

ANILAM

4200T CNC Programming and Operations Manual

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. ANILAM, Inc. reserves the right to make changes to improve reliability, function or design without notice. ANILAM, Inc. assumes no liability arising out of the application or use of the product described herein. All rights reserved. Subject to change without notice.

Copyright 2005 ACU-RITE COMPANIES, Inc.

Section 1 - Programming Concepts

Introduction 1-1
 Programs 1-1
Axis Descriptions..... 1-2
 Z Axis..... 1-2
 C Axis 1-2
 X Axis 1-3
Part Zero Definition 1-3
Polar Coordinates 1-4
Absolute Positioning..... 1-5
Incremental Positioning 1-6
Diameter or Radius Programming..... 1-6
Developing Part Programs 1-7

Section 2 - CNC Console and Software Basics

CNC Console 2-1
Keypad 2-2
 Alphanumeric Keys 2-3
 Editing Keys 2-5
 Using the Keypad to Program Moves 2-5
 Soft Keys 2-6
 Accessing Keypad Commands Using a PC Keyboard (Optional)..... 2-6
Software Basics 2-6
 Pop-up Menus 2-7
 Screen Saver 2-7
 Highlight Bars 2-7
 Cursors 2-7
 Clearing Entries 2-7
 Operator Prompts 2-7
 Typeover and Insert Modes 2-8
 Deleting Text..... 2-8
 Messages/Error Messages 2-8

Section 3 - Manual Operation and Machine Setup

Manual Panel 3-1
 Manual Panel Keys 3-2
 Manual Panel LEDs 3-3
Manual Mode Screen 3-4
 Machine Status Display Area Labels 3-5
 Program Area Labels 3-6
Powering on the CNC 3-6
Shutting Down the CNC 3-7
Setting/Resetting the Emergency Stop (E-STOP)..... 3-7
Activating/Resetting the Servos 3-7

Manual Mode Settings	3-7
Activating Manual Mode Rapid or Feed	3-8
Adjusting the Feedrate	3-8
Adjusting Rapid Move Speed	3-9
Setting Absolute Zero	3-10
Part Zero	3-10
Jog Moves	3-11
Changing the Jog Mode	3-11
Selecting an Axis	3-11
Jogging the Machine (in Increments)	3-12
Jogging the Machine (Continuous Moves)	3-12
Manual Data Input (MDI) Mode	3-12
Using Manual Data Input Mode	3-13
Handwheel Operation	3-13
Operating a Handwheel Mounted on the Manual Panel	3-13
Operating Handwheels Mounted on the Machine	3-14

Section 4 - Preparatory Functions: G-Codes

Preparatory Functions: G-Codes	4-1
Rapid Traverse (G0)	4-3
Linear Interpolation (G1)	4-4
Angular Motion Programming Example	4-5
Circular Interpolation (G2 and G3)	4-6
Partial Arcs	4-8
Dwell (G4)	4-8
Ellipse (G5)	4-9
Non-modal Exact Stop Check (G9)	4-10
Setting Software Limits (G22)	4-11
Maximum Spindle Speed (G24)	4-12
Probe Move (G31)	4-13
Thread Cutting (G33)	4-13
Offsets (Work Coordinate System Select) (G53)	4-14
Modal Corner Rounding/Chamfering (G59)	4-16
Cancel Modal Corner Rounding (G60)	4-18
In-Position Mode (Exact Stop Check) (G61)	4-18
Contouring Mode (Cutting Mode) (G64)	4-19
Using Macros (G65, G66, G67)	4-20
Inch Mode (G70) or MM Mode (G71)	4-22
Absolute Mode (G90) or Incremental Mode (G91)	4-22
Presetting the Axes (G92)	4-24
Cutting Feedrates in Feed per Minute (G94)	4-25
Cutting Feedrates in Feed Per Revolution (G95)	4-25
Setting Spindle Speed to Constant Surface Speed (G96)	4-26
Setting Spindle Speed to Constant Revolutions Per Minute Speed (G97)	4-26

Section 5 - Canned Cycles

Programming a Z-Axis Turning/Boring Cycle with a Defined Profile (G73)	5-1
Programming an X-Axis Facing/Boring Cycle with Defined Profile (G74)	5-4
Programming a Z-Axis Rough Turning or Boring Cycle (G76)	5-7
Programming an X-Axis Radial (Facing) Roughing Cycle (G77)	5-9

Programming a Pecking Cycle (G78).....	5-11
Programming a Chip Break Cycle (G79).....	5-12
Programming a Longitudinal Grooving Cycle (G81).....	5-13
Programming a Radial Grooving Cycle (G82).....	5-15
Types of Threading	5-17
Cutting Longitudinal Threads with Canned Cycles (G83, Uni-directional)	5-18
Cutting Longitudinal Threads with Canned Cycles (G84, Bi-directional).....	5-20
Cutting Radial Threads with Canned Cycles (G85, Uni-directional).....	5-23
Cutting Radial Threads with Canned Cycles (G86, Bi-directional).....	5-26
Tapping a Hole with a Canned Cycle (G87).....	5-28
Programming a Boring Cycle (G88)	5-30
Lathe Tool Probe Cycles (Option)	5-32
Tool Probe G-code Cycle Designations	5-33
Macro Variable and System Variable Settings.....	5-33
Probe Orientation.....	5-35
Lathe Tool Probe G-code Cycle Designations	5-35
Description of Tool Probe Cycles.....	5-37

Section 6 - Program Editor

Activating the Program Editor.....	6-1
Activating the Edit Mode from the Manual Screen	6-1
Activating the Edit Mode from the Program Directory	6-1
Activating the Edit Mode from Draw Graphics	6-1
Edit Soft Keys.....	6-3
Editing Shift Soft Keys	6-4
Deleting a Character	6-4
Inserting Text to Overwrite Previous Text	6-4
Inserting Text without Overwriting Previous Text.....	6-4
Deleting a Program Block.....	6-5
Page Up/Down through the Program Listing	6-5
Going to a Line of the Program Listing	6-6
Advancing to the Beginning or End of a Block	6-6
Advancing to the First or Last Block of a Program	6-6
Searching the Program Listing for Selected Text.....	6-7
Inserting a Blank Line in the Program Listing	6-7
Restoring a Block.....	6-8
Replacing Typed Text with New Text.....	6-8
Abbreviating Statements	6-9
Marking (Highlighting) Programming Blocks	6-10
Unmarking Program Blocks	6-10
Canceling Edits to a Program Block.....	6-10
Scrolling Through the Program Listing	6-10

Copying Program Blocks.....	6-11
Printing a Portion of a Program.....	6-12
Pasting Copied Blocks into a Program Listing.....	6-12
Numbering (or Re-numbering) Program Blocks.....	6-13
Writing Blocks to Another Program.....	6-13
Reading a Program into Another Program.....	6-14
Recording Keystrokes.....	6-14
Playing Recorded Keystrokes.....	6-15
Repeating a Command or Key.....	6-15
Printing an Entire Program Listing.....	6-15
Accessing the Most Recently Used Programs.....	6-16
Opening Another Program from the Program Listing.....	6-16
Saving Edits.....	6-16
Canceling Unsaved Edits.....	6-17
Including Comments in a Program Listing.....	6-17

Section 7 - Edit Help

Edit Help Menu.....	7-4
Edit Help Soft Keys.....	7-8
Help Template Menu.....	7-9
Help Graphic Screens.....	7-11
Using Help Graphic Screens to Enter Program Blocks.....	7-13
Available Help Graphic Screens.....	7-14
Programming Modal G-Codes in Edit Help.....	7-37
Programming LINE Moves in Edit Help.....	7-38
Angle Direction for Front and Rear Tool Posts.....	7-39
Using Angles to Program Moves.....	7-41
Programming MULTIPLE Move Commands in Edit Help.....	7-42

Section 8 - Draw Mode

Introduction.....	8-1
Starting Draw.....	8-2
Draw Screen Description.....	8-2
Putting Draw in Hold.....	8-3
Restarting Draw When in Hold.....	8-3
Canceling Draw.....	8-3
Draw Parameters.....	8-4
Tool On or Off.....	8-4
Drawing Compensated Moves.....	8-5
Showing Rapid Moves.....	8-5
Setting Grid Line Type.....	8-5
Setting Grid Size.....	8-6
Putting Draw in Motion, S.Step, or Auto Mode.....	8-6

Automatic Draw Restart	8-7
Erasing the Draw Display	8-7
Running Draw for Selected Blocks	8-7
Starting Draw at a Specific Block	8-8
Ending Draw at a Specific Block	8-8
Adjusting Draw Display	8-8
Fitting the Display to the Viewing Window	8-9
Halving Display Size	8-9
Doubling Display Size	8-9
Scaling the Display by a Factor	8-9
Using the Window Zoom	8-10
Erasing Display	8-10
Changing the Viewing Area without Changing the Scale	8-11

Section 9 - Tool Page and Tool Management

Activating the Tool Page	9-1
Using the Tool Page	9-1
Entering Values in the Tool Page	9-3
Adjusting a Single Value	9-3
Finding Tools by Number	9-3
Clearing Tool Information (Delete Row)	9-4
Clearing a Value from a Column	9-4
Clearing the Whole Tool Page	9-4
Tool-Length Offsets	9-4
Setting Tool-Length Offsets	9-5
Entering Tool Page TLO's Using the Calibration Function	9-6
Entering Tool Page TLO's Manually	9-6
Tool Offset Modification	9-7
Activating a Tool Number in the Manual Mode	9-8
Deactivating TLO's in the Manual Mode	9-8
Tool Nose Radius (TNR) Compensation	9-8
Selecting a Tool Location Code	9-11
Tool Nose Radius Programming Examples	9-12
Left of Path TNR Program (G41)	9-13
Right of Path TNR Program (G42)	9-15
Canceling Tool Nose Radius Compensation (G40)	9-17
Choosing the Correct TNR Compensation	9-18
Changing TNR Compensation in a Program	9-18
Motion of Tool during TNR Compensation	9-20
Compensation around Acute Angles	9-21
Changing the Numbers of Look Ahead Blocks	9-22
Wear Offsets	9-22
Setting Wear Offset Adjustment While Running a Program	9-23

Section 10 - Program Management

Changing the Program Directory's Display	10-1
Creating a New Part Program	10-2
Choosing Program Names	10-2

Selecting a Program for Running	10-2
Selecting a Program for Editing and Utilities	10-3
Listing a Program	10-3
Deleting a Program	10-3
Logging to Other Drives	10-4
Marking and Unmarking Programs.....	10-5
Marking Programs	10-5
Unmarking Marked Programs	10-5
Marking All Programs	10-5
Unmarking All Marked Programs	10-5
Deleting Groups of Programs.....	10-6
Restoring Programs	10-6
Copying Programs to Floppy Disks	10-6
Renaming Programs	10-7
Printing Programs	10-7
Formatting Floppy Disks.....	10-8
Checking Disks for Lost Program Fragments.....	10-8
Displaying System Information.....	10-9
Copying Programs from/to Unspecified Locations	10-10
Renaming Programs from/to Unspecified Locations	10-10
Printing Programs from Unspecified Locations	10-11
Creating Subdirectories.....	10-11
Deleting an Unspecified Program.....	10-12
Listing an Unspecified Program	10-12
Editing an Unspecified Program.....	10-12
Optimizing Your Hard Disk	10-13
Accessing the Disk Optimizer	10-13
Maximizing Program Storage Space.....	10-14

Section 11 - Running Programs

Running a Program One Step at a Time	11-1
Toggling Between Motion and Single-Step Mode	11-2
Holding or Canceling a Single-Step Run	11-2
Single-Step Execution of Selected Program Blocks.....	11-3
Using Arrow Keys to Select Starting Block	11-3
Using Search to Select Starting Block	11-3
Switching from Single-Step to Auto	11-3
Position Display.....	11-4
Automatic Program Execution.....	11-4
Holding or Canceling an Auto Run.....	11-5

Automatic Execution Starting at a Specific Block	11-5
Using Arrow Keys to Select Starting Block	11-5
Using SEARCH to Select Starting Block.....	11-5
Clearing a Halted Program.....	11-5
Using Draw while Running Programs.....	11-6
Setting the CNC to Display an Enlarged Position Display	11-6
Teach Mode	11-7
Initiating Teach-In	11-7
Soft Keys of Teach-In	11-7
Inputting Data into a Program	11-7
Canceling Teach Mode.....	11-8
About Teach Mode	11-8
Parts Counter and Program Timer	11-9
Background Mode	11-10

Section 12 - S Function, M Functions, and C-Axis Programming

Introduction	12-1
Spindle Speed Control (S-Function).....	12-1
Miscellaneous Functions (M-Codes).....	12-2
Control M-Codes	12-2
Order of Execution.....	12-3
C-Axis Programming	12-3
Enabling C-Axis Programming Mode.....	12-3
Disabling C-Axis Programming Mode	12-3
C-Axis Jog	12-4
Programming C-Axis Feedrate	12-4
Synchronizing/Unsynchronizing C-Axis	12-4

Section 13 - Simplified Command Interface (SCI) Mode

Activating the SCI Mode.....	13-1
SCI Mode Soft Keys.....	13-3
Activating the SHIFT SCI Soft Key Menu	13-3
Deactivating the SCI Mode.....	13-4
Canceling an SCI Command.....	13-4
Canceling a Running SCI Command	13-4
Clearing Messages	13-4
Clearing an Entry Field.....	13-4
Returning to the Software Selection Screen.....	13-4
Exiting SCI Mode	13-5
Commanding Moves in SCI Mode.....	13-5
Performing Rapid Moves (G00)	13-5
Activating and Changing a Tool.....	13-5
Performing Line Moves (G01).....	13-6

Performing Arc Moves (G02 or G03)	13-6
Setting Software Limits (G22)	13-7
Homing the Machine (G28).....	13-9
Making a Rough Turning Cut (G76).....	13-9
Making a Rough Facing Cut (G77)	13-10
Cutting Threads (G83).....	13-11
Activating Absolute (G90) or Incremental (G91) Mode	13-12
Setting the Feedrate to Feed per Minute (G94)	13-12
Setting the Feedrate to Feed per Revolution (G95)	13-13
Setting Spindle Speed to Constant Surface Speed (G96)	13-13
Setting the Spindle Speed to Revolutions per Minute (G97).....	13-14
Slave Mode	13-14
Cutting a Taper	13-15
Cutting a Chamfer.....	13-16
Cutting a Radius	13-17

Section 14 - Shape Editor

Activating the Shape Editor	14-2
Shape Terminology	14-3
Using the Pointer.....	14-3
Selecting Editing Tools.....	14-4
Line Tools.....	14-6
Arc Tools	14-8
Arc Tools	14-9
Chamfering Corners	14-9
Example #1, Creating a Basic Shape.....	14-10
Reversing an Arc's Direction	14-13
Deleting a Shape.....	14-13
Projecting Line Segments (Restoring Sharp Corners)	14-14
Joining Line Segments	14-14
Importing Shapes from Other Programs.....	14-14
Deleting a Segment.....	14-15
Changing the Shape Editor View	14-15
Viewing a List of Shape Segment Details	14-15
Deactivating and Activating Shapes.....	14-16
Using Construction Geometry	14-16
Accessing Geometry Tools	14-16
Point Tools	14-17
Line Tools.....	14-18
Circle Tools	14-19
Example #2, Creating Construction Geometry and Chaining it Into a Shape	14-20
Creating Construction Geometry	14-20

Chaining Geometry Elements to Create a Shape	14-26
Viewing a Listing of Geometry Elements.....	14-28
Deleting Geometry Elements	14-28
Deleting All Geometry Elements	14-29
Activate/Deactivating Geometry Elements	14-29
Shape Editor Hot Keys	14-29
Exiting the Shape Editor.....	14-30
Executing Shapes in a Program.....	14-30
Managing Shape Files	14-31
Naming Shape Files	14-31
Viewing Shape Files in the Program Directory.....	14-31
Deleting Shape Files.....	14-32
Backing Up Shapes.....	14-32
Shape Editor Settings	14-32
Verifying Shapes	14-33

Section 15 - Communication

Introduction	15-1
Installing the RS-232 Cable.....	15-1
Accessing the Communication Utilities	15-2
Setting Communication Parameters.....	15-3
Communication Port	15-3
Baud	15-3
Parity	15-4
Data Bits	15-4
Stop Bits	15-4
Settings.....	15-4
Data Type	15-4
Testing the Data Link	15-5
Activating the Test Link Screen.....	15-5
Setting Test Link Display Modes.....	15-5
Testing the Link	15-6
Clearing the Receive Area	15-6
Clearing the Transmit Area	15-6
Sending a Program	15-6
Receiving a Program.....	15-6
Setting the Transmission and Receiving Display	15-7
Holding Transmission/Receiving Operations	15-7
Using Data Control (DC) Codes	15-8
Using DC Codes in Receive Mode	15-8
Using DC Codes in Send Mode	15-8

Section 16 - Advanced Programming Features

Block Separator.....	16-1
Modifiers.....	16-1
Expressions and Functions	16-2
System Variables	16-3
User Variables.....	16-4
Local Variables	16-4
Common (Global) Variables.....	16-4
Read Only Variables	16-5
Block Skip Variables	16-5
Setting Selective Block Skips.....	16-6
Parameters and Variable Registers	16-6
Contents of Variables	16-7
Setting and Transferring Variables.....	16-7
Setting Variables and Direct Transfer.....	16-7
Indirect Transfer.....	16-8
Storing the Results of Computation.....	16-9
Parametric Programs	16-9
Applications of Parametric Programming	16-10
Families of Parts	16-10
Complex Geometric Shapes.....	16-10
Parametric Programming Versus Subprogramming	16-10
Call Statements	16-10
User Macro (G65, G66, G67).....	16-11
Programming Macros	16-11
Setting and Passing Parameters	16-12
Probe Move (G31).....	16-13
Conditional Statements	16-14
Unconditional: The LOOP repeat	16-15
Abbreviating Statements.....	16-16
Logical Terms	16-17
Comparative Terms	16-18
File Inclusion	16-19
Inclusion File Requirements.....	16-19
Inclusion File Formats.....	16-19
Sample Parametric Program	16-20

Section 17 - Machine Software and Peripherals Installation

Passwords.....	17-1
Machine Software Installation	17-1
Printer Installation	17-1
Keyboard Installation.....	17-1

Keypad Equivalent Keyboard Keys 17-2

Section 18 - Off-line Software

Passwords..... 18-1

Exiting the Software 18-1

Windows Off-line Software Installation 18-3

 Running Off-line Software from Windows 18-3

System Settings 18-3

 Maximum Memory Allocated..... 18-3

 Disabled Features..... 18-4

Section 19 - Sample Programs

Introduction 19-1

Sample Program #1 19-2

Sample Program #2 19-4

Sample Program #3 19-6

Sample Program #4 19-8

Sample Program #5 19-10

Sample Program #6 19-12

Sample Program #7 19-14

Sample Program #8 19-16

Sample Program #9 19-18

Index Index-1

Section 1 - Programming Concepts

Introduction

This section contains programming concepts for the beginning programmer. It contains terminology and concepts that should be mastered before writing programs.

Programs

A program is a precise sequence of machine instructions. Each program consists of blocks of information that instruct the movements and sequence of events to machine a workpiece. These blocks include all necessary events, including activating the spindle, coolant, tool selection, and all axis movement required to cut the workpiece in a safe and productive manner.

Every program is given a unique name for identification. Programs are stored in the CNC's memory and accessed from the CNC's Program Directory. You can create, delete, undelete, copy, and rename programs in the CNC's Program Directory.

Axis Descriptions

Z Axis

Refer to **Figure 1-1**. Motion along the Z-axis is linear. Positive Z motion is toward the tailstock and negative Z motion is toward the headstock.

C Axis

Refer to **Figure 1-1**. The C-axis is reserved for rotary spindle motion, which can be clockwise or counterclockwise. The builder determines C axis capability.

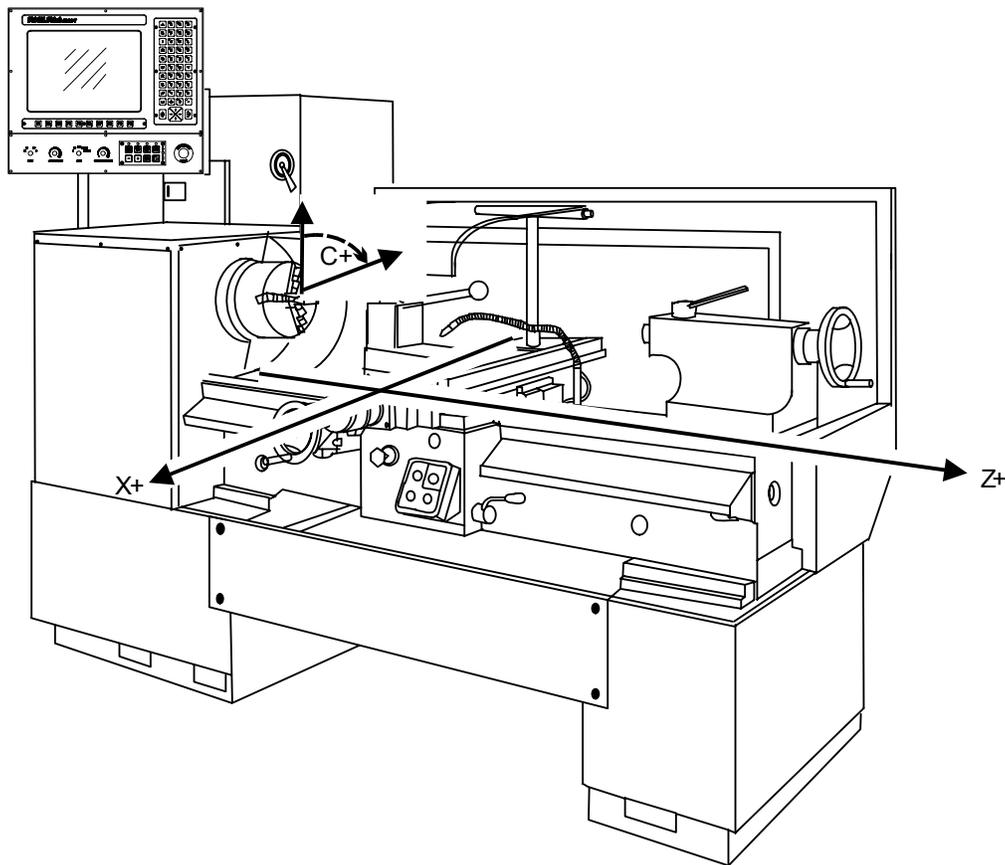


Figure 1-1, Axes, Directions of Motion

X Axis

Refer to **Figure 1-2**. In relation to the tool, positive X motion is away from the centerline of the part. Negative X motion is toward the centerline of the part. If the tool passes the centerline, the value of X will be negative. Positive X direction varies, depending on whether a Front or Rear Tool Post is used.

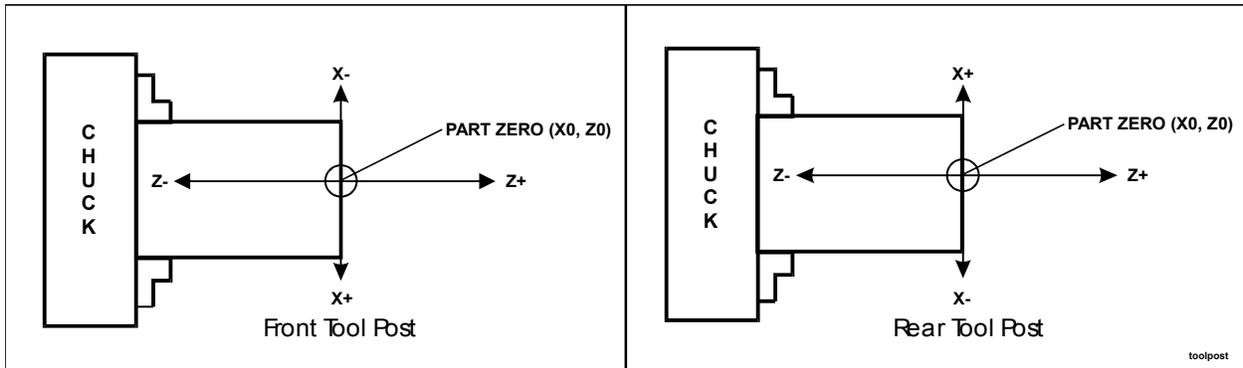


Figure 1-2, X-Axis Front/Rear Tool Post Directions of Motion

Part Zero Definition

NOTE: In general, all drawings in this manual assume a front tool post configuration. The builder determines the tool post configuration.

Refer to **Figure 1-3**. The intersection of the zero X and zero Z point is the reference point that defines part zero. In X, this is usually at the part centerline; in Z, it is at the front face of the part.

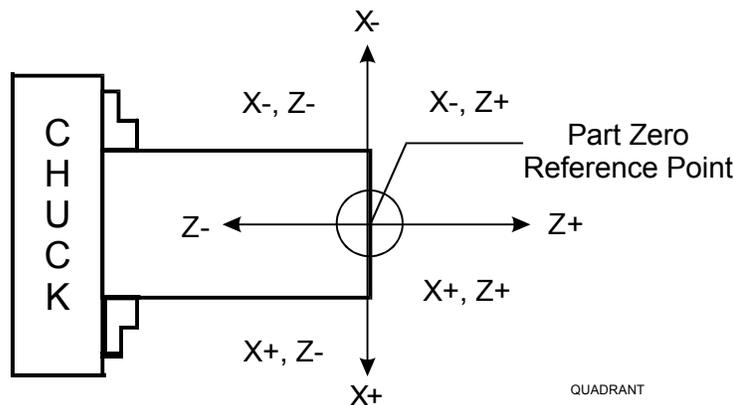


Figure 1-3, Part Zero

The reference point for moves in Absolute Mode is Part Zero; in Incremental Mode, it is the current position.

- **Example (using front tooling and Incremental Mode):** In X, to travel 2 inches toward the part centerline (negative direction), program X-4. (In Incremental Mode, an X-4 command moves the tool 2 inches per side.) In Z, to travel 6 inches towards the chuck (negative direction), program Z-6.

Polar Coordinates

Refer to **Figure 1-4**. If it is inconvenient to use X and Z coordinates, you can use the Polar Coordinate System. Polar Coordinates can only define points that lie on a single plane. Polar coordinates use the distance from the origin and an angle to locate points.

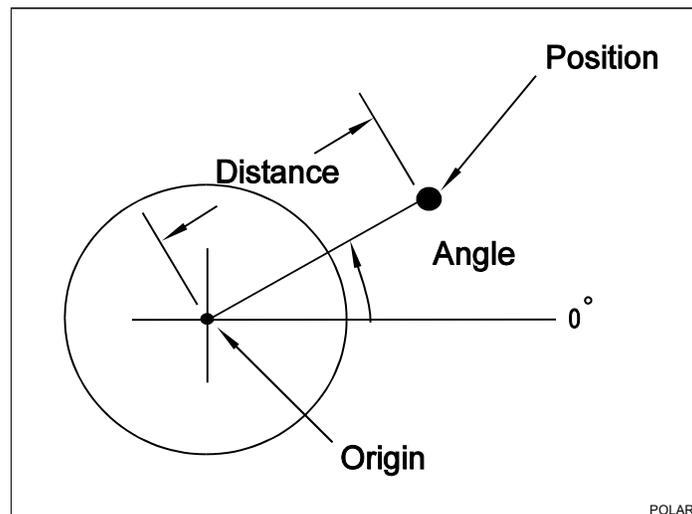


Figure 1-4, Polar Coordinate System

Absolute Positioning

Refer to **Figure 1-5**. When the CNC is in the Absolute Mode, all positions are measured from Absolute Zero. The Absolute Zero Reference, also called Part Zero, is not a fixed position on the machine; you select the point.

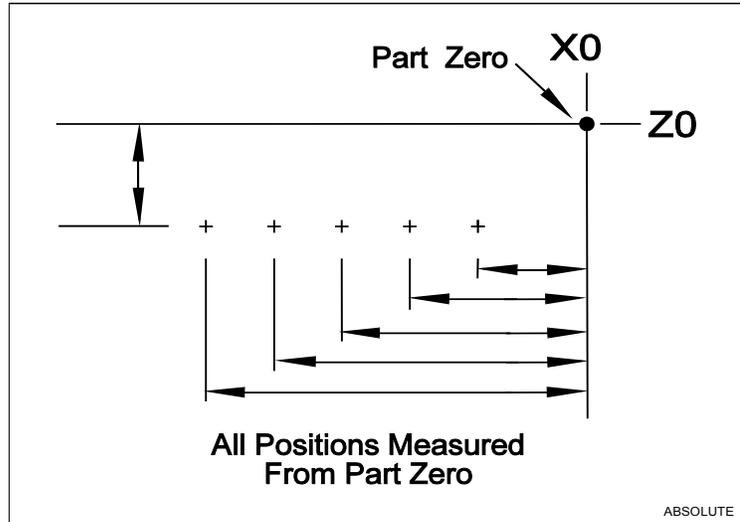


Figure 1-5, Absolute Positioning

You can set the Absolute Zero (X0, Z0) anywhere. Absolute Zero Reference is frequently at a position that makes it easy to use blueprint dimensions.

NOTE: In Absolute Mode, a move to X0, Z0 will move the machine to Absolute Zero.

Incremental Positioning

Refer to **Figure 1-6**. Incremental positions are measured from the machine's current position. This is convenient for performing an operation at regularly spaced intervals.

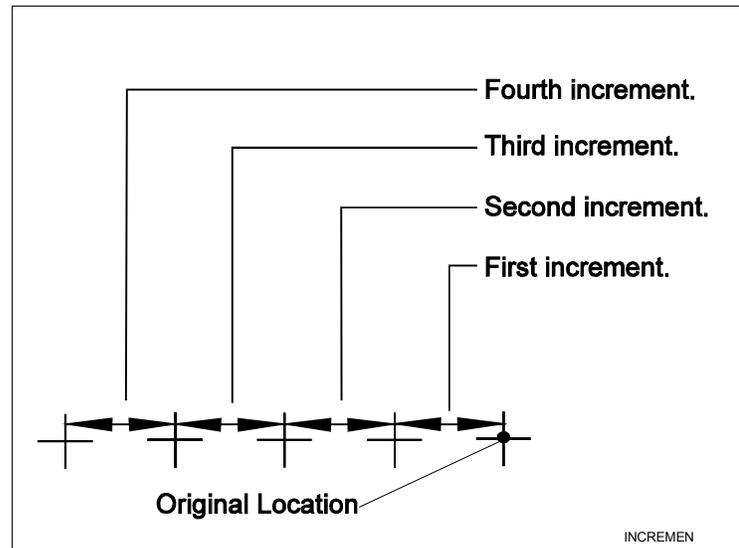


Figure 1-6, Incremental Positioning

NOTE: If an Incremental command to X0, Z0 is programmed, the CNC remains at the current position.

Diameter or Radius Programming

NOTE: All program examples in this manual use Diameter Programming.

Refer to [Figure 1-7, Diameter and Radius Programming](#). X Dimensions can be given in Diameter or Radius Mode:

Diameter X absolute dimensions are given as diameter values. X incremental dimensions are given as the diametric distance from the current diameter. X arc centers (I) are always given as actual distances.

Radius X absolute dimensions are given as radius values. X incremental dimensions are given as distance from the current position.

Since X0 is usually the centerline of the workpiece, diameter programming can simplify the programming of the workpiece.

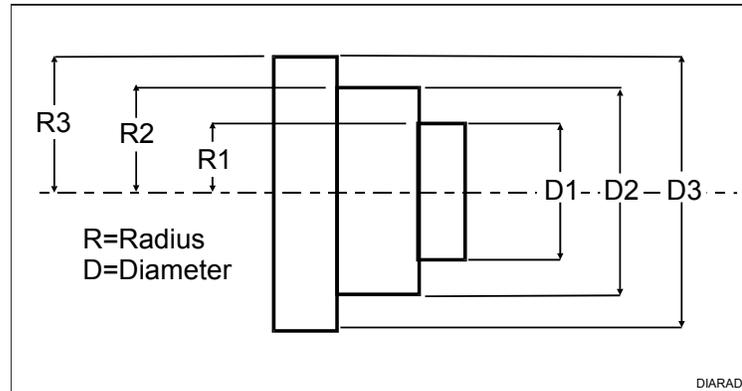


Figure 1-7, Diameter and Radius Programming

The default, set by the machine builder, is usually **Diameter**.

NOTE: In canned cycles (with the exception of area clearance), all X values are given as radial, regardless of whether diameter or radius programming is used.

Developing Part Programs

The most important part of writing a CNC program is the planning. The following guidelines are provided to help a first-time programmer organize the process.

First, decide how to hold the workpiece in the machine and where to set Part Zero. Part Zero is best located on the front face of the workpiece along the Z-axis and at the centerline of the workpiece in the X-axis. Try to select a Part Zero that facilitates the direct application of part print dimensions.

Develop a program using a procedure similar to the one that follows:

1. To enter the Program Directory from the Manual screen, press **Program (F2)**.
2. Create a program name for the part by pressing **Create (F2)**.
3. Enter the program name in the entry field for **New Program**.
Refer to "[Section 10 - Program Management](#)" for additional in-depth information.
4. Enter the Program Editor (press **Edit (F8)**) to open the new program and start writing blocks as described in steps 6 – 9 below.
Refer to "[Section 6 - Program Editor](#)" for additional in-depth information.
5. The first block of any program is usually a safe start position and tool change position (a position away from the workpiece to which the axes can return for safe tool-changing). The first block is also normally used to state the units of measurement (Inch/MM), the mode of operation (Absolute), the move type (Rapid), and to cancel all auxiliary functions (Tool Offsets, Spindle, Coolant).

Typical first block: G70 G90 G0 X0 Z0 T0 M5

6. Subsequent blocks in the program set spindle information, call tool number, turn on coolant, etc., and make the initial move toward the workpiece.
7. The remaining blocks in the program describe the moves, canned cycles, and tool changes required to complete the machining of the workpiece.
8. 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 is an M2 code that ends the program.

Typical final blocks: M5
 G0 T0 X0 Y0 Z0
 M2

9. Once you write a program, use Draw Graphics Mode to verify it. Make sure that correct machining procedures are being followed. Verify that all programmed moves are safe and accurate to the part print dimensions. Refer to "[Section 8 - Draw Mode](#)" for additional in-depth information.
10. At this point, you can load the stock material into the selected work-holding device.
11. Set the Tool Offsets for each tool in the Tool Page. Refer to "[Section 9 - Tool Page and Tool Management](#)" for additional in-depth information.
12. Prior to running the part in the Auto Mode, run it in Single Step to verify that both the program and the setting of tool offsets have been completed correctly. The Single Step Mode enables you to execute the program block by block.
13. After testing, the program is ready for production. Save the program on a floppy disk for future use. Refer to "[Section 10 - Program Management](#)" for additional in-depth information.

Section 2 - CNC Console and Software Basics

CNC Console

Refer to **Figure 2-1**. The CNC console consists of a 14-inch color screen and two keypads: one to the right of the screen and the other one underneath. Optionally, a 12.1-inch color flat panel console with a liquid crystal display (LCD) is available.

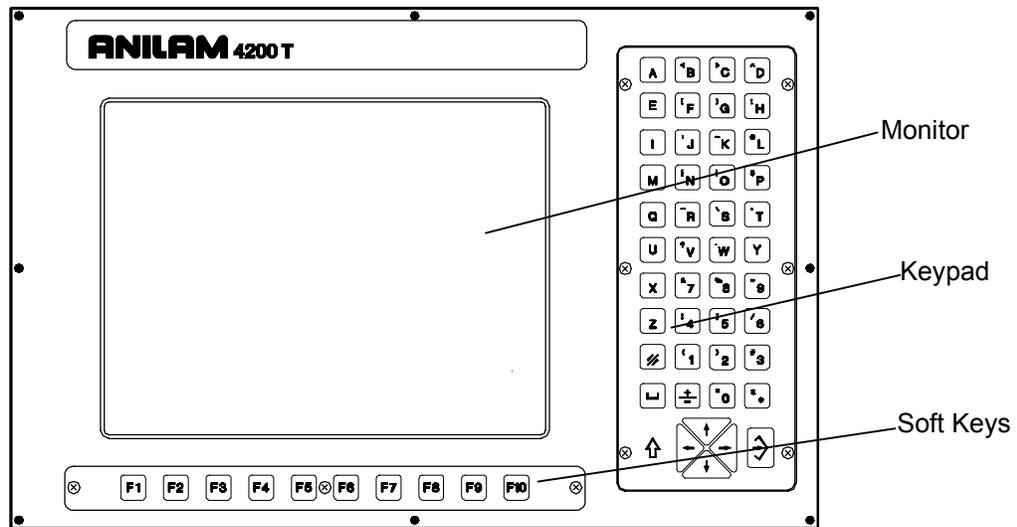


Figure 2-1, CNC Console

Keypad

Refer to **Figure 2-2**. The keypad to the right of the monitor is laid out in two groups.

- [Alphanumeric Keys](#)
- [Editing Keys](#)

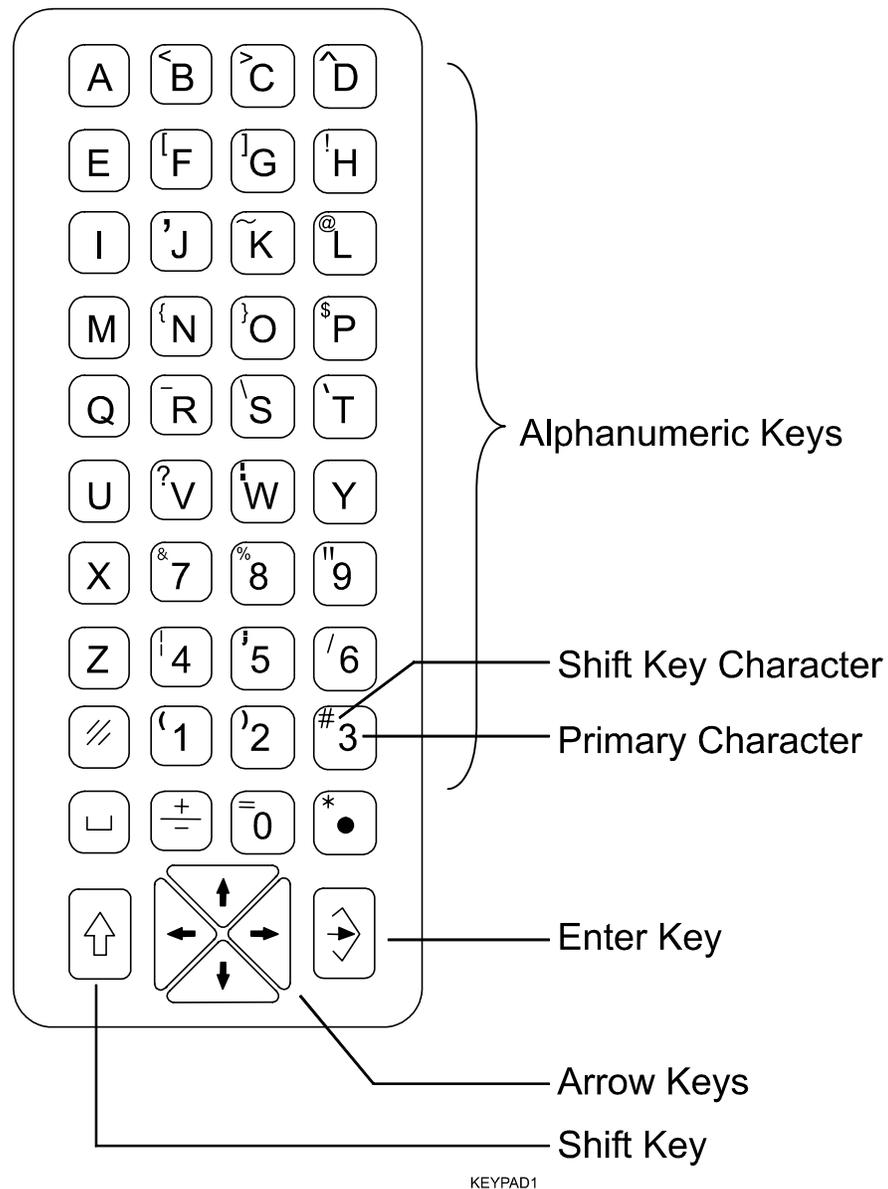


Figure 2-2, Keypad Keys

Alphanumeric Keys

Refer to [Figure 2-2. Keypad Keys](#). Alphanumeric Keys enable you to key-in position coordinates (X and Z) and program G, M, S and T codes. Some keys contain two symbols or characters: a large character in the middle of the key and a smaller one in the upper left corner. The large characters are the primary characters. The small characters are the **SHIFT** characters.

To type a primary character, press the key that displays that character. To type a **SHIFT** character, do the following:

1. Press **SHIFT**.
2. Press the key that displays the character in the upper left corner.

Refer to **Table 2-1**. Primary functions and **SHIFT** Key functions are listed.

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/Feed rate	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
{N	Letter N	Left Curly Bracket
}O	Letter O/Program Number Designator	Right Curly Bracket
\$P	Letter P	Dollar Sign
Q	Letter Q	None

Key Face	Primary Function	Shift Function
	Letter R	Underscore
	Letter S/Spindle Speed Designator	Backslash
	Letter T/Tool words	Single Quote
	Letter U	None
	Letter V	Question Mark
	Letter W	Colon
	Letter X/X Axis Coordinate	None
	Letter Y	None
	Letter Z/Z Axis Coordinate	None
	Number one	Left Parenthesis
	Number two	Right Parenthesis
	Number three	Pound or Number Sign
	Number four	Vertical Bar; Used To Separate Parts Of A Blueprint Programming Block For Angles/Chamfers/Radii.
	Number five	Semi-Colon
	Number six	Slash
	Number seven	Ampersand
	Number eight	Percent Symbol
	Number nine	Inch Symbol
	Number 0	Equal Sign
	Minus sign/dash	Plus Sign
	Period/Decimal sign	Asterisk: Used To "Comment Out" All Or Part Of A Block (Any Characters To The Right Of The Asterisk Are Ignored). The CNC Ignores These Blocks.

Editing Keys

Refer to **Table 2-2**. Use the editing keys to edit programs and move around the screen.

Table 2-2, Editing Keys

Label or Name	Key Face	Purpose
CLEAR		Clears messages, values, commands and program blocks that you select.
ARROW		Allows you to move highlight bars and cursor around the screen.
ENTER		Selects blocks for editing; activates menu selections; activates number entry.
SHIFT		Selects the SHIFT characters on the keypad. Remains ON until the next key is pressed.
SPACE		Types a space at the cursor.

Using the Keypad to Program Moves

Refer to **Table 2-3**. These keys appear on the keypad in pale yellow or white. They are the main keys used to program moves, positions and other commands.

Table 2-3, Programming Word Keys

Label or Name	Key Face	Function
G-CODES		Program G-codes.
MISCELLANEOUS FUNCTIONS		Activate Miscellaneous Functions.
X AXIS COORDINATE		Indicates X Axis position.
Z AXIS COORDINATE		Indicates Z Axis position.
SPINDLE SPEED		Provides Spindle Speed or provides maximum speed in a Constant Surface Speed move.
TOOL WORDS		Activates a tool and associated offsets.

(Continued...)

Table 2-3, Programming Word Keys (Continued)

Label or Name	Key Face	Function
POSITIVE/NEGATIVE INDICATOR		Used to indicate positive or negative direction or value, plus or minus; in Manual Data Input (MDI) mode, press key to insert a minus (-) sign.
DECIMAL/ASTERISK		<p>Decimal (.): In Manual Mode, marks decimal place in coordinates.</p> <p>Asterisk (*): Use to “comment out” all or part of a block (characters to the right of the asterisk will be ignored).</p> <p>NOTE: To type an asterisk, press SHIFT + .</p>

Soft Keys

The soft keys (also called function keys) are located on a keypad just below the screen. They are labeled **F1** through **F10**. Soft key functions are not hardwired. Their functions change with changes in mode.

The current function of each soft key appears on the screen in a label directly above each key. If a soft key label is **blank**, the soft key has no active function.

Accessing Keypad Commands Using a PC Keyboard (Optional)

The CNC supports an optional external keyboard. Most standard PC computer keyboards are compatible.

All alphanumeric keypad keys are available by pressing the corresponding key on the PC keyboard.

Software Basics

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

Pop-up Menus

Refer to **Figure 2-3**. Pop-up menus are temporary menus that enable you to make additional selections. Pop-up menus automatically appear where needed. Each pop-up menu contains a highlight bar. Use the **ARROW** keys to move the highlight bar up and down the menu. To activate a highlighted selection, press **ENTER**. To close a pop-up menu, press the key that activated it or press **CLEAR**.

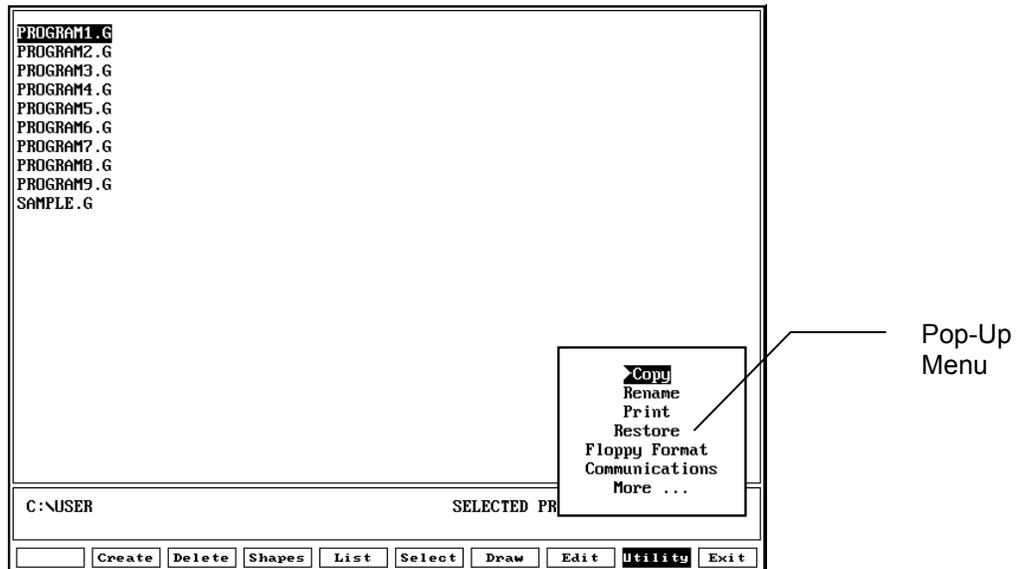


Figure 2-3, Pop-Up Menu

Screen Saver

After a set period of inactivity, the CNC's screen dims to preserve the monitor. Press any key to restore the CNC to a ready status. Keystrokes are not recognized until after you reactivate the monitor.

Highlight Bars

Each menu screen is equipped with a highlight bar. The highlight bar enables you to choose one of the selections. To move the highlight bar up, down, left, or right, press the appropriate **ARROW** key. Press **ENTER** to activate the highlighted selection.

Cursors

Some screens use a white underscore to indicate where to insert numbers and letters.

Clearing Entries

Press **CLEAR** key to delete all portions of an entry.

Operator Prompts

The CNC displays a prompt when it requires specific information. When the CNC prompts for a text entry, enter the information required to complete the operation.

Typeover and Insert Modes

The edit mode has two text entry modes: “typeover” and “insert”. By default, the CNC runs in the typeover mode. In the typeover mode, new characters replace characters marked by the cursor (the white underline). In **Ins (F3)** mode, new characters appear at the cursor and existing characters move to the right. The **Ins (F3)** soft key label highlights when the CNC is in “insert” mode.

To put the CNC in the Insert Mode:

1. Go to the Edit screen.
2. In Edit Mode, press **Ins (F3)**. The **Ins** soft key label highlights, indicating that Insert Mode is ON.

Deleting Text

To delete text:

1. Go to the Edit screen.
2. In edit mode, press **Del (F2)**. The character to the right of the cursor is then deleted.

Messages/Error Messages

Messages generated by the CNC are displayed in the MESSAGE area of the screen. It appears in all operating modes (S. Step, Auto, and Manual). When the CNC generates more than one message, the message with the highest priority is displayed in the MESSAGE area. Lower priority messages remain in memory. The **MESSAGE** label is highlighted while messages are stored in memory. There are two ways to review pending messages:

- ❑ Press **CLEAR** key. The current message clears and the next message appears.
- ❑ Go to the Manual screen and press **SHIFT + Message (F1)**. The Message Screen activates.

Refer to [Figure 2-4. Message Window](#). The MESSAGE screen enables you to view the last eight messages, whether or not they have been displayed. As new messages are generated, the oldest messages are cleared. Messages that have not been displayed appear at the top of the MESSAGE screen. Old messages appear below. To close the MESSAGE screen, press **Cont (F10)**.

To clear stored messages, close the MESSAGE screen and press **CLEAR** until the MESSAGE highlight clears.

Some messages are advisory in nature, others put the CNC in hold. If an error holds the CNC, solve the problem before you restart the program.

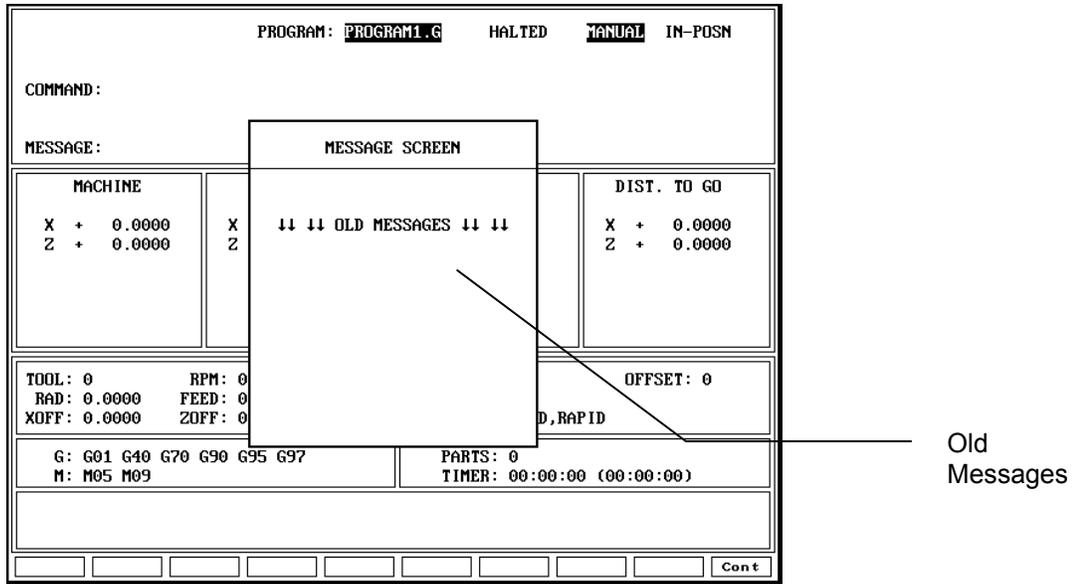


Figure 2-4, Message Window

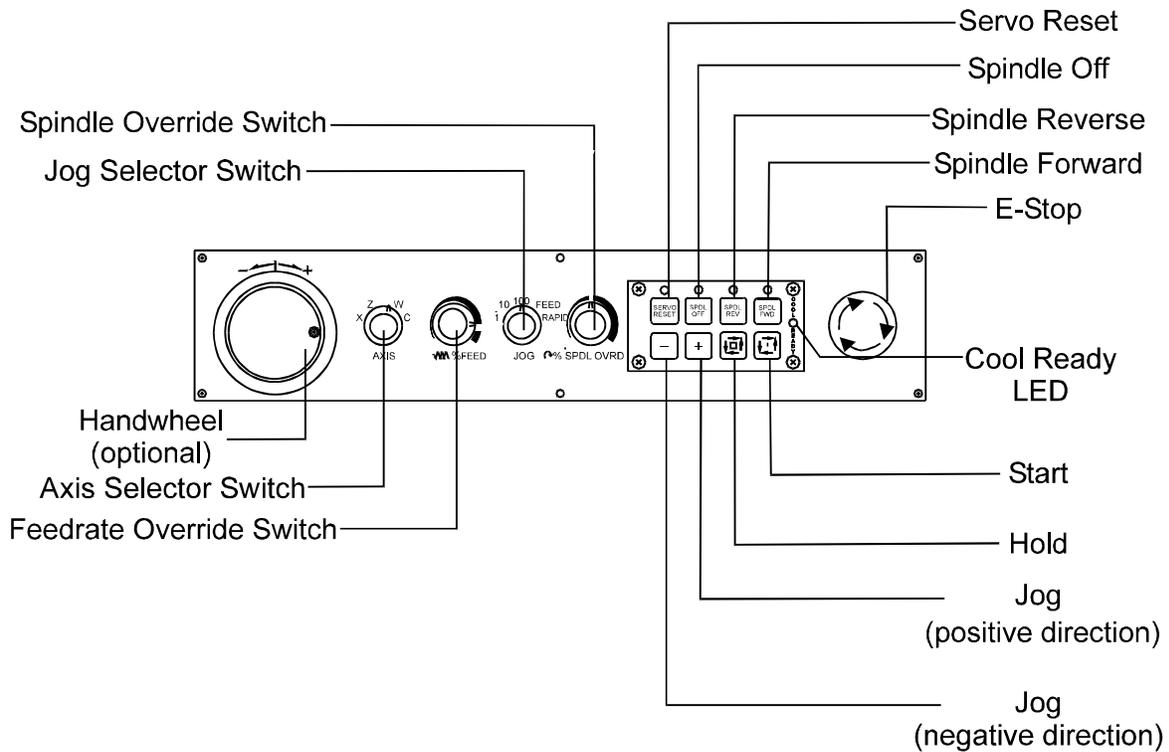
Section 3 - Manual Operation and Machine Setup

This section describes the functions of the keys and switches on the Manual Panel. It also provides information on manual operations.

Manual Panel

Refer to **Figure 3-1**. The Manual Panel contains the Manual Operation keys. Use these keys to move the machine manually.

NOTE: Some machines have the handwheels mounted to the machine tool. On these types of lathes, the handwheel shown on the manual panel is omitted.



MANPAN

Figure 3-1, Manual Panel

Manual Panel Keys

Manual Operation Keys enable you to control machine movements manually. These keys are located on the Manual Panel. See **Table 3-1**.

Table 3-1, Manual Operation Keys

Label or Name	Key Face	Purpose
JOG		Cycles the CNC through manual movement modes (JOG: RAPID, JOG: FEED, JOG: 100, JOG: 10, JOG: 1). The machine builder sets the default rapid and feed rates at setup and determines the actual speed of the machine during a move.
AXIS SELECT		In Manual Mode, selects the axis to be jogged.
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.
HANDWHEEL (optional)		Moves selected controlled axis while in the Manual Mode. Jog must be set to 1, 10, or 100.
SERVO RESET		Activates servo motors.
SPINDLE FORWARD		Starts spindle in a forward direction. NOTE: On some machines, gear range and RPM may need to be specified before you activate this key.
SPINDLE REVERSE		Starts spindle in a reverse direction. NOTE: On some machines, gear range and RPM may need to be specified before you activate this key.
SPINDLE OFF		Stops the spindle.
START		Starts all machine moves, except Jog.

(Continued...)

Table 3-1, Manual Operation Keys (Continued)

Label or Name	Key Face	Purpose
JOG +		Moves selected axis in positive direction. Used in all Jog modes. Feedrate determined by machine builder.
JOG -		Moves selected axis in negative direction. Used in all Jog modes. Feedrate determined by machine builder.
HOLD		Halts any running program or programmed move. Press START to continue.
EMERGENCY STOP (E-STOP)		The Emergency Stop button is bright red with white arrows. Press E-STOP to halt all axes and machine-related functions. When E-STOP is activated, the servo motors and any programming operations are shut down. The CNC defaults to Manual Mode. Use E-STOP for emergency shutdowns or to shut down the servos intentionally.

Manual Panel LEDs

The following keys have LEDs located directly above them on the Manual Panel. When the key is activated, the corresponding LED lights up.

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

Refer to [Figure 3-1, Manual Panel](#). The Coolant Ready LED is also located on the Manual Panel. Some CNCs are equipped with 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.

Manual Mode Screen

Refer to **Figure 3-2**. When the CNC is in Manual Mode, it displays the Manual screen. The Manual screen is the first screen displayed when you turn on the CNC. All other operating screens are similar in appearance and are selected from the Manual screen soft keys. In Manual Mode, the Manual (F4) soft key label highlights.

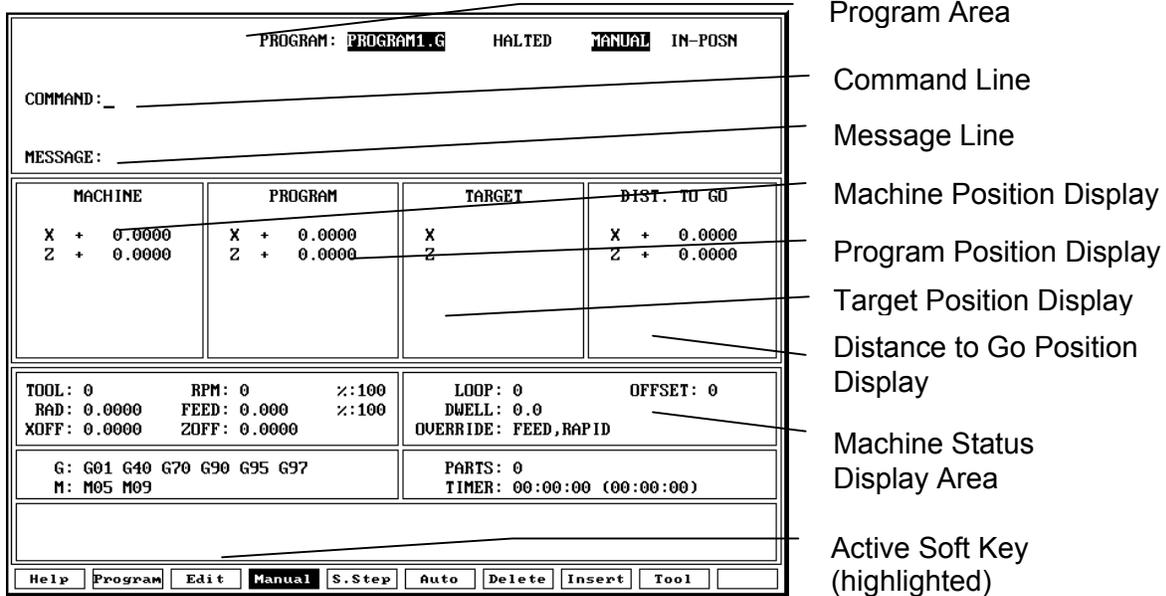


Figure 3-2, Manual Screen

The Manual screen is divided into the following areas:

- Program Area** Displays the storage device access indicator, the working program name, running status, mode of operation, and in-position status.
- Command Line** Enables you to enter commands manually.
- Message Line** Displays messages, prompts, and reminders.
- Machine Position Display** Displays machine's X and Z position coordinates in reference to Machine Home.
- Program Position Display** Displays machine's X and Z position coordinates in reference to Program.
- Target Position Display** Displays machine's X and/or Z programmed target position.

Distance To Go Displays distance to go with respect to X and Z commanded coordinates.

Machine Status Display Area

Displays operating information.

Active Soft Key Refer to **Table 3-2**. Soft key functions change from screen to screen. An active function identifies the function of the soft key. A highlighted label indicates an active mode.

Table 3-2, Manual Mode Soft Keys

Soft Key Name	Soft Key Number	Description
Help	F1	Help Mode activates.
Program	F2	List of user programs.
Edit	F3	Edit Mode activates. A program must first be selected.
Manual	F4	Press to enter Manual Mode from Auto and S.Step.
S.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 line in Manual Mode.
Insert	F8	Puts the cursor in Insert Mode. Typed text is inserted without overwriting existing text.
Tool	F9	Brings Screen to the Tool Page. Tool Page is used to store tool diameter and length offsets, and wear factors.

Machine Status Display Area Labels

TOOL: Active tool.

DIA: Active tool radius.

XOFF: X-axis tool length offset for active tool.

RPM: Current spindle speed in revolutions per minute.

FEED: Current feed rate (in Inch/MM per minute).

ZOFF: Z-axis tool length offset for active tool.

% (RPM): Spindle override setting (40% to 160%).

% (FEED): Feedrate override setting (0% to 120% for Feed moves and 0% to 100% for Rapid moves).

LOOP: Loop counter. Counts down the number of times a loop command has been performed.

DWELL:	A timed pause, indicated in number of seconds or revolutions. Units are specified as seconds or revolutions, during builder setup.
OVERRIDE:	Indicates whether the feedrate override setting applies to both feed and rapid moves or to feed moves only.
OFFSET:	Active G53 offset.
PARTS:	Number of parts. Resets to zero when you enter Auto or Single Step mode.
TIMER:	Indicates 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.
G:	Active G-codes.
M:	Active M-codes.

Program Area Labels

BLK:	Block number (in S. Step or Auto Mode).
PROGRAM:	Name of loaded program.
HALTED/*HALTED:	No asterisk: Indicates machine is in a programmed hold or has completed its program. With asterisk: Indicates hold was activated by an event or HOLD was pressed.
MANUAL/AUTO/S.STEP:	Current operating mode displayed.
IN-POSN:	Indicates whether the machine has reached an endpoint (in-posn).
COMMAND:	Used to enter commands in Manual Mode.

Powering on the CNC

Refer to [Figure 3-1, Manual Panel](#). The **E-STOP** can be in or out when powering up. Turn ON the CNC according to the builder's instructions. When the switch is ON, the CNC completely resets. All parameters and variables reset to their default values.

To power on the CNC:

1. Turn the power switch ON. The startup screen activates and prompts you to "**Press F10 to continue**".
2. Press **F10**. The CNC displays the **Software Options** menu.
3. Highlight **Control Software**, and press **ENTER**. Manual Mode activates.

Shutting Down the CNC

To shut down the CNC:

1. Press **E-STOP**. The servos disengage and the CNC reverts to Manual Mode.
2. Press **SHIFT (F10)**. The CNC displays the **Software Options** menu.
3. Follow the builder's instructions for turning the CNC OFF.

Setting/Resetting the Emergency Stop (E-STOP)

1. Press **E-STOP** to take all axes and spindle servos offline and halt all machine movement.
2. To reset the **E-STOP**, pull out the **E-STOP** rotary switch and rotate it clockwise in the direction of the arrows.
The switch makes a clicking sound when it resets. Resetting the **E-STOP** does not automatically reactivate the servos. The servos must be reset to move the machine.
3. Press **SERVO RESET** to reset the servos.

Activating/Resetting the Servos

For safety reasons, the CNC powers up with the servo motors disengaged. While the servos are off, the CNC cannot move the machine. A "**SERVO OFF!**" message is generated when the servos are disengaged. The servos are also disengaged when you press **E-STOP**, or if the machine attempts to move beyond a limit switch.

To reset the servos:

1. If the servos were disengaged by a limit switch, manually reposition the machine inside its normal range of travel.
2. Press **E-STOP**. "**MESSAGE: E-STOP IN - SERVO OFF!**" is displayed in the MESSAGE area.
3. Reset the **E-STOP** by rotating the **E-STOP** rotary switch in the direction of the arrows.

The switch makes a clicking sound when it resets.

4. Press **SERVO RESET**. "**MESSAGE: SERVO DELAY, PLEASE WAIT...**" is displayed while the servos are being reset. The message disappears when the servos reset.

Manual Mode Settings

Features (or settings) that remain active for more than one operation are said to be "modal". Modal features remain active until changed or canceled. The CNC's function is modal.

For example, when in Rapid Mode, the CNC executes all moves in Rapid until put in the Feed Mode. The CNC can be in two modes, such as the Absolute Mode and the Inch Mode, at the same time, as long as the modes do not conflict.

Before you make a manual move, make any necessary mode settings. Modes set from the Manual screen remain active if the CNC is put in a program operating mode until the mode is changed by the program or operator.

The following modes can be set from the Manual screen:

- Position mode: either Absolute or Incremental Positioning.
- Move mode: (Rapid or Feed) – for either Rapid move or a move at a specified feedrate.
- The active tool (includes tool length offsets and tool nose radius compensation).
- The measurement mode (Inch/MM).

The following settings can be made in the Manual screen:

- The location of Machine Home position.
- The location of Part Zero.

There are three types of moves available in Manual Mode:

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

Activating Manual Mode Rapid or Feed

Turn the **JOG** rotary switch to cycle through all available jog modes. Choose Rapid or Feed Mode. The active feedrate/rapid mode is displayed in the Machine Status Display Area.

NOTE: In Manual Mode, press **R + ENTER** to toggle the override setting between the following choices:

- FEED and RAPID rate override (FEED, RAPID)
- FEEDRATE override (FEED)

Switch the setting to apply the current override selection to programmed rates.

Adjusting the Feedrate

NOTE: If G95 (feedrate per revolution) is used, the spindle must be turning. Otherwise, the axes cannot be moved.

The basic feedrate is manually keyed in as Feedrate Per Minute (FPM, G94) or Feedrate Per Revolution (FPR, G95) command. You can change the feedrate at any time, manually or in a program. Use F-codes to set the feedrate. The F-code follows the G94 or G95 command in the following format:

G94 Fnnn or G95 Fnnn

Refer to "[Section 4 - Preparatory Functions: G-Codes](#)" for details pertaining to G-codes.

When the **FEEDRATE OVERRIDE** is set to 100%, the CNC makes all Feed moves at the programmed feedrate. To run the CNC at a percentage of the programmed feedrate, adjust the **FEEDRATE OVERRIDE**. Each click of the **FEEDRATE OVERRIDE** adjusts the feedrate setting by increments of 10% each. The override rotary switch has a range of 0% to 120%. Align the rotary switch pointer with the white stripe to set the override to 100%. All moves are affected by the **FEEDRATE OVERRIDE**.

CAUTION: When the CNC is shut down, the manufacturer's setup file will reload a default feedrate when it powers up again.

Adjusting Rapid Move Speed

The **FEEDRATE OVERRIDE** also adjusts Rapid moves. If FEED, RAPID is set, every click of the **FEEDRATE OVERRIDE** adjusts the Rapid rates by increments of 10% of the default speed. The override rotary switch gives you a range of 0% to 100%. Align the rotary switch pointer with the white stripe to set the override to 100%. The maximum override rate for rapid speeds is 100%.

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

NOTE: In Manual Mode, press **R + ENTER** to toggle the override setting between the following choices:

- FEED and RAPID rate override (FEED, RAPID)
- FEEDRATE override (FEED)

Toggle the setting to apply the current override selection to programmed rates.

Setting Absolute Zero

Refer to **Figure 3-3**. In Absolute Mode, all positions are measured from Absolute Zero. Absolute Zero is the point the CNC recognizes as X0, Z0 when the Absolute Positioning Mode is active. You can set Absolute Zero anywhere within or outside a machine's range of travel. It can be moved as many times as necessary. All absolute X and Z positions are measured from this point.

Setting the Absolute Zero Reference to a location on the part is referred to as "setting the Part Zero".

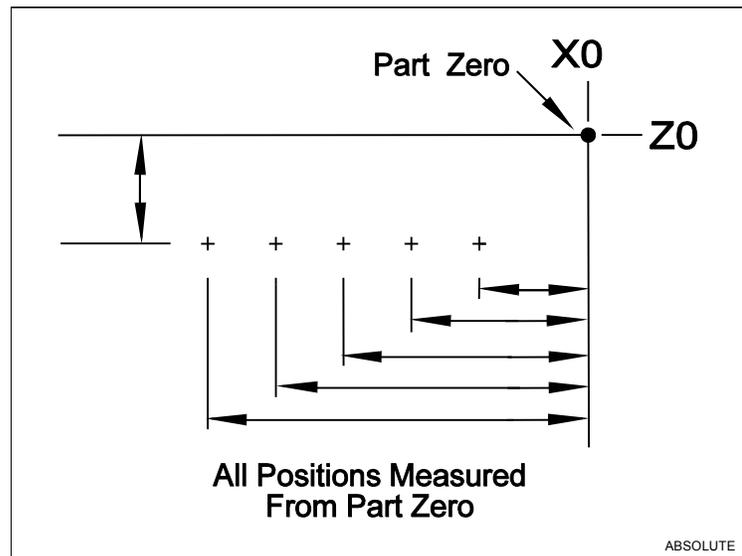


Figure 3-3, Absolute Positioning

Part Zero

In all programming examples in this manual place, Part Zero is the centerline (X) and front face (Z) of the part.

There are three ways to establish Part Zero on a workpiece for a particular tool:

- Tool length offsets (TLO)
- G53 offsets
- Absolute Presets (G92)

TLOs (T-codes) are the best way to set Part Zero. Tool length offsets enable you to program the moves from Part Zero, regardless of the type or size of the tool. Enter tool length offsets into the Tool Page; then, activate as needed.

Refer to "[Section 9 - The Tool Page and Tool Management](#)" for details on how to use tool length offsets to set Part Zero.

G53 offsets are used to change Zero location. G92 absolute presets make the current position set to what you enter, i.e., the X, Z entry becomes the current position when G92 is entered.

When the machine is powered up, its present location is automatically set as the Absolute Zero Reference. When the CNC is shut down, the position of the Absolute Zero Reference is lost.

Jog Moves

Jog moves are available in Manual Mode, Teach Mode, or when the Tool Page is active and the servos are on.

Refer to **Table 3-3**. There are five available jog modes. The actual rate for each mode is determined at machine setup. Use **JOG** to cycle the CNC through the jog mode choices.

NOTE: You can change the Jog Mode at any time when the CNC is in Manual Mode.

Table 3-3, Jog Moves

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 resolution of machine.
Jog: 10	Conventional Jog Mode. Increment set to 10 times resolution of machine.
Jog: 1	Conventional Jog Mode. Increment set to actual resolution of machine.

Changing the Jog Mode

Jog moves, with the exception of Rapid, are performed in Feed.

To change the Jog Mode:

1. In Manual Mode, turn the JOG rotary switch to select a jog feedrate.

NOTE: You can change the Jog Mode in Manual Mode, Teach Mode, or when the Tool Page is active.

Selecting an Axis

To select an axis in the Manual Mode:

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

Jogging the Machine (in Increments)

In Manual Mode, you can position the machine with jog increments. Jog increments advance the machine in selected increments: 1, 10, or 100 times the machine resolution.

To make a jog increment move:

1. Use the **AXIS SELECT** rotary switch to select an axis.
2. Use the **JOG** switch to cycle through the move mode choices and choose a conventional Jog Mode (1, 10 or 100 times machine resolution).
3. Press the **JOG+** or **JOG-** key to choose a direction. Do not hold down the key. Each time you press the key, the machine jogs along the selected axis by the selected increment.

Jogging the Machine (Continuous Moves)

In Manual Mode, you can move the machine at feed rate or at the jog rapid rate. The machine builder determines the effective jog and feed rates at setup.

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

Manual Data Input (MDI) Mode

NOTE: In MDI mode, you cannot execute canned cycles or activate Tool Nose Radius Compensation.

The Manual Data Input mode (MDI) enables you to command moves without creating a part program. MDI is also a quick way to program a single move, or a series of moves that will be used only once.

To execute commands, type an instruction on the **COMMAND:** line of the Program Area and press the **START** key. In Manual Mode, the cursor rests on the command line.

You can program more than one command at a time. Use a block separator (;) to separate the commands.

One shot moves can be paused by pressing **HOLD**, continued by pressing **START**, or canceled altogether by pressing **MANUAL (F4)**. MDI moves are executed only once. To recall a previously commanded block, press the **UP ARROW** key as many times as necessary. Approximately 2K (2048) bytes of MDI commands are stored in memory and can be recalled by using Up (Previous) and Down (Next) arrow keys.

CAUTION: It is important that you know where the Absolute Zero is before you make Absolute Mode moves.

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**. The CNC will 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; G97 S600; M3

M43	Activates Gear Range defined by M43 in setup
G97 S600	Activates specified Spindle Speed
M3	Activates Spindle Forward

Handwheel Operation

NOTE: The manual panel can include an optional handwheel. This handwheel is omitted if a customer has elected to purchase one or more handwheels mounted to the machine.

The sub-sections below describe handwheel operation for:

- ❑ Standard handwheel configuration, consisting of one handwheel mounted on the manual panel.
- ❑ Purchased option, consisting of one or two handwheels mounted on the machine.

Operating a Handwheel Mounted on the Manual Panel

In some 4200T systems, one handwheel is mounted on the Manual Panel. In Manual Mode, you can use the handwheel to jog any controlled axis (X or Z). The machine builder must enable the handwheel option in the Setup Utility.

NOTE: The handwheel soft key (**F10**) is labeled "**HandWI**". If the Handwheel feature has not been enabled in the Setup Utility, the **F10** soft key will not be labeled "**HandWI**".

To operate the standard handwheel:

1. In Manual Mode, press **HandWI (F10)**. The **HandWI (F10)** soft key activates and highlights.
2. Use **AXIS SELECT** to select an axis.
3. Use the **JOG** to select a conventional Jog Mode (1, 10 or 100).
4. Rotate the handwheel in a positive or negative direction.
5. To return to the Manual Mode, press **HandWI (F10)**. The handwheel option deactivates and the soft key is no longer highlighted.

Refer to **Table 3-4** for **JOG** selections available with the handwheel.

Table 3-4, Handwheel Jog Resolution Settings

Jog Mode Setting	Handwheel Resolution
FEED	Not Available
RAPID	Not Available
1	Machine Resolution
10	10 times Machine Resolution
100	20 times Machine Resolution

Operating Handwheels Mounted on the Machine

As a purchased option, two handwheels can be mounted on the machine, one for each axis (X and Z). The handwheel is mounted so that it directly controls axis movement. While in the Manual Mode, you can use the handwheels to jog the X and Z-axis. The machine builder must enable the handwheel feature in the Setup Utility. Refer to **Table 3-4** for **JOG** selections available with the handwheel option.

To operate a Handwheel mounted on the machine:

1. Use **JOG** to select a conventional jog mode (1, 10 or 100).
2. Rotate the handwheel in a positive or negative direction.

Section 4 - Preparatory Functions: G-Codes

Preparatory Functions: G-Codes

G-codes initiate motion commands, canned cycles, various machine functions and other CNC capabilities. More than one G-code may be specified per block. If a block contains conflicting G-codes, an error message appears. Refer to [Table 4-1, Modal and Non-Modal G-Codes](#) for a list of modal and non-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. Once the non-modal block is completed, the Modes active before the non-modal command take effect again. Non-modal G-codes have no preset defaults.

In the subsections that describe individual G-codes, every attempt has been made to show the most convenient method of programming each code. There are usually several ways to program a G-code. Typically, if a move type or G-code is covered in the Edit Help Menus or Edit Help G-Code Menu, you should use these sources to program the code. In all cases, you can program G-codes in the Edit Mode.

Table 4-1, Modal and Non-Modal G-Codes

Modal		Non-Modal	
G-Code	Function	G-Code	Function
G0	Positioning-Rapid Traverse	G4	Dwell
G1	Linear Interpolation	G5	Ellipse
G2	Circular Interpolation-CW	G9	Exact Stop Check
G3	Circular Interpolation-CCW	G31	Probe Move
G22	Stored Stroke Limit ON (Programmable Stroke Limit)	G33	Threading
G24	Spindle Speed Clamp	G65	User Macro Single Call
G40	Tool Tip Radius Compensation, Cancel	G74	Area Clearance (Face) Cycle
G41	Tool Tip Radius Compensation, Left of Path	G76	Rough Turning/Boring Cycle
G42	Tool Tip Radius Compensation, Right of Path	G77	Rough Facing Cycle
G53	Work Coordinate System (1-99, select)	G78	Peck Drill Cycle
G59	Modal Corner Rounding	G79	Chip Break Drill Cycle
G60	Modal Corner Rounding Off	G81	Longitudinal Grooving Cycle (Turn)
G61	Exact Stop Check Mode (Continuous Path OFF)	G82	Face Grooving Cycle
G64	Cutting Mode (Continuous Path ON)	G83	Longitudinal Thread Cycle, Plunge
G66	User Macro Modal Call	G84	Longitudinal Thread Cycle, Compound
G67	User Macro Modal Call Cancel	G85	Face Thread Cycle, Plunge
G68	Coordinate System Rotation	G86	Face Thread Cycle, Compound
G70	Inch Programming	G87	Tapping Cycle
G71	MM Programming	G88	Boring
G72	Axis Scaling	G92	Absolute Zero Preset
G90	Absolute Programming		
G91	Incremental Programming		
G94	Per Minute Feed		
G95	Per Revolution Feed		
G96	Constant Surface Speed Function (CSS)		
G97	Revolutions Per Minute (RPM)		

Rapid Traverse (G0)

Format: G0

G0 initiates rapid traverse. The machine builder sets the actual rapidrate in the Setup Utility. Use rapid traverse to position the tool prior to or after a cut. Do not use rapid to cut a part.

One or two axes can be included on a block with G0. The two axes reach target simultaneously.

Rapid Traverse is modal and remains in effect until canceled or changed.

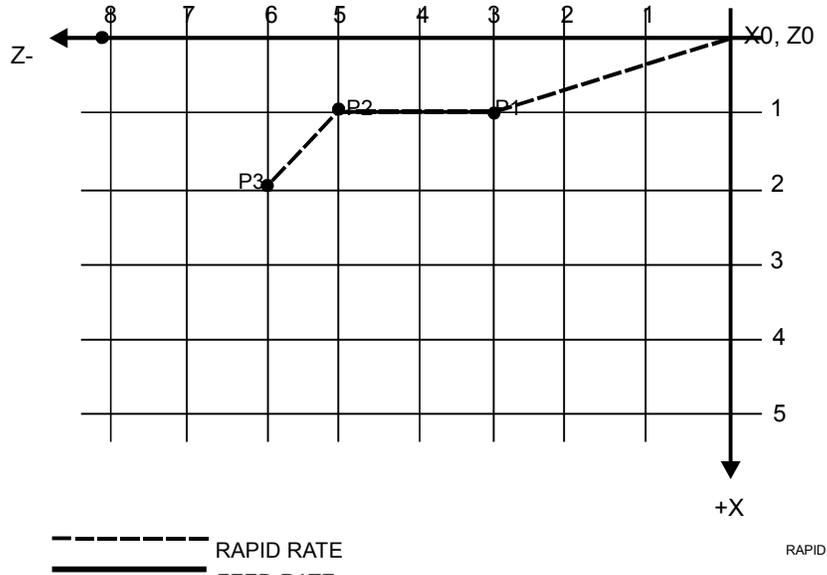


Figure 4-1, Rapid Traverse

Refer to **Figure 4-1**. Two examples of rapid motion are diagrammed. **Table 4-2** provides and explains the program blocks required to complete the moves described in **Figure 4-1**.

Table 4-2, Rapid Traverse

N1	G90	G0	X1	Z -3	Rapid to X1, Z-3 (P1) in Absolute Mode.
N2	G1	Z-5			Z-axis feeds to Z-5 (P2).
N3	G0	X2	Z-6		Rapid to X2, Z-6 (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.](#)"

Linear Interpolation (G1)

Format: G1

Linear Interpolation (G1) initiates feed motion in a straight line. It is used for cutting a part. Straight-line motion can be linear or angular. The block could contain any combination of the available axes.

G1 is modal and remains in effect until changed. A feedrate must be given on or prior to the G1 block.

Linear moves require only one dimension to define the endpoint of motion. Provide the dimension (X or Z) for the axis in which motion occurs. **Table 4-3** demonstrates how G1 is used in a program to perform linear moves. MM equivalents are provided in parentheses following the Inch measurements. **Figure 4-2** is a diagram of the part programmed.

Table 4-3, Straight Line Programming, Absolute/Inch (MM) Mode

N1	G90 G70 (G71) G1 X0 Z0 G95 F.010 (.25)	Feed to P1.
N2	X1 (25.4)	Feed from P1 to P2.
N3	Z-1.5 (-38.1)	Feed from P2 to P3.
N4	X2 (50)	Feed from P3 to P4.
N5	Z-4.25 (-107.95)	Feed from P4 to P5.
N6	X3 (76.2)	Feed from P5 to P6.
N7	M2	End program, return to N1.

