not hardwired; their functions change with changes in mode. Labels indicate the function of each soft key. Unlabeled soft keys are inactive.

Manual Panel

Refer to "Section 3 - Manual Operation and Machine Setup" for information on the manual panel and the optional handwheel.

All rights reserved. Subject to change without notice.

Software Basics

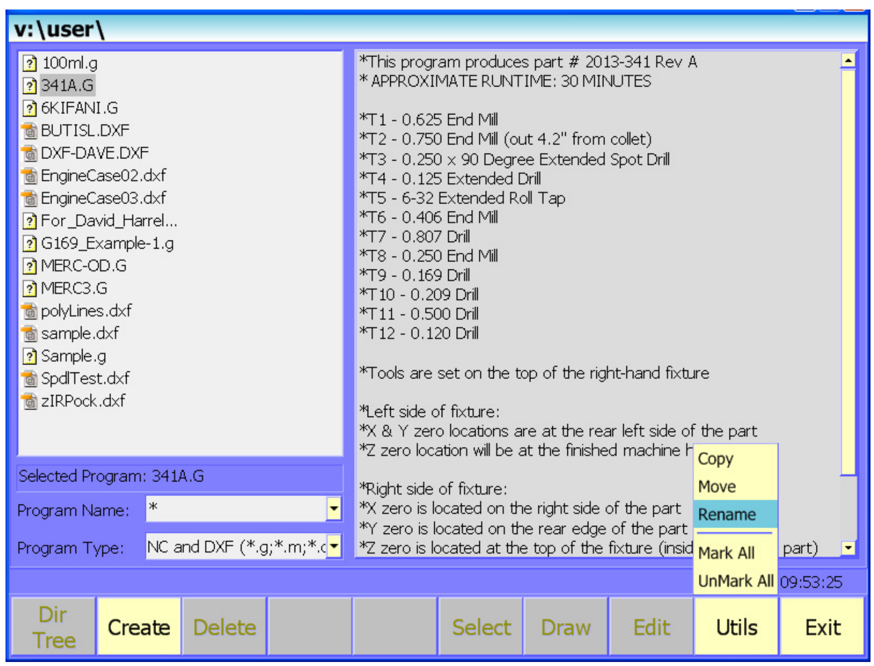
The CNC's screens change as different modes are activated. Basic procedures and features of the software remain the same, regardless of the CNC's mode.

The following topics are described:

- **Pop-Up Menus**
- Clearing Entries
- Operator Prompts
- Cursor
- Overwrite and Inserting Text
- Deleting Text

Pop-Up Menus

Pop-up menus are temporary menus that allow you to make additional selections. Each pop-up menu contains a highlight bar. The **ARROWS** move the highlight bar up and down the menu. Press **ENTER** to activate a highlighted selection. Press the soft key that activated the pop-up menu again to deactivate the function. Refer to **Figure 2-3**.



UTILITIES POP-UP

Figure 2-3, Pop-Up Menu

All rights reserved. Subject to change without notice.

Clearing Entries

Press **CLEAR** to clear an entry in an entry field or a character from a program.

Operator Prompts

The CNC sometimes prompts for required information. Enter numbers from the keypad.

Cursor

The CNC uses either a cursor or highlight to mark an item for selection or editing. The highlight displays in the Edit Mode, Program Manager, and Manual Mode. Use the **ARROWS** to move the highlight. The software highlights a selected item in a menu or window. Selected items can be activated or changed.

For instance, highlight a program block in Edit Mode to edit it. Highlight an entry field label in a graphic menu to enter a value or toggle between the available selections.

The cursor displays when the Tool Page activates. The cursor is a white underline that indicates where letters and numbers will be inserted.

Overwrite and Inserting Text

The Editor has two text-entry modes, **Insert Off** (overwrite) and **Insert On** [Default: **Insert On**]. In the Insert Off (overwrite) mode, new characters replace characters marked by the cursor.

In the Insert On mode, new characters appear at the cursor and existing characters move to the right. When the Insert On mode is active, **Insert On** (**F8**) highlights. To put the CNC in the Insert On mode:

1. When the CNC prompts for a name, toggle to **Insert On** (**F8**). The CNC highlights **Insert On** (**F8**).

Deleting Text

To delete text:

1. Move the cursor to underline the text to be deleted.
2. Press **Delete** (**F7**) to delete the selected text.

All rights reserved. Subject to change without notice.
25-January-08

Section 3 - Manual Operation and Machine Setup

The following topics are described in this section:

Powering On the CNC

NOTE:	When you power-on the CNC, ensure that the **E-STOP** switch is in the in position.

1. Turn on the CNC according to the builder's instructions. When the power switch is turned on, the CNC completely resets.
2. Turn the power switch ON. The startup screen activates (see **Figure 3-1**).

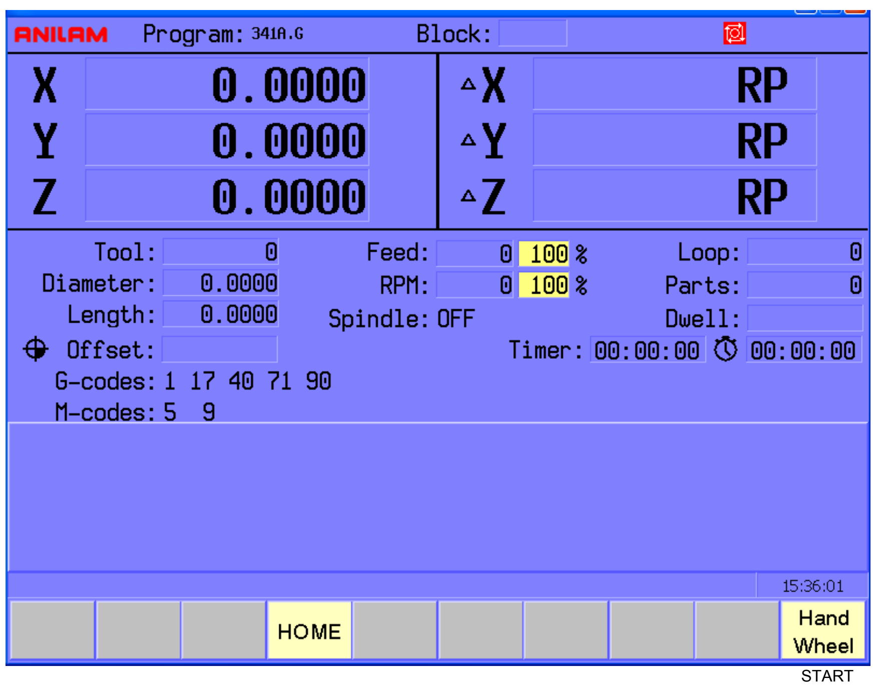


Figure 3-1, Start Screen

All rights reserved. Subject to change without notice.

3. Reset the servo drive by pressing the **SERVO RESET** button with the **EMERGENCY STOP** button Out.
4. Press **Home** (**F4**) and then press (**START**) to start. The CNC displays the Manual screen (see **Figure 3-2**).

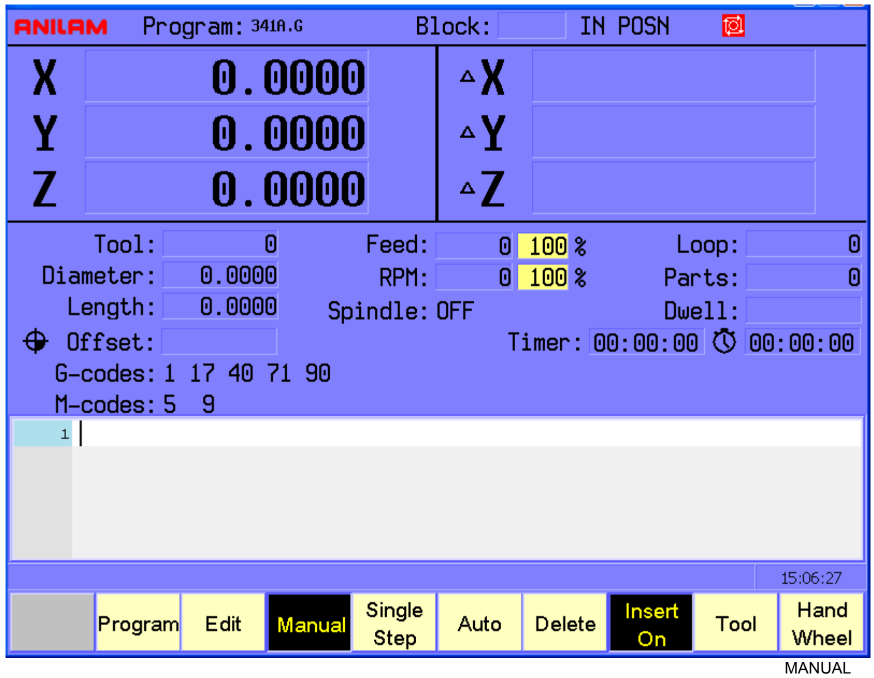
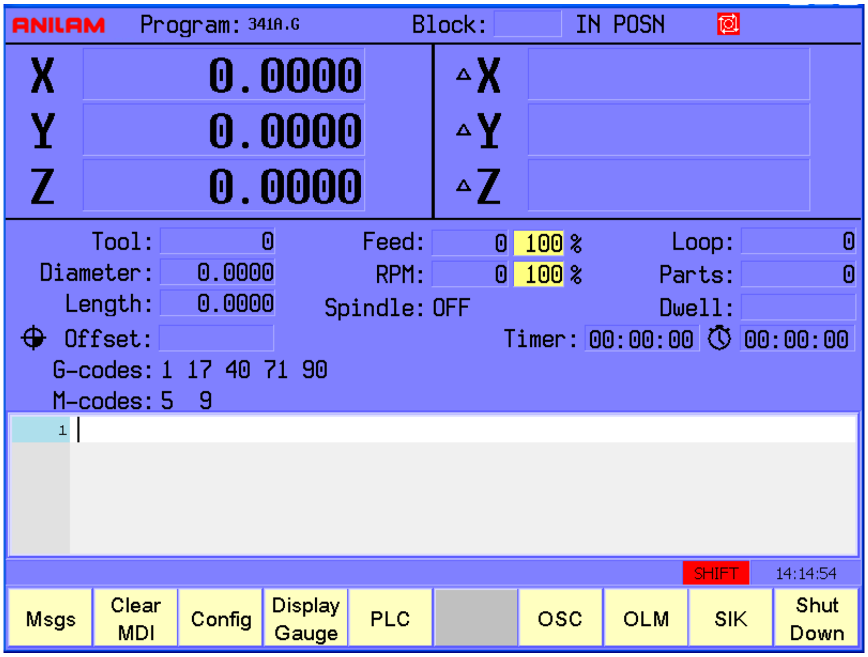


Figure 3-2, Manual Screen

5. Select **Manual** (**F4**) to display the Manual screen (refer to Figure 3-7, Manual Screen for illustration with callouts). Refer to Table 3-3, Manual Screen Soft Keys.
6. Press the **SHIFT** key on the keyboard to display the Manual Shift screen (refer to **Figure 3-3, Shift Screen from Manual Screen**). Refer to Table 3-4, Manual Screen Secondary Soft Keys.

All rights reserved. Subject to change without notice.
25-January-08

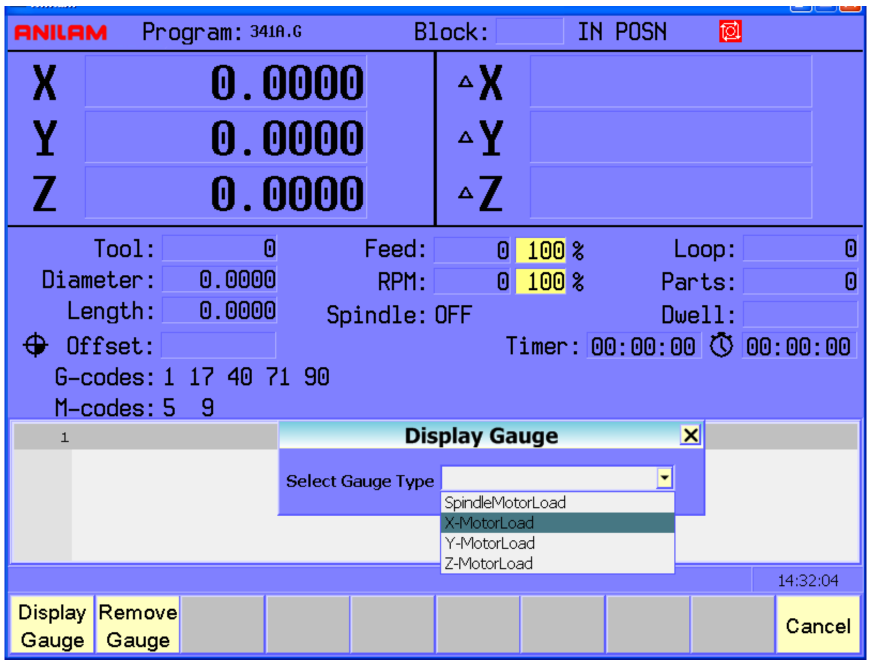


SHIFT MANUAL

Figure 3-3, Shift Screen from Manual Screen

All rights reserved. Subject to change without notice.

7. Press the **Display Gauge** (**F4**) soft key to display the Display Gauge screen (refer to **Figure 3-4**). Select the Gauge information that you want to display on the Manual screen:
   - SpindleMotorLoad — Spindle Motor Load
   - X-MotorLoad — X Axis Motor Load
   - Y-MotorLoad — Y Axis Motor Load
   - Z-MotorLoad — Z Axis Motor Load



DISPLAY GAUGE

Figure 3-4, Display Gauge from SHIFT Manual Screen

8. Press the **Display Gauge** (**F1**) soft key to display the Gauge information on the Manual screen (refer to Figure 3-5, Manual Screen with Gauge Display). Refer to **Table 3-1**.

   Table 3-1 describes the Display Gauge soft keys.

Table 3-1, Display Gauge Screen Soft Keys

Label	Soft Key	Function
Display Gauge	**F1**	Displays the gauge information selected on the Manual screen.
Remove Gauge	**F2**	Removes the gauge information field and label from the Manual screen.
Cancel	**F10**	Exits the Display Gauges screen and does not save changes.

All rights reserved. Subject to change without notice.

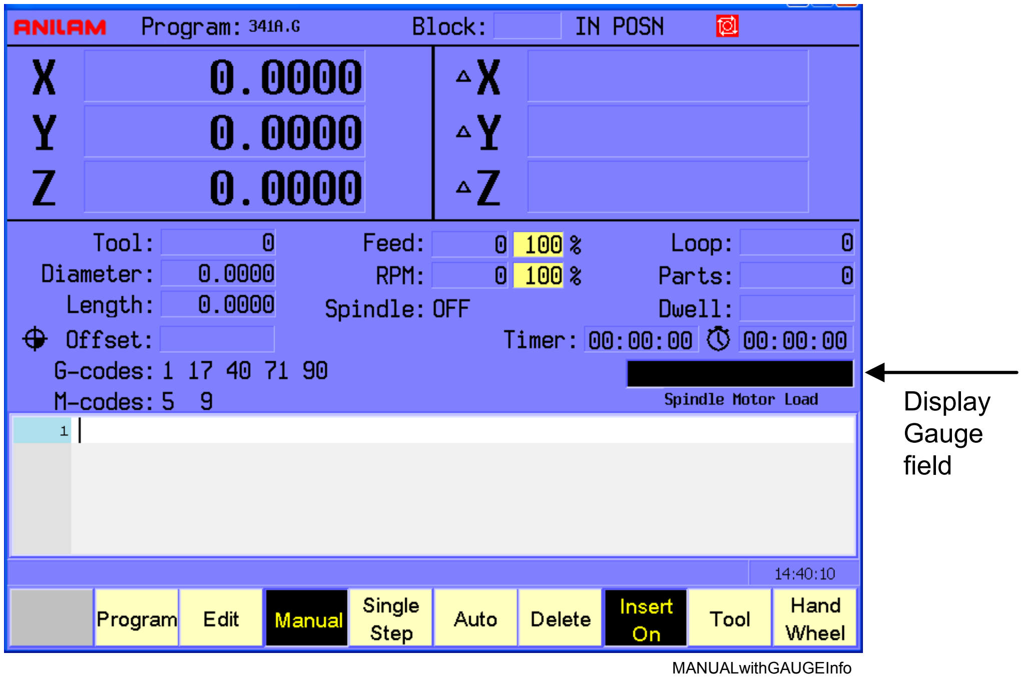


Figure 3-5, Manual Screen with Gauge Display

The Display Gauge field is dynamic. When the spindle motor load increases, the field in the Display Gauge increases.

All rights reserved. Subject to change without notice.

Shutting Down the CNC

1. Press **E-STOP** to disengage the servos and then revert to Manual Mode.
2. Press **Shut Down** (**SHIFT**+**F10**) to display the Shut Down soft keys. Press **Shut Down** (**F1**) to display the power down the CNC. Press **Cancel** (**F2**) to cancel the shut down.
3. Follow the builder's instructions for turning off the CNC.

Emergency Stop (E-STOP)

Press **E-STOP** to take all axes and spindle servos offline. This ends all machine movement.

To reset **E-STOP**, pull out and turn the rotary switch clockwise in the direction of the arrows. The switch makes a clicking sound when it resets.

Resetting **E-STOP** does not automatically reactivate the servos. The servos must be reset to move the machine. Press **SERVO RESET** to reset the servos.

Activating/Resetting the Servos

For safety reasons, the CNC powers up with the servomotors disengaged. While the servos are off, the CNC cannot move the machine. The servos are also disengaged when you press **E-STOP**.

Reset the servos as follows:

1. If a limit switch disengaged the servos, manually reposition the machine inside its normal range of travel.
2. Press **E-STOP** to display the message **External emergency stop**.
3. Rotate the **E-STOP** switch in the direction of the arrows to reset it. The **E-STOP** switch makes a clicking sound when it resets.

All rights reserved. Subject to change without notice.

Manual Panel

Use the keys on the manual panel to move the machine manually. Refer to **Figure 3-6**.

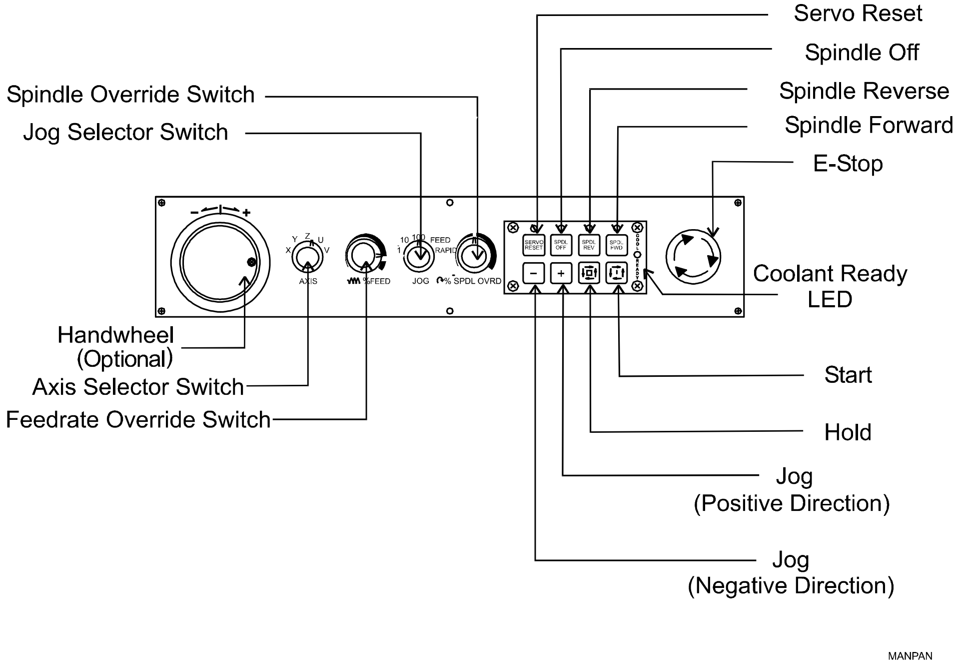


Figure 3-6, Manual Panel

The following topics are described:

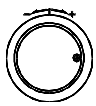
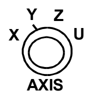
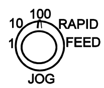
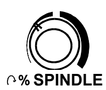
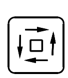
- Manual Panel Keys
- Manual Panel LEDs

All rights reserved. Subject to change without notice.

Manual Panel Keys

Manual panel keys allow you to control machine movements manually. These keys are located on the Manual Panel. Refer to **Table 3-2**.

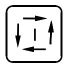
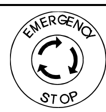
Table 3-2, Manual Operation Keys

Label/Name	Key Face	Purpose
Handwheel		Moves the selected controlled axis while in the Manual Mode. Jog must be set to 1, 10, or 100. Optional.
Axis Select		In Manual Mode, selects the axis to be jogged.
JOG		Cycles the CNC through manual movement modes (**FEED, RAPID**, **100**, **10**, **1**). The machine builder sets Default rapid and feed rates at setup. **NOTE:** The machine builder determines the actual speed of the machine during a move.
SPINDLE OVERRIDE		Overrides the programmed spindle RPM rate. It is a 13-position rotary switch that ranges from 40 to 160 percent. (Each increment adjusts the spindle override by 10%.) This feature can be used only on machines with programmable spindles.
FEEDRATE OVERRIDE		Overrides the feed and/or rapid rate of the axes in Manual, Auto, and Single Step modes. It is a 13-position rotary switch, which ranges from 0 to 120 percent. (Each increment adjusts the feedback override by 10%.) **NOTE:** The override range for rapid rate is 100%. The CNC will not exceed the maximum rapid rate.
SERVO RESET		Activates the servomotors.
SPINDLE FORWARD		Starts the spindle in a forward direction. **NOTE:** On some machines, you must provide the gear range and RPM before you activate this key.
SPINDLE REVERSE		Starts the spindle in a reverse direction. **NOTE:** On some machines, you must provide gear range and RPM before you activate this key.
SPINDLE OFF		Stops the spindle.
START		Starts all machine moves except jog.

(Continued…)

All rights reserved. Subject to change without notice.

Table 3-2, Manual Operation Keys (Continued)

Label/Name	Key Face	Purpose
JOG –	–	Moves the selected axis in a negative direction. Available in all modes. The machine builder specifies Feedrate.
JOG +	+	Moves the selected axis in the positive direction. Available in all modes. The machine builder specifies Feedrate.
HOLD		Halts any running program or programmed move. Press **START** to continue.
E-STOP		Press **E-STOP** to halt all axes and machine-related functions. When you activate **E-STOP**, the servomotors and any programming operations shut down. The CNC defaults to Manual Mode. Use **E-STOP** for emergency shutdown or intentional servo shutdown.

Manual Panel LEDs

The following keys have LEDs located directly above them on the Manual Panel. When any of the keys is activated, the corresponding LED lights up. Refer to Figure 3-6, Manual Panel.

- Servo Reset
- Spindle Off
- Spindle Forward
- Spindle Reverse

The Coolant Ready LED is also located on the Manual Panel. Some CNCs have a coolant ready M-function. For these CNCs, the Coolant Ready LED lights when the coolant is ready. The coolant is programmed to come on when the machine receives a **SPINDLE ON** command.

All rights reserved. Subject to change without notice.

Manual Mode Screen

In Manual Mode, the CNC displays the Manual screen. The Manual screen is the basic operating screen and is displayed when the CNC is turned on. All other operating screens are similar in appearance and selected from the Manual screen soft keys. When the Manual Mode is active, the **Manual** (**F4**) soft key label highlights. Refer to **Figure 3-7**.

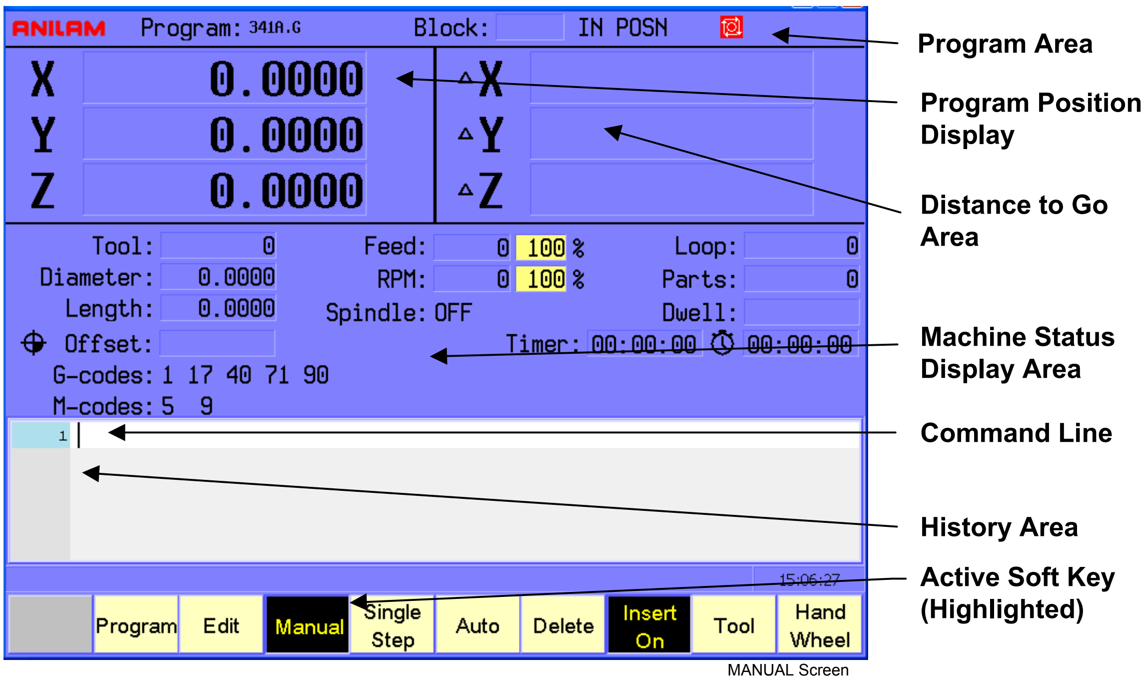


Figure 3-7, Manual Screen

The Manual screen is divided into the following areas.

Program Area Displays the working program name, running status, mode of operation, and in-position check.

Program Position Display

Displays programs X, Y, and Z position coordinates in reference to Part Home.

Distance to Go

Displays the Distance to Go in reference to:

- Part Zero
- Target

Machine Status Display Area

Displays operating information.

Command Line Allows you to enter commands manually.

All rights reserved. Subject to change without notice.

Active Soft Key	Identifies the function of the soft key. Soft key functions change from screen to screen. A highlighted label indicates an active mode.

The following topics are described:

- **Machine Status Display Area Labels**
- **Program Area Labels**

Machine Status Display Area Labels

Tool:	Active tool
Diameter:	Active tool diameter
Length:	Z-Axis Tool-Length Offset for active tool
Offset:	Active fixture offset
G-Codes:	Active G-Codes
M-Codes:	Active M-Codes
Feed:	Current feed rate (in inch/mm per minute)
% Feed:	Feedrate override setting (0% to 120% for Feed moves and 0% to 100% for Rapid moves)
RPM:	Current spindle speed in revolutions per minute
% RPM:	Spindle override setting (40% to 160%)
Spindle:	Current spindle status: OFF, FORWARD, REVERSE, or LOCKED
Loop:	Loop counter. Counts subprogram repetitions.
Parts:	Number of parts. Resets to zero when you enter Auto or Single Step mode.
Timer	Indicates the amount of time per part and accumulated amount of time (in parentheses) for all parts. Resets to zero when you enter Auto or Single Step mode.
Dwell:	Seconds remaining in a dwell

Program Area Labels

Program:	Name of loaded program
Block:	Block number (displays in S.Step or Auto Mode only)

IN POSN	All axis at target position
Green Icon:	Program is running.
Red icon:	Machine is in a programmed hold, has completed its program, external hold has been activated by an event, or **HOLD** was pressed.

All rights reserved. Subject to change without notice.

Manual Mode Settings

Features (or settings) that remain active for more than one operation are said to be modal. Modal features remain active until you change or cancel them. Most CNC functions are modal.

For example, if the CNC is in Rapid Mode, it executes all moves at the rapid rate until you initiate Feed Mode. The CNC can be in several modes, as long as the modes do not conflict.

Before making a manual move, make any necessary mode settings. Modes set from the Manual screen remain active if the CNC is put in a program mode (Auto, S.Step) until the program or operator changes the mode.

Set the following modes from the Manual screen:

Position Mode	Absolute or Incremental Mode
Move Mode	Rapid or Feed Mode
Active tool	Active tool, tool-length offsets, and tool radius compensation
Measurement Mode	Inch or MM Mode

The Manual screen determines the following:

- The location of Part Zero

Manual Mode provides the following types of moves:

- Jog (Conventional)
- Jog (Continuous)
- Manual Data Input (MDI)
- Handwheel (optional)

The following topics are described:

All rights reserved. Subject to change without notice.

Table 3-3 describes the active soft keys in Manual Mode.

Table 3-3, Manual Screen Soft Keys

Label	Soft Key	Function
Program	**F2**	Lists the user programs.
Edit	**F3**	Activates the Edit Mode. A program must first be selected.
Manual	**F4**	Activates Manual Mode from Auto and S.Step.
Single Step	**F5**	Changes to Single Step Mode.
Auto	**F6**	Changes to Auto Mode. Use to run part programs for production.
Delete	**F7**	Deletes a character from the command
Insert On/ Insert Off	**F8**	Toggle between Insert On and Insert Off (overwrite). For Insert On, the typed text replaces the existing text. For Insert Off (overwrite), the typed text overwrites the existing text.
Tool	**F9**	Displays the Tool Page. The Tool Page stores tool diameter, length offsets, and wear factors.
Hand Wheel	**F10**	Activates or deactivates Handwheel Mode. Use to jog any controlled axis in Manual Mode.

Press **SHIFT** while in the Manual screen to activate the secondary soft key functions (refer to Figure 3-3, Shift Screen from Manual Screen). Refer to **Table 3-4**.

Table 3-4, Manual Screen Secondary Soft Keys

Label	Soft Key	Function
Msgs	(**SHIFT + F1**)	Displays the last 10 messages, both old (already read) and new (not yet read)
Clear MDI	(**SHIFT + F2**)	Clear Manual Data Input (MDI)
Config	(**SHIFT + F3**)	Displays the Configuration screen
Display Gauge	(**SHIFT + F4**)	Display the gauge information on the Manual screen. See Figure 3-4, Display Gauge for Shift Manual Screen.
PLC	(**SHIFT + F5**)	Programmable Logic Controller (PLC)
OLM	(**SHIFT + F7**)	Online Monitor (OLM)
OSC	(**SHIFT + F8**)	Oscilloscope (OSC)
SIK	(**SHIFT + F9**)	Software Identification Key (SIK)
Shut Down	(**SHIFT +F10**)	Shut down the CNC

All rights reserved. Subject to change without notice.

PLC, OLM, OSC, and SIK Descriptions

For more detailed information on **PLC**, **OLM**, **OSC**, and **SIK**, refer to *6000i CNC Technical Manual*, P/N 627787-21.

PLC (**SHIFT** + **F5**)	Refer to "Section 7, Selecting the PLC Mode" in P/N 627787-21.
OLM (**SHIFT** + **F7**)	Refer to "Section 6, Diagnosis with the Online Monitor (OLM)" in P/N 627787-21.
OSC (**SHIFT** + **F8**)	Refer to "Section 5, Integrated Oscilloscope" in P/N 627787-21.
SIK (**SHIFT** + **F9**)	Refer to "Section 1, 6000i Overview" in P/N 627787-21.

Shut Down (SHIFT + F10)

On the Manual screen (refer to Figure 3-2, Manual Screen), press the **SHIFT** key on the keyboard to display the Manual Shift screen (refer to Figure 3-3, Shift Screen from Manual Screen).

Table 3-5 describes the Shutdown screen active soft keys.

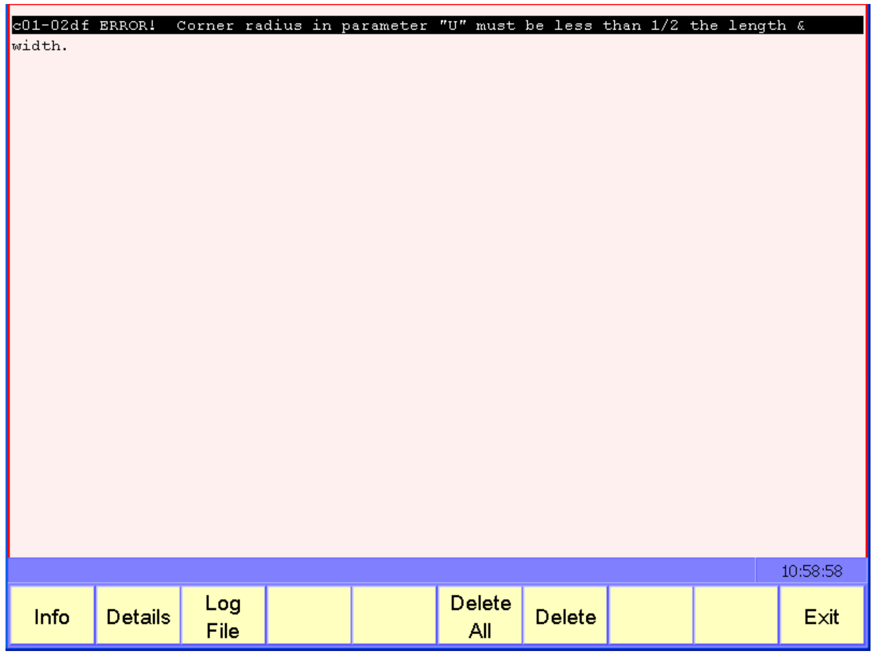
Table 3-5, Shut Down (SHIFT + F10) Screen Soft Keys

Label	Soft Key	Function
Shut Down	**F1**	Shutdown the CNC
Cancel	**F2**	Cancel the shutdown and return to Manual screen

All rights reserved. Subject to change without notice.

Messages (Msgs) (SHIFT + F1)

On the Manual screen (refer to Figure 3-2, Manual Screen), press the **SHIFT** key on the keyboard to display the Manual Shift screen (refer to Figure 3-3, Shift Screen from Manual Screen). Refer to **Figure 3-8**.



WITH MESSAGE

Figure 3-8, Messages Screen

Table 3-6 describes the Messages screen active soft keys.

Table 3-6, Msgs (SHIFT + F1) Screen Soft Keys

Label	Soft Key	Function
Info	**F1**	The Messages information is displayed
Details	**F2**	The Messages internal information is displayed
Log Files	**F3**	The Messages Log File is displayed. For more detailed information on **Log Files**, see *6000i CNC Technical Manual*, P/N 627787-21, "Section 6, Error Messages and Log Files."
Delete All	**F6**	Delete all messages
Delete	**F7**	Delete the message highlighted
Exit	**F10**	Displays the Manual screen.

All rights reserved. Subject to change without notice.

Activating Manual Mode Rapid or Feed

Turn the **JOG** rotary switch to cycle through all available Jog Modes. Choose **Rapid** or **Feed** mode. The CNC displays the active Feed or Rapid Mode in the Machine Status Display Area.

Adjusting Rapid Move Speed

The **FEEDRATE OVERRIDE** rotary switch also adjusts the speed of Rapid moves. The switch provides a range of 0% to 100%. Set the switch to 100 to set the rapid rate. The maximum override rate for rapid speeds is 100%.

NOTE: The machine builder determines the default rapid rate at setup.

Adjusting Feedrate

You can run the CNC at a percentage of the programmed feedrate by adjusting the **FEEDRATE OVERRIDE** switch. Each click of the **FEEDRATE OVERRIDE** switch adjusts the feedrate by an increment of ten percent; the range is 0 to 120%. Set **FEEDRATE OVERRIDE** to 100 to set the feedrate to 100% of the programmed feedrate.

CAUTION: If the CNC is shut down, the setup file will reload a default feedrate at the next power-on.

All rights reserved. Subject to change without notice.

Absolute Mode

In Absolute Mode, all positions are measured from Absolute Zero. Absolute Zero is X0, Y0, Z0 when the Absolute Mode is active. You can move Absolute Zero to any convenient location. All absolute XYZ positions are measured from this point. Refer to **G53** and **G92** in "Section 4 - Preparatory Functions: G-Codes" for more information on setting absolute zero. Setting Absolute Zero to a location on the part is referred to as setting Part Zero. Refer to **Figure 3-9**.

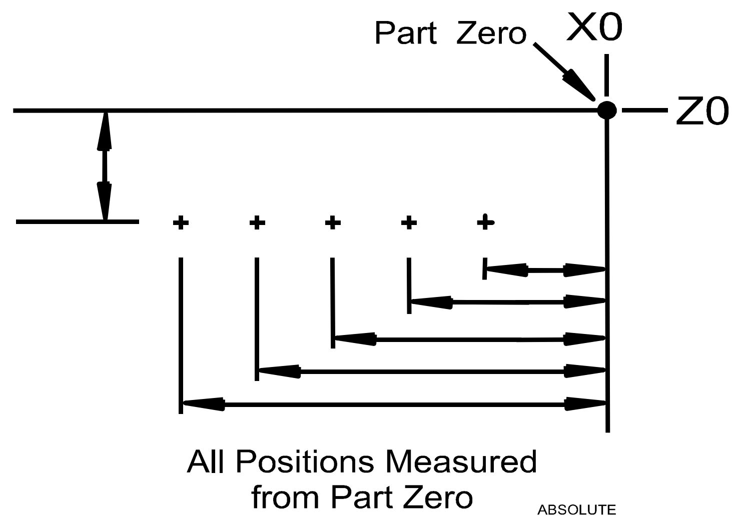


Figure 3-9, Absolute Positioning

NOTE:	To determine the Z-axis location of Part Zero, set tool length offsets for each tool.

NOTE:	The location of Absolute Zero can be restored after a shutdown if the machine has the Home function installed.

CAUTION:	**If Part Zero is not correctly located, the CNC will not position correctly in Absolute Mode.**

All rights reserved. Subject to change without notice.

Jog Moves

You can make or change jog moves when:

- The CNC is in Manual Mode, the Teach Mode, or the Tool Page; and
- The servos are on.

The actual rate for each mode is determined at machine setup. Use the **JOG** rotary switch to cycle the CNC through the Jog Mode choices. Refer to **Table 3-7** for the available Jog Modes.

Table 3-7, Jog Modes

Mode	Description
Rapid	Default rapid speed for continuous jogs. Actual speed determined at machine setup.
Feed	Continuous jog at feedrate determined at machine setup.
Jog: 100	Conventional Jog Mode, increment set to 100 times machine resolution.
Jog: 10	Conventional Jog Mode, increment set to 10 times machine resolution.
Jog: 1	Conventional Jog Mode, increment set to actual machine resolution.

You can change the Jog Mode any time the CNC is in Manual Mode.

The following topics are described:

- **Changing the Jog Mode**
- **Selecting an Axis**
- Jogging the Machine (Incremental Mode)
- Jogging the Machine (Continuous Mode)

Changing the Jog Mode

NOTE:	Jog move modes, with the exception of Jog Rapid Mode, are performed in Feed Mode.

To change the Jog Mode:

1. In Manual Mode, turn the **JOG** switch to select a jog feed rate.

Selecting an Axis

To select an axis in the Manual Mode:

1. Use the **AXIS SELECT** rotary switch to cycle through the available axes. Turn the switch until the indicator points to the required axis.

All rights reserved. Subject to change without notice.

Jogging the Machine (Incremental Moves)

In Manual Mode, position the machine with jog increments. To make a jog increment move:

1. Use **AXIS SELECT** to select an axis.
2. Use **JOG** to cycle through the move mode choices and choose a Jog Mode.
3. Press **JOG+** or **JOG-** to choose a direction. Do not hold down the key. Each time the key is pressed, the machine jogs along the selected axis by the selected increment.

Jogging the Machine (Continuous Moves)

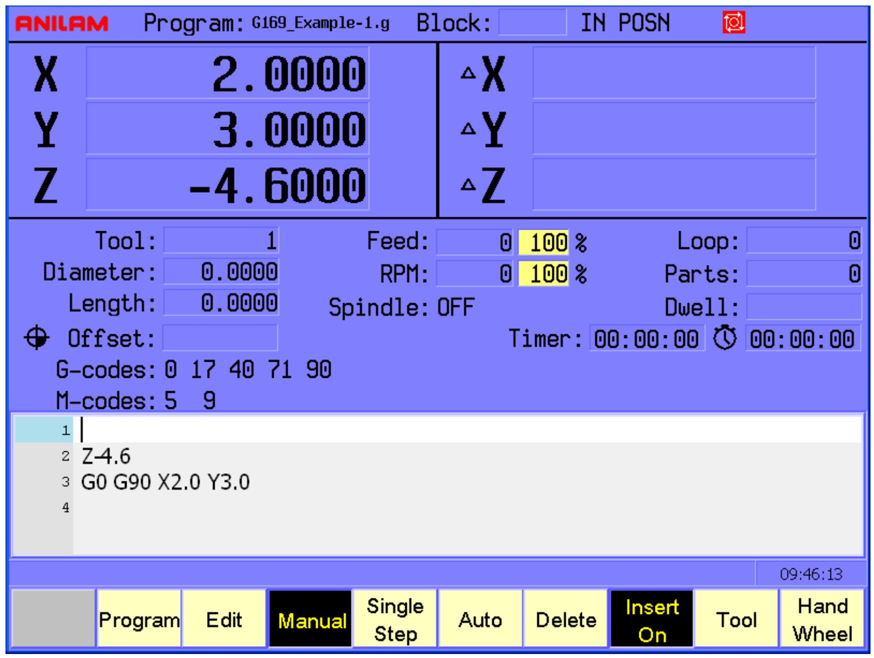
From the Manual screen, move the machine at feedrate or at the Jog Rapid Rate. The machine builder determines the effective jog and feed rates at setup.

1. In Manual Mode with the Manual screen active, use the **AXIS SELECT** to select an axis.
2. Use **JOG** to select a Continuous Jog Mode (Feed or Rapid).
3. Press and hold down **+** or **-** to jog the machine in the desired direction. The machine jogs along the selected axis. To stop the machine, release the key.

All rights reserved. Subject to change without notice.

Manual Data Input Mode

Manual Data Input (MDI) Mode allows you to command moves without creating a part program. MDI also is a quick way to program one move, or a series of moves that will be used only one time. Refer to **Figure 3-10**.



MDI

Figure 3-10, MDI Screen

To execute a command, type an instruction on the command line of the Program Area, and press **START**. (In Manual Mode, the cursor rests on the command line.)

More than one command can be programmed at a time. Use a semicolon (;) to separate the commands.

Press **HOLD** to pause one-shot moves.

Press **START** to continue. Press **Manual** (**F4**) to cancel. MDI moves are executed only once. To recall a previously commanded block, press **UP ARROW** or **DOWN ARROW** to go to the line that you want. You can execute from any line on the MDI history page. After you execute, the CNC takes you to the top line of the history page. You can store up to 150 lines on the MDI history page (two lines stores above). Use (**SHIFT** + **F2**) **Clear MDI** to clear the history page.

CAUTION:	**You must know the location of the Absolute Zero before making Absolute Mode moves.**

The following topic is described:

- Using Manual Data Input Mode

All rights reserved. Subject to change without notice.

Using Manual Data Input Mode

To use Manual Data Input Mode:

1. In Manual Mode, type the command block(s) at the COMMAND: line.
2. Press **START** to execute the typed commands.

Most functions that can be commanded in a part program can also be commanded in MDI Mode. These include:

- G00, G01, G02, G03 moves
- M-Codes, T-Codes (tool activation), S-Codes (spindle speed)
- Modal commands (G90, G91, G70, G71, etc.)
- G-Codes (G92, G28, G53, etc.)

The following example demonstrates how MDI Mode might be used to activate the spindle.

COMMAND: M43; S600; M3

M43	Activates Gear Range defined by M43 in setup
S600	Activates Specified Spindle Speed
M3	Activates Spindle Forward

All rights reserved. Subject to change without notice.

Operating the Handwheel (Optional)

NOTE:	The handwheel operation described here assumes that the handwheel has been properly installed and configured in the Setup Utility. The handwheel soft key will not display unless the Setup Utility has been configured for handwheel use.

The CNC supports an option that allows you to move a selected axis via a remote handwheel.

The resolution of the handwheel depends on the Jog Mode. Refer to **Figure 3-11**.

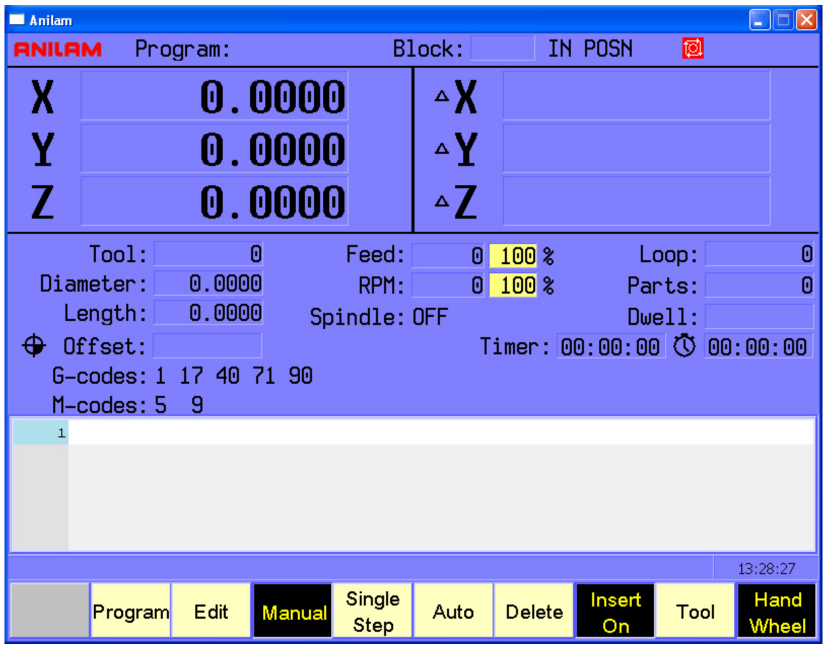


Figure 3-11, Handwheel Operation

To select a Jog Mode:

1. Turn the rotary switch to select an axis.
2. Select a conventional Jog Mode (100, 10, or 1).
3. Press **-** or **+** to move in a negative or positive direction, respectively.

To operate the handwheel:

1. From the Manual screen, press **HandWheel** (**F10**). The soft key highlights and the other soft keys are blank.
2. On the Manual Panel, select the axis that will be moved using the remote handwheel. Press **ENTER**. The selected axis can now be moved using the remote handwheel.
3. On the Manual Panel, select a Jog Mode (100, 10, 1) at a speed proportional to the 100, 10, and 1 setting.
4. Move the handwheel clockwise to move the selected axis in a positive direction or counterclockwise to move the axis in a negative direction.

All rights reserved. Subject to change without notice.

Section 4 - Preparatory Functions: G-Codes

G-Codes initiate motion commands, canned cycles and various machine and CNC functions. More than one G-Code may be specified per block. If a block contains conflicting G-Codes, an Error message is displayed.

Table 4-1 lists non-modal and modal G-Codes. Modal G-Codes remain in effect until canceled by the appropriate code. Non-modal G-Codes affect only the block in which they are programmed.

Edit Help provides graphic menus and labeled entry fields to aid those unfamiliar with G-Code programming. Refer to "Section 7 - Edit Help" for information.

Table 4-1, G-Codes

Modal		Non-Modal	
G-Code	**Function**	**G-Code**	**Function**
G0	Rapid Move – End-Point	**G4**	Dwell
G1	Feed Move – End-Point	**G9**	Exact Stop
G2	Arc CW Radius and End-Point	**G28**	Reference Point Return
G3	Arc CCW Radius and End-Point	**G29**	Return from Reference Point
G17	XY plane	**G30**	Move Reference from Machine Home
G18	XZ plane	**G31**	Probe Move
G19	YZ plane	**G65**	Macro Call, Single
G22	Stroke Limit	**G73**	Draft Angle Pocket Cycle
G40	Compensation OFF	**G75**	Frame Pocket Cycle
G41	Compensation LEFT	**G76**	Hole Mill Cycle
G42	Compensation RIGHT	**G77**	Circular Pocket Cycle
G53	Fixture Offset	**G78**	Rectangular Pocket Cycle
G59	Modal Radius/Chamfer	**G79**	Drill Bolt Hole Cycle
G60	Cancel Modal Radius or Chamfer	**G140**	Spindle Probe Calibration Cycle
G61	Exact Stop Mode	**G141**	Single Surface Measure/Edge Find
G64	Contouring Mode	**G142**	Outside Part Corner Find
G66	Macro Call, Modal	**G143**	Inside Part Corner Find
G67	Cancel Modal Macro	**G144**	Inside or Outside Hole or Boss Center Find
G68	Rotation (Axis)	**G145**	Inside or Outside Web or Slot Center Find
G70	Inch	**G146**	Protected Positioning Move
G71	MM	**G147**	Skew Error or Angle Find
G72	Scaling	**G150**	Tool Probe Calibration Cycle
G80	Drilling Off	**G151**	Tool Length and Diameter Offset Preset
G81	Basic Drill Cycle	**G152**	Manual Tool-Length Offset Preset

(Continued…)

All rights reserved. Subject to change without notice.

Table 4-1, G-Codes (Continued)

Modal		Non-Modal	
G-Code	**Function**	**G-Code**	**Function**
G82	CounterBore Drill Cycle	**G153**	Manual Tool Diameter Preset
G83	Peck Drill Cycle	**G154**	Tool Breakage, Length and Diameter Wear Protection
G84	Tapping Cycle	**G162**	Islands
G85	Boring Bidirectional Cycle	**G169**	Irregular Pocket Cycle
G86	Boring Unidirectional Cycle	**G170**	Face Mill Cycle
G87	Chip Break Cycle	**G171**	Circular Profile Cycle
G89	Flat Bottom Boring Cycle	**G172**	Rectangular Profile Cycle
G90	Absolute	**G175**	Mill Cycle
G91	Incremental	**G176**	EndMill Cycle
G92	Zero Set	**G177**	Plunge Circ Pocket Cycle
G100	Mirroring	**G178**	Plunge Rect Pocket Cycle
G120	BlockForm	**G179**	Drill Pattern Cycle
FEED	Feedrate	**G181**	Thread Mill Cycle
		G1000	Programmable Temporary Path Tolerance

For G-Codes not described in this Section, refer to "Section 5 – Canned Cycles and Subprograms."

All rights reserved. Subject to change without notice.

The following topics are described in this section:

All rights reserved. Subject to change without notice.

Rapid Move – End-Point (G0)

Format: G0

G0 initiates rapid traverse. The machine builder sets the actual rapid rate in the Setup Utility. Use rapid to position the tool prior to or after a cut. Do not use rapid to cut a part. Refer to **Figure 4-1**.

One to four axes can be included on a block with G0. X, Y, and Z will reach target simultaneously.

G0 is modal and remains in effect until canceled or changed.

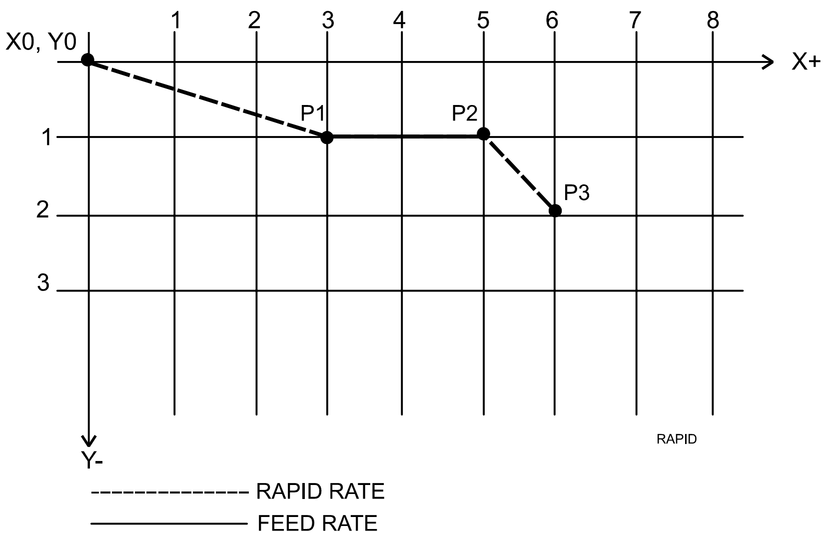


Figure 4-1, Rapid Traverse

Table 4-2 lists the program blocks required to complete the moves illustrated in **Figure 4-1**.

Table 4-2, Rapid Traverse

N1	G90 G0 X3 Y-1	Rapid move to X3, Y-1 (P1) in Absolute Mode.
N2	G1 X5.0	X-axis feeds to X5 (P2).
N3	G0 X6 Y-2	XY rapid to X6, Y-2 (P3).

NOTE: To override rapid, use the **FEEDRATE OVERRIDE**. For more information on using **FEEDRATE OVERRIDE**, refer to "Section 3 - Manual Operation and Machine Setup."

All rights reserved. Subject to change without notice.

Feed Move – End-Point (G1)

Format: G1

Feed move (**G1**) initiates straight-line feed motion and is used to cut a part. Straight-line motion occurs in one or more axes. The block may contain any combination of available axes. **G1** moves can be straight-line or angular moves.

G1 is modal and remains in effect until changed. Specify the feedrate on or prior to the **G1** block.

In **Figure 4-2** and **Table 4-3**, MM equivalents are in parentheses following the Inch measurements.

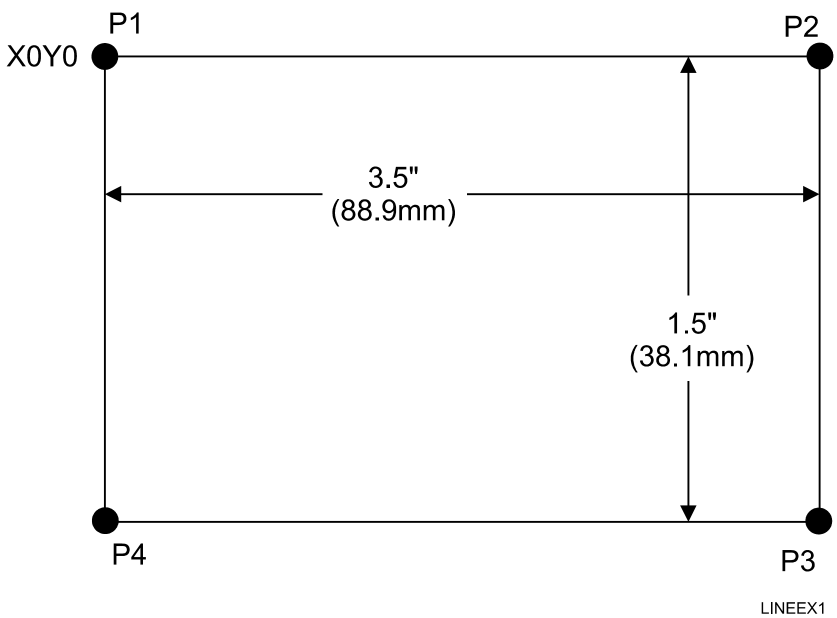


Figure 4-2, Feed Move

Table 4-3, Straight-Line Programming Example

N1	G90 G70 (G71) G1 X0 Y0 Z0	Feed to starting position.
N2	G1 F10 (254) X3.5 (88.9)	Feed to P2.
N3	Y-1.5 (-38.1)	Feed to P3.
N4	Z-1.5 (-38.1)	Move Z down.
N5	X0 (X0)	Feed to P4.
N6	Y0 (Y0)	Feed to P1.
N7	M2	End program, return to N1.

All rights reserved. Subject to change without notice.

Angular Motion Programming Example

Angular moves involve motion in two or more axes. In Absolute Mode, all dimensions are referenced to Part Zero (X0, Y0). In Incremental Mode, all dimensions are referenced to the current tool position. Refer to **Table 4-4**.

Table 4-4, Angular Programming Example, Absolute/Inch Mode

N1	G70 G90 G0 X0 Y0	Feed to starting position (X0, Y0).
N2	G1 F10 X3	Absolute, Inch Mode feed to P2.
N3	Y-2	Feed to P3.
N4	X0 Y-3	Feed to P4 (angular move).
N5	Y0	Feed to P1.
N6	M2	End program, return to N1.

In **Figure 4-3**, MM equivalents are in parentheses following the Inch measurements.

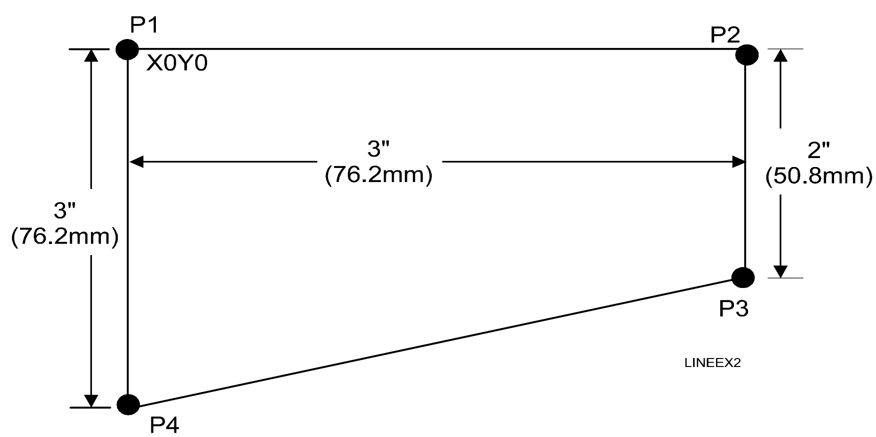


Figure 4-3, Angular Motion

All rights reserved. Subject to change without notice.

Circular Interpolation (G2 and G3)

Circular interpolation initiates circular moves, including arcs. **G2** commands a clockwise motion. **G3** commands a counterclockwise motion.

Arc input Format: G2 Xx Yy Zz Ii Jj Kk

Arc input Format: G3 Xx Yy Zz Ii Jj Kk

Radius Format: G02 Xx Yy Rr

Radius Format: G03 Xx Yy Rr

Refer to **Table 4-5** for parameter descriptions.

NOTE: For circular interpolation in another plane, make the plane changes prior to the **G2** or **G3** block. Refer to "Plane Selection (G17, G18, G19)" for information on planes. Arc examples use the most common plane, **G17** (**XY**).

NOTE: If the value of **X**, **Y**, **Z**, **I**, **J**, or **K** is zero, omit it.

Table 4-5, Parameters for Circular Interpolation

Parameter	Description
G2 **G3**	Arc CW Radius and End-Point (clockwise) motion. Arc CCW Radius and End-Point (counterclockwise) motion.
XYZ	Endpoint of arc motion in Absolute or Incremental Mode.
I (X) **J (Y)** **K (Z)**	Distance from the tool location to the arc center. **I** = X center, **J** = Y center, and **K** = Z center. **NOTE:** Arc centers are incremental by default. This is set up in the Setup Utility.
R	Arc Radius. **NOTE:** If Arc is greater than 180°, enter the **R**-value as a negative value (For example, **R-.5**).

The following topic is described:

- Examples of Circular Interpolation

All rights reserved. Subject to change without notice.

Examples of Circular Interpolation

Partial Arcs (XYIJ)

Figure 4-4 illustrates an arc move between P2 and P3.

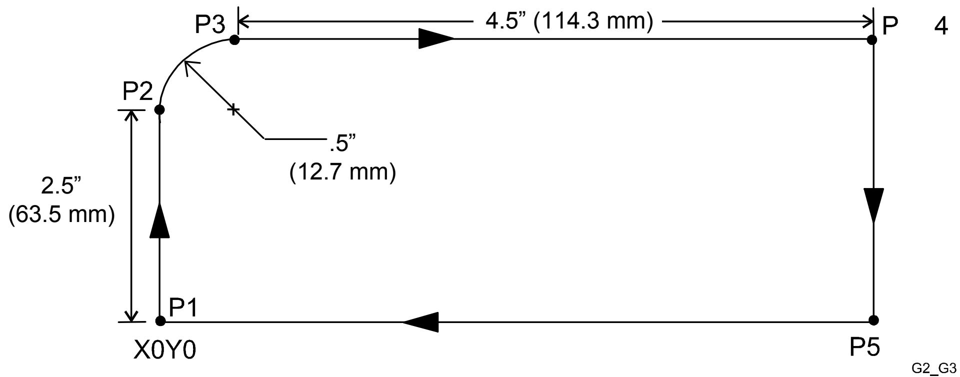


Figure 4-4, Circular Interpolation

Absolute Mode: Refer to **Table 4-6**.

Table 4-6, Circular Interpolation in Absolute Mode, Inches

Address Word	Format	Description
N1	G70 G90 G17 G1 Y2.5 F3	Activate Inch and Absolute Mode and set feedrate to IPR. Activate plane. Feed to P2.
N2	G2 X.5 Y3.0 I.5 J0	Arc move to P3.
N3	G1 X5	Feed to P4.
N4	Y0	Feed to P5.
N5	X0	Feed to P1.
N6	M2	End Program.

Incremental Mode: Refer to **Table 4-7**.

Table 4-7, Circular Interpolation in Incremental Mode, Inches

Address Word	Format	Description
N1	G70 G91 G17 G1 Y2.5 F3	Activate Inch and Absolute Mode and set feedrate to IPR. Activate plane. Feed to P2.
N2	G2 X.5 Y.5 I.5 J0	Arc move to P3.
N3	G1 X4.5	Feed to P4.
N4	Y-3	Feed to P5.
N5	X-5	Feed to P1.
N6	M2	End Program.

All rights reserved. Subject to change without notice.

Any arc of less than 360 degrees is a partial arc. Use Address Words **X**, **Y**, **I, J** together.

To program a move from P1 to P2, calculate arc centers (**I** and **J**) and endpoints (**X** and **Y**). Refer to **Figure 4-5**.

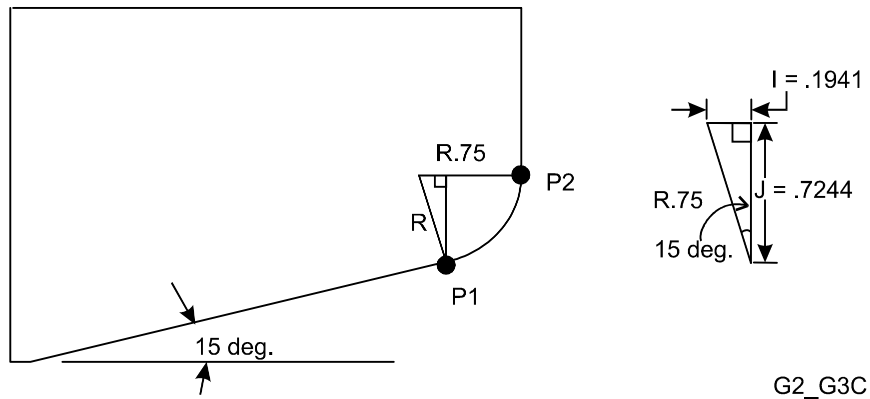


Figure 4-5, Partial Arc Sample

From P1 to P2, the block format is: **G91 G3 X.5559 Y.7244 I-.1941 J.7244**

Construct a triangle at a right angle to the given angle (15 deg.). Using the given angle (15) and the hypotenuse (.75, radius), calculate the lengths of the unknown sides I (opposite side) and J (adjacent side).

A. Sine (15 deg.) times hypotenuse = I

   .2588 x .75 = .1941

   Since **I** is in an X minus direction, **I (X arc center) = -.1941**

B. Cosine (15 deg.) times hypotenuse = J

   .9659 x .75 = .7244

   Since **J** is in a Y positive direction, **J (Y arc center) = .7244**

C. Radius - I = X

   .750 - .1941 = .5559

   X moves in a positive direction. **X (endpoint) = .5559**

D. **Y** (endpoint) = **J** (Y arc center)

   Y = J = .7244

NOTE:	If the endpoint (P2) does not lie along the arc path, the CNC displays an error message.

All rights reserved. Subject to change without notice.

Circles

Since the endpoint and starting point of a circle are the same, you do not need to program an endpoint for a circle. Position the tool at the required starting point before you execute the arc move. Refer to **Figure 4-6**.

Format: G91 G3 J.5

Since X, Y, and I equal 0, omit these parameters.

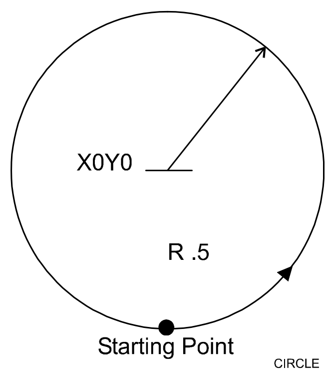


Figure 4-6, Circle Sample

Helical Interpolation (XYZIJK)

Format: G17 G2 Xn Yn Zn In Jn Ln

Helical interpolation adds a third dimension to **G2** or **G3** moves.

For the XY plane (**G17**), the tool will move in a circular motion in the XY axes and linearly in Z, simultaneously.

The added Z parameter provides the Z endpoint. **L** is the number of complete plus partial revolutions, referenced from the start point.

You can use helical interpolation for threading and rough boring applications. Additional linear or rotary axes (**U**, **W**) can also be specified. Refer to **Table 4-8**.

Table 4-8, Helical Interpolation Program

Block	Description
N5 G17 G90 G70 G0 X0 Y0 Z0	Sets XY plane, Absolute, Inch, Rapid Modes. Moves axes to zero.
N6 G02 X2.0 Y0 Z-.5 I1.0 J0 L1 F20	Programs CW helical move to X2 Y0 Z-.5, with center point at I1J0 and 0 complete turns. The tool will execute a half turn at feedrate F20. If L2 were programmed, the tool would make 1-1/2 turns.
N7 G01	Next block.

