WinMax Lathe
NC Programming

The information in this document is subject to change without notice and does not represent a commitment on the part of Hurco Companies, Inc. (Hurco). The software described in this document is furnished under the License Agreement to customers. It is against the law to copy the software on any medium except as specifically allowed in the license agreement. The purchaser may make copies of the software for backup purposes.

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

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

Patents:U.S. Patents B14,477,754; 5,453,933; Canadian Patent 1,102,434; Japanese Patents 1,649,006 and 1,375,124; other Patents pending.

Hurco, Max, Ultimax, and WinMax are Registered Trademarks of Hurco Companies, Inc.

AutoCAD, Autodesk, and DXF are registered trademarks of Autodesk, Inc.

MS-DOS, Microsoft, and Windows are registered trademarks of Microsoft Corporation.

Many of the designations used by manufacturers and sellers to distinguish their products are claimed as trademarks. Hurco has listed here all trademarks of which it is aware. For more information about Hurco products and services, contact:

Hurco Companies, Inc.
One Technology Way
P.O. Box 68180
Indianapolis, IN 46268-0180

For Hurco subsidiary contact information, go to Hurco's Web site:
www.hurco.com

Table of Contents

DOCUMENTATION CONVENTIONS

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

Sample Screens

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

Softkeys

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

Screen Areas

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

Data Entry

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

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

Prompts and Error/Status Area

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

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

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

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

Status Bar

The status bar contains

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

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

Console Buttons and Keys

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

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

Using the Touchscreen

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

Icons

This manual may contain the following icons:

Caution/Warning

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

Important

Ensures proper operation of the machine and control.

Troubleshooting

Steps that can be taken to solve potential problems.

Hints and Tricks

Useful suggestions that show creative uses of the WinMax features.

Where can we go from here?

Lists several possible options the operator can take.

Programming and Operation Information

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

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

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

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

Using the On-screen Help

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

- Buttons in the upper left-hand corner of the Help screen are used to move through Help topics and print screens.
    - Use the **Hide** button to hide the navigation pane.
    - Use the **Back** button to return to the previous Help screen.
    - Use the **Print** button to print the current displayed Help topic, if a printer is attached and configured. See *Printing the Programming Manuals, on page - xiii* for more information about printing.
- Use the arrow buttons to move between pages within a Help topic and to move through topics.
- Use the **Contents** tab for a list of information sorted by subject:
    1. Select the "+" to expand the topic and view sub-topics.
    2. Select the topic to display it.
- Use the **Index** tab to show the Help index:
    1. Quickly scroll to an index topic by typing the topic in the box at the top of the index.
    2. Select a topic and the Display button to view the topic.

- Use the **Search** tab to search the Help for a word or phrase:
  1. Type the search word(s) into the text box at the top of the pane.
  2. Select the List Topics button. A list of topics that contain the search word(s) is displayed.
  3. Select a topic and the Display button to view that topic.
- Use the **Favorites** tab to save Help topics for quick access:
  1. Select the Add button at the bottom of the pane to add the current topic.
  2. Select a topic from the Favorites list, and select the Display button to view it.
     - Select a topic from the Favorites list, and select the Remove button to remove it from the list.

Printing the Programming Manuals

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

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

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

   ⇨ The SHOW ALL FILE TYPES field in User Interface Settings must be set to YES (default is NO) in order to see the PDF files in the directory. Access the SHOW ALL FILE TYPES field in Auxiliary Mode, Utilities/ User Preferences/ User Interface Settings.
4. Highlight the PDF file(s) in the right-hand pane, and select the COPY *F2* softkey.
5. Ensure that your media is loaded (either a floppy disk in the disk drive or a USB memory device in the USB port), and navigate to the proper location in the left-hand pane of the DISK OPERATIONS screen (either the floppy drive A: or the USB port E:). Highlight the desired location.
6. Place the cursor in the right-hand pane and select the PASTE *F3* softkey to paste the PDF file(s) to the desired location.

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

OVERVIEW

This documentation describes the BNC (Basic Numerical Control) and ISNC (Industry Standard Numerical Control) Part Programming Editor portion of the CNC software as it is used on the machine tool console.

NC programming is designed to use as much of the Conversational system as possible. As a result, the screen layout is the same in both the NC and the Conversational systems. This allows a smooth transition between the two editors.

The primary difference between Conversational and NC programming is the program editors. The NC programming uses standard G and M codes; whereas, Conversational programming uses a supported programming language, such as plain English.

⇨ The CNC software can read NC files from the serial port directly into dynamic memory. NC files can be serially loaded to the hard disk.

NC part programs can be created using the CNC on the machine tool or off-line CNC programming software running on a personal computer. NC programs cannot be converted to Conversational programs, nor can NC programs be converted automatically to any other NC format.

Please refer to Getting Started with WinMax Lathe, *NC Settings, on page 2 - 17* for information about default NC Settings.

NC Part Program Components

NC programs are a series of characters and words that form program blocks. These program blocks tell the machine tool how and where to move. The operator needs to understand the basic program structure and the types of codes in order to create, edit, and run a program successfully.

Program Start

All NC programs begin with a "%" (percent) character. When a percent character is received, the control starts to accept, check, and load blocks into its memory. If you are creating a new part program at the control, the percent character is automatically inserted at the beginning of the program Address Characters

In the following sample screen, the **S [%** indicates the beginning of the program and the **E] %** line indicates the end of the program. Both characters appear automatically when beginning a new NC program.

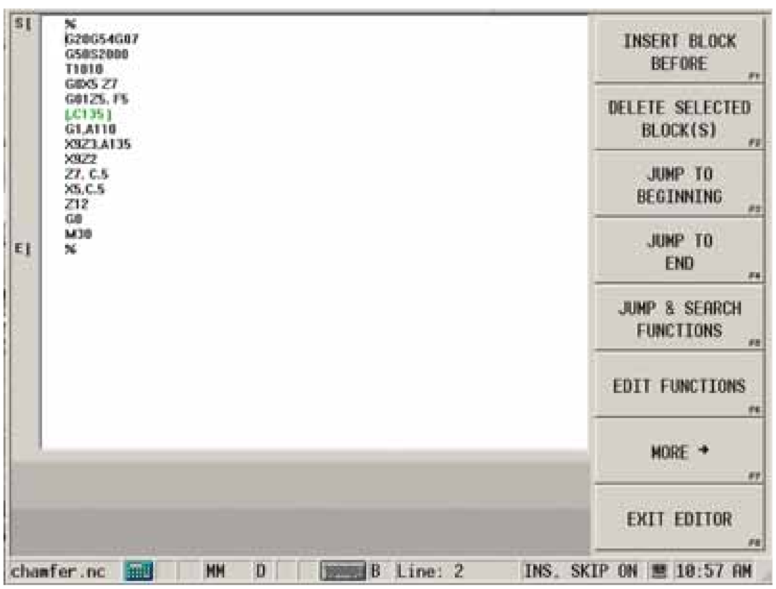


Figure 1–1. NC Editor Screen

Sequence Number

A sequence number serves as a block label; it has no other significance within the part program except being required with GOTOs in the NCPP option and the M99 jump command. Sequence numbers are often used to mark the beginning of milling sequences so you can restart at a given sequence number or recall specific operations within the program. The maximum sequence number is 9999999.

When programming on an off-line system, sequence numbers should be used sparingly. Sequence numbers (N words) are optional in the NC Editor, and they are useful in programs sent over the RS-232 link. However, the absence of sequence numbers permits faster processing (loading, syntax checking, and parsing) of the part program and can result in improved part program execution. In addition, omission of these numbers increases the amount of the program that can fit into memory.

⇨ If you request renumbering of part program sequence numbers, any sequence numbers in GOTO statements will not be updated. In general, you will not want to renumber part programs that use GOTO statements. Refer to *Renumbering and Tagging Menu, on page 1 - 22* for renumbering information.

Address Characters

An address character is the first character of a word in a program block. The following is a list of the address characters recognized by this system:

/	**Block Skip Command**. Specify blocks of code that are skipped when this control feature is enabled. The specified block, or portion of a block, begins with a “/” (forward slash). Refer to *Block Skip Code, on page 1 - 6* for more information about Block Skip.
( )	**Comment Command**. Comment statements provide information about the part program.You can insert comment statements at the end of any part program block by enclosing the comment within parentheses. You may make an entire block of code a comment statement by enclosing it within parentheses.The Comment Command characters are used to delimit comments.
F	**Feedrate**. Sets the modal feedrate for cutting moves. Pitch. Sets the pitch for threading.
G	**Preparatory Functions**. G codes have two basic functions: • specify a modal condition. For example, G20 establishes Inch mode, G21 establishes Millimeter mode, G90 establishes Absolute mode and G91 establishes Incremental mode. Programming G20 Inch mode and G90 Absolute mode in the first block of a part program tells the control to remain in inch mode until G21 Millimeter mode is programmed, and to remain in Absolute mode until G91 Incremental mode is programmed. • specify the type of tool motion (example: G00 programs a rapid move, G01 programs a linear feed move, G02 and G03 program circular moves).
I	**X-axis Center / Offset coordinate** for programming geometric information needed to determine the endpont of a motion command.
K	**Z-axis Center / Offset coordinate** for programming geometric information needed to determine the endpoint of a motion command.

Address Characters (continued)

M	**Miscellaneous Functions**. M Codes perform miscellaneous functions such as turning on the spindle, turning on coolant, specifying a program stop, or specifying an end of program.
N	**Sequence Number**. Each block may contain a nine-digit integer N Number. N Numbers are not required; however, programmers can use them as sequence numbers to order the part program. It is not required for the numbers to appear in order within the part program.
P	**Thread Angle**. Starting thread angle for threading, G33 with P Value.
R	**Radius Programming**. Generally used for programming the Radius. Can also be used with G73, Pattern Repeating for programming the number of repetitions. Can also be used with G74 G75 for programming the retract distance. See G Code descriptions for parameter definitions.
S	S**pindle Speed Function**. Sets the spindle speed for two-axis machines (TM and TMX). If using the S command on machines with multiple spindles (including live-tooling), the S command refers to ALL spindles. • S:M19 orients all spindles on a multiple spindle machine. • S1:M19 orients only the main spindle. • S:S2000 ramps multiple spindles to 2000 RPM. • S1:S2000 commands only the main spindle.
S1	**Spindle Speed Function**. Sets the spindle speed for main spindle for live-tooling machines. Please refer to the **S** address character definition above for examples.
S2	**Spindle Speed Function**. Sets the spindle speed for live-tooling spindle 2. Please refer to the **S** address character definition above for examples.
S3	**Spindle Speed Function**. Sets the spindle speed for sub-spindle. Please refer to the **S** address character definition above for examples.
T	**Tool Select**. Identifies the active tool and activates the offsets for the tool.
X	**Primary X Motion Dimension, Dwell Time coordinat**e for programming geometric information needed to determine the endpoint of a motion command.
Z	**Primary Z Motion Dimension coordinate** for programming geometric information needed to determine the endpoint of a motion command.
C	**Primary C Motion Dimension** coordinate for programming geometric information needed to determine the endpoint of a motion command.This code is used for Live Tooling only.
W	**Primary W Motion Dimension** coordinate for programming geometric information needed to determine the endpoint of a motion command.
Y	**Primary Y Motion Dimension** coordinate for programming geometric information needed to determine the endpoint of a motion command.

Table 1–1. Address Characters

Special Characters

Special characters are ASCII characters within a file which have special meaning to the system and cannot be edited. The following special characters are recognized by the NC software:

- **% (Beginning/End of tape)**—signals the system that all of the following characters are part of the program. The system automatically adds this character to the beginning of a new program. You can also include the % character to signal the End of Tape.
- **E. of tape (EOT)** (optional)—signals the NC system that no more legal program characters follow. This character is optional to provide compatibility with existing programs that include EOT characters at the end.
- **[CR] (Carriage Return)**—signals the End of a Program Block.
- **[CRLF] (Carriage Return/Line Feed Pair)**—signals the End of a Program Block (identical to [CR]).

⇨ [CRLF] is not shown when the program is viewed in the NC Editor.

Words

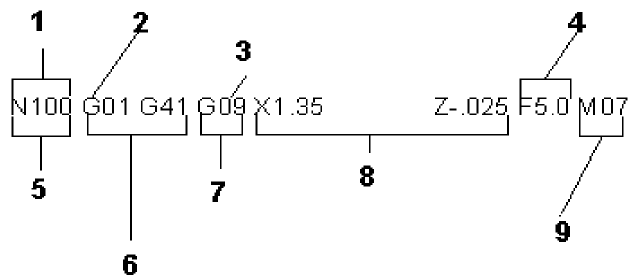
A word is a group of alphanumeric characters. The first character of a word is an address character—a letter such as M or G. The address character is followed by a signed or unsigned numeric value. Some sample NC words are “X-.03” and “G00.” One word or group of words form a program block.

Block

A block is a group of words terminated by the end-of-block character. Each block within a part program must be terminated with either a carriage return [CR] or a carriage return line feed [CRLF].

When changing the cutting type (for example, hole drilling or profile turning) for a data block, a different data block must be created to describe the new cutting type. When a different tool is needed in a program, a new data block must be created to describe the actions of that tool.

The following illustration shows a typical NC block and its components:



1	Word
2	Address character
3	Numeric indicator
4	Feedrate word
5	Sequence number
6	Modal preparatory functions
7	One-shot preparatory functions
8	Dimension words
9	Miscellaneous function

Figure 1–2. Typical NC Block

Block Skip Code

Block Skip will skip specified blocks of code when turned on. To skip a block of code, place a slash (/) at the front of the block.

```
N120 G00 Z4 (rapid to Z)
/N125 G42   (cutter compensation right, skip if block skip on)
```

Please refer to *NC Editor Settings Menu, on page 1 - 26* for information about enabling the Block Skip state.

The program processes blocks before they appear on the screen. If you change block skip mode while a part program is running, the control may have already processed blocks marked to be skipped. The control will not go back and reprocess these blocks.

⇨ When programming with **M200**, program BNC M200 - Block Skip Synchronization or ISNC Block Skip Sync (ISNC M200) so it can be activated when the tool is off the part. Using this code when in a cutting mode may cause the tool to leave dwell marks.

Off-line Part Program Formats

This section describes how an NC post processor part program code should be configured on an off-line CAM system. The order in which the control processes a block of code (block code processing) and the data format for each code are important to a post processor.

Block Code Processing

A block of code can contain 0 to 10 G codes and 0 to 10 M codes. The table below shows the order in which the control will process a block of part program code. The order in which each entry appears within a block of code does not affect the order of processing, except for block delete codes, which are processed at the point they appear in a block.

Order for Processing Blocks of Code

Block	Description
Block Delete Codes	Place a slash at the front of a block to skip it during processing. Block Delete Codes are processed at the point they appear in a block. Example— **/N125 G42 (cutter compensation right, skip if block del on)**
1. N Numbers	N Numbers can contain up to 9 digits. Example— **N123456789**
2. Comment Statements	Enclose comments in parentheses. Example— **(this is an example of a Comment Statement)**
3. Specific G Codes	Refer to *Basic NC G Codes, on page 2 - 1* or *ISNC G Codes, on page 2 - 1* for a list of G Codes and their descriptions.
4. Feedrate Override	Choose between M48 or M49.
5. Beginning of Block M Codes	Choose between M07 and M08.
6. Tool (T Code) Commands	Tool motion does not occur unless axis motion is programmed in the block containing the T Code, or until motion is programmed in a following block. **T0303** **M03 S1200** T00 cancels tool offsets.

Order for Processing Blocks of Code (continued)

Block	Description
7. After Tool Change, Before Motion M Codes	Choose between M3 and M4.
8. Spindle Speed (S Code) Command	Set the spindle speed (rpm and css) using an S code. Example— **G92 S5000 (prevent the spindle from exceeding 5000 rpm)**
9. Dwell Command	Enter **G04**. Refer to BNC G04 - Dwell or ISNC G04 - Dwell for information about this code.
10. Feedrate (F code) or Dwell	Choose between G94 and G95 to set up the feedrate values. Dwell commands programmed with G94 or G95 are interpreted as the number of seconds or spindle revolutions to dwell.
11. Motion Commands	Choose among G00, G01, G02, G03, and G33.
12. Exact Stop	Enter G09.
13. Spindle Stop	Enter M5 to stop spindle motion.
14. End of Block System M Codes	Choose among M0, M1, M2, and M30.

Table 1–2. Order of Block Code Processing

Block Data Formats

The table below shows the format (ranges) for the different types of blocks of part program code.

Entry	Description	Format (range)
G	G Code	5 (0-65535)
M	M (Miscellaneous) Code	5 (0-65535)
S	Spindle Speed (rpm) and (css)	9 (0 - 2 x 10^9) (no practical limit)
T	Tool Code + Offset T00 cancels tool offsets	0 (0-999999999)
N	N Numbers	9 (0-999999999)
F	G94 FPM Mode G95 FPR Mode	Refer to BNC G20 - Inch Mode (default) and G21 - Millimeter Mode or ISNC G20 - Inch Mode (default) and G21 - Millimeter Mode for these formats.
F	Dwell in G94 fpm mode Dwell in G95 fpr mode	3.4 seconds (0.0001 - 999.9999) 3.4 revs (0.0001 - 999.9999)

Table 1–3. Block Data Formats

⇨ Where decimals are included in the format, the number on the left of the decimal (3) represents the number of places to the left and the number on the right (4) of the decimal represents the number of places to the right of the decimal.

3.4 = nnn.nnnn

Default M and G Codes

Upon power up, control reset, initial entry into the NC Editor, or after erasing a program, the system presets these M codes as defaults:

M05	Spindle Off
M09	Both Coolant Systems Off

The system also presets certain G codes as the default active codes. The default G codes are highlighted in the BNC *G Code Table, on page 2 - 4* and the ISNC *G Code Table, on page 2 - 3*.

The system uses the units specified when the NC Editor is selected, not the G codes, for graphics display and running the part program.

Navigation

To move the cursor from a block to the beginning of the next block, press the down arrow (↓). Use the right/advance arrow (→) and the left/back arrow (←) to move the cursor within a block. Use the Enter key to move the cursor between words and blocks.

To move to the beginning of the current block, press the Home key or the up arrow (↑). If the cursor is already at the beginning of the block, pressing the up arrow moves the cursor to the beginning of the last word in the previous block.

To move from a word to the beginning of the next word, press the Enter key. If the cursor is at the end of the current block when the Enter key is pressed, the editor automatically presents the next legal address character. To move from one character to the next, press the right arrow. If the cursor is at the end of the current block, the cursor wraps around to the beginning of the block.

To move from one character to the preceding character, press the left arrow. If the cursor is at the start of the current block, it wraps around to the end of the current block.

Delete characters or words from a block using these methods:

- To delete numeric data, position the cursor on the number and press the Delete key.
- To erase the entire word, position the cursor on the address character and then press the left arrow or the Delete key. The entire word is removed since numeric data is not allowed in an NC program without an address character to introduce it.

NC Editor

The NC Editor contains information about the program. You can view the NC Editor by either opening an existing NC program or creating a new NC program and accessing the Part Programming screen.

NC Editor provides a wide range of tools to review, create, and modify NC part programs. The editor has the following features:

- Several ways of selecting the code, such as by dragging the pointing device across the screen, and by using keyboard keys (if available).
- Copying/Pasting/Cutting of a block of selected code, including to applications outside WinMax.
- Undo/Redo functionality.
- Unlimited number of tags.
- Real-time syntax check. Incorrect syntax is indicated by showing the incorrect text in red. Comments are shown in a user-defined color (green in this example):

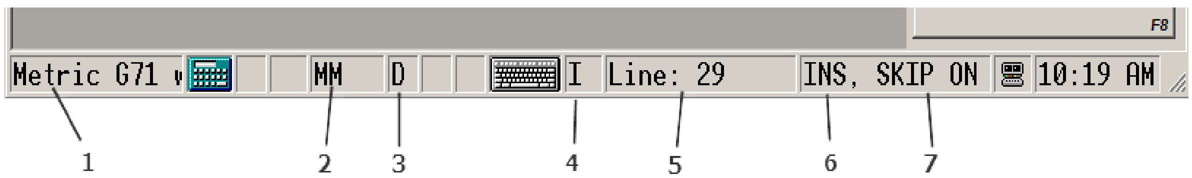
```
N9 Z-44.45
N10 X101.6
G1 G40 X104.14 Z2.54
M9
N65[#1]
(FINISH WITH A DIFFERENT TOOL)
G91 G28 U0. W0.
G90
N75GE]
T1212
G97 S2000 M3
M8
```

See *NC Editor Settings, on page 1 - 27* for information about the comment color.

- Real-time indicators of the meanings of the G codes and M codes. Place the cursor on the G code or M code and the definition is displayed in the prompt area of the screen.
- Simplified and friendlier access to common editing tasks, such as jumping and searching operations.
- Keyboard shortcuts:
  - Ctrl + Home—jump to the beginning of program.
  - Ctrl + End—jump to the end of program.
  - Ctrl + C—copy selected text to the clipboard.
  - Ctrl + F—opens a Find window. Enter text to find in the current program.
  - Ctrl + G—opens a Goto window. Enter a block number to navigate to that block.
  - Ctrl + H—opens a Find and Replace window.
  - Ctrl + V—paste the copied or cut text

- Ctrl + X—cut the selected text from screen and copy to clipboard
- Ctrl + Y—redo the last undo. This command can also be disabled using the Disable Undo and Redo Yes/No field selections on the NC Editor Settings screen.
- Ctrl + Z—undo the last change. This command can also be disabled using the Disable Undo and Redo Yes/No field selections on the NC Editor Settings screen.

The status bar provides real-time updates on the status on the program being edited. The information shown in the status bar is as follows:



1	Part Program Name
2	Units
3	Programming Mode
4	Coding Standard
5	Current Line Number
6	Insert/Overstrike Mode
7	Block Skip On/Off

Figure 1–3. NC Editor Status Bar

The Part Program name is shown first in the status bar. Programming Mode can either be radius (R) or diameter (D). Coding standard can either be Industry Standard (I) or Basic NC (B). Current Line number indicates the current location of the cursor in the part program. Insert mode is indicated as INS, while overstrike mode is indicated as OVR. SKIP On indicates Block Skip is turned on; SKIP OFF indicates Block Skip is turned off.

Refer to *NC Editor Menus, on page 1 - 17* for information about the NC Editor softkey menus.

Starting a New NC Program

Procedures for entering the program name, the part setup information, the tool descriptions, and the program parameters are described in Getting Started with WinMax Lathe, *Project Manager, on page 3 - 1* and *Programming Basics, on page 4 - 1*, in addition to information about saving files.

The parameters, part setup (except work offsets), and tool setup (except tool offsets) used for NC programs are not stored as part of the NC program. This information can be loaded from a Conversational program before going into the NC mode.

Part and tool setup information is not automatically deleted or replaced when loading a new NC program. If part and tool setup information appears on the setup screens after loading in a new NC program, it is the setup information from the previously displayed part program.

When there is no NC program loaded in memory, the system automatically assigns the filename NONAME#, where the # represents a sequential number from 0 to 99. The file extension is set in User Preferences. Refer to Getting Started with WinMax Lathe, *Project Manager, on page 3 - 1*.

These steps for creating an NC part program help determine the most efficient tool movement and basic program structure to save time during programming:

1. Determine the tool path on the print and label the points where the path direction changes.
2. Make a chart showing the coordinates of each point identified in the previous step.
3. Identify the turret movements that will be necessary during cutting.

NC Programming Rules

Here are some basic rules to follow when creating NC part programs:

- The axis letter always precedes the numeric information.
- In most cases an integer works the same as a decimal or real number. In the following cases an integer is scaled by the appropriate scaling factor to maintain compatibility with existing NC programs:

  Feedrate: F (BNC only)

  Rotation: R (ISNC only)

  Dwell: P, X (Both BNC and ISNC)

  Scaling: P (ISNC only)

  ⇨ If an integer is below the acceptable range after scaling, a “Below Minimum Value” error message occurs.

- All axis dimensions are considered to be positive unless a minus sign is entered. When describing axis motion, the codes for the program block must contain the following information in order to move properly:
  - Axis identification (e.g., X, Y, Z).
  - Direction the axis will move (+ or -).
  - Distance the axis will move (e.g., 4.0).
  - Enter the speed preceded by the F address character to program a feedrate in a block.
  - Include a Z parameter to permit the system to draw the part on the graphics screen. An absolute Z command must occur after a tool change before making another move command.

Program Editing Features

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

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

The console keyboard contains these keys:

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

  Please refer to Getting Started with Your WinMax Lathe, *Standard Keys, on page 1 - 34* for details about using these keys for specific functions for Tool Setup or Tool Review.

  Refer to *Program Review, on page 1 - 16* and for details about using these keys with the Program Review screen.

- **Cursor Control keys**—Arrows, Enter, alt, C, and F function keys. Refer to Getting Started with WinMax Lathe, *Cursor Control Keys, on page 1 - 35*.
- **Draw key**—displays the graphic. Refer to Getting Started with WinMax Lathe, *Verifying Part Programs—Graphics, on page 4 - 86* for more information about Wireframe Graphics.
- **Verify key**—displays the graphic in 2D or 3D view. Refer to Getting Started with WinMax Lathe, *Verifying Part Programs—Graphics, on page 4 - 86* for more information about Verification Graphics.
- **Optional AT-Keyboard**—allows you to enter data using a keyboard. Refer to Getting Started with WinMax Lathe, *AT-Keyboard, on page 1 - 36*.

The software provides these features:

- **Pop-up Text Entry window**—allows you to enter text, such as naming a part program, if the console does not have an optional AT-keyboard. Access this feature by selecting the keyboard icon on the bottom of the screen.
- **NC Subprogram Review screen**—lists the data blocks in a program and the tools programmed for each block. Refer to *Program Review, on page 1 - 16* for information about using this screen.
- **Tool Review screen**—lists the tool number, type, orientation, size, programming mode (diameter or radius), speed, and feed for each tool programmed in the data block. Please refer to Getting Started with WinMax Lathe, *Tool Review, on page 4 - 77* for details about using Tool Review.

Program Review

Press the **Review** console key to display the NC Program Review screen. A summary of the NC subroutines appears by number.