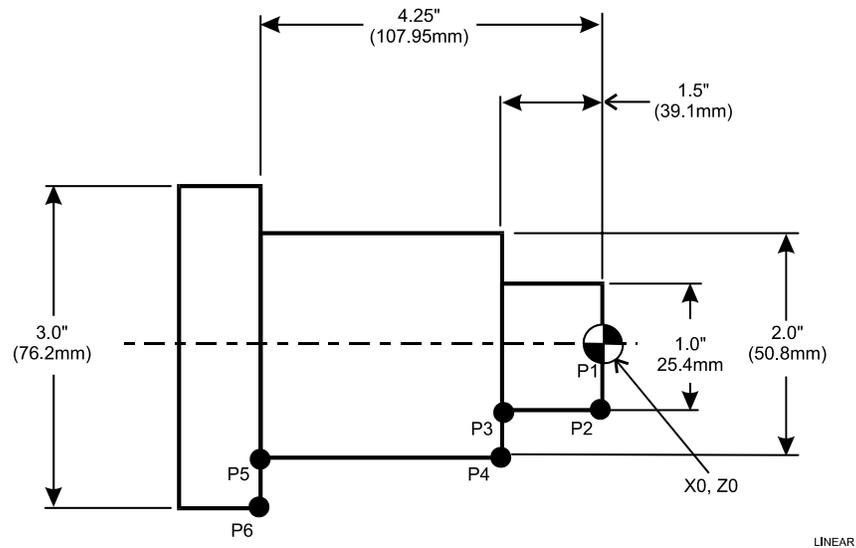


Figure 4-2, Linear Motion

Angular Motion Programming Example

Angular (vectored) motions are straight-line motions involving motion in both the X- and Z-axes. They require two dimensions (X and Z) to define the endpoint of motion. **Table 4-4** demonstrates how G1 is used in a program. MM equivalents are provided in parentheses following the Inch measurements.

Table 4-4, Angular Programming, Absolute/Inch (MM) Mode

N1	G70 (G71) G90 G1 X0 Z0 G95 F.010 (.25)	Absolute, Inch Mode feed to P1.
N2	X.5 (12.7)	Feed to P2.
N3	X1 (25.4) Z-.25 (-6.35)	Feed to P3. (angle endpoint)
N4	Z-1.5 (-38.1)	Feed to P4.
N5	X2 (50.8) Z-2.75 (-69.85)	Feed to P5. (angle endpoint)
N6	Z-4.25 (-107.95)	Feed to P6.
N7	X3 (76.2)	Feed to P7.
N8	M2	End program, return to N1.

Figure 4-3 is a diagram of the part programmed in **Table 4-4**. Angular moves are defined by the start and endpoints of the move. The diagram shows Absolute Mode (Inch and MM measurements provided). All dimensions are referenced to Part Zero (X0, Z0).

In Incremental Mode (G91), Z dimensions are defined by the distance from the current to target positions. X dimensions are the difference in diameter between the current and target positions.

When more than one axis endpoint is defined (Xn, Zn), the CNC uses Linear Interpolation to produce a vector along the programmed path.

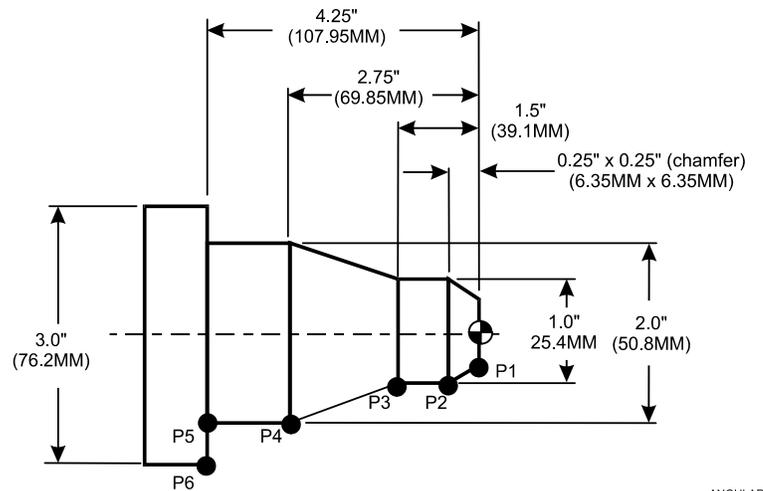


Figure 4-3, Angular Motion

Circular Interpolation (G2 and G3)**Arc input format:** G2 (G3) Xn Zn In Kn**Radius format:** G2 (G3) Xn Zn Rn

Circular Interpolation initiates circular moves, including arcs. **Table 4-5** contains the parameters necessary to program these moves. G2 commands a clockwise motion. G3 commands a counterclockwise motion.

Table 4-5, Parameters for Circular Interpolation

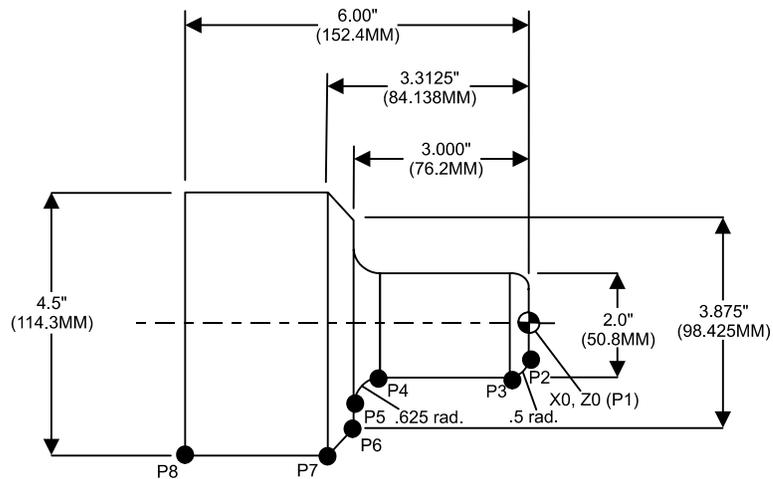
G2	CW (clockwise) motion.
G3	CCW (counterclockwise) motion.
X, Z	Endpoint of Arc motion in Absolute or Incremental Mode.
I K	Distance from current tool position to arc center in each axis. X arc center. Z arc center.
R	Arc Radius.
n	Some value.

Examples of Circular Interpolation

Refer to **Table 4-6** and **Figure 4-4**. Both the MM and Inch equivalents are provided.

Table 4-6, Circular Interpolation in Absolute Mode, Inches (MM)

N1	G70 (G71) G90 G1 X0 Z0 G95 F.010 (.25)	Activate Inch and Absolute Mode and set feedrate to IPR.
N2	X1 (25.4)	Feed to P2.
N3	G2 X2 (50.8) Z-.5 (-12.7) R.5	Arc move to P3.
N4	G1 Z-2.375 (-60.325)	Feed to P4.
N5	G3 X3.25 (82.55) Z-3 (-76.2) R.625	Arc move to P5.
N6	G1 X3.875 (98.425)	Feed to P6.
N7	X4.5 (114.3) Z-3.3125 (-84.138)	Vector move feed to P7.
N8	Z-6 (-152.4)	Feed to P8.
N9	M2	End Program.



CIRCLE1

Figure 4-4, Circular Interpolation

Table 4-7 provides the incremental version of the part program diagrammed in [Figure 4-4, Circular Interpolation](#).

Table 4-7, Circular Interpolation in Incremental Mode, Inches (MM)

N1	G70 (G71) G90 G1 X0 Z0 G95 F.010 (25.4)	Activate Inch and Absolute Mode. Set feedrate to .010 inches per revolution.
N2	G91 X1 (25.4)	Activate Incremental Mode and feed to P2.
N3	G2 X1 (25.4) Z-.5 (-12.7) R.5	Arc move to P3.
N4	G1 Z-1.875 (-47.625)	Feed to P4.
N5	G3 X1.25 (31.75) Z-.625 (-15.875) R.625	Arc move to P5.
N6	G1 X.625 (15.875)	Feed to P6.
N7	X.625 (15.875) Z-.3125 (-7.938)	Vector move to P7.
N8	Z-2.6875 (-68.263)	Feed to P8.
N9	M2	End Program.

In block N1, the CNC positions the tool at Part Zero (X0, Z0). This must be done in Absolute Mode. N2 activates Incremental Mode for the rest of the program.

Partial Arcs

Any arc with a value of less than 360 degrees is a partial arc. Most arcs require four Address Words (X, Z, I, K) to define the arc move.

A Circle does not require an endpoint input because it was the same starting point and endpoint.

Dwell (G4)

Timed Dwell Format: G4 Tx.x

Infinite Dwell Format: G4 T0

Dwell Per Revolution Format: G4 Tx

Dwell (G4) can be used to program a delay between moves and is specified in setup as a timed dwell or as dwell per revolution. At setup (via Builder Setup\Miscellaneous\ #11 - Dwell RPMs in FPR Mode), if Dwell RPMs in FPR Mode is set to No, all Dwells will be in seconds; e.g., G4T4.5 will cause a dwell of 4.5 seconds before the program continues to the next block. If Dwell RPMs in FPR Mode is set to Yes, Dwell will be in seconds if the control is in Feed Per Minute (G94), or it will be in number of spindle revolutions if the control is in Feed Per Revolution mode (G95). When Dwell pertains to number of revolutions, revolutions expressed in decimal fractions will be ignored; e.g., G4T50 will produce a dwell of fifty (50.0) revolutions before the program continues execution.

For a timed dwell, T represents seconds. For dwell per revolution, T represents the number of turns where one complete turn is a revolution.

A timed dwell is a timed stop. An infinite dwell is a stop that can only be canceled by pressing **START**. With a dwell activated, the CNC halts motion on all axes, but other functions (coolant on/off, spindle control, etc.) 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 Example: N20 G4 T2.1

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

Infinite Dwell Example: N21 G4 T0

Block 21 commands an infinite dwell.

The time or revolution countdown is displayed in the Machine Status area of the Manual screen.

NOTE: ANILAM recommends that you use the Programmed Stop M-code (M00) instead of G4T0.

Ellipse (G5)

Format: G5 Xn Zn In Kn An Bn Ln

Use G5 to program a full or partial ellipse. The following variables must be known and programmed after the G5 code. Refer to **Table 4-8**.

Program the starting point prior to G5. It must lie along the ellipse.

Table 4-8, G5 Address Words

Address Word	Description
X	Incremental Z end pt. (or dist. from start to end)
Z	Incremental X end pt. (or dist. from start to end)
I	Incremental X ctr pt. (or dist. from start to ctr)
K	Incremental Z ctr pt. (or dist. from start to ctr)
A	Half-length of ellipse in Z axis *
B	Half-width of ellipse in X axis *
L	Direction of tool travel: 1 is CCW; -1 is CW
*Half-length is the dimension of a quadrant of the ellipse. For a full ellipse, it is half the Z length (for A variable), and half the X width (for B variable). A and B are always positive.	

Non-modal Exact Stop Check (G9)

Format G9

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

Refer to **Table 4-9** for a list of G-codes related to Exact Stop Check (G9).

Table 4-9, Exact Stop Check G-Codes

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

The Non-Modal Exact Stop Check (G9) initiates the Non-Modal In-Position Mode. With the In-Position Mode activated, the CNC approaches target and performs an in-position check before the next move is executed. The CNC comes to a complete stop at the completion of each command (end of block). This could cause witness marks to appear on the workpiece, but prevents the CNC from rounding-off sharp corners.

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.

In-Position Mode only remains active for the block containing the G9 command. Use G61 to initiate modal Exact Stop Check (In-Position Mode).

Setting Software Limits (G22)

Format: G22 Xn Zn In Kn
Activates Software Limits, Modal

Format: G22
Cancels Software Limits and allows free movement within the machine limits.

Refer to **Table 4-10**.

Table 4-10, G22 Address Words

Address Word	Format	Description
G22	---	Stored Stroke Limit (Programmable Travel Limits).
X	xxx.xxxx	Coordinate of the X positive software limit.
Z	xxx.xxxx	Coordinate of the Z positive software limit.
I	xxx.xxxx	Coordinate of the X negative software limit.
K	xxx.xxxx	Coordinate of the 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 positive X or Z limits represent the extreme distance the tool can travel in a positive X or Z direction. The negative X and Z limits represent the extreme distance the tool can travel in a negative X and Z direction.

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

Before you set software limits, make sure the tool is within the "envelope" defined by the software limits.

Refer to [Figure 4-5, Front Tooling Software Limits](#) and [Figure 4-6, Rear Tooling Software Limits](#). The diagrams show that the location of positive and negative software limits (+X, -X, +Z, -Z) reverses from front to rear tooling.

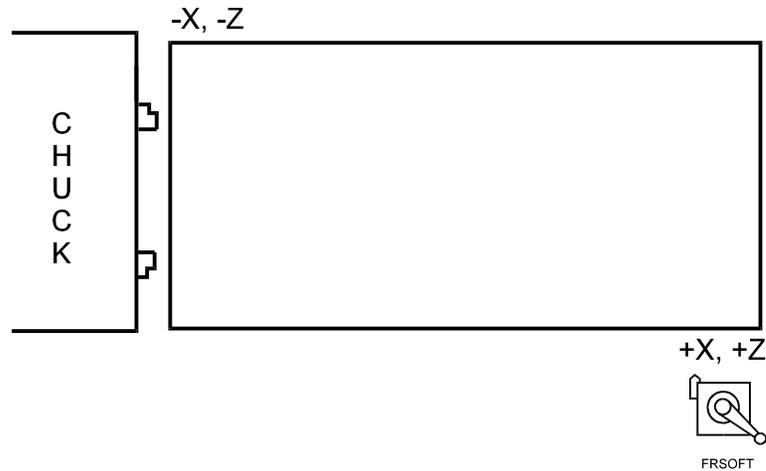


Figure 4-5, Front Tooling Software Limits

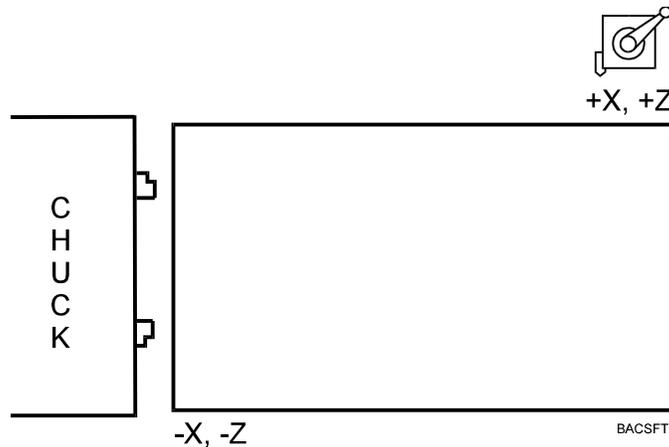


Figure 4-6, Rear Tooling Software Limits

Maximum Spindle Speed (G24)

Format: G24 Sxxxx

Refer to Table 4-11.

Table 4-11, G24 Address Words

Address Word	Format	Description
G24	---	Sets the maximum programmable spindle speed.
S	xxxx	Maximum programmable spindle speed, maximum four digits.

G24 sets the maximum programmable spindle speed, when the CNC is in Constant Surface Speed Mode (G96). The G24 command stops the spindle from accelerating to the machine maximum spindle speed when facing to the centerline of the part. G24 is not required when using G97 (direct RPM value).

The S-word can contain up to four digits (Sxxxx, range 0001 to 9999). Leading zeros can be omitted.

Once you program a G24 command, the CNC prevents the programmed spindle speed from exceeding the entered amount (Sxxxx). The G24 command must be stated after each Gear Range statement.

Probe Move (G31)

Refer to [“Section 16 - Advanced Programming Features,” “Probe Move \(G31\).”](#)

Thread Cutting (G33)

Format: G33 Xn Zn (En or Fn) An Cn

Refer to **Table 4-12**.

Table 4-12, G33 Address Words

Address Word	Description
Xn	X-axis end position.
Zn	Z-axis end position.
En	Threads per inch
Fn	Feed per revolution (pitch).
An	Incremental X retract distance.
Cn	Incremental Z retract distance.

Refer to **Figure 4-7**. The diagram describes the G33 Single-Pass Threading Cycle. Use G33 to program tapers and compound threading. G33 cuts to the depth to which the tool is positioned in the block preceding the G33 block. It does not contain an option for a finish pass.

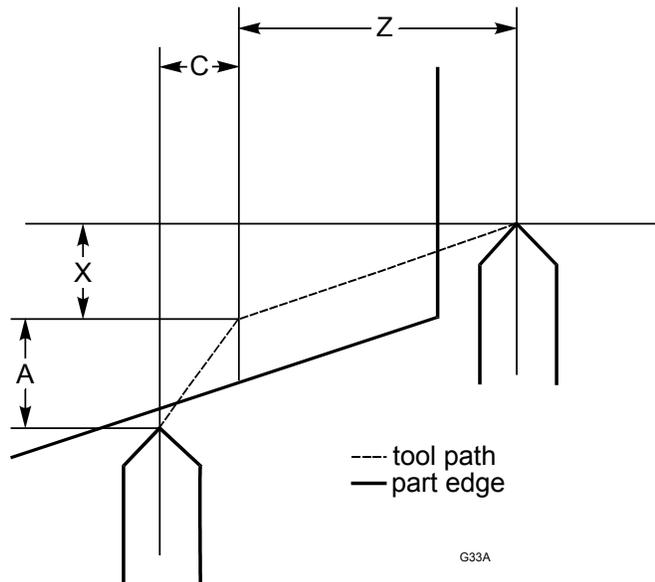


Figure 4-7, G33 Threading Cycle

NOTE: Pitch (F) = $\frac{1}{\text{Threads Per Inch (E)}}$

Refer to **Table 4-13** for a sample Program Listing containing the G33 command. E or F, not both, must be programmed. At least one axis address must accompany F/E value. F (pitch) is usually used in MM programming. E (Threads per Inch) is usually used in Inch programming. All other Address Words are optional.

A and C are always incremental, regardless of whether the Absolute (G90) or Incremental (G91) Mode is active.

Table 4-13, G33 Program Listing

Block	Format	Description
N33	G33 Z-3.0 F.1 (or E10) A.2	This block will execute a threading cycle along Z to Z-3.0, with a pitch of 0.1 and an X pullout distance of 0.2.

NOTE: All variables are sign (+/-) dependent. Moves are Outer Diameter or Inner Diameter, and move toward or away from the chuck or part centerline. The sign of the variable determines what kind of move the CNC will make. Make sure to program the appropriate sign (+/-).

Offsets (Work Coordinate System Select) (G53)

Format: G53 Oxx Xn Zn

The work coordinate system (absolute zero) of the CNC can be shifted to a preset dimension from machine zero using the G53 function. This function differs from G92 in that it is always in reference to machine zero, not current zero.

When programmed, G53 will automatically cancel M100 (mirror image) and G72 (scaling).

99 offsets (zero shifts) are available to the user. Offsets are stored in a table in the CNC accessed by F1 from the Tool Table Screen. This table can be updated through the program. If programmed in the part program, the values programmed will override and overwrite those in the table.

Each of the 99 offset values are designated with the letter O, followed by 1-99 (i.e., O1, O2, etc.), then the axes and their values.

The G53 block can be used in several ways as follows:

1. Use offset Number 3 from the preset table by programming G53 O3.
2. Clear any active offset by programming G53 O0.
3. Do update offset table, and do activate the shift by programming G53 On Xn Zn C. The word C tells the CNC to activate and update.
4. Do update offset table, but do not activate the shift by programming G53 On Xn Zn.

Notes:

1. All of the above will clear any G92 active Tool Table.
2. Typically, the table found in the Tool Table is used (**F1**) to set the values and the values activated in the part program by programming G53 with O# (such as G53 O1).
3. If Table values are programmed in the part program, the values programmed will override and overwrite those in the table under **F1**.
4. G53 O3 is used to activate a zero point previously described in the table and is one of the most typically used.
5. G53 O0 is used to clear any offset of G53 or G92. It resets the current zero to that of machine home.
6. G53 On Xn Zn C is used as an immediate absolute zero-shift.
7. G53 On Xn Zn is used when offsets are defined at the beginning of a program. It is used strictly to set-up the offset table. No activation occurs.
8. G92 may be used in reference to (after) any G53 active, or without any G53 active (G53 O0).
9. G53 is modal, and G53 O0 (use none) is active at power-up.
10. Only axes required to be shifted need to be programmed.
11. G41/G42 should be cancelled before programming G53.

Changing Fixture Offsets in the Table

To change a fixture offset to a manually entered coordinate:

1. Go to the Tool Table.
2. Press **Offsets (F1)**.
3. Highlight Offset (row 1 to 99).
4. Press an axis key (**X** or **Z**).
5. Enter 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** 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.

Changing Fixture Offsets Using Calibrate Keys

To change offset using the calibrate keys:

1. Move the machine to the desired position.
2. Highlight the offset.
3. Press **CalibX**.

You will be prompted to enter a part diameter in the entry field that pops up.

4. Enter the part diameter in the entry field, and press **ENTER**.
5. Press **CalibZ**. The value will be entered to make the present location Z0.

Modal Corner Rounding/Chamfering (G59)

Corner Rounding Format: G59 Rn

Chamfer Format: G59 En

Refer to **Table 4-14**.

Table 4-14, G59 Address Words

Address Word	Description
G59	Modal corner rounding/chamfering command.
R	Radius of arc used to round off each corner.
E	Chamfer distance used to round off each corner.

Use Modal Corner Rounding/Chamfering (G59) to round corners (blend radii) or to chamfer intersections of line-to-line, arc-to-arc and line-to-arc moves. Apply G59 to any consecutive moves for nontangent lines and arcs. You can change the blend radius or chamfer between intersections.

Use G59 on inside or outside diameter moves. The command can be used with Tool Nose Radius (TNR) Compensation. When using TNR and G59 together, make sure that the blend radius is greater than the tool radius.

The modal G59 remains in effect until canceled with a G60 code. G59 automatically initiates linear interpolation (G1).

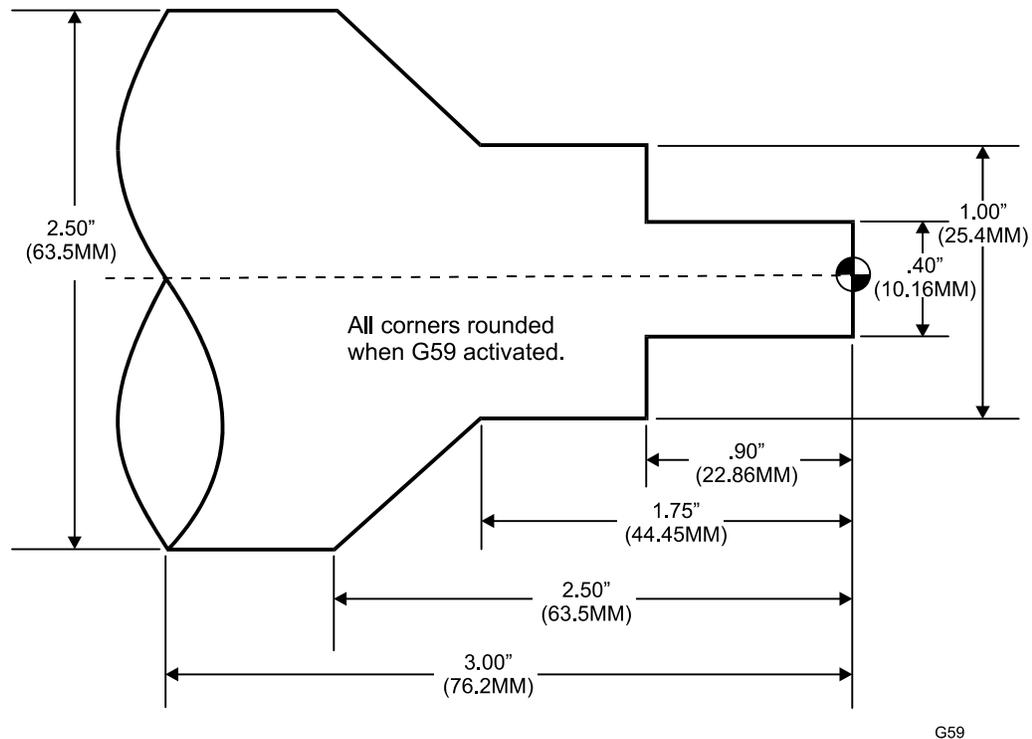


Figure 4-8, G59 Part Program

Refer to **Figure 4-8**. The diagram illustrates the part programmed in **Table 4-15**. In this example, G59 is used to command modal corner rounding. Whenever the CNC encounters an intersection between non-tangent line-to-line, arc-to-arc or line-to-arc moves, it rounds off the intersection to the specified radius.

Table 4-15, G59 Part Program (Inch Mode)

N20	G90 G0 T3	Set Absolute/Rapid. Activate tool #3.
N25	G41 X0.4 Z0.1	Rapid to start point and turn on Tool Nose Radius compensation.
N30	G59 R0.1	Turn on G59, set radius value R.
N35	G1 Z-0.9 G95 F.006	Feed to Z-0.9 at 0.006 Inch Per Revolution.
N40	X1.0	Feed to X1.0.
N45	Z-1.75	Feed to Z-1.75.
N50	X2.5 Z-2.5	Feed to X2.5 Z-2.5.
N55	G60 Z-3.0	Feed to Z-3, turn off corner rounding.
N60	G40 G0 X2.7	Turn off compensation, rapid to X2.7.
N65	X0 Z0 T0	Rapid to machine zero.
N70	M2	Program end.

To change the radius between intersections, refer to the previous program. If you wanted to change the radius of the corner rounding of N45 to R0.25, you would need to insert a G59 command (G59 R0.25) between lines N40 and N45. Refer to **Table 4-16**.

Type the G59 command for modal corner rounding or chamfer. Include the new arc or chamfer radius. Insert the command one move before the new radius applies.

Table 4-16, G59, Changing Radius Value (Inch Mode)

N30	G59 R0.1	Turn on G59, set radius value R.
N35	G1 Z-0.9 G95 F.006	Feed to Z-0.9 at 0.006 Inch Per Revolution.
N40	X1.0	Feed to X1.0.
N43	G59 R0.25	Round corner to radius 0.25.
N45	Z-1.75	Feed to Z-1.75 (Apply Corner Rounding R.1 from N30, active G59 command.)
N50	X2.5 Z-2.5	Feed to X2.5 Z-2.5 (Apply Corner Rounding R0.25, commanded in N43).

Cancel Modal Corner Rounding (G60)

Format: G60
Cancels Modal Corner Rounding/Chamfering. Next intersection will not be rounded off/chamfered.

Format: G60 Xn Zn
Cancels Modal Corner Rounding/Chamfering. Move (intersection) programmed in the same line as G60 will be rounded off/chamfered. Following intersections will not be rounded off/chamfered.

G60 cancels G59.

Use the Edit Help Menus and Graphic screens to program a G60 command.

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

Refer to **Table 4-17**.

Table 4-17, G61 and Associated G-Codes

Code	Format	Action
G9	G9 Xx.x Zx.x	Activates non-modal In-Position Mode. Complete stop only in this block.
G61	G61 Xx.x Zx.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.

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

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

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.

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

Contouring Mode (Cutting Mode) (G64)

Refer to **Table 4-18**.

Table 4-18, G64 and Associated G-Codes

Code	Format	Action
G9	G9 Xx.x Zx.x	Activates non-modal In-Position Mode. Complete stop only in this block.
G64	G64	Activates the Contouring Mode (also called Continuous Path Mode). The CNC approaches target and comes within the Continuous Path Tolerance of the target position. No in-position check is made.
G61	G61 Xx.x Zx.x	Cancels G64 and activates the In-Position Mode.

Format: G64

NOTE: Rapid moves always execute in In-Position Mode.

The Contouring Mode (G64), also known as Continuous Path Mode, or Cutting Mode, is active at power ON. 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. (During an in-position check, the CNC comes to a stop within the in-position tolerance of the target position.) This enables smooth contouring of a profile or surface.

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 (G1) or circular interpolation (G2, G3).

Using Macros (G65, G66, G67)

NOTE: Before using macros, you must learn how variables and parameters are used in a program or subprogram. Refer to [“Section 16 - Advanced Programming Features”](#) for an explanation of these features.

A macro is a group of instructions stored in memory and called by the main program. Macros are sophisticated, flexible subprograms which can be modal (G66) or non-modal (G65). Refer **Table 4-19** for g-codes related to macro calling.

Table 4-19, Macro G-Codes

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

Table 4-20 lists and describes the Address Words and M-codes used with macros.

Table 4-20, Macro Address Words

Address Word	Format	Description
Pn	Pxxxx	Used in M98 command. Macro label number (O) called.
Ln	Lxxxx	Optional repeat command. Specify number of times macro should repeat (1 to 9999).
On	N(block Number) Oxxxx	Macro label used in first line of macro for identification.
M98	M98 Pxxxx	Call macro xxxx.
M99	M99	End macro and return to line following M98.

A subprogram consists of fixed dimensions, but a macro contains variables and parameters that can change every time the macro is used.

Macros can be stored in the same file as the main program or in a separate file. Macros stored in a separate file must be called using the File Inclusion feature.

Macros stored in the same file as the main program are defined in the same way as a subprogram (Oxxxx). Terminate the macro with M99. Enter the macro after the main program. Refer to **Table 4-21**.

Refer to "[Section 16 - Advanced Programming Features](#)" for information on File Inclusion and passing parameters.

Table 4-21, Macro Program Listing

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 (M98) Macro call in the main program.

Refer to **Table 4-22**. Use the M98 Macro call to call a macro from the main body of the program.

Table 4-22, Macro Call in Main Program

Program Block	Description
N40	
N50 M98 P201	CNC executes macro O201 once, at present location.
N60	After it executes the macro (M99 encountered), the CNC returns to the main program and performs the next programmed command.

The CNC executes the called macro (201) at Block 50 (no programmed loops). At the M99 command, the CNC returns to the next line of the main program (N60).

Inch Mode (G70) or MM Mode (G71)

Inch Mode Format: G70

MM Mode Format: G71

You can change the unit of measurement display via the Inch Mode (G70) or MM Mode (G71) commands. G70 switches the CNC to Inch Mode. Use G71 to switch to MM Mode within a program. Inch/MM Mode is usually specified at the start of a program. Refer to **Table 4-23**.

Table 4-23, Activating the Inch/MM Mode

Block	Format	Description
N2	G70 G90 G0	Activates Absolute Mode and Inch Mode. Activates Rapid Traverse.
N2	G71 G90 G0	Activates Absolute Mode and MM Mode. Activates Rapid Traverse.

All blocks following N2 will be in Inch (MM) Mode.

Absolute Mode (G90) or Incremental Mode (G91)

Absolute Mode Format: G90

Incremental Mode Format: G91

In Absolute programming, all dimensions are referenced to the current Part Zero (Program Zero). In Incremental programming, all dimensions are referenced to the current tool position.

When you turn on the CNC, the Absolute or Incremental Mode specified by the machine builder is active. Absolute Mode (G90) is usually the default.

Absolute vs. Incremental Programming

In Absolute Mode (G90), all dimensions are referenced to Absolute Zero. In Incremental Mode (G91), all dimensions are referenced to the current machine position. G90 and G91 are modal and remain in effect until canceled.

Refer to [Figure 4-9, Absolute vs. Incremental Programming](#) and [Table 4-24, Absolute vs. Incremental Programming Examples](#). The table lists the Absolute and Incremental position coordinates for the diagram.

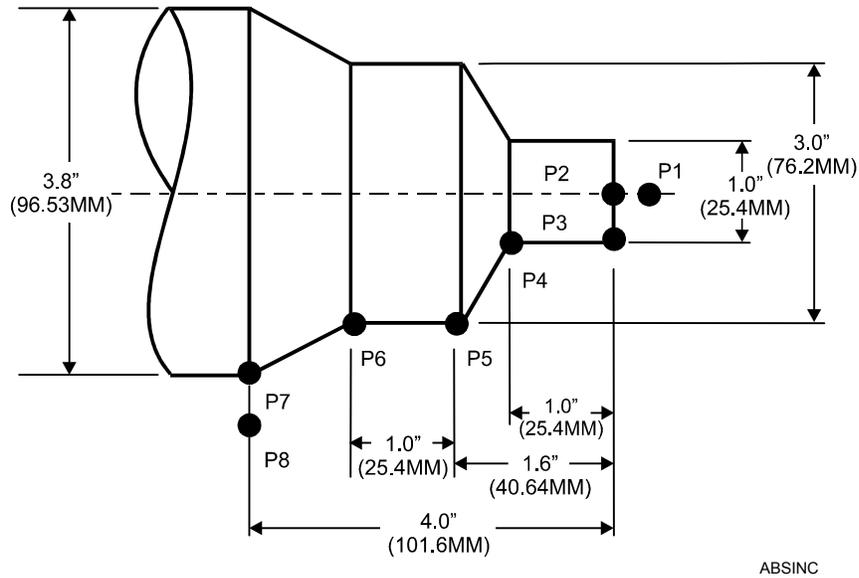


Figure 4-9, Absolute vs. Incremental Programming

Table 4-24, Absolute vs. Incremental Programming Examples

	Absolute		Incremental	
	X	Z	X	Z
Point 1	X0	Z.1	X0	Z.1
Point 2	X0	Z0	X0	Z-.1
Point 3	X1	Z0	X1.0	Z0
Point 4	X1	Z-1	X0	Z-1
Point 5	X3	Z-1.6	X2	Z-.6
Point 6	X3	Z-2.6	X0	Z-1
Point 7	X3.8	Z-4	X.8	Z-1.4
Point 8	X4	Z-4	X.2	Z0

Presetting the Axes (G92)

Format: G92 Xn Zn

X is the distance in X from the tool tip to Part Zero (usually a diameter value). Z is the distance in Z from the tool tip to Part Zero. These distances are measured with the tool sitting at Machine Home.

Refer to **Table 4-25**. The Absolute Zero Preset (G92) code resets axes to X0, Z0 or to any other pre-determined position. On a G92 command, the CNC resets the absolute position displays for X and Z. This adjusts the axes' displays to the actual position of the tool to be used.

Table 4-25, Absolute Zero Preset (G92) Listing

Program Example	Description
G92 X0 Z0	Sets the current tool position as Part Zero. The axes' position displays change to read X0 and Z0.
G92 X2 Z4	Sets Part Zero 2 inches in X and 4 inches in Z from the tool tip. The axes' position displays read X2 and Z4. NOTE: Make sure the tool is resting at Machine Home.

Part Zero (Program Zero) is usually the front face, centerline of a part. All programmed moves are referenced from Part Zero. When it is used to set Part Zero (X0, Z0), the G92 command tells the control the distance from Part Zero to the tip of the tool. You have to measure this distance, referenced to Machine Home, in X and Z prior to programming G92. The measured distances become part of the G92 command.

Before you complete the measurements, place the tool at Machine Home to ensure that the tool is in the appropriate spot. If you do not position the tool properly, the measurements will be incorrect.

NOTE: There are three ways to establish Part Zero on a workpiece for a particular tool: Tool Length Offsets (T-codes), work coordinate offsets (G53), or Absolute Zero Presets (G92). T-codes are the best way to set Part Zero. Refer to ["Section 9 - Tool Page and Tool Management"](#) for information on how to set Part Zero with Tool Length Offsets.

Refer to **Table 4-26**. The example program moves the tool to Machine Home and changes the current tool position (Machine Home) to X4.0000 and Z6.0000.

Table 4-26, G92, Presetting Axes' Position Display

N01 O2001 *SAMPLE-G92	Program # and name.
N02 G28 XZ	Homes X and Z-axes.
N03 G92 X4 Z6	Sets current position to X4, Z6.

NOTE: Anilam recommends that you do not use G92 to set Part Zero. If the machine loses power, this reference point will be lost.

Cutting Feedrates in Feed per Minute (G94)

G94: (FPM) Inch or MM per minute feedrate programming.

Refer to **Table 4-27**. Feedrates programmed in Feed Per Minute (FPM) (G94) are either in Inch (G70) or MM (G71) Mode. In FPM Mode, the moves are made at a specified rate per minute. The G-code is followed by an F-code that designates the corresponding feed rate.

Table 4-27, FPM (G94) Inch and MM Formats

Mode	Format	Feedrate
Inch Mode (G70)	G94 F1	1.0 Inch/Min.
Millimeter (G71)	G94 F1	1.0 MM/Min.

When programming lathes and turning centers, feedrates are usually programmed in Inch Per Revolution (G95).

NOTE: In all modes of operation, if the spindle is OFF, G94 must be active to move the axes with any one of the following modals: G1, G2, G3, G5.

NOTE: If jogging or using the Handwheel, G94 and G95 are ignored and the Manual Panel settings defined by the builder are used.

Cutting Feedrates in Feed Per Revolution (G95)

Format: G95
(FPR) Inch or MM per revolution feedrate programming.

Refer to **Table 4-28**. Feedrate Per Revolution (FPR) (G95) is the default feedrate most commonly used in lathes and turning centers. In FPR Mode, moves are programmed in reference to the distance moved per revolution of the spindle. In order to use Feed Per Revolution, a spindle encoder (with a Zero Marker (Index Pulse)) must be installed on the machine.

Table 4-28, Inch and MM Feedrates

Mode	Format	Feedrate
Inch Mode (G70)	F 0.008	0.008 In./Rev.
Millimeter (G71)	F 0.20	0.20 MM/Rev.

Maximum Feed Values

The maximum available feedrate (for G94 and G95) is set by the machine builder and cannot be exceeded.

Setting Spindle Speed to Constant Surface Speed (G96)

CAUTION: Program a G24 Sxxxx (Maximum Spindle RPM) prior to G96. This prevents the RPM of the spindle from reaching maximum speed when a move to the centerline (X0) is programmed.

CAUTION: Do not use CSS when threading. This operation requires the threading tool to be able to retrace its motion over and over again, at changing diameters, with each pass.

NOTE: The capability to use this feature is determined by the builder.

Format: G96 Sxxxx
Constant Surface Spindle Speed (CSS)

Most turning operations are performed in CSS. In the Constant Surface Speed (CSS) Mode (G96), the CNC adjusts the spindle RPM to maintain a constant surface speed. The S Word specifies the surface feet per minute (or, in meters per minute if in metric mode) at which the spindle rotates. To find the actual RPM, use the following formula:

RPM = 3.8 X (Surface feet per minute divided by the diameter)

Spindle speed is inversely proportional to the distance from the part centerline. If the calculated RPM exceeds the machine's maximum RPM, the CNC generates an error message.

Setting Spindle Speed to Constant Revolutions Per Minute Speed (G97)

Format: G97 Sxxxxx
Revolutions Per Minute (RPM)

In Revolutions per Minute (RPM) Mode (G97), the CNC rotates the spindle at a constant rate. The S-Word provides the rate. If the entered RPM exceeds the machine's maximum RPM for the active gear range, the CNC generates an error message.

Threading passes and center cutting operations (tapping, drilling and reaming) must be performed in RPM Mode. Most other turning operations are performed in Constant Surface Speed (G96) Mode.

When the program is run and the CNC encounters the G97 command, it activates the Revolutions per Minute Mode. The CNC will remain in RPM Mode until the command is changed or canceled.

Section 5 - Canned Cycles

Programming a Z-Axis Turning/Boring Cycle with a Defined Profile (G73)

Format G73 Wn An Rn Sn Cn Bn Pn In Jn Kn

Refer to **Table 5-1**. The table lists the Address Words that define the G73 (longitudinal) and G74 (radial) area clearance cycles.

Table 5-1, G73 and G74 Address Words

Address Word	Description
W	Input Subprogram Number. Subprogram number in which the part profile is programmed. Store in the same file as the main program or use shape created in the shape editor. Required.
A	Depth per Pass. Depth of material removed per rough pass (per side). Required.
R	X Finish Stock. Amount of material left in X after roughing cycle is finished. Removed during finish pass. Optional.
S	Z Finish Stock. Amount of material left in Z after roughing cycle is finished. Removed during finish pass. Optional.
C	Finish pass indicator. 0 (no finish pass) or 1 (a finish contour pass will be completed after the roughing cycle). Required.
B	OD=1, ID=0: 1 (completes passes along the Z-axis, away from the centerline) or 0 (completes passes along the Z-axis, toward the centerline). Required.
P	Undercut. Use value of 1 if undercuts are present in the profile to be rough/finish cut. Leave the parameter unassigned if no undercuts are present. Optional.
I	X In Feed. Plunge Feedrate. (G94=Z In Feed) Optional.
J	Rough feed. Feedrate for the roughing cycle. Optional.
K	Finish feed. Feedrate for the finish cycle. Optional.

Refer to [Figure 5-1, G73 Turning/Boring with Defined Profiles](#). G73 rough machines a workpiece into a given profile, along the Z-axis (longitudinal). The profile shape is defined by a subprogram. The subprogram number is called in the cycle using the parameter W.

The subprogram profile is programmed after the end of the main program and uses the letter O. The profile program can contain lines, arcs and angles. You can use the shape editor. An optional finish pass removes the finish stock amount defined by parameters R (X) and S (Z).

NOTE: The X ending point of the shape should be the largest diameter point for OD and smallest diameter point for ID on the shape.

A different feedrate can be programmed for rough (J) and finish (K) passes. Parameter B determines whether the roughing cycle is an outer diameter profile (B = 1) or an inner diameter profile (B = 0). The finish pass is taken directly along the part profile.

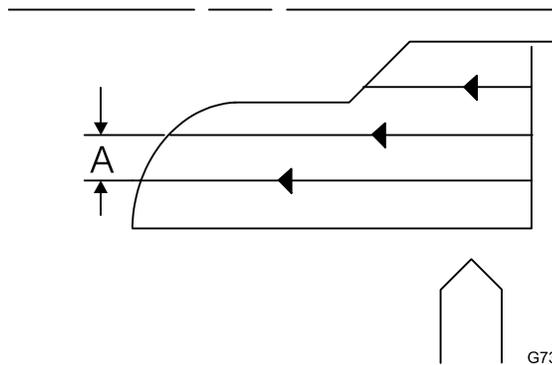


Figure 5-1, G73 Turning/Boring with Defined Profiles

Refer to **Figure 5-2**. The diagram shows the part programmed in [Table 5-2, G73 Part Program](#). The program demonstrates how to use G73 to perform a longitudinal Area Clearance (turning/boring) cycle.

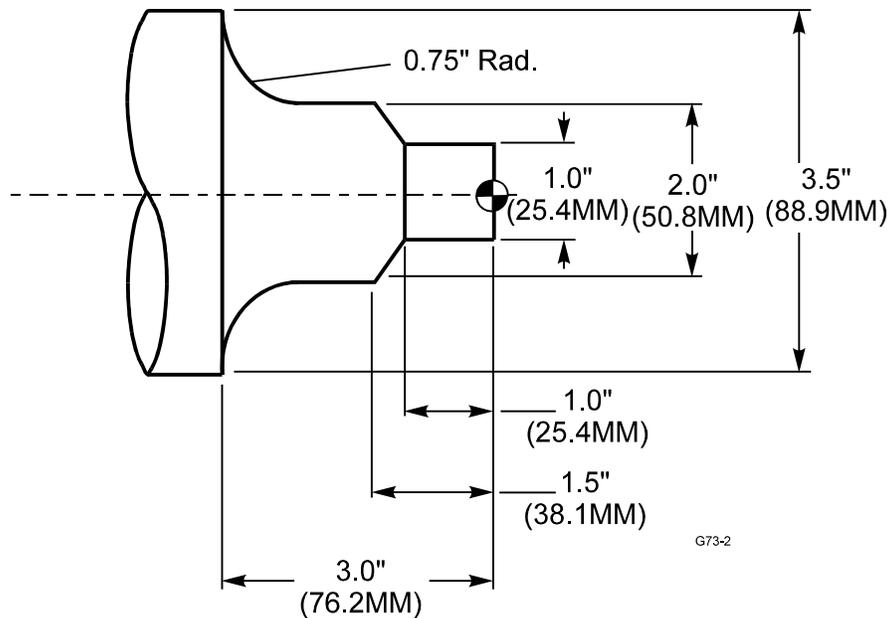


Figure 5-2, G73 Part Program

Table 5-2, G73 Part Program

Program Block	Description
N1 G70 G90 G0 X0 Z0 T0	Sets Inch/MM Mode and Absolute Mode. Places machine at X0,Z0 with no tool offsets active.
N2 X3.7 (93.9) Z0 T1	Activates T1, rapids to X start position (.2" standoff).
N3 G1 X-.03 (-1) G95 F.008 (0.2)	Faces to X-0.03" (X-1mm) past centerline at 0.008" (.2mm) per revolution.
N4 G0 X3.7 (93.9) Z.1 (2.0)	Rapids to start position of Area Clearance Cycle.
N5 G73 W9 A.2 (5) R.02 (.5) S.005 (.1) C1 B1 J.02 (.5) K.01 (.25)	Sets parameters for G73 and activates cycle.
N6 T0 X0 Z0	Cancels T1 offsets, rapids to home position.
N7 M2	Ends Program, returns to block N1.
N8	Blank for easy distinction between Main and Sub-programs.
The following subprogram describes the part profile called into the main program above. The CNC calculates actual moves based on the profile described in the subroutine.	
N10 O9 (Subprogram)	Establishes name of Subprogram (O9).
N11 G0 X1 (X25.4) Z.1 (2)	Rapids to start point of profile.
N12 Z-1 (-25.4)	Feeds to Z-1" (Z-25.4mm).
N13 X2 (50.8) Z-1.5 (-38.1)	Feeds up angle to X2 Z-1.5 (X50.8 Z-38.1).
N14 Z-2.25 (-57.15)	Feeds to start of radius.
N15 G3 X3.5 (88.9) Z-3 (-76.2) R.75 (19.05)	Feeds around radius CCW.
N16 G1 X3.7 (93.9)	Feeds away from part diameter to clear the part.
N17 M99	Ends Subprogram.

NOTE: All variables are sign (+/-) dependent. Moves can be defined as Outer Diameter/Inner Diameter, and toward or away from the chuck or part centerline. The positive/negative direction (sign of the variable) determines what kind of move is made by the CNC. Ensure that the appropriate sign (+/-) is programmed.

Programming an X-Axis Facing/Boring Cycle with Defined Profile (G74)

Format G74 Wn An Rn Sn Cn Bn Pn In Jn Kn

Refer to [Table 5-1, G73 and G74 Address Words](#). The table lists the Address Words that define the G73 (longitudinal) and G74 (radial) Area Clearance Cycles.

Refer to **Figure 5-3**. The G74 Area Clearance cycle rough machines a workpiece into a given profile, along the X-axis (radial). The tool must be positioned properly before the cycle is executed.

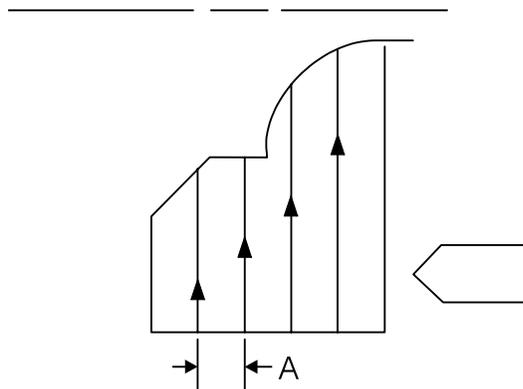


Figure 5-3, G74 Facing/Boring Cycle with Defined Profile

The profile shape is defined by a subprogram. The subprogram profile is programmed after the end of the main program and uses the letter O. The profile program contains lines, arcs and angles. Use the shape editor. An optional finish pass removes the finish stock amount given by parameter R (X) or S (Z).

A different feedrate can be programmed for rough (J) and finish (K) passes. The B address word determines whether the roughing cycle passes are toward (B = 1) or away from (B = 0) the part centerline. The finish pass is taken directly along the part profile.

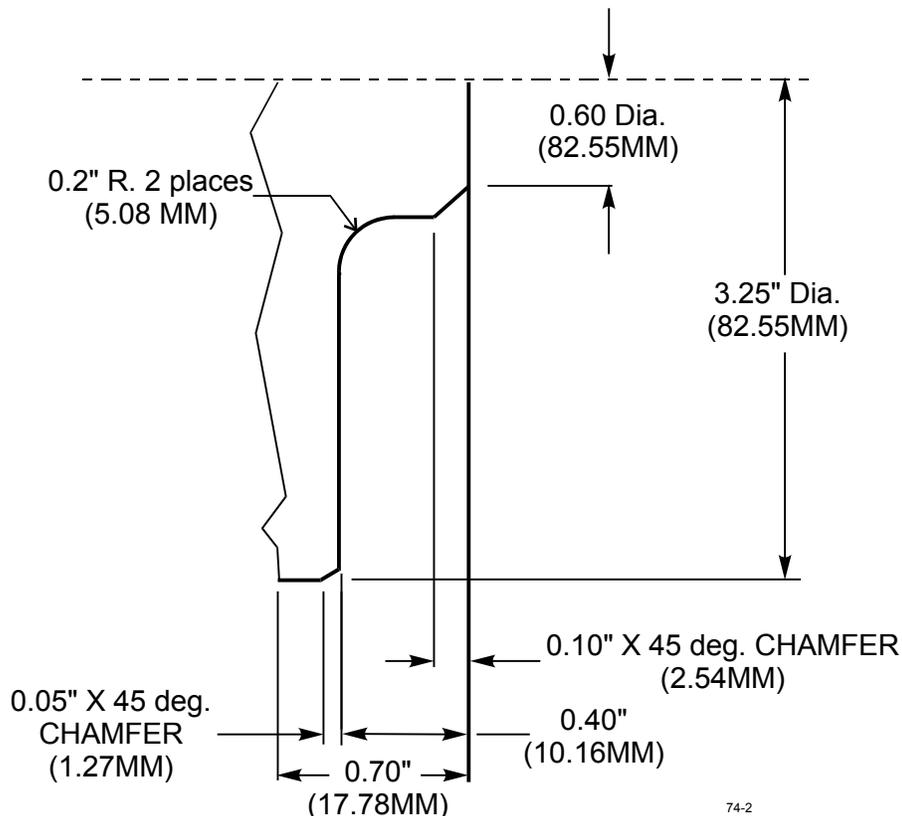


Figure 5-4, G74 Part Program

Refer to **Figure 5-4**. The diagram shows the part programmed in **Table 5-3**. The program demonstrates how to use G74 to perform a radial Area Clearance (facing/boring) cycle.

Table 5-3, G74 Part Program

Part Program	Description
N1 G70 (G71) G90 G0 X0 Z0 T0	Sets Inch/MM and Absolute Modes. Sends tool to Machine Home (X0,Z0) and cancels tool offsets.
N2 X3.45 (87.63) Z0 T1	Activates T1, rapids to start position of Area Clearance Cycle.
N3 G74 W8 A.05 (1.28) R.02 (.508) S.005 (0.127) C1 B1 J.02 (0.508) K.01 (0.254)	Sets parameters for G74 and activates cycle.
N4 G0 X0 Z0 T0	Cancels T1 offsets, rapids to Machine Home.
N5 M2	Ends Program, returns to block N1.
N6	Blank for easy distinction between Main and Subprograms.

The following subprogram describes the part profile called into the main program above. The CNC calculates actual moves based on the profile described in the subroutine.	
N7 O8 (Subprogram)	Establishes name of Subprogram (O8).
N8 G0 X3.45 (87.63) Z-0.45 (-11.43)	Rapids to start point of profile.
N9 G1 X3.25 (82.55)	Feeds to X position.
N10 X3.15 (80.01) Z-0.4 (-10.16)	Feeds down chamfer to X2 Z-1.5 (X80.01 Z-11.43).
N11 X1.2 (30.48)	Faces down to start of radius.
N12 G2 X.8 (20.32) Z-0.2 (-5.08) R.2 (5.08)	Feeds around radius CW.
N13 G1 Z-0.1 (-2.54)	Feeds to start of 2nd chamfer Z-0.1" (Z-2.54).
N14 X0.6 (15.24) Z0	Feeds down chamfer.
N15 M99	Ends Subprogram.

NOTE: All variables are sign (+/-) dependent. Moves can be defined as Outer Diameter/Inner Diameter, and toward or away from the chuck or part centerline. The positive/negative direction (sign of the variable) determines what kind of move is made by the CNC. Make sure to program the appropriate sign (+/-).

G81 Address Words Refer to **Figure 5-5**. The G76 turning cycle removes stock material along the Z-axis, from the outer diameter of the work. The tool must be positioned appropriately before the canned cycle begins. B determines the standoff position in X from which the tool advances to the start of the turning cycle.

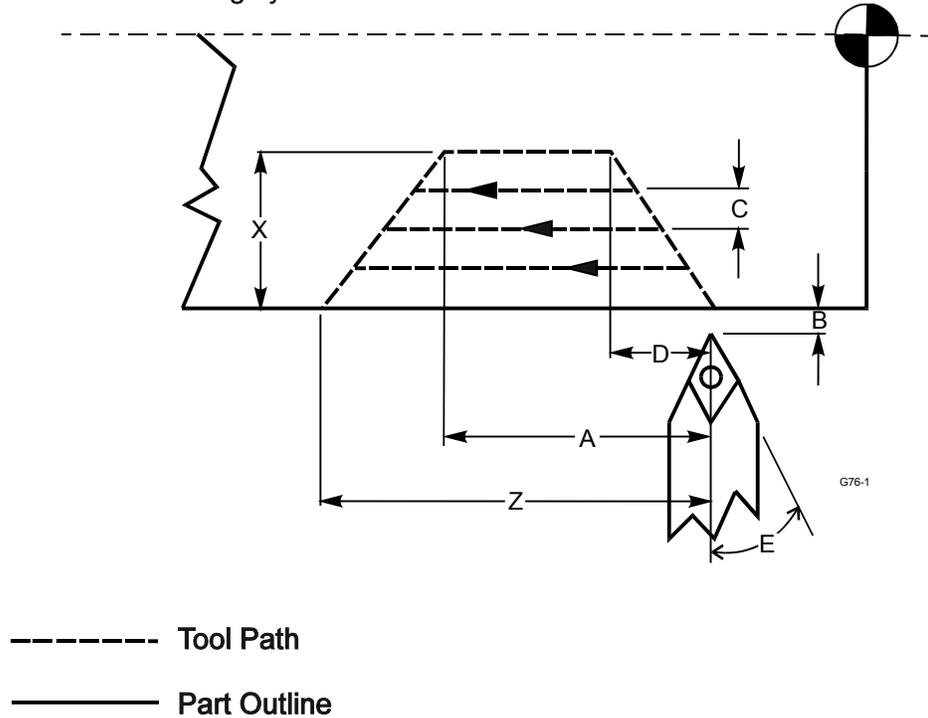


Figure 5-5, G76 Z-Axis Roughing Cycle

The CNC calculates the depth of each pass and the number of passes necessary to reach finish depth (X). The actual depth per pass will not exceed the programmed maximum depth (C). The CNC advances the tool along at the calculated depth per pass, and at the desired profile (angular or rectangular). The rear angle of the cutting tool (E) cannot exceed the slope of the finished cut (X divided by D).

Once the cycle is completed, the CNC returns to the G76 starting coordinates. If you want a finish pass, you must program it separately.

Programming an X-Axis Radial (Facing) Roughing Cycle (G77)

Format: G77 Xn Zn Cn An Bn Dn En

Refer to **Table 5-5** for a list and description of the Address Words used to define G77.

Table 5-5, G77 Address Words

Address Word	Description
X	Incremental X distance from start to finish (length). Required.
Z	Incremental Z distance from start to end diameter (excludes B). Required.
C	Maximum incremental depth per cut. Required.
A	Incremental X distance from start to final pullout. Optional.
B	Incremental Z standoff distance from face. Default: O.D. 0.1" or 2MM. Optional.
D	Incremental X distance from start to final pull-in. Optional.
E	Incremental rear angle of tool, referenced to the positive Z direction (Z-axis to trailing edge of tool). Must be a positive value. Optional.

Refer to **Figure 5-6**. The G77 radial facing cycle removes stock material along the X-axis. The tool must be positioned appropriately before the canned cycle begins.

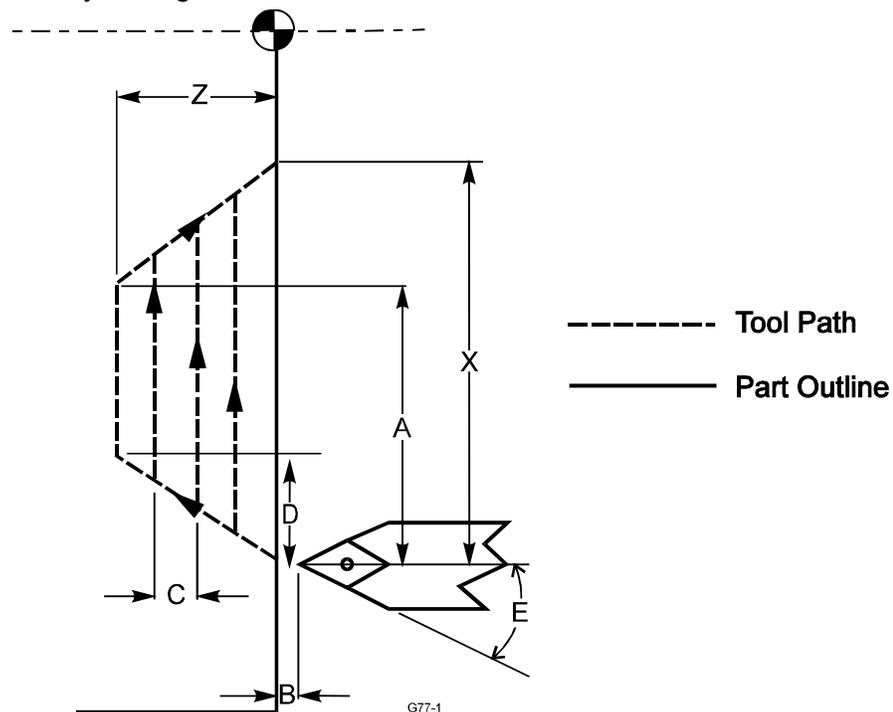


Figure 5-6, G77 Radial Roughing/Boring Cycle

G77 can be used for I.D. or O.D cuts. B determines the standoff position in Z, from which the tool advances to the start of the facing cycle.

The CNC calculates the depth of each pass and the number of passes necessary to reach finish depth (Z). The actual depth per pass will not exceed the programmed maximum depth (C). The CNC advances the tool along at the calculated depth per pass, and the desired profile (angular or rectangular). The rear angle of the cutting tool (E) cannot exceed the slope of the finished cut.

No compensation can be performed during the G77 canned cycle (including Tool Nose Radius Compensation). Once the cycle is completed, the CNC returns to the starting coordinates. If a finish pass is desired, it must be programmed separately.

Refer to **Figure 5-7**. The diagram shows how G77 can alternately be used for inner (boring) and outer (roughing) diameter cutting.

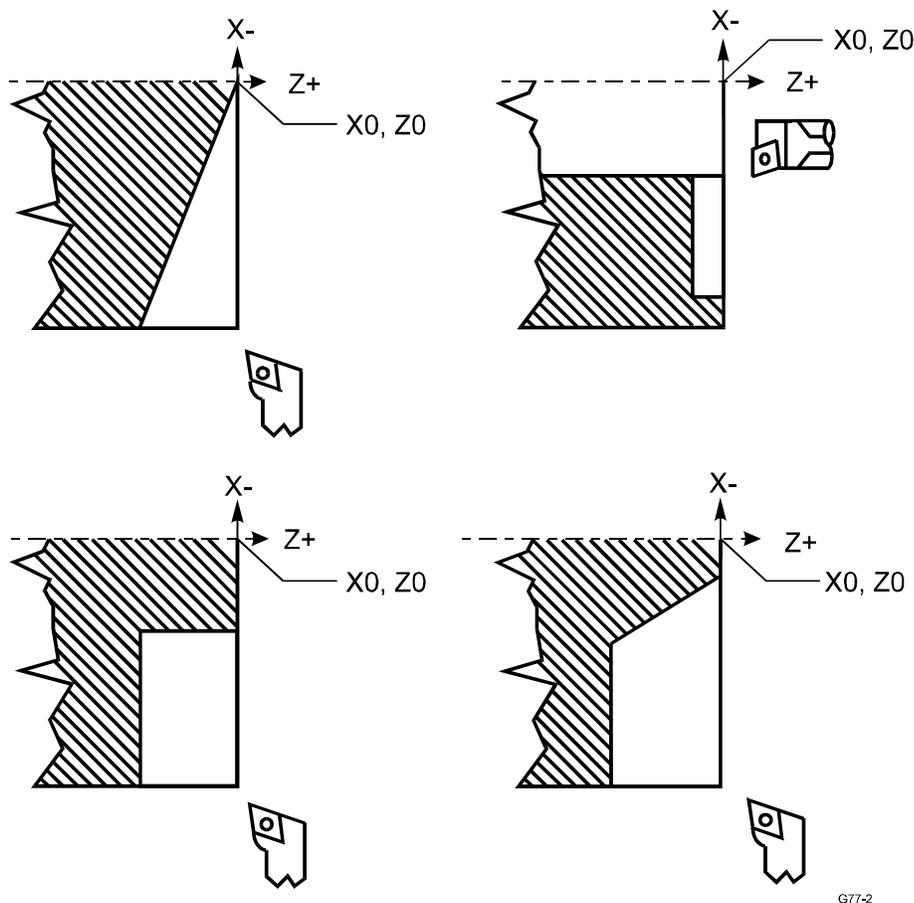


Figure 5-7, G77 Roughing and Boring Cycles

NOTE: All variables are sign (+/-) dependent. Moves can be defined as Outer Diameter/Inner Diameter, and toward or away from the chuck or part centerline. The positive/negative direction (sign of the variable) determines what kind of move is made by the CNC. Make sure that the appropriate sign (+/-) is programmed.

Programming a Pecking Cycle (G78)

Format G78 Zn Rn Fn In Pn

Refer to **Table 5-6** for a list and description of the Address Words used to define a Pecking Cycle (G78).

Table 5-6, G78 Address Words

Address Word	Description
Z	Absolute hole depth in Z. Optional.
R	Distance from starting position to surface to be drilled, in Z. (Default: .10"/2MM). Optional.
F	Feedrate. Optional.
I	Maximum Peck distance. Required.
P	Retract distance from surface of work to end position in Z. Point to which tool retracts after hole depth is reached. Optional.

Refer to **Figure 5-8** for a description of the peck drilling cycle (G78). G78 is used for drilling holes along the Z-axis. No motion occurs along the X-axis.

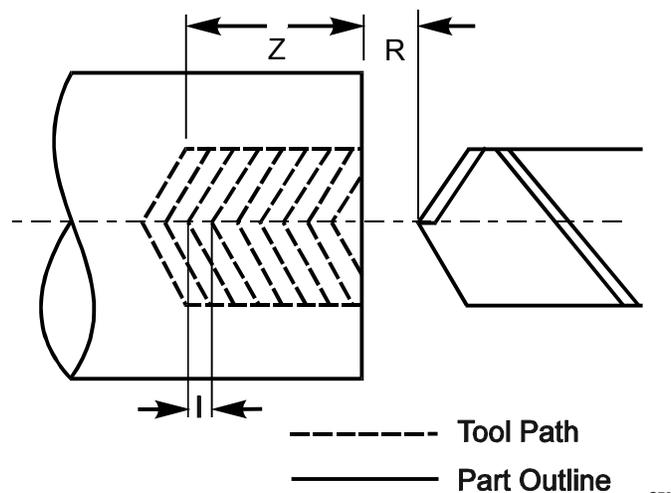


Figure 5-8, G78 Pecking Cycle

Before running the cycle, you must position the tool at the start position.

The CNC feeds from the Z start position (R) to the specified hole depth (Z), at a given feedrate (F). The number of pecks and depth per peck depend on the hole depth (Z) and number of pecks necessary to complete the cycle. After the peck drilling cycle is complete, the CNC rapids to the return height (P), before proceeding to the next program block.

NOTE: Drilling cycles G78 (Peck Drilling) and G79 (Chip Break Drilling) require the Z start (R) to be 0.10"/2MM from the surface to be drilled. The Z return (P) is optional. If P is not given, the CNC returns to the Z start position at the end of the cycle.

Programming a Chip Break Cycle (G79)

Format G79 Zn Rn Fn In Jn Kn Wn Un Pn

Refer to **Table 5-7** for a list and description of the Address Words used to define a Chip Break Cycle (G79).

Table 5-7, G79 Address Words

Address Word	Description
Z	Absolute hole depth in Z. Required.
R	Starting position of cycle in Z. Required.
F	Feedrate. Optional.
I	Z depth of first peck (positive dimension). Required.
J	Amount to subtract from subsequent pecks (I-J) (positive dimension). Required.
K	Minimum peck distance (positive dimension). Required.
W	Chip break increment (positive dimension). Optional.
U	Incremental depth between full retracts (positive dimension). Optional.
P	Point to which tool retracts after hole depth is reached. Optional.

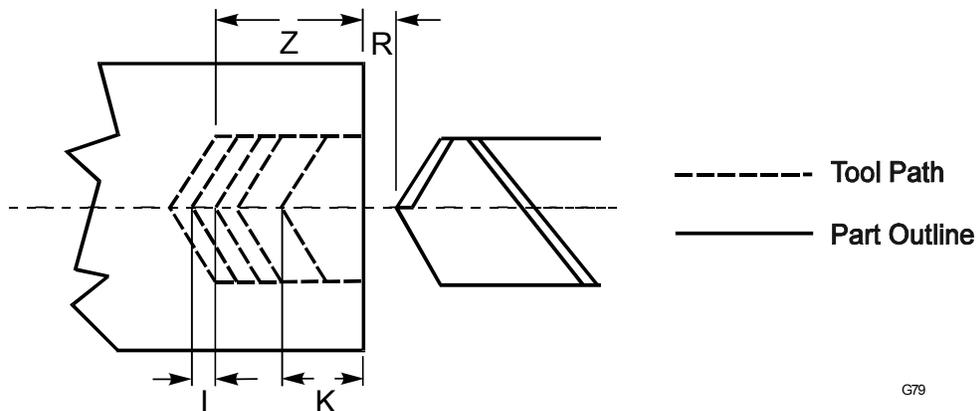


Figure 5-9, G79 Chip Break Cycle

Refer to **Figure 5-9**. G79 is the chip-break peck-drilling cycle, generally used for peck drilling medium to deep holes. G79 allows you to tailor Address Words to the material and tool types used in deep hole drilling.

During the chip break cycle, the CNC feeds from the starting position to the first peck depth. Before commencing the next peck, the CNC rapid retracts the tool by the chip break increment (W). Then, the tool advances to the next calculated peck depth (I minus J). This procedure continues until the tool reaches retract depth (U), or until final hole depth is reached. At the end of the cycle, the CNC advances to the next programming block.

The peck distance will never be more than I or less than K.

To prevent binding of the chips, tool, and workpiece in deep-hole drilling, as the depth increases, the peck distance (J) decreases. The retract depth (U) variable allows the programmer to fully retract the tool from the hole, at specified incremental depths.

Programming a Longitudinal Grooving Cycle (G81)

Format G81 Zn Xn Wn Bn Dn

Refer to **Table 5-8** for a list and description of the Address Words used to define a Longitudinal Grooving Cycle (G81).

Table 5-8, G81 Address Words

Address Word	Description
Z	Total incremental Z dimension of groove (length). Required.
X	Incremental X distance (depth) from start radius to end radius (excludes B). Required.
W	Tool width. Required.
B	Incremental X standoff distance from diameter (0.1 in/2mm outer diameter; -0.1in/2mm inner diameter). Optional.
D	Dwell time at bottom of groove, in seconds. Optional.

Refer to **Figure 5-10**. G81 creates a groove along the longitudinal (Z) axis. Use a groove-cutting tool for this cycle. The cycle has built-in compensation for tool width. Position the tool appropriately (with the tool width taken into consideration) before the canned cycle begins.

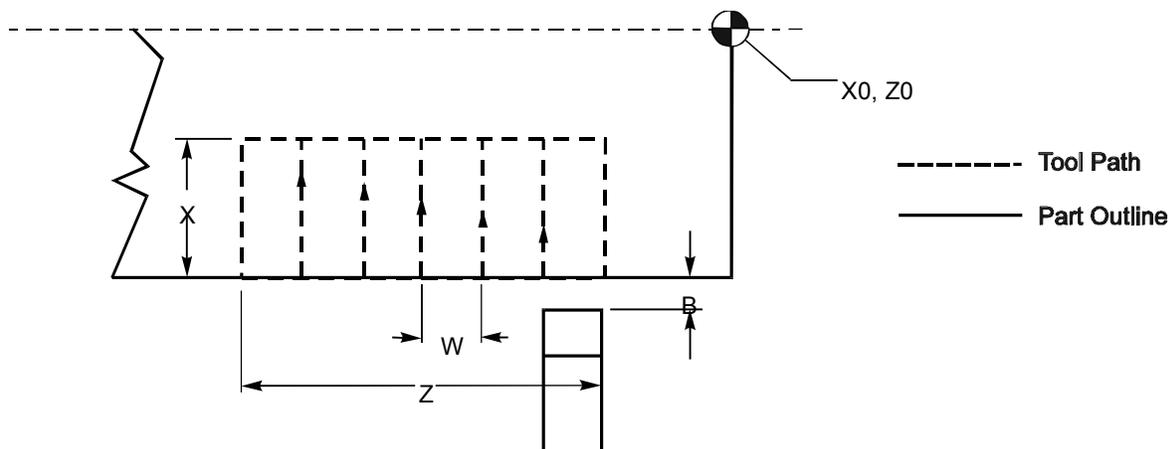


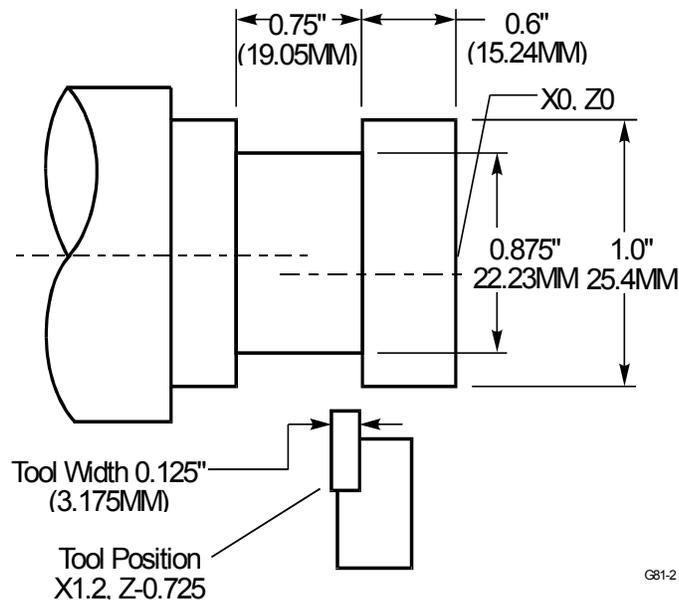
Figure 5-10, G81 Address Words

Use G81 for inner diameter or outer diameter cuts. B determines the standoff position in X, from which the tool advances to the start of the grooving cycle.

After the tool reaches total depth (X), it remains at the bottom of the groove until the dwell time (D, optional) elapses. The CNC calculates the number and width of passes from the total length (Z) required and the tool width (W). After the cycle terminates, the CNC returns to the starting coordinates of the canned cycle.

NOTE: All variables are sign (+/-) dependent. Moves can be defined as Outer Diameter/Inner Diameter, and toward or away from the chuck or part centerline. The positive/negative direction (sign of the variable) determines what kind of move is made by the CNC. Make sure to program the appropriate sign (+/-).

Refer to **Figure 5-11**. The diagram shows the part programmed in [Table 5-9, G81 Programming Example](#). Metric equivalents are given in parentheses.



G81-2

Figure 5-11, G79 Chip Break Cycle

Table 5-9, G81 Programming Example

Program Block	Description
N1 O7 *GROOVY	Establishes program name and number.
N2 G90 G70 G0 T0 X0 Z0 M5	Sets Absolute/Inch/Rapid Modes. Cancels tool offsets and positions tool at X0, Z0.
N3 T1 *.125 (3.175) WIDE GROOVE INSERT	Activates tool length offsets for Tool 1. Contains a comment regarding tool width.
N4 X1.2 (30.48) Z-.725 (-18.415) M3 S980	Positions axes, turns on spindle and sets spindle speed to 980.
N5 G95 F.0012 (.03)	Sets feedrate.
N6 G81 Z-.75 (-19.05) X-.0625 (-1.588) W.125 (3.175) B.1(2)	Defines and activates G81 canned cycle.
N7 G0 T0 X0 Z0 M5	Cancels tool offset, positions axes at Machine Home and turns off the spindle.
N8 M2	End of program. Return to N1.

Programming a Radial Grooving Cycle (G82)

Format G82 Zn Xn Wn Bn Dn

Refer to **Table 5-10** for a list and description of the Address Words used to define a Radial Grooving Cycle (G82).

Table 5-10, G82 Address Words

Address Word	Description
X	Incremental X distance (length) from start radius to end radius (excludes B). Required.
Z	Total incremental Z dimension of groove (depth). Required.
W	Tool width. Required.
B	Incremental X standoff distance from diameter (0.1 in/2mm outer diameter; -0.1in/2mm inner diameter). Optional.
D	Dwell time at bottom of groove, in seconds. Optional.

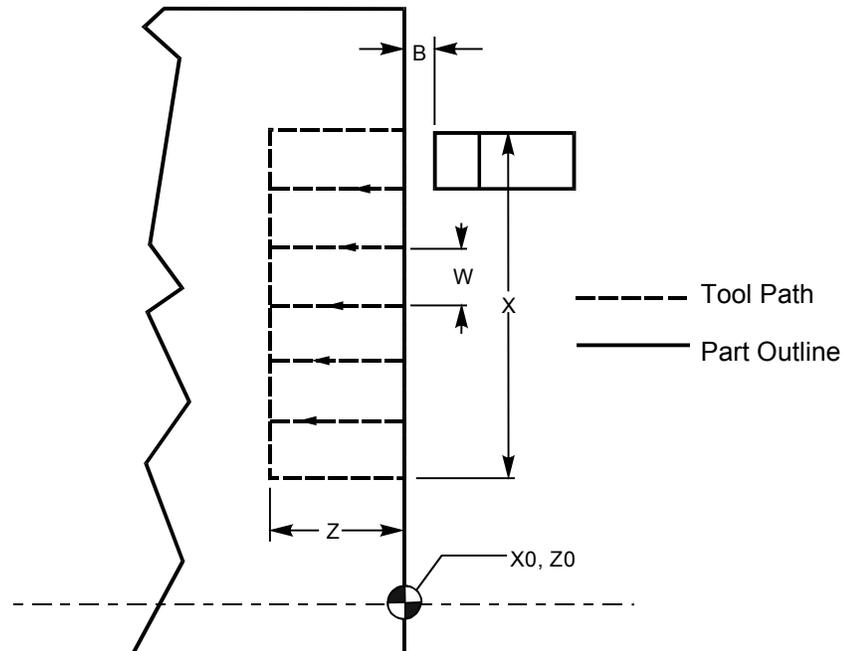


Figure 5-12, G82 Address Words

Refer to **Figure 5-12**. The G82 canned cycle creates a groove along the radial (X) axis. Use a groove-cutting tool for this cycle. The cycle has built-in compensation for tool width. Position the tool appropriately before the cycle begins.

B determines the standoff position in Z, from which the tool advances to the start of the grooving cycle. G82 can be used for I.D. or O.D cuts.

After the tool reaches total depth (Z), it remains at the bottom of the groove until the dwell time (D, optional) elapses. The CNC calculates the number and width of passes from the total length (X) required and the tool width (W). After the cycle terminates, the CNC returns to the starting coordinates of the cycle.

NOTE: All variables are sign (+/-) dependent. Moves can be defined as Outer Diameter/Inner Diameter, and toward or away from the chuck or part centerline. The positive/negative direction (sign of the variable) determines what kind of move is made by the CNC. Make sure to program the appropriate sign (+/-).

Types of Threading

Refer to **Table 5-11**. The following terms will help explain the threading cycles given in subsequent sections.

Table 5-11, Types of Threading

Threading	Description
Compound Threading	The tool enters the work on an angle (A) specified by the programmer.
Plunge Cut Threading	The tool enters the work "straight on" with no thread angle (A) specified.
Uni-directional Threading	The tool cuts only along its leading edge.
Bi-directional Threading	The tool alternates cuts along its leading and trailing edges.
Longitudinal Threading	Threading passes occur along the Z-axis.
Radial Threading	Threading passes occur along the X-axis.

Cutting Longitudinal Threads with Canned Cycles (G83, Uni-directional)

Format G83 En Zn Cn Dn An Sn Rn Xn Vn Bn Wn

Refer to **Table 5-12** for a list and description of the Address Words used to define a Uni-directional Longitudinal Threading Cycle (G83).

Table 5-12, G83 Address Words

Address Word	Description
E	Threads per inch. Used in Inch programming.
F	Pitch (single pass threading) or amount of travel in Z per revolution (multi-pass threading). Used in MM programming. NOTE: Use E or F, but not both.
Z	Incremental distance in the Z-axis from Tool start position to finish point (length of threading pass).
C	Incremental depth of the first cut (depth of cut per side for first pass). Does not include B parameter.
D	Incremental depth of the thread (depth of thread per side). Does not include B parameter. Default: Pitch x 0.6134 (60 degree thread).
A	Thread angle, 60 degrees for standard threads. Default: 0 degrees for a straight plunge thread.
S	Number of passes at final depth. Default: 1.
R	Incremental Z-axis pullout distance. Creates a 2-axis pullout move. Default: 0 causes the tool to pullout at 90 degrees to the threading pass.
X	Distance moved in the X-axis during a threading pass. Default: 0 straight thread.
V	Angle of tapered thread. Default: 0 NOTE: When programming a tapered thread, use X or V parameter, but not both.
B	Distance in the X-axis from the tool tip to the work diameter per side. Default: Inch = 0.1. Outside diameter default: Inch = -0.1. Inside diameter default: MM = 2. Outside diameter default: MM = -2 Inside diameter. Tool must be positioned the standoff distance away from the diameter in the block prior to the threading canned cycle.
W	Used to program the number of starts in a multiple start thread. Default: 1 (single start thread).

Refer to **Figure 5-13**. The figure describes the motion of the CNC during a G83 canned cycle. The G83 canned cycle creates a longitudinal thread along the Z-axis, utilizing multiple-pass threading. G83 supports an X-axis taper distance or angle.

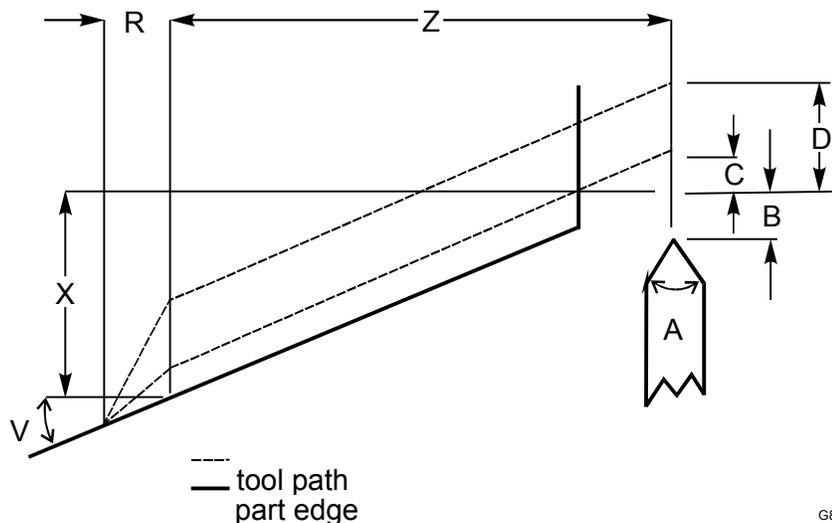


Figure 5-13, G83 Uni-direction Compound or Plunge Thread Cutting

NOTE: All variables are sign (+/-) dependent. Moves can be defined as Outer Diameter/Inner Diameter, and toward or away from the chuck or part centerline. The positive/negative direction (sign of the variable) determines what kind of move is made by the CNC. Make sure to program the appropriate sign (+/-).

Cutting Longitudinal Threads with Canned Cycles (G84, Bi-directional)

Format G84 En (Fn) Zn Cn Dn An Sn Rn Bn Wn

Refer to **Table 5-13** for a list and description of the Address Words used to define a Bi-directional Longitudinal Threading Cycle (G84).

Table 5-13, G84 Address Words

Address Word	Description
E	Threads per inch. Used in Inch programming.
F	Pitch (single pass threading) or amount of travel in Z per revolution (multi-pass threading). Used in MM programming. NOTE: Use E or F, but not both.
Z	Incremental distance in the Z-axis from Tool start position to finish point (length of threading pass).
C	Incremental depth of the first cut (depth of cut per side for first pass). (Does not include B parameter).
D	Incremental depth of the thread (depth of thread per side). (Does not include B parameter.) Default: Pitch x 0.6134 (60 degree thread).
A	Thread angle, 60 degrees for standard threads. Default: 0 degrees for a straight plunge thread.
S	Number of passes at final depth. Default: 1.
R	Incremental Z-axis pullout distance. Creates a 2-axis pullout move. Default: 0 causes the tool to pullout at 90 degrees to the threading pass.
B	Distance in the X-axis from the tool tip to the work diameter per side. Default: Inch = 0.1 Outside Diameter Default: Inch = -0.1 Inside Diameter Default: MM = 2 Outside Diameter Default: MM = -2 Inside Diameter Tool must be positioned the standoff distance away from the diameter in the block prior to the threading canned cycle.
W	Used only in multiple pass threading, the number of staggered starts used to reach depth. Default: 1. NOTE: Specifying the number of starts changes the way the CNC interprets all other threading variables.