All rights reserved. Subject to change without notice.

Dwell (G4)

Dwell (**G4**) can be used to program a delay between blocks. A Timed Dwell is a timed stop. An Infinite Dwell is a stop that can be canceled only by pressing **START**. With a dwell activated, the CNC halts motions on all axes, but other functions (coolant on/off, spindle control) remain active. Do not program any other commands on a **G4** block. T is the time in seconds that the machine will remain at the current location. The range of T is .1 to 9999.9 seconds.

Timed **Dwell** Format: **G4 Tx.x (timed)**

Infinite **Dwell** Format: **G4 T0 (infinite)**

Example: N20 G4 T2.1

Block 20 commands a timed dwell with duration of 2.1 seconds.

Example: N21 G4 T0

Block 21 commands an infinite dwell.

The time countdown is displayed in the Machine Status Area of the Manual screen.

NOTE:	ANILAM recommends that you use the Programmed Stop M-function (**M00**) instead of an infinite dwell.

All rights reserved. Subject to change without notice.

Programming Non-modal Exact Stop (G9)

With the In-Position Mode activated, the CNC approaches target and performs an in-position check before it executes the next move. The CNC comes to a complete stop at the end of every block. This could cause witness marks to appear on the work, but prevents the CNC from rounding off sharp corners. Refer to **Table 4-9**.

Format: G9

NOTE: Rapid moves are always performed in In-Position Mode.

Table 4-9, Exact Stop G-Codes

Code	Format	Action
G9	G9 Xx.x Yx.x	Activates non-modal In-Position Mode. Complete stop only in this block.
G61	G61 Xx.x Yx.x	Activates Modal In-Position Mode. The CNC stops to verify location for each targeted position. In-Position Mode remains active until changed.
G64	G64	Cancels **G61** and activates the Contouring Mode (also called Continuous Path Mode).

NOTE: In-Position and Continuous Path Tolerances are defined in the Setup Utility. The In-Position Tolerance should be closer to target than the Continuous Path Tolerance.

The In-Position Mode will be active only for the block containing the **G9** command. Use **G61** to initiate modal Exact Stop (In-Position Mode).

Plane Selection (G17, G18, G19)

Make plane changes prior to circular interpolation (**G2**, **G3**) blocks. Refer to **Table 4-10** for the G-Codes that activate different planes. XY (**G17**) is the default plane at power-on. Refer to Figure 4-7, Plane Selection.

Table 4-10, Plane Selection G-Codes

G-Code	Cutting Plane
G17	XY plane
G18	XZ plane
G19	YZ plane

All rights reserved. Subject to change without notice.

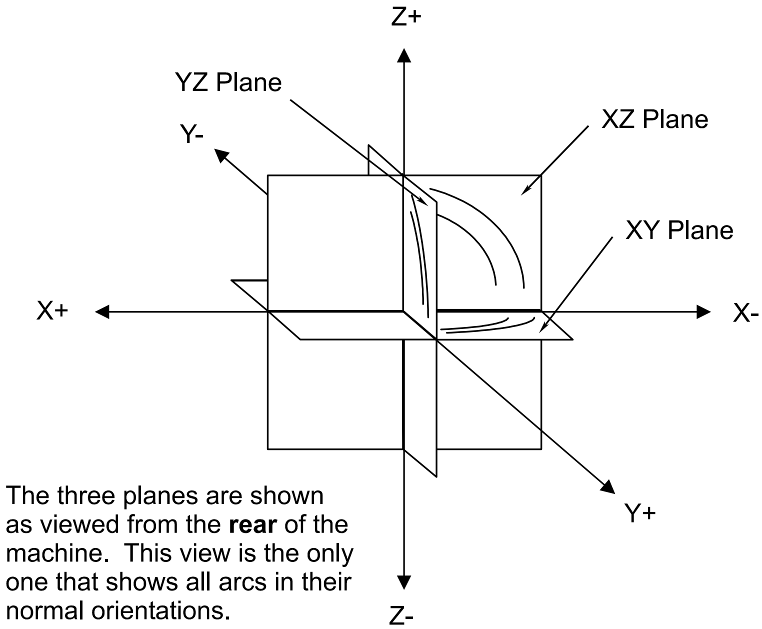


Figure 4-7, Plane Selection

To determine arc direction, look toward the negative direction of the non-used axis. Refer to **Figure 4-8**. (Example: for XY plane, look along Z-.)

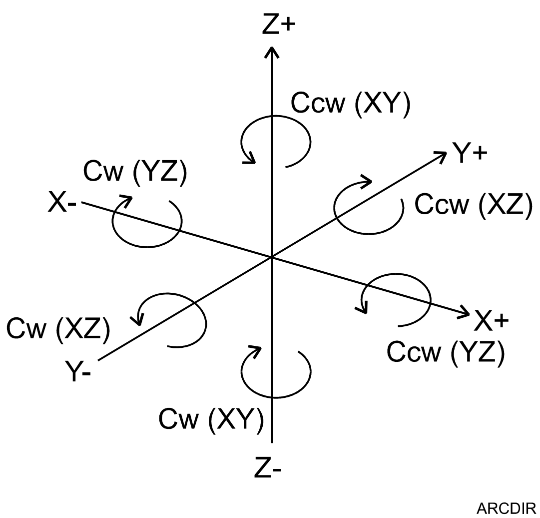


Figure 4-8, Arc Direction

All rights reserved. Subject to change without notice.

Setting Stroke Limit (G22)

The G22 Xn Yn Zn In Jn Kn format (activate software limits) is modal. Use **G22** (alone) to cancel software limits. Refer to **Table 4-11**.

Format: G22 Xn Yn Zn In Jn Kn

Activates software limits.

Format: G22

Cancels software limits and enables free movement within the machine limits.

Table 4-11, G22 Address Words

Address Word	Format	Description
G22	See above	Stored Stroke Limit (Programmable Travel Limits)
X	xxx.xxxx	X positive software limit.
Y	xxx.xxxx	Y positive software limit.
Z	xxx.xxxx	Z positive software limit.
I	xxx.xxxx	X negative software limit.
J	xxx.xxxx	Y negative software limit.
K	xxx.xxxx	Z negative software limit.

The software limits feature creates an envelope that limits the tool's range of travel. It is also called the Stored Stroke Limit feature. The X, Y, and Z limits represent the extreme distance the tool can travel in the positive X, Y, and Z directions. The I, J, and K limits represent the extreme distance the tool can travel in the negative X, Y, and Z directions. Refer to **Figure 4-9**.

Software limits are referenced to Absolute Zero (Machine Home). The values of the positive and negative limits depend on where you locate Machine Home.

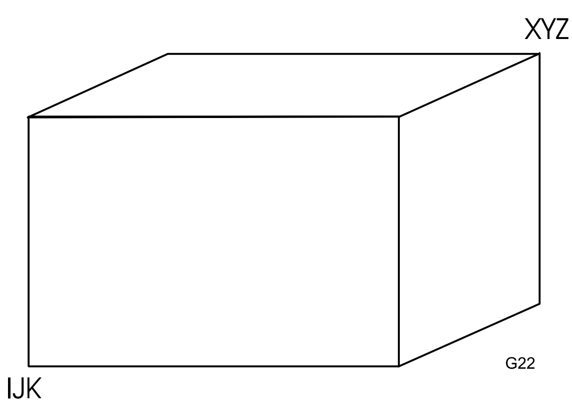


Figure 4-9, Software Limits Envelope Parameters

All rights reserved. Subject to change without notice.

To set software limits:

1. Make sure the tool is within the envelope defined by the software limits (XYZIJK).
2. In Edit Mode or Manual Mode, type the **G22** command in the proper format (G22 Xn Yn Zn In Jn Kn). All the Address Words must be accompanied by the appropriate values or the CNC will not activate software limits.

   In Edit Mode, make sure the appropriate Program Listing is displayed. Type the **G22** command into any program block.

   In the Manual Mode, type the **G22** command next to the command line. Press **START**.

Reference Point Return (G28)

With the **G28** XYZ format, the Reference Point Return command (**G28**) returns the CNC to a permanent reference position. Refer to **Table 4-12**.

Format: G28 XYZU

Returns the machine directly to its X, Y, Z, and U reference point (Machine Home). Axes will return at the current feedrate if in **G1** mode or rapid if in **G0** mode.

Format: G28 Xn Yn Zn Un

n = coordinates X, Y, Z, and U of intermediate point. Return to reference point (Machine Home) through an intermediate point.

Table 4-12, Return to Reference Point, Address Words

Label	Address Word	Format	Description
Mid Point X	Xn	xxx.xxxx	Intermediate point in X, if used
Mid Point Y	Yn	xxx.xxxx	Intermediate point in Y, if used
Mid Point Z	Zn	xxx.xxxx	Intermediate point in Z, if used
Mid Point U	Un	xxx.xxxx	Intermediate point in U, if used

NOTE: At least one axis must be specified, or no motion will occur.

With the G28 Xx Yy Zz Uu format, the machine rapids or feeds (depending on if **G0** or **G1** is active) to the intermediate point and then rapids or feeds to Machine Home. The intermediate point is always in reference to Machine Home.

All offsets and transformations (rotation, mirror, and scaling) are automatically canceled by **G28** before traversing to Machine Home.

All rights reserved. Subject to change without notice.

Return from Reference Point (G29)

Return from Reference Point (Machine Home) (**G29**) is used in conjunction with Reference Point Return (**G28**). **G29** returns the CNC to the intermediary point programmed in **G28**, then to the coordinates programmed in the **G29** block. Return move will be rapid or feed depending on active **G0** or **G1** to a program start position. Refer to **Table 4-13**.

Format: G29 Xx Yy Zz Uu

xyzu = coordinates X, Y, Z, and U of **G29** move. The CNC commands a move from Machine Home to an intermediate point (specified in G28 command), then to the **G29** coordinates.

Table 4-13, G29 Address Words

Label	Address Word	Description
Mid Point X	X	G29 move in X
Mid Point Y	Y	G29 move in Y
Mid Point Z	Z	G29 move in Z
Mid Point U	U	G29 move in U

Table 4-14, G29 Program List

Block	Format	Description of Variables
N1	G28 Xx1 Yy1 Zz1	N1 = coordinates of intermediary point. The CNC traverses to the programmed coordinate (N1), then traverses to Machine Home.
N2	G29 Xx2 Yy2 Zz2	N2 = coordinates of point to which CNC will return after reaching the intermediary point. The machine traverses to the G28 intermediate point (N1), then to the programmed coordinate (N2).

All rights reserved. Subject to change without notice.
25-January-08

Move Reference from Machine Home (G30)

Move Reference from Machine Home (**G30**) is used to move an axis in relation to machine home without being influenced by tool or fixture offsets. Move will be rapid or feed depending on active **G0** (Rapid Move) or **G1** (Feed Move) to a program start position. Refer to **Table 4-15**.

Format: G30 Xx Yy Zz Uu

xyzu = coordinates X, Y, Z, and U of G29 move. The CNC commands a move from Machine Home to the **G30** coordinates.

Table 4-15, G30 Address Words

Label	Address Word	Description
X	X	G30 move in X
Y	Y	G30 move in Y
Z	Z	G30 move in Z
U	U	G30 move in U

Probe Move (G31)

Refer to "Section 17 - Advanced Programming Features," "Probe Move (G31)."

All rights reserved. Subject to change without notice.

Fixture Offset (Work Coordinate System Select) (G53)

Format: G53 Oxx Xn Yn Zn Un Wn C

Use the work coordinate system (**G53)**, commonly known as fixture offsets, to shift Absolute Zero to a preset dimension. **G53** dimensions are referenced to Machine Zero.

G53 cancels Mirroring (**G100**), Axis Rotation (**G68**) and Scaling (**G72**).

99 offsets (zero shifts) are available. Offsets are stored in a table. To activate the Fixture Offset Table in Manual Mode, press **F9** (**Tool**) + **F1** (**Offsets**). You can update this table through the program. If you use a G53 command to change the offsets in the table, the CNC will overwrite the values in the Fixture Offset Table.

The letter O followed by the Fixture Offset Table number (1 to 99) defines an offset.

The following topics are described:

- **Fixture Offset Table**
- Activating the Fixture Offset Table
- Changing Fixture Offsets in the Table
- Adjusting Fixture Offsets in the Table
- G53 Programming Examples

Fixture Offset Table

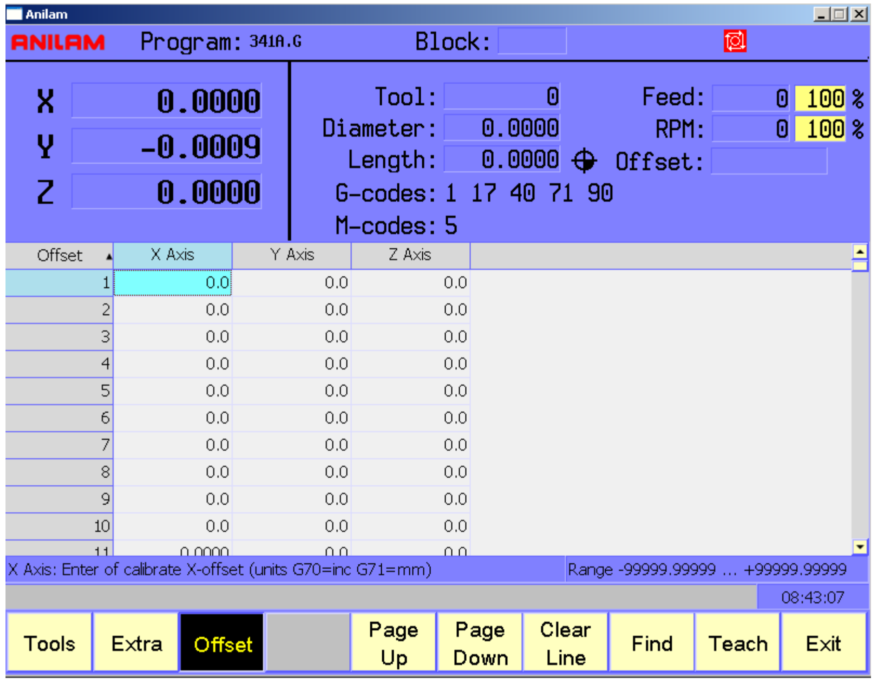
The Fixture Offset Table, accessed via the Manual screen, contains the entered values for Fixture Offsets 1 to 99.

All rights reserved. Subject to change without notice.

Activating the Fixture Offset Table

To activate the Fixture Offset Table:

1. In Manual Mode, press **F9** (**Tool**) + **F3** (**Offset**). The Fixture Offset Table activates. Refer to **Figure 4-10**.



OFFSET

Figure 4-10, Fixture Offset Table

Changing Fixture Offsets in the Table

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. Press an axis key (**X**, **Y**, or **Z**).
3. Type a value. Press **ENTER**. The CNC stores the value 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. Press the letter **A** key to display the message, "Enter axis and adjustment value."
3. Type the axis to adjust (**X**, **Y**, or **Z**) and the amount of the adjustment. The adjustment value may be positive or negative.
4. Press **ENTER** to adjust the value, and display the adjusted value in the table.

All rights reserved. Subject to change without notice.

G53 Programming Examples

G53 examples #1 to #3 below will clear any active **G92**.

1. Use offset number three from preset table: G53 O3

   Activates a zero point previously entered in the table.

2. Clear any active offset: G53 O0

   Use to clear any offset of **G53** or **G92**. It resets the current zero to Machine Home.

3. Update offset table, shift coordinates: G53 On Xn Yn Zn Un C

   The C word tells the CNC to update the table. Use as an immediate command to shift Absolute Zero and save values in the offset table.

4. Update offset table, but do not activate the shift:
   G53 On Xn Yn Zn Un
   is used when offsets are defined at the beginning of a program. It is strictly to set up the offset table.

G92 can be used in reference to (after) any **G53** active, or without any **G53** active (G53 O0). **G53** is modal, and G53 O0 (use none) is active at power-up.

NOTE: Use **G40** to cancel **G41**/**G42** before programming **G53**.

All rights reserved. Subject to change without notice.

Modal Corner Radius/Chamfering (G59, G60)

Use **G59** to program modal corner rounding or chamfering. The corner-rounding format blends the intersection of two moves. The chamfer format chamfers the intersection of two moves. You can use **G59** at the intersection of non-tangent line-line, line-arc, arc-line, and arc-arc moves (provided a blend radius or chamfer is possible). You cannot blend radii at the intersection of a line tangent to an arc.

G59 can be used to blend inside or outside radii. Tool diameter compensation can be active during modal corner rounding. When you blend inside radii with diameter compensation active, the blend radius must be greater than the tool radius.

R defines the radius value for corner rounding. E defines a chamfer size. Refer to **Table 4-16**. **G59** is modal. It will stay active until canceled with a **G60** code. The CNC activates linear interpolation (**G1**) with **G59**. You do not have to program **G1** prior to the **G59** block.

Corner Rounding Format: G59 Rn

Chamfer Format: G59 En

Cancel G59: G60 (Cancels G59 immediately.)

Cancel G59: G60 Xn Yn Zn (Cancels G59 after move.)

Table 4-16, G59 Address Words

Address Word	Description
R	Corner radius
E	Chamfer distance

G60 cancels **G59** immediately. G60 Xn Yn Zn cancels **G59** at the end of the move it contains (as in N13). For example, if **G60** were programmed on a block prior to the X0 move, the lower-left corner would not be rounded.

You can change the blend radii or chamfer value between moves. To change the radius to .25 for the bottom two corners, insert **G59 R.25** between Blocks N10 and N11. The new radius would apply on the next move (after Block N10).

All rights reserved. Subject to change without notice.

In the example in **Figure 4-11**, **G59** is used to command modal corner rounding. Whenever the CNC encounters an intersection between line-line, arc-arc, or line-arc moves, it will round off the intersection to the specified radius.

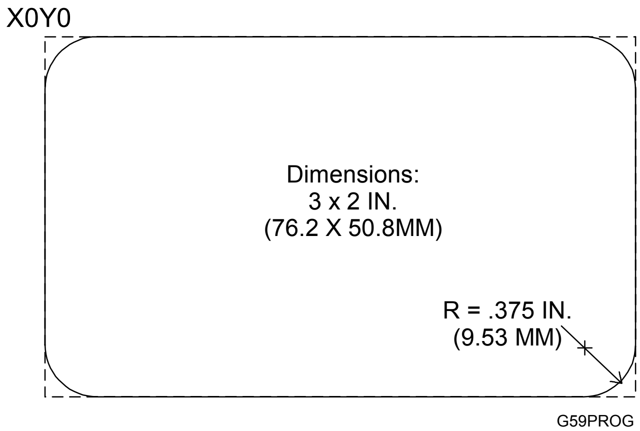


Figure 4-11, G59 Programming Example

Table 4-17 describes the required program blocks.

Table 4-17, G59 Programming Example, Inch

Blk. #	Block	Description
N4	G17 G90	* Set plane and absolute
N5	G0 X-.5 Y-.5	* Move to point
N6	Z-.25	* Lower Z-axis
N7	G1 X0 F20	* Move to X0 and set feedrate
N8	G59 R.375	* Set G59 and radius value
N9	Y0	* Move to Y0
N10	X3	* Move to X3
N11	Y-2	* Move
N12	X0	* Move
N13	G60 Y.5	* Move to Y.5, then deactivate modal corner rounding
N14	G0 Z.1	* Raise Z-axis

All rights reserved. Subject to change without notice.

In-Position Mode (Exact Stop Check) (G61)

While the In-Position Mode (**G61**) is active, the CNC approaches target and performs an in-position check before the next move is executed. Refer to **Table 4-18**. The CNC comes to a complete stop at the completion of each command. This could cause tool dwell marks to appear on the work, but prevents the CNC from rounding off sharp corners.

Table 4-18, G61 and Associated G-Code Formats

Code	Format	Action
G9	G9 Xx.x Yx.x	Activates Non-modal In-Position Mode. Complete stop only in this block.
G61	G61 Xx.x Yx.x	Activates Modal In-Position Mode. CNC stops to verify location of each endpoint.
G64	G64	Cancels **G61** and activates the Contouring Mode (Continuous Path Mode).

NOTE: Rapid moves are always performed in In-Position Mode.

G61 is modal and remains in effect until canceled. Use Contouring Mode (**G64**) to cancel the **G61**. Non-modal In-Position Mode (**G9**) remains active only for a single block.

NOTE: The In-Position and Continuous Path Tolerances are defined in the Setup Utility. The In-Position Tolerance should be closer to target than the Continuous Path Tolerance.

All rights reserved. Subject to change without notice.

Contouring Mode (Cutting Mode) (G64)

The Contouring Mode (**G64**), also known as Continuous Path Mode or Cutting Mode, is active at power on. Refer to **Table 4-19**. It is used for feed moves. With the Contouring Mode activated, the CNC approaches target and comes within the Continuous Path Tolerance of the target position. No in-position check is made before the next move is executed. This enables the smooth contouring of a profile or surface.

Format: G64

Table 4-19, G64 and Associated G-Code Formats

Code	Format	Action
G9	G9 Xx.x Yx.x	Activates Non-modal In-Position Mode. Complete stop only in this block.
G61	G61 Xx.x Yx.x	Activates Modal In-Position Mode. CNC stops to verify location of each endpoint.
G64	G64	Cancels **G61** and activates the Contouring Mode (Continuous Path Mode).

NOTE: Rapid moves are always performed in In-Position Mode.

NOTE: the machine builder defines the In-Position and Continuous Path Tolerances in the Setup Utility.

G64 is modal and remains in effect until canceled. Use Exact Stop Check (**G61**) to cancel the Contouring Mode. **G64** initiates linear interpolation (**G1**).

All rights reserved. Subject to change without notice.

User Macros (G65, G66, G67)

NOTE:	Before using macros, you must understand how variables and parameters are used in a program or subprogram. Refer to "Section 17 - Advanced Programming Features" for an explanation of these features.

NOTE:	**G65** or **G66** codes always contain some letter variable(s) (Pn, An, Bn, etc.) to be passed to the macro (subprogram).

A macro is a group of instructions stored in memory and called by the main program when needed. Think of macros as sophisticated, flexible subprograms, which can be modal (**G66**) or Non-modal (**G65**). Refer to **Table 4-20**.

Macros might consist of:

- Customized canned cycles to simplify the programming of a particular part or entire programs for similar part production.
- Parameters (Pn, An, Bn, etc.) passed to the subprogram by letter address, similar to canned cycles.
- Automatic measuring sequences using sensors, such as probes, for feedback to the CNC.

Table 4-20, Macro G-Codes

Format	M/NM	Action
G65 Pn Ln **G65 Pn** **G65 Pn, An, Bn, etc.**	Non-modal	Executes Non-modal Macro (Pn), with optional repeat loop, at current location. Macro is repeated number of times specified in command (Ln). If the **L** address word is omitted, the macro will be executed only once. Pn = macro number (O). Ln = optional loop. Specify number of times the macro should repeat (n).
G66 Pn **G66 Pn, An, Bn, etc**	Modal	Executes called macro (Pn) after each programmed move until canceled with a **G67** command. Pn = called macro.
G67	Cancel	Cancels Modal Macro (**G66**).

All rights reserved. Subject to change without notice.

Table 4-21 lists and describes the Address Words and M-Codes used with macros.

Table 4-21, Macro Address Words

Address Word	Format	Description
Pn	Pxxxx	Used in **G65** and **G66** commands. Lists macro number (O) to be called.
Ln	Lxxxx	Used only in **G66**. Optional repeat command. Specify number of times macro should repeat (1 to 9999).
On	N(block Number) Oxxxx	Macro number that occurs in the first line of the macro; for identification.
M99	M99	End macro (subprogram) and return to line following **G65** or **G66** in main program.

A subprogram consists of fixed dimensions, but a macro contains variables and parameters that can change every time the macro is used. The CNC can pass values to variables in the **G65** or **G66** command. to variables

Macros can be stored in the same file as the main program or in a separate file. Use the File Inclusion feature to call Macros stored in a separate file.

Refer to "Section 17 - Advanced Programming Features" for a more detailed explanation of Parameter Passing and Variables and File Inclusion.

Macros stored in the same file as the main program are defined in the same way as a subprogram; with the **O** address word followed by a label number. The macro is terminated with the **M99** code and entered into the Program Listing after the main program. Refer to Table 4-22, Macro Program List.

If the command contains an **L** address word, the macro is repeated the specified number of times before the CNC returns to the main program.

All rights reserved. Subject to change without notice.

Table 4-22, Macro Program List

Program Block	Description
N200 M2	End main program
N210 O201	Macro number assigned
N220 [Enter macro here]	Macro program
N230	
N240	
N250	
N260 M99	End macro, return to next line of main program. The CNC returns to the line following the Macro call (G65 or G66) in the main program.

Use the **G65** Macro call to call a macro into the main body of the program. Refer to **Table 4-23**.

Table 4-23, Macro Call in Main Program

Program Block	Description
N40	
N50 G65 P201	CNC executes Macro O201 once, at present location.
N60	After executing the macro (**M99** encountered), the CNC returns to the main program and performs the next programmed command.

The CNC executes the macro (201) at block 50, with or without repeated loops, as programmed. When the CNC detects the **M99** (End Macro) Code, it returns to the next line of the main program (N60).

All rights reserved. Subject to change without notice.

Axis Rotation (G68)

G68 is modal and remains active until canceled. Refer to **Table 4-24**. The CNC automatically cancels rotation if you program **S** and **L**. Use only the listed codes.

Activate Format: G68 In Jn Sn Cn Pn Ln

Cancel Format: G68

Table 4-24, G68 Address Words

Label	Address Word	Description
Angle	**C**	Angle of Rotation. (Required)
Rotation Ctr.	**I**	Center of rotation (polar origin) in X-axis.
Rotation Ctr.	**J**	Center of rotation (polar origin) in Y-axis.
First Angle	**S**	Start angle (referenced original programmed angular position). This variable is used only if **L** and **P** are programmed.
SubProgram	**P**	Subprogram number to call.
# of Times	**L**	Number of loops. Number of times **C** will increment, and subprogram **P** will be called.

Patterns commanded by the program can be rotated using polar coordinates. Any angle can be described as positive or negative, depending on how it is referenced. CCW from 0 degrees is positive. CW from 0 degrees is negative. Refer to **Figure 4-12**.

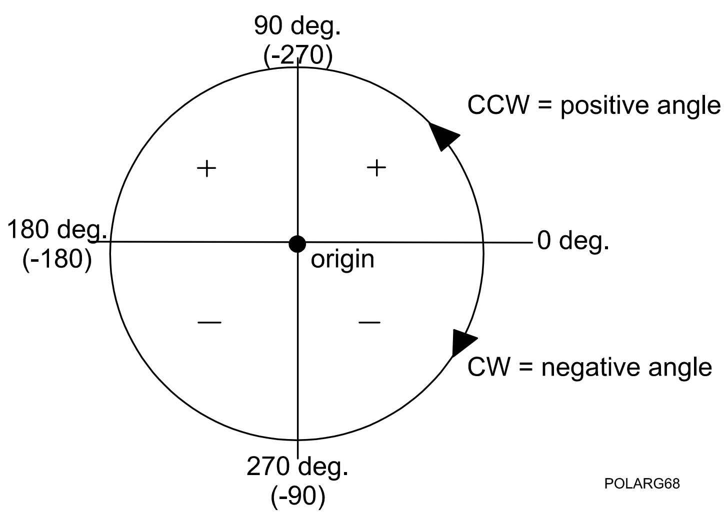


Figure 4-12, G68 Angle Rotation Guide

All rights reserved. Subject to change without notice.

Minimum data entry for G68 rotation is: G68 Cn. If **I** and **J** are not given, the current position is used. **S** angle is referenced to the original programmed position. For example: If a slot is programmed at the 90-degree position, **S** is referenced from 90 degrees. **S** should be used only if **L** and **P** are programmed. **C** must be programmed. **P** and **L** are optional. They enable a loop to be executed, so the subprogram will be called at each angle increment. **G17**, **G18**, or **G19** must be commanded prior to programming **G68**.

The following topics are described:

- **G68 Programming Examples**

G68 Programming Examples

Example 1: Refer to **Figure 4-13** and **Table 4-25**.

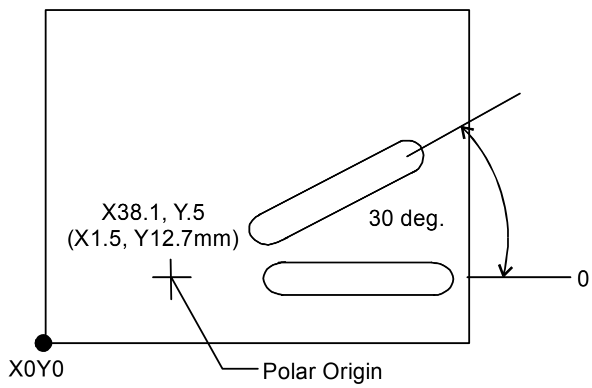


Figure 4-13, G68 Programming Example 1

Table 4-25, G68 Programming Example 1

Blk. #	Block	Description
N21	G17 G90	* Set plane and absolute
N22	G68 I1.5 J.5 C30	* Activate rotation to values
N23	M98 P1001	* Execute subprogram
N24	G68	* Cancel rotation
Required Subprogram: O1001 G90 G0 X2.5 Y.375 G1 Z-.125 F5 X3.5 G3 Y.625 J.125 G1 X2.5 G3 Y.375 J-.125 G0 Z.1 M99		

Example 1 does not use **S**, **C**, **P**, or **L**. (No loop is required.)

All rights reserved. Subject to change without notice.

N21 sets the XY plane and Absolute Mode. N22 enables rotation angle of 30 degrees, the origin is X1.5 Y.5. N23 executes sub 1001 at the rotated position. The sub is programmed at the 3 o'clock position. N24 cancels polar rotation.

Example 2: Refer to **Figure 4-14** and **Table 4-26**.

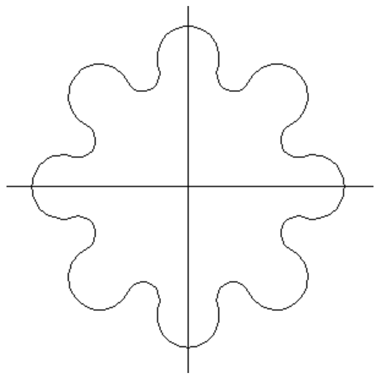


Figure 4-14, G68 Programming Example 2

Table 4-26, G68 Programming Example 2

Blk. #	Block
N1	O688 * G68-2
N2	G90 G70 G17 G0 T0 Z0
N3	X0 Y0
N4	T1 * .25 MILL
N5	Y2.5 Z.1
N6	G1 Z-.125 F5
N7	Y2.0 F14
N8	G68 I0 J0 S0 C-45 P1 L8
N9	G90 G1 Y2.5
N10	G0 G28 Z0
N11	X0 Y0
N12	M2
N13	
N14	
N15	
N16	O1 * 45 DEG. SECTION
N17	G91 G1 G41 Y-.125
N18	G2 X.3542 Y-.4981 I0 J-.375
N19	G3 X.3689 Y-.1528 I.1889 J-.0657
N20	G2 X.6027 Y.1017 I.3375 J-.1634
N21	G1 G40 X.0884 Y.0884
N22	M99

All rights reserved. Subject to change without notice.

Example 2 uses all variable words of the **G68** function. Only the path from the 12 o'clock position (90 deg.) to the 1:30 position (45 deg.) is programmed in the subprogram. The **G68** loop increments the angle and recalls the subprogram to complete the shape.

N1 through N4 set program number, modals, position and tool activation.

N5 and N6 move the tool to the starting position.

N7 moves to the 12 o'clock position one-half of the cutter away from the part.

Note that cutter compensation cannot be active prior to calling **G68**. **G41** or **G42** cutter radius compensation must be activated at the beginning of the subprogram, and then canceled with a **G40** prior to the end of the subprogram.

N8 calls the **G68** rotation function: origin (**I**,**J**) at X0 Y0, starting angle (**S**) of zero degrees (First call of subprogram will not be rotated), angle increment (**C**) of -45 deg. (CW is negative), call subprogram (**P**) 1, and loop count (**L**) equals eight.

N9 returns back to the starting position in the Y axis.

N10 to N12 brings the Z axis to machine home, then move the X and Y axes to position and ends the program.

N16 to N22 define the 45-degree section for the shape, from 12 o'clock position to 1:30 position, turning on and off cutter radius compensation for each iteration of the rotation loop.

NOTE:	If you are using **S** and **L** format, you do not need to cancel **G68**.

NOTE:	Select the plane prior to **G68** (default is **G17**). Program dimensions for both axes of the active plane.

The CNC interprets **IJKABC** values in the current Absolute/Incremental Mode. If **C** is absolute, the 3 o'clock position is 0 degrees. If **C** is incremental (**G91**), the current angle is 0 degrees. It is better to use incremental (**G91**) dimensions.

All rights reserved. Subject to change without notice.

Activating Inch (G70) or MM (G71) Mode

Inch Mode Format: G70

MM Mode Format: G71

Change the unit of measurement displayed by the CNC by using Inch Mode (G70) or MM Mode (G71). Refer to **Table 4-27**. The Inch/MM Mode is usually specified at the start of a program.

Table 4-27, Activating the Inch/MM Mode

Block	Format	Description
N2	**G70** G90 G0	Activates **Inch** Mode.
N2	**G71** G90 G0	Activates **MM** Mode.

NOTE: The display resolution (number of decimal places shown) is set up in the Setup Utility. The default resolution is four decimal places for Inch Mode and also four decimal places for MM Mode.

Scaling (G72)

Use Scaling (**G72**) to enlarge or reduce patterns commanded by the program. Refer to **Table 4-28**. **G72** is modal. If a variable word is not given, it is assumed to be x1 factor. Axes for circular motion must have the same factor.

Activating Format: G72 Xn Yn Zn

Table 4-28, Cancel Format: G72

Label	Address Word	Description
Scale Factor X	**X**	Scaling factor for X-axis
Scale Factor Y	**Y**	Scaling factor for Y-axis
Scale Factor Z	**Z**	Scaling factor for Z-axis

Tool length offsets, diameter offsets, tool wear factors, and cutter compensation are not affected by **G72**. No other codes are allowed on a **G72** block.

WARNING: Never program a T-Code (T0, T1) while in G72.

Activate the Tn command before **G72**, and then deactivate **G72** before deactivating the Tn command. This applies to all tools (T0 to T99).

Example: G72 X2 Y2 Z1

The CNC will scale all X and Y moves to twice their programmed size. **Z** moves will not be scaled (times 1). **Z** could have been omitted.

All rights reserved. Subject to change without notice.

Activating Absolute (G90) or Incremental (G91) Mode

You can change the program mode to **G90** or **G91**. Specify Absolute or Incremental Mode at the start of a program. Refer to **Table 4-29**.

Absolute Mode Format: G90

Incremental Mode Format: G91

Table 4-29, Activating the Absolute/Incremental Mode

Block	Format	Description
N2	G70 G90 G0	Activates the Absolute Mode
N2	G70 G91 G0	Activates the Incremental Mode

Absolute Zero Point Programming (G92)

The **G92** code is used to set axes to zero (reset) or to new coordinates (preset). It is sometimes used to set Part Zero. You can use **G92** to set Part Zero on a vise or a fixture. Anilam recommends using **G53** (Fixture Offset) instead of **G92**.

G92 cancels Mirroring (**G100**), Axis Rotation (**G68**), and Axis Scaling (**G72**).

NOTE: **G92** should only be used in Auto or Single Step mode. If programmed in MDI, the **G92** cancelled when the CNC switches to Auto.

All rights reserved. Subject to change without notice.

Mirroring (G100)

Format:

G100 XYZUVW

G100 programmed with axis (G100 X) activates "mirroring" (ON) for that axis. Mirroring reverses the sign (+/-) of subsequent numbers. More than one axis can be mirrored at once (G100 XY). To cancel mirroring, program **G100** on a block by itself. Refer to **Table 4-30**.

Table 4-30, G100 Address Words

Label	Address Word	Description
X Axis	**X**	Activates "mirroring" (ON) for the X-axis.
Y Axis	**Y**	Activates "mirroring" (ON) for the Y-axis.
Z Axis	**Z**	Activates "mirroring" (ON) for the Z-axis.
U Axis	**U**	Activates "mirroring" (ON) for the U-axis.
V Axis	**V**	Activates "mirroring" (ON) for the V-axis.
W Axis	**W**	Activates "mirroring" (ON) for the W-axis.

All rights reserved. Subject to change without notice.
25-January-08

BlockForm (G120)

Format:

G120 Xnn.nnnn Ynn.nnnn Znn.nnnn Inn.nnnn Jnn.nnnn Knn.nnnn

G120 is used to define a window in relation to the part zero. This is used by the Draw function to present a solid model on the raw stock. Refer to **Table 4-31**.

Note: Even if there is no cutter compensation being used, you must have an active tool with a diameter before draw will graphically show material being machine from the above stock definition.

G120 can be placed only once any where within the program and must be accompanied by all of the following parameters:

Table 4-31, G120 Address Words

Label	Address Word	Description
XMax	**X**	The positive most edge in the X-axis of the stock from part zero. (Required)
YMax	**Y**	The positive most edge in the Y-axis of the stock from part zero. (Required)
ZMax	**Z**	The top of the stock in relation to part zero. (Required)
XMin	**I**	The negative most edge in the X-axis of the stock from part zero. (Required)
YMin	**J**	The negative most edge in the Y-axis of the stock from part zero. (Required)
KMin	**K**	The depth of the part from top to bottom in relation to part zero. (Required)

All rights reserved. Subject to change without notice.

Programmable Temporary Path Tolerance (G1000)

Format:

G1000 Xx

G1000 is used to temporarily override the parameter for path tolerance. G1000 should only be used in program and should be programmed by itself. The value in configuration is restored at the end of the program. The typical default is 0.010 mm (0.0004"). This can be useful if the CNC hesitates between small moves, such as a 3-D surface output from CAD-CAM.

All rights reserved. Subject to change without notice.
25-January-08

Feedrate (FEED)

Format:

Fn.n

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. Refer to **Table 4-32**.

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

Table 4-32, FEED Address Word

Label	Address Word	Description
FEED	**F**	Feedrates for Inch mode (**G70**) are programmed as inch/minute (IPM). F1 = 1.0 IPM Feedrates for MM (metric mode) (**G71**) in mm/min: F1 = 1 mm/min (Required)

All rights reserved. Subject to change without notice.

Section 5 - Canned Cycles and Subprograms

The following topics are described in this section:

Canned Cycles

A canned cycle is a preset sequence of events initiated by a single block of data. Canned cycles are part of the CNC software and cannot be altered. They simplify the programming of complicated cycles. One block of data can instruct the CNC to perform the necessary moves to drill a hole, or mill a pocket.

A canned cycle consists of a G-Code and variable words. The variable words describe parameters, such as peck distance, retract height, pocket depth and tool stepover. Each canned cycle has its own set of variable words.

The variable words in a canned cycle allow you to customize the cycle to include the necessary dimensions, feedrates, etc.

Canned cycles greatly reduce program blocks. Use them whenever applicable.

Canned cycles are usually entered into the part program from the Main Edit Help Menu. Edit Help contains graphics and labeled entry fields to make programming canned cycles quick and easy. Refer to "Section 7 - Edit Help" for details.

All rights reserved. Subject to change without notice.

Drilling, Tapping, and Boring Canned Cycles (G81 to G89)

When you activate a drilling cycle, it executes after each programmed position, until you cancel it.

NOTE:	The **P** entry (return height) is optional, and you do not need to provide it. If you do not specify **P**, the CNC will set it to **R**.

Keep the following in mind for drill cycles:

- **P** dimension is optional. If it is not given, the retract height will be the same as the Z start height (**R** dimension).
- **F** feedrate is optional. If it is not given, the current feedrate is used.
- All start heights (**R**) and finish heights (**P**) as well as Z dimensions are absolute dimensions.
- **P** must be greater than **R**, or an alarm will be given.
- For all peck drill cycles (**G83** and **G87**), **R** (start height) must be 0.1" (or 2 mm) above the work surface.
- **G84** (Tapping) uses **S** word for Spindle **Yes**/**No**. Your machine must be equipped with spindle M-functions to use **G84**.
- Z-axis depth can be changed by placing a new Z depth on the same line as the X- and/or Y-axis location of the hole you want the new depth applied. A Z address on a line of its own will cause the control to drill the new depth at the current location.

The following topics are described:

All rights reserved. Subject to change without notice.

Drilling Off (G80)

Format: G80

Modal cycles remain active until canceled. Use **G80** to cancel drill, tap, and bore canned cycles (**G81** to **G89**). **G80** can be included with other commands on a block.

Basic Drill Cycle (G81)

Format: G81 Zn Rn Fn Pn

G81 is a basic drilling cycle, generally used for center drilling or hole drilling that does not require a pecking motion. It feeds from the start height (**R**) to the specified hole depth (**Z**) at a given feedrate (**F**), then rapids to the return height (**P**). Refer to Figure 7-12, Basic Drill Cycle Screen. Refer to **Table 5-1**.

Table 5-1, G81 Address Words

Label	Address Word	Description
ZDepth	**Z**	Absolute hole depth. (Required)
StartHgt	**R**	Initial Z start point, in rapid. (Required)
Feed	**F**	Feedrate
ReturnHgt	**P**	Z return point after hole depth, in rapid. **P** must be higher than **R**.

CounterBore Drill Cycle (G82)

Format: G82 Zn Rn Dn Fn Pn

G82 is the counter bore cycle, generally used for counterboring. It feeds from the R-plane to **Z** depth, dwells for specified time, then rapids to the return point. Refer to Figure 7-13, CounterBore Drill Cycle Screen. Refer to **Table 5-2**.

Table 5-2, G82 Address Words

Label	Address Word	Description
Finish Depth	**Z**	Absolute hole depth. (Required)
Start Height	**R**	Initial Z start point, in rapid. (Required)
Dwell Time	**D**	Dwell time (in seconds). (Required)
Feed Rate	**F**	Feedrate
Return Height	**P**	Z return point after hole depth, in rapid. **P** must be higher than **R**.

All rights reserved. Subject to change without notice.

Peck Drill Cycle (G83)

Format: G83 Zn Rn In Fn Pn

G83 is the peck drilling cycle, generally used for peck drilling relatively shallow holes. **G83** feeds from the R-plane to the first peck depth (calculated so that all pecks are equal and do not exceed the maximum peck distance programmed in I word). Then rapid retracts to R-plane (to clear chip), rapids down to previous depth less .02", and continues this loop until it reaches the final hole depth. It then rapid retracts to the **P** dimension. Refer to **Table 5-3**.

Table 5-3, G83 Address Words

Label	Address Word	Description
ZDepth	**Z**	Absolute whole depth. (Required)
StartHgt	**R**	Initial Z start point, in rapid. (Required)
Peck	**I**	Maximum peck distance (positive dimension). (Required)
Feed	**F**	Feedrate
ReturnHgt	**P**	Z return point after hole depth, in rapid. **P** must be higher than **R**.

All rights reserved. Subject to change without notice.

Tapping Cycle (G84)

Format: G84 Zn Rn Vn Sn Pn Dn

NOTE:	The machine must be equipped with spindle M-functions (FWD, REV, OFF) to use this cycle. Do not use G84 if the machine does not have spindle commands available.

G84 is the tapping canned cycle, used for tapping holes. During a **G84** cycle: the tool feeds from the R-plane to **Z** depth; the spindle stops and reverses; the tool feeds to the retract plane; and the spindle stops and reverses again. Refer to **Table 5-4**.

F (TPIorLead): Enter Threads per Inch when in Inch mode. Enter Lead when in MM (**G71**) mode. Lead is the distance from one thread to the next. You must program a spindle RPM. The Feedrate is calculated based on the spindle RPM and the TPI or Lead specified.

S (Spindle sync): To enable Spindle sync, enter a value of 1. The machine must have direct spindle control to use this feature. The spindle rotation and Z-axis movement will be synched together, as in a threading cycle.

D (Dwell): A dwell time value in seconds can be entered. You may require this feature because of the time required to stop and reverse the spindle.

NOTE:	If **S**=0, the programmed Dwell (**D**) will be active when the spindle reverses at the bottom and top of each hole. If **S**=1, the programmed Dwell (**D**) will be at the top of each hole.

Table 5-4, G84 Address Words

Label	Address Word	Description
ZDepth	**Z**	Absolute hole depth (Required)
StartHgt	**R**	Initial Z start point, in rapid. (Required)
TPIorLead	**V**	Threads per Inch (TPI) in Inch mode, or Lead (Distance between threads) in MM mode. (Required)
SynSpn	**S**	Spindle, **No** (0), or **Yes** (1).
ReturnHgt	**P**	Z retract height after hole depth, in rapid.
Dwell	**D**	Dwell time

All rights reserved. Subject to change without notice.

Boring Bidirectional Cycle (G85)

Format: G85 Zn Rn Fn Pn

G85 is a boring cycle, generally used to make a pass in each direction on a bore or to tap with a self-reversing tapping head. It feeds from the R-plane to **Z** depth, and then feeds back to the retract height. Refer to **Table 5-5**.

Table 5-5, G85 Address Words

Label	Address Word	Description
ZDepth	**Z**	Absolute hole depth. (Required)
StartHgt	**R**	Initial Z start point, in rapid. (Required)
Feed	**F**	Feedrate
ReturnHgt	**P**	Z return point after hole depth, in rapid.

Boring Unidirectional Cycle (G86)

Format: G86 Zn Rn Fn In Dn Pn Cn

G86 is a boring cycle that allows the X-axis to back off the bore surface after the spindle has stopped and oriented itself. The cycle will feed from the R-plane to **Z** depth, dwell for the specified time, stop and orient the spindle to the specified angle **C**, back off in X, rapid retract in Z, re-position in X, and restart the spindle. Refer to **Table 5-6**.

NOTE: Your machine must be equipped with spindle M-functions (Spindle Forward [**M3**], Spindle Reverse [**M4**], Spindle Off [**M5**]) and spindle orientation (**M19**) to use this cycle. Do not use the **G86** cycle if the machine does not have the spindle commands and spindle orientation.

Table 5-6, G86 Address Words

Label	Address Word	Description
Finish Depth	**Z**	Absolute hole depth. (Required)
Start Height	**R**	Initial Z start point, in rapid. (Required)
Feed Rate	**F**	Feedrate
X Backoff	**I**	X-axis incremental backoff distance in X (positive or negative dimension).
Dwell Time	**D**	Dwell time (in seconds)
Return Height	**P**	Z return point after hole depth, in rapid.
Index Angle	**C**	**M19** index angle. If no angle is given, the angle in MC_5003, Default Spindle Orientation Angle, is used.

All rights reserved. Subject to change without notice.

Chip Break Cycle (G87)

Format: G87 Zn Rn In Jn Kn Fn Wn Un Pn

G87 is the chip-breaker peck-drilling cycle, generally used to peck-drill medium to deep holes. The cycle feeds from the R-plane to the first peck depth in **Z**, rapid retracts the chip-break increment (**W**), feeds to the next calculated peck depth (initial peck less **J**), and continues this sequence until it reaches a **U** depth, or until final hole depth is reached. The peck distance will never be more than **I** or less than **K**. Refer to **Table 5-7**.

This cycle enables optimum drilling conditions for holes. For maximum efficiency in deep hole drilling, set parameters to accommodate the material and tool types used. Generally, the deeper the hole, the smaller the peck distance (**J**). This prevents the binding of chips, tool, and workpiece. Set **U** to retract the drill completely at set depth intervals.

Table 5-7, G87 Address Words

Label	Address Word	Description
ZDepth	**Z**	Absolute hole depth. Required.
StartHgt	**R**	Initial Z start point, in rapid. Required.
FirstPeck	**I**	First peck distance (positive dimension). Required.
PeckDecr	**J**	Amount to subtract from previous peck (positive dimension). Required.
MinPeck	**K**	Minimum peck distance (positive dimension). Required.
Feed	**F**	Feedrate.
ChipBrkInc	**W**	Chip break increment (positive dimension).
RetractDep	**U**	Incremental depth between full retracts (positive dimension).
ReturnHgt	**P**	Z return point after hole depth, in rapid. **P** must be higher than **R**.

All rights reserved. Subject to change without notice.

Flat Bottom Boring Cycle (G89)

Format: G89 Zn Rn Dn Fn Pn

G89 is a boring cycle, generally used to program a pass in each direction with a dwell at the bottom. The tool feeds from the R-plane to **Z** depth, dwells for specified time, then feeds to the retract (**P**) dimension. Refer to **Table 5-8**.

Table 5-8, G89 Address Words

Label	Address Word	Description
Finish Depth	**Z**	Absolute hole depth. (Required)
Start Height	**R**	Initial Z start point (0.1 inch or 2 mm), in rapid. (Required)
Dwell Time	**D**	Dwell time (in seconds). (Required)
Feed Rate	**F**	Feedrate
Return Height	**P**	Z return point after hole depth, in rapid.

Drilling Example

The following example assumes that the machine has no automatic tool changer (ATC). If your machine has an ATC, check your machine manual for proper tool changer programming procedures. Refer to **Figure 5-1** and Table 5-9, Drilling Example, Inch (Metric).

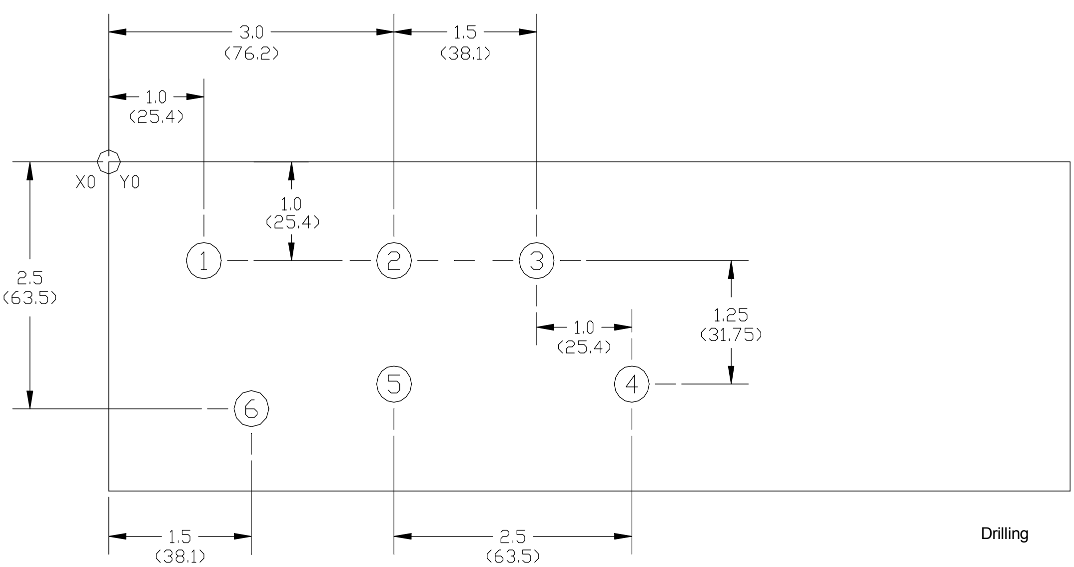


Figure 5-1, Drilling Example

All rights reserved. Subject to change without notice.

Table 5-9, Drilling Example, Inch (Metric)

Blk #	Block	Description
N1	O1 * DRIL-X1	Program number (1) and name (DRILL-EX1).
N2	G90 G70 (G71) G0 T0 Z0	Sets absolute dimensions (G90), inch input (G70), rapid (G0), cancel any active tool (T0), and bring Z to zero (Z0).
N3	X-3.0 (X-75) Y1.0 (Y25)	Move to X-3 Y1.
N4	T01 * 1/4" DRILL (6.35 DRILL)	Activate Tool #1 length offset.
N5	G83 Z-.55 (Z-14) R.1 (R2) F12 (F300) I.08 (I2) P.1 (P2)	Initiates peck drill cycle G83: Z = hole depth, R = start height, F = feedrate, I = maximum peck, P = return height.
N6	X1.0 (X25.4) Y-1.0 (Y-25.4)	Hole location #1 (Rapid and Absolute).
N7	X3.0 (X76.2)	Hole #2.
N8	G91 X1.5 (X38.1)	Hole #3 (moves from #2 to #3 in incremental: G91).
N9	X1.0 (X25.4) Y -1.25 (Y-31.75)	Hole #4 (Incremental).
N10	X-2.5 (X-63.5)	Hole #5 (Incremental).
N11	G90 X1.5 (X38.1) Y -2.5 (Y-63.5)	Hole #6 (Absolute).
N12	G80 T0 Z0	Cancel drill cycle (G80), cancel tool (T0), and rapid Z to zero (Z0).
N13	X-3.0 (X-75) Y1.0 (Y25)	Move to X-3 Y1 for part change.
N14	M02	End Program.

All rights reserved. Subject to change without notice.

Pattern Drill Cycles

Use the drill bolt hole cycle (**G79**) to drill a partial or full bolt circle. A drill cycle (**G81** to **G89**) must be programmed prior to **G79**. You can move around the pattern clockwise or counterclockwise, either point to point or along a radius. **G79** calculates the hole locations. The cycle uses the Polar Coordinate System for dimensions. When the **G79** cycle is completed, you must cancel the cycle (**G80**).

Drill Bolt Hole Cycle (G79)

Format: G79 An Hn Dn Xn Cn Yn Bn Rn

Table 5-10, G79 Address Words

Label	Address Word	Description
StartAngle	**A**	Angle of the first hole. (Required)
#Holes	**H**	Number of holes in full bolt circle. (Required)
Diameter	**D**	Diameter of bolt circle. Tool will normally move from hole to hole in a CCW (positive) direction. For CW direction, **D** = negative. (Required)
XCenter	**X**	Absolute X center of the bolt-circle. Defaults to current position.
IndexAngle	**C**	Rotates the Polar Coordinate System by entered angle. Default: 0 degrees (3 o'clock). CCW = positive, CW = negative.
YCenter	**Y**	Absolute Y center of the bolt-circle. Defaults to current position.
EndAngle	**B**	Angle of the last hole. If there is no **B** value, the CNC will execute a full bolt hole circle.
Radial Path	**R**	Move from hole to hole on a radius. Set to 1.0 to activate circular path between holes. Defaults to straight-line path between holes.

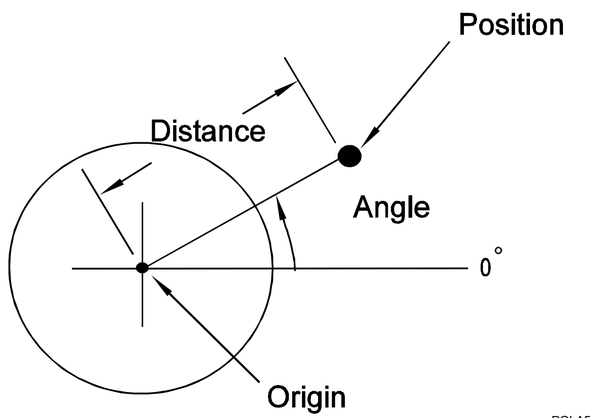


Figure 5-2, Polar Coordinates

All rights reserved. Subject to change without notice.

Drill Pattern Cycle (G179)

Format: G179 Xn Yn Bn En Un Vn Cn An Dn Wn

NOTE: Do not program G68 with G179.

Use the automatic hole pattern canned cycle (**G179**) to program partial or full pattern hole grids. You can use **G179** for a corner pattern when holes are required only on four corners. It calculates the hole locations from the entered variables. You can also rotate the pattern around the starting hole location. A drill cycle (**G81–G89**) must be programmed prior to G179. You must cancel the cycle (**G80**) after the pattern is completed. Refer to Figure 7-14, Drill Pattern Cycle Screen. Refer to **Table 5-11**.

You can use [**A** and **D**] or [**U** and **V**], but not both combinations. Positive and negative values are allowed in all variable words except: **B**, **E**, and **W**.

Table 5-11, G179 Address Words

Label	Address Word	Description
X	**X**	Absolute X position of start hole. (Required)
Y	**Y**	Absolute Y position of start hole. (Required)
#XHoles	**B**	Number of holes in X-axis. (Required)
#YHoles	**E**	Number of holes in Y-axis. (Required)
XIncr	**U**	Increment between holes in X-axis. Can be used instead of **A**.
YIncr	**V**	Increment between holes in Y-axis. Can be used instead of **D**.
Angle	**C**	Angle to rotate the hole pattern. Default is 0 degrees (3 o'clock position).
Length	**A**	Length of pattern in X-axis. If used, **U** cannot be given.
Width	**D**	Width of pattern in Y-axis. If used, **V** cannot be given.
Perimeter	**W**	Pattern or Square. If **W** is 0, then a matrix pattern will be drilled. If **W** is 1, then a perimeter pattern (edges only) will be drilled. Refer to **Figure 5-3**.

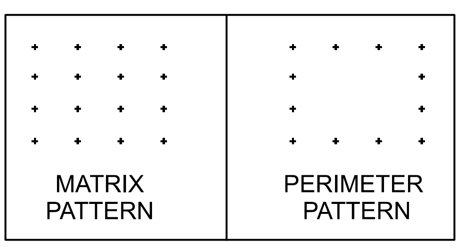


Figure 5-3, Matrix vs. Perimeter Pattern

All rights reserved. Subject to change without notice.

Example:

G81 Z-.1 R.1 F15

G179 X2 Y1 C30 B6 E4 U.5 V.375 W0

G80

These blocks rotate a bolt hole pattern 30 degrees counterclockwise. Refer to **Figure 5-4**.

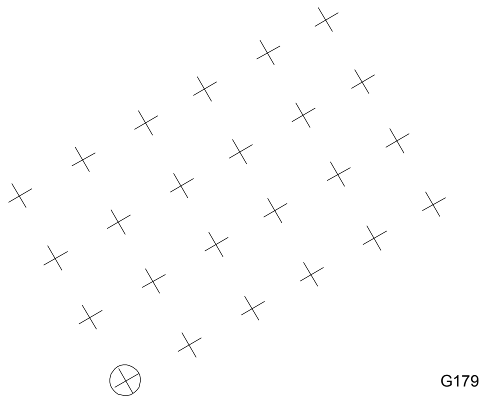


Figure 5-4, G179 Programming Example

All rights reserved. Subject to change without notice.

Pocket Cycles

Pocketing cycles eliminate extensive programming. One block of programming will mill out the described pocket. Activate a tool before programming a pocket cycle. All pockets use the current tool diameter from the Tool Page.

XY positioning may be necessary prior to programming a pocket cycle.

Programmer is responsible for all **Z** moves in Hole Mill (**G76**) cycle.

Cutting direction is reversible in the pocketing cycles.

Always check that tool-to-corner radii do not conflict.

Z and **P** dimensions are absolute.

On all cycles with variable **A** (tool stepover), **A** must be less than the tool diameter. In **G78** and **G178**, **A** must be 70% or less of tool diameter.

Alarm messages will occur if the CNC detects program errors.

G41 and **G42** are not permitted during pocket cycles. Pocket cycles use "built-in" cutter compensation.

Stock variable #1030 is not permitted and will be ignored.

WARNING:	**When you cut one pocket inside another, make sure to set P above the highest pocket. At the end of each pocket, the tool will rapid to P, then rapid to the start position.**

For plunge pockets (**G177**, **G178**), drill a start hole prior to activating the pocket; position the axes over the start hole prior to **G177** or **G178**.

The following topics are described:

All rights reserved. Subject to change without notice.

Draft Angle Pocket Cycle (G73)

Format: G73 Xn Yn Hn Zn In En An Bn Cn Dn Qn Vn Sn Kn Wn Jn

Use the draft pocket milling cycle (**G73**) to machine a draft angle on a pocket. The tool must be at the center point of the lower-left corner radius. This is where the machining begins. You can use **G78** to mill out an initial pocket prior to the **G73** block. Refer to **Table 5-12**.

Table 5-12, G73 Address Words

Label	Address Word	Description
Bottom Length	**X**	X length at the bottom of the pocket. (Required)
Bottom Width	**Y**	Y width at the bottom of the pocket. (Required)
Start Height	**H**	Z absolute rapid start height (must be 0.1 inch or 2 mm above surface). (Required)
Z Depth (abs)	**Z**	Z absolute pocket depth. (Required)
Z Step Rough	**I**	Z-axis roughing step-down. (Required)
Draft Angle	**E**	Draft angle to be machined on vertical walls of pocket. (Required)
Lower Left Rad	**A**	Lower-left corner radius. Cannot be less than tool radius. (Required) If only **A** is used, then the **A** value is used for all four corners.
Lower Right Rad	**B**	Lower-right corner radius. Cannot be less than tool radius.
Upper Left Rad	**C**	Upper-right corner radius. Cannot be less than tool radius.
Upper Right Rad	**D**	Upper-left corner radius. Cannot be less than tool radius.
Z Step Finish	**Q**	Z-axis finishing step-down.
Max XY Step	**V**	Maximum XY tool stepover. Used if angle is so great that the amount of XY step per Z step exceeds 70 % of the tool diameter.
Finish STK XY	**S**	XY finish stock amount, sides only.
Finish Feed	**K**	Finish-pass feedrate.
Flat 0, Ball 1	**W**	Flat end mill = 0. Ball end mill = 1. Default is flat end mil (0)
RoughFeed	**J**	Roughing feedrate

All rights reserved. Subject to change without notice.

Example:

This program will cut the draft angle pocket shown in the figure. The drawing does not show the finish pass. Assume an existing rectangular pocket (4 in. long x 2 in. wide x 1 in. deep) with a theoretical sharp lower-left corner at X2 Y2. The following program will machine a draft angle onto the existing pocket. Refer to **Figure 5-5** and **Table 5-13**.

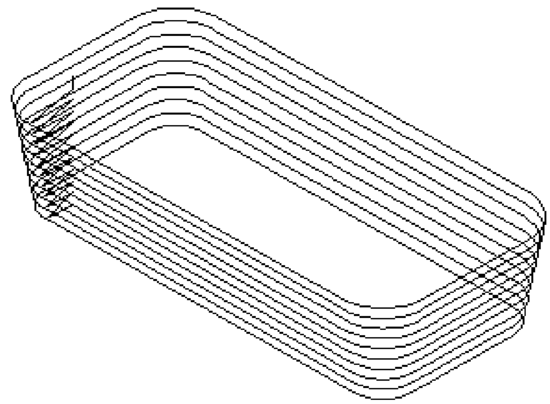


Figure 5-5, G73 Programming Example

Table 5-13, G73 Programming Example

T1 M3 S2000 ***** 1/2" FLAT END MILL
G90 G0 X2.5 Y2.5 F30 ***** 4" x 2" x 1" DP RCT. PKT ALREADY EXISTS
G73 X4 Y2 H.1 Z-1 A.5 E10 I.1 S.01 Q.02 R35
G0 T0 Z0 M5
X0 Y0 M2

Position the tool above the center of the lower-left corner radius. The tool path starts and ends at the center of the lower-left corner radius (after each perimeter pass) for all roughing passes. During finish passes, the tool will step down the draft angle and make passes around the perimeter.

If a ball-end mill is programmed (**W**=1), the following points must be considered: If **W**=1, the length (**X**) and width (**Y**) at the bottom of the pocket is measured at the tangency point of the ball radius and the draft angle. If **W**=1, the start height (**H**) must be set to (.1 + ball radius) above surface to be cut. If **W**=1, set the tool-length offset so that the ball is buried up to its centerline when at the part surface (touch off the tip and add the ball radius, or touch off tip and use a negative length wear equal to the ball radius).

All rights reserved. Subject to change without notice.

Frame Pocket Cycle (G75)

Format: G75 Mn Wn Zn Un Hn Cn Xn Yn Bn In Jn Vn Kn Sn An Pn

Frame milling (**G75**) will mill a frame or trough around an island of material. You must position the XY axes at the lower-left (theoretical sharp) corner of the island before you program **G75**. Refer to **Table 5-14**.

Prior to **G75**, activate a tool so that the CNC will consider the tool diameter. The relation of the outside corner radius to the frame width must be geometrically possible or an Error Message is displayed.

Table 5-14, G75 Address Words

Label	Address Word	Description
Length	**M**	Length of island in X-axis (Required)
Width	**W**	Width of island in Y-axis (Required)
ZDepth	**Z**	Absolute depth of frame (Required)
InsideRad	**U**	Inside corner radius of frame (corner radius of island) (Required)
StartHgt	**H**	Z absolute starting (rapid) height (must be 0.1 inch or 2 mm above surface to be cut into) (Required)
FrameWidth	**C**	Frame width (Required)
XCenter	**X**	Center of island in X-axis. Default: Current position.
YCenter	**Y**	Center of island in Y-axis. Default: Current position.
DepthCut	**B**	Maximum **Z** depth per pass (For example, if **Z** is programmed to be -1, and **B** to be .5, the frame will be roughed out in two levels.) **B** is programmed as a positive dimension. Defaults to tool diameter (depth) less finish stock.
RampFeed	**I**	Ramp in feed: The tool will ramp into the first depth of cut with a YZ move from the I.D. of the frame to the O.D. of the frame. Defaults to last programmed feedrate.
RoughFeed	**J**	After the ramp-in move described above, the tool will rough-mill the frame, at feedrate **J**. Defaults to last programmed feedrate.
OutsideRad	**V**	Outside corner radius of frame. Defaults to value of **U** (InsideRad). Must be equal to or greater than tool radius.
FinFeed	**K**	Finish-pass feedrate. Defaults to last programmed feedrate.
FinStock	**S**	Finish stock amount per side (including bottom). If you enter a negative value, stock will be left, but no finish pass will occur. If you do not enter a value, finish stock will not be left.
Stepover	**A**	Maximum tool stepover (must be less than tool diameter): +**A** dimension = climb (CCW) -**A** dimension = conventional (CW) Defaults to half tool diameter.
RetractHgt	**P**	Z-axis absolute (rapid) retract height (must be equal to or above **H**). Defaults to **H** (StartHgt) value.