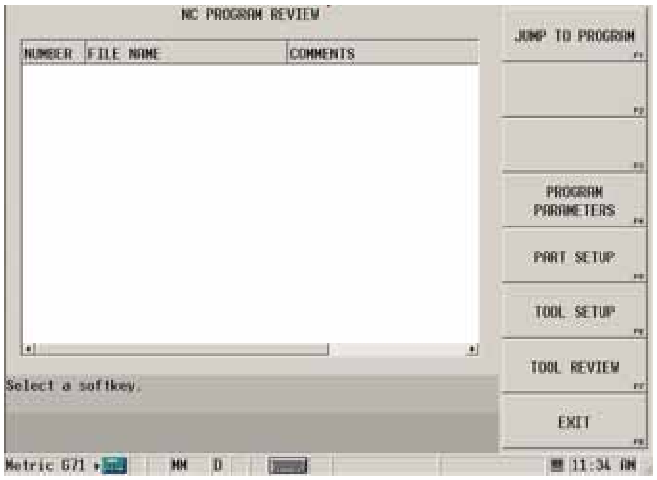


Figure 1–4. NC Program Review screen

To go directly to a subroutine, highlight the subroutine and press the **Enter** key or the JUMP TO SUBPROGRAM *F3* softkey. Refer to *NC Editor Menus, on page 1 - 17* for information about the NC Editor's screen and softkeys.

These Standard Console keys perform these functions with the Program Review screen:

- The **Home** console key jumps to the first subroutine listed in the Program Review screen.
- The **End** console key jumps to the last subroutine listed in the Program Review screen.
- The **Page Up** console key jumps to the first subroutine visible on the screen.
- The **Page Down** console key jumps to the last subroutine visible on the screen.

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

You can clear, delete, copy, and move data blocks to another location in a part program or even another program using the Edit Functions described in the *Edit Functions Menu, on page 1 - 20*.

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

With the optional AT-keyboard, you can use **Ctrl** + **select** (using the pointing device for selecting) to select non-consecutive data blocks.

NC Editor Menus

The Hurco NC system provides many levels of program editing, as well as editing tools, to simplify the task. Press the MORE ➔ *F7* softkey on each menu to move to the next menu selection. The menus appear in a circular loop; every time you press the MORE ➔ *F7* softkey you access the next menu.

The NC Editor contains these top level menus:

Basic Programming Menu

While entering NC codes to create blocks, you may wish to insert new blocks, delete blocks, or display different sections of the part program on the screen.

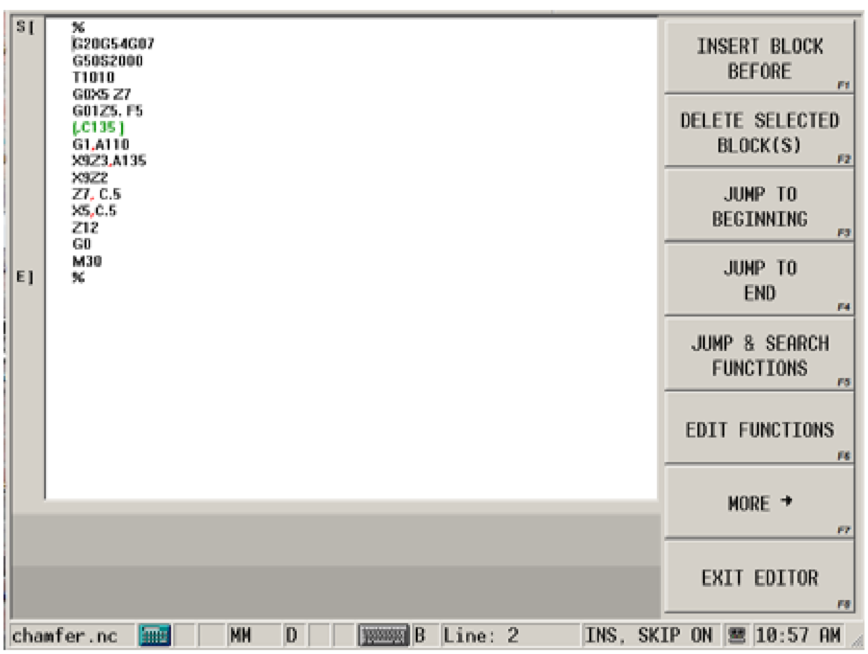


Figure 1–5. ISNC Basic Programming Menu

The Basic Programming softkeys provide these functions:

- **Insert Block Before**—inserts a blank line before the block where the cursor is located. This permits addition of a new block of data. This softkey will be disabled if text can't be inserted at the current cursor location.

- **Delete Block**—removes the block where the cursor is positioned. This softkey will be disabled if the block can't be deleted.
- **Jump to Beginning**—moves the cursor to the beginning of the first program block in memory. If a keyboard is available, Ctrl + Home combination will result in the same action.
- **Jump to End**—moves the cursor to the beginning of the last program block in memory. If the keyboard is available, Ctrl + End combination will result in the same action.
- **Jump and Search Functions**—invokes Jump and Search Functions menu.
- **Edit Functions**—invokes Edit Functions menu
- **More ➔**—displays the next menu.
- **Exit Editor**—exits the NC Editor and moves to the Input screen. Select Part Programming to return to the NC Editor screen.

Jump and Search Functions Menu

The Jump and Search provides the flexibility to locate hard-to-find items in program memory using the block or sequence number or searching for specific address characters, numeric parameters, or words.

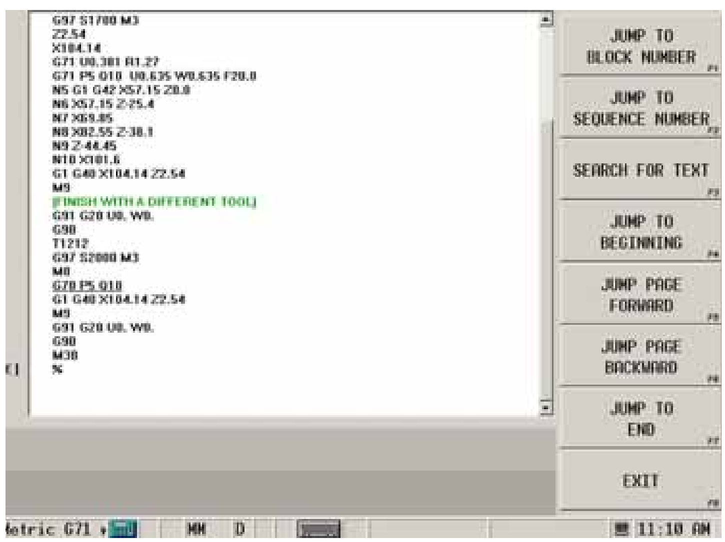


Figure 1–6. ISNC Jump and Search Functions Menu

When the Jump & Search Functions softkey is selected from the Basic Programming menu, these softkeys appear:

- **Jump to Block Number**—enter a block number in the popup box. The cursor is positioned on the specified block. The number entered refers to the position that block has in NC memory.

  ⇨ You can also use the touchscreen to select the Line number in the Editor status bar to open the Jump to Block popup box.

- **Jump to Sequence Number**—enter a sequence number (N code) in the popup bo.The cursor is positioned on that block.
- **Search for Text**—activates the Search submenu. Refer to *Search Submenu, on page 1 - 19* for more information.
- **Jump to Beginning**—moves the cursor to the beginning of the first program block in memory. If a keyboard is available, Ctrl + Home combination will result in the same action.
- **Jump Page Forward**—moves the cursor to show the screen one page forward.
- **Jump Page Backward**—moves the cursor to show the screen one page backward.
- **Jump To End**—moves the cursor to the beginning of the last program block in memory. If the keyboard is available, Ctrl + End combination will result in the same action.
- **Exit**—returns to the previous menu.

Search Submenu

The Search submenu allows you to search for specific text in the part program.

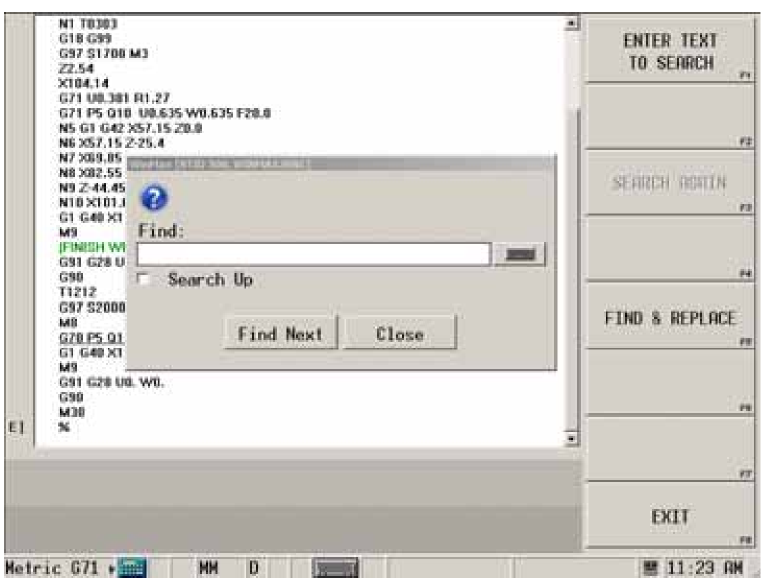


Figure 1–7. ISNC Search Submenu

The menu contains the following softkeys:

- **Enter Text To Search**—opens a popup box where search text is entered. Type search term and select the Find Next button. The NC editor finds the next occurrence of the entered text following the current cursor position. If found, the text is highlighted. Selecting the Search Up checkbox will search for the search term prior to the current cursor position.
- **Search Again**—repeats the last search operation without prompting the user.
- **Find and Replace**—opens a popup box where search text is entered and replacement term is specified.
  - The **Replace Next** screen button finds the next instance of the search term and replaces it with the new term. To find and replace the next instance you must select the Replace Next button again.
  - The **Replace All** screen button finds and replaces all instances of the search term.
- **Exit**—invokes the Jump and Search menu.

Edit Functions Menu

The Edit menu provides advanced editing functionality to the user.

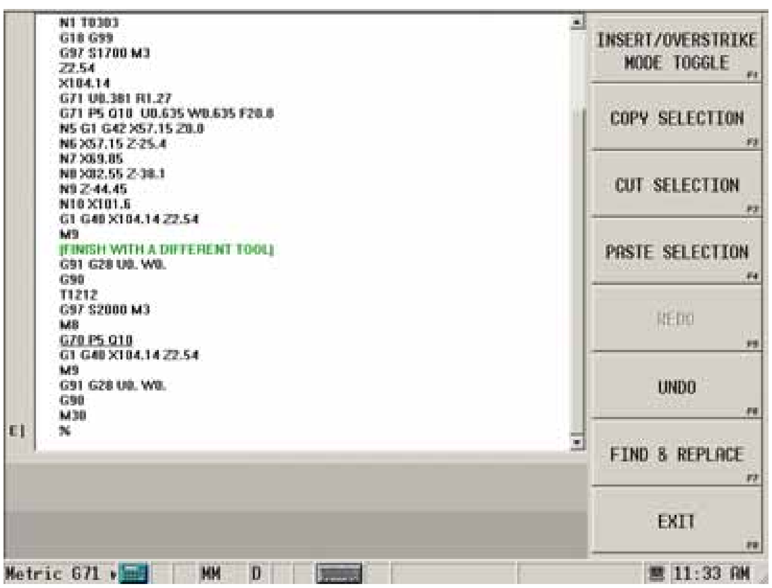


Figure 1–8. ISNC Edit Functions Menu

When the Edit Functions softkey is selected from the Basic Programming menu, these softkeys appear:

- **Insert/Overstrike Mode Toggle**—switches the data entry style between insert and overwrite. Currently active mode is shown on the status bar of the window - “INS” indicates Insert and “OVR” indicates Overwrite.

  ⇨ You can also use the touchscreen to toggle the Insert/Overstrike Mode in the Editor status bar.

  Or, you can press the **Insert** console key to toggle between the two modes.

- **Copy Selection**—copies the selected text to the Windows' clipboard memory. If the keyboard is available, this action can be achieved by pressing Ctrl + C combination. Text can be selected by dragging the pointing device (up/down/right/left) across the editing area of the screen, or by holding down the keyboard Shift key and using the arrow keys to select text.
- **Cut Selection**—copies the selected text to the Windows’ clipboard. In addition, the selected text is deleted from the part program. If the keyboard is available, Ctrl + X combination will cut the selected text and place it in the Windows’ clipboard.
- **Paste Selection**—text previously copied to the Windows' clipboard is inserted at the current cursor position. If the keyboard is available, this action can be achieved by pressing Ctrl + V keys.
- **Redo**—redo the editing operation(s) previously undone by the Undo softkey. If the keyboard is available, Ctrl + Y combination will result in the same action. The feature can be disabled using the NC Editor Settings screen and the Disable Undo and Redo field.
- **Undo**—undo the previous editing operations starting from the latest to the earliest. The depth of the undo buffer is 100. If more than one text modification was performed in a single step (e.g. “Replace All”), all those modifications will be undone in one step as well. If the keyboard is available, Ctrl + Z combination will result in the same action.The feature can be disabled using the NC Editor Settings screen and the Disable Undo and Redo field.
- **Find and Replace**—activates a popup box where search text and replacement text are entered. NC editor finds occurrences of the search text and replaces it with the replacement text.
- **Exit**—loads the Basic Programming menu.

Renumbering and Tagging Menu

Renumbering and tagging menu offers access to renumbering and tagging functionality. This menu can be accessed by selecting More➔ from the Basic programming menu.

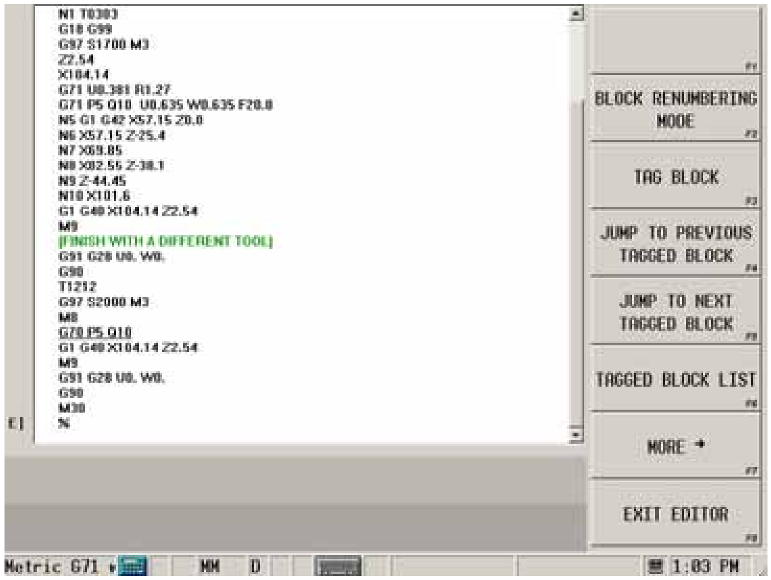


Figure 1–9. ISNC Renumbering and Tagging Menu

The menu contains the following softkeys:

- **Block Renumbering Mode**—invokes Block Renumbering Mode submenu. See *Block Renumbering Submenu, on page 1 - 23*.
- **Tag Block**—tags a block by underlining it when selected. Select the Tag Block softkey a second time to remove the tag from the block. There is no limit on the amount of tagged blocks you can have.
- **Jump to Previous Tagged Block**—NC editor tries to find the closest tagged block before the current cursor's position. If such block is found, the cursor is placed in the beginning of the block.
- **Jump to Next Tagged Block**—NC editor tries to find the closest tagged block after the current cursor's position. If such block is found, the cursor is placed in the beginning of the block.
- **Tagged Block List**—invokes Tagged Blocks List screen, which allows the user to review the tagged block and manage the tags. See *NC Tag List, on page 1 - 24*.
- **More ➔**—invokes Program Execution menu.
- **Exit Editor**—exits the NC Editor and moves to the Input screen. Select Part Programming to return to the NC Editor screen.

Block Renumbering Submenu

The Block Renumbering mode submenu allows the user to search for specific text in the part program.

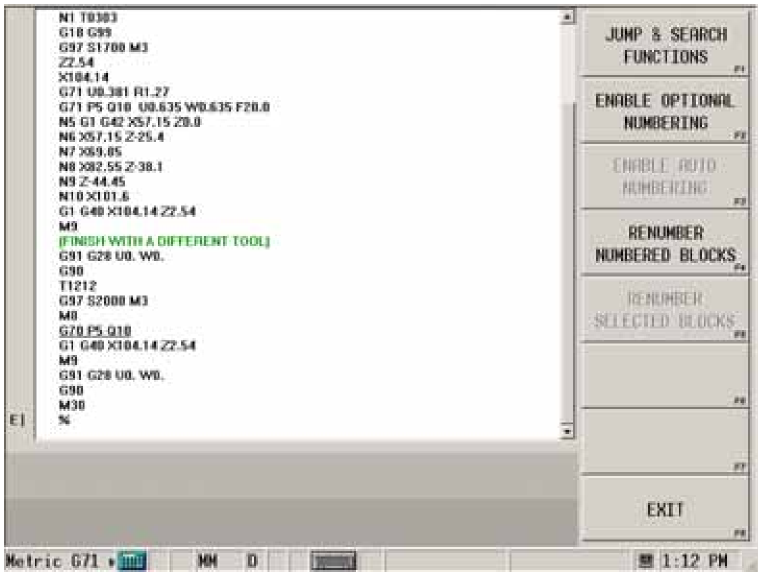


Figure 1–10. ISNC Block Renumbering Submenu

The menu contains the following softkeys:

- **Jump & Search Functions**—opens the Jump and Search Functions menu. Refer to *Jump and Search Functions Menu, on page 1 - 18*.
- **Enable Optional Numbering**—enables user-assigned block numbering mode.
- **Enable Auto Numbering**—enables automatic block numbering mode.
- **Renumber Numbered Blocks**—enables block renumbering. The renumbering interval is entered into the popup box and the blocks are automatically renumbered. All numbered blocks will be renumbered in the program.
- **Renumber Selected Blocks**—enables renumbering of selected blocks. First select the blocks to be renumbered by dragging the pointing device (up/down/right/left) across the editing area of the screen, or by holding down the keyboard Shift key and using the up/down/left/right arrow keys to select text. Then select the softkey and specify the renumbering interval. The selected blocks are renumbered.
- **Exit**—invokes the Renumbering and Tagging menu.

NC Tag List

The NC Tag List screen is accessed with the Tagged Block List softkey on the Renumbering and Tagging Menu. This softkey is not active (grayed out) if there are no tagged blocks. The tagged blocks in a program are displayed:

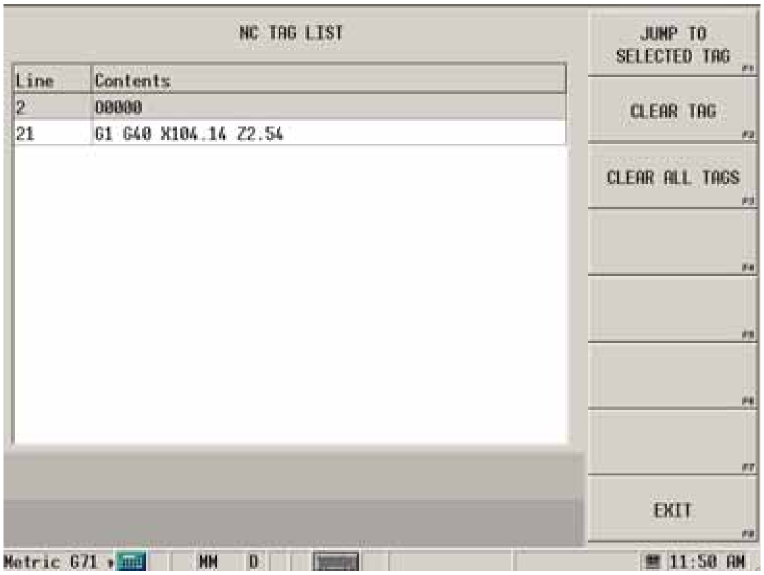


Figure 1–11. ISNC NC Tag List Screen

Softkeys are:

- **Jump to Selected Tag**—goes to the selected tag in the program.
- **Clear Tag**—deletes the selected tag.
- **Clear All Tags**—deletes all tags in the list.
- **Exit**—invokes the Renumbering and Tagging menu.

Program Execution Menu

Program Execution menu allows selection of the part program to be processed for graphics and for execution. This menu can be accessed by twice pressing More➔ (first from the Basic Programming menu and then from the Renumbering and Tagging menu).

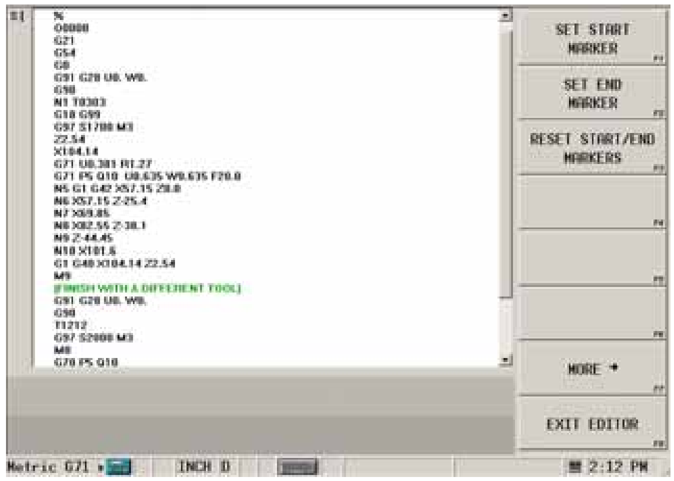


Figure 1–12. ISNC Program Execution Menu

The menu contains the following softkeys:

- **Set Start Marker**—indicates the block that the system should use to start program execution when running and verifying the program. A letter S is inserted to the left of the block. All blocks previous to the Start block are grayed out.
- **Set End Marker**—indicates the block that system should use to end program verification and execution. A letter E is inserted to the left of the block. All blocks following the End block are grayed out.

  ⇨ If the Start and End markers overlap, the dollar ($) sign is displayed to the left of the line.

- **Reset Start/End Markers**—restores start and end markers to their defaults. The defaults are at the beginning and end of the program.
- **More ➔**—invokes NC Editor Settings menu.
- **Exit Editor—**exits the NC Editor and moves to the Input screen. Select Part Programming to return to the NC Editor screen.

NC Editor Settings Menu

NC Editor Settings allows you to modify the editor's behavior.

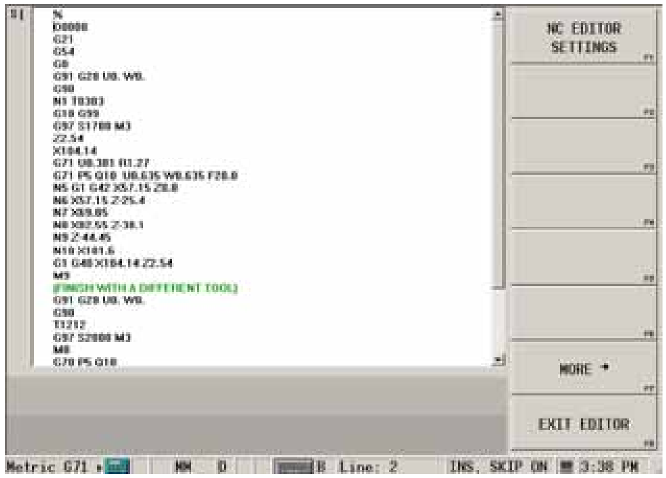


Figure 1–13. NC Editor Settings Menu

Softkeys are

- **NC Editor Settings**—invokes NC Editor Settings screen. See *NC Editor Settings, on page 1 - 27*.
- **More →**—invokes Basic Programming menu.
- **Exit Editor**—exits the NC Editor and moves to the Input screen. Select Part Programming to return to the NC Editor screen.

NC Editor Settings

The NC Editor Settings screen allows you to set the Block Skip Enable state, adjust the color of commented text, and disable the Undo and Redo feature.

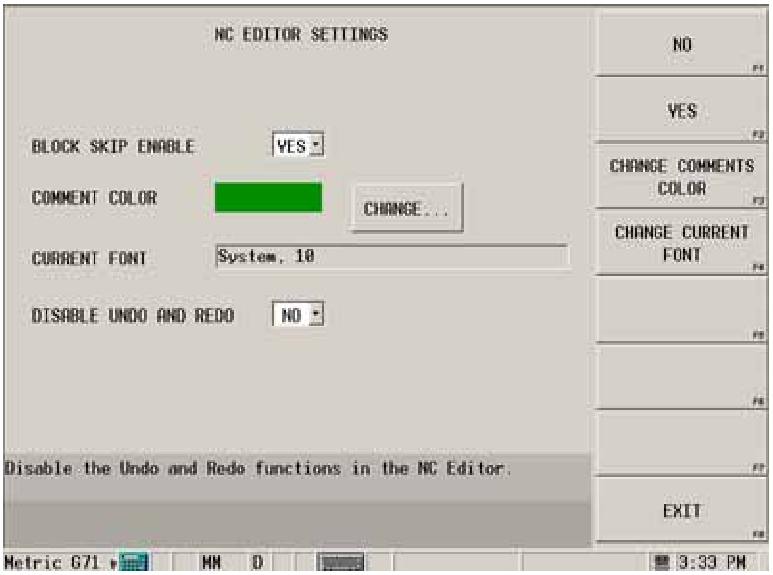


Figure 1–14. ISNC NC Editor Settings screen

Fields and softkeys are

- **Block Skip Enable**—indicates whether blocks with the Ignore character (**/**) are interpreted and executed or ignored. Yes (Skip Off) and No (Skip On) softkeys appear when the cursor is in this field. To change the Block Skip state,
  - Select the **Yes** softkey to ignore the blocks with the Ignore character (**/**). **Skip On** appears in the status bar.
  - Select the **No** softkey to execute the blocks with the Ignore character (**/**). **Skip Off** appears in the status bar.
- **Comment Color**—displays the current comment color. The color can be changed by selecting the **Change...** screen button or the **Change Comments Color** softkey. A window opens with Red, Green, and Blue slide adjustments for changing the color of comment text.
  - Select **OK** to save changes and return to the NC Editor Settings screen.
  - Select **Cancel** to close the window without saving changes.
- **Current Font**—displays the current NC Editor font. To change the font, select the **Change Current Font** softkey. A Choose Font window opens with Font and Size selections. A Sample of the selected font appears in the lower portion of the window.
  - Select **OK** to save changes and return to the NC Editor Settings screen.
  - Select **Cancel** to close the window without saving changes.