Refer to **Figure 5-14**. The figure describes the motion of the CNC during a G84 canned cycle. The G84 canned cycle creates a longitudinal thread along the Z-axis, utilizing multiple-pass threading. G84 does not support an X-axis taper distance or angle, but allows bi-directional threading.

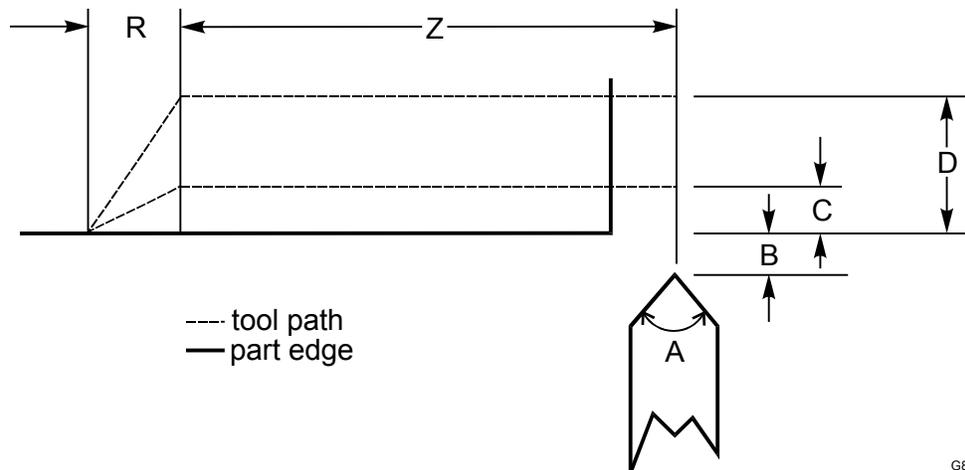


Figure 5-14, G84 Bi-directional Compound Threading

NOTE: All variables are sign (+/-) dependent. Moves can be defined as Outer Diameter/Inner Diameter, and toward or away from the chuck or part centerline. The positive/negative direction (sign of the variable) determines what kind of move is made by the CNC. Make sure to program the appropriate sign (+/-).

G84 Programming Example

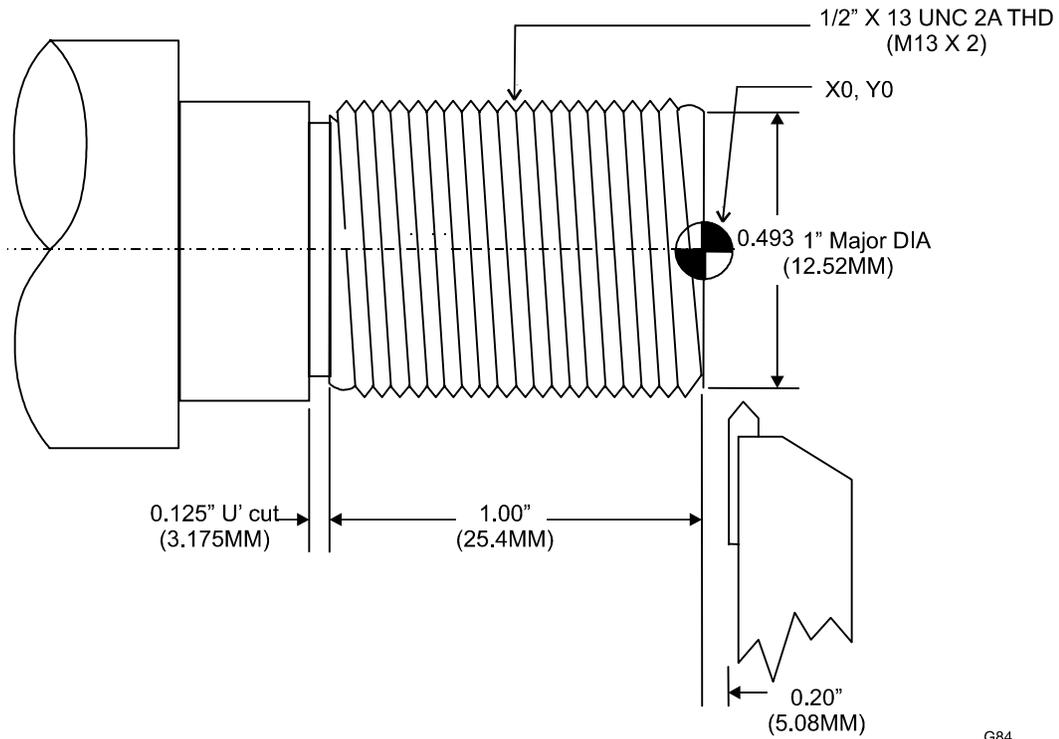


Figure 5-15, G84 Programming Example

Refer to **Figure 5-15** for a diagram of the part programmed in **Table 5-14**. The Listing provides a sample Program Listing for a G84 threading canned cycle. Metric equivalents are provided in parentheses following the inch measurements.

Table 5-14, G84 Program Listing

Program Block	CNC Response
G0 X0.7 (17) Z0.2 (5.08) T3	Positions tool to starting point, activates Tool #3 offsets.
N27 G84 E13 (F2) Z-1.2625 (32.068) C-.015 (-0.4)	Completes programmed threading pass. Threading Cycle E (F)= Threads Per Inch or pitch/mm (metric) Z = Incremental Thread Length C = 1st pass depth of cut
N28 X0 Z0 T0	Cancels offsets, rapids home.

Cutting Radial Threads with Canned Cycles (G85, Uni-directional)

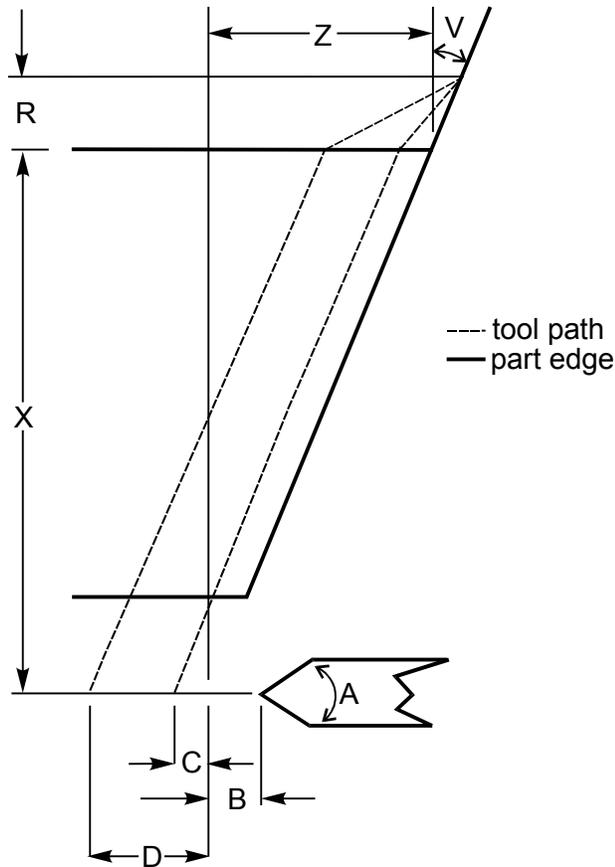
Format: G85 En (Fn) Xn Cn Dn An Sn Rn Zn Vn (Bn) Wn

Refer to **Table 5-15** for a listing and description of Address Words associated with the G85 command.

Table 5-15, G85 Address Words

Address Word	Description
E	Threads per inch. Used in Inch programming.
F	Pitch (single pass threading) or amount of travel in X per revolution (multi-pass threading). Used in MM programming. NOTE: Use E or F, but not both.
X	Incremental distance in the X-axis from Tool start position to finish point (length of threading pass).
C	Incremental depth of the first cut (depth of cut for first pass). (Does not include B parameter.)
D	Incremental depth of the thread (depth of thread). (Does not include B parameter) Default: Pitch x 0.6134 (60 degree thread).
A	Thread angle, usually 60 degrees. Default: 0 degrees for a straight plunge thread.
S	Number of passes at final depth. Default: 1.
R	Incremental X-axis pullout distance. Creates a 2-axis pullout move. Default: 0, causes the tool to pullout at 90 degrees to the threading pass.
Z	Distance moved in the Z-axis during a threading pass. Default: 0 straight thread.
V	Angle of tapered thread. Default: 0. NOTE: When programming a tapered thread use Z or V parameters, but not both.
B	Distance in the Z-axis from the tool tip to the work face. Default: Inch = 0.1 Default: MM = 2. (Tool must be positioned the Standoff distance away from the face in the block prior to the threading canned cycle.)
W	Used to program the number of starts in a multiple start thread. Default: 1 (single start thread).

Refer to [Figure 5-16. G85 Canned Cycle](#). The figure describes the motion of the CNC during a G85, multiple-pass threading cycle. During a G85 cycle, a plunge or compound thread is cut along the X-axis (part face). The programmer must position the tool before running the cycle. After the cycle is executed, the tool returns to the starting coordinates for the cycle.



G85

Figure 5-16, G85 Canned Cycle

NOTE: All variables are sign (+/-) dependent. Moves can be defined as Outer Diameter/Inner Diameter, and toward or away from the chuck or part centerline. The positive/negative direction (sign of the variable) determines what kind of move is made by the CNC. Make sure to program the appropriate sign (+/-).

G85 Radial (Face) Plunge Thread Example

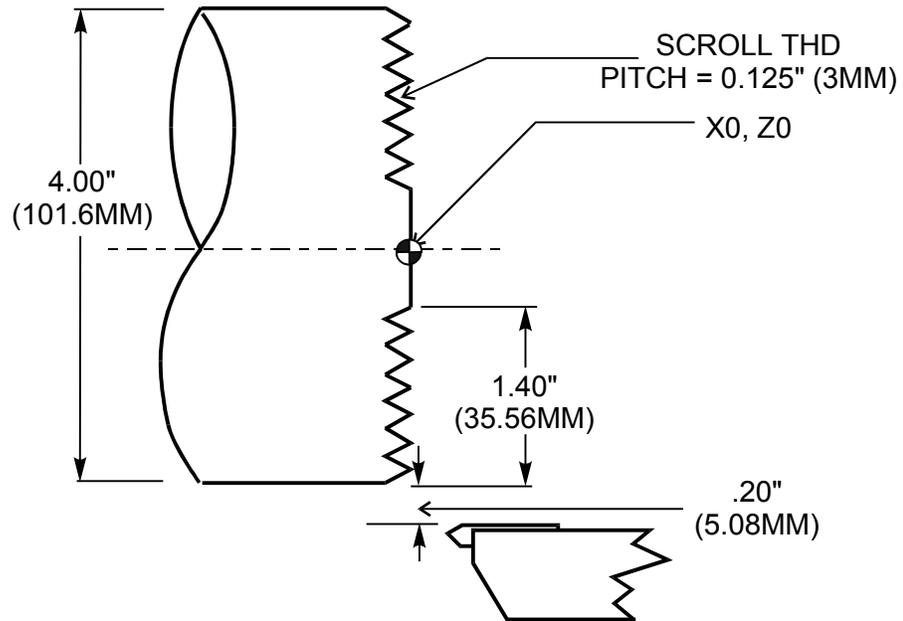


Figure 5-17, Radial (Face) Threading Example

Refer to **Figure 5-17**. The figure shows a part diagram for a G85 cycle. Refer to **Table 5-16** for the Program Listing. Metric equivalents are provided in parentheses.

Table 5-16, G85 Program Listing

Program Block	Description
N26 G0 X4.4 (111.76) Z0.1 (2) T5	Position Tool to Start Point.
N27 G85 E.125 (F3) X-1.5 (-37.64) C-.025 (-.635) R-.1 (-.3)	G85 Threading Cycle E = Pitch F = Lead (MM) X = Incremental Thread Length C = 1st pass depth of cut R = Incremental pull-out distance
N28 X0 Z0 T0	Cancel Offsets, Rapids Home.

Cutting Radial Threads with Canned Cycles (G86, Bi-directional)

Format G86 En (Fn) Xn Cn Dn An Sn Rn Bn Wn

Refer to **Table 5-17** for a listing and description of the Address Words associated with the G86 canned cycle.

Table 5-17, G86 Address Words

Address Word	Description
E	Threads per inch. Used in Inch programming.
F	Pitch (single pass threading) or amount of travel in X per revolution (multi-pass threading). Used in MM programming. NOTE: Use E or F, but not both.
X	Incremental distance in the X-axis from tool start position to finish point (length of threading pass).
C	Incremental depth of the first cut (depth of cut for first pass). (Does not include B parameter).
D	Incremental depth of the thread (depth of thread). (Does not include B parameter) Default: Pitch x 0.6134 (60 degree thread).
A	Thread angle, 60 degrees for standard threads. Default: 0 degrees for a straight plunge thread.
S	Number of passes at final depth. Default: 1.
R	Incremental X-axis pullout distance. Creates a 2-axis pullout move. Default: 0 causes the tool to pullout at 90 degrees to the threading pass.
B	Distance in the Z-axis from the tool tip to the work face. Default: Inch = 0.1 Default: MM = 2 Tool must be positioned the Standoff distance away from the face in the block prior to the threading canned cycle.
W	Used only in multiple pass threading, the number of staggered starts used to reach depth. Default: 1. NOTE: Specifying number of starts changes the way the CNC interprets

Refer to **Figure 5-18**. The diagram describes the motion of the CNC during the G86, multiple-pass thread cutting. In Radial Bi-directional threading (G86), passes are performed along the X-axis (face). G86 is designed for compound threading. The operator must position the tool before running the G86 canned cycle. After the cycle is executed, the tool returns to the starting coordinates for the canned cycle.

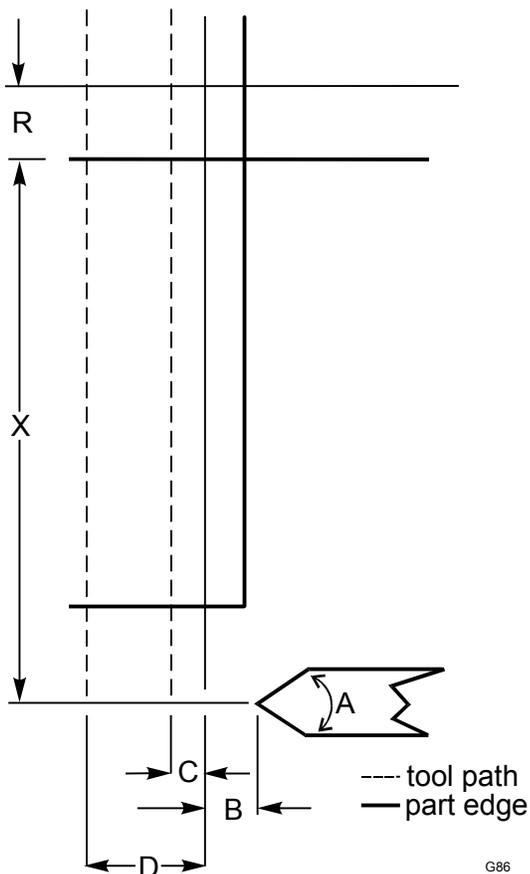


Figure 5-18, G86 Canned Cycle

NOTE: All variables are sign (+/-) dependent. Moves can be defined as Outer Diameter/Inner Diameter, and toward or away from the chuck or part centerline. The positive/negative direction (sign of the variable) determines what kind of move is made by the CNC. Make sure to program the appropriate sign (+/-).

Tapping a Hole with a Canned Cycle (G87)

NOTE: The machine must support spindle M-codes (spindle forward, spindle reverse and spindle off) in order to use this cycle.

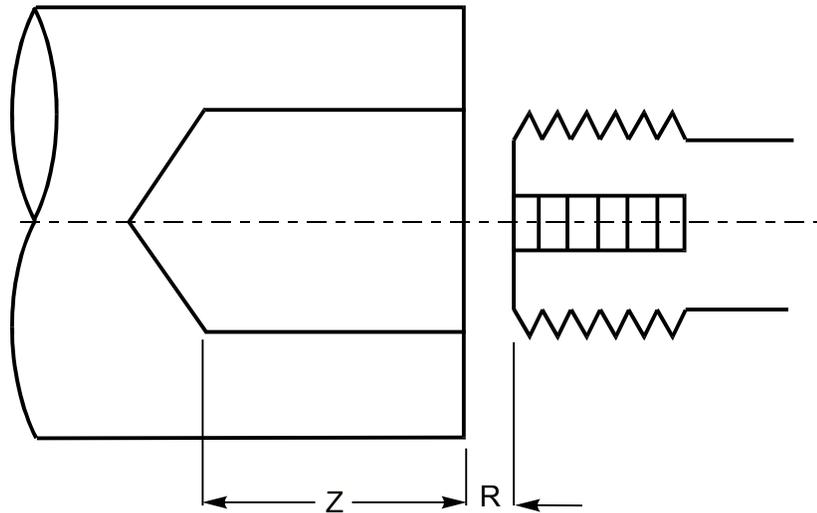
Format G87 Zn Rn Fn Pn Dn

Refer to **Table 5-18** for a listing and description of the Address Words associated with the G87 canned cycle.

Table 5-18, G87 Address Words

Address Word	Description
Z	Absolute hole depth
R	Initial Z start point (CNC rapids to R)
F	Threads per Inch (TPI) in Inch mode, or Lead (Distance between threads) in MM mode.
P	Alternate Z return point after hole depth (CNC feeds to P, if specified after completing the cycle). If no P value is given, the CNC returns to the starting point (R) at the conclusion of the cycle.
D	Dwell time. 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. If "D" is set, the control will dwell at the start and bottom of the hole.

Refer to **Figure 5-19**. The diagram describes the Address Words used to define a G87 Tapping Cycle. G87 is used for tapping holes. The CNC feeds from the retract plane to the depth of the hole in Z. The spindle stops and reverses motion, and retracts the tool from the hole. The CNC returns the tool to the starting position (R, or to alternate finish position, P) reversing spindle motion before advancing to the next program block.



G87

Figure 5-19, G87 Tapping Cycle

NOTE: Not all machines contain the spindle encoder necessary for Inch Per Revolution (G95) programming.

All variables are sign (+/-) dependent. Moves can be defined as Outer Diameter/Inner Diameter, and toward or away from the chuck or part centerline. The positive/negative direction (sign of the variable) determines what kind of move is made by the CNC. Make sure to program the appropriate sign (+/-).

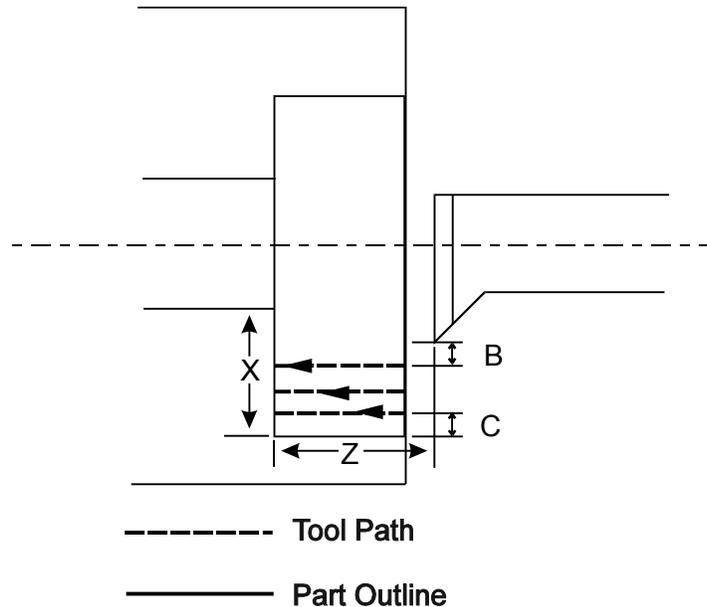
Programming a Boring Cycle (G88)**Format** G88 Zn Xn Cn An Bn

Refer to **Table 5-19** for a listing and description of the Address Words associated with the G88 canned cycle.

Table 5-19, G88 Address Words

Address Word	Description
Z	Incremental Z distance from start to finish (length). Required.
X	Incremental X distance from start to end diameter (excludes B). Required.
C	Maximum incremental depth per cut. Required.
A	Incremental Z distance from start to final pull-out. Optional.
B	Incremental X standoff distance from the diameter. Default: 0.1inch/2mm for O.D. and -0.1inch/-2mm for I.D. If no value provided, the CNC assumes the default. Optional.

Refer to **Figure 5-20**. The G88 boring cycle removes stock material along the Z-axis, from the inner diameter of the work. The tool must be positioned appropriately before the canned cycle begins. Perform boring cycles only on a workpiece that has been pre-drilled. B determines the standoff position in X, from which the tool advances to the start of the boring cycle.



G76-3

Figure 5-20, G88 Boring Cycle

The CNC calculates the depth of each pass and the number of passes necessary to reach finish depth (Z). The actual depth per pass will not exceed the programmed maximum depth (C). The CNC advances the tool along at the calculated depth per pass.

Once the cycle is completed, the CNC returns to the starting coordinates for the canned cycle. If a finish pass is desired, it must be programmed separately.

Lathe Tool Probe Cycles (Option)

This document describes operation and an overview of the lathe tool probe canned cycles available on the 4200T CNC product. Probing is an option on the 4200T CNC. The cycles provided perform the most common tool probing functions. Custom cycles to perform specific functions can be written using the **G31** primitive and parametric programming. Refer to "[Section 16, Probe Move \(G31\)](#)" for more details. If Probing has been added post-sale, besides Setup Utility changes, there may be Integral Programmable Intelligence (IPI) program modifications required.

The lathe tool probe cycles are only supported on machines with homing with a permanent X and Z machine position. The method described assumes the use of negative tool-length offsets. In this method, the Tool-Length Offset (TLO) for each tool represents the distance from the tool tip at machine home to the centerline of the spindle in X and to a fixed known point in Z such as the spindle nose or chuck face. This method requires the use of a Z work coordinate offset.

Tool probing does not allow scaling to be active. If scaling is on before calling the canned cycle, it will be deactivated when the cycle is complete.

IMPORTANT: Check with machine builder for operation of probing hardware.

Probes in lathes typically have an arm. If the arm is automatic, an M-code is commonly used to move the arm down and another M-code is used to move the arm up. The machine builder or probe installer will provide these details.

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
- Macro Variable and System Variable Settings
- Detailed description of all tool probe cycles

The lathe 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 probing system parameter setup information, refer to [4200T CNC Setup Utility Manual, P/N 70000414](#), and "[Macro Variable and System Variable Settings](#)."

Tool Probe G-code Cycle Designations

Before running any of these probing cycles you must go into Probing Setup and set the probing setup parameters.

G150 Tool Probe Calibration Cycle

This is used to set the X and Z datum for diameter, length, and establish 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 unless the probe is moved or the Z offset reference surface is changed.

G151 Tool Preset Cycle

Preset the tool length (Z) and diameter (X) in the tool table. The probe must be calibrated before you run this cycle.

G154 Tool Wear/Breakage Cycle

Checks the tool and gives an alarm if not within tolerance.

If the tool wear is greater than allowed tolerance, then the tool must be changed. If less than the allowed tolerance, the cycle updates the wear register(s).

G146 Protected Probe Positioning Cycle

This is used to position the tool around the probe during a program for automated tool preset (**G151**) and wear adjustment (**G154**). The tool arm must be in the proper position before trying to use any of these cycles.

Macro Variable and System Variable Settings

Before you set the parameters for the tool probe you must:

- Know the size of the calibration standard.
- Ensure when entering the values in the parameters that you are in the same units (inch/metric) as when you are running the tool preset canned cycle. Set the units in the Setup Utility to the units in which you are going to run the cycles. When changing from inch to mm or from mm to inch. First change on the Setup Options screen, then go back to Setup Options, Operator Setup and Control Software, change Default units (#2) to inch or mm as required.

To set the Probe Parameters:

- On the Software Options menu, select Software Utility.
- On the Setup Options menu, select Builder Setup.
- On the Builder Setup menu, select Probing.
- On the Probing Setup menu, set the following options:
 1. Set **Inspection probe type**, this should be set to Corded, Cordless, or Cordless SG for a lathe type probe.
 2. Do not set **Inspection probe stylus diameter** – Not used.
 3. Do not set **Inspection probe standard diameter** – Not used.

NOTE: The parameter labels starting with “Inspection” do not relate to these cycles and are only present for future use.

4. Set **Tool probe stylus width**, the lathe probe width. This needs to be an exact value and is used by the cycle to calibrate the probe stylus. It is also used to position the tool properly. The probe stylus should be a perfectly square rectangle. [For example, set to 0.5906 inch (15 mm).]
5. Set **Positioning FeedRate while probing**, the feedrate the control will use while positioning the probe around the part. This is the feed-rate used to position the tool around the probe. This should be the fastest feed-rate to save time but not so fast as to jeopardize the safety of the probe. [For example, set to 70 in/min (1778 mm/min).]
6. Set **First pick feedrate**, the feedrate the control will use while making its initial touch finding the surface it is measuring. [For example, set to 20 in/min (508 mm/min).]
7. Set **Second pick feedrate**, the feedrate the control will use after making its initial touch finding the surface it is measuring. This is the feed-rate used for the final probe pick. This should be relatively slow. [For example, set to 1 in/min (25.4 mm/min).]
8. Set **Retract amount after first touch**, the amount the tool retracts from the probe stylus after the first pick. [For example, set to 0.0200” (0.5080 mm).]
9. Set **Invert probe logic** to **No** for standard Normally Open (N.O.) probe interface and to **Yes** for a Normally Closed (N.C.) probe interface. This parameter should not be changed except by a qualified probe installer or the OEM.

All defaults, i.e. **G90/91**, **G95/94**, etc., must be reset by the programmer or operator after running a probing canned cycle because they can and will be changed by the canned cycle.

Tools must be reactivated after a probing cycle is run to insure the new offset has taken effect.

Probe Orientation

Probe orientation is the position the tool is placed. **G146** may be used to position the tool. If orientation **2** is to be used, the tool would be placed at the back right corner of the stylus at the appropriate distance from X and Z. This is parameter (**D**) when required in a cycle.

See **Figure 5-21** for the tool orientations to the probe stylus.

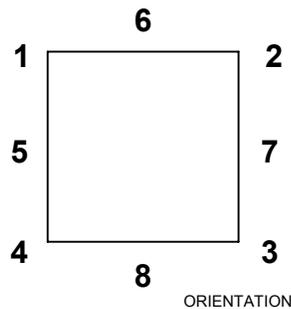


Figure 5-21, Probe Orientations

The orientation parameter (**D**) must be entered in cycles requiring this parameter. The most commonly used orientations are:

- D2** For the rear tool post turning tools
- D3** For the front tool post turning tools

Lathe Tool Probe G-code Cycle Designations

The following summarizes the cycles available:

G150 Tool Probe Calibration Cycle

The Tool probe must be calibrated before presetting a tool offset. To calibrate the tool probe you must have a calibration standard in the tool turret or holder that can touch the probe stylus on at least two adjacent sides. Once mounted in the tool turret, the calibration standard must have an offset activated to a known fixed point on the lathe relative to machine home. This is usually the chuck face and the center of spindle rotation to the edge of the calibration standard that will contact the probe stylus.

Before calling the **G150** calibration cycle, you must have that tool offset active. Next, bring the probe arm down and jog the calibration standard to within 0.2"/5mm of the probe stylus into one of the orientations listed above, preferably this should be orientation 2 or 3 so that the **G150** cycle can touch two adjacent sides of the probe stylus to calculate and store the calibration data. Example: **G150 D2**. The cycle will only touch two sides and then will calculate the other two sides of the stylus based on the width of the stylus entered probe setup utility or in the "**S**" parameter. Once the cycle has calibrated the probe stylus, it will store the data that represents the distance from the fixed point on the machine, (the chuck face and spindle center line), to each side of the probe stylus. Once that data is stored, it will then be used to calculate the tool offset when using the **G151** and **G154** cycles.

G151 Tool Preset Cycle

Bring the probe arm down and jog the tool into the orientation position. The tool that is being set must be active before calling this cycle. Execute the **G151** cycle with at least the “**D**” parameter set to the tool orientation to the probe.

Example: **G151 D 2**. This will load the X and/or Z offset into the tool table, which is the distance from machine home to the spindle center-line in X and the chuck face (or where ever you had chosen as a reference point when calibrating the tool) in Z. After setting all the tools, the operator MUST put in a **G53** offset, which is the positive distance from the chuck face in the Z-axis only, to the front face of the part. This can be done by placing the part in the chuck and calling any tool, then manually jogging the tool to the face of the part. Then write the number in the “Program” section Z register and entering that number into the Z offset register of the work coordinate you wish to use.

CAUTION: This work coordinate MUST be active at all times when machining or the Z zero position will be the face of the chuck instead of the part face causing a crash. When a new part is being setup and machined and the tools are remaining the same, the operator only needs to adjust the Z-axis work coordinate for all the tools to be properly set. It is not necessary to re-set all the tool offsets. If the tool offset needs to be somewhere other then the edge of the part that is touching the probe stylus, you will need to enter an amount into the “**C**” parameter to add a shift moving the offset over to another place on the tool such as the center of a threading tool.

G154 Tool Wear/Breakage Cycle

Bring the probe arm down and jog or use the **G146** protected probe positioning cycle to move the tool into the orientation position. The tool that is being set must be active before calling this cycle. Execute the **G154** cycle with at least the “**D**” to set the tool orientation to the probe and one or both “**I**” or “**K**” parameters.

Example: **G154 D 2 I .001 K .002** where “**I**” and “**K**” are the maximum amounts that the X and Z offsets can be off before generating an error. This will check to make sure the amount of wear for X and/or Z is not over the specified amount and update the wear registers if the “**U**” parameter is set to “**1**”. If the tool offset is somewhere other then the edge of the part that is touching the probe stylus, you will need to enter the same amount into the “**C**” parameter as was used when doing the initial tool set using the **G151** Tool Preset Cycle.

The tool must be positioned in relation to the probe stylus using the **G146** protected probe move cycle prior to calling this cycle.

G146 Protected Probe Positioning Cycle

This is intended to be used to position the tool around the probe during a program for automated tool wear adjustment (**G154**). The tool arm must be in the proper position before trying to use any of these cycles. The machine builder or probe installer will be able to tell you the codes associated to bringing the probe arm up or down if automatic.

Description of Tool Probe Cycles

This section contains detailed descriptions of the tool probe cycles:

- Establishing a Position on the Machine
- Tool Probe Calibration Cycle (**G150**)
- Tool Preset (**G151**)
- Tool Wear/Breakage Cycle (**G154**)
- Protected Probe Positioning Cycle (**G146**)

Establishing a Position on the Machine

X will normally be set to the center of the spindle.

1. Turn a part in the machine and measure the diameter and set the tool-length offset (TLO) for X in the normal way using the offset of any active tool.
2. Use the same tool for the Z-axis and **G150** (Tool Probe Calibration Cycle).

This must be completed before **G150** is run.

A nominal position must first be set before using these cycles. This can be done the same way as setting a TLO; but in this case, it is put **G53** Offset Table. The face of chuck or any convenient position along the Z-axis can be selected. The position should be a repeatable position. This position is also set in G53 Offset Table. When this is done, activate the G53 offset. You are now ready to set the TLO's.

Note after **G151** Tool preset as been run for all tools, it will be necessary to adjust **Z G53** offset to the front of part or to wherever Z zero is required to be set.

Tool Probe Calibration Cycle (G150)**Format:** G150 An Bn Sn Dn

This cycle is used to calibrate the probe. This is used to set the X and Z datums establishing the center of the probe stylus and the effective probe stylus size for setting tool diameter registers. Refer to **Table 5-20**.

Table 5-20, G150 Address Word

Address Word	Description
A	If n=1, calibrates all four sides of the stylus. When n=0, calibrates two sides. [Default: 1] (Optional)
B	Distance away from the probe stylus. [Default: 0.2" (5.08mm)] [Range: 0.01" (0.254 mm) – 1.0" (25.4 mm)] (Optional)
S	Stylus width and length, if set in the Setup Utility the value is passed from there. (Optional)
D	Tool orientation to the probe stylus. See Figure 5-21. Probe Orientations . (Required) [Range: 1 to 8]

To calibrate the tool probe:

1. Manually jog the tool to less than 0.2" (5.08 mm) of the stylus. This is the start position to calibrate the tool.
2. From the manual mode, type **G150 D(n)**, and press the **START** button. Where **D** is the tool orientation to the probe stylus.

Tool Preset (G151)**Format:** G151 Bn Cn Sn Dn

- This tool preset (**G151**) can be run from within a program or from the manual mode. Refer to **Table 5-21**.

Table 5-21, G151 Address Words

Address Word	Description
B	(Distance away) The distance away from the probe stylus. [Default: 0.2" (5.08 mm)] [Range: 0.01"/0.254mm to 1.0"/25.4mm] (Optional)
C	(Shift Amount) Offset value where the center of the tool is required to be zero. (Optional)
S	(Stylus Size) The stylus width and length, if set the Setup Utility the value is passed from there. (Optional)
D	Tool orientation to the probe stylus. See Figure 5-21, Probe Orientations . (Required) [Range: 1 to 8]

Jog machine to within 0.2" (5.08 mm) of stylus X and Z and on the tool page place the highlight on the tool to be calculated. Enter the program into the control and run in Auto.

To enter value into the offset tool page:

1. The Tool# that is active will automatically have its TLO updated.

Tool Wear/Breakage Cycle (G154)

Format: G154 Bn Cn Sn Dn In Kn Un

Refer to **Table 5-22**.

Table5-22, G154 Address Word

Address Word	Description
B	(Distance away) The distance away from the probe stylus. [Default: 0.2" (5.08 mm)] [Range: 0.01"/0.254mm to 1.0"/25.4mm] (Optional)
C	(Shift Amount) Offset value where the center of the tool is required to be zero. (Optional)
S	(Stylus Size) The stylus width and length, if set the Setup Utility the value is passed from there. (Optional)
D	Tool orientation to the probe stylus. See Figure 5-21, Probe Orientations . (Required) [Range: 1 to 8]
I	(Maximum Xwear Adjustment) Maximum X-axis wear allowed. If less than I maximum value, the system will adjust the wear offset in the tool page. If the X-axis wear is over the I maximum value, the control will alarm when the maximum limit set in I or K has been exceeded. If undefined, will not adjust X wear table. (Required – One or both I and/or K)
K	(Maximum Zwear Adjustment) Maximum Z-axis wear allowed. If less than K maximum value, the system will adjust the wear offset in the tool page. If the X-axis wear is over the K maximum value, the control will alarm when the maximum limit set in I or K has been exceeded. If undefined, will not adjust Z wear table. (Required – One or both I and/or K)
U	(1=Update, 0=No) If set to "1", it will update the wear registers and the control will alarm when the maximum limit set in I or K has been exceeded. If this is undefined or set to 0 , the G154 cycle will not update the wear registers each time it checks a tool. (Optional)

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

Jog to position as for **G151**. If **C** is used in the original setting, it should be entered at this time.

It is possible to use **G146** to position the probe before using the **G154** cycle.

Protected Probe Positioning Cycle (G146)

Format: G146 Xn Zn Fn

- ❑ When an X and Z move is programmed using the **G146** (Protected Probe Positioning Cycle), the control will stop and alarm if the probe stylus is triggered before reaching the target set in the X and Z parameters.
- ❑ Before setting this parameter, the probe should be calibrated. Jog each tool to within 0.2" (5.08mm) of the stylus and enter the numbers into the X and Z parameters.
- ❑ **G146** can be used to position tools close to the probe before using G154 (Tool Wear/Breakage Cycle). The probe must be in position before the G146 cycle is run.
- ❑ The **G146** Protected Probe Positioning Cycle can be run from within a program or manual mode. Refer to **Table 5-23**.

Table 5-23, G146 Address Words

Address Word	Description
X	X position from the original position set
Z	Z position from the original position set
F	Feedrate at which to travel to this position. (Optional)

To use the Protected Probe Positioning Cycle:

1. Type **G146 Xn Zn Fn**. If this is run from inside a program, this line needs to be repeated for every move you wish to make.
2. Execute that line in Manual or Auto by pressing **START**.

Section 6 - Program Editor

Activating the Program Editor

Program blocks are written with the Program Editor. Activate the Program Editor to put the CNC in Edit Mode. You must create program files before you can write or edit programs. Refer to "[Section 10 - Program Management](#)" for instructions on how to create new programs. You can activate the Program Editor from the Program Directory or the Manual screen.

When the Program Editor activates from the Program Directory, the highlighted program opens for editing.

When the Program Editor is activated from the Manual screen, the selected program opens for editing. All programming procedures written in this section take place in Edit Mode.

Activating the Edit Mode from the Manual Screen

To activate the Edit screen from the Manual screen:

1. Select the appropriate program and press **Edit (F3)**. The Edit screen activates and **Ins (F3)** highlights.

Activating the Edit Mode from the Program Directory

To activate the Edit screen from the Program Directory:

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

Activating the Edit Mode from Draw Graphics

To activate the Edit screen from Draw Graphics, press **Edit (F2)**. The Edit screen activates and **Ins (F3)** highlights. See [Figure 6-1, Edit Screen](#).

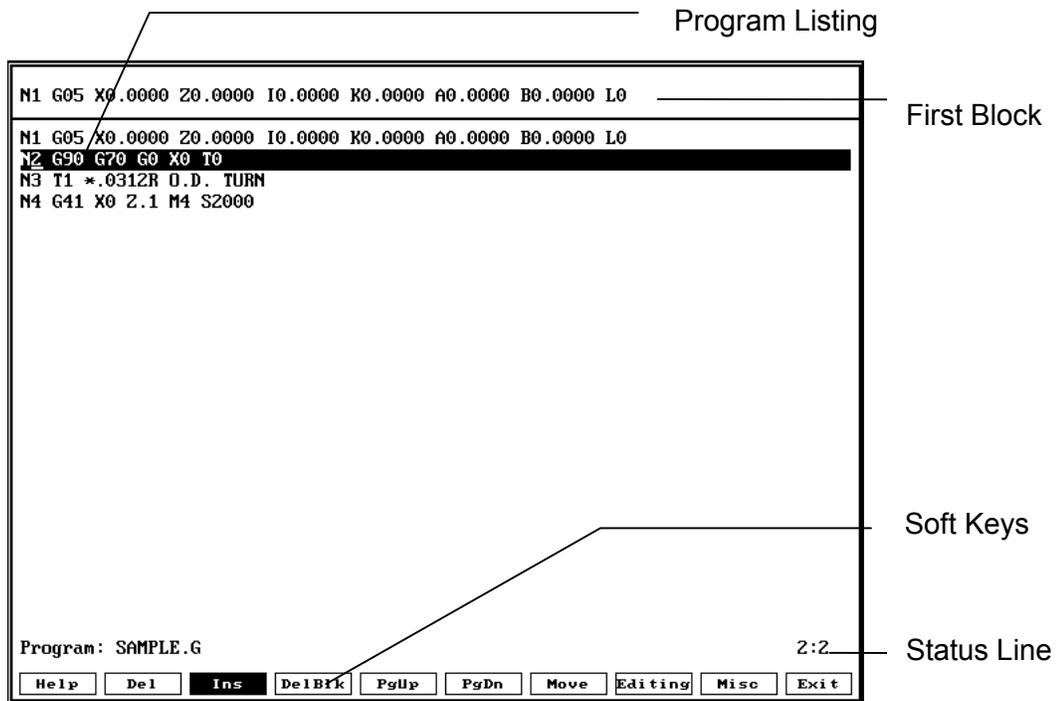


Figure 6-1, Edit Screen

Refer to **Figure 6-1** to write and edit programs from the Edit Screen. Refer to **Table 6-1** for an overview of the Edit screen.

Table 6-1, Edit Screen Overview

Program Name	The name of the program listed on the screen.
Insertion Point Location (cursor)	Indicates where typed text will be entered on the screen.
Soft Key Labels	These labels define soft key functions. Some soft keys, when pressed, activate Pop-Up Menus containing additional features.
(edited) Marker	The (edited) marker indicates that the program has been edited and that the edits have not been saved.
Marked Block	Highlighted block(s) to which the activated editing feature (cut, paste, delete, etc.) will be applied.
Program Listing	Area of the screen where the program is listed.

Edit Soft Keys

Refer to **Table 6-2** for a list of the soft keys in the edit mode.

Table 6-2, Edit Soft Keys

Soft Key Label	No.	Function
Help	F1	Activates Edit Help Menu.
Del	F2	Deletes the single character located at the insertion point.
Ins	F3	Sets the CNC to the Insert Mode. Typed characters will be inserted at the insertion point, without overwriting the existing text.
DelBlk	F4	Deletes a single block, located at the insertion point.
PgUp	F5	Returns the display to the previous page of the Program Listing.
PgDn	F6	Advances the display to the next page of the Program Listing.
Move	F7	Activates the Move Pop-Up Menu. The Move Pop-Up Menu contains features which enable you to go to the beginning or end of a line or program. It also contains word and line search features.
Editing	F8	Activates the Editing Pop-Up Menu. The Editing Pop-Up Menu contains features which enable you to perform editing functions within a single program or between two programs. This includes: inserting, undeleting blocks, and canceling unsaved edits to a block. It also includes cutting and pasting blocks within a program or writing and reading blocks between programs.
Misc	F9	Activates the Misc Pop-Up Menu. The Misc Pop-Up Menu contains features that allow you to record and recall recorded keystrokes, repeat programming commands, print the Program Listing, and display or edit another program.
Exit	F10	Exits the editor.

Editing Shift Soft Keys

The Edit screen contains ten soft keys when it activates. Four additional soft keys activate when you press **SHIFT**. Refer to **Table 6-3** for a list of shift soft keys in the edit mode. To activate any **SHIFT** soft key:

1. In Edit Mode, press **SHIFT** and then press the appropriate soft key (**F4**, **F7**, **F8**, or **F10**).

Table 6-3, Edit Shift Soft Keys

Soft Key Label	No.	Function
UnDelBlk	SHIFT + F4	Restores up to 128 deleted blocks, beginning with the last one deleted.
FinNext	SHIFT + F7	Advances to the next occurrence of the text specified in the Find Word feature. Find Word is located in the MOVE Pop-Up Menu.
ChaNext	SHIFT + F8	Advances to the next occurrence of the text specified in the Change Word feature and changes the text to the replacement text specified in Change Word . Change Word is located in the MOVE Pop-Up Menu.
Quit	SHIFT + F10	Exits the editor without saving edits made to the Program Listing.

Deleting a Character

To delete a character:

1. In Edit Mode, use the **ARROWS** to place the insertion point on the character being deleted.
2. Press **Del (F2)**. The character is deleted.

Inserting Text to Overwrite Previous Text

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

1. In Edit Mode, press **Ins (F3)** to switch it OFF. **Ins (F3)** is no longer highlighted.
2. Place the insertion point where the text is to be inserted. Enter the new text. The CNC replaces previously typed text with the new text.

Inserting Text without Overwriting Previous Text

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

1. In Edit Mode, press **Ins (F3)**, unless it is already highlighted.
2. Place the insertion point where the text is to be inserted. Type the new text. The new text is placed at the insertion point. Previously typed text is advanced forward as new text is typed.

Deleting a Program Block

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

- Press **DelBlk (F4)** to delete one block at a time.
- Use the BLOCK Operations **Delete** feature to delete several blocks at a time.

To delete a program block using **DelBlk (F4)**:

1. In Edit Mode, use the **ARROW** keys to place the insertion point on the program block being deleted.
2. Press **DelBlk (F4)**. The CNC deletes the block.

Delete program blocks by first marking the block and then using the BLOCK Operations **Delete** feature. To mark the program block and delete it:

1. In Edit Mode, first mark the program blocks to be deleted. Press **Editing (F8)**. The soft key highlights and the EDITING Pop-up Menu activates listing **Mark Block** as one of the options.
2. Highlight **Mark Block**, and press **ENTER**.
3. Use the **ARROW** keys to scroll to the desired blocks to be marked. Press **ENTER**. The blocks are now marked.
4. Press **Editing (F8)**. The soft key highlights and the EDITING Pop-Up Menu again activates.
5. Use the **ARROW** keys to highlight **BLOCK Operations**. Press **ENTER**. The BLOCK operation Pop-Up Menu activates.
6. Use the **ARROW** keys to highlight **Delete**. Press **ENTER**. The CNC deletes the marked blocks.

<p>NOTE: To unmark blocks that are marked (without doing any block operation), go to Edit, then Mark Block, and press ENTER.</p>
--

Page Up/Down through the Program Listing

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

1. In Edit Mode, press **PgUp (F5)** to advance forward or **PgDn (F6)** to go back in the program. The CNC advances or goes back a page at a time.

Going to a Line of the Program Listing

The **Go to Line** feature enables you to go to any line in the Program Listing. **Go to Line** operates independently of block numbering. Blocks can be numbered sequentially by any increment (1, 5, 10...). **Go to Line** counts the lines of the program in increments of 1 (1 [starting block], 2, 3...). When this feature activates, the CNC goes to the line number you have specified, 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)**. The soft key highlights and the MOVE Pop-Up Menu activates.
2. Highlight **Go to Line**.
3. The following message is displayed, "**Go to line:**" Type the appropriate line number, and press **ENTER**. The CNC places the insertion point at that line number.

Advancing to the Beginning or End of a Block

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

1. In Edit Mode and with the insertion point on any block of the program, press **Move (F7)**. The soft key highlights and the MOVE Pop-Up Menu activates.
2. Highlight **End of Line**, and press **ENTER**. The insertion point advances to the end of the block.

– or –

Highlight **Start of line**, and press **ENTER**. The insertion point 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)**. The soft key highlights and the MOVE Pop-Up Menu activates.
2. Highlight **End of program**, and press **ENTER**. The insertion point advances to the last block of the program.

– or –

Highlight **Start of program**, and press **ENTER**. The insertion point returns to the first block of the program.

Searching the Program Listing for Selected Text

Use the **Find Word** and **Find Next** features to search blocks for selected text. Enter the text to be found. The CNC searches out the first (**Find Word**) and subsequent occurrences (**Find Next**) in the Program Listing.

To find all occurrences of text in a program:

1. In Edit Mode, place the insertion point at the beginning of the program. (**Find Word** and **Find Next** only search forward in the program.)
2. Press **Move (F7)**. The soft key highlights and the Move Pop-Up Menu activates.
3. Highlight **Find Word**, and press **ENTER**.
4. The following message is displayed, "**Enter Word to Find:**" Type the text to be found, and press **ENTER**. The insertion point advances to the first occurrence of the text in the program.
5. To advance to the next occurrence of the text, press **Move (F7)**. The soft key highlights and the Move Pop-Up Menu activates.
6. Highlight **Find Next**, and press **ENTER**. The CNC advances to the next occurrence of the text in the program.
7. Use this method to search for all occurrences of the text in the Program Listing.

NOTE: You can also activate **Find Next** from the **SHIFT** soft key menu. To activate **Find Next** from the **SHIFT** soft key menu:

1. In Edit Mode, press **SHIFT**. This activates the **SHIFT** soft key menu.
2. Press **FinNext (F7)**. This soft key performs the same function as **Find Next**, located in the **MOVE** Pop-Up Menu. Each time you press the soft key, the CNC advances to the next occurrence of the text specified in **Find Word**.

Inserting a Blank Line in the Program Listing

You can insert a line at the insertion point. All subsequent lines will be moved down one line in the program. To insert a blank line:

1. In Edit Mode and with the insertion point where the blank line is to appear, press **Editing (F8)**. The soft key highlights and the **EDITING** Pop-Up Menu activates.
2. **Insert Block** is highlighted when the menu activates. Press **ENTER**. A blank line is inserted at the insertion point.
3. You can use the line to type a new program block.

Restoring a Block

If a block has been deleted, you can restore it to the program by using **Undelete Block**. The last block deleted is the first block restored. There are two ways to undelete a block:

To restore a block using the **Editing (F8)** Pop-up Menu:

1. In Edit Mode, use the ARROWS to position the insertion point at the point where the undeleted block will appear.
2. Press **Editing (F8)**. The soft key highlights and the EDITING Pop-Up Menu activates.
3. Highlight **Undelete Block**, and press **ENTER**. The last line deleted from the program is displayed at the insertion point.

To undelete a block using the SHIFT soft key menu:

1. In Edit Mode, use the **ARROW** keys to position the insertion point where the undeleted block will appear.
2. Press **SHIFT** and then press **UnDelBlk (SHIFT + F4)**. The last line deleted from the program is inserted at the insertion point.

NOTE: You can restore up to 128 consecutively deleted blocks.

Replacing Typed Text with New Text

The **Change Word** feature allows you to replace occurrences of specific text selectively. Type in the appropriate text and the CNC searches the Program Listing for all occurrences of that text. You can edit or skip each occurrence.

To edit each occurrence of the typed text selectively:

1. In Edit Mode, press **Editing (F8)**. The soft key highlights and the EDITING Pop-Up Menu activates.
2. Highlight **Change Word**, and press **ENTER**.
3. The following message is displayed, "**Enter Word to Change:**" Type the text to be replaced, and press **ENTER**.
4. The following message is displayed, "**Change <<TEXT>> with:**" Type the replacement text, and press **ENTER**. The CNC goes to the first occurrence of the text in the program. The following message is displayed, "**Change <<TEXT>> with <<REPLACEMENT TEXT>>?"** and the soft keys change. The CNC finds each occurrence of the text in the program. For each occurrence, you can choose one of the following features:

Yes	(F1)	Replacement text is inserted in place of text.
No	(F2)	Text remains unchanged.
All	(F3)	Replacement text is inserted in place of the text for all occurrences.
Only	(F4)	Replacement text is inserted in place of text and Change Word feature is deactivated.

If **Yes (F1)** or **No (F2)** is chosen, the CNC replaces (**F1**) or leaves the text unchanged (**F2**), then moves to the next occurrence of the entered text.

The **Change Word** feature deactivates when all occurrences of the text have been found or if you press **Only (F4)**. After you deactivate the **Change Word** feature, the following message is displayed, "**Change complete; [# of changes] occurrences changed.**" The CNC returns to Edit Mode.

Abbreviating Statements

Expand key enables you to use statements without typing in the entire statement. Enter an abbreviation assigned to a statement and activate the **Expand key**.

For conditional statements, the CNC displays the statement and waits for you to key in the condition under which the statement will be activated. Not all statements require you to enter a condition.

Refer to **Table 6-4** for a list of the statements to which **Expand key** can be applied and the abbreviations assigned to each statement. In the table, the statements are displayed inside brackets. These brackets do not appear on the screen.

Table 6-4, Statement Abbreviations

Abbreviation	Statement
D	[DO] [END]
E	[END] or [ENDIF] or [ELSE]
G	[GOTO]
I	[IF (_) THEN] [ENDIF]
L	[LOOP] [END]
P	[PRINT]
W	[WHILE (_) DO] [END]

To generate a statement from an abbreviation:

1. In Edit Mode, type the abbreviation for the conditional statement.
2. Press **Editing (F8)**. The EDITING Pop-Up Menu activates.
3. Highlight **Expand key**, and press **ENTER**. The CNC generates the statement on the screen.
4. If the insertion point is displayed inside parentheses (_), you must enter a condition under which the statement will be activated. Enter the condition, if applicable.

Marking (Highlighting) Programming Blocks

Many editing features require you to mark (highlight) affected program blocks before the edit is performed. To mark program blocks:

1. In Edit Mode, place the insertion point at the beginning of the first block to be marked.
2. Press **Editing (F8)**. The soft key highlights and the EDITING Pop-Up Menu activates.
3. Highlight **Mark Block**, and press **ENTER**. The block is marked.
4. Use the **ARROW** keys to mark the appropriate blocks. Press **ENTER**.

Unmarking Program Blocks

To unmark blocks:

1. In Edit Mode, press **Editing (F8)**. The soft key highlights and the EDITING Pop-Up Menu activates.
2. Highlight **Mark Block**, and press **ENTER**. Previously marked blocks will no longer be highlighted.

Canceling Edits to a Program Block

Use **Restore Block** to cancel edits to a program block and restore the block to its original form. The feature must be activated before the insertion point has been moved to another block. To cancel edits to a program block:

1. In Edit Mode and with the insertion point on the program block to be restored, press **Editing (F8)**. The soft key highlights and the EDITING Pop-Up Menu activates.
2. Highlight **Restore Block**, and press **ENTER**. The CNC cancels the edits and restores the original block.

Scrolling Through the Program Listing

In Edit Mode, press the **UP** (Forward) and **DOWN** (Backward) **ARROW** keys to display different portions of the Program Listing.

Copying Program Blocks

NOTE: You can cut, save and paste blocks within a Program Listing. **Paste** features works for pasting blocks between two different programs as long as you don't exit the editor. Use the Edit Program or Pick Program to edit another program within the editor.

You can 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 save blocks	Copies and stores marked blocks. Leaves original blocks unchanged.
Mark and cut blocks	Copies and stores marked blocks. Deletes original blocks.

To copy one or more program blocks while leaving the original blocks unchanged:

1. In Edit Mode, place the insertion point at the beginning of the first block to be copied.
2. You must mark the blocks to be copied. Press **Editing (F8)**. The soft key highlights and the EDITING Pop-up Menu activates listing **Mark Block** as one of the options.
3. Highlight **Mark Block**, and press **ENTER**.
4. Use the **ARROW** keys to scroll to the desired blocks to be marked. The blocks are now marked.
5. Press **Editing (F8)**. The soft key highlights and the EDITING Pop-Up Menu activates.
6. Highlight **BLOCK Operations**, and press **ENTER**. The BLOCK Operations Pop-Up Menu activates.
7. Highlight **Copy**, and press **ENTER**. The CNC saves the blocks in memory and the original blocks remain in the Program Listing.

To copy program blocks and delete the original blocks:

1. In Edit Mode, place the insertion point at the beginning of the first block to be copied.
2. You must mark the program blocks to be copied. Press **Editing (F8)**. The soft key highlights and the EDITING Pop-up menu activates listing **Mark Block** as one of the options.
3. Highlight **Mark Block**, and press **ENTER**.
4. Use the **ARROW** keys to scroll to the desired blocks to be marked. The blocks are now marked.
5. Press **Editing (F8)**. The soft key highlights and the EDITING Pop-Up Menu again activates.

6. Highlight **BLOCK operations**, and press **ENTER**. The BLOCK operations Pop-Up Menu activates.
7. Highlight **Cut**, and press **ENTER**. The CNC saves the blocks in memory and deletes the original blocks from the Program Listing.

Printing a Portion of a Program

NOTE: To print the entire Program Listing, use the **Print Program** feature from the MISC Pop-Up Menu.

Use the **Print** feature, from the BLOCK operations Pop-Up Menu, to print part of a Program Listing. Select one or more blocks from the Listing. The CNC prints out the selected blocks.

1. In Edit Mode, mark all blocks to be printed.
2. Press **Editing (F8)**. The soft key highlights and the EDITING Pop-Up Menu activates.
3. Highlight **BLOCK operations**, and press **ENTER**. The BLOCK operations Pop-Up Menu activates.
4. Highlight **Print**, and press **ENTER**. The soft key menu changes.
5. The following message is displayed, "**Print selected block?**" Press **Yes (F1)** to print the program, or **No (F2)** to cancel the feature.
6. When **Yes (F1)** is pressed, the CNC prints the selected program blocks. A status screen is displayed, containing the program name and the line, page, and number of copies being printed. At the completion of the print job, the CNC displays a message and the soft keys change.
7. Press **Continue (F10)** to return to the Edit screen.

NOTE: While the program is printing, press **Cancel (F9)** to cancel the print job.

Pasting Copied Blocks into a Program Listing

NOTE: You can cut, save and paste blocks within a Program Listing. **Paste** features works for pasting blocks between two different programs as long as you don't exit the editor. Use the Edit Program or Pick Program to edit another program within the editor.

You can paste blocks into another section of the same Program Listing. To paste copied blocks into the Listing:

1. In Edit Mode, place the insertion point where the copied blocks are to be pasted.
2. Press **Editing (F8)**. The soft key highlights and the EDITING Pop-Up Menu activates.
3. Highlight **BLOCK operations**, and press **ENTER**. The BLOCK operations Pop-Up Menu activates.
4. Highlight **Paste**, and press **ENTER**. The CNC pastes the copied blocks into the Program Listing at the insertion point.

Numbering (or Re-numbering) Program Blocks

To number or re-number blocks in a program:

1. In Edit Mode, mark all the blocks in the program.
2. Press **Editing (F8)**. The soft key highlights and the EDITING Pop-Up Menu activates.
3. Highlight **BLOCK operations**, and press **ENTER**. The BLOCK operations Pop-Up Menu activates.
4. Highlight **Renum**, and press **ENTER**. The soft key menu changes.
5. The CNC displays the following message, “**Enter starting N#:**” Enter a value (1, 5, 10...) . The CNC assigns the number to the starting block of the program. Press **ENTER**.
6. The following message is displayed, “**Enter N# increment:**” Enter an incremental value (1 ,5 , 10...). The CNC uses the entered values to number the program. The numbering sequence assigned to the listing starts at the number assigned to the starting block. Subsequent block numbers are assigned based on the incremental value entered (1, 2, 3...; 5, 10, 15...; 10, 20, 30...).
7. Unmark highlighted blocks.

Writing Blocks to Another Program

Any blocks that have been saved using the Block Copy feature can be pasted into another program as long as the editor has not been exited. Use the **Write** feature, located in the BLOCK Operations Pop-Up Menu, to copy one or more blocks between programs. You select the program to which the copied blocks are written. The **Write** feature deletes any other information currently in the selected program and replaces it with the copied blocks. The copied blocks remain in the original program.

To write blocks from one program to another:

1. In Edit Mode, mark all blocks to be copied.
2. Press **Editing (F8)**. The soft key highlights and the EDITING Pop-Up Menu activates.
3. Highlight **BLOCK Operations**, and press **ENTER**. The BLOCK Operations Pop-Up Menu activates.
4. Highlight **Write**, and press **ENTER**. The soft key menu changes.

NOTE: Press Cancel (F9) to cancel the operation.
--

5. The following message is displayed, “**Write block as:**” Type in the name of the program to which the blocks are being copied. (Type the entire program name or the CNC generates an error message.) Press **ENTER**.

If the program name entered already exists, you will be prompted to overwrite (options: **Yes, No, Cancel**). If the program name does not exist, it will be created, and block will be written to it.

The blocks are copied to the selected program.

NOTE: The **Write** feature does not use or overwrite information in the buffer, where cut and saved blocks are stored. When **Write** is used, the information in the buffer remains unchanged.

Reading a Program into Another Program

Use the **Read** feature, in the BLOCK operations Pop-Up Menu, to copy an entire program into the Program Listing automatically. The copied program will appear at the insertion point.

To copy an entire program into another Program Listing:

1. In Edit Mode, choose **Editing (F8)**. The soft key highlights and the EDITING Pop-Up Menu activates.
2. Highlight **BLOCK operations**, and press **ENTER**. The BLOCK operation Pop-Up Menu activates.
3. Highlight **Read**, and press **ENTER**.

The following message is displayed, "**Enter program to read:**" Enter the name of the program to be copied into the Program Listing (wild cards may be used for program name). The CNC copies the entered program into the Program Directory at the insertion point. The following message is displayed when the operation is finished, "**Block read from <<PROGRAM>>**"

Recording Keystrokes

The **Record Keys** feature records keystrokes as they are entered. It is useful for recording program sequences that are used in several areas of the program. When you activate **Record Keys**, the keystrokes are saved in a part of memory called the "macro buffer". **Play keys** recalls recorded keystrokes.

To record keystrokes as they are typed:

1. In Edit Mode, press **Misc (F9)**. The soft key highlights and the MISC Pop-Up Menu activates.
2. Highlight **Record Keys**, and press **ENTER**.
3. If any data has been stored in the macro buffer, the following message is displayed, "**Overwrite existing macro buffer?**" If the message appears, choose **Yes (F1)**.
4. Type the appropriate text. The CNC records the typed text.
5. Highlight **Record Keys**, and press **ENTER**. The Record Keys feature deactivates.

NOTE: Information in the macro buffer remains there until you overwrite it with new data or until you exit the editor..

Playing Recorded Keystrokes

The **Play keys** feature retrieves recorded keystrokes and prints them on the screen. To retrieve recorded keystrokes:

1. In Edit Mode, press **Misc (F9)**. The soft key highlights and the MISC Pop-Up Menu activates.
2. Highlight **Play Keys**, and press **ENTER**. The CNC displays the recorded keystrokes at the insertion point.

Repeating a Command or Key

NOTE: **Repeat Command** is used in conjunction with other features. Make sure you understand how a feature works before duplicating it with **Repeat Command**.

The **Repeat Command** feature repeats activated commands or repeats a single typed character.

To repeat a command or character:

1. In Edit Mode, press **Misc (F9)**. The soft key highlights and the MISC Pop-Up Menu activates.
2. Highlight **Repeat Command**, and press **ENTER**.
3. The following message is displayed, "**Enter Repeat Count:**" Enter the number of times the command will be repeated.
4. The CNC prompts you to "**Select Command (key) To Repeat.**"
5. Select the command (or key) to be repeated. The CNC repeats the command. The command is repeated the number of times entered after the **Enter Repeat Count:** inquiry.

NOTE: The maximum Repeat Count is 99,999.

Printing an Entire Program Listing

NOTE: Use **Print Program** (located in the BLOCK operations Pop-Up Menu) to print part of a Program Listing.

Use **Print Program**, in the MISC Pop-Up Menu, to print an entire program.

To print an entire program:

1. In Edit Mode, press **Misc (F9)**. The soft key highlights and the MISC Pop-Up Menu activates.
2. Highlight **Print Program**, and press **ENTER**.
3. The following message is displayed, "**Print <<PROGRAM.G>>?**" Press **Yes (F1)** to print the program.

– or –

Press **No (F2)** to cancel the feature.

4. When **Yes (F1)** is pressed, the CNC prints the program. A status screen is displayed, containing the program name, the line, page, and number of copies being printed. At the completion of the print job, the CNC displays a message and the soft keys change.

5. Press **Continue (F10)** to return to the Edit screen.

NOTE: While the program is printing, press **Cancel (F9)** to cancel the print job.

Accessing the Most Recently Used Programs

Use the **Pick Program** feature, from the MISC Pop-Up Menu, to access and display any of the last ten programs opened in the Edit Mode. The feature lists the programs in descending order, with the most recently opened program at the top of the list. The program currently open and displayed on the Edit screen is not displayed on the list.

To access and display any of the last ten programs opened in Edit Mode:

1. In Edit Mode, press **Misc (F9)**. The soft key highlights and the MISC Pop-Up Menu activates.
2. Highlight **Pick Program**, and press **ENTER**. The Pick program screen activates.
3. Highlight a program, and press **ENTER**. The CNC replaces the current Program Listing with the selected program.

NOTE: You can paste any blocks previously cut or copied into the newly opened program.

Opening Another Program from the Program Listing

Once final edits have been made to a program, use the **Edit Program** feature to open and edit another program from the Program Listing. To use the **Edit Program** feature to open a program:

1. In Edit Mode, press **Misc (F9)**. The soft key highlights and the MISC Pop-Up Menu activates.
2. Highlight **Edit program**, and press **ENTER**.
3. The following message is displayed, "**Enter program name:**" Enter the program name (wild cards may be used for program name).
4. If the current program has been edited, the following message is displayed, "**Program <<PROGRAM>> has been edited. Save before LOAD?**" Choose **Yes (F1)** to save the edits.

– or –

Press **No (F2)** to cancel edits to the current Listing.

5. If **Yes (F1)** is selected, press **ENTER**. The CNC displays the entered program.

Saving Edits

Although the Program Listing displays edits as soon as they are made, the edits are not saved until you exit the Program Editor. If the **(edited)** Marker appears beside the Program Name, the program contains unsaved edits.

To save edits:

1. In Edit Mode, press **Exit (F10)**. The CNC returns to the screen from which edit was entered and all edits are saved.

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 (SHIFT + F10)**.
2. The following message is displayed, "**Program edited, sure you want to Quit?**" and the soft keys change. Press **Yes (F1)** to cancel edits and return to the Program Directory.

– or –

Press **No (F2)** to return to the Edit Mode.

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 * is placed before a string of text, the CNC ignores all the text to the right of and on the same block as the asterisk. Refer to **Table 6-6** for various ways in which the * can be used in a Program Listing.

Table 6-6, Commenting Blocks

Commented Block	Ignored Text
*N20 G1 X5 Z6 N30 ...	Entire block is ignored by the CNC. Next block is executed normally.
N20 G1 *X5 Z6 N21 ...	G1 (linear interpolation) activated. The programmed move to X5 Z6 is ignored and the next block is executed normally.
N10 G70 G90 G0 X0 Z0 T0 N20 T1 *FACE/TURN TOOL	Block N20 activates Tool #1. The comment reminds you what type of tool this should be.

Section 7 - Edit Help

Edit Help provides diagrams and entry fields to program move types and canned cycles for blueprint programming.

NOTE: To select menu items (**2** through **9**, **+/-**, or **.**), press the key corresponding to the desired item, and then press **ENTER**. Menu items **2** through **9** correspond to the numeric keypad keys **2** through **9**. Menu item **+/-** corresponds to the \pm key. Menu item **.** corresponds to the "." key.

Refer to [Figure 7-1, Overview of the Edit Help Screens](#). Refer to [Table 7-1](#) for the functionality of the different screens in Edit Help.

Table 7-1, Edit Help Screen References

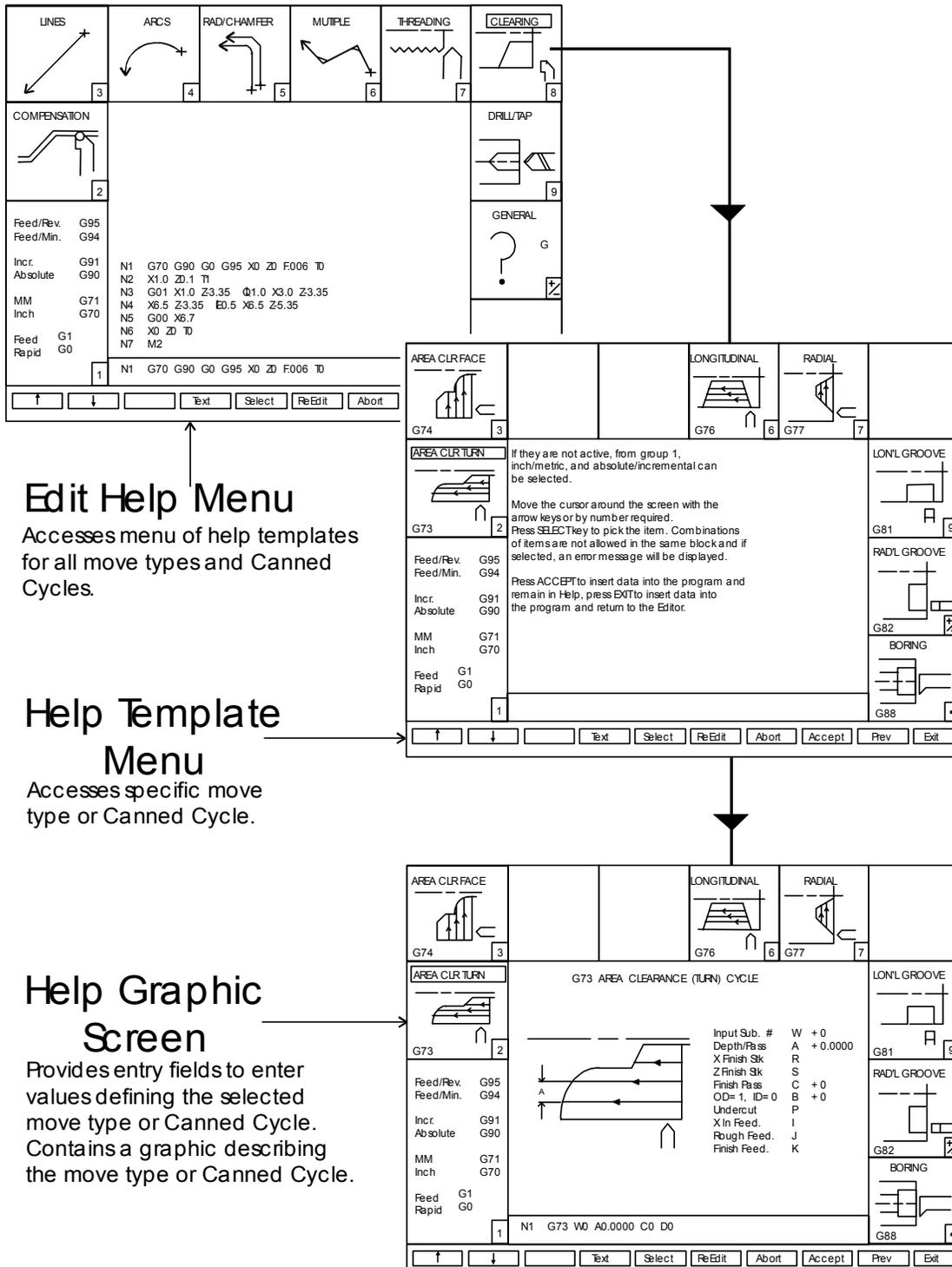
Edit Help Menu	Figure Reference	Function
Main Edit Help Menu	Figure 7-2, Main Edit Help Menu	Accesses help templates.
Help Template Menus	Figure 7-3, Sample Help Template Menu	Accesses Graphic Screens
Help Graphic screens	Figure 7-4, Sample Help Graphic Screen	Accesses entry fields to program moves and canned cycles.

The following instructions show how to activate a Help Graphic Screen for a G-code command and enter values in the appropriate entry fields:

1. In Edit Mode, open the appropriate program. Press **Help (F1)**. The Main Edit Help Menu activates. Refer to [Figure 7-2, Main Edit Help Menu](#).
2. Highlight **2 (Compensation)**, and press **ENTER**. The Compensation Help Template Menu activates. Refer to [Figure 7-3, Sample Help Template Menu](#).
3. Highlight Menu Item **7, Scaling (G72)**, and press **ENTER**. The Scaling Help Graphic screen activates. Refer to [Figure 7-4, Sample Help Graphic Screen](#).

NOTE: Most Help Template Menus contain some inactive menu items. Inactive menu selections contain no graphic and no menu item number.

4. Enter the appropriate values for the G72 command. Address word entry fields containing a "0.0000" value must be filled in. All other entries are optional.
5. When all appropriate values have been entered, press **ENTER**. The program block appears in the Input Box. Press **Accept (F8)** or **ENTER** to insert the block in the Program Listing.



Edit Help Menu
 Accesses menu of help templates for all move types and Canned Cycles.

Help Template Menu
 Accesses specific move type or Canned Cycle.

Help Graphic Screen
 Provides entry fields to enter values defining the selected move type or Canned Cycle. Contains a graphic describing the move type or Canned Cycle.

Figure 7-1, Overview of the Edit Help Screens

FLOWCHRZ

Menu items 2, 3, 4, 5 and 6 are referred to as move types. Menu items 7, 8 and 9 are Canned Cycles.

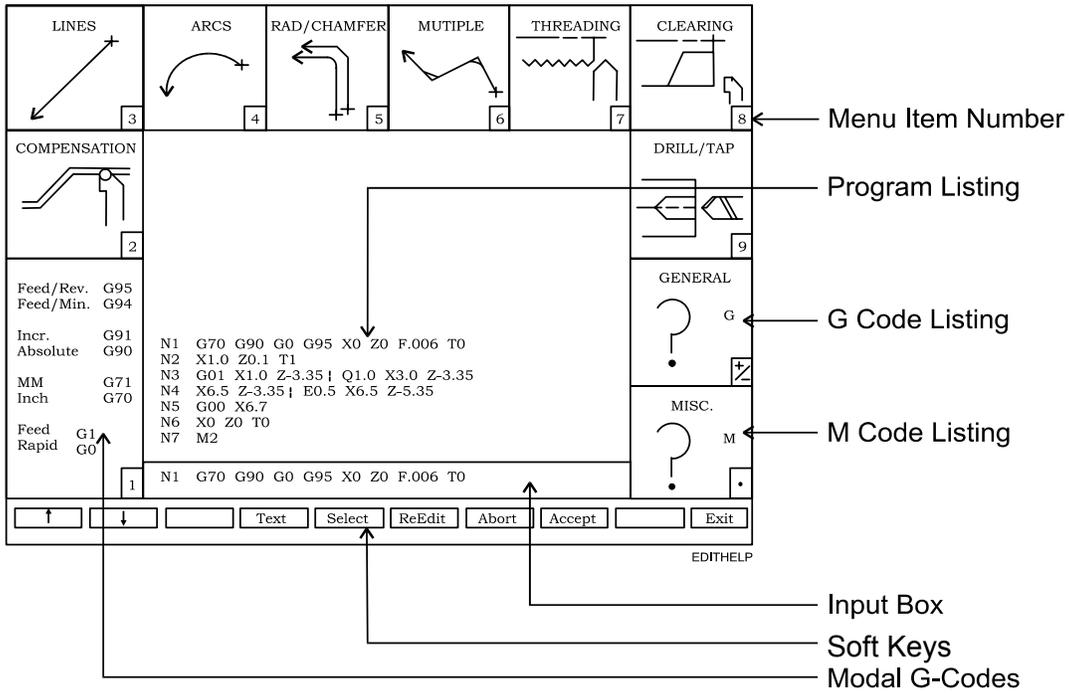


Figure 7-2, Main Edit Help Menu

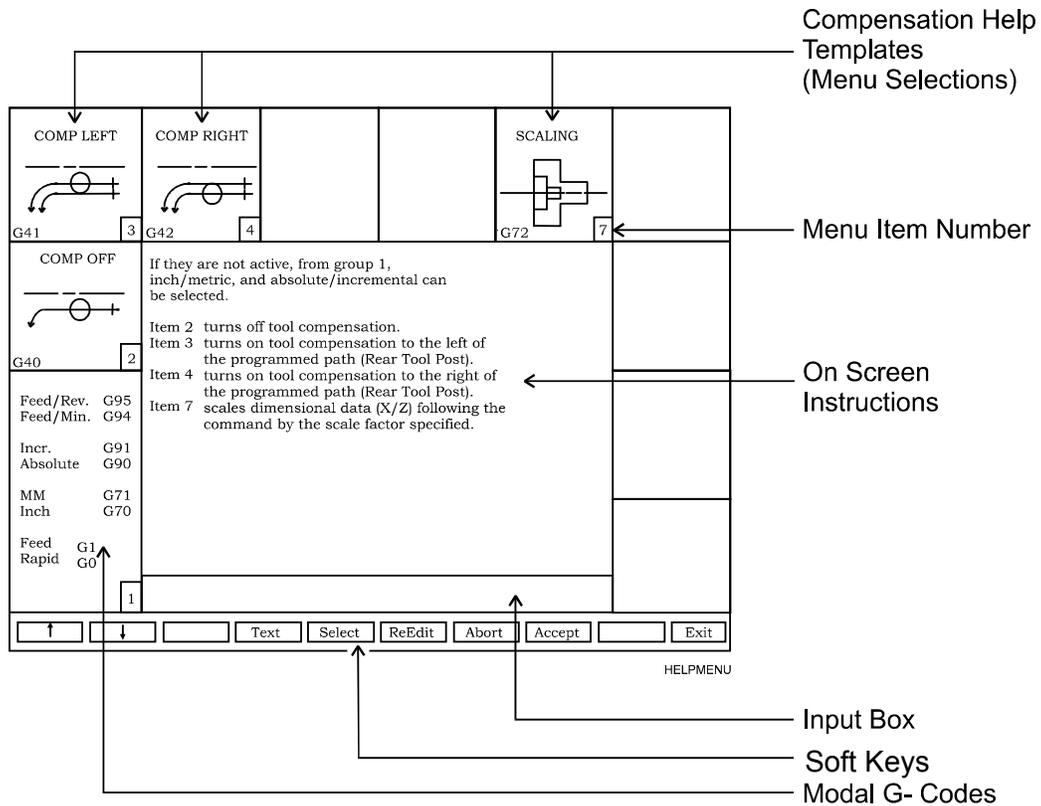


Figure 7-3, Sample Help Template Menu

Edit Help Menu

The [Main Edit Help Menu \(Figure 7-2\)](#) displays categories for which Help Menus are available. Refer to [Table 7-2, Edit Help Menu Features](#) for a description of Main Edit Help Menu features.

To activate the Main Edit Help Menu:

1. In Edit Mode, highlight the appropriate program listed, and press **Help (F1)**. The Main Edit Help Menu activates.

Refer to:

- [Table 7-2, Edit Help Menu Features](#)
- [Table 7-3, Edit Help G-Code Menu](#)
- [Table 7-4, Edit Help M-Code Listing](#)

for features that appear in the Main Edit Help Menu.

Table 7-2, Edit Help Menu Features

Feature	Description
Menu Item Number	Use this number to select a menu item.
Help Templates	In the Edit Help Menu, help templates access template help menus. (Template help menus access Help Graphic screens containing instructions for inputting move types and canned cycles.)
Move Type	Indicates what help template menu is accessed when the template is activated. Help template menus provide move types or Canned Cycles selections that, when activated, provide instructions for inputting code into a program.
Program Listing	The program being edited or written.
G-Code Listing	Lists and describes commonly used G-Codes (Preparatory Functions). Does not contain a complete listing of G-Codes. Refer to Table 7-3, Edit Help G-Code Menu for applicable G-Codes.
M-Code Listing	Lists and describes commonly used M Codes (Miscellaneous Functions). Does not contain a complete listing of M-Codes. Refer to Table 7-4, Edit Help M-Code Listing for applicable M-Codes.
Input Box	The CNC inserts the selected commands at the block displayed in the Input Box and displays commands as they are entered by the programmer. NOTE: The block displayed in the Input Box depends upon the location of the insertion point in the Program Listing when Help (F1) is pressed.
Input Box	Displays G-Codes and other commands selected in the Edit Help Menu. The CNC inserts the selected commands at the block displayed in the Input Box. NOTE: The block displayed in the Input Box depends upon the location of the insertion point in the Program Listing when Help (F1) is pressed.
Soft Keys	Use the soft keys to access, select, and accept programming blocks or to exit the Edit Help menus.

Table 7-3, Edit Help G-Code Menu

G-Code	Label and Description
G4	Dwell. Programs a timed or infinite dwell.
G9	Exact Stop (Single Block). Non-modal exact stop check. Activates exact stop check for a single block.
G22	Stroke Limit. Activates/deactivates software limits.
G24	Set Maximum Spindle Speed. Sets the maximum programmable spindle speed when the CNC is in Constant Surface Speed Mode (G96).
G53	Offset(s) (Coor. 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 Offsets Table.
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.
G92	Preset Zero. Shifts the location of Absolute Zero to a preset location. The preset location, measured from Machine Home, is specified in the G92 command.
G96	Constant Surface Speed (CSS). Sets the CNC's spindle to Constant Surface Speed (CSS). The CNC adjusts the spindle RPM to maintain a constant surface speed.
G97	Constant RPM. Sets the CNC's spindle speed to the Revolutions per Minute (RPM) mode.

Table 7-4, Edit Help M-Code Listing

M-Code	Function
M0	Program stop
M1	Optional program stop
M2	End of program
M3	Spindle ON FWD
M4	Spindle ON REV
M5	Spindle OFF
M8	Coolant ON
M9	Coolant OFF
M30	Jump to new program
M40	Open gear range
M41	First gear range
M42	Second gear range
M43	Third gear range
M98	Call subprogram
M99	End subprogram
M100	Mirror image

Edit Help Soft Keys

Refer to **Table 7-5** for a list of the soft keys available in the Main Edit Help Menu.

Table 7-5, Edit Help Soft Keys

Soft Key Label	No.	Description
↑ (UP)	F1	Moves highlight to the next Help Template.
↓ (DOWN)	F2	Moves highlight to the previous Help Template.
Text	F4	Switches the text shown in the center of the Edit Help Menu between two choices. The CNC displays either: <ul style="list-style-type: none"> – the active program and input box, or – instructions for using the Edit Help Menu
Select	F5	Selects the highlighted Help Template or G Code choice.
ReEdit	F6	When the insertion point is placed on a block containing a Canned Cycle, pressing ReEdit (F6) automatically activates the Help Graphic screen pertaining to the Canned Cycle. Once all the fields have been edited, press Accept (F8) .
Abort	F7	Deactivates the Edit Help Menu and returns the user to the Program Listing. NOTE: When the Edit Help Menu is deactivated, the CNC saves all accepted edits. The CNC does not save unaccepted edits to the block displayed in the Input Box.
Accept	F8	Inserts the block displayed in the Input Box into the program.
Exit	F10	Returns the CNC to the Program Listing.