All rights reserved. Subject to change without notice.
25-January-08

Example:

G75 M3 W1.125 H.1 Z-.375 A.25 B.36 I5 J18 U.25 V.5 C1 S.015 K30 P.1

Figure 5-6 illustrates the moves output by the CNC to mill the frame. cycle:

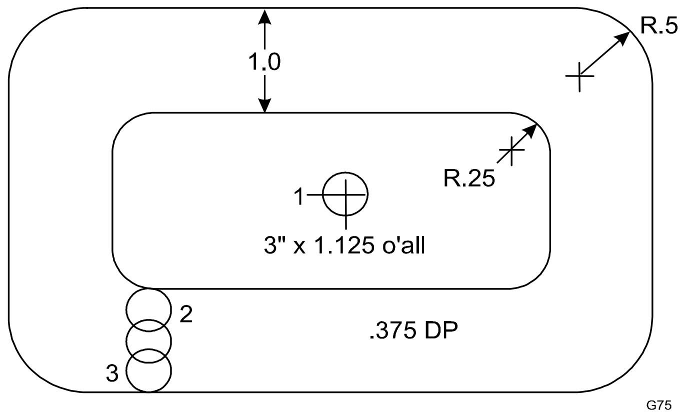


Figure 5-6, G75 Programming Example

The tool will perform the following operations:

NOTE:	If **X** and **Y** are not provided, position the tool at the center of the island prior to **G75**.

1. Tool will rapid from position 1 to position 2: X is the center of the inside corner radius (**U**), and **Y** is the corner radius plus tool radius plus finish stock.
2. Tool will feed -.1 (or 2 mm) in **Z** to the part surface.
3. Tool performs a ramp-in move to O.D. of frame minus tool radius minus finish stock (position 3).
4. Tool then moves 360 degrees CCW around frame back to position 3.
5. Tool then steps over calculated amount, and mills CW until position 2 is reached again at depth.
6. Tool then mills 360 degrees CW (climb-milling) around the island.

NOTE:	The number of times the tool repeats Steps 3 through 6 depends on the **Z** and **B** dimensions.

7. When the frame is completed, the tool rapids first to the **P** dimension, then to the center of the island.

All rights reserved. Subject to change without notice.

Hole Mill Cycle (G76)

Format: G76 Dn Xn Yn Bn Zn Hn Jn Kn Sn

Use the hole milling cycle (**G76**) to machine through holes or counter-bores. You can position the tool at the hole center prior to the **G76** block. Activate a tool prior to **G76** so that the CNC knows the tool diameter. Refer to **Table 5-15**.

If you do not provide **Z** and **H**, program a separate **Z** move to raise the tool out of the hole after the cycle.

Table 5-15, G76 Address Words

Label	Address Word	Description
Diameter	**D**	Diameter of hole. Negative D = CW direction. The direction CCW (climb milling) is reversible: +**D** dimension = climb (CCW) -**D** dimension = conventional (CW) (Required)
XCenter	**X**	X coordinate of the center. Default: Pocket centers at present position.
YCenter	**Y**	Y coordinate of the center. If no coordinate is provided, default is set to present position.
DepthCut	**B**	Z-axis increment used for each pass.
ZDepth	**Z**	The absolute depth of the finished pocket.
StartHgt	**H**	Absolute Z position to which the CNC rapids before feeding into the workpiece.
RoughFeed	**J**	Rough-pass feedrate. Defaults to last programmed feedrate.
FinFeed	**K**	Finish-pass feedrate. Defaults to last programmed feedrate.
FinStock	**S**	Finish-stock amount per side. If you enter a negative value, stock will be left, but no finish pass will occur. If you do not enter a value, no finish stock will be left.

Example:

G76 D2.5 J12 S.01 K20

In Figure 5-7, G76 Programming Example, the tool will perform the following operations:

1. Tool moves from position 1 to a position 45 degrees from center, at half the radius (position 2).
2. Tool then arcs onto the O.D. tangential (CCW) (position 3).
3. Tool mills O.D. CCW (position 4).

All rights reserved. Subject to change without notice.

4. Tool leaves O.D. tangentially to a point 135 degrees from the center at half the radius. CCW (position 5).
5. Tool returns to center (position 6).
6. If you have programmed a finish pass, the process repeats at the finish dimensions.

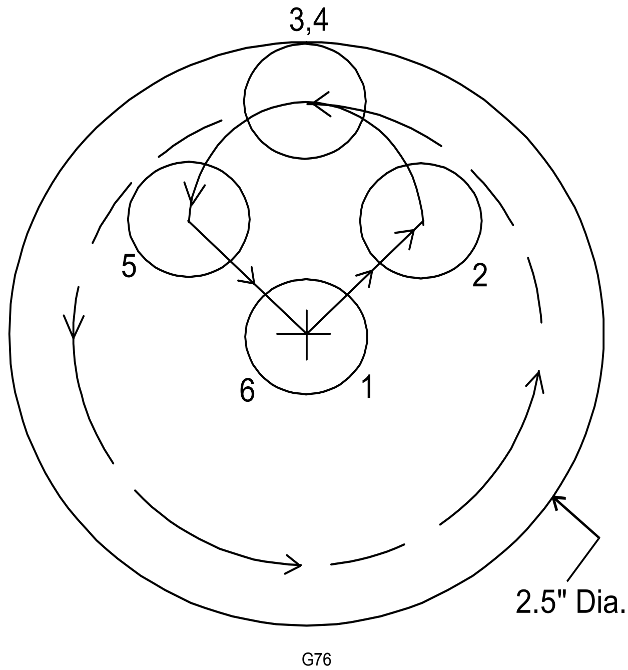


Figure 5-7, G76 Programming Example

All rights reserved. Subject to change without notice.

Circular Pocket Cycle (G77)

Format: G77 Zn Hn Dn Xn Yn Bn In Kn Sn An Pn

Use the circular pocket canned cycle (**G77**) to mill round pockets. You do not have to place the tool at the center of the pocket, since the cycle has variable words for **X** and **Y** center. If **X** and **Y** variable words are not programmed, the CNC will use the current position as the pocket center. Refer to **Table 5-16**.

Activate a tool prior to programming **G77** so that the CNC will know the cutter diameter. You can position the tool at the pocket center and omit the XY words. By default, the CNC will use the current position as the pocket center.

Table 5-16, G77 Address Words

Label	Address Word	Description
ZDepth	**Z**	Absolute depth of pocket. (Required)
StartHgt	**H**	Z absolute starting height (0.1 inch or 2 mm above surface). Executed in rapid. (Required)
Diameter	**D**	Diameter of pocket. The direction CCW (climb milling) is reversible (Required): +**D** dimension = climb (CCW) -**D** dimension = conventional (CW)
XCenter	**X**	Center of the pocket in X-axis. Defaults to current position.
YCenter	**Y**	Center of the pocket in Y-axis. Defaults to current position.
DepthCut	**B**	Maximum **Z** depth per pass (Example: If **Z** is programmed to be -1, and **B** to be .5, the pocket will be roughed out in two levels.) **B** is programmed as a positive dimension. Defaults to tool diameter (depth), less finish stock.
RoughFeed	**I**	Ramp in/rough feed: The tool will ramp into the first depth of cut with a spiral move from the I.D. of the pocket to the O.D. of the pocket. The feedrate for this move is programmed as **I**. After the ramp-in move, the tool will rough-mill the pocket, at feedrate **I**. Defaults to last programmed feedrate.
FinFeed	**K**	Finish-pass feedrate. Defaults to last programmed feedrate.
FinStock	**S**	Finish-stock amount per side (including bottom). If you enter a negative value, stock will be left, but no finish pass will occur. If not programmed, no finish stock is left.
Stepover	**A**	Maximum tool stepover (must be less than tool diameter). If +**A** dimension = outward spiral. If -**A** dimension = inward spiral. On inward spirals, the tool moves to O.D. at 0 degrees, and begins the roughing process there (3 o'clock). Defaults to tool radius.
RetractHgt	**P**	Z-axis absolute-retract height (must be equal to or above **H**). Executed in rapid. Defaults to **H** (**StartHgt**) dimension.

All rights reserved. Subject to change without notice.

Example:

G77 X2 Y2 H.1 Z-.25 D3 A.35 B.25 I12 S.01 K20 P.1

In **Figure 5-8,** the tool will perform the following operations:

NOTE: **Figure 5-8** shows only the tool path.

1. Tool will move to X2 Y2 (position 1) in current modes: G0/1, G90/91, G70/71 (position 1).
2. Tool will feed .1 in. (2 mm) down in Z-axis.
3. Tool will move to O.D. (less finish stock) in a 3-axis spiral motion (position 2).
4. Tool will make a full circle (position 2).
5. Tool then spirals inward to complete the roughing cycle, at the first level.
6. If you have specified a finish pass, repeat steps 3 through 5 at the finish feedrate.
7. Tool rapids to P dimension, then to the original XY location.

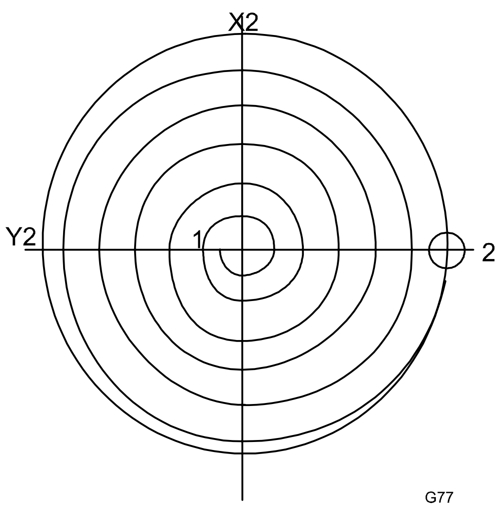


Figure 5-8, G77 Programming Example

All rights reserved. Subject to change without notice.

Rectangular Pocket Cycle (G78)

Format: G78 Mn Wn Zn Hn Xn Yn Bn In Jn Kn Sn An Un Pn

Use the rectangular pocket cycle (G78) to mill square or rectangular pockets. You must position the tool directly over the center of the pocket prior to the G78 block, or use the X Y words. Refer to **Table 5-17**.

Activate a tool prior to programming G78, so cutter diameter is known.

Table 5-17, G78 Address Words

Label	Address Word	Description
Length	**M**	Length of pocket in X-axis (Required)
Width	**W**	Width of pocket in Y-axis (Required)
ZDepth	**Z**	Absolute depth of pocket (Required)
StartHgt	**H**	Z absolute starting height (must be 0.1 inch or 2 mm above surface to be cut). Executed in rapid. (Required)
XCenter	**X**	X coordinate of the center. Default: pocket centers at present position.
YCenter	**Y**	Y coordinate of the center. If no coordinate is provided, default is set to present position.
DepthCut	**B**	Maximum **Z** depth per pass (Example: if you program **Z** to be -1, and **B** to be .5, the CNC will rough out the pocket in two levels.) **B** is programmed as a positive dimension. Defaults to tool diameter (depth), less finish stock.
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.
RoughFeed	**J**	After the ramp-in move, the tool will rough-mill the pocket, at feedrate **J**. Defaults to last programmed feedrate.
FinFeed	**K**	Finish-pass feedrate. Defaults to last programmed feedrate.
FinStock	**S**	Finish stock amount per side (including bottom). If entered as negative, stock will be left, but no finish pass will occur. If not programmed, no finish stock is left.
Stepover	**A**	Maximum tool stepover (must be 70% or less of tool diameter). +**A** dimension = climb (CCW). -**A** dimension = conventional (CW). Defaults to half tool diameter.
CornerRad	**U**	Actual corner radius of pocket (all four corners will be same). Must be equal to or greater than tool radius. Defaults to tool radius.

(Continued…)

All rights reserved. Subject to change without notice.

Table 5-17, G78 Address Words (Continued)

Label	Address Word	Description
RetractHgt	**P**	Z-axis absolute finish height (must be equal to or above **H**). Executed in rapid. Defaults to **H** (**StartHgt**) value. WARNING: **When you cut a pocket inside another pocket, you must set P above the highest pocket. At the end of each pocket, the tool will rapid to P, then rapid to the start position.**

Example:

G78 M4 W2 H.1 Z-.5 U.75 A.35 B.25 I7 J12 S.01 K20 P.1

NOTE: If you do not use **X** and **Y** words, you must position the tool at the center of the pocket before the **G78** block.

In **Figure 5-9**, the tool will perform the following operations:

NOTE: **Figure 5-9** shows only the tool path.

1. Tool moves to the center of the radius in the lower-left corner (position 1).
2. Tool feeds -.1" (2 mm) to meet the part surface.
3. Tool moves in XYZ toward center of pocket (position 2) to the first roughing depth, at feedrate I.
4. Tool mills out the pocket with straight lines and arcs (if necessary), using feedrate **J**, until first level is completed.
5. Tool repeats this process if necessary to achieve full depth (less finish stock).
6. Optional finish pass is made in the same manner at feedrate **K**.
7. When the pocket is complete, tool rapids to **P**, then to the center of the pocket.

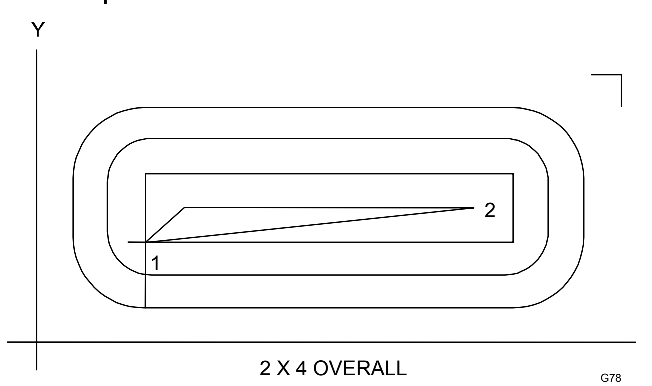


Figure 5-9, G78 Programming Example

All rights reserved. Subject to change without notice.

Irregular Pocket Cycle (G169)

Format: G169 Wn Xn Yn Hn Zn Mn An Bn In Jn Sn Kn Pn

Use **G169** 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 **G41** (left) or **G42** (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 parametric programming or 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 5-18**.

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 **G169** line using **G162**, Islands.

Table 5-18, G169 Address Words

Label	Address Words	Description
Sub#	**W**	The number of the subprogram that contains the perimeter of the pocket. Must be a closed shape. (Required)
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)
ZDepth	**Z**	The Absolute depth of the pocket (Required)
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)
XStart	**X**	A rapid to a position to start pocket. Cycle will rapid the Z-axis to position **P** (**RetractHgt**); then, **X** and **Y** to the starting position before beginning pocket. If not given, the cycle will use the current position.
YStart	**Y**	A rapid to a position to start pocket. (Same as **XStart** above)

(Continued…)

All rights reserved. Subject to change without notice.

Table 5-18, G169 Address Words (Continued)

Label	Address Words	Description
RampFeed	**I**	The feedrate at which the tool will "ramp" into the pocket in all three axes.
RoughFeed	**J**	Rough-cycle feedrate
FinFeed	**K**	Finish-cycle feedrate
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. The finish stock amount applies to the sides and bottom unless **M** (**SideStock**) is defined; then, **S** (**FinStock**) will only apply to the bottom. If you do not specify a value, finish stock is not left.
SideStock	**M**	Finish stock side. If not set, the cycle will use the **S** (**FinStock**) value.
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.
RetractHgt	**P**	Retract height. The Absolute Z position at the start and end of the cycle. **Caution: The Z-axis will rapid to this position before traversing to the X and Y staging position.**

All rights reserved. Subject to change without notice.

Islands (G162)

Format: G162 An Bn Cn Dn En

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

More than one **G162** Island 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 5-19**.

Islands that are defined to be avoided on the inside of an irregular pocket are done so by using the **G162** followed by a list of up to 5 subprogram label names. If more than 5 islands need to be defined, the **G162** 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:

G162 A 2 B 3 C 4 D 5 E 6

G162 A 7 B 8 C 9 D 10 E 11

G162 A 12 B 13 C 14 D 15 E 16

G162 A 17 B 18

and so forth … prior to calling the **G169** 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 **G41** (left) or **G42** (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 **G162** and **G169**, so cutter diameter is known. **G162** is for use with **G169** only. Program **G162** before **G169**.

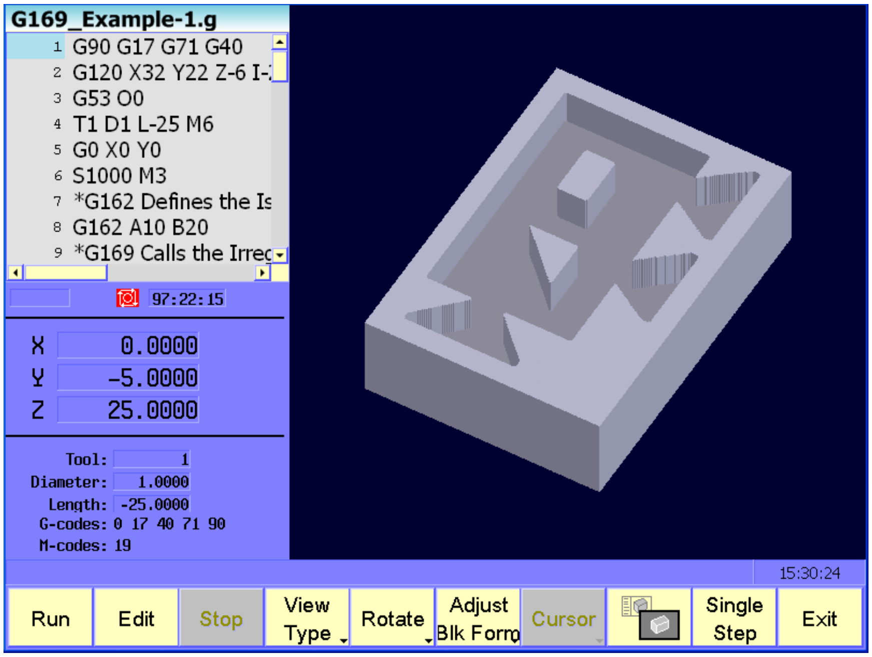
Table 5-19, G162 Address Words

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

All rights reserved. Subject to change without notice.

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 5-10** and Table 5-20, Pockets with Islands Subroutines Programming Example.



SUBROUTINES PISLANDS

Figure 5-10, Subroutines Pockets with Islands Example Workpiece

All rights reserved. Subject to change without notice.

Table 5-20, Pockets with Islands Subroutines Programming Example

1	G90 G17 G71 G40
2	G120 X32 Y22 Z-6 I-2 J-2 K-15
3	G53 O0
4	T1 D1 L-25 M6
5	G0 X0 Y0
6	S1000 M3
7	*G162 Defines the Islands
8	G162 A10 B20
9	*G169 Calls the Irregular Pocket Cycle
10	G169 W1 X5 Y5 A.69 Z-10.0 H2.0 I500 J2000 K1500 M.1 S.2 P25
11	G0 G90 Z25
12	M5
13	G0 X0 Y-5
14	M2
15	O1 *Subroutine for outside of pocket
16	*G41 to Indicate which side the cutter is on
17	G41
18	G0X5 Y5
19	G1 X13
20	X10 Y0
21	X20
22	X16 Y5
23	X24
24	X21 Y0
25	X31
26	X27 Y5
27	X30
28	Y20
29	X5
30	Y15
31	X0 Y20
32	Y7
33	X5 Y12
34	Y5
35	M99
36	
37	O10 *Subroutine for first island
38	*G42 to indicate cutter path is outside of the island
39	G42
40	G0 G90 X10 Y10

All rights reserved. Subject to change without notice.

41	X15
42	Y15
43	X10 Y10
44	M99
45	
46	O20 *Subroutine for second island
47	*G41 to indicate cutter path is also outside of the island
48	G41
49	G0 G90 X20 Y12
50	Y15
51	X25
52	Y12
53	X20 Y12
54	M99

All rights reserved. Subject to change without notice.

Irregular Pocket Examples

Example 1:

This example uses an irregular pocket cycle to cut the pocket shape. Refer to **Figure 5-11**. Program the perimeter of the pocket in a subprogram. The CNC calculates the moves to mill out the pocket. Enter a 3/8" diameter tool in the Tool Page. This part program consists of a main program and a subprogram. Refer to **Table 5-21**.

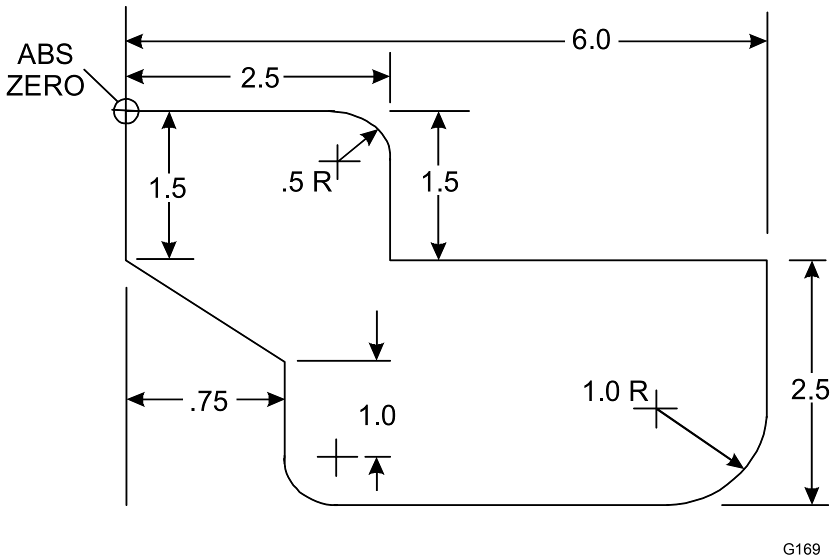


Figure 5-11, G169 Programming Example 1

Table 5-21, G169 Programming Example 1

1	G90 G70 G17
2	T1
3	G169 W1 X0 Y0 H.1 Z-.125 A.15 M.01 S.01 I7.5 J12.5 K9.5 P2.0
4	G90 G00 T0 Z0
5	M2
6	O1
7	G42
8	G90 G00 X0.0 Y0.0
9	G01 X2
10	G2 X2.5 Y-.5 R.5
11	G1 Y-1.5
12	X6
13	Y-3
14	G2 X5 Y-4 R1
15	G1 X1.25
16	G2 X.75 Y-3.5 R.5

All rights reserved. Subject to change without notice.

17	G91 G1 Y1
18	G90 X0 Y-1.5
19	Y0
20	M99

Example 2:

Use an irregular pocket cycle to cut the pocket shape. Input the "perimeter" of the pocket into a subprogram. The CNC will calculate the moves to mill out the pocket. Input a 3/8" diameter tool in the Tool Page. This part program consists of a main program and a subprogram. Refer to **Figure 5-12** and **Table 5-22**.

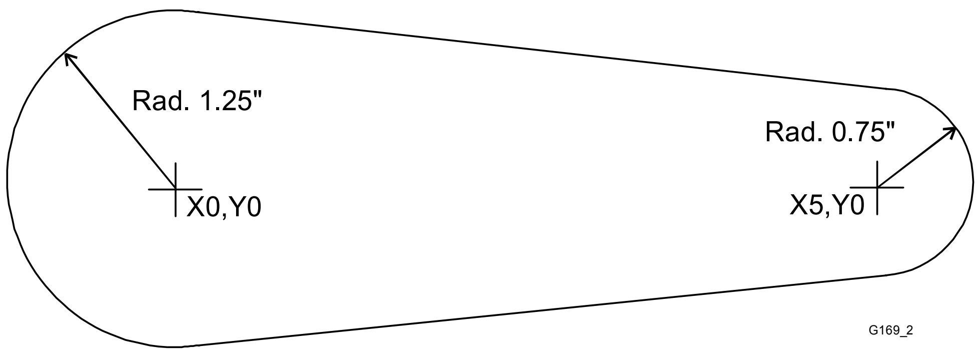


Figure 5-12, G169 Programming Example 2

Table 5-22, G169 Programming Example 2

1	G90 G70 G0 T0 Z0
2	T1 M6
3	G169 W99 X0 Y0 H.1 Z-.25 A.16 B.125 M.01 S.01 I7.5 J12.5 K9.5 P2
4	G90 G00 T0 Z0
5	M2
6	O99
7	G42
8	G90 G00 X-1.25 Y0
9	G2 X .125 Y 1.2437 R1.25
10	G1 X 5.075 Y .7462
11	G2 X5.075 Y-.7462 R.75
12	G1 X .125 Y -1.2437
13	G2 X -1.25 Y0.0 R1.25
14	M99

All rights reserved. Subject to change without notice.

Face Mill Cycle (G170)

Format: G170 Xn Yn An Bn Fn Hn Zn Dn En

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

Execution begins one tool radius from the **D** and **E** (start point). The selected stepover determines the approach axes. Refer to **Figure 5-13**.

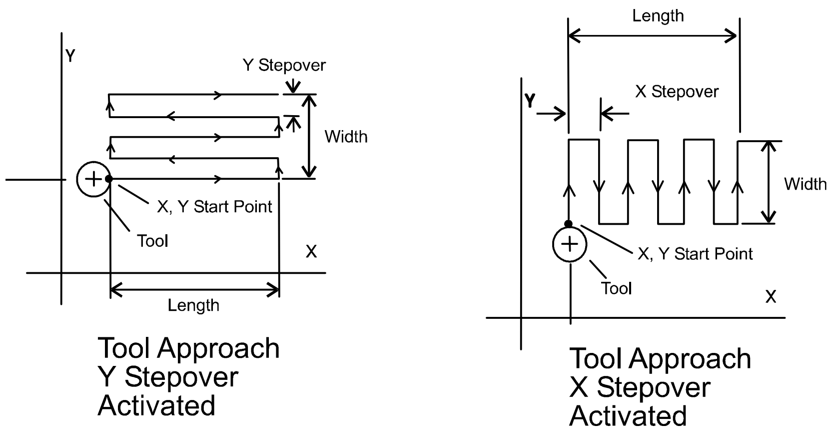


Figure 5-13, Face Cycle Tool Approach

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

At the end of the cycle, the tool rapids to **H**, then rapids back to **D** and **E** (start position).

All rights reserved. Subject to change without notice.
25-January-08

Refer to Figure 7-5, Face Mill Cycle Screen. **Table 5-23** describes the **FACE MILL** entry fields.

Table 5-23, G170 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 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.
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.
Feed	**F**	Feedrate used in cycle.
XStart	**D**	X coordinate of the starting point. Defaults to current position. **NOTE:** Type the required absolute **XStart** and **YStart** coordinates when possible.
YStart	**E**	Y coordinate of the starting point. Defaults to current position. **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.

All rights reserved. Subject to change without notice.

Circular Profile Cycle (G171)

Format: G171 Xn Yn Hn Dn Zn An Rn Bn Sn In Jn Kn Pn

The Circular Profile Cycle cleans up the inside or outside profile of an existing circle.

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

The machine feeds into the profile along Ramp #1, cuts the circle to the specified **D** (**Diameter**) 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 illustrated in **Figure 5-14**.

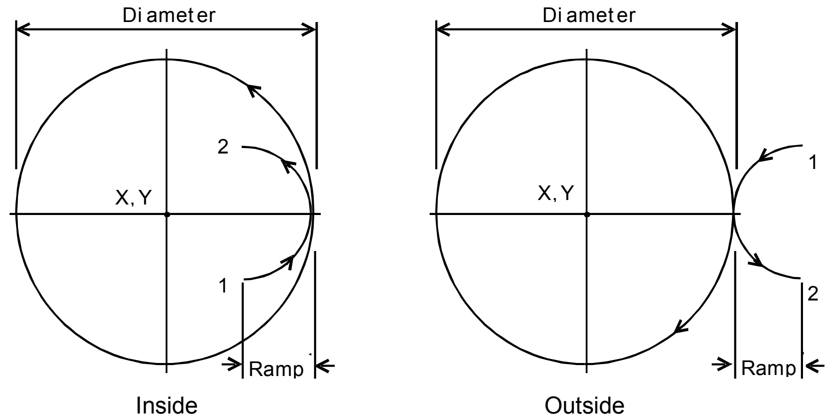


Figure 5-14, Ramp Position for Inside and Outside Profile

The Circular Profile Cycle automatically compensates for tool diameter. Activate the correct tool diameter before the **G171** block.

Refer to Figure 7-6, Circular Profile Cycle Screen. **Table 5-24** describes the **CIRCULAR PROFILE** entry fields.

Table 5-24, G171 Address Words

Label	Address Word	Description
StartHgt	**H**	Z absolute starting (rapid) height (must be 0.1 inch or 2 mm above surface to be cut into). Executed in rapid. Required.
ZDepth	**Z**	Absolute depth of the finished profile. Required.
Diameter	**D**	Finished diameter of circle. If you enter a negative value, both the direction of cut and the starting and endpoints reverse. Required.
Ramp	**R**	Ramp distance used for each pass. Required.

(Continued…)

All rights reserved. Subject to change without notice.

Table 5-24, G171 Address Words (Continued)

Label	Address Word	Description
XCenter	**X**	X coordinate of the center. Default: Present position.
YCenter	**Y**	Y coordinate of the center. Default: Present position.
DepthCut	**B**	Z-axis increment used for each pass
ZFeed	**I**	Z-axis feedrate
RoughFeed	**J**	Rough-pass feedrate
FinFeed	**K**	Finish-pass feedrate
FinStock	**S**	Amount of stock left by the machine before the finish pass. Default: 0. Enter a negative value to leave the stock without making a finish pass.
Side	**A**	Setting for cutting on the inside of the profile (**In**) or the outside (**Out**). Selection required. 0=In, 1=Out.
RetractHgt	**P**	Retract height

If you enter a Depth Cut = **B** (**DepthCut**), the CNC executes the number of passes required to get from the **H** (**StartHgt**) to **Z** (**ZDepth**), cutting to the Depth Cut = **B** (Z Max.cut) on each pass.

When you enter an **S** (**FinStock**) value, the CNC leaves the specified stock on the profile and depth for a finish pass. The CNC finishes to the entered diameter on the finish pass. Enter a negative **S** (**FinStock**) to leave the finish stock without making a finish pass.

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

All rights reserved. Subject to change without notice.

Rectangular Profile Cycle (G172)

Format: G172 Mn Wn Hn Zn Rn An Xn Yn Un In Jn Kn Bn Sn Pn

The Rectangular Profile Cycle cleans up the inside or outside profile of a rectangle. When run, the CNC rapids to the Ramp #1 starting position, rapids to H (Z StartHgt), and then feeds to the depth of the first cut.

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

When cutting an inside profile, the Graphic Menu displays ramp moves.

When cutting an outside profile, the tool ramps into the profile along Ramp #1 and away from the profile along Ramp #2, as illustrated in **Figure 5-15**.

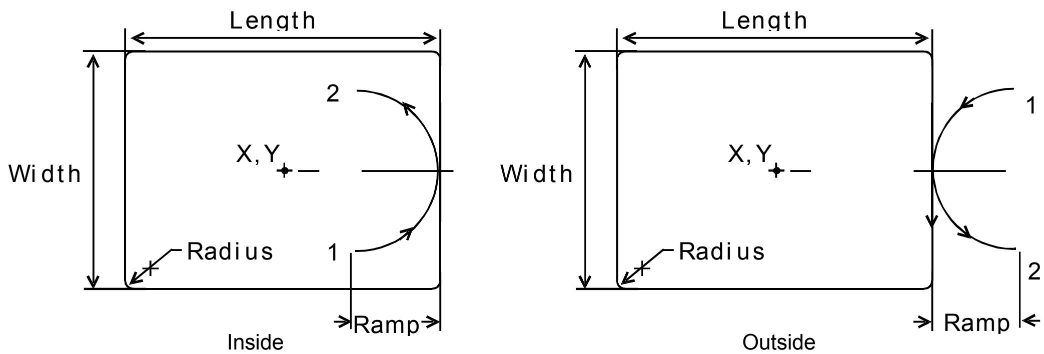


Figure 5-15, Inside and Outside Profile Ramp Moves

The Rectangular Profile Cycle automatically compensates for tool diameter. Activate the correct tool diameter before the **G172** block.

Refer to Figure 7-7, Rectangular Profile Cycle Screen. Refer to **Table 5-25**.

Table 5-25, G172 Address Words

Label	Address Word	Description
Length	**M**	Finished length of rectangle (Required)
Width	**W**	Finished width of rectangle (Required)
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)
ZDepth	**Z**	Absolute depth of the finished profile (Required)
Ramp	**R**	Radius of the ramping moves (Required)
Side	**A**	Setting for cutting on the inside of the profile (**In**) or the outside (**Out**). (Required) 0=Inside, 1=Outside

(Continued…)

All rights reserved. Subject to change without notice.

Table 5-25, G172 Address Words (Continued)

Label	Address Word	Description
XCenter	**X**	X coordinate of the center. If no coordinate is entered, the CNC centers the profile at its present position.
YCenter	**Y**	Y coordinate of the center. If no coordinate is entered, the CNC centers the profile at its present position.
CornerRad	**U**	Corner radius setting. If the programmer enters a negative value, both direction of cut and the starting and endpoints reverse.
ZFeed	**I**	Z-axis feedrate
RoughFeed	**J**	Rough-pass feedrate
FinFeed	**K**	Finish-pass feedrate
DepthCut	**B**	Maximum Z-axis increment used for each pass.
FinStock	**S**	Amount of stock left by the machine before the finish pass. Default: 0. If the programmer enters a negative value, the CNC will leave the stock without making a finish pass.
RetractHgt	**P**	Retract Height

When you enter a value, the CNC executes the number of passes required to get from the **H** (**StartHgt**) to the **Z** (**ZDepth**), cutting the **B** (**DepthCut**) on each pass.

When you enter an **S** (**FinStock**) value, the CNC leaves the specified stock on the profile and depth for a finish pass. The CNC cuts the rectangle to the **M** (**Length**), **W** (**Width**), and **Z** (**ZDepth**) dimensions on the finish pass. Enter a negative **S** (**FinStock**) to leave the finish stock without making a finish pass.

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

All rights reserved. Subject to change without notice.

Mill Cycle (G175)

Format: G175 Xn Yn Hn Zn Bn Dn In Jn Kn Sn

The Mill Cycle (**G175**) 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 **ToolComp** "**D**" parameter is used) rapid to the start height and then feed to the **ZDepth** (**Z**) or **DepthCut** (**B**) using the **ZFeed** (**I**). Subsequent milling blocks are then executed using the **ToolComp** (**D**) parameter and Feed specified. The feedrate can be changed in the blocks that are being milled. Cutter Compensation cannot be changed from within the cycle. The cycle is terminated with the **EndMill** (**G176**) block; at which point, it rapids up to the **StartHgt** (**H**) and returns to the un-comped **XStart** (**X**) YStart (**Y**) location. Activate a tool prior to **G175** so the CNC knows the tool diameter.

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.

Refer to Figure 7-8, Mill Cycle Screen. Refer to **Table 5-26**.

Table 5-26, G175 Address Words

Label	Address Word	Description
XStart	**X**	X coordinate of the center. If no coordinate is provided, default is set to the present position. (Uncompensated) (Required)
YStart	**Y**	Y coordinate of the center. If no coordinate is provided, default is set to present position. (Uncompensated) (Required)
StartHgt	**H**	Absolute Z position to which the CNC rapids before feeding into the workpiece. (Required)
ZDepth	**Z**	The absolute depth of the finished pocket. (Required)
DepthCut	**B**	Z-axis increment used for each pass.
ToolComp	**D**	Tool Compensation. Use **41** or **42** only. All other numbers are no compensation. **41** Compensation LEFT **42** Compensation RIGHT
ZFeed	**I**	Z-axis feedrate (plunging feedrate)
RoughFeed	**J**	XY axes roughing feedrate. Defaults to last programmed feedrate.

All rights reserved. Subject to change without notice.
25-January-08

Table 5-26, G175 Address Words (Continued)

Label	Address Word	Description
FinFeed	**K**	XY axes finish feedrate. Defaults to last programmed feedrate.
FinStock	**S**	Finish-stock amount per side (including bottom). If you enter a negative value, stock will be left, but no finish pass will occur. If not programmed, no finish stock is left.

In G-Code, the mill cycle starts with **G175** and ends with **G176** as in the example below.

```
G70 G90 G0 G17
M5
G28 Z0
G120 X.1 Y1.1 Z.0 I-1.1 J-.1 K-.25
T1 M6
G90 G0 X-.5 Y.5
G175 X-.5 Y.5 H.1 Z-.25 B.125 D41 I25 J35 K45 S.01
G91 G1 X-.5
Y-.5
X1
Y1
X-1
Y-.5
G176 X-.5 Y.5
G90 G0 Z1
X0 Y0
M2
```

EndMill Cycle (G176)

Format: G176 Xn Yn

The mill cycle is terminated with the EndMill (**G176**) block; at which point, it rapids up to the **StartHgt** and returns to the uncompensated X and Yours truly, location. Refer to Figure 7-9, EndMill Cycle Screen. Refer to **Table 5-27**.

Table 5-27, G176 Address Words

Label	Address Word	Description
X	**X**	X ending point. Default: Path ends at the starting point. (Uncompensated)
Y	**Y**	Y ending point. Default: Path ends at the starting point. (Uncompensated)

All rights reserved. Subject to change without notice.

Thread Mill Cycle (G181)

Format: G181 Zn Hn Pn Dn Cn Bn Xn Yn Rn Sn Jn Kn En

WARNING: The first move in this cycle is a rapid move to the center of the thread before moving the Z axis. Make sure the tool is properly located before calling up this cycle.

Use the thread milling for cutting inside or outside threads. It will cut either Inch or MM, left or right hand, and Z movement up or down. A single tooth or multi-toothed tool may be used. Start can be at the top or bottom of the hole or boss. The tools are set, as you would normally set TLO.

Programming the Thread Mill Cycle

To program the Thread Mill Cycle:

1. In Edit mode, press **Help** (**F1**), then select **THREAD MILL CYCLE** display the G181 Thread Mill Cycle menu (refer to Figure 7-10, Thread Mill Cycle Screen).
2. Complete the entry fields (refer to **Table 5-28**), and press **USE** (**F10**).

Table 5-28 describes the Thread Mill Cycle entry fields.

Table 5-28, G181 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)

(Continued…)

All rights reserved. Subject to change without notice.

Table 5-28, G181 Address Words (Continued)

Label	Address Word	Description
TPIor Lead	**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.
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.
ArcInRad	**R**	Size of radius arcing into start of thread. **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	**S**	Amount to leave for a finish pass after the roughing passes.
RoughFeed	**J**	Feedrate for roughing. (If not set (blank), the cycle will use the current active feedrate.)
FinFeed	**K**	Feedrate for the finish pass. (If not set (blank), the cycle will use the current active feedrate.)
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.

Tool Length Offset is set the same as with any other tool or operation. A tool diameter also has to be set in the tool table, as cutter compensation is built into this cycle (cutter compensation is not allowed during the use of this cycle).

If **X** (**XCenter**) and **Y** (**YCenter**) are not programmed, position tool center of the thread before the **G181** line:

- **X** and **Y** will rapid to the starting position of the thread.
- Z will rapid to the safe height specified in **P** (**ZSafePosn**).

All rights reserved. Subject to change without notice.

- The Z-axis will feed down to the start cut position **H** (**ZStar**t). This could be above or below the Z position specified in the **Z** (**ZFinish**) finish position.
- Depending on what is in the **R** (**ArcInRad**) parameter the tool will arc into the first cut position.
- Spiral up or down, depending on the difference between "**Z**" and "**H**" and go counterclockwise or clockwise depending if **D** (**MajorDia**) is plus or minus.
- Then arc-out and feed to the thread center for inside threads or a safe distance away from the thread for outside threads depending on the value in **R** (**ArcInRad**).
- Then feed back to the "**H**" height.
- Then feed X and Y to the next depth of cut. The depth of each roughing pass will be the thread depth specified in the **C** (**ThdDepth**) parameter minus the stock amount specified in the **S** (**StockAmt**) parameter, divided by the number of roughing passes specified in the E (Passes) parameter.
- The cycle repeats this process until the final finish pass.
- It will then cut the thread at the full thread major diameter.

Sample Thread Mill Cycle Program

```
1    G0 G90 G70 G17
2    T1 M6
3    S2000 M3
4    X0 Y0
5    G181 Z-1. H0.1 P.5 D1. C.0625 B8. R.1 S.002 E2 J20.0 K5.0
6    Z5
7    M5
8    M2
```

With a cutter diameter of 0.625, this program will cut a 1-8 inside thread at X0 Y0. The tool will spiral down the thread pitch of 8 threads per inch, finishing at a depth of –1. The starting height is 0.1, the safe rapid **Z** (**ZFinish**) height is 0.5, the major thread diameter is 1 inch, and depth of thread is 0.0625. The arc-in radius is 0.1 and the stock amount for the finish pass is 0.002. The rough feedrate is 20.0 and the finish feedrate is 5.0.

NOTE: If you would like all non-cutting positioning moves to be rapid, set **E** (**Passes**) to a negative number. The idea is to initially set "**E**" as a positive number and after proving out the program, change "**E**" to a negative number for faster production. If you only need one pass to size and you want the positioning moves to be rapid, set "**E**" to -1.

All rights reserved. Subject to change without notice.

Plunge Circular Pocket Cycle (G177)

Format: G177 Zn Hn Dn Xn Yn Bn In Jn Kn Sn Pn An

Use the plunge circular pocket cycle (**G177**) for carbide tooling, when a multiple-axis ramp-in move is not possible. The Z-axis will plunge (single-axis) to programmed depths. You must drill a start hole prior to using this cycle. Activate the tool prior to **G177** so that tool diameter is known. The tool is not required to be at the center of the pocket, as the cycle has variable words for **X** and **Y** center. If you do not program **X** and **Y** variable words with **G177**, then the CNC will use the current position as the pocket center. Refer to Figure 7-15, Plunge Circ Pocket Cycle Screen. Refer to **Table 5-29**.

Table 5-29, G177 Address Words

Label	Address Word	Description
Z Depth (abs)	**Z**	Absolute depth of pocket (Required)
Z Start Height	**H**	The Absolute Z position before beginning to mill the pocket. This must be 0.1 inch (or 2 mm) above the surface. Executed in rapid. (Required)
Diameter	**D**	Diameter of pocket: +**D** dimension = climb (CCW) -**D** dimension = conventional (CW) (Required)
Center X	**X**	Center of the pocket in X-axis. Defaults to current position.
Center Y	**Y**	Center of the pocket in Y-axis. Defaults to current position.
Z Max Cut	**B**	Maximum Z depth per pass (For example, if **Z** = -1, and **B** = 0.5, the pocket will be roughed out in two levels.) **B** is positive. Defaults to tool diameter (depth), less finish stock.
Z Feed Rate	**I**	Z feedrate for plunge move. The tool will feed to the first depth of cut with a plunge move. Defaults to last programmed feedrate.
RoughFeed	**J**	Feedrate of rough cycle used to mill out the pocket. Defaults to last programmed feedrate.
Finish Feed	**K**	Finish-pass feedrate. Defaults to last programmed feedrate.
Finish Stock	**S**	Finish stock amount per side (including bottom). Positive = leave stock and execute finish pass. Negative = leave stock without executing a finish pass. If no value is given, no finish stock is left.
Retract Height	**P**	Z-axis absolute-retract height (must be equal to or above **H**). Executed in rapid. Defaults to **H** (Z Start Height) value.
Stepover	**A**	Maximum tool stepover (must be less than tool diameter). Defaults to tool radius. (If not set, uses 0.5 diameter of the active tool.)

All rights reserved. Subject to change without notice.

You must position the start hole at the center of the pocket prior to **G177** and drill to a sufficient depth.

The required position of the start hole is as follows:

1. For inward to outward cutting (+**A**) (**Stepover**): at the hole center.
2. For outward to inward cutting (-**A**): start hole must be at the 3 o'clock position on the pocket perimeter, less finish stock, less tool radius.
3. Drilled to a sufficient depth.
4. The axes must be positioned over the start hole prior to programming this cycle.
5. If you position the tool at the pocket center and omit XY words from **G177** block, the CNC will use current position as pocket center.

Plunge Rectangular Pocket (G178)

Format: G178 Mn Wn Hn Zn Xn Yn Bn In Jn Kn Sn Pn An Un

Use the plunge rectangular pocket cycle (**G178**) for carbide tooling, where a multiple-axis ramp-in move is not possible. The Z-axis will plunge (single-axis) to the programmed depth. You must drill a start hole previous to the **G178** cycle. Activate a tool prior to programming **G178**, so cutter diameter will be known. Position the tool at the center of the pocket prior to **G178**, or use the **X** and **Y** words. Refer to Figure 7-16, Plunge Rect Pocket Screen. Refer to **Table 5-30**.

Table 5-30, G178 Address Word

Label	Address Word	Description
Length	**M**	Length of pocket in X-axis (Required)
Width	**W**	Width of pocket in Y-axis (Required)
Z Start Height	**H**	Z absolute starting (rapid) height (0.1 inch or 2 mm above surface). Required. Executed in rapid. (Required)
Z Depth (abs)	**Z**	Absolute depth of pocket (Required)
Center X	**X**	X center of pocket.
Center Y	**Y**	Y center of pocket.
Z Max Cut	**B**	Maximum Z depth per pass. (For example, if you program **Z** to be -1, and **B** to be .5, the CNC will rough out the pocket in two levels.) **B** is positive. Defaults to tool diameter (depth), less finish stock.
Ramp Feed	**I**	Z Plunge feed. The tool will plunge to the first depth of cut with a single-axis Z move from the centerline of the lower-left radius. Defaults to last programmed feedrate.
Rough Feed	**J**	After the plunge move, the tool will rough mill the pocket, at feedrate **J**. Defaults to last programmed feedrate.

(Continued…)

All rights reserved. Subject to change without notice.

Table 5-30, G178 Address Word (Continued)

Label	Address Word	Description
Finish Feed	**K**	Finish-pass feedrate. Defaults to last programmed feedrate.
Finish Stock	**S**	Finish stock amount per side (including bottom). Negative = stock will be left, but no finish pass will occur. Positive = leave stock and execute finish pass. If not programmed, no finish stock is left.
Retract Height	**P**	Z-axis absolute finish height (must be equal to or above **H**). Defaults to **H** (Z Start Height value). Executed in rapid. **WARNING: When you cut a pocket inside another pocket, you must set P above the highest pocket. At the end of each pocket, the tool will rapid to P, then rapid to the start position.**
Stepover	**A**	Maximum tool stepover (must be 70% or less of tool diameter). Positive = CCW. Negative = CW. Defaults to half tool diameter.
Corner Radius	**U**	Actual corner radius of pocket (all four corners). Must be equal or greater than tool radius. Defaults to tool radius.

You must position the start hole at the center of the pocket prior to **G178** and drill to a sufficient depth.

All rights reserved. Subject to change without notice.

Engrave Cycle (G190)

Format: G190 A("Text") Hn Zn En Xn Yn Cn Un Vn Fn

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** (the "**H**" parameter). It then feeds to the **ZDepth** (the "**Z**" parameter) specified and begins cutting the **Text** selected.

Programming the Engrave Cycle

To program the Engrave Cycle:

1. In Edit mode, press **Help** (**F1**), then select **G190 Engrave Cycle** and press **ENTER** to display the **G190** Engrave Cycle menu (refer to Figure 7-11, Engrave Cycle Screen).
2. Complete the entry fields (refer to **Table 5-31**), and press **USE** (**F10**).

Table 5-31, G190 Address Word

Label	Address Word	Description
Text	**A**	When the cursor is on Text (the "A" parameter), 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) (maximum 80 characters)
StartHgt	**H**	Z absolute start height. Must be higher than **ZDepth** (the "**Z**" parameter). (Required)
ZDepth	**Z**	Z absolute depth of engraving. Must be below **StartHgt** (the "**H**" parameter). (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)

All rights reserved. Subject to change without notice.

Sample Engrave Cycle Program

```
G90 G70
G0  X0 Y0
T1
X1.0 Y1.0
Z0.1
G190 A("ABCD") H 0.1 Z-.01 E0.5
G0 Z1.0
X0 Y0
M2
```

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.

All rights reserved. Subject to change without notice.

Subprograms

Program repetitive sequences or patterns in a subprogram. Enter subprograms in the program after the end of the main program. Call subprograms from the main program. Refer to **Table 5-32**.

Table 5-32, Subprogram Addresses

M98 Pn	Jump to subprogram.
M99	Return to main program.
Oxxxx	Subprogram label. Up to 4 digits.
Pn	Subprogram number to jump to.

A subprogram can use any code or move type. For example, to cut a contour twice (one rough pass and one finish pass), program it as a subprogram. You can call the subprogram from the main program as many times as required, but you enter the parameters only once.

Subprogram(s) must be stored in the same file as the main program that calls them.

The following topics are described:

All rights reserved. Subject to change without notice.
25-January-08

Subprogram Addresses

Examples:

M98 P2000 commands a jump to subprogram O2000.

Following the program number, blocks in a subprogram are numbered as in normal programming, as in the following example:

```
N2000  O2000  * SUBPROGRAM  #2000
N2001 * blocks in program
N2002
N2003  etc.
```

You can store subprograms anywhere in the program after the main program. They do not have to be entered in numerical order or begin on any specific block number. Refer to **Table 5-33**.

Table 5-33, Subprogram Called from a Main Program

Main Program	
N1 O3 *SUB-EX1	
N2	
N3 M98 P100	Jump to N67 to execute subprogram 100.
N4	
N5	
N6	
N7	
N8 M02	
Subprogram	
N67 O100	CNC jumps to here at N3, completes subprogram until it reaches M99 (N71), and then returns to the main program at N4.
N68	
N69	
N70	
N71 M99	

Repetition of Subprogram (Loop)

Format: M98 Pxxx Lxx

L is the number of repetitions of the subprogram.

Example:

M98 P2000 L12

The block commands twelve repetitions of subprogram number 2000. The maximum number of repetitions is 9999.

All rights reserved. Subject to change without notice.

Calling a Subprogram from a Subprogram

Calling a subprogram from another subprogram is referred to as nesting. The maximum number of programs that can be nested is ten.

The method of calling an additional subprogram is similar to calling the first. Refer to **Table 5-34**.

Table 5-34, Nesting Subprograms

Main Program	Flow of Program During Call of Additional Subprogram
N1 O9 *SUB-EX2	
N2	
N3	
N4 M98 P101	Jump to 1st subprogram N501 from main program at N4.
N5	Return from 1st subprogram.
N6	
N7	
N8 M02	Return to N1 after all subprograms are complete.
1st Subprogram	
N501 O101	
N502	
N503	
N504	
N505 M98 P200	Jump from 1st subprogram to 2nd subprogram occurs at N505. Executes N600 to M606 (M99).
N506	Return to N506 after 2nd subprogram is completed (M99). Finish 1st subprogram.
N507	
N508	
N509 M99	Return to main program at N5.
2nd Subprogram	
N600 O200	
N601	
N602	
N603	
N604	
N605	
N606 M99	Jump to N506.

All rights reserved. Subject to change without notice.

Example:

Mill out a series of identical slots in a plate. Each slot is 1/2" wide and 0.3750" deep. Slot 1 is programmed in a subprogram. All XY dimensions will be incremental to enable you to position the slot anywhere on the coordinate system. Refer to **Figure 5-16**.

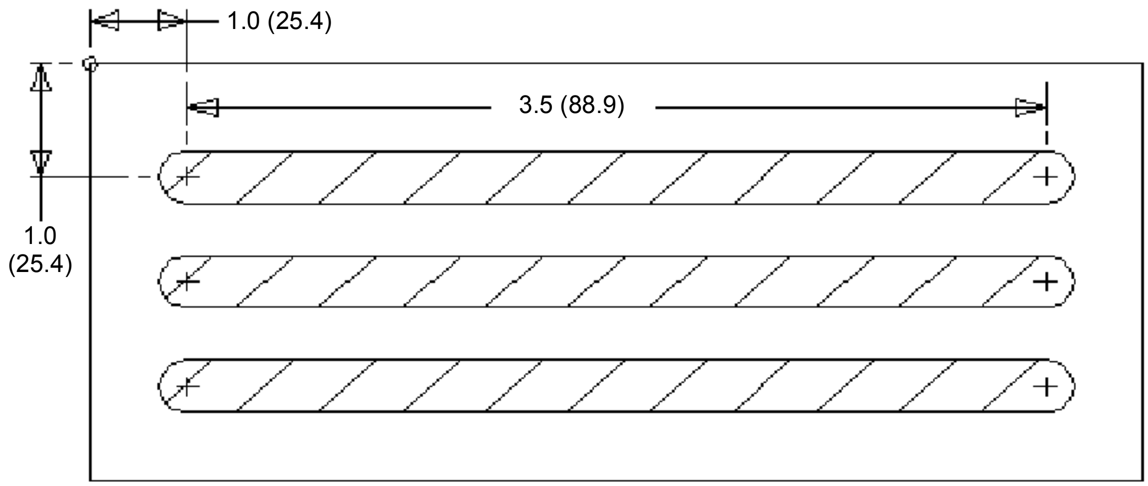


Figure 5-16, Subprogram Programming Example

All rights reserved. Subject to change without notice.

The main program will position the cutter for each slot and call the subprogram that mills out the slots. Subprogram O100 uses incremental values to enable you to position the slot at various positions on the work. For all three slots, you must position the cutter before you call the subprogram. Refer to **Table 5-35**.

Table 5-35, Subprogram Programming Example

Blk #	Block	Description
N1	O12 *SLOTS-MAIN PROGRAM	Define program #12, program name.
N2	G90 G70 (G71) G0 G17 T0 Z0	Set absolute inch, rapid, XY plane, cancel tool, Z0.
N3	X-2 (X-50) Y2 (Y50)	Move to X-2 Y2.
N4	T1 * 1/2" MILL	Activate Tool #1.
N5	X1 (X25.4) Y-1 (Y-25.4) Z.1 (Z2.54) M98 P100	Move to slot location #1 and call sub.
N6	Y-2 (Y-50.8) M98 P100	Move to slot 2 and calls sub.
N7	Y-3 (Y-76.2) M98 P100	Move to slot 3 and calls sub.
N8	T0 Z0	Cancel tool offset and raises Z.
N9	X-2 (X50.8) Y2 (Y50.8)	Move to X-2 Y2.
N10	M02	End program, reset to N1.
N11		
N12	O100 *SLOTS-SUBPROGRAM	Define this as program #100, and gives name.
N13	G90 G1 Z-.375 (Z-9.53) F3.5 (F89)	Feed Z to -.3750" in absolute.
N14	G91 X3.5 (X88.9) F10 (F254)	Feed X 3.5" incrementally.
N15	G90 G0 Z.1 (Z2)	Rapid Z to 0.1" absolute.
N16	G91 X-3.5 (X-88.9)	Rapid X-3.5000", return to start point.
N17	G90 M99	Set Absolute Mode, end sub, return to main.

All rights reserved. Subject to change without notice.
25-January-08

End of Subprogram (M99) with a P-Code

M99 Pxxx

When the End of Subprogram (**M99**) command contains a P-Code, the P-Code refers to the block number in the main program to which the subprogram returns.

Example:

M99 P70

At N30, the CNC will execute the subprogram and then return to N70 (skipping N40 to N70) in the main program. From N70 it will resume main program execution.

The subprogram will return the program to N70 of the main program, skipping N40 through N60. Refer to **Table 5-36**.

Table 5-36, M99 P-Code Usage

Main Program	
N10	
N20	
N30 M98 P100	Call subprogram.
N40	
N50	
N60	
N70	
N80	
N90 M2	
Subprogram	
N110 O100	
N120	
N130	
N140 M99 P70	After complete subprogram, return to N70 in main.

Subprogram for Multiple Parts Programming

To set up a subprogram to machine multiple parts, follow this method.

In this example, a table has two vises installed. Each table holds a part of identical configuration. The same tool does all the work.

1. Program the machining sequence as a subprogram in Absolute or Incremental Mode.
2. When the sequence is finished on Part #1, program a coordinate shift (**G92** or **G53**) and recall the subprogram.
3. If you program the subprogram incrementally, you do not have to shift the zero point. If programmed in absolute, then use a coordinate (zero) shift.

All rights reserved. Subject to change without notice.

Loop Function

In some cases, it is simpler to command a program block or series of blocks to loop (repeat), rather than to program the block(s) several times.

Format: N680 LOOP nnnn

```
N685 .
      .
      .
N695 END
```

LOOP instructs the CNC to execute the blocks following block N680 until **END**. The block is repeated nnnn times. Subprogram calls, axes moves, M-Codes, etc. are all available within a "loop".

Each **LOOP** must have an **END**. Nesting loops (one loop inside of another) is possible, to 10 levels deep.

The full body of the LOOP command (**LOOP** to **END**) must be contained in the main program or the subprogram in which it was initiated.

Example:

In the following program example, M and S codes are omitted. The program assumes that a manual tool change machine is used (no ATC). Check your machine tool manual for details on programming M, S, and T codes. Refer to **Figure 5-17** and Table 5-37, Loop Programming Example.

Tooling to be used:

Tool 1 = #4 centerdrill

Tool 2 = 1/4" diameter twist drill

Tool 3 = 3/8" diameter end mill

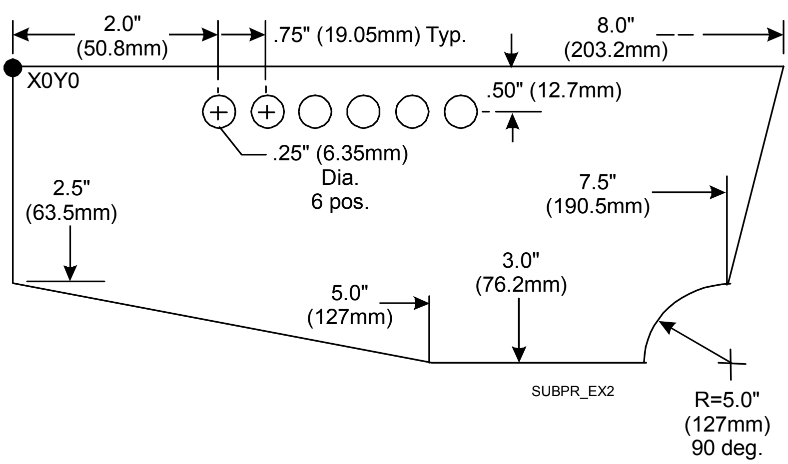


Figure 5-17, Loop Programming Example

All rights reserved. Subject to change without notice.

Table 5-37, Loop Programming Example

Blk. #	Block	Description
N1	O100 * EXAMPLE	Program name and number.
N2	G90 G70 (G71) G0 T0 Z0	Set modes. Cancel tool. Rapid to Z0.
N3	X-2 (X-50) Y2 (Y50)	Rapid to tool change position.
N4	T1 * #4 CTR-DRL (6.35 CTR-DRL)	Activate tool 1, centerdrill.
N5	G81 Z-.23 (Z-.5.84) R.1 (R2) F8 (F203)	Activate spot drill cycle 1.
N6	M98 P1	Call subprogram 1.
N7	T2 * 1/4" DRL (* 6.35 DRILL)	Activate tool 2, twist drill.
N8	G83 Z-.45 (Z-11.43) R.1 (R2) F10 (F254) I.15 (I3.81)	Activate peck drill cycle.
N9	M98 P1	Call subprogram 1.
N10	T3 * 3/8" MILL (* 9.525 MILL)	Activate tool 3, end mill.
N11	G41 X-.3 (X-76.2) Y0	Activate cutter compensation. Feed to XY position.
N12	Z.1 (Z2)	Retract move in Z.
N13	G1 Z-.38 (Z-9.65) F13 (F330)	Feed to cutting depth.
N14	X8 (X203.2)	Cut top of part.
N15	X7.5 (X190.5) Y-2.5 (Y-63.5)	Cut right side of part (vectored path).
N16	G3 X7 (X177.8) Y-3 (Y-76.2) I0 J-.5 (J-12.7)	Activate circular interpolation. Make arc move.
N17	G1 X5 (X127)	Feed to X position (bottom of part).
N18	X0 Y-2.5 (Y-63.5)	Return to start position (cut left side of part).
N19	Y.3 (Y8)	Move off part in Y.
N20	G0 Z.1 (Z2)	Retract move in Z.
N21	G40 X-.3 (X-8)	Cancel cutter compensation.
N22	T0 Z0	Cancel tool offsets and tools. Retract to Z home.
N23	X-2 (X-50) Y2 (Y50)	Move off in X.
N24	M2	End Main.
N25		
N26		
N27	O1 * HOLE LOCATIONS SUB.	Subprogram.

(Continued…)

All rights reserved. Subject to change without notice.

Table 5-37, Loop Programming Example (Continued)

Blk. #	Block	Description
N28	G90 G0 X2 (X50.80) Y-.5 9Y-12.7)	Activate Absolute and Rapid Modes. Move to first hole location.
N29	LOOP 5	Repeat following moves 5 times.
N30	G91 X.75 (X19.05)	Distance between holes.
N31	END	End of loop.
N32	G80 G90 T0 Z0	Cancel drill cycle. Activate Absolute, Raise Z.
N33	X-2 (X-50) Y2 (Y50)	Rapid to tool change position.
N34	M99	Return to main program.

All rights reserved. Subject to change without notice.

Probing Cycles

This section describes operation and an overview of the tool and spindle probe canned cycles available on the 6000i CNC products. The cycles provided perform the most common tool and spindle probing functions. Custom cycles to perform specific functions can be written using the **G31** primitive and parametric programming. Refer to "Section 4, Probe Move (G31)" for more details. If Probing has been added post-sale, beside Machine Parameter changes, there may 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 does not allow rotation, scaling, and mirroring. Plane will be set to **XY G17** when these cycles are complete.

The following topics are described:

- Tool Probe Cycle
- Spindle Probe Cycle

All rights reserved. Subject to change without notice.

Tool Probe Cycles

Before using your tool probe and tool probe cycles, you must setup the probe following the probe manufacturer's specifications.

This section covers the following topics:

- Tool Probe G-Code Cycle Designations
- Description of Tool Probe Cycles

For more probing system parameter setup information, refer to the *6000i CNC Technical Manual*, P/N 627787-21.

The probing parameters can be found on the control by going into the machine configuration as follows (refer to **Figure 5-18**):

1. From the Manual mode, press **SHIFT** then **F3** (**Config**).
2. When asked for a password, simply press the **ENTER** key.
3. Go into System>Probing>CfgProbingParameters
4. Remember that all numeric values are in metric.

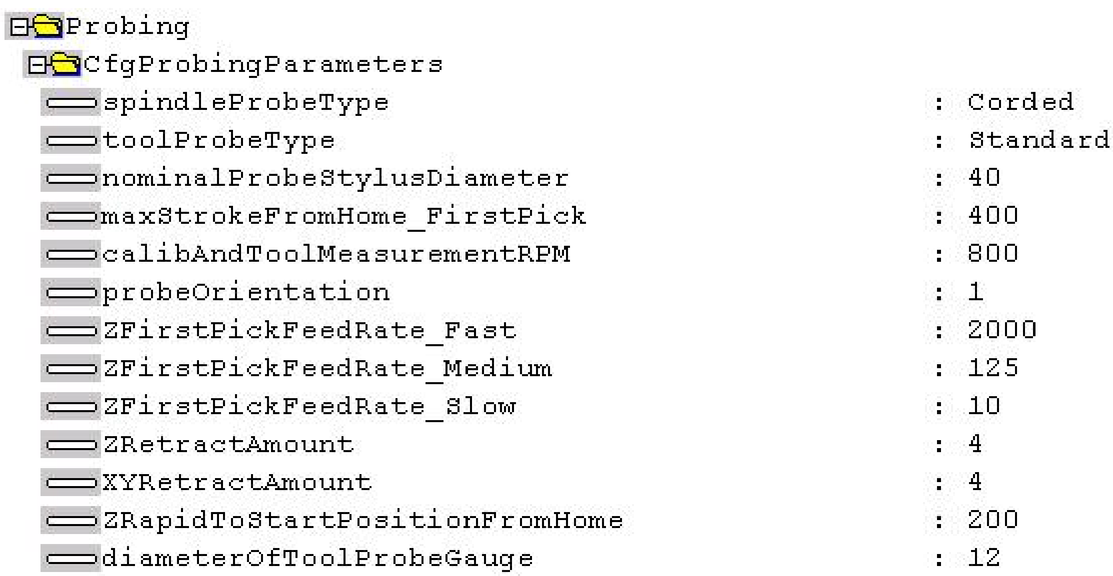


Figure 5-18, Config Data Parameter Screen Capture

- Set **toolProbeType**, Standard or Laser (Standard is the default).
- Set **nominalProbeStylusDiameter**, the overall nominal probe stylus diameter. For example, 12mm for the Renishaw®[**1] probe, or for the Heidenhain probe use 40mm. On a laser style probe this value would normally be zero. This is dependent on the probe style and specifications (refer to your probe documentation).

- - - -
**1 Renishaw® is a registered trademark of Renishaw plc.

All rights reserved. Subject to change without notice.

- Set **maxStrokeFromHome_FirstPick**, represents the distance from machine Z home with the shortest tool or the spindle face to just below the probe stylus top as the maximum stroke for the initial probe pick.
- Set **calibAndToolMeasurementRPM**, the spindle RPM for tool touch. [For example, set to 800 for standard probe type or 4000 for laser probe type tool presetter.]
- Set **probeOrientation**, the proper probe orientation. For example, if set to –1, the probe should be installed on the right side of the table pointing toward the left in the –X direction. See **Table 5-38**.