⇨ The **OK** screen button is inactive if a font size other than a pre-defined size selected from the list is entered.

- **Disable Undo and Redo**—indicates whether the NC Editor's Undo and Redo functions are disabled.
  - When **Yes** is selected, the Undo and Redo softkeys are not available (grayed out) on the Edit Functions softkey menu.
  - When **No** is selected, the Undo and Redo softkeys are available on the Edit Functions softkey menu for use following an editing operation.

Importing and Exporting M Code Data

The import and export functions feature allows you to import NC files written for machine types for use with controls other than WinMax Lathe Max controls. In addition, this feature makes it possible for you to export NC files written for WinMax Lathe Max controls to machine types for use with other controls.

From the Input screen, select the Import/Export Functions F7 softkey. The Import and Export Functions screen appears.

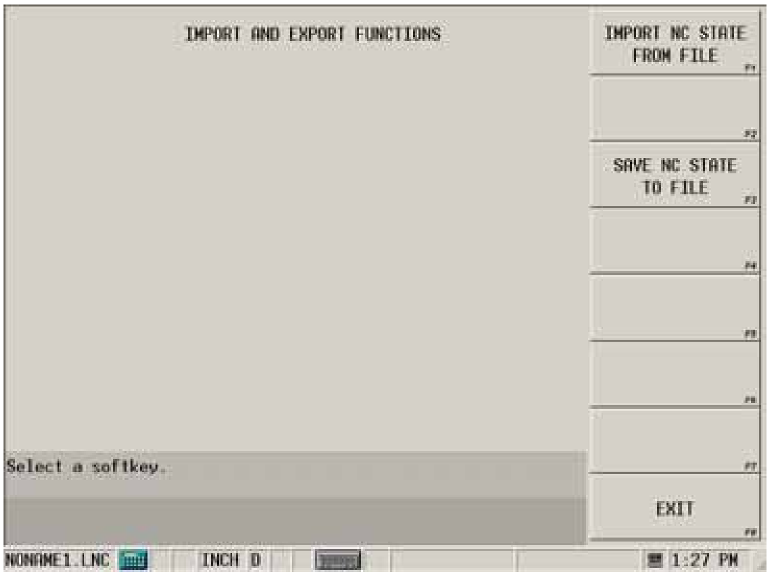


Figure 1–15. Import and Export Functions screen

Softkeys are:

- **Import NC State From File** *F1*—imports M Codes from a file.
- **Save NC State To File** *F3*—saves NC setup data to a file.

Select Save NC State To File *F3* softkey and this screen appears:

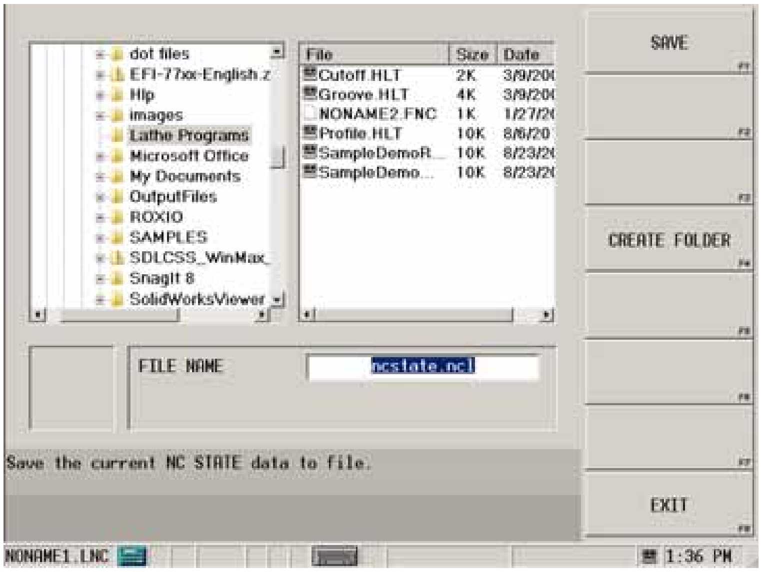


Figure 1–16. Save NC State to File screen

This screen shows folders on the left pane and file names on the right pane. The File Name field defaults to the name “ncstate.ncl.”

You can select the Create Folder *F4* softkey to set up a folder to hold the NC State File. A pop-up window appears for you to enter the folder name. Select OK and the folder appears in the left pane. When Save NC State to File is selected, the screen opens to the last folder used.

To name the file and save it to the folder,

1. Open the folder.
2. Type the file name in the File Name field.
3. Select the Save *F1* softkey. A message appears confirming the file was saved successfully.
4. Select OK to clear the message. The Input screen appears.

To view the file in the directory listing,

1. Select the Import/Export Functions *F7* softkey on the Input screen
2. Select the Save NC State to File *F3* softkey. The screen shows directory information, including the folder and file.

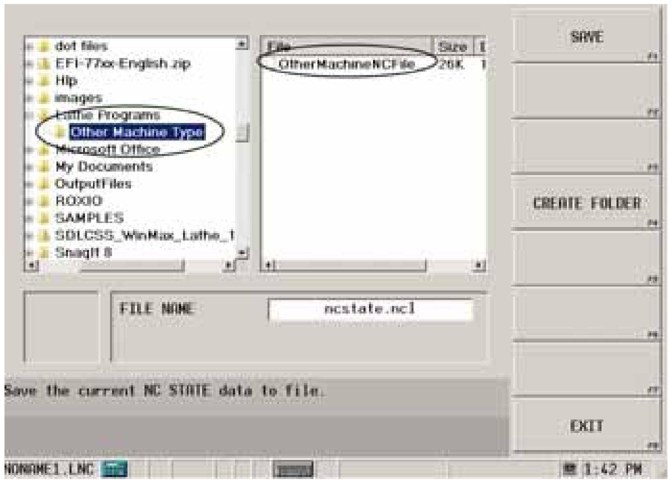


Figure 1–17. Save NC State to File screen with new folder and NC State file

To load this file from the Import and Export Functions screen,

1. Select the Import NC State From File *F1* softkey. The screen opens to the last folder used.
2. Select the file name from the right-hand pane.
3. Select the Load Selected File(s) *F1* softkey. The Import NC State to Conversational screen appears.

⇨ The Import NC State to Conversational screen appears if a conversational program (.LNC) is currently loaded. Load an NC file or create a new NC file if a conversational program is loaded.

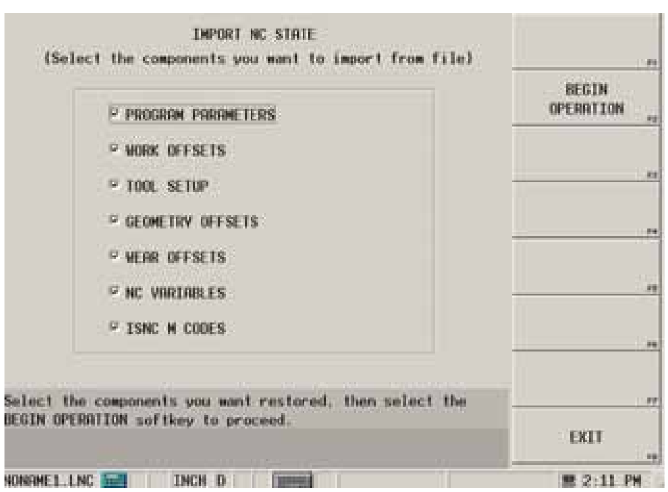


Figure 1–18. Import NC State screen

4. Select the program components to be restored.

   ⇨ To import only the M Codes from an NC file, select only ISNC M Codes. This selection retains existing tool offsets, work offsets, and parameters.

5. Select the Begin Operation *F2* softkey. A pop-up window appears confirming the NC State being successfully restored. Select OK to close the window.

ISNC G Codes

The following ISNC G Codes are defined in this section:

Overview

G Codes initiate axis motion, plane changes, and feedrate changes. This information is often needed when using an off-line CAM or CAD/CAM system to create NC part programs.

⇨ Some G Codes are on by default at the start of a program until another G Code cancels it. Defaults are identified below.

Any unrecognized code will stop execution and display an error message. More than one G code can be specified in the same block. If more than one is from the same group, the last G code entered is active.

G Code Groups

The G codes are grouped by function. These groups are listed in the following table:

Group	Function
00	One-Shot
01	Interpolation
03	Dimension
02	Spindle Speed
05	Feed
06	Units of Measure
07	Cutter Compensation
10	Return from Canned Cycles
14	Coordinate System Selection
16	Plane Selection
21	Programming Mode
30	Lathe Axis Map Coordinates

Table 2–1. G Code Groups

Specifying a Group 01 (Interpolation) G code in a canned cycle automatically enters the G80 - Canned Cycle Cancel (default) condition. Conversely, a Group 01 G code is not affected by the canned cycle G codes.

G Code Table

The following table lists the G codes, identifies the defaults in bold text, identifies groups, lists Modal or Non-modal types, and describes the G code functions.

G Code	Group	Type	Function
G00	01	Modal	Rapid Traverse **(default)**
G01			Linear Interpolation
G02			Clockwise Circular Motion at Feed
G03			Counterclockwise Circular Motion at Feed
G04	00	Non-modal	Dwell
G07	21	Modal	Radius Programming
G08			Diameter Programming **(default)**
G09	00	Non-modal	Exact Stop
G10	00	Modal	Data Setting— Work Offsets or Tool Offsets or Tool Geometry Offsets
G11			Data Setting Cancel **(default)**
G17	16	Modal	XY Plane
G18			XZ Plane **(default)**
G19			YZ Plane
G20	06	Modal	Inch Mode **(default)**
G21			Millimeter Mode
G28		Non-modal	Automatic Return to reference Point
G32	01	Modal	Constant Lead Thread Cutting
G33			Threading
G40	07	Modal	Cutter Radius Compensation Off **(default)**
G41			Cutter Radius Compensation Left
G42			Cutter Radius Compensation Right
G50 or G92	00	Non-Modal	Set Part Setup or Set Max RPM
G53	14	Non-modal	Machine Coordinates
G54 - G59			Work Coordinates **(default)**
G54.1 - G54.93			Aux Work Coordinates
G61		Modal	Precision Cornering
G64			Precision Cornering Cancel **(default)**
G65	00		Subprogram Call
G66	12	Modal	Modal Subprogram Call
G67			Modal Subprogram Call Cancel **(default)**
G70	00	Non-Modal	Finish Cycle
G71			Stock Removal in Turning
G72			Stock Removal in Facing
G73			Stock Removal with Repeat Pattern
G74			Z Direction Grooving
G75			X Direction Grooving
G76	01	Modal	Multiple Thread Cutting
G77			Stock Removal in Turning Cycle
G78 or G92			Threading Cycle
G79			Stock Removal in Facing Cycle

G Code	Group	Type	Function
G80	10	Modal	Canned Cycle Cancel **(default)**
G81			Drill Cycle
G82			Drill Cycle with Dwell
G83			Face Drilling with Pecks
G83.1			Chip Breaker Peck Drilling
G84			Face CW Tapping
G84.2 (or G84 M29)			Rigid CW Tapping
G85			Face Boring Cycle
G86			Bore Rapid Out Cycle
G87			Side Drilling with Pecks/Dwell
G88			Side CW Tapping
G89			Side Boring Cycle
G90	03	Modal	Absolute Programming **(default)**
G91			Incremental Programming
G92	14	Modal	Work Coordinate Offsets or Spindle Max Speed or Threading Cycle
G93	05	Modal	Inverse Time
G94			Feed per Minute **(default)**
G95			Feed per Revolution
G96	02	Modal	Constant Surface Speed (CSS)
G97			Direct Spindle Speed **(default)**
G98	11	Modal	Return to Initial Level **(default)**
G99			Return to R Point Level
G101	30	Modal	Axial Surface of the Part - Main Spindle without Linear Y
G102		Modal	Radial Surface of the Part - Main Spindle without Linear Y
G103		Modal	Axial Surface of the Part - Main Spindle with Linear Y
G104		Modal	Radial Surface of the Part - Main Spindle with Linear Y
G105		Modal	Axial Surface of the Part - Sub-spindle without Linear Y
G106		Modal	Radial Surface of the Part - Sub-spindle without Linear Y
G107		Modal	Axial Surface of the Part - Sub-spindle with Linear Y
G108		Modal	Radial Surface of the Part - Sub-spindle with Linear Y
G109		Modal	Turning / Lathe Mode **(default)**
G152	00	Non-modal	Moves the W-axis

Table 2–2. G Codes

G00 - Rapid Traverse (default)

G00 sets rapid traverse as the machine's modal condition. Any block that executes while G00 is modal moves the axes at rapid feedrate to the programmed endpoint. The feedrate of each axis is controlled to ensure that all programmed axes reach their respective endpoints simultaneously.

Default—Yes

Modal—Yes

Cancels G01 - Linear Interpolation, G02/G03 - Clockwise/Counterclockwise Arc

Format

G0 X(u) ____ Y(v) ____ Z (w) ____ C(h)____

- X indicates Absolute X and (u) indicates Incremental X.
- Y indicates Absolute Y and (v) indicates Incremental Y.
- Z indicates Absolute Z and (w) indicates Incremental Z.
- C indicates Absolute C and (h) indicates Incremental C.

G01 - Linear Interpolation

G01 sets feed linear interpolation as the machine's modal condition. Any block that executes while G01 is modal moves the programmed axes directly to their programmed endpoint at a specified feedrate using the F command. G01 requires programming X or Z or both and you may include F if not specified previously. The vector feedrate of each axis is controlled to ensure that all axes reach their endpoints simultaneously.

Default—No

Modal—Yes

Cancels G00 - Rapid Traverse (default), G02/G03 - Clockwise/Counterclockwise Arc

Format

G1 X(u) ____ Y(v) ____ Z(w) ____ C(h) ____

- X indicates Absolute X and (u) indicates Incremental X.
- Y indicates Absolute Y and (v) indicates Incremental Y.
- Z indicates Absolute Z and (w) indicates Incremental Z.
- C indicates Absolute C and (h) indicates Incremental C.

Examples

```
N5 G90 G07 G01 X0.25 Z-0.5 F ___ (G90 = Absolute; G07 = Radius)
N7 G90 G08 G01 X1.00 Z-0.5 F ___ (G90 = Absolute; G08 = Diameter)
N5 G91 G07 G01 X0.25 Z-0.5 F ___ (G91 = Incremental; G07 = Radius)
N7 G91 G08 G01 X0.50 Z-0.5 F ___ (G91 = Incremental; G08 = Diameter)
```

G02/G03 - Clockwise/Counterclockwise Arc

G02 sets clockwise arc as the modal condition. G03 sets counterclockwise arc as the modal condition.

Default—No

Modal—Yes

Cancels:

- G00 - Rapid Traverse (default)
- G01 - Linear Interpolation,
- G02/G03 - Clockwise/Counterclockwise Arc

Either program the arc endpoint and center coordinates, or program the arc endpoint and radius. Refer to *Program Arc with Endpoint and Center Coordinates, on page 2 - 9* and *Program Arc with Endpoint and Radius, on page 2 - 11* for programming examples.

Format

G02 X(u) ____ Z(w) ____ I ____ K ____ or R ____

G03 X(u) ____ Z(w) ____ I ____ K ____ or R ____

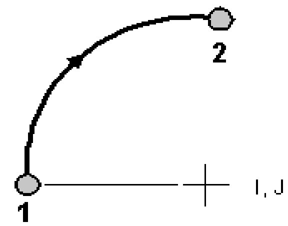
⇨
- X indicates Absolute X and (u) indicates Incremental X.
- Z indicates Absolute Z and (w) indicates Incremental Z.

Parameters

- **I**—X-axis Center/Offset coordinate for programming geometric information needed to determine the endpont of a motion command.
- **K**—Z-axis Center/Offset coordinate for programming geometric information needed to determine the endpoint of a motion command.
- **R**—radius

Example

The following figure illustrates G02 (Clockwise) arc at feed:



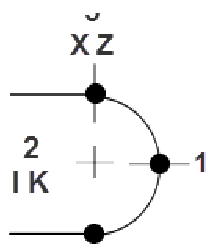
1	Start
2	End

Figure 2–1. G02 (Clockwise) circular motion at feed

Program Arc with Endpoint and Center Coordinates

This method requires you to define the arc endpoint and center coordinates. Dimensions for the arc endpoint (X, Z) may be defined with absolute or incremental dimensions, depending on the modal G90 - Absolute Programming (default) or G91 - Incremental Programming condition.

All arc center dimensions (I, K) must be defined using signed incremental dimensions (the distance from the arc start point to the arc center).



1	Start Point
2	I K Arc Center Coordinate
3	X Z End Point

Figure 2–2. Arc with Endpoint and Center Coordinates

If G07 - Radius Programming is active, the arc endpoint (X) and center (I) coordinates must be programmed as radius values.

If G08 - Diameter Programming (default) is active the arc endpoint (X) must be programmed as a diameter value, but the arc center (I) must be programmed as a radius value.

Example G02/G03 Arcs

The following example part program shows arcs programmed with X, Z, I, and K.

```
%
N1 G90 G20 G40 G54 (MAKE OFFSET 10 INCHES)
N2 G00 T0101 G07 X.0 Z.1 (G07 = RADIUS PROGRAMMING) (MAKE SURE OFFSET IS
LARGER THAN 0.5")
N3 G01 Z0 F10
N4 X.2
N5 G03 X0.8 Z-0.6 I0 K-0.6
N6 G01 G91 Z-1.0
N7 G90 G08 G02 X1.9512 Z-2.0243 R.6 (G08 = DIAMETER PROGRAMMING)
N8 G01 X2.2 Z-2.1487
N9 Z-2.5
N1 G00 X2.5
N11 Z.1
M30
```

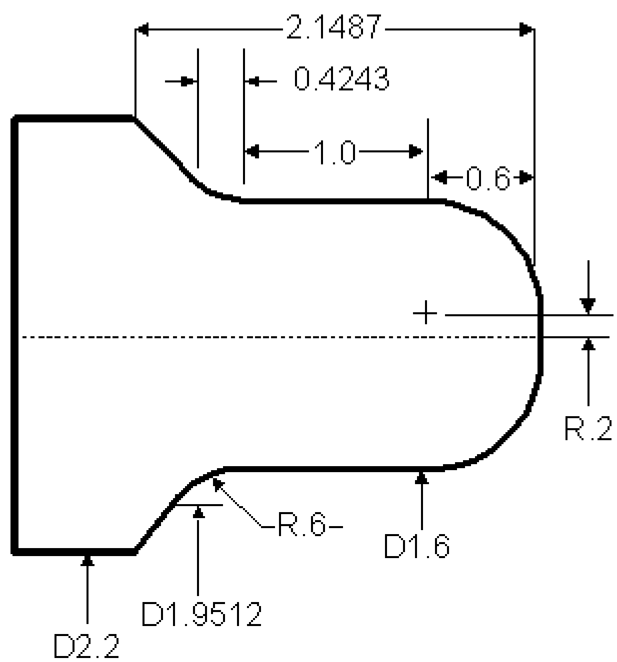
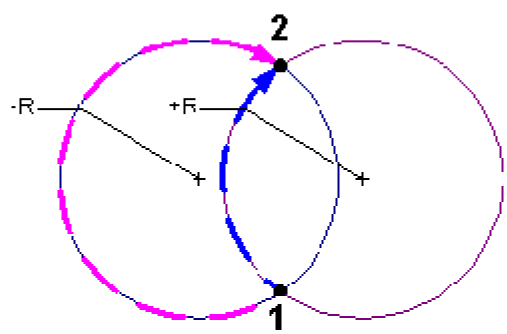


Figure 2–3. Arc with Endpoint and Center Coordinates

Program Arc with Endpoint and Radius

The second method of programming a G02 or G03 block is to define the arc endpoint and radius. There are two possible arcs when you program X, Z, and R (radius). The sign of the R radius value determines whether you want the arc that spans less than 180° (by programming R as a positive value), or the arc that spans more than 180° (by programming R as a negative value).

```
N2 G90 G07 G02 X__ Z__ R+__ (G90 = Absolute; G07 = Radius; G02 = CW)
N2 G90 G07 G03 X__ Z__ R-__ (G90 = Absolute; G07 = Radius; G03 = CCW)
N7 G90 G08 G02 X__ Z__ R+__ (G90 = Absolute; G08 = Diameter; G02 = CW)
N7 G90 G08 G03 X__ Z__ R-__ (G90 = Absolute; G08 = Diameter; G03 = CCW)
N2 G91 G07 G02 X__ Z__ R+__ (G91 = Incremental; G07 = Radius G02 = CW)
N2 G91 G07 G03 X__ Z__ R-__ (G91 = Incremental; G07 = Radius; G03 = CCW)
N7 G91 G08 G02 X__ Z__ R+__ (G91 = Incremental; G08 = Diameter G02 = CW)
N7 G91 G08 G03 X__ Z__ R-__ (G91 = Incremental; G08 = Diameter; G03 = CCW)
```



1	Start
2	X, Z End

Figure 2–4. Arc with Endpoint and Radius

G04 - Dwell

G04 programs a pause in part program execution.

Modal—No

Format

G04 P ____ or X ____ or U ____

X and U can include a decimal point. For example, X10 is X10.0.

P cannot include a decimal point, and the value is divided by 1000 or 10000. For example, P3000 is 3.0 seconds.

If Active G Code	Then P, X, or U in the G04 Block Programs
G93 Inverse Time Feedrate Refer to *G93 - Inverse Time Feedrate, on page 2 - 89* for information about G93.	The actual feedrate is the distance divided by the time. The dwell is in seconds.
G94 Feed Per Minute Refer to *G94 - Feed per Minute (default), on page 2 - 90* for information about G94.	The number of seconds to dwell. For example, the block G04 X2.1 pauses for 2.1 seconds before it executes the next block of code.
G95 Feed Per Revolution Refer to *G95 - Feed per Revolution, on page 2 - 90* for information about G95.	The number of spindle revolutions to dwell. For example, the block G04 X10 pauses for 10 spindle revolutions before it executes the next block of code.

Table 2–3. G04 Dwell Examples

G06 - Probe/Block Skip

G06 arms the Probe mechanism and provides a means for stopping the axis, for instance, when the torque is detected to be too high. Remaining Motion programmed after a high torque detection is deleted from memory. Please refer to *W Torque Mon On (ISNC M186), on page 3 - 15* for details.

G07 - Radius Programming

You can program X axis dimensions using the radius or diameter of the part. Program **G07** when you wish to program radius dimensions.

Default—No

Modal—Yes

Cancels—G08

Format

G07 X ____ (X is interpreted as a radius value.)

This mode will affect live-tool (milling) when running G codes. When G07 is active, live-tool programming is the same as programming a 3-axis milling machine.

The table below provides information about how the modal setting affects part programs.

Programmed Value	G07 Modal
Endpoint of linear move	X use radius
Endpoint of circular move	X use radius
Center and radius of circular move	IKR use radius
X axis infeed distances	infeed distance use radius

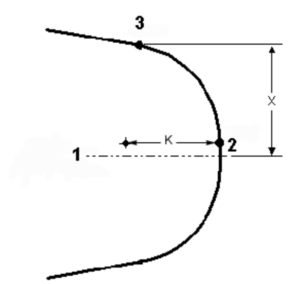
Table 2–4. Effects of Using G07

G07 Radius Programming Example

The following example part program shows arcs programmed with X, Z, I, and K.

```
G90 G07 X0.8 Z-0.6 I0 K-0.6 (G90 = Absolute; G07 = Radius)
G91 G07 X0.8 Z-0.6 I0 K-0.6 (G91 = Incremental; G07 = Radius)
```

G07, Radius programming, establishes a modal setting that interprets X-axis dimensions in radius values. The following figure illustrates radius programming:



1	Spindle Center Line
2	Start
3	End

Figure 2–5. G07 Radius Programming

Programming Display Mode

G07 does not affect the Programming Display Mode. The correct units are interpreted from the part program.

⇨ The Programming Display Mode appears on the screen's Status bar. Select the D (diameter) on the screen to toggle the mode to R (radius). The display toggles between choices.

Figure 2–6. WinMax Status bar

G08 - Diameter Programming (default)

You can program X axis dimensions using the radius or diameter of the part. Program **G08** when you wish to program diameter dimensions.

Default—Yes

Modal—Yes

Cancels—G07

This mode will affect live-tool (milling) when running G codes. When G07 is active, live-tool programming is the same as programming a 3-axis milling machine.

The table below provides information about how the modal setting affects part programs.

Programmed Value	G08 Modal
Endpoint of linear move	X use diameter
Endpoint of circular move	X use diameter
Center and radius of circular move	IKR use radius
X axis infeed distances	infeed distance use radius

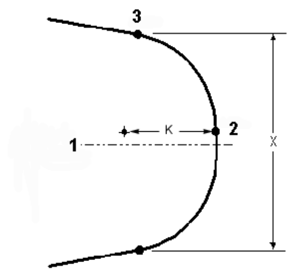
Table 2–5. Effects of Using G08

G08 Diameter Programming Example

The following example part program shows arcs programmed with X, Z, I, and K.

```
G90 G08 X1.6 Z-0.6 I0 K-0.6 (G90 = Absolute; G08 = Diameter)
G91 G08 X1.6 Z-0.6 I0 K-0.6 (G91 = Incremental; G08 = Diameter)
```

G08, Diameter programming, specifies that all X-axis positions are interpreted in diameter values. The following figure illustrates diameter programming:



1	Spindle Center Line
2	Start
3	End

Figure 2–7. G08 Diameter Programming

The arc center dimension (K) stays the same for both commands, but the X-axis dimension (X) changes.The program may switch between **G07** Radius and **G08** Diameter at any time.

Programming Display Mode

G08 does not affect the Programming Display Mode. The correct units are interpreted from the part program.

Figure 2–8. WinMax Status bar

G09 - Exact Stop

G09 causes the axes to decelerate to zero velocity at the end of a block. Since G09 is non-modal, you must program it in every block that requires the tool to come to a complete stop.