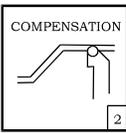
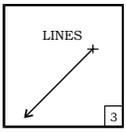
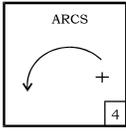
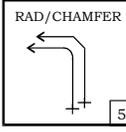
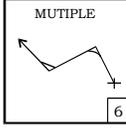
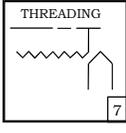
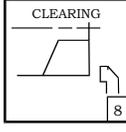
Help Template Menu

Selecting a help template from the Edit Help Menu activates the Help Template Menu for the selected move type or Canned Cycle. (Refer to [Figure 7-3, Sample Help Template Menu](#)). Help Template Menus are available for the move types described in **Table 7-6**.

To activate a help template menu:

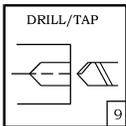
1. With the Edit Help Menu active, enter the menu item number, and press **ENTER**. The help template menu for the selected item number activates.

Table 7-6, Help Template Menus

Template	Description	Reference Table
	COMPENSATION (Compensated Moves)	Table 7-7, Compensation Help Template Menu
	LINES (Line Moves)	Table 7-8, LINES Help Template Menu
	ARCS (Arcs)	Table 7-9, ARCS Help Template Menu
	RAD/CHAMFER (Radius or Chamfer Moves)	Table 7-10, RAD/CHAMFER Help Template Menu
	MULTIPLE (Moves Containing Multiple Radius and/or Chamfer Moves)	Table 7-11, MULTIPLE Help Template Menu
	THREADING (Threading Cycles)	Table 7-12, THREADING Help Template Menu
	CLEARING (Clearing Cycles)	Table 7-13, CLEARING Help Template Menu

(Continued...)

Table 7-6, Help Template Menus (Continued)

Template	Description	Reference Table
	DRILL/TAP (Drill and Tapping Cycles)	Table 7-14, DRILL/TAP Help Template Menu

The following features appear in each of the Help Template Menus:

- Menu Item Number** You select a menu item for which the appropriate Help Graphic screen will be displayed.
NOTE: To select a menu item, press the corresponding number on the keypad, and press **ENTER**.
- Help Templates** Menu items containing available move types and Canned Cycles. When selected, these templates activate a Help Graphic screen. (Help Graphic screens contain instructions for inputting commands into the Program Listing.) Help template menus are available for the move types listed in [Figure 7-3, Sample Help Template Menu](#).
- On Screen Instructions** Press **TEXT (F4)** to switch the text displayed in the middle of the screen between on-screen instructions and the active Program Listing. On screen instructions may contain a description of each menu item (template) in the active Help Template Menu or general instructions on using Edit Help.
- Input Box** The CNC inserts the selected commands at the block displayed in the Input Box and displays commands as they are entered by the programmer.
- Soft keys** Use the soft keys to access, select, and accept programming blocks or to exit the screen. The Help Template Menu contains many of the same soft keys as the Edit Help Menu, with the following exceptions:
 The **ReEdit (F6)** soft key is inactive in the Help Template Menu. The **Prev (F9)** soft key is active. When the **Prev (F9)** soft key is activated, the CNC returns to the previous screen.
- Modal G-Codes** Modal G-Codes do not program actual moves. Use these commands to change the Rapid/Feed Mode, Inch/MM mode, Absolute/Incremental Mode, and Feed Per Revolution/Feed Per Minute Mode.

Help Graphic Screens

The Help Graphic screens list and give instructions on entering move types and Canned Cycles into the Program Listing.

Refer to **Figure 7-4**. Help Graphics screens are available for templates listed in [Table 7-6, Help Template Menus](#).

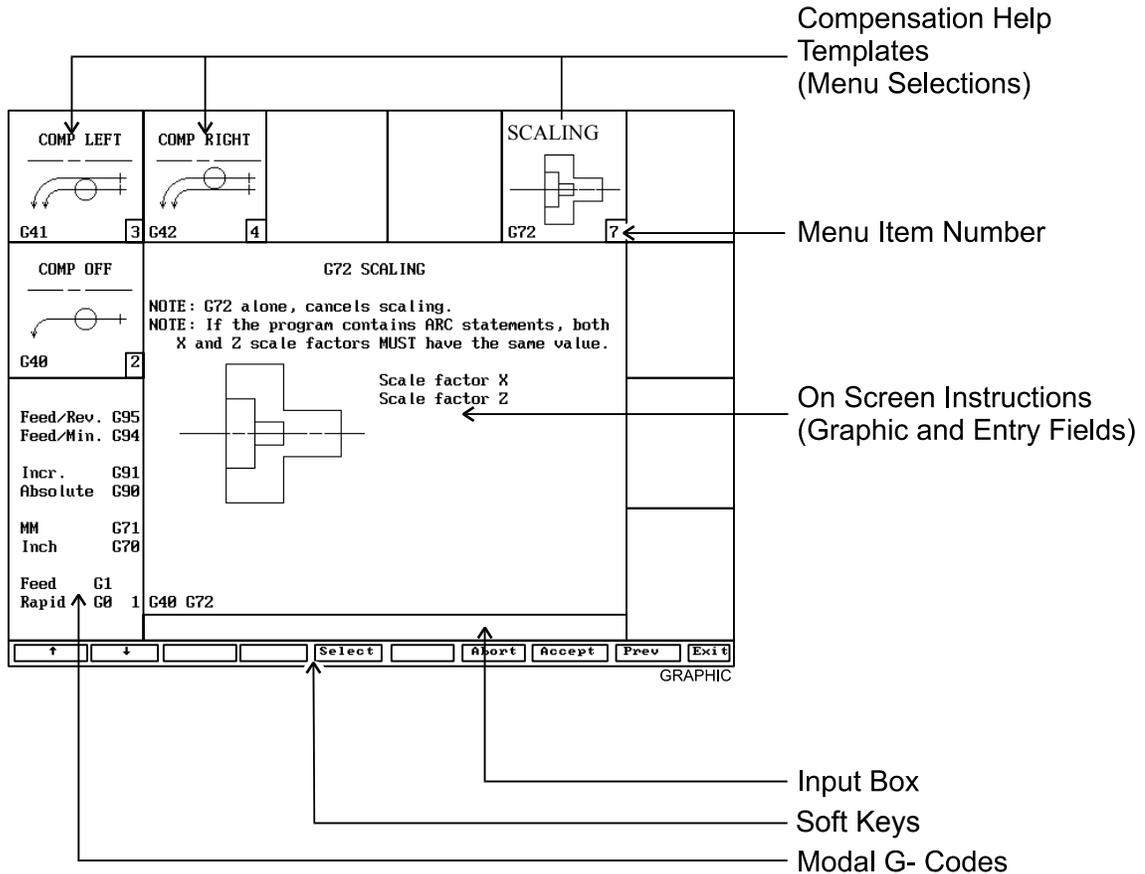


Figure 7-4, Sample Help Graphic Screen

To activate a Help Graphics screen:

1. With the Edit Help Menu active, enter the appropriate menu item number, and press **ENTER**. (Refer to [Figure 7-2, Main Edit Help Menu](#).) The help template menu for the selected move type or Canned Cycle activates.

From the activated Help Template Menu, enter the appropriate menu item number, and press **ENTER**. The Help Graphic screen for the selected template activates. It will contain labeled entry fields and a graphic describing the move type or Canned Cycle being programmed.

The following features are available in the Help Graphic screens:

Menu Item Number	Menu items 2 through 9 are inactive in the Help Graphic screen. A new selection can be made when the block for the active Help Graphic screen has been inserted into the Program Listing or canceled.
Help Templates	<p>The Help Templates displayed on the screen remain inactive until the values for the selected move type or Canned Cycle have been inserted into the Program Listing.</p> <p>Press Exit (F10) to insert the block into the program and return to the Program Listing.</p>
On Screen Instructions	On-screen instructions include a description of the selected menu item (move, G Code or Canned Cycle) being programmed, the entry fields, and graphic pertaining to the selected menu item.
Input Box	The CNC inserts the selected commands at the block displayed in the Input Box and displays commands as they are entered by the programmer.
Soft keys	<p>Use the soft keys to access, select, and accept programming blocks or to exit the screen. The Help Template Menus contain many of the same soft keys as the Edit Help Menu, with the following exceptions:</p> <p>The ReEdit (F6) and Text (F4) soft keys are inactive in the Help Graphic screens. The Prev (F9) soft key is active. When the Prev (F9) soft key is activated, the CNC returns to the previous screen.</p>
Modal G-Codes	Modal G-Codes do not program actual moves. Use these commands to change the Rapid/Feed Mode, Inch/MM mode, Absolute/Incremental Mode, and Feed Per Revolution/Feed Per Minute Mode.

Using Help Graphic Screens to Enter Program Blocks

The Program Editor displays Help Graphic screens for writing and editing program blocks.

When a Help Graphic screen is first displayed, the first entry field is highlighted. Once a field is highlighted, it will accept a value or selection made by the operator. Pressing **ENTER** advances the highlight to the next entry field. Pressing **ENTER** from the last entry field closes the block, and adds it to the Program Listing. Pressing the **Accept (F8)** soft key from any entry field also closes the block, and adds it to the program. Move the highlight from field to field using the **ARROW** keys. Entry fields can be filled in any order. Pressing the **CLEAR** key clears the value in the highlighted field.

There are two types of entry fields in a Help Graphic screen:

- Required entry fields - These fields contain “**0.000**” when the Help Graphic screen is first displayed.
- Optional entry fields - These fields are completely blank when the Help Graphic screen is first displayed.

Required entry fields cannot be left blank. The value can remain “**0.000**” (zero), if appropriate. If a required entry field is left blank, the CNC still writes the block, but when the block is executed, it holds the machine, and generates an error message.

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

The operator must remember to enter decimal points and negative signs where needed. The CNC assumes a positive value if no negative sign is keyed in.

Use the **(+/-)** key, to insert a negative sign. The **(+/-)** key is also used to switch the selection in entry fields that offer specific choices.

Available Help Graphic Screens

Help Graphic Screens are available for each of the templates shown and described in **Table 7-7**. The table shows the Edit Help Menu Templates and Template Menus. The input labels and entry fields are defined for each move type available in Edit Help.

NOTE: All examples used in this manual refer to machines with a front tool post orientation.

Table 7-7, Compensation Help Template Menu

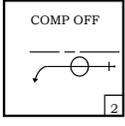
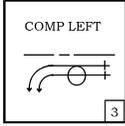
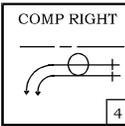
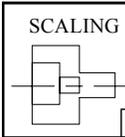
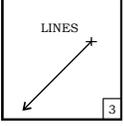
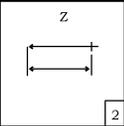
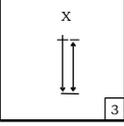
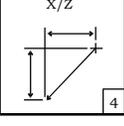
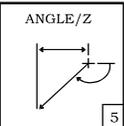
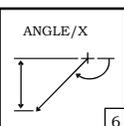
 COMPENSATION	
Templates and Parameters	Move Description
 <p>G40</p> <p>Select the COMP OFF template, and press ENTER automatically inserts the G40 command into the Input Box.</p>	<p>No move programmed. Tool Nose Radius Compensation canceled.</p>
 <p>G41</p> <p>Select the COMP LEFT template, and press ENTER automatically inserts the G41 command into the Input Box.</p>	<p>No move programmed. Tool Nose Radius Compensation, left of path, activated.</p>
 <p>G42</p> <p>Select the COMP LEFT template, and press ENTER automatically inserts the G42 command into the Input Box.</p>	<p>No move programmed. Tool Nose Radius Compensation, right of path, activated.</p>
 <p>G72</p> <p>Select SCALING Template, and press ENTER.</p>	<p>Scales dimensional data (X/Z) following the command by the scale factor specified.</p>

Table 7-8, LINES Help Template Menu

Templates and Parameters	Move Description
 <p>LINES</p>	
 <p>Z End Pt.: Z endpoint</p>	<p>CNC moves in a straight line along the Z-axis.</p>
 <p>X End Pt.: X endpoint</p>	<p>CNC moves in a straight line along the X-axis.</p>
 <p>X End Pt.: X endpoint Z End Pt.: Z endpoint</p>	<p>CNC moves along a vectored path from the current location to the X, Z endpoint.</p>
 <p>Angle: B, angle measured from the Z-axis (3 o'clock equals 0 degrees) Z End Pt.: Z endpoint</p>	<p>CNC moves along a vectored path from the current location to the endpoint.</p>
 <p>Angle: B, angle measured from the Z-axis (3 o'clock equals 0 degrees) X End Pt.: X endpoint</p>	<p>CNC moves along a vectored path from the current location to the endpoint.</p>

(Continued...)

Table 7-8, LINES Help Template Menu (Continued)

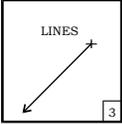
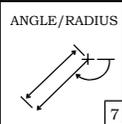
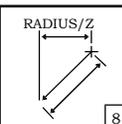
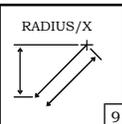
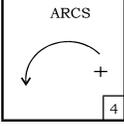
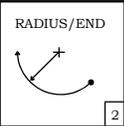
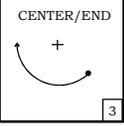
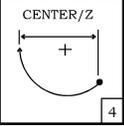
 LINES	
Templates and Parameters	Move Description
 <p>Angle: B, angle measured from the Z-axis (3 o'clock equals 0 degrees).</p> <p>Radius: Q, distance from the centerpoint of an arc to the circumference.</p>	<p>CNC moves along a vectored path from the current location to the endpoint.</p>
 <p>Radius: Q, distance from the centerpoint of an arc to the circumference.</p> <p>Z End Pt.: Z endpoint.</p>	<p>CNC moves along a vectored path from the current location to the endpoint.</p>
 <p>Radius: Q, distance from the centerpoint of an arc to the circumference.</p> <p>X End Pt.: X endpoint.</p>	<p>CNC moves along a vectored path from the current location to the endpoint.</p>

Table 7-9, ARCS Help Template Menu

Templates and Parameters	Move Description
 <p>ARCS</p>	
 <p>X End Pt.: X endpoint Z End Pt.: Z endpoint Radius: R, distance from the current location (centerpoint of an arc) to the endpoint (end of arc move).</p>	<p>CNC moves along a radial path from the current location to the programmed endpoint.</p>
 <p>X End Pt.: X endpoint Z End Pt.: Z endpoint X Center Pt.: I, incremental centerpoint of the arc in X. Z Center Pt.: K, incremental centerpoint of the arc in Z.</p>	<p>CNC moves along a radial path from the current location to the programmed endpoint.</p>
 <p>X Center Pt.: I, incremental centerpoint of the arc in X. Z Center Pt.: K, incremental centerpoint of the arc in Z. Z End Pt.: Z endpoint.</p>	<p>CNC moves along a radial path from the current location to the programmed endpoint.</p>

(Continued...)

Table 7-9, ARCS Help Template Menu (Continued)

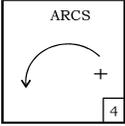
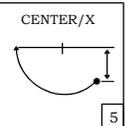
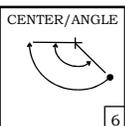
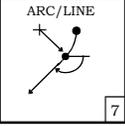
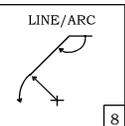
 ARCS	
Templates and Parameters	Move Description
 <p>X Center Pt.: I, incremental centerpoint of the arc in X. Z Center Pt.: K, incremental centerpoint of the arc in Z. X End Pt.: X endpoint.</p>	<p>CNC moves along a radial path from the current location to the programmed endpoint.</p>
 <p>X Center Pt.: I, centerpoint of the arc in X. Z Center Pt.: K, centerpoint of the arc in Z. Angle: B, angle measured from the Z-axis (3 o'clock equals 0 degrees).</p>	<p>CNC moves along a radial path from the current location to the programmed endpoint.</p>
 <p>Radius: Q, distance from the centerpoint of the arc to the circumference. Angle: B, angle measured from the Z-axis (3 o'clock equals 0 degrees). X End Pt.: X endpoint. Z End Pt.: Z endpoint.</p>	<p>The CNC performs an ARC move to the tangent point of the LINE intersection, and then moves at the specified angle to the endpoints.</p>
 <p>Angle: B, angle measured from the Z-axis (3 o'clock equals 0 degrees). Radius: Q, distance from the centerpoint of an arc to the circumference. X End Pt.: X endpoint. Z End Pt.: Z endpoint.</p>	<p>The CNC performs a LINE move to the tangent point of the ARC intersection, and then moves at the specified radius to the endpoints.</p>

Table 7-9, ARCS Help Template Menu (Continued)

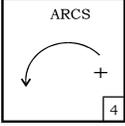
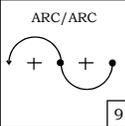
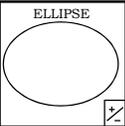
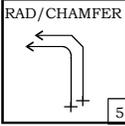
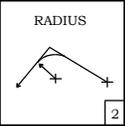
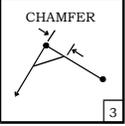
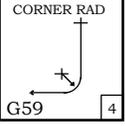
Templates and Parameters	Move Description
 <p>ARCS</p>	
 <p>R1 Center X: I, centerpoint of the first arc in X. R1 Center Z: K, centerpoint of the first arc in Z. R2 Center X: I, centerpoint of the second arc in X. R2 Center Z: K, centerpoint of the second arc in Z. X End Pt.: X endpoint. Z End Pt.: Z endpoint.</p>	<p>CNC performs an arc move into another arc move. The first arc ends and the second arc begins at the point where the two arcs are tangent.</p>
 <p>GO5 X End Pt.: X endpoint. Z End Pt.: Z endpoint. X Center Pt.: I, centerpoint of the arc in X. Z Center Pt.: K, centerpoint of the arc in Z. Half Width: A, half the width in X of the entire ellipse. Half Length: B, half the length in Z of the entire ellipse. CCW+, CW -: L, direction of tool travel. +1 for counterclockwise and -1 for clockwise.</p>	<p>CNC moves along an elliptical path from the current location to the programmed endpoint. This path could be a whole or partial ellipse in a clockwise or counterclockwise direction.</p>

Table 7-10, RAD/CHAMFER Help Template Menu

Templates and Parameters	Move Description
 <p>RAD/CHAMFER</p>	
 <p>RADIUS</p> <p>X Midpoint: X intersection point of two lines. Z Midpoint: Z intersection point of two lines. Radius: Q, radius of arc tangent to two lines. X End Pt.: X endpoint. Z End Pt.: Z endpoint.</p>	<p>The CNC performs a LINE move to the intersecting ARC, moves around the arc, and then moves along the second LINE to the specified endpoints.</p>
 <p>CHAMFER</p> <p>X Midpoint: X intersection point of two lines. Z Midpoint: Z intersection point of two lines. Chamfer: Distance from the XZ midpoint to the start/end of the Chamfer move. X End Pt.: Endpoint in X. Z End Pt.: Endpoint in Z.</p>	<p>The CNC performs a LINE move to the intersecting CHAMFER, moves across the chamfer, and then moves along the second LINE to the specified endpoints.</p>
 <p>CORNER RAD</p> <p>G59</p> <p>Radius: R, radius of arc used to round off each corner. NOTE: Possible in non-tangent line to line, line to arc, arc to line, and arc to arc moves.</p>	<p>The CNC performs an automatic corner rounding (where practical) at all intersecting elements (LINES, ARCS) with the specified radius. G59 (activate corner rounding) should be programmed on the block prior to the first required corner radius. G60 (cancel corner rounding) should be programmed on the block immediately following the last required corner radius. NOTE: Corner Rounding cannot be used in conjunction with Blue Print Programming (multiple moves programmed on the same block).</p>

(Continued...)

Table 7-10, RAD/CHAMFER Help Template Menu (Continued)

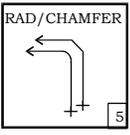
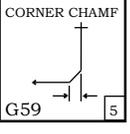
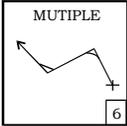
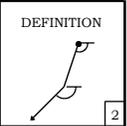
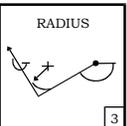
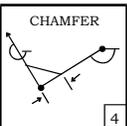
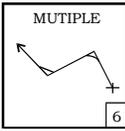
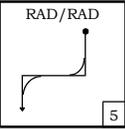
RAD/CHAMFER	
Templates and Parameters	Move Description
 <p>RAD/CHAMFER</p>	<p>The CNC performs an automatic corner chamfer (where practical) at all intersecting elements (LINES, ARCS). G59 (activate corner chamfering) should be programmed on the block prior to the first required corner chamfer. G60 (cancel corner chamfering) should be programmed on the block immediately following the last required corner chamfer.</p> <p>NOTE: Corner Chamfering cannot be used in conjunction with Blue Print Programming (multiple moves programmed on the same block).</p>
 <p>Chamfer: E, radius of chamfer used to round off each corner.</p> <p>NOTE: Possible in non-tangent line to line, line to arc, arc to line, and arc to arc moves.</p>	

Table 7-11, MULTIPLE Help Template Menu

 <p>MULTIPLE</p>	
<p>Templates and Parameters</p>	<p>Move Description</p>
 <p>DEFINITION</p> <p>First Angle: B, angle measured from the Z-axis (3 o'clock equals 0 degrees).</p> <p>Second Angle: B, angle measured from the Z-axis (3 o'clock equals 0 degrees).</p> <p>X End Pt.: Endpoint in X.</p> <p>Z End Pt.: Endpoint in Z.</p>	<p>The CNC performs a LINE move along the First Angle to the intersection of the Second Angle, and then continues along the Second Angle to the specified endpoints.</p>
 <p>RADIUS</p> <p>First Angle: B, angle measured from the Z-axis (3 o'clock equals 0 degrees).</p> <p>Radius: Q, radius of intersecting arc.</p> <p>Second Angle: B, angle measured from the Z-axis (3 o'clock equals 0 degrees).</p> <p>X End Pt.: Endpoint in X.</p> <p>Z End Pt.: Endpoint in Z.</p>	<p>The CNC performs a LINE move along the First Angle to the intersecting ARC, moves around the arc, and then moves along the Second Angle to the specified endpoint.</p>
 <p>CHAMFER</p> <p>First Angle: B, angle measured from the Z-axis (3 o'clock equals 0 degrees).</p> <p>Chamfer: E, length of intersecting chamfer.</p> <p>Second Angle: B, angle measured from the Z-axis (3 o'clock equals 0 degrees).</p> <p>X End Pt.: Endpoint in X.</p> <p>Z End Pt.: Endpoint in Z.</p>	<p>The CNC performs a LINE move along the First Angle to the intersecting CHAMFER, moves across the chamfer, and then moves along the Second Angle to the specified endpoints.</p>

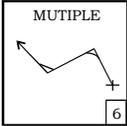
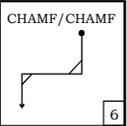
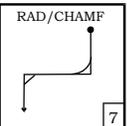
(Continued...)

Table 7-11, MULTIPLE Help Template Menu (Continued)

	
Templates and Parameters	Move Description
 <p> 1st Angle: B, angle measured from the Z-axis (3 o'clock equals 0 degrees). 1st Radius: Q, radius of arc tangent to 1st and 2nd Angle. 2nd Angle: B, angle measured from the Z-axis (3 o'clock equals 0 degrees). X Mid Pt.: Midpoint in X. Z Mid Pt.: Midpoint in Z. 2nd Radius: Q, radius of arc tangent to 2nd Angle and endpoints. X End Pt.: Endpoint in X. Z End Pt.: Endpoint in Z. </p>	<p>The CNC performs a LINE move along the 1st Angle to the intersecting arc. After completing the arc, the CNC moves along the 2nd Angle to the intersection of the 2nd arc. After completing the 2nd arc, the CNC moves to the specified endpoints.</p>

(Continued...)

Table 7-11, MULTIPLE Help Template Menu (Continued)

 <p>MULTIPLE</p>	
<p>Templates and Parameters</p>	<p>Move Description</p>
 <p>1st Angle: B, angle measured from the Z-axis (3 o'clock equals 0 degrees).</p> <p>1st Chamfer: E, length of intersecting chamfer between 1st and 2nd Angle.</p> <p>*These two lines are lines 1 and 2.</p> <p>2nd Angle: B, angle measured from the Z-axis (3 o'clock equals 0 degrees).</p> <p>X Mid Pt.: Midpoint in X.</p> <p>Z Mid Pt.: Midpoint in Z.</p> <p>2nd Chamfer: E, length of intersecting chamfer between 2nd Angle and endpoints.</p> <p>*These two lines are line 2 and the line between the midpoint and the endpoint.</p> <p>X End Pt.: Endpoint in X.</p> <p>Z End Pt.: Endpoint in Z.</p>	<p>The CNC performs a LINE move along the 1st Angle to the intersecting chamfer. After completing the chamfer, the CNC moves along the 2nd Angle to the intersection of the 2nd chamfer. After completing the 2nd chamfer, the CNC moves to the specified endpoints.</p>
 <p>1st Angle: B, angle measured from the Z-axis (3 o'clock equals 0 degrees).</p> <p>Radius: Q, radius of arc tangent to 1st and 2nd Angle.</p> <p>2nd Angle: B, angle measured from the Z-axis (3 o'clock equals 0 degrees).</p> <p>X Mid Pt.: Midpoint in X.</p> <p>Z Mid Pt.: Midpoint in Z.</p> <p>Chamfer: E, length of intersecting chamfer between 2nd Angle and endpoints.</p> <p>End Pt.: endpoint in X.</p> <p>Z End Pt.: endpoint in Z.</p>	<p>The CNC performs a LINE move along the 1st Angle to the intersecting arc. After completing the arc, the CNC moves along the 2nd Angle to the intersection of the chamfer. After completing the chamfer the CNC moves to the specified endpoints.</p>

(Continued...)

Table 7-11, MULTIPLE Help Template Menu (Continued)

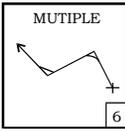
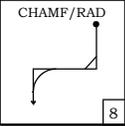
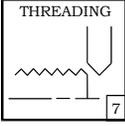
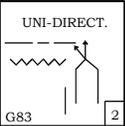
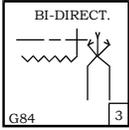
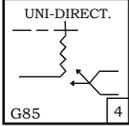
 MULTIPLE	
Templates and Parameters	Move Description
<div style="display: flex; align-items: flex-start;"> <div style="margin-right: 10px;">  </div> <div> <p>1st Angle: B, angle measured from the Z-axis (3 o'clock equals 0 degrees).</p> <p>Chamfer: E, length of intersecting chamfer between 1st and 2nd Angle.</p> <p>*These are lines 1 and 2.</p> <p>2nd Angle: B, angle measured from the Z-axis (3 o'clock equals 0 degrees).</p> <p>X Mid Pt.: Midpoint in X.</p> <p>Z Mid Pt.: Midpoint in Z.</p> <p>Radius: Q, radius of arc tangent to 2nd Angle and endpoints.</p> <p>X End Pt.: Endpoint in X.</p> <p>Z End Pt.: Endpoint in Z.</p> </div> </div>	<p>The CNC performs a LINE move along the 1st Angle to the intersecting chamfer. After completing the chamfer, the CNC moves along the 2nd Angle to the intersection of the arc. After completing the arc the CNC moves to the specified endpoints.</p>

Table 7-12, THREADING Help Template Menu

 THREADING	
Templates and Parameters	Move Description
<div style="display: flex; align-items: flex-start;"> <div style="margin-right: 10px;">  </div> <div> <p>G83 Longitudinal Uni-Directional Threading Entered Values</p> <p>Length: Z, incremental distance in the Z-axis from Tool start position to finish point (length of threading pass).</p> <p>First Cut: C, incremental depth of the first cut (depth of cut per side for first pass). Does not include B parameter.</p> <p>Thread Depth: D, incremental depth of the thread (depth of thread per side). Does not include B parameter. Default: Pitch x 0.6134 (60 degree thread).</p> <p>Thread Angle: A, thread angle, 60 degrees for standard threads. Default: 0 degrees for a straight plunge thread.</p> <p>Z Pullout: R, incremental Z-axis pullout distance. Creates a 2-axis pullout move. Default: 0 causes the tool to pullout at 90 degrees to the threading pass.</p> <p>Taper X: X, distance moved in the X-axis during a threading pass. Default: 0 straight thread.</p> <p>Taper Angle: V, Angle of tapered thread. Default: 0</p> <p>NOTE: When programming a tapered thread, use X or V parameters, but not both.</p> <p>X Standoff: B, distance in the X-axis from the tool tip to the work diameter per side. Default: Inch = 0.1 Outside Diameter Default: Inch = -0.1 Inside Diameter Default: MM = 2 Outside Diameter Default: MM = -2 Inside Diameter</p> <p>Tool must be positioned the Standoff distance away from the diameter in the block prior to the threading Canned Cycle.</p> <p>Num. Starts: W, used to program the number of starts in a multiple start thread. Default: 1 (Single start thread)</p> </div> </div>	<p>The G83 cycle creates a longitudinal thread along the Z-axis using multiple passes to reach full thread depth. The CNC moves from the X standoff (B) to the diameter of the work, plus the first depth of cut (C) and completes the first pass. The tool returns to the start point, and subsequent passes are taken at reducing depths of cut until the completed depth is reached.</p> <p>The number of passes necessary to complete a thread is calculated by the CNC based on the Thread Depth (D), and the depth of the first pass (C). The thread may be cut by a plunge motion (A=0), or by a compound motion (A=Angle of cutting tool*.)</p> <p>A number of finish passes (S=# of passes taken at final depth) may be specified. Default: 1</p> <p>R creates a 2-axis pullout move. The tool moves the distance programmed (R) in the Z-axis while retracting to the B standoff in the X-axis.</p> <p>If the taper angle (V) or distance (X) is specified the threading passes are completed at specified taper.</p> <p>The W parameter specifies the number of thread starts.</p> <p>A second use for the W parameter (programming a W-1) will cause all threading passes to be performed at the same depth of cut. D divided by C. Threads are normally cut in reducing depths of cut.</p> <p>*In the G83 cycle (uni-directional threading) if the A parameter is programmed the cycle will performed the passes using a compound cutting motion. The tool enters the thread on an angle (described by the A parameter), and cuts only on the leading edge of the tool.</p>

<p>TPI: E, Threads per inch. Used in Inch programming.</p> <p>Lead: F, pitch (single pass threading) or amount of travel in Z per revolution (multi-pass threading). Used in MM programming.</p> <p>NOTE: Use E or F, but not both.</p> <p># Fini. Pass: S, number of passes at final depth. Default: 1.</p>	
 <p>G84 Longitudinal Bi-Directional Threading</p> <p>TPI: E, Threads per inch. Used in Inch programming.</p> <p>Lead: F, pitch (single pass threading) or amount of travel in Z per revolution (multi-pass threading). Used in MM programming.</p> <p>NOTE: Use E or F, but not both.</p> <p>Length: Z, incremental distance in the Z-axis from Tool start position to finish point (length of threading pass).</p> <p>First Cut: C, incremental depth of the first cut (depth of cut per side for first pass). (Does not include B parameter).</p> <p>Thread Depth: D, incremental depth of the thread (depth of thread per side). (Does not include B parameter.) Default: Pitch x 0.6134 (60 degree thread).</p> <p>Thread Angle: A, thread angle, 60 degrees for standard threads. Default: 0 degrees for a straight plunge thread.</p> <p># Fini. Pass: S, number of passes at final depth. Default: 1.</p> <p>Z Pullout: R, incremental Z-axis pullout distance. Creates a 2-axis pullout move. Default: 0 causes the tool to pullout at 90 degrees to the threading pass.</p> <p>X Standoff: B, distance in the X-axis from the tool tip to the work diameter per side. Default: Inch = 0.1 Outside Diameter Default: Inch = -0.1 Inside Diameter. Default: MM = 2 Outside Diameter Default: MM = -2 Inside Diameter</p> <p>(Tool must be positioned the Standoff distance away from the diameter in the block prior to the</p>	<p>The G84 cycle creates a longitudinal thread along the Z-axis using multiple passes to reach full thread depth. The CNC moves from the X standoff (B) to the diameter of the work, plus the first depth of cut (C), and completes the first pass. The tool returns to the start point, and subsequent passes are taken at reducing depths of cut until the completed depth is reached. The number of passes necessary to complete a thread is calculated by the CNC based on the Thread Depth (D), and the depth of the first pass (C). The thread may be cut by a plunge motion (A=0), or by a compound motion (A=Angle of cutting tool*.) A number of finish passes (S=# of passes taken at final depth) may be specified. Default: 1</p> <p>R creates a 2-axis pullout move. The tool moves the distance programmed (R) in the Z-axis while retracting to the B standoff in the X-axis.</p> <p>*In the G84 cycle (uni-directional threading) if the A parameter is programmed, the cycle will perform the passes using a compound cutting motion. The tool enters the thread on an angle (described by the A parameter), and cuts alternating passes on the leading and trailing edges of the tool.</p>

<p>threading Canned Cycle.</p> <p>Num. Starts: W, used only in multiple pass threading, the number of staggered starts used to reach depth. Default: 1.</p> <p>NOTE: Specifying the Num. of Starts changes the way the CNC interprets all other threading variables.</p>	
 <p>G85 Radial Uni-directional Threading</p> <p>TPI: E, Threads per inch. Used in Inch programming.</p> <p>Lead: F, pitch (single pass threading) or amount of travel in X per revolution (multi-pass threading). Used in MM programming.</p> <p>NOTE: Use E or F, but not both.</p> <p>Length: X, incremental distance in the X-axis from Tool start position to finish point (length of threading pass).</p> <p>First Cut: C, incremental depth of the first cut (depth of cut for first pass). (Does not include B parameter.)</p> <p>Thread Depth: D, incremental depth of the thread (depth of thread). (Does not include B parameter) Default: Pitch x 0.6134 (60 degree thread).</p> <p>Thread Angle: A, thread angle, usually 60 degrees. Default: 0 degrees for a straight plunge thread.</p> <p># Fini. Pass: S, number of passes at final depth. Default: 1.</p> <p>X Pullout: R, incremental X-axis pullout distance. Creates a 2-axis pullout move. Default: 0, causes the tool to pullout at 90 degrees to the threading pass.</p> <p>Taper Z: Z, distance moved in the Z-axis during a threading pass. Default: 0 straight thread.</p> <p>Taper Angle: V, Angle of tapered thread. Default: 0.</p> <p>NOTE: When programming a tapered thread use Z or V parameters, but not both.</p> <p>Z Standoff: B, distance in the Z-axis from the tool tip to the work face. Default: Inch = 0.1 Default: MM = 2. (Tool must be positioned the Standoff distance away from the face in the block prior to the threading Canned Cycle.)</p>	<p>The G85 cycle creates a radial thread along the X-axis using multiple passes to reach full thread depth. The CNC moves from the Z standoff (B) to the face of the work, plus the first depth of cut (C), and completes the first pass. The tool returns to the start point, and subsequent passes are taken at reducing depths of cut until the completed depth is reached.</p> <p>The number of passes necessary to complete a thread is calculated by the CNC based on the Thread Depth (D), and the depth of the first pass (C).</p> <p>The thread may be cut by a plunge motion (A=0), or by a compound motion (A=Angle of cutting tool*.)</p> <p>A number of finish passes (S=# of passes taken at final depth) may be specified. Default: 1</p> <p>R creates a 2-axis pullout move. The tool moves the distance programmed (R) in the X-axis while retracting to the B standoff in the Z-axis.</p> <p>If the taper angle (Z) or distance (V) is specified the threading passes are completed at specified taper.</p> <p>The W parameter specifies the number of thread starts.</p> <p>A second use for the W parameter (programming a W-1) will cause all threading passes to be performed at the same depth of cut. D divided by C. Threads are normally cut in reducing depths of cut.</p> <p>*In the G85 cycle (uni-directional threading) if the A parameter is programmed, the cycle will perform the passes using a compound cutting motion. The tool enters the thread on an</p>

<p>Num. Starts: W, used to program the number of starts in a multiple start thread. Default: 1 (Single start thread).</p>	<p>angle (described by the A parameter), and cuts only on the leading edge of the tool.</p>
<div data-bbox="203 409 332 535" data-label="Image"> </div> <p>G86 Radial Bi-directional Threading</p> <p>TPI: E, Threads per inch. Used in Inch programming.</p> <p>Lead: F, pitch (single pass threading) or amount of travel in X per revolution (multi-pass threading). Used in MM programming.</p> <p>NOTE: Use E or F, but not both.</p> <p>Length: X, incremental distance in the X-axis from Tool start position to finish point (length of threading pass).</p> <p>First Cut: C, incremental depth of the first cut (depth of cut for first pass). (Does not include B parameter).</p> <p>Thread Depth: D, incremental depth of the thread (depth of thread). (Does not include B parameter) Default: Pitch x 0.6134 (60 degree thread).</p> <p>Thread Angle: A, thread angle, 60 degrees for standard threads. Default: 0 degrees for a straight plunge thread.</p> <p># Fini. Pass: S, number of passes at final depth. Default: 1.</p> <p>X Pullout: R, incremental X-axis pullout distance. Creates a 2-axis pullout move. Default: 0 causes the tool to pullout at 90 degrees to the threading pass.</p> <p>Z Standoff: B, distance in the Z-axis from the tool tip to the work face. Default: Inch = 0.1 Default: MM = 2</p> <p>Tool must be positioned the Standoff distance away from the face in the block prior to the threading Canned Cycle.</p> <p>Num. Starts: W, used only in multiple pass threading, the number of staggered starts used to reach depth. Default: 1.</p> <p>NOTE: Specifying Num. of Starts changes the way the CNC interprets</p>	<p>The G86 cycle creates a radial thread along the X-axis using multiple passes to reach full thread depth. The CNC moves from the Z standoff (B) to the face of the work, plus the first depth of cut (C), and completes the first pass. The tool returns to the start point, and subsequent passes are taken at reducing depths of cut until the completed depth is reached. The number of passes necessary to complete a thread is calculated by the CNC based on the Thread Depth (D), and the depth of the first pass (C). The thread may be cut by a plunge motion (A=0), or by a compound motion (A=Angle of cutting tool*.) A number of finish passes (S=# of passes taken at final depth) may be specified. Default: 1</p> <p>R creates a 2-axis pullout move. The tool moves the distance programmed (R) in the X-axis while retracting to the B standoff in the Z-axis.</p> <p>*In the G86 cycle (Bi-directional threading) if the A parameter is programmed, the cycle will perform the passes using a compound cutting motion. The tool enters the thread on an angle (described by the A parameter), and cuts alternating passes on the leading and trailing edges of the tool.</p>

Table 7-12, THREADING Help Template Menu (Continued)

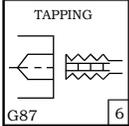
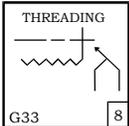
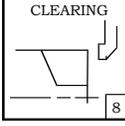
THREADING	
Templates and Parameters	Move Description
 <p>G87 Tapping Cycle Finish Depth: Z, Absolute depth of thread. Z Start: R, initial start position, in rapid. Feed./Pitch: F, feed in Inches per Revolution. Z Return: P, absolute Z return position.</p>	<p>The CNC feeds the tap from the initial start position (R) into a pre-drilled hole along the Z-axis. When the finish depth (Z) is reached, the spindle stops and reverses direction. The tap is then retracted in the Z-axis at the same feedrate and spindle speed.</p>
 <p>G33 Single-pass Thread Cutting X End Pt.: X end point. Z End Pt.: Z end point. TPI: E, Threads per inch. Used in Inch programming. Lead: F, pitch. Used in MM programming. NOTE: Use E or F, but not both. X Pullout: A, incremental X retract distance. Z Pullout: C, incremental Z retract distance.</p>	<p>The G33 command creates a longitudinal, radial or tapered thread using single pass threading. To obtain multiple passes use the G33 command in a Loop. The tool must be positioned at the first cutting depth prior to initiating the G33 command.</p> <p>For straight longitudinal threading, enter only the Z endpoint. For straight radial threading enter only the X endpoint. For tapered threading, enter both the X and Z endpoints.</p> <p>G33 command gives only one threading pass to the X, Z, or XZ endpoints. The tool retracts from the threading pass using the X, Z, or XZ pullout distances given by the A and/or C parameters. If no pullout dimensions are stated, the tool retracts at 90 degrees to the threading pass.</p>

Table 7-13, CLEARING Help Template Menu

 CLEARING	
Templates and Parameters	Move Description
<div style="display: flex; align-items: flex-start;"> <div style="margin-right: 10px;">  </div> <div> <p>G73 Area Clearance (Turn) Cycle</p> <p>Input Sub #: W, subprogram number in which the part profile is programmed. The subprogram is programmed after the end of the main program.</p> <p>Depth/Pass: A, depth of material in X to be removed (per side) on each roughing pass.</p> <p>X Finish Stk: R, amount of material to remain after roughing for a finish pass (X-axis).</p> <p>Z Finish Stk: S, amount of material to remain after roughing, for a finish pass (Z-axis).</p> <p>Finish Pass: C=0 (no finish pass taken) C=1 (a finish contour pass will be taken after the roughing passes are completed).</p> <p>OD=1, ID=0: B, 1 (completes passes along the Z-axis, away from the centerline) or 0 (completes passes along the Z-axis, toward the centerline). Required.</p> <p>Undercut: P, Use value of 1 if there are undercuts in the profile to be rough/finish cut. Leave unassigned if no undercuts are used.</p> <p>X In Feed.: I, plunge feedrate.</p> <p>Rough Feed.: J, feedrate for the roughing pass.</p> <p>Finish Feed.: K, feedrate for the finish pass.</p> </div> </div>	<p>The G73 Area Clearance Cycle rough turns a work piece to a profile described in a subprogram. The subprogram is named by a subprogram number using the O (letter O) parameter followed by a number 1–999. The W parameter in the G73 cycle calls the subprogram number.</p> <p>Roughing passes are made along the Z-axis, and may be either an OD profile (B=1) or an ID profile (B=0), the feedrate for the roughing passes is set by the (J) parameter. The amount of material (stock) to be removed by a finish pass is described by the parameters (R) for the X-axis, and (S) for the Z-axis. A finish feedrate may also be described by the (K) parameter.</p>

(Continued...)

Table 7-13, CLEARING Help Template Menu (Continued)

Templates and Parameters	Move Description
<div data-bbox="207 289 334 415" style="border: 1px solid black; padding: 2px; margin-bottom: 5px;"> </div> <p style="margin-left: 20px;">CLEARING</p> <div data-bbox="207 495 334 621" style="border: 1px solid black; padding: 2px; margin-bottom: 5px;"> </div> <p style="margin-left: 20px;">G74</p> <p>G74 Area (Face) Clearance Cycle Input Sub #: W, subprogram number in which the part profile is programmed. The subprogram is programmed after the end of the main program. Depth/Pass: A, depth of material in Z to be removed on each roughing pass. X Finish Stk: R, amount of material to remain after roughing for a finish pass (X-axis). Z Finish Stk: S, amount of material to remain after roughing for a finish pass (Z-axis). Finish Pass: C=0 (no finish pass taken) C=1 (a finish contour pass will be taken after the roughing passes are completed). OD=1, ID=0: B=1, (completes roughing passes in the X-axis towards the part centerline). B=0, (completes roughing passes in the X-axis away from the part centerline). Undercut: P, Use value of 1 if there are undercuts in the profile to be rough/finish cut. Leave unassigned if no undercuts are used. X In Feed.: I, plunge feedrate. Rough Feed: J, feedrate for roughing passes. Finish Feed: K, feedrate for finish pass.</p>	<p>The G74 Area Clearance Cycle rough faces a work piece to a profile described in a subprogram. The subprogram is named by a subprogram number using the O (letter O) parameter followed by a number 1–999. The W parameter in the G74 cycle calls the subprogram number.</p> <p>Roughing passes are made along the X-axis, and may be either toward part centerline (B=1) or away from part centerline (B=0), the feedrate for the roughing passes is set by the (J) parameter.</p> <p>The amount of material (stock) to be removed by a finish pass is described by the parameters (R) for the X-axis, and (S) for the Z-axis. A finish feedrate may also be described by the (K) parameter.</p>
<div data-bbox="207 1570 334 1696" style="border: 1px solid black; padding: 2px; margin-bottom: 5px;"> </div> <p style="margin-left: 20px;">LONGITUDINAL</p> <div data-bbox="207 1705 334 1831" style="border: 1px solid black; padding: 2px; margin-bottom: 5px;"> </div> <p style="margin-left: 20px;">G76</p> <p>G76 Rough Turning Cycle Total Length: Z, incremental Z distance from start to finish (length). Total Depth: X, incremental depth of material in X to be removed per side. (Excludes B standoff).</p>	<p>The G76 longitudinal cycle removes stock material along the Z-axis in a series of passes. The tool must be positioned appropriately (starting OD + 0.1" / 2mm per side for OD roughing, and starting ID -0.1" / -2mm per for ID roughing) prior to the canned cycle.</p> <p>The CNC calculates both the depth and number of passes required to rough turn the workpiece using the programmed parameters. The actual depth of each pass</p>

<p>Depth/Pass: C, maximum incremental depth of cut per side for each pass. Pull Out: A, incremental distance in Z from start position to beginning of pull out. X Standoff: B, incremental X standoff distance from part diameter per side. Default: Inch OD = 0.1 Default: Inch ID = -0.1 Default: MM OD = 2 Default: MM ID = -2 (Tool must be positioned the Standoff distance away from the diameter in the block prior to the G76 Canned Cycle). Pull In: D, incremental distance in Z from start position to the end of pull in. Tool Angle: E, rear angle of tool in reference to positive X direction. Must be a negative value.</p>	<p>will not exceed the maximum depth of cut (C). The rear angle of the tool (E) is checked against the angle of the pull in move to avoid interference. If the tool cannot cut the angle due to interference, an error message will be generated. G76 can be used for either OD or ID roughing, all parameters are incremental and sign (+ or -) sensitive. The G76 is a roughing cycle only, and no finish pass completed. If a finish pass is required, it must be programmed separately. Tool Nose Compensation is not supported in the G76 canned cycle.</p>
<div data-bbox="203 779 332 907" data-label="Image"> </div> <p>G77 Rough Turning Cycle (Facing, Back Facing) Total Length: X, incremental X distance from start to finish (length). Total Depth: Z, incremental depth of material in Z to be removed. (Excludes B standoff). Depth/Pass: C, maximum incremental depth of cut per pass. Pull Out: A, incremental distance in X from start position to beginning of pull out. Z Standoff: B, incremental Z standoff distance from part face. Default: Inch = 0.1 Default: MM OD = 2 (Tool must be positioned the Standoff distance away from the face in the block prior to the G77 Canned Cycle). Pull In: D, incremental distance in X from start position to the end of pull in. Tool Angle: E, incremental top angle of tool, referenced to the positive Z direction (Z-axis to trailing edge of tool). Must be a positive value.</p>	<p>The G77 Radial cycle removes stock material along the X-axis in a series of passes. The tool must be positioned appropriately (0.1" / 2mm from the face) prior to the canned cycle. The CNC calculates both the depth and number of passes required to rough face the workpiece using the programmed parameters. The actual depth of each pass will not exceed the maximum depth of cut (C). The rear angle of the tool (E) is checked against the angle of the pull in move to avoid interference. If the tool cannot cut the angle due to interference an error message will be generated. All G77 parameters are incremental and sign (+ or -) sensitive. The G77 is a roughing cycle only, and no finish pass is completed. If a finish pass is required, it must be programmed separately. Tool Nose Compensation is not supported in the G77 canned cycle.</p>

(Continued...)

Table 7-13, CLEARING Help Template Menu (Continued)

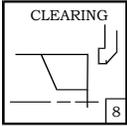
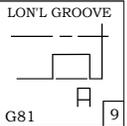
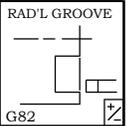
Templates and Parameters	Move Description
<div data-bbox="207 289 334 415" style="border: 1px solid black; padding: 2px; margin-bottom: 5px;">  </div> <p style="text-align: center;">CLEARING</p> <div data-bbox="207 495 334 621" style="border: 1px solid black; padding: 2px; margin-bottom: 5px;">  </div> <p>G81 Longitudinal (Diameter) Grooving Cycle</p> <p>Total Length: Z, total incremental length of groove.</p> <p>Total Depth: X, incremental depth of groove per side (excludes B standoff).</p> <p>Tool Width: W, tool width.</p> <p>X Standoff: B, incremental X standoff distance from part diameter per side. Default: Inch OD = 0.1 Default: Inch ID = -0.1 Default: MM OD = 2 Default: MM ID = -2</p> <p>(Tool must be positioned the Standoff distance away from the diameter in the block prior to the G81Canned Cycle).</p> <p>Dwell time: D, dwell time at bottom of the groove, in seconds.</p>	<p>The G81 cycle creates a groove along the Z-axis using the length (Z), and depth (X) parameters. The cycle automatically compensates for the width of the grooving tool.</p> <p>Prior to the cycle the tool must be positioned to the standoff position (0.1" / 2mm) from the diameter of the workpiece.</p> <p>When the tool reaches the total depth (X), it will dwell for the time programmed in parameter D, and then retract. If no dwell is programmed the tool will retract immediately. The CNC calculates the required number of grooving cuts and the width of the passes from the total length (Z). The tool returns to the starting coordinates after completion of the cycle.</p>
<div data-bbox="207 1199 334 1325" style="border: 1px solid black; padding: 2px; margin-bottom: 5px;">  </div> <p>G82 Radial (Facing) Grooving Cycle</p> <p>Total Length: X, total incremental length of groove.</p> <p>Total Depth: Z, incremental depth of groove (excludes B standoff).</p> <p>Tool Width: W, tool width.</p> <p>Z Standoff: B, incremental Z standoff distance from part face. Default: Inch = 0.1 Default: MM = 2</p> <p>(Tool must be positioned the Standoff distance away from the face in the block prior to the G81Canned Cycle.)</p> <p>Dwell time: D, dwell time at bottom of the groove, in seconds.</p>	<p>The G82 cycle creates a groove along the X-axis using the length (X), and depth (Z) parameters. The cycle automatically compensates for the width of the grooving tool.</p> <p>Prior to the cycle the tool must be positioned to the standoff position (0.1" / 2mm) from the face of the workpiece.</p> <p>When the tool reaches the total depth (Z), it will dwell for the time programmed in parameter D, and then retract. If no dwell is programmed, the tool will retract immediately. The CNC calculates the required number of grooving cuts and the width of the passes from the total length (X). The tool returns to the starting coordinates after completion of the cycle.</p>

Table 7-14, DRILL/TAP Help Template Menu

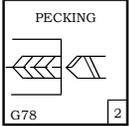
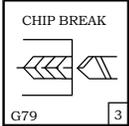
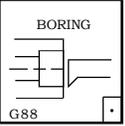
DRILL/TAP	
Templates and Parameters s	Move Description
 <p>G78 Peck Drilling Cycle Finish Depth: Z, Absolute depth of hole Z Start: R, Initial start position. Feedrate: F, feed in Inches per revolution Max Peck: I, incremental maximum depth per peck. Z Return: P, Absolute Z return position.</p>	<p>The Pecking cycle drills a hole using a pecking action. The depth of each peck remains the same, and therefore, this cycle should be used for relatively shallow holes.</p> <p>The CNC drills from the initial start position (R) into the workpiece to the calculated maximum peck depth, and then retracts from the hole to clear chips. The cycle, at a rapid feedrate, returns the drill to +0.01” from the previous hole depth, and feeds to the next peck position. The pecking continues until the Finish Depth (Z) is reached at which point the CNC will return the tool to the initial starting position, or to the (P) parameter if programmed.</p>
 <p>G79 Chip Break Cycle Finish Depth: Z, Absolute depth of hole Z Start: R, Initial start position. Feedrate: F, feed in Inches per revolution First Peck: I, first peck distance (length) Delta Peck: J, amount to be subtracted from previous peck length. Min. Peck: K, minimum depth per peck. Chp Brk. Inc: W, distance the drill retracts to break chip. Retract Dpt: U, incremental depth between full retracts to clear chips. Z Return: P, Absolute Z return position.</p>	<p>The Chip Breaking cycle drills a hole using a pecking action. The depth of each peck is reduced allowing for the drilling of deep holes without chip build up problems.</p> <p>The CNC drills from the initial start position (R) into the workpiece to the First peck depth, and then retracts by the Chip Break amount (W) to break chips. The cycle then feeds the drill to the next calculated peck depth (I-J). The pecking continues until the drill reaches the Retract Depth (U), or the Finish Depth (Z) at which point the CNC will return the tool to the initial starting position or to the (P) parameter if programmed.</p>
 <p>G87 Tapping Cycle Finish Depth: Z, Absolute depth of thread Z Start: R, Initial start position, in rapid Feed./Pitch: F, feed in Inches per revolution Z Return: P, Absolute Z return position</p>	<p>The CNC feeds the tap from the initial start position (R) into a predrilled hole along the Z-axis. When the finish depth (Z) is reached, the spindle stops, and reverses direction. The tap is then retracted in the Z-axis at the same feedrate and spindle speed.</p>

Table 7-14, DRILL/TAP Help Template Menu (Continued)

 DRILL/TAP	
Templates and Parameters s	Move Description
 <p>G88 Boring Canned Cycle (Inner Dimension cutting)</p> <p>Total Length: Z, incremental Z length</p> <p>Total Depth: X, incremental depth of material in X to be removed per side. (Excludes B standoff).</p> <p>Depth/Pass: C, maximum incremental depth of cut per side for each pass.</p> <p>Pull Out: A, incremental distance in Z from start position to beginning of pull out.</p> <p>X Standoff: B, incremental X standoff distance from part diameter per side.</p> <p>Default: Inch OD = 0.1 Default: Inch ID = -0.1 Default: MM OD = 2 Default: MM ID = -2 (Tool must be positioned the Standoff distance away from the diameter in the block prior to the G88 Canned Cycle.)</p>	<p>The G88 Boring cycle removes stock material along the Z-axis in a series of passes. The tool must be positioned appropriately (starting ID -0.1" / -2mm per side for ID roughing) prior to the canned cycle.</p> <p>The CNC calculates both the depth and number of passes required to rough bore the workpiece using the programmed parameters. The actual depth of each pass will not exceed the maximum depth of cut (C).</p> <p>G88 is used for ID roughing, all parameters are incremental and sign (+ or -) sensitive.</p> <p>The G88 is a roughing cycle only and no finish pass completed. If a finish pass is required, it must be programmed separately. Tool Nose Compensation is not supported in the G88 canned cycle</p>

Programming Modal G-Codes in Edit Help

Certain G-Codes define the way the CNC will interpret commands entered by an operator. These G-Codes set the CNC to interpret measurement units as Inch or Metric, numerical values as Absolute or Incremental, set tool movement to feed or rapid rate, set spindle speed to feedrate per minute or per revolution of the spindle. Refer to **Table 7-15**. Use Edit Help to set these types of modes.

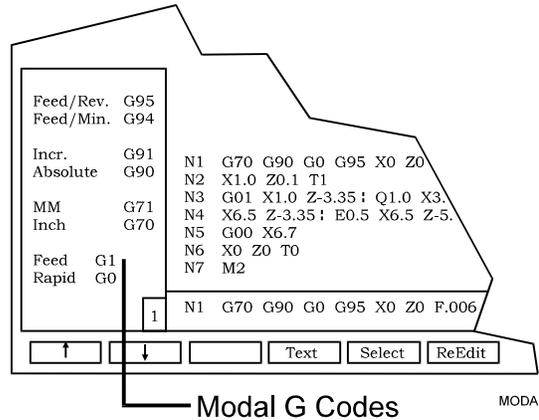


Figure 7-5, Modal G-Codes

Refer to **Figure 7-5**. The drawing shows a portion of the Edit Help Menu displaying Modal G-Codes.

When the Edit Menu is activated, the **Rapid G0** mode is highlighted. Use the **ARROWS** to highlight any of the displayed Modal G-Codes. Press **ENTER** or **Select (F5)** to insert the G-Code into the Program Listing.

Table 7-15, Most Common Modal G-Codes

G Code	Function
Rapid G0	Axis moves made at rapid rate.
Feed G1	Axis moves made at feedrate.
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.
Feed/Min. G94	Feedrates programmed in units per minute.
Feed/Rev. G95	Feedrates programmed in units per revolution.

Each of these G-Codes is described in a separate section elsewhere in the manual. Refer to the **Index** or **Contents**.

Programming LINE Moves in Edit Help

NOTE: Refer to [Table 7-6, Help Template Menus](#). This table gives a list of all Help Graphic Templates, and the Table location for the corresponding Input Labels and Entry Fields.

LINE moves can be vectored moves (motion in two axes, X and Z) or straight-line moves (motion in one axis, X or Z). LINE moves are programmed by entering values for angles, endpoints, and/or radii that define a LINE move. These features are found in the Edit Help Menu.

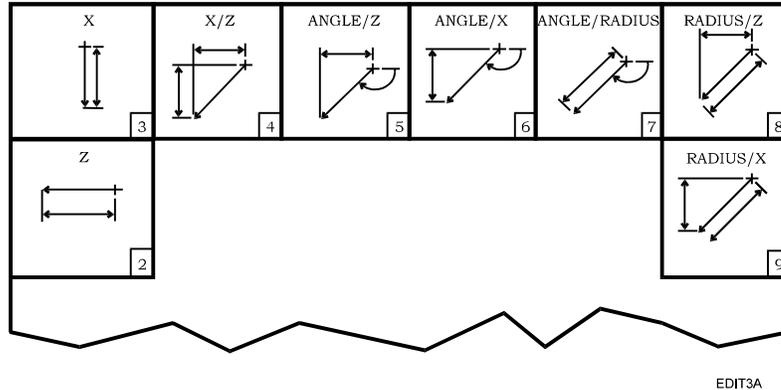


Figure 7-6, LINE Help Template Menu

Refer to **Figure 7-6**. The drawing shows the LINE Help Template Menu, which contains the various types of LINE moves that can be programmed in the Edit Help Menu. Activating a LINE Help Graphic screen allows the programmer to insert a program block into the Listing for the appropriate LINE move.

Refer to [Table 7-16, LINE Move Types](#). The table lists the types of LINE moves available in the LINE Help Template Menu. It describes the entry fields (information) that must be supplied by the programmer in order to program a LINE move.

Menu item numbers reference the LINE Help Template used to program the line move defined.

Table 7-16, LINE Move Types

LINE Move Types	Defined By
Vector	<ul style="list-style-type: none"> – An XZ endpoint (menu item 4) OR – An angle measured from the Z-axis (3 o'clock position equal to 0 degrees), and X or Z endpoint (menu item 5 or 6, respectively) OR – An angle measured from the Z-axis (3 o'clock position equal to 0 degrees), and the distance from the centerpoint of an arc to the circumference (radius) (menu item 7) OR – The distance from the centerpoint of an arc to the circumference (radius), and an X or Z endpoint (menu item 9 or 8, respectively).
Straight Line Moves	<ul style="list-style-type: none"> – An X or Z endpoint (menu item 3 or 2, respectively)

Angle Direction for Front and Rear Tool Posts

Refer to [Figure 7-7, Rear Tool Post Positive and Negative Directions](#) and [Figure 7-8, Front Tool Post Positive and Negative Directions](#). Angles can be used to program LINE moves or define lines used to program other types of moves.

The diagrams show the positive and negative directions of X coordinates and angles for front and rear tool posts. In the Incremental Mode, angles are measured from the Z-axis of the current position (3 o'clock or 0 degrees). In the Absolute Mode, angles are measured from the Z-axis of the Absolute Zero Reference (3 o'clock or 0 degrees).

The direction of positive and negative X-axis motion varies for front and rear tool post machines.

NOTE: All programming examples in this manual use Front Tool Post orientation.

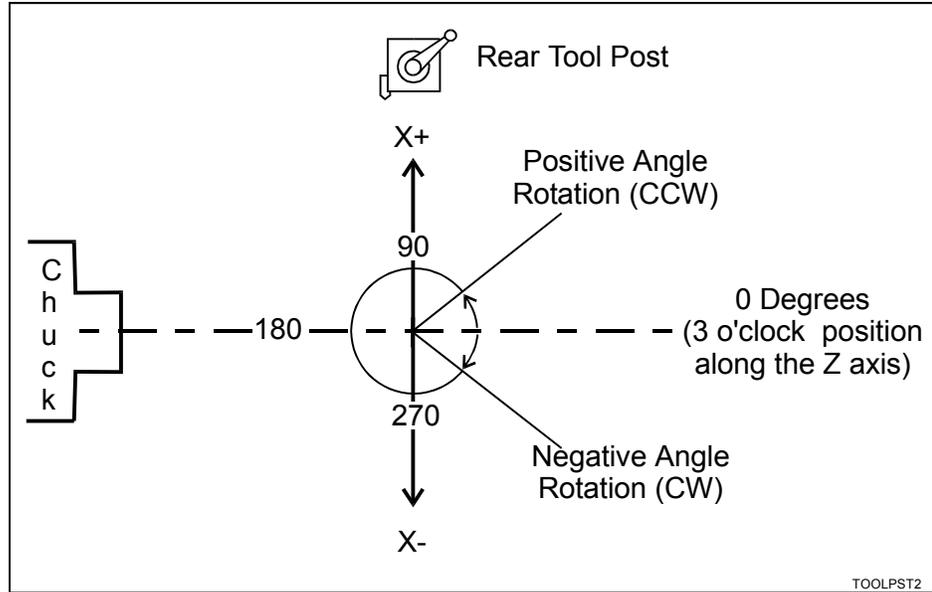


Figure 7-7, Rear Tool Post Positive and Negative Directions

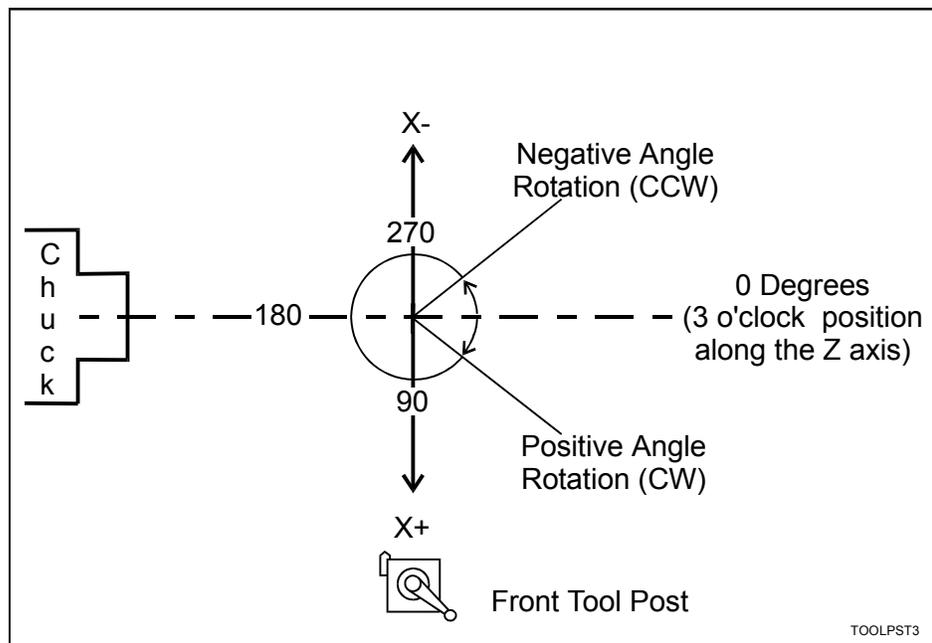


Figure 7-8, Front Tool Post Positive and Negative Directions

Using Angles to Program Moves

Refer to **Figure 7-9**. Angles can be used to program LINE Moves or lines used to define other types of moves.

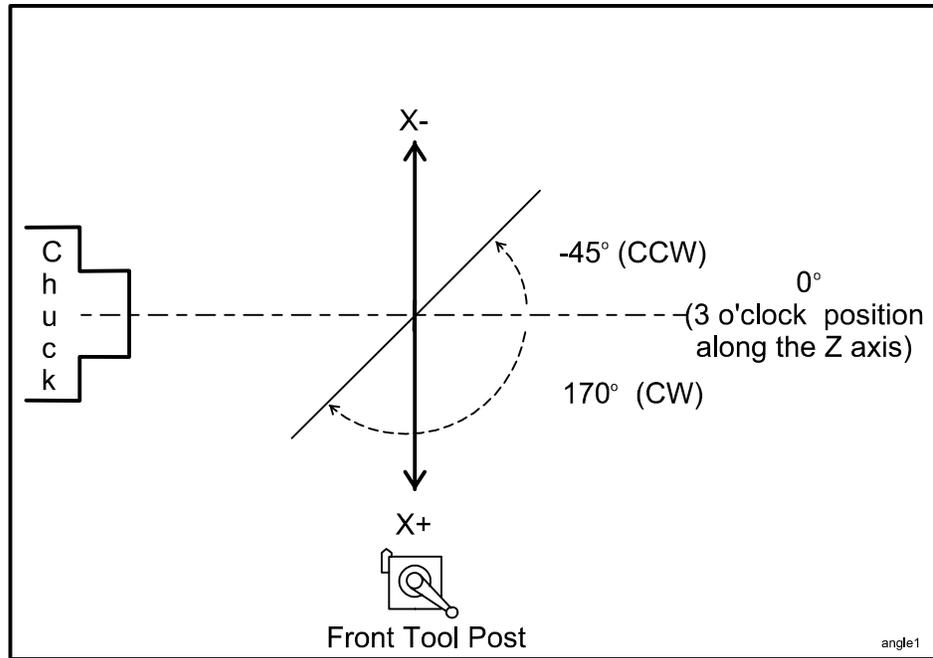


Figure 7-9, Using Angles to Program Moves

Refer to **Table 7-17**. The sign (+/-) of an entered angle depends on the direction of rotation (clockwise or counterclockwise) used. This varies for Front and Rear Tooling.

Table 7-17, Programming Positive/Negative Angle Direction

Front/Rear Tooling	Negative/Positive Angle	CW/CCW
Front	Negative	CCW
Front	Positive	CW
Rear	Negative	CW
Rear	Positive	CCW

In the Incremental Mode, angles are measured from the Z-axis of the current position. In the Absolute Mode, the angles are measured from the Z-axis of the Absolute Zero Reference.

Programming MULTIPLE Move Commands in Edit Help

Refer to **Figure 7-10**. The figure shows the MULTIPLE Help Template Menu, accessed through the Edit Help Menu.

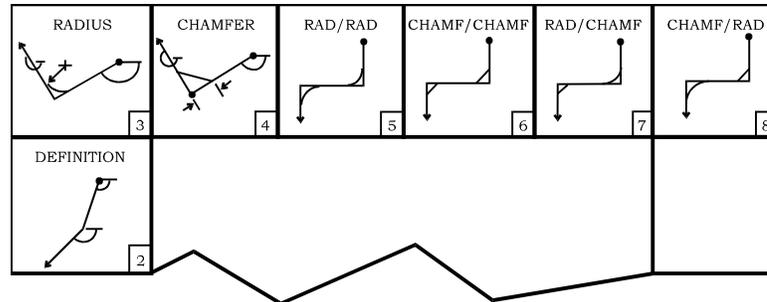


Figure 7-10, MULTIPLE Help Template Menu

MULTIPLE moves allow the programmer to input more than one move on a single program block. The permissible combinations include line, arc, and chamfer moves, in various combinations, detailed below:

- Two consecutive line moves (menu item 2)
- Line to arc to line moves (menu item 3)
- Line to chamfer to line moves (menu item 4)
- Line to arc to line to arc to line moves (menu item 5) line to chamfer to line to chamfer to line moves (menu item 6)
- Line to arc to line to chamfer to line moves (menu item 7)
- Line to chamfer to line to arc to line moves (menu item 8)

Menu item numbers reference the MULTIPLE Help Template used to program the MULTIPLE move described. Refer to [Table 7-11, MULTIPLE Help Template Menu](#) for definitions and input instructions regarding these moves.

Section 8 - Draw Mode

Introduction

The CNC has two Draw Modes:

- Draw Simulation Mode
- Real Time DRAW Mode.

NOTE: In this manual, **Draw** (with a capital D) refers to the CNC's Draw Simulation Mode. **DRAW** (all capitalized) refers to the CNC's Real Time Draw Mode. The same spelling convention is used on the soft key labels.

- In **Draw** Simulation Mode, the CNC runs programs to simulate machine movements in the viewing area. The machine itself does not move.
- In the Real Time **DRAW** Mode, the CNC displays the machine moves in the viewing area as it makes them.

This section of the manual explains how to use the Draw Simulation Mode to view programs. It also explains how to change the view settings to permit a detailed inspection of the programmed moves.

The viewing settings for both modes are changed from the Draw Simulation Mode soft keys. Refer to "[Section 11 - Running Programs](#)" for instructions on running DRAW while cutting a part.

Draw enables you to test run a program before cutting any material. In Draw, you can zoom in or out, and change scales, to better inspect the program.

NOTE: You can change to the display settings at any time in Draw Simulation Mode. The display settings cannot be changed in Real Time Draw Mode.

When **Draw** is started from the Program Directory, Draw Simulation Mode activates.

When **DRAW** is started in **Auto** or **S.Step Modes**, Real Time Draw Mode activates.

Starting Draw

Start Draw Simulation Mode from the Program Directory. The **Display (F5)** and **Parms (F9)** settings determine how Draw looks and runs. You can use the soft keys to make some changes during a program simulation. However, it is better to adjust settings before the simulation.

In Draw Simulation Mode, the CNC does not hold program operation for Dwells or tool mounts.

To activate Draw Simulation Mode:

1. In the Program Directory, highlight a program. Press **Draw (F7)**. The Draw Graphic Screen activates.
2. Press **Display (F5)**. A pop-up activates; **Fit** is already highlighted.
3. Press **ENTER**. **Fit** scales the image to fit in the viewing area.

The soft key menu changes to display the available simulation modes: **Auto (F1)**, **S.Step (F2)**, or **Motion (F3)**, and related settings. Choose an operation Mode. Auto Mode runs the entire program without pause. S.Step Mode executes the program a block at a time. Motion Mode executes the program a move at a time, without pausing between non-motion blocks. Draw runs the highlighted program, the tool path is displayed in viewing area and the machine remains idle.

Draw Screen Description

Refer to **Figure 8-1**.

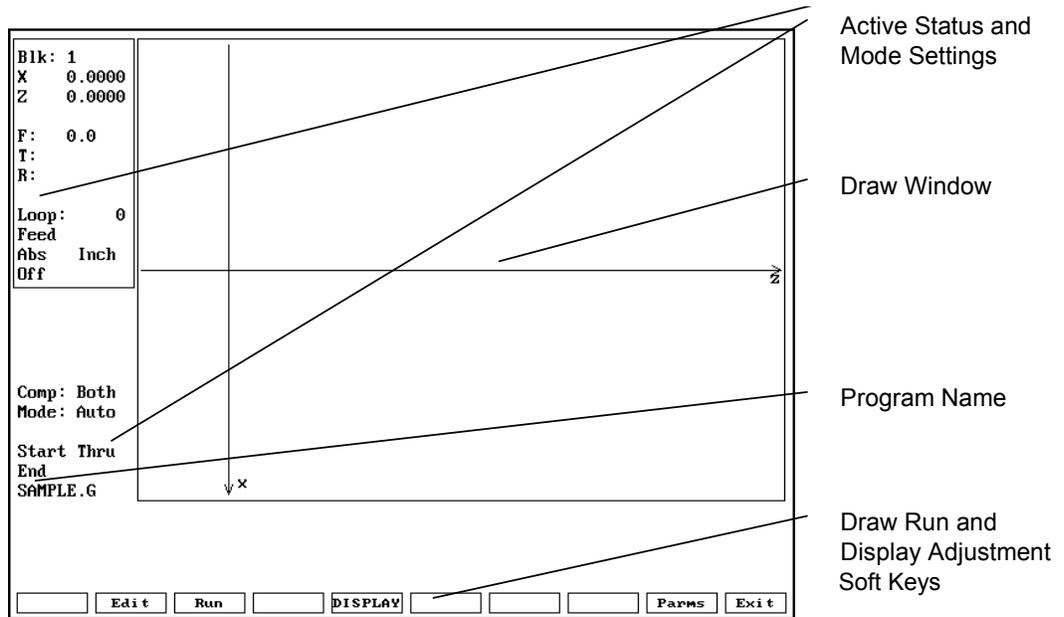


Figure 8-1, Draw Simulation Mode

The Draw screen displays the following information to the left of the Draw window:

Blk:	Block Number currently being simulated.
X	X Current Position.
Z	Z Current Position.
F:	Currently active feed rate.
T:	Currently active tool number.
R:	Currently active Tool Nose Radius.
Loop:	Loop countdown, if loop is programmed.
Rapid	Type of current movement (rapid, feed, arc CW, arc CCW).
Abs/Inc	Absolute/Incremental Mode active.
Inch/MM	Inch/MM Mode active.
Off/Left/Right	Active compensation, if any (off, left of path or right of path).
Comp:	Compensation setting (Both, Ignore or Use).
Mode:	Current operating mode: S.Step, Motion, or Auto.
Start and End Block	The range of the program that will be run in draw. Example: Start Thru End. (Based on Start N# and End N# in Params Pop-Up Menu.)
Program Name	Program Name.

Putting Draw in Hold

To pause Draw any time, press **Hold (F8)** or the **HOLD** key.

Restarting Draw When in Hold

When Draw is in hold, press **Start (F7)** or **START** to continue.

Canceling Draw

The execution of a program in Draw Simulation Mode can be canceled by pressing **Cancel (F9)**.

Draw Parameters

Refer to **Figure 8-2**.

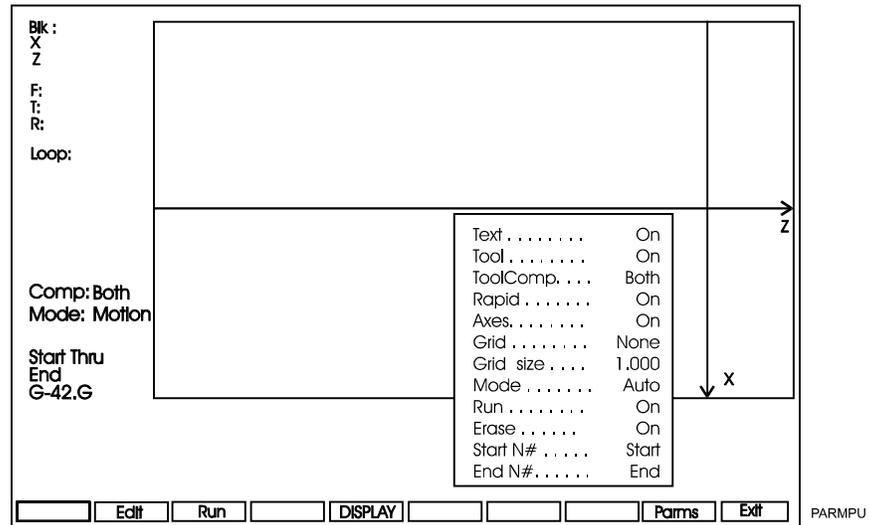


Figure 8-2, Draw Parameters Pop-up Menu

Draw shows the following features during simulation:

Feature	Displayed as
Rapid moves	Dotted Lines - when turned on
Feed moves	Solid Lines
Tool	Based on Tool Location Code and Radius, if turned on

Viewing parameters are set two ways. Before the program is run, set the parameters from the **Parms (F9)** Pop-Up Menu. While the program is running, change the parameters from the soft key menu. Active soft key features are highlighted.

Tool On or Off

Run Draw with **Tool On** to display a drawing of the tool as it moves through the part. A tool must be active and have a Radius and Tool Location Code entered on the Tool Page in order for the CNC to display it. The displayed tool is the shape of the Tool Location Code. The size of the tool is scaled to the tool's Radius. Draw runs faster with **Tool Off**.

The default setting is **On**.

To switch **Tool On** or **Off**:

1. With the Draw Mode active, press **Parms (F9)**. The Parameter Pop-Up Menu activates.
2. Highlight **Tool**, and press **ENTER**. The **Tool** setting switches **On** and **Off**.
3. Press **Parms (F9)**. The **Parms** pop-up closes.

NOTE: To change the **Tool** parameter while Draw is running, press **Tool (F5)**.

Drawing Compensated Moves

The **ToolComp** setting determines if and how Draw displays compensated moves. This lets you see the effects of compensation on the moves in a program. The Draw **ToolComp** setting does not affect program execution to cut a part.

There are three **ToolComp** options:)

- **Ignore** a program's compensated moves.
- **Use** (display a program's compensated moves.
- **Both** runs the program twice. (First without compensation then with compensation.) The moves from both are displayed simultaneously.

Both is the only selection that enables a direct comparison between the two options. [Default: **Both**]

To set the compensation parameter:

1. With the Draw Mode active, press **Parms (F9)**. Parameter Pop-Up Menu activates.
2. Highlight **ToolComp**, and press **ENTER**. A second pop-up activates.
3. Move the highlight to the desired selection, and press **ENTER**. The smaller pop-up closes.
4. Press **Parms (F9)**. **Parms** pop-up closes.

Showing Rapid Moves

Draw displays Rapid moves as dotted lines. You can turn the dotted lines off to eliminate screen clutter. This parameter does not affect program execution, only the display. [Default: **On**]

To switch **Rapid On** or **Off**:

1. With the Draw Mode active, press **Parms (F9)**. The CNC displays parameter Pop-Up Menu.
2. Highlight **Rapid** and press the **ENTER**. **Rapid** switches **On** and **Off**.
3. Press **Parms (F9)**. **Parms** pop-up closes.

NOTE: To change the **Rapid** parameter while Draw is running, press **Rapid (F6)**.

Setting Grid Line Type

Draw can display a two dimensional grid with dots or solid lines. The default **Grid** setting is **None**.

To set the **Grid** parameter:

1. With the Draw Mode active, press **Parms (F9)**. The parameter Pop-Up Menu activates.
2. Highlight **Grid**, and press **ENTER**. **Grid** Pop-Up Menu activates.
3. Move the highlight bar to the desired selection, and press **ENTER**. **Grid** Pop-Up closes.
4. Press **Parms (F9)**. **Parms** Pop-Up closes.

Setting Grid Size

You can adjust the grid size. The units are determined by the CNC's current mode. [Default: **1.00**]

To set the **Grid size**:

1. With the Draw Mode active, press **Parms (F9)**. **Parameter Pop-Up Menu** activates.
2. Highlight **Grid size**, and press **ENTER**. The CNC displays the number entry field.
3. Type desired size, and press **ENTER**. The new value displays on pop-up menu.
4. Press **Parms (F9)**. The **Parms Pop-Up** closes.

Putting Draw in Motion, S.Step, or Auto Mode

The Draw Simulation Mode executes programs three ways:

- In the Automatic Mode, fully automatic (**Auto**)
- In the Single Step Mode, one program block at a time (**S.Step**)
- In the Motion Mode, one move at a time (**Motion**)

In the Automatic Mode (**Auto**), blocks are executed sequentially without pause until the program is finished, the CNC is put in a hold, or an error stops the program.

NOTE: Press **Auto (F1)** to switch to Auto Mode.

In the Single Step Mode (**S.Step**), one block of the program is executed each time you press **START**. This enables you to run the program one block at a time.

NOTE: Press **S. Step (F2)** to switch to Single Step Mode.

In the Motion Mode (**Motion**), a program is executed from one motion to the next, executing one motion each time you press **START**.

NOTE: Press **Motion (F3)** to switch to Motion Mode.

Select the default mode as follows:

1. With the Draw Mode active press **Parms (F9)**. The **Parameter Pop-Up Menu** activates.
2. Highlight **Mode**, and press **ENTER**. **Mode Pop-Up Menu** activates.
3. Highlight the desired mode, and press **ENTER**. **Mode Pop-Up** closes.
4. Press **Parms (F9)**. **Parms Pop-Up** closes.

NOTE: Off-line keyboard users running Draw in the Single Step or Motion Modes can use the **SPACEBAR** to continue the program after each hold.

Automatic Draw Restart

The **Run** parameter determines whether Draw automatically restarts after a **DISPLAY** setting change. This enables you to make more than one setting change before restarting Draw.

When **Run** is **Off**, Draw starts when you press **Run (F3)**.

When **Run** is **On**, Draw starts automatically after each **DISPLAY** change.

[Default: **On**]

1. With the Draw Mode active, press **Parms (F9)**. Parameter Pop-Up Menu activates.
2. Highlight **Run**, and press **ENTER**. **Run** switches **On** and **Off**.
3. Press **Parms (F9)**. **Parms** Pop-Up closes.

Erasing the Draw Display

The **Erase** parameter sets Draw to clear the display when it starts a program. When **Erase** is **Off**, the old drawing remains in the viewing area and new moves are drawn over it. The default setting is **On**. Set the **Erase** parameter as follows:

1. With the Draw Mode active, press **Parms (F9)**. Parameter Pop-Up Menu.
2. Highlight **Erase**, and press **ENTER**. **Erase** switches **On** and **Off**.
3. Press **Parms (F9)**. **Parms** Pop-Up closes.

Running Draw for Selected Blocks

NOTE: Program blocks must have block numbers to use this feature.

Draw can run any portion of a program or a subprogram. Run a portion of a program by specifying the **Start N#** and **End N#** block settings on the **Parms** menu. The default settings are **Start** and **End**.

To run a portion of a subprogram, both the starting and ending blocks must be within the subprogram. To run an entire subprogram, select starting and ending blocks that include the subprogram call from the main program.