Table 5-38, Probe Orientation Settings

Probe Orientation Setting	Direction
1	Probe is pointing to the right as you are facing the machine in +X direction
-1	Probe is pointing to the left of the machine in the -X direction.
2	Probe is pointing away from you, toward the back of the machine in the +Y direction
-2	Probe is pointing toward you, toward the front of the machine in the -Y direction

- Set **ZFirstPickFeedRate_Fast**, the Z fast feedrate. [For example, set to 2000 mm per minute (mm/min).]

Warning: When using **G151**, the tool will travel down beyond the top of the probe after the probe is tripped. For this reason, make sure that the fast feedrate is not so high as to cause the tool to travel past the probe travel causing damage to the probe. The maximum feedrate that can be used is specific to the machine and may need to be set much lower to prevent damage to the probe.

- Set the **ZFirstPickFeedRate_Medium**, the Z medium feedrate. [For example, set to 125 mm/min.]
- Set **ZFirstPickFeedRate_Slow**, the Z slow federate for the actual probe pick. [For example, set to 10.0 mm/min.]
- Set **ZRetractAmount**. Amount to retract from top of probe after pick. [For example, set to 4.0 mm.]
- Set **XYRetractAmount**. Amount to retract from side of probe after pick. [For example, set to 4.0 mm.]

All rights reserved. Subject to change without notice.

- Set **ZRapidToStartPositionFromHome**. Install the longest tool in the spindle and bring the Z-axis to machine home. With a tape measure, measure the distance from the tool tip to within 13 mm above the top of the probe stylus and enter that number into **ZRapidToStartPositionFromHome**. When using **G151**, this will cause the tool to rapid to this position in the Z-axis before starting the initial probe touch in the Z-axis. This will save time especially if the Z-feed must be set relatively slow to prevent probe over travel after the probe has been tripped.
- Set **diameterOfToolProbeGauge**. The default gauge diameter of the tool calibration standard. **diameterOfToolProbeGauge** can be overwritten by the **D** word in the **G150** cycle. **diameterOfToolProbeGauge** is used in the **G150** calibration only. [For example, set to 12 mm.]

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 or tool length presetting, 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.

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.

All rights reserved. Subject to change without notice.

Tool Probe G-Code Cycle Designations

The following summarizes the cycles available:

G150 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.

G151 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 **G152** for manual length preset or **G153** for manual diameter preset.

G152 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.

G153 Manual Tool Diameter Preset
Updates tool diameter register for irregular shaped tools or tools with a hole in the center of the bottom.

G154 Tool Breakage, Length and Diameter Wear Detection
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 the program will stop.

All rights reserved. Subject to change without notice.

Description of Tool Probe Cycles

This section contains detailed descriptions of the tool probe cycles:

Tool Probe Calibration Cycle (G150)

Format: G150 Dn En

This cycle is used to calibrate the probe. This is used to set the Z datum for length preset, establishing the center of the probe stylus, and the effective probe stylus diameter for setting tool diameter registers. Refer to **Table 5-39**.

Table 5-39, G150 Address Words

Address Word	Description
D	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 the **diameterOfToolProbeGauge** machine setup parameter)
E	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 **E** value is 0.55" (13.97 mm). Without any **E** 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, 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.

All rights reserved. Subject to change without notice.

3. From the manual mode, type **G150 D(n)**, and press the **START** button. Where **D** is the exact diameter of the calibration standard. (For example, G150 D.5)
4. The Z-axis will initially go down and touch the top of the probe stylus at the feedrate specified in the **ZFirstPickFeedRate_Medium** machine setup parameter. Then retouch at the slow feedrate, specified in the **ZFirstPickFeedRate_Slow** machine setup parameter, establishing the zero probe stylus top.
5. Then incrementally rapid up whatever value that is in the **ZRetractAmount** machine setup parameter.
6. The spindle will come on at the RPM specified in the **calibAndToolMeasurementRPM** machine setup parameter and then the calibration standard will move over an incremental amount that is equal to (Half the value entered in the **D** cycle parameter + Half the value entered in the **nominalProbeStylusDiameter** machine setup parameter + The value in the **XYRetractAmount** machine setup parameter). The direction the probe will move over depends on what is placed in the **probeOrientation** 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 **E** 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 **G151** to set your tool-length offsets or tool diameter registers.

All rights reserved. Subject to change without notice.

Tool Length and Diameter Offset Preset (G151)

Format: G151 Tn Dn Qn En Fn Mn Sn Rn

- Each tool must have the length set once before trying to set the diameter. Call this cycle up the first time using Q2 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 (**G151**) can be run from within a program or from the manual mode. Refer to **Table 5-40**.

Table 5-40, G151 Address Words

Address Word	Description
T	Tool number. (Required) With only the **T** 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.
D	This is the rough diameter of the tool. This should be within 0.04" (1.0 mm). (Optional) If the **D** 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 **D** 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 **D** cycle parameter.
Q	This option specifies to measure length, diameter, or both and the appropriate tool registers are updated. (Optional) [Default: **Q1**] **Q0** Measure the diameter only **Q1** Measure the length only **Q2** Measure both length and diameter If **Q** is not set, the cycle will measure the tool length only. If **Q0** or **Q2** are programmed, you must also have a **D** cycle parameter or the control will display an error message.

(Continued…)

All rights reserved. Subject to change without notice.

Table 5-40, G151 Address Words (Continued)

Address Word	Description
E	The distance to go down along the side of the probe stylus when doing a diameter pick. The maximum **E** 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 **E** 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.
F	This is the override for the fast Z feedrate that was set in the machine setup parameter **ZFirstPickFeedRate_Fast**. 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)
M	This is the override for the medium feedrate that was set in the machine setup parameter **ZFirstPickFeedRate_Medium**. This is used for the same reason as the **F** cycle parameter. This can only be set slower. Trying to set this higher will only result in the software using the original feedrate. (Optional)
S	This is the override for the slow feedrate that was set in the machine setup parameter **ZFirstPickFeedRate_Slow**. This is used for the same reason as the **F** cycle parameter. This can only be set slower. Trying to set this higher will only result in the software using the original feedrate. (Optional)
R	This is the override for the RPM that was set in the machine setup parameter **calibAndToolMeasurementRPM**. This is used for the same reason as the **F** 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 automatic tool preset:

1. Install all the tools you wish to set, in the tool changer.
2. Type in:
   G151 T(tool#) D(tool rough diameter) Q2
   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 Manual, or run the program if you have set all the tools up in a program.

All rights reserved. Subject to change without notice.

4. If you have done a single tool in Manual, 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 **G152 Manual Tool Length Measure for Special Tools** for length and **G153 Manual Tool Diameter Measure for Special Tools** for diameter. See Table 5-40, G151 Address Words. 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.

The following examples are described for machining centers with automatic tool changers.

Format: G151 T(tool#)

With **T** cycle parameter only set:

1. The machine will rapid the Z-axis up and pick up the tool designated in the **T** parameter and rapid directly over the center of the probe stylus.
2. The Z-axis will rapid down the distance placed in the **ZRapidToStartPositionFromHome** machine setup parameter, then start feeding down toward the probe for the initial touch at the feedrate that was placed in the **ZFirstPickFeedRate_Fast** machine setup parameter, then will back up and retouch the probe at the feedrate that is in the **ZFirstPickFeedRate_Slow**.
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 Manual, 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.

All rights reserved. Subject to change without notice.

Format: G151 T(tool#) D(tool rough diameter)

With **T** and **D** cycle parameters only set:

1. The machine will rapid the Z-axis up and pick up the tool designated in the **T** cycle parameter and rapid directly over the center of the probe stylus.
2. The Z-axis will rapid down the distance placed in the **ZRapidToStartPositionFromHome** machine setup parameter, then start feeding down toward the probe for the initial touch at the feedrate that was placed in the **ZFirstPickFeedRate_Fast** 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 **probeOrientation** machine setup parameter.
4. The spindle will then come on in reverse at the RPM specified in the **calibAndToolMeasurementRPM** machine setup parameter and retouch the probe twice, once at the feedrate that is in the **ZFirstPickFeedRate_Medium** machine setup parameter and again at the **ZFirstPickFeedRate_Slow** 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 Manual, 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: G151 T(tool#) D(tool rough diameter) Q2

With **T**, **D**, and **Q** cycle parameters set:

1. The machine will rapid the Z-axis up and pick up the tool designated in the **T** cycle parameter and rapid directly over the center of the probe stylus.
2. The Z-axis will rapid down the distance placed in the **ZRapidToStartPositionFromHome** machine setup parameter then start feeding down toward the probe for the initial touch at the feedrate that was placed in the **ZFirstPickFeedRate_Fast** 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 **probeOrientation** machine setup parameter.

All rights reserved. Subject to change without notice.

4. The spindle will then come on counter clockwise at the RPM specified in the **calibAndToolMeasurementRPM** machine setup parameter and retouch the probe twice, once at the feedrate that is in the **ZFirstPickFeedRate_Medium** machine setup parameter and again at the **ZFirstPickFeedRate_Slow** 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 **ZRetractAmount** machine setup parameter. Then it will 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 **probeOrientation** machine setup parameter, before making a guarded move down 0.1" (2.54 mm) or whatever value has been placed in the **E** cycle parameter.
8. The machine will then touch the tool to the probe stylus on two opposite sides at the feedrate specified in the **ZFirstPickFeedRate_Medium** machine setup parameter with the spindle running at the RPM specified in the **calibAndToolMeasurementRPM** machine setup parameter, backing up 0.02" (0.508 mm) after each first touch then retouching at the feedrate specified in the **ZFirstPickFeedRate_Slow** 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 Manual, 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.

All rights reserved. Subject to change without notice.

Manual Tool-Length Measure for Special Tools (G152)

Format: G152 Tn Dn Mn Sn Rn

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 5-41**. for special tools, desc

Table 5-41, G152 Address Word

Address Word	Description
T	Tool number. (Required) With only the **T** 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 **T** parameter must the same as the current tool in the spindle.
D	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)
M	This is the override for the medium feedrate that was set in the machine setup parameter **ZFirstPickFeedRate_Medium**. 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)
S	This is the override for the slow feedrate that was set in the machine setup parameter **ZFirstPickFeedRate_Slow**. This is used for the same reason as the **M** cycle parameter. This can only be set slower. Trying to set this higher will only result in the software using the original feedrate. (Optional)
R	This is the override for the RPM that was set in the machine setup parameter **calibAndToolMeasurementRPM**. This is used for the same reason as the **M** cycle parameter. This can only be set slower. Trying to set this higher will only result in the software using the original RPM. If you would like the cycle to run without the spindle turning, enter a negative value in the "R" parameter. (Optional)

WARNING: Large tools can result in probe damage if the touch feedrate is set too fast. For this reason, the cycle parameters: **M**, **S**, and **R** have been added to enable the programmer/operator to override the values in the machine setup parameters for the specific tool being checked or set.

All rights reserved. Subject to change without notice.

You must have the tool positioned over the probe stylus so the tooth that sticks down the furthest is directly over the center of the probe stylus and above the stylus less then 0.100" (2.0 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 manual mode, type **G152 Tn Dd** and press the **START** button. Where **T** is the tool number and **D** is roughly the diameter of the special tool. (For example, G152 T3 D3.5)
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 setup parameter. 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.

All rights reserved. Subject to change without notice.

Manual Tool Diameter Measure for Special Tools (G153)

Format: G153 Tn Dn En Mn Sn Rn

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 5-42**.

Table 5-42, G153 Address Words

Address Word	Description
T	Tool number. (Required) The **T** cycle parameter must be the same as the current tool in the spindle.
D	This is the rough diameter of the tool. (Required) The diameter specified in this cycle 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 the forward direction.
E	The distance to go down along the side of the probe stylus when doing a diameter pick. The maximum **E** 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 **E** 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.
M	This is the override for the medium feedrate that was set in the machine setup parameter **ZFirstPickFeedRate_Medium**. 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)
S	This is the override for the slow feedrate that was set in the machine setup parameter **ZFirstPickFeedRate_Slow**. This is used for the same reason as the **M** cycle parameter. This can only be set slower. Trying to set this higher will only result in the software using the original feedrate. (Optional)
R	This is the override for the RPM that was set in the machine setup parameter **calibAndToolMeasurementRPM**. This is used for the same reason as the **M** cycle parameter. This can only be set slower. Trying to set this higher will only result in the software using the original RPM. (Optional)

All rights reserved. Subject to change without notice.

WARNING: Large tools can result in probe damage if the touch feedrate is set too fast. For this reason, the cycle parameters: **M**, **S**, and **R** have been added to enable the programmer/operator to override the values in the machine setup 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 **E** if different from 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 manual mode and the spindle off, input:

   "G153 Tn Dn En" and press the **START** button. Where **T** is the tool number, **D** is roughly the diameter of the special tool (this should be larger but not more then 0.100" (2.54 mm) larger), and **E** 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, G153 T3 D3.5 E.25 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 **E** or 0.1" (2.54 mm) if **E** 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 then 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.

All rights reserved. Subject to change without notice.
25-January-08

Tool Breakage, Length, and Diameter Wear Detection (G154)

Format: G154 Tn Dn Kn Jn En Un Mn Sn Rn

Refer to **Table 5-43**.

Table 5-43, G154 Address Word

Address Word	Description
T	Tool number. (Required) The **T** cycle parameter will be the tool number you want checked.
D	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.
K	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, **K** or **J** must be set or the cycle will alarm.
J	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, **K** or **J** must be set or the cycle will alarm.
E	The distance to go down along the side of the probe stylus when doing a diameter check. The maximum **E** 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 **E** 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.
U	If this is undefined or set to 0, the **G154** cycle will not update the diameter or length wear register each time it checks a tool. If set to one, the cycle will update the wear registers. In both cases, the control will alarm when the maximum limit set in **K** or **J** has been exceeded. (Optional)

(Continued…)

All rights reserved. Subject to change without notice.

Table 5-43, G154 Address Word (Continued)

Address Word	Description
M	This is the override for the medium feedrate that was set in the machine setup parameter **ZFirstPickFeedRate_Medium**. 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)
S	This is the override for the slow feedrate that was set in the machine setup parameter **ZFirstPickFeedRate_Slow**. This is used for the same reason as the **M** cycle parameter. This can only be set slower. Trying to set this higher will only result in the software using the original feedrate. (Optional)
R	This is the override for the RPM that was set in the machine setup parameter **calibAndToolMeasurementRPM**. This is used for the same reason as the **M** 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 cycle parameters: **M**, **S**, and **R** have been added to enable the programmer/operator to override the values in the machine setup 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. **G150 Calibration** and **G151 Automatic Tool Length and Diameter set**, or **G152 Manual Tool Length Measure for Special Tools** and **G153 Manual Tool Diameter Measure for Special Tools,** must be run first before using the **G154** cycle.

The **G154** 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 **E** 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 **Tn M6** command, use:

G154 Tn Dn Kn Jn En Un" 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 "T(Tool#) M6" after this cycle has been run.

All rights reserved. Subject to change without notice.

Spindle Probe Cycles

This section describes operation and an overview of the spindle probing cycles available in 6000i CNCs. It is designed to assist in part setup.

Before using your spindle probe for part setup, you must set the probe up according to the probe manufacturer's specification so that 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.

Rotation, mirroring and scaling is not allowed while running these cycles. Plane will be set to **XY G17** when these cycles are complete.

This section contains the following topics:

- **Spindle Probe G-Code Cycle Designations**
- Canned Cycle Parameter Settings
- Description of Spindle Probe Cycles

Spindle Probe G-Code Cycle Designations

The following summarizes the cycles available:

G140 **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 **G140** 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. In this case the accuracy of the spindle probe is only as good as the stylus concentricity to the spindle. 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.

G141 **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.

G142 **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.

G143 **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.

All rights reserved. Subject to change without notice.

G144 **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.

G145 **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.

G146 **Protected Positioning Move**
This cycle allows for safe positioning of the probe around the part and will generate an alarm and stop the program if an obstruction is encountered.

G147 **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 the **SkewComp** at the same time as it is measured or in a subsequent call at another place in the program without measuring again by using **Q2**.

Canned Cycle Parameter Settings

Before you set the cycle parameters for the Spindle probe you must:

- Know the exact diameter of the Ring Gauge (calibration standard).
- Know that the Ring Gauge is a standard that is specifically designed for calibrating the probe. The **D** cycle parameter is the diameter of hole that comes in contact with the probe stylus during calibration and should be an exact measurement.
- 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).

The probing parameters can be found on the control by going into the machine configuration as follows (refer to **Figure 5-19**):

1. From the Manual mode, press **SHIFT** then **F3** (**Config**).
2. When asked for a password, simply press the **ENTER** key.
3. Go into System>Probing>CfgProbingParameters
4. Remember that all numeric values are in metric.

All rights reserved. Subject to change without notice.
25-January-08

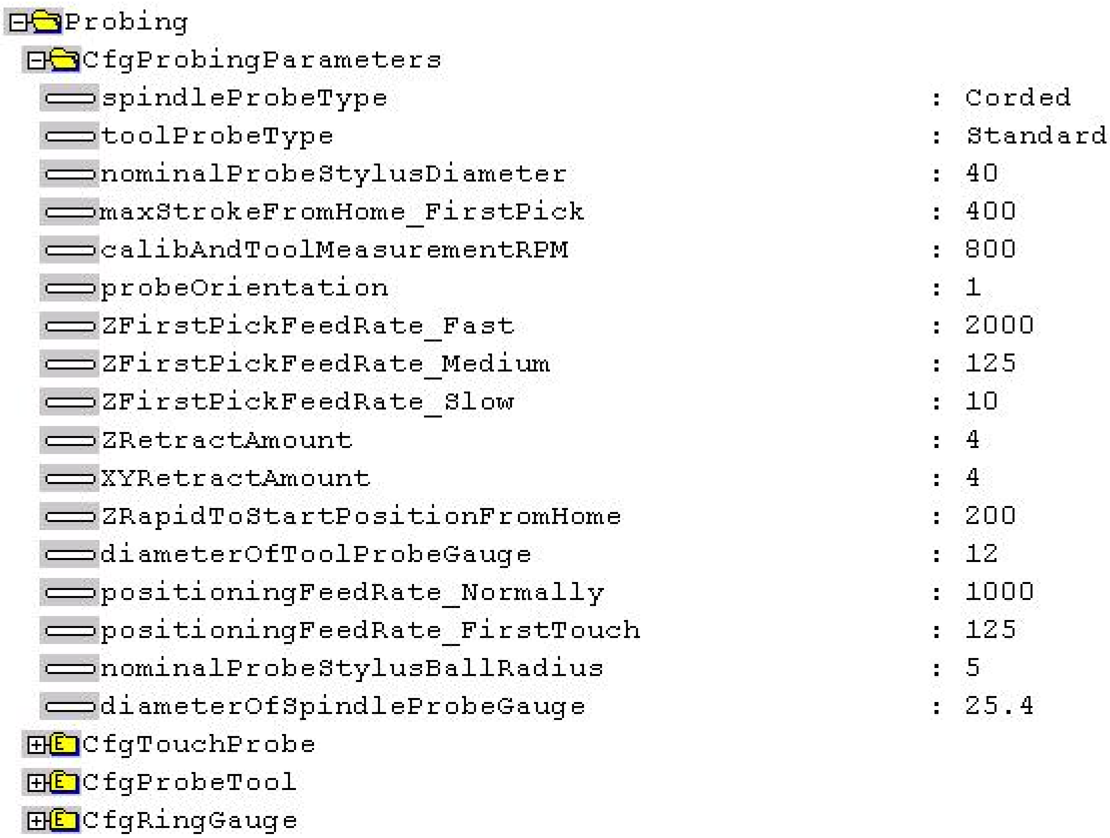


Figure 5-19, Config Data Parameter Screen Capture (Additional Parameters Displayed)

- Set **spindleProbeType**, to Corded, Cordless, or Cordless SG (Strain Gauge) depending on the probe style (refer to *6000i CNC Technical Manual*, P/N 627787-21).
- Set **positioningFeedrate_Normally**, the feedrate the control will use while positioning the probe around the part. [For example, set to 1000 mm/min.]
- Set **positioningFeedrate_FirstTouch**, the feedrate the control will use while making its initial touch finding the surface it is measuring. [For example, 125 mm/min.]
- Set **nominalProbeStylusBallRadius**, the diameter as measured with a micrometer. [For example, set to 5 mm.]
- Set **diameterOfSpindleProbeGauge**, the exact diameter of the ring gauge used to calibrate the spindle probe. [For example, 25.4 mm.]

All rights reserved. Subject to change without notice.

Description of Spindle Probe Cycles

This section contains detailed descriptions of the spindle probe cycles:

- Spindle Probe Calibration (G140)
- Edge Finding (G141)
- Outside Corner Finding (G142)
- Inside Corner Finding (G143)
- Out/Inside Boss/Hole Finding (G144)
- Out/Inside Web Finding (G145)
- Protected Probe Positioning (G146)
- Skew Error Find (G147)

All rights reserved. Subject to change without notice.

Spindle Probe Calibration (G140)

Format: G140 Qn Hn En Vn Dn An Bn

Refer to **Table 5-44**.

Table 5-44, G140 Address Word

Address Word	Description
Q	Set **Q** to **1** if you are calibrating to a boss verses a ring gauge. Otherwise, do not set or set to **0**. Default is: **0**. (Optional)
H	If set to **1**, the cycle will find the top of the part before calibrating the probe. If **Q** parameter is set to **1**, **H** is forced to **1** as well; otherwise, the Default is: **0**. (Optional)
E	The distance to go down from the top of the ring gauge or standing boss for calibration. This is only used if **H** parameter is set to **1**. Without any **E** value, the cycle will bring the probe down past the top of the ring gauge after finding the top, 0.1". Note: If the stylus ball is greater than .2" (5.08 mm), E must be set to at least half the ball diameter. (Optional)
V	The **V** 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)
D	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 **diameterOfSpindleProbeGauge** if needed and <u>should be an exact measurement</u>. (Optional)
A	The distance from the starting point to move in the X-axis to find the top of the gauge. The default, if **Q** is not set or set to **0**, is 0.1" (2.54 mm) beyond the edge of the ring gauge hole. If **Q** is set to **1**, the default is the current probe position. (Optional)
B	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.
2. The Ring Gauge mounted on the machine table.

All rights reserved. Subject to change without notice.

To calibrate the probe:

1. 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.
2. From the manual mode, type **G140** and press **START**.
3. The probe will touch four sides of the inside of the hole. The spindle will rotate 180 degrees (if machine has spindle orientation) 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 or machines that cannot orient the spindle, 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.

All rights reserved. Subject to change without notice.
25-January-08

Edge Finding (G141)

Format: G141 Qn Wn

- 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 "Section 9, Tool-Length Offsets" 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 "Section 9, Tool-Length Offsets" for setting and activating work coordinate offsets.
- The G141 Edge Finding Cycle can be run from within a program or from the manual mode. Refer to **Table 5-45**.

Table 5-45, G141 Address Words

Address Word	Description
Q	Axis and direction to find edge. **0** = X+, **1** = X-, **2** = Y+, **3** = Y-, **4** = Z+, **5** = Z- (Required)
W	Work Coordinate to update with edge location in X- or Y-axes. If set, work coordinate will be updated if 1 through 4 are specified for **Q** or tool-length offset will be set for the current tool number if **Q** is set to **5** or **6**. **NOTE:** Before any tool-length offset is active, you must re-call that tool. Work coordinate register or Tool-length register will not be updated if **W** is not set and a warning message will tell the operator no update has taken place. (Optional)

To use the Edge Finding Cycle:

1. Place the probe in the spindle.
2. Manually jog the probe stylus less then 0.1" (2.54 mm) away from the surface to be found.
3. Type **G141 Qn Wn**. 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 **G146 (Protected Probe Positioning)** cycle (see **G146** instructions later in this document).

4. Execute that line in Manual by pressing **START**.

All rights reserved. Subject to change without notice.

Outside Corner Finding (G142)

Format: G142 Qn Hn En Dn Vn An Bn In Jn Kn Wn

- 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 "Section 9, Tool-Length Offsets" 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 "Section 9, Tool-Length Offsets" for setting and activating work coordinate offsets.
- The G142 Outside Corner Finding Cycle can be run from within a program or from the manual mode. Refer to **Table 5-46**.

Table 5-46, G142 Address Words

Address Word	Description
Q	Quadrant of corner to find. **0** = +,+ (upper right) **1** = -,+ (upper left) **2** = -,- (lower left) **3** = +,- (lower right) (Required)
H	If set to **1**, the cycle will find the top of the part before finding the X & Y corner coordinate. Default is: **0**. If **H** is not set or is set to **0**, the Z-axis must be at the picking depth. If **H** = **1**, 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)
E	The distance to go down from the top of part to find X & Y coordinate of the corner. This is only used if **H** parameter is set to **1**. Without any **E** 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)
D	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)
V	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)
A	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)

(Continued…)

All rights reserved. Subject to change without notice.

Table 5-46, G142 Address Words (Continued)

Address Word	Description
B	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)
I	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)
J	Same as **I** only for the Y-axis. (Optional)
K	Same as **I** only for the Z-axis. (Optional)
W	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 **W** is not set. (Optional)

To use the Outside Corner Finding Cycle:

1. Place the probe in the spindle.
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 **H** = **1**, 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. Type **G142 Qn Wn**. 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 **G146 (Protected Probe Positioning)** cycle (see **G146** instructions later in this document) or use the **I**, **J**, or **K** cycle parameters for the same purpose.

4. Execute that line in Manual by pressing **START**.

All rights reserved. Subject to change without notice.

Inside Corner Finding (G143)

Format: G143 Qn Hn En Dn Vn An Bn In Jn Kn Wn

- 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 "Section 9, Tool-Length Offsets" 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 "Section 9, Tool-Length Offsets" for setting and activating work coordinate offsets.
- The **G143** Inside Corner Finding Cycle can be run from within a program or from the manual mode. Refer to **Table 5-47**.

Table 5-47, G143 Address Words

Address Word	Description
Q	Quadrant of corner to find. **0** = +,+ (upper right) **1** = -,+ (upper left) **2** = -,- (lower left) **3** = +,- (lower right) (Required)
H	If set to **1**, the cycle will find the top of the part before finding the X & Y corner coordinate. Default is: **0**. If **H** is not set or is set to **0**, the Z-axis must be at the picking depth. If **H** = **1**, 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 inside of the corner in X & Y. (Optional)
E	The distance to go down from the top of part to find X & Y coordinate of the corner. This is only used if **H** parameter is set to **1**. Without any **E** 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)
D	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" or 10.16 mm. (Optional)
V	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)
A	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" or 10.16 mm. (Optional)

(Continued…)

All rights reserved. Subject to change without notice.

Table 5-47, G143 Address Words (Continued)

Address Word	Description
B	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" or 10.16 mm. (Optional)
I	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)
J	Same as **I** only for the Y-axis. (Optional)
K	Same as **I** only for the Z-axis. (Optional)
W	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 **W** is not set. (Optional)

To use the Inside Corner Finding Cycle:

1. Place the probe in the spindle.
2. Manually jog the probe stylus 0.1" (2.54 mm) away from the inside of the corner you wish to find in X & Y. If **H** = **1**, 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. Type **G143 Qn Wn**. 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 **G146** (**Protected Probe Positioning**) cycle (see **G146** instructions later in this document) or use the **I**, **J**, or **K** cycle parameters for the same purpose.

4. Execute that line in Manual by pressing **START**.

All rights reserved. Subject to change without notice.

Inside/Outside Boss/Hole Finding (G144)

Format: G144 Qn Xn Yn Hn En Vn An Bn In Jn Kn Rn Wn

- 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 "Section 9, Tool-Length Offsets" 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 "Section 9, Tool-Length Offsets" for setting and activating work coordinate offsets.
- The G144 Inside or Outside Boss/Hole Finding Cycle can be run from within a program or from the manual mode. Refer to **Table 5-48**.

Table 5-48, G144 Address Words

Address Word	Description
Q	Inside or Outside. **0** = Inside Hole **1** = Outside Boss (Required)
X	Estimated length of boss/hole if rectangular or the Diameter if round. (Required)
Y	Estimated width of boss/hole. **Y** is only specified if boss or hole is rectangular in shape. (Optional)
H	If set to **1**, the cycle will find the top of the part before finding center of hole or boss. If **Q** parameter is set to **1**, **H** is forced to **1** as well; otherwise, the Default is: **0**. (Optional)
E	The distance to go down from the top of part to find X & Y coordinate of the center. This is only used if **H** parameter is set to **1**. Without any **E** 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)
V	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)
A	The distance from the starting point to move in the X-axis to find the top of the part. The default, if **Q** is not set or set to **0**, is 0.1" beyond the edge of the boss/hole. If **Q** is set to **1**, the default is the current probe position. (Optional)

(Continued…)

All rights reserved. Subject to change without notice.

Table 5-48, G144 Address Words (Continued)

Address Word	Description
B	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)
I	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)
J	Same as **I** only for the Y-axis. (Optional)
K	Same as **I** only for the Z-axis. (Optional)
W	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 **W** is not set. (Optional)
R	If set to **1**, 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 **0**, the cycle will only measure "X" once for a total of 4 touches. Default is: **0**. (Optional)

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

1. Place the probe in the spindle.
2. Manually jog the probe stylus the approximate center in X & Y within 0.1" (2.54 mm). If **H** = **1**, 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. Type **G144 Qn Xn Yn Wn**. 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 **G146** (**Protected Probe Positioning**) cycle (see **G146** instructions later in this document) or use the **I**, **J**, or **K** cycle parameters for the same purpose.

4. Execute that line in Manual by pressing **START**.

All rights reserved. Subject to change without notice.

Inside/Outside Web Finding (G145)

Format: G145 Qn Xn Yn Hn En Vn An Bn In Jn Kn Wn

- 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 "Section 9, Tool-Length Offsets" 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 "Section 9, Tool-Length Offsets" for setting and activating work coordinate offsets.
- The G145 Inside or Outside Web Finding Cycle can be run from within a program or from the manual mode. Refer to **Table 5-49**.

Table 5-49, G145 Address Words

Address Word	Description
Q	Inside or Outside. **0** = Inside Hole **1** = Outside Boss (Required)
X	Estimated X width of Web if measuring in the X-axis. X or Y must be specified; but only one, not both.
Y	Estimated Y width of Web if measuring in the Y-axis. X or Y must be specified; but only one, not both.
H	If set to **1**, the cycle will find the top of the part before finding center of Web. If **Q** parameter is set to **1**, **H** is forced to **1** as well; otherwise, the Default is: **0**. (Optional)
E	The distance to go down from the top of part to find X or Y coordinate of the center. This is only used if **H** parameter is set to **1**. Without any **E** 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)
V	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)
A	The distance from the starting point to move in the X-axis to find the top of the part. The default, if **Q** is not set or set to **0**, is 0.1" beyond the edge of the web. If **Q** is set to **1**, the default is the current probe position. (Optional)

(Continued…)

All rights reserved. Subject to change without notice.

Table 5-49, G145 Address Words (Continued)

Address Word	Description
B	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)
I	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)
J	Same as **I** only for the Y-axis. (Optional)
K	Same as **I** only for the Z-axis. (Optional)
W	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 **W** is not set. (Optional)

To use the Inside/Outside Web Finding Cycle:

1. Place the probe in the spindle.
2. Manually jog the probe stylus the approximate center in X or Y within 0.1" (2.54 mm). If **H** = **1**, 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. Type **G145 Qn Xn Wn**. 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 **G146** (**Protected Probe Positioning**) cycle (see **G146** instructions later in this document) or use the **I**, **J**, or **K** cycle parameters for the same purpose.

4. Execute that line in Manual by pressing **START**.

All rights reserved. Subject to change without notice.

Protected Probe Positioning (G146)

Format: G146 Xn Yn Zn Fn

- When an X, Y, and/or Z move is programmed using the **G146** (Protected Positioning Cycle), the control will stop the axis travel and program 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 "Section 9, Tool-Length Offsets" 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 "Section 9, Tool-Length Offsets" for setting and activating work coordinate offsets.
- The **G146** Protected Probe Positioning Cycle can be run from within a program or from the manual mode. Refer to **Table 5-50**.

Table 5-50, G146 Address Words

Address Word	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.
F	Feedrate at which to travel to target. **F** 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 **positioningFeedrate_Normally**. (Optional)

To use the Protected Probe Positioning Cycle:

1. Place the probe in the spindle and make sure that its tool and work coordinate offsets are active.
2. Type **G146 Xn Yn Zn Fn**. 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 Manual by pressing **START**.

All rights reserved. Subject to change without notice.

Skew Error Find (G147)

Format: G147 Qn Sn Dn Hn En Vn An Bn In Jn Kn

- G68, axis rotation, cannot be used with **G147**, 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 "Section 9, Tool-Length Offsets" 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 "Section 9, Tool-Length Offsets" 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 **S** parameter as described below or an **I**, **J**, and/or **K** should be included for pre-positioning.
- The **G147** Skew Error Finding Cycle can be run from within a program or from the manual mode. Refer to **Table 5-51**.

Table 5-51, G147 Address Words

Address Word	Description
Q	**Q0** Finds the skew angle, but does not activate skew compensation. **Q1** Finds the skew angle, and activates skew compensation. **Q2** Activates skew compensation with the current skew value, but will not rerun the cycle on the part. **NOTE:** If **Q2** is used, all other **G147** parameters are ignored. **NOTE:** Before using **G147 Q2**, you must have called **G147** at least once with **Q0** or **Q1**, 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 **G147** from the MDI mode. The operator can run the **G147** from MDI but must place **G147 Q2** inside the program for skew compensation to take effect. A G53 work coordinate call will deactivate skew compensation, necessitating a re-issuance of **G147 Q2** to activate skew compensation. Using **Q1** or **Q2** will default the control to **G90** (Absolute). If you are in **G91** (Incremental), you will need to switch back after the cycle has been run. (Optional)

(Continued…)

All rights reserved. Subject to change without notice.

Table 5-51, G147 Address Words (Continued)

Address Word	Description
S	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: **S**=90 would start in the lower-left side, picking in the X positive direction, finding the skew of the left side of the part. **S**=-90 would start in the upper-right side, picking in the X negative direction, finding the skew of the right side of the part. **S**=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 is: **0**. (Optional)
D	The distance from the first pick to the second pick. Default is: 2.0" (50.8 mm) (Optional)
H	If set to **1**, the cycle will find the top of the part before finding part skew angle. Default is: **0**. If **H**" is set to **1**, the probe stylus should be pre-positioned within 0.1" (2.54 mm) above the part. If **H** is set to **0**, the probe stylus should be positioned at the Z-axis depth from which you want to make side picks. (Optional)
E	The distance to go down from the top of part to find part skew angle. This is only used if **H** parameter is set to **1**. Without any **E** 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)
V	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 **S** cycle parameter is relatively accurate, this parameter will not be needed because the default will be good enough. (Optional)
A	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 **S** cycle parameter. (Optional)
B	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 **S** cycle parameter. (Optional)
I	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)

(Continued…)

All rights reserved. Subject to change without notice.

Table 5-51, G147 Address Words (Continued)

Address Word	Description
J	Same as **I** only for the Y-axis. (Optional)
K	Same as **I** only for the Z-axis. (Optional)

To use the Skew Error Finding Cycle:

1. Place the probe in the spindle.
2. Manually jog the probe stylus to the appropriate start position relative to the part as specified by the **S** parameter in Table 5-51, G147 Address Words above. X or Y should be within 0.1" (2.54 mm) of the part edge. If **H** = **1**, 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. Type **G147 Qn Sn**. 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 **G146** (**Protected Probe Positioning**) cycle (see **G146** instructions above in this document) or use the **I**, **J**, or **K** parameters for the same purpose.

4. Execute that line in Manual by pressing **START**.

All rights reserved. Subject to change without notice.

Section 6 - Program Editor

The following topics are described in this section:

Activating the Program Editor

Program blocks are written using the Program Editor. The Program Editor can be activated from the Manual screen, Program Manager, or Draw Graphics.

When you activate the Program Editor, the selected program opens for editing.

The following topics are described:

All rights reserved. Subject to change without notice.

Activating Edit Mode from the Manual Screen

To activate the Edit Mode from the Manual screen:

1. With the appropriate program loaded, press **Edit** (**F3**). The Edit Screen activates. See **Figure 6-1**.

Activating Edit Mode from the Program Manager

To activate the Edit Mode from the Program Manager:

1. Highlight a program in the Program Manager.
2. Press **Edit** (**F8**). The Edit screen activates and **Insert** (**F3**) highlights.

Activating Edit Mode from Draw Graphics

To activate the Edit Mode from Draw Graphics:

1. Press **Edit** (**F2**). The Edit screen activates. Refer to **Figure 6-1**.

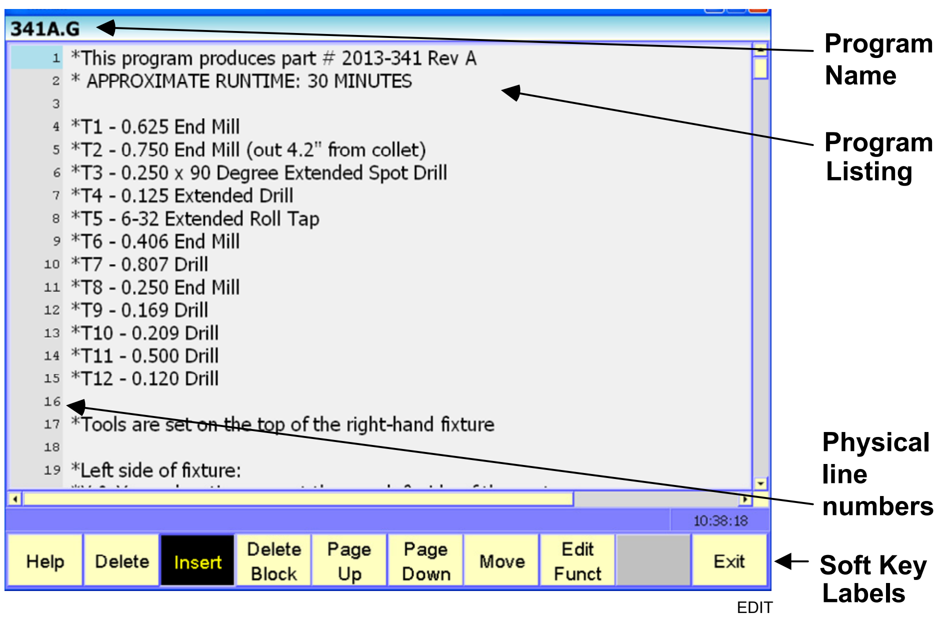


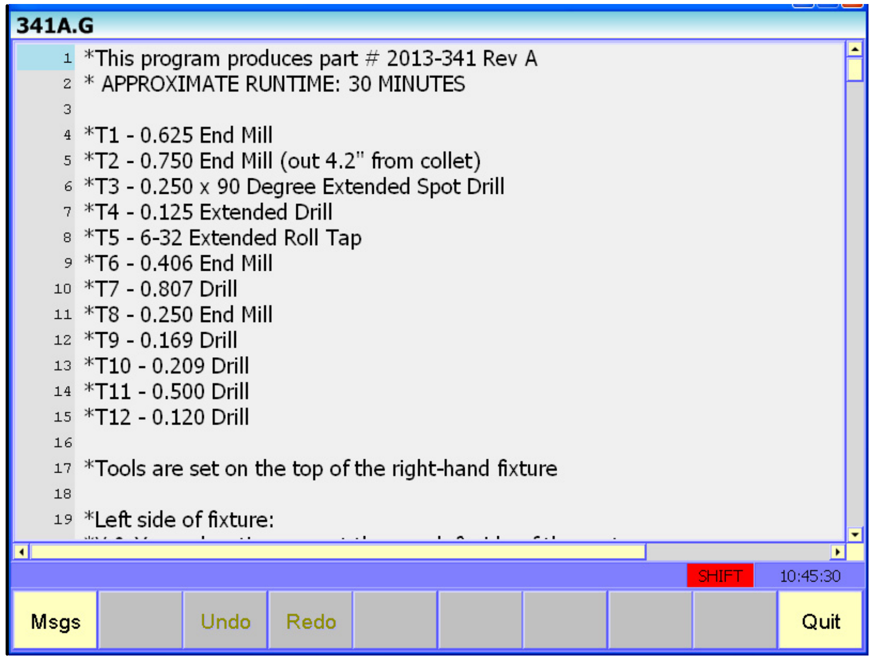
Figure 6-1, Edit Screen

You can write and edit programs from the Edit screen. The Edit screen provides the following areas:

Program Name	The name of the program listed on the screen.
Program Listing	Area of the screen where the program is listed.
Soft Key Labels	These labels define soft key functions. Some soft keys, when pressed, activate screens that contain additional features.

2. Press the **SHIFT** key on the keyboard to display the Manual Shift screen (refer to **Figure 6-2**). Refer to Table 6-1, Edit Soft Keys.

All rights reserved. Subject to change without notice.
25-January-08



SHIFT EDIT

Figure 6-2, Program Editor SHIFT Screen

Editing Soft Keys

The Edit screen contains 14 soft keys, four of which are activated by pressing the **SHIFT** key. See Table 6-1, Edit Soft Keys.

To activate any **SHIFT** soft key:

1. In Edit Mode, press **SHIFT** and then press the appropriate soft key.

Table 6-1, Edit Soft Keys

Label	Soft Key	Function
Help	**F1**	Activates Edit Help screen.
Delete	**F2**	Deletes a single character located to the right of the cursor.
Insert	**F3**	Activates Insert Mode. Use to insert typed characters at the cursor position without overwriting the existing text.
Delete Block	**F4**	Deletes a single block located at the cursor.
Page Up	**F5**	Scrolls the display up one page.
Page Down	**F6**	Advances the display to the next page of the Program Listing.
Move	**F7**	Activates the Move pop-up menu. Refer to "Move (F7) Description from Edit Screen." Use this pop-up menu to return to the beginning or advance to the end of a block or program.
Edit Funct	**F8**	Activates the Edit Function pop-up menu. Refer to "Edit Funct (F8) Description from Edit Screen." Use this pop-up menu to perform various editing functions within a single program or between two programs. This includes inserting blocks. It also includes cutting and pasting blocks within a program or writing and reading blocks between programs. The pop-up menu contains word and line search features.
Exit	**F10**	Saves the program, closes the Edit screen, and returns the CNC to the Program Manager.
Msgs	(**SHIFT + F1**)	Displays the last 10 messages, both old (already read) and new (not yet read).
Undo	(**SHIFT + F3**)	Use to cancel edits made to a program block and restore the block to its original form. See "Undeleting a Block" and "Canceling Edits to a Program Block."
Redo	(**SHIFT + F4**)	Use to reverse canceled edits to a program block and restore the block to its original form. See "Restore Canceled Edits to a Program Block."
Quit	(**SHIFT + F10**)	Returns the CNC to the Program Manager without saving edits made to the Program Listing.

The following topics are described:

- Move (F7) Description from Edit Screen
- Edit Funct (F8) Description from Edit Screen

All rights reserved. Subject to change without notice.
25-January-08

Move (F7) Description from Edit Screen

Press **Move** (**F7**) to display the Move pop-up menu. Refer to **Figure 6-3** and **Table 6-2**.

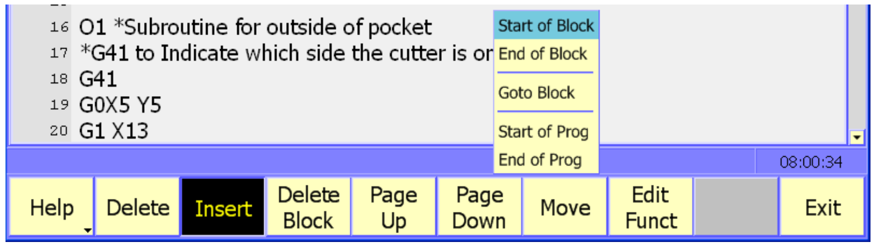


Figure 6-3, Move (F7) Pop-up Menu

Table 6-2, Move (F7) Pop-up Menu Description

Label	Description
Start of Block	The cursor returns to the beginning of the block. See "Advancing to the Beginning or End of a Block."
End of Block	The cursor advances to the end of the block. See "Advancing to the Beginning or End of a Block."
Goto Block	Use to move to any line in the Program Listing. See "Going to a Block of the Program Listing."
Start of Prog	The cursor returns to the first block of the program. See "Advancing to the First or Last Block of a Program."
End of Prog	The cursor advances to the last block of the program. See "Advancing to the First or Last Block of a Program."

All rights reserved. Subject to change without notice.

Edit Funct (F8) Description from Edit Screen

Press **Edit Funct** (**F8**) to display the Edit Funct pop-up menu. Refer to **Figure 6-4** and **Table 6-3**.

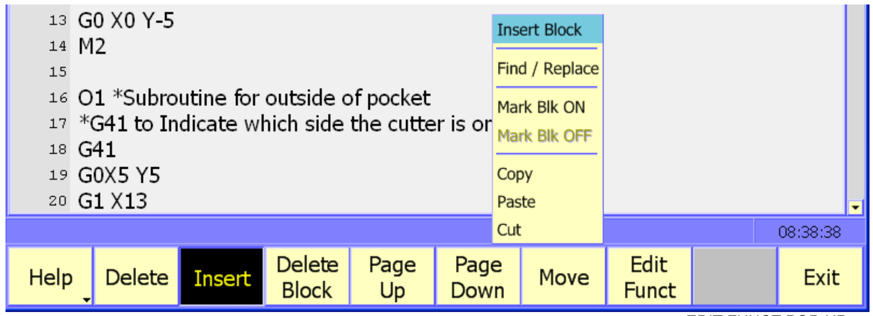


Figure 6-4, Edit Funct (F8) Pop-up Menu

Table 6-3, Edit Funct (F8) Pop-up Menu Description

Label	Function
Insert Block	Insert a program block at the cursor. See "Inserting a Program Block."
Find / Replace	Use to search blocks for specific text; searches forward and reverse in the program. See "Searching the Program Listing for Specific Text" and "Find/Replace Description from Edit Funct (F8) Screen."
Mark Blk ON **Mark Blk OFF**	Use to mark and unmark program blocks. See "Marking Programming Blocks" and "Unmarking Program Blocks."
Copy	Copy program blocks using the BLOCK operations **Copy** feature. See "To Mark and Copy Blocks:"
Paste	Paste copied or cut blocks into another section of the program using the BLOCK operations **Paste** feature. See "Pasting Blocks within a Program."
Cut	Delete program blocks using the BLOCK operations **Cut** feature. See "Deleting a Program Block" and "To Mark and Cut Blocks:"

All rights reserved. Subject to change without notice.
25-January-08

Marking Programming Blocks

For many editing features, you must mark the affected program blocks before the edit is performed. To mark program blocks:

1. In Edit Mode, place the cursor at the beginning of the first block to be marked.
2. Press **Edit Funct** (**F8**) to display the pop-up menu. Refer to Figure 6-4, Edit Funct (F8) Pop-up Menu.
3. Select **Mark Blk ON**. The block is marked.
4. Use **ARROW** keys to mark additional blocks up or down from the cursor position.

Unmarking Program Blocks

1. In Edit Mode, press **Edit Funct** (**F8**) to display the pop-up menu. Refer to Figure 6-4, Edit Funct (F8) Pop-up Menu.
2. Select **Mark Blk OFF**. Previously marked blocks will no longer be highlighted.

Saving Edits

The Program Listing displays edits as soon as they are made, but the edits are not saved until you exit the Program Editor.

To save edits:

1. In Edit Mode, press **Exit** (**F10**). The CNC returns to the Program Manager or Draw Graphics screen and saves all edits.

Canceling Unsaved Edits

If edits have not been saved, they can be canceled.

To cancel unsaved edits:

1. In Edit Mode, press **SHIFT** and then press **Quit** (**F10**).
2. The message **ProgramFilename has changes. Would you like to quit without saving?** displays on the screen, and the soft keys change. Press **Yes** (**F1**) to cancel edits and return to the Program Manager. Press **No** (**F3**) to return to the Edit Mode.

Deleting a Character

To delete a character:

1. In Edit Mode, use **ARROWS** to place the cursor to the left of the character you want to delete.
2. Press **Delete** (**F2**) to delete the character.

All rights reserved. Subject to change without notice.

Deleting a Program Block

There are two ways to delete program blocks from a Program Listing.

- Use the **Delete Block** (**F4**) soft key to delete blocks one at a time.
- Use **Edit Funct** (**F8**) pop-up menu, **Mark Blk ON**, to mark a block, and then press the **Delete** (**F2**) soft key to delete marked block.

To delete a program block using the **Delete Block** (**F4**) soft key:

1. In Edit Mode, place the cursor on the program block to be deleted.
2. Press **Delete Block** (**F4**). The CNC deletes the block.

To delete program blocks using the **Edit Funct** (**F8**) pop-up menu **Cut** feature:

1. In Edit Mode, mark the blocks to be deleted.
2. Press **Edit Funct** (**F8**) to display the pop-up menu. Refer to Figure 6-4, Edit Funct (F8) Pop-up Menu.
3. Select **Mark Blk ON** and the cursor will highlight. Then use the **ARROW** keys to move up or down to the number of the lines you want to mark for delete.
4. Select **Delete** (**F2**) and the CNC deletes the marked blocks.

Inserting a Program Block

Insert a program block at the cursor by using either method:

- **Edit Funct** (**F8**) pop-up menu **Insert Block** feature
- Edit soft key **Insert** (**F3**) feature

Using either method all subsequent lines will be moved down one line in the program.

To insert a program block using the **Edit Funct** (**F8**) pop-up menu **Insert Block** feature:

1. In Edit Mode, place the cursor at the beginning of the line where you want the program block to appear.
2. Select **Insert Block**, press **ENTER**, and a blank line is inserted at the cursor. You can type a new program block on the line.

To insert a program block using the Edit soft key **Insert** (**F3**) feature:

1. In Edit Mode, place the cursor at the beginning of the line where you want the program block to appear.
2. Press **Insert** (**F3**) to highlight it, then press **ENTER**, and a blank line is inserted at the cursor. You can type a new program block on the line.

All rights reserved. Subject to change without notice.

Undeleting a Block

You can restore deleted blocks with the Edit **Undo** (**SHIFT + F3**) soft key. Refer to Figure 6-2, Program Editor SHIFT Screen. The last block deleted is the first block restored.

To restore a block:

1. In Edit Mode, place the cursor at the point where the restored block is displayed.
2. Press the **Undo** (**SHIFT + F3**) soft key and the last line deleted from the program displays at the cursor.

NOTE: Up to 128 consecutively deleted blocks can be restored.

Canceling Edits to a Program Block

Use the **Undo** feature to cancel edits made to a program block and restore the block to its original form.

To cancel edits to a program block:

1. In Edit Mode, press **SHIFT** to display the **SHIFT** Edit soft keys. Refer to Figure 6-2, Program Editor SHIFT Screen.
2. Press **Undo** (**SHIFT + F3**) soft key. The last line(s) deleted will be restored.

Restore Canceled Edits to a Program Block

Use the **Redo** feature to reverse cancel edits made to a program block and restore the block to its original form.

To restore cancel edits to a program block:

1. In Edit Mode, press **SHIFT** to display the **SHIFT** Edit soft keys. Refer to Figure 6-2, Program Editor SHIFT Screen.
2. Press **Redo** (**SHIFT + F4**) soft key. The CNC reverses the canceled edits one character at a time to restore the original block.

Inserting Text without Overwriting Previous Text

Use **Insert** (**F3**) to activate the Insert Mode. In Insert Mode, the CNC inserts typed text at the cursor, without overwriting previously entered text.

To insert text into a program without overwriting previously entered text:

1. In Edit Mode, press **Insert** (**F3**). Make sure the Insert soft key is highlighted.
2. Place the cursor where you want to insert the text. Enter the new text. The new text is inserted to the left of the cursor. The CNC does not delete previously typed text as you type.

All rights reserved. Subject to change without notice.

Inserting Text and Overwriting Previous Text

To insert text into a program while overwriting previously entered text:

1. In Edit Mode, press **Insert** (**F3**) so the soft key is no longer highlighted.
2. Place the cursor where the text will be inserted. Enter the new text. The new text is inserted overwriting the character to the right of the cursor. The CNC deletes previously typed text as you type.

Advancing to the Beginning or End of a Block

To advance to the beginning or end of a program block:

1. In Edit Mode, place the cursor on the desired block of the program. Press **Move** (**F7**) to display the pop-up menu. Refer to Figure 6-3, Move (F7) Pop-up Menu.
2. Select **End of Block**. The cursor advances to the end of the block.

   – or –

   Select **Start of Block**. The cursor returns to the beginning of the block.

Advancing to the First or Last Block of a Program

To advance to the first or last block of a program:

1. In Edit Mode, press **Move** (**F7**) to display the pop-up menu. Refer to Figure 6-3, Move (F7) Pop-up Menu.
2. Select **End of Prog**. The cursor advances to the last block of the program.

   – or –

   Select **Start of Prog**. The cursor returns to the first block of the program.

All rights reserved. Subject to change without notice.

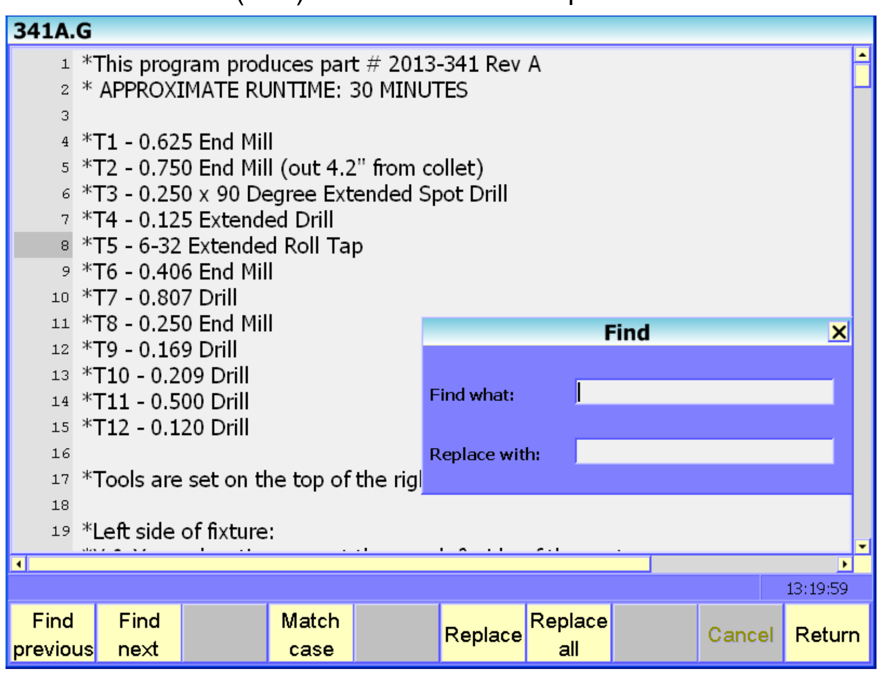
Searching the Program Listing for Specific Text

Use **Edit Funct** (**F8**) pop-up menu **Find / Replace** feature to search blocks for specific text.

To find all references of text in a program:

1. In Edit Mode, place the cursor at the beginning of the program. (**Find / Replace** searches forward and reverse in the program.)
2. Press **Edit Funct** (**F8**) to display the pop-up menu. Refer to Figure 6-4, Edit Funct (F8) Pop-up Menu.
3. Select **Find / Replace** and press enter to display the Find screen. Refer to **Figure 6-5**.
4. The message **Find what:** displays on the screen. Enter the text to be found, and press **Find next** (**F2**) or **Find prev** (**F1**). The cursor advances to the first occurrence of the text in the program or regresses to the previous occurrence of the text in the program from the current cursor position. Use **Match Case** (**F4**) for a case-sensitive search.
5. To advance to the next occurrence of the text, press **Find next** (**F2**) again. The CNC advances to the next occurrence of the text in the program.

   To regress to the previous occurrence of the text, press **Find prev** (**F1**). The CNC regresses (Page Up) to the previous occurrence of the text in the program.
6. Use this method to search for all occurrences of the text in the Program Listing.
7. Press **Return** (**F10**) to exit the Find / Replace screen.



FIND_REPLACE

Figure 6-5, Edit Funct (F8) Find/Replace Screen

All rights reserved. Subject to change without notice.

The following topic is described:

- **Find/Replace Description for Edit Funct (F8) Pop-up Menu**

Find/Replace Description from Edit Funct (F8) Pop-up Menu

Press **Find/Replace** to display the Find/Replace screen (refer to Figure 6-5, Edit Funct (F8) Find/Replace Screen) and **Table 6-4**.

Table 6-4 describes the Find/Replace screen active soft keys.

Table 6-4, Find/Replace Screen Soft Keys

Label	Soft Key	Function
Find previous	**F1**	Finds the previous occurrence (Page up) of the **Find what:** text in the program; highlighted text does not change.
Find next	**F2**	Finds the next occurrence (Page down) of the **Find what:** text in the program; highlighted text does not change.
Match Case	**F4**	Use a case-sensitive search.
Replace	**F6**	Inserts replacement text (**Replace with:**) for highlighted text (**Find what:**) and goes to the next occurrence of the **Find what:** text in the program.
Replace all	**F7**	Inserts replacement text (**Replace with:**) for all occurrences of the highlighted text (**Find what:**) in the program.
Cancel	**F9**	Cancels the find/replace search.
Return	**F10**	Returns the CNC to the original **Edit Funct** (**F8**) Screen.

All rights reserved. Subject to change without notice.

Replacing Typed Text with New Text

Use **Replace with:** to replace selected occurrences of text. Enter the appropriate text and the CNC searches the Program Listing for all occurrences of the text. You can edit or skip each occurrence. Use **Match Case** (**F4**) for a case-sensitive replacement. Refer to Figure 6-5, Edit Funct (F8) Find/Replace Screen.

To replace specific occurrences of the typed text:

1. In Edit Mode, press **Edit Funct** (**F8**) to display the pop-up menu. Refer to Figure 6-4, Edit Funct (F8) Pop-up Menu.
2. Select **Find / Replace** and press **ENTER** to display the Find screen. Refer to Figure 6-5, Edit Funct (F8) Find/Replace Screen.
3. Type in **Find what:** and **Replace with:** texts. Press **Replace** (**F6**). Use **Match Case** (**F4**) for a case-sensitive search.
4. Each time you press **Replace** (**F6**) the CNC finds the next occurrence of the text in the program. You can search forward [**Find next** (**F2**)] or backward [**Find previous (F1**)] in the program. The text is not replaced until you press the **Replace** (**F6**) soft key. A description of the Find / Replace soft keys follows:

Going to a Block of the Program Listing

Use **Goto Block** to move to any line in the Program Listing. **Goto Block** operates independently of block numbering. Blocks can be numbered sequentially by any increment (**1**, **5**, **10**...). **Goto Block** counts the blocks of the program in increments of 1 (**1** [starting block], **2**, **3**...). When the feature is activated, the CNC goes to the line number specified by the user, regardless of how the blocks are numbered.

Block #	Line #
N10	Line 1
N20	Line 2
N30	Line 3

1. In Edit Mode, press **Move** (**F7**) to display the pop-up. Refer to Figure 6-3, Move (F7) Pop-up Menu.
2. Highlight **Goto Block**. And press enter to display the Goto dialog prompt. Refer to Figure 6-6, Goto Block Dialog Prompt.

All rights reserved. Subject to change without notice.

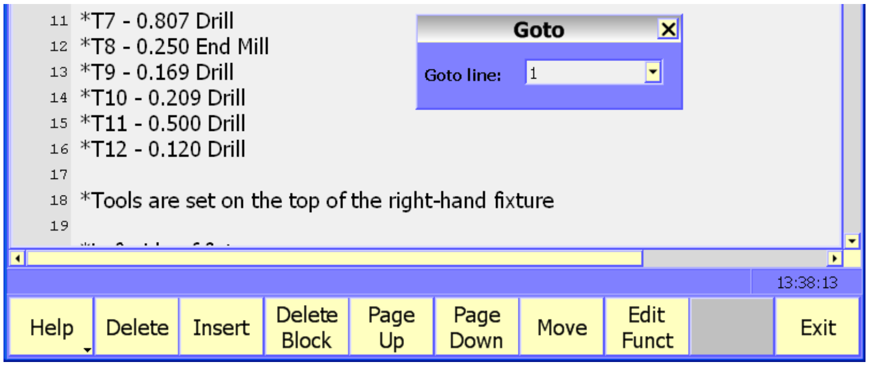


Figure 6-6, Goto Block Dialog Prompt

3. Type in the appropriate line number. Press **ENTER**. The CNC places the cursor at that line number.

Scrolling Through the Program

In Edit Mode, press the up and down **ARROWS** to scroll up and down in the Program Listing.

Paging Through the Program

With long programs, it is convenient to move the Program Listing display up and down a whole page at a time.

1. In Edit Mode, press **Page Up** (**F5**) to go backward one page or press **Page Down** (**F6**) to advance down one page in the program. The CNC advances or goes back one page at a time.

Inserting a Blank Line

Insert a line at the cursor position with the Insert Line feature. All subsequent lines will be moved down one line in the program. To insert a blank line:

1. In Edit Mode, place the cursor at the beginning of the string and press **ENTER** to add a blank line before the string. Place the cursor at the end of the string and press **ENTER** to add a blank line after the string.
2. You can type a new program block on the blank line.

All rights reserved. Subject to change without notice.
25-January-08

Copying Program Blocks

NOTE: You can cut, copy, and paste blocks within a Program Listing. The Cut, Copy, and Paste features work for copying and pasting blocks between two different programs.

Copy one or more program blocks and place them elsewhere in the same Program Listing. **Table 6-5** describes two ways to copy program blocks.

Table 6-5, Copying Program Blocks

Method	Description
Mark and copy blocks	Copies and stores marked blocks. Leaves original blocks unchanged.
Mark and cut blocks	Cuts and stores marked blocks. Deletes original blocks.

To Mark and Copy Blocks:

1. In Edit Mode, place the cursor at the beginning of the first block to be copied.
2. Mark the blocks to be copied. See "Marking Programming Blocks."
3. Press **Edit Funct** (**F8**) to display the pop-up menu. Refer to Figure 6-4, Edit Funct (F8) Pop-up Menu.
4. Select **Copy**. The CNC saves the blocks in memory and the original blocks remain in the Program Listing.

To Mark and Cut Blocks:

1. In Edit Mode, place the cursor at the beginning of the first block to be copied.
2. Mark the blocks to be copied. See "Marking Programming Blocks."
3. Press **Edit Funct** (**F8**) to display the pop-up menu. Refer to Figure 6-4, Edit Funct (F8) Pop-up Menu.
4. Select **Cut**. The CNC saves the blocks in memory and deletes the original blocks from the Program Listing.

Pasting Blocks within a Program

To paste copied or cut blocks into another section of the program:

1. In Edit Mode, place the cursor where you want to paste the copied blocks.
2. Press **Edit Funct** (**F8**) to display the pop-up menu. Refer to Figure 6-4, Edit Funct (F8) Pop-up Menu.
3. Select **Paste**. The CNC pastes the copied blocks into the Program Listing behind the current cursor position.

All rights reserved. Subject to change without notice.

Including Comments in a Program Listing

Use an asterisk (*) to make comments within a Program Listing or to mask all or part of a block from the CNC. When an asterisk is placed before a string of text, the CNC ignores all the text to the right of the asterisk. **Table 6-6** shows various ways to use the asterisk in a Program Listing.

Table 6-6, Comment Blocks

Commented Block	Ignored Text
*N20 G1 X5 Z6 N30 …	CNC ignores the entire block. The next block is executed.
N20 G1 *X5 Z6 N21 …	CNC activates Linear Interpolation (G1). Then, programmed move to X5 Z6 is ignored. The next block is executed.
N10 G70 G90 G0 X0 Z0 T0 N20 T1 *FACE/TURN TOOL	Block N20 activates Tool #1. The comment contains the type of tool used as a note to the operator only.

All rights reserved. Subject to change without notice.

Section 7 - Edit Help

Edit Help provides diagrams and entry fields to program move types and Canned Cycles. The following section describes how to activate a Help Screen for a G-Code command and type values in the appropriate entry fields. Refer to **Figure 7-1**.

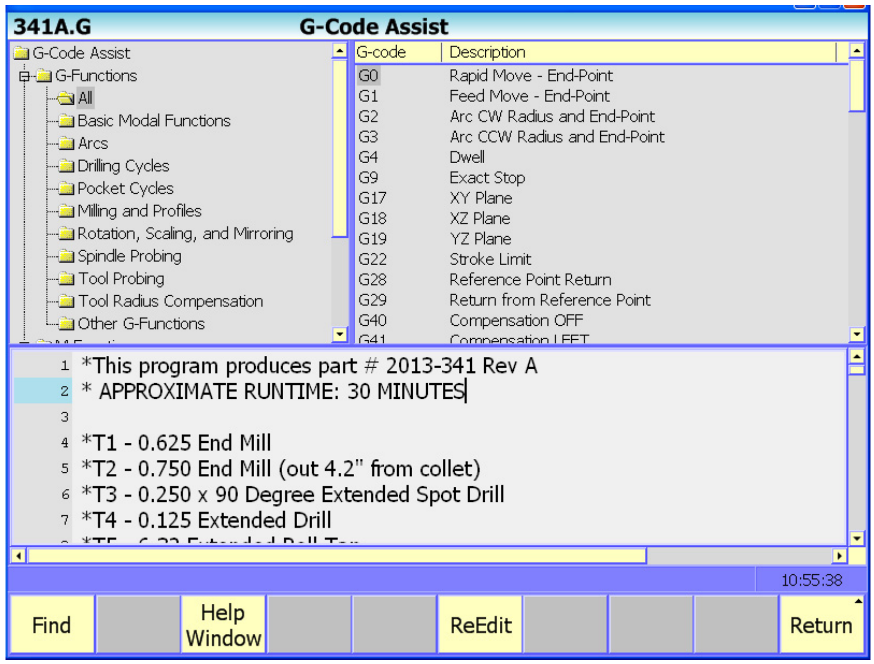


Figure 7-1, Edit Help Screen

To access the Edit Help Screen:

1. In Edit Mode, open the appropriate program. Press **Help** (**F1**). The Edit Help screen activates.
2. Select the Help group from the top left and the specific helps are displayed on the top right.
3. Select the specific code that you want to add to your code and complete the required and optional fields displayed.
4. Press **Use** to add the code to your program.