Modal—No

⇨ G09 is not used in conjunction with G00 - Rapid Traverse (default), G01 - Linear Interpolation or G02/G03 - Clockwise/Counterclockwise Arc commands.

Format

G09 X ____ Y ____ Z ____ (machine decelerates to stop at specified point.)

G10 - Data Setting (Work Offsets, Tool Offsets, or Tool Geometry)

G10 used in conjunction with specific parameters provides the following data setting functions:

Default—No

Modal—Yes

Cancels G11 - Data Setting Cancel (default)

⚠ G10 commands can not be performed by graphics at the same time the program is running.

Work Offsets

G10 L2 is used for setting work coordinate systems. One of six work coordinate systems can be changed as shown below where P is used to select the external work zero point offset value (P = 0), or one of the work coordinate systems (P = 1 to 6), and X, Y, Z is the work zero point offset value of each axis.

For Work Offsets, absolute values (X, Y, Z) replace work offset values. Incremental values (U, V, W) are added to the current work offset values.

Format—Work Offset

G10 L2 P0 or P1-6 X ____ Y ____ Z ____

G10 L2 P1-6 U ____ V ____ W ____

Tool Offset or Tool Geometry

G10 is used for setting tool length and radius offsets. G10 is used with the P and R parameters. P is the offset number 01 through 99, and R is the offset amount which may be absolute or incremental depending on G90 or G91.

For Tool Offsets, absolute values (X, Y, Z) replace tool offset values. Incremental values (U, V, W) are added to the current tool offset values.

Format—Tool Offset or Tool Geometry

G10 P01-99 X ____ Y ____ Z ____ R ____ (Tool Offset)

G10 P10001-10099 U ____ V ____ W ____ C ____ (Tool Geometry)

Parameters—Tool Offset or Tool Geometry

- **P**—program with a number 01 through 99 for the tool wear offset value.
- **P**—program with a number 10001 to 10099 for the tool geometry offset value.

G11 - Data Setting Cancel (default)

Default—Yes

Modal—Yes

Cancels—G10 - Data Setting (Work Offsets, Tool Offsets, or Tool Geometry)

G17 - XY Plane

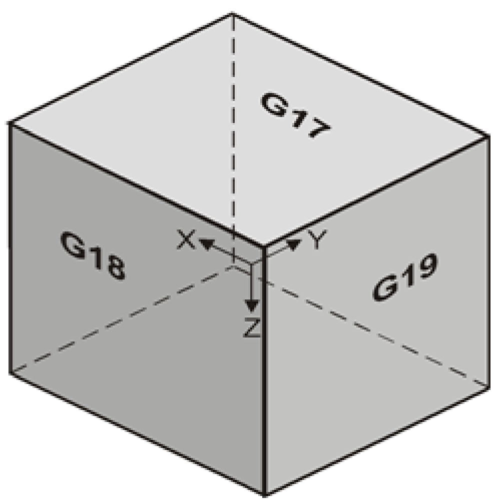
G17, the XY Plane Selection code, sets the plane for circular interpolation modes G02/ G03 - Clockwise/Counterclockwise Arc.

Default—No

Modal—Yes

G17 cancels G18 - XZ Plane (default) or G19 - YZ Plane.

The three codes in the plane selection group and their relationships to each other are illustrated below:



G17	XY Plane
G18	XZ Plane (default)
G19	YZ Plane

Figure 2–9. G17, G18, and G19 Planes

Format

The format of the XY plane selection command is as follows:

G17 X____ Y____

⇨ G17 is only available for turning centers with Y-axis motion.

- The X, Y, Z, I, and J words are valid in circular interpolation blocks.
- K words are invalid.
- If a Z word is programmed in the circular interpolation block, a helix is generated in the XY plane.
- The direction of an arc or helix in the XY plane can be determined by looking at the XY plane with positive X to the right and positive Y going up.
- The XY plane is a right-handed coordinate system (thumb points to positive Z, and fingers wrap in counterclockwise direction).
- In G17, the arc end point is defined by the X and Y words in the block. The arc center point is defined by the I and J words in the block.

Example

The diagram below illustrates XY plane selection:

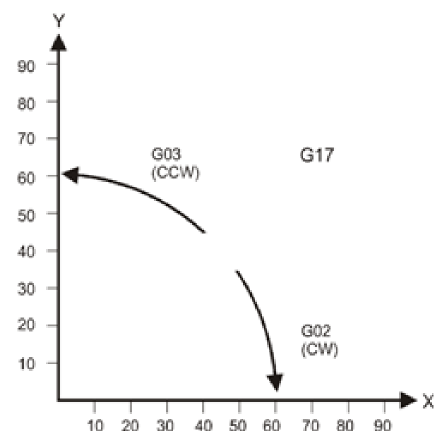


Figure 2–10. XY Plane Selection

G18 - XZ Plane (default)

G18, the XZ Plane Selection code, is the default and sets the plane for the circular interpolation modes G02/G03 - Clockwise/Counterclockwise Arc to the XZ plane.

Default—Yes

Modal—Yes

G18 cancels G17 - XY Plane or G19 - YZ Plane.

Format

The format of the XZ plane selection command is as follows:

G18 Z____ X____

- The X, Y, Z, I, and K words are valid in circular interpolation blocks.
- J words are invalid.
- If a Y word is programmed in the circular interpolation block, a helix is generated in the XZ plane.
- The direction of an arc or helix in the XZ plane can be determined by looking at the XZ plane with positive X to the right and positive Z going up.
- Basic NC and ISNC handle the XZ plane differently. For Basic NC, the XZ plane is a left-handed coordinate system (thumb points to positive Y, and fingers wrap in clockwise direction). For ISNC, the XZ plane is a right-handed coordinate system (thumb points to positive Y, and fingers wrap in a counterclockwise direction).
- In G18, the arc end point is defined by the X and Z words in the block. The arc center point is defined by the I and K words in the block.

Example

The diagram below illustrates XZ plane selection:

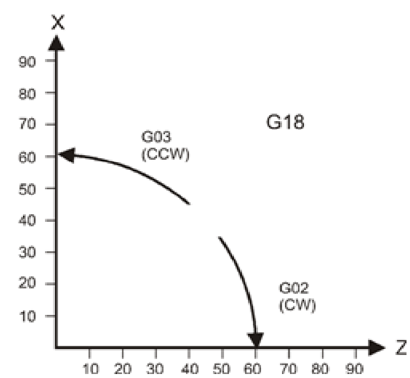


Figure 2–11. G18 XZ Plane Selection

G19 - YZ Plane

G19, the YZ Plane Selection code, sets the plane for circular interpolation modes G02/ G03 - Clockwise/Counterclockwise Arc to the YZ plane.

⇨ G19 is only available for turning centers with Y-axis motion.

Default—No

Modal—Yes

G19 cancels G17 - XY Plane or G18 - XZ Plane (default).

- The X, Y, Z, J, and K words are valid in circular interpolation blocks.
- I words are invalid.
- If an X word is programmed in the circular interpolation block, a helix is generated in the YZ plane.
- The direction of an arc or helix in the YZ plane can be determined by looking at the YZ plane with positive Y to the right and positive Z going up.
- The YZ plane is a right-handed coordinate system (thumb points to positive X, and fingers wrap in counterclockwise direction).
- In G19, the arc end point is defined by the Y and Z words in the block. The arc center point is defined by the J and K words in the block.

Format

The format of the YZ plane selection command is as follows:

G19 Y____ Z____

Example

The diagram below illustrates YZ plane selection:

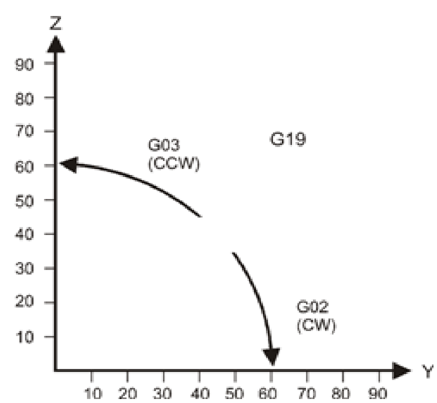


Figure 2–12. G19 YZ Plane Selection

G20 - Inch Mode (default)

G20 sets Inch mode. Program this command in the first block of each part program that uses inches for the dimension.

Default—Yes

Modal—Yes

Cancels—G21 - Millimeter Mode

Format

The default format for Inch mode for different types of blocks is shown in the table below:

Blocks	Default Format	Range
Geometric Entries X, Z, I, K, R	3.4	-845.0000 to 845.0000
Feedrate (Feed/min) and (Feed/rev) F	3.4	000.0001 to 845.0000
Thread Lead (distance/rev) K	2.6	00.000001 to 10.000000

Table 2–6. Inch Mode formats

The default formats are defined as follows:

- 3.4 represents 3 places to the left of the decimal point and 4 places to the right.
- 2.6 represents 2 places to the left of the decimal point and 6 places to the right.

Unit Display Mode

G20 does not affect the Unit Display Mode as viewed in the position or offset screens. The correct units are interpreted from the part program.

G21 - Millimeter Mode

G21 sets Millimeter mode. Program this command in the first block of each part program that uses millimeters for the dimension.

Default—No

Modal—Yes

Cancels—G20 - Inch Mode (default)

Format

The default format for Millimeter mode for different types of blocks is shown in the table below:

Blocks	Default Format	Range
Geometric Entries X, Z, I, K, R	5.3	-21463.000 to 21463.000
Feedrate (Feed/min) and (Feed/rev) F	5.3	0000.001 to 21463.000
Thread Lead (distance/rev) K	3.5	000.00001 to 10.000000

Table 2–7. Millimeter Mode formats

The default formats are defined as follows:

- 5.3 represents 5 places to the left of the decimal point and 3 places to the right.
- 3.5 represents 3 places to the left of the decimal point and 5 places to the right.

Unit Display Mode

G21 does not affect the Unit Display Mode as viewed in the position or offset screens. The correct units are interpreted from the part program.

G28 - Automatic Return to Reference Point

Any point within the machine coordinate system can be selected as the reference point. The return reference point is often used to move the turret to the maximum machine position at the end of the program to get the turret out of the way to load and unload parts.

The G28 Automatic Return To Reference Point command specifies an automatic return to the reference point for the designated axes. An intermediate point can be specified with the X____Y____Z____ parameters. If no intermediate point coordinates are specified, the system uses the previous intermediate point coordinates. If no intermediate point coordinates are specified during the current program execution, the machine returns directly to the reference point.

Modal—No

Format

G28 U____ V____ W____

⇨

- X indicates Absolute X and (u) indicates Incremental X.
- Y indicates Absolute Y and (v) indicates Incremental Y.
- Z indicates Absolute Z and (w) indicates Incremental Z.

Parameters

- V—used with machines configured with Y-axis motion.

⚠ A milling map must be active in order for V parameter to be used, if it is not, the machine will be unable to complete the motion.

For information about milling maps, please refer to the following sections:

- G101 - Axial Surface of the Part - Main Spindle without Linear Y
- G102 - Radial Surface of the Part – Main Spindle without Linear Y
- G103 - Axial Surface of the Part – Main Spindle with Linear Y
- G104 - Radial Surface of the Part – Main Spindle with Linear Y
- G105 - Axial Surface of the Part – Sub-spindle without Linear Y
- G106 - Radial Surface of the Part – Sub-spindle without Linear Y
- G107 - Axial Surface of the Part – Sub-spindle with Linear Y
- G108 - Radial Surface of the Part – Sub-spindle with Linear Y
- G109 - Turning / Lathe Mode

Examples

These parameters specify the absolute or incremental location of the intermediate point in coordinates relative to the current coordinate system. The G28 command is only performed for the axis which follows the G28. For example, if an X value follows the G28, the machine moves to the X reference point, not the Y or Z reference point.

As another example, a typical method to send the turret home using incremental moves is shown below. This example will send the turret to the maximum machine position.

```
G28 U0. W0.
```

G32 - Constant Lead Thread Cutting

G32 allows you to perform single edge ID and OD thread cutting. The cutting amount per depth is held constant, and either straight or tapered threads can be cut using this cycle. Refer to *G78 - Threading Cycle, on page 2 - 64* for information about programming the more complex, multi-pass Threading Cycle for OD and ID threads.

Default—No

Modal—Yes

Cancels:

- G33 - Threading
- G76 - Multiple Threading Cycle
- G77 - Outer Diameter/Inner Diameter Cutting Cycle
- G78 - Threading Cycle
- G79 - Stock Removal in Facing Cycle

The starting angle is measured in 0.001 degree increments. If no decimal point is specified the scale factor is 0.001. For example, Q180000 is equivalent to 180.000 degrees.

This cycle can be interrupted.

- When the **Feed Hold** button is pressed, cutting will continue until the first non-thread cutting move. Then the tool will retract to Z start after completing the lead out move for the current depth of cut.
- When the **Interrupt Cycle** button is pressed, the tool will complete the motion for the G32 move as well as any thread cutting move following that line, and then stop the spindle and turn off the coolant.

The feedrate override is fixed at 100% during thread cutting.

When multiple threading commands are issued in sequence, the control will not wait for a spindle before continuing the next thread command.

Format

G32 X(u) ____ Z(w) ____ F or E ____ Q ____

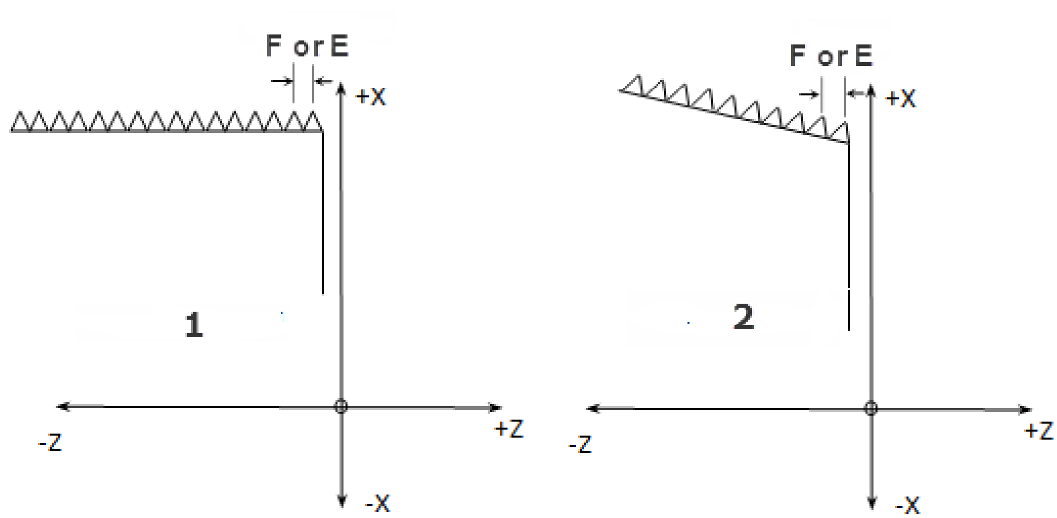
⇨
- X indicates Absolute X and (u) indicates Incremental X.
- Z indicates Absolute Z and (w) indicates Incremental Z.

Parameters

- **X(u)**—X end point
- **Z(u)**—Z end point; only used for tapered threading.
- **F or E**—pitch
- **Q**—thread starting angle

Example

The following figure illustrates G32:



1	Straight Thread
2	Tapered Thread

Figure 2–13. G32 Constant Lead Thread Cutting

G33 - Threading

G33 allows you to program a simple threading cycle. Refer to *G78 - Threading Cycle, on page 2 - 64* for information about programming the more complex, multi-pass Threading Cycle for OD and ID threads,

Modal—No

Cancels:

- G32 - Constant Lead Thread Cutting
- G76 - Multiple Threading Cycle
- G77 - Outer Diameter/Inner Diameter Cutting Cycle
- G78 - Threading Cycle
- G79 - Stock Removal in Facing Cycle

Format

G33 X ____ Z ____ (F ____ or E ____)

Parameters

- **Z**—only used for tapered threading.
- **F**—pitch
- **E**—thread lead

Example—G33

To program a threading move, X and Z and either I or K are required. P (the thread start angle) and F (feedrate) may be specified.

1. Program the thread's endpoint (using Z only, X only, or a combination of X and Z for a taper thread).
2. Program the pitch of the thread using either of these codes:
   - **F** (modal feedrate)
   - **K** (the pitch along the Z axis)
   - **I** (the pitch along the X axis)

   ⇨ If you are programming a tapered thread (both X and Z axes move), program the lead (I or K) for the axis with the longest move.

```
G00 X3.5 Z4 (Position for thread)
G33 Z1 K0.2 (Cut thread)
G01 X3.6 (Pullout of thread)
```

Example—G33 with P Value

You can also start a thread at a given angle. The angle value may range from -360.0000 to 360.0000. A zero angle starts the thread at the index mark of the spindle encoder, and is the default. The P value is not modal and must appear in the same block as the G33. The thread start angle is specified by a P value in a G33 block, such as:

```
G00 X3.5 Z4 (Position for thread)
G33 Z1 K0.2 P180 (Cut thread using 180° thread start angle)
G01 X3.6 (Pullout of thread)
```

Cutter Compensation (G40–G42)

Cutter compensation may be used for two purposes. First, it may be used when the dimensions in the program and the part surface are the same. The system calculates the proper tool path by using the part surface and the tool diameter information.

Second, cutter compensation corrects the difference between the diameters of the tool specified and the tool actually used to cut the part. This situation often occurs when the program originates from an off-line device. Note that the coordinates of those programs are usually tool center line data.

Cutter compensation is based on the direction of travel of the tool. To determine which type of cutter compensation to use, look at the part as if you are moving around the part always keeping the tool ahead of you. Then it becomes obvious whether the tool needs to be on the right or the left of the programmed line or the boundary of the part

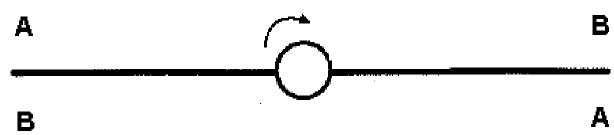


Figure 2–14. G40 Cutter Compensation Off

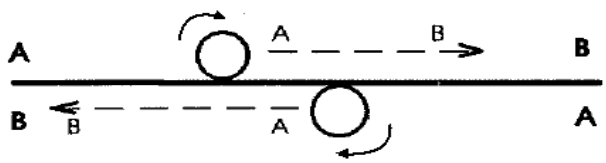


Figure 2–15. G41 Cutter Compensation Left

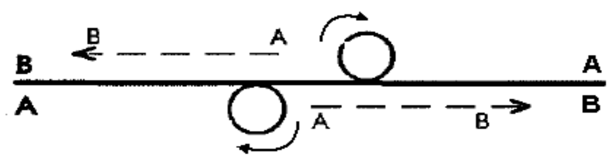


Figure 2–16. G42 Cutter Compensation Right

A	Beginning point of programmed direction (clockwise tool movement from point A to point B)
B	Ending point of programmed direction (clockwise tool movement from point A to point B)

G40 - Cutter Radius Compensation Off (default)

G40 turns off cutter compensation. When G40 is active, the tool center will move along the path defined in the part program. Refer to *Figure 2–14. G40 Cutter Compensation Off, on page 2 - 28* for an illustration of the tool path with G40.

Default—Yes

Modal—Yes

Cancels G41 - Cutter Radius Compensation Left or G42 - Cutter Radius Compensation Right

G40 must be active to perform G53 - Machine Coordinates.

G00 - Rapid Traverse (default) or G01 - Linear Interpolation must be selected in order for this command to cancel the offset compensation. Each axis moves straight (G01) or at rapid traverse (G00) from the point of the old vector at the start point toward the end point. The machine should be in G40 mode before the end of a program. Otherwise, when the program ends in the offset mode, positioning cannot be made to the terminal point of the program, and the tool position will be separated from the terminal position by the vector value.

Examples

Please refer to *Examples—Cutter Compensation, on page 2 - 30*.

G41 - Cutter Radius Compensation Left

G41 turns cutter compensation on to the left of the programmed part profile. When G41 is active, the tool path is offset by the tool radius value stored in the tool table. The offset will be to the left side of the part profile. Refer to *Figure 2–15. G41 Cutter Compensation Left, on page 2 - 28* for an illustration of the tool path with G41.

Default—No

Modal—Yes

Cancels G40 - Cutter Radius Compensation Off (default), G42 - Cutter Radius Compensation Right

Either G00 - Rapid Traverse (default), G01 - Linear Interpolation, or G02/G03 - Clockwise/Counterclockwise Arc must be active.

Examples

Please refer to *Examples—Cutter Compensation, on page 2 - 30*.

G42 - Cutter Radius Compensation Right

G42 turns cutter compensation on to the right of the programmed part profile. When G42 is active, the tool path is offset by the tool radius value stored in the tool table. The offset will be to the right side of the part profile. Refer to *Figure 2–16. G42 Cutter Compensation Right, on page 2 - 28* for an illustration of the tool path with G42.

Default—No

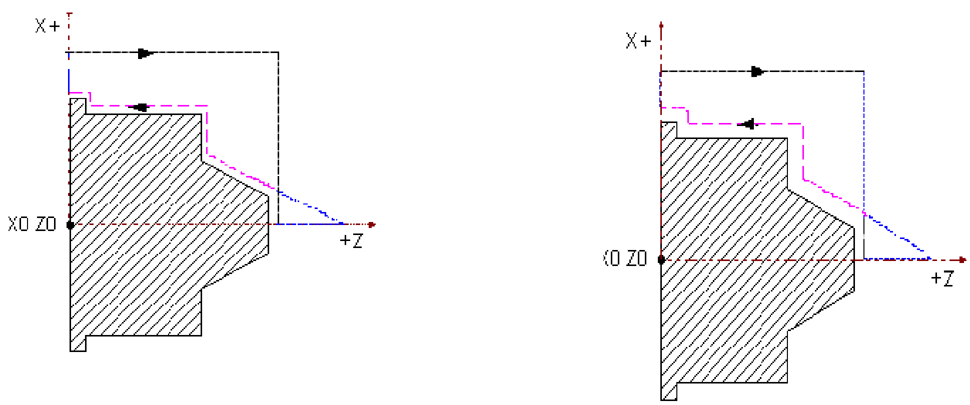
Modal—Yes

Cancels G40 - Cutter Radius Compensation Off (default), G41 - Cutter Radius Compensation Left

Either G00 - Rapid Traverse (default), G01 - Linear Interpolation, or G02/G03 - Clockwise/Counterclockwise Arc must be active.

Examples—Cutter Compensation

The following examples use the part program below. The figures illustrate the tool path with G40 and with G42. Notice that the tool travels very close to the part with G40. The nose radius of the tool programmed in Tool Setup determines the space between the part and the tool.



G40 **G42**

Figure 2–17. Cutter Compensation Off (G40) and Right (G42)

```
N10 (msg, lathe doc sample, abs G42, tool rad 0.1)
N20 G90 G40 G94 T0000 (absolute mode, cutter compensation off, offset 0)
N30 G07 G00 X2.5 T0101 (radius programming, rapid to X, tool 1 offset 1)
N40 Z3 (rapid to Z)
N60 X.5 (rapid to X)
N70 G96 R.5 S350 M03 (CSS, axis at R.5 in, 350 SFM and clockwise)
N80 G01 G95 F.01 X0 (face, IPR 0.01)
N120 G00 Z4 (rapid to Z)
N125 G42 (cutter compensation right)
N130 X.5 Z3 (rapid to XZ)
```

```
N160 G01 X1 Z2
N170 X1.7
N180 Z0.3
N190 X1.9
N200 Z0
N230 G00 G40 X2.5 (cutter compensation off, rapid to X)
N240 Z3 (rapid to Z)
N900 M30 (end of program, rewind)
```

G50 - Set Part Setup or Set Max RPM

G50 used in conjunction with specific parameters provides the following programming functions.

ISNC Mode A

Modal—No

Set Part Setup

G50 sets part setup RPM when an S Parameter is not specified. Please also refer to *G92 - Work Coordinate Offsets / Spindle Max Speed / Multiple Thread Cutting Cycle, on page 2 - 87* for information about using G92 in setting work offsets.

Set Part Setup Format

G50 X ____ Y ____ Z ____

Set Part Setup Parameters

- No **S** parameter is used.

Set Max RPM

G50 in conjunction with an S Parameter sets the Maximum Spindle RPM. Please also refer to *G92 - Work Coordinate Offsets / Spindle Max Speed / Multiple Thread Cutting Cycle, on page 2 - 87* for information about using G92 in setting maximum spindle RPM.

G97 - Direct Spindle Speed (default) must be active.

Set Max RPM Format

G50 S ____

Set Max RPM Parameters

- **S**—spindle speed (for Max RPM)

Set Max RPM Example

```
G97
G50 S1000
G94
```

G53 - Machine Coordinates

This one-shot usage code causes the part program to ignore the following offsets for one block so you can program the axes to a position that you define using machine coordinates:

- active work coordinate offsets (G50 - Set Part Setup or Set Max RPM, G92 - Work Coordinate Offsets / Spindle Max Speed / Multiple Thread Cutting Cycle)
- active tool offset
- active fixture offset

Default—No

Modal—No

Cancels:

- G54-G59 - Work Coordinates (G54 default)
- G54.1 - G54.93 - Aux Work Coordinates
- G92 - Work Coordinate Offsets / Spindle Max Speed / Multiple Thread Cutting Cycle

Before you program G53 in a block, the program must meet the following conditions:

- G40 - Cutter Radius Compensation Off (default) must be active.
- G90 - Absolute Programming (default) must be active if using X instead of U.
- Either G00 - Rapid Traverse (default) or G01 - Linear Interpolation mode must be active.

Format

G53 X(u) ____ Y(v) ____ Z(w) ____

⇨
- X indicates Absolute X and (u) indicates Incremental X.
- Y indicates Absolute Y and (v) indicates Incremental Y.
- Z indicates Absolute Z and (w) indicates Incremental Z.

Example

```
N10 G90 G00 Z0 X0 (Absolute Mode)
N30 G53 Z2 X1 (Move to X1 Z2)
```

⇨ When the N30 block is finished, the machine will return to the G90 - Absolute Programming (default) mode.