If a starting block is inside a subprogram and the ending block is in the main program, the CNC will stop and generate an error message at the end subprogram block, because it cannot find the subprogram start block.

Starting Draw at a Specific Block

1. With the Draw Mode active, press **Parms (F9)**. Parameter Pop-Up Menu activates.
2. Highlight **Start N#**, and press **ENTER**. **Start N#** Pop-Up Menu activates.
3. Move the highlight to mark the desired selection, and press **ENTER**. If **Start Of Program** is selected, Draw will start at the first block of the program. If **Other Block** is selected, type the block number, and press **ENTER**.
4. Press **Parms (F9)**. **Parms** Pop-Up closes.

Ending Draw at a Specific Block

1. With the Draw Mode active, press **Parms (F9)**. Parameter Pop-Up Menu activates.
2. Highlight **End N#**, and press **ENTER**. **Start N#** Pop-Up Menu activates.
3. Move the highlight to mark the desired selection, and press **ENTER**. If **End Of Program** is selected, Draw will stop at the last program block. If **Other Block** was selected, type the block number, and press **ENTER**.
4. Press **Parms (F9)**. **Parms** pop-up closes.

Adjusting Draw Display

Draw has several display settings to enable you to have the best view of the moves in the viewing window. Refer to **Figure 8-3**. Activate these settings from the **Display (F5)** Pop-Up Menu.

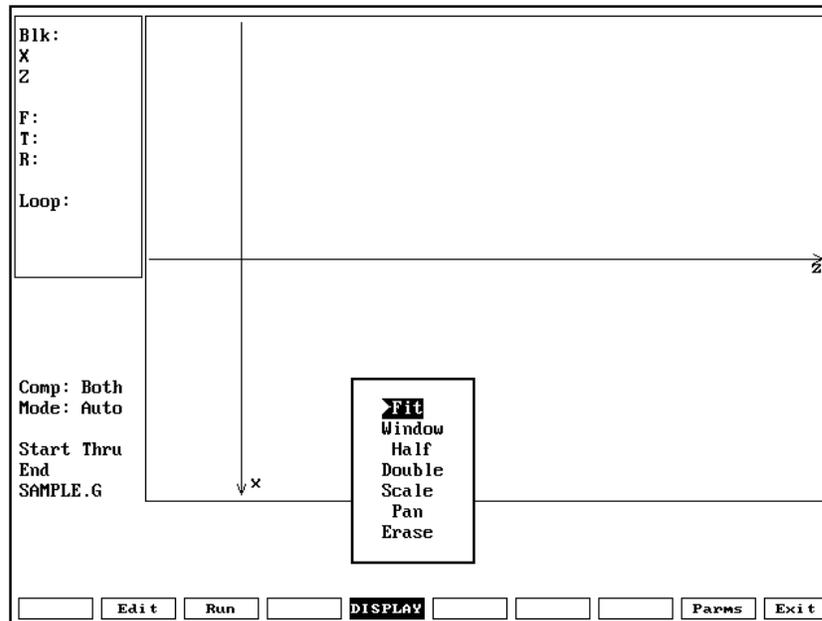


Figure 8-3, Display Pop-up Menu

Fitting the Display to the Viewing Window

Draw can automatically scale the display to fit in the viewing area.

To fit the display to the viewing area:

1. In Draw Mode, press **Display (F5)**. A pop-up activates.
2. Highlight **Fit**, and press **ENTER**. The pop-up closes and the display adjusts to fit in the viewing window.

Halving Display Size

Draw can reduce the size of the display to half the existing size.

To reduce the display size by one-half:

1. In Draw Mode, press **Display (F5)**. A pop-up menu activates.
2. Highlight **Half**, and press **ENTER**. The pop-up closes. The next time Draw runs, the display will be one-half its present size.

Doubling Display Size

Draw can double the size of the display.

To double the size of the display:

1. In Draw Mode, press **Display (F5)**. A pop-up menu activates.
2. Highlight **Double**, and press **ENTER**. The pop-up closes. The next time Draw runs, the display will be twice its present size.

Scaling the Display by a Factor

Draw can enlarge or reduce the display size.

To scale the Draw display:

1. In Draw Mode, press **Display (F5)**. A pop-up menu activates.
2. Highlight **Scale**, and press **ENTER**. The pop-up closes. The CNC prompts for a scale factor.

NOTE: Type a decimal value to reduce the size. Type a whole number to enlarge the size.
--

3. Type the desired factor, and press **ENTER**. The prompt disappears. The next time Draw runs, the display will be scaled by the specified factor.

Using the Window Zoom

Refer to **Figure 8-4**.

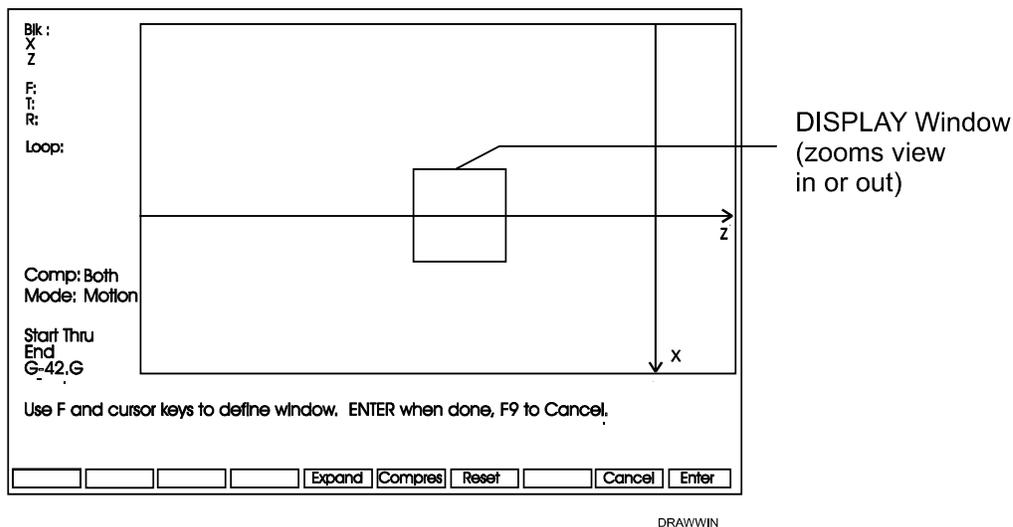


Figure 8-4, Display Window (Zoom)

Draw lets you zoom in on any portion of the display.

To zoom in on a portion of the display:

1. In Draw Mode, press **Display (F5)**. A pop-up menu activates.
2. Highlight **Window**, and press **ENTER**. A white window appears inside of viewing window.
3. Using the **ARROWS**, center the window over the area of interest.
4. To enlarge or reduce the window, press **Expand (F5)** or **Compress (F6)**.

NOTE: Press **Reset (F7)** to restore window to its original size. Press **Cancel (F9)** to cancel the Window command.

5. Once the window is sized and positioned, press **Enter (F10)**. The window closes. The next time you run **Draw**, the portion of the display framed by the window will fill the viewing window.

Erasing Display

To erase the display:

1. In Draw Mode, press **Display (F5)**. The **Display Pop-Up Menu** activates.
2. Highlight **Erase**, and press **ENTER**. The display is erased.

Changing the Viewing Area without Changing the Scale

Refer to **Figure 8-5**.

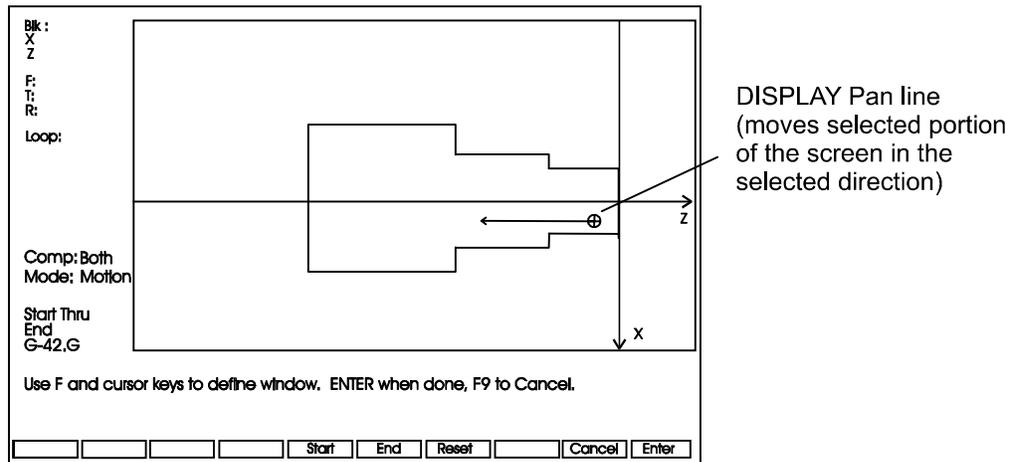


Figure 8-5, Display Pan

To shift a portion of the screen in a desired direction without changing the scale factor, use the Draw **Pan** command. This is especially useful on long parts which do not entirely fit in the Draw window.

When you activate the Pan command from the DISPLAY pop-up menu, the Pan line appears on the screen and the soft keys change. Press **Start (F5)** to place the beginning of the Pan line (circle) on the portion of the screen to be shifted. Press **End (F6)** to point the End of the Pan line (arrow) in the direction and distance the screen will be shifted.

To change the viewing area without changing the Scale factor:

1. In Draw Mode, press **Display (F5)**. A Display pop-up menu activates.
2. Highlight **Pan**, and press **ENTER**.
3. The Pan line appears on the screen. Press **Start (F5)** and use the **ARROWS** to place the start of the Pan line (circle) on a portion of the screen.
4. Press **End (F6)** and use the **ARROWS** to place the tip of the arrow to indicate the appropriate direction and distance in which the view will be shifted.
5. Press **Enter (F10)**. The CNC shifts the selected portion of the screen in the selected direction.

NOTE: Press **Reset (F7)** to restore the graphic to its original size (size when **Pan** selected). Press **Cancel (F9)** to cancel the Pan command.

Section 9 - Tool Page and Tool Management

This section discusses entering information into the Tool Page, including Tool Nose Radius compensations (G40, G41 and G42) and Tool-Length Offsets for X and Z.

The Tool Page is used by the CNC as a storage area for information on the tools used in a program. The user enters the following information:

- Tool nose radius value
- X and Z Tool-Length Offsets
- X and Z tool wear offsets
- Tool nose location codes

Information can be entered for up to 99 tools. You must enter relevant information before running a program for part production so that the CNC can correctly position the axes.

Activating the Tool Page

Activate the Tool Page as follows:

1. Go to the Manual screen.
2. From the Manual screen, press **Tool (F9)**. The Tool Page activates.

Using the Tool Page

Refer to [Figure 9-1, Tool Page](#). Activating Tool #0 (T0) cancels all Tool-Length Offsets (TLO's) and Tool Nose Radius Compensations. The tool change position from which TLO's are measured is usually Machine Home.

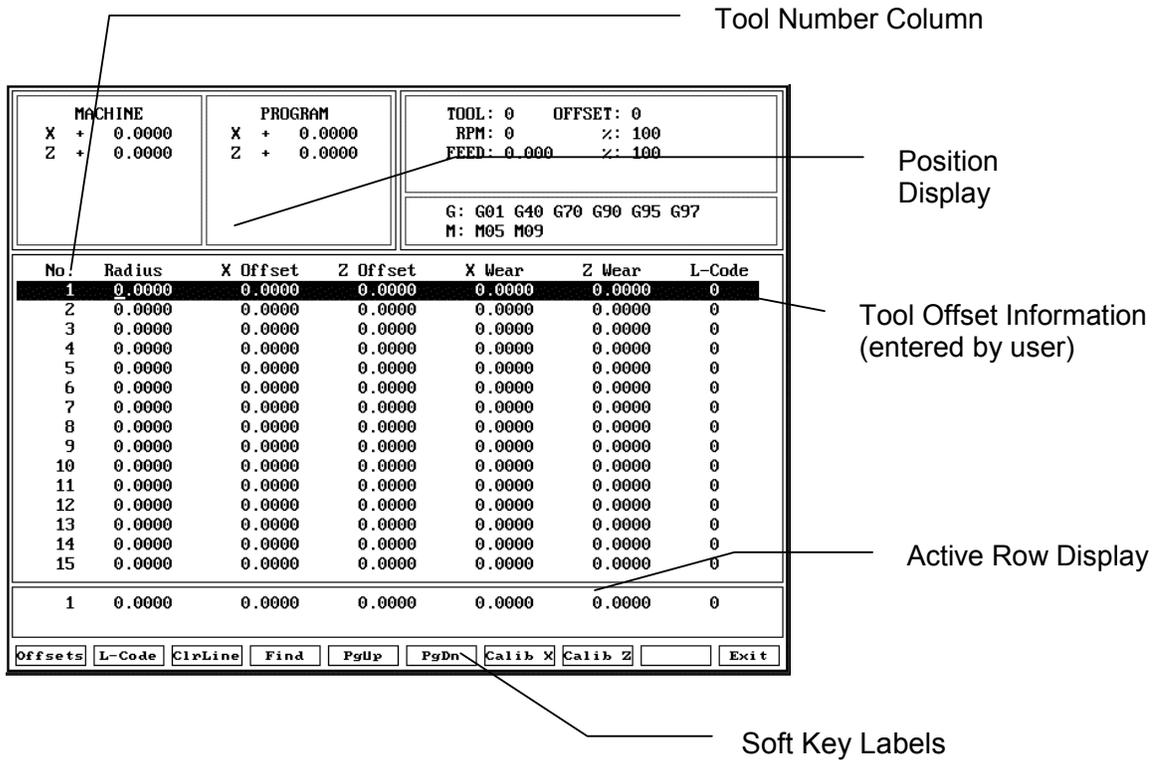


Figure 9-1, Tool Page

All of the CNC's Jog features can be run from the Tool Page. The handwheels (if present) can also be used.

The following features appear on the Tool Page.

No. 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.

Position Display Gives information regarding current machine position and active Units Mode (Inch/MM).

Radius Tool Nose Radius Offset applied when G41 (left of path) or G42 (right of path) compensation is activated.

X Offset X Tool-Length Offset applied at tool activation.

Z Offset Z Tool-Length Offset applied at tool activation.

X Wear X wear offset applied at tool activation.

Z Wear Z wear offset applied at tool activation.

L Code Tool location code (0-8).

Active Row Display Shows the row where the cursor is located.

Soft Key Labels Identify the functions of the active soft keys.

Entering Values in the Tool Page

The row numbers shown on the Tool Page correspond to Tool Numbers. When the CNC executes a program block that activates a tool number, the values entered on that row of the Tool Page are also activated.

Tool information can only be entered or changed in the row containing the cursor. The information in the row containing the cursor will be displayed at the bottom of the screen in the **Active Row Display**.

To enter a value:

1. Use the up and down **ARROW** keys to position the cursor and select a tool (row) number.
2. Use the left and right **ARROW** keys to position the cursor in the appropriate column, enter the value, and press **ENTER**. The CNC accepts the entered value.

Tool page values are automatically converted to their Inch/MM equivalents when the active CNC Units Mode changes. Enter values in the active Units Mode.

CAUTION: Be sure the CNC is set for the units (Inch or MM) being entered. The active Units Mode is displayed in the Position and Status Display area of the Tool Page.

Press **PgUp (F5)** or **PgDn (F6)** to scroll through the Tool Page.

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, "**Enter adjustment 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.

Finding Tools by Number

To find a specific tool number in the Tool Page:

1. Press **Find (F4)**. The CNC displays the following prompt, "**Enter tool number:**" and the cursor appears.
2. Type a tool number and press **ENTER**. The cursor moves to the selected tool number.

Clearing Tool Information (Delete Row)

To clear a row:

1. Go to the Tool Page and position the cursor at the row being cleared.
2. Press **CirLine (F3)**. All values in the row return to zero.

Clearing a Value from a Column

To clear a value from a column:

1. Go to the Tool Page and move the cursor to the appropriate tool number. Using the **ARROW** keys to go to the column that will be cleared.
2. Press **CLEAR**. The value reverts to 0.0000 (Inch or MM).

Clearing the Whole Tool Page

To clear all values stored in the Tool Page:

1. Go to the Tool Page and press **CirTabl (SHIFT + F3)**.
2. The CNC displays the following prompt, "**Sure you want to clear entire tool-table?**" Choose **Yes (F1)**. All values in the Tool Page revert to 0.0000 (Inch or MM).

NOTE: Choose No (F2) or Cancel (F9) to cancel the command.

Tool-Length Offsets

A Tool-Length Offset is the distance from the tip of the tool, with the tool at the Machine Home position, to the axis Part Zero position.

TLO's are used to preset each tool to the same position on the component (Part Zero). In Absolute Mode, all X and Z dimensions are measured from this position. In X, Part Zero is usually the centerline of the part. In Z, Part Zero is usually the front face of the part. TLO's also give the direction of the offset, usually negative.

Refer to [Figure 9-2, Tool-Length Offsets](#). In the figure:

X Tool-Length Offset = -6 inches

Z Tool-Length Offset = -4 inches

After these values are entered into the Tool Page under the appropriate Tool number, they may be activated in the program.

When a tool number is activated, the entered Tool-Length Offsets are applied. This makes it possible to adjust the offset values without altering the program.

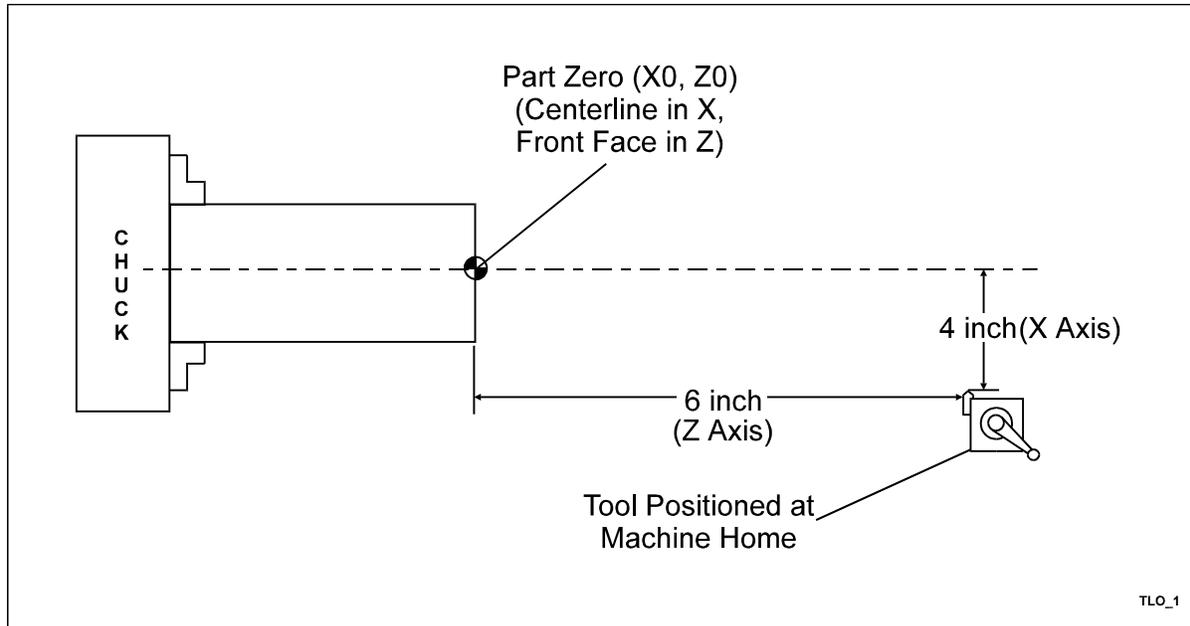


Figure 9-2, Tool-Length Offsets

Refer to **Table 9-1**. The T code used to activate a tool number consists of the Word Address T and the corresponding tool number. T codes can be programmed alone on a block or with axis commands, M-codes and other data.

Table 9-1, Programming T Codes

Block #	T Code	Additional Commands
N4	T1	
N50	T14	X1Z1
N16	T0	

T(n) activates the tool's offsets, T0 cancels all tool offsets and Machine Home becomes the reference point.

Setting Tool-Length Offsets

NOTE: Each tool used in a program should have Tool-Length Offsets. The Tool Page allows up to 99 tool numbers.

After selecting the types of tools and the order in which they will be used, determine the X and Z Tool-Length Offsets and enter them into the Tool Page. The Tool-Length Offsets establish Part Zero (X0, Z0) in reference to each tool number.

There are two methods for setting TLO's, using the X and Z Calibration functions, or entering the TLO's manually.

NOTE: Both the Jog keys and handwheels (optional) can be used to move the machine while the Tool Page is active.

NOTE: If desired, a skim cut can be made prior to determining the Tool-Length Offset. In Z, the skim cut is a shallow face cut to ensure an even work surface. In X, the skim cut can be made to ensure a consistent diameter.

Entering Tool Page TLO's Using the Calibration Function

Ensure that Tool 0 is in effect when Tool-Length Offsets are being set.

To enter X and Z Tool-Length Offsets using the calibration function:

1. Load the tool in the holder and position the tool at Machine Home.
2. Move the machine in Z until the tip of the tool rests against the face of the part.
3. In the Tool Page, place the cursor on the appropriate tool number.
4. Press **Calib Z (F8)**. The CNC stores the current Z position (distance from Machine Home to part face) as the **Z Offset**.
5. Move the machine in X until the tool clears the part, and then jog the machine until the tip of the tool rests against the part outer diameter (inner diameter for boring tools).
6. In the Tool Page, place the cursor on the appropriate tool number.
7. Press **Calib X (F7)**.
8. The CNC displays the following prompt, "**Enter Part Diameter:**" Type the diameter of the part (or bore). Press **ENTER**.
9. The CNC divides the diameter by two and subtracts that amount from the current X position. This is the **X Offset**.
10. Move off the outer diameter in X.
11. Repeat the procedure for all tools used in the program.

Entering Tool Page TLO's Manually

To enter X and Z Tool-Length Offsets manually:

1. Load the tool in the holder and position the tool at Machine Home.
2. Move the machine in Z until the tip of the tool rests against the face of the part.
3. In the Tool Page, place the cursor on the appropriate tool number. Go to the **Z Offset** column.
4. Type in the Z Tool-Length Offset (usually current Z position, including direction).
5. Move the machine in X until the tool clears the part, and then jog the machine until the tip of the tool rests against the part outer diameter (inner diameter for boring tools).
6. In the Tool Page, place the cursor on the appropriate tool number. Go to the **X Offset** column. The X Offset is half the part diameter plus the current X position. Type the calculated X Offset value (entered as negative number).
7. Move off the outer diameter in X.
8. Repeat the procedure for all tools used in the program.

Tool Offset Modification

Refer to **Table 9-2** and **Table 9-3**.

Table 9-2, Tool Offset Address Words

Address Word	Description
Q	X offset
I	X wear offset
L	Z offset
K	Z wear offset
P	Tool Nose Location Code
R	Tool Nose Radius
H	Indicates that the Tool Page values should be updated per values entered in the program.

Table 9-3, Tool Offset Modification via the Program

Format	Description
T1 Q-2.480 I.0005 L-3.9950 K.0015 R.0312 P3	Changes Tool No.1's X and Z offsets to (X)-2.480 (X wear offset 0.0005) and (Z)-3.9950 (Z wear offset 0.0015) and will not update the tool table for Tool 1.
T1 Q-2.480 I.0005 L-3.9950 K.0015 H	Changes Tool No.1's X and Z offsets to (X)-2.480 (X wear offset 0.0005) and (Z)-3.9950 (Z wear offset 0.0015) and will update the tool table for Tool 1.

NOTE: If no radius (R) or Tool Location Code (P) are given in the block, the CNC uses the radius and Tool Location Code given in the Tool Page for the active tool.

The Tool-Length Offset may be modified by the user through the program without using the Tool Page. This modification can be either temporary or permanent. The user decides whether to update the Tool Page.

The Q and L values are the tool offsets applied. It is not a value to be added to the current offset, but it is a new offset. The I and K wear offset values are used to make small adjustments to the X length and Z length offsets, respectively. (See "[Wear Offsets](#)" in this section for more detailed information.) The H address word forces the Tool Page to be updated to the programmed values.

Activating a Tool Number in the Manual Mode

NOTE: Make sure the correct offsets have been entered in the Tool Page before manually activating a tool.

To activate a tool number in the Manual Mode:

1. Go to the Manual screen.
2. Type **T**, followed by the appropriate tool number. (Example: T1)
3. Press **START**.

Deactivating TLO's in the Manual Mode

To deactivate a tool number in the Manual Mode:

1. Go to the Manual screen.
2. Type **T0**.
3. Press **START**.

Tool Nose Radius (TNR) Compensation

CAUTION: Tool Nose Radius compensation must be used when cutting arcs and angles or the part geometry will not be correct. Be sure to enter the correct Tool Nose Radius for each tool number in the Tool Page.

NOTE: Tool compensation cannot be used with some shapes (For example, the ellipse.) because of its effect on the geometry.

NOTE: You must program a tool number prior to G41/G42.

NOTE: Tool compensation is not active in Manual mode.

The CNC performs look ahead Tool Nose Radius (TNR) compensation. Program the actual part dimensions, and the control calculates all correct tangent points based on the Tool Nose Radius value, the Tool Location Code and the direction of travel.

Refer to [Figure 9-3, Tool Nose Radius](#). All forms of turning and boring tools have some form of radius on the very cutting edge of the tool. Though the radius may be quite small, when a cutting tool is used for machining contours on a workpiece, the radius may be sufficient to cause a deviation from the required shape of the finished work.

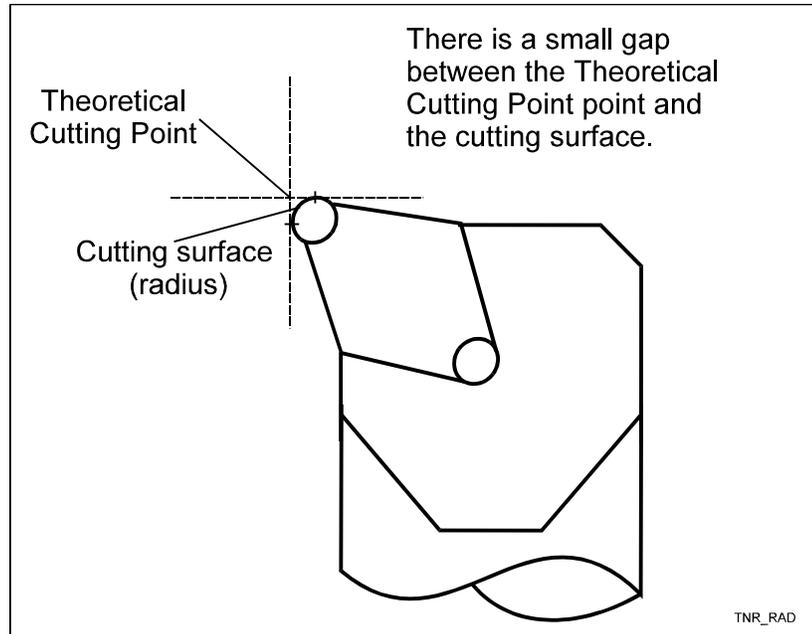


Figure 9-3, Tool Nose Radius

When tool compensation is not active, the CNC positions the tool's Theoretical Cutting Point on the programmed path. This creates a problem when programming a contoured part because Tool Nose Radius will introduce an error on angular and arc moves. This problem is overcome by using Tool Nose Radius compensation.

This enables you to program the moves that cut the surface of a part by writing moves that follow the part's profile.

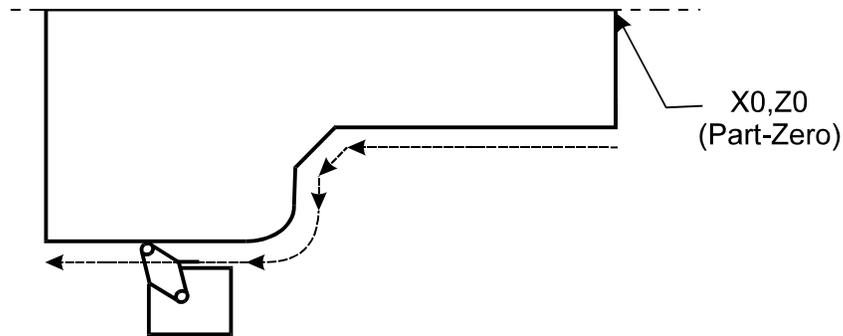
Most moves can be compensated. You must specify right or left compensation. Right or left refers to the side of the part the tool is on, looking from behind the tool as it moves.

The endpoints of a compensated move are offset perpendicular to the programmed endpoint.

There are three G-codes associated with TNR compensation. (Refer to **Table 9-4.**)

Table 9-4, Tool Nose Radius G-Codes

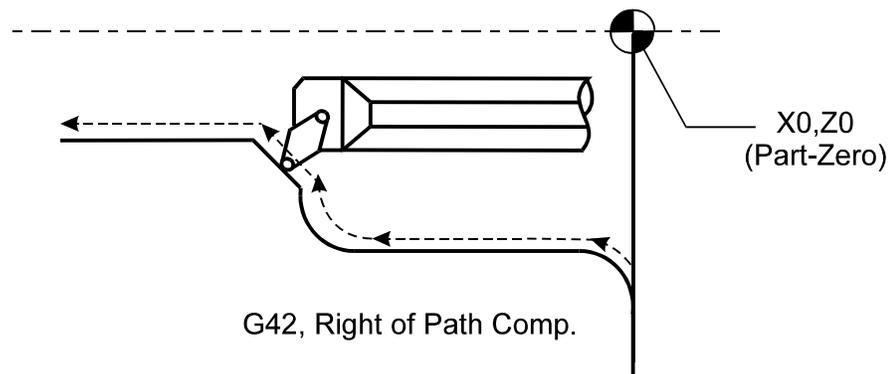
Format	TNR Function
G40 Xn Zn	Cancels TNR compensation.
G41 Xn Zn	Activates compensation to the left of the tool path.
G42 Xn Zn	Activates compensation to the right of the tool path.



G41, Left of Path TNR Comp.

Figure 9-4, Left of Path Compensation (G41)

Refer to **Figure 9-4**. When left hand tool compensation is activated, the tool offsets to the left of the programmed path (looking from behind the tool as it moves).



G42, Right of Path Comp.

Figure 9-5, Right of Path Compensation (G42)

Refer to **Figure 9-5**. When right hand tool compensation is activated, the tool offsets to the right of the programmed path (looking from behind the tool as it moves).

Selecting a Tool Location Code

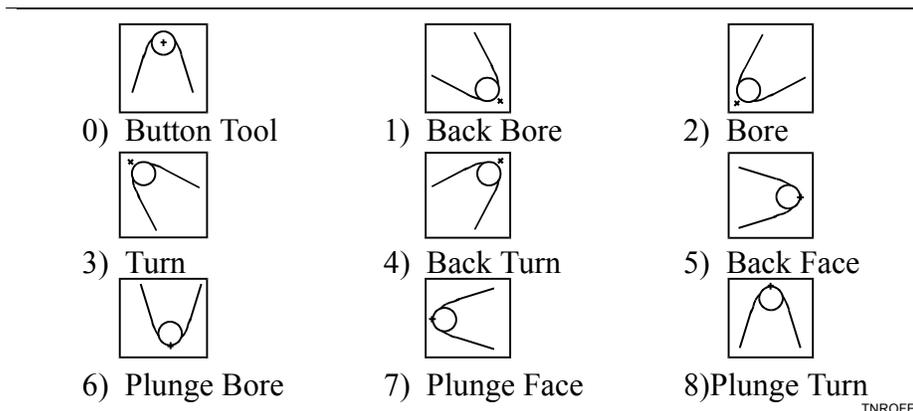


Figure 9-6, Tool Location Code Key (front tool post)

The Tool Location Code indicates the orientation of the tool tip to the workpiece. The CNC uses the tool location code with the entered radius to determine what compensation to apply.

In the column titled **L-Code**, enter the appropriate tool location code. To display a key of the tool location codes used, press **L-Code (F2)**. (Refer

Example: Tool 1 is an O.D. turning tool with a 0.0156 radius. The **Radius** value entered in the Tool Page for Tool 1 is .0156. The location-offset code is 3. (Refer to **Figure 9-6**.)

To enter a tool location code in the Tool Page:

1. At the appropriate tool number, use the **ARROWS** to move the cursor to the **L-Code** column.
2. Type in the tool location code that applies to that tool.

To enter the Radius:

1. At the appropriate tool number, use the **ARROWS** to move the cursor to the **Radius** column.
2. Enter in the appropriate Radius value. The CNC compensates for the given Radius when the tool number is activated.

Tool Nose Radius Programming Examples

Refer to **Table 9-5**. G40 or G41/G42 commands must be accompanied by a motion statement within the same block. The motion must be in rapid or feed (G0 or G1). Programming an arc move (G2/G3) will generate an alarm message.

The Tool Nose Radius compensation for the active Tool deactivates (G40) or activates (G41/G42) during the motion to the XZ coordinates on that block. The move (X, Z or XZ) must be at least the value of the Tool Nose Radius.

Table 9-5, Activating Tool Nose Radius Compensation

Block	Description
N3 T1	Activates Tool 1 length offsets.
N4 G00 G41(G42) X1 Z-1	Activates left(right) of path Tool Nose Radius compensation beginning with move programmed in same block.
N5 ...	
N6 ...	
N7 G40 X3 Z-3	Cancels Tool Nose Radius compensation beginning with move programmed in same block.

Left of Path TNR Program (G41)

Refer to **Figure 9-7**. The diagram shows a part design for an O.D. turning tool. It uses Absolute measurements and shows Inch and MM equivalents. The contoured shape of the part requires Tool Nose Radius Compensation to ensure accurate dimensions on the finished work.

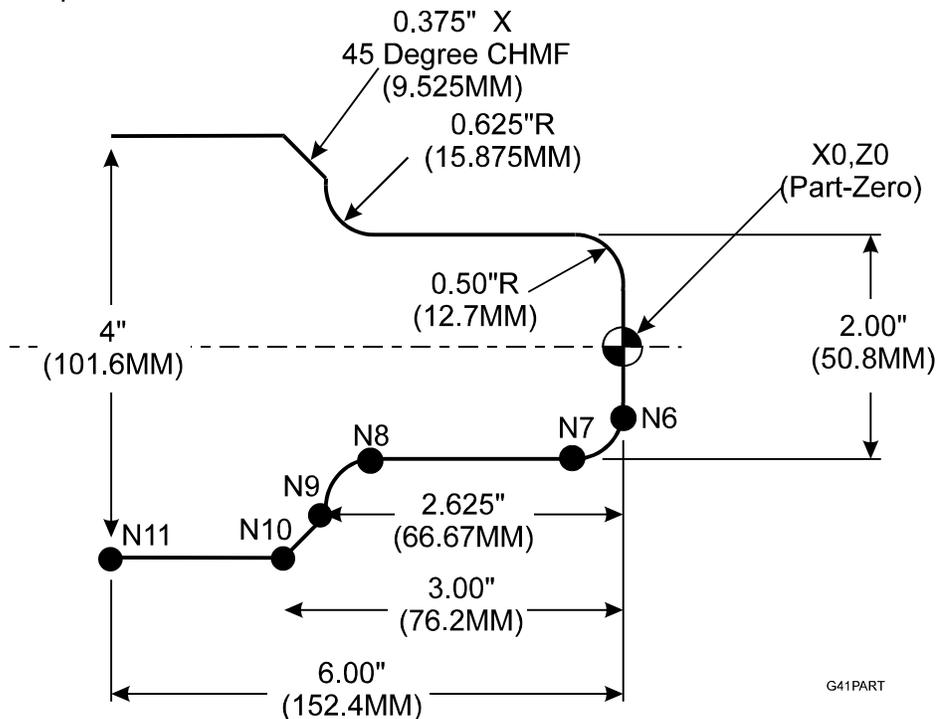


Figure 9-7, G41 Programming Example

Refer to **Table 9-6**. The table lists and describes the program required to program the part in the diagram. Diameter Mode is used.

Table 9-6, G41 Programming Example

Block	Description
N1. O1010 * TNR-COMP-EX1	Program name.
N2. G90 G70 G0 X0 Z0 T0	Activate Absolute Mode, Inch Mode, Rapid to Machine Home, cancel offsets.
N3. T1 * .0312R O.D. TURN	Activate tool 1 (O.D. turning tool with .0312 tool tip radius).
N4. G41 X0 Z.1 M3 S1200	Activate tool nose radius compensation, left of path. Position tool to start point. Activate spindle forward at 1200 RPM.
N5. G1 Z0 G95 F.01	Feed to part face at .01 inch per revolution.
N6. X1.0	Feed to first radius.
N7. G2 X2 Z-.5 R.5	Activate circular feed (CW) and perform arc move.
N8. G1 Z-2	Activate linear feed. Feed to position.
N9. G3 X3.25 Z-2.625 R.625	Activate circular feed (CCW) and perform arc move.
N10. G1 X4 Z-3	Activated linear feed. Feed to position.
N11. Z-6.0	Feed to position.
N12. G40 G0 X4.2	Cancel Tool Nose Radius compensation and rapid away from the part.
N13. T0 X0 Z0 M5	Cancel Tool-Length Offsets, return to home position, and deactivate spindle.
N14. M2	End program, return to N1.

Right of Path TNR Program (G42)

Refer to **Figure 9-8**. The diagram shows a part design for a boring tool (inner diameter), using Absolute measurements and showing Inch and MM equivalents. The contoured shape of the part requires Tool Nose Radius Compensation to ensure accurate dimensions on the finished work.

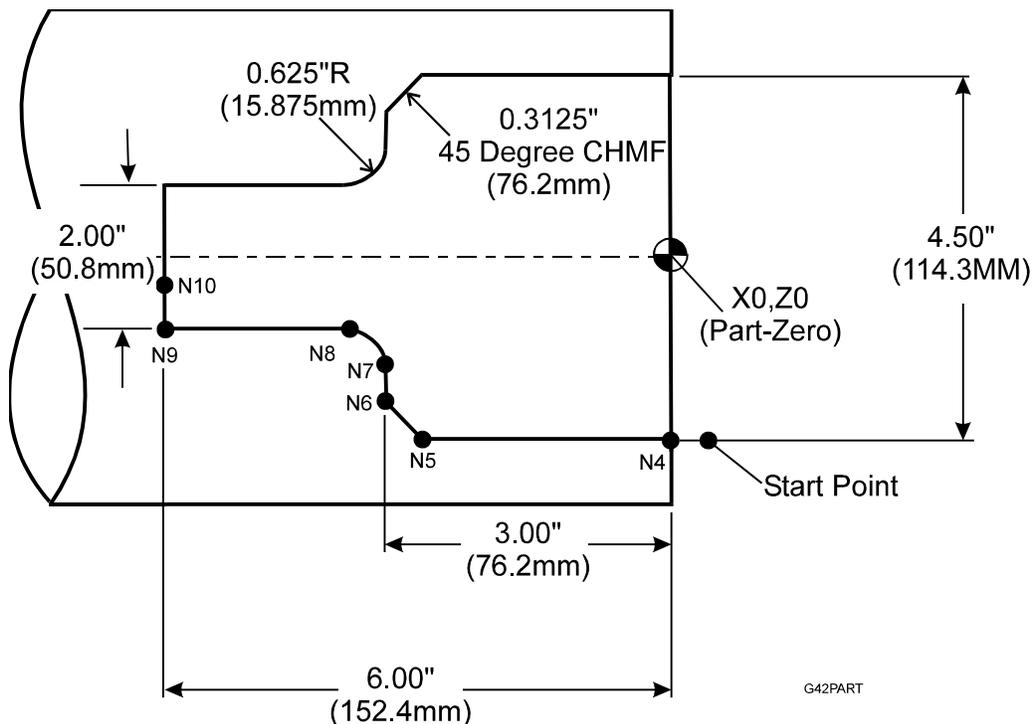


Figure 9-8, Tool Nose Radius Programming Example (G42)

Refer to **Table 9-7**. The table lists and describes the program required to machine the part in the diagram. Diameter mode is used.

Table 9-7, G42 Programming Example

Block	Description
N1 O1011 * TNR-COMP-EX2	Program name.
N2 G90 G70 G0 T0 X0 Z0	Activate Absolute Mode, Inch Mode, Rapid to Machine Home, cancel offsets.
N3 T1 *.0312R I.D. BORE	Activate Tool 1 (I.D. boring tool with .0312 tool tip radius).
N4 G42 X4.5 Z.1 M3 S1000	Activate tool nose radius compensation, right of path. Position tool at start point. Activate spindle forward at 1000 RPM.
N5 G1 Z-2.6875 G95 F.01	Feed to Z position at .01 inch per revolution.
N6 X3.875 Z-3	Feed to position.
N7 X3.25	Feed to position.
N8 G3 X2.0 Z-3.625 R.625	Activate circular feed (CCW) and perform arc move.
N9 G1 Z-6.0	Activate linear feed. Feed to position.
N10 G40 X1.8	Cancel compensation at programmed move.
N11 G0 Z.1	Rapid out of bore.
N12 T0 X0 Z0 M5	Cancel tool offsets and TNR comp. Rapid away from the part. Deactivate spindle.
N13 M2	End program, return to N1.

Canceling Tool Nose Radius Compensation (G40)

NOTE: The G40 code must be programmed with a linear feed code (G00 or G01) currently active or indicated in the G40 program block. G40 programmed with or immediately following circular feed codes (G02/G03) will result in an alarm message.

Also, the active tool cannot be changed until after the G40 command.

At power on, no Tool Nose Radius compensation is active. At the end of any cutting sequence using TNR compensation by G41 (left of path) or G42 (right of path), the compensation must be canceled by G40 (refer to **Table 9-8**).

Table 9-8, Canceling TNR

Block	Description
N4 G01 G41 X__ Z__	Activate TNR comp. Left of path at programmed move.
N5 etc...	
...	
...	
N20 G01 G40 X__ Z__	Cancel TNR comp. At programmed move.
N21 etc...	

Choosing the Correct TNR Compensation

Refer to **Figure 9-9**. There is only one question a programmer needs to ask to determine which Tool Nose Radius Compensation to use. Looking from behind the tool as it moves, is the tool to the left or the right of the programmed path? If the tool is to the left of the programmed path, choose G41. If the tool is to the right of the programmed path, choose G42.

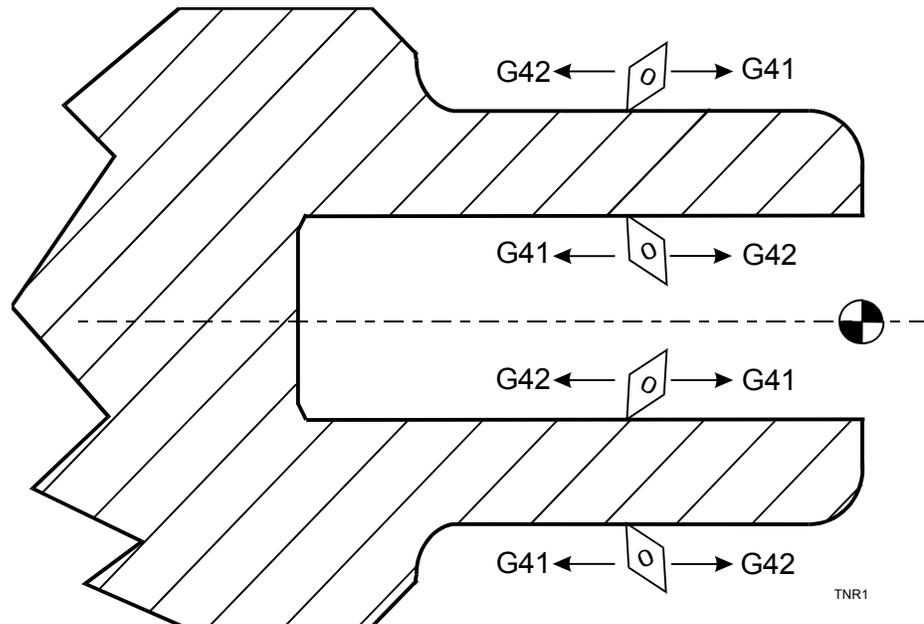


Figure 9-9, Choosing G41 or G42

Changing TNR Compensation in a Program

NOTE: Do not change compensation direction in the startup compensation block or the block immediately following it. Do not change compensation direction in the G40 block or the block immediately preceding it.

Sometimes it is necessary to change TNR Compensation within a program from G41 to G42, or from G42 to G41.

If the compensation direction changes, you must program a move in the same block as the new compensation command, using G00 or G01. It is not necessary to cancel compensation (G40) before changing the G41/G42 direction.

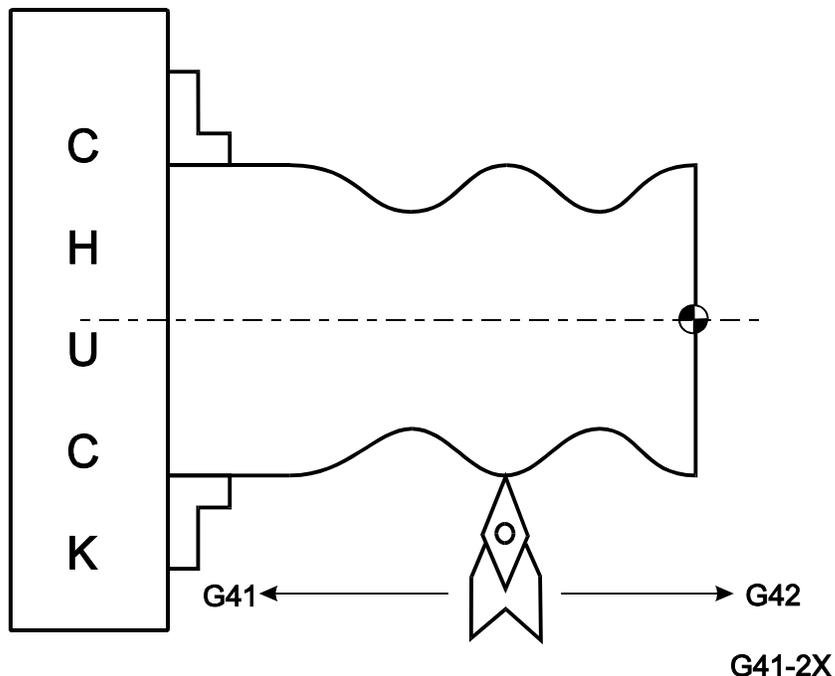


Figure 9-10, Changing Compensation

Refer to **Figure 9-10** and **Table 9-9**. In this example, the tool makes an O.D. cut toward the chuck with G41 active and clears the part (X+). Then, the tool contacts the part O.D. (X-) and makes an O.D. cut away from the chuck with G42 active.

This example would use Location Code 0 or 8.

Table 9-9, Changing TNR Compensation

Block	Description
G41	Xn Zn Activate compensation.
...	Machine programmed contour.
G91 X.1	Clear the part.
G90 X(part O.D.)	Contact the part O.D.
G42 Xn Zn	Change TNR compensation.

Motion of Tool during TNR Compensation

(1) At start-up of compensation

Refer to **Figure 9-11**. In linear to linear, or linear to circular moves, the position at the end of the start-up block (block with G41 or G42) will be perpendicular to the next programmed move in the axis. When two consecutive moves are compensated, the tool follows its offset path until it reaches the offset path for the second move.

The move which activates compensation (X, Z, or XZ) must move the machine a distance no less than the tool nose radius in length. For example, if nose radius is .0312", the vector of the move with G41 or G42 must be at least .0312" long.

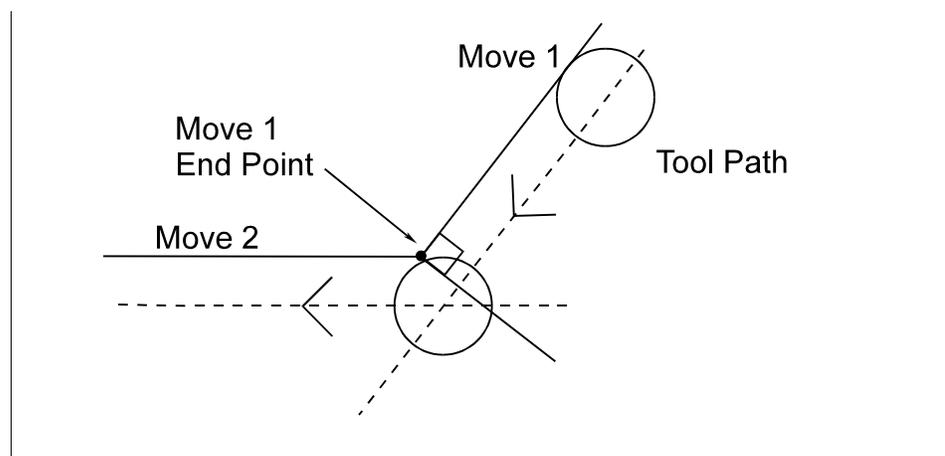


Figure 9-11, Consecutive Compensated Moves

(2) During TNR compensation

During TNR compensation, offset is performed correctly and automatically. Non-positioning moves such as dwells, coolant, or other auxiliary functions are allowed.

(3) In Offset cancel (G40)

The offset cancel code is G40. The TNR cancel block must contain a move command (X, Z or XZ) no less than the tool nose radius value in length. For example, if nose radius is .0312", the vector of the move in the G40 block must be at least .0312" long.

The G40 block move must be linear (G00 or G01).

The control looks ahead to following blocks in order to compensate correctly. When the control sees a G40 block coming up, the CNC positions the tool tip perpendicular to the last move before the G40 block.

Compensation around Acute Angles

During compensated moves around sharp angles, the CNC “rounds off” the intersection of the two moves to prevent wasted movement. The CNC will do this for any angles less than or equal to the Compensation Cutoff Angle. The Compensation Cutoff Angle is setup in the Setup Utility and is configurable. The default is 15 degrees.

Refer to **Figure 9-12**. Assume all programmed moves are made with Tool Nose Radius Compensation active.

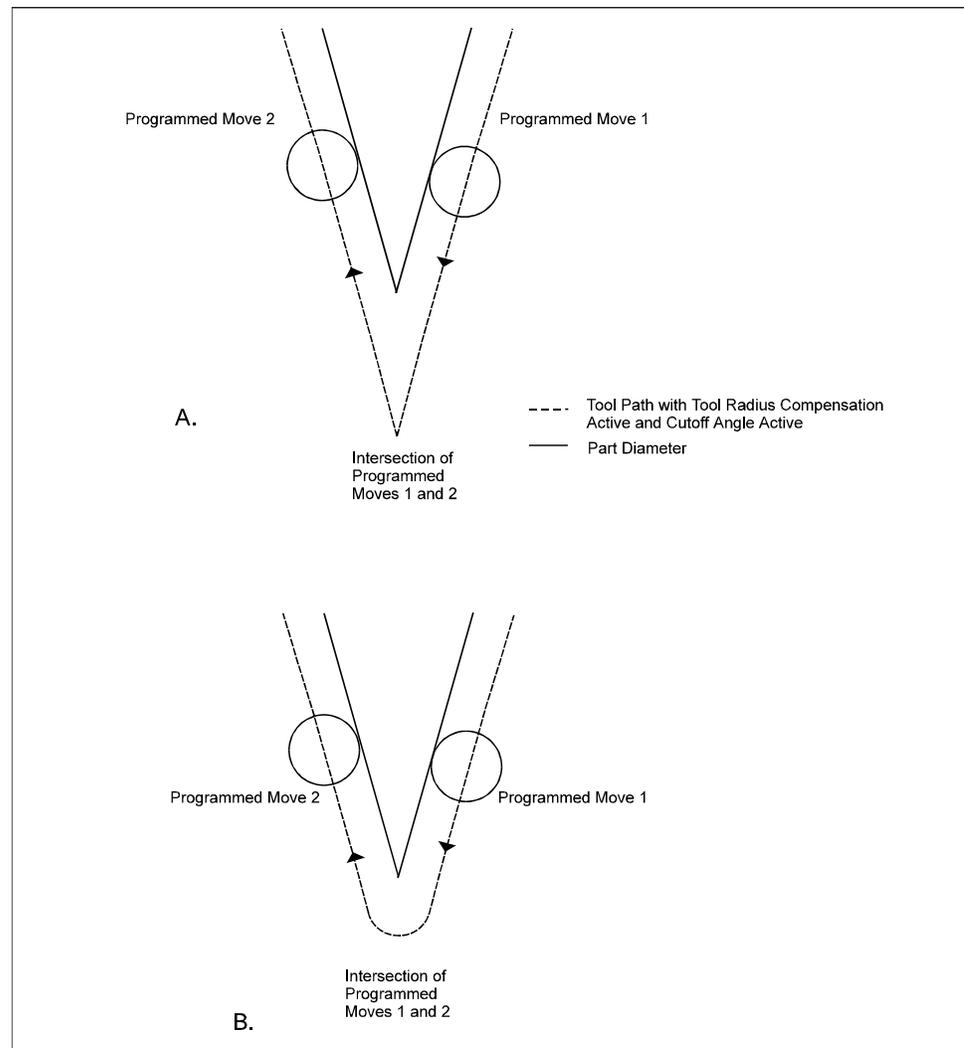


Figure 9-12, Compensation Cutoff Angle

The diagram describes two scenarios:

Diagram A shows the tool path resulting with no Compensation Cutoff angle (Compensation Cutoff Angle = 0). The tool path travels beyond the part diameter to a point where compensated moves 1 and 2 intersect, before the CNC executes move 2.

Diagram B shows the tool path resulting when a Compensation Cutoff angle is used. The CNC introduces a 0 degree radius arc move between programmed moves 1 and 2. This alters the tool path, eliminating wasted motion.

The CNC handles all angles less than the set default as shown in Diagram B.

NOTE: The Compensation Cutoff Angle can also be changed in a program via M Code #1031. Example: to change angle to 10 degrees, program: #1031=10. This value should be re-programmed to 15 degrees when finished (#1031=15).

Changing the Numbers of Look Ahead Blocks

A system variable is available to change the number of blocks the CNC will look ahead during TNR compensation. The variable must be programmed prior to the G40/G41/G42 block.

The variable is #1032 and the default is 1. At this setting, the CNC will look ahead however many blocks are necessary to find a valid intersection of the current and following moves.

Changing this value can change the compensated tool path. The intent of this option is to allow further look ahead capability to help prevent undercut due to excessive tool radius.

Wear Offsets

Wear offsets provide a means to correct for tool tip wear. Use wear offsets when the dimensions of the finished part are larger or smaller than the programmed dimensions. This method also does not require you to change the Tool-Length Offset.

For example (assume diameter mode), if a part's X-axis dimensions after machining are 0.005" too large, change the X wear offset to -0.005". This will remove 0.005" from the diameters.

If a part's X-axis dimensions after machining are 0.005" too small, change the X wear offset to 0.005". Subsequent runs of the program produce diameters 0.005" larger.

If Z-axis lengths are short, use a negative Z wear offsets. If Z-axis lengths are too long, use a positive Z wear offset.

To enter wear offsets:

1. In the Tool Page, go to the appropriate tool number. Place the cursor in the X Wear or Z Wear column.
2. Type in the wear offset value, and press **ENTER**.

Setting Wear Offset Adjustment While Running a Program

Wear offset adjustment is used to increment or decrement an X or Z tool offset for adjustment purposes. This can only be used when you are in single-step or auto mode (program run). This feature allows adjustment of tool-length offsets while making parts.

To set the wear offset adjustment while running a program:

1. Press **ToolWr (F8)**. The tool table screen will appear.
2. Move the cursor up or down to select the tool number that needs adjusting.
3. Press **X+ (F6)**, **X- (F7)**, **Z+ (F8)**, or **Z- (F9)** as needed to adjust for tool wear.
4. Exit the Tool Table screen by pressing **(F10)** and continue running parts.

<p>NOTE: Each key press will adjust the tool wear by an amount that is preset in setup. The new TLOs will take effect the next time the tool is activated. Default is 0.0005”</p>
--

Section 10 - Program Management

The Program Directory provides access to all of the program management and disk utilities. These functions include creating, loading, deleting, undeleting, and copying programs. The Program Directory also provides access to the floppy disk drives and the communications utilities.

To activate the Program Directory from the Manual screen, press **Program (F2)**.

Refer to **Figure 10-1**. The Program Directory's **USER** listing displays a list of programs stored in the CNC. All CNC programs have ".G" extensions. Change the display mode to view programs in other formats.

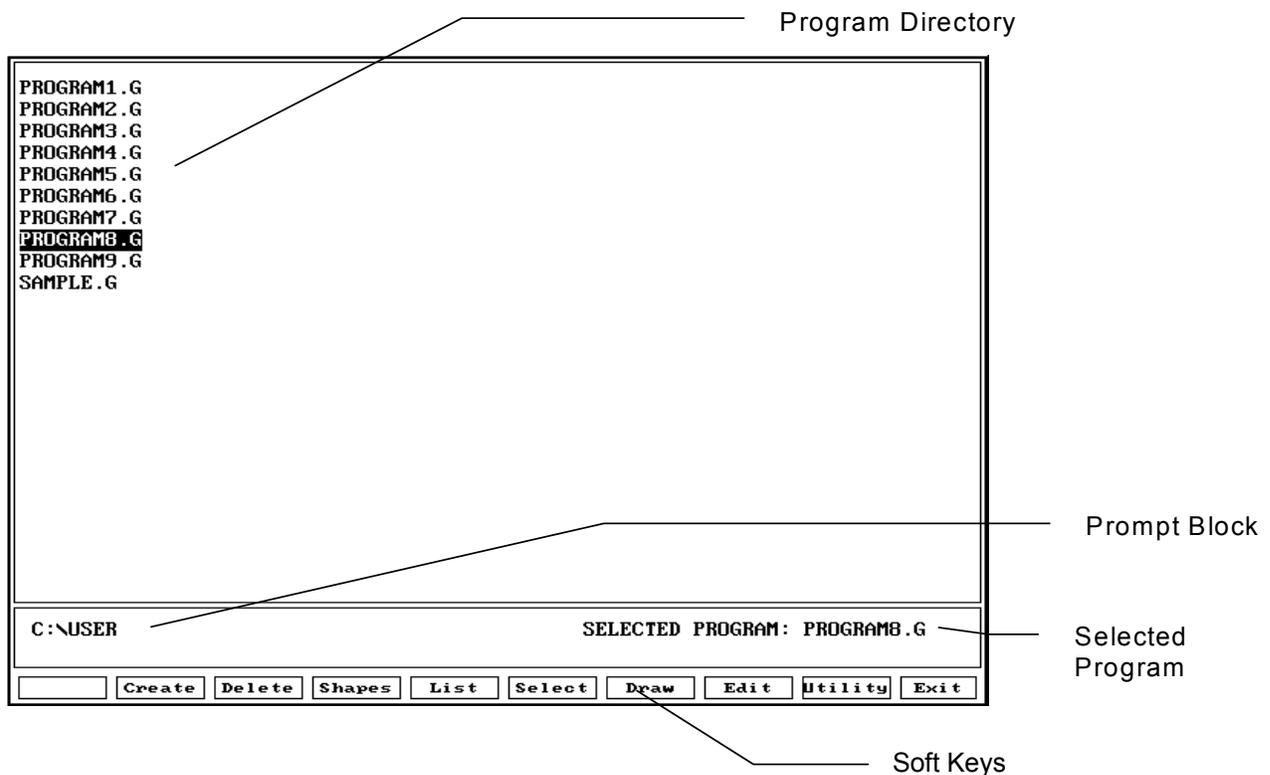


Figure 10-1, Program Directory

Changing the Program Directory's Display

The Program Directory has four display modes:

- Display only part program names (ending with ".G" extensions)
- Display only part program names (ending with ".G" extensions) along with size, date and time of last edit.
- Display all programs.
- Display all programs along with size, date, and time of last edit.

To switch the Program Directory display mode, press **Display (SHIFT + F9)**. The display setting showing only part program names is usually the easiest to use. Press (**SHIFT + F9**) to cycle through the four program display modes.

CAUTION: The Program Directory can provide access to internal CNC programs. Tampering with internal programs can cause the control to malfunction.

Creating a New Part Program

To create a new part program:

1. In Manual Mode, press **Program (F2)**. Program Directory activates.
2. Press **Create (F2)**. The control displays “**NEW PROGRAM: _**” prompt.
3. Type the new program name.
4. Press **ENTER**. The new program name is placed in the Program Directory alphabetically.

Choosing Program Names

The maximum length for a program name is eight characters. Additional characters are ignored by the CNC. The CNC displays all program names in capital letters. Each program must have a unique name. The CNC will automatically place the “.G” extension after the name.

Selecting a Program for Running

A program must be loaded before it can be executed by the CNC. Only one program can be loaded at a time.

To Load a program:

1. In the Program Directory, highlight a program.
2. Press **Select (F6)**. The CNC loads the program. The name of the currently loaded program activates next to the “**Selected Program**” label at the bottom of the screen.

Selecting a Program for Editing and Utilities

When you press **Edit (F8)**, the highlighted program opens for editing.

NOTE: If you activate the Program Editor while running a program, the editor will open the loaded program.

To select a program for editing:

1. In the Program Directory, highlight the name of the program being edited or written.
2. Press **Edit (F8)**. The Program Editor activates with the selected program open.

Listing a Program

List (F5) displays a program without allowing you to edit it. The List feature only works on CNC part programs (programs with a “.G” extension on their name).

The soft keys available on the List screen are **PgUp (F5)**, **PgDn (F6)**, **Home (F7)**, **End (F8)**, **Find (F9)**, and **Exit (F10)**.

PgUp (F5) and **PgDn (F6)** scroll through the Program Listing a page at a time, backward and forward, respectively. **Home (F7)** and **End (F8)** move the cursor to the beginning and end of a block. **Find (F9)** allows you to search for specific text or numbers in the program. **Exit (F10)** returns you to the Program Directory.

To list a program:

1. Highlight a program in the Program Directory.
2. Press **List (F5)**. The CNC displays the program blocks.

Deleting a Program

To delete a program:

1. From the Program Directory, highlight a program.
2. Press **Delete (F3)**. The CNC prompts you to confirm the deletion and the soft keys change.
3. Press **Yes (F1)**. The CNC deletes the selected program.

– or –

Press **No (F2)**. The Delete command is canceled.

NOTE: Deleting a program automatically deletes the associated “.S” file.

Logging to Other Drives

The Program Directory displays the programs in the C:\USER directory by default, but it can be set to show programs stored in other drives or subdirectories. When you press **Log (SHIFT + F7)**, the Log Pop-Up Menu activates. The pop-up lists the following choices: A:, C:, or Other. Choose **A:** to display programs stored on a floppy drive inserted in the machine's floppy disk drive. Choose **C:** to display user part programs.

To list the programs in another drive or directory, choose **Other**. A prompt activates. Enter the full pathname of the drive and directory whose programs will be listed.

To set the Program Directory to display the programs in a selected drive:

1. From the Program Directory, press **SHIFT**. The soft keys menu changes.
2. Press **Log (F7)**. The Log Pop-Up activates.
3. Highlight **A:** or **C:**, and press **ENTER**. The CNC displays the programs stored in the drive.

To set the Program Directory to display the programs in a selected directory:

1. From the Program Directory, press **SHIFT**. The soft keys menu changes.
2. Press **Log (F7)**. The Log Pop-Up activates.
3. Highlight **Other**, and press **ENTER**. The CNC displays the following prompt, "**Log to:**".
4. Type the full pathname, including drive, of the directory whose programs will be listed, and press **ENTER**. The CNC displays the programs stored in the specified directory.

Marking and Unmarking Programs

Some operations can be performed on more than one program at a time. The Program Directory allows you to select (Mark) one, some or all of the programs in the **USER** listing.

Marking Programs

To mark a program:

1. From the Program Directory, highlight a program.
2. Press **ENTER**. The marked program highlights and the highlight bar advances to the next program.
3. Press **ENTER** to mark the next program.

– or –

Highlight another program, and press **ENTER**.

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.

Marking All Programs

To mark all programs in the Program Directory:

1. In the Program Directory, press **Utility (F9)**. The Utility Pop-Up Menu activates.
2. Highlight **More**, and press **ENTER**. The second page of the Utility Pop-Up displays.
3. Highlight **Mark All**, and press **ENTER**. The CNC marks all the programs in the Program Directory. Programs are highlighted.

Unmarking All Marked Programs

To unmark all programs in the Program Directory:

1. In the Program Directory, press **Utility (F9)**. The Utility Pop-Up Menu activates.
2. Highlight **More**, and press **ENTER**. The second page of the Utility Pop-Up activates.
3. Highlight **Unmark All**, and press **ENTER**. The CNC unmarks all the programs in the Program Directory. Programs are no longer highlighted.

Deleting Groups of Programs

1. From the Program Directory, mark all programs to be deleted.
2. Press **Delete (F3)**. The CNC prompts you to confirm the deletion and the soft keys change.
3. Press **Yes (F1)** and the CNC deletes the selected programs.
– or –
Press **No (F2)** and the deletion is canceled.

Restoring Programs

A deleted program can be restored (recovered) if the space it occupied has not been reused. Sometimes only a portion of a deleted program is recoverable.

To restore a program:

1. From the Program Directory, press **Utility (F9)**. The Utility Pop-Up activates.
2. Highlight **Restore**, and press **ENTER**. If the control finds programs to restore, it displays a pop-up menu. If the control does not find any deleted programs, a “**No programs available for restore.**” message is displayed.
3. When deleted programs are found, the control will list them in a pop-up menu. Highlight a deleted program name, and press **ENTER**. The CNC prompts for the first letter of the deleted program’s name.
4. Type in the letter, and press **ENTER**. The CNC presents an information message telling the user whether the program can be restored.
5. If the program disk space has not been reused, press **Cont (F10)**. The program is restored.

<p>NOTE: Restored programs may not contain all of the original information. Review any restored programs for accuracy before attempting to use it.</p>

Copying Programs to Floppy Disks

Copying programs to floppy disks provides an easy way to transfer programs to other machines. Programs can also be copied to floppy disks for storage.

To copy programs to floppy disks:

1. From the Program Directory, mark all of the programs to be copied.
2. Press **Utility (F9)**. The Utility Pop-Up Menu activates. **Copy** is highlighted. Press **ENTER**. The **Copy to:** pop-up menu activates.
3. Highlight the target drive, and press **ENTER**. The marked programs are copied to the target drive.

Renaming Programs

To rename a program:

1. From the Program Directory, highlight a program.
2. Press **Utility (F9)**. The Utility Pop-Up Menu activates.
3. Highlight **Rename**, and press **ENTER**. The CNC prompts, "**Rename [PROGRAM] to ?:**"
4. Type in the new program name, and press **ENTER**. The CNC renames the program.

Printing Programs

The CNC can print to any standard IBM PC compatible printer. The print feature can be used to make paper copies of part programs. All marked programs will be sent to the printer.

1. From the Program Directory, mark all the programs to be printed.

– or –

Highlight a program to select it.

2. Press **Utility (F9)**. The Utility Pop-Up activates.
3. Highlight **Print**, and press **ENTER**. The CNC prompts you to confirm the command and the soft keys change.
4. Press **Yes (F1)** to send the program(s) to the printer. Press **No (F2)** to cancel.

Formatting Floppy Disks

Before programs can be copied to a floppy disk, the disk must be formatted. Most disks come pre-formatted. The CNC can format a disk when necessary.

To format a floppy diskette:

1. From the Program Directory, press **Utility (F9)**. The Utility Pop-Up activates.
2. Highlight **Floppy Format**, and press **ENTER**. A warning message is displayed and the CNC prompts you to confirm the format. The soft keys change.

CAUTION: Formatting a floppy disk erases all of the data on the disk.

3. Press **Yes (F1)** and the CNC prompts for disk insertion. Press **No (F2)** to cancel.
4. If **Yes (F1)** is chosen, insert the floppy disk into the CNC's disk drive and press **Cont (F10)**. The CNC displays disk formatting statistics as it formats.

NOTE: Press Cancel (F9) to cancel.

5. Press **Cont (F10)** to return to the Program Directory.

Checking Disks for Lost Program Fragments

Sometimes computer disks contain program fragments that have been lost. This might happen if a computer loses power while it is saving a program. Lost program fragments are invisible in the Program Directory, but use up valuable program space.

To check for lost program fragments:

1. From the Program Directory, press **Utility (F9)**. The Utility Pop-Up activates.
2. Highlight **More**, and press **ENTER**. The second page of the pop-up menu activates.
3. Highlight **Check Disk**, and press **ENTER**. The CNC prompts you to select the drive to be checked.
4. Highlight the desired drive, and press **ENTER**. The CNC checks the disk. If lost clusters are found, the CNC prompts the user for recovery instructions and the soft keys change.
5. Press **Yes (F1)** to recover lost disk space. Press **No (F2)** to cancel.
6. If **Yes (F1)** is chosen, the CNC attempts to recover lost disk space. At the end of the procedure, press **Cont (F10)** to return to the Program Directory.

Displaying System Information

Refer to **Figure 10-2**. The System Information screen displays specific details about the CNC and software package. Most of the information on this screen is only required when the machine is setup or when troubleshooting a problem.

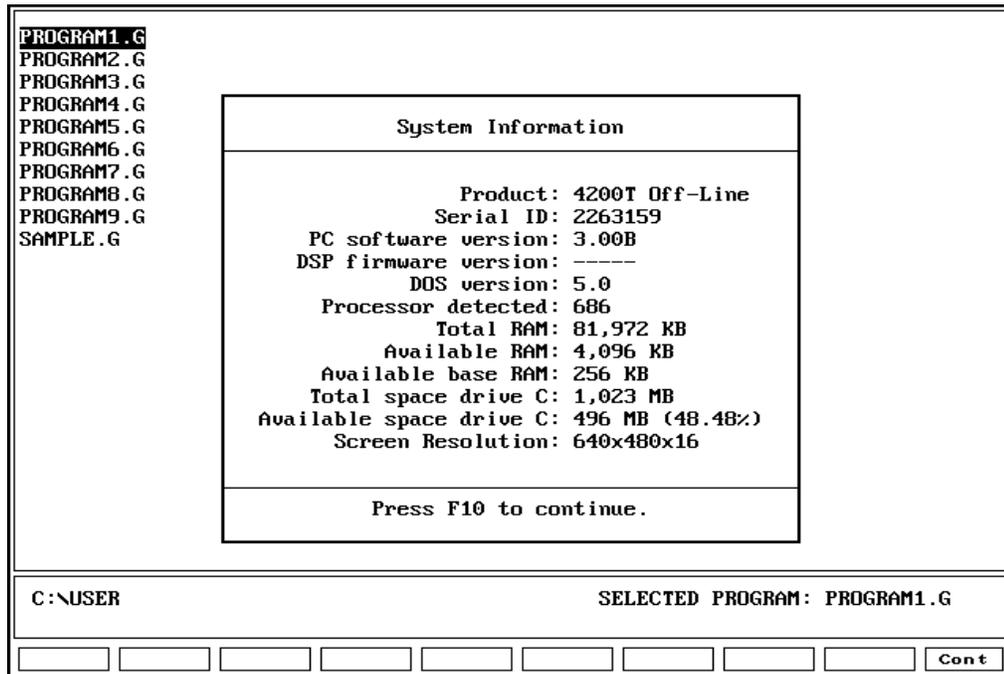


Figure 10-2, System Information Screen

To display the System Information Screen:

1. From the Program Directory, press **Utility (F9)**. The Utility Pop-Up menu activates.
2. Highlight **More**, and press **ENTER**. The second page of pop-up menu activates.
3. Highlight to select **System Info**, and press **ENTER**. The System Information Screen displays.

Copying Programs from/to Unspecified Locations

To copy programs to or from an unspecified location:

1. From the Program Directory, press **Utility (F9)**. The Utility Pop-Up activates.
2. Highlight **More**, and press **ENTER**. The second page of pop-up menu activates.
3. Highlight **Copy ?**, and press **ENTER**. The CNC prompts for the name and location of source program. (Wild cards may be used.)
4. Type the name and location (complete path) of the program being copied, and press **ENTER**. A pop-up prompts the user for the destination drive or **Other** destination.
5. Highlight **Other**, and press **ENTER**. The CNC prompts for destination.
6. Type in the new location (complete path), and press **ENTER**. The program is copied into the new location.

<p>NOTE: Sometimes it is easier to Log onto the floppy disk drive containing the program, mark the program, and use the Copy to: feature.</p>

Renaming Programs from/to Unspecified Locations

Rename a program at any location as follows:

1. From the Program Directory, press **Utility (F9)**. The Utility Pop-Up menu activates.
2. Highlight **More**, and press **ENTER**. The second page of pop-up menu activates.
3. Highlight **Rename ?**, and press **ENTER**. The CNC prompts for the name and location of the source program.
4. Type the name and location (complete path) of the program being renamed, and press **ENTER**. The CNC prompts for new name and location.
5. Type the new name and location (complete path) of the program, and press **ENTER**. The program is renamed.

<p>NOTE: Use the Rename ? feature to move a program to a different drive by typing a different program destination.</p>

<p>NOTE: Sometimes it is easier to Log onto the floppy drive containing the program, mark the program, and use the Rename feature.</p>
--

Printing Programs from Unspecified Locations

The CNC can print to any standard IBM PC compatible printer. To print programs from unspecified locations:

1. From the Program Directory, press **Utility (F9)**. The Utility Pop-Up activates.
2. Highlight to select **More**, and press **ENTER**. The second page of pop-up menu activates.
3. Highlight **Print ?**, and press **ENTER**. The CNC prompts for name and location of source program.
4. Type the name and location (complete path) of the program being renamed, and press **ENTER**. The CNC prompts you to confirm the instruction and the soft keys change.
5. Press **Yes (F1)** to send the program to the printer. Press **No (F2)** to cancel the print job.

NOTE: Sometimes it is easier to Log onto the desired drive, mark the desired program, and use the Print feature.
--

Creating Subdirectories

Use the **Sub Dir (SHIFT + F2)** soft key to create subdirectories. Ensure that the CNC is in the desired drive before creating a subdirectory (default is C:\USER).

To create a subdirectory:

1. In the Program Directory, press **SHIFT**.
2. The soft key menu changes. Press **Sub Dir (F2)**.
3. The CNC prompts for the new subdirectory. Type the subdirectory name, and press **ENTER**. The CNC creates the subdirectory.

Deleting an Unspecified Program

Use the **Del ? (SHIFT + F3)** soft key to delete an unspecified program. It enables the user to delete programs in another drive without logging into that drive.

To delete an unspecified program:

1. In the Program Directory, press **SHIFT**. The soft key menu changes.
2. Press **Del ? (F3)**. The CNC prompts for the name of the program to be deleted.
3. Type the name and location of the program (complete path), and press **ENTER**. The CNC requests confirmation of the delete command and the soft keys change.
4. Press **Yes (F1)** to delete the program. Press **No (F2)** to cancel the command.

If **Yes (F1)** is chosen, the CNC deletes the entered program.

Listing an Unspecified Program

Use the **List ? (SHIFT + F5)** soft key to list an unspecified program. It enables the user to list programs in another drive without logging into that drive. Listing a program allows the user to review program without editing capability.

To list an unspecified program:

1. In the Program Directory, press **SHIFT**. The soft key menu changes.
2. Press **List ? (F5)**. The CNC prompts for the name of the program to be deleted.
3. Type the name and location of the program (complete path), and press **ENTER**. The CNC displays the Program Listing for the entered program.

Editing an Unspecified Program

Use the **Edit ? (SHIFT + F5)** soft key to edit an unspecified program. It enables the user to edit programs in another drive without logging into that drive.

To edit an unspecified program:

1. In the Program Directory, press **SHIFT**. The soft key menu changes.
2. Press **Edit ? (F5)**. The CNC prompts for the name of the program to be deleted.
3. Type the name and location of the program (complete path), and press **ENTER**. The CNC displays the program. The CNC activates the Edit Mode.

Optimizing Your Hard Disk

When data is stored on a hard disk, it is stored in areas known as tracks and sectors, similar to those on a CD. As more and more programs are created, deleted, copied, renamed, etc., the hard disk becomes “fragmented”. Information is stored in random unoccupied spaces. Fragmentation slows down the performance of the hard disk. Therefore, it will take longer to access information. To minimize fragmentation, you must optimize your hard disk periodically. Your CNC has a built-in disk optimizer. ANILAM recommends that you optimize your hard disk bi-monthly, or at the very least, once every six months.

Accessing the Disk Optimizer

To access the Disk Optimizer:

1. In Manual Mode, press **Program (F2)**. Press **Utility (F9)**. A pop-up is displayed.
2. Highlight **MORE....**, and press **ENTER**. The first entry in the window, Disk Optimize highlights.
3. Press **ENTER**. The Optimizer automatically scans the hard disk directories and examines the hard disk. This process takes up to three minutes.
4. When the process is complete, a pop-up menu is displayed. You can **Begin Optimization** or **Exit Optimizer**.

<p>NOTE: Optimization is an automatic process; do not interfere with it while it is running. If an emergency arises, press Cancel (F9) to halt optimization.</p>
--

5. The optimization process normally takes fifteen to ninety minutes, depending on the size of the hard disk. To minimize run time, optimize your hard disk as recommended.
6. During optimization, the CNC displays the various processes that are taking place.
7. When optimization is complete, the CNC displays: **OPTIMIZATION COMPLETE...Press any key to exit.**
8. Press any key to return to the Program Directory.

Maximizing Program Storage Space

The CNC has a fixed amount of space available for programs. Check the space available with the System Information feature.

Refer to [Figure 10-2. System Information Screen](#).

- ❑ **Total Space Available for the System** is the total amount of program storage space available in the CNC.
- ❑ **Total Free User Space** is the available space remaining for new programs.

When a program is executed on the machine (or in Draw), the control generates a second program of the same name followed by “.S”. The “.S” programs contain information required by the CNC. Normally an “.S” program is slightly larger than the part program that generated it. When a part program is deleted, the associated “.S” file is also deleted.

The CNC is not the best place for long term storage of part programs. Part programs should be periodically copied to floppy disks for backup and storage.

If you run out of disk space, you can delete the “.S” files of programs that are not currently being used to increase storage space. The CNC will automatically replace the “.S” files the next time the programs are run.

Display all program files in all formats to view the extraneous *.S files.

Section 11 - Running Programs

NOTE: All programs should be verified in Draw prior to running them. Refer to [“Section 8 - Draw Mode.”](#)

There are three modes of programmed operation.

Single-Step Mode Runs a program one block at a time.

Motion Mode Runs a program from motion block to motion block.

Automatic Mode Runs a program automatically, without pausing.

All screens for running programs are variations of the Manual Mode screen. The soft key for the active mode will be highlighted.

You must load a program before you can use it to cut a part. Refer to [“Section 10 - Program Management”](#) for information on loading programs.

Use the Manual Data Input Mode (MDI) to program a few quick moves without creating and saving a program. MDI is usually associated with manual operation and it is only available in the Manual Mode. Refer to [“Section 3 - Manual Operation and Machine Setup”](#) for information on MDI.

All programming tools, moves and cycles are available in MDI.

Running a Program One Step at a Time

The Single-Step screen provides access to two different execution modes: the Single-Step Mode (**S.STEP**) and the Motion Mode (**MOTION**). Both of these modes allow the operator to step through the program and verify the moves before cutting an actual part.

Refer to [Figure 11-1, Single-Step/Motion Screen](#). The S.Step screen looks like a Manual screen with fewer soft keys and **S.STEP (F5)** highlighted.

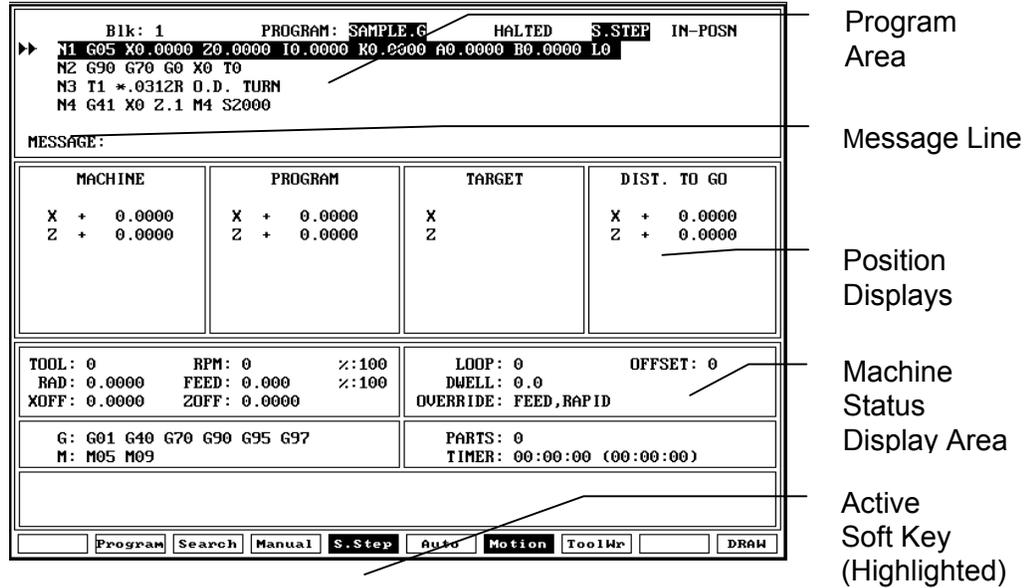


Figure 11-1, Single-Step/Motion Screen

To run a program in Single-Step Mode, do the following:

1. Go to the Program Directory and select the required program.
2. In the MANUAL screen, press **S.STEP (F5)**. The Single-Step Mode activates.
3. Press **START**. The CNC executes a single block or motion.
4. Repeat step 3 above, as required.