All rights reserved. Subject to change without notice.

The following topics are described in this section:

Edit Help Soft Keys

The Edit Help Menu contains the following soft keys. Refer to **Table 7-1**.

Table 7-1, Edit Help Soft Keys

Label	Soft Key Number	Description
Find	**F1**	Moves highlight to the requested Help Template.
Help Window	**F3**	Switches between selecting codes for help on the top portion of the screen and directly editing the program "free form" on the bottom portion of the screen.
ReEdit	**F6**	To edit a typed command (G-Code or canned cycle), place the cursor on the appropriate block and press **ReEdit** (**F6**). Once all the fields have been edited, press **Use** (**F10**).
Return	**F9**	Deactivates the Edit Help Menu and returns you to the Program Listing.

All rights reserved. Subject to change without notice.
25-January-08

Using Help Graphic Screens to Enter Program Blocks

The Program Editor displays help graphic screens, in which you write and edit program blocks.

When the CNC activates a help graphic screen, its first entry field is highlighted. A highlight indicates that you can type values in an entry field or make the appropriate selection. Press **ENTER** to move the highlight to the next entry field. In the last entry field of the help graphic screen, press **Use** (**F10**) or **ENTER** to add the block to the Program Listing. Press the **ARROWS** to move the highlight between entry fields without typing values. Press **CLEAR** to clear an entry field.

There are two types of entry fields:

Required entry fields	Contain **0.000**. You must type a value for operation of canned cycle or other command.
Optional entry fields	Blank. Entry optional.

If a required entry field is left blank, the CNC writes the block using the 0.0000 default. This may generate an error message when the program runs.

Optional entry fields do not require a value. When left **blank**, a default value or position is usually assumed.

You must remember to type: decimal points and negative signs where needed. The CNC assumes a positive value if no negative sign is typed.

Press the (**+/-**) key to insert a negative sign.

Press the **CLEAR** key to clear an entry.

All rights reserved. Subject to change without notice.

G-Functions

The G-Code functions have the following functional groups:

- All – All G-Codes are listed (including user defined G-Codes)
- Basic Modal Functions
- Arcs
- Drilling Cycles
- Pocket Cycles
- Milling and Profiles
- Rotation, Scaling, and Mirroring
- Spindle Probing
- Tool Probing
- Tool Radius Compensation
- Other G-Functions

The following topics are described:

- Basic Modal Functions
- Tool Radius Compensation
- Arcs
- Milling and Profiles
- Drilling Cycles
- Pocket Cycles
- Rotation, Scaling, and Mirroring
- Other G-Functions

All rights reserved. Subject to change without notice.
25-January-08

Basic Modal Functions

The Basic Modal Functions enables:

G0 Rapid Move – End-Point
Refer to "Section 4, Rapid Move (G0)"

G1 Feed Move – End-Point
Refer to "Section 4, Feed Move (G1)"

G17 XY Plane
Refer to "Section 4, Plane Selection (G17, G18, G19)"

G18 XZ Plane
Refer to "Section 4, Plane Selection (G17, G18, G19)"

G19 YZ Plane
Refer to "Section 4, Plane Selection (G17, G18, G19)"

G59 Modal Radius/Chamfer
Refer to "Section 4, Modal Corner Radius/Chamfer (G59, G60)"

G60 Cancel Modal Radius or Chamfer
Refer to "Section 4, Modal Corner Radius/Chamfer (G59, G60)"

G70 Inch
Refer to "Section 4, Activating Inch (G70) or MM (G71) Mode"

G71 MM
Refer to "Section 4, Activating Inch (G70) or MM (G71) Mode"

G90 Absolute
Refer to "Section 4, Activating Absolute (G90) or Incremental (G91) Mode"

G91 Incremental.
Refer to "Section 4, Activating Absolute (G90) or Incremental (G91) Mode"

Feed Feedrate.
Refer to "Section 4, Feedrate (FEED)"

Tool Radius Compensation

The Tool Radius Compensation enables:

G40 Compensation OFF
Refer to "Section 9, Compensation (G40, G41, G42)"

G41 Compensation LEFT
Refer to "Section 9, Compensation (G40, G41, G42)"

G42 Compensation RIGHT
Refer to "Section 9, Compensation (G40, G41, G42)"

All rights reserved. Subject to change without notice.

Arcs

The Arcs enables:

G2 Arc CW
Refer to "Section 4, Circular Interpolation (G2 and G3)"

G3 Arc CCW
Refer to "Section 4, Circular Interpolation (G2 and G3)"

Refer to "Programming Concepts" in "Section 1 - Introduction" for information on planes and arc directions. The CNC executes arcs in the XY (**G17**) plane by default. For an arc in the XZ (**G18**) or YZ (**G19**) plane, program the plane change before the arc move. After you make all the required moves in the XZ or YZ plane, return the CNC to the XY plane.

Refer to **Figure 7-2**. There are two arcs that 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.

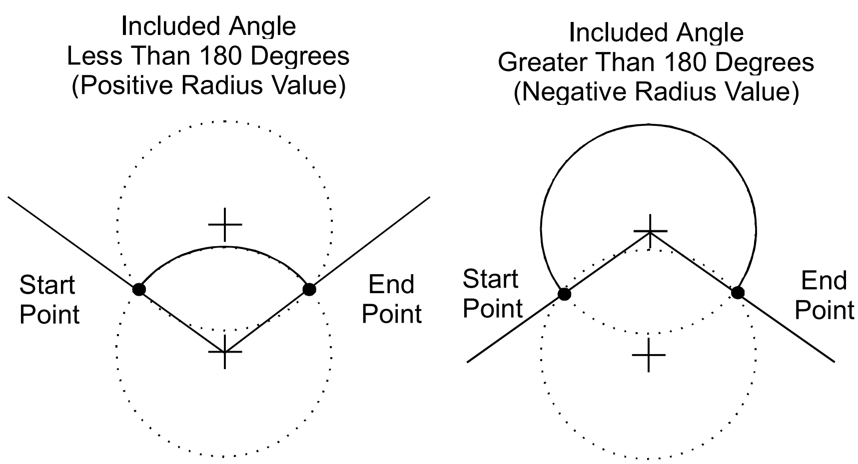


Figure 7-2, Endpoint Radius Arc Types

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

All rights reserved. Subject to change without notice.
25-January-08

Refer to **Figure 7-3** and **Figure 7-4**. Specify the appropriate Absolute or Incremental Mode for the angle and center point. 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 positions differ, the arc will be a helix.

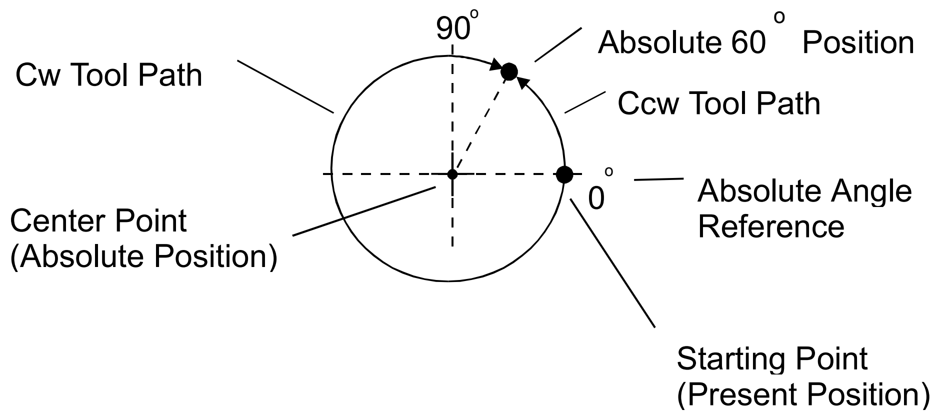


Figure 7-3, Absolute Mode, Center-Angle Arc

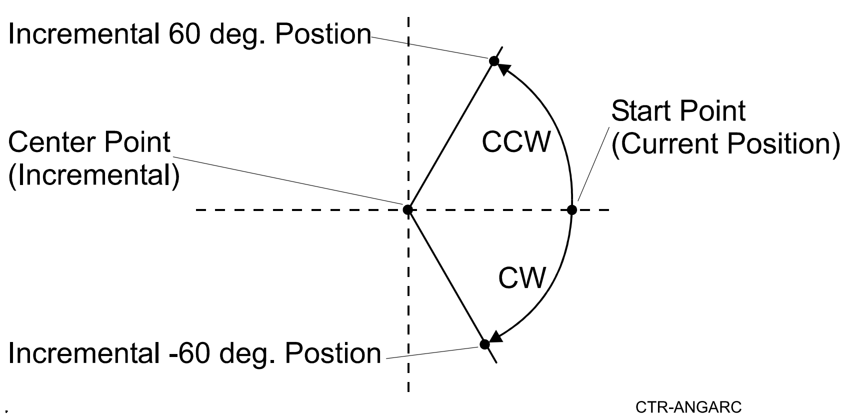


Figure 7-4, Incremental Mode, Center-Angle Arc

All rights reserved. Subject to change without notice.

Table 7-2, G2 Address Words

Label	Address Word	Description
End Horizontal	**X**	X endpooint (Required)
End Vertical	**Y**	Y endpoint (Required)
Radius	**R**	Radius of arc (Required)

Table 7-3, G3 Address Words

Label	Address Word	Description
End Horizontal	**X**	X endpoint (Required)
End Vertical	**Y**	Y endpoint (Required)
Radius	**R**	Radius of arc (Required)

All rights reserved. Subject to change without notice.

Milling and Profiles

The Milling and Profiles enables:

G170 Face Mill Cycle
Refer to "Section 5, Face Mill Cycle (G170)"

G171 Circular Profile Cycle
Refer to "Section 5, Circular Profile Cycle (G171)"

G172 Rectangular Profile Cycle
Refer to "Section 5, Rectangular Profile Cycle (G172)"

G175 Mill Cycle
Refer to "Section 5, Mill Cycle (G175)"

G176 EndMill Cycle
Refer to "Section 5, EndMill Cycle (G176)"

G181 Tread Mill Cycle
Refer to "Section 5, Tread Mill Cycle (G181)"

G190 Engrave Cycle
Refer to "Section 5, Engrave Cycle (G190)"

Drilling Cycles

The Drilling Cycles enables:

G79 Drill Bolt Hole Cycle
Refer to "Section 5, Drill Bolt Hole Cycle (G79)"

G80 Drilling Off
Refer to "Section 5, Drilling Off (G80)"

G81 Basic Drill Cycle
Refer to "Section 5, Basic Drill Cycle (G81)"

G82 CounterBore Drill Cycle
Refer to "Section 5, CounterBore Drill Cycle (G82)"

G83 Peck Drill Cycle
Refer to "Section 5, Peck Drill Cycle (G83)"

G84 Tapping Cycle
Refer to "Section 5, Tapping Cycle (G84)"

G85 Boring Bidirectional Cycle
Refer to "Section 5, Boring Bidirectional Cycle (G85)"

G86 Boring Unidirectional Cycle
Refer to "Section 5, Boring Unidirectional Cycle (G86)"

G87 Chip Break Cycle
Refer to "Section 5, Chip Break Cycle (G87)"

G89 Flat Bottom Boring Cycle
Refer to "Section 5, Flat Bottom Boring Cycle (G89)"

G179 Drill Pattern Cycle
Refer to "Section 5, Drill Pattern Cycle (G179)"

All rights reserved. Subject to change without notice.

Pocket Cycles

The Pocket Cycles enables:

G73 Draft Angle Pocket Cycle
Refer to "Section 5, Draft Angle Pocket Cycle (G73)"

G75 Frame Pocket Cycle
Refer to "Section 5, Frame Pocket Cycle (G75)"

G76 Hole Mill Cycle
Refer to "Section 5, Hole Mill Cycle (G76)"

G77 Circular Pocket Cycle
Refer to "Section 5, Circular Pocket Cycle (G77)"

G78 Rectangular Pocket Cycle
Refer to "Section 5, Rectangular Pocket Cycle (G78)"

G162 Islands
Refer to "Section 5, Islands (G162)"

G169 Irregular Pocket Cycle
Refer to "Section 5, Irregular Pocket Cycle (G169)"

G177 Plunge Circular Pocket Cycle
Refer to "Section 5, Plunge Circular Pocket Cycle (G177)"

G178 Plunge Rectangular Pocket Cycle
Refer to "Section 5, Plunge Rectangular Pocket (G178)"

Rotation, Scaling, and Mirroring

The Rotation, Scaling, and Mirroring enables:

G68 Rotation (Axis)
Refer to "Section 4, Axis Rotation (G68)"

G72 Scaling
Refer to "Section 4, Scaling (G72)"

G100 Mirroring
Refer to "Section 4, Mirroring (G100)"

All rights reserved. Subject to change without notice.

Other G-Functions

The Other G-Functions enables:

G04 Dwell
Refer to "Section 4, Dwell (G4)"

G09 Exact Stop
Refer to "Section 4, Programming Non-modal Exact Stop (G9)"

G22 Stroke Limit
Refer to "Section 4, Setting Stroke Limit (G22)"

G28 Reference Point Return
Refer to "Section 4, Reference Point Return (G28)"

G29 Return from Reference Point
Refer to "Section 4, Return from Reference Point (G29)"

G53 Fixture Offset
Refer to "Section 4, Fixture Offset (Work Coordinate System Select), (G53)"

G61 Exact Stop Mode
Refer to "Section 4, In-Position Mode (Exact Stop Check) (G61)"

G64 Contouring Mode
Refer to "Section 4, Contouring Mode (Cutting Mode) (G64)"

G65 Macro Call, Single
Refer to "Section 4, User Macros (G65, G66, G67)"

G66 Macro Call, Modal
Refer to "Section 4, User Macros (G65, G66, G67)"

G67 Cancel Modal Macro
Refer to "Section 4, User Macros (G65, G66, G67)"

G92 Zero Set
Refer to "Section 4, Absolute Zero Point Programming (G92)"

G120 BlockForm
Refer to "Section 4, BlockForm (G120)"

All rights reserved. Subject to change without notice.

M-Functions

The M-Code functions have the following functional groups:

- All – All M-Codes are listed (including user defined M-Codes)
- Basic M-Functions
- Cooling, Cleaning, and Lubrication
- Spindle Functions
- Tool Change

The following topics are described:

Basic M-Functions

The Basic M-Functions enables the following M-Codes:

M0	Program Stop Mode
M1	Optional Program Stop
M2	End of Program
M30	Jump to New Program
M98	Call SubProgram
M99	End of SubProgram
M105	Dry Run, All Axes
M106	Dry Run, No Z Axis
M107	Dry Run, Off - Cancel M105 and M106

Cooling, Cleaning, and Lubrication

The Cooling/Cleaning/Lubrication M-functions enables the following M-Codes:

M8	Coolant On
M9	Coolant Off

Spindle Functions

The Spindle Functions M-functions enables the following M-Codes:

M3	Spindle Forward
M4	Spindle Reverse
M5	Spindle Off
M19	Spindle Orientation
SPEED	Spindle Speed

Tool Change

The Tool Change M-function enables the following M-Code:

M6	Tool Mount

All rights reserved. Subject to change without notice.

Tools

The Tools enables the following:

TOOL Tool Mount

All rights reserved. Subject to change without notice.

G-Code Listing

When a G-Code is selected from the list, an input screen activates. It contains instructions and entry fields that pertain to the selected G-Code. Use the screens to input G-Codes. Refer to **Table 7-4**.

Table 7-4, Most Common Modal G-Codes

G-Code	Function
Rapid Move G0	Axis moves made at rapidrate
Feed Move G1	Axis moves made at feedrate
Arc CW G2	Sets clockwise circular interpolation
Arc CCW G3	Sets counterclockwise circular interpolation
XY Plane G17	Sets default XY plane
XZ Plane G18	Sets default XZ plane
YZ Plane G19	Sets default YZ plane
Inch G70	Sets CNC to Inch measurements
MM G71	Sets CNC to MM measurements
Absolute G90	Sets CNC to Absolute Mode
Incremental G91	Sets CNC to Incremental Mode

Table 7-5 describes the other G-Codes in the menu.

Table 7-5, Edit Help G-Code Listing

G-Code	Label and Description
G0	**Rapid Move.** Axis moves made at rapidrate. See also **Table 7-4, Most Common Modal G-Codes**.
G1	**Feed Move.** Axis moves made at feedrate. See also **Table 7-4, Most Common Modal G-Codes**.
G2	**Arc CW.** Sets clockwise circular interpolation. See also **Table 7-4, Most Common Modal G-Codes**.
G3	**Arc CCW.** Sets counterclockwise circular interpolation. See also **Table 7-4, Most Common Modal G-Codes**.
G4	**Dwell.** Programs a timed or infinite dwell.
G9	**Exact Stop.** Non-modal exact stop check. Activates exact stop check for a single block.
G17	**XY Plane.** Sets default XY plane. See also **Table 7-4, Most Common Modal G-Codes**.
G18	**XZ Plane.** Sets default XZ plane. See also **Table 7-4, Most Common Modal G-Codes**.

All rights reserved. Subject to change without notice.

Table 7-5, Edit Help G-Code Listing (Continued)

G-Code	Label and Description
G19	**YZ Plane.** Sets default YZ plane. See also Table 7-4, Most Common Modal G-Codes.
G22	**Stroke Limit.** Activates/deactivates software limits.
G28	**Reference Point Return.** Return to Machine Home directly or through an intermediary point.
G29	**Return from Reference Point.** Return from Machine Home to the coordinates specified. (G29 Xn Zn)
G40	**Compensation OFF.** Tool radius compensation cancel G41 or G42.
G41	**Compensation LEFT.** Tool radius compensation LEFT.
G42	**Compensation RIGHT.** Tool radius compensation RIGHT.
G53	**Fixture Offset.** Shifts the location of Absolute Zero to a preset location. The preset location is the specified fixture offset, measured from Machine Home and stored in the Fixture Offsets Table.
G59	**Modal Radius/Chamfer.** Use to program modal corner rounding or chamfering.
G60	**Cancel Modal Radius or Chamfer.** Use to cancel the program modal corner rounding or chamfering.
G61	**Exact Stop Mode.** Contouring Mode OFF. Modal Exact Stop Check. Activates In-Position Mode.
G64	**Contouring Mode.** Exact Stop Mode OFF. Modal Contouring Mode. De-activates In-Position Mode.
G65	**Macro Call, Single.** (Non-Modal) Used in a program to call a stored macro. Macros can be entered after the main program (subprogram) or in another file (must use file inclusion to call to active program). In non-modal macro (**G67**) call, the variables can be changed at each call.
G66	**Macro Call, Modal.** Used in a program to call a macro. Macros can be entered after the main program (subprogram) or in another file (must use file inclusion to call to active program). In Modal macro (**G66**) call, the variables will always contain the same values.
G67	**Cancel Modal Macro.** Cancels a **G66** Modal Macro call.
G68	**Rotation (Axis).** Axis rotation is modal and remains active until canceled.
G70	**Inch.** Sets CNC to Inch measurements. See also Table 7-4, Most Common Modal G-Codes.

(Continued…)

All rights reserved. Subject to change without notice.

Table 7-5, Edit Help G-Code Listing (Continued)

G-Code	Label and Description
G71	**MM.** Sets CNC to MM measurements. See also Table 7-4, Most Common Modal G-Codes.
G72	**Scaling.** Use Axis Scaling to enlarge or reduce patterns commanded by the program.
G73	**Draft Angle Pocket Cycle.** Use the draft angle pocket cycle (**G73**) to machine a draft angle on a pocket.
G75	**Frame Pocket Cycle.** Frame pocket cycle (**G75**) will mill a frame or trough around an island of material.
G76	**Hole Mill Cycle.** Use the hole mill cycle (**G76**) to machine through holes or counter-bores.
G77	**Circular Pocket Cycle.** Use the circular pocket cycle (**G77**) to mill round pockets.
G78	**Rectangular Pocket Cycle.** Use the rectangular pocket cycle (**G78**) to mill square or rectangular pockets.
G79	**Drill Bolt Hole Cycle.** Use the automatic drill bolt hole cycle (**G79**) to drill a partial or full bolt circle.
G80	**Drilling Off.** Use **G80** to cancel drill, tap, and bore canned cycles (**G81** to **G89**).
G81	**Basic Drill Cycle. G81** is a basic drilling cycle, generally used for center drilling or hole drilling that does not require a pecking motion.
G82	**CounterBore Drill Cycle. G82** is the counterbore drill cycle, generally used for counterboring.
G83	**Peck Drill Cycle. G83** is the peck drilling cycle, generally used for peck drilling relatively shallow holes.
G84	**Tapping Cycle. G84** is the tapping canned cycle, used for tapping holes.
G85	**Boring Bidirectional Cycle. G85** is a boring cycle, generally used to make a pass in each direction on a bore or to tap with a self-reversing tapping head.
G86	**Boring Unidirectional Cycle. G86** is a boring cycle that allows the X-axis to back off the bore surface after the spindle has stopped and oriented itself.
G87	**Chip Break Cycle. G87** is the chip-breaker peck-drilling cycle, generally used to peck-drill medium to deep holes.
G89	**Flat Bottom Boring Cycle. G89** is a boring cycle, generally used to program a pass in each direction with a dwell at the bottom.
G90	**Absolute.** Sets CNC to Absolute Mode
G91	**Incremental.** Sets CNC to Incremental Mode

(Continued…)

All rights reserved. Subject to change without notice.

Table 7-5, Edit Help G-Code Listing (Continued)

G-Code	Label and Description
G92	**Zero Set.** Shifts the location of Absolute Zero to a preset location. The preset location, measured from Machine Home, is specified in the **G92** command.
G100	**Mirroring.** G100 programmed with axis (G100 X) activates "mirror image" (ON) for that axis. Mirror image reverses the sign (+/-) of subsequent numbers. More than one axis can be mirrored at once (G100 XY). To cancel mirror image, program **G100** on a block by itself.
G120	**BlockForm. G120** is used to define a window in relation to the X and Y machine home and the Z part zero.
G162	**Islands.** This cycle allows islands in irregular pockets.
G169	**Irregular Pocket Cycle.** Use **G169** to mill irregular pockets.
G170	**Face Mill Cycle.** Facing cycles simplify the programming required to face the surface of a part.
G171	**Circular Profile Cycle.** The Circular Profile Cycle cleans up the inside or outside profile of an existing circle.
G172	**Rectangular Profile Cycle.** The Rectangular Profile Cycle cleans up the inside or outside profile of a rectangle.
G175	**Mill Cycle.** Use the mill cycle (**G175**) to machine through holes or counter-bores.
G176	**EndMill Cycle.** The mill cycle is terminated with the EndMill block; at which point, it rapids up to the StartHgt and returns to the uncompensated X and Y location.
G177	**Plunge Circ Pocket Cycle.** Use the plunge circular pocket cycle (**G177**) for carbide tooling, when a multiple-axis ramp-in move is not possible. The Z-axis will plunge (single-axis) to programmed depths.
G178	**Plunge Rect Pocket.** Use the plunge rectangular pocket cycle (**G178**) for carbide tooling, where a multiple-axis ramp-in move is not possible. The Z-axis will plunge (single-axis) to the programmed depth.
G179	**Drill Pattern Cycle.** Use the automatic hole pattern canned cycle (**G179**) to program partial or full pattern hole grids. You can use **G179** for a corner pattern when holes are required only on four corners.
G181	**Thread Mill Cycle.** Use the thread mill cycle for cutting inside or outside threads. It will cut either Inch or MM, left or right hand, and Z movement up or down. A single tooth or multi-toothed tool may be used.

(Continued…)

All rights reserved. Subject to change without notice.

Table 7-5, Edit Help G-Code Listing (Continued)

G-Code	Label and Description
G190	**Engrave Cycle.** Use the engrave cycle to engrave part numbers, legends, or any alpha/numeric inscription. The usual type of cutter is a sharp point or center-drill type tool.
FEED	**Feedrate.** Use to set the feedrate.

Entry Fields

When a G-Code is selected from the G-Code Listing, instructions and entry fields are listed on the screen. Type values for the required parameters.

All rights reserved. Subject to change without notice.

M-Code Listing

You can program M-Codes by selecting them from the list. If the M-Code requires a parameter, the software displays the Help Graphic for the entered M-Code. Only **M30** and **M98** require parameters. Fill in the entry fields for these M-Codes. Press **Use** (**F10**) to insert the selected code in the block or **Cancel** (**F9**) to cancel.

For other M-Codes, select the code and press **Use** (**F10**) to insert the code in the block. Press **Cancel** (**F9**) to cancel.

Table 7-6 describes the M-Codes in the menu.

Table 7-6, Edit Help M-Code Listing

M-Code	Function
M0 or **M00**	Program Stop Mode
M1 or **M01**	Optional Program Stop
M2 or **M02**	End of Program
M3 or **M03**	Spindle Forward
M4 or **M04**	Spindle Reverse
M5 or **M05**	Spindle Off
M6 or **M06**	Tool Mount
M8 or **M08**	Coolant On
M9 or **M09**	Coolant Off
M19	Spindle Orientation
M30	Jump to New Program
M98	Call SubProgram
M99	End of SubProgram
M105	Dry Run, All Axes
M106	Dry Run, No Z Axis
M107	Dry Run, Off - Cancel M105 and M106
SPEED	Spindle Speed

All rights reserved. Subject to change without notice.

Typing in Address Words

You can manually type in most address words without exiting Edit Help. Address words that can be typed into the program via Edit Help include: dimension coordinates (**XYZUW**); spindle codes (**S**); feedrates (**F**); tool codes (**T**); and preparatory codes (**G**). Use the following procedure:

1. From the Main Edit Help screen or from a Help Template Menu, type the required commands. Edit Help displays the typed commands in the center of the screen. If the address word requires a parameter, the software displays an entry field in which you type the appropriate value or selection.
2. Type the value or selection, if required. You can accept or cancel commands just as you would in the Help Graphic Menus. Press **Use** (**F10**) to enter the block into the program. Press **Cancel** (**F9**) to cancel your entry and clear the screen. Accepted commands are inserted in the program.

Example: Entering G-Codes

From the Main Edit Help screen, type **G77**, and press **ENTER**. The CNC activates the Help Graphic for Circular Pocket Milling (**G77**).

Typing in M-Codes

You can manually type in M-Codes listed in the table. Refer to Table 7-5, Edit Help M-Code Listing. Most of these M-Codes (except M30 and M98) do not require additional parameter settings.

For M-Codes that do require additional parameter settings (**M30** and **M98**), Edit Help displays the Help Graphic for the M-Code. Type the required parameters and press **Use** (**F10**) to insert the M-Code into the program.

1. From the Main Edit Help screen or from a Help Template Menu, type the entire M-Code. (Example: **M2**, etc.) The Edit Help displays the typed M-Code.

NOTE:	If the M-Code requires a parameter, the software displays the Help Graphic for the typed M-Code. Only **M30** and **M98** require parameters. Fill in the entry fields for these M-Codes.

2. Press **Use** (**F10**) to enter the block into the program. Press **Cancel** (**F9**) to cancel your entry and clear the screen. Accepted commands are inserted in the program.

All rights reserved. Subject to change without notice.

Examples of G-Code Help Screens

Some examples of the G-Code Help screens are illustrated below. For example, from Milling and Profiles select Face Mill Cycle (**G170**) to display the Help screen (refer to **Figure 7-5**):

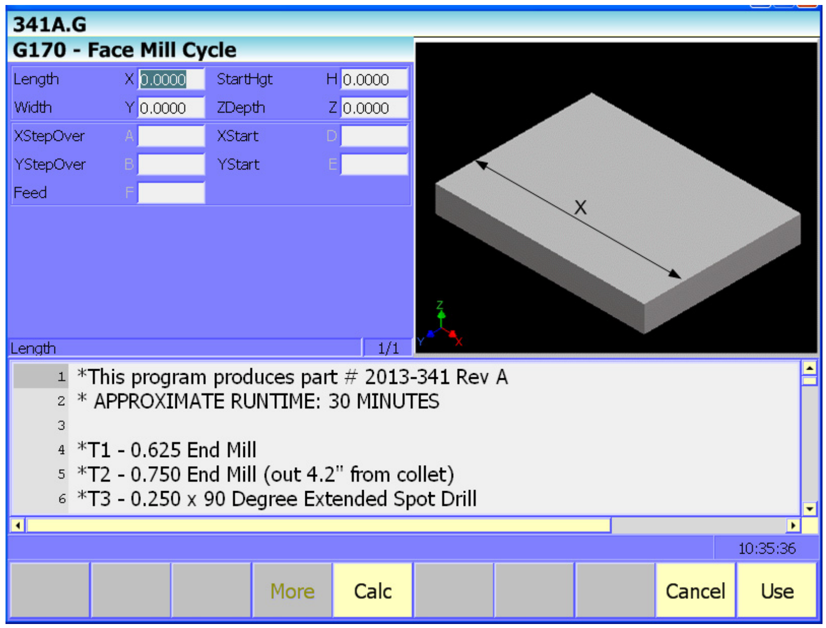


Figure 7-5, Face Mill Cycle Screen

From Milling and Profiles select Circular Profile Cycle (**G171**) to display the Help screen (refer to **Figure 7-6**):

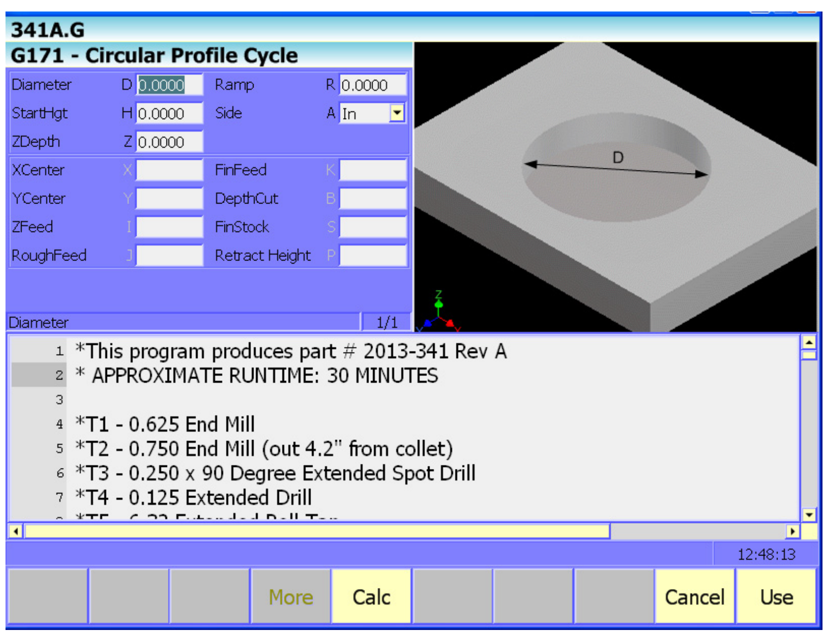


Figure 7-6, Circular Profile Cycle Screen

From Milling and Profiles, select Rectangular Profile Cycle (**G172**) to display the Help screen (refer to **Figure 7-7**):)

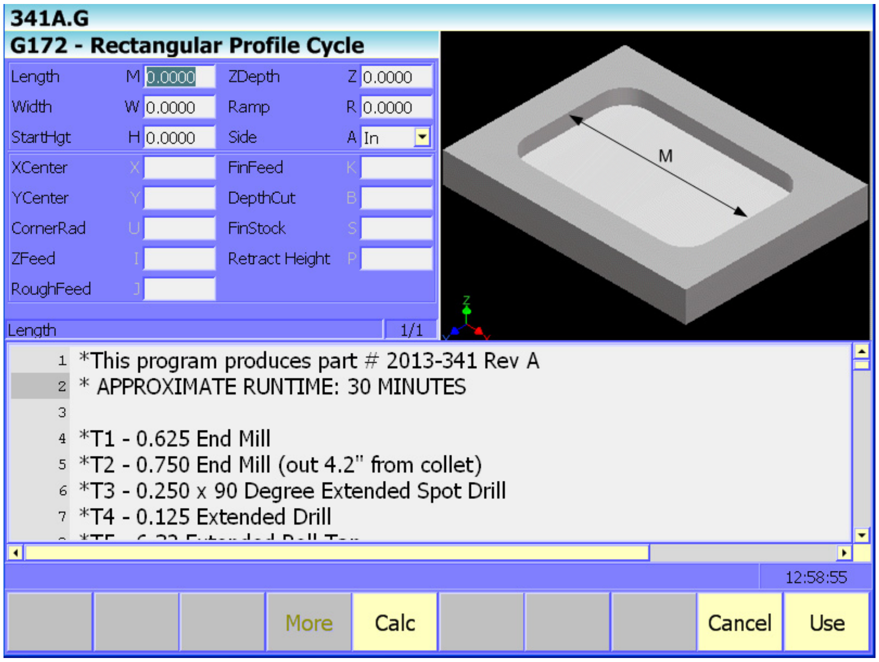


Figure 7-7, Rectangular Profile Cycle Screen

From Milling and Profiles, select Mill Cycle (**G175**) to display the Help screen (refer to **Figure 7-8**):

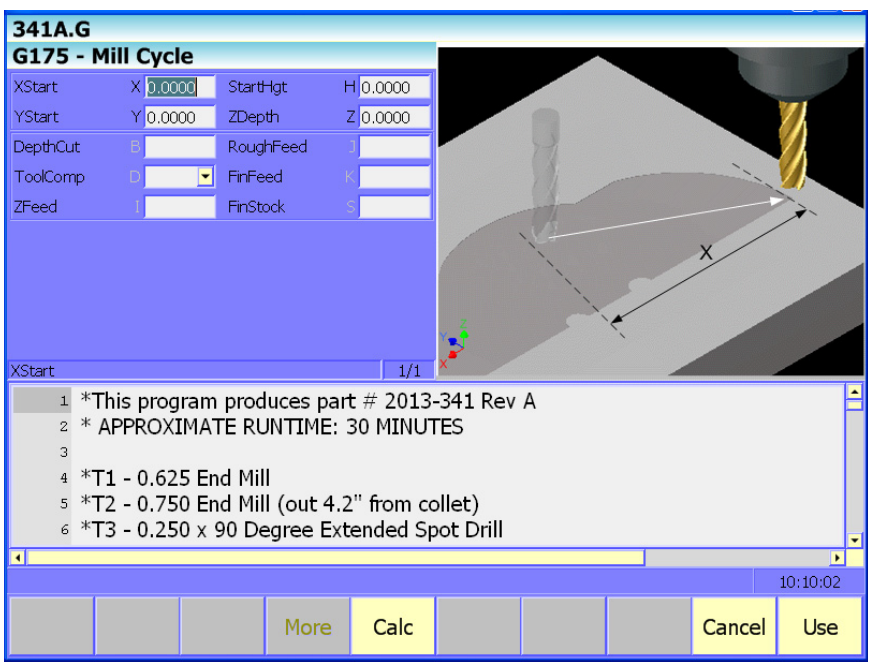


Figure 7-8, Mill Cycle Screen

All rights reserved. Subject to change without notice.

From Milling and Profiles, select EndMill Cycle (**G176**) to display the Help screen (refer to **Figure 7-9**):

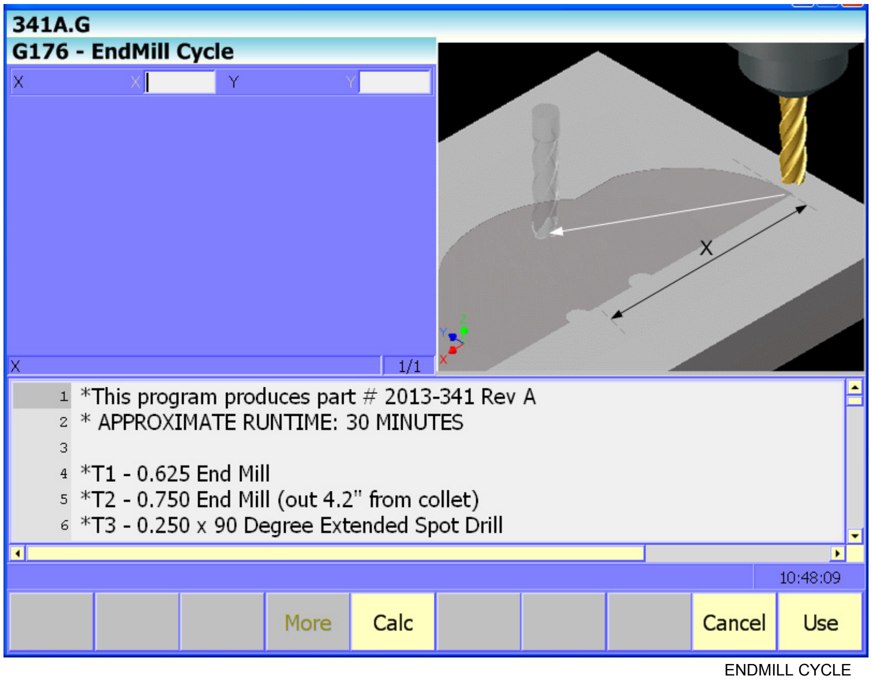


Figure 7-9, EndMill Cycle Screen

From Milling and Profiles, select Thread Mill Cycle (**G181**) to display the Help screen (refer to **Figure 7-10**):

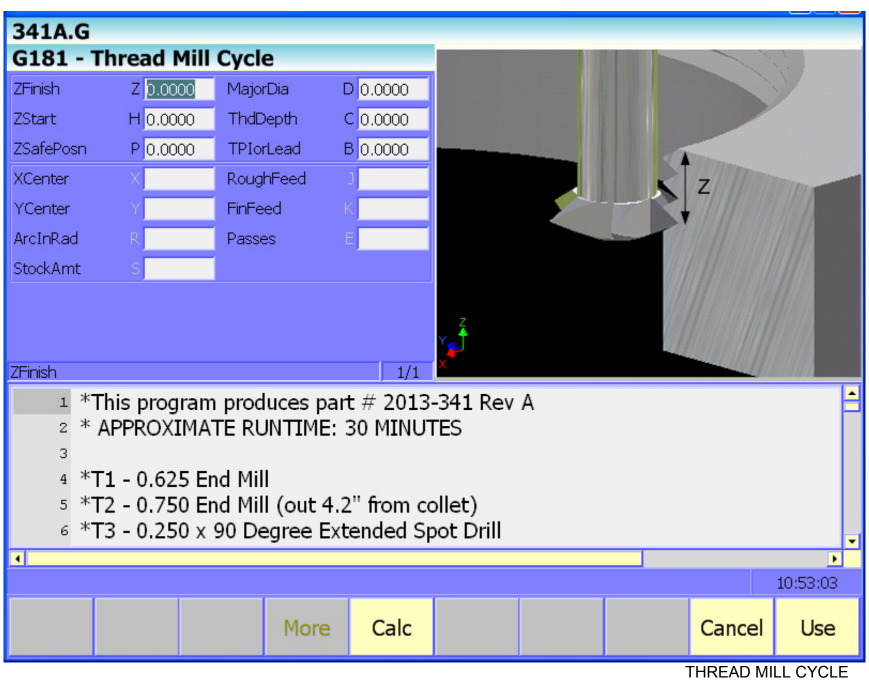


Figure 7-10, Thread Mill Cycle Screen

All rights reserved. Subject to change without notice.

From Milling and Profiles, select Engrave Cycle (**G190**) to display the Help screen (refer to **Figure 7-11**):

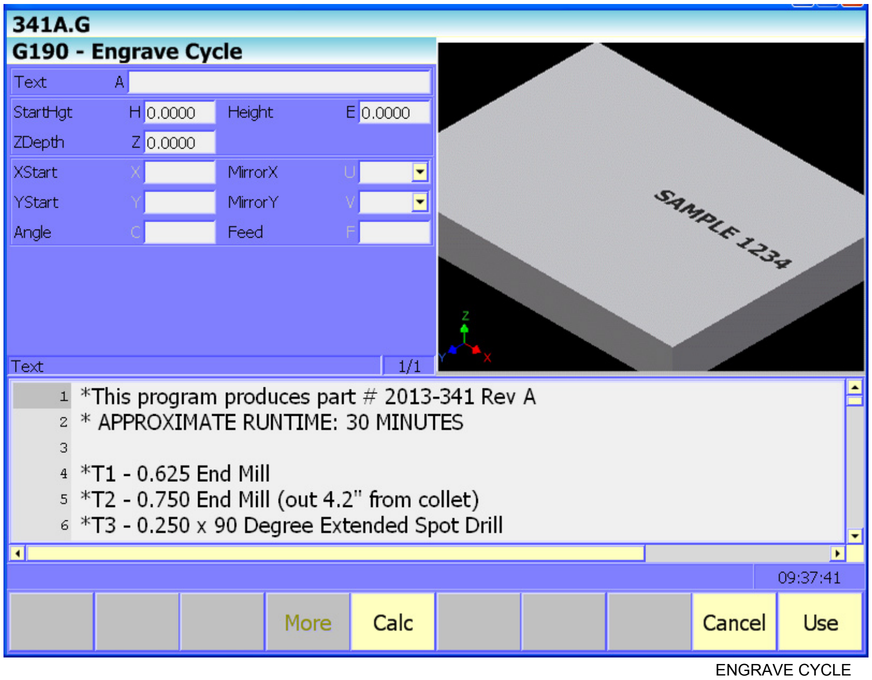


Figure 7-11, Engrave Cycle Screen

From Drilling Cycles, select Basic Drill Cycle (**G81**) to display the Help screen (refer to **Figure 7-12**):

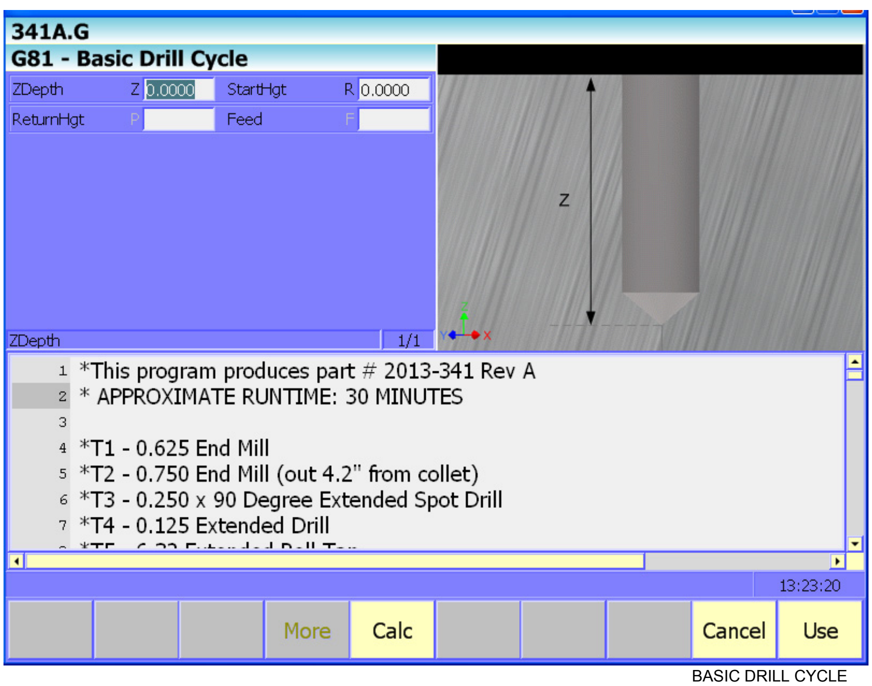
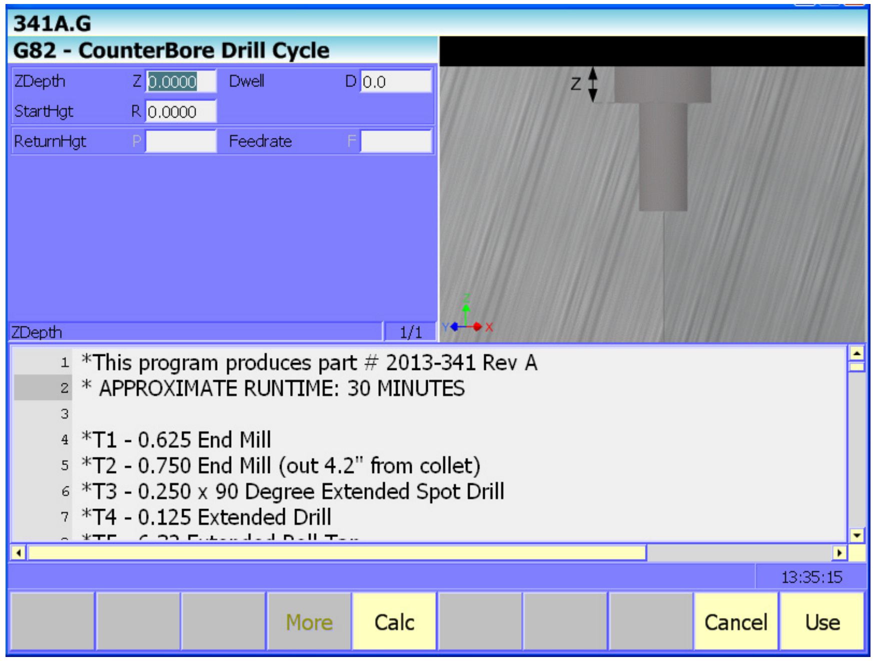


Figure 7-12, Basic Drill Cycle Screen

All rights reserved. Subject to change without notice.

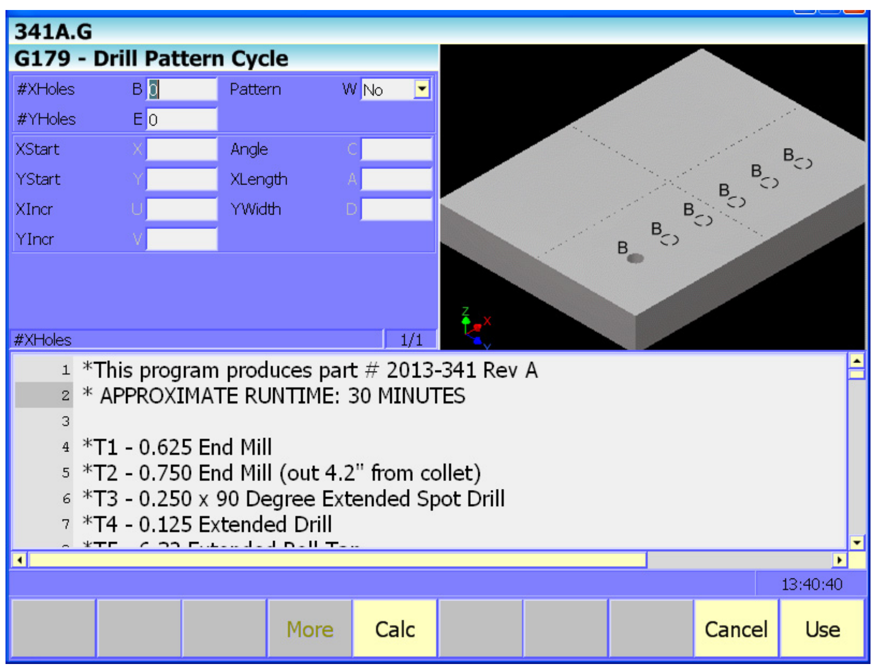
From Drilling Cycles, select CounterBore Drill Cycle (**G82**) to display the Help screen (refer to **Figure 7-13**):



COUNTER BORING CYCLE

Figure 7-13, CounterBore Drill Cycle Screen

From Drilling Cycles, select Drill Pattern Cycle (**G179**) to display the Help screen (refer to **Figure 7-14**):

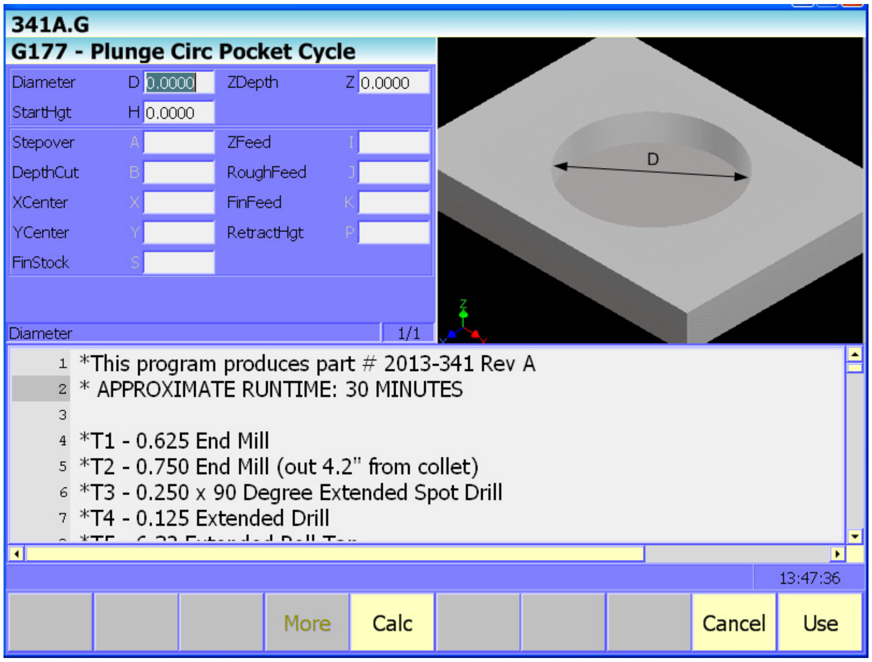


DRILL PATTERN CYCLE

Figure 7-14, Drill Pattern Cycle Screen

All rights reserved. Subject to change without notice.

From Pocket Cycles, select Plunge Circ Pocket Cycle (**G177**) to display the Help screen (refer to **Figure 7-15**):



PLUNGE CIRC POCKET CYCLE

Figure 7-15, Plunge Circ Pocket Cycle Screen

From Pocket Cycles, select Plunge Rect Pocket (**G178**) to display the Help screen (refer to **Figure 7-16**):

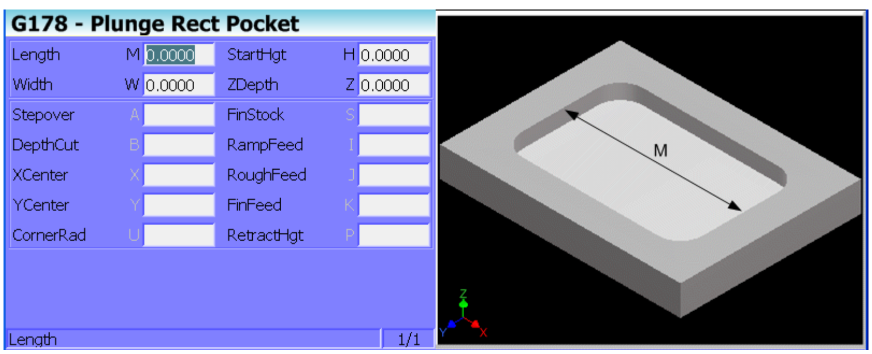


Figure 7-16, Plunge Rect Pocket Screen

All rights reserved. Subject to change without notice.

Section 8 - Viewing Programs with Draw

Draw Graphics (part graphics) is a method by which to prove a program before you cut any material. It allows you to view the part edge and/or tool path from different angles, inspect the moves the machine is programmed to make, without necessarily moving the axes. This reduces waste and the chance of damaging a part.

The CNC has two Draw Modes: Draw Simulation Mode and Real-Time Draw Mode. This section explains how to use Draw Simulation Mode to view programs. It also explains how to set the display for a detailed inspection of the programmed moves.

NOTE:	Draw (lowercase letters with an uppercase **D**) refers to the CNC's Draw Simulation Mode; DRAW (all uppercase letters) refers to the CNC's Real-Time Draw Mode.

- In Draw Simulation Mode, the CNC runs programs and simulates machine movements in the viewing area. The machine does not move.
- In the Real-Time DRAW Mode, the CNC displays the machine moves in the viewing area as it runs the program in Auto or Single Step Mode.

Refer to "Section 11 - Running Programs" for instructions on how to run DRAW while cutting a part.

When you select **Draw** (**F7**) from the Program Manager, Draw Simulation Mode is activated. The CNC draws the part without machine movement. When you start DRAW from Auto or S.Step Modes, Real-Time Draw Mode is activated. The CNC draws the part while it is machining it.

The following topics are described in this section:

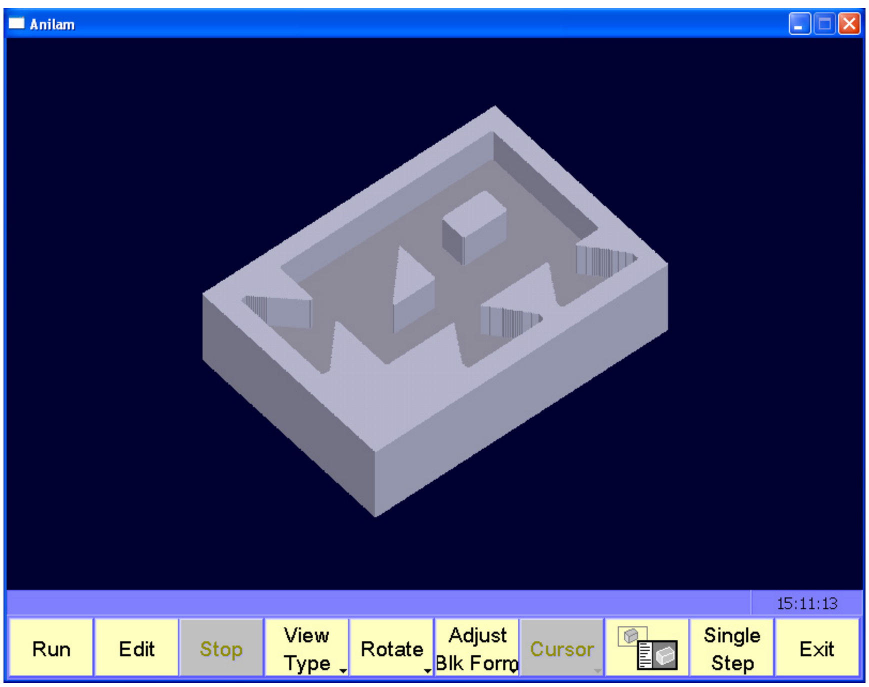
Starting Draw

Draw Simulation Mode is started from the Program Manager. You can make some changes from the soft keys while a simulation is running. In Draw Simulation Mode, the CNC does not hold the operation of the program for Dwells and tool mounts and other machine related features.

NOTE: G120 (BlockForm) must be defined in the program that is using Draw and a tool with a diameter defined must be active in the program for Draw to work.

To activate Draw Simulation Mode:

1. In the Program Manager, highlight a program and press **Draw** (**F7**). The Draw graphic screen activates.
2. Press **Run** (**F1**). Refer to **Figure 8-1**. Refer to Table 8-1, Draw Screen Soft Keys for a description of the Draw screen soft keys.
3. Draw runs the highlighted program and the machine remains idle.



DRAW1

Figure 8-1, Draw Screen

All rights reserved. Subject to change without notice.

Table 8-1, Draw Screen Soft Keys

Label	Soft Key	Soft Key Label and Function
Run	**F1**	Run the program and start Draw Simulation Mode.
Edit	**F2**	Edit the program
Stop	**F3**	Stop the program
View Type	**F4**	Open the type view screen. See Figure 8-3, View Type Screen.
Rotate	**F5**	Open the rotate screen. See Figure 8-4, Rotate Screen.
Adjust Blk Form	**F6**	Changes the work piece size to simulate a cut away feature. See Figure 8-5, Adjust Block Form Screen.
Cursor	**F7**	Change a cross-section in 3 View Type only.
Display Program	**F8**	Displays program and dashboard screen. See **Figure 8-2**.
Single Step	**F9**	Execute the program one block at a time
Exit	**F10**	Return to Program screen

Press **Display Program** (**F8**) to open the Display Program and dashboard screen. Refer to **Figure 8-2**.

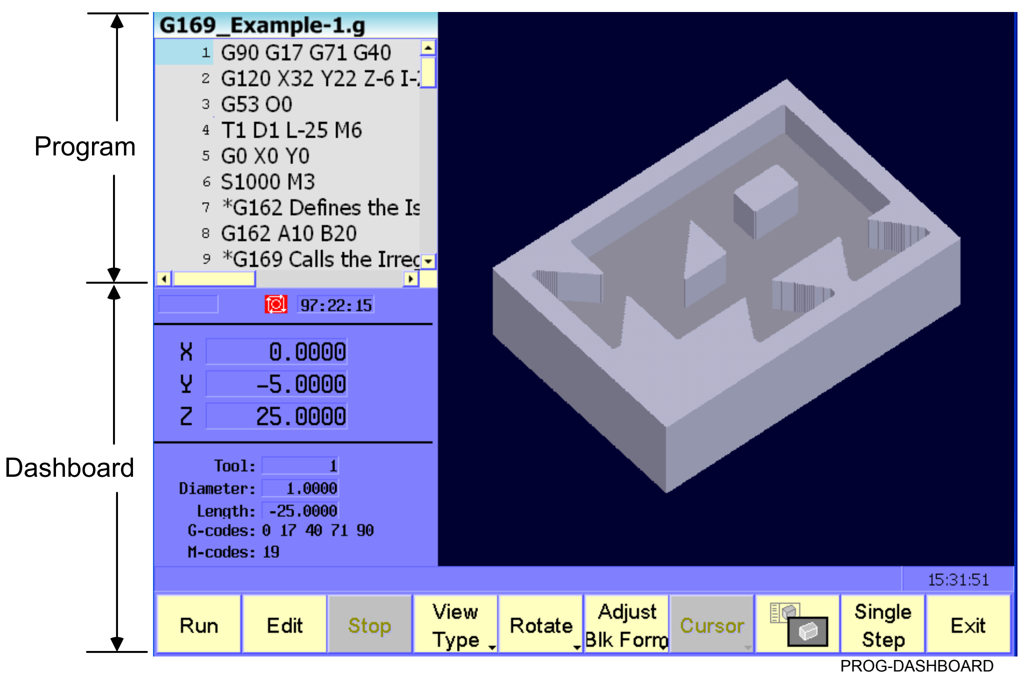
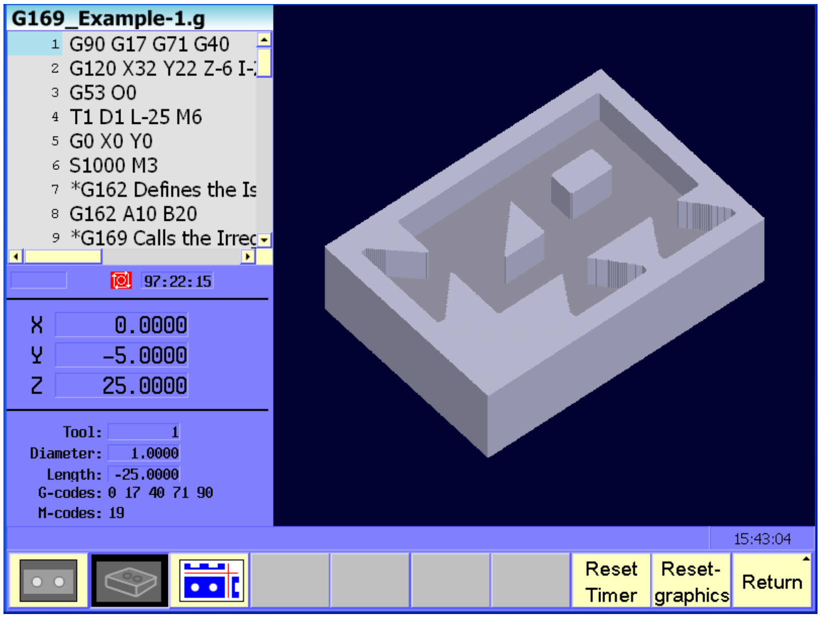


Figure 8-2, Display Program Screen

All rights reserved. Subject to change without notice.

Press **View Type** (**F4**) on the Draw screen to open the View Type screen. Refer to **Figure 8-3**. Refer to **Table 8-2** for a description of the View Type screen soft keys.



DRAW3

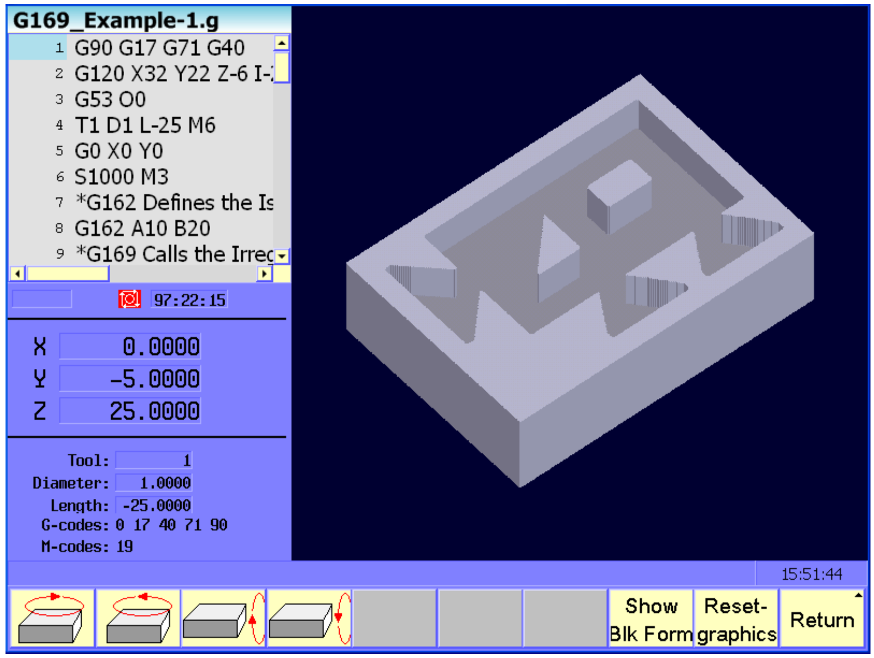
Figure 8-3, View Type Screen

Table 8-2, View Type Screen Soft Keys

Label	Soft Key	Soft Key Label and Function
Plain View	**F1**	Plain View – no features
3-D View	**F2**	3-D View – rotate and adjust view form features
3 View Type	**F3**	3 View – adjust view form and cursor features
Reset Timer	**F8**	Resets the timer
Reset graphics	**F9**	Resets the graphic
Return	**F10**	Return to Program screen

All rights reserved. Subject to change without notice.

Press **Rotate** (**F5**) on the Draw screen to open the Rotate screen. Refer to **Figure 8-4**. Refer to **Table 8-3** for a description of the Rotate screen soft keys.



ROTATE

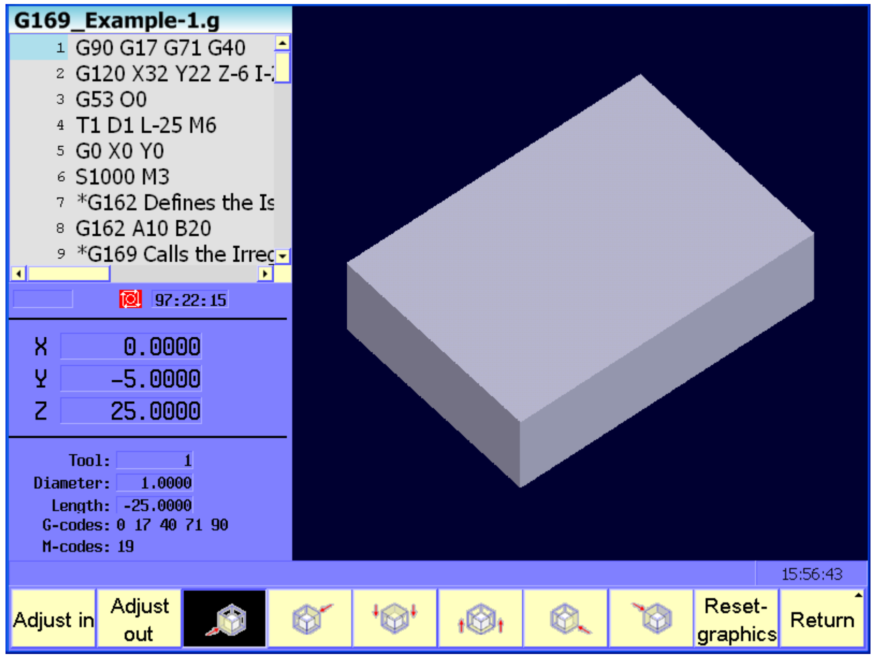
Figure 8-4, Rotate Screen

Table 8-3, Rotate Screen Soft Keys

Label	Soft Key	Soft Key Label and Function
Rotate Right	**F1**	Rotate the image right
Rotate Left	**F2**	Rotate the image left
Rotate Up	**F3**	Rotate the image up
View Down	**F4**	Rotate the image down
Show Blk Form	**F8**	Outline the block form in red
Reset graphics	**F9**	Resets the graphic
Return	**F10**	Return to View Type screen

All rights reserved. Subject to change without notice.

Press **Adjust Blk Form** (**F6**) on the Draw screen to open the Adjust Blk Form screen. Refer to **Figure 8-5**. Refer to **Table 8-4** for a description of the View Type screen soft keys.



ADJUST BLK FORM

Figure 8-5, Adjust Block Form Screen

Table 8-4, Adjust Block Form Screen Soft Keys

Label	Soft Key	Soft Key Label and Function
Adjust in	**F1**	Adjust the block form – in
Adjust out	**F2**	Adjust the block form – out
	F3	Adjust the block form – left side
	F4	Adjust the block form – right side
	F5	Adjust the block form – top
	F6	Adjust the block form – bottom
	F7	Adjust the block form – front
	F8	Adjust the block form – back
Reset graphics	**F9**	Resets the graphic
Return	**F10**	Sets and applies the block form adjustments

All rights reserved. Subject to change without notice.