G54-G59 - Work Coordinates (G54 default)

These modal commands select the work coordinate systems 1–6. The work coordinate systems are affected by the work offsets, the shift offset, and the G50 - Set Part Setup or Set Max RPM or G92 - Work Coordinate Offsets / Spindle Max Speed / Multiple Thread Cutting Cycle command. Coordinate system 1 is the same as the part setup and it is the default coordinate system. Coordinate systems 1–6 are established by manually entering work offset values for G55–G59 on the Work Offset screen or with the G10 command.

Default—Yes

Modal—Yes

Cancels:

- G53 - Machine Coordinates
- G54-G59 - Work Coordinates (G54 default)
- G54.1 - G54.93 - Aux Work Coordinates
- G92 - Work Coordinate Offsets / Spindle Max Speed / Multiple Thread Cutting Cycle

Use the G10 - Data Setting (Work Offsets, Tool Offsets, or Tool Geometry) command to **change** work coordinate systems, and use the G50 - Set Part Setup or Set Max RPM or G92 - Work Coordinate Offsets / Spindle Max Speed / Multiple Thread Cutting Cycle command to **set** part zero. All six work coordinate systems can be moved an equal distance and direction by using the G50 or G92 command.

Format

The format of the multiple work coordinates command is as follows:

G54 (Select work coordinate system 1)

G55 (Select work coordinate system 2)

G56 (Select work coordinate system 3)

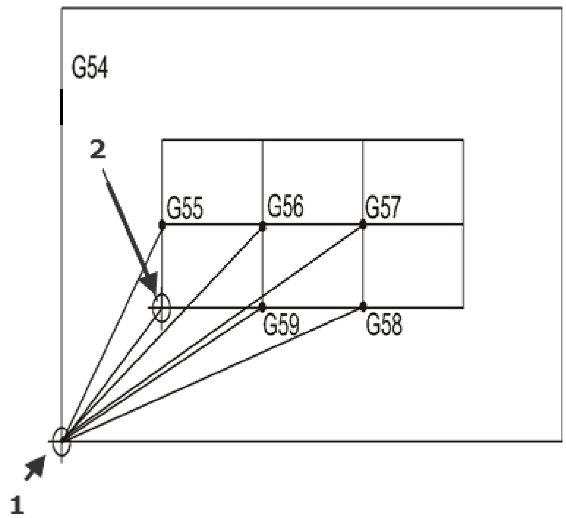
G57 (Select work coordinate system 4)

G58 (Select work coordinate system 5)

G59 (Select work coordinate system 6)

Example

When in NC mode, the Part Setup screen displays up to six work coordinates (G54–G59). As shown below, these codes are used to set multiple part zeroes for multiple parts using the same part program.



1	Machine Zero
2	Part Zero

Figure 2–18. G54-G59 Work Offset G Codes for Multiple Parts

The coordinates defining G54 are the part zero coordinates for the original part defined on the Part Setup screen. Set the X, Z, C, W, and Y values for the G54 to G59 codes. These work offsets are stored in memory, but not with the part program.

The G54 work offsets are the same registers as those in the Part Setup screen for Part Zero X, Z, C, W, and Y. Editing G54 work offsets for multiple coordinate systems updates the part setup for X, Z, C, W, and Y on the Part Setup screen.

G54.1 - G54.93 - Aux Work Coordinates

There are 93 additional X, Z, C, W, and Y work offsets available in NC programming.

Default—No

Modal—Yes

Cancels:

- G53 - Machine Coordinates
- G54-G59 - Work Coordinates (G54 default)
- G92 - Work Coordinate Offsets / Spindle Max Speed / Multiple Thread Cutting Cycle

Format

G54.1 P1- P93 X ____ Y ____ Z ____

Parameters

- **P**—auxiliary work offset

Examples

To access any of these offsets call G code G54.1 Pn, where n is 1 through 93. For example, to **change to** auxiliary work offset 46,

G54.1 P46

To update work offset values, use data setting G code G10 Pn to set the Auxiliary work offsets values. For example, to **update** work offset 46,

G10 P46 X12.5 Y3.0 Z-0.5

G61 - Precision Cornering

Precision cornering causes the subsequent line to come to an exact stop.

Default—No

Modal—Yes

Cancels G64 - Precision Cornering Cancel (default)

G64 - Precision Cornering Cancel (default)

Default—Yes

Modal—Yes

Cancels G61 - Precision Cornering.

G65 - Subprogram Call

The G65 subprogram command has the following form:

G65 P____ L_____ [followed by optional arguments]

The P represents the subprogram number and the L represents the number of iterations that the subprogram must perform. These two methods of argument passing can be used together.

Arguments

In a G65 subprogram call, the local variables in the calling program are not copied to the local variables in the called subprogram. Arguments which follow the G65 command are copied to the local variables in the subprogram as illustrated in the following command:

G65 P5080 A0.0 B8 C2.3 S6 T2 H81 I9 J3.5 K0 Z-1 R.1

The value which follows A is copied to the local variable #1 in the subprogram. *Table 2–8.Macro Mode B Local Variables and Subprogram Arguments, on page 2 - 38* shows the relationships between the subprogram arguments and the local variables in the subprograms.

Multiple Arguments

Multiple I, J, and K arguments can also be used as subprogram arguments. For example, if three I arguments are used in the subprogram call, the first I maps to the #4 variable, the second I maps to the #7 variable, and the third I maps to the #10 variable. The following subprogram call is legitimate:

G65 P2000 A2.3 B3.2 I2.0 J3. K5.4 I3. I5. J2. I6. W3. U3

Only numbers may be used as arguments in a G65 subprogram call; no variables or expressions can be used. If multiple iterations of the subprogram are to be performed, the local variables will be initialized to the same argument values.

Passing Argument Lists to Subprograms in Macro Mode B

There are several methods for passing arguments and parameters to subprograms. The G65 - Subprogram Call and G66 - Modal Subprogram Call subprogram calls allow an argument list to be provided after the G65 and G66, respectively. The user defined M Code and the user defined G Code allow an argument list to be provided after the user defined Code. The argument list consists of various letters followed by values. The values are then stored as local variables within the subprogram.

The table below lists the correspondence between the arguments and the local variables in Macro Mode B. The argument list is optional. Any arguments which are not included in the list are given vacant status.

Macro Mode B			
Local Variables	**Subprogram Arguments**	**Local Variable**	**Subprogram Arguments**
#1	Argument A	#18	Argument R or K5
#2	Argument B	#19	Argument S or I6
#3	Argument C	#20	Argument T or J6
#4	Argument I or I1	#21	Argument U or K6
#5	Argument J or J1	#22	Argument V or I7
#6	Argument K or K1	#23	Argument W or J7
#7	Argument D or I2	#24	Argument X or K7
#8	Argument E or J2	#25	Argument Y or I8
#9	Argument F or K2	#26	Argument Z or J8
#10	Argument I3	#27	Argument K8
#11	Argument H or J3	#28	Argument I9
#12	Argument K3	#29	Argument J9
#13	Argument M or I4	#30	Argument K9
#14	Argument J4	#31	Argument I10
#15	Argument K4	#32	Argument J10
#16	Argument I5	#33	Argument K10
#17	Argument Q or J5		

Table 2–8. Macro Mode B Local Variables and Subprogram Arguments

Layering of Local Variables within Subprogram Calls

M98 subprogram calls use local variables differently from other subprogram calls since the called subprogram does not get a new set of local variables. Changes made to the local variables within the current subprogram will be retained when the calling program is re-instated.

Changes can be made to the local variables within the current subprogram, but when program execution returns to the calling program, the values of the local variables of the calling program are reinstated. The local variables in the subprogram can be changed, however, without affecting the local variables in the calling program. With other subprogram calls, unless an argument list is passed to the subprogram, the local variables are given vacant status.

Specifying Subprogram Iterations

The number of iterations for a subprogram to perform are specified with G65, G66, and M98 subprogram calls.

Using G65 and G66

When making G65 and G66 subprogram calls, the L parameter is used to specify iterations. The maximum number of iterations which can be specified with the G65 and G66 subprogram calls is 999.

Using M98

When making M98 subprogram calls, the P parameter is used to specify iterations as well as the subprogram number. Up to four digits can be used to specify iterations for a maximum of 9999 iterations. Leading zeros are not required when specifying iterations; however, leading zeros are required with a subprogram number that is less than 1000.

G65 Subprogram Example

In the following example a line of holes will be drilled along a line. The type of canned cycle can be determined along with the distance between the holes in both the X and Y axes. The type of canned cycle and various canned cycle parameters can also be set.

ISNC Part Program 1 Inch
BOLT_LN.LNC

```
O4000
T1 M06
M03 G00 G90 X0 Y0 Z0 S1800
/
(B REPRESENTS THE NUMBER OF BOLT HOLES)
(H REPRESENTS THE DESIRED CANNED CYCLE)
(X,Y REPRESENTS THE INCREMENTAL DISTANCE BETWEEN HOLES)
(Z REPRESENTS THE HOLE DEPTH)
(R REPRESENTS THE R PLANE LEVEL)
/
#500=99 (RETURN TO R LEVEL)
G65 P5070 X.5 Y.75 B10 H81 Z-1 Q0. R.1 F20.
M30
(*************************************************)
(BOLT HOLE LINE PATTERN - SUBPROGRAM 5070)
(*************************************************)
O5070
(#2 IS THE NUMBER OF HOLES)
(#11 IS THE CANNED CYCLE NUMBER)
(#26 IS THE HOLE DEPTH)
(#500 IS 99 FOR RETURN TO R LEVEL MODE OR 98 FOR RETURN TO INITIAL POINT)
(#5003 IS THE Z COORDINATE BEFORE THE CANNED CYCLE IS PERFORMED)
/
WHILE [#2GT0] DO250
G91 G#500 G#11 Z#26 Q#17 R[#5003-#18] F#9
G00 X#24 Y#25
#2 = #2-1
END250
M99
```

View the part using the Verify console key to verify that the part is programmed correctly.

Macro Instruction (G65)

G65 Macro instructions are G65 commands which are used to perform mathematical, trigonometric, or program control functions instead of subprogram calls. These commands are intended to support existing programs which use this program format.

The value in the H parameter defines the operation being performed. In all instructions except the GOTO commands H80 through H86, a variable number follows the P parameter. The operation's result is stored in that variable number. In the following command the value stored in variable #100 is added to the number 1 and the resultant value is stored in variable #115.

```
G65 H02 P#115 Q#100 R1
```

For the GOTO commands, the value which follows the P is a positive or negative integer. If the number is negative, the software begins searching for the sequence number at the beginning of the file and continues to search for the sequence number until reaching the end of the file. If the number is positive, the search for the sequence numbers begins with the block after the GOTO command and continues until reaching the end of the file. The software then searches from the beginning of the file until reaching the GOTO command block.

The values which follow Q and R are general purpose parameters which are used in mathematical, logical, or GOTO operations. The specific operations are listed in the following table.

Format

The following is the G65 Macro Instruction format:

```
G65 H ____, P #a, Q #b, R #c,.
```

For H80 through H86, if "a" has a negative value, the software performs a GOTO but begins looking for the sequence number at the beginning of the program. No variables can be used for the P parameter for H80 through H86.

⇨ The G65 Macro Instructions are intended to support existing Macro A programs. Use equations and regular GOTO statements in place of these instructions when developing new programs.

For example use #100 = 4.56 OR #110 instead of G65 #11 P#100 Q4.56 R#110.

And use IF [#150 EQ #160] GOTO 100 instead of G65 H81 P100 Q#150 R#160.

These commands can be used in either Macro A or B mode.

This table lists the Descriptions and Instruction Functions for the H codes used in the G65 macro instructions:

H Code	Description	Instruction Function
H01	Definition, Substitution	#a = #b
H02	Addition	#a = #b + #c
H03	Subtraction	#a = #b - #c
H04	Product	#a = #b * #c
H05	Division	#a = #b / #c
H11	Logical Sum	#a = #b .OR. #c
H12	Logical Product	#a = #b .AND. #c
H13	Exclusive OR	#a = #b .XOR. #c
H21	Square Root	#a = Ö#b
H22	Absolute Value	#a = |#b|
H23	Remainder	#a = #b - trunc (#b/#c) * #c trunc: discard fractions less than 1.
H24	Conversion from BCD to Binary	#a = BIN(#b)
H25	Conversion from binary to BCD	#a = BCD(#b)
H26	Combined Multiplication/Division	#a = (#a * #b) / #c
H27	Combined Square Root 1	#a = Ö(#b2 + #c2)
H28	Combined Square Root 2	#a = Ö(#b2 - #c2)
H31	Sine	#a = #b * SIN(#c)
H32	Cosine	#a = #b * COS (#c)
H33	Tangent	#a = #b * TAN(#c)
H34	Arc tangent	#a = TAN(#b/#c)
H80	Unconditional Divergence (GOTO)	GOTO a
H81	If Statement, Equals	IF #b = #c, GOTO a
H82	If Statement, Not Equal	IF #b ¹ #c, GOTO a
H83	If Statement, Greater Than	IF #b > #c, GOTO a
H84	If Statement, Less Than	IF #b < #c, GOTO a
H85	If Statement, Greater Than/Equals	IF #b >= #c, GOTO a
H86	If Statement, Less Than/Equals	IF #b <= #c, GO TO a

Table 2–9. H Code Descriptions and Instruction Functions for G65 Macro Instructions

Example

This example shows how to use G65 macro instructions in a Bolt Hole Circle subprogram:

ISNC Part Program 1 Inch
G65INST.LNC

```
%
(#600 IS BOLT HOLE CIRCLE X COORD)
(#601 IS BOLT HOLE CIRCLE Y COORD)
(#602 IS BOLT HOLE CIRCLE RADIUS)
(#603 IS STARTING ANGLE)
(#604 IS NUMBER OF BOLT HOLES)
#600 = 0
#601 = 0
#602 = 2.
#603 = 30.
#604 = 12.
T1 M6
G00 X0 Y0 Z0.05
M98 P3030
G00 X0 Y0 Z0.05
M02
O3030
(#110 IS BOLT HOLE COUNTER)
(#112 IS ANGLE OF CURRENT HOLE)
(#113 IS X COORD OF CURRENT HOLE)
(#114 IS Y COORD OF CURRENT HOLE)
N10 G65 H01 P#110 Q0
G65 H22 P#111 Q#604
N20 G65 H04 P#112 Q#110 R360
G65 H05 P#112 Q#112 R#604
G65 H02 P#112 Q#603 R#112
G65 H32 P#113 Q#602 R#112
G65 H02 P#113 Q#600 R#113
G65 H31 P#114 Q#602 R#112
G65 H02 X#114 Q#601 R#114
G90 H00 X#113 Y#114
G81 Z-1. F20.
G80
G65 H02 P#110 Q#110 R1
G65 H84 P-20 Q#110 R#111
M99
```

G66 - Modal Subprogram Call

Default—No

Modal—Yes

Cancels—G67 - Modal Subprogram Call Cancel (default).

Modal subprograms are executed every time a motion is performed (i.e., after a Move command). Use them for performing repetitive tasks at different locations. The repetitive tasks can be put inside a modal subprogram. A subprogram call can be made to a program which contains X, Y, and Z locations and will be executed at each of these locations.

A modal subprogram will not be modal within another modal subprogram. If the modal subprogram contains Move commands, the modal subprogram will not be performed after Move commands within the modal subprogram. This allows Move commands to be contained within modal subprograms.

These methods allow the subprogram call to be modal:

- A Modal Subprogram Call (G66 - Modal Subprogram Call) Command
- A Modal user defined G code

In a G66 Modal subprogram call, the subprogram is repeatedly executed after each Move command until the G67 - Modal Subprogram Call Cancel (default) command is performed.

Modal Subprogram Call (G66) Example

The following program draws a series of squares and rectangles:

NC Part Program 1 Inch
G66.LNC

```
%
(EXAMPLE OF MODAL SUBPROGRAM CALL G66)
(P6010 IS USED AS MODAL SUBPROGRAM)
(THE VALUES AFTER I AND J ARE PASSED TO)
(THE SUBPROGRAM. THE SUBPROGRAM IS ONLY)
(EXECUTED AFTER BLOCKS WITH MOVE COMMANDS.)
X0 Y0 Z0
X5 G66 P6010 I1. J1.5
Y-3
X-5
Y0
(MODAL SUBPROGRAM IS NOW CANCELED WITH G67)
G67
Y3
(THE MODAL SUBPROGRAM IS STARTED AGAIN WITH NEW PARAMETERS.)
X0 G66 P6010 I3.J1.
Y0
Y-2
M02
:6010
(THIS SUBPROGRAM CREATES A SIMPLE BOX SHAPE.)
G91
X#4
Y#5
X-#4
Y-#5
G90
M99
```

View the part using the Verify console key to verify that the part is programmed correctly.

G67 - Modal Subprogram Call Cancel (default)

The G67 command is used to cancel modal subprograms initiated with either the G66 or with a modal user defined G code.

Default—Yes

Modal—Yes

Cancels—G66 - Modal Subprogram Call

G70 - Finishing Cycle

G70 causes the tool to make a finish pass along the profile. Individual feeds and speeds are used in the profile. Remaining roughing stock is removed.

Modal—No

F, S, and T functions specified in Profile blocks using G71 - Stock Removal in Turning, G72 - Stock Removal in Facing, and G73 - Pattern Repeating are ignored until a G70 is active.

Format

G70 P ____ Q ____ F ____ S ____ T ____

Parameters

- **P**—first sequence number of Profile
- **Q**—last sequence number of Profile
- **F**—cutting feedrate
- **S**—spindle speed
- **T**—tool

Example

The following sample program and figure illustrate G70:

```
%
O1234
G21
G54
T303
G0 X52 Z2
G90
G94
G96 S190
G70 P10 Q40 F200
N10 G1 G42 X50 Z0
N20 W-10
N30 G2 U6 Z-13 I3 K0 F150
N40 G40 G1 X60 Z-11
M8
M5
M30
%
```

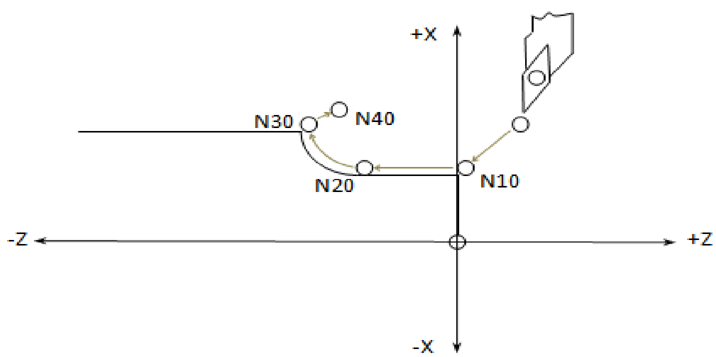


Figure 2–19. G70 Finish Cycle

G71 - Stock Removal in Turning

G71 removes stock in a turning move.

Modal—No

F, S, and T functions specified in Profile blocks using G71 - Stock Removal in Turning, G72 - Stock Removal in Facing, and G73 - Pattern Repeating are ignored until a G70 is active.

G71 allows Linear and Circular interpolation.

Sign of U and W

The sign of the U and W parameters determines the stock removed from point B to C, as shown below:

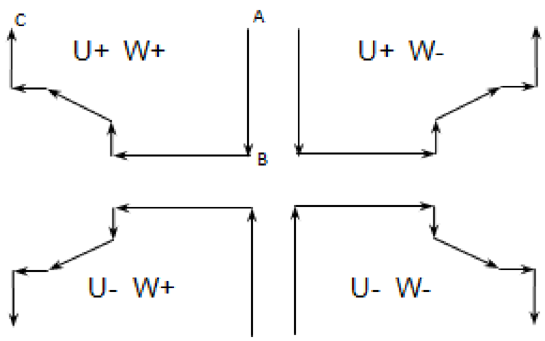


Figure 2–20. Sign of U and W determines stock removal

Format

G71 U ____ R ____

G71 P ____ Q ____ U ____ W ____ F ____ S ____ T ____

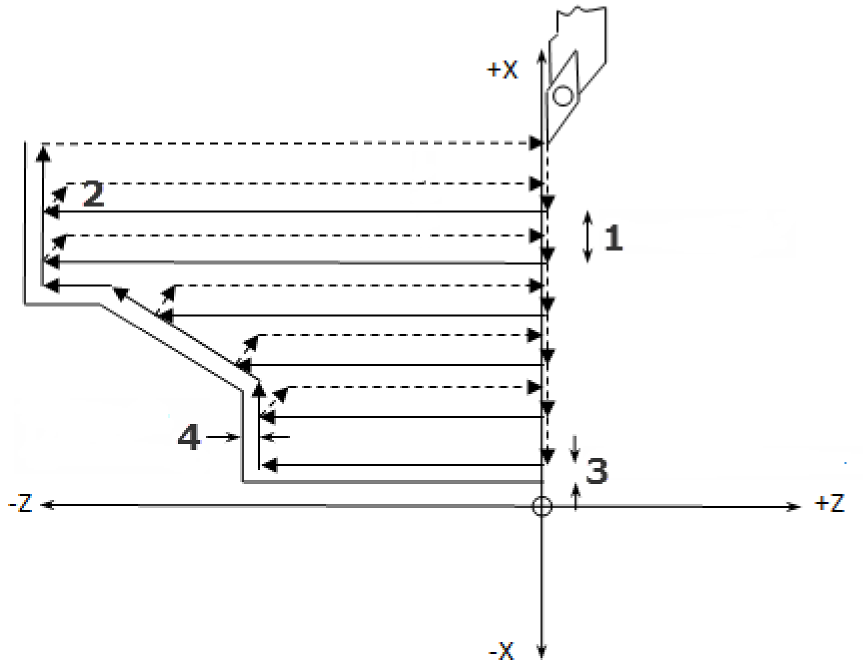
Parameters

- **U**—depth of cut (U/2 specified as diameter)
- **R**—retract distance (retracts at 45° angle)
- **P**—first sequence number of Profile
- **Q**—last sequence number of Profile
- **U**—second line with U is X stock finish (or finish tolerance)
- **W**—Z stock finish (or finish tolerance)
- **F**—cutting feedrate for roughing
- **S**—roughing speed
- **T**—tool with offset xxxx

Example

The following sample program and figure illustrate G71:

```
%
O0000
G0 G21
G54
G28 U0. W0.
T0303
G97 S1700 M3
Z2.54
X104.14
G71 U4. R2
G71 P5 Q10 U2 W3 F200
N5 G1 G42 X57.15 Z0.0
N6 X57.15 Z-25.4
N7 X69.85 S500
N8 X82.55 Z-38.1
N9 Z-44.45 F190
N10 G40 X101.6 S300
G0 X104.14 Z2.54
M9
(FINISH WITH A DIFFERENT TOOL)
G28 U0. W0.
M8
G70 P5 Q10 T1212
G0 X104.14 Z2.54
G28 U0. W0.
M30
%
```



1	Cutting depth
2	Retract distance
3	Finish tolerance in X
4	Finish tolerance in Z

Figure 2–21. G71 Stock Removal in Turning

G72 - Stock Removal in Facing

G72 removes stock in a facing move. The cutting passes are determined by the profile.

Modal—No

F, S, and T functions specified in Profile blocks using G71 - Stock Removal in Turning, G72 - Stock Removal in Facing, and G73 - Pattern Repeating are ignored until a G70 is active.

G72 allows Linear and Circular interpolation.

Format

G72 U ____ R ____

G72 P ____ Q ____ U ____ W ____ F ____ S ____ T ____

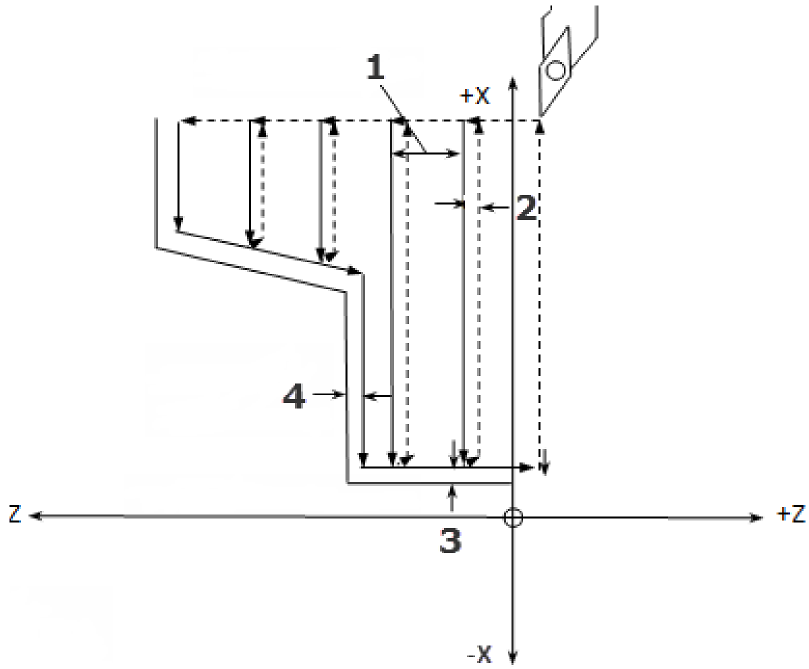
Parameters

- **U**—depth of cut
- **R**—retract distance (retracts at 45° angle)
- **P**—first sequence number of Profile
- **Q**—last sequence number of Profile
- **U**—second line with U is X stock finish (or finish tolerance)
- **W**—Z stock finish (or finish tolerance)
- **F**—cutting feedrate for roughing
- **S**—roughing spindle speed
- **T**—tool with offset xxxx

Example

The following sample program and figure illustrate G72:

```
%
O1000
G21
T0303
G97 S1700 M3
X115
Z2
G72 U6 R1
G72 P5 Q10 U-2 W-2 F250
N5 G1 X115 Z-25
X105
X85
X75 Z-12
X25
Z0.
N10 G1 Z2
M5
M8
M09
M30
```



1	Cutting depth
2	Retract distance
3	Finish tolerance in X
4	Finish tolerance in Z

Figure 2–22. G72 Stock Removal in Facing

G73 - Pattern Repeating

G73 cuts identical profiles in each pass, shifting by the current relief amount as each pass gets closer to the final profile.

Modal—No

F, S, and T functions specified in Profile blocks using G71 - Stock Removal in Turning, G72 - Stock Removal in Facing, and G73 - Pattern Repeating are ignored until a G70 is active.