NOTE: In Auto Mode, press **S.STEP (F5)** to activate Single-Step Mode.

Toggling Between Motion and Single-Step Mode

To switch to Motion Mode, press **Motion (F7)**. **Motion (F7)** highlights.

- ❑ In Single-Step Mode, the CNC holds before the execution of each block. Press **START** to execute each block.
- ❑ In Motion Mode, the CNC holds before each machine move. Press **START** to execute each move.

Holding or Canceling a Single-Step Run

Press **HOLD** to hold program execution. To restart a program that is on hold, press **START**. To cancel a program that is on hold, press **Manual (F4)**. When a program is canceled, tool compensation is canceled and canned cycles are terminated. All other modal settings remain active.

Single-Step Execution of Selected Program Blocks

Using Arrow Keys to Select Starting Block

Select the starting block before starting the program:

1. Load the desired program and return to the Manual screen.
2. Press **S.STEP (F5)**. Single-Step Mode activates.

NOTE: To switch to Motion Mode, press **Motion (F7)**.

3. Highlight the desired starting block in the program.
4. Press **START**. CNC executes the next block or motion.

Using Search to Select Starting Block

Use **SEARCH** to locate a specific block number or entered text. **SEARCH** directs the CNC to search the program for a block number, a block containing a number, or a block containing specific text. The first block found containing the specified information is highlighted and displayed. Search only searches forward in the program. Perform the Search before starting the program.

1. From the Program Directory, select the desired program and return to the Manual screen.
2. Press **S.Step (F5)**. Single-Step Mode activates.
3. Press **Search (F3)**. CNC prompts for search number or text.
4. Type the required number or text, and press **ENTER**. The CNC runs the search. The first block containing the number or text will be highlighted.
5. Press **START**. The CNC executes the program from the highlighted block.

Switching from Single-Step to Auto

To change the CNC from Single-Step to Auto Mode, do the following:

1. In Single-Step Mode, press **Auto (F6)**. The CNC completes current move then holds.
2. Press **START**. The CNC restarts and runs the rest of the program in the Automatic Mode.

Position Display

The Position Displays for X and Z show:

Machine	Movement to the programmed (commanded) position in reference to Machine Home.
Program	Movement to the programmed (commanded) position in reference to Part Zero.
Target	The commanded position.
Distance To Go	Distance to go to commanded position.

Automatic Program Execution

The Auto Mode is the CNC's production mode. All or any portion of a program can be executed in the Auto Mode. The CNC can be put in Auto Mode from either Manual or Single-Step Modes.

Refer to **Figure 11-2**. The AUTO screen is similar to the Manual screen, with fewer soft keys. **Auto (F6)** highlights when Auto Mode activates.

To run a program in Automatic Mode, do the following:

1. In the Program Directory, select the required program and return to the Manual screen.
2. Press **Auto (F6)**. Automatic Mode activates.
3. Press **START**. The CNC begins executing program blocks.

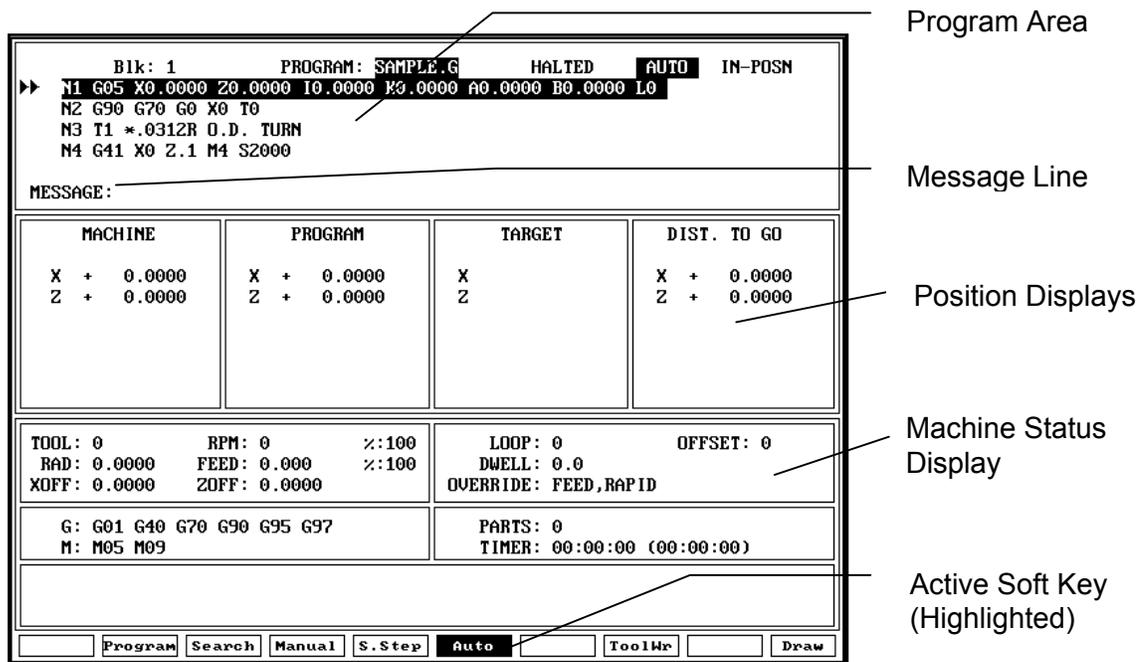


Figure 11-2, Auto Screen

Holding or Canceling an Auto Run

Press **HOLD** to hold program execution. To restart a program that is on hold, press **START**. To cancel a program that is on hold, press **Manual (F4)**. When program execution is canceled, tool compensation is canceled and canned cycles are terminated. All other modal settings remain active.

Automatic Execution Starting at a Specific Block

Using Arrow Keys to Select Starting Block

Select the starting block before starting the program.

1. From the Program Directory, load the required program and return to the Auto screen.
2. Highlight the required starting block.
3. Press **START**. CNC begins automatic program execution from the selected block.

Using SEARCH to Select Starting Block

Search directs the CNC to search the program for a block number, a block containing a number, or a block containing specific text. The first block found containing the specified information is highlighted and displayed. Search only searches forward in the program. Perform the Search before starting the program.

1. From the Program Directory, load the required program and return to the Manual screen.
2. Press **Auto (F6)**. Auto Mode activates.
3. Press **Search (F3)**. The CNC prompts for search number or text.
4. Type in the number or text, and press **ENTER**. The CNC performs the search. The first block containing the number or text is highlighted.
5. Press **START**. The CNC executes the program from the highlighted block.

Clearing a Halted Program

When the CNC encounters a program block that generates an error, it displays a Warning message and the CNC stops program execution. Return 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 error messages.

When the program is correct, 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. **S.Step (F5)** or **Auto (F6)** and the **Draw (F10)** soft key will be highlighted.

Draw Simulation and Real Time Draw Mode settings must be made from the Draw Simulation Mode screen prior to program execution. (Refer to [“Section 8 - Draw Mode.”](#))

NOTE: Press **CLEAR** at any time to clear the Draw display.

To activate **Draw** while running a program, do the following:

1. Load the required program and put the CNC in required execution mode (**S.Step** or **Auto**).
2. Press **Draw (F10)**. The Real Time Draw screen activates and the soft keys change.
3. Press **START**. Program execution starts and moves are displayed as executed.

Setting the CNC to Display an Enlarged Position Display

Refer to **Figure 11-3**. In the Manual, Auto and S.Step Modes, the CNC can be set to display a Large Position Display. The Large Position Display appears in the middle of the screen and will show motion to programmed positions.

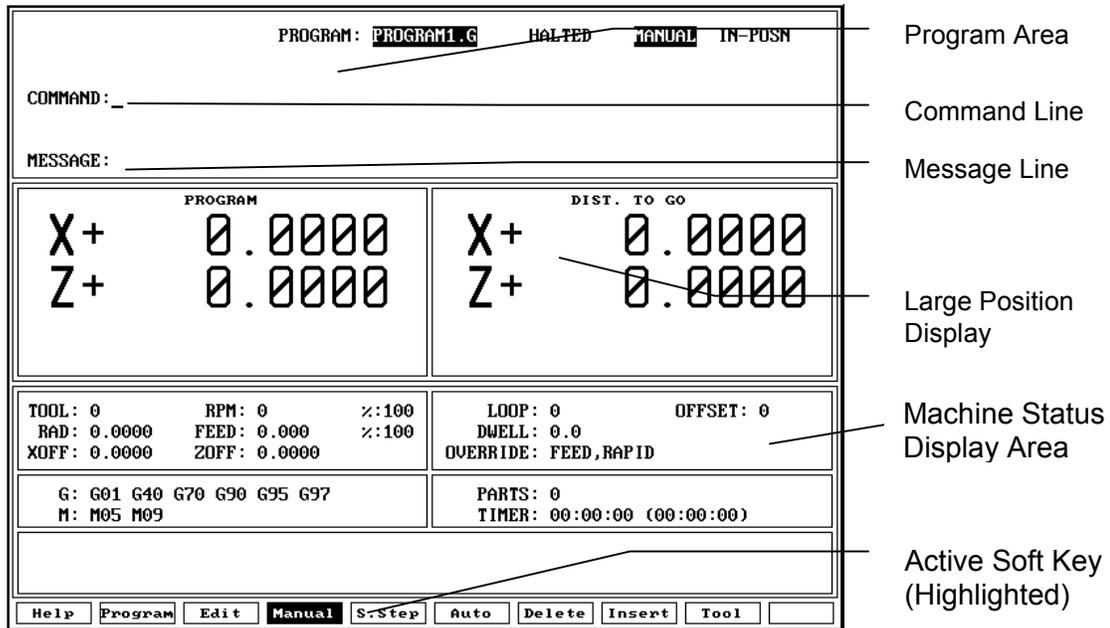


Figure 11-3, Large Position Display

To switch the display between the Large Position Display and the default position display, in Manual Mode, at the command line, type **B**, and press **ENTER**.

To switch the display between the Large Position Display and the default position displays in **S.Step** or **Auto** Mode, press **B**.

Teach Mode

Use Teach Mode to input data into a program from Manual Mode. Axes positions, modal status and MDI commands can be input directly into the program.

A series of manual moves can be incorporated into a new program or added to an existing program. Use Teach Mode to input data when the target position is difficult to calculate. Move the tool via the jog functions on the manual panel or via MDI.

To store input data into a new program, you must first create and load the program.

Initiating Teach-In

In Manual Mode and with a program selected, press **Teach (SHIFT + F5)** to activate Teach Mode. **Manual (F4)** and **Teach (F5)** will be highlighted. The mode designator in the upper-right area of the screen also shows that TEACH Mode is active.

Soft Keys of Teach-In

Press **Teach (SHIFT + F5)** from Manual Mode to activate the following soft keys listed in **Table 11-1**.

Table 11-1, Teach Soft Keys

Label	Soft Key
Help	F1
Manual	F4
Teach	F5
Delete	F7
Insert	F8
Quit	F10

Inputting Data into a Program

Data will be input to or executed from the highlighted block on the Teach Mode screen. If you input a move, all subsequent blocks move down in the program. To add blocks at the end of a program, position the cursor on the last program block.

Press **START** to input or activate MDI blocks.

Command data = G-codes, F-codes

Axis data = XZ positions

1. If no command or axis data is typed onto the command line and **START** is pressed, the CNC will store all axes positions with a G90 code.
2. If no axis data, only command data is typed onto the command line and **START** is pressed, the command data will be stored with the axes positions, and a G90 code.

Example: G1 F.007

When the above line is typed onto the command line and **START** is pressed, that information and the axes positions will be input into the program with G90. G91 will give error if used here.

3. If axis data with or without command data is typed onto the command line and **START** is pressed;

Example: G91 G1 X1.5 F.005

The block will be executed and stored in the program simultaneously. The mode will change to G91, depicted by active G-codes at bottom of screen. If axis data only is typed, the current modals (G90-G91) will remain in use.

Canceling Teach Mode

To cancel Teach Mode and save any additions to the program, press **Manual (F4)**. To cancel Teach Mode and discard any additions, press **QUIT (F10)**. Either will return the CNC to Manual Mode.

About Teach Mode

1. Axes positions stored via Teach Mode are always referenced to Program Zero. Axis data is considered to be any axis move or location (XZ).
2. Press **HandWI (F10)** to activate the handwheel prior to activating Teach Mode. The handwheel cannot be activated in Teach Mode.
3. S, M, and T codes can be input but not executed in Teach Mode. They must be input on a block by themselves.
4. No canned cycles may be executed in Teach Mode.
5. G91 will change the mode status, shown at the bottom of the screen in the active G-codes. Press **START** to activate G90.
6. Press **Manual (F4)** during a move to abort the move without canceling Teach Mode.

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 S.Step Modes. Refer to **Figure 11-4**.

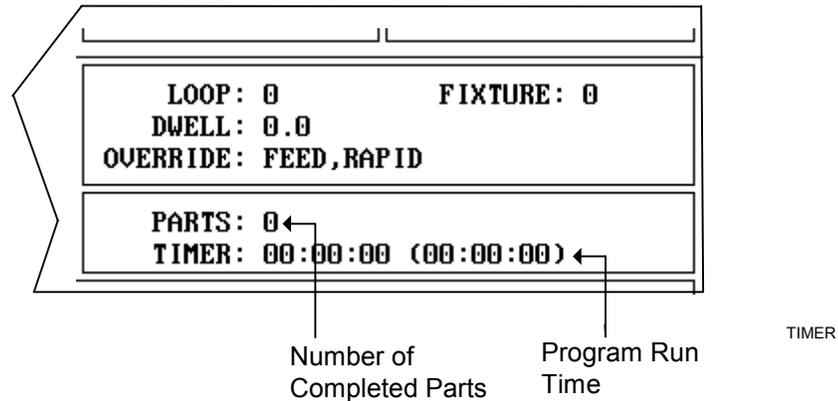


Figure 11-4, Program Timer and Parts Counter

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 S.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. The Parts Counter value can be modified via M-Codes. Refer to [Table 11-2, M-Codes Used with Parts Counter and Program Timer](#).

Table 11-2, M-Codes Used with Parts Counter and Program Timer

M-Code	Function
M9355 X0	Prevents the parts counter from resetting to zero.
M9356 X0	Disables the Timer and Counter.
M9376 Xx	Presets any value into the parts counter register. For example, program M9376 X5 to preset 5 in the parts counter register.
M9377 Xx	Adds entered number to the parts counter. For example, if the current parts counter value is 4 and you then program M9377 X6, the new parts counter value will be 10.

Background Mode

In the Background Mode, you can create, edit, delete, copy, or print other programs while you run the selected program.

Most program management utilities are available in the Background Mode. You can also send or receive (but not execute) programs through RS-232 communications. The CNC does not display the soft keys for prohibited functions in Background Mode. The following functions are not available:

- Draw Graphics
- DNC execution
- Selecting another program to run
- Editing the currently running (selected) program

To activate the Background Mode:

1. While the selected program is running in the Auto or Single-Step Mode, press **Program (F2)**. The CNC activates the Program Directory.

In Background Mode, the CNC displays messages for the running program in a message box. To clear the messages, press **CLEAR**.

For information on the Edit Mode, refer to "[Section 6 - Program Editor](#)." For information on RS-232 communications, refer to "[Section 15 - Communication](#)." For information on the program management utilities, refer to "[Section 10 - Program Management](#)."

Section 12 - S Function, M Functions, and C-Axis Programming

Introduction

This section provides S and M code formats. Refer to **Table 12-1**. The codes are included in the part program or activated in Manual Mode. A description of C-axis programming is also provided.

Table 12-1, S and M Codes

Code	Function
S (Spindle)	Commands spindle speeds (S).
M (Miscellaneous)	Performs miscellaneous (M) functions such as spindle forward/reverse/off and coolant ON/OFF.

Spindle Speed Control (S-Function)

Format: Sxxxx

Spindle speed is programmed via S-code. The S-code range is determined by the RPM range of the machine.

There are two types of spindle speed:

1. Constant surface speed (G96), where S value represents surface feet per minute (SFM).
2. Direct revolutions per minute (G97), where S value represents an actual RPM value.

Refer to "[Section 4 – Preparatory Function: G-Codes](#)" of this manual for further information pertaining to G96 and G97.

In determining spindle speeds there may also be gear ranges selected by M-codes. Refer to **Table 12-2**. M40, M41, M42, M43, and M44 are used to select a gear range on machines that have this capability. The manufacturer specifies the RPM range available for each gear. The programmer must select the spindle speeds that are within the currently selected gear range.

Table 12-2, M-Function Selected Ranges

M-Code	Range Selected
M40	Open gear range
M41	1 st Gear range
M42	2 nd Gear range
M43	3 rd Gear range
M44	4 th Gear range

Check the machine tool manual for further details.

Miscellaneous Functions (M-Codes)

Refer to **Table 12-3**. Miscellaneous codes control a variety of machine tool functions. 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 the machine tool manual for a complete list of M-codes.

Table 12-3, M-Code Controlled Functions

M-Code	Function
M2	Program end
M3	Spindle on forward
M4	Spindle on reverse
M5	Spindle off
M8	Coolant on
M9	Coolant off

Control M-Codes

Refer to **Table 12-4**. 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.

Table 12-4, Control M-Codes

M-Code	Function
M00	Program stop. Program stops indefinitely. Press START to resume.
M01	Optional program stop. If corresponding hardware switch is ON, M01 acts as M00. If switch is OFF, program will ignore M01. NOTE: Appropriate hardware is required for M01.
M02	Program end. At M02, the program stops and returns to the first program block.
M19	Orient Spindle and Enable C-Axis Mode. Spindle will orient at orientation speed defined by the builder.
M30	Program end. 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	Subprogram call. 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. M98 is programmed on a block by itself.

(Continued...)

Table 12-4, Control M-Codes (Continued)

M-Code	Function
M100	Mirror image. M100 programmed with axis (M100 X) activates “mirror image” (ON) for that axis. Mirror image reverses the sign (+/-) of subsequent numbers. More than one axis may be mirrored at once (M100 XY). To cancel mirror image, program M100 on a block by itself. NOTE: Cutter compensation is mirrored; therefore, switching from G41 to G42 is unnecessary.
M900 C	Synchronize C to X and Z.
M901 C	Un-Synchronize C from X and Z.
M9351 X302	Clear. Use to clear the Draw Graphics screen at any time. No other code is allowed on this block.

Order of Execution

The order of execution for available codes is T, M, S, F, G, X, Z, and C.

C-Axis Programming

NOTE: The builder determines if the machine will have C-Axis programming capability.

Enabling C-Axis Programming Mode

M19 orients the spindle and enables C-Axis programming. The spindle orientation will be done in the forward (positive) direction at orientation speed defined by the builder. The C-Axis position display is only enabled when C-Axis programming mode is active. G94 (Feed Per Minute) must be enabled when using C-Axis programming. C-Axis programming will not work in G95 (Feed Per Revolution). C-Axis programming can be used in MDI or in a program. C-Axis programming mode allows the spindle to be used as a rotary axis. Thus, the units displayed and used in programming the C-Axis are degrees. **Example:**

G94	* Enable Feed Per Minute
M19	* Enable C Axis mode (spindle will orient)
C45	* Position C Axis at 45 degrees

Disabling C-Axis Programming Mode

C-Axis programming mode is disabled by programming an M5 (Spindle Off). In C-Axis programming mode, M3 (Spindle Forward) and M4 (Spindle Reverse) commands should not be programmed. If any spindle command is programmed, an error message is generated.

C-Axis Jog

The Manual Panel can be used to jog the C-Axis to position. To set the Manual Panel for jogging the C-Axis, set the Axis Selector to **C** and use the **Jog+** and **Jog-** keys as needed. The feedrate for jogging is set by the builder.

Programming C-Axis Feedrate

The builder sets the default feedrate for programmed moves. To change the default feedrate, use the address FC. The feedrate unit is in degrees/min. **Example:**

FC 360 * Set the C-Axis feedrate to 360 degrees/min.

Synchronizing/Unsynchronizing C-Axis

C-Axis can be programmed with X and Z in the same block. The C-Axis is by default not synchronized to X and Z. To synchronize the C Axis to X and Z use M900 C. To unsynchronize the C-Axis from X and Z, use M901 C. M900 C and M901 C should be programmed in a block by themselves.

Section 13 - Simplified Command Interface (SCI) Mode

SCI provides an additional set of soft keys that contain the most commonly used functions for a two-axis lathe. It is designed to simplify programming and does not require G-codes. SCI Mode simplifies manual operation. It is used in conjunction with the CNC handwheel for the X- and Z-axes.

Activating the SCI Mode

To activate SCI Mode:

1. In Manual Mode, press **SHIFT**. The soft keys change. Press **SCI (F4)**. In Teach Mode, press **SCI (F3)**. The CNC will activate and display the SCI soft keys.

NOTE: The machine builder can configure SCI to be the default mode, instead of Manual Mode.

Refer to **Figure 13-1**.

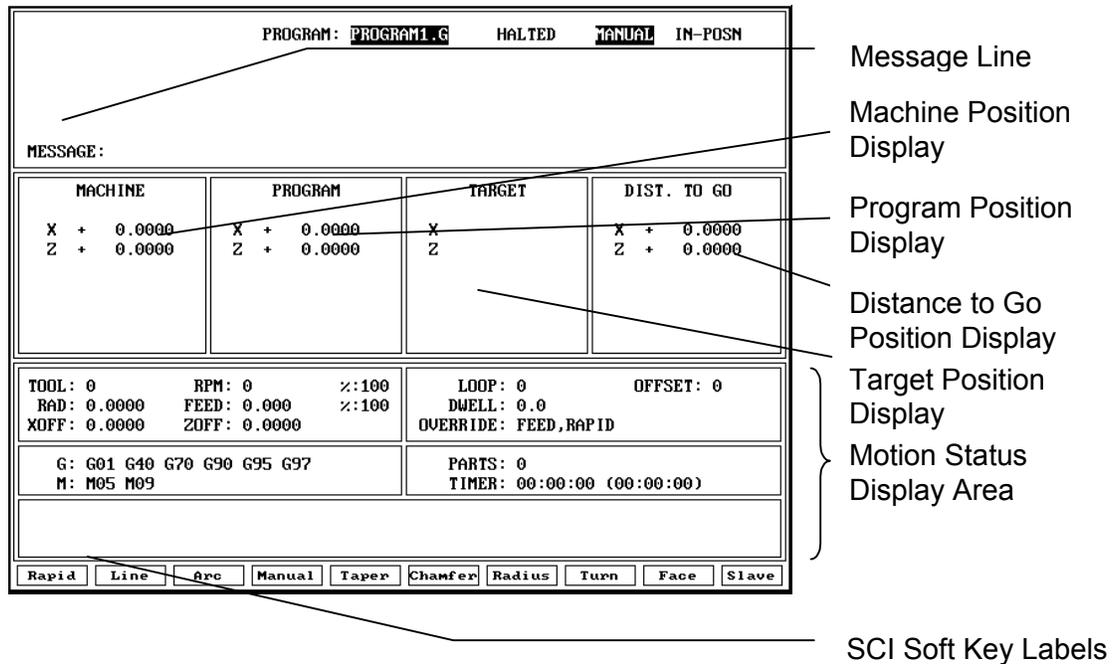


Figure 13-1, Simplified Command Interface Screen Display

Table 13-1 lists the features available in SCI Mode. The SCI Mode can only be activated from the Manual screen. When SCI Mode activates, the soft key labels change.

Table 13-1, Screen Features

Feature	Description
Move Type Being Performed	Specifies the activated SCI soft key.
Input Labels	Specifies entry fields required to complete the command.
Entry Field	Enter values here.
Message Line	Messages, prompts and reminders are displayed here.
Motion Display	Displays the machine's X and Z position coordinates in reference to Part Zero (Program); programmed endpoint (Target) and Distance To Go (distance to go to reach Target position).
Machine Home Position Display	Displays machine's X and Z position coordinates in reference to Machine Home.
Machine Status	Displays operating information.
SCI Soft Key Labels	These labels identify the function of the soft key. Soft key functions change when the SHIFT key is pressed.
Active Soft Key (highlighted)	The currently active SCI soft key will be highlighted on the screen.

SCI Mode Soft Keys

Table 13-2 lists the SCI soft key functions. All moves are manual operations.

Activating the SHIFT SCI Soft Key Menu

There are ten SCI soft keys that do not appear when the mode is activated. To activate the additional soft keys, press **SHIFT**. The soft keys change. Press the appropriate soft key [**Limits (F1)** to **Exit (F10)**].

Table 13-2, SCI Mode Soft Keys

Soft Key Label	Soft Key Number	Function
Rapid	F1	Executes a linear move to target location at rapid speed.
Line	F2	Executes a linear move to target location at a specified feedrate.
Arc	F3	Executes a circular move to the target location based on arc center or radius input at the specified feedrate.
Manual	F4	Cancels an active SCI command.
Taper (Handwheel)	F5	Inputs a taper move in the positive/negative direction specified in the command.
Chamfer (Handwheel)	F6	Executes a 45-degree taper move in the positive/negative direction specified in the command.
Radius (Handwheel)	F7	Executes an arc move according to specified XZ arc centers, direction and sweep angle.
Turn	F8	Executes rough turning cycle on a piece of stock to a specified length and depth. You specify the depth per pass and feedrate.
Face	F9	Inputs a rough facing cycle on a piece of stock to a specified depth and width. You specify the depth per pass and feedrate.
Slave	F10	Activates/deactivates the Slave Mode. When the Slave Mode is active the X-axis is slaved to the Z-axis. The Slave Mode can only be enabled with the Taper (F5) , Chamfer (F6) , and Radius (F7) SCI commands.
Limits	SHIFT + F1	Sets software limits.
FPM	SHIFT + F2	Activates Feed Per Minute (G94) Mode.
FPR	SHIFT + F3	Activates Feed Per Revolution (G95) Mode.
CSS	SHIFT + F4	Activates Constant Surface Speed (CSS) and sets gear range.

(Continued...)

Table 13-2, SCI Mode Soft Keys (Continued)

Soft Key Label	Soft Key Number	Function
RPM	SHIFT + F5	Sets the spindle to RPM Mode and sets spindle speed and gear range.
Home	SHIFT + F6	Executes a homing sequence.
Thread	SHIFT + F7	Executes a threading move to the target location according to the parameters specified.
Abs/Inc	SHIFT + F8	Switches the CNC between Absolute and Incremental Modes.
Prev	SHIFT + F9	Deactivates SCI soft key menu. Switches back to Manual or Teach Mode.
Exit	SHIFT + F10	Exits the CNC Control software and returns the user to the software selections screen.

Deactivating the SCI Mode

In SCI Mode, press **SHIFT**. The soft keys change. Press **Prev (F9)**. The CNC reverts to the screen that was previously active, Manual.

Canceling an SCI Command

To deactivate any of the Simplified Command Interface commands, press the soft key used to activate the command. The command deactivates and the command's input labels and entry fields will no longer be displayed.

Canceling a Running SCI Command

To abort any running SCI command, press **Manual (F4)**.

Clearing Messages

If the CNC displays a message, press the **CLEAR** key to clear the message.

Clearing an Entry Field

To clear an SCI entry field, press the **SPACE** key. The CNC clears the entry field where the insertion point is located.

Returning to the Software Selection Screen

Press **SHIFT**. The soft keys change. Press **Exit (F10)** to return to the Software Options screen. This is the first screen encountered when the software is activated, containing Software Options:

1. CNC Control
2. Setup Utility
3. Motion Setup/Testing

Exiting SCI Mode

Press **SHIFT + F10 (Exit)** to cancel SCI Mode. The CNC returns to the Manual screen.

Commanding Moves in SCI Mode

NOTE: The X End option can be accessed by pressing the **X** key. The Z End option can be accessed by pressing the **Z** key.

The servos must be on to execute any SCI command. As in the Manual Data Input (MDI) Mode, values cannot be stored in a program while the SCI Mode is active. SCI moves are one-shot moves. You can store data in a program only in Edit Mode, with the appropriate Program Listing activated.

Performing Rapid Moves (G00)

Refer to **Table 13-3**. Input a linear move to the target location.

Table 13-3, Rapid Movement Input Labels

Input Label	Entry Field Input Format	Description
X End	xxx.xxxx	coordinate of X endpoint
Z End	xxx.xxxx	coordinate of Z endpoint
Tool	Nnn	activate tool # n

Input a rapid move:

1. With the SCI soft key menu active, press **Rapid (F1)**. The soft key highlights and the Rapid move input labels and entry fields are displayed.
2. Enter the parameters listed above and press **START**. The command is executed.

Activating and Changing a Tool

A tool can only be activated from the **Rapid (F1)** entry field. To activate a tool:

1. If necessary, stop the spindle by pressing **SPINDLE OFF**.
2. Input the rapid move X and Z end coordinates and the tool number.
3. Press **START**. The CNC performs the move. Tool offsets take effect.
4. Perform the tool change, if necessary.
5. Restart the spindle by pressing **SPINDLE FORWARD**.

NOTE: Make sure the spindle is off before performing a tool change. The machine builder decides whether the spindle must stop for tool changes. This parameter is in the Setup Utility.

Performing Line Moves (G01)

Refer to **Table 13-4**. Executes a linear move to the target location.

Table 13-4, Line Move Input Labels

Input Label	Entry Field Input Format	Description
X End	xxx.xxxx	Coordinate of X end point
Z End	xxx.xxxx	Coordinate of Z end point
Feed	xxxx.x or xx.xxx	IPM (G94) or IPR (G95), current active feedrate type.

The Feedrate will be in Inches/MM per Minute (G94) or Inches/MM per Revolution (G95), whichever is active before the Line move is made.

To perform a linear move:

1. With the SCI soft key menu active, press **Line (F2)**. The soft key highlights and the Line move input labels and entry fields are displayed.
2. Enter the parameters listed above and press **START**. When **START** is pressed, the command is executed.

Performing Arc Moves (G02 or G03)

Refer to **Table 13-5**. Executes an arc move to the target location.

Table 13-5, Arc Move Input Labels

Input Label	Entry Field Input Format	Description
Direction	Cw/Ccw	Activates Cw (G2) or Ccw (G3) circular motion
X End	xxx.xxxx	Coordinate of X end point
Z End	xxx.xxxx	Coordinate of Z end point
Radius	xxx.xxxx	Arc radius
X Cent	xxx.xxxx	Coordinate of X center of arc
Z Cent	xxx.xxxx	Coordinate of Z center of arc
F Feed	xxxx.x or xx.xxx	IPM (G94) or IPR (G95), current active feedrate type.

Press the **(+/-)** key to switch the **Direction** between Cw (Clockwise) and Ccw (Counterclockwise). Use only one of the following parameters to define the arc:

- Arc radius, or
- Arc centers (X Cent, Z Cent)

In either case, the endpoint must be specified. The centerpoint of the arc will be either Absolute or Incremental. This parameter is set up in the Setup Utility by the machine builder. The default Arc Centerpoint is Incremental.

If no feedrate is set, the CNC uses the currently active feedrate.

To input an arc move:

1. With the SCI soft key menu active, press **Arc (F3)**. The soft key highlights and the Arc move input labels and entry fields are displayed.
2. Enter the parameters listed above and press **START**. The command is executed.

Setting Software Limits (G22)

NOTE: This command is similar in execution and function to the G22 Software Limits command.

Refer to **Table 13-6**. Set the CNC software limits by specifying the following parameters:

Table 13-6, Arc Move Input Labels

Input Label	Entry Field Input Format	Description
X+ Limit	xxx.xxxx	Sets the positive X software limit.
Z+ Limit	xxx.xxxx	Sets the positive Z software limit.
X- Limit	xxx.xxxx	Sets the negative X software limit.
Z- Limit	xxx.xxx	Sets the negative Z software limit.
Event	Enable/Disable	Enables or disables the software limits.

The software limits feature creates an envelope limiting the tool's range of travel. The positive X and Z limits represent the extreme distance the tool can travel in a positive X and Z direction. The negative X and Z limits represent the extreme distance the tool can travel in a negative X and Z direction.

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

The location of the positive and negative software limits (+X, -X, +Z, -Z) reverses from front to rear tooling. Refer to [Figure 13-2, Rear Tool Post Software](#) and [Figure 13-3, Front Tool Post Software Limits](#) for the location of positive/negative software limits.

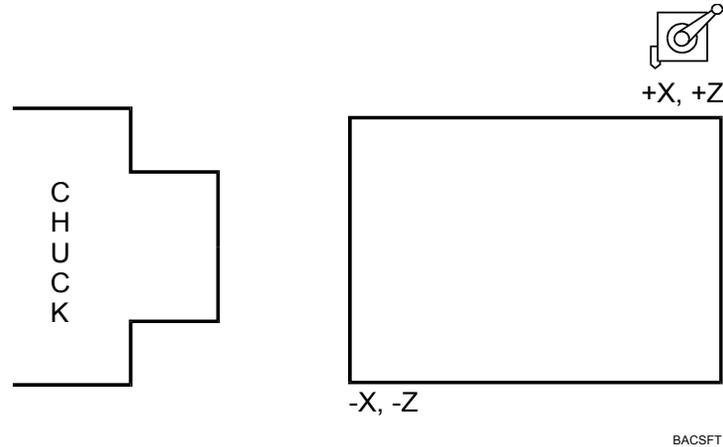


Figure 13-2, Rear Tool Post Software

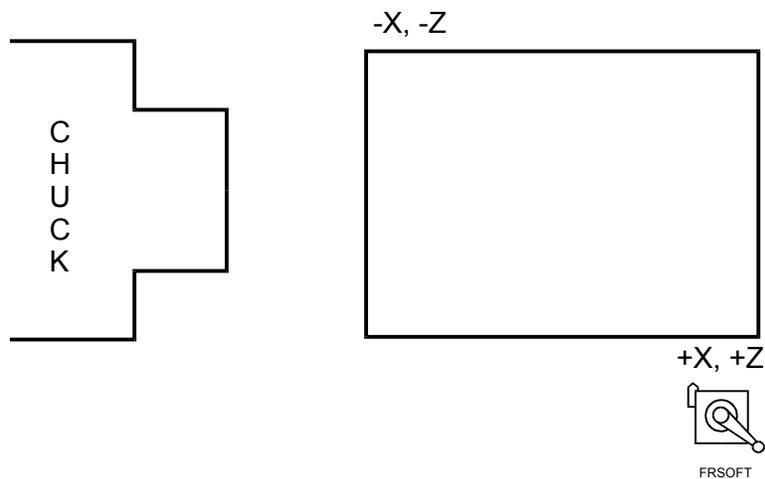


Figure 13-3, Front Tool Post Software Limits

The CNC will not travel outside the specified software limits once they are enabled. Use the (+/-) key to switch the **Event** entry field between enable and disable.

To enable the entered software limits:

1. With the SCI soft key menu active, press **SHIFT**. The soft keys change. Press **Limits (F1)**. The soft key highlights and the Software Limits input labels and entry fields are displayed.
2. Enter the applicable positive and/or negative software limits. Highlight the **Event** entry field. Switch the **Event** field to Enable and press **START**. The CNC enables the software limits.

To disable the software limits:

1. With the SCI soft key menu active, press **SHIFT**. The soft keys change. Press **Limits (F1)**. The soft key highlights and the Software Limits input labels and entry fields are displayed.
2. Highlight the **Event** entry field. Switch the **Event** field to Disable and press **START**. The CNC disables the software limits.

Homing the Machine (G28)

To home all axes, press **SHIFT**. The soft keys change. Press **Home (F6)**.

NOTE: The CNC will only perform a homing sequence on axes that have a homing parameter specified in the Setup Utility.

Making a Rough Turning Cut (G76)

NOTE: This cycle is similar in format and execution to a G76 Rough Turning Cycle. Only use this feature for rectangular cuts (90 degree corners).

Refer to **Table 13-7**. Make a rough cut by specifying the following parameters:

Table 13-7, Rough Cut Input Labels

Input Label	Entry Field Input Format	Description
Length	xxx.xxxx	The length of the roughing pass in Z.
Depth	xxx.xxxx	The depth in X to which the stock will be cleared.
Step	xxx.xxxx	The depth in X which will be removed in a single pass.
Feed	xxx.x or xx.xxx	IPM (G94) or IPR (G95), current active feedrate type.

All parameters are sign dependent, incremental values. In Diameter Mode, the CNC automatically adds 0.2 inch/4 MM (0.1inch/2MM, radius) in X to the first pass depth. (You must position the tool in X before beginning the first pass.) Subsequent passes are made at the depth entered by the operator.

Feedrate is optional. If no feedrate is entered, the CNC executes the cycle at the current feedrate.

To make a rough turning cut:

1. Position the tool at the starting point of the roughing cycle. Make sure the tool is 0.1 inch/2 MM from the work in X.
2. With the SCI soft key menu active, press **Turn (F8)**. The soft key highlights and the Turning cycle input labels and entry fields are displayed.
3. Enter the parameters listed above and press **START**. The CNC executes the rough turning cut.
4. The CNC returns to the starting point after completing the cycle.

Making a Rough Facing Cut (G77)

NOTE: This cycle is similar in format and execution to a G77 Rough Facing Cycle. Perform only rectangular cycles (90 degree corners).

Refer to **Table 13-8**. Perform a rough facing cut on a piece of stock by specifying the following parameters:

Table 13-8, Rough Facing Cut Input Labels

Input Label	Entry Field Input Format	Description
Length	xxx.xxxx	The length of the roughing pass in X.
Depth	xxx.xxxx	The final depth in Z to which the stock will be cleared.
Step	xxx.xxxx	The depth in Z which will be removed in a single pass.
Feed	xxx.x or xx.xxx	IPM (G94) or IPR (G95), current active feedrate type.

All parameters are sign dependent, incremental values. The CNC automatically adds 0.1 inch/2 MM (diameter values) in Z to the first pass depth. (You must position the tool in Z before beginning the first pass.) Subsequent passes are made at the specified depth (step).

Feedrate is optional. If no feedrate is entered, the CNC executes the cycle at the current feedrate.

To make a rough facing cut:

1. Position the tool at the starting point of the facing cycle. Make sure the tool is 0.1 inch/2 MM from the work in Z.
2. With the SCI soft key menu active, press **Face (F9)**. The soft key highlights and the Facing cycle input labels and entry fields are displayed.
3. Fill in the entry fields listed above and press **START**. The CNC executes the rough facing cycle. 2MM (0.1 inch) is added to the depth of the first pass.
4. The CNC returns to the starting point after completing the cycle.

Cutting Threads (G83)

NOTE: This cycle is similar in function and execution to the G83 Threading Cycle.

Refer to **Table 13-9**. Perform a threading cycle to the target location by specifying the following entry fields:

Table 13-9, Threading Cycle to the Target Location

Input Label	Entry Field Input Format	Description
TPI**	xxx.xxxx	Threads per Inch. Specifies the number of threads per inch that will be cut.
Lead**	xxx.xxxx	Incremental Z distance between one thread and the next.
Length	xxxx.x or xx.xxx	Incremental Z distance specifying the length of the threading pass.
FirstCut	xxx.xxxx	Incremental X distance, specifying the depth of the first threading pass.

**Specify TPI or Lead, but not both.

All entry fields are sign dependent, incremental values. In Diameter Mode, the CNC automatically adds 0.2 inch/4 MM (0.1inch/2MM, radius) in X to the depth of the first threading pass. (You must position the tool in X before beginning the first pass.) The number and depth of subsequent passes are determined by the CNC based on the pitch of the thread and the depth of the first cut.

To cut threads:

1. Position the tool at the starting point of the threading cycle. Make sure the tool is 0.1 inch/2 MM from the work in X.
2. With the SCI soft key menu active, press **SHIFT**. The soft keys change. Press **Thread (F7)**. The soft key highlights and the Threading cycle input labels and entry fields are displayed.
3. Fill in the entry fields listed above and press **START**. The CNC executes the threading cycle.
4. The CNC returns to the starting point after completing the cycle.

Activating Absolute (G90) or Incremental (G91) Mode

Press **Abs/Inc (SHIFT + F8)** to switch the CNC between Absolute and Incremental Modes. The active mode (Absolute, G90 or Incremental, G91) will appear in the Machine Status Display Area of the screen.)

To switch the **Abs/Inc** Mode:

1. In SCI Mode, press **SHIFT**. The soft keys change.
2. Press **Abs/Inc (F8)**. The CNC switches the Absolute/Incremental Mode setting.

To change the Absolute/Incremental Mode while another SCI command is active:

1. With another SCI command activated, press **SHIFT**. The soft keys change.
2. Press **Abs/Inc (F8)**. The CNC switches the mode.

Setting the Feedrate to Feed per Minute (G94)

Refer to **Table 13-10**. Set the CNC to the Feed per Minute (G94) mode. The feedrate for Line and Arc moves will be at the specified **Feed**.

Table 13-10, Setting Feedrate to Feed per Minute

Input Label	Entry Field Input Format	Description
Feed	xxxx.x	Sets the CNC to the FPM (G94) rate specified in the entry field.

To set the feedrate to FPM (G94):

1. With the SCI soft key menu active, press **SHIFT**. The soft keys change. Press **FPM (F2)**. The soft key highlights and the FPM input labels and entry fields are displayed.
2. Enter the desired feedrate (inch/MM) per minute. Press **START**. The CNC switches the active feedrate to FPM Mode. Line and Arc moves will be performed at the specified FPM.

Setting the Feedrate to Feed per Revolution (G95)

Refer to **Table 13-11**. Set the CNC to Feed Per Revolution (G95) Mode. The feedrate for Line and Arc moves depends upon the programmed spindle RPM. For each revolution of the spindle, the CNC advances linearly by the **Feed** amount.

Table 13-11, Setting Feedrate to Feed per Revolution

Input Label	Entry Field Input Format	Description
Feed	xxxx.x	Sets the CNC to the FPR (G95) rate specified in the entry field.

To set the feedrate to FPR (G95):

1. With the SCI soft key menu active, press **SHIFT**. The soft keys change. Press **FPR (F3)**. The soft key highlights and the FPR input labels and entry fields are displayed.
2. Enter the desired feedrate (inch/MM) per revolution. Press **START**. The CNC switches the active feedrate to FPR Mode. For every revolution of the spindle, the CNC advances the axes of motion by the specified FPR.

Setting Spindle Speed to Constant Surface Speed (G96)

Refer to **Table 13-12**. Set the CNC's spindle speed to the Constant Surface Speed (CSS, G96) Mode. In this mode, the CNC adjusts the spindle RPM to maintain a constant surface speed. Spindle speed is inversely proportional to the distance from the part centerline.

Table 13-12, Setting Spindle Speed to Constant Surface Speed

Input Label	Entry Field Input Format	Description
GearCode	M4X (41, 42, 43)	Use an M-code to specify the gear range to be used. Gear code is optional.
Speed	xxxx.x	Sets the CNC to the surface feet (MM/Meter) per minute specified in the entry field. Speed must be given.

To set the CNC's spindle speed to CSS Mode:

1. With the SCI soft key menu active, press **CSS (SHIFT + F4)**. The soft key highlights and the CSS input labels and entry fields are displayed.
2. Enter the desired speed in feet (Meter/MM) per minute and the gear range to be used. Press **START**. The CNC switches the spindle to CSS Mode. The CNC constantly adjusts the spindle speed to maintain a constant surface speed, specified in the **Speed** entry field, for the axes of motion.

Setting the Spindle Speed to Revolutions per Minute (G97)

Refer to **Table 13-13**. Set spindle speed to the Revolutions Per Minute (RPM, G97) Mode. The spindle rotates at a constant RPM, specified in the entry field.

Table 13-13, Setting Spindle Speed to Revolutions per Minute

Input Label	Entry Field Input Format	Description
Gear Code	M4X (41, 42, 43)	Use an M-code to specify the gear range to be used. Optional.
Speed	xxxx.x	Sets the CNC to the FPR (G95) specified in the entry field. Required.

To set the CNC's spindle speed to RPM Mode:

1. With the SCI soft key menu active, press **SHIFT**. The soft keys change. Press **RPM (F5)**. The soft key highlights and the RPM input labels and entry fields are displayed.
2. Enter the desired spindle speed (inch/MM) per revolution and the gear range to be used. Press **START**. The CNC switches the spindle to the RPM mode. The CNC applies a constant RPM, specified in the **Feed** entry field, to the spindle.

Slave Mode

In Slave Mode, the CNC does not move the machine. The X-axis is slaved to the movement of the Z-axis handwheel. This means that both axes move when you rotate the Z-axis handwheel.

Press **Slave (F10)** to switch the Slave Mode OFF or ON. The soft key highlights when Slave Mode activates. When Slave Mode deactivates, the soft key is no longer highlighted.

For **Taper (F5)**, **Chamfer (F6)**, and **Radius (F7)** moves, the CNC enters the Slave Mode when the **START** key is pressed.

When the Slave Mode is deactivated, you can move the X and Z handwheels independently.

Cutting a Taper

Refer to **Table 13-14**. Cut a Taper by specifying the following entry fields:

Table 13-14, Cutting a Taper

Input Label	Entry Field Input Format	Description
Type	PosAngle/NegAngle	Specifies whether the angle is measured in a positive or negative direction from the Z-axis. The 3 o'clock position along the Z-axis is zero degrees.
Angle	xxx.xxx	The angle of the taper, measured from the Z+ axis.

The **PosAngle/NegAngle** option is switched by pressing the (+/-) key. The given angle is always measured from the standard 3 o'clock reference. Refer to **Figure 13-4**.

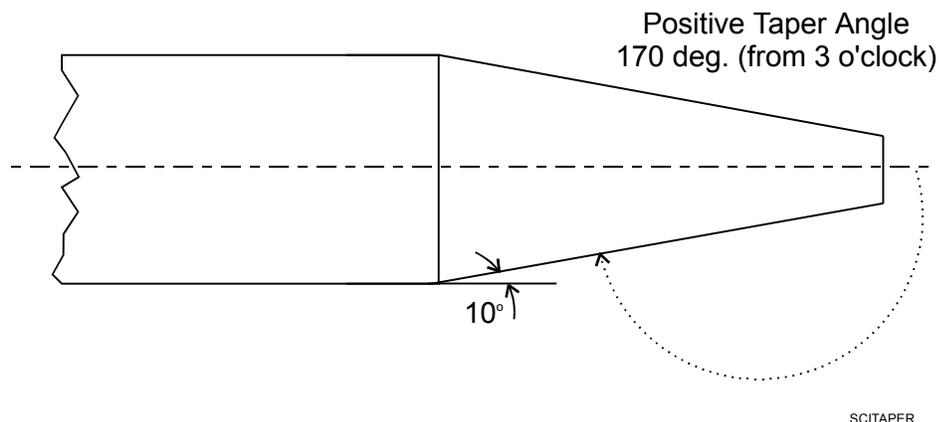


Figure 13-4, Defining a Taper Angle and Direction

To cut a taper:

1. Position the tool at the starting point of the first taper pass.
2. With the SCI soft key menu active, press **Taper (F5)**. The soft key highlights and the Taper move input labels and entry fields are displayed.
3. Enter the parameters listed above and press **START**. Slave Mode activates.
4. Using the Z-axis handwheel, move the tool to perform the first taper pass. Stop turning the handwheel when the tool has cleared the stock.

5. Press **Slave (F10)** to deactivate the Slave Mode and manually re-position the tool at the starting point of the next taper pass.
6. Press **Slave (F10)** to activate the Slave Mode and perform the next taper pass.
7. Repeat steps 5 and 6 until the taper is complete.

Cutting a Chamfer

Refer to **Table 13-15**. Cut a Chamfer move by specifying the following entry fields:

Table 13-15, Cutting a Chamfer

Input Label	Entry Field Input Format	Description
Type	Positive/Negative	Specifies whether the angle is measured in a positive or negative direction from the Z-axis. The 3 o'clock position along the Z-axis is zero degrees.

A Chamfer is actually a 45-degree taper, measured from the Z-axis.

Use the (+/-) key to switch the **Type** between Pos (positive) and Neg (negative). To cut a chamfer, follow the instructions for cutting a taper pass. Refer to **Table 13-16** to define chamfer types.

Table 13-16, Positive and Negative Chamfers

Positive Chamfers	Negative Chamfers
I.D. Left	I.D. Right
O.D. Right	O.D. Left

Refer to **Figure 13-5**. The diagram shows the different chamfer types possible.

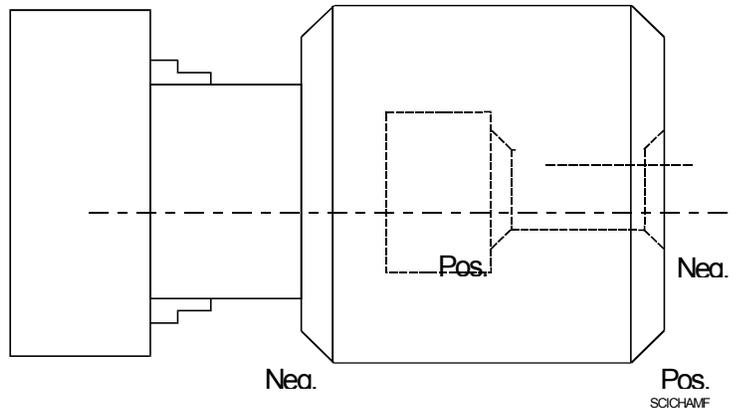


Figure 13-5, Positive and Negative Chamfers

Cutting a Radius

Refer to **Table 13-17**. Cut a radius by specifying the following entry fields:

Table 13-17, Cutting a Radius

Input Label	Entry Field Input Format	Description
X Cent	xxx.xxxx	The X-axis center of the arc.
Z Cent	xxx.xxxx	The Z-axis center of the arc.
Cw/Ccw	Cw/Ccw	Defines the direction of an optional sweep angle (in conjunction with a beginning angle).
SweepAng	xxx.xxxx	Together with the beginning angle, defines an area beyond which the tool cannot travel during the Radius move.

The starting angle is determined by the position of the tool at the beginning of the Radius move and the location of the arc centerpoint (X Cent, Z Cent or XZ Cent).

To cut an arc:

1. Position the tool at the starting point of the arc move.
2. With the SCI soft key menu active, press **Radius (F7)**. The soft key highlights and the Radius move input labels and entry fields are displayed.
3. Enter the parameters listed above and press **START**. Slave Mode activates.
4. Using the Z-axis handwheel, move the tool to perform the first radius pass. Stop turning the handwheel when the tool has cleared the stock.
5. Press **Slave (F10)** to deactivate the Slave Mode and manually reposition the tool at the starting point of the next pass, if more than one pass is required to complete the arc to the proper dimensions.
6. Press **Slave (F10)** to activate the Slave Mode and perform the next pass.
7. Repeat steps 5 and 6 until the arc is complete.

Section 14 - Shape Editor

The Shape Editor is a graphic CNC programming tool. The Shape Editor enables you to write part subprograms without using G-codes. This section describes the Shape Editor's features and outlines its use.

Before using the Shape Editor, the user should be familiar with programming techniques, including G-codes. Refer to **Figure 14-1**.

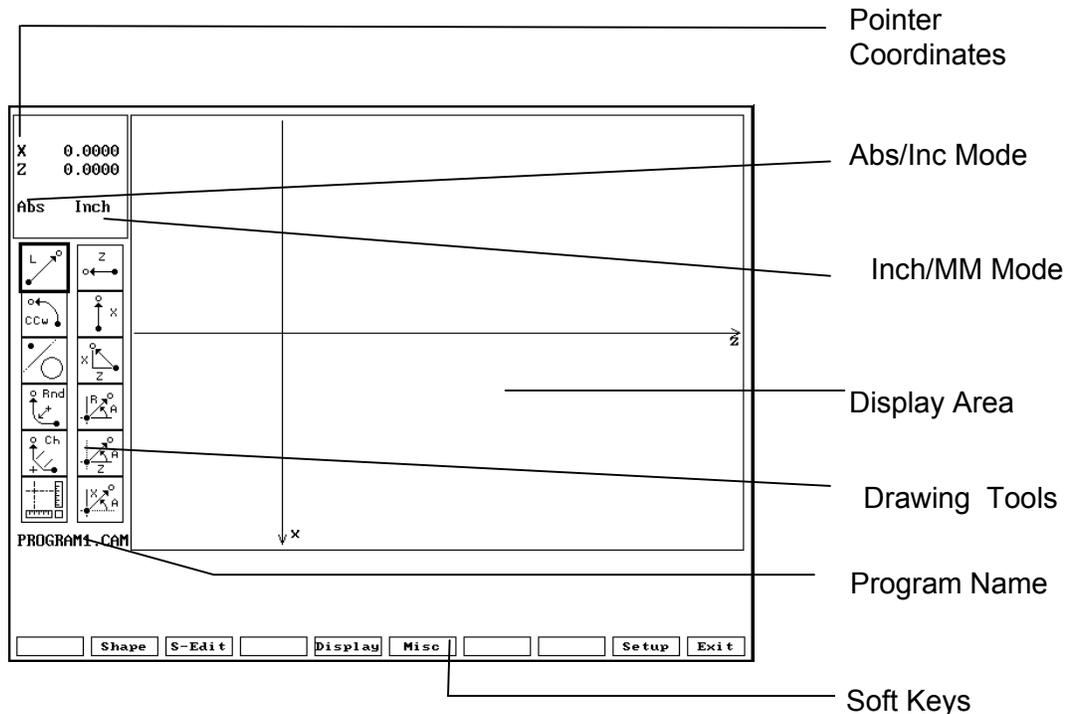


Figure 14-1, Shape Editor Screen

The Shape Editor's features are described below:

Pointer Coordinates

Indicate X, Z position of the pointer.

Abs/Inc Mode indicator

Indicates the Absolute or Incremental Mode of the Shape Editor.

NOTE: The resulting subprogram will execute moves using absolute dimensions.

Inch/MM Mode Indicator

Indicates Inch or MM Mode active.

Pointer

Used to select specific points for editing.

Display Area	Viewing area.
Drawing Tools	Tools used to create and edit shapes.
Soft Keys	<p>Shape Editor has two sets of soft keys: a default set and the Shape (F2) set.</p> <p>The default soft keys provide tools for changing the setup, adjusting the display and managing shapes.</p> <p>The Shape soft keys provide tools for moving the pointer, deleting shape segments and deleting geometry elements. Shape soft keys are switched on and off by pressing the Shape (F2) soft key. The Shape (F2) soft key label is highlighted when these soft keys are active. It is best to leave the Shape (F2) soft keys active while editing a shape.</p>
Program Name	Name of program linked to the shapes.

Activating the Shape Editor

To activate the Shape Editor, go to the Program Directory and press **Shapes (F4)**. The program that highlights when the Shape Editor activates will be linked to shapes created or edited during that session.

Shapes are called from the main part program just like a G-code is called from a subprogram. Shapes are not in, nor viewable from the main program. Shapes files have the same filename as the part program minus the “.G” extension.

To activate the Shape Editor:

1. From the Program Directory, move the highlight to select the program that will use the new or edited shapes. Create the program if necessary.
2. Press **Shapes (F4)**. The Shape Editor activates.

Shape Terminology

The Shape Editor works with two types of screen objects “construction geometry” and “shapes”. Only shapes are executed by the CNC. Construction geometry is a calculation tool. Once a path is laid out using construction geometry, you must convert it to a shape for use by the CNC.

Refer to **Figure 14-2**. The parts of a shape are given names to simplify editing instructions. Lines, Arcs, Rounded Corners and Chamfers are all shape segments. Shapes are built from segments connected end to end. Nodes are the connecting points between two segments.

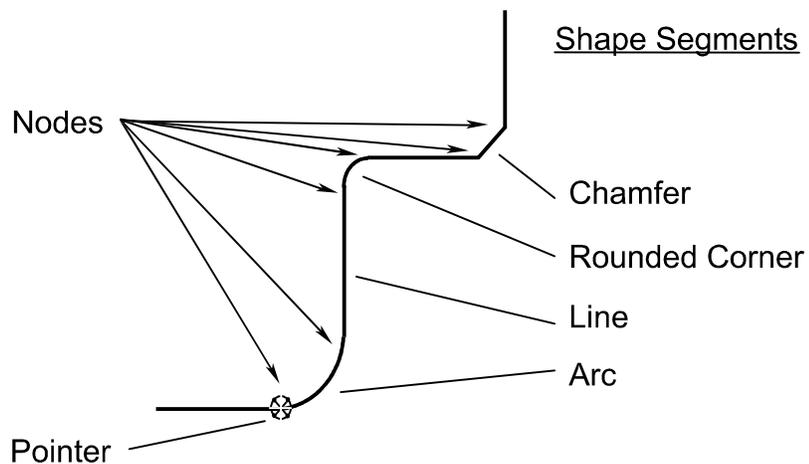


Figure 14-2, Elements of a Shape

SHAPE

Using the Pointer

The pointer can be positioned only on nodes or endpoints. It is used to select items for editing.

The forward end of a shape is the end away from the origin (starting point). The origin is always the back endpoint of the shape. The pointer is moved from node to node along a shape by using the **Back (F4)** and **Forw (F5) Shape** soft keys (- and + keys, respectively, for off-line keyboard programming).

Use **Prev-S (F6)** and **Next-S (F7)** to move the pointer between shapes on the screen.

Position the pointer at the forward end of a shape to add a segment. Shape segments cannot be added to the back end of a shape.

Use the pointer to select nodes for chamfering, rounding, deleting, or moving shapes.

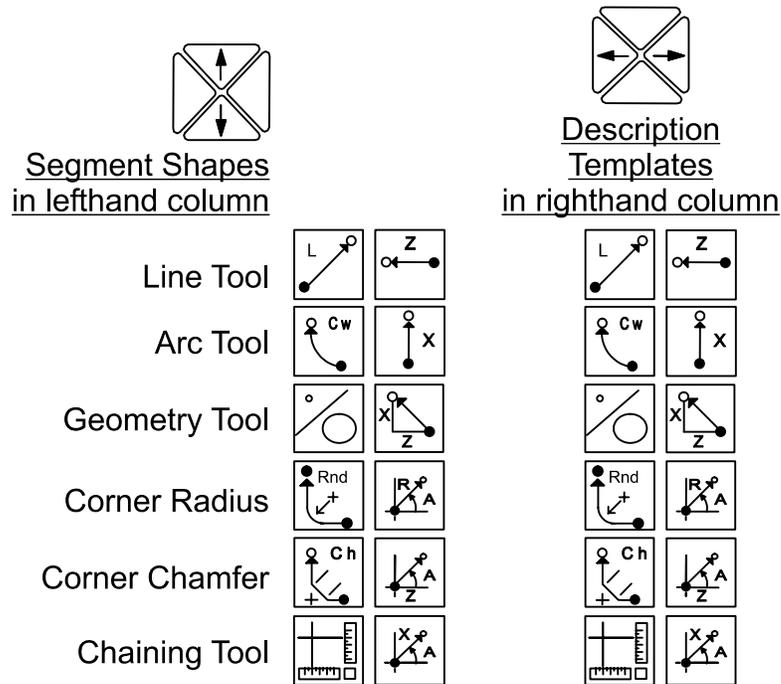
Selecting Editing Tools

Refer to [Figure 14-3, Shape Editing Tools](#). Editing tools create or edit shapes. Select an editing tool from the two columns of templates. Templates in the left column determine a new segment's shape. Templates in the right column determine how the new segment will be defined.

Templates use a standard convention to describe moves. Each template depicts a segment with a hollow ball at one end and a solid ball at the other. The solid ball represents the pointer. The hollow ball is the endpoint of the segment being added. The Shape Editor will prompt you for the letter values shown in the template. Refer to **Table 14-1**.

Table 14-1, Template Lettering Convention

Value Required
X - X-axis endpoint coordinate (usually a diameter).
Z - Z-axis endpoint coordinate.
I - X-axis coordinate of a circle or radius center.
K - Z-axis coordinate of a radius or circle center.
R - radius value.
A - an included or absolute angle.
C - circle.
D - distance.



OPST

Figure 14-3, Shape Editing Tools

Select the shape of a new segment by moving the highlight up or down the left column of templates. Once the desired shape is highlighted, use the **ENTER** key or **ARROWS** to activate a description method in the right column. Only the Line and the Arc have more than one description method. (The geometry tool is covered later.)

Use the Line and Arc tools to rough out the main features of the shape. Once the major features have been drawn, add chamfers and corner rounding, as needed.

Line Tools

Refer to **Table 14-2**. In addition to the six line definition templates in the right column, the endpoint of a line can be described using endpoint definition tools. Refer to [Table 14-3, Line Segment Endpoint Definition Tools](#). To select this option, highlight the Line template, and press **ENTER**. This activates the point definition templates displayed in the right template column.

Table 14-2, Line Segment Tools

Line Segment Template	Line Definition Templates	Values Required
 <p>Right arrow permits use of these line definition templates.</p>	 <p>Line from pointer to endpoint parallel to Z-axis. Defined by Z position.</p>	Use absolute or incremental Z-axis endpoint.
	 <p>Line from pointer to endpoint parallel to X-axis. Defined by X position.</p>	Use absolute or incremental X-axis endpoint.
	 <p>Line from pointer to endpoint in any direction. Defined by X Z endpoint.</p>	Use absolute or incremental X and Z-axis coordinates to define endpoint.
	 <p>Line from pointer to endpoint in any direction. Defined by Radius and angle (polar coordinates).</p>	Use radius and absolute angle.
	 <p>Line from pointer to endpoint in any direction. Defined by Z endpoint coordinate and absolute angle.</p>	Use absolute Z position and angle.
	 <p>Line from pointer to endpoint in any direction. Defined by X endpoint coordinate and absolute angle.</p>	Use absolute X position and angle.

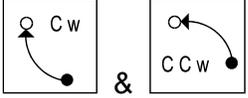
Table 14-3, Line Segment Endpoint Definition Tools

Template	Purpose	Requirements
 <p>Pressing ENTER activates use of these point definition templates.</p> <p>Geometry elements can be used as a reference.</p>	 <p>Defines a point in a X, Z coordinate system.</p>	<p>Use absolute X and Z position of the point.</p>
	 <p>Defines a point at X & Z increment from existing point.</p>	<p>Use incremental X and Z distance from existing point.</p>
	 <p>Defines a point at radius and angle from existing point.</p>	<p>Use distance from existing point and number of degrees from 3 o'clock position.</p>
	 <p>Identifies the center point of an existing circle.</p>	<p>Use center of existing circle.</p>
	 <p>Identifies the intersection points of any two elements in a geometry. Prompts user to select a point when more than one intersecting point exists.</p>	<p>Use one intersection point between two elements.</p>
	 <p>Identifies existing points, usually used when construction of other element requires a reference point.</p>	<p>Use existing point.</p>

Arc Tools

Refer to **Table 14-4**. There are two types of Arcs, clockwise (Cw) and counterclockwise (Ccw). To change arc direction, highlight a template, and press **ENTER**. The definition templates are accessed with the right **ARROW** key, and selected using the up/down **ARROW** keys.

Table 14-4, Arc Segment Tools

Arc Template	Arc Definition Template	Values Required
 <p>Press ENTER to switch between clockwise (Cw) & counter-clockwise (Ccw).</p>	 <p>Arc from pointer to endpoint. Endpoint and center typed by operator.</p>	<p>Use absolute or incremental X, Z to define endpoint. Use absolute or incremental I, K to define center.</p>
	 <p>Arc from pointer to endpoint. Endpoint and radius typed by operator. Center calculated by CNC.</p>	<p>Use radius and absolute or incremental X, Z endpoint.</p>
	 <p>Arc from pointer to endpoint. Center and included angle typed by operator. Endpoint calculated by CNC.</p>	<p>Use absolute or incremental I, K centerpoint. Use incremental angle.</p>
	 <p>Tangent Arc from pointer to endpoint. Endpoint and radius typed by operator. Center is calculated by CNC. Center and radius values must be correct for tangent arc.</p>	<p>Use incremental radius and absolute or incremental X, Z endpoint.</p>
	 <p>Line from pointer followed by tangent arc. Line direction defined by the angle. Arc defined by X, Z endpoint and radius. Arc starts where tangent to line from pointer.</p>	<p>Use radius and absolute or incremental X, Z endpoint. Use absolute angle.</p>

(Continued...)

Table 14-4, Arc Segment Tools (Continued)

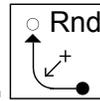
Arc Template	Arc Definition Template	Values Required
	 <p data-bbox="477 447 1117 575">Arc followed by tangent line from pointer to endpoint. Line is defined from pointer to X, Z endpoint. Arc from pointer to tangent intersection with line is defined by radius.</p>	<p data-bbox="1182 342 1398 470">Use radius and absolute or incremental X, Z endpoint.</p> <p data-bbox="1182 485 1354 548">Use absolute angle.</p>

Arc Tools

Use the Rounding tool to insert a radius segment in place of the sharp corner at the node between two line segments.

To round a corner:

1. Move the pointer to the corner being rounded.



2. Highlight the ROUNDING  template, and press **ENTER**. The CNC will prompt for the blend radius.
3. Type the blend radius, and press **ENTER**. A new radius segment will be inserted in place of the original corner.

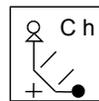
NOTE: Rounding can be performed only on a node located between two shape segments. The forward end or back end of a shape cannot be rounded.

Chamfering Corners

Use the Chamfering tool to chamfer selected corners on the shape.

To Chamfer a corner:

1. Move the pointer to the node to be chamfered.



2. Highlight the CHAMFERING  template, and press **ENTER**. The CNC will prompt for the chamfer distance.
3. Type the desired chamfer distance, and press **ENTER**. A new line segment chamfering the corner will be added to the shape in place of the original corner.

NOTE: Use chamfering only on a node located between two shape segments. The start or end of a shape cannot be chamfered.

Example #1, Creating a Basic Shape

Refer to **Figure 14-4**. This example demonstrates the construction of a simple shape. Naturally, the shape to cut the part will extend beyond the actual dimensions at the ramp on and ramp off moves.

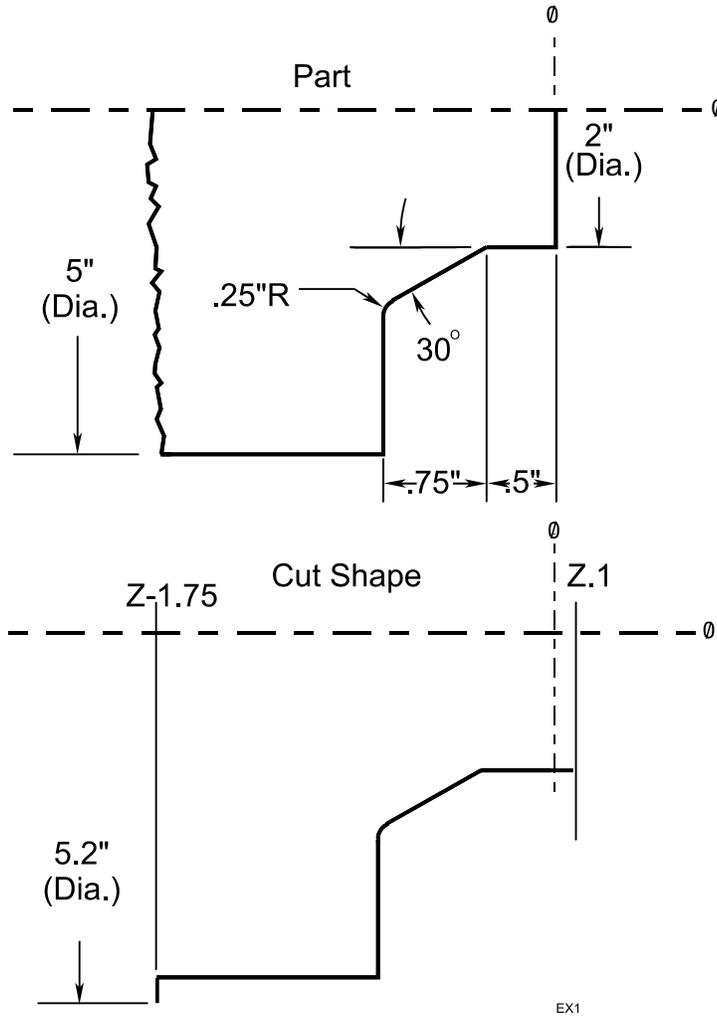


Figure 14-4, Example #1 Shape

All new shapes must start from a point (the shape origin) defined somewhere in the XZ coordinate system. In this example, X1, Z.1 is the starting point. The starting point must be defined using one of the point definition tools. This shape will be the only shape in the program.

NOTE: Point definition templates for shape segments look and act exactly like point definition templates for geometry elements. The difference is in how the point definition template is activated. Point definition templates activated from shape editing tools are always used to designate shape starting point or segment endpoints.

Refer to **Table 14-5**. The only way to start a new shape is with the **Create** selection found in the **S-EDIT** Pop-Up Menu. This example starts in the Shape Editor with the default soft keys on and the Inch/Abs modes active and the CNC in the diameter mode. The XZ point definition tool defines the origin.

Table 14-5, Example #1 Keystrokes

Step 1: Activate SHAPE soft keys and create shape.	
Action	Description
1. From the Program Directory, with the appropriate program name highlighted, press Shapes (F4) .	1. Shape Editor activates.
2. Press Shape (F2) .	2. The Shape soft keys activate.
3. Press S-Edit (F3) .	3. A pop-up activates with Create highlighted.
4. Press ENTER .	4. Creates new shape.
5. The  and  templates are highlighted. Press ENTER .	5. Activates template and prompts, " Enter X value: "
6. Type 1 . Press ENTER .	6. CNC accepts value and prompts, " Enter Z value: "
7. Type .1 . Press ENTER .	7. CNC accepts value.
8. The  template is highlighted. Use ARROWS to select  . Press ENTER .	8. CNC activates template and prompts, " Enter Z value: "
9. Type -.5 . Press ENTER .	9. Creates a line from the pointer to the entered endpoint in Z.
10. The  template is highlighted. Use ARROWS to select  . Press ENTER .	10. Activates template and prompts, " Enter Z value: "
11. Type -1.25 . Press ENTER .	11. CNC accepts Z value and prompts, " Enter angle: "
12. Type 150 . Press ENTER .	12. CNC accepts angle value and creates a line from the pointer to the endpoint. The endpoint is defined by the entered Z coordinate and absolute angle.

Action	Description
13. The  template is highlighted. Use ARROWS to select  template.	13. Activates selected template and prompts, " Enter X value: "
14. Type 5 . Press ENTER .	14. CNC accepts value.
15. Use ARROWS to select  template. Press ENTER .	15. Activates selected template and prompts, " Enter Z value: "
16. Type -1.75 . Press ENTER .	16. CNC accepts Z value.
17. Use ARROWS to select  template. Press ENTER .	17. Activates selected template and prompts, " Enter X value: "
18. Type 5.2 . Press ENTER .	18. CNC accepts X value.

Refer to **Table 14-6**.

Table 14-6, Example #1 Fit Shape to Window

Step 2: Fit shape to window for visual inspection.	
Action	Description
1. Press Shape (F2) .	1. The Shape soft keys activate.
2. Press Display (F5) .	2. A pop-up activates with Fit highlighted.
3. Press ENTER .	3. CNC fits shape to window.

Refer to **Table 14-7**.

Table 14-7, Example #1 Round Radius

Step 3: Insert blend radius.	
Action	Description
1. Press Shape (F2) .	1. The shape soft keys activate.
2. Press Back (F4) three times.	2. The pointer moves to the intersection to be rounded.
3. Use ARROWS to select  template. Press ENTER .	3. CNC activates corner rounding and prompts, Enter blend radius:
4. Type .25 . Press ENTER .	4. CNC rounds intersection at specified radius.

Because there is only one shape in the example, it is Shape #1. Refer to **Table 14-8** that shows how to use this in a program. Refer to [“Executing Shapes in a Program”](#) later in this section of this manual.

Table 14-8, Example #1, G-code Program

Block	Description
N1.G70 G90 G0 X0 Z0 T0	Rapids home, cancels all offsets and activates Inch/Abs Modes.
N2.T1	Activate T1 offsets.
N3.X1 Z.1 G95 F.01 S800 M3	Rapid to start position. Set spindle speed and activate spindle forward.
N4.G73 W1 A.1 R.02 S.01 C1 B1 J.02 K.008	Activate clearing cycle with subprogram call to Shape #1 (W1).
N5.G0 X0 Z0 T0 M5	Cancel T1. Deactivate spindle. Rapid Home.
N6.M2	End program.

Reversing an Arc's Direction

Occasionally you might program an Arc move in the wrong direction. Instead of deleting the segment and redrawing it, reverse the Arc's direction. Arcs at the end of a shape or between any two segments can be reversed.

To reverse an Arc's direction:

1. Move the pointer to the Arc's forward node and press **S-Edit (F3)**. S-Edit Pop-Up Menu will activate.
2. Highlight **Rev Arc**, and press **ENTER**. The arc will be drawn in the opposite direction.

Deleting a Shape

When necessary, an entire shape can be deleted.

To delete an entire shape:

1. Use **Prev-S (F6)** and **Next-S (F7)** to move the pointer to occupy any node within the shape to be deleted.
2. Press **S-Edit (F3)**. The **S-Edit** Pop-Up Menu activates.
3. Highlight **Delete**, and press **ENTER**. The CNC will prompt you to confirm the deletion.
4. Press **Yes (F1)** or **No (F2)** as required. **Yes** causes the shape to be deleted. **No** cancels the process.

Sometimes small bits of a shape will remain on the screen after deletion. Press the **“R” REDRAW** hot key to refresh the screen.

Projecting Line Segments (Restoring Sharp Corners)

Use the **Project** feature to remove a blend radii and restore the sharp corners. This operation is called projecting because projections are added to line segments on both sides of a removed radius.

To remove a radius and restore a sharp corner:

1. Move the pointer to the forward node of the radius being removed.
2. Press **S-Edit (F3)**. The **S-Edit** Pop-Up Menu will activate.
3. Highlight **Project**, and press **ENTER**. The CNC removes the radius and adds line projections forming a sharp angle. The editor will prompt you to join the lines.
4. To join the projections press **Yes (F1)**. The collinear line segments are joined into a single, straight-line segment.

NOTE: Collinear lines are end to end lines that lie in the same direction.

Joining Line Segments

Sometimes what activates to be a single line segment is more than one line segment drawn end to end. The presence of the extra nodes within the segment can be detected by moving the pointer along the segment.

Collinear segments do not affect the execution of the move. Usually it is desirable to join collinear lines to keep the subprogram, and future editing sessions, as simple as possible.

To join collinear lines:

1. Position the pointer at the node between the collinear lines.
2. Press **S-Edit (F3)**. The **S-Edit** Pop-Up Menu activates.
3. Highlight **Join**, and press **ENTER**. The unnecessary node is removed from the line.

Importing Shapes from Other Programs

Sometimes the same shape is used in more than one program. Rather than programming the shape more than once, you can import the shape from its original program.

To import an existing shape from another program:

1. Press **S-Edit (F3)**. The **S-Edit** Pop-Up Menu activates.
2. Highlight **Import**, and press **ENTER**. The Shape Editor prompts operator for the name of the program containing the shape to be imported.
3. Type the program name, and press **ENTER**. The Shape Editor displays a list of the shapes contained in the source program.

4. Highlight the desired shape, and press **ENTER**. The editor will prompt you to find out if the origin of the shape should be changed.
5. Press **Yes (F1)** and type a new origin as prompted or press **No (F2)** to import the shape at its original position.

Deleting a Segment

You can delete segments from an end or the middle of a shape. When a segment is deleted, the node marked by the pointer and the preceding segment are removed. When a segment is deleted from the middle of the shape, the segment in front of the pointer is connected to the preceding node.

To delete the segment at the forward end of the shape:

1. Move the pointer to the forward end of the shape.
2. Press **DelMove (F8)**, and press **ENTER**. The last shape on the segment is removed.

Sometimes small bits of a deleted shape will remain on the screen. Press **"R"**, REDRAW hotkey, to refresh the screen.

Changing the Shape Editor View

Display (F5) accesses all display functions. These functions perform the same operations as described in the Draw Mode, which enables you to test-run a program before cutting any material. Refer to "[Section 8 - Draw Mode.](#)"

Viewing a List of Shape Segment Details

To view the details of the moves in any shape:

1. Press **Misc (F6)**. The **Misc** Pop-Up Menu activates.
2. **Shapes List** is highlighted by default. Press **ENTER**. The list of the shapes available in the file activates.
3. Highlight the shape to be viewed, and press **ENTER**. The details of the moves in the shape activate in a box on the screen.

The following details are presented:

- Starting point (origin)
- Line endpoint and lengths
- Arc directions, centers, and radius

Deactivating and Activating Shapes

Shapes can be deactivated to prevent screen clutter. When a shape is deactivated, it is not deleted from the program. It is simply not displayed. A deactivated shape can be activated at any time.

To deactivate/activate a shape:

1. From the default soft key line press **Setup (F9)**. A pop-up menu activates.
2. Highlight **Shapes**, and press **ENTER**. The Shape listing activates.
3. Highlight the shape being deactivated/activated and switch its status by pressing **ENTER**. The shape status will switch **Off** or **On** accordingly.
4. Press **Setup (F9)** to close the pop-up menus. When all pop-ups have been closed, the CNC activates/deactivates the shape, as indicated.

Using Construction Geometry

Construction Geometry can be used to find path intersections not given on the blueprint. The use of construction geometry can also eliminate the need for many trigonometric functions.

The geometry of a shape can be constructed on the screen using the geometry tools. Once the tool path is solved, the intersecting segments of the geometry are chained together to form a shape.

Construction geometry is useful when working with blueprints that contain limited information. Many combinations of geometry can be entered to help the programmer find intersections.

Points, lines and circles are the basic elements of all construction geometry.

Accessing Geometry Tools

To access the Geometry tools:

1. Move the left column highlight to select the GEOMETRY  template.
2. Press **ENTER** to cycle the right column through three sets of geometry templates. When the desired templates are displayed, use the right **ARROW** key to highlight a template in the right column.
3. Highlight the desired tool, and press **ENTER**.

Points, lines and circles can be defined more than one way. Each geometry tool defines a geometry element a different way. Tools that require a point position to start will prompt you to select and activate an additional point definition tool.

Lines and circles are drawn on the display with dotted lines. Points are marked by X's. All geometry elements are numbered.

Geometry tools generally produce one of the three different elements. Some tools require certain elements to already be part of the geometry. First-time operators are encouraged to experiment with all Shape Editor drawing tools. The following tables (Table 14-9, [Table 14-10, Geometry Line Tools](#), and [Table 14-11, Geometry Circle Tools](#)) are provided as a reference to the features of each tool. Look over each of the tables before attempting the examples.

Point Tools

Some of the point tools define the position of a point in the coordinate system, others allow the user to select an existing point in the geometry.

When using tools that require a starting point, a through point, or a from/to point, the Shape Editor will display a, "**Select point definition . . .**," message. You must then activate an additional point tool and define the required point.

Tools that identify center points or element intersections will prompt for necessary element numbers, highlight all possible points, and prompt the user to select one.

Table 14-9, Geometry Point Tools

Template	Purpose	Requirements
	Defines a point in a geometry.	<ul style="list-style-type: none"> <input type="checkbox"/> Must know absolute X and Z position of the point.
	Defines a point at X & Z increment from existing point.	<ul style="list-style-type: none"> <input type="checkbox"/> One endpoint must already be an element of the geometry. <input type="checkbox"/> Must know incremental X and Z distance from existing point.
	Defines a point at radius and angle from existing point.	<ul style="list-style-type: none"> <input type="checkbox"/> One endpoint must already be an element of the geometry. <input type="checkbox"/> Must know distance from existing point. <input type="checkbox"/> Must know number of degrees from 3 o'clock position.
	Identifies the center point of an existing circle.	<ul style="list-style-type: none"> <input type="checkbox"/> The circle must already be an element of the geometry.

Template	Purpose	Requirements
	Identifies the intersection points of any two elements in a geometry. Prompts user to select a point when more than one intersecting point exists.	<ul style="list-style-type: none"> <input type="checkbox"/> Geometry must already contain an intersection of two elements.
	Identifies existing points, usually used when construction of other element requires a reference point.	<ul style="list-style-type: none"> <input type="checkbox"/> Desired point must already be an element of the geometry.

Line Tools

Some line tools require point definition or identification to start. The Shape Editor will display a message when this is necessary. Line tools that draw tangents to circles will display all possible lines and prompt you to select one. Refer to **Table 14-10**.