Use the Draw screen to change the operation mode:

- **Run** (**F1**) [Auto Mode (the default operation mode)] runs the entire program without pause.
- Select **Single Step** (**F9**) to run the program in Single Step. Single Step Mode executes the program one block at a time. Toggle **F9** to return to Auto Mode.

Draw Screen Description

Information is displayed on the screen. In the Dashboard on the left side of the screen, axis position, Tool#, Diameter, Length, G-Code, and M-Code are displayed. Refer to Figure 8-2, Display Program and Dashboard Screen.

Exiting Draw

To exit Draw and return to the Program Manager, press **Exit** (**F10**).

All rights reserved. Subject to change without notice.

Section 9 - Tool Page and Tool Management

The Tool Page stores data on tools, such as: tool number, diameter, length offset, diameter wear, length wear, and tool type. Refer to **Figure 9-1**. For a description of the Tool Page soft keys, see Table 9-1, Tool Page Soft Keys.

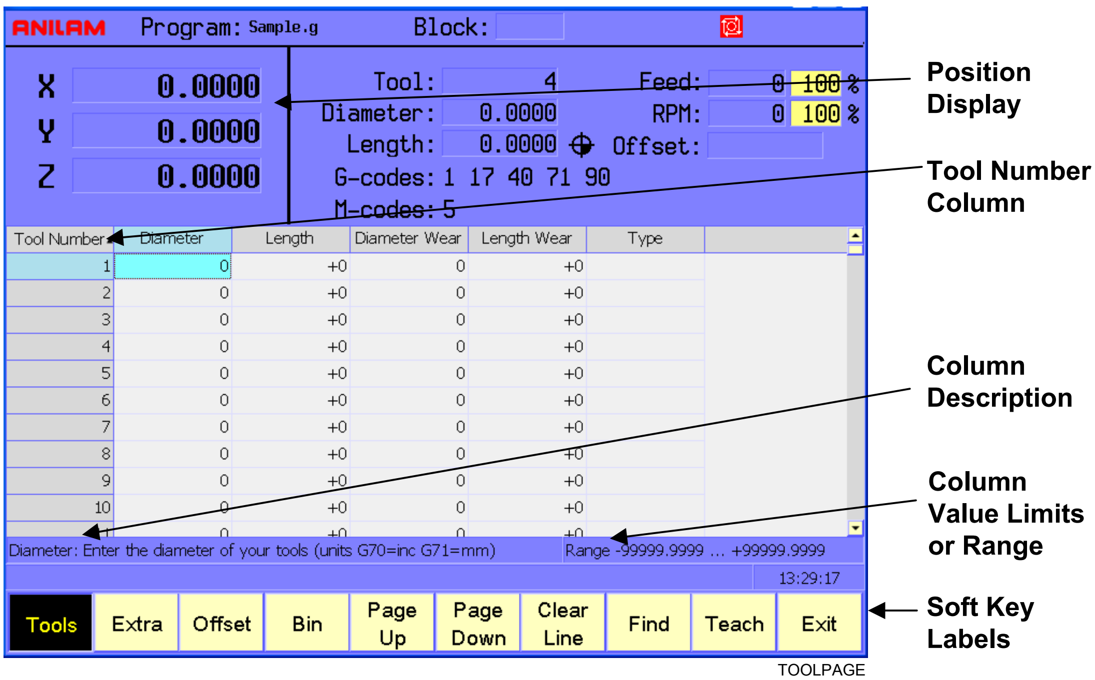


Figure 9-1, The Tool Page

Other functions related to the Tool Page may be accessed using the **SHIFT** key. These functions are described in this section. Press the **SHIFT** key to redisplay the Tool Shift screen. Refer to Figure 9-2, Shift Screen from Tool Page.

All rights reserved. Subject to change without notice.

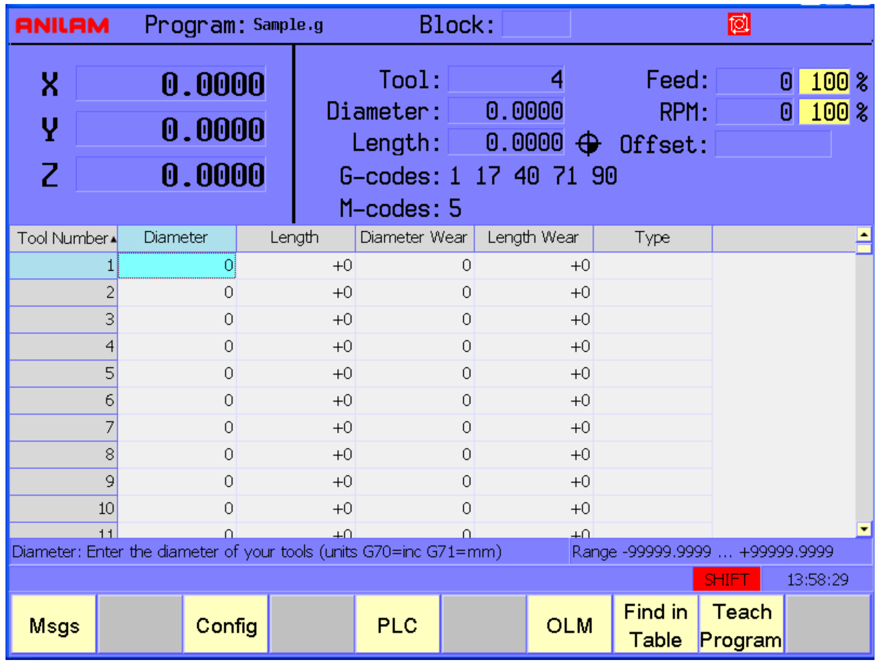


Figure 9-2, Shift Screen from Tool Page

For a description of the Tool Page soft keys, see Table 9-2, Tool Page Secondary Soft Keys.

The following topics are described in this section:

- **Activating the Tool Page**
- Using the Tool Page
- Finding Tools by Number
- Changing Tool Page Values
- Tool Page Soft Keys and Secondary Soft Keys
- T-Codes and Tool Activation
- Tool-Length Offsets
- Diameter Offset in Tool Page
- Compensation (G40, G41, G42)
- Activating Offsets via the Program

Activating the Tool Page

Activate the Tool Page as follows:

1. Go to the Manual screen.
2. In the Manual screen, press **TOOL** (**F9**). The Tool Page activates (refer to Figure 9-1, The Tool Page).
3. Press the **SHIFT** key on the keyboard to display the Shift screen from Tool Page (refer to **Figure 9-2**). The **SHIFT Message** (**F1**) functions are described in "Section 3, Messages (Msgs) (SHIFT + F1)." Press **SHIFT** again to return to the Tool Page.

All rights reserved. Subject to change without notice.

Using the Tool Page

Press **UP** and **DOWN ARROWS** to highlight and select tool numbers (row numbers). You can type tool information only in a highlighted row. Highlight the appropriate row and column to enter the selected fields, and type the values. The cursor marks the location of information to be typed.

Numbered rows correspond to tool numbers. When the CNC executes a program block that activates a tool number, the values on that row of the Tool Page are activated.

Press **RIGHT** and **LEFT ARROWS** to move from column to column. Tool Page values are automatically converted to their inch or millimeter equivalents when you change the CNC's unit mode. All typed values must match the CNC's current unit mode.

NOTE:	The Tool Page is the only place where the CNC converts values from Inch Mode to MM Mode, and vice-versa. Programmed positions are not converted when you change the unit mode.

Press **Page Up** (**F5**) or **Page Down** (**F6**) to scroll through the tool table one page at a time.

When you activate Tool #0, you cancel the active tool diameter and length offset of the CNC. The Tool #0, Z0 position is usually set as the fully retracted position of the Z-axis.

All of the CNC's Jog features can be run from the Tool Page. The handwheels (if installed) can also be used if activated prior to entering the Tool Page.

The Figure 9-1, The Tool Page labels:

Position Display	Displays information regarding current machine position and active Units Mode (Inch/MM).
Tool Number Column	The tool numbers are listed in the first column (CNC provides).
Column Description	Displays a brief description of the column on which the cursor is located.
Column Value Limits or Range	Displays the column value type or limits range.
Soft Key Labels	Identify the functions of the active soft keys.

All rights reserved. Subject to change without notice.

The following tool attributes display on the Tool Page:

Tool Number	Row Numbers link the values on a row of the Tool Page to a tool number. A program block that activates a tool number activates the values and settings on that row of the Tool Page (CNC provides).
Diameter	Tool diameter applied when you activate tool diameter compensation or use some canned cycles.
Length	Tool-length offset, which enable the CNC to adjust the Z-axis tool tip reference position.
Diameter Wear	Diameter wear offset, which compensate for wear on the tool diameter or an incorrectly sized tool.
Length Wear	Length wear offset, which compensate for wear on the tool length or an incorrectly sized tool.
Type	Type any character or press **ENTER** to display the options: Milling cutter, Touch probe, or Undefined. Use the **UP** and **DOWN ARROWS** to highlight a type and press **ENTER** to select the type.

Finding Tools by Number

To find a specific tool number in the Tool Page:

1. Press **Find** (**F8**). The CNC display the "Find Tool #:" line below the Column Description. Refer to Figure 9-3, Find (F8) from Tool Page.
2. Type a tool number in the "Find Tool #:" line that you want to locate and press **ENTER**. The cursor moves to the selected tool number.

All rights reserved. Subject to change without notice.

Changing Tool Page Values

1. In the Tool Page, highlight the desired row. Position the cursor on the desired column.

CAUTION:	**Ensure that the CNC is in the same unit mode, MM or Inch, as the value you enter. To verify the unit setting, look at the G-Code area of the Tool Page where either G70 (Inch) or G71 (MM) is displayed.**

2. Type the new value with all appropriate decimal values and press **ENTER** (or press any **ARROW**). The value will be entered.

The following topics are described:

- **Clearing a Tool (Whole Row)**
- **Clearing a Single Value**
- **Adjusting a Single Value**

Clearing a Tool (Whole Row)

To clear a row:

1. In the Tool Page, highlight the row to be cleared.
2. Press **Clear Line** (**F7**). All values in the row return to zero.

Clearing a Single Value

To clear a single value:

1. In the Tool Page, highlight an entire row.
2. Position the cursor on the value you wish to clear and press the **CLEAR** key (on auxiliary keyboard press **DELETE** or (**ALT** + **C**)). The value changes to zero.

Adjusting a Single Value

To adjust a single value:

1. In the Tool Page, highlight the desired row. Position the cursor on the desired column.
2. Press the letter **A** key to display the message, "**Add value**."
3. Type the amount of the adjustment. The adjustment value may be positive or negative.
4. Press **ENTER** to adjust the value, and display the adjusted value on the table.

All rights reserved. Subject to change without notice.

Tool Page Soft Keys and Secondary Soft Keys

Refer to **Table 9-1**.

Table 9-1, Tool Page Soft Keys

Label	Soft Key	Function
Tools	**F1**	The Tools soft key label is highlighted for standard tool.
Extra	**F2**	Provides access to additional tool attributes. Refer to "Extra Tool Information."
Offset	**F3**	Enables entry to the G53 Offset pop-up menu. Refer to "Section 4, Fixture Offsets (Work Coordinate System Select), (G53)" and "Offset Tool Information."
Bin	**F4**	Soft key displays only when machine is equipped with a random tool changer. Provides access to the pocket table. Refer to "Bin Tool Information."
Page Up	**F5**	Moves the cursor one page backward.
Page Down	**F6**	Moves the cursor one page forward.
Clear Line	**F7**	Clears the entire single line. Refer to "Clearing a Tool (Whole Row)."
Find	**F8**	Enables "search" of a tool number. Refer to "Finding Tools by Number" and "Find Tool Number."
Teach	**F9**	Sets the current machine Z axis value into the tool length column of the highlighted row. On the offset table, however, the machine axis that is copied to the table depends on which column is highlighted.
Exit	**F10**	Exits the Tool Page. (The changes are saved as they are completed.)

All rights reserved. Subject to change without notice.

Press **SHIFT** while in the Tool Page to activate the secondary soft key functions (refer to Figure 9-2, Shift Screen from Tool Page). Refer to **Table 9-2**.

Table 9-2, Tool Page Secondary Soft Keys

Label	Soft Key	Function
Msgs	(**SHIFT** + **F1**)	Displays messages, prompts, and reminders. Refer to "Section 3, Messages (Msgs) (SHIFT + F1)."
Config	(**SHIFT** + **F3**)	Provides access to machine parameters.
PLC	(**SHIFT** + **F5**)	Programmable Logic Controller (PLC) Refer to "PLC and OLM Descriptions."
OLM	(**SHIFT** + **F7**)	On-line Monitor (OLM) Refer to "PLC and OLM Descriptions."
Find in Table	(**SHIFT** + **F8**)	Enables "search" of the tool table. It enables you to search for content inside the table. Refer to "Find in Table."
Teach Program	(**SHIFT** + **F9**)	Sets the current program Z axis value into the tool length column of the highlighted row. On the offset table, however, the program axis that is copied to the table depends on which column is highlighted.

The following topics are described:

- Extra Tool Information
- Bin Tool Information
- Offset Tool Information
- Find Tool Number
- Find in Table
- PLC and OLM Descriptions

All rights reserved. Subject to change without notice.

Extra Tool Information

On the Tool Screen (refer to Figure 9-1, The Tool Page), press **Extra** (**F2**) to display the Extra screen. The **Extra** (**F2**) soft key highlights and new screen field attributes display which can be optionally set. These screen field attributes may be necessary for specific canned cycles.

The **Extra** (**F2**) screen field attributes follow:

Tool Number	Row Numbers link the values on a row of the Tool Page to a tool number. A program block that activates a tool number activates the values and settings on that row of the Tool Page. (CNC provides)
Tool Age	Time the tool has been in use.
Max Length	Maximum length of tool to be used for roughing cycle. (Default = 0)
Plunge Angle	Plunge angle (used by some canned cycles)
Comment	You type any comment you want (for user reference only). (up to 17 alphanumeric characters)

Bin Tool Information

Soft key displays only when machine is equipped with a random tool changer. On the Tool Screen (refer to Figure 9-1, The Tool Page), press **Bin** (**F4**) to display the Bin screen or pocket table. The **Bin** (**F4**) soft key highlights and new screen field attributes display which can be optionally set. These screen field attributes may be necessary for random style tool changers.

The **Bin** (**F4**) screen field attributes follow:

Bin Number	Bin number (CNC provides)
Tool Number	Row Numbers link the values on a row of the Tool Page to a tool number. A program block that activates a tool number activates the values and settings on that row of the Tool Page. (CNC provides)
Reserved	Press **ENTER** to assign R (Reserved) to the Bin number. Press **ENTER** again to clear the Reserved field.
Fixed Tool	Press **ENTER** to toggle between "Not Fixed" and "Fixed". When a tool is fixed, it is assigned to a specific pocket.
Bin Locked	Press **ENTER** to toggle between "Not Locked" and "Locked".

All rights reserved. Subject to change without notice.

Offset Tool Information

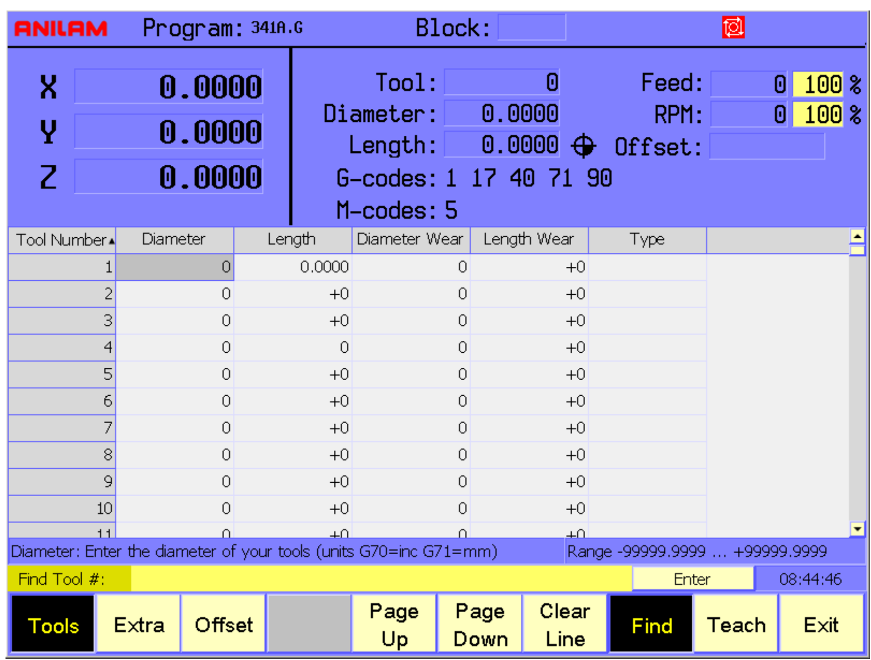
On the Tool Screen (refer to Figure 9-1, The Tool Page), press **Offset** (**F3**) to display the Offset screen. The **Offset** (**F3**) soft key highlights and new screen field attributes display which can be optionally set. These screen field attributes are necessary for fixture offsets (G53).

The **Offset** (**F3**) screen field attributes follow:

Offset	Fixture offset number
X Axis	X-axis shift from Machine Home
Y Axis	Y-axis shift from Machine Home
Z Axis	Z-axis shift from Machine Home

Find Tool Number

On the Tool Screen (refer to Figure 9-1, The Tool Page), press **Find** (**F8**) to display the "Find Tool #:" line below the Column Description. Type in the "Find Tool #:" line the tool number that you want to locate. Press **ENTER** on the right of the "Find Tool #:" line to start your search. Refer to **Figure 9-3**.



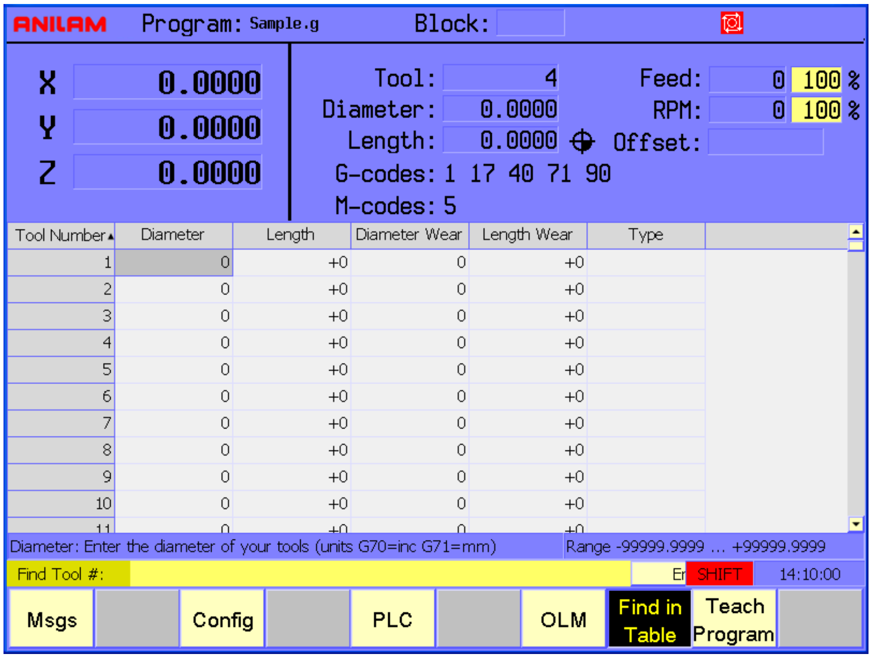
TOOLFIND

Figure 9-3, Find (F8) from Tool Page

All rights reserved. Subject to change without notice.

Find in Table

On the **SHIFT** Tool Screen (refer to Figure 9-2, Shift Screen from Tool Page), press **Find in Table** (**SHIFT** + **F8**) to display the "Find in Table:" line below the Column Description. Type in the "Find in Table:" line the table content that you want to locate. Press **ENTER** on the right of the "Find in Table:" line to start your search. Refer to **Figure 9-4**.



FIND IN TABLE

Figure 9-4, Find in Table (F8) from Shift Tool Page

PLC and OLM Descriptions

Refer to *6000i CNC Technical Manual*, P/N 627787-21, for PLC and OLM descriptions.

PLC (**SHIFT** + **F5**) Refer to "Section 7, PLC Functions, Selecting the PLC Mode" in P/N 627787-21

OLM (**SHIFT** + **F7**) Refer to "Section 6, Diagnosis with the Online Monitor (OLM)" in P/N 627787-21

All rights reserved. Subject to change without notice.

T-Codes and Tool Activation

To activate a tool, program a T-Code followed by the tool number. The tool number corresponds to the row of the Tool Page that contains the Tool-Length Offsets (TLOs) and other required values for the active tool.

Format: Txx

Two-digit T-Codes are used if the machine tool is not equipped with an automatic tool changer (ATC). If the machine is equipped with an ATC, then the four-digit T-Code system can be used.

The following topics are described:

- **Tool Definition Blocks**

Tool Definition Blocks

Example: N3 T1 R1.25 L-1 M6

A tool definition block defines the tool radius in the program, rather than via the Tool Page. The block assigns Tool 1:

- Tool radius of 1.25 (diameter equals 2.50)
- Tool length of -1
- **M6** is the tool activation

All rights reserved. Subject to change without notice.

Tool-Length Offsets

Tool-length offset (TLO) is the distance from Z0 Machine Home to the tip of the tool at the part Z0 (usually the surface of the work). Refer to **Figure 9-5**.

Tool-length offsets allow each tool used in the part program to be referenced to the part surface. In an idle state, the CNC does not have a tool-length offset active. Therefore, Tool #0 (T0) is active. When T0 is active, all Z dimensions are in reference to the Z Home position. When you program T1, all Z dimensions become referenced to the surface on which the tool-length offset of Tool #1 was activated.

For machines that do not have a Z-axis automatic homing feature, you must set the Z0 position of the Z-axis. Usually, it is the fully retracted (Up) position of the quill or machine head. Tool-Length Offsets are referenced to this position.

Because tools differ in length, Z0 axis (Part Zero) is not set the same way as X0 or Y0. The tool-length offset is the distance from the tip of the tool to the top of the part. Enter a length offset for each tool in the Tool Page.

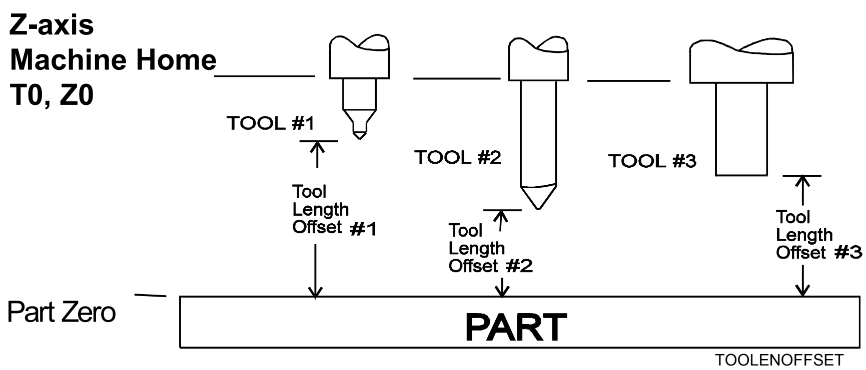


Figure 9-5, Tool-Length Offset

With tool-length offsets active, the Z-axis position display reads 0.00 when the active tool moves to Part Zero. Tool-length offsets simplify programming.

The following topics are described:

- Entering Offsets in the Tool Page
- Setting Tool-Length Offsets
- Entering the Z Position Manually

All rights reserved. Subject to change without notice.

Entering Offsets in the Tool Page

After you choose the type of tools and the order of their use in the program, and you know the diameter and length offsets of tools, type the data into the Tool Page.

1. In Manual Mode, press **TOOL** (**F9**) to open the Tool Page.
2. In the Tool Page, you must highlight a line before you can edit it.

Typically, you type diameter offsets in the Tool Page directly, after measuring the tool with a micrometer.

To measure length offset:

1. In Manual Mode, put the tool in the spindle and carefully jog the tool down until it touches surface (top of the work).
2. In the Tool Page, highlight that tool's tool number, and press **Teach** (**F9**). This will take the dimension from Z Machine Home position, and input it into the Length Offset column for that tool.
3. Exit the Tool Page, raise the Z-axis and continue.

   – or –

   Jog the tool(s) as described above, write down each offset(s) and type it into the Tool Page.

In case of errors, to identify and correct:

1. If the value entered in out of range, the field changes to a different color and you cannot exit the field.
2. Correct the value entered. A correct value (within the range) is the same color as the other fields and you can exit the field.

All rights reserved. Subject to change without notice.

Setting Tool-Length Offsets

Before you run a job in production, perform the following steps:

1. Review the part drawing.
2. Make a machining plan. Include fixturing, tooling and machine sequence.
3. Write the program on a program sheet.
4. Input the program.
5. Set the tool offsets.

The following sequence describes tool offsets for a job that center-drills, drills and mills a part:

Tools:	#1	No. 3 center drill
	#2	0.3750 diameter Twist drill
	#3	0.5000 diameter End-mill

Typically, you would perform the following steps to activate TLOs:

1. Insert and tighten all tools in their appropriate holders.
2. Set the CNC to Manual Mode.
3. Ensure the Z-axis has been homed or Machine Zero is set to its fully retracted position.
4. If the machine is a vertical knee mill, place the longest tool in the spindle and raise the knee until the tip of the tool is approximately half an inch from the top of the work.
5. Place Tool #1 in the spindle.
6. Jog the tool over the work.
7. Carefully jog the tip of the tool down to meet the top of the work. Use MDI moves and/or the Manual Panel Jog selections.
8. Press **TOOL** (**F9**) to open the Tool Page.
9. Ensure that the cursor is on Tool #1 (row 1).
10. Press **Teach** (**F9**) to input the Z value in the Length column.
11. Press **Exit** (**F10**).
12. Raise the tool from the work to Z Home (Z0).
13. Repeat Steps 7 to 12 for all tools.
14. Use a micrometer to measure tool diameters and type those values in their respective columns.
15. Press **Exit** (**F10**) to return to Manual Mode.

All rights reserved. Subject to change without notice.

Entering the Z Position Manually

1. Retract the Z-axis to the Machine T0, Z0 position.
2. Load the tool and manually position its tip at the Part Z0 position.
3. Manually type the plus or minus Z position as it displays in the position display in the Length offset column of the Tool being set and press **ENTER**.

NOTE: The value of a tool-length offset is usually a negative number.

Diameter Offset in Tool Page

When you activate a tool, you automatically activate the length offset and diameter values recorded on the Tool Page. When a tool is activated, the length offset is applied immediately to provide an accurate Z-axis position display.

The active diameter value is important when you program compensated moves and use cycles with built-in tool compensation. If tool diameter is correct, compensated moves and cycles will be executed accurately too.

Enter tool-length offsets and tool diameter values on the numbered lines of the Tool Page. The numbered lines on the Tool Page identify the tool number (T-Code) that activates those values.

You can program a tool activation as a separate block or include it within the block for most moves and cycles. Tool activation's programmed, as separate blocks are easier to find in a Program Listing.

The Tool Page can store information for up to a number of tools specified by the machine builder.

On machines equipped with collet-type tool holders, it is impractical to use the Tool Page to store tool-length offsets. You can set tool-length offset at tool change. Tool Page diameters are still required for compensated moves and when using cycles that have built-in compensation. You can run all Jog features from the Tool Page.

Tool Page offsets activate when you program a T-Code.

For example:

```
N3  T1
N4  G0 G41 XnYn
N5  etc...
```

Block N3 activates Tool #1 length offset. N4 activates tool compensation for the following blocks.

NOTE: In Block N4, the **G41** command must be accompanied by a move (XYZ) to take effect. The motion must be in rapid (**G0**) or feedrate (**G1**). The tool diameter activates when the CNC executes the move programmed on the block. **G40** and **G42** must also be accompanied by moves, and activate in the same manner.

All rights reserved. Subject to change without notice.

The following topics are described:

- **Tool Path Compensation (G41, G42)**
- Using Tool Diameter Compensation and Length Offsets with Ball-End Mills

Tool Path Compensation (G41, G42)

NOTE: Be familiar with basic CNC principles before you attempt to write compensated moves.

When tool compensation is not active, the CNC positions the tool's center on the programmed path. This creates a problem when programming a part profile because the cutting edge is half a diameter away from the path. Use tool diameter compensation to overcome this problem.

When tool compensation is active, the CNC offsets the tool by half a diameter to position the cutting edge of the tool on the programmed path.

This enables you to program the coordinates along the part profile. You do not need to adjust the path to compensate for tool diameter.

Most moves can be compensated. Specify right-hand or left-hand compensation. "Right" or "left" refers to the side of the path to which the tool offsets, as viewed from behind a moving tool. If the tool is to the left of the work, use **G41**. If the tool is to the right of the work, use **G42**.

NOTE: Use tool compensation with lines and arcs only.

With left-hand tool diameter compensation (**G41**) active, the tool offsets to the left of the programmed path (as viewed from behind a moving tool). Refer to Figure 9-6, Left Hand Tool Compensation.

All rights reserved. Subject to change without notice.

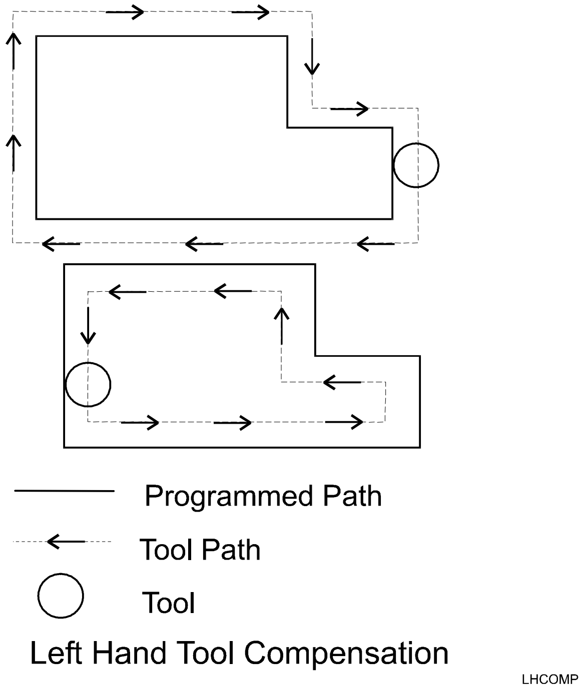


Figure 9-6, Left Hand Tool Compensation

With right-hand tool diameter compensation (**G42**) active, the tool offsets to the right of the programmed path (as viewed from behind a moving tool). Refer to **Figure 9-7**.

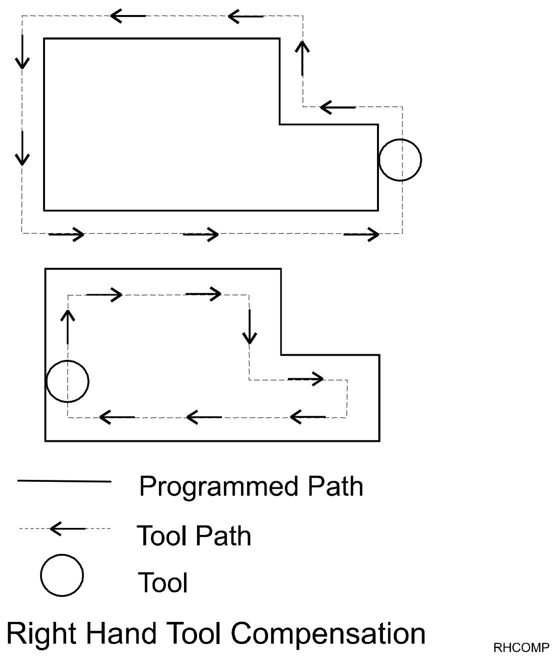


Figure 9-7, Right Hand Tool Diameter Compensation

When the CNC encounters two consecutive, compensated moves, the tool follows the offset path for the first move until it reaches the offset path for the second move. The tool may intersect the offset path for the second move, either before or after the endpoint of the first move, depending on the geometry. Refer to **Figure 9-8**.

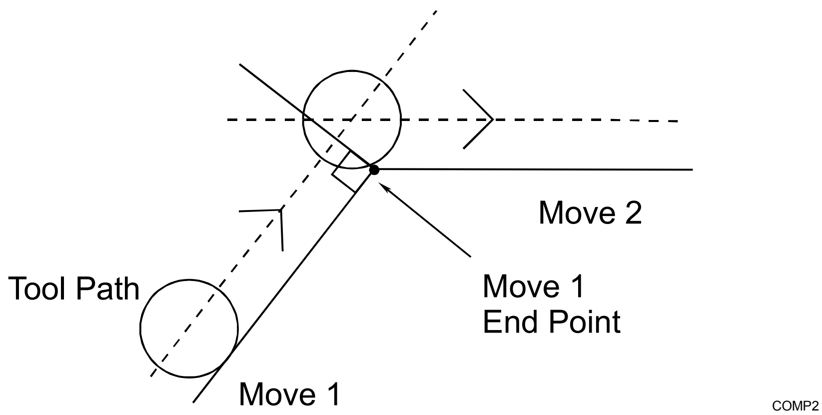


Figure 9-8, Consecutive Compensated Moves

The moves to and from compensated moves are called ramp moves. Ramp moves give the CNC time to position the tool. The ramp move must be at least half the active tool's diameter in length. Refer to Figure 9-9, Ramping into a Compensated Move.

All rights reserved. Subject to change without notice.

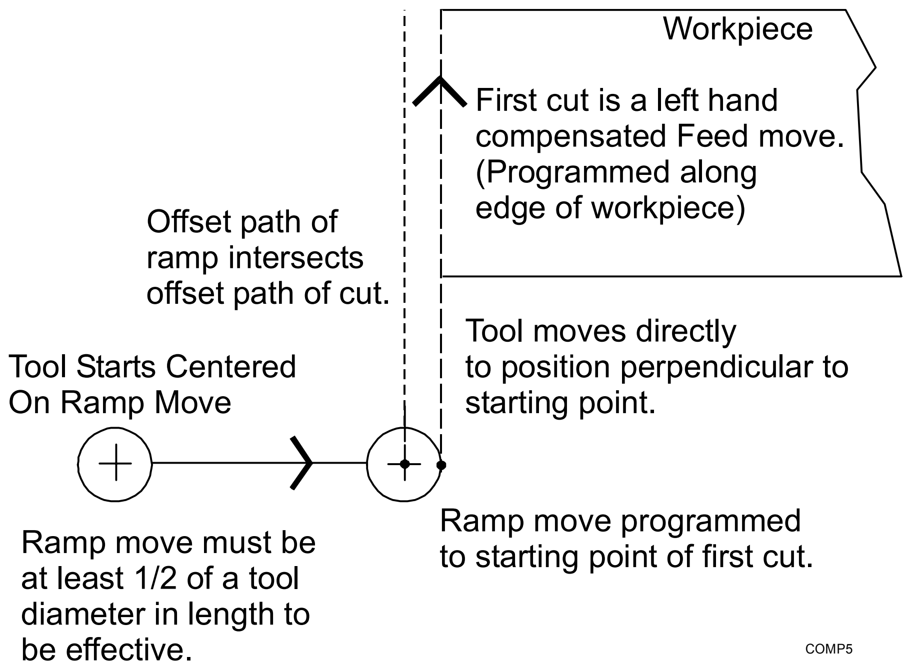


Figure 9-9, Ramping into a Compensated Move

At the start of a ramp move, the tool centers on the programmed path. At the end of the ramp move (starting point of the compensated move), the tool centers perpendicular to the starting point, offset by half the tool's diameter.

When a compensated move starts and stops in a corner, the tool gouges the work because the tool offsets to a position perpendicular to the endpoint. Begin ramp moves at the side to avoid gouging the work. Refer to Figure 9-10, Ramp On/Off Choices for Milling Inside a Square.

NOTE:	Use canned cycles to cut profiles and pockets, when possible. The CNC automatically selects Ramp On/Off positions in a canned cycle.

All rights reserved. Subject to change without notice.

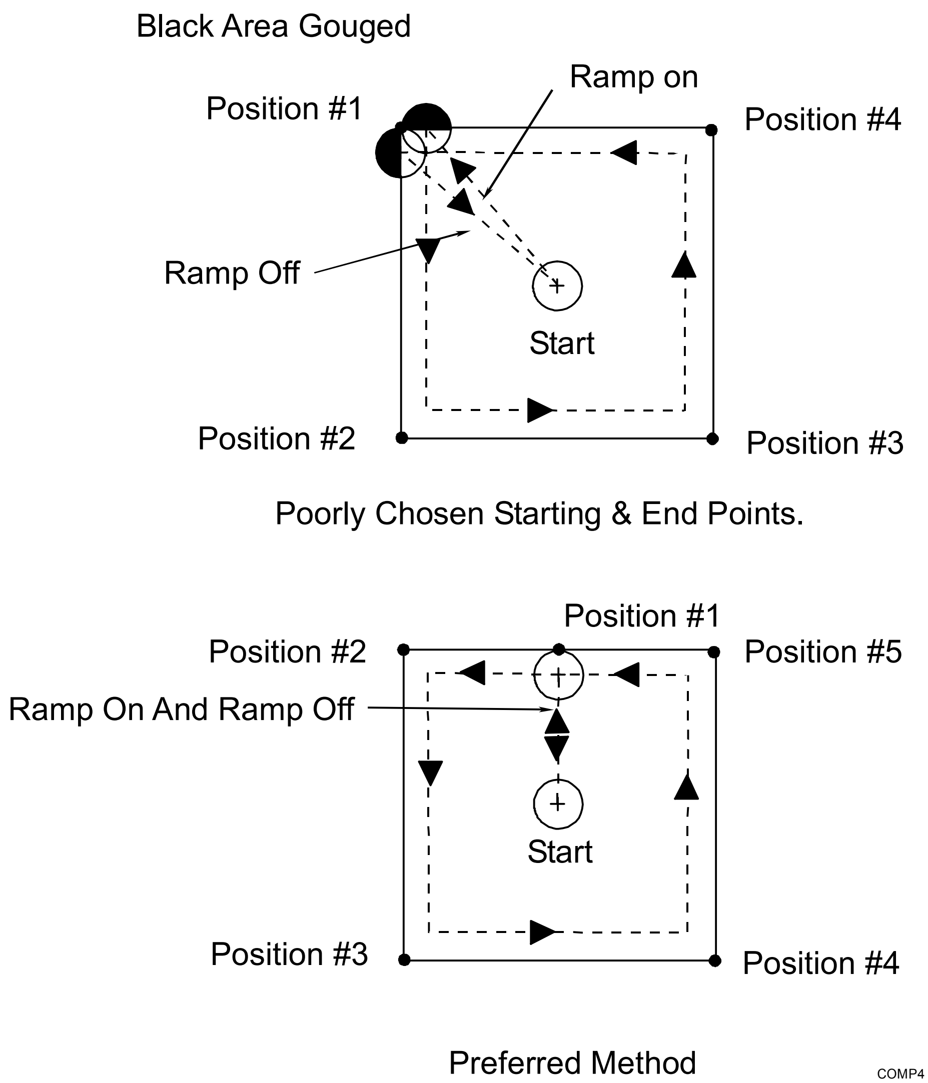


Figure 9-10, Ramp On/Off Choices for Milling Inside a Square

All rights reserved. Subject to change without notice.

Using Tool Diameter Compensation and Length Offsets with Ball-End Mills

When you use a ball-end mill to cut contoured surfaces, use tool diameter compensation and tool-length offset together, if at all. Unlike a flat-bottom tool, the tool-length offset for a ball-end mill is not necessarily set to the tip of the tool.

In most cases, set the tool-length offset for a ball-end mill half the tool's diameter back from the tip. Refer to **Figure 9-11**.

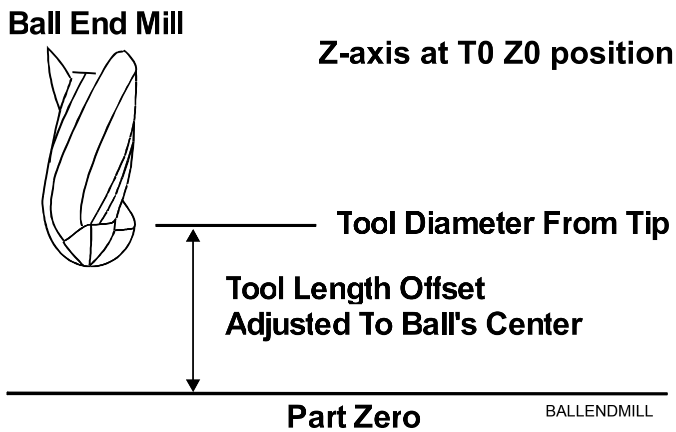


Figure 9-11, Setting Tool-Length Offset for Ball End Mill

Compensation (G40, G41, G42)

The following topics are described:

All rights reserved. Subject to change without notice.

Cancel Mode in Tool Compensation (G40)

At the end of a cutting sequence that performs tool compensation (**G41** or **G42**); you must use **G40** to cancel compensation. The following example describes a part programmed in the XY plane using **G41**.

NOTE:	You must make an X and/or Y move with or after **G40**, before changing the active tool number.

Example:

```
N4    G17 G0 G41 Xn Yn
N5    etc...
.
.
N20   G0 G40 Xn Yn
N21   etc...
```

Program **G40** on a line with **G0** or **G1** (unless **G0** or **G1** is already active). G40 programmed with or immediately following **G2** or **G3** will generate an alarm message.

Startup and Movement in Z Axis

The CNC "looks ahead" far enough to determine the next planar intersection. Z-axis moves, even many consecutive Z moves, are permitted at any time after a compensation block.

Refer to **Example 1**. N10 contains compensation block, properly accompanied by an XY move. N11 contains a Z move.

Refer to **Example 2**. N10 contains the compensation block. N11 and N12 contain two consecutive Z moves.

Example 1: Single Z move in a compensated program

```
N10   G0 G41 X0 Y-.5
N11   G1 Z-.125 F3
N12   Y3.625 F7.5
N13   X5.5
N14   etc...
```

Example 2: Two Z moves in a compensated program

```
N10   G0 G41 X0 Y-.5
N11   Z.1
N12   G1 Z-.125 F3
N13   Y3.625 F7.5
N14   X5.5
N15   etc...
```


All rights reserved. Subject to change without notice.

Temporary Change of Tool Diameter

To change the tool radius in order to leave stock for a finish pass, program the "stock-variable". The variable assigned for this function is #1030.

Example: N120 #1030 = .015

When the CNC reads the above block, .015 will be added to the active tool radius. The value in the Tool Page for that tool # will not be updated, and tool compensation will be affected only until the tool is cancelled. #1030 is temporary.

When the tool is cancelled (T0), #1030 is also cancelled.

Example: N120 #1030 = -.015

In this case, .015 will be subtracted from the active tool's radius.

You must program the variable after the tool is activated.

#1030 is ignored for pocket canned cycles.

The variable must be programmed before programming **G41** (Cutter Compensation LEFT) or **G42** (Cutter Compensation RIGHT) and cannot be changed while **G41** or **G42** is active.

All rights reserved. Subject to change without notice.

Motion of Tool During Tool Compensation

In linear-to-linear or linear-to-circular moves, the position at the end of the startup block (block with **G4**1 [Compensation LEFT] or **G42** [Compensation RIGHT]) will be perpendicular to the next programmed move in the plane. Refer to **Figure 9-12** and **Figure 9-13**.

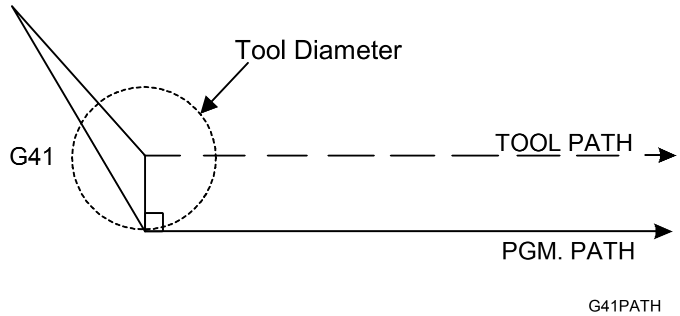


Figure 9-12, A Linear-to-Linear Move

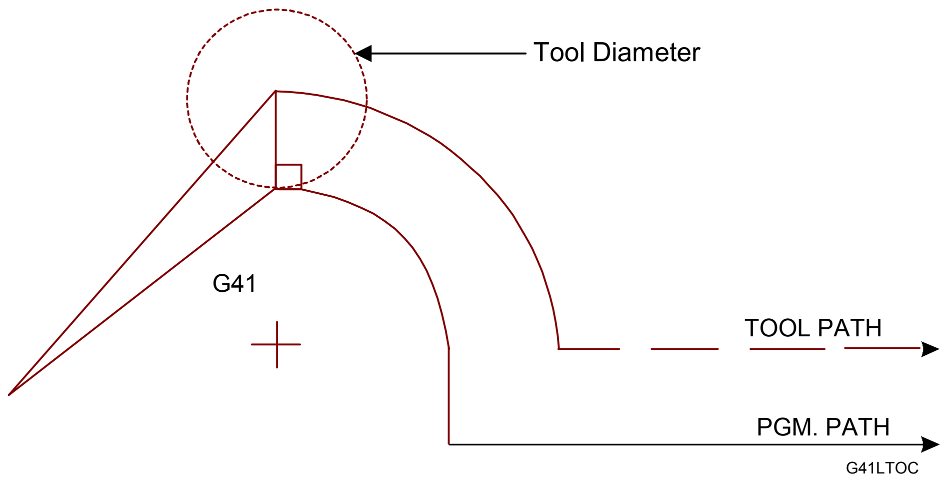


Figure 9-13, A Linear-to-Circular Move

In either case, the axes will move to a point perpendicular to the next move during the startup block.

The length of the XY move that activates compensation must be equal to or greater than the tool radius value. Example: If tool radius equals 0.3750", the vector length of the XY move that activates compensation must be 0.3750" or greater.

The same applies to the **G40** (Compensation OFF) move.

Refer to Figure 9-14, Paths During Tool Compensation. During tool compensation, the CNC performs offset correctly and automatically. Non-positioning moves such as dwells (except dwell zero or infinite dwells), coolant, or other auxiliary functions are allowed (except **M1** [Optional Program Stop] and **M0** [Program Stop Mode], these are not allowed). Moves in the third axis are also allowed during compensation.

All rights reserved. Subject to change without notice.

You cannot program a plane change (**G17**, **G18**, or **G19**) during tool compensation. However, a 2-axis move off the currently active plane is allowed.

For example: **G17** is the active plane (compensation in XY). You program an XZ or YZ move. The Z-axis will reach the programmed target as X/Y reaches its compensated target. Helical moves in the active plane are also allowed.

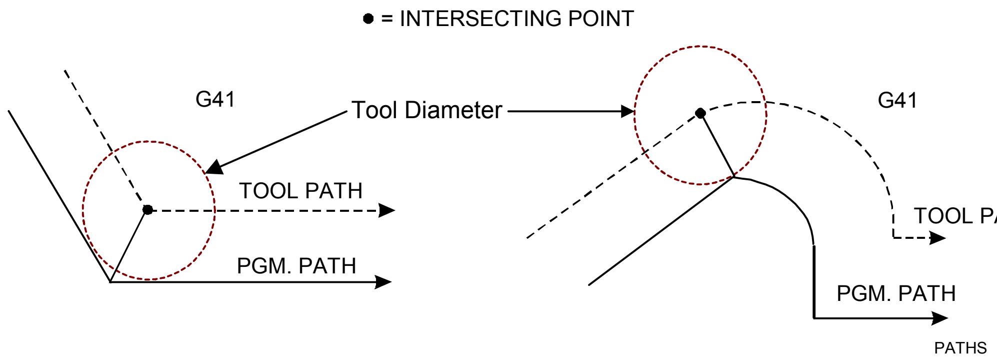


Figure 9-14, Paths during Tool Compensation

Program **G40** (cancel compensation) alone or with a move in the active plane. The move must be in rapid (**G0**) or feedrate (**G1**). Deactivation with **G2**/**G3** is not permitted. The move must be at least the tool radius in length.

The CNC "looks ahead" to following blocks in order to compensate correctly. When it "sees" an upcoming **G40** block, the CNC positions the tool perpendicular to the last move before the **G40** block.

All rights reserved. Subject to change without notice.

Figure 9-15 shows tool movement as compensation is deactivated.

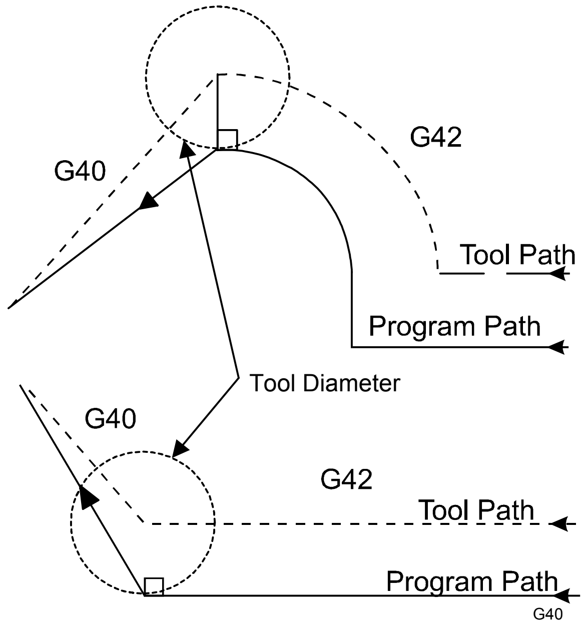


Figure 9-15, Offset Cancel

The tool moves to a point perpendicular to the last move before the **G40** (deactivation) move.

All rights reserved. Subject to change without notice.

Compensation Around Acute Angles

Refer to "Temporary Change of Tool Diameter" in this section. During compensation, the CNC finds the compensated intersection of moves and travels to that point.

On very sharp angles, this is not always desirable. For example, if you compensate along the outside of a 15-degree corner angle, the compensated intersection point will be far away from the actual point on the work. Time is wasted by "cutting air" until the compensated point is reached. To save time, the CNC creates an arc around the end of the point on the work.

The CNC applies the arc where there are angles of 15 degrees or less. This can be set in the Setup Utility or in the program. To change the angle via program, set #1031.

Example: to change an angle to 10 degrees, program: #1031=10. Re-program this value to 15 degrees (default) when finished.

The bottom part of **Figure 9-16** shows how the CNC will automatically "round" the compensated intersection. The work will remain a sharp corner.

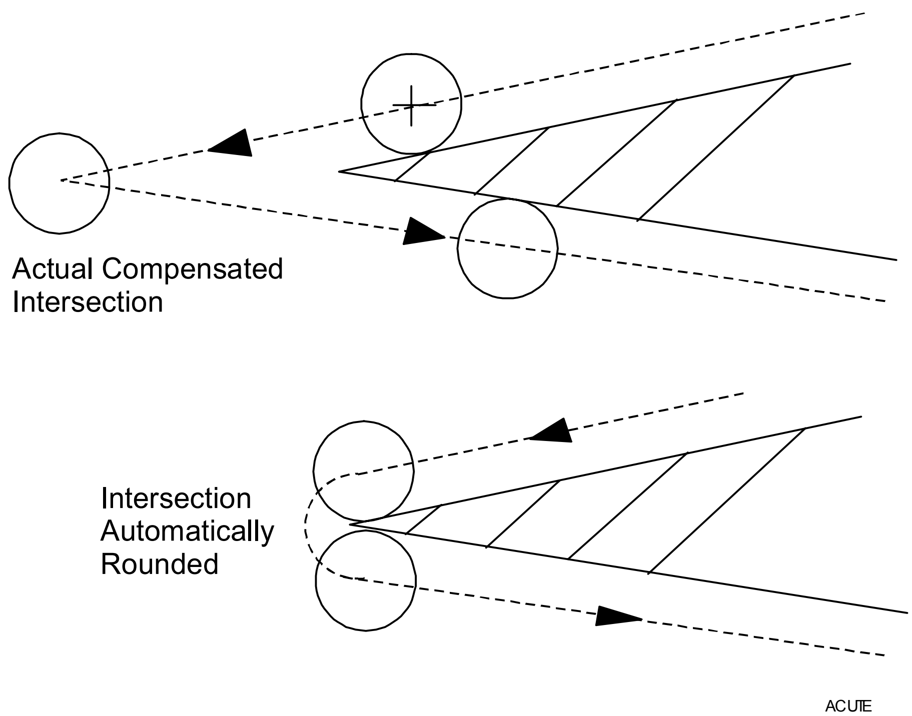


Figure 9-16, Compensation around an Acute Angle

All rights reserved. Subject to change without notice.

General Precautions

1. When you program tool path instead of part edge, a negative diameter in the Tool Page effectively changes **G41** to **G42** in the moves during compensation.
2. Third axis moves (not in the active plane) are permitted during compensation.
3. The CNC automatically rounds off the compensated intersection of acute angles of 15 degrees or less. To change this value, program #1031.
4. It is possible to change the tool diameter currently in use with "stock" variable #1030.
5. Startup (Ramp On) and cancellation (Ramp Off) blocks must be of **G0** or **G1** type, and must be at least the tool radius in length.
6. You must enter proper diameter value in the Tool Page before you use tool compensation.
7. Compensated arcs must be on the active plane (G17 = XY, G18 = XZ, G19 = YZ).
8. **G53** (Fixture Offset) and **G92** (Zero Set) are not permitted during compensation.
9. In Manual Mode, any active compensation deactivates.
10. **Jog/Return** is permitted during compensation.
11. System variable #1032 is available to change the number of blocks the CNC can "look-ahead" while in tool-comp.

CAUTION:	**Changing this value can change the compensated tool path. This variable enables further look ahead to prevent undercut (excessive tool diameter). At default, the CNC looks ahead far enough to find a valid intersection between the current and next move. Set the variable #1032 before you turn on the compensation (G40, G41 or G42).**

All rights reserved. Subject to change without notice.

G41 Programming Example

Tool compensation can be activated with **G41** or **G42**. Therefore you can program the part-edge directly, rather than having to calculate the offset manually. Refer to **Figure 9-17** and **Table 9-3**. On a 3/8" diameter end mill, the diameter value in the Tool Page for Tool #1 is .3750".

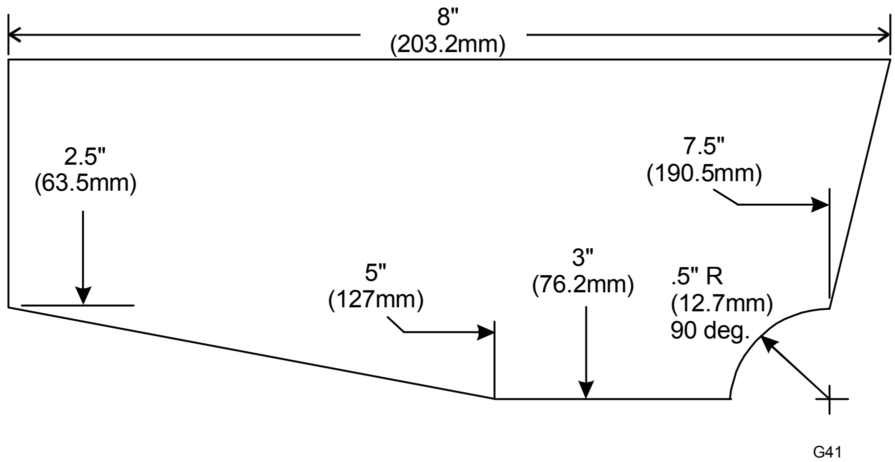


Figure 9-17, Motion Example using G41: Absolute (G90)

Table 9-3, Motion Example Using G41

Standard	Metric
N1 O1010 * COMP-EX-1	N1 O1010 * COMP-EX-1
N2 G90 G70 G0 T0 Z0	N2 G90 G71 G0 T0 Z0
N3 X-3.0 Y1.0	N3 X-76 Y25
N4 T1 * .375 MILL	N4 T1 * 9.52 MILL
N5 G41 X-.5 Y0	N5 G41 X-12 Y0
N6 Z.1	N6 Z2
N7 G1 Z-.125 F5.0	N7 G1 Z-3.175 F125
N8 X8.0 F12.0	N8 X203.2 F300
N9 X7.5 Y-2.5	N9 X190.5 Y-63.5
N10 G3 X7.0 Y-3.0 I0 J-.5	N10 G3 X177.8 Y-76.2 I0 J-12.7
N11 G1 X5.0	N11 G1 X127
N12 X0 Y-2.5	N12 X0 Y-63.5
N13 Y.5	N13 Y12
N14 G0 Z.1	N14 G0 Z2
N15 G40 X-3.0 Y1.0	N15 G40 X-76 Y25
N16 T0 Z0	N15 T0 Z0
N17 M2	N17 M2

All rights reserved. Subject to change without notice.

Refer to **Table 9-4** for line by line details of Table 9-3, Motion Example Using G41.

Table 9-4, Line by Line Description of Table 9-3, Motion Example Using G41

N-Code	Function
N1	Establishes program # and name.
N2	Sets Absolute, Inch, Rapid; cancels tool offset, raises Z-axis.
N3	Moves to tool change position.
N4	Activates tool-length comp., also contains comment (*).
N5	Activates tool diameter compensation and positions tool.
N6	Positions Z above part.
N7	Feeds Z to depth, at feedrate of 5.
N8	Feeds first element of contour at new feedrate.
N9	N9 to N13 feeds around contour.
N14	Rapids Z above part.
N15	Disables diameter compensation during rapid move to X-3.0 Y1.0.
N16	Cancels tool, moves Z to home position.
N17	Ends program, resets CNC to N1.

G42 Programming Example

Refer to **Figure 9-18** and Table 9-5, Milled Pocket Using G42 for an example of a milled pocket created using **G42**.

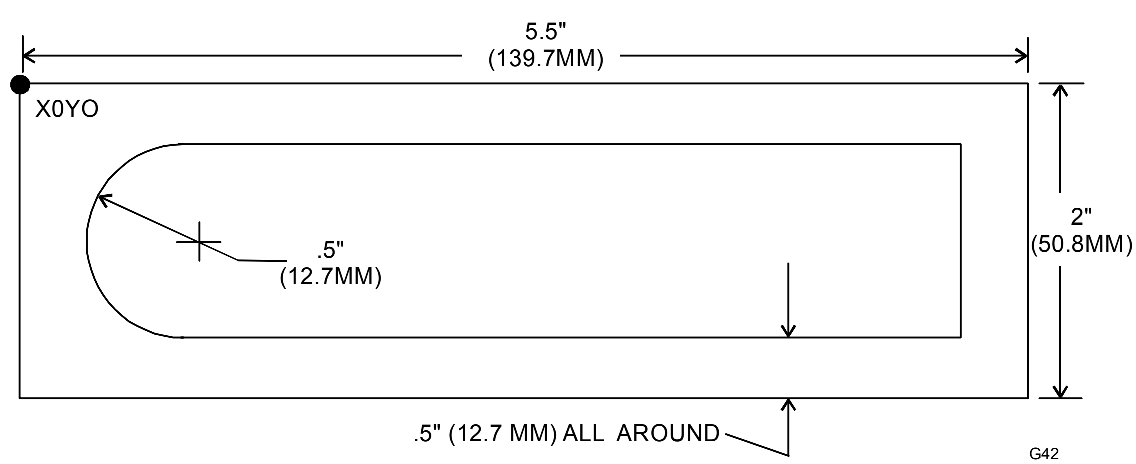


Figure 9-18, A Milled Pocket Using G42

All rights reserved. Subject to change without notice.

Table 9-5, Milled Pocket Using G42

Standard	Metric
N1 O1011 * COMP-EX-2	N1 O1011 * COMP-EX-2
N2 G90 G70 G0 T0 Z0	N2 G90 G71 G0 T0 Z0
N3 X-2 Y2	N3 X-50 Y50
N4 T1 * .5000 MILL	N4 T1 * 12.7 MILL
N5 X1.5 Y-1.0	N5 X38.1 Y-25.4
N6 Z.1	N6 Z2
N7 G1 Z-.25 F5	N7 G1 Z-6.35 F127
N8 G42 X.5 F12	N8 G42 X12.7 F300
N9 G2 X1 Y-.5 I.5 J0	N9 G2 X25.4 Y-12.7 I12.7 J0
N10 G1 X5.0	N10 G1 X127
N11 Y-1.5	N11 Y-38.1
N12 X1	N12 X25.4
N13 G2 X.5 Y-1 I0 J.5	N13 G2 X12.7 Y-25.4 I0 J12.7
N14 G40 G1 X4.7	N14 G40 G1 X119
N15 T0 G0 Z0	N15 T0 G0 Z0
N16 X-2 Y2	N16 X-50 Y50
N17 M2	N17 M2

All rights reserved. Subject to change without notice.

Refer to **Table 9-6** for line by line details of Table 9-5, Milled Pocket Using G42.

Table 9-6, Line by Line Description of Table 9-5, Milled Pocket Using G42

N-Code	Function
N1	Establishes program # and name.
N2	Sets Absolute, Inch, Rapid, cancels tool offset, and raises Z-axis.
N3	Moves to tool change position.
N4	Activates tool-length comp., block also contains comment (*).
N5	Positions to inside of pocket.
N6	Position Z above part.
N7	Feeds Z to depth at feedrate of 5.
N8	Initiates compensation during feed move to arc start point.
N9 to N13	N9 to N13 feeds around slot's contour.
N14	Deactivates comp during move to clean-up center of pocket.
N15	Cancels tool offset and rapids Z home.
N16	Moves to tool (part) change position.
N17	Ends program, resets CNC to N1.

Activating Offsets via the Program

In a program, T1 (by itself) calls the Tool Page diameter and length offsets for the specified tool. T1 with D, R, and L address words programs a temporary diameter/radius and length offset independent of the Tool Page. The entered D (diameter) or R (radius) and L (tool-length) offsets remain active until you cancel the active tool. Refer to **Table 9-7**.

Table 9-7, Activating Offsets Using T1

T1 Format	Description
T1	Activates Tool #1 diameter offset listed in the Tool Page.
T1 D.5000 L-1.2500	Applies a diameter offset of .5000 and length offset of -1.2500 to the active tool.
T1 R.2500 L-1.2500	Applies a tool radius value of .2500 and length offset of -1.2500 to the active tool.

The diameter offset takes effect when you program **G41** or **G42**. All dimensions are in reference to the work surface.

CAUTION: If you use T1 to activate a tool later in the program, the Tool Page offsets for Tool #1 will be used (not the values programmed via T1 Dn/Rn Ln).

NOTE: ANILAM recommends that you use the Tool Page to avoid confusion or possible entry errors on the offsets.

All rights reserved. Subject to change without notice.

Section 10 - Program Management

The Program Manager provides access to all of the program management utilities. These functions include creating, selecting, deleting, and copying programs. The Program Manager also provides access to the network or USB.

Press **Program** (**F2**) to activate the Program Manager from the Manual screen.

The Program Manager's **USER** listing lists the programs stored in the CNC. Refer to **Figure 10-1**. All CNC programs have **.G** extensions after their names. Change the Program Type mode to view programs with other formats.

NOTE:	The folder where the user programs are stored is V:\USER. All programs should be created and saved in this folder. Additional folders can be created in V:\USER to better organize programs. Do not delete or alter files or folders outside of V:\USER. Some files and folders outside of V:\USER may be needed for normal operation of the CNC.

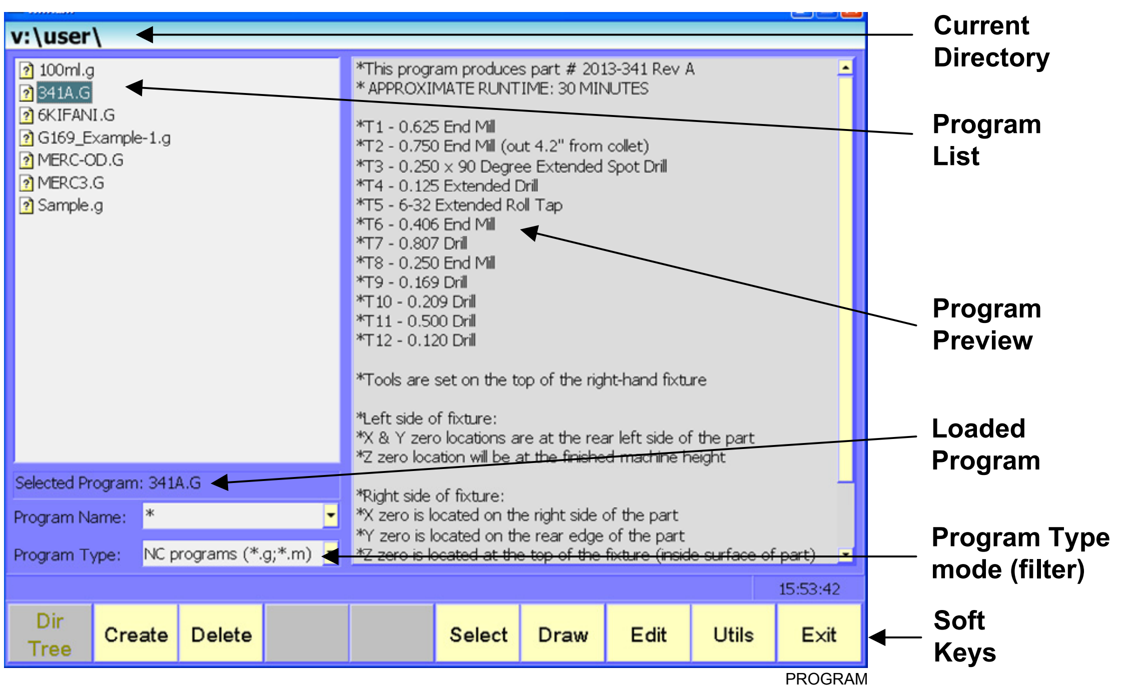


Figure 10-1, Program Screen

Other functions related to the Program screen may be accessed using the **SHIFT** key. These functions are described in this section. Press the **SHIFT** key to display the Program Shift screen. Refer to Figure 10-2, Shift Screen from Program Screen.