G73 allows Linear and Circular interpolation.

Format

G73 U ____ W ____ R ____

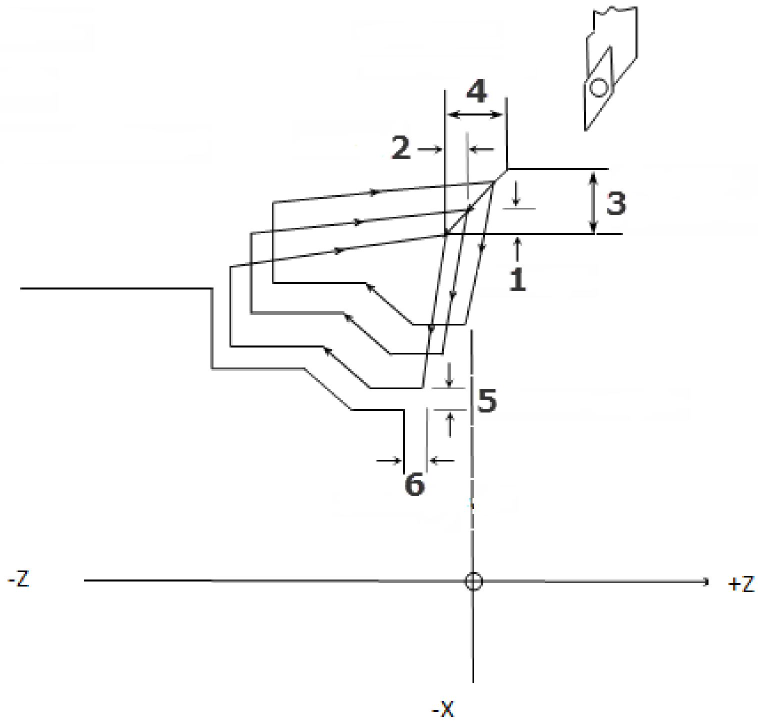
G73 P ____ Q ____ U ____ W ____ F ____ S ____ T ____

Parameters

- **U**—depth of cut in X axis
- **W**—first line with W is depth of cut in Z axis
- **R**—number of repetitions for roughing passes
- **P**—first sequence number of Profile
- **Q**—last sequence number of Profile
- **U**—second line with U is finish allowance in X/2
- **W**—second line with W is finish allowance in Z
- **F**—cutting feedrate for roughing
- **S**—spindle speed
- **T**—tool with offset xxxx

Example

The following figure illustrates G73:



1	X Cut Depth
2	Z Cut Depth
3	X Cut Depth + X Finish Allowance/2
4	Z Cut Depth + Z Finish Allowance
5	X Finish Allowance/2
6	Z Finish Allowance

Figure 2–23. G73 Pattern Repeating

G74 - End Face Peck Drilling

G74 performs grooving in the Z Direction.

Modal—No

Format

G74 R ____

G74 X ____ Z ____ P ____ Q ____ R ____ F ____ S ____ T ____

Parameters

- **R**—retract distance between pecks
- **X**—total X distance X/2
- **Z**—total Z depth
- **P**—peck cutting depth in X
- **Q**—peck cutting depth in Z
- **R**—second line with R is relief distance for retract move
- **F**—cutting feedrate for roughing
- **S**—spindle speed
- **T**—tool with offset xxxx

⇨ **Groove Width** is the starting Z value minus ending Z value plus tool width.

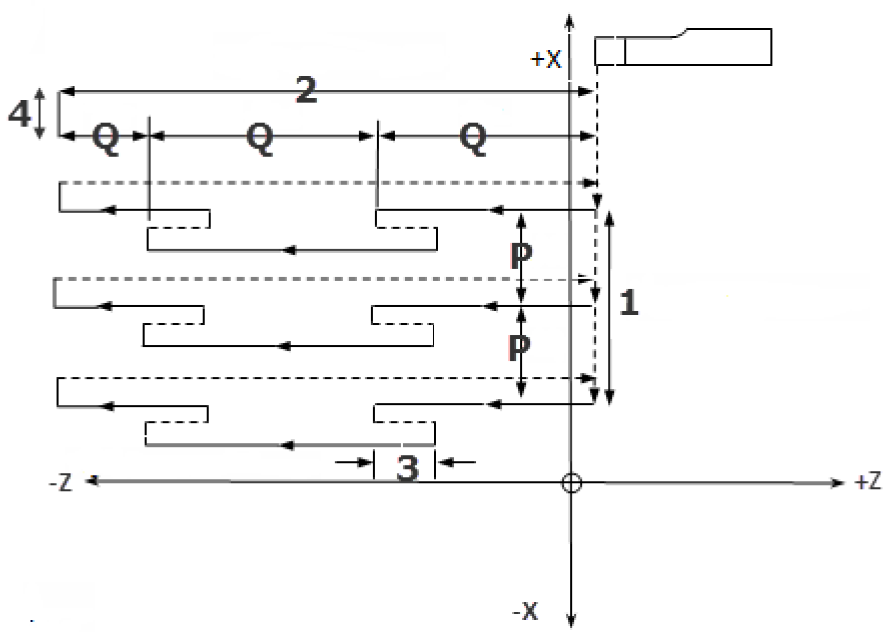
⇨ The X and P parameters are not required when making single grooves.

⇨ If P or Q are integers (no decimal point), the values will be divided by 10000. For example, P10000 becomes P1.0.

Example

The following sample program and figure illustrate G74:

```
%
O1000
G21 G54
T1313
M8
G97 S1700 M3
G0 X90.0 Z3.0
X60.0
G74 R1.0
G74 Z-6. X40.0 P3 Q4 F200
M9
M05
M30
%
```



1	X Total X distance x/2
2	Z Total Z depth
3	R Retract distance
4	R Relief distance for retract move

Figure 2–24. G74 End Face Peck Drilling

G75 - Outer Diameter/Inner Diameter Drilling

G75 performs grooving in the X Direction.

Modal—No

Format

G75 R ____

G75 X ____ Z ____ P ____ Q ____ R ____ F ____ S ____ T ____

Parameters

- **R**—retract distance between pecks
- **P**—peck cutting depth in X
- **Q**—peck cutting depth in Z
- **X**—total distance x/2
- **Z**—total Z depth
- **R**—second line with R is relief distance for retract move
- **F**—cutting feedrate for roughing
- **S**—spindle speed
- **T**—tool with offset xxxx

⇨ **Groove Width** is the Z Start value minus Z End value plus tool width.

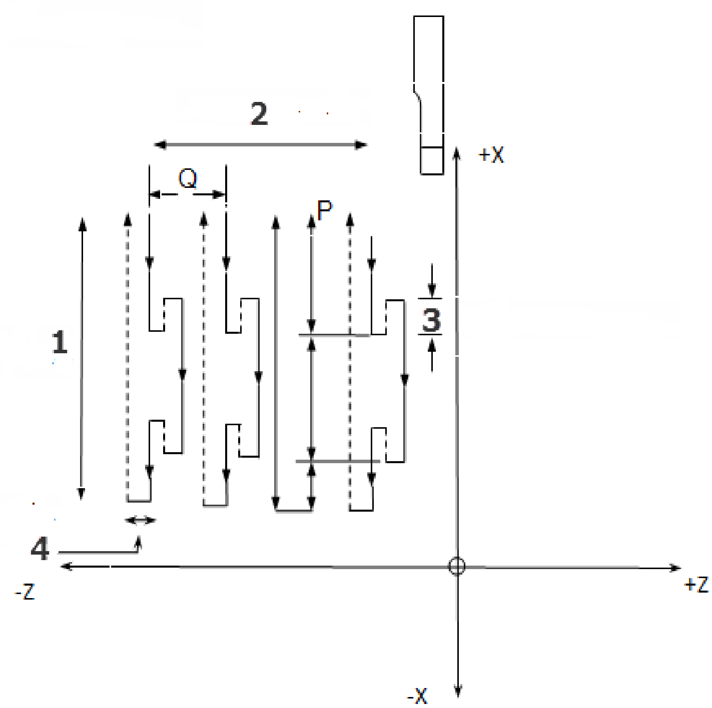
⇨ The Z and Q parameters are not required when making single grooves.

⇨ If P or Q are integers (no decimal point), the values will be divided by 10000. For example, P10000 becomes P1.0.

Example

The following sample program and figure illustrate G75:

```
%
O1000
G21 G54 G90
T0202
M8
G97 S1700 M3
G0 X105 Z3.0
Z-20.0
G75 R1.0
G75 X97.75 Z-35.0 P2 Q3 F200
G28 U0
M9
M05
M30
%
```



1	X Total X Distance X/2
2	Z Total Z Depth
3	R Retract Distance
4	R Relief Distance for Retract Move

Figure 2–25. G75 Outer Diameter/Inner Diameter Drilling

G76 - Multiple Threading Cycle

G76 performs single edge ID and OD thread cutting. The cutting amount per depth is held constant.

Default—No

Modal—Yes

Cancels:

- G32 - Constant Lead Thread Cutting
- G33 - Threading
- G77 - Outer Diameter/Inner Diameter Cutting Cycle
- G78 - Threading Cycle
- G79 - Stock Removal in Facing Cycle

This cycle can be interrupted. If either the **Feed Hold** or **Interrupt Cycle** button is pressed, the tool will retract to Z start after completing the lead-out move for the current depth of cut.

Format

G76 Pxxxxxx Q ____ R ____

G76 X(u) ____ Z(w) ____ R0 P ____ Q ____ F ____

⇨
- X indicates Absolute X and (u) indicates Incremental X.
- Z indicates Absolute Z and (w) indicates Incremental Z.

⇨ Please refer to *Sign of U and R, on page 2 - 62* for information about determining the angle of the taper.

Parameters

- **P**—first line with P, P contains 6 digits. Moving from left to right, the digits represent the following values:
    - The first two digits represent the number of repetitive passes for the finish pass. The value ranges from 1 to 99.
    - The second two digits represent the chamfering distance along the Z axis for the lead-out move. This value ranges from 0.1 to 9.9 in increments of 0.1 expressed as a value from 00 to 99 and is a multiple of the thread lead specified by F.
    - The last two digits represent the angle of the tool tip.

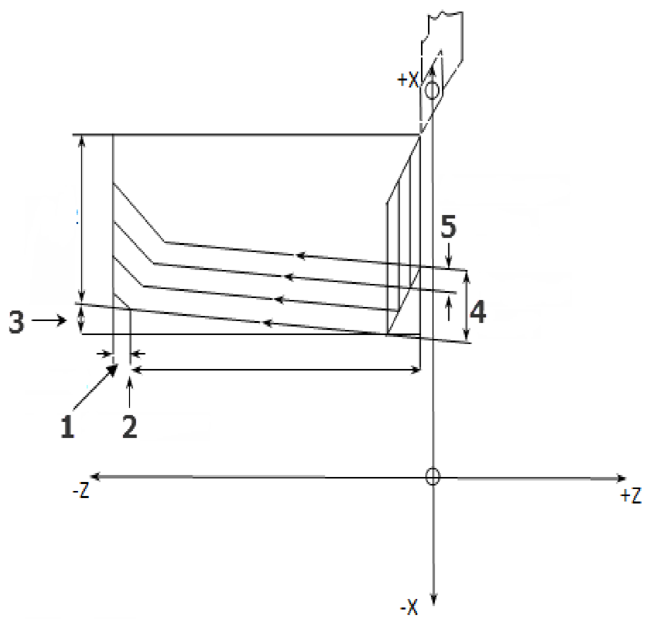
      For example 2 finish passes, a chamfer of 1.5F using a tool tip angle of 55 would be expressed as: P021555
- **Q**—minimum depth of cut.

- **R**—finish allowance. This is the amount of material to leave for the finish pass.
- **X(u)**—minimum diameter for the thread depth.
- **Z(w)**—Z end position of the cycle.
- **R**—second line with R is start thread radius minus end thread radius.
- **P**—second line with P is thread height.
- **Q**—second line with Q is depth of cut for the first pass.
- **F**—thread lead.

⇨ The retract move is approximately 45° or less depending upon servo lag.

Example

The following figure illustrates G76:



1	Chamfer distance along Z axis for lead-out move
2	Z end of thread
3	R start thread radius minus end thread radius
4	P thread height
5	Q depth of cut for first pass

Figure 2–26. G76 - Multiple Threading Cycle

G77 - Outer Diameter/Inner Diameter Cutting Cycle

G77 performs external or internal diameter cutting.

Default—No

Modal—Yes

Cancels:

- G32 - Constant Lead Thread Cutting
- G33 - Threading
- G76 - Multiple Threading Cycle
- G78 - Threading Cycle
- G79 - Stock Removal in Facing Cycle

Sign of U and R

The sign of U and R determines the angle of the taper for canned cycles. In general, the value of R is added to the start point of the cycle.

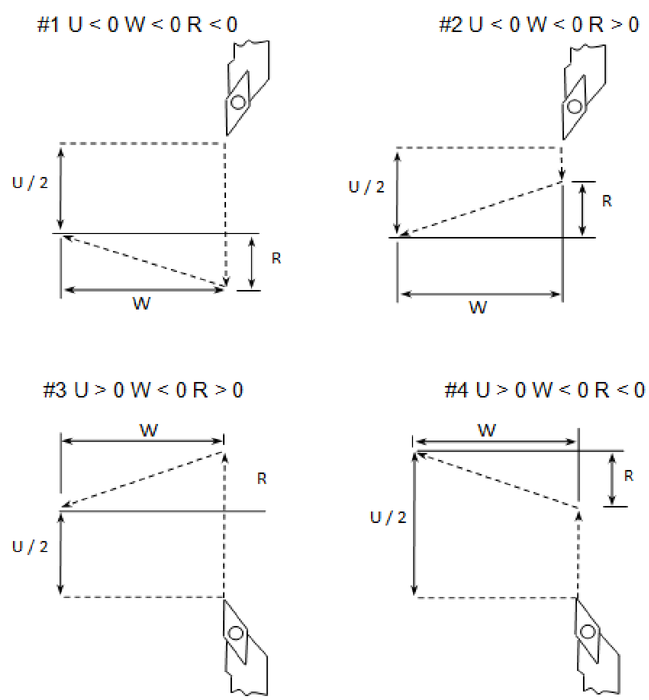


Figure 2–27. Sign of U and R determines the angle of the taper for canned cycles

Format

G77 X(u) ____ Z(w) ____ R ____ F ____

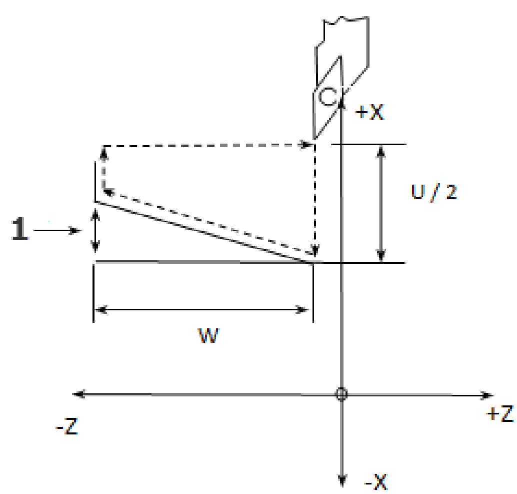
- X indicates Absolute X and (u) indicates Incremental X.
- Z indicates Absolute Z and (w) indicates Incremental Z.

Parameters

- **X(u)**—X end position
- **Z(w)**—Z end position
- **R**—start radius minus end radius
- **F**—cutting feedrate

Example

The following figure illustrates G77:



1	R: start radius minus end radius

Figure 2–28. G77 External Diameter/Internal Diameter Cutting Cycle

G78 - Threading Cycle

G78 has a box shape. The cycle starts at the previous XZ point and goes to the XZ point specified in the current block. This block is modal and allows cutting threads with multiple passes. Please also refer to G92 - Work Coordinate Offsets / Spindle Max Speed / Multiple Thread Cutting Cycle for information about using G92 with Threading Cycles.

Default—No

Modal—Yes

Cancels:

- G32 - Constant Lead Thread Cutting
- G33 - Threading
- G76 - Multiple Threading Cycle
- G77 - Outer Diameter/Inner Diameter Cutting Cycle
- G79 - Stock Removal in Facing Cycle

Format

G78 X ____ Z ____ R ____ F ____

⇨ Please refer to *Sign of U and R, on page 2 - 62* for information about determining the angle of the taper.

Parameters

- **R**—taper distance.
  - **R** is positive if the X Start position is larger than the X End position.
  - **R** is negative if the X Start position is smaller than the X End position.
- **F**—pitch

G79 - Stock Removal in Facing Cycle

G79 removes stock in a facing move. Each cutting pass is programmed for this cycle.

Default—No

Modal—Yes

Cancels:

- G32 - Constant Lead Thread Cutting
- G33 - Threading
- G76 - Multiple Threading Cycle
- G77 - Outer Diameter/Inner Diameter Cutting Cycle
- G78 - Threading Cycle

Format

G79 X(u) ____ Z(w) ____ R ____

- X indicates Absolute X and (u) indicates Incremental X.
- Z indicates Absolute Z and (w) indicates Incremental Z.

Please refer to *Sign of U and R, on page 2 - 62* for information about determining the angle of the taper.

Parameters

- **R**—amount of taper, if any.

Example

G79 is modal so stock removal can be done as follows:

```
G79 X ____ Z ____ R ____
U ____
U ____
W ____
W ____
```

G80 - Canned Cycle Cancel (default)

G80 cancels any active drilling cycle. When G80 is active, X and Z motion commands move the tool directly to the programmed endpoint without performing a drilling cycle. In addition to turning off the drilling cycle, G80 activates Linear Motion at Rapid and clears the modal drilling parameters such as R, Z, Q, and P.

If the drill cycle depth is not programmed after G80, then the drill cycle depth defaults to the R plane.

Default—Yes

Modal—Yes

Cancels:

- G81 - Drill Cycle
- G82 - Drill Cycle with Dwell
- G83 - Face Drilling with Pecks
- G83.1 - Chip Breaker Peck Drilling
- G84 - Face CW Tapping
- G84.2 (or G84 M29) - Rigid CW Tapping
- G85 - Boring Cycle
- G86 - Bore Rapid Out Cycle
- G87 - Side Drilling with Pecks/Dwell
- G88 - Side CW Tapping
- G89 - Side Boring Cycle

G00 - Rapid Traverse (default) cannot be programmed in the same block.

G81 - Drill Cycle

G81 sets drilling as the modal drill cycle.

Default—No

Modal—Yes

Cancels:

- G80 - Canned Cycle Cancel (default)
- G82 - Drill Cycle with Dwell
- G83 - Face Drilling with Pecks
- G83.1 - Chip Breaker Peck Drilling
- G84 - Face CW Tapping
- G84.2 (or G84 M29) - Rigid CW Tapping
- G85 - Boring Cycle
- G86 - Bore Rapid Out Cycle
- G87 - Side Drilling with Pecks/Dwell
- G88 - Side CW Tapping
- G89 - Side Boring Cycle

Format

G81 X ____ Z ____ F ____ S ____

Parameters

- **F**—Feedrate
- **S**—Spindle speed

The tool motion for the drill cycle is shown below.

1. Position to XZ
2. Rapid to reference plane
3. Feed to depth
4. Rapid to reference plane

Example—G81 Drill Cycle

```
G90 (Absolute)
G81 X2 Z-1.2 R.1 F10
```

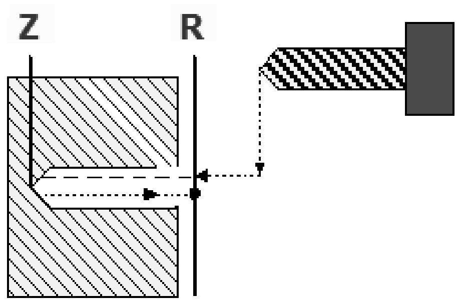


Figure 2–29. G81 Drill Cycle

G82 - Drill Cycle with Dwell

G82 sets drilling with dwell as the modal drill cycle.

Default—No

Modal—Yes

Cancels:

- G80 - Canned Cycle Cancel (default)
- G81 - Drill Cycle
- G83 - Face Drilling with Pecks
- G83.1 - Chip Breaker Peck Drilling
- G84 - Face CW Tapping
- G84.2 (or G84 M29) - Rigid CW Tapping
- G85 - Boring Cycle
- G86 - Bore Rapid Out Cycle
- G87 - Side Drilling with Pecks/Dwell
- G88 - Side CW Tapping
- G89 - Side Boring Cycle

Format

G82 X ____ Z ____ P ____ F ____ S ____

Parameters

- **P**—Seconds to dwell at bottom of hole
- **F**—Feedrate
- **S**—Spindle speed

When P is an integer (no decimal point), the value will be divided by 10000. For example, P10000 becomes P1.0.

The tool motion for this cycle is the same as the drill cycle except a dwell for P seconds occurs at the bottom of the hole.

1. Position to XZ.
2. Rapid to reference plane.
3. Feed to depth.
4. Dwell for **P** seconds or revolutions (depending on FPM or FPR mode).
5. Rapid to reference plane.

Example—G82 Drill Cycle with Dwell

```
G90 (Absolute)
G82 X2 Z-1.2 R.1 F10 P2.2 G99
```

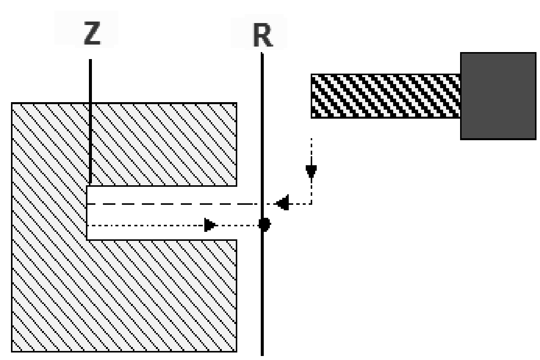


Figure 2–30. G82 Drill Cycle with Dwell

G83 - Face Drilling with Pecks

G83 sets face drilling with pecks as the modal drill cycle.

Default—No

Modal—Yes

Cancels:

- G80 - Canned Cycle Cancel (default)
- G81 - Drill Cycle
- G82 - Drill Cycle with Dwell
- G83.1 - Chip Breaker Peck Drilling
- G84 - Face CW Tapping
- G84.2 (or G84 M29) - Rigid CW Tapping
- G85 - Boring Cycle
- G86 - Bore Rapid Out Cycle
- G87 - Side Drilling with Pecks/Dwell
- G88 - Side CW Tapping
- G89 - Side Boring Cycle

Format

G83 X ____ Z ____ P ____ Q ____ F ____ S ____

Parameters

- **P**—Seconds to dwell at bottom of hole
- **Q**—Distance for feed
- **F**—Feedrate
- **S**—Spindle speed

⇨ If P or Q are integers (no decimal point), the values will be divided by 10000. For example, P10000 becomes P1.0.

The tool motion for the chip break drill cycle is illustrated in *Figure 2–31. G83 Peck Drill Cycle, on page 2 - 71*This cycle requires programming X or Z, R (Retract Clearance Plane), and Q. This cycle causes the tool to feed by the amount programmed in Q, then rapid back to the R plane. Next the tool rapids to a distance slightly above (2.5mm) the last infeed depth, then feeds another Q distance into the part. This sequence repeats until the tool reaches the programmed Z depth.

The following figure illustrates tool motion for G83 peck drill cycle. The short broken lines represent rapid moves; the long broken lines represent feed moves.

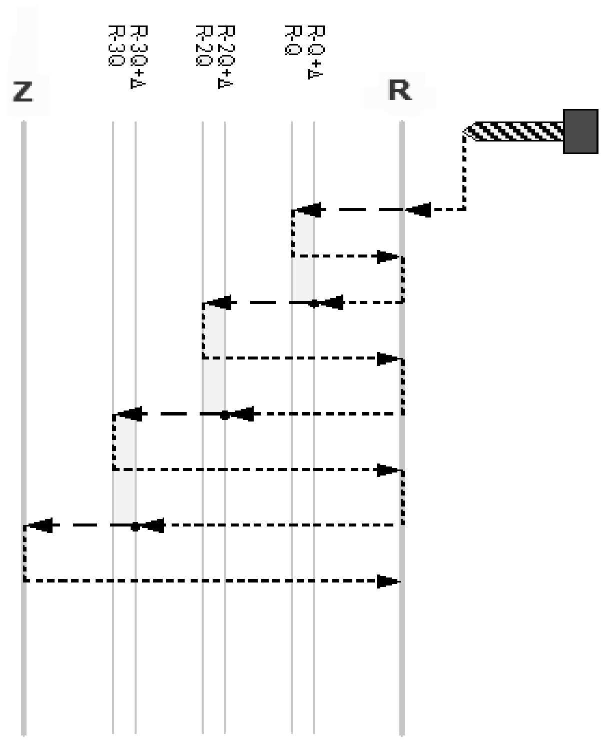


Figure 2–31. G83 Peck Drill Cycle

Example—G83 Peck Drill Cycle

```
G90 (Absolute)
G83 X2.0 Z-1.2 R0.1 Q0.5
```

G83.1 - Chip Breaker Peck Drilling

G83.1 sets chip breaker peck drilling as the modal drill cycle.

Default—No

Modal—Yes

Cancels:

- G80 - Canned Cycle Cancel (default)
- G81 - Drill Cycle
- G82 - Drill Cycle with Dwell
- G83 - Face Drilling with Pecks
- G84 - Face CW Tapping
- G84.2 (or G84 M29) - Rigid CW Tapping
- G85 - Boring Cycle
- G86 - Bore Rapid Out Cycle
- G87 - Side Drilling with Pecks/Dwell
- G88 - Side CW Tapping
- G89 - Side Boring Cycle

Format

G83.1 X ____ Z ____ P ____ Q ____ F ____ S ____

Parameters

- **P**—Seconds to dwell at bottom of hole
- **Q**—Distance for feed
- **F**—Feedrate
- **S**—Spindle speed

⇨ If P or Q are integers (no decimal point), the values will be divided by 10000. For example, P10000 becomes P1.0.

G84 - Face CW Tapping

G84 sets Face CW tapping as the modal drill cycle.