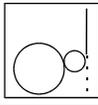
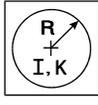
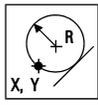
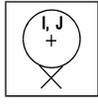
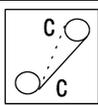
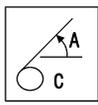
Table 14-10, Geometry Line Tools

Template	Purpose	Requirements
	Constructs a line parallel to X-axis, at given Z position. Prompts operator for Z value.	<ul style="list-style-type: none"> <input type="checkbox"/> Must know absolute position the line crosses the Z-axis.
	Constructs a line parallel to Z-axis, at given X position. Prompts operator for X value.	<ul style="list-style-type: none"> <input type="checkbox"/> Must know absolute position the line crosses the X-axis.
	Constructs a line between any two points. Prompts user to select any convenient point tool when activated.	<ul style="list-style-type: none"> <input type="checkbox"/> Use any method to locate the two endpoints.
	Constructs a line through a point, rotated specified number of degrees from 3 o'clock position. Prompts user to select any point tool to define point.	<ul style="list-style-type: none"> <input type="checkbox"/> Must know angle. <input type="checkbox"/> Use any method to locate point of rotation.
	Constructs line parallel to existing line at specified (positive or negative) distance.	<ul style="list-style-type: none"> <input type="checkbox"/> Line must already be an element of the geometry.
	Constructs line tangent to circle that passes through selected point. Prompts user to select any point tool for point. Always draws tangent lines to both sides of circle; user must select one.	<ul style="list-style-type: none"> <input type="checkbox"/> Circle must already be an element of the geometry. <input type="checkbox"/> Use any method to locate the point.

Circle Tools

Some circle tools require point definition or identification when used, the Shape Editor will display a message when this is necessary. Circle tools that draw circles tangent to other circles, lines or points will construct all possible circles and prompt you to select one. Refer to **Table 14-11**.

Table 14-11, Geometry Circle Tools

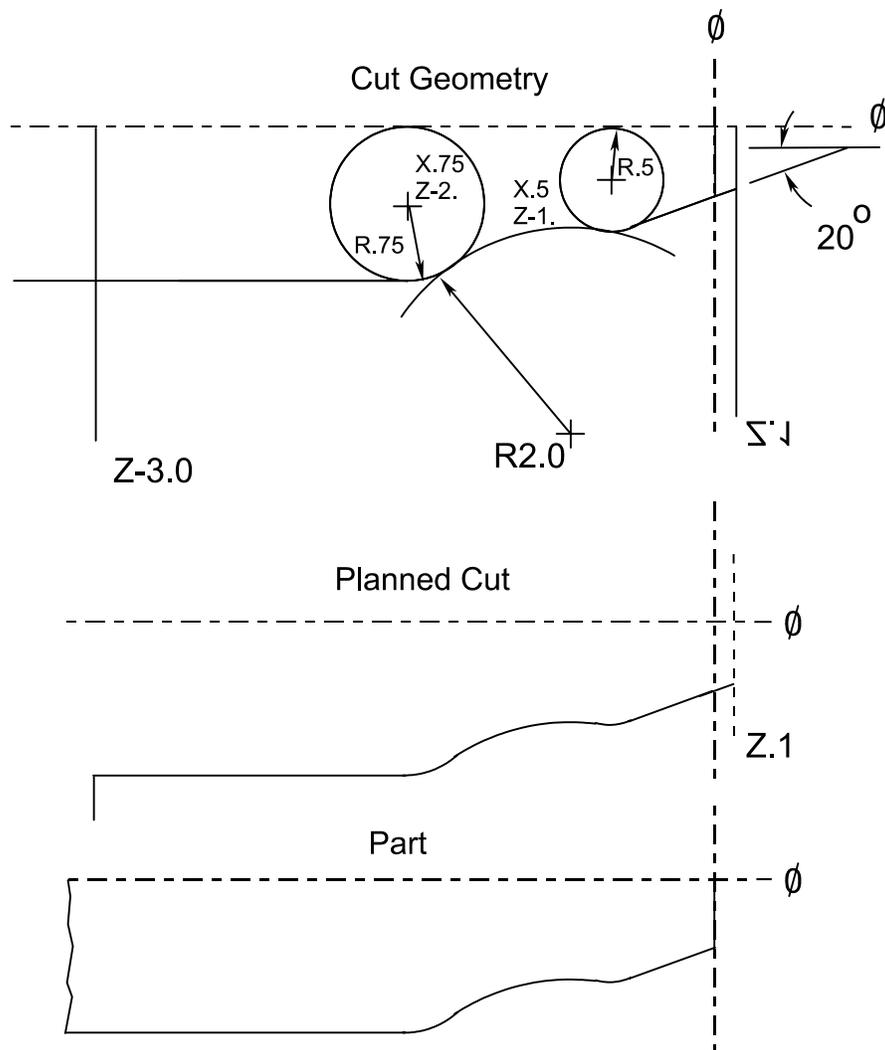
Template	Purpose	Requirements
	Constructs circle of specified radius tangent to any two points, lines or circles. Draws all possible solutions, user must select one.	<ul style="list-style-type: none"> <input type="checkbox"/> Geometry must contain at least two elements.
	Constructs circle of specified radius centered at any position. Prompts user to select any point tool to define circle center point.	<ul style="list-style-type: none"> <input type="checkbox"/> Use any method to locate center point. <input type="checkbox"/> Must know radius.
	Constructs circle of specified radius tangent to line and point. Prompts user to select any point tool to define point.	<ul style="list-style-type: none"> <input type="checkbox"/> Line must already be an element of the geometry.
	Constructs circle tangent to line with center at defined point. Prompts user to select any point tool to define center point.	<ul style="list-style-type: none"> <input type="checkbox"/> Line must already be an element of the geometry. <input type="checkbox"/> Use any method to locate center point.
	Constructs line tangent to any two circles. Always draws the four possible lines; user must select one.	<ul style="list-style-type: none"> <input type="checkbox"/> Two circles must already be elements of the geometry.
	Constructs line rotated specified number of degrees from 3 o'clock position and tangent to existing circle.	<ul style="list-style-type: none"> <input type="checkbox"/> Circle must already be an element of the geometry. <input type="checkbox"/> Must know number of degrees of rotation from 3 o'clock position.

Example #2, Creating Construction Geometry and Chaining it Into a Shape

To better understand geometry tools, work through the following example. Note the templates used and the prompts requesting additional information. Refer to **Figure 14-5** and [Table 14-12, Exercise 2 - Steps 1 through 5](#).

Creating Construction Geometry

This example illustrates the use of construction geometry to calculate the path required to form a shape. When the geometry is complete, segments of the geometry are chained together to form a shape with the chaining tool.



EX2

Figure 14-5, Example #2, Construction Geometry Details

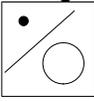
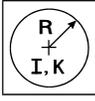
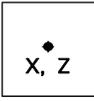


All geometry tools are accessed with the left column GEOMETRY template highlighted. Highlight the template, and press **ENTER** to cycle through three sets of the geometry tool templates in the right column.

This six-step example starts in the Shape Editor with no other shapes or geometry present and the default soft keys showing.

Geometry should be constructed so each node of the required shape is an intersection of two geometry elements.

Table 14-12, Exercise 2 - Steps 1 through 5

Step 1: Create .75" and .5" radius circles. Fit geometry to inspect.	
Action	Description
1. Use ARROWS to highlight the geometry tool template  . Press ENTER until the  template activates in the right column.	1. When ENTER is pressed the CNC switches the available geometry tool menus.
2. Use ARROWS to highlight the  template. Press ENTER .	2. Activates the template and prompts, " Enter R value: "
3. Type .75 (radius of first circle). Press ENTER .	3. Accepts Radius value and prompts, " Select center definition... "
4. The  template is highlighted. Press ENTER .	4. Activates template and prompts, " Enter X value: "
5. Type .75 (X circle center). Press ENTER .	5. Accepts X value and prompts, " Enter Z value: "
6. Type -2 (Z circle center). Press ENTER .	6. Creates first circle.
7. The  template is highlighted. Press ENTER .	7. Activates template and prompts, " Enter R value: "
8. Type .5 (second circle radius). Press ENTER .	8. Accepts radius and prompts, " Select center definition... "

<p>9. The  template is highlighted. Press ENTER.</p>	<p>9. Activates template and prompts, "Enter X value:"</p>
<p>10. Type .5 (X circle center). Press ENTER.</p>	<p>10. Accepts value and prompts, "Enter Z value:"</p>
<p>11. Type -1 (Z circle center). Press ENTER.</p>	<p>11. Creates second circle.</p>
<p>12. Press Display (F5).</p>	<p>12. Pop-up activates.</p>
<p>13. Fit is highlighted. Press ENTER.</p>	<p>13. Fits geometry to display window.</p>

Step 2: Create 2" tangent circle. Fit geometry to inspect.	
Action	Description
<p>1. With  highlighted, press ENTER until the  template activates. Use ARROWS to highlight the template. Press ENTER.</p>	<p>1. Activates template and prompts, "Enter R value:"</p>
<p>2. Type 2 (radius of tangent circle). Press ENTER.</p>	<p>2. Accepts value and prompts, "Enter number of first element:"</p>
<p>3. Type 1 (circle #1). Press ENTER.</p>	<p>3. Accepts value and prompts, "Enter number of second element:"</p>
<p>4. Type 2 (circle #2). Press ENTER.</p>	<p>4. Displays all possible tangent circles and prompts, "Select 1-4:"</p>
<p>5. Type 4 (choice #4). Press ENTER.</p>	<p>5. Selects specified tangent circle and deletes other choices.</p>
<p>6. Press Display (F5).</p>	<p>6. Pop-up activates.</p>
<p>7. Fit is highlighted. Press ENTER.</p>	<p>7. Fits geometry to display window.</p>

Step 3: Create tangent lines. Fit geometry to inspect.	
Action	Description
<p>1. With  highlighted, press ENTER until the  template activates. Use ARROWS to highlight the template. Press ENTER.</p>	<p>1. Activates template and prompts, "Enter angle:"</p>
<p>2. Type -20 (absolute angle from 3 o'clock). Press ENTER.</p>	<p>2. Accepts angle value and prompts, "Enter number of circle:"</p>
<p>3. Type 2 (circle #2). Press ENTER.</p>	<p>3. Displays all possible tangent lines and prompts "Select 1-2:"</p>
<p>4. Type 2 (line #2). Press ENTER.</p>	<p>4. Selects line 2 and deletes other choices.</p>
<p>5. With  highlighted, press ENTER until the  template activates. Use ARROWS to highlight the template. Press ENTER.</p>	<p>5. Activates template and prompts, "Enter angle:"</p>
<p>6. Type 0 (absolute angle from 3 o'clock). Press ENTER.</p>	<p>6. Accepts angle value and prompts, "Enter number of circle:"</p>
<p>7. Type 1 (circle #1). Press ENTER.</p>	<p>7. Displays all possible tangent lines and prompts "Select 1-2:"</p>
<p>8. Type 2 (line #2). Press ENTER.</p>	<p>8. Selects line 2 and deletes other choices.</p>
<p>9. Press Display (F5).</p>	<p>9. Pop-up activates.</p>
<p>10. Fit is highlighted. Press ENTER.</p>	<p>10. Fits geometry to display window.</p>

Step 4: Locate shape starting point.	
Action	Description
<p>1. The  template is highlighted.</p> <p>Press ENTER until the  template activates. Use ARROWS to highlight the template and press ENTER.</p>	<p>1. Activates template and prompts, "Enter Z value:"</p>
<p>2. Type .1 (Z position). Press ENTER.</p>	<p>2. Creates line parallel to X-axis at entered Z position.</p>
<p>3. With  template highlighted,</p> <p>press ENTER until the  template activates. Use ARROWS to highlight the template and press ENTER.</p>	<p>3. Activates template and prompts, "Enter number of first element:"</p>
<p>4. Type 4 (line #4). Press ENTER.</p>	<p>4. Prompts, "Enter number of second element:"</p>
<p>5. Type 6 (line #6). Press ENTER.</p>	<p>5. Creates point (element #7) marked by an X.</p>

Step 5: Create ramp off geometry.	
Action	Description
<p>1. Select the  template. Press ENTER until the  template activates. Use ARROWS to highlight the template and press ENTER.</p>	<p>1. Activates template and prompts, "Enter Z value:"</p>
<p>2. Type -3 (Z position). Press ENTER.</p>	<p>2. Creates line parallel to X-axis at entered Z position.</p>
<p>3. With  template highlighted, press ENTER until the  template activates. Use ARROWS to highlight the template, and press ENTER.</p>	<p>3. Activates template and prompts, "Enter X value:"</p>
<p>4. Type 2.5. Press ENTER.</p>	<p>4. Creates line parallel to Z-axis at entered X position.</p>

Chaining Geometry Elements to Create a Shape

Refer to [Table 14-13, Exercise 2, Step 6](#). A new shape must be created before chaining can occur, the shape starting point (origin) should be a point on an element of the construction geometry. The new shape will be chained, from the starting point, around various geometry elements.



When the CHAIN  template is highlighted and **ENTER** is pressed, the Shape Editor prompts you to “**Select element:**”. Type a chain of intersecting geometry elements, each separated by a space and press **ENTER**. The editor will trace the new shape from intersection to intersection. Each intersection will become a node in the new shape.

The path around a circular geometry element can be either in a clockwise or counter clockwise direction.

If the intersecting geometry element is a circle, a positive element number creates a counterclockwise path, a negative element number creates a clockwise path.

NOTE: This refers to front tooling applications. All examples in the manual are front tooling. For rear tooling, enter a negative element number for counterclockwise and positive element number for clockwise path.

If the direction of the path around any element is incorrectly chained, the direction can always be reversed using the shape editing tools. Elements can be chained one at a time or all at once.

Table 14-13, Exercise 2, Step 6

Step 6: Chain segments of geometry to form shape.	
Action	Description
1. Press S-EDIT (F3) .	1. A pop-up activates.
2. Create is highlighted. Press ENTER .	2. Creates a new shape.
3. Use ARROWS to select the  template. Press ENTER .	3. Activates template and prompts, " From point: "
4. Type 7 (point #7). Press ENTER .	4. The pointer activates at point 7.
5. Use ARROWS to select the  template. Press ENTER .	5. Activates template and prompts, " Select element: "
6. Enter the elements to be chained into a single shape. Include direction for circles or arcs (- equals CW path; + or no sign equals CCW path): 4 space -2 space 3 space -1 space 5 space 8 space 9 ENTER .	6. Chains objects into a single shape.
7. Press Cancel (F9) .	8. End exercise 2.

Viewing a Listing of Geometry Elements

A complete listing of all the geometry elements and relevant position information is available.

To view the Geometry list:

1. From the default soft key line press **MISC (F6)**. A pop-up menu activates.
2. Move the highlight to select **Geometry List**, and press **ENTER**. A listing of geometry elements and position data activates.

The following information is displayed:

- Element number
- Points - X and Z positions
- Lines parallel to X-axis - Z-axis position
- Lines parallel to Z-axis - X-axis position
- Lines through a point - point position and angle
- Circles - The center point and radius

Deleting Geometry Elements

After a shape has been chained together from Construction Geometry, the geometry can be deleted, if desired. Elements can be left in the program along with the shape.

To delete geometry elements:

1. Press **DelGeom (F9)**. The Shape Editor prompts operator for the number of the geometry element to be deleted.
2. Type the element number, and press **ENTER**. The element is removed from the display.

Sometimes small bits of an element will remain on the screen after deletion. Press the "**R**" **REDRAW** hot key to refresh the screen.

NOTE: Deleted geometry cannot be undeleted.
--

Deleting All Geometry Elements

To delete all geometry elements:

1. With the **Shape** soft keys active, press **SHIFT** and **DelAll (F9)**. The Shape Editor prompts operator to confirm deletion.
2. Press **Yes (F1)** to delete or **No (F2)** to cancel.

Sometimes small bits of a deleted element will remain on the screen after deletion. Press the "**R**" **REDRAW** hot key to refresh the screen.

NOTE: Deleted geometry cannot be undeleted.

Activate/Deactivating Geometry Elements

Geometry elements can be deactivated to eliminate clutter in the viewing area. Turning elements off is not the same as deleting them. Geometry elements that have been turned off can be turned back on at any time. Geometry elements are deactivated and reactivated by group.

To deactivate/activate geometry elements:

1. From the default soft key line press **Setup (F9)**. A pop-up menu activates).
2. Move the highlight to select **Geometry**, and press **ENTER**. The second pop-up menu activates.
3. Move the highlight to select the category of elements to be activated/deactivated and switch the display status by pressing **ENTER**. Status is indicated by **On/Off** label adjacent to listing.

Geometry elements can be activated/deactivated in the following categories:

All	All geometry elements
Lines	Lines only
Circles	Circles only
Points	Points Only

Shape Editor Hot Keys

Several keypad keys have additional functions to adjust the display. These functions are available when the Shape Editor is in use. These hot key functions are available in the Draw View Mode. See the section on Draw for more details on working with the display hotkeys. Refer to [Table 14-14, Shape Editor Hot Keys](#).

Table 14-14, Shape Editor Hot Keys

Keypad Key	Function
"A"	Automatic Fit - Adjusts the scale of the image and centers the image so the image fills the entire display area.
"D"	Double Size - Adjusts the scale of the image to double its size.
"H"	Half size - Adjusts the scale of the image to reduce its size by one half.
"M"	Abs/Inc - Switches the Shape Editor between absolute and incremental mode.
"W"	Windows - Activates a zoom window to permit better viewing of any particular spot.
"R"	Redraw - Refreshes the image in the display area.
"U"	Inch/MM - Switches the Shape Editor between Inches and Millimeters mode.
"P"	Pan - Adjusts the position of the center of the image within the display area.
"S"	Scale - Adjusts the scale of the drawing by a factor.
"+"	Move pointer forward along shape (for keyboard programmers).
"-"	Move pointer backward along shape (for keyboard programmers).

Exiting the Shape Editor

To exit the Shape Editor and return to the Program Directory, press **Exit (F10)**.

Executing Shapes in a Program

Shape subprograms are not visible in the regular G-code program editor. They can be viewed only from the Shape Editor. Shape numbering always starts at 1 and is always consecutive.

To avoid confusion, G-code subprograms can be assigned 100 series numbers ex. 101, 102 103 etc.

If a G-code subprogram and a shape both have the same subprogram number, the G-code subprogram will be the one executed.

Shape subprograms are called from the main program by their shape number. The actual call is done exactly as it would be for a subprogram written in G-code. Thus, shapes can be used in G73, G74, M98, G65, and G66.

Refer to **Table 14-15**.

Table 14-15, G-code for Example #2

Block	Description
N1.G70 G90 G0 X0 Z0 T0	Activate Inch/Abs Modes. Rapid to Machine Home. Cancel offsets.
N2.T1	Activate T1 offsets.
N3.X1 Z.1 G95 F.01 S800 M3	Rapid to start position at 800 RPM. Activate spindle forward.
N4.M98 P1	Call Subprogram 1 (shape).
N5.G0 X0 Z0 T0 M5	Rapid to Machine Home. Deactivate tool and spindle.
N6.M2	End program.

Managing Shape Files

Naming Shape Files

The CNC automatically names shape files created in the Shape Editor. The Shape files contain the same primary name as the G-code file with which they are associated. The Shape Editor generates several files with the G-code file but different extensions. Refer to **Table 14-16** for a list of these file extensions.

Table 14-16, G-Code File Extensions

FILENAME.CAM	CAM mode settings
FILENAME.GEO	Construction Geometry
FILENAME.1	(or .2, .3) are Shapes
FILENAME.\$*	Temporary files

Viewing Shape Files in the Program Directory

The types of files generated by the Shape Editor are discussed in the previous section. To view program names of all formats in the Program Directory:

1. From the Program Directory, press the SHIFT key. The soft key menu changes.
2. Press **SHIFT + F9 (Display)** until all shape files are viewed.

Deleting Shape Files

CAUTION: Do not delete a shape file unless the associated shape is no longer needed to produce the part. If the main program calls a shape that has been deleted, the CNC generates an error message.

To delete .CAM and .GEO files:

1. From the Program Directory, highlight the filename and press **Delete (F3)**. The shape file is deleted.

To delete the .1, .2, .3 files:

1. From the Shape Editor, press **Shape (F2)**. The soft keys change.
2. Press **S-EDIT (F3)**. Pop-up activates.
3. Highlight **Delete**. Prompt activates, “**Sure you want to delete current shape?**”
4. Press **Yes (F1)**. Shape is deleted. Associated file will no longer activate in the Program Directory. Other shape files for the program are not affected.

To delete \$ files:

1. .\$ files will never remain in the user’s directory under normal circumstances. If found, they can be deleted from the Program Directory.
2. Highlight the program name and press **Delete (F3)**. The file is deleted.

Backing Up Shapes

Make sure to back up all necessary shape files when backing up the main G-code program.

Shape Editor Settings

The **Settings** menu allows you to configure various Shape Editor characteristics.

To access the **Settings** menu:

1. From the default soft key menu press **SETUP (F9)**. The pop-up menu activates.
2. **Settings** is highlighted. Press **ENTER** and the settings menu activates. Individual settings are made by highlighting a choice and toggling the setting with the **ENTER** key.

The various Shape Editor settings are as described in [Table 14-17, Shape Editor Settings](#). Shape Editor settings are preserved from session to session.

Table 14-17, Shape Editor Settings

Action	Description
Dimensions	Switches dimensioning mode and angles between absolute and incremental.
Input Units	Switches units mode between inches and Millimeters.
Arrows	Switches arrows that activate on the 1st move of any shape, to be turned On/Off. These arrows tell the programmer the direction of the shape (default is on).
Labels	Switches labels On/Off. The labels refer to the numbers of each shape, path or geometry element, such as 1, 2, 3, etc. (default is on).
Axes	Switches axes lines On/Off (default is on).
Grid	Allows selection of solid, dotted, or no grid lines.
Length	Allows selection of grid spacing.

Verifying Shapes

Run the program containing the shapes in Draw Mode. If problems arise, edit the shapes in the Shape Editor and re-run the program in Draw to verify corrections.

Section 15 - Communication

Introduction

The CNC is equipped to exchange data with other RS-232 compatible devices. The baud rate, parity, data bits, stop bits, and software parameters of the CNC and the device it is communicating with must agree for RS-232 communications to work.

Default communication parameters are entered in the device's setup utility when the CNC is first setup. These parameters can also be changed from the Communications screen. Parameter changes made in the Communications screen are not permanent. Every time the control is powered up, the CNC's default settings are reloaded.

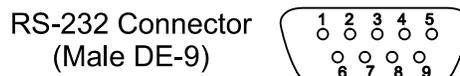
CAUTION: Only qualified personnel should attempt to alter settings in the CNC's setup utility. Incorrect settings may disable the device.

NOTE: The setup file is reloaded whenever setup changes are made and every time the CNC is powered on.

Installing the RS-232 Cable

RS-232 communication requires a cable connection between the sending and receiving devices. The machine builder determines the location of the DE-9 connector. Typically, a DE-9 connector is used on the machine. The other end of the cable must be connected to the other device.

Refer to **Figure 15-1**. It is important to use a cable designed for RS-232 communication. Cables designed for RS-232 communications have the wires between pins 2 and 3 internally crossed.



Pin 2 - Data Receive Pin
Pin 3 - Data Transmit Pin

DE-9

Figure 15-1, RS-232 Communication Connector

Data sent from pin 3 (transmit) of one device must go to pin 2 (receive) of the other. This is the only way RS-232 can work.

Computer cables designed as extension cords for computer peripherals cannot be used for RS-232 communications (pins 2 & 3 are not crossed).

Accessing the Communication Utilities

To access the Communication screen:

1. With the CNC in the Manual Mode press **Program (F2)**. The Program Directory activates.
2. Press **Utility (F9)**. A pop-up displays.
3. Move highlight to select **Communications**, and press **ENTER**. The communication screen is displayed. Refer to **Figure 15-2**.

NOTE: The program marked by the highlight when the Communications screen is activated is the default selection for sending.

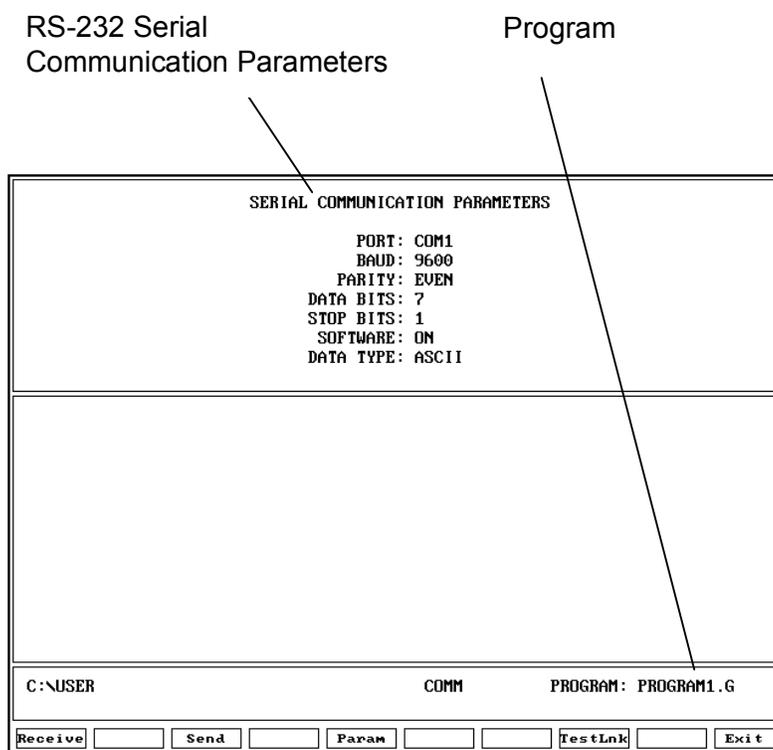


Figure 15-2, RS-232 Communication Screen

PROGRAM

The program selected for transmission.

SERIAL COMMUNICATIONS PARAMETERS

Settings required for the two devices to send and receive programs.

Setting Communication Parameters

This manual does not attempt to discuss the merits of the different parameter choices. Refer to an appropriate computer communications reference for this information.

NOTE: Both the sending and receiving devices must have the same baud, parity, data bits, stop bits, and software parameter settings.

Refer to **Figure 15-3**.

Parameter Settings	
1. ▶Port.....	COM1
2. Baud rate..	9600
3. Parity.....	EVEN
4. Data bits..	7
5. Stop bits..	1
6. Software...	On
7. Data Type..	ASCII

Figure 15-3, Parameter Settings Pop-Up

To change communication parameters:

1. From the Communications screen, press **Param (F5)**. Parameter Settings pop-up is displayed.
2. Highlight the setting to be changed, and press **ENTER**.
3. When you select the **Port**, **Baud**, or **Parity** parameter, the CNC activates an additional pop-up. In the pop-up, select the appropriate setting, and press **ENTER** to store the setting.
4. For the **Data Bits**, **Stop Bits**, **Software**, or **Data Type** parameter, press **ENTER** to cycle through the available choices.

Communication Port

Most CNC installations have at least one RS-232 connector. It is usually labeled and mounted on the back of the console. The CNC sends signals via its **COM1** port to the RS-232 connector. This configuration cannot be changed. Personal computers that run the offline software can use **COM2** if they are equipped with the necessary hardware.

Baud

The CNC supports the following baud rates: 110, 150, 300, 600, 1200, 2400, 4800, 9600, or 19,200. [Default: **9600**]

Parity

The CNC supports the following parity settings: Odd, Even, or None.
[Default: **Even**]

Data Bits

The CNC supports the following data bit settings: 7-bit or 8-bit.
[Default: **7-bit**]

Stop Bits

The CNC supports the following stop bit settings: 0-bit or 1-bit.
[Default: **1-bit**]

Settings

The CNC supports the following protocol settings: ON or OFF.
[Default: **ON**] Software protocol is frequently referred to as “Xon” or “Xoff”, or “handshaking” in commercial communications packages.

Data Type

The CNC supports the following data display types: ASCII or BINARY.
[Default: **ASCII**] This setting does not affect the data exchanged, only how it appears on the screen during transmission.

Testing the Data Link

The CNC has a utility to test the data link. With the CNC connected to another machine, set the parameters on both machines. If the other machine has a manual test screen, activate it.

Activating the Test Link Screen

Refer to **Figure 15-4**. With the Communications screen active, press **TestLnk (F8)**. The Test Link screen displays.

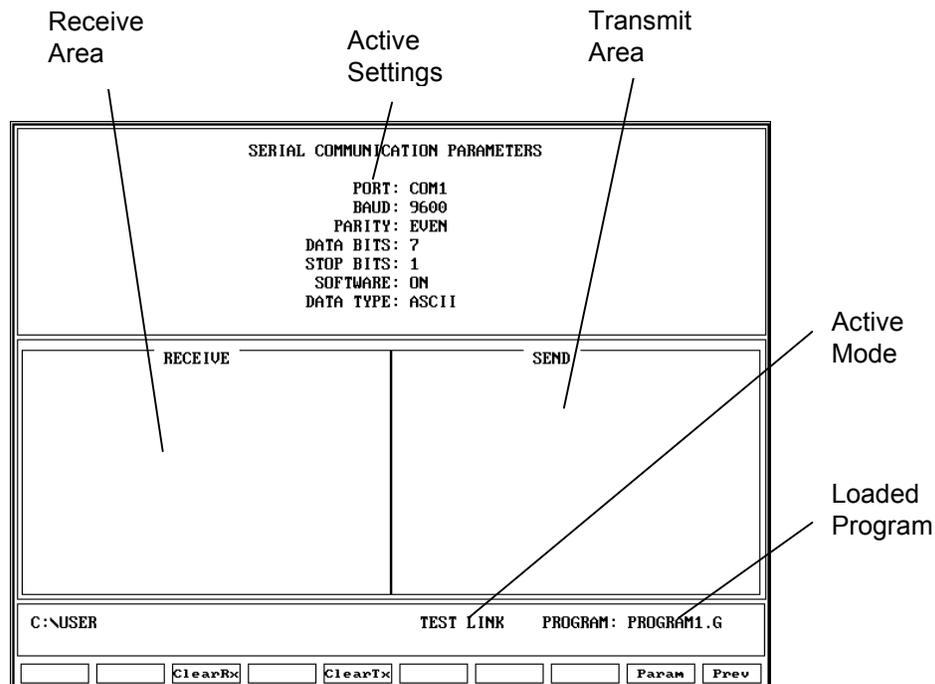


Figure 15-4, Test Link Screen

Setting Test Link Display Modes

The operator tests the link by visually verifying that the test data sent matches the test data received. The data type setting determines how characters appear on the screen.

If the data type is set to ASCII, letters and numbers are displayed. If the data type is set to binary, the hexadecimal equivalent is displayed. Set both devices to use the same data display type.

NOTE: Hexadecimal characters appear as pairs of numbers or numbers and letters.

To change the Test Link screen data display:

1. Press **PARAM (F9)**, and Down Arrow to **#7- Data Type**. Press **ENTER** to toggle. The current mode is displayed in the settings area of the Test Link screen.

Testing the Link

To test the link:

1. Set up an RS-232 connection with another device.
2. Set the other device to receive.
3. With the Link Test screen active use the CNC's keypad to type in any series of numbers. These characters will appear in the transmit area.
4. Verify that the characters are correctly received at the other device.
5. Set the other device to transmit.
6. Manually transmit a series of characters from the other device.
7. Verify the characters sent to the CNC correctly appear in the receive area.

Clearing the Receive Area

Clear the receive area by pressing the **ClearRx (F3)** key.

Clearing the Transmit Area

Clear the transmit area by pressing the **ClearTx (F5)** key.

Sending a Program

The name of the program marked by the highlight when the Communications screen was activated is listed next to the PROGRAM label on the screen. The CNC will send this program unless another program is selected. The CNC will prompt the operator to select another program during the sending process.

1. With the Communications screen active, the parameters set and the link tested, press **Send (F3)**. The CNC prompts the operator to send the selected program or to change the program being sent.
2. Press **Select (F1)**. The CNC sends the program.

– or –

Press **Change (F3)**. The CNC prompts the operator to type in the name of the desired program.

Receiving a Program

The operator must key in a name for the received program before receiving it from the other device.

To receive a program:

1. With the Communications screen active, the parameters set and the link tested, press **Receive (F1)**. The CNC prompts operator to key in a name for the incoming program.
2. Key in the desired name, and press **ENTER**. The CNC displays a "**READY TO RECEIVE . . .**" message.
3. Start transmission from sending device.

Setting the Transmission and Receiving Display

If the CNC is transmitting or receiving with the Text Mode active, the exchanged program will be displayed on the screen. If the Text Mode is off, the display area will remain blank.

The **Text (F4)** soft key label is highlighted when the Text Mode is active.

The “Transfer In Progress” symbol (just above the soft key line) cycles when data is exchanged.

Activate or deactivate text mode by pressing **Text (F4)** while transmitting or receiving.

NOTE: In general, transmission will occur faster with Text mode Off.
--

Holding Transmission/Receiving Operations

When sending or receiving programs the operation can be paused by pressing the **Hold (F1)** key. Press **Resume (F2)** to continue the exchange.

NOTE: If the CNC is receiving a program and either device’s software parameter (Xon/Xoff) is set to OFF , there is a possibility a hold will overload the CNC’s buffer causing portions of the program to be lost. Operating with the software parameter (Xon/Xoff) ON is recommended.

Using Data Control (DC) Codes

Data Control (DC) codes are sometimes required to automate the operation of a paper tape reader or punch. Refer to **Table 15-1** for the available DC Codes.

Table 15-1, DC Codes

ASCII	Function	Hex Code	CNC Key
DC1	Reader Start	0x11	Q
DC2	Punch Start	0x12	R
DC3	Reader Stop	0x13	S
DC4	Punch Stop	0x15	T

The “ASCII” column lists the codes required to perform the corresponding “Function.” The “Hex Code” is the hexadecimal equivalent of the ASCII code. The “CNC Key” column lists the key you press on the CNC keypad or a PC keyboard to transmit the required DC code.

A reader or punch will turn ON (start) or OFF (stop) in response to these codes. To test reader or punch communication, activate the Communications screen and press any of these keys. The paper tape reader or punch should respond appropriately.

Using DC Codes in Receive Mode

Usually, a receive operation involves the paper tape reader. You must start the reader, thereby initiating the reading of the paper tape. Do this after the CNC has been set up to receive the file.

1. Set up to the CNC to receive a program.
2. Press 1 to send the DC1 code to the reader. The reader starts.
3. After the CNC has read the tape, press 3 to stop the reader, if necessary. The CNC sends the DC3 code to the reader and the reader stops.

Using DC Codes in Send Mode

Usually, a send operation involves the paper tape punch. Set up and start the punch prior to initiating the send operation.

1. Select the program that will be sent to the tape punch.
2. Press 2 to send the DC2 code to the punch. The punch starts.
3. Press **Send (F3)** to start sending the program. If necessary, change the name of the program.
4. When the program has been completely transferred to the punched tape, press 4 to stop the punch, if necessary. The CNC sends the DC4 code to the punch and the punch stops.

Section 16 - Advanced Programming Features

Block Separator

Format G90 G01 G95 X0 Z0 F.003 ; X1 Z-1 ; Z-2 ; X0 ; Z0
Description CNC feeds to X0, Z0 in Abs/Inch mode with Feed Per Revolution active. It feeds to four separate coordinates in subsequent moves.

Block separators (;) may be used to place several functions on a single line of program. Use a semi-colon (;) to designate a block separation. This can be particularly useful while inputting M.D.I moves in the Manual Mode. With block separators, several moves can be made from one input to the command line.

NOTE: The number of separated steps in a program file is only limited by the memory available.

Modifiers

Format G02 X1.5 Z-1.5 I&A1.5 K&A-1.0
Description The A modifier is used to make the I and K center points of an arc abs. (Arc centerpoints are incremental by default.)

Modifiers are addresses that will allow a CNC user to modify the way a word address is interpreted by the CNC. For example, one value in an Inch program may be changed to MM without programming G71. An arc center I or K may also be changed to be an Absolute value using address modifiers. Refer to **Table 16-1** for a list of possible modifiers.

Table 16-1, Address Modifiers

Modifier	Description
A	Force the address word to be ABSOLUTE.
D	Force the address word to be INCREMENTAL.
E	Force the address word to be INCH.
M	Force the address word to be MILLIMETER.

The address and modifier must be accompanied by the ampersand (&) character. The ampersand character must be placed between the address to be modified and the modifier. The address is programmed first, followed by the (&), followed by modifier, followed by the value.

The modifier is non-modal, and is only used for the address it accompanies.

Expressions and Functions

Expression commands can be used within a program to do complex mathematical calculations. Parentheses should enclose each and every expression. The order of hierarchy of commands follows the standard of mathematics. Refer to **Table 16-2** for a list of operators and functions.

Table 16-2, Operators and Functions

Operator	Function	Format	Description
()	expression function (parentheses)	G01 X(#100 + #101)	All calculations must be enclosed in parentheses. This defines an expression function.
*	Multiplication	G00 Z(#102 *#103)	Moves Z to product of #102 and #103
/	division	Loop (5/2/0.1)	Loops 25 times
%	modulo gives the remainder of a function	G01 X(8 % 3)	Feeds X2. The number 2 comes from the remainder of 3 into 8
+ , -	addition, subtraction	G01 X(3+2) G01 X(3-2)	Addition and subtraction.
>	relational, greater than	(#101 > #103)	true if #101 is greater than #103
<	relational, less than	(#101 < #103)	true if #101 is less than #103
=	relational, equal to	(#100 = #101)	True if #100 is equal to #101
!=	relational, not equal	(#100 != #101)	True if #100 does not equal #101
round	round up or down, automatically	#101=1.5; G01 X(round(#101))	Moves X2
		#102=1.498; G01 X(round(#102))	Will round fractions greater than .500000 up and will round fractions less than .500000 down. Moves X1.
fix	discard fraction less than 1	#101=(5/2); G01 X(fix(#101))	Moves X2. Fix will round (n) down to the next integer.
fup	raise fraction up 1	#101=(5/2); G01 X(fup(#101))	Moves X3. Fup will round (n) up to the next integer.
var	true if variable defined, false otherwise	Var(#100)	True if #100 has been defined false if not. Var (n) is used to check if a user variable has been defined in a program.
sin	sine trigonometric function	G01 X(cos(15)) Z(sin(15))	Moves along the hypotenuse of a 15 degree angle with a hypotenuse of 1. Sin (n) will give the sine of (n).
cos	cosine trigonometric function	Cos (n)	Cos (n) will give the cosine of (n).

(Continued...)

Table 16-2, Operators and Functions (Continued)

Operator	Function	Format	Description
tan	tangent trigonometric function	Tan (n)	Tan (n) will give the tangent of (n).
asin	arc sine	asin(n)	Will give the inverse of sine or 1/sin
acos	arc cosine	acos(n)	Will give the inverse of cosine or 1/cos.
atan	arc tangent	atan(n)	Will give the inverse of tangent or 1/tan.
abs	absolute value	abs(-n)= n.	Will give the absolute value of (n).
sqrt	square root	Sqrt(9) = 3	Gives the square root of (n).
ln	natural logarithm	Ln (n)	Gives the natural logarithm of (n)
log	logarithm base 10	Log (n)	Gives the logarithm of (n).
exp	exponential	EXP (n)	Gives the exponent of (n).
trun	truncate	TRUN (n)	Truncates the value of (n).
!	unary logical not	(! = (#100))	Not equal to value in #100.
+	unary logical positive	(+ (#101))	Means positive whatever value is in #101.
-	unary logical negative	(- (#100))	Means negative whatever value is in #100
#	indirection	##102	One level of indirection

System Variables

System variables are variables that are set aside for the CNC system to use. Some of these variables may be useful for the user to know when programming macros. System variables range from #1000 to #1099. Refer to **Table 16-3**.

Most of these variables are read only and cannot be changed (written to). There are a few exceptions to this rule (block skip variables).

Table 16-3, System Variables

Variable Address	Description
#1000 to #1009	Block skip variables (Read/write allowed).
#1010 and #1012	Commanded ABS tool position #1010 = X #1012 = Z
#1016	Current G motion code: 0=rapid, 1=feed, 2=cw arc, 3=ccw arc, 5=ellipse
#1019	Current X tool length offset.
#1020	Current tool radius times 2. (diameter)

(Continued...)

Table 16-3, System Variables (Continued)

Variable Address	Description
+	Current Z tool length offset.
#1022	Current feedrate.
#1023	Current rapid rate.
#1024	Current RPM.
#1030	Stock variable
#1031	Acute angle for rounding compensated intersections (default = 15.0 degrees).
#1032	Number of look-ahead blocks for tool nose compensation.
#1040	Real time tool compensation (for ellipse, 0=off, 1=outside, 2 inside).
#1041	Current program tool compensation (40=off, 41=left, 42=right).
#1050 and #1052	Actual ABS position #1050 = X #1052 = Z
#1070	Current XZ dimension (70=inch, 71=mm).
#1090	Current XZ dimension (90=ABS, 91 =incremental)

User Variables

User variables are variables that are set aside for the programmer to use. They consist of local variables, common variables, read only variables, and block skip variables. These variables can be written to, or read from.

Local Variables

Local variables range from #1 to #99. These variables can only be used inside the body of a subprogram (or macro or parametric program). Errors will occur if they are used in the main body of the program. Their values will not hold from one subprogram to another. This prevents the operator from assigning conflicting values to the variables.

Common (Global) Variables

Common variables range from #100 to #220 and are read and write. These variables can be used anywhere in the program or subprogram and their value will remain.

Read Only Variables

These variables range from #221 to #250 and are read only in macros. Read only variables must be assigned in the main program. Once set, the contents can be used in subprograms or macros as read only variables.

Block Skip Variables

Refer to **Table 16-4**. If the machine is provided with a block skip switch, all blocks containing the slash character will be skipped or not skipped, depending on the switch position. A variable does not need to be set for this to occur.

Selective block skips (/ (n)) are not affected by the switch. They are always turned off or on by setting their variable (0 = off; 1 = on).

If no block skip switch is available, M9345 X1 must be programmed as the first block of any program that will use skip functions and variable #1000 = 1 must be programmed to skip regular slash blocks.

Table 16-4, Block Skip

Block	Description
M9345 X 1	Enables block skip
N11 #1000 = 1	Sets block skip to On
N12 G01 X1.125 G95 F.003	
N13 X1.125 Z-.125	
N14 Z-2.0	
/N15 X1.1375 Z-2.125	This line is skipped by the CNC
N16 Z-4.0	
N17 G00 X4.0	

The slash (/) can appear anywhere in the block. It is recommended that the operator put the slash (/) before the block number.

Setting Selective Block Skips

Format #100n, where n is a number 1 - 9
 The programmer can use up to 9 optional block skips. Refer to **Table 16-5**.

Table 16-5, Selective Block Skip

Program	Description
M9345 X 1	Enables block skip
N11 #1002 = 1	Sets block skip variable
N12 G01 X1.125 G95	
N13 X1.125 Z-.125	
N14 Z-2.0	
/2 N15 X1.1375 Z-2.125	This line is skipped by the CNC. If #1002 = 0, block N15 will be executed.
N16 Z-4.0	
N17 G00 X4.0	

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 simple component of any size will always have a stock diameter from which it was machined, a length of the turned diameter and a depth or difference of that diameter. It follows then, that many turned diameters of different sizes can be cut using a similar tool-path with longer or shorter moves for the tool-path. If the constraints of the diameter are processed by a suitable program, a tool-path can be calculated by the control to cut the particular diameter. Such a program may be called a macro or parametric program.

The constraints of the diameter, or the features required, are its parameters. Because particular parameters for any feature will vary as dimensions are changed, the parameters are often called variables.

Contents of Variables

It is often useful to know the contents of a variable when programming to assist in testing. Using the PRINT command, the contents of a variable can be displayed on the screen in Manual, Single Step, Automatic and Graphic modes. The format is PRINT number, number. Refer to **Table 16-6**.

Table 16-6, Print Display

Format	Description
PRINT 200	Displays the contents of the variable (#200)
N180 PRINT 110	Display contents of the variable(#110) as part of program execution

If the numbers are separated by commas, several variables can be printed simultaneously, but too many will cause numbers to print off the screen. Print will not wrap output.

In Manual mode, type **PRINT 110**, then **START** to display the contents on the SCREEN.

Instead of typing the whole word PRINT, a short form is available. **JP** is interpreted as print.

Setting and Transferring Variables

Setting Variables and Direct Transfer

Variables are loaded or set when they appear on the left side of an equation. Refer to **Table 16-7** to see how variables are used in a program.

Table 16-7, Setting Variables and Direct Transfer

Format	Description
N200 #100 = 5.56	#100 contains the number 5.560000 until changed.
N200 #100 = 25.4m	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).
N200 #100 = #20	#100 contains the number held by #20 until changed. Equating one variable with another is called a direct transfer.

NOTE: When a variable is used on the right side of the equal sign, the variable must already be assigned a value. Otherwise, the CNC displays an error message.

Indirect Transfer

Variables can be indirectly assigned to a depth of four levels by introducing extra pound (#) symbols before the variable number. Indirection means that a value can be assigned to a variable through another variable or variables. Refer to **Table 16-8** for an example of single indirection.

Table 16-8, Indirection

Command Line	Description
N201 G90 G71 G0	
N202 #101 = 51.456	sets variable (#101) to 51.456
N203 #102 = 101	sets variable (#102) to 101
N204 X(##102)	the X axis will move to 51.456

The contents of variable #101 are used by variable #102. The actual content of variable #102 is constant 101. The indirection is used at block N204 by the addition of the (#) symbol to variable #102.

In **Table 16-9, Second Level Indirection**, indirection is taken to two levels. The example shows how the contents from several variables can be assigned to a command or expression.

Table 16-9, Second Level Indirection

Command Line	Description
N210 G90 G71 G0	
N211 #101 = 1	Sets variable (#101) to 1
N212 #102 = 2	Sets variable (#102) to 2
N213 #103 = 3	Sets variable (#103) to 3
N214 #104 = 4	Sets variable (#104) to 4
N215 #119 = 100	Variable #119 is set to constant 100
N216 LOOP 4	
N217 #119 = #119 + 1	One is added to the contents of variable #119
N218 #120 = 119	Variable #120 is set to constant 119
N219 X(###120)	Moves the X axis to the position contained in variable #120 through two levels of indirection
N220 END	
N221 M2	

The first level is the content of variable #119, and the second level is the content of variable #100 which is incremented in the loop at block N217 to introduce the contents of variables #101, #102, #103 and #104.

Storing the Results of Computation

Variables on the left side of an equation store the result of computation when a mathematical expression is programmed. Refer to **Table 16-10**.

Table 16-10, Computation Results

Command Line	Description
N250 #110 = #20 + #35	#110 contains the sum of the contents of #20 and #35.
N260 #120 = #18 / 2	#120 contains the result of the contents of #18 divided by 2.
N300 #140 = (#11 * #115) / 2	Divides the product of (#11) and (#115) by 2
N310 #141 = SIN (45)	Sets (#141) to the sine of 45 degrees
N320 #142 = (#141 * #140)	Sets (#142) to the product of (#141) times (#140)

Parentheses are not required for calculation except where priority of calculation is required, special functions are used, or a multiplication is used (to avoid ambiguity with the comments symbol, *).

NOTE: Spaces are included in the examples for clarity and may be omitted.

Parametric Programs

Parametric programs are a type of subprogram that can be called from within a main program. Parametric programming can be compared with computer programming languages such as Basic. The strength of parametric programming is that a generalized program can be written that will accept variable values so that a whole family of parts can be created using a parametric program instead of programming each individual part separately.

Applications of Parametric Programming

Families of Parts

Parametric programming can be a powerful tool in cutting down programming time when manufacturing similar parts or part families. Any manufacturing application that incorporates a large amount of repetition with slight variations between repetitions would be more readily accomplished with a parametric program than with traditional subprograms.

Complex Geometric Shapes

Parametric programming can be used to create extremely complex geometric shapes because parametric programming allows arithmetic calculations to be done within the parametric program. Any shape that can be mathematically defined can be created by a parametric program.

Parametric Programming Versus Subprogramming

One way to help you understand parametric programming is to compare it to subprogramming. Subprograms must be totally redundant, that is to say that if anything changes from one job to the next, subprogramming techniques cannot be used. For example, if a subprogram is written to machine a 0.25 inch-wide by 0.25 inch-deep groove for a turning center, the subprogram will only machine a groove with these specifications. If a 0.187 in-wide by 0.125 in-deep groove must be machined this subprogram cannot be used. Of course, another similar subprogram could be written, but it would also only machine one groove size.

The same can be said of any subprogram, subprograms can only be used to make one specific part.

The flexibility of parametric programming is the ability to generate general-purpose subprograms. Once the parametric program is developed, any number of similar parts or variations could be created with little or no extra programming time. This is because parametric programs allow variables to be passed from the main program to the parametric program.

Call Statements

Call statements are commands that call the parametric program or macro from within the body of the main program. Refer to **Table 16-11**.

Table 16-11, Call Statements

Command Line	Description
N05 G00 X3. Z3. T1 X2.5 Z3.5	
N10 G65 P100 X3. Z4. W.25	The call statement that calls program 100 and passes the variables X, Z, and W to the parametric program.

User Macro (G65, G66, G67)

Refer to **Table 16-12**. **G66** is modal and **G67** cancels a macro.

Table 16-12, User Macro Calls

G-code	Description
G65 Pn	Calls macro n . The cycle will execute at the current position. It is non-modal.
G66 Pn	Calls macro n . The cycle will execute at any X and/or Z location given after the G66 code, until G67 (cancel) is called. G66 is modal, and will stay in effect until G67 is called.
G67	Cancels user macro modal call.

Programming Macros

The programmer identifies the macro in the first block of the macro with the address word O, followed by a number.

Format N200 O100

The macro is terminated with the M99 code in the last line of the macro.

As previously described, local variables can be used only inside the body of a macro or subprogram, and will not transfer data to other macros or subprograms. If further subprogram calls are made from the macro body, it is necessary to transfer data from the local variables to common variables in order to make the data available to other subprograms.

Format N220 #50 = #20

Common variables range from #100 to #220. Because the CNC's canned cycles use local variables in the range #1 up to and including #40, it is recommended that user macros be restricted to variables #50 up to and including #199 to eliminate the risk of a user defined variable being overwritten during the use of a CNC canned cycle.

The macro (subprogram) must either be part of the program it is executed from, or be included using the file inclusion code.

Setting and Passing Parameters

When parameters are passed to a macro body by letter address, the contents of the parameters are stored in local variables according to the following table. Refer to **Table 16-13**.

Table 16-13, Local Variables

A = #1	I = #4	T = #20
B = #2	J = #5	U = #21
C = #3	K = #6	V = #22
D = #7	M = #13	W = #23
E = #8	Q = #17	X = #24
F = #9	R = #18	Y = #25
H = #11	S = #19	Z = #26

Refer to **Table 16-14**. Parameters for a macro may be set before the subprogram call (M98 Pn) is programmed. However, it may be more convenient to use macro call G65 Pn or G66 Pn (modal macro call), as the parameters may be passed to the subprogram by letter address in a similar fashion to a canned cycle call.

Table 16-14, Passing Parameters

Format	Description
N10 #151 = 2	N10 through N12 set variables to constant values.
N11 #152 = 3	
N12 #153 = 3.4	
N13 M98 P1	subprogram call for program 1

Refer to **Table 16-15**. Macro call G65 Pn can be repeated through use of the loop command (format Ln, n is the number of loops).

Table 16-15, G65 Parameter Passing

Format	Description
N20 G65 P100 A2. B3. C3.4 D.25	Macro call for program 100. Passes the values of 2, 3, 3.4, and .25 to #1, #2, #3 and #7, respectively.
N20 G65 P100 A2. B3. C3.4 D.25 L3	Macro call for program 100. Macro call is instructed to repeat the call 3 times.

NOTE: Letter addresses G, L, N, O, and P **cannot** be used for parameter passing.

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 and #1062 for X and Z).

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

Using the **G31** primitive, parametric programming and the M-code described above, it is possible to write additional cycles to perform custom probing functions.

Conditional Statements

Conditional statements **IF**, **THEN**, **ELSE**, **GOTO**, and **WHILE** are available for the programmer to use. Refer to **Table 16-16**.

Table 16-16, Conditional Statements

Statement	Format	Description
IF - THEN - ENDIF	N300 IF (expression) THEN N310 (body) N360 ENDIF	If the expression is true, program execution continues on through block N300 to N310. If the expression is false, blocks N310 to block N360 are skipped.
IF - THEN - ELSE - ENDIF	N400 IF (expression) THEN N410 (body) N440 ELSE N450 (body) N470 ENDIF N480	If the expression is true, program execution continues from block N400 to N410 - N440 where a jump is made to N480. If the expression is false, blocks N410 to N440 are skipped. Blocks N450 through N470 are executed. In place of an expression, a non-zero variable can be used. The variable will be treated as a true expression
IF - GOTO	N500 IF (expression) GOTO Nnnn N510	If the expression is true, program execution jumps to the block number specified by Nnnn. If the expression is false, program execution continues on to block N510. A variable can be substituted for expression.
WHILE - DO -END	N550 WHILE (expression) DO nnnn N560 ----- N590 END nnnn N600 -----	If the expression is true, the program loops between N550 and N590 until the expression becomes false. Similarly, if the expression is false when block N550 is executed, a jump will be made to block N600.

(Continued...)

Table 16-16, Conditional Statements (Continued)

Statement	Format	Description
DO – END	N620 DO nnnn N630 ----- N650 IF (expression) GOTO 1111 N660 ----- N670 END nnnn	N620 sets the program into an infinite loop. N650 ends loop when the expression is true. N670 ends DO command programmed in N620.

DO - END sets the program into an infinite loop that can only be ended by programming GOTO (1111) another block number. DO and END must be paired with labels (nnnn). When executed, the program will repeat blocks N630 through to N660 until the expression at N650 becomes true. Then, the program execution resumes at block 1111.

Unconditional: The LOOP repeat

The conditional statements above require that a test is either true or false for a particular course of action to be taken. Unconditional statements are acted on without a logical condition being met. Refer to **Table 16-17**.

Table 16-17, Unconditional Statements

Statement	Format	Description
LOOP - END	N680 LOOP nnnn N685 ----- N695 END	LOOP instructs the control to execute the blocks following block N680 until End, the number of times specified (nnnn). The number of loops can be a variable assignment (LOOP #121).
GOTO	N698 GOTO nnnn	Continue program at line nnnn. GOTO is an instruction to continue program execution at the block specified (nnnn).

Abbreviating Statements

The **Expand key** feature allows the operator to use statements without typing-in the entire statement. The operator keys in an abbreviation assigned to a statement and activates the **Expand key** feature.

Table 16-18 lists statements to which the **Expand key** feature can be applied and the abbreviations assigned to each statement. In the table, the statements are displayed inside brackets. These brackets do not appear on the screen.

Table 16-18, Abbreviating Statements

Abbreviation	Statement
D	[DO [END]
E	[END] or [ENDIF] or [ELSE]
G	[GOTO]
I	[IF () THEN] [ENDIF]
L	[LOOP] [END]
P	[PRINT]
W	[WHILE () DO] [END]

For conditional statements, the CNC displays the statement and waits for the operator to key in the condition under which the statement will be activated. Not all statements require the operator to enter a condition.

To generate a statement from an abbreviation:

1. In Edit Mode, type the abbreviation corresponding to the conditional statement.
2. Press **Editing (F8)**. The Editing pop-up menu activates.
3. Use the **ARROWS** to highlight the **Expand key** feature, and press **ENTER**. The CNC generates the statement on the screen.
4. If the insertion point appears inside parentheses (), the operator must enter a condition under which the statement will be activated. Type in the condition, if applicable.

Logical Terms

All required logical operations can be carried out using the following command characters or combinations of characters. Refer to **Table 16-19**.

Table 16-19, Truth Table

Statement	Symbol	True/False Table	Description
OR		0-0 = False 0-1 = True 1-0 = True 1-1 = True	0 represents a false statement. 1 represents a true statement. 0-1 would mean a false first statement and a true second statement
EXCLUSIVE OR	^	0-0 = False 0-1 = True 1-0 = True 1-1 = False	The truth table shows the truth value of the overall statement containing two statements.
AND	&	0-0 = False 0-1 = False 1-0 = False 1-1 = True	Ex: (The sky is blue) AND (the earth is flat) would be a false statement. (the sky is blue) = 1 (the earth is flat) = 0 1-0=false

Comparative Terms

Comparisons can be made between variables, and between variables and constants. Refer to **Table 16-20**.

Table 16-20, Comparative Statements

Statement	Format	Description
Equality	N700 IF (#120 = #125) THEN (or GOTO)	Compares the contents of variable #120 with the contents of variable #125.
	N710 -----	If the contents are equal, then the expression is true and the program is directed by THEN or GOTO.
	N740 IF (#130 = 360) THEN (or GOTO)	Variable #130 is compared with the constant 360.
	N750 -----	The consequence of the comparison is identical to the first case.
Inequality NOT	N760 WHILE (#135 != #137) DO 10 N790 END 10	Continue at 790 while #136 is not equal to #137. The symbol (!=) means not equal to. Block N760 instructs the program to continue at block N790 while the contents of #135 and #137 are not equal.
GREATER THAN	N800 IF (#122 > #134) GOTO 820	Continue at line 820 if #122 is greater than #134. The symbol (>) represents greater than . The program is instructed to skip to block 820 if the contents of #122 are greater than #134. If the expression is false program, execution continues at 810.
LESS THAN	N850 IF (#123 < #135) GOTO 880	Continue at line 880 if #123 is less than #135. The symbol (<) represents less than . The program is instructed to skip to block 880 if the content of #123 is less than the content of #135. Program execution will continue at block 860 if the expression is false.

NOTE: Both **greater than** and **less than** expressions become false if the variable contents are equal.

File Inclusion

File inclusion is a function that allows a second program to be called or included in the program that is being run. If the function is encountered in a main program, the entire included program will be processed. The program will then continue running the original program. The function may also be placed after the end of the main program. It will then enable the use of an included subroutine.

Format ["FILENAME.G

- ❑ Open square bracket, double quote, filename and extension are required.

Inclusion File Requirements

- ❑ The program must exist in the current program directory.
- ❑ The contents will vary depending on the intended use.

Inclusion File Formats

- ❑ Typically, program lines that are to be included whenever the function is encountered, will not have an ending code, e.g. M2, M30 or M99. This would be useful when several program lines that never change are used several times in many programs. (see Toolchg.g)
- ❑ Used as a subroutine that is accessible to all programs, the program would start with a program number (Oxxx) and end with M99. The Include function statement must then be after the end of the main program. (after M2 or M30) The included subroutine then can be used by the main program as any other subroutine, using M98 Pxxx, G65 Pxxx, etc. (see Mysub.g)

Example Program

```

N01 ....
N02 ....
N03 ["TOOLCHG.G   Executes blocks in program Toolchg.g
N04 ...
N05 M98 P200      Executes included subroutine Mysub.g
N06 ...
N07 ["TOOLCHG.G   Executes blocks in program Toolchg.g
N08 ...
N09 M2            End of program
N10 ["MYSUB.G     Includes subroutine Mysub.g

```

Toolchg.g

```

N01 .....
N02 .....

```

N03	No end of program used
Mysub.g	
N01 O200	Subroutine format
N02	
N03	
N04	
N05 M99	End of subroutine

Sample Parametric Program

Refer to **Figure 16-1**. The diagram represents the part that will be programmed. Refer to **Table 16-21**. The program uses parametric programming features to create a G81 longitudinal grooving cycle.

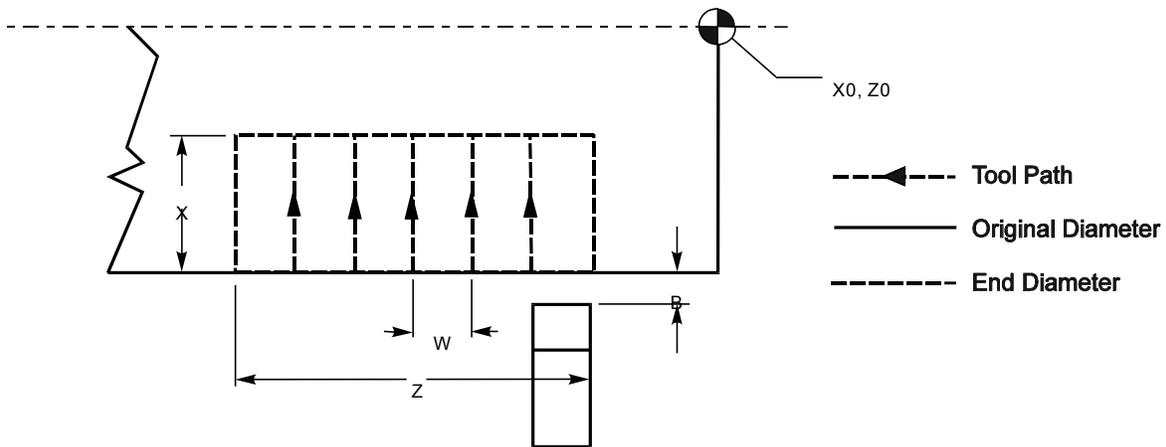


Figure 16-1, G81 Longitudinal Grooving Cycle

Table 16-21, Parametric Programming Example

Program Body	Comments
N005 G90 G70 G0 T0 X0 Z0	
N010 G94 F20.0	
N030 T1	
N050 X2.1 Z-1.	
N055 G65 P100 Z-.500 X-.500 W.200	*Calls Macro 100
N060 T0 G0 X0 Z0	*CNC returns here when the macro is completely executed.
N100 M2	*Ends main program
N105 O100 *Begin Macro for Canned Cycle	*The O address word identifies the program as a macro (or subprogram).
*PARAMETERS PASSED *X #24 = depth of groove *Z #26 = width of groove	*External groove only. Z positioned at start of groove. *Groove size negative, Z moves toward spindle.

Program Body	Comments
*W #23 = width of tool	*Groove size positive, Z moves away from spindle.
N125 #51 = .100	*Clearance distance for X.
N130 #23 = ABS(#23)	*Tool width is absolute.
N135 #60 = 0	*Set variable for test. (0 for fail, 1 for pass).
N140 IF (#26 > 0) THEN	*If groove direction (and size) is positive.
N145 #50 = #26 - #23	*Calculates stock left after one pass
N150 IF (#50 > 0 #50 = 0) THEN; #60 = 1; ENDIF	*If STK (Stock Remaining) is greater than zero then pass test. If pass programs jumps to line 170.
N155 ELSE	*If groove direction (and size) is negative.
N160 #50 = #26 + #23	*STK remaining after one pass
N165 IF (#50 < 0 #50 = 0) THEN; #60 = 1; ENDIF	*If there is STK remaining or if STK = 0 then pass test, move to line 170
N170 ENDIF	*Ends if statement from line 140
N175 IF (#60 = 0) THEN	*program continues at line 190 if #60 != 0
N180 PRINT (TOOL IS WIDER THAN GROOVE!)	*Error Message
N185 M30	*Terminate subroutine
N190 ENDIF	*Ends if statement from line 175
N195 #52 = #1012 + #50	*Absolute end position of groove width
N200 #86 = #1012; #87 = #1010	*Save current X and Z position
N205 IF (#24 < 0) THEN	*If X depth is negative, set clearance negative
N210 #51 = -(#51)	
N215 ENDIF	*Ends IF statement from line 205
N220 #53 = #51 + #24	*Total depth in X
N225 #55 = FIX(ABS(#50/#23))	*Number of loops
N230 IF ((#55*ABS(#23))= ABS(#50)) THEN; #55 = #55 -1 ENDIF	*Checks how many loops are needed to finish groove.
N235 #56 = #50/(#55+1)	*Calculates distance to move along Z axis after each cut is made.
N240 G91 G01 X#53	*Cutting motion begins
N245 G00 X(-#53)	*Rapid out
N250 LOOP #55	*#55= number of passes to cut
N255 G91 G00 Z#56	*Move Z position one tool width over
N260 G01 X#53	
N265 G00 X(-#53)	*Rapid out

Program Body	Comments
N270 END	*End Loop
N275 IF (#50 != 0) THEN	
N280 G90 G00 Z#52	
N285 G91 G01 X#53	
N290 G90 G00 X#87 Z#86	
N295 ENDIF	*Ends IF statement from line 275
N300 M99	*Ends macro, return to main program

Section 17 - Machine Software and Peripherals Installation

Passwords

The Setup Utility requires a password for restricted parameters. The CNC limited access password is 159. The service access password is Z48.

Machine Software Installation

The CNC software is installed when the machine is set up and during software updates.

To install the machine software, do the following:

1. In Manual Mode, press **SHIFT + Exit (F10)**. The CNC's startup screen is displayed.
2. Highlight **Setup Utility**, and press **ENTER**. The Setup Menu is displayed.
3. Highlight **Builder Setup**, and press **ENTER**. The Builder Setup Menu is displayed.
4. Highlight **Software Update**, and press **ENTER**. The installation sequence begins.
5. Follow the instructions on the screen.

Printer Installation

The machine builder determines whether the system will support a printer option. To connect a printer, you must provide the cable and connect it to the DB-25 printer port on the machine. The connector is usually labeled "printer".

Keyboard Installation

The machine builder determines whether the system will support a keyboard option. If the system supports a keyboard, plug the keyboard DIN connector into the side of the computer chassis, in the connector labeled "Keyboard". While the keyboard is installed, the console keypad remains inactive.

WARNING: There is no keyboard equivalent for the E-STOP. Therefore, emergency shutdowns (E-STOP) cannot be performed via keyboard. You must use the console E-STOP.

NOTE: Industrial grade keyboards are recommended for shop environments.

Keypad Equivalent Keyboard Keys

Refer to Table 17-1.

Table 17-1, Keyboard Equivalents

Function	CNC Key Face	Keyboard Keystroke Equivalent
CLEAR		Z
ARROWS		ARROWS
ENTER		ENTER
SHIFT		SHIFT
X (axis)		(X)
Y (axis)		(Y)
Z		(ALT + C)
START		(ALT + S)
HOLD		(ALT + H)
Activate System Information Screen	Refer to " Displaying System Information " in " Section 10 - Program Management ."	(ALT + I)
Activate Large Position Display		(B)
Switch to Rapid Override		(R + ENTER)
Activate IPI Monitor		(P)

Section 18 - Off-line Software

The off-line version of the software requires an **Intel®** based Personal Computer (PC) or 100% compatible. A minimum of 4MB of RAM is required. The hard disk drive must have a minimum of 4MB of space available.

The off-line software is also compatible with **Microsoft® Windows®** Operating Systems, such as **Windows NT®**, Windows 95, Windows 98, or Windows 2000.

Passwords

Access to some parameters is restricted. Refer to **Table 18-1**. Where required, the CNC prompts the user to enter a password.

To access protected parameters:

1. Type the service password, and press **ENTER**. The CNC will enable you to change the protected settings.
2. There are four access levels protected by passwords. Operators are assigned limited passwords, which enable them to set parameters used in normal machine operations. Service passwords enable a higher level of access. The factory password is not used outside the manufacturer's plant.

Table 18-1, Off-line Passwords

Restriction Level	Password
Limited access	Q1
Service access	Q2
Integrated Programmable Intelligence (IPI)	Q3

Exiting the Software

Press **ESC** to exit the software from the Main Menu selection screen.

 ** Intel® is a registered trademark of Intel Corporation in the United States and/or other countries.

** Microsoft®, Windows®, and Windows NT® are registered trademarks of Microsoft Corporation in the United States and/or other countries.

Windows Off-line Software Installation

1. Insert the installation disk in the floppy drive.
2. Go to the task bar, and click on the **Start** button. Select **Run**. The Run window activates.
3. In the **Open** entry field, type "**A:setup**". Click on **OK**. The installation procedure will then begin.
4. Follow the on-screen prompts as they appear.

NOTE: Substitute " B:setup " for " A:setup " if your 3.5 inch floppy is in the B:drive.
--

Running Off-line Software from Windows

1. If you selected "Desktop Icon" (recommended) during the installation, click the 4200T icon on your desktop.
2. If you selected Start Menu, start from there.

System Settings

Maximum Memory Allocated

In the Setup Utility, you can adjust the amount of memory allocated to the CNC software. Set the **Maximum Memory Allocated** parameter between 2 MB and 18 MB. This feature limits the amount of memory available to the software, preventing the CNC program from tapping into Windows' large virtual memory supply. Allocating too much memory to the control software will dramatically increase startup time and make the software run slower.

To change the **Maximum Memory Allocated** parameter:

1. In the CNC's startup screen, select **Setup Utility**, and press **ENTER**. The Setup Options menu is displayed.
2. Select **Operator Setup**, and press **ENTER**. The General Software menu is displayed.
3. Select **Control Software**, and press **ENTER**. The Control Software menu is displayed.
4. Select **Maximum Memory Allocated**, and press **ENTER**. Type the appropriate value, and press **ENTER**.

The only time it might be necessary to increase this parameter is when editing a program that is larger than this value. In this case, the CNC will generate an error message indicating that there is not enough memory to edit the program. To correct this problem, change this parameter to be the size of program plus 1MB.

Disabled Features

The following software features, found in the Program Directory's **Utility (F9)** pop-up are not available under any Windows operating system:

- Check Disk
- Floppy Format
- Disk Optimization
- Program Restore

If you select one of these features, the CNC displays a message to inform you that the feature is disabled.

Section 19 - Sample Programs

Introduction

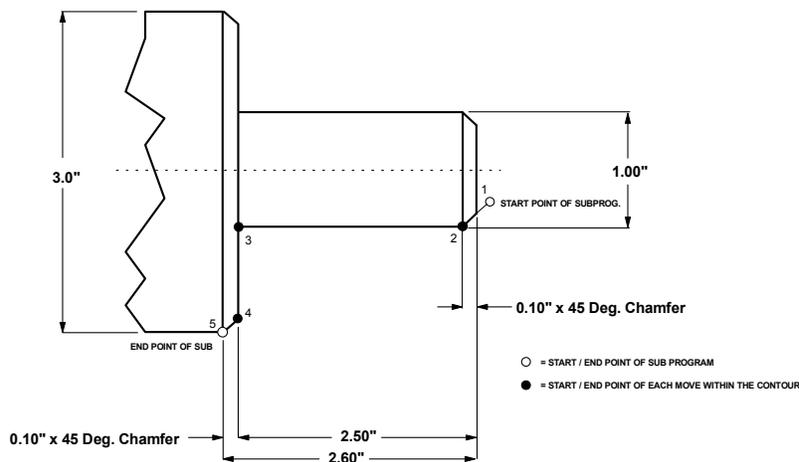
The following sample programs are examples of typical programming formats used in the 4200T CNC. The sample programs are included in your machine and offline installation disks.

All program blocks are commented. The comments are located to right of the asterisk (*) and are used to describe the program code and operations of that block.

Sample Program #1

One operation is required to complete this component. Part must be extended a minimum of 2.75 inches from chuck jaws with stock size 3.00 inches diameter.

Refer to **Figure 19-1**. The program shows a typical format used to face and turn a component. A sub-program is shown for the Area Clear Cycle (G73). The sub-program describes the Finish Profile of the component.



TOOL REQUIRED



TOOL No.1
ROUGH TURN

Figure 19-1, Layout for Sample Program #1

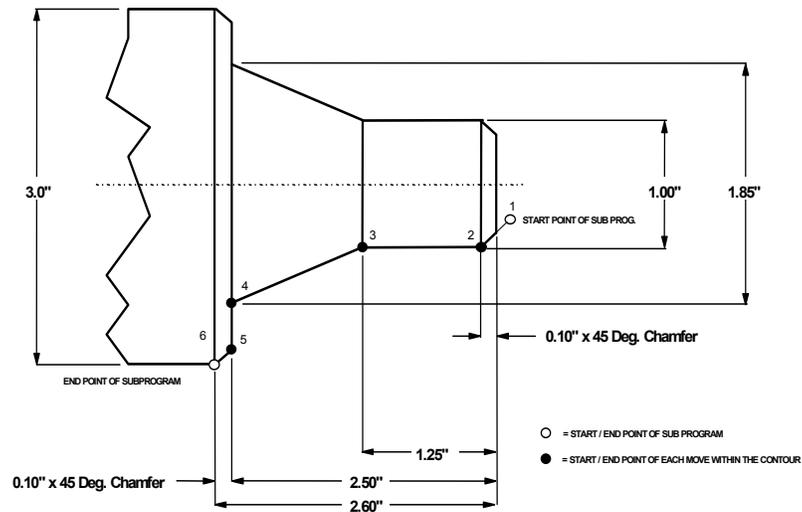
SAMPLE-1.G

G70 G90 G0 X0 Z0 T0	*INCH, ABSOLUTE, RAPID X Z, TOOL 0
M43	*HIGH-SPINDLE RANGE
G24 S1500	*RPM LIMIT WITH CONSTANT SURFACE SPEED
T1	*CALL TOOL No.1 (ROUGH/FINISH FACE/TURN TOOL)
G96 S400	*400 SURFACE FEET PER MINUTE
X3.2 Z0 M3	*RAPID TO START POSITION TO FACE OFF, SPINDLE FWD
G1 X-.065 F.007	*FACE OFF PART AT 0.007 INCHES PER REVOLUTION
G0 X3 Z.1	*RAPID TO START OF AREA CLEAR CYCLE
G73 W22 A0.1 R0.025 S0.005 C1 B1 J0.01 K.007	*AREA CLEARANCE CYCLE
G0 X0 Z0 T0 M5	*RAPID TO TOOL CHANGE POSITION, SPINDLE STOP
M2	*END OF MAIN PROGRAM
O22	*SUB-PROGRAM No.22 (OD PROFILE)
G0 X0.6 Z.1	*START POINT OF FINISH PROFILE
G1 X1 Z-0.1	*END POINT OF 1.00" DIA CHAMFER
Z-2.5	*FEED ACROSS 1.00" DIA. TO SHOULDER
X2.8	*FACE SHOULDER TO 2.8" DIA.
X3 Z-2.6	*CHAMFER STOCK DIAMETER
M99	*END SUB-PROGRAM

Sample Program #2

One operation is required to complete this component. Part must be extended a minimum of 2.75 inches from chuck jaws with stock size 3.00 inches diameter.

Refer to **Figure 19-2**. The program below shows a typical format used to face and turn a component. A sub-program is shown for the Area Clear Cycle (G73). The sub-program describes the Finish Profile of the component.



TOOL REQUIRED



TOOL No.1
ROUGH TURN

Figure 19-2, Layout for Sample Program #2

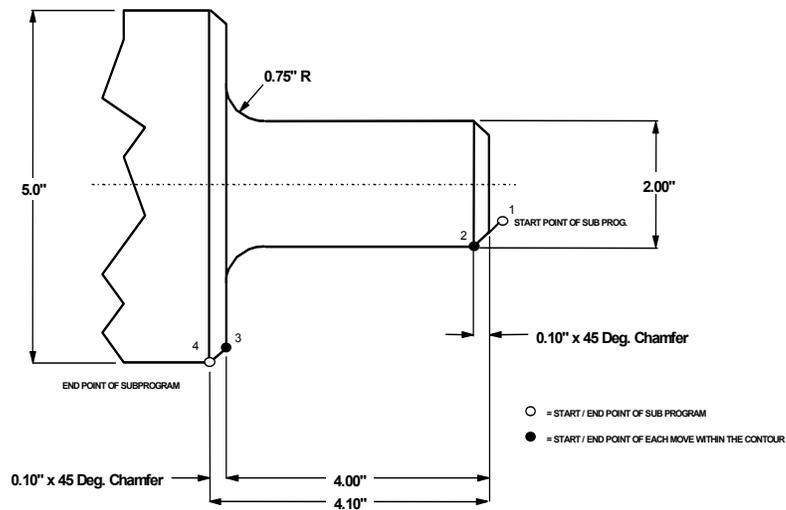
SAMPLE-2.G

G70 G90 G0 X0 Z0 T0	*INCH, ABSOLUTE, RAPID X Z, TOOL 0
M43	*HIGH SPINDLE RANGE
G24 S1500	*RPM LIMIT WITH CONSTANT SURFACE SPEED
T1	*CALL TOOL No.1 (ROUGH/FINISH FACE/TURN TOOL)
G96 S400	*400 SURFACE FEET PER MINUTE
X3.2 Z0 M3	*RAPID TO START POSITION TO FACE OFF, SPINDLE FWD
G1 X-.065 F.007	*FACE OFF PART AT 0.007" PER REVOLUTION
G0 X3 Z.1	*RAPID TO START OF AREA CLEAR CYCLE
G73 W3 A.1 R.025 S.005 C1 B1 J.01	*AREA CLEARANCE CYCLE
G0 X0 Z0 T0 M5	*RAPID TO TOOL CHANGE POSITION, SPINDLE STOP
M2	*END OF MAIN PROGRAM
O3	*SUB-PROGRAM NO.3 (OD PROFILE)
G0 X0.6 Z.1	*START POINT OF FINISH PROFILE
G1 X1 Z-0.1	*END POINT OF 1.00" DIA CHAMFER
Z-1.25	*FEED ACROSS 1.00" DIA. TO START POINT OF ANGLE
X1.85 Z-2.5	*FEED TO END POINT OF ANGLE
X2.8	*FACE OUT TO START OF CHAMFER ON STOCK DIAMETER
X3 Z-2.6	*CHAMFER STOCK DIAMETER
M99	*END SUB-PROGRAM

Sample Program #3

One operation is required to complete this component. Part must be extended a minimum of 4.50 inches from chuck jaws with stock size 5.00 inches diameter.

Refer to **Figure 19-3**. The program below shows a typical format used to face and turn a component. A sub-program is shown for the Area Clear Cycle (G73). The sub-program describes the Finish Profile of the component. The sub-program uses the Blue Print Programming (available via the HELP menu) format to describe the Finish Profile. Blue Print Programming provides for stating multiple moves in the same block without having to calculate end points.



TOOL REQUIRED



TOOL No.1
ROUGH TURN

Figure 19-3, Layout for Sample Program #3

SAMPLE-3.G

G70 G90 G0 X0 Z0 T0	*INCH, ABSOLUTE, RAPID X Z, TOOL 0
M43	*HIGH SPINDLE RANGE
G24 S1500	*RPM LIMIT WITH CONSTANT SURFACE SPEED
T1	*CALL TOOL No.1 (ROUGH/FINISH, FACE/TURN TOOL)
G96 S350	*350 SURFACE FEET PER MINUTE
X5.2 Z0 M3	*RAPID TO START POSITION TO FACE OFF, SPINDLE FWD
G1 X-.065 F.007	*FACE OFF PART AT 0.007" PER REVOLUTION
G0 X5 Z.1	*RAPID TO START OF AREA CLEAR CYCLE
G73 W1 A.1 R.025 S.005 C1 B1 J.01	*AREA CLEARANCE CYCLE
G0 X0 Z0 T0 M5	*RAPID TO TOOL CHANGE POSITION, SPINDLE STOP
M2	*END OF MAIN PROGRAM
O1	*SUB-PROGRAM No.1 (OD PROFILE)
G0 X1.6 Z.1	*START POINT OF FINISH PROFILE
G1 X2 Z-0.1	*END POINT OF 1.00" DIA CHAMFER
B180 Q.75 B90 X4.8 Z-4	*TURN, RADIUS AND FACE TO GIVEN X,Z, END POINT
X5 Z-4.1	*CHAMFER STOCK DIAMETER
M99	*END SUB-PROGRAM

Sample Program #4

One operation is required to complete this component. Part must be extended a minimum of 4.50 inches from chuck jaws with stock size 4.50 inches diameter.

Refer to **Figure 19-4**. The program below shows a typical format used to face, rough and finish turn a component. A sub-program is shown for the Area Clear Cycle (G73). The sub-program describes the Finish Profile of the component. The sub-program uses the Blue Print Programming (available via the HELP menu) format to describe the Finish Profile. Blue Print Programming provides for stating multiple moves in the same block without having to calculate end points.