All rights reserved. Subject to change without notice.

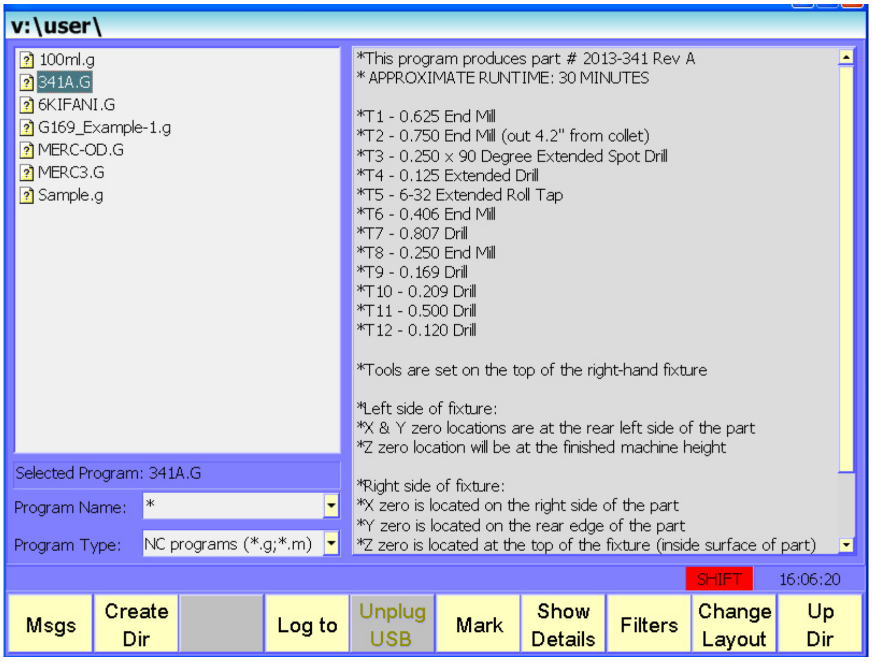


Figure 10-2, Shift Screen from Program Screen

The following topics are described in this section:

- Program Screen Soft Keys and Secondary Soft Keys
- Activating the Program Screen
- Changing the Program Manager Display
- Creating a New Part Program
- Choosing Program Names
- Selecting a Program for Running
- Selecting a Program for Editing
- Deleting a Program
- Utils Function Pop-Up Menus
- Copy Programs from/to Other Directories
- Moving Programs from/to Other Directories
- Renaming Programs
- Marking and Unmarking Programs
- Deleting Groups of Programs
- Creating Subdirectories

All rights reserved. Subject to change without notice.

Program Screen Soft Keys and Secondary Soft Keys

Refer to **Table 10-1**.

Table 10-1, Program Screen Soft Keys

Label	Soft Key	Soft Key Label and Function
DirTree	**F1**	Toggles between tree and list control
Create	**F2**	Creates a new blank program
Delete	**F3**	The CNC deletes the selected program
Select	**F6**	You must load a program before you can run it. Only one program can be loaded at a time.
Draw	**F7**	Enables Draw functionality
Edit	**F8**	Select a program for editing
Utils	**F9**	Displays the Utils pop-up menu. See Table 10-3, Utils (F9) Pop-up Menu Description.
Exit	**F10**	Return to Manual screen

Press **SHIFT** while in the Program screen to activate the secondary soft key functions (refer to Figure 10-2, Shift Screen from Program Screen). Refer to Table 10-2, Program Screen Secondary Soft Keys.

All rights reserved. Subject to change without notice.

Table 10-2, Program Screen Secondary Soft Keys

Label	Soft Key	Soft Key Label and Function
Msgs	(**SHIFT** + **F1**)	Displays messages, prompts, and reminders. See "Section 3, Messages (Msgs) (SHIFT + F1)"
Create Dir	(**SHIFT** + **F2**)	Creates a new directory.
Log to	(**SHIFT** + **F4**)	Changes the current directory to the selected directory: o Expanded by using the right **ARROW** o Compressed by using the left **ARROW** o Traverse the directory using up and down **ARROW** keys o Go to parent directory of current directory by using the **ARROW** keys
Mark	(**SHIFT** + **F6**)	To mark a program. See "Marking and Unmarking Programs."
Show Details	(**SHIFT** + **F7**)	Show the Program Manager details. Refer to Figure 10-4, Show Details Screen.
Filters	(**SHIFT** + **F8**)	Used to move the highlighter to Program Name to enable search.
Change Layout	(**SHIFT** + **F9**)	Shows the Program Manager structure. Refer to Figure 10-3, Change Layout Screen.
Up Dir	(**SHIFT** + **F10**)	Shows the Program Manager structure. Refer to Figure 10-5, Up Dir Screen.

Activating the Program Screen

Activate the Program screen as follows:

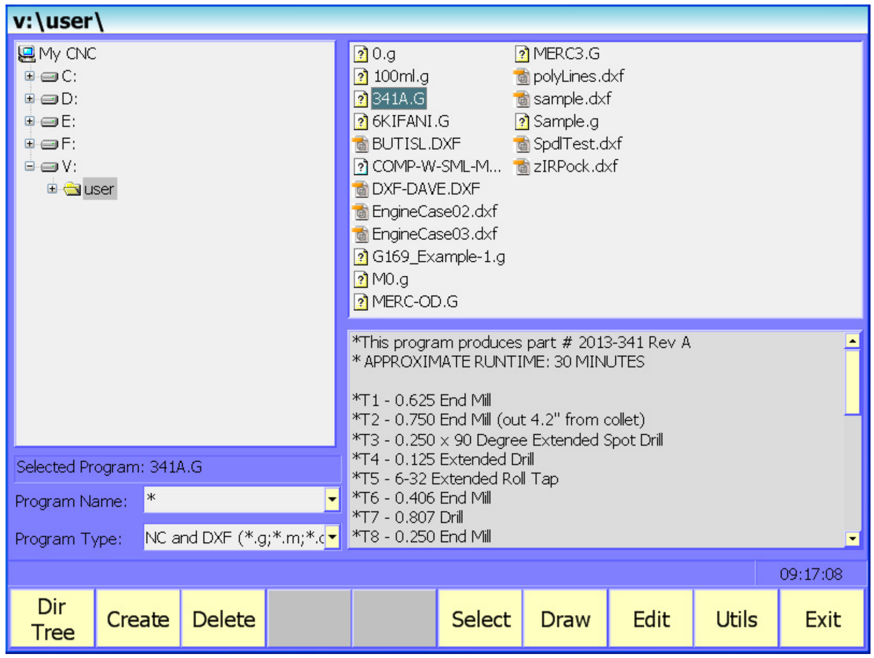
1. Go to the Manual screen.
2. In the Manual screen, press **Program** (**F2**). The Program screen activates (refer to Figure 10-1, Program Screen).
3. Press the **SHIFT** key on the keyboard to display the Shift screen from Program screen (refer to Figure 10-2, Shift Screen from Program Screen). Press **SHIFT** again to return to the Program screen.

All rights reserved. Subject to change without notice.

Changing the Program Manager Display

You can change the Program Manager display to one of the following:

- Select **Change Layout** (**SHIFT** + **F9**) to show the Program Manager structure. Refer to **Figure 10-3**.



CHANGE LAYOUT1

Figure 10-3, Change Layout Screen

- Select **Show Details** (**SHIFT** + **F7**) to show the Program Manager details (part program names (ending with .G extensions) along with size, date, and time of last edit). Refer to Figure 10-4, Show Details Screen.

All rights reserved. Subject to change without notice.

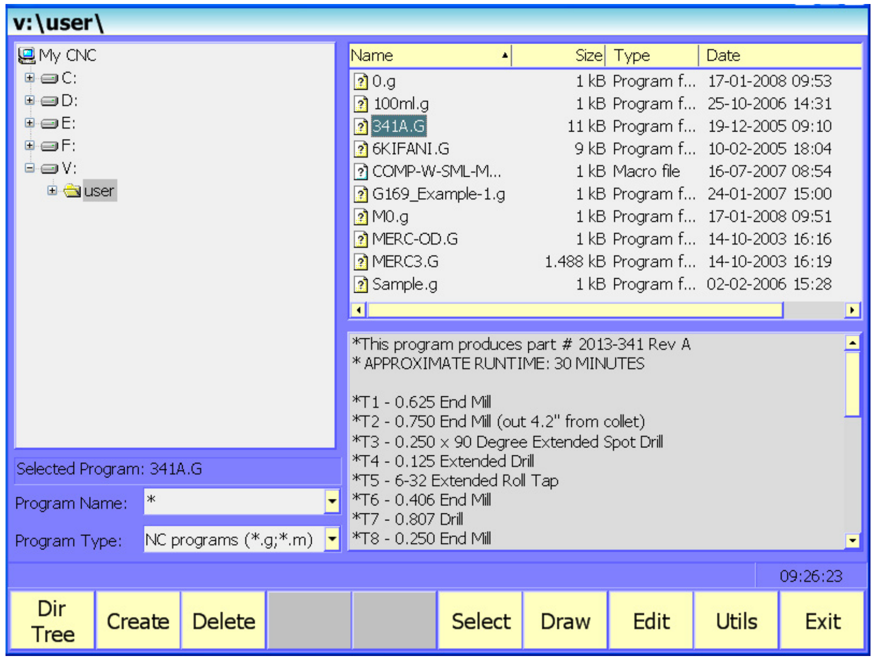
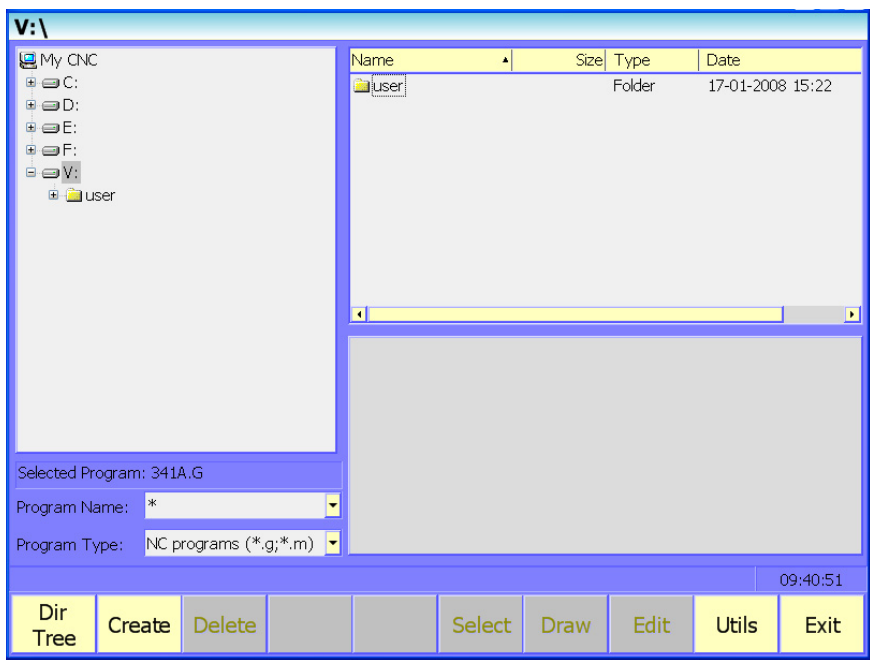


Figure 10-4, Show Details Screen

- Select **Up Dir** (**SHIFT** + **F10**) to show the source directory without the tree structure. Refer to **Figure 10-5**.



UP DIR1

Figure 10-5, Up Dir Screen

The display setting that shows only part program names is usually the easiest to use.

All rights reserved. Subject to change without notice.

Creating a New Part Program

To create a new part program:

1. In Manual Mode, press **Program** (**F2**). The Program Manager activates.
2. Press **Create** (**F2**). The control displays the Create pop-up with the prompt, **Enter the name of the new Program**
   Program Name: _.
3. Type the new program name.
4. Press **ENTER**. The new program name is inserted in the Program Manager.

Choosing Program Names

A name cannot be longer than 60 alphanumeric characters. The CNC displays program names as they were entered. No two programs can have the same name. The CNC automatically places the **.G** extension after the name.

Selecting a Program for Running

You must select a program before you can run it. Only one program can be selected at a time.

To Select a program:

1. In the Program Manager, use **ARROWS** to highlight a program.
2. Press **Select** (**F6**). The CNC loads the program. The name of the currently loaded program displays in the **Program Name** field at the bottom left of the screen.

Selecting a Program for Editing

When the required program is highlighted, press **Edit** (**F8**) to activate the Editor.

NOTE:	If the Program Editor is activated in Manual Mode, the Editor will open the loaded program.

To select a program for editing:

1. In the Program Manager, use **ARROWS** to highlight the program name.
2. Press **Edit** (**F8**). The Program Editor activates. The CNC displays the listing for the selected program.

All rights reserved. Subject to change without notice.

Deleting a Program

To delete a program:

1. Highlight a program in the Program Manager.
2. Press **Delete** (**F3**). The CNC prompts to confirm the deletion and the soft keys change for your response.
3. Press **Yes** (**F1**). The CNC deletes the selected program.

   – or –

   Press **No** (**F3**). The Delete command is canceled.

NOTE: Deleting a program automatically deletes the associated **.S** file.

Utils Function Pop-Up Menus

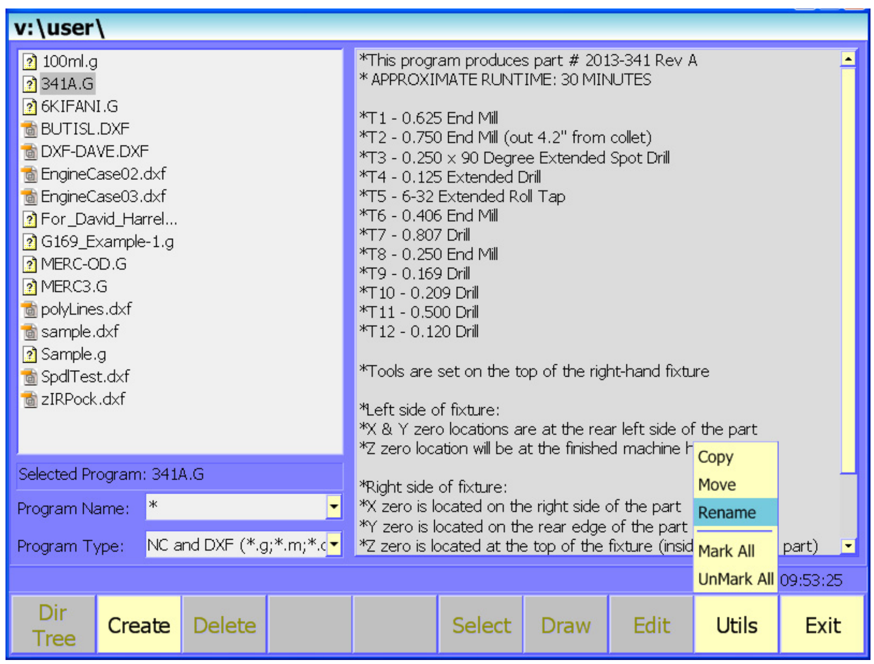


Figure 10-6, Utils Pop-up Menu

All rights reserved. Subject to change without notice.

On the Program screen, select **Utils** (**F9**) to display the Utilities pop-up menu. Refer to Figure 10-6, Utils Pop-up Menus. Refer to **Table 10-3**.

Table 10-3, Utils (F9) Pop-up Menu Description

Label	Description
Copy	To copy programs to another drive or to floppy disks. See "Copying Programs from/to Other Directories." Copies currently highlighted file and files that were previously marked. **o** Expanded by using the right **ARROW** **o** Compressed by using the left **ARROW** **o** Traverse the directory using up and down **ARROW** keys **o** Go to parent directory of current directory by using the **ARROW** keys **o** Display directory name in status area at bottom, which defaults to current drive **o** If copying to current location, need to create file with "copy of …" **o** If copying to a new location and file exists, need to prompt to overwrite: **OK** (**F1**) and **Cancel** (**F10**)
Move	To move a program. Copies currently highlighted file and files that were previously marked. **o** Expanded by using the right **ARROW** **o** Compressed by using the left **ARROW** **o** Traverse the directory using up and down **ARROW** keys **o** Go to parent directory of current directory by using the **ARROW** keys **o** Display directory name in status area at bottom, which defaults to current drive **o** If copying to a new location and file exists, need to prompt to overwrite: **OK** (**F1**) and **Cancel** (**F10**) **o** Deletes source file after copying
Rename	Use to rename a program. See "Renaming Programs."
Mark All	To mark all programs in the Program Manager. See "Marking and Unmarking Programs."
UnMark All	To unmark all marked programs. See "Marking and Unmarking Programs."

All rights reserved. Subject to change without notice.

Copying Programs from/to Other Directories

Use **Copy** to copy programs to or from another directory, such as a subdirectory or a Universal Serial Bus (USB). Refer to Figure 10-6, Utils Pop-up Menu.

To copy programs to or from another directory:

1. In the Program Manager, highlight the program or mark all programs to be copied.
2. Press **Utils** (**F9**) to display the Utils pop-up menu (refer to Figure 10-6, Utils Pop-up Menu and Table 10-3, Utils Soft Keys from Program Screen). Select **Copy** to display the **Select Copy to destination:** pull up menu.
3. Highlight the target drive and press **ENTER**. The CNC copies marked programs to the target drive.

   – or –

   Type the new location (complete path), and press **ENTER**. The program is copied into the new location.

Moving Programs from/to Other Directories

To move a program copies the currently highlighted file or files that were previously marked to or from another directory, such as a subdirectory or a Universal Serial Bus (USB). Refer to Figure 10-6, Utils Pop-up Menu.

To move programs to or from another directory:

1. In the Program Manager, highlight the file or mark all files to be moved.

4. Press **Utils** (**F9**) to display the Utils pop-up menu (refer to Figure 10-6, Utils Pop-up Menu and Table 10-3, Utils Soft Keys from Program Screen). Select **Move** to display the **Select Move to destination:** pull up menu.
5. Highlight the target drive and press **ENTER**. The CNC moves marked programs to the target drive.

   – or –

   Type the new location (complete path), and press **ENTER**. The program is moved into the new location.

All rights reserved. Subject to change without notice.

Renaming Programs

To rename a program:

1. In the Program Manager, highlight a program.
2. Press **Utils** (**F9**) to display the Utils pop-up menu (refer to Figure 10-6, Utils Pop-up Menu and Table 10-3, Utils Soft Keys from Program Screen). Select **Rename**, and press **ENTER**.
3. Type new program name, and press **ENTER**. The new name replaces the old name.

Marking and Unmarking Programs

You can perform some operations on more than one program at a time. The Program Manager enables you to select (Mark) one, some, or all of the programs in the **USER** listing.

The following topics are described:

- **Marking Programs**
- **Unmarking Marked Programs**
- Marking All Programs
- Unmarking All Marked Programs

Marking Programs

To mark a program:

1. Highlight a program in the Program Manager.
2. Select **Mark** (**SHIFT** + **F6**).
3. Press **ARROWS** to highlight another program in the list, and select **Mark** (**SHIFT** + **F6**).
4. Repeat these steps to mark as many program as required.

Unmarking Marked Programs

To unmark a program:

1. Highlight any marked program, and press **ENTER**. The program is no longer marked. The CNC unmarks the program in the Program Manager. (The Program is no longer highlighted.)

All rights reserved. Subject to change without notice.

Marking All Programs

To mark all programs in the Program Manager:

1. In the Program Manager, press **Utils** (**F9**) to display the pop-up menu. Refer to Figure 10-6, Utils Pop-up Menu and Table 10-3, Utils Soft Keys from Program Screen). Select **Mark All**.
2. The CNC displays a **Select a Filter for Marking Files** pull up menu and another pull up menu with the file extensions. Highlight the file extension that you want to mark all.
3. Press **ENTER** to mark all of the files in the file extension you have highlighted.

Unmarking All Marked Programs

To unmark all programs in the Program Manager:

1. In the Program Manager, select **Utils** (**F9**) to display the pop-up menu. Refer to Figure 10-6, Utils Pop-up Menu and Table 10-3, Utils Soft Keys from Program Screen). Select **UnMark All**.
2. Press **ENTER** to unmark all of the marked files.
3. The CNC unmarks all programs in the Program Manager. (Programs are no longer highlighted.)

Deleting Groups of Programs

1. From the Program Manager, mark all of the programs to be deleted.
2. Select **Delete**. The CNC prompts to confirm the deletion and the soft keys change for your response.
3. Press **Yes** (**F1**) to delete the selected programs.

   – or –

   Press **No** (**F2**) to cancel.

Creating Subdirectories

Press **Create Dir** (**SHIFT** + **F2**) to create subdirectories. Ensure that the CNC is in the desired drive before you create a subdirectory.

[Default: **V:\USER**]

To create a subdirectory:

1. Press **Create Dir** (**SHIFT** + **F2**).
2. The CNC prompts for the new subdirectory. Type the subdirectory name, and press **ENTER**. The CNC creates the subdirectory.

All rights reserved. Subject to change without notice.
25-January-08

Section 11 - Running Programs

NOTE:	Verify all programs in Draw before you run them. Refer to "Section 8 - Viewing Programs with Draw."

There are two modes of programmed operation:

Single-Step Mode	Runs a program one block at a time.
Automatic Mode	Runs a program automatically, without pausing.

The screens for both modes resemble the Manual screen. Use the soft key labels to distinguish between modes. The CNC highlights the label for the active mode.

You must load a program before you can run it to cut a part. Load programs from the Program Manager. Refer to "Section 10 - Program Management" for information on how to load programs.

The Manual Data Input Mode (MDI) enables you to program a few quick moves without having to create and save a program. MDI is usually used for manual operation. It is available only in Manual Mode.

All programming tools, moves and cycles are available in MDI.

Refer to "Section 3 - Manual Operation and Machine Setup" for additional information.

The following topics are described in this section:

- Running a Program One Step at a Time
- Position Display Modes
- Automatic Program Execution
- Clearing a Halted Program
- Using Draw While Running Programs
- Parts Counter and Program Timer
- Jog/Return

All rights reserved. Subject to change without notice.

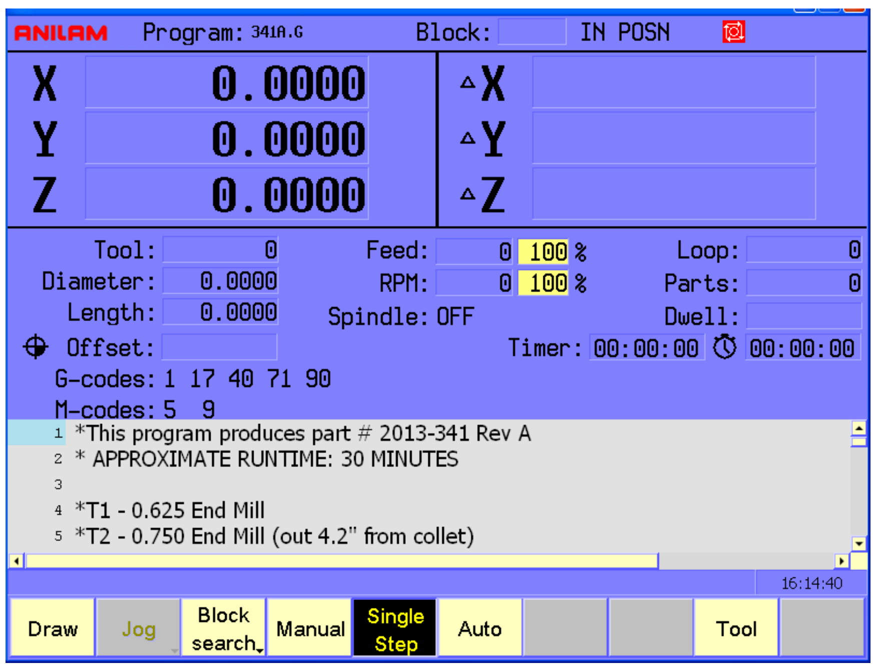
Running a Program One Step at a Time

Single-Step Mode runs a program block by block. This mode enables you to step through the program and verify the moves before you cut an actual part. Refer to **Figure 11-1**.

To run a program in Single-Step Mode:

1. Go to the Program Manager, select a program and press **Select** (**F6**) to load the required program.
2. Press **Exit** (**F10**) to return to the Manual screen.
3. In Manual Mode, press **Single Step** (**F5**) to activate Single-Step Mode.
4. Press **START** to execute each block or motion.

NOTE: In Auto Mode, press **Single Step** (**F5**) to activate Single-Step Mode.



SINGLE STEP

Figure 11-1, Single Step Screen

All rights reserved. Subject to change without notice.
25-January-08

Table 11-1 describes the active soft keys on the Single Step screen and Auto screen (refer to Figure 11-2, Auto Screen).

Table 11-1, Single-Step and Auto Screen Soft Keys

Label	Soft Key	Function
Draw	**F1**	Activates the Draw function
Jog	**F2**	Displays the Jog screen
Block search	**F3**	Activates the Block Search. Refer to "Using Block Search to Select a Starting Block."
Manual	**F4**	Activates Manual Mode from Auto and Single Step
Single Step	**F5**	Changes to Single-Step Mode
Auto	**F6**	Changes to Auto Mode. Use to run part programs for production.
Tool	**F9**	Activates the Tool page.

Press the **SHIFT** key on the keyboard to display the Single-Step and Auto Mode Shift screen (refer to Figure 11-2, Auto Screen). Refer to **Table 11-2**.

Table 11-2, Single-Step and Auto Mode Screen Secondary Soft Keys

Label	Soft Key	Function
Msgs	(**SHIFT + F1**)	Displays the last 10 messages, both old (already read) and new (not yet read)
BG Prog	(**SHIFT + F3**)	Background programming displays Figure 10-1, Program Screen and enables to change the program while it is running.
Parts Counter	(**SHIFT + F4**)	Activates the Parts Counter pop-up window to reset the New Value.
OSC	(**SHIFT + F7**)	Oscilloscope. For details see *6000i CNC Technical Manual*, P/N 627787-21.
OLM	(**SHIFT + F8**)	On-line Monitor. For details see *6000i CNC Technical Manual*, P/N 627787-21.

The following topics are described:

- Using Single-Step Mode
- Holding or Canceling a Single-Step Run
- Single-Step Execution of Selected Program Blocks

All rights reserved. Subject to change without notice.

Using Single-Step Mode

When Single-Step is active, **Single Step** (**F5**) highlights.

- In Single-Step Mode, the CNC holds before it executes each block. Press **START** to execute each block.

Holding or Canceling a Single-Step Run

Press **HOLD** to halt the execution of the program. Press **START** to restart a program that is on hold. Press **MANUAL** (**F4**) to cancel a program that is on hold. When you cancel a program, the CNC terminates tool compensation and canned cycles. All other modal settings remain active.

Single-Step Execution of Selected Program Blocks

The following topics are described:

- **Using Arrow Keys to Select a Starting Block**
- Using Block Search to Select a Starting Block
- Switching from Single-Step Mode to Auto Mode

Using Arrow Keys to Select a Starting Block

Select the starting block before you start program.

1. Load the required program and return to the Manual screen.
2. Press **Single Step** (**F5**) to activate Single-Step Mode.
3. Press **Block Search** (**F3**), and then press **Scroll** (**F9**).
4. Highlight the desired starting block using the **ARROW** keys.
5. Press **Return** (**F10**).
6. Press **START** to execute the next block or motion.

All rights reserved. Subject to change without notice.
25-January-08

Using Block Search to Select a Starting Block

Use **Block search** to locate a specific block number or entered text. The CNC highlights the first block found that contains the specified information. **Block search** only searches forward in the program. Begin the search from the starting block to search through the entire program.

1. From the Program Manager, load the required program and return to the Manual screen.
2. Press **Single Step** (**F5**) or **Auto** (**F6**).
3. Press **Block search** (**F3**). The CNC prompts for search number or text. Or press **Goto** (**F4**) to search for a line in the program.
4. Type the required number or text, and press **ENTER**. The CNC runs the search and highlights the first block it finds that contains the number or text.
5. Press **START** to run the program from the highlighted block.

NOTE: After you start the program, it will execute normally.

Table 11-3 describes the active soft keys on the Block Search screen.

Table 11-3, Block Search Screen Soft Keys

Label	Soft Key	Function
Find previous	**F1**	Moves the program display one page backward.
Find next	**F2**	Moves the program display one page forward.
Match case	**F3**	Displays the Find pop-up with the message **Find what:** and an entry field for you to type the case-sensitive string that you want.
Goto	**F4**	Displays the Goto pop-screen with the message, **Go to line:** and an entry field.
Start of Prog	**F5**	The cursor returns to the first block of the program. See "Section 6, Advancing to the First or Last Block of a Program."
End of Prog	**F6**	The cursor advances to the last block of the program. See "Section 6, Advancing to the First or Last Block of a Program."
Find	**F8**	Displays the Find screen. The Find pop-up screen displays with the message, **Find what:** and an entry field. For the Find screen soft keys, refer to Table 11-4, Block Search>Find (F8) Screen Soft Keys.
Scroll	**F9**	Scrolls the display
Return	**F10**	Return to the Single-Step Screen.

All rights reserved. Subject to change without notice.

Table 11-4 describes the active soft keys on the Block Search>Find screen.

Table 11-4, Block Search>Find (F8) Screen Soft Keys

Label	Soft Key	Function
Find previous	**F1**	Moves the program display one page backward.
Find next	**F2**	Moves the program display one page forward.
Match case	**F4**	Displays the Find pop-up with the message **Find what:** and an entry field for you to type the case-sensitive string that you want.
Return	**F10**	Return to the Block Search Screen.

Switching from Single-Step Mode to Auto Mode

To switch from Single-Step Mode to Auto Mode:

1. In Single-Step Mode, press **Auto** (**F6**) to complete the current move, then hold.
2. Press **START** to restart the CNC and run the rest of the program in Auto Mode.

Position Display Modes

Position Displays for X, Y, Z, and U show:

Program Movement to the programmed (commanded) position in reference to Part Zero or datum.

Distance to Go Distance to go to reach the commanded position.

Automatic Program Execution

The Auto Mode is the CNC's production mode. All or any part of a program can be executed in the Auto Mode. Put the CNC in Auto Mode from either the Manual or Single-Step Modes.

The Auto screen is similar to a Manual screen, but has fewer soft keys. The **Auto** (**F6**) soft key label highlights when the Auto Mode is active. Refer to Figure 11-2, Auto Screen.

All rights reserved. Subject to change without notice.

To run a program in Auto Mode:

1. In the Program Manager, load the required program and return to the Manual screen.
2. Press **Auto** (**F6**) to activate Automatic Mode.
3. Press **START**. The CNC begins to execute program blocks.

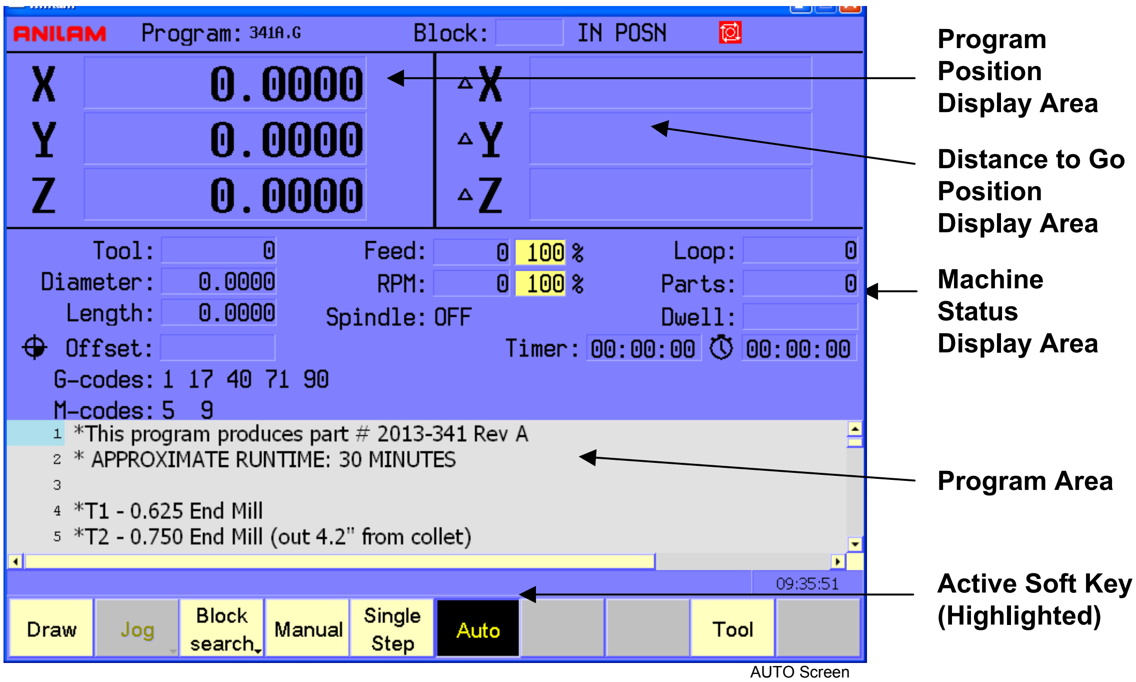


Figure 11-2, Auto Screen

The following topics are described:

- **Holding or Canceling an Auto Run**
- **Starting at a Specific Block**

Holding or Canceling an Auto Run

Press **HOLD** to halt the program. To restart a program on hold, press **START**. To cancel a program that is on hold, press **MANUAL** (**F4**). The CNC cancels any active tool compensation and canned cycles. Modal settings (such as Absolute Mode or Inch Mode) remain active.

Starting at a Specific Block

The following topic is described:

- Using Arrow Keys to Select a Starting Block

All rights reserved. Subject to change without notice.

Using Arrow Keys to Select Starting Block

1. From the Program Manager, select the required program and return to the Auto screen.
2. Press **Block Search** (**F3**), and then press **Scroll** (**F9**).
3. Highlight the required starting block using the **ARROW** keys.
4. Press **Return** (**F10**).
5. Press **START** to begin automatic program execution from the selected block.

Clearing a Halted Program

When the CNC encounters a program block that generates an error, it displays a Warning or error message and halts the program. Go back to Manual Mode to correct the problem.

A program error could generate more than one message. Refer to "Section 2 - CNC Console and Software Basics" for instructions on reviewing undisplayed error messages.

After you correct the program, load and restart it at the appropriate block.

Using Draw While Running Programs

In Real-Time Draw, the CNC displays moves as it executes them. The active **Single Step** (**F5**) or **Auto** (**F6**) highlights as does **Draw** (**F1**). Refer to Figure 11-3, Draw (Real-Time Mode).

All display options in Draw Simulation Mode are available in the Real-Time Draw Mode.

To activate **Draw** while running a program:

1. Load the required program and put the CNC in **Single Step** or **Auto** Mode.
2. Press **Draw** (**F1**) to activate the Real-Time Draw screen and change the soft keys.
3. Press **START** to run the program. The CNC displays moves as it executes them.

All rights reserved. Subject to change without notice.

DRAW REAL TIME

Figure 11-3, Draw (Real-Time Mode)

All rights reserved. Subject to change without notice.

Parts Counter and Program Timer

The CNC keeps track of program run-time (**Timer**) and the number of completed parts (**Parts**). The CNC displays Run-time in hours, minutes, and seconds. These two features are available in the Manual, Auto, and Single-Step Modes. Refer to **Figure 11-4**.

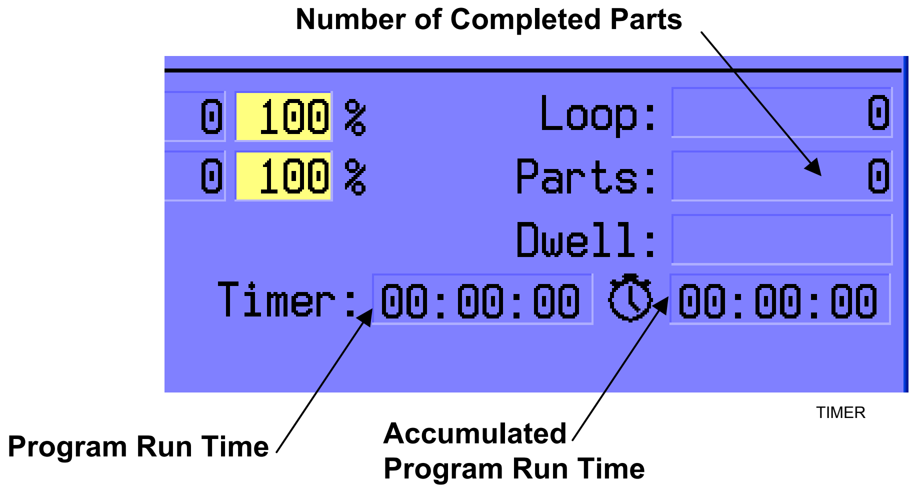


Figure 11-4, Program Timer and Parts Counter

The Timer begins timing the program run when you press **START** and stops when it encounters an **M2** block. Therefore, ensure that an **M2** block has been included at the end of the program.

The timer pauses if the CNC holds. The timer stops if you switch to Manual Mode. If you re-run the program before going back to Manual, the total time for all runs is displayed. The Timer values remain the same until you switch to Auto or Single-Step Mode again. Then, the timers reset to zero.

The Parts counter starts at zero and increments by one every time the CNC runs an **M2** block. Therefore, ensure that an **M2** block has been included at the end of the program. The CNC continues to count parts when you re-run the program in Auto or Single-Step. The parts counter value is maintained when you switch to Manual Mode, but will reset to 0 when you switch back to Auto or Single-Step Mode.

Refer to Figure 11-1, Single Step Screen. Press (**SHIFT + Parts Counter** (**F4**)) to display the Part Counter pop-up window. Type the New Value to display the new value in the Parts field.

All rights reserved. Subject to change without notice.
25-January-08

Jog/Return

Jog/Return is a function in the CNC that allows the tool to be removed from the cut while in Auto or Single-Step Modes, without switching the CNC to Manual. It has an 'automatic return' capability that will return the tool to its departure point.

It is generally used to check the tool's wear and to change a cutting tool/offset in case of tool breakage or excessive wear in the middle of an operation, or inspect a critical dimension or cut.

The following topics are described:

- **Initiating Jog/Return**
- **Operations Allowed While "In" Jog/Return**
- Jog/Return Soft Keys
- EXAMPLES:
- Notes on Jog/Return

Initiating Jog/Return

The Jog/Return function may be initiated from the Auto or Single-Step modes of the CNC.

To use the Jog/Return feature, the **HOLD** key must be pressed first, before pressing **JOG** (**F2**). This ensures that the machine cannot be stopped accidentally while cutting by pressing **JOG** (**F2**). In this way, the axes must be halted before the Jog/Return feature may be used.

Operations Allowed While "In" Jog/Return

Several motions/functions are allowed after the CNC has been put into "Jog/Return". The axes may be moved using the Manual Panel or by the soft keys. Manual Date Input (MDI) moves are not allowed.

Any Tool-Length Offset can be changed while in Jog/Return mode. Diameter offsets SHOULD NOT be altered with in Jog/Return mode (if the current diameter offset is altered, the new value WILL NOT take affect until the next time it is activated). ONLY THE CURRENT TOOL LENGTH OFFSET should be altered with in this special mode. In this way, if a tool breaks while in an operation, the user may replace the tool, re-set the tool length offset, and "return" the tool to the cut without aborting the program.

This is very useful and saves a great deal of time, if a tool breaks while in the middle of a canned cycle or an extremely long cut.

All rights reserved. Subject to change without notice.

Jog/Return Soft Keys

After the axes are halted by the **HOLD** key, and **JOG** (**F2**) is pressed, a new strip of soft keys related to the Jog/Return function is displayed:

Restart Pos (**F1**)	Sends the axes to a pre-determined point, the position before halted by the **HOLD** key.
Tool (**F7**)	Activates the Tool screen
Handwheel (**F8**)	Enable or disables handwheel moves
Return (**F10**)	Return to the Single Step or Auto screen

The following topics are described:

- **TOOL (F7)**
- **HANDWHEEL (F8)**
- **RETURN (F10)**

TOOL (F7)

Tool (**F7**) when pressed displays the CNC's Tool Page. This allows the operator to make a change to the tool length or wear offsets.

If these values in the Tool Page are changed, the new values will <u>not</u> be invoked until that tool is (re-) activated in the part program.

HANDWHEEL (F8)

Enable or disables handwheel moves.

RETURN (F10)

Once any axis is moved, the **Restart Pos** (**F1**) highlights and **Return** (**F10**) grays out. **Return** (**F10**) can only be used if no axis has changed position and after pressing **Return** (**F10**); you simply press **START** to continue.

If any axis has been moved in the jog mode, **Restart Pos** (**F1**) highlights and when you are done, you press **Restart Pos** (**F1**) and a new set of soft keys are displayed. Use these soft keys to return the axes moved in the order that you want. Once all axes are returned, the control automatically switches back.

All rights reserved. Subject to change without notice.
25-January-08

EXAMPLES:

The following are typical scenarios as to how and when to use the Jog/Return function. Assume the CNC is running the program in Auto or Single-Step Modes.

SITUATION 1:

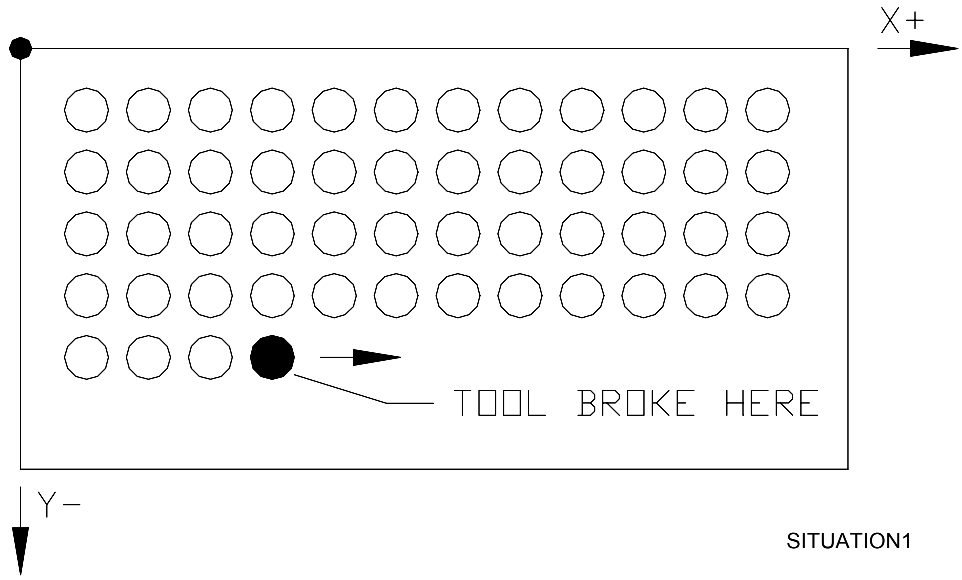


Figure 11-5, Drilling Illustration

Refer to **Figure 11-5**. The tool is drilling in an X+ row of holes in a workpiece. The tool becomes dull and breaks.

All rights reserved. Subject to change without notice.

Keystrokes/operations:

1. **HOLD**
2. **JOG** (**F2**)
3. Raise the Z-axis using jogging keys
4. Press **SPINDLE OFF** to stop spindle
5. Remove drill from holder
6. Place new drill in holder
7. Jog tool over workpiece with Manual Panel
8. Jog tool down to offset surface
9. **TOOL** (**F9**)
10. **TEACH** (**F8**)
11. **EXIT** (**F10**)
12. Jog Z+ with Manual Panel
13. Restart spindle by pressing **SPINDLE FWD**
14. Press **Restart Pos** (**F1**) and use soft keys (**F1–F4**) to return the axes to their positions
15. Press **START** to continue program

The axes will return to the position they were jogged from when the Jog/Return function was initiated, in the described path.

SITUATION 2:

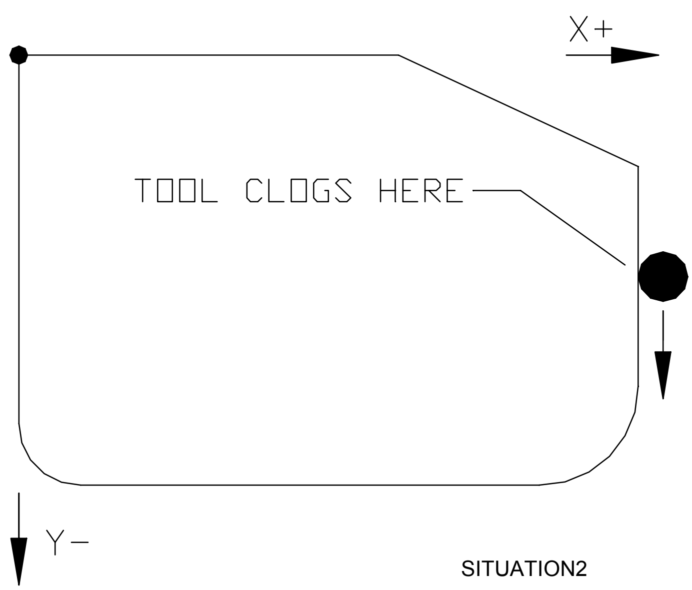


Figure 11-6, Cutting Illustration

Refer to **Figure 11-6**. The tool is feeding along a Y– cut on the right side of a workpiece. The tool becomes clogged with materials and is no longer able to cut.

All rights reserved. Subject to change without notice.
25-January-08

Keystrokes/operations:

1. **HOLD**
2. **JOG** (**F2**)
3. Press **SPINDLE OFF** to stop spindle
4. Remove end mill from holder
5. Place new end mill in holder
6. Jog tool over workpiece with Manual Panel
7. Jog tool down to offset surface
8. **TOOL** (**F9**)
9. **TEACH** (**F8**)
10. **EXIT** (**F10**)
11. Jog Z+ with Manual Panel
12. Restart spindle by pressing **SPINDLE FWD**
13. Press **Restart Pos** (**F1**) and use soft keys (**F1–F4**) to return the axes to their positions
14. Press **START** to continue program

The axes will return to the position they were jogged from when the Jog/Return function was initiated, in the described path.

Notes on Jog/Return

- Jog/Return is generally only used in trouble situations, where a tool breaks or a tolerance must be checked. It allows the program to be interrupted in Auto or Single-Step mode, without having to switch to Manual.
- The **HOLD** key must be pressed prior to **JOG**.
- Manual Panel moves are allowed while in Jog/Return.
- Manual Data Input (MDI) moves are not allowed.
- Tool length or wear offset on the tool may be altered while in Jog/Return. However, the new values will <u>not</u> take effect until the tool is reactivated again in the part program.
- The mode can be cancelled at any time by pressing **F4** (**Manual**).
- The Manual Panel is fully active (Handwheel also if present).

All rights reserved. Subject to change without notice.

Section 12 - S and M Functions

This section covers S and M code formats. Refer to **Table 12-1**. The codes are included in the part program or activated in Manual Mode.

Table 12-1, S and M Codes

Code	Function
S (Spindle Speed)	Commands spindle speeds (S).
M (Miscellaneous)	Performs miscellaneous (M) functions such as spindle ON/OFF and coolant ON/OFF.

The following topics are described in this section:

- **Speed Spindle Control (S-Function)**
- Miscellaneous Functions (M-Code)
- Control M-Codes
- Order of Execution

Speed Spindle Control (S-Function)

Format: Sxxxxx

Spindle speed is programmed via S-Code. The RPM range of the machine determines the S-Code range. Refer to Table 12-3, Control M-Codes.

In determining spindle speeds there also may be gear ranges selected by M-Codes. For example, you may be able to select four ranges of speed by programming the corresponding M-Code for the required range. The machine tool builder specifies the range. Check your machine tool manual for details.

All rights reserved. Subject to change without notice.

Miscellaneous Functions (M-Code)

Miscellaneous codes control a variety of machine tool functions. Refer to **Table 12-2**. The machine builder assigns them. Be familiar with the M-Codes available on your machine-control combination. M-function availability varies from one machine to another. Refer to your machine tool manual for a complete list of M-Codes.

Table 12-2, M-Code Controlled Functions

M-Code	Function
M3 or **M03**	**Spindle Forward**.
M4 or **M04**	**Spindle Reverse**.
M5 or **M05**	**Spindle Off**.
M6 or **M06**	**Tool Mount**.
M8 or **M08**	**Coolant On**.
M9 or **M09**	**Coolant Off**.

All rights reserved. Subject to change without notice.

Control M-Codes

Control M-Codes execute or alter certain CNC functions, such as program end, subprogram call, mirror image, etc.

These M-Codes are part of the CNC software. To use them, write the appropriate M-Code into the program. Refer to **Table 12-3**.

Table 12-3, Control M-Codes

M-Code	Function
M0 or **M00**	**Program Stop Mode**. Program stops indefinitely. Press **START** to resume.
M1 or **M01**	**Optional Program Stop**. Optional program stops indefinitely. Press **START** to resume.
M2 or **M02**	**End of Program**. At **M02**, the program stops and returns to the first program block.
M19	**Spindle Orientation**. (Followed by the C word for a spindle orient to a specific angle [i.e., M19 C45 orients the spindle to 45 degrees].)
M30	**Jump to New Program**. Return to other program. M30 O75 programmed, as the last block of a main program will return the CNC to program #75. O75 must be in the same file.
M98	**Call SubProgram**. A block in the main program with M98 P100 will execute subprogram 100. O100 must be in the file after the end of the main program.
M99	**End of SubProgram**. **M99** ends a subprogram and returns to the main program at the block preceding the last executed program call.
M105	**Dry Run, All Axes**. Program **M105** in a program file or in MDI to set Dry Run Mode. CNC executes all feed moves at a rate set by the builder. It enables you to run programs through quickly to check for mistakes. M107disables Dry Run. **NOTE:** Making and saving a change to the Setup Utility will cancel M105.
M106	**Dry Run, No Z Axis**. **M106** in a program file or in MDI sets Dry Run (No Z) Mode. All feed moves are executed at a rate set by the builder, and all Z moves are ignored during the dry-run. This enables you to run through a program quickly, without Z-axis movement. M107 disables Dry Run, No Z Axis. **NOTE:** Making and saving a change to the Setup Utility will cancel M106.
M107	**Dry Run, Off - Cancel M105 and M106**. This returns the CNC to normal operating mode.
SPEED	**Spindle Speed**. Commands spindle speeds (S).

All rights reserved. Subject to change without notice.

Order of Execution

The order of execution for available codes is as follows:

T, M, S, F, G, and XYZ (M98 P {sub call} is the exception)

NOTE: Subprogram call (M98 Pn) will always execute last.

All rights reserved. Subject to change without notice.
25-January-08

Section 13 - Machine Software and Peripherals Installation

The following topics are described in this section:

- **Keyboard Installation (Option)**
- **Keypad Equivalent Keyboard Keys**
- Peripherals Supported

Keyboard Installation (Option)

The machine builder determines whether the system will support a keyboard option. If the system supports a keyboard, plug the keyboard USB connector into the computer chassis.

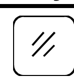
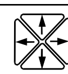
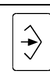
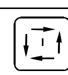
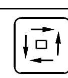
WARNING:	**There is no keyboard equivalent for the E-STOP. Therefore, emergency shutdowns (E-STOP) cannot be performed via keyboard.**

NOTE:	Industrial grade keyboards are recommended for shop environments.

Keypad Equivalent Keyboard Keys

Refer to **Table 13-1**.

Table 13-1, Keyboard Equivalents

Function	CNC Key Face	Keyboard Keystroke Equivalent
CLEAR		(**ALT** + **C**) *– or –* **DELETE**
ARROWS		**ARROWS**
ENTER		**ENTER**
X (axis)	X	(**X**)
Y (axis)	Y	(**Y**)
Z (axis)	Z	(**Z**)
U (axis)	U	(**U**)
START		(**ALT** + **S**)
HOLD		(**ALT**+ **H**)

All rights reserved. Subject to change without notice.

Peripherals Supported

The 6000i also supports other Universal Serial Bus (USB) devices:

- USB Memory Sticks
- USB Floppy Drives
- USB CD–ROM Drives
- USB Mouse

All rights reserved. Subject to change without notice.

Section 14 - Off-line Software

The off-line software is compatible with **Microsoft® **Windows® XP Operating System.

The hard disk drive must have a minimum of 1.5 GB of space available.

The following topics are described in this section:

- **Running and Shutting Down**

Contact your local ANILAM dealer or Sales office to obtain off-line software.

Running and Shutting Down

The software will automatically start running when the control is powered-up. When it is time to turn-off the machine the CNC software must be shutdown first. This is necessary to make sure that any data that may be in RAM is saved to the hard-drive. The shutdown process is very easy; pressing **Shut Down** (SHIFT + **F10**) displays the Shut Down screen with the following soft keys. Refer to **Table 14-1**.

Table 14-1, Shut Down Screen Soft Keys

Label	Soft Key	Description
Shut Down	**F1**	Confirm the shutdown.
Cancel	**F2**	Cancel the shutdown.

Follow the same procedure to shutdown the 6000i off-line as on the machine. If for some reason the 6000i off-line stops responding, try using the "Shutdown 6000i" option in the program group. (Left-click on Start at the bottom-right of the Windows display, select All Programs, highlight Anilam, and select SHUTDOWN 6000I.) Refer to **Figure 14-1**.

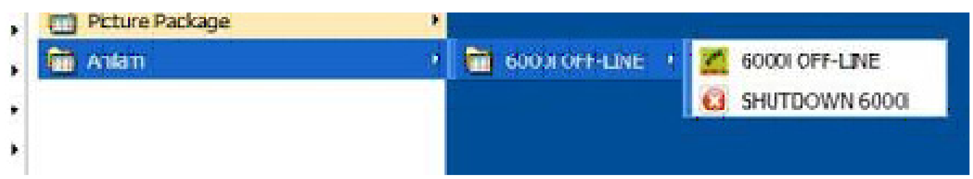


Figure 14-1, 6000i Off-line Program Group

As the software starts to come up (both machine and off-line), a counter is displayed showing the progress of the start-up sequence.

The counter reaches into the 70s when the main application screen (i.e.: manual mode) is displayed. At this point, a homing sequence must be done by pressing **Home** (**F4**) and START [or for the off-line (ALT + **S**)]. For off-line, use (ALT + **H**) to **Hold** the program.

- - - - -

** Microsoft® and Windows® are registered trademarks of Microsoft Corporation in the United States and/or other countries.

The off-line software will have a desktop icon or program group entry as shown in **Figure 14-1**.

Select either 6000i Off-Line option (desktop icon or program group entry) to start the software.

All rights reserved. Subject to change without notice.
25-January-08

Section 15 - Four-Axis Programming

The following topics are described in this section:

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: FU 500.0 = 500 dpm for the U axis.

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

All rights reserved. Subject to change without notice.

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:

All rights reserved. Subject to change without notice.

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 15-1 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 15-1, Four-Axis Example 1

```
* 4-AX-DRL
* SET shortestDistance to “off”
G90 G70 G0 M5
G28 Z0
G53 O1
G0 X0 Y0 U0
T1 *#3 CENTERDRILL
M3 S2400
G81 Z-.22 R.1 F12
M98 P1
T2 * 3/8" DRILL
M3 S1850
G53 O1 *RE-ACTIVATE OFFSET CANCELED IN SUBR #1
G87 Z-1 R.1 F14 I.18 J.012 K.1 U.3334
M98 P1
M2

O1 * ROTARY HOLE LOCATIONS
G0 G90 X-.75 Y0 U0
LOOP 9
G0 G91 U36
END
G0 G90 X-2 U0
LOOP 9
G0 G91 X-.5 U-36
END
G80
M5
G0 G90 G28 Z0 *CANCELS G53 OFFSET
X0 Y0 U0
M99
```

All rights reserved. Subject to change without notice.

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 15-2 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 15-2, Four-Axis Example 2

```
* 4-AX-MILL
* SET shortestDistance TO “off”
G90 G70 G0 M5
G28 Z0
G53 O1
G0 X0 Y0 U0
T1 *.25 BALL-END-MILL
S2400
M3
M98 P1 L6
G90 G0 M5
G28 Z0
G0 X0 Y0 U0
M2

O1 * GROOVE
G90 G0 X.225
G0 Z2.625
G1 X.125 F5
G18 G91 G2 X-.25 Z.25 I-.25 K0 U-2.
G17 G1 X-3.25 Z.125 U-13
G90 G0 Z3.225
G0 X.225
G91 G0 U-45
M99
```


All rights reserved. Subject to change without notice.
25-January-08

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 15-3 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 15-3, Four-Axis Example 3

```
* 4-AX-THD
* SET shortestDistance TO “on"
G90 G70 G0 M5
G28 Z0
G53 O1
G0 X0 Y0 U0
T1 * SPECIAL THD-TOOL
S3500
M3
G0 X.125 Y0 U0
Z.1
G1 Z-.075 F20
* SET shortestDistance TO “on"
* THIS IS TO PREVENT THE NEED TO UNWIND U
* 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
G91 G1 X-6.125 U((360*8*6)+360)
G90 G0 M5
G28 Z0
X0 Y0 U0
M2
```

All rights reserved. Subject to change without notice.

Section 16 - DXF Converter Feature

The DXF Converter feature allows information in a Drawing Exchange Format (.DXF extension) to be used to create a CNC conversational (.M extension) or G-Code (.G extension) file.

Contours and drill hole locations in the DXF file can be put in the CNC file in the form of subroutines, using a mouse and "point and click" approach.

The DXF Converter feature creates a CNC program that must be edited to be usable, but most of the program creation is already done.

The following topics are described in this section:

Requirements

The following topics are described:

- **Off-line Software**
- **Machine Software**

Off-line Software

The Personal Computer (PC) must have a mouse installed. The Anilam Off-line Software is required. The Anilam Off-line Software will run in a Windows environment. (See "Section 14 - Off-line Software.")

Machine Software

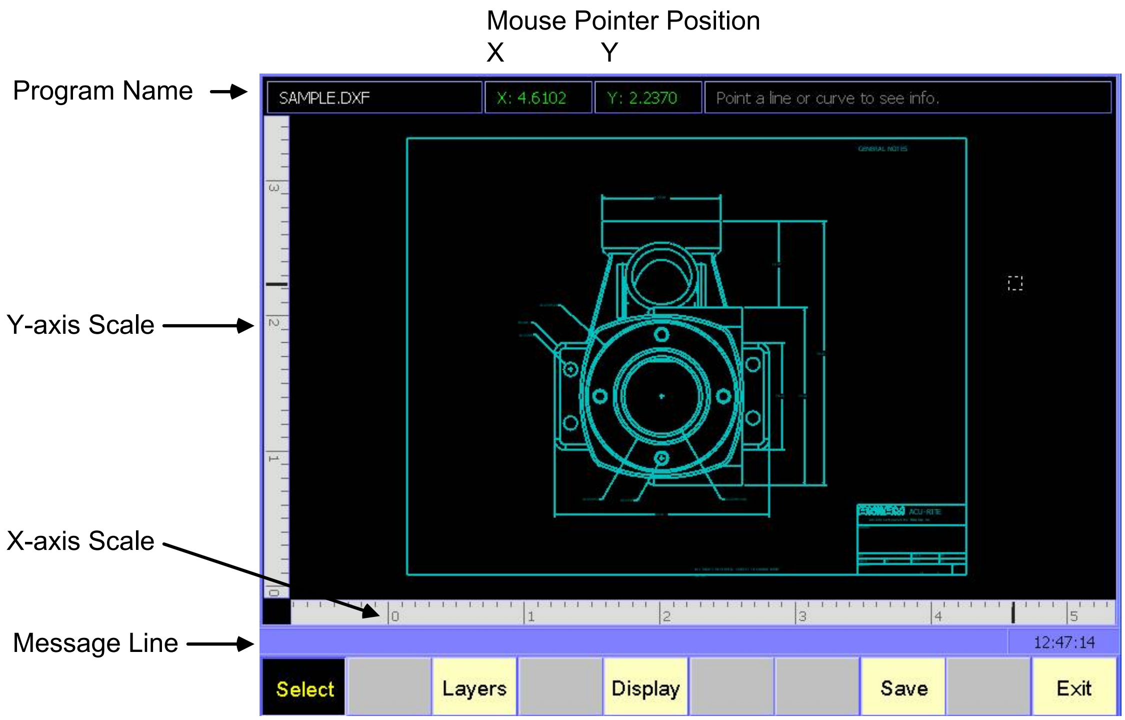
A mouse or other pointing device (for example, track ball) must be installed to properly operate the DXF converter on the machine. Depending on the mouse, it may be necessary to have the mouse connected before turning the CNC on.

All rights reserved. Subject to change without notice.

Entry to the DXF Converter

To open the DXF Converter (off-line software):

1. Open the Anilam Off-line Software
2. Gain access to the Program page, select **Program Type:** DXF drawings (*.dxf), and highlight the DXF file you wish to convert. For details on how to work with the Program page see "Section 10 - Program Management."
3. Select the **Edit** (**F8**) soft key to open and bring the drawing into the DXF converter. Refer to **Figure 16-1**.



DXF SCRN

Figure 16-1, DXF Screen

The drawing display screen shows the file name in the upper-left border, the current X Y position of the mouse pointer position at the top center, Y-axis vertical scale, X-axis horizontal scale, DXF message line, and several soft keys along the bottom edge. Refer to Table 16-2, Soft Key Description.

The following topics are described:

- Creating Shapes
- Contours
- Drilling

All rights reserved. Subject to change without notice.

Creating Shapes

The part drawing is used to create shapes. Shapes are then output to CNC programs as subroutines. Converting to DXF edit creates the subroutines to change the G-Code file. You change the features (subroutines) attached to the G-Code file. There are many features to make the drawing screen easier to use. Layers may be turned on or off. Any area of the screen can be zoomed in or out. Refer to Table 16-1, Mouse Operations.

Each shape is given a sequential number. The number is displayed on the screen at the beginning of the shape. Press **Select** (**F1**) to toggle Select mode on. With Select mode on, entities can be selected to be part of a shape. Left click with the mouse to make selections. When an entity is selected, it will change color.

When selecting entities, direction is determined by where the mouse pointer is positioned. If nearer the left side of a line, direction will be left to right.

There are two types of shapes:

- One used for contouring or feed motion
- One for drilling a series of holes

Contours

Pick an entity where the shape will begin. Pick the last entity in the shape. All entities that are connected will be chained together and change color to verify this. Some shapes have to be selected one entity at a time. This is determined by the way the part was drawn in the DXF file.

If an entity is selected that is not connected to the previous one, a message is displayed, "**SHIFT** + Select to start a new contour". This message means you have selected an unchainable entity, maybe by mistake. You must press and hold **SHIFT** if the intent is to start a new contour. When a chainable entry is selected, a new shape number is displayed.

Entities in a shape can be un-selected by clicking them again. This un-selects everything previously selected to that point. To delete a shape, click on the first entity. The first entity is typically colored differently for this purpose.

Drilling

When circle entities are selected, they are assumed to be drilling hole locations. As circles are selected, a dotted line shows the rapid path between holes. Selecting anything other than a circle, ends the drilling shape and produces the message, "**SHIFT** + Select to start a new contour"

If you wish to create a second drilling shape, select the first hole of the next shape while holding the **SHIFT** key. This tells the application that you want to start a new contour.

All rights reserved. Subject to change without notice.

CNC Code

Each shape that is created is made into a subroutine. For each subroutine, there is a call in the main program. Running the CNC program in Draw mode allows the tool paths to be seen.

The file must be edited to add tool numbers, feed rates, cutter comp on or off, and so forth.

The tool paths are only as accurate as the DXF drawing file used.

Mouse Operations

Refer to **Table 16-1**.

Table 16-1, Mouse Operations

Button	Event	Function
Left	Press–Drag–Release	Zoom Window
Right	Press–Drag-Release	Pan
Left	**SHIFT** + Click	New contour signal
Left	Click (Select mode on)	Select Entity

All rights reserved. Subject to change without notice.

DXF Soft Keys

Refer to **Table 16-2**.

Table 16-2, Soft Key Descriptions

Soft Key	Function	Description
F1	**Toggle Select Mode**	Select mode must be on when chaining shapes.
F3	**Layers Menu**	Pop-up menu has: ▪ All Layers on ▪ Invert Layers ▪ Toggle Layers Layers can be turned on or off as desired.
F5	**Display Menu**	Pop-up menu has: Fit, Window, Half, and Double. Select the desired display.
F8	**Save**	Creates CNC code. The message, "Successfully created (filename) (.M or .G)." is displayed when Save is activated. If no shapes are defined, a warning message is displayed.
F9	**Setup**	Set the parameters for the DXF conversion.
F10	**Exit**	**F10** exits the Setup menus, exits the DXF Converter, and returns to the Program page. Be sure to Save (**F8**) any work done before exiting. Anything not saved will be lost. If shapes have been created, a message, "Exit (Y/N)?" is displayed. This is a reminder to be sure you have saved your work.

The following topics are described:

- Fitting the Display to the Viewing Window
- Using the Window Zoom
- Halving Display Size
- Doubling Display Size

All rights reserved. Subject to change without notice.

Fitting the Display to the Viewing Window

The DXF Converter can automatically scale the display to fit into the viewing area.

To fit the display in the viewing area:

1. In DXF, press **Display** (**F5**). A pop-up menu displays.
2. Highlight **Fit**, and press **ENTER**. The pop-up closes and the display adjusts to fit into the viewing window.

Using the Window Zoom

The DXF Converter allows you to zoom in on any part of the display.