Default—No

Modal—Yes

Cancels:

- G80 - Canned Cycle Cancel (default)
- G81 - Drill Cycle
- G82 - Drill Cycle with Dwell
- G83 - Face Drilling with Pecks
- G83.1 - Chip Breaker Peck Drilling
- G84.2 (or G84 M29) - Rigid CW Tapping
- G85 - Boring Cycle
- G86 - Bore Rapid Out Cycle
- G87 - Side Drilling with Pecks/Dwell
- G88 - Side CW Tapping
- G89 - Side Boring Cycle

Format

G84 X ____ Z ____ P ____ Q ____ F ____ S ____

Parameters

- **P**—Seconds to dwell at bottom of hole
- **Q**—Distance for feed
- **F**—Feedrate
- **S**—Spindle speed

⇨ If Q is an integer (no decimal point), the value will be divided by 10000. For example, Q10000 becomes Q1.0.

The tool motion for this cycle is described below. The feedrate and spindle override controls are forced to 100% are disabled while this cycle is active.

⚠ To stop tool motion when the tap is in the bore, either press the Emergency Stop button or use Retract.

The control will not respond to a motion-stop command until the tool returns to the reference plane.

This drawing shows right-hand tapping tool motion:

1. Tool rapids to R plane.
2. Tool feeds CW to the programmed depth.
3. Spindle reverses direction (CW).
4. If **G94** is active, the axis will dwell for P seconds; if **G95** is active, the axis will dwell for P revolutions.
5. Tool feeds to R plane.
6. Spindle reverses direction (CCW).

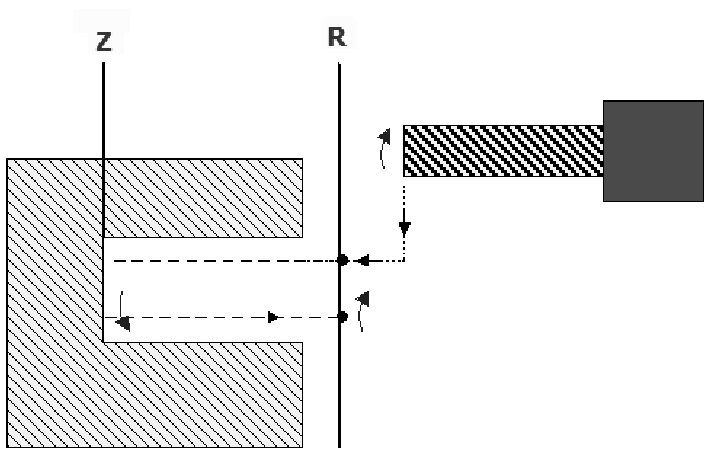


Figure 2–32. G84 Face CW Tapping

G84.2 (or G84 M29) - Rigid CW Tapping

Rigid tapping allows the same hole to be tapped repeatedly with precision. The rigid tapping feature increases accuracy by synchronizing the rotation of the spindle with the feed of the Z axis. G84.2 is used for right-handed tapping. M29 is required for Rigid Tapping.

Default—No

Modal—Yes

Cancels:

- G80 - Canned Cycle Cancel (default)
- G81 - Drill Cycle
- G82 - Drill Cycle with Dwell
- G83 - Face Drilling with Pecks
- G84 - Face CW Tapping
- G85 - Boring Cycle
- G86 - Bore Rapid Out Cycle
- G87 - Side Drilling with Pecks/Dwell
- G88 - Side CW Tapping
- G89 - Side Boring Cycle

Format

G84.2 (or G84 M29) X ____ Z ____ P ____ Q ____ F ____ S ____

Parameters

- **P**—Seconds to dwell at bottom of hole
- **Q**—Distance for feed
- **F**—Feedrate
- **S**—Spindle speed

⇨ If P or Q are integers (no decimal point), the values will be divided by 10000. For example, P10000 becomes P1.0.

G85 - Boring Cycle

The Boring cycle provides a feed-in and feed-out sequence suitable for boring.

Default—No

Modal—Yes

Cancels:

- G80 - Canned Cycle Cancel (default)
- G81 - Drill Cycle
- G82 - Drill Cycle with Dwell
- G83 - Face Drilling with Pecks
- G84 - Face CW Tapping
- G84.2 (or G84 M29) - Rigid CW Tapping
- G86 - Bore Rapid Out Cycle
- G87 - Side Drilling with Pecks/Dwell
- G88 - Side CW Tapping
- G89 - Side Boring Cycle

Format

G85 X ____ Z ____ P ____ F ____ S ____

Parameters

- **P**—Seconds to dwell at bottom of hole
- **F**—Feedrate
- **S**—Spindle speed

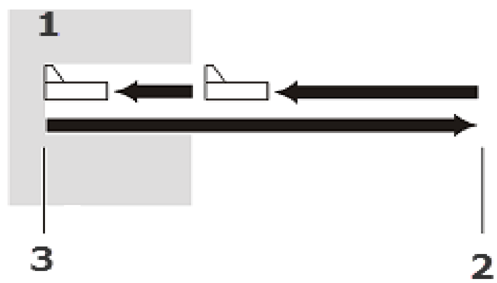
The face boring cycle moves the axes in this manner:

1. The spindle should already be switched on using an M3 code.
2. The tool is positioned over the hole location.
3. At the G85, the spindle feeds to Z Bottom as specified.
4. At Z Bottom, the spindle feeds to the Z Start position.

⇨ It is possible to have an XY position move with the G85 code.

Example

The diagram below illustrates tool movement for the Boring cycle (G85):



1	Stock
2	Z Start; Return Point
3	Z Bottom

Figure 2–33. G85 Tool Movement for the Boring Cycle

G86 - Bore Rapid Out Cycle

The Bore Rapid Out canned cycle is a feed-in, rapid-out sequence. The spindle stops at the bottom of the hole and is retracted at the rapid traverse rate.

Default—No

Modal—Yes

Cancels:

- G80 - Canned Cycle Cancel (default)
- G81 - Drill Cycle
- G82 - Drill Cycle with Dwell
- G83 - Face Drilling with Pecks
- G84 - Face CW Tapping
- G84.2 (or G84 M29) - Rigid CW Tapping
- G85 - Boring Cycle
- G87 - Side Drilling with Pecks/Dwell,
- G88 - Side CW Tapping
- G89 - Side Boring Cycle

Format

G86 X ____ Z ____ P ____ F ____ S ____

Parameters

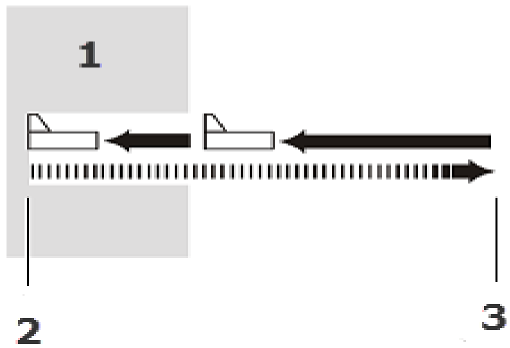
- **P**—Seconds to dwell at bottom of hole
- **F**—Feedrate
- **S**—Spindle speed

The Bore Rapid Out canned cycle moves the axes in this manner with the spindle switched on:

1. The tool is positioned at the Initial Z location and moves at the rapid speed to XY if it is in the block.
2. The spindle bores down to Z Bottom at the specified feedrate.
3. The spindle turns off.
4. The spindle moves up to Z Start at the rapid speed.
5. The spindle turns on.

Example

This diagram illustrates tool movement for the Bore Rapid Out cycle:



1	Stock
2	Z Start (Basic); Return Point (Industry standard)
3	Z Bottom Spindle Stop

Figure 2–34. G86 Tool Movement for the Bore Rapid Out Cycle

G87 - Side Drilling with Pecks/Dwell

Default—No

Modal—Yes

Cancels:

- G80 - Canned Cycle Cancel (default)
- G81 - Drill Cycle
- G82 - Drill Cycle with Dwell
- G83 - Face Drilling with Pecks
- G84 - Face CW Tapping
- G84.2 (or G84 M29) - Rigid CW Tapping
- G85 - Boring Cycle
- G86 - Bore Rapid Out Cycle
- G88 - Side CW Tapping
- G89 - Side Boring Cycle

Format

G87 X ____ Z ____ P ____ Q ____ F ____ S ____

Parameters

- **P**—Seconds to dwell at bottom of hole
- **Q**—Distance for feed
- **F**—Feedrate
- **S**—Spindle speed

⇨ If P or Q are integers (no decimal point), the values will be divided by 10000. For example, P10000 becomes P1.0.

G88 - Side CW Tapping

Default—No

Modal—Yes

Cancels:

- G80 - Canned Cycle Cancel (default)
- G81 - Drill Cycle
- G82 - Drill Cycle with Dwell
- G83 - Face Drilling with Pecks
- G84 - Face CW Tapping
- G84.2 (or G84 M29) - Rigid CW Tapping
- G85 - Boring Cycle
- G86 - Bore Rapid Out Cycle
- G87 - Side Drilling with Pecks/Dwell
- G89 - Side Boring Cycle

Format

G88 X ____ Z ____ P ____ F ____ S ____

Parameters

- **P**—Seconds to dwell at bottom of hole
- **F**—Feedrate
- **S**—Spindle speed

⇨ If P is an integer (no decimal point), the value will be divided by 10000. For example, P10000 becomes P1.0.

G89 - Side Boring Cycle

Default—No

Modal—Yes

Cancels:

- G80 - Canned Cycle Cancel (default)
- G81 - Drill Cycle
- G82 - Drill Cycle with Dwell
- G83 - Face Drilling with Pecks
- G84 - Face CW Tapping
- G84.2 (or G84 M29) - Rigid CW Tapping
- G85 - Boring Cycle
- G86 - Bore Rapid Out Cycle
- G87 - Side Drilling with Pecks/Dwell
- G88 - Side CW Tapping

Format

G89 X ____ Z ____ F ____ S ____

Parameters

- **F**—Feedrate
- **S**—Spindle speed

G90 - Absolute Programming (default)

G90 applies to Macro Mode B and establishes a modal setting that tells the control to interpret all XZ endpoint dimensions in absolute coordinates. G90 Absolute Programming specifies that all tool endpoint positions are measured from the current part zero position.

Default—Yes

Modal—Yes

Cancels G91 - Incremental Programming (Mode B)

You can use G90 absolute or G91 - Incremental Programming dimensions directly from the blueprint. Most part programs should begin with a G90 Absolute command that moves the tool to a fixed position on the machine. Once the start point is defined, you can study the part blueprint and determine whether to use G90 Absolute or G91 Incremental dimensions.

Most part programs should begin with a G90 command to establish part zero. This setting remains active until you program G91 to select incremental dimension programming. A program may switch between G90 Absolute mode and G91 Incremental mode at any time.

Absolute dimensions are measured from part zero. The figure shows the path that the tool center will follow.

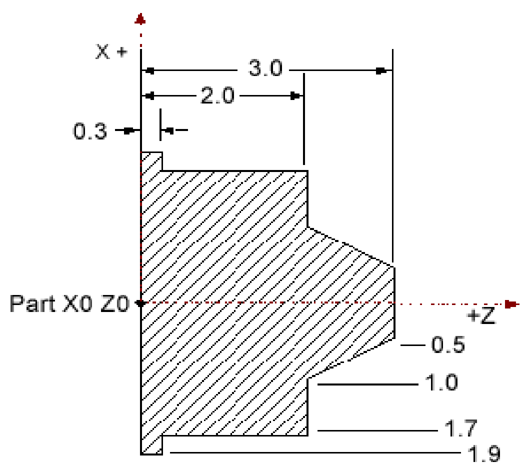


Figure 2–35. G90 Absolute Dimensions (Mode B)

Example

The following sample NC program is in Absolute mode (Mode B):

```
N10 (msg, lathe doc sample, abs G42, tool radius 0.1)
N20 G90 G40 G94 T0000 (absolute mode, cutter compensation off, offset 0)
N30 G07 G00 X2.5 T0101 (radius programming, rapid to X, tool 1 offset 1)
N40 Z3 (rapid to Z)
N60 X.5 (rapid to X)
N70 G96 S350 M03 (CSS, X axis at 5 in, 350 SFM and clockwise)
N80 G01 G95 F.01 X0 (face, IPR 0.01)
N120 G00 Z4 (rapid to Z)
N125 G42 (cutter compensation right, skip if block del on)
N130 X.5 Z3 (rapid to XZ)
N160 G01 X1 Z2 (taper)
N170 X1.7 (face)
N180 Z0.3 (turn)
N190 X1.9 (face)
N200 Z0 (turn)
N230 G00 G40 X2.5 (cutter compensation off, rapid to X, skip if block del
on)
N240 Z3 (rapid to Z)
N900 M30 (end of program, rewind)
```

G91 - Incremental Programming

G91 applies to Macro Mode B and establishes a modal setting that tells the control to interpret all XZ endpoint dimensions in incremental coordinates. G91 Incremental Programming specifies that all tool endpoint positions are measured from the tool's position at the start of the motion. This setting remains active until you program G90 - Absolute Programming (default) to select absolute dimension programming. A program may switch between G90 Absolute and G91 Incremental at any time within a part program.

Default—No

Modal—Yes

Cancels G90 - Absolute Programming (default) (Mode B)

Incremental dimensions specify the distance that the tool must move during each block (i.e. the distance from the start of the move to the end of the move).

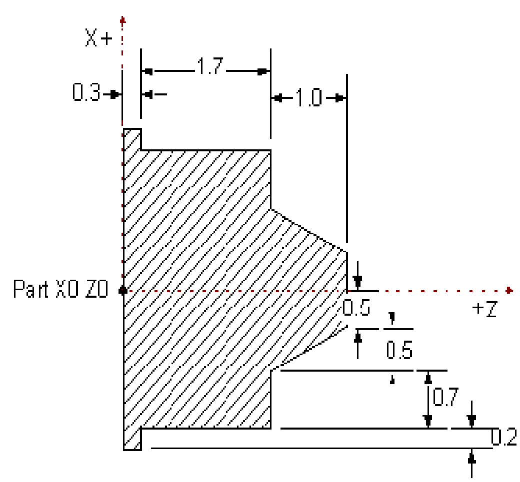


Figure 2–36. G91 Incremental Dimensions (Mode B)

Example

The following sample NC program changes from G90 to G91 (Mode B):

```
N10 (msg, lathe doc sample inc G42 tool rad 0.1)
N20 G90 G40 G94 T0000 (absolute mode, cutter compensation off, offset 0)
N30 G07 G00 X2.5 T0101 (radius programming, rapid to X, tool 1 offset 1)
N40 Z3 (rapid to Z)
N60 G91 X-2 (incremental mode, rapid to X)
N70 G96 S350 M03 (CSS, X axis at 5 in, spindle speed 350 and clockwise)
N80 G01 G95 F.01 X-.5 (face, RPM 0.01)
N120 G00 Z1 (rapid to Z)
N125 G42 (cutter compensation right, skip if block del on)
N130 X.5 Z-1 (rapid to XZ)
N160 G01 X.5 Z-1 (taper)
N170 X.7 (face)
N180 Z-1.7 (turn)
N190 X.2 (face)
N200 Z-.3 (turn)
/
N230 G00 G40 X.6 (cutter compensation off, rapid to X, skip if block del on)
N240 Z3 (rapid to Z)
N900 M30 (end of program, rewind)
```

G92 - Work Coordinate Offsets / Spindle Max Speed / Multiple Thread Cutting Cycle

ISNC Mode B

Default—No

Modal—Yes

Cancels:

- G53 - Machine Coordinates
- G54-G59 - Work Coordinates (G54 default)
- G54.1 - G54.93 - Aux Work Coordinates

G92 used in conjunction with specific parameters provides the following programming functions:

Work Coordinate Offsets

G92 lets you establish the part program coordinates at the current position without generating any tool motion.

G92 Work Coordinates Offsets cannot include X and Z coordinates.

Work Coordinate Offsets Format

G92

Spindle Max Speed

G92 can also set a maximum spindle speed for direct spindle speed or constant surface speed mode when an S parameter is used.

G92 Spindle Max Speed cannot include X and Z coordinates.

Spindle Max Speed Format

G92 S____

Spindle Max Speed Parameters

- **S**—spindle speed

Spindle Max Speed Example

```
G92 S5000 (prevent the spindle from exceeding 5000 rpm)
```

Multiple Thread Cutting Cycle

G92 can perform a multiple thread cutting cycle when an F parameter is used.

Please refer to *Sign of U and R, on page 2 - 62* for information about determining the angle of the taper.

Multiple Thread Cutting Cycle Format

G92 X(u) ____ Z(w) ____ R or I ____ F ____

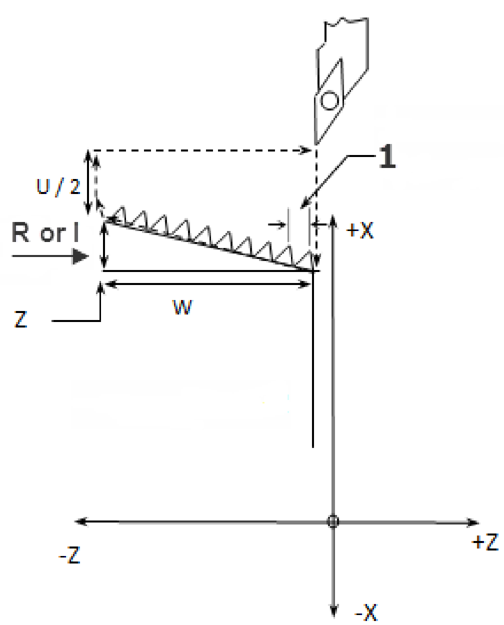
- X indicates Absolute X and (u) indicates Incremental X.
- Z indicates Absolute Z and (w) indicates Incremental Z.

Multiple Thread Cutting Cycle Parameters

- **X(u)**—minimum thread depth diameter
- **Z(w)**—Z end cycle position
- **R or I**—thread radius start minus thread radius end
- **F**—cutting feedrate

Multiple Thread Cutting Cycle Example

The following figure illustrates G92 Multiple Thread Cutting Cycle:



1	Lead is based on commanded spindle RPM and F: cutting feedrate
R or I	Thread radius start minus thread radius end

Figure 2–37. G92 Multiple Thread Cutting—Tapered Thread

G93 - Inverse Time Feedrate

G93 can be specified to change the feedrate as a function of time and distance. If the time is unchanged but the distance changes, then the actual feedrate will change proportionally. The time is computed by dividing one by the Inverse Time programmed. The actual feedrate is the distance divided by the time.

Default—No

Modal—Yes

Cancels:

- G94 - Feed per Minute (default)
- G95 - Feed per Revolution

The feedrate must be specified for every move.

If the inverse time value falls outside the contouring limits of the machine, then the control will instead use the minimum or maximum value.

Format

G93 X____ Z____ F____

The format for Inverse Time is F6.3 (maximum of six digits before the decimal point and maximum of three digits after the decimal point) and the units are minutes. Feedrates of up to 999999.999 can be programmed using G93.

Parameters

- **F**—feedrate as a function of time and distance

Example

Time is 1/10.0min = 0.1min

Actual Feedrate for first line is 5.0in/0.1 min = 50 ipm.

Actual Feedrate for second line is 7.0in/0.1 min = 70 ipm.

```
G93 G1 X5.0 F10.0
Y7.0 F10.0
```

G94 - Feed per Minute (default)

G94 puts the control into feed per minute mode. When G94 is active, all F feedrate values are interpreted in inch/minute (G20 - Inch Mode (default)) or millimeter/minute (G21 - Millimeter Mode) units. Any dwell command that is programmed while G94 is active will be interpreted as number of seconds to dwell.

Default—Yes

Modal—Yes

Cancels:

- G93 - Inverse Time Feedrate
- G95 - Feed per Revolution

Format

G94 X____ Z____ F____

Parameters

- **F**—feedrate in inch/minute or millimeter/minute

G95 - Feed per Revolution

G95 puts the control into feed per revolution mode. When G95 is active, all F feedrate values are interpreted in inch/revolution (G20 - Inch Mode (default)) or millimeter/revolution (G21 - Millimeter Mode) units. Any dwell command that is programmed while G95 is active will be interpreted as number of spindle revolutions to dwell.

Default—No

Modal—Yes

Cancels:

- G93 - Inverse Time Feedrate
- G94 - Feed per Minute (default)

Format

G95 X____ Z____ F____

Parameters

- **F**—feedrate in inch/revolution or millimeter/evolution

G96 - Constant Surface Speed (CSS)

G96 sets feet per minute or meters per minute as the modal spindle command mode. When G96 is programmed, the spindle speed will increase as the tool tip moves toward the spindle centerline and decrease as the tool moves away from the spindle centerline. When G96 is active, the programmed S value is read as a feet per minute or meters per minute command.

Default—No

Modal—Yes

Cancels G97 - Direct Spindle Speed (default)

To specify a radius of the tool tip that corresponds to the surface speed given in S, program the R value in the same block as G96.

- If you do not program an R value or the R value is 0, the control uses the X-axis position as the radius.
- If an R value has been previously programmed, and you program a G96 without an R value, the previously programmed R value will be used.

Active Programming Mode	R Value
G07 - Radius Programming	distance from the tool tip to the spindle centerline (radius)
G08 - Diameter Programming (default)	twice the distance from the tool tip to the spindle centerline (diameter)

Table 2–10. R Values and G96 Programming

G97 - Direct Spindle Speed (default)

G97 sets direct RPM as the modal spindle command mode. When G97 is active, the programmed S value is revolutions per minute.

Default—Yes

Modal—Yes

Cancels G96 - Constant Surface Speed (CSS)

G98 - Return to Initial Level (default)

When G98 is active, all G81-G89 drill cycles command the tool to return to the Z axis coordinate where the tool was when the cycle was initiated. If the initial Z coordinate is changed for subsequent cycles, the Z coordinate for the first cycle after G80 - Canned Cycle Cancel (default) is used as the reference plane.

Default—Yes

Modal—Yes

Cancels G99 - Return to R Point Level

Examples

In the following example, the tool will rapid to Z10 in the second block even though Z20 (init plane) is specified:

```
N005 G98
N010 G00 Z10
N020 G81 Z5 F60 (feed to Z5, rapid to Z10)
N030 G00 Z20
N040 G81 Z15 (rapid to Z10, feed up to Z15, rapid to Z10)
N050 G80
```

In the following example, to return to the init plane, **G98** requires a **G80** Cancel Drill Cycle to change to the init plane.

```
N005 G98
N010 G0 Z10
N020 G81 Z5 F60 (feed to Z5, rapid to Z10)
N030 G80 Z20
N040 G81 Z15 (rapid to Z10, feed up to Z15, rapid to Z20)
N050 G80
```

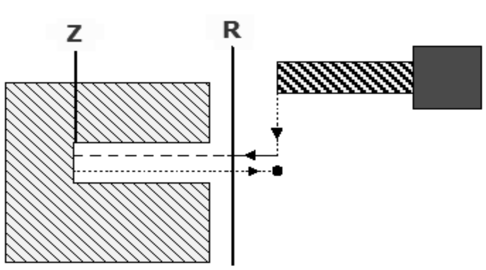


Figure 2–38. G98 Drill Cycle Initial Level Return (default)

G99 - Return to R Point Level

When G99 is active, all G81-G89 drill cycles command the tool to return to the R Reference plane at the end of the drill cycle.

Default—No

Modal—Yes

Cancels G98 - Return to Initial Level (default)

Example

The tool returns to the R Reference Plane.

```
N005 G99
N020 G81 Z5 (feed to Z5, rapid to R)
```

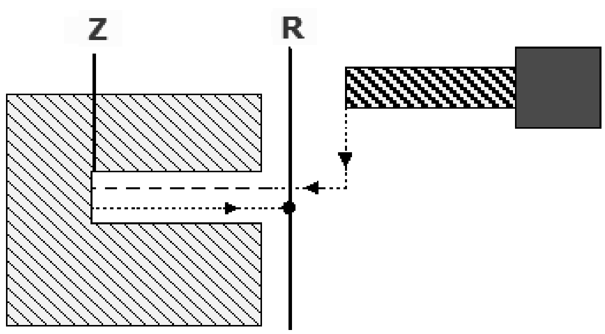


Figure 2–39. G99 General Drill Cycle Tool Motion

G101 - Axial Surface of the Part - Main Spindle without Linear Y

G101 sets up the machine for Axial Milling without Linear Y Motion and activates the coordinate map shown below.

Default—No

Modal—Yes

Cancels—G102, G103, G104, G105, G106, G107, G108, G109

Example

```
G101 (AXIS MAP)
M131 (MILLING MODE)
G00 Z.25
```

This part-relative, tool motion map is used when the Y Axis travel is inadequate for the geometry of the feature. The Y-axis positions are accomplished by coordinating the C axis and X axis motion. On TMM machines this map is always used for Axial Milling.

⚠ This map is relative to the TMX MY and TMX MYS Main Spindle Only. The Axial Milling map for the TMX MYS Sub-spindle uses G105.

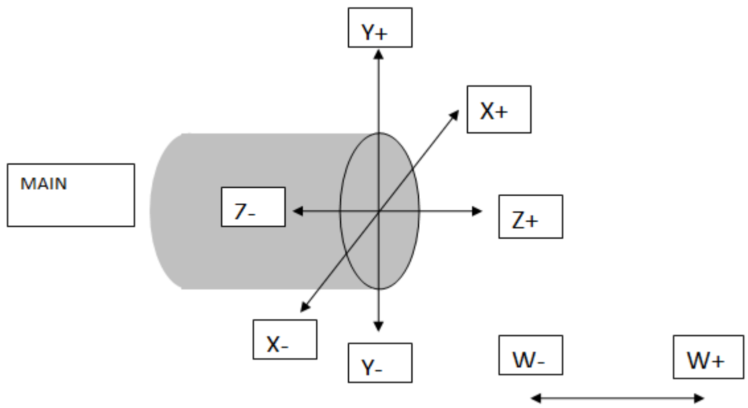


Figure 2–40. G101 TMX Axis Coordinates

G102 - Radial Surface of the Part – Main Spindle without Linear Y

G102 sets up the machine for Radial Milling without Linear Y Motion and activates the coordinate map shown below.

Default—No

Modal—Yes

Cancels—G101, G103, G104, G105, G106, G107, G108, G109

Example

```
G102 (AXIS MAP)
M131 (MILLING MODE)
G25 Q4. (DIAMETER OF PART)
G00 X.25
```

G25 Q must be used to establish the Z part-relative location. The surface of the part is Z.0, and a positive Z value is above the part in the air; a negative Z value moves into the stock.

This part-relative, tool motion map is used for wrapping the Y Axis coordinate system around the circumference of the part. The bottom surface of the feature will be parallel to the circumference of the part. On the TMM machine this map is always used for Radial Milling.

> ⚠ This map is relative to the TMX MY and TMX MYS Main Spindle only. The Radial Milling map for the TMX MYS Sub-spindle uses G106.
>
> This map swaps the Z and X axes from G101.
>
> The Y-axis travel limits are limited to +/- 800 inches (20320 mm) when using this map.

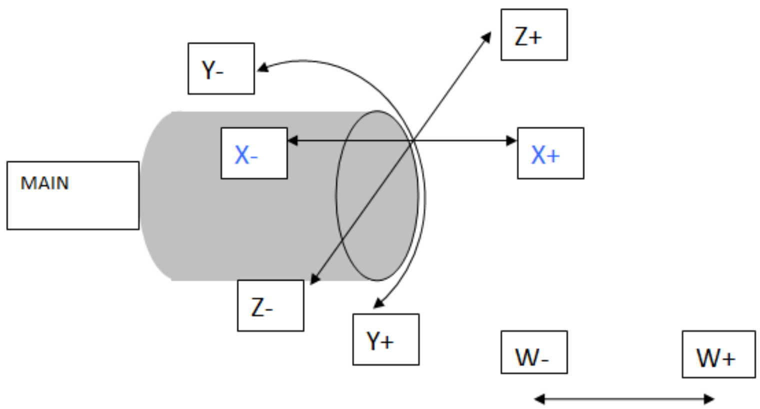


Figure 2–41. G102 ***TMX Axis Coordinates***

In the following figure, the Cylinder Diameter is 4 inches (102 mm), and the Cylinder Circumference is 12.544 inches (319 mm).

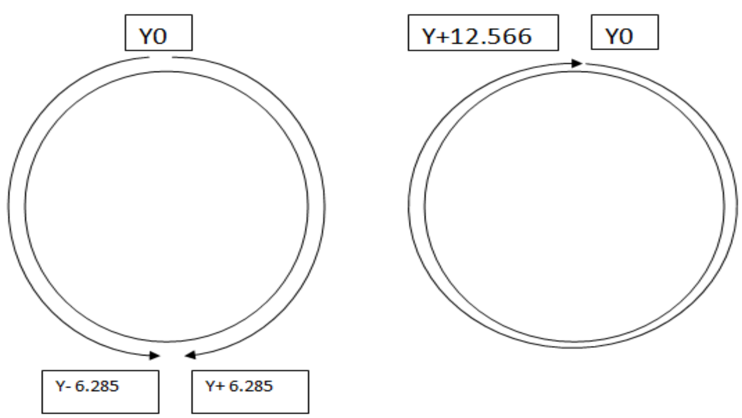


Figure 2–42. Y Axis Coordinate System Detail for G102

G103 - Axial Surface of the Part – Main Spindle with Linear Y

G103 sets up the machine for Axial Milling with true linear Y-Axis Motion and activates the coordinate map shown below.

Default—No

Modal—Yes

Cancels—G101, G102, G104, G105, G106, G107, G108, G109

Example

```
G103 (AXIS MAP)
M131 (MILLING MODE)
Z.25
Y.1
```

When using tool holders with multiple turn stations that require a Y move on a tool change, the G103 axis map must be active.

Tool Change Example

```
G103 (AXIS MAP)
T0101
Y0.
G109
```

The linear Y-axis motion for this part-relative, tool motion map is created by using the combined motion of the X and X' physical axes. The coordinate system for this axis map is the same as the G101 coordinate system.

⚠ This map is relative to the TMX MY and TMX MYS Main Spindle only. The Axial Milling map for the TMX MYS Sub-spindle uses G107.

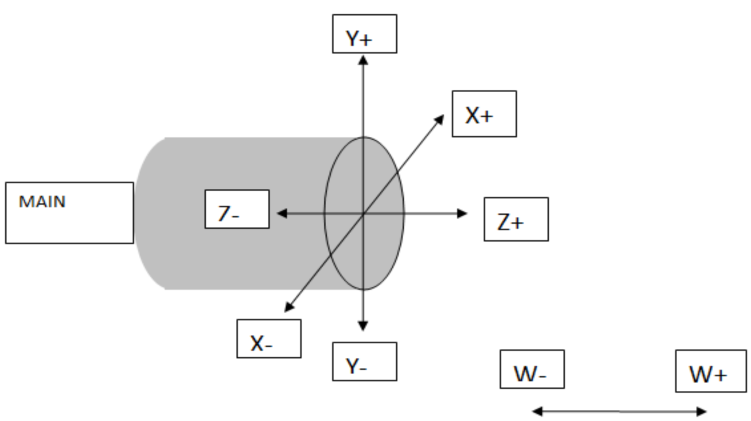


Figure 2–43. G103 TMX Axis Coordinates

G104 - Radial Surface of the Part – Main Spindle with Linear Y

G104 sets up the machine for Radial Milling with true linear Y-Axis Motion and activates the coordinate map shown below.

Default—No

Modal—Yes

Cancels—G101, G102, G103, G105, G106, G107, G108, G109

Example

```
G104 (AXIS MAP)
M131 (MILLING MODE)
G25 Q4. (DIA. OF PART)
```

G25 Q must be used to establish part-relative location. The surface of the part is Z.0, and a positive Z value is above the part in the air; a negative Z value moves into the stock.

This part-relative, tool motion map is used for Radial Milling with true linear Y-Axis motion. The bottom surface of the feature will be perpendicular to the X physical axis. On TMM machines this map is always used for Radial Milling.

This map is relative to the TMX MY and TMX MYS Main Spindle only. The Radial Milling map for the TMX MYS Sub-spindle uses G109.

This map swaps the Z and X axes from G101.

The Y+ and Y- axis has reversed direction from the G101 and G103 axis maps.

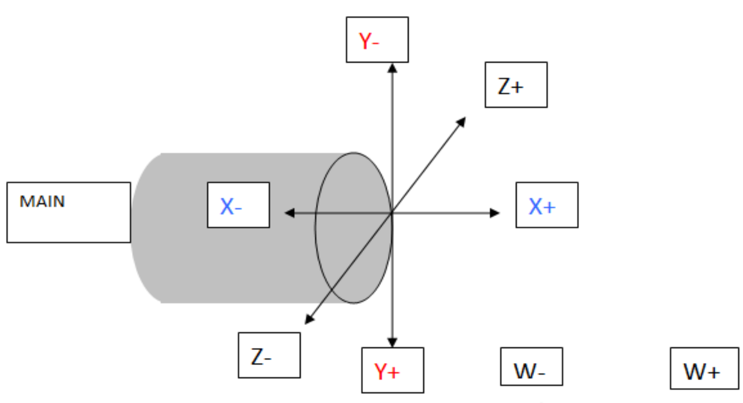


Figure 2–44. G104 TMX Axis Coordinates

G105 - Axial Surface of the Part – Sub-spindle without Linear Y

G105 sets up the machine for Axial Milling on the Sub-spindle and activates the coordinate map shown below.

Default—No

Modal—Yes

Cancels—G101, G102, G103, G104, G106, G107, G108, G109

Example

```
G105 (AXIS MAP)
M131 (MILLING MODE)
```

This part-relative, tool motion map is used when the Y-axis travel is inadequate for the geometry of the feature. The Y-axis positions are accomplished by coordinating the C and X axis motion.

⚠ This map is relative to the TMX MYS Sub-Spindle only. The Axial Milling map for the TMX MYS Main spindle uses G101.

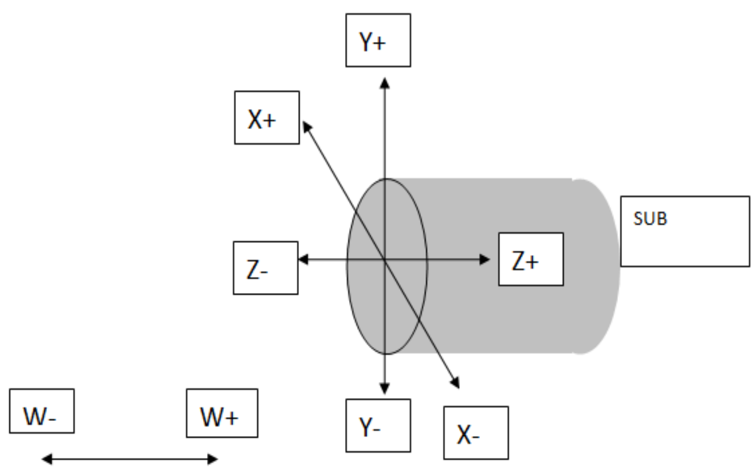


Figure 2–45. G105 TMX Axis Coordinates

G106 - Radial Surface of the Part – Sub-spindle without Linear Y

G106 sets up the machine for Radial Milling on the sub-spindle and activates the coordinate map shown below.

Default—No

Modal—Yes

Cancels—G101, G102, G103, G104, G105, G107, G108, G109

Example

```
G106 (AXIS MAP)
M131 (MILLING MODE)
G25 Q4. (DIA. OF PART)
```

G25 Q must be used to establish part-relative location. The surface of the part is Z.0, and a positive Z value is above the part in the air; a negative Z value moves into the stock.

This part-relative, tool motion map is used to wrap the Y-axis coordinate system around the circumference of the part and the part is in the sub-spindle. The bottom surface of the feature will be parallel to the circumference of the part.

> ⚠ This map is relative to the TMX MYS Sub-Spindle only. The Radial Milling map for the TMX MYS Main spindle uses G102.
>
> This map swaps the Z and X axes from G101.
>
> The Y-axis travel limits are limited to +/- 800 inches (20320 mm) when using this map.

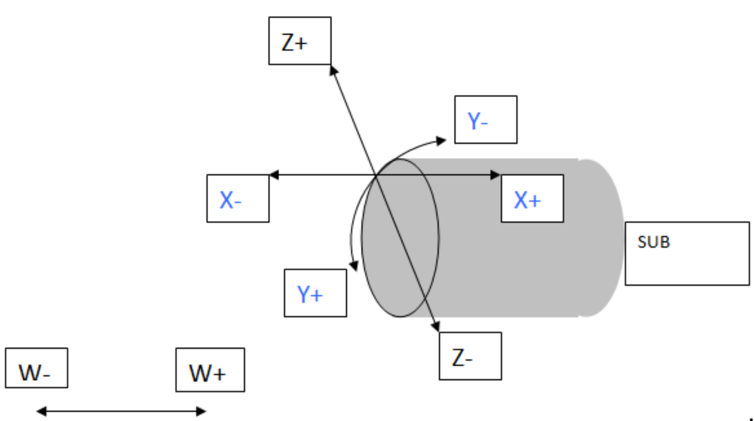


Figure 2–46. G106 TMX Axis Coordinates

In the following figure, the Cylinder Diameter is 4 inches (102 mm), and the Cylinder Circumference is 12.544 inches (319 mm).

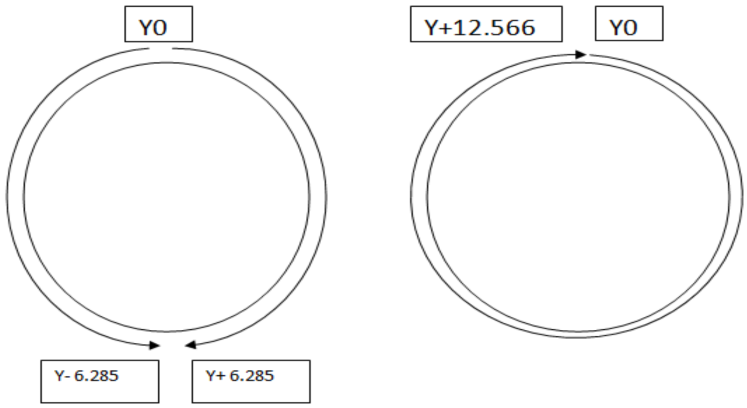


Figure 2–47. Y Axis Coordinate System Detail for G106

G107 - Axial Surface of the Part – Sub-spindle with Linear Y

G107 sets up the machine for axial milling on the sub-spindle with true linear Y-Axis Motion and activates the coordinate map shown below.

Default—No

Modal—Yes

Cancels—G101, G102, G103, G104, G105, G106, G108, G109

Example

```
G107 (AXIS MAP)
M131 (MILLING MODE)
```

The linear Y-axis motion for this part-relative, tool motion map is created by using the combined motion of the X and X' physical axes.

This map is relative to the TMX MYS Sub-Spindle only. The Axial Milling map for the TMX MYS Main spindle uses G103.

The G107 coordinate system is the same as the G105 axis map.

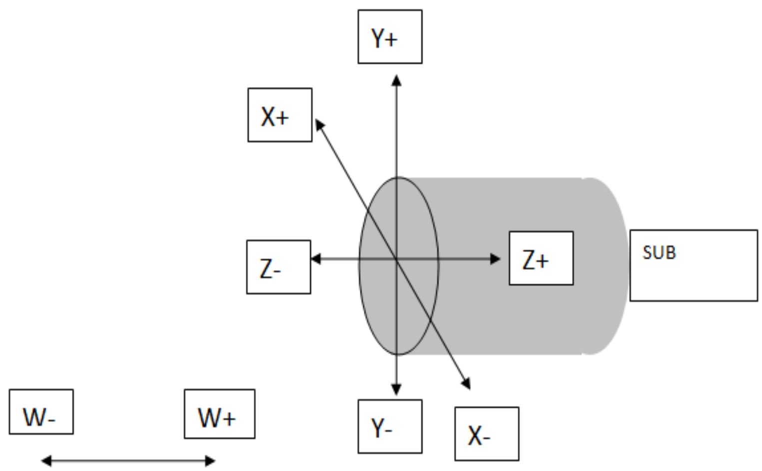


Figure 2–48. G107 TMX Axis Coordinates

G108 - Radial Surface of the Part – Sub-spindle with Linear Y

G108 sets up the machine for Radial Milling on the sub-spindle with true linear Y-Axis Motion and activates the coordinate map shown below.

Default—No

Modal—Yes

Cancels—G101, G102, G103, G104, G105, G106, G107, G109

Example

```
G108 (AXIS MAP)
M131 (MILLING MODE)
G25 Q4. (DIA. OF PART)
```

G25 Q must be used to establish part-relative location. The surface of the part is Z.0, and a positive Z value is above the part in the air; a negative Z value moves into the stock.

This part-relative, tool motion map is used for Radial Milling on the sub-spindle with true linear Y-Axis motion. The bottom surface of the feature will be perpendicular to the X physical axis.

This map is relative to the TMX MYS Sub-Spindle only. The Radial Milling map for the TMX MYS Main spindle uses G104.

This map swaps the Z and X axes from G101.

This map reverses the direction of the Y-axis from the direction G101.

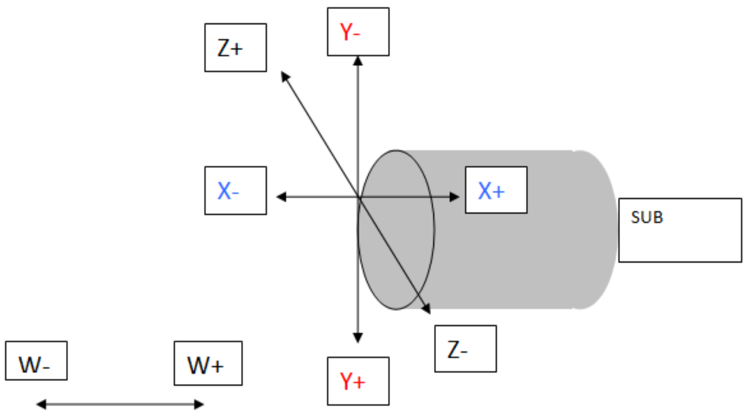


Figure 2–49. G108 TMX Axis Coordinates

G109 - Turning / Lathe Mode

G109 sets up the machine in Lathe Mode and activates the coordinate map shown below.

Default—Yes

Modal—Yes

Cancels—G101, G102, G103, G104, G105, G106, G107, G108

Example (Turning Operations)

```
M130
G109
```

Example (Drilling operations with the C axis mapped instead of Y)

```
M131
G109
```

When G109 is used with M131 (Live Tool Mode) it allows for performing live tooling operations using polar coordinates. The G109 must be preceded by either the M130 (Lathe Mode) or M131 (Live-tool Mode – Main Spindle/C Axis).

This axis map applies to all machine machines.

Polar Coordinate programming is not available for the sub spindle.

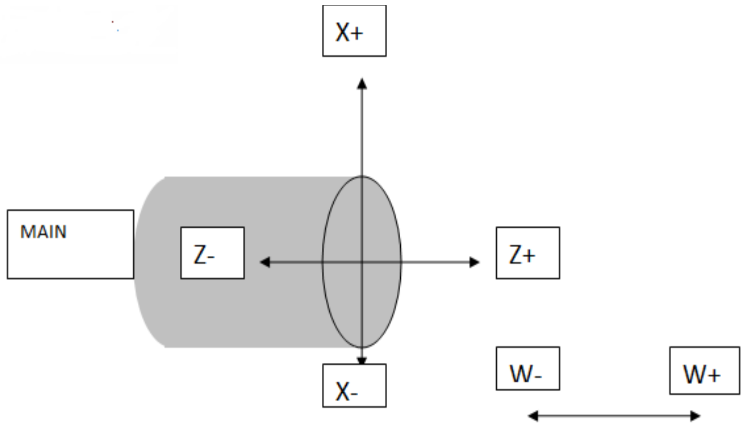


Figure 2–50. G109 Turning/Lathe Mode (Main Spindle Axis Map)

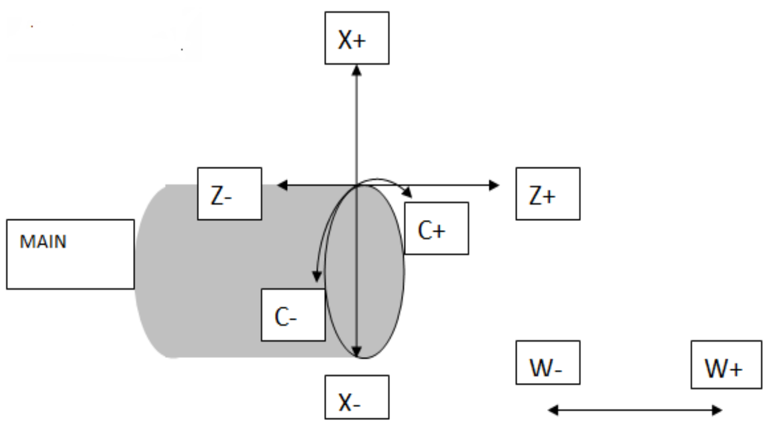


Figure 2–51. G109 Turning/Live-tool Mode (Main Spindle Axis Map)

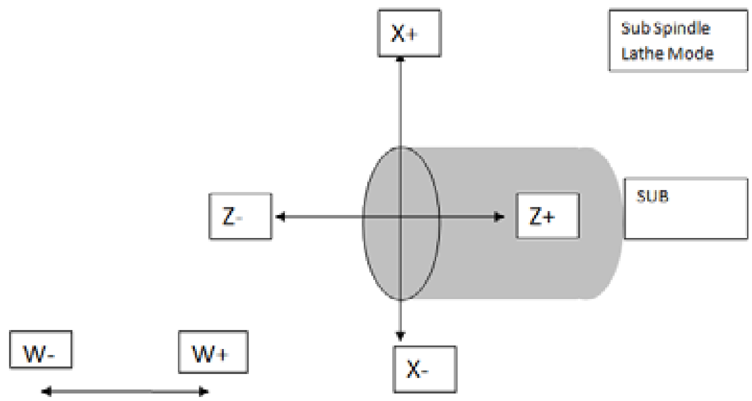


Figure 2–52. G109 Turning/Lathe Mode (Sub-Spindle Axis Map)

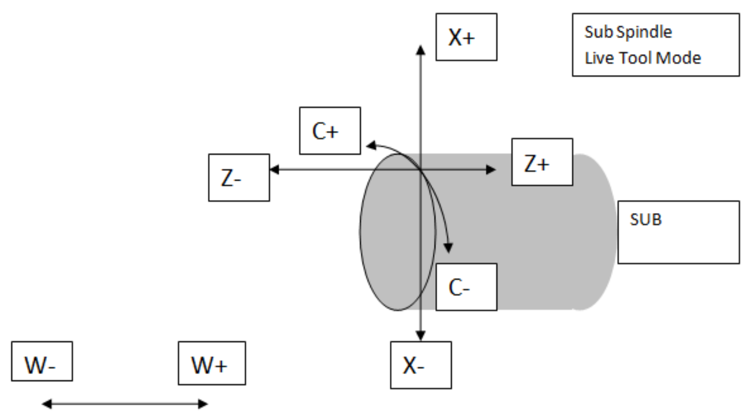


Figure 2–53. G109 Turning/Live-tool Mode (Sub-spindle Axis Map)

G152 - Move the W-axis

G152 moves the W-axis for a block.

Default—No

Modal—No

Format

G152 W____

G152 is used to move the W-axis in a particular block. When a W command is used elsewhere, it is treated as an incremental Z move.

ISNC M CODES

This section contains information about the following topics:

ISNC M Code Overview

M codes belong to one of the following four execution categories:

- Execute before tool change or motion.
- Execute after T and S codes, but before motion.
- Execute after tool change or motion.
- Execute after tool change or motion (system M codes).

Understanding each M code's execution category is important when the M code is programmed in a block that also contains a T code, S code, E code, and XZ axes motion.

M Code Translation

The ISNC option provides M Code translation, allowing you to customize M Codes from existing programs and controls for use on the WinMax Lathe Max Control. Many M Codes are named by the machine builder and vary from one machine to another. These M Codes can be converted to a default set of codes, or the codes can be customized and translated to the M Code that the WinMax Lathe Max control uses.

In addition, if an existing program contains M Codes for machine functions that are not supported, you can specify which M Codes to ignore.

Setting up M Codes

From the Utilities, User Preferences, NC Settings screen with an ISNC program open, select the ISNC M CODES *F5* softkey.This softkey appears when the cursor is in the NC Dialect field.

The ISNC M Code Translation Map screen opens.

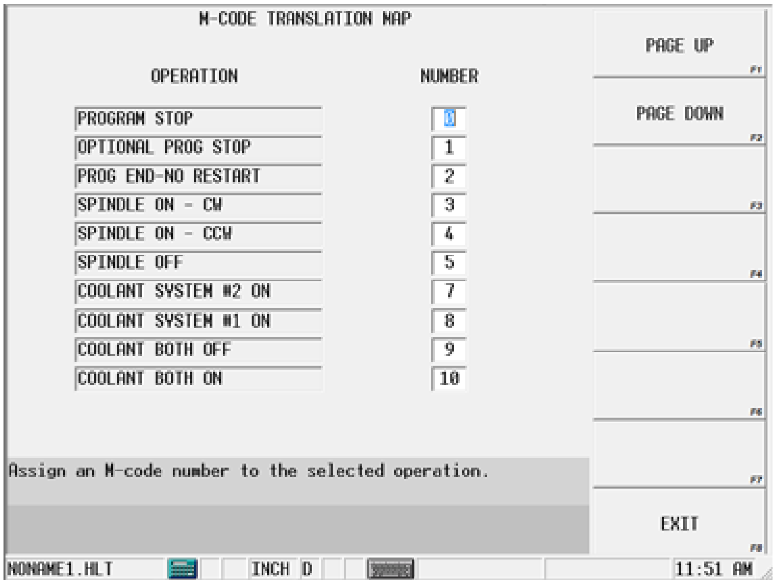


Figure 3–1. M Code Translation Map

This screen lists M Code names and a column listing the corresponding default ISNC code. Please refer to *ISNC M Code Definitions, on page 3 - 4* for details about each M Code.

You can change the default ISNC codes, unless the field is inactive. Inactive fields contain commonly used code numbers.

ISNC M Code Definitions

Following are definitions for Hurco M Codes. A corresponding ISNC M Code for M Code Translation or Importing and Exporting M Code Data is provided for each M Code.

The following M codes are available on most systems.

ISNC M Code Translation List - Page 1

ISNC M Code Translation List - Page 2

ISNC M Code Translation List - Page 3

ISNC M Code Translation List - Page 4

HURCO

ISNC M Code Translation List - Page 1

Program Stop (ISNC M0)

M0 causes part program execution to stop and the spindle to turn off. Press the **START CYCLE** button to resume program execution and restart the look-ahead program. Execute after tool change or motion (system M codes).

Optional Stop (ISNC M1)

M1 performs the same functions as M0, but can be toggled on or off through the **Opt Stop** key on the control.

- When the **Opt Stop key** is **Off**, the program ignores M1.
- When the **Opt Stop key** is **On**, the program stops on M1 so you can inspect the part. Press the **START CYCLE** button to resume running the program.

Execute after tool change or motion.

Main Spindle CW (ISNC M3)

M3 turns the main spindle (S1) on in the clockwise direction. M3 will execute after Tool, Spindle, and E (Work Offset) codes execute, but before motion programmed in the block occurs.

An S code specifying the spindle speed must be programmed before the spindle will turn on. The advanced RPM Look Ahead feature that reduces part cycle time by ramping the spindle up to speed before the feed move begins improves the cycle time because the control does not dwell while the spindle ramps up to the programmed speed. When this feature is active, it is possible to create a part program containing an M3 and spindle speed command that does not turn on the spindle.

Execute after T and S codes, but before motion.

Format

S1:M3

Main Spindle CCW (ISNC M4)

M4 is the same as M0 except it commands the main spindle (S1) to rotate in the counterclockwise direction. Execute after T and S codes, but before motion.

Format

S1:M4

Main Spindle Stop (ISNC M5)

M5 turns the main spindle (S1) off. M5 executes at the end of the block after all motion occurs. Execute after tool change or motion (system M codes).

Format

S1:M5

Live Tool CW (ISNC M33)

⚠ M33 is currently not