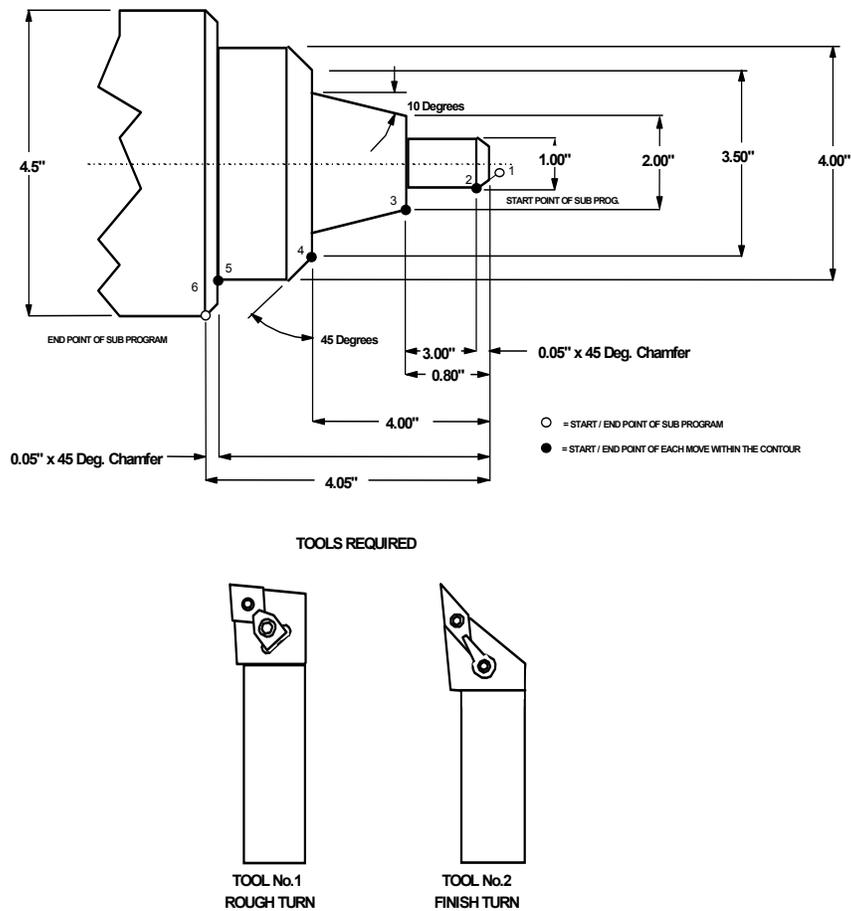


Figure 19-4, Layout for Sample Program #4

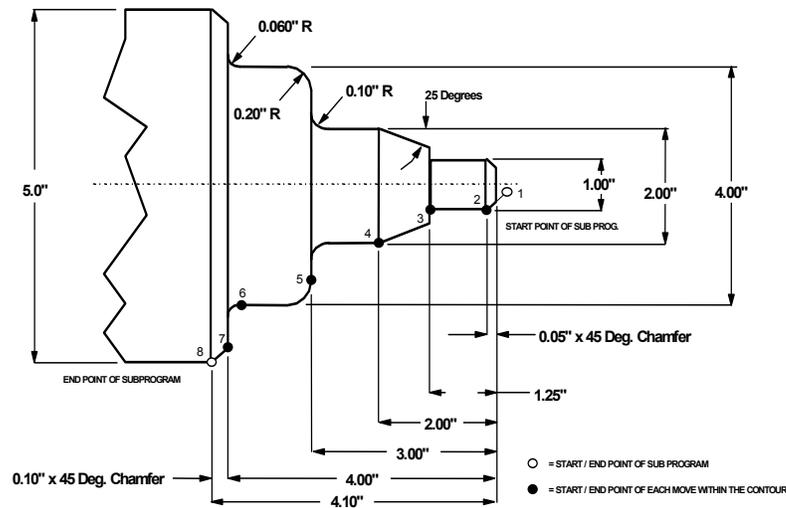
SAMPLE-4.G

G70 G90 G0 X0 Z0 T0	*INCH, ABSOLUTE, RAPID X Z, TOOL 0
M43	*HIGH SPINDLE RANGE
G24 S1800	*RPM LIMIT WITH CONSTANT SURFACE SPEED
T1	*CALL TOOL NO.1 (FACE AND ROUGH TURN TOOL)
G96 S400	*400 SURFACE FEET PER MINUTE
X4.7 Z0 M3	*RAPID TO POSITION TO FACE OFF, SPINDLE FWD
G1 G95 X-.065 F.006 M8	*FACE OFF AT 0.006" PER REV. COOLANT ON
G0 X4.5 Z.1	*RAPID TO START POSN FOR AREA CLEAR. CYCLE
G73 W1 A0.15 R0.02 S0.005 C0 B1 J0.015 K0.008	*AREA CLEARANCE CYCLE
M5	*COOLANT OFF
G0 X0 Z0 T0 M9	*RAPID TO TOOL CHANGE POSITION, SPINDLE OFF
T2	*CALL TOOL NO.2 (FINISH TURN TOOL)
M43	*HIGH SPINDLE RANGE
G96 S450	*450 SURFACE FEET PER MINUTE
G41 X.7 Z.1 M3	*TNC LEFT, RAPID POSN FOR FINISH PASS, SPINDLE FWD
M98 P1	*CALL SUB-PROGRAM NO.1 FOR FINISH PASS
G40 X4.7 M9	*CANCEL TNC, FEED OFF PART O.D. COOLANT OFF
G0 X0 Z0 T0 M5	*RAPID TO TOOL CHANGE POSITION, SPINDLE OFF
M2	*END MAIN PROGRAM
O1	*SUB-PROGRAM O1 (OD PROFILE)
G0 X.7 Z.1	*START POINT OF FINISH PROFILE
G1 X1 Z-.05	*END POINT OF 1.00" CHAMFER
B180 B90 X2 Z-.8	*TURN 1.00" DIA. FACE OUT TO 2.00" DIA.
B170 B90 X3.5 Z-3	*TURN ANGLE, FACE OUT TO 3.5" DIA.
B135 B180 X4 Z-4	*TURN 45 DEGREE ANGLE AND 4.00" DIA.
B90 B135 X4.5 Z-4.05	*FACE OUT AND CHAMFER 4.5" DIA.
M99	*END OF SUB-PROGRAM

Sample Program #5

One operation is required to complete this component. Part must be extended a minimum of 4.50 inches from chuck jaws with stock size 5.00 inches diameter.

Refer to **Figure 19-5**. The program below shows a typical format used to face, rough and finish turn a component. A sub-program is shown for the Area Clear Cycle (G73). The sub-program describes the Finish Profile of the component. The sub-program uses the Blue Print Programming (available via the HELP menu) format to describe the Finish Profile. Blue Print Programming provides for stating multiple moves in the same block without having to calculate end points.



TOOLS REQUIRED

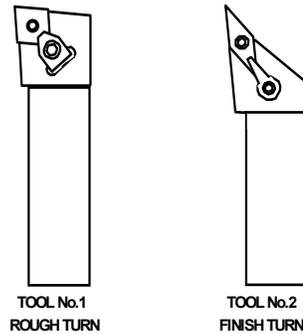


Figure 19-5, Layout for Sample Program #5

SAMPLE-5.G

G70 G90 G0 X0 Z0 T0	*INCH, ABSOLUTE, RAPID X Z, TOOL 0
M43	*HIGH SPINDLE RANGE
G24 S1600	*RPM LIMIT WITH CONSTANT SURFACE SPEED
T1	*CALL TOOL NO.1 (ROUGH FACE AND TURN TOOL)
G96 S350	*350 SURFACE FEET PER MINUTE
X5.2 Z0 M3	*RAPID TO POSITION TO FACE OFF
G1 G95 X-.065 F.006	*FACE OFF PART AT 0.006" PER REV.
G0 X5 Z.1	*RAPID TO START POINT OF AREA CLEAR CYCLE
G73 W1 A0.125 R0.02 S0.005 C0 B1 J0.015 K0.008	*AREA CLEAR CYCLE
G0 X0 Z0 T0 M5	*RAPID TO TOOL CHANGE POSITION, SPINDLE STOP
T2	*CALL TOOL NO.2 (FINISH TURN TOOL)
M43	*HIGH SPINDLE RANGE
G96 S450	*450 SURFACE FEET PER MINUTE
G41 X.7 Z.1 M3	*TNR COMP RIGHT, RAPID TO PROFILE START POSN
M98 P1	*CALL SUB-PROGRAM NO.1 FOR FINISH PASS
G40 X5.2	*CANCEL TOOL NOSE RADIUS COMP. MOVE OFF OD
G0 X0 Z0 T0 M5	*RAPID TO TOOL CHANGE POSITION, SPINDLE OFF
M2	*END OF MAIN PROGRAM
O1	*SUB-PROGRAM NO.1 (FINISH PROFILE)
G0 X.7 Z.1	*START POINT OF FINISH PROFILE
G1 X1 Z-.05	*CHAMFER 1.0" DIA
Z-1.25	*TURN 1.0" DIA
B90 B155 X2 Z-2	*FACE AND TURN ANGLE TO GIVEN X,Z END POINTS
B180 Q.1 B90 X3.6 Z-3	*TURN, RADIUS AND FACE TO GIVEN X,Z END POINTS
Q0.2 B180 X4 Z-3.94	*RADIUS AND TURN TO GIVEN X,Z END POINTS
Q0.06 B90 X4.8 Z-4	*RADIUS AND FACE TO GIVEN X,Z END POINTS
X5 Z-4.1	*CHAMFER 5.0" DIA
M99	*END OF SUB-PROGRAM

Sample Program #6

One operation is required to complete this component. Part must be extended a minimum of 4.50 inches from chuck jaws with stock size 5.00 inches diameter.

Refer to **Figure 19-6**. The program below shows a Program to face, center drill, drill and rough and finish bore a component. A Sub-Program is shown for the Area Clear Cycle (G73). The Sub-Program describes the Finish Profile of the component. All TEXT to the right of the star (*) are ignored when running the program and are only shown to describe the Program code and operations on that block.

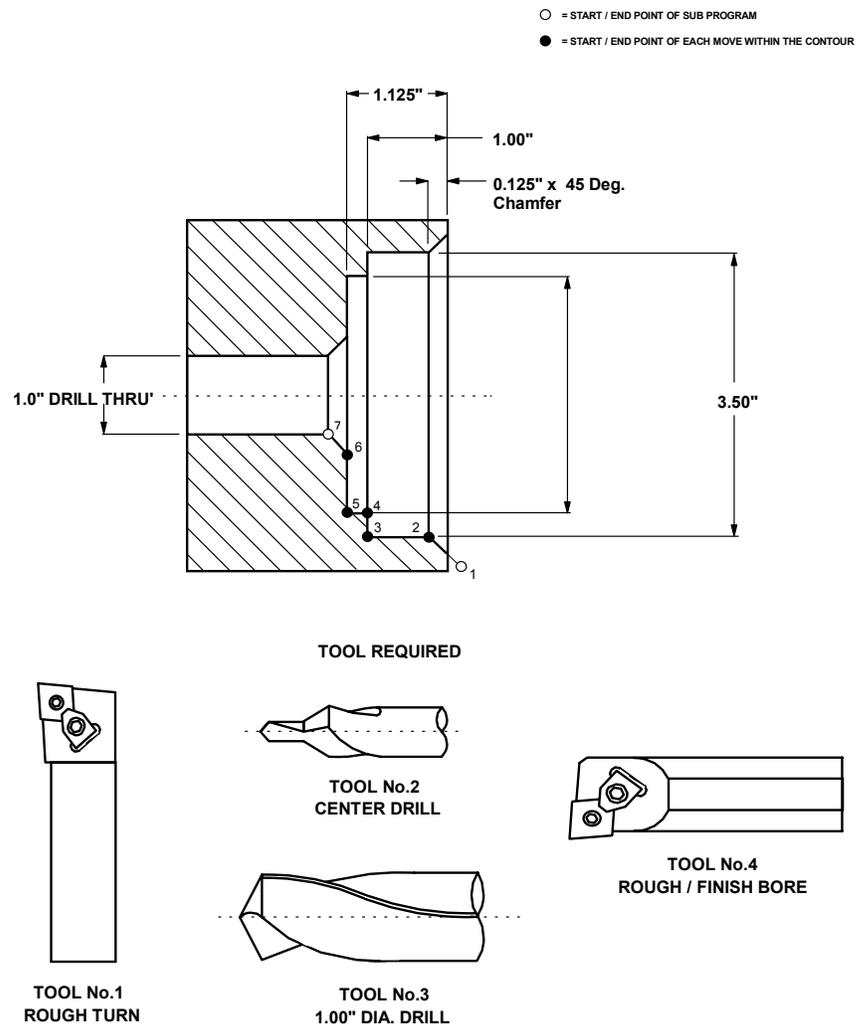


Figure 19-6, Layout for Sample Program #6

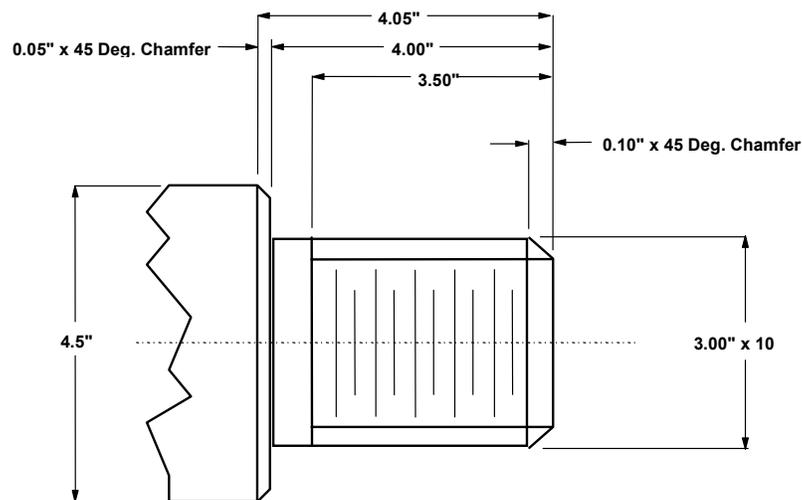
SAMPLE-6.G

G70 G90 G0 X0 Z0 T0	*INCH, ABSOLUTE, RAPID X Z, TOOL 0
M43	*HIGH SPINDLE RANGE
G24 S1600	*RPM LIMIT WITH CONSTANT SURFACE SPEED
T1	*CALL TOOL NO.1 (FACING TOOL)
G96 S400	*400 SURFACE FEET PER MINUTE
X3.7 Z0 M3	*RAPID TO POSITION TO FACE OFF
G1 X-.065 G95 F.006 M8	*FACE OFF AT 0.006" PER REV. COOLANT ON
G0 Z.1 M5	*RAPID OFF FRONT FACE, SPINDLE OFF
X0 Z0 T0 M9	*RAPID TO TOOL CHANGE POSITION, COOLANT OFF
T2	*CALL TOOL NO.2 (CENTER DRILL)
M42	*MID SPINDLE RANGE
G97 S500	*500 RPM
X0 Z.1 M3	*RAPID TO PART CENTER, 0.1 FROM FRONT FACE
G1 Z-.275 M8 F.005	*CENTERDRILL TO DEPTH AT 0.005" PER REV. COOLANT ON
G0 Z.1 M9	*RAPID OUT OF CENTERDRILLED HOLE, COOLANT OFF
X0 Z0 T0 M5	*RAPID TO TOOL CHANGE POSITION, SPINDLE OFF
T3	*CALL TOOL NO.3 (1.0" DIA DRILL)
M42	*MID SPINDLE RANGE
G97 S500	*500 RPM
X0 Z.1 M3	*RAPID TO PART CENTER, 0.1" FROM FACE, SPINDLE FWD
M8	*COOLANT ON
G78 Z-3 R0.1 F0.005 I0.3	*PECK DRILLING CANNED CYCLE
M9	*COOLANT OFF
G0 X0 Z0 T0 M5	*RAPID TO TOOL CHANGE POSITION, SPINDLE OFF
T4	*CALL TOOL NO.4 (ROUGH/FINISH BORING BAR)
M43	*HIGH SPINDLE RANGE
G96 S375	*375 SURFACE FEET PER MINUTE
X1 Z.1 M3	*RAPID TO START POINT OF CLEAR.CYCLE, SPINDLE FWD
M8	*COOLANT ON
G73 W1 A0.1 R0.02 S0.005 C1 B0 J0.012 K0.007	*AREA CLEARANCE CYCLE
G0 Z.1 M9	*RAPID CLEAR OF BORE, COOLANT OFF
X0 Z0 T0 M5	*RAPID TO TOOL CHANGE POSN. SPINDLE OFF
M2	*END MAIN PROGRAM
O1	*SUB-PROGRAM O1 (ID PROFILE)
G0 X3.95 Z.1	*START POINT OF FINISH PROFILE
G1 X3.5 Z-.125	*CHAMFER 3.5" ID
Z-1	*BORE 3.5" DIA
X2.75	*FACE TO 2.75" ID
Z-1.125	*BORE 2.75" ID
X1	*FACE DOWN TO DRILL DIA.
M99	*END SUB-PROGRAM

Sample Program #7

One operation is required to complete this component. Part must be extended a minimum of 4.50 inches from chuck jaws with stock size 4.50 inches diameter.

Refer to **Figure 19-7**. The program below shows a typical format to face, turn and OD thread a component. A Sub-Program is shown for the Area Clear Cycle (G73). The Sub-Program describes the Finish Profile of the component.



**TOOLS REQUIRED
OPERATION No.1**

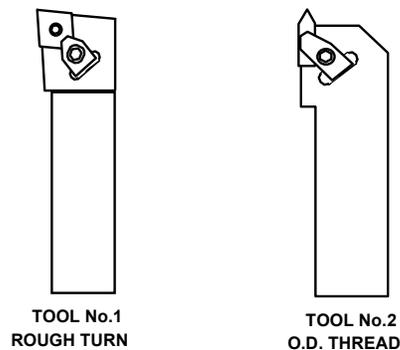


Figure 19-7, Layout for Sample Program #7

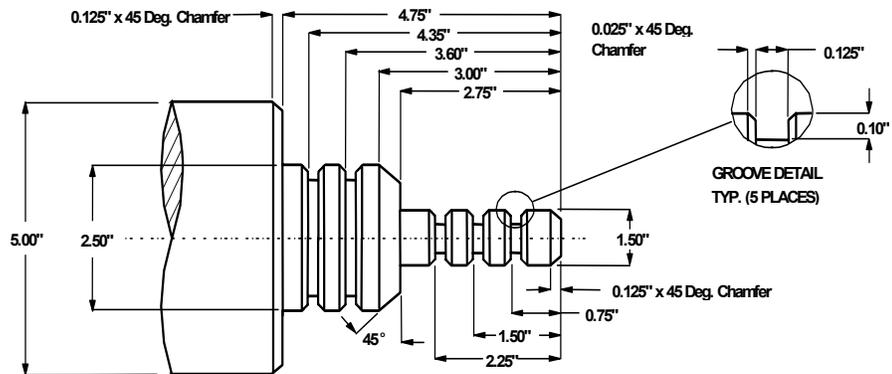
SAMPLE-7.G

G70 G90 G0 X0 Z0 T0	*INCH, ABSOLUTE, RAPID X Z, TOOL 0
M43	*HIGH SPINDLE RANGE
G24 S1600	*RPM LIMIT WITH CONSTANT SURFACE SPEED
T1	*CALL TOOL NO.1 (ROUGH/FINISH FACE/TURN TOOL)
G96 S400	*400 SURFACE FEET PER MINUTE
X4.7 Z0 M3	*RAPID POSITION TO FACE OFF, SPINDLE FORWARD
M8	*COOLANT ON
G1 X-.065 G95 F.007	*FACE OFF AT 0.007" INCH PER REV.
G0 X4.5 Z.1	*RAPID POSITION FOR AREA CLEAR. CYCLE
G73 W10 A.1 R.02 S.005 C1 B1 J.015 K.008	*AREA CLEAR. CYCLE
G0 X0 Z0 T0 M9	*RAPID TO TOOL CHANGE POSITION, COOLANT OFF
M5	*SPINDLE STOP
T2	*CALL TOOL NO.2 (THREADING TOOL)
M42	*MID SPINDLE RANGE
G97 S500	*500 RPM
X3.2 Z.5 M3 G95 F.04	*RAPID POSN FOR THD CYCLE, SET FEED 0.04" PER REV
M8	*COOLANT ON
G83 E10 Z-4 C-0.015 S2 R-0.1	*THREADING CANNED CYCLE
G0 X0 Z0 T0 M9	*RAPID TO TOOL CHANGE POSITION, COOLANT OFF
M5	*SPINDLE OFF
M2	*END OF MAIN PROGRAM
O10	*SUB-PROGRAM NO.10 (OD PROFILE)
G0 X2.6 Z.1	*START POSITION OF FINISH PROFILE
G1 X3 Z-.1	*CHAMFER 3" DIAMETER
Z-4	*TURN 3" DIAMETER
X4.4	*FACE OUT TO 4.4" DIA.
X4.5 Z-4.05	*CHAMFER 4.5" DIAMETER
M99	*END OF SUB-PROGRAM

Sample Program #8

One operation is required to complete this component. Part must be extended a minimum of 5.00 inches from chuck jaws with stock size 5.00 inches diameter.

Refer to **Figure 19-8**. The program below shows a typical format to face, rough and finish turn and groove (in multiple places) using an incremental Sub-Program. A Sub-Program is also shown for the Area Clear Cycle (G73). The Sub-Program describes the Finish Profile of the component. The Sub-Program uses the Blue Print Programming (available via the HELP menu) format to describe the Finish Profile. Blue Print programming provides for stating multiple moves in the same block without having to calculate end points.



TOOLS REQUIRED FOR OPERATION No.1

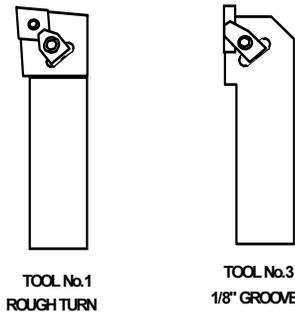


Figure 19-8, Layout for Sample Program #8

SAMPLE-8.G

G70 G90 G0 X0 Z0 T0	*INCH, ABSOLUTE, RAPID X Z, TOOL 0
M42	*MID-SPINDLE RANGE
G24 S1800	*RPM LIMIT WITH CONSTANT SURFACE SPEED
T1	*CALL TOOL NO.1 (ROUGH/FINISH FACE/TURN TOOL)
G96 S550	*550 SURFACE FEET PER MINUTE
X5.2 Z0 M3	*RAPID POSITION TO FACE OFF, SPINDLE FORWARD
G1 X-.04 G95 F.01	*FACE OFF AT 0.01" PER REV.
G0 X5 Z.1	*RAPID POSITION TO START OF AREA CLEAR.CYCLE
G73 W1 A0.15 R.02 S.005 C1 B1 J.015 K.008	*AREA CLEAR.CYCLE
G0 X0 Z0 T0 M5	*RAPID TO TOOL CHANGE POSITION, SPINDLE OFF
M43	*HIGH SPINDLE RANGE
T2	*CALL TOOL NO.2 (1/8" WIDE GROOVE TOOL)
G96 S650 M3	*650 SURFACE FEET PER MINUTE
G0 X1.7 Z-.75 G95 F.003	*RAPID POSITION FOR 1ST GROOVE ON 1.5" DIA.
G95 F.003	*SET FEEDRATE TO 0.003" PER REV.
M98 P99	*CALL SUB-PROGRAM NO.99 (GROOVING ROUTINE)
G0 Z-1.5	*RAPID POSITION FOR 2ND GROOVE ON 1.5" DIA.
M98 P99	*CALL SUB-PROGRAM NO.99 (GROOVING ROUTINE)
G0 Z-2.25	*RAPID POSITION FOR 3RD GROOVE ON 1.5" DIA.
M98 P99	*CALL SUB-PROGRAM NO.99 (GROOVING ROUTINE)
G0 X2.7	*RAPID POSITION X AXIS FOR GROOVES ON 2.5" DIA.
Z-3.6	*RAPID POSITION Z AXIS FOR 1ST GROOVE ON 2.5" DIA.
M98 P99	*CALL SUB-PROGRAM NO.99 (GROOVING ROUTINE)
G0 Z-4.35	*RAPID POSITION FOR 2ND GROOVE ON 2.5" DIA.
M98 P99	*CALL SUB-PROGRAM NO.99 (GROOVING ROUTINE)
G0 X0 Z0 T0 M5	*RAPID TO TOOL CHANGE POSITION, SPINDLE OFF
M2	*END OF MAIN PROGRAM
O1	*SUB-PROGRAM NO.1 (OD CONTOUR)
G0 X1.05 Z.1	*START POINT OF FINISH CONTOUR
G1 X1.5 Z-.125	*CHAMFER 1.5" DIAMETER
B180 B90 X2 Z-2.75	*TURN 1.5" DIA. AND FACE OUT TO 2.0" DIA
B135 B180 X2.5 Z-4.75	*TURN 45 DEGREE ANGLE AND 2.5" DIA.
B90 B135 X5 Z-4.85	*FACE OUT AND CHAMFER 5.0" DIAMETER
M99	*END OF SUB-PROGRAM
O99	*SUB-PROGRAM NO.99 (GROOVING ROUTINE)
G91 G1 X-.4	*INCREMENTAL, FEED TO GROOVE DEPTH
G0 X.22	*RAPID OUT OF GROOVE
Z-.035	*POSITION FOR LEFT SIDE CHAMFER
G1 X-.07 Z.035	*45 DEGREE CHAMFER
G0 X.07	*RAPID OUT OF GROOVE
Z.035	*POSITION FOR RIGHT SIDE CHAMFER
G1 X-.07 Z-.035	*45 DEGREE CHAMFER
G0 X.25	*RAPID TO ORIGINAL START POSITION
G90	*ABSOLUTE MODE
M99	*END OF SUB-PROGRAM

Sample Program #9

One operation is required to complete this component. Part must be extended a minimum of 10.5 inches from chuck jaws; support with live center stock size 3.00 inches diameter by 12.5 inches long.

Refer to **Figure 19-9**. The program below shows a typical format to rough and finish turn a shaft component (including grooving and threading). A Sub-Program is shown for the Area Clear Cycle (G73). The Sub-Program describes the Finish Profile of the component. The Sub-Program uses the Blue Print Programming (available via the HELP menu) format to describe the Finish Profile. Blue Print Programming provides for stating multiple moves in the same block without having to calculate end points. Sub-Program O2 describes the groove profile, this is written in a sub-program so that multiple grooves can be machined.

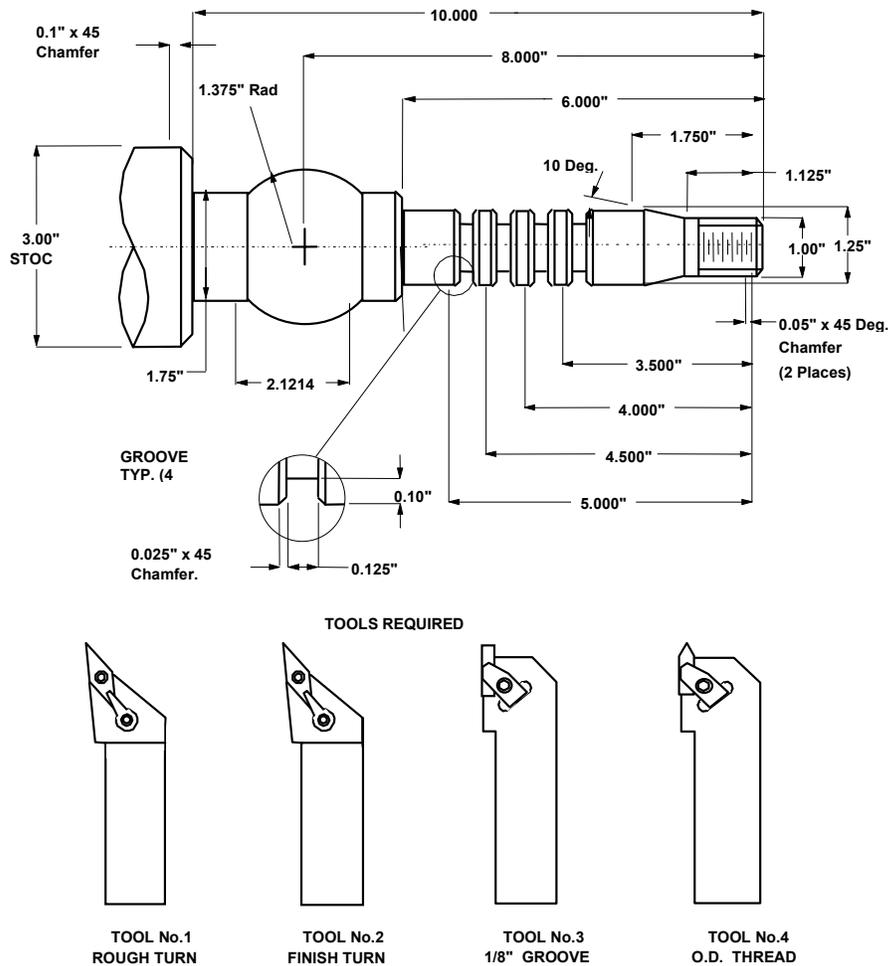


Figure 19-9, Layout for Sample Program #9

SAMPLE-9.G

G70 G90 G0 X0 Z0 T0 *INCH, ABSOLUTE, RAPID X Z, TOOL 0
 M43 *HIGH SPINDLE RANGE
 G24 S1600 *RPM LIMIT WITH CONSTANT SURFACE SPEED
 T1 *CALL TOOL NO.1 (ROUGH TURN TOOL)
 G96 S400 *400 SURFACE SPEED PER MINUTE
 X3 Z.1 M3 *RAPID TO START POINT OF AREA CLEAR CYCLE
 G73 W1 A0.15 R.02 S.01 C0 B1 P1 I.005 J.012 *AREA CLEARANCE CYCLE
 G0 X0 Z0 T0 M5 *RAPID TO TOOL CHANGE POSITION, SPINDLE OFF
 T2 *CALL TOOL NO.2 (FINISH TURN TOOL)
 M43 *HIGH SPINDLE RANGE
 G24 S1800 *RPM LIMIT WITH CONSTANT SURFACE SPEED
 G96 S475 *475 SURFACE SPEED PER MINUTE
 G41 G0 X.6 Z.1 M3 G95 F.008 * TNR COMP. LEFT, RAPID TO START POINT, SPD FWD
 M98 P1 *CALL SUB-PROGRAM NO.1 (OD PROFILE)
 G40 X3.2 *CANCEL TNR COMP. MOVE OFF PART DIAMETER
 G0 X0 Z0 T0 M5 *RAPID TO TOOL CHANGE POSITION, SPINDLE OFF
 T3 *CALL TOOL NO.3 (1/8" WIDE GROOVE TOOL)
 M43 *HIGH SPINDLE RANGE
 G97 S750 *750 RPM
 X1.45 Z-3.5 M3 *RAPID POSITION FOR 1ST GROOVE, SPINDLE FORWARD
 M98 P2 *CALL SUB-PROGRAM NO.2 (GROOVING SUB-PROGRAM)
 G0 Z-4 *RAPID POSITION FOR 2ND GROOVE
 M98 P2 *CALL SUB-PROGRAM NO.2 (GROOVING SUB-PROGRAM)
 G0 Z-4.5 *RAPID POSITION FOR 3RD GROOVE
 M98 P2 *CALL SUB-PROGRAM NO.2 (GROOVING SUB-PROGRAM)
 G0 Z-5 *RAPID POSITION FOR 4TH GROOVE
 M98 P2 *CALL SUB-PROGRAM NO.2 (GROOVING SUB-PROGRAM)
 G0 X0 Z0 T0 M5 *RAPID TO TOOL CHANGE POSITION, SPINDLE OFF
 T4 *CALL TOOL NO.4 (SINGLE POINT THREADING TOOL)
 M43 *HIGH SPINDLE RANGE
 G97 S675 *675 RPM
 X1.2 Z.2 *RAPID TO START POSITION FOR THREADING CYCLE
 G83 E16 Z-1.125 C-.018 D-.042 A60 S2 R-.1 *THREADING CANNED CYCLE
 G0 X0 Z0 T0 M5 *RAPID TO TOOL CHANGE POSITION, SPINDLE OFF
 M2 *END OF MAIN PROGRAM

O1 *SUB-PROGRAM NO.1 (OD PROFILE)
 G0 X0.6 Z.1 *START POINT OF FINISH PROFILE
 G1 X1 Z-.1 *CHAMFER 1.0" DIA.
 B180 | B170 X1.25 Z-1.75 *TURN AND 10 DEGREE ANGLE TO GIVEN X,Z, END POINTS
 Z-6 *TURN 1.25" GROOVE DIAMETER
 B90 | B135 X1.75 Z-6.05 *FACE AND CHAMFER 1.75" DIA.
 Z-6.9393 *TURN TO START OF 1.375" RADIUS
 G2 X1.75 Z-9.0607 R1.375 *TURN 1.375" RADIUS
 G1 Z-10 *TURN 1.75" DIAMETER
 B90 | B135 X3 Z-10.1 *FACE AND CHAMFER 3" DIAMETER
 M99 *END OF SUB-PROGRAM

O2 *SUB-PROGRAM NO.2 (GROOVING SUB-PROGRAM)

G1 X1.05	*FEED TO GROOVE DEPTH
G4 T2	*DWELL AT BOTTOM OF GROOVE (2 SECONDS)
G0 X1.27	*RAPID OUT OF THE GROOVE
G91 Z-.035	*INCREMENTAL, POSITION FOR LEFT GROOVE CHAMFER
G1 X-.07 Z.035	*CHAMFER LEFT SIDE OF GROOVE
G90 G0 X1.27	*ABSOLUTE, RAPID OUT OF THE GROOVE
G91 Z.035	*INCREMENTAL, POSITION FOR RIGHT GROOVE CHAMFER
G1 X-.07 Z-.035	*CHAMFER RIGHT SIDE OF GROOVE
G90 G0 X1.45	*ABSOLUTE, RAPID TO GROOVE START POSN
M99	*END OF SUB-PROGRAM

% (RPM), machine status display, 3-5
 % Feed, machine status display, 3-5
 .G extension, 10-1, 10-2, 10-3
 .S files, 10-14
4200T CNC Setup Utility Manual, P/N
 70000414, referenced, 5-32

A

abbreviating statements
 description, 6-9, 16-16
 listed, 6-9
 absolute mode
 (G90), 4-22
 activating Absolute, (G90), 13-12
 description, 1-5, 3-10
 absolute zero
 defined, 3-10
 preset (G92) listing, 4-24
 reference point, 1-5, 3-10
 accessing
 communication software, 15-2
 geometry tools, 14-16
 most recently used programs, 6-16
 activate
 background mode, 11-10
 geometry elements, 14-29
 activating
 manual mode, feed, 3-8
 manual mode, rapid, 3-8
 servos, 3-7
 shape editor, 14-2
 shapes, 14-16
 test link screen, 15-5
 active soft key, manual screen area, 3-5
 acute angles, compensation, around, 9-21
 adjusting
 Draw display, 8-8
 feedrate, 3-8
 fixture offsets, in a table, 4-15
 rapid move speed, 3-9
 advanced programming features
 block, separators, 16-1
 modifiers, description, 16-1
 modifiers, listed, 16-1
 probe move (G31), 16-13
 advancing
 to beginning or end of a block, 6-6
 to first or last block of a program, 6-6
 alphanumeric keys
 illustration, 2-2

listed, 2-3
 angular motion programming, example, 4-5
 applications, parametric programming,
 16-10
 arc
 move, input labels, listed, 13-6, 13-7
 reversing, direction, 14-13
 to cut, 13-17
 arc tools
 description, 14-8
 rounding tool, description, 14-9
 template table, 14-8
 ARCS Help Template Menu, 7-17
 ARROW keys, 2-5
 ASCII, data type, 15-4
 auto mode
 program, cancel, 11-5
 program, hold, 11-5
 program, to run, 11-4
 screen, illustration, 11-4
 starting block, select
 using arrow keys, 11-5
 using SEARCH, 11-5
 switch from, single-step mode, 11-3
 automatic mode, defined, 11-1
 automatic, Draw restart, 8-7
 axes, direction of motion, illustration, 1-2
 axis
 descriptions, 1-2
 select key, illustration, 3-2
 selecting, 3-11

B

background mode
 description, 11-10
 to activate, 11-10
 backing up, shapes, 14-32
 baud rate, setting, 15-3
 bi-directional
 cutting radial threads, (G86), 5-26
 longitudinal threading cycle, (G84), 5-20
 BINARY, data type, 15-4
 Blank Entry Fields, 7-13
 Blank fields, 7-13
 BLK, program area label, 3-6
 block
 advancing, to beginning or end of, 6-6
 number, program area label, 3-6
 restoring, 6-8
 selected, run Draw, 8-7

- separators, description, 16-1
 - skip variables, description, 16-5
 - skip variables, listed, table, 16-5
 - skips, setting selective, 16-6
 - blocks, writing, to another program, 6-13
 - boring cycle, (G88), 5-30
 - Both, Draw, description, 8-5
- ## C
- Calibrate X TLO, 9-5
 - Calibrate Z TLO, 9-5
 - call statements, description, 16-10
 - cancel
 - auto mode program, 11-5
 - Draw, program, 8-3
 - In-Position Mode, modal, exact stop check, (G64), 4-10
 - single-step run, execution, 11-2
 - canceling
 - edits to a program block, 6-10
 - tool nose radius (TNR), 9-17
 - unsaved edits, 6-17
 - canned cycle
 - cutting
 - longitudinal threads, (G83, uni-directional), 5-18
 - longitudinal threads, (G84, bi-directional), 5-20
 - radial threads, (G85, uni-directional), 5-23
 - radial threads, (G86, bi-directional), 5-26
 - lathe tool probe cycles
 - description, 5-32
 - G-codes, designations, 5-35
 - macro variable, settings, 5-33
 - system variable, settings, 5-33
 - programming
 - boring cycle, (G88), 5-30
 - chip break cycle, (G79), 5-12
 - longitudinal grooving cycle, (G81), 5-13
 - pecking cycle, (G78), 5-11
 - radial grooving cycle, (G82), 5-15
 - X-axis, radial (facing) roughing cycle, (G77), 5-9
 - X-axis, facing/boring cycle with defined profile, (G74), 5-4
 - Z-axis, rough turning or boring cycle, (G76), 5-7
 - Z-axis, turning/boring cycle
 - longitudinal, (G73), 5-1
 - tapping a hole, (G87), 5-28
 - C-axis programming
 - description, 12-3
 - disabling, 12-3
 - enabling, 12-3
 - feedrate, 12-4
 - jog, to position, 12-4
 - synchronize/unsynchronize, 12-4
 - C-axis, description, 1-2
 - chaining, geometry elements to create a shape, 14-26
 - chamfer, cutting, 13-16
 - chamfer, positive and negative, illustration, 13-16
 - chamfering rounding, modal, (G59 En), 4-16
 - chamfering, corners, 14-9
 - change word, feature description, 6-8
 - change, scales, 8-1
 - changing
 - fixture offsets, in a table, 4-15
 - fixture offsets, using calibrate keys, 4-16
 - jog mode, 3-11
 - link test link screen data display, 15-5
 - shape editor view, 14-15
 - the numbers, of look ahead blocks, 9-22
 - TNR compensation, in a program, 9-18
 - chip break cycle, (G79), 5-12
 - choosing correct, TNR compensation, 9-18
 - circle tools
 - description, 14-19
 - templates, listed, 14-19
 - circular interpolation
 - absolute mode, 4-7
 - incremental mode, 4-8
 - parameters, 4-6
 - partial arc, description, 4-8
 - CLEAR key, 2-5
 - clear, (M9351 X302), 12-3
 - clearing
 - a halted program, 11-5
 - entries, 2-7
 - messages, 2-8
 - receive area, 15-6
 - transmit area, 15-6
 - CLEARING, help template menu, 7-31
 - clockwise motion (G2), description, 4-6
 - CNC
 - enlarged motion display, to set, 11-6
 - parts counter, description, 11-9
 - timer, description, 11-9
 - codes, order of execution, 12-3

- COM1, 15-3
 - COM2, 15-3
 - Comm Port, selecting, 15-3
 - command line, manual screen area, 3-4
 - COMMAND, program area label, 3-6
 - Commanding Moves in SCI Mode, 13-5
 - comments, including in a program listing, 6-17
 - common (global) variables, description, 16-4
 - communication
 - description, 15-1
 - parameters, setting, 15-3
 - port, to select, 15-3
 - screen layout, description, 15-2
 - screen, illustration, 15-2
 - software, accessing, 15-2
 - Communications, 15-2
 - comparative terms, description, 16-18
 - compensation
 - around acute angles, 9-21
 - left, 9-9
 - right, 9-9
 - COMPENSATION, help template menu, 7-14
 - completed program, 3-6
 - conditional statements, listed, 16-14
 - console, illustration, 2-1
 - constant surface speed. *See* CSS
 - construction geometry
 - creating, 14-20
 - using, 14-16
 - Cont (F10), 2-8
 - continuous
 - jog, 3-8
 - jog, Feed mode, 3-11
 - jog, Rapid mode, 3-11
 - move, execute, 3-12
 - contouring mode (cutter mode), (G64), 4-19
 - control M-codes
 - description, 12-2
 - functions, table, 12-2
 - conventional jog, 3-8
 - conventional jog mode, 3-11
 - coolant off, (M9), 12-2
 - coolant on, (M8), 12-2
 - coolant ready, 3-3
 - copy
 - programs, other directories, 10-10
 - programs, to floppy disk, 10-6
 - copying, program blocks, 6-11
 - corner rounding
 - cancel, modal, (G60), 4-18
 - modal, (G59 Rn), 4-16
 - corners
 - chamfering, 14-9
 - restoring, sharp, 14-14
 - rounding, 14-9
 - counterclockwise motion (G3), description, 4-6
 - create
 - new programs, 10-2
 - subdirectory, 10-11
 - Create (F2), 10-2
 - creating, basic shape, 14-10
 - CSS mode, to set, 13-13
 - current, operating mode, 3-6
 - cursor, edit screen, location, 6-2
 - cursors, 2-7
 - cutting
 - a chamfer, 13-16
 - a radius, 13-17
 - a taper, 13-15
 - feedrates in FPM, G94, 4-25
 - feedrates in FPR, G95, 4-25
 - longitudinal threads, (G83, uni-directional), 5-18
 - longitudinal threads, (G84, bi-directional), 5-20
 - radial threads, (G85, uni-directional), 5-23
 - radial threads, (G86, bi-directional), 5-26
 - threads (G83), 13-11
 - threads, input label, listed, 13-11
- ## D
- data bits, to set, 15-4
 - data control codes
 - function, table, 15-8
 - receive mode, to use, 15-8
 - send mode, to use, 15-8
 - to use, 15-8
 - data link, testing, 15-5
 - data type, to set, 15-4
 - date and time, 10-1
 - DE-9, 15-1
 - deactivate, geometry elements, 14-29
 - deactivating, shapes, 14-16
 - Decimal Points, 7-13
 - Default Feedrate, 3-9
 - Del (F4), 2-8

- delete
 - groups of programs, 10-6
 - program on another drive, 10-12
 - text, 2-8
 - Delete (F3), 10-3
 - deleting
 - a character, 6-4
 - a program block, 6-5
 - a segment, 14-15
 - a shape, 14-13
 - all geometry elements, 14-29
 - geometry elements, 14-28
 - shape files, 14-32
 - developing, part programs, 1-7
 - DIA, machine status display, 3-5
 - diameter programming, description, 1-6
 - direct transfer, variables, 16-7
 - disabled, features, off-line, 18-4
 - disclaimer, iii
 - disengage, servos, 3-7
 - disk optimizer, to access, 10-13
 - display
 - Draw adjusting, 8-8
 - Draw, fit window, 8-9
 - Draw, scale, 8-9
 - erase, 8-7, 8-10
 - zoom, 8-10
 - DISPLAY (F5), description, 8-2
 - display size
 - change, 8-11
 - half, 8-9
 - distance to go, manual screen area, 3-5
 - Draw
 - Auto mode, 8-6
 - display size, double, 8-9
 - display size, half, 8-9
 - display size, scale, 8-9
 - display, adjusting, 8-8
 - display, erase, 8-10
 - display, fit window, 8-9
 - end at, specific block, 8-8
 - erase display, 8-7
 - grid parameter, to set, 8-5
 - grid size, to set, 8-6
 - Motion mode, 8-6
 - parameters, viewing, 8-4
 - rapid moves, description, 8-5
 - real time mode, description, 8-1
 - restart, 8-7
 - run, selected block:, 8-7
 - S.Step mode, 8-6
 - screen, illustration, 8-2
 - simulation mode
 - cancel, program, 8-3
 - description, 8-1
 - execution options, listed, 8-6
 - screen, illustration, 8-2
 - status items, listed, 8-3
 - to activate, 8-2
 - to pause, 8-3
 - to restart, 8-3
 - start at, specific block, 8-8
 - starting, 8-2
 - tool compensation, description, 8-5
 - Tool Off, description, 8-4
 - Tool On, description, 8-4
 - ToolComp, description, 8-5
 - using while running programs, 11-6
 - view area, change, 8-11
 - window, zoom, 8-10
 - DRILL/TAP Help Template Menu, 7-35
 - drives, log on, 10-4
 - dwel (G4), description, 4-8
 - dwel, description, 5-28
 - DWELL, machine status display, 3-6
- ## E
- edit
 - canceling, unsaved, 6-17
 - keys, illustration, 2-2
 - program in another directory, 10-12
 - saving, 6-16
 - screen, description, 6-2
 - edit help
 - description, 7-1
 - screens, illustration, 7-2
 - soft keys, listed, 7-8
 - edited marker, description, 6-2
 - editing
 - keys, listed, 2-5
 - SHIFT soft keys, description, 6-4
 - soft keys, description, 6-3
 - editing tools
 - description, 14-4
 - selecting, 14-4
 - ellipse, G5, 4-9
 - ELSE, conditional statement, 16-14
 - emergency stop, reset, 3-7
 - End N#, parameter, 8-7
 - enlarged motion display, set the CNC, 11-6
 - equality operators, description, 16-18
 - erase, display, 8-10

Erase, parameter, 8-7
 error messages, displayed, 2-8
 establishing, position on the machine, 5-37
 E-STOP, 3-7
 key, illustration, 3-3
 no keyboard equivalent, 17-1
 exact stop check
 G-Code, formats, 4-10
 in-position mode, (G61), 4-18
 non-modal, G9, 4-10
 executing, shapes in a program, 14-30
 exit, shut down, 3-7
 exiting, off-line software, 18-1
 exiting, shape editor, 14-30
 Expand key, 16-16
 expand key, description, 6-9
 expressions
 description, 16-2
 listed, functions, 16-2

F

F2, Create, 10-2
 F3, Delete, 10-3
 families of parts, description, 16-10
 features, disabled, off-line, 18-4
 FEED, machine status display, 3-5
 Feed, move, 3-11
 feedrate
 adjustment, 3-8
 set to feed per minute, (G94), 13-12
 set to feed per revolution, (G95), 13-13
 FEEDRATE OVERRIDE, 3-9
 setting, 3-5
 switch, illustration, 3-2
 file inclusion, description, 16-19
 first block, 1-7
 first pick feedrate, to set, 5-34
 fixture offset table
 changing, using calibrate keys, 4-16
 to adjust offset, 4-15
 to change offset, 4-15
 floppy disk
 checking for lost data, 10-8
 copy programs, to, 10-6
 display, 10-4
 rename programs, 10-10
 floppy disks
 formatting, 10-8
 formats, file inclusion, 16-19
 formatting, floppy disks, 10-8

front tool, post software limits, illustration, 13-8
 functions
 description, 16-2
 listed, operators, 16-2

G

G, machine status display, 3-6
 G0, 4-2
 G0, rapid traverse, description, 4-3
 G00, perform rapid move, 13-5
 G01, perform line move, 13-6
 G02 or G03, perform arc move, 13-6
 G1, 4-2
 G1, linear interpolation, description, 4-4
 G146, protected probe positioning cycle
 defined, 5-33
 description, 5-41
 designations, 5-36
 G150, 5-37
 G150, tool probe calibration cycle
 defined, 5-33
 description, 5-38
 designations, 5-35
 G151, tool preset cycle
 defined, 5-33
 description, 5-39
 designations, 5-36
 G154, tool wear/breakage cycle
 defined, 5-33
 description, 5-40
 designations, 5-36
 G2, 4-2, 4-6
 G2, clockwise motion, description, 4-6
 G22, 4-2
 G22, set software limits, 4-11, 13-7
 G24, 4-2
 G24, maximum spindle speed, 4-12
 G28, homing the machine, 13-9
 G3, 4-2, 4-6
 G3, counterclockwise motion, description, 4-6
 G31, 4-2
 G31, probe move, 5-32, 16-13
 G33, 4-2
 G33, tread cutting, 4-13
 G4, 4-2
 G4, dwell, description, 4-8
 G40, 4-2
 G40, tool nose radius (TNR) compensation, 9-17

- G41, 4-2
- G41, left of path compensation, illustration, 9-10
- G42, 4-2
- G42, right of path compensation, illustration, 9-10
- G5, 4-2
- G5, ellipse, 4-9
- G53, 4-2, 5-37
- G53, offsets (work coordinate system select), 4-14
- G59, 4-2
- G59 En, modal, chamfering, 4-16
- G59 Rn, modal, corner rounding, 4-16
- G60, 4-2
- G60, cancel, modal, corner rounding, 4-18
- G61, 4-2
- G61, exact stop check, in-position mode, 4-18
- G61, In-Position Mode, modal, exact stop check, 4-10
- G64, 4-2
- G64, cancel, In-Position Mode, modal, exact stop check, 4-10
- G64, contouring mode (cutter mode), 4-19
- G65, 4-2
- G65, G66, G67, user macros, description, 16-11
- G65, G66, G67, using macros, 4-20
- G66, 4-2
- G67, 4-2
- G68, 4-2
- G70, 4-2
- G70, inch mode, 4-22
- G71, 4-2
- G71, mm mode, 4-22
- G72, 4-2
- G73, Z-axis turning/boring cycle, longitudinal, 5-1
- G74, 4-2
- G74, X-axis facing/boring cycle with defined profile, 5-4
- G74, Z-axis turning/boring cycle, radial, 5-1
- G76, 4-2
- G76, rough turning cut, making, 13-9
- G76, Z-axis rough turning or boring cycle, 5-7
- G77, 4-2
- G77, rough facing cut, making, 13-10
- G77, X-axis radial (facing) roughing cycle, 5-9
- G78, 4-2
- G78, pecking cycle, 5-11
- G79, 4-2
- G79, chip break cycle, 5-12
- G81, 4-2
- G81, longitudinal grooving cycle, 5-13
- G82, 4-2
- G82, radial grooving cycle, 5-15
- G83, 4-2
- G83, cutting threads, 13-11
- G83, uni-directional longitudinal threading cycle, 5-18
- G84, 4-2
- G84, bi-directional longitudinal threading cycle, 5-20
- G84, program listing, 5-22
- G85, 4-2
- G85, uni-directional cutting radial threads, 5-23
- G86, 4-2
- G86, bi-directional cutting radial threads, 5-26
- G87, 4-2
- G87, tapping a hole, 5-28
- G88, 4-2
- G88, boring cycle, 5-30
- G9, 4-2
- G9, exact stop check, non-modal, 4-10
- G9, In-Position Mode, non-modal, exact stop check, 4-10
- G90, 4-2
- G90, absolute mode, 4-22
- G90, absolute mode, activating Absolute, 13-12
- G91, 4-2
- G91, incremental mode, 4-22
- G91, incremental mode, activating, 13-12
- G92, 4-2, 4-24
- G92, presetting the axes, 4-24
- G94, 4-2
- G94, cutting feedrates in FPM, 4-25
- G94, feedrate, set to feed per minute, 13-12
- G95, 4-2
- G95, cutting feedrates in FPR, 4-25
- G95, feedrate, set to feed per revolution, 13-13
- G96, 4-2
- G96, constant surface speed, spindle, 4-26
- G96, spindle speed, set to constant surface speed, 13-13
- G97, 4-2

-
- G97, spindle speed, set to RPM, (G97), 13-14
 - G97, spindle, setting speed to constant RPM, 4-26
 - G-code
 - defined, 4-1
 - exact stop check, formats, 4-10
 - file extensions, listed, 14-31
 - tool nose radius (TNR), listed, 9-9
 - tool probe cycles, listed, 5-33
 - user macros, listed, 16-11
 - geometric shapes, complex, 16-10
 - geometry
 - circle tools, templates, 14-19
 - line tools, templates, 14-18
 - point tools, templates, listed, 14-17
 - tools, accessing, 14-16
 - geometry elements
 - activate / deactivating, 14-29
 - deleting, 14-28
 - deleting, all, 14-29
 - viewing, listing of, 14-28
 - go to a line, program listing, 6-6
 - GOTO, conditional statement, 16-14
 - GREATER THAN operator, description, 16-18
 - grid
 - parameter, to set, 8-5
 - size, to set, 8-6
- H**
- halted program, clearing, 11-5
 - HALTED/*HALTED/RUNING, program area label, 3-6
 - handshaking, 15-4
 - handwheel
 - jog mode setting, table, 3-14
 - key, illustration, 3-2
 - operation, mounted on machine, 3-14
 - operation, mounted on manual panel, 3-13
 - to operate, 3-13
 - to select, 3-13
 - hard drive, optimize, 10-13
 - Help Graphic Screen
 - illustration, 7-11
 - Help Graphic screens
 - description, 7-11
 - entry field, types, 7-13
 - to activate, 7-11
 - using, to enter program blocks, 7-13
 - Help Template Menu, illustration, 7-3
 - help templates, help template menus, 7-10
 - highlight bars, 2-7
 - hold
 - auto mode program, 11-5
 - Draw, description, 8-3
 - single-step run, execution, 11-2
 - Hold (F1), 15-7
 - HOLD key, illustration, 3-3
 - holding, transmission / receiving operations, 15-7
 - homing the machine, (G28), 13-9
 - hot keys, shape editor, description, 14-29
- I**
- I.D. (inner diameter), 5-7
 - IF, conditional statement, 16-14
 - Ignore, Draw, description, 8-5
 - importing, shapes from other programs, 14-14
 - inch mode, (G70), 4-22
 - inclusion file
 - formats, 16-19
 - requirements, 16-19
 - incremental jog mode, 3-11
 - incremental mode
 - (G91), 4-22
 - activating, (G91), 13-12
 - incremental move, execute, 3-12
 - incremental, positioning, 1-6
 - indirect transfer
 - second level, 16-8
 - variables, 16-8
 - inequality operators, description, 16-18
 - In-Position Mode
 - modal, exact stop check, (G61), 4-10
 - IN-POST, program area label, 3-6
 - input box, help template menus, 7-10
 - Ins (F3), 2-8
 - insert mode, 2-8
 - inserted characters, 2-8
 - inserting
 - blank line in program listing, 6-7
 - text to overwrite previous text, 6-4
 - text without overwriting previous text, 6-4
 - inspecting, programmed moves, 8-1
 - inspection probe type, to set, 5-34
 - install
 - keyboard, 17-1
 - machine software, 17-1
 - RS-232 cable, 15-1
-

invert probe logic, to set, 5-34

J

jog

:1, mode, 3-11

:10, mode, 3-11

:100, mode, 3-11

continuous, 3-8

continuous move, execute, 3-12

conventional, 3-8

incremental move, execute, 3-12

JOG - key, illustration, 3-3

JOG +key, illustration, 3-3

JOG key, illustration, 3-2

jog mode, changing, 3-11

jog moves, description, 3-11

joining, line segments, 14-14

K

keyboard

equivalent keypad keys, table, 17-2

to install, 17-1

keypad

equivalent keyboard keys, table, 17-2

illustration, 2-2

L

lathe tool probe cycles

description, 5-32

G-codes, designations, 5-35

macro variable, settings, 5-33

probe orientation, 5-35

system variable, settings, 5-33

left hand tool compensation, description, 9-10

left of path compensation, (G41), 9-10

left of path TNR, (G41), programming example, 9-13

LESS THAN operator, description, 16-18

limit switch, 3-7

limited access password, 17-1

line move

input labels, listed, 13-6

to perform, 13-6

line tools

description, 14-6, 14-18

template table, 14-6

templates, listed, 14-18

line, joining, segments, 14-14

linear interpolation (G1)

description, 4-4

illustration, 4-4, 4-5

LINES Help Template Menu, 7-15

link test screen data display, to change, 15-5

link, testing, 15-6

list, program in another source, 10-12

loading, program for running, 10-2

local variables, description, 16-4

logical symbols, listed, 16-17

logical terms, description, 16-17

longitudinal

grooving cycle, (G81), 5-13

Z-axis turning/boring cycle, (G73), 5-1

look ahead blocks, changing, the numbers, 9-22

LOOP - END, description, 16-15

loop counter, 3-5

LOOP, machine status display, 3-5

M

M, machine status display, 3-6

M00, program stop, 12-2

M01, optional program, 12-2

M02, program end, 12-2

M100, mirror image, 12-3

M19, orient spindle and enable C-axis mode, 12-2

M2, program end, 12-2

M3, spindle on forward, 12-2

M30, program end, 12-2

M4, spindle on reverse, 12-2

M40, 12-1

M41, 12-1

M42, 12-1

M43, 12-1

M44, 12-1

M5, spindle off, 12-2

M8, coolant on, 12-2

M9, coolant off, 12-2

M900 C, synchronize C to X and Z, 12-3

M901 C, un-synchronize C to X and Z, 12-3

M9351 X302, clear, 12-3

M9387, M-code, probe select, 16-13

M98, subprogram call, 12-2

machine

position display, manual screen area, 3-4

setup, 3-1, 3-6

software, to install, 17-1

status display area, manual screen area, 3-5

- status display, area labels, 3-5
 - macros
 - address words, listed, 4-20
 - call in main program, 4-21
 - defined, 16-6
 - program listing, 4-21
 - programming, 16-11
 - using, (G65, G66, G67), 4-20
 - Main Edit Help Menu
 - description, 7-4
 - features, descriptions, 7-5
 - illustration, 7-3
 - managing, shape files, 14-31
 - Manual Data Input. *See* MDI, *See* MDI
 - manual mode
 - jog, continuous move, 3-12
 - jog, incremental move, 3-12
 - move types, 3-8
 - screen, illustration, 3-4
 - settings, 3-7
 - manual operation, 3-1, 3-6
 - manual panel
 - illustration, 3-1
 - keys, listed, 3-2
 - LEDs, 3-3
 - manual screen, 3-4
 - MANUAL/AUTO/S.STEP, program area
 - label, 3-6
 - mark
 - all programs, 10-5
 - program, 10-5
 - marked block, description, 6-2
 - marking, program blocks, 6-10
 - maximize, program storage, 10-14
 - maximum, available feedrate, 4-25
 - maximum, memory allocated
 - description, 18-3
 - to change, 18-3
 - maximum, spindle speed, (G24), 4-12
 - M-code
 - description, 12-2
 - function, description, 12-1
 - functions, table, 12-2
 - M9387, probe select, 16-13
 - to move arm, 5-32
 - MDI
 - defined, 3-12, 11-1
 - manual mode, 3-8
 - to use, 3-13
 - memory, maximum allocated
 - description, 18-3
 - to change, 18-3
 - menu item number, help template menus, 7-10
 - menus
 - Compensation Help Template, 7-1
 - Display, pop-up, 8-8
 - Draw parameters, pop-up, 8-4
 - End N#, pop-up, 8-8
 - Help Template Menu, illustration, 7-3
 - Main Edit Help Menu, 7-3
 - pop-up, 2-7
 - Start N#, pop-up, 8-8
 - message
 - displayed, 2-8
 - error, displayed, 2-8
 - line, manual screen area, 3-4
 - priority, 2-8
 - storage, 2-8
 - mirror image, (M100), 12-3
 - miscellaneous codes. *See* M-codes
 - mm mode, (G71), 4-22
 - modal function, 3-7
 - modal G-codes
 - defined, 4-1
 - listed, table, 4-2
 - modal G-codes, help template menus, 7-10
 - modifiers
 - description, 16-1
 - listed, 16-1
 - motion mode
 - defined, 11-1
 - switch to single-step mode, 11-2
 - motion of tool, during TNR compensation, 9-20
 - MULTIPLE Help Template Menu, 7-22
- ## N
- naming, shape files, 14-31
 - Negative Signs, 7-13
 - new program, creating, 10-2
 - non-modal G-codes
 - defined, 4-1
 - listed, table, 4-2
 - NOT operator, description, 16-18
 - number of parts, counter, 11-9
 - numbering, program blocks, 6-13
- ## O
- O.D. (outer diameter), 5-7

- off-line
 - passwords, listed, 18-1
 - software
 - installation, 18-1
 - to exit, 18-1
 - Windows, to install, 18-3
 - Windows, to run, 18-3
 - OFFSET, machine status display, 3-6
 - offsets (work coordinate system select), (G53), 4-14
 - on screen instructions, help template menus, 7-10
 - opening, another program, from program listing, 6-16
 - operating mode, current, 3-6
 - operator, prompts, 2-7
 - operators, listed, functions, 16-2
 - optimize, hard drive, 10-13
 - optional program stop, (M01), 12-2
 - order of execution, codes, 12-3
 - orient spindle and enable C-axis mode, (M19), 12-2
 - orientation, probe, 5-35
 - Other, list programs, 10-4
 - OVERRIDE, machine status display, 3-6
 - overwrite, previous text, 6-4
- P**
- P/N 70000414, *4200T CNC Setup Utility Manual*, referenced, 5-32
 - page up/down, through program listing, 6-5
 - Pan, command, 8-11
 - parameters
 - description, 16-6
 - protected, to access, 18-1
 - setting and passing, 16-12
 - viewing, 8-4
 - parametric programs
 - applications, 16-10
 - description, 16-9
 - families of parts, 16-10
 - geometric shapes, complex, 16-10
 - sample program, 16-20
 - subprogramming, versus, 16-10
 - parity, to set, 15-4
 - Parms (9), description, 8-2
 - part diameter, enter in entry field, 4-16
 - part programs, developing, 1-7
 - part programs, display, 10-4
 - part zero
 - defined, 3-10
 - definition, 1-3
 - illustration, 1-3
 - location, 1-7
 - parts counter, CNC, description, 11-9
 - PARTS, machine status display, 3-6
 - passing, parameters, 16-12
 - password
 - limited access, 17-1
 - service access, 17-1
 - passwords, off-line, listed, 18-1
 - Paste, 6-12
 - pasting, copied blocks into a program listing, 6-12
 - pecking cycle, (G78), 5-11
 - pending, messages, 2-8
 - perform
 - arc move, (G02 or G03), 13-5, 13-6
 - line move, (G01), 13-6
 - rapid move, (G00), 13-5
 - Play keys, 6-15
 - playing, recorded keystrokes, 6-15
 - point tools
 - description, 14-17
 - templates, listed, 14-17
 - Point Tools, 14-17
 - pointer, using, 14-3
 - polar coordinates, description, 1-4
 - pop-up menu, illustration, 2-7
 - position, display modes, listed, 11-4
 - positioning feedrate with probing, to set, 5-34
 - Positive Value, 7-13
 - powering
 - off, 3-7
 - on, 3-6
 - presetting the axes, (G92), 4-24
 - Print, 6-12
 - print display, contents, description, 16-7
 - Print Program, 6-15
 - printer, installation, 17-1
 - printing
 - a portion of a program, 6-12
 - entire program listing, 6-15
 - programs, 10-7
 - programs, from other locations, 10-11
 - probe
 - first pick feedrate, to set, 5-34
 - G-codes, designations, 5-35
 - inspection probe type, to set, 5-34
 - invert probe logic, to set, 5-34
 - M9387, M-code, probe select, 16-13

- move, (G31), 16-13
- move, G31, 5-32
- orientation, 5-35
- parameters, to set, 5-34
- positioning feedrate with probing, to set, 5-34
- retract amount after first touch, to set, 5-34
- second pick feedrate, to set, 5-34
- tool cycles
 - description, listed, 5-37
- tool probe G-code cycle designations, 5-33
- tool probe stylus width, to set, 5-34
- tool-length offset, description, 5-32
- menus, 5-34
- program
 - .G extensions, 10-1
 - area, labels, listed, 3-6
 - area, manual screen area, 3-4
 - cancel, Draw, 8-3
 - definition, 1-1
 - display area, manual screen area, 3-4
 - halted, clearing, 11-5
 - management, 10-1
 - name, 3-6
 - name, description, 6-2
 - names, choosing, 10-2
 - parts counter, description, 11-9
 - parts counter, illustration, 11-9
 - printing, a portion of, 6-12
 - run-time timer, description, 11-9
 - sample, #1, 19-2
 - sample, #2, 19-4
 - sample, #3, 19-6
 - sample, #4, 19-8
 - sample, #5, 19-10
 - sample, #6, 19-12
 - sample, #7, 19-14
 - sample, #8, 19-16
 - sample, #9, 19-18
 - stop, (M00), 12-2
 - timer, description, 11-9
 - timer, illustration, 11-9
 - to delete, 10-3
 - to list contents, 10-3
 - viewing, 8-1
- program blocks
 - canceling, edits, 6-10
 - copying, 6-11
 - deleting, 6-5
 - displaying, 10-3
 - marking, 6-10
 - numbering, 6-13
 - re-numbering, 6-13
 - unmarking, 6-10
- program directory
 - description, 10-1
 - display, changing, 10-1
 - illustration, 10-1
- program editor
 - activating, 6-1
 - activating, from Draw Graphics, 6-1
 - activating, from Manual Screen, 6-1
 - activating, from Program Directory, 6-1
- program end
 - M02, 12-2
 - M2, 12-2
 - M30, 12-2
- program listing
 - description, 6-2
 - go to a line, 6-6
 - including, comments, 6-17
 - inserting, blank line, 6-7
 - opening, another program, from, 6-16
 - page up/down, through, 6-5
 - pasting, copied blocks into, 6-12
 - printing, entire, 6-15
 - reading, into another program, 6-14
 - scrolling, through, 6-10
 - searching, for selected text, 6-7
- PROGRAM, program area label, 3-6
- programmed, hold, 3-6
- programming
 - angular motion, example, 4-5
 - block separators, description, 16-1
 - boring cycle, (G88), 5-30
 - C-axis, description, 12-3
 - chip break cycle, (G79), 5-12
 - concepts, 1-1
 - diameter, description, 1-6
 - examples, tool nose radius (TNR), 9-12
 - expressions, description, 16-2
 - expressions, listed, 16-2
 - functions, description, 16-2
 - functions, listed, 16-2
 - LINE moves in Edit Help, 7-38
 - longitudinal grooving cycle, (G81), 5-13
 - macros, 16-11
 - modal G-codes in Edit Help, 7-37
 - modifiers, listed, 16-1
 - pecking cycle, (G78), 5-11

radial grooving cycle, (G82), 5-15
 radius, description, 1-6
 single moves, 3-12
 system variables, listed, 16-3
 T codes, 9-5
 user variables
 block skip, description, 16-5
 common (global), description, 16-4
 description, 16-4
 local, description, 16-4
 read only, description, 16-5
 word keys, 2-5
 X-axis, facing/boring cycle with defined profile, (G74), 5-4
 X-axis, radial (facing) roughing cycle, (G77), 5-9
 Z-axis turning/boring cycle
 longitudinal, (G73), 5-1
 Z-axis, rough turning or boring cycle, (G76), 5-7
 programs
 .G extensions, 10-1
 block separators, description, 16-1
 copy, other directories, 10-10
 copy, to floppy disk, 10-6
 create, new program, 10-2
 delete, groups, 10-6
 delete, on another drive, 10-12
 display, soft keys listed, 10-3
 edit, in another directory, 10-12
 fragment, checking disk, 10-8
 list, in another source, 10-12
 loading, 10-2
 mark, 10-5
 mark all, 10-5
 names, choosing, 10-2
 parametric, description, 16-9
 parametric, sample, 16-20
 printing, 10-7
 printing, from other locations, 10-11
 receiving, 15-6
 rename, 10-7, 10-10
 restore, 10-6
 running, 11-1
 running, one step at a time, 11-1
 select, for editing, 10-3
 sending, description, 15-6
 storage, maximize, 10-14
 undelete, 10-6
 unmark, 10-5
 unmark all, 10-5

using real time Draw, while running programs, 11-6
 projecting, line segments, 14-14
 prompts, 3-4
 prompts, operator, 2-7
 protected parameters, to access, 18-1
 protected probe positioning cycle, G154
 defined, 5-33
 description, 5-41
 designations, 5-36

R

RAD/CHAMFER Help Template Menu, 7-20
 radial grooving cycle, (G82), 5-15
 radial, Z-axis turning/boring cycle, (G74), 5-1
 radius programming, description, 1-6
 radius, cutting, 13-17
 rapid move
 input, 13-5
 viewing, 8-5
 rapid traverse (G0), description, 4-3
 Rapid, mode, 3-11
 read only variables, description, 16-5
 reading, program into another program
 listing, 6-14
 real time Draw, using while running programs, 11-6
 rear tool, post software limits, illustration, 13-8
 receive, 15-1
 receive area, to clear, 15-6
 receive mode, data control codes, to use, 15-8
 receiving
 display, to set, 15-7
 operations, to hold, 15-7
 program, 15-6
 programs, through RS-232 communications, 11-10
 recording, keystrokes, 6-14
 remaining, seconds, 3-6
 rename
 program, 10-7
 programs, 10-10
 Renum, 6-13
 re-numbering, program blocks, 6-13
 repeating, a command or key, 6-15
 replacing, text with new text, 6-8
 requirements, file inclusion, 16-19
 resetting, the servos, 3-7

-
- restart, Draw, 8-7
 - restart, Draw, description, 8-3
 - restarting, program on hold, 11-2
 - restore, programs, 10-6
 - restoring, a block, 6-8
 - restoring, sharp corners, 14-14
 - Resume (F2), 15-7
 - retract amount after first touch, to set, 5-34
 - reversing, an arc's direction, 14-13
 - reviewing, messages, 2-8
 - right of path compensation, (G42), 9-10
 - right of path TNR, (G41), programming
 - example, 9-15
 - rough cut, input labels, listed, 13-9
 - rough facing cut, input labels, listed, 13-10
 - rough facing cut, making, (G77), 13-10
 - rough turning cut, making (G76), 13-9
 - rounding, corners, 14-9
 - RPM formula, spindle, 4-26
 - RPM mode, to set, 13-14
 - RPM, machine status display, 3-5
 - RS-232 cable, 15-1
 - RS-232 communications, 11-10
 - Run, parameter, 8-7
 - running
 - program, one step at a time, 11-1
 - programs, 11-1
- S**
- sample program
 - #1, 19-2
 - #2, 19-4
 - #3, 19-6
 - #4, 19-8
 - #5, 19-10
 - #6, 19-12
 - #7, 19-14
 - #8, 19-16
 - #9, 19-18
 - saving, edits, 6-16
 - scale, Draw, display size, 8-9
 - SCI mode
 - canceling, command, 13-4
 - canceling, running command, 13-4
 - clearing, entry field, 13-4
 - clearing, messages, 13-4
 - commanding moves, 13-5
 - deactivating, 13-4
 - description, 13-1
 - exiting, 13-5
 - SHIFT soft keys, listed, 13-3
 - soft keys, listed, 13-3
 - to activate, 13-1
 - S-code, description, 12-1
 - S-code, function, description, 12-1
 - screen clutter, eliminate, 8-5
 - screen saver, 2-7
 - screens
 - Auto, illustration, 11-4
 - Draw simulation mode, 8-2
 - edit, illustration, 6-2
 - Enlarged Position Display, illustration, 11-6
 - Help Graphic Screen, illustration, 7-11
 - manual mode, 3-4
 - RS-232 communication, illustration, 15-2
 - shape editor, illustration, 14-1
 - Single-Step/Motion, illustration, 11-2
 - system information, 10-14
 - system information, illustration, 10-9
 - Test Link, 15-5
 - scrolling, through program listing, 6-10
 - SEARCH
 - to select, auto mode, starting block, 11-5
 - using to select, single-step mode, starting block, 11-3
 - searching, program listing for selected text, 6-7
 - second pick feedrate, to set, 5-34
 - seconds, remaining, 3-6
 - Select point definition, 14-17
 - selecting
 - axis, 3-11
 - communication port, 15-3
 - editing tools, 14-4
 - program for editing, 10-3
 - program for utilities, 10-3
 - send mode, data control codes, to use, 15-8
 - sending programs, through RS-232
 - communications, 11-10
 - sending, program, 15-6
 - service access password, 17-1
 - SERVO RESET key, illustration, 3-2
 - servos
 - activating, 3-7
 - disengage, 3-7
 - reset, 3-7
 - set software limits, (G22), 4-11
 - setting
 - baud rate, 15-3
 - data bits, 15-4
 - data type, 15-4
-

- parameters, 16-12
- parity, 15-4
- receiving display, 15-7
- software, 15-4
- stop bits, 15-4
- test link display modes, 15-5
- transmission display, 15-7
- variables, 16-7
- shape
 - creating, basic, 14-10
 - deleting, 14-13
 - deleting, a segment, 14-15
 - editing tools, illustration, 14-5
 - elements, illustration, 14-3
 - segment details, viewing, listing of, 14-15
 - template, lettering convention, 14-4
 - terminology, 14-3
- shape editor
 - changing, view, 14-15
 - description, 14-1
 - example, 14-21
 - exiting, 14-30
 - hot keys, description, 14-29
 - hot keys, listed, 14-30
 - screen, illustration, 14-1
 - settings, listed, 14-32
 - to activate, 14-2
- shape files
 - deleting, 14-32
 - managing, 14-31
 - naming, 14-31
 - viewing, in program directory, 14-31
- shapes
 - activating, 14-16
 - backing up, 14-32
 - deactivating, 14-16
 - executing, in a program, 14-30
 - importing, from other programs, 14-14
 - verifying, 14-33
- shutting down, CNC, 3-7
- Simplified Command Interface mode. See SCI mode
- simulating, programs, 8-1
- simulation size
 - change, 8-11
 - double, 8-9
 - half, 8-9
- single-step mode
 - auto mode, switch to, 11-3
 - defined, 11-1
 - MOTION mode, 11-1
 - motion mode, switch to, 11-2
 - program, to run, 11-2
 - S.STEP mode, 11-1
 - starting block, select
 - using arrows, 11-3
 - using SEARCH, 11-3
- single-step run, hold execution, 11-2
- single-step/motion screen, 11-2
- slave mode, description, 13-14
- soft keys
 - display program, listed, 10-3
 - edit help, listed, 7-8
 - editing keys, description, 6-3
 - help template menus, 7-10
 - labels, description, 2-6, 6-2
 - manual mode, listed, 3-5
 - SCI mode, listed, 13-3
 - SHIFT editing keys, description, 6-4
 - SHIFT, SCI mode, listed, 13-3
 - teach mode, listed, 11-7
- software
 - basics, 2-7
 - basics, description, 2-6
 - exiting, off-line, 18-1
 - parameters, 15-1
 - selection screen, returning to, 13-4
 - setting, 15-4
- software limits
 - front tooling, illustration, 4-12
 - rear tooling, illustration, 4-12
 - setting, (G22), 13-7
 - to disable, 13-8
 - to enable, 13-8
- spindle
 - constant surface speed, (G96), 4-26
 - off, (M5), 12-2
 - on forward, (M3), 12-2
 - on reverse, (M4), 12-2
 - override, setting, 3-5
 - RPM formula, 4-26
 - setting speed to constant RPM, (G97), 4-26
 - speed control, description, 12-1
 - speed, set to constant surface speed, (G96), 13-13
 - speed, set to RPM, (G97), 13-14
- SPINDLE FORWARD key, illustration, 3-2
- SPINDLE OFF key, illustration, 3-2
- SPINDLE OVERRIDE switch, illustration, 3-2
- SPINDLE REVERSE key, illustration, 3-2
- START key, illustration, 3-2

- Start N#, parameter, 8-7
 - starting block, select
 - auto mode, using SEARCH, 11-5
 - single-step mode, using SEARCH, 11-3
 - using arrow keys, 11-5
 - using arrows, 11-3
 - starting, Draw, 8-2
 - statements, abbreviating, 16-16
 - status items, listed, 8-3
 - stop bits, to set, 15-4
 - storing, the results of computation, 16-9
 - subdirectory, creating, 10-11
 - subprogram
 - call, (M98), 12-2
 - file inclusion, description, 16-19
 - synchronize C to X and Z, (M900 C), 12-3
 - system
 - information screen, illustration, 10-14
 - information, displaying, 10-9
 - variables, listed, 16-3
- T**
- T codes, programming, 9-5
 - tapper, cutting, 13-15
 - tapping a hole, (G87), 5-28
 - target position display, manual screen area, 3-4
 - teach mode
 - data, inputting, 11-7
 - description, 11-7, 11-8
 - soft keys, listed, 11-7
 - to cancel, 11-8
 - to start, 11-7
 - templates
 - circle tools, 14-19
 - lettering convention, 14-4
 - line tools, 14-18
 - point tools, listed, 14-17
 - test link display modes, to set, 15-5
 - test link screen, to activate, 15-5
 - testing, data link, 15-5
 - testing, link, 15-6
 - text, deleting, 2-8
 - THEN, conditional statement, 16-14
 - THREADING Help Template Menu, 7-26
 - threading, types of, listed, 5-17
 - timer, CNC, description, 11-9
 - TIMER, machine status display, 3-6
 - TLO. See tool-length offset
 - TLO, defined, 5-32
 - tool, 3-5
 - activating, 13-5
 - changing, 13-5
 - compensation, Draw, description, 8-5
 - management, 9-1
 - tool location code
 - key, illustration, 9-11
 - selecting, 9-11
 - to enter, in tool page, 9-11
 - tool nose radius (TNR)
 - canceling, 9-17
 - compensation, (G40), 9-17
 - compensation, changing in program, 9-18
 - compensation, choose correct, 9-18
 - compensation, description, 9-8
 - compensation, motion of tool, 9-20
 - G-codes, listed, 9-9
 - illustration, 9-8
 - programming, examples, 9-12
 - tool number
 - to activate, in manual mode, 9-8
 - to deactivate, in manual mode, 9-8
 - to find, 9-3
 - Tool Off, Draw, description, 8-4
 - tool offset, address words, listed, 9-7
 - tool offset, modification, by program, 9-7
 - Tool On, Draw, description, 8-4
 - tool page
 - all values, to clear, 9-4
 - definition, 9-1
 - row, to clear, 9-4
 - single value, to adjust, 9-3
 - specific tool number, to find, 9-3
 - to activate, 9-1
 - to enter, tool location code, 9-11
 - tool-length offset, 9-4
 - using, 9-1
 - value from a column, to clear, 9-4
 - values, entering, 9-3
 - tool preset cycle, G151
 - defined, 5-33
 - description, 5-39
 - designations, 5-36
 - tool probe calibration cycle, G150
 - defined, 5-33
 - description, 5-38
 - designations, 5-35
 - tool probe cycles
 - description, listed, 5-37
 - G-code cycle designations, 5-33
 - tool probe stylus width, to set, 5-34
 - tool wear/breakage cycle, G152

defined, 5-33
description, 5-40
designations, 5-36
TOOL, machine status display, 3-5
ToolComp, Draw, description, 8-5
tool-length offset
description, 9-4
entering, manually, 9-6
entering, using calibration, 9-6
probe, description, 5-32
setting, 9-5
total free user space, 10-14
total space, available, system, 10-14
transferring, variables, 16-7
transmission display, to set, 15-7
transmission operations, to hold, 15-7
transmit, 15-1
transmit area, to clear, 15-6
tread cutting, (G33), 4-13
truth table, logical symbols, listed, 16-17
typeover mode, 2-8
types of threading, listed, 5-17

U

unconditional LOOP repeat, description, 16-15
Undelete Block, 6-8
undelete, programs, 10-6
uni-directional
cutting radial threads, (G85), 5-23
longitudinal threading cycle, (G83), 5-18
unmark
a program, 10-5
all programs, 10-5
unmarking, program blocks, 6-10
un-synchronize C to X and Z, (M901 C), 12-3
Use, Draw, description, 8-5
USER listing, 10-1
user macro G-codes, listed, 16-11
user macros (G65, G66, G67), description, 16-11
user variables
block skip, description, 16-5
common (global), description, 16-4
description, 16-4
local, description, 16-4
read only, description, 16-5
using
construction geometry, 14-16
data control codes, 15-8

the pointer, 14-3

V

variable registers, description, 16-6
variable, contents, description, 16-7
variables
direct transfer, 16-7
indirect transfer, 16-8
indirect transfer, second level, 16-8
setting, 16-7
verifying, shapes, 14-33
viewing
feed moves, 8-4
listing, of geometry elements, 14-28
listing, of shape segment details, 14-15
programs, 8-1
rapid moves, 8-4
shape files, in program directory, 14-31

W

warranty, iii
wear offsets
address words, listed, 9-7
description, 9-22
setting adjustment, running program, 9-23
WHILE, conditional statement, 16-14
window, sized, 8-10
Windows, off-line software
to install, 18-3
to run, 18-3
word keys, programming, 2-5
Write, 6-13
writing, blocks to another program, 6-13

X

X-axis
description, 1-3
direction of motion, illustration, 1-3
facing/boring cycle with defined profile, (G74), 5-4
radial (facing) roughing cycle, (G77), 5-9
Xoff, 15-4
XOFF, machine status display, 3-5
Xon, 15-4

Z

Z-axis
description, 1-2
rough turning or boring cycle, (G76), 5-7

turning/boring cycle
 longitudinal, (G73), 5-1
ZOFF, machine status display, 3-5
zoom, Draw window, 8-10
zoom, in or out, 8-1

ANILAM

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.**

France

ANILAM S.A.R.L.

2 Ave de la Cristallerie
B.P. 68-92316

Serves Cedex, France

☎ +33-1-46290061

☎ (FAX) +33-1-45072402

✉ courrier@acu-rite.fr

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, ROC

☎ +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