To zoom in on part of the display:

1. In DXF, press **Display** (**F5**). A pop-up menu displays.
2. Highlight **Window**, and press **ENTER**. A window displays inside the viewing window.
3. Use the mouse to center the window over the area of interest.
5. Once the window is positioned, press **ENTER**. The part of the display framed by the window will fill the viewing window.

Halving Display Size

The DXF Converter can reduce the size of the display to half the existing size.

To reduce the display size by half:

1. In DXF, press **Display** (**F5**). A pop-up menu displays.
2. Highlight **Half**, and press **ENTER**. The display will be half its present size.

Doubling Display Size

The DXF Converter can double the size of the display.

To double the size of the display:

1. In DXF, press **Display** (**F5**). A pop-up menu displays.
2. Highlight **Double**, and press **ENTER**. The display will be twice its present size.

All rights reserved. Subject to change without notice.

DXF Entities Supported

See **Table 16-3** for the DXF entities supported.

Table 16-3, DXF Entities Supported

Entities	Drawing	Transformation	Chaining	Information
Line	X	X	X	X
Point	X	X		X
Circle	X	X	X	X
Arc	X	X	X	X
Trace	X	X		X
Solid	X	X		X
Text	X	X		
Shape	X	X		
Insert	X	X		
Attdef	X	X		
Attribute	X	X		
Vertex	X	X		X
Polyline	X	X	X	X
Line3d	X	X	X	X
Face3d	X	X		X

The following topics are described:

- **Drawing Entities Not Supported**

Drawing Entities Not Supported

See **Table 16-3**. Note that the Extrusion, Dimension, and Viewpoint entities are not supported. Dimensions may be seen on the displayed DXF file. Some DWG (an AutoCad drawing file) to DXF converters convert the dimension entities into lines and arcs, which are supported entities.

All rights reserved. Subject to change without notice.

Files Created

The DXF Converter creates the CNC file, .G for G-Code and .M for conversational, based on the setting of the **Output format** parameter.

A file is also created with the extension .sel. This file saves the status of parameter settings that were used in Setup.

DXF Examples

From the Program listing open the DXF file. Refer to **Figure 16-2**.

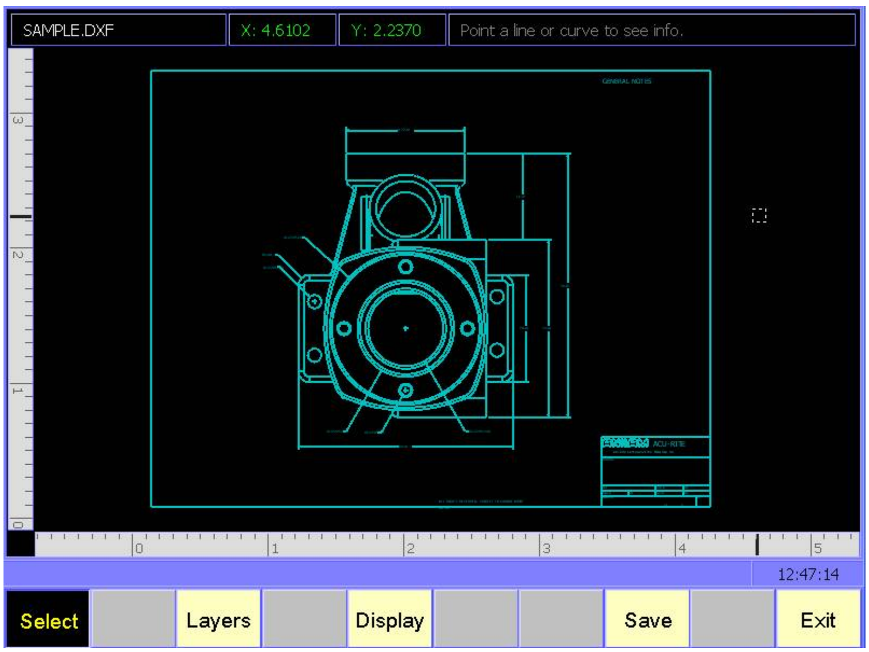
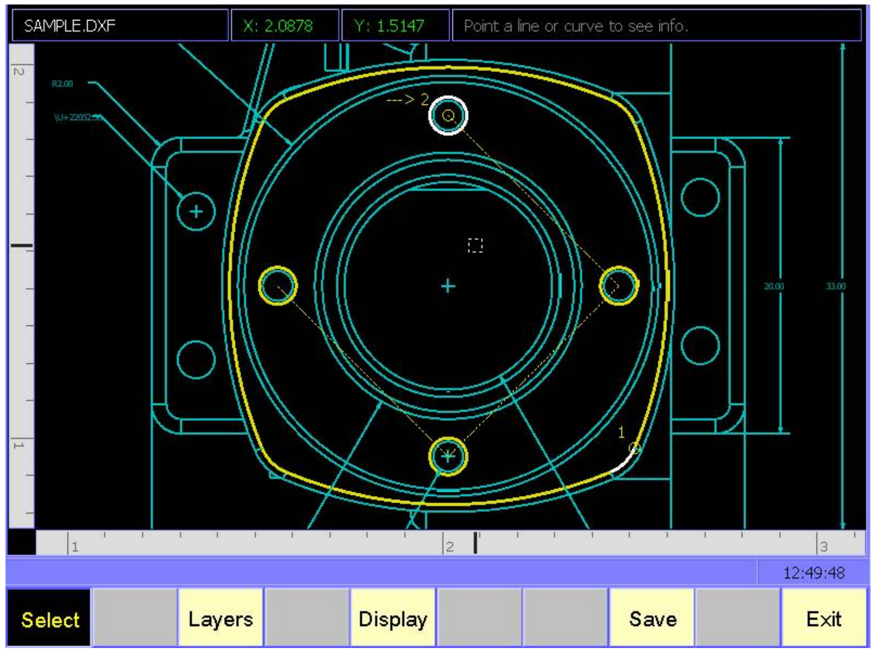


Figure 16-2, Example DXF File

All rights reserved. Subject to change without notice.

Refer to **Figure 16-3**. Many unneeded layers have been turned off. The Figure shows the drill locations and the contour selected (numbered 1 and 2).



ZOOM PART

Figure 16-3, Zoomed Part

The following topics are described:

- Unedited Conversational Program Listing
- Unedited G-Code Program Listing
- Unedited Program Run in Draw
- Edited Conversational Program Listing
- Edited G-Code Program Listing

All rights reserved. Subject to change without notice.

Unedited Conversational Program Listing

The CNC conversational program is created and must be edited to be usable. An unedited conversational program created from Figure 16-3, Zoomed Part follows. See **Table 16-4**.

– or –

An unedited G-Code program created from the Figure 16-3, Zoomed Part example is listed in Table 16-5, Unedited G-Code Program Listing.

Table 16-4, Unedited Conversational Program Listing

```
* Simple setup for easy testing
Dim Abs
Unit Inch
Feed 80
@ T1 D.01 L.01
Line Z -1.0

* Call sub program(s)
Call 1
Call 2
EndMain

* Sub program definition(s) ...

Sub 1
Rapid   X 2.51037 Y 0.97667
Arc Cw  X 2.44669 Y 0.91299 XCenter -0.10905 YCenter 0.04536
Arc Cw  X 1.58477 Y 0.91299 XCenter -0.43096 YCenter 1.03598
Arc Cw  X 1.52108 Y 0.97667 XCenter 0.04536 YCenter 0.10905
Arc Cw  X 1.52108 Y 1.83859 XCenter 1.03598 YCenter 0.43096
Arc Cw  X 1.58477 Y 1.90228 XCenter 0.10905 YCenter -0.04536
Arc Cw  X 2.44669 Y 1.90228 XCenter 0.43096 YCenter -1.03598
Arc Cw  X 2.51037 Y 1.83859 XCenter -0.04536 YCenter -0.10905
Arc Cw  X 2.51037 Y 0.97667 XCenter -1.03598 YCenter -0.43096
EndSub

Sub 2
Rapid   X 2.01573 Y 1.86039
Rapid   X 2.46848 Y 1.40763
Rapid   X 2.01573 Y 0.95488
Rapid   X 1.56297 Y 1.40763
EndSub


* Approximated stock for easy 3D simulated draw
BlockForm XMax 2.71258 YMax 2.10448 ZMax 0.00000 XMin 1.31888
YMin 0.71078 ZMin -2.00000
```


All rights reserved. Subject to change without notice.
25-January-08

The unedited conversational program generated will automatically have sample tool mode and stock information that can be used as guidelines to create the desired program. Also these sample commands enable the user to instantly run the generated program in Draw and visualize the tool path without having the edit it first. The unedited conversational program can be edited to fix and/or add tool numbers, feed rates, cutter comp on or off, and so forth. When the edits are complete, use Draw to check the tool path. See Figure 16-4, Unedited Program Run in Draw.

Unedited G-Code Program Listing

The CNC G-Code program is created that must be edited to be usable. An unedited G-Code program created from Figure 16-3, Zoomed Part follows. See **Table 16-5**.

Table 16-5, Unedited G-Code Program Listing

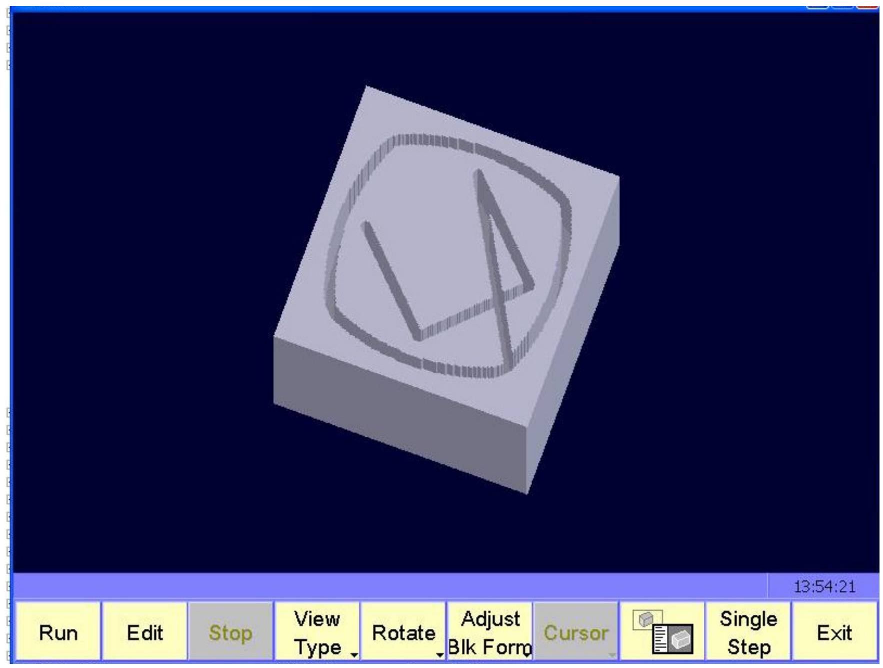
```
* Simple setup for easy testing
G90G70F80
T1 D.01 L.01
G1 Z-1.0

* Call sub program(s)
M98 P1
M98 P2
M2

* Sub program definition(s) ...

O1
G0 X2.51037 Y0.97667
G2 X2.44669 Y0.91299 I-0.10905 J0.04536
G2 X1.58477 Y0.91299 I-0.43096 J1.03598
G2 X1.52108 Y0.97667 I0.04536 J0.10905
G2 X1.52108 Y1.83859 I1.03598 J0.43096
G2 X1.58477 Y1.90228 I0.10905 J-0.04536
G2 X2.44669 Y1.90228 I0.43096 J-1.03598
G2 X2.51037 Y1.83859 I-0.04536 J-0.10905
G2 X2.51037 Y0.97667 I-1.03598 J-0.43096
M99

O2
G0 X2.01573 Y1.86039
G0 X2.46848 Y1.40763
G0 X2.01573 Y0.95488
G0 X1.56297 Y1.40763
M99


* Approximated stock for easy 3D simulated draw
G120 X2.71258 Y2.10448 Z0.00000 I1.31888 J0.71078 K-2.00000
```

All rights reserved. Subject to change without notice.

The unedited G-Code program generated will automatically have sample tool mode and stock information that can be used as guidelines to create the desired program. Also these sample commands enable the user to instantly run the generated program in Draw and visualize the tool path without having the edit it first. The unedited G-Code program can be edited to fix and/or add tool numbers, feed rates, cutter comp on or off, and so forth. When the edits are complete, use Draw to check the tool path. See **Figure 16-4**.

Unedited Program Run in Draw

The unedited program run is draw is illustrated in **Figure 15-4**.



UNEDITED RUN IN DRAW

Figure 16-4, Unedited Program Run in Draw

The edited program listings are shown in the following Tables:

- Table 16-6, Edited Conversational Program Listing
- Table 16-7, Edited G-Code Program Listing

All rights reserved. Subject to change without notice.

Edited Conversational Program Listing

See **Table 16-6**.

Table 16-6, Edited Conversational Program Listing

```
Dim Abs
Unit Inch
DrillOff
MCode 5
*FIXTURE OFFSET
Offset    Fixture# 1
*TOOL CALL SET OFFSET IN TOOL TABLE .375 DIA MILL
Tool# 1    MCode 6
*SET SPINDLE SPEED
RPM 1500
*TURN ON SPINDLE
MCode 3
*TURN ON COOLANT
MCode 8
*POCKET USING SUBR #1
Pocket   Sub# 1 X 2.00 Y 1.500 StartHgt 0.100 ZDepth -0.375 Stepover
0.250 DepthCut 0.125 FinStock 0.005 RampFeed 10.0 RoughFeed 30.0
FinFeed 20.0
*TURN OFF SPINDLE
MCode 5
*TURN OFF COOLANT
MCode 9
*CALL CENTER DRILL FOR HOLES
Tool# 2    MCode 6
RPM 3500
MCode 3
MCode 8
*SETUP CENTER DILLING CYCLE
BasicDrill ZDepth -0.5 StartHgt -0.275 ReturnHgt 1.0 Feed 15.0
*CALL SUBR FOR HOLES
Call 2
DrillOff
MCode 5
MCode 9
*CALL .125 DRILL FOR HOLES
Tool# 3    MCode 6
RPM 2500
MCode 3
MCode 8
*SETUP PECK DILLING CYCLE
PeckDrill  ZDepth -1.25 StartHgt -0.275 ReturnHgt 1.0 Peck 0.50 Feed
20.0
Call 2
DrillOff
MCode 5
MCode 9
```

All rights reserved. Subject to change without notice.

```
*CALL .25 COUNTER BORE FOR HOLES
Tool# 4    MCode 6
RPM 2000
MCode 3
MCode 8
*SETUP  COUNTERBORE CYCLE
Boring     ZDepth -0.625 StartHgt -0.275 ReturnHgt 1.0 Dwell 0.1 Feed
20.0
Call 2
DrillOff
MCode 5
MCode 9
Dim Abs
Rapid    Z 5.0
Rapid    X -10.0 Y 5.0
EndMain

* Sub program definition(s) ...

Sub 1
Line    X 2.51037 Y 0.97667 ToolComp Right
Arc Cw  X 2.44669 Y 0.91299 XCenter -0.10905 YCenter 0.04536
Arc Cw  X 1.58477 Y 0.91299 XCenter -0.43096 YCenter 1.03598
Arc Cw  X 1.52108 Y 0.97667 XCenter 0.04536 YCenter 0.10905
Arc Cw  X 1.52108 Y 1.83859 XCenter 1.03598 YCenter 0.43096
Arc Cw  X 1.58477 Y 1.90228 XCenter 0.10905 YCenter -0.04536
Arc Cw  X 2.44669 Y 1.90228 XCenter 0.43096 YCenter -1.03598
Arc Cw  X 2.51037 Y 1.83859 XCenter -0.04536 YCenter -0.10905
Arc Cw  X 2.51037 Y 0.97667 XCenter -1.03598 YCenter -0.43096
EndSub

Sub 2
Rapid   X 2.01573 Y 1.86039
Rapid   X 2.46848 Y 1.40763
Rapid   X 2.01573 Y 0.95488
Rapid   X 1.56297 Y 1.40763
EndSub


* Approximated stock for easy 3D simulated draw
BlockForm XMax 2.71 YMax 2.1 ZMax 0 XMin 1.32 YMin 0.71
ZMin -1.125
```


All rights reserved. Subject to change without notice.

Edited G-Code Program Listing

Table 16-7, Edited G-Code Program Listing

```
G0 G90 G70 G40 G80
G53 O1       *FIXTURE OFFSET
T1 M6        *TOOL CALL SET OFFSET IN TOOL TABLE .375 DIA MILL
S1500 M3    *TURN ON SPINDLE
M8           *TURN ON COOLANT
*POCKET USING SUBR #1
G169 W1 X2.0 Y1.5 H.1 Z-.375 M.01 S.005 A.25 B.125 I10.0 J30.0
K20.0 P1.0
M5           *TURN OFF SPINDLE
M9           *TURN OFF COOLANT
T2 M6        *CALL CENTER DRILL FOR HOLES
S3500 M3  *TURN ON SPINDLE
M8          *TURN ON COOLANT
*SETUP CENTER DILLING CYCLE
G81 Z-.5 R-.275 F15.0 P1.0
M98 P2       *CALL SUBR FOR HOLES
G80
M5
M9
T3 M6        *CALL .125 DRILL FOR HOLES
S2500 M3
M8
*SETUP PECK DILLING CYCLE
G83 Z-1.25 R-.275 I.5 F20.0 P1.0
M98 P2       *CALL SUBR FOR HOLES
G80
M5
M9
T4 M6        *CALL .25 COUNTER BORE FOR HOLES
S2000 M3
M8
*SETUP  COUNTERBORE CYCLE
G82 Z-.625 R-.275 D.1 F20.0 P1.0
M98 P2       *CALL SUBR FOR HOLES
G80
M5
M9
G0 G90 Z5.0
X-10 Y5
M2 *END PROGRAM

* Sub program definition(s) ...
O1
G42 *ADDED FOR IRREG POCKET
G1 X2.51037 Y0.97667
G2 X2.44669 Y0.91299 I-0.10905 J0.04536
G2 X1.58477 Y0.91299 I-0.43096 J1.03598
```

All rights reserved. Subject to change without notice.

```
G2 X1.52108 Y0.97667 I0.04536 J0.10905
G2 X1.52108 Y1.83859 I1.03598 J0.43096
G2 X1.58477 Y1.90228 I0.10905 J-0.04536
G2 X2.44669 Y1.90228 I0.43096 J-1.03598
G2 X2.51037 Y1.83859 I-0.04536 J-0.10905
G2 X2.51037 Y0.97667 I-1.03598 J-0.43096
G1 X2.51037 Y0.97667
M99

O2
G0 X2.01573 Y1.86039
G0 X2.46848 Y1.40763
G0 X2.01573 Y0.95488
G0 X1.56297 Y1.40763
M99
```

*** Approximated stock for easy 3D simulated draw**

```
G120 X2.71 Y2.1 Z0 I1.32 J0.71 K-1.125
```


All rights reserved. Subject to change without notice.
25-January-08

Section 17 - Advanced Programming Features

The following topics are described in this section:

Modifiers

Use modifiers to alter the way the CNC interprets a word address. For example, a single value in an Inch Mode program may be forced to Metric Mode, without programming G71. Or, arc center values (I, J, or K) may be forced to an absolute value.

The address and modifier must be accompanied by an ampersand (&). Place the ampersand (&) between the address word to be modified and the modifier. The address word is programmed first, followed by &, followed by the modifier, followed by the value.

The modifier is non-modal and is applied only to the address word it accompanies.

Example

G02 X2.0 Y1.0 I&A1.5 J&A1.0

The example forces the I and J center of an arc to be in Absolute Mode. I and J are incremental by default. Assume the axes are at X1 Y1. **Table 17-1** lists the available modifiers.

Table 17-1, Modifiers

A	Force the address word to be in Absolute Mode.
D	Force the address word to be in Incremental Mode.
E	Force the address word to be in Inch Mode.
M	Force the address word to be in Millimeter Mode.

All rights reserved. Subject to change without notice.

Block Separators

Block separators (;) can be used to place several functions on one line of a program. This is useful in Manual Data Input (MDI) Mode because you can combine several commands on one line at the command line.

Example 1 will execute five moves on the machine when you press **START**. Each move is separated by the (;) block separator.

Example 1:

```
G90 G01 X0 Y0 F30 ; X3 ; Y-2 ; X0 ; Y0
```

Example 2 will move the axes linearly to X0 Y0, then CW to X1 Y1, then linearly to X2.

Example 2:

```
G90 G01 X0 Y0 F10 ; G02 X1 Y1 I1 J0 F8 ; G01 X2
```

The number of separate steps in a program file is limited only by the available memory.

Block separators can also be used in programs.

Tool Offset Modification

You can modify a tool diameter or length offset in the program without using the Tool Page. This is useful when rough-milling a profile where cutter diameter compensation requires different diameter definitions for the same tool to step the width of the cut. Tool modification can be either temporary or permanent. To make it temporary, choose not to update the Tool Page. To make it permanent, choose to update the Tool Page. Refer to **Figure 17-1**.

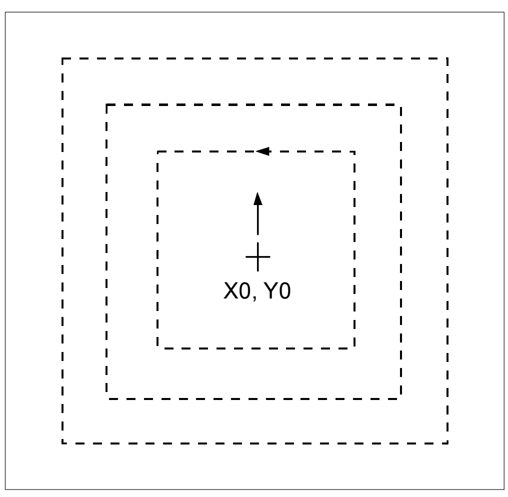


Figure 17-1, Tool Modification Programming Example

All rights reserved. Subject to change without notice.

Temporary Format:

T1 D.5500 L-1.1000

Changes Tool 1 diameter offset to .5500 and length offset to -1.1000. Do not update the Tool Page for Tool 1.

Permanent Format:

T1 D.5500 L-1.1000 H M6

Changes Tool 1 diameter offset to 0.5500 and length offset to -1.1000. Updates the Tool Page for Tool 1 to entered values.

D and L values are absolute and replace the previous offsets. They are not added to existing offsets. The H command instructs the CNC to update the Tool Page offsets to the programmed values and must come between the M6 and the D & L addresses, if an M6 is required by the tool change of your machine.

All rights reserved. Subject to change without notice.

Tool Modification Programming Example

This program will mill the square shape four times. The CNC executes the first pass using the tool diameter entered in the Tool Page. Each subsequent pass will use a different, "modified" tool diameter, as programmed in Blocks 8, 10, and 12. T, D, L, and H are the only word addresses allowed on the block.

N1	O41 * TOOL-MOD.G
N2	G90 G70 G0 G17
N3	T0
N4	Z0
N5	X0 Y0
N6	T1 * .8000 DIA.
N7	M98 P1
N8	T1 D.6
N9	M98 P1
N10	T1 D.4
N11	M98 P1
N12	T1 D.2
N13	M98 P1
N14	T0
N15	G0 Z0
N16	M2
N17	
N18	O1 * SUBPGM-1
N19	G1 Z-.25 F10
N20	G41 Y1
N21	X-1
N22	Y-1
N23	X1
N24	Y1
N25	X0
N26	G40 Y0
N27	M99

The main program calls the subprogram that contains the compensation on/off commands between each tool modification.

NOTE:	When tool modifiers are activated, the CNC still applies any wear offset entered in the Tool Page.

All rights reserved. Subject to change without notice.

Expressions and Functions

You can program some values as expressions. Parentheses enclose expressions. The CNC displays an error message if the expression is incorrectly entered. Expressions follow the standard mathematics order of operations (multiplication, division, addition and subtraction).

An expression must contain an operator or use a function. Refer to **Table 17-2**.

Table 17-2, Operators and Functions

Ref.	Expression	Function
a)	( )	Expression function (parenthesis)
b)	* / &	Multiplication, division, modification
c)	+ -	Addition, subtraction
d)	> <	Relation greater than, less than
e)	= !=	Relation equal, not equal
f)	tomm	Convert to mm
g)	toin	Convert to inch
h)	tode	Convert to inch if inch, mm if mm
i)	tonu	Force to current modal
j)	round	Round up or down, automatically
k)	fix	Discard fraction less than 1
l)	fup	Raise fraction 1
m)	var	True if defined, false otherwise
n)	sin	Sine
o)	cos	Cosine
p)	tan	Tangent
q)	asin	Arcsine
r)	acos	Arcosine
s)	atan	Arctangent
t)	abs	Absolute value
u)	sqrt	Square root
v)	ln	Natural logarithm
w)	log	Logarithm
x)	exp	Exponential
y)	trun	Truncate
z)	! + - #	Unary logical not, positive, negative, indirection

The following topic is described:

- Examples

All rights reserved. Subject to change without notice.

Examples

Ref. from Previous Table	Example
a)	G01 X(#100 + #101). All calculations must be enclosed in parentheses. This defines an expression.
b)	G00 Y&A(#102 * #103) LOOP (5 / 2 / .01) Example of multiplication, division and modification.
c)	G01 X(3 + 2) #100 = (#122 - #105). Addition and Subtraction.
d)	IF (#101 > 0) THEN .... Greater than (>), less than (<).
e)	IF (#144 = #143) GOTO ..... Equal to, not equal to (!=).
f)	TOMM (n); convert n to mm. If n's type is inch, TOMM (n) = n * 25.4.
g)	TOIN (n); convert n to inch. If n's type is mm, TOIN (n) = n / 25.4.
h)	TODE (n); convert to current (IN or MM) mode.
i)	TONU (n); force the type of (n) to the modal (inch or mm).
j)	ROUND (n) will round the value of (n) up or down, depending if its fractional part is equal or greater than 0.500000, or less than 0.500000. #100 = 1.500 ; G01 X(round(#100)) will move to X2.0000 #101 = 1.499 ; G01 X(round(#101)) will move to X1.0000
k)	FIX (n) will round the value down to the next whole number. #100 = (5/2) ; G01 X(fix(#100)) will move to X2.0000
l)	FUP (n) will round the value up to the next whole number. #100 = (5/2) ; G01 X(fup(#100)) will move to X3.0000
m)	VAR (n) is used to check if a user variable has been defined in a program. IF (var(#100)) THEN .... If #100 has been defined by the user, then true. If not, then false.
n)	SIN (n) will give the sine of (n). (n) is assumed to be in degrees. G01 X(cos(15)) Y(sin(15)) will move along the hypotenuse of a 15-degree angle with a hypotenuse of 1.
o)	COS (n) will give the cosine of (n).
p)	TAN (n) will give the tangent of (n).
q)	ASIN (n) will give the arcsine of (n).
r)	ACOS (n) will give the arccosine of (n).
s)	ATAN (n) will give the arctangent of (n).
t)	ABS (n) will give the absolute value of (n).
u)	SQRT (n) will give the square root of (n).
v)	LN (n) is natural logarithm.
w)	LOG (n) is logarithm.
x)	EXP (n) is exponential function.

All rights reserved. Subject to change without notice.

Ref. from Previous Table	Example
y)	TRUN (n) will truncate the value of (n).
z)	! unary logical not, != (not equal to). Positive, (+(#100)) means positive whatever value is in #100. Negative, (-(#100)) means negative whatever is in #100. Example of indirection: N30 #200 = 51.456 N40 #201 = 200 N50 G90 G1 X ##201 F200 At Block N40 variable #201 = 200. Only when the second level of indirection is used at N50 does variable #201 contain the contents of variable #200 causing the X-axis to move to position 51.456. Up to four levels of indirection can be used.

All rights reserved. Subject to change without notice.

System Variables

Certain variables are set aside as CNC system variables. Some may be useful for you to know when programming macros. The system variables range from #1000 to #1099. Most of these variables are "read only". You cannot write information to them. There are a few exceptions to this rule. Refer to **Table 17-3** for a list of available system variables.

Table 17-3, System Variables

Variable	Description
#1000	Block skip variables (read/write)
#1001 to #1009	Selective block skip
#1010 to #1015	Commanded ABS tool position (x,y,z,u,v,w) **NOTE:** These variables are not valid during compensations such as tool radius, scaling, mirroring and rotation or while in transitional moves such as corner rounding and chamfering.
#1016	Current G motion mode (0=rapid, 1=feed, 2=cw arc, 3=ccw arc, 5=ellipse, 6=spiral)
#1017	Current XYZ plane (17=XY, 18=XZ, 19=YZ)
#1020	Current tool diameter
#1021	Current tool length offset
#1022	Current feedrate
#1023	Current rapidrate
#1024	Current RPM
#1030	Stock variable (R/W)
#1031	Acute angle for rounding compensated intersections (default = 15.0)
#1032	# of look-ahead blocks for cutter comp (R/W)
#1041	Current program tool compensation (40=off, 41=left, 42=right)
#1050 to #1055	Actual absolute position (X,Y,Z,U,V,W) **NOTE:** These variables are not valid during compensations such as tool radius, scaling, mirroring and rotation or while in transitional moves such as corner rounding and chamfering.
#1070	Current XYZ dimension (70=inch, 71=mm)
#1071	Current UVW dimension (70=inch, 71=mm)
#1090	Current XYZ dimension (90=abs, 91=incr)
#1091	Current UVW dimension (90=abs, 91=incr)

All rights reserved. Subject to change without notice.

User Variables

Certain variables are set aside for the programmer to use. These may be useful when programming macros. You can read from or write to these variables. They are divided into four categories:

- Local variables: #1 to #99

  These variable numbers can be used only within the body of a subprogram (or macro). The CNC generates an error message if you program these variables in the main program. Values do not hold from one subprogram to another. In this way, the same variables can be used in separate subprograms, with different values.
- Common (global) variables: #100 to #219. (Read/Write)

  These variables can be used anywhere in the program or subprogram and their value will remain.
- Read only variables: #220 to #249

  These variables can only be set in the main program. Once set, the variables can be used in subprograms or macros as "read only" variables.
- Static (global) variables: #260 to #279. (Read/Write)

  These variables can be used anywhere in the program or subprogram and their value will remain across shutdowns or software resets.

WARNING:	**OEM and machine tool builders should use #100–#150 and #260–#269 for any custom macros. End users should use #151–#220 and #270–#279 for your custom macros. This avoids conflicting usage of the global variables.**

- Block skip variables: #1000 to #1009 (Refer to "Block Skip" and "Selective Block Skip" in this section.)

The following topic is described:

- Variable Programming (Parametric Programming)

All rights reserved. Subject to change without notice.

Variable Programming (Parametric Programming)

Variable, or parametric, programming enables you to create macros to generate geometric shapes that are not already available in a canned cycle.

Conditional loops, jumps, and GOTO commands can be used to control program execution.

The following topics are described:

Block Skip

Any block preceded by a slash (/) code is omitted if the corresponding block skip 'switch' is set "ON" in the program, previous to the (/) code. The switch is set on by programming variable #1000 to the value of "1". A value of "0" sets the switch "OFF".

Example:

```
N11     #1000 = 1                *Note:  0=OFF, 1= ON
N12     G81  Z-.5  R.1  F12  P.1
N13     X1 Y1
N14     X2
/N15     X3
N16     X5
N17     G80
```

In the above example, the hole at N15 will be skipped. If N11, read

```
N11     #1000 = 0
```

then N15 would be executed.

The (/) slash code can be placed anywhere in the block, provided it does not exclude code necessary to complete that operation. It is therefore recommended the (/) code be used as shown above.

#1000 is reserved for block skip use.

All rights reserved. Subject to change without notice.
25-January-08

Selective Block Skip

The 6000i control has nine (9) optional block skip 'switches'. The (/) code followed by a number 1 through 9 will activate the corresponding switch.

Example:

```
N11     #1002 = 1                *Note:  0=OFF, 1= ON
N12     G81  Z-.5  R.1  F12  P.1
N13     X1 Y1
N14     X2
/2  N15   X3
N16     X5
N17     G80
```

In the above example, the hole at N15 will be skipped. If N11, read

```
N11     #1002 = 0
```

then N15 would be executed.

#1001 through #1009 are reserved for optional block skip use.

Parameters and Variable Registers

A macro is a series of instructions designed to achieve a specific result for a given set of constraints. For example, a rectangular pocket of any size always has four sides, four corner radii and a depth. Therefore, you can cut many pockets of different sizes using a similar tool path with longer or shorter moves for the tool path. If a suitable program processes the constraints of the pocket, the CNC calculates a tool path to cut a particular pocket. Such a program is called a macro. The G78 rectangular pocket cycle is an example of a macro that cannot be edited.

The constraints of the pocket, or the feature required, are its parameters. Parameters for any feature will vary as dimensions change, therefore the parameters are often called variables. The data for each parameter must be stored as an entity, known as a variable register, also called variables.

Parameters passed to a macro will be called parameters in this manual.

All rights reserved. Subject to change without notice.

Contents of Variables (PRINT)

Format:

PRINT xxx(variable)

Format:

N(Block number) PRINT xxx(variable)

You can verify the contents of a variable. This is useful when you are debugging a program. Use the PRINT command to display the contents of a variable on the screen in Manual, Single-Step and Auto Modes.

Example 1:

PRINT 200

Displays the contents of the variable (#200).

Example 2:

N180 PRINT 110

To display variable contents during program execution, use the PRINT command as part of the program. Example 2 will print the contents of variables #110 to the screen.

If commas separate the numbers, several variables can be printed simultaneously.

In Manual Mode, type **PRINT 110** then press **START** to display the contents on the screen.

The PRINT variable can be abbreviated, as follows:]P

See "Short Form Addressing," Table 17-6, Abbreviations, for a list of abbreviations to activate the corresponding command.

All rights reserved. Subject to change without notice.

Setting and Transferring Variables

When using parametric programming with axis addresses and expressions (including unary minus), the complete expression needs to be in parenthesis. For example, X(-#151) is correct. X-#151 or X-(#151) are not correct.

Setting and Direct Transfer:

Variables are loaded or set when they appear on the left side of an equation. (That is, the left side of the equal sign.)

Example 1:

```
N200   #100 = 5.56
```

Variable #100 contains number 5.560000 until changed.

Example 2:

```
N200   #100 = 25.4m
```

Variable #100 sets variable 100 to 25.4mm. Similarly, #100 = 5i sets variable 100 to 5 inches. If neither "i" nor "m" are used when assigning a variable, then the value of the variable is modal. For example, #100 = 8 sets variable #100 to 8 (no unit).

Example 3:

```
N200   #100 = #20
```

Variable #100 contains the number held by #20 until changed. Equating one variable with another is called a direct transfer.

NOTE:	When a direct transfer is requested, the variable on the right side of the equation must contain a value. Otherwise, the CNC displays an error message(#nn not defined).

Indirect Transfer:

You can indirectly transfer variables to a depth of four levels by introducing extra hatch marks (#) before the variable number. In an indirect transfer, a value is transferred to one variable via another.

Example 1:

```
N201   G90 G17 G71 G0
N202   #101 = 51.456
N203   #102 = 101
N204   X##102
```

At Block N204, the X-axis moves to 51.456. Example 1 shows single indirection. The contents of variable #101 are used by variable #102. The actual content of variable #102 is constant 101. The indirection is activated at Block N204 by the addition of the (#) symbol to variable #102.

All rights reserved. Subject to change without notice.

Example 2:

```
N210    G90 G17 G71 G0
N211    #101 = 1
N212    #102 = 2
N213    #103 = 3
N214    #104 = 4
N215    #119 = 100
N216    LOOP 4
N217    #119 = #119 + 1
N218    #120 = 119
N219    X###120
N220    END
N221    M2
```

Example 2 contains two levels of indirection (N219) and shows how the contents from multiple variables can be assigned to a command or expression.

At Block N215, variable #119 is set to constant 100.

At Block N217 one is added to the contents of variable #119.

At Block N218 variable #120 is set to constant 119.

Block N219 moves the X-axis to the position contained in variable #120 via two levels of indirection. The first level is the content of variable #119. The second level is the content of variable #101, which is incremented in the loop at Block N217 to introduce the contents of variables #102, #103 and #104.

The X-axis will move to X1; X2; X3; and X4.

Storing Result of Computation

When a mathematical expression is programmed, variables on the left side of an equation store the computed result.

N250 #110 = #20 + #35

N260 #120 = #18 / 2

At N250, #110 contains the sum of the contents of #20 and #35. At N260, #120 contains the result of the contents of #18 divided by 2.

Parentheses establish an order of operations or denote special functions.

NOTE:	Multiplication operations MUST be in parentheses or the CNC treats the multiply command (*) as a comment sign and disregards the rest of the line following the sign (*).

N300 #140 = (#11 * #115) / 2

N310 #141 = sin (45)

N320 #142 = (#141 * #140) ; * #142 is shortest side

All rights reserved. Subject to change without notice.
25-January-08

Variable Programming Examples

Example 1

This program uses common variables in the range of #50 to #149. The program mills a pocket with a three-degree draft angle on the sidewalls. The dimensions at the bottom of the pocket are:

15.5730 (X axis) x 13.8850 (Y axis). The pocket is 1.0000 in. deep.

The tool begins at the upper-left corner of the pocket and at full depth. Part Zero is set in the center of the pocket.

```
O 28 * 3-DEG. DRFT PKT
G90 G70 G0 G17
T0
Z0
X0 Y0
T1
X-7.7865 Y6.9425          * MOVE TO UP-LEFT CORNER
Z.5
G1 Z-1 F10                * FEED TO DEPTH
F40
#101 = 15.5730            * LENGTH (X) OF POCKET
#102 = 13.8850            * WIDTH OF POCKET
#103 = .0200              * DESIRED "STEP-UP" IN Z AXIS
#104 = (#103*TAN(3))      * CALCULATE "STEP-OVER" IN X-Y AXES
M98 P100                  * CALL SUBPROGRAM 100
T0
G90 G0 Z0
X0 Y0
M2

O100
LOOP((1/#103)+1);         * SET LOOP NUMBER (1 IN. DP / .02 STEP) + 1
G91                       * SET INCREMENTAL MODAL
G1 Y(-#102);              * MILL L.H. SIDE
X#101;                    * MILL BOTTOM SIDE
Y#102;                    * MILL R.H. SIDE
X(-#101);                 * MILL TOP SIDE, BACK TO START POS'N
X(-#104) Y#104 Z#103;     * STEP UP/OVER IN X-Y AND Z
#101 = #101 + (#104*2);   * ADD STEPOVER TIMES 2 TO LONG SIDES
#102 = #102 + (#104*2);   * ADD STEPOVER TIMES 2 TO SHORT SIDES
END                       * END LOOP
M99
```

All rights reserved. Subject to change without notice.

The pocket will be milled with a side draft angle of three degrees. Z is set to a step-up increment of .02 in. #152 can be set to a desired value, perhaps to determine the finish on the sidewalls of the pocket. In this example, the pocket will always have a depth of 1 in., and a draft angle of 3 degrees. The side lengths and Z step may be changed.

To make this program totally independent, the Z depth and draft angle can be set to variables, and the additional calculations must then be made.

Example 2

This program requires the length and width of a rectangle, the cut per side on the rectangle, and the number of passes around the rectangle. Variables #150 to #199 are read only. They can be set only in the main program.

```
N10     O 1000
N20     G0 G17 G70 G90 F80
N30     T0
N40     Z0
N50     X0 Y0 ;* START POSITION OF RECTANGLE
N60     #151 = 3 ;* SET READ ONLY VARIABLE, X LENGTH OF SIDE
N70     #152 = 3;* SET READ ONLY VARIABLE, Y LENGTH OF SIDE
N80     #153 = .25 ;* SET READ ONLY VARIABLE, CUT PER SIDE
N90     #154 = 5 ;* SET READ ONLY VARIABLE, NUMBER OF PASSES
N100    M98 P1 ;* CALL SUBPROGRAM BODY
N110    T0
N120    Z0
N130    X0 Y0
N140    M2

N160    O1
N170    G91 G1 X#151  ;* MOVE X AXIS LENGTH OF SIDE
N180    Y#152  ;* MOVE Y AXIS LENGTH OF SIDE
N190    X(-#151) ;* MOVE X NEGATIVE
N200    Y(-#152) ;* MOVE Y NEGATIVE
N210    #111 = 0  ;* SET SIDE CUT INCREMENT TO 0
N220    LOOP #154  ;* LOOP #154 NUMBER OF TIMES
N230    X#153 Y#153  ;* SET SIDE CUT
N240    #111 = #111 - #153  ;* DECREMENT SIDE CUT EACH LOOP
N250    #101 = #151 + (#111 * 2 ) ;* CALCULATE NEW X LENGTH
N260    #102 = #152 + (#111 * 2 ) ;* CALCULATE NEW Y LENGTH
N270    X#101  ;* MOVE AROUND SQUARE USING NEW SIDE * LENGTHS
N280    Y#102
N290    X(-#101)
N300    Y(-#102)
N310    END
N320    M99
```

The read only variables are set in Blocks N60 to N90. Then, the subprogram is called. At Block N170, the first move is made along the X-axis, followed by a move along the Y-axis. At Blocks N190 and N200, the

All rights reserved. Subject to change without notice.
25-January-08

logical negative sign makes the axis move in the opposite direction. The contents of the variables remain the same.

At Block N220, a loop, which ends at Block N310, is set up. The loop runs the number of times contained in variable #154. The first move in the loop is in the X and Y axes to the side of cut value in #153. In Block N240, #111 decrements at each pass through the loop, by the value of the side cut. This value, in turn, is used to calculate a new length of cut for each side.

User Macros (G65, G66, G67)

Use G66 when you want to use a modal macro subprogram. These groups of instructions can be special canned cycles made up by the user to simplify the programming of the particular part, or master programs for similar part families, programmed with variables rather than fixed dimensions.

Macros can contain automatic measuring sequences for sensors, such as a probe, for feedback to the CNC. Refer to **Table 17-4**.

Table 17-4, Macro G-Codes

Format	Function
G65 Pn Ln	Non-modal macro call. Call macro n. Execute macro, at the current position, only once. The subprogram can be looped (L).
G66 Pn	Modal macro call. Call macro n. Execute macro at any X and/or Y location given after the G66 code, until G67 (cancel) is called. G66 will stay active until G67 is called.
G67	Cancel Modal Macro Call (G66).

The following topics are described:

- Macro Body Structure
- Setting and Passing Parameters

All rights reserved. Subject to change without notice.

Macro Body Structure

The macro body is defined in the same way as a subprogram.

Format: Oxxx

O identifies it as a macro.

xxx is the label number.

Example:

```
N200 O 201
N210 ------
```

Terminate the macro with an M99 code.

Use local variables within the body of a macro or subprogram only. You cannot use them to transfer data to other macros or subprograms. If further subprogram calls are made from the macro body, you must transfer data from the local variables to common variables. The common variables can then be referenced to transport data to the further subprogram.

```
N220 #100 = #20
N230 -------
```

Common variables range from #100 to #220.

The macro must either be part of the program from which it is called or "included" using the file inclusion code. Refer to "File Inclusion" in this section.

Setting and Passing Parameters

The following topics are described:

- G65 Macro Programming, Main
- G65 Macro Programming, Macro (Subprogram)
- G66/G67 Macro Programming
- SLOTMAC.G Program
- Macro Programming (Hole Milling Macro)

You can set parameters for a macro before the subprogram call (M98 Pn). Refer to Example 1. Blocks 10 to 12 define variable values for the subprogram called in Block 13.

Example 1

```
N10    #151 = 2
N11    #152 = 3
N12    #153 = 3.4
N13    M98 P1
N14    -------
```


All rights reserved. Subject to change without notice.

It may be more convenient to use macro call G65 Pn or G66 Pn to pass variables to the subprogram by letter address. This is how a canned cycle operates. Refer to Example 2. Values are passed on for parameters A, B and C.

Example 2

```
N20    G65 P1 A2 B3 C3.4
N21    -------
```

Macro call G65 Pn contains a loop option (Ln). Where, n is the number of repetitions of the subprogram called.

```
N20  G65 P1 A2 B3 C3.4 L3
N21  -------
```

Macro 1 will be called three times (Ln equals 3).

When parameters are passed to a macro body by letter address, the contents of the parameters are stored in local variables. Refer to **Table 17-5**.

Table 17-5, Letter Addresses

A = #1	B = #2	C = #3	D = #7	E= #8	F = #9,
H = #11	I = #4	J = #5	K = #6	M = #13	Q = #17,
R = #18	S = #19	T = #20	U = #21	V = #22	W = #23,
X = #24	Y = #25	Z = #26			

Letter addresses G, L, N, O, and P cannot be used for parameter passing.

All rights reserved. Subject to change without notice.

G65 Macro Programming, Main

The following is an example of a simple macro program. In this example, the macro is a "window milling" cycle designed to mill a square or rectangular window through a part.

Example:

```
N1    O99  * WINDOW-MACRO-CALL
N2    G90 G70 G0 G17
N3    T0
N4    Z0
N5    X0 Y0
N6    T1  *** .5000 MILL
N7    G90 G0 X1 Y1
N8    Z.1
N9    F40
N10   G65 P3 X4 Y4 Z-.55
N11   G90 G0 Z.1
N12   T0
N13   Z0
N14   X0 Y0
N15   M30 O99
```

* parameters passed:

* X (#24) = length of window in X axis

* Y (#25) = width of window in Y axis

* Z (#26) = absolute tool depth

G65 Macro Programming, Macro (Subprogram)

This macro can mill any size window (L x W), at any Z depth. To change the pocket size, change the parameters on Block 10 (X,Y,Z). The CNC will execute the macro only once, at the current position. (G65 is not modal.)

Example:

```
N22   O3  * WINDOW-MACRO
N23   G90 G1 Z#26
N24   G91 G41 Y(#25/2)
N25   X(-(#24/2))
N26   Y(-#25)
N27   X#24
N28   Y#25
N29   X(-(#24/2))
N30   G40 Y(-(#25/2))
N31   M99
```


All rights reserved. Subject to change without notice.

G66/G67 Macro Programming

This example is a modal macro program to mill slots in a plate at various locations. In contrast to the G65 (single-call macro) in Example 1, G66 (modal macro call) applies the macro to all subsequent moves, until canceled by G67. Program G67 after the last slot location.

Example:

```
N1          O101  * SLOTCALL.G
N2          G90 G70 G0 G17
N3          T0 Z0
N4          X0 Y0
N5          T1 D.25 L-1 F30
N6          G66 P1255 X5 Y1 Z-.1 A5 B12 C5
N7          X1 Y2
N8          X2 Y4
N9          G67
N10         G90 G0 T0 Z0
N11         X0 Y0
N12         M2
N13
N14         ["SLOTMAC.G
```

This program calls SLOTMAC.G, a program in another file. The "file inclusion" block (N14) calls the program from another file in the Program Manager.

All rights reserved. Subject to change without notice.

SLOTMAC.G Program

In the following Blocks 1260 through 1400 are comment blocks that regard the macro's structure and concept.

Example:

```
N1255    O1255  * SLOTMAC.G
N1260
N1270    * EXAMPLE: G65 P1255 X-3 Y1 Z-.125 A5 B12 C10
N1280    * P = SUB #
N1290    * X = X DIM OF SLOT       #24
N1300    * Y = Y DIM OF SLOT       #25
N1310    * Z = ABS DEPTH OF SLOT   #26
N1320    * A = Z FEEDRATE          #1
N1330    * B = XY FEEDRATE         #2
N1340    * C = ANGLE FROM 3 o'clock #3
N1350
N1360    * NOTES:
N1370    * 1.  SLOT WILL HAVE FULL RAD.

N1380    * 2.  MUST POS'N XY OVER CENTER OF L.LEFT RAD.
N1390    * 3.  PROGRAM SLOT LENGTHWISE IN X, ANGLE C WILL ROTATE
N1400
N1410    G90 G0 Z.1
N1420    G61 Z#26 F#1
N1430    G68 C#3
N1440    G91 G41 G64 X.1 Y(#25/2) F#2
N1450    X-.1
N1460    G3 X0 Y(-(#25)) I0 J(-(#25/2))
N1470    G1 X(ABS((ABS(#24))-(ABS(#25))))
N1480    G3 X0 Y#25 I0 J(#25/2)
N1490    G1 X(-(ABS((ABS(#24))-(ABS(#25)))))
N1500    G1 G40 Y(-(#25/2))
N1510    G68
N1520    G90 G0 Z.1
N1530    M99
```


All rights reserved. Subject to change without notice.

ANILAM

Macro Programming (Hole Milling Macro)

Example 3 machines a CW or CCW hole. A move is made to the hole center and to the required Z depth before calling the macro. After the macro is completed, the Z-axis moves to the clearance plane. The macro contains tangential entry to and exit from the hole surface. It uses error checking and messages. When the macro is finished, machine parameters return to the their previous status.

String variables (Examples: EPSI, SAVEFRT) can be set and used in place of regular variables.

Symbol or Name Variables

Symbol or name variables can be used to make a macro program easier to understand. They can represent a value or a variable. They can be used only in subprograms.

Symbol or name variables must be defined before use, in the following format:

[TEXT   value or variable

Examples:

[PI 3.141592654   *PI will be read as the value given

[TFLAG #1041   *TFLAG will represent system variable #1040 (current tool comp)

NOTE:	Open bracket must start line. Do not use equal signs (=) in string variables.

You can use a variable to print values.

#35= PI ;print 35   *3.141592654 will be printed

There must be at least one space preceding and following the string variable in a program. In the following examples, PI is the variable.

#35/PI+#23   Produces error.

#35/ PI +#23   Correct format.

Once set, string variables can be used in any macro within the same program.

All rights reserved. Subject to change without notice.

Example:

```
G90 G70 G0 G17
 T0 Z0
 X0 Y0
 T1 F30
 X1.5 Y0  * MOVE TO HOLE CENTER
 Z.1
 G1 Z-.5  * MOVE Z TO DEPTH
 G65 P76 D2.0 S.010 J35 K20
 G0 Z.1   * RAISE Z TO CLEARANCE PLANE
 TO Z0
 X0 Y0
 M2

 O76  **  HOLE MILLING MACRO.
 *
 *  D#7 = HOLE DIAMETER (+=CCW,-=CW), J#5 = ROUGH FEEDRATE,
 *  S#19 = FINISH STOCK AMOUNT, K#6 = FINISH FEEDRATE.
 *  #1020 = TOOL DIAMETER.
 *
 [SAVEG90  #99   * SET STRING SAVEG90 TO VAR 99
 [SAVEG00  #98   * SET STRING SAVEG00 TO VAR 98
 [SAVEFRT  #97   * SET STRING SAVEFRT TO VAR 97
 [TDIA     #96   * SET STRING TDIA TO VAR 96
 [EPSI   .00001  * SET STRING EPSI TO .00001

 SAVEG90 = #1090   * SAVE CURRENT DIM MODE (ABS=90,INCR=91)
 SAVEG00 = #1016   * SAVE CURRENT MOVE MODE (RAPID=0,FEED=1)
 SAVEFRT = #1022   * SAVE CURRENT FEEDRATE
 TDIA = ABS(#1020) * SAVE CURRENT ABSOLUTE TOOL DIA
 IF(!VAR(7)) THEN
 PRINT (ERROR! HOLE DIA. NOT GIVEN)
 M30
 ENDIF
 IF(!VAR(5)) THEN; #5=#1022; ENDIF * DEFAULT ROUGH FEEDRATE.
 IF(!VAR(6)) THEN; #6=#5; ENDIF    * DEFAULT FINISH FEEDRATE.
 IF(!VAR(19)) THEN; #19=0.; ENDIF  * DEFAULT NO FINISH STOCK.
 IF(ABS(#7/2)<ABS(#19)) THEN
 PRINT (ERROR! TOOL DIA. TOO BIG)
 M30
 ENDIF
 #33 = (ABS(#7)/2-ABS(#19)- TDIA /2); * ROUGHING PASS RADIUS.
       IF(#33<0|#33=0) THEN
       PRINT (ERROR! ROUGH AMOUNT TOO BIG)
       M30
       ENDIF
       IF( #1041 > 40+ EPSI ) THEN    * CHECK IF TOOL COMP IS ON
       PRINT (ERROR! TOOL COMP NOT ALLOWED)
 M30
 ENDIF
 IF( TDIA < EPSI  ) THEN
```


All rights reserved. Subject to change without notice.

```
PRINT (WARNING: TOOL DIA.= 0)
M00 * DWELL UNTIL START KEY.
ENDIF
#34 = (#33/2); * INTERMEDIATE RADIUS.
#35 = (ABS(#7)/2- TDIA /2); * FINISH PASS RADIUS.
#36 = (#35/2); * INTERMEDIATE RADIUS.
G64; * CONTOURING MODE.
IF(#7>0) THEN * COUNTER-CLOCKWISE.
G91 F#5
G01 X#34 Y#34
G03 X(-#34) Y#34 I(-#34) J0
G03 X0 Y0 I0 J(-#33)
G03 X(-#34) Y(-#34) I0 J(-#34)
G01 X#34 Y(-#34)
IF((#19> EPSI ) & (#6> EPSI )) THEN * IF FINISH PASS.
G91 F#6
G01 X#36 Y#36
G03 X(-#36) Y#36 I(-#36) J0
G03 X0 Y0 I0 J(-#35)
G03 X(-#36) Y(-#36) I0 J(-#36)
G01 X#36 Y(-#36)
ENDIF
ELSE * CLOCKWISE.
G91 F#5
G01 X(-#34) Y#34
G02 X#34 Y#34 I#34 J0
G02 X0 Y0 I0 J(-#33)
G02 X#34 Y(-#34) I0 J(-#34)
G01 X(-#34) Y(-#34)
IF((#19> EPSI ) & (#6> EPSI )) THEN * IF FINISH PASS.
G91 F#6
G01 X(-#36) Y#36
G02 X#36 Y#36 I#36 J0
G02 X0 Y0 I0 J(-#35)
G02 X#36 Y(-#36) I0 J(-#36)
G01 X(-#36) Y(-#36)
ENDIF * FINISH PASS.
ENDIF * CLOCKWISE

IF( SAVEFRT > EPSI ) THEN; F( SAVEFRT ); ENDIF * RESTORE FEEDRATE.
G SAVEG90 ;  * RESTORE G90/91.
G SAVEG00 ;  * RESTORE G00/01.
M99
```

All rights reserved. Subject to change without notice.

Probe Move (G31)

G31 is to be issued with an associated axis move (i.e. G31 X10). When the G31 is executed, it moves at current feedrate selected for G1 until the touch probe selected is deflected. At this point, the move is stopped, and the position where the probe touched the part is read and passed to system variables (#1060 to #1063 for X to U).

G31 is aborted if any of the following events occur:

- The primitive is issued while the probe is still deflected (touching the part).
- The ready signal is not present.
- Hardware malfunction: Trigger signal engaged, but no position is latched.
- Start pulse is issued, but probe is not ready after 2 seconds. (Only cordless probes).
- Cordless probe still in "sleeping mode."
- Low battery signal becomes active (Only cordless probes).

M-code **M9387** is provided to select the probe **G31** will use and probe activation:

M9387X0	Selects the Tool touch probe (X13)
M9387X1	Selects the 3-D touch probe (X12) (default)
M9387Y0	Copies Tool touch probe state (deflected or not) into a system variable (#1066)
M9387Y1	Copies 3-D touch probe state (deflected or not) into a system variable (#1066)
M9387Z0	Turns off cordless probe
M9387Z1	Turns on cordless probe

Canned cycles are available for the most common probe functions. Refer to "Section 5, Probing Cycles," for details. Using the **G31** primitive, parametric programming and the M-code described above, it is possible to write additional cycles to perform custom probing functions.

All rights reserved. Subject to change without notice.

Conditional Statements

This subsection discusses the conditional statements IF, THEN, ELSE, GOTO and WHILE.

IF - THEN - ENDIF

```
N300 IF (expression) THEN
N310 -------------------
        ::
        ::
N360 ENDIF
N370 ---------
```

If the expression in N300 is true, the program continues at N310. If the expression is false, the program continues at N370.

In place of an expression, you can use a variable that while not zero will be treated as a true expression. (Zero equals false. Any other value equals true.)

IF - THEN - ELSE - ENDIF

```
N400 IF (expression) THEN
N410 -------------------
        ::
        ::
N440 ELSE
N450 -------------------
        ::
        ::
N470 ENDIF
N480 ---------
```

If the expression is true, the program continues at N410, then to N440, where a jump is made to N480.

If the expression is false, the CNC skips Blocks N410 to N440 and executes Blocks N450 to N470. In place of an expression, you can use a variable that while not zero will be treated as a true expression. (Zero equals false. Any other value equals true.)

IF - GOTO

```
N500 IF (expression) GOTO nnnn
N510 ---------
```

NOTE:	When you program IF-GOTO statements do not precede the block number with the character "N". For example, IF-GOTO 487 skips to block number N487.

If the expression is true, the program jumps to the block number specified (nnn). If the expression is false, the program continues at Block N510. In place of an expression a variable can be used which while not zero will be treated as a true expression. (Zero equals false. Any other value equals true.)

All rights reserved. Subject to change without notice.

WHILE - DO - END

```
N550 WHILE (expression) DO nnnn
N560 ------------------------
                             ::
                             ::
N590 END nnnn
N600 ---------
```

If the expression is true, the program repeats between N550 and N590 until the expression becomes false. Similarly, if the expression is false when Block N550 is executed, the CNC jumps to Block N600. The number after DO is a label (identifier only) and the same number must be used to identify the END of the loop.

In place of an expression, you can use a variable that while not zero will be treated as a true expression. (Zero equals false. Any other value equals true.)

DO - END

```
N620 DO nnnn
N630 ---------
          ::
          ::
N650 IF ( expression ) GOTO 1111
N660 -------------------
N670 END nnnn
```

DO - END sets the program into an infinite loop that can only be ended by programming a GOTO (1111) command to another block. DO and END must be paired with labels (nnnn). When executed the program will repeat Blocks N630 to N660 until the expression at N650 becomes true and program execution continues at block (1111).

All rights reserved. Subject to change without notice.

Unconditional LOOP Repeat

Conditional statements require that a test be strictly true or false in order for a particular course of action to be taken. Unconditional statements are acted on without a logical precondition.

LOOP - END

```
N680 LOOP nnnn
N685 ---------
        ::
        ::
N695 END
```

LOOP instructs the control to execute the following blocks (N685) until it reaches an END command. The sequence is repeated nnnn times. The number of loops can be a variable assignment (LOOP #121).

GOTO

```
\N698 GOTO nnnn
N699 ----------
```

GOTO is an instruction to continue program execution at the block specified (nnnn). You should not require this instruction in a user macro. It is intended for use in conjunction with the block skip symbol (\), as shown in the example. When block skip is ON, Block N698 is not executed. When block skip is OFF, Block N698 is executed and program execution jumps to the block specified.

NOTE: When you program GOTO statements do not precede the block number with the character "N".
For example, GOTO 610 skips to block number N610.

Short Form Addressing

The appropriate abbreviation instructs the CNC to activate the corresponding command. Refer to **Table 17-6**.

Table 17-6, Abbreviations

Command	Abbreviation
DO	]D
END	]E
GOTO	]G
IF	]I
LOOP	]L
PRINT	]P
THEN	]T
WHILE	]W

All rights reserved. Subject to change without notice.

Logical and Comparative Terms

The following topics are described:

- **Logical Terms**
- **Comparative Terms**

Logical Terms

All logical operations can be carried out using the following command characters or combinations of characters. Refer to **Table 17-7**.

Table 17-7, Logical Symbols

Statement	Symbol	True/False Table
OR	¦	0-0 = False 0-1 = True 1-0 = True 1-1 = True
EXCLUSIVE OR	^	0-0= False 0-1= True 1-0= True 1-1 = False
AND	**&**	0-0= False 0-1= False 1-0= False True

Comparative Terms

You can compare variables with variables and variables with constants using equality and inequality operators.

The following topics are described:

- **Equality Operators**
- Inequality Operators

Equality Operators

N700 IF (#120 = #125) THEN (or GOTO)
N710 --------------------
::
::
N740 IF (#130 = 360) THEN (or GOTO)
N750 --------------------

Block N700 compares the contents of variable #120 with the contents of variable #125. If the contents are equal, then the expression is true and THEN or GOTO directs the program. Otherwise, the expression is false. At Block N740, the contents of variable #130 are compared with the constant 360. The result of the comparison is identical to the first case.

All rights reserved. Subject to change without notice.

Inequality Operators

NOT

```
N760 WHILE (#135 != #137) DO 10
N770 -----------------------
                    ::
N790 END 10
```

The exclamation mark (!) symbolizes NOT. Therefore, Block N760 instructs the CNC to continue the loop to N790 while the contents of variables #135 and #137 are not equal (condition true). When the contents of the variables become equal the expression is false and the loop terminates.

GREATER THAN

```
N800 IF (#122 > #134) GOTO 830
N810 --------------------
```

The symbol (>) symbolizes GREATER THAN. Therefore, Block N800 instructs the control to go to (GOTO) or jump to Block N830 if the contents of variable #122 are greater than the contents of variable #134 (condition true). If the expression is false, execution continues to Block N810.

LESS THAN

```
N840 IF (#123 < #135) GOTO 880
N850 --------------------
```

The symbol (<) symbolizes LESS THAN. The function is the opposite of GREATER THAN and the expression is true when the contents of variable #123 are less than the contents of variable #135.

NOTE:	Greater than (>) and less than (<) expressions become false if the contents of the compared variables are equal.

File Inclusion

Example 1: ["FILENAME.G

File inclusion is a function that allows a subprogram that is not actually part of the file to be called from the main program, or from another subprogram in the file.

In this way, a tool change subprogram or a macro can be stored in the .G directory, and called from any other program that has the proper "file inclusion" code, which will allow the execution of the external subprogram.

Example 1 shows the syntax necessary to "include" a file into another file.

Format: open left bracket ([), then double quote character ("), then the filename and its extension. This line must appear somewhere in the program that is to call the "included" program.

All rights reserved. Subject to change without notice.

Example 2:

```
N1      O23 * TEST.G
N2      M98 P9
N3      T1 * 1.0000 MILL
N4          G0 X-.6 Y.6
N5      Z.1
N6      .
N7      .
.
.
.
N33     M98 P9
N34     T2
N35     * .368 DRILL
N36
.
.
.
N50     M98 P9
N51     M30 O23
N52     ["TOOLCHNG.G
```

In Example 2, a program named TOOLCHNG.G can be called from the main program (or from an existing subprogram). It is made possible by line N52. The file inclusion function is programmed on N52.

In this way, the same subprogram can be used in many programs, but you do not need to type it into each program. Each program must, contain the proper "file inclusion" block.

The program to be included must be in the form of a subprogram, beginning with Onnn, and ending with the M99 code.

The format for file inclusion is: ["FILENAME.G

It is especially useful for including tool change subprograms, zero-setting subprograms and macros.

All rights reserved. Subject to change without notice.

A

All rights reserved. Subject to change without notice

B

All rights reserved. Subject to change without notice.

All rights reserved. Subject to change without notice

D

All rights reserved. Subject to change without notice.

All rights reserved. Subject to change without notice

 All rights reserved. Subject to change without notice.

All rights reserved. Subject to change without notice

All rights reserved. Subject to change without notice.

All rights reserved. Subject to change without notice

All rights reserved. Subject to change without notice.

All rights reserved. Subject to change without notice

All rights reserved. Subject to change without notice.

All rights reserved. Subject to change without notice

All rights reserved. Subject to change without notice.

All rights reserved. Subject to change without notice.

All rights reserved. Subject to change without notice

All rights reserved. Subject to change without notice.

All rights reserved. Subject to change without notice

All rights reserved. Subject to change without notice.

Q

R

All rights reserved. Subject to change without notice

S

All rights reserved. Subject to change without notice.
25-January-08

All rights reserved. Subject to change without notice

All rights reserved. Subject to change without notice.

All rights reserved. Subject to change without notice

U

All rights reserved. Subject to change without notice.

All rights reserved. Subject to change without notice

U.S.A.
ANILAM
One Precision Way
Jamestown, NY 14701
☎ **(716) 661-1899**
FAX **(716) 661-1884**
🖂 **anilaminc@anilam.com**

ANILAM, CA
16312 Garfield Ave., Unit B
Paramount, CA 90723
☎ **(562) 408-3334**
FAX **(562) 634-5459**
🖂 **anilamla@anilam.com**

Dial "011" before each number when calling from the U.S.A.

Germany
ANILAM GmbH
Fraunhoferstrasse 1
D-83301 Traunreut
Germany
☎ **+49 8669 856110**
FAX **+49 8669 850930**
🖂 **info@anilam.de**

Italy
ANILAM Elettronica s.r.l.
10043 Orbassano
Strada Borgaretto 38
Torino, Italy
☎ **+39 011 900 2606**
FAX **+39 011 900 2466**
🖂 **info@anilam.it**

Taiwan
ANILAM, TW
No. 246 Chau-Fu Road
Taichung City 407
Taiwan, R.O.C.
☎ **+886-4 225 87222**
FAX **+886-4 225 87260**
🖂 **anilamtw@anilam.com**

United Kingdom
ACI (UK) Limited
16 Plover Close, Interchange Park
Newport Pagnell
Buckinghamshire, MK16 9PS
England
☎ **+44 (0) 1908 514 500**
FAX **+44 (0) 1908 610 111**
🖂 **sales@aciuk.co.uk**

China
Acu-Rite Companies Inc.(Shanghai Representative Office)
Room 1986, Tower B
City Center of Shanghai
No. 100 Zunyi Lu Road
Chang Ning District
200051 Shanghai P.R.C.
☎ **+86 21 62370398**
FAX **+86 21 62372320**
🖂 **china@anilam.com**

January 2008

Ve 02

627785-21 · 1/2008 · VPS · Printed in USA · Subject to change without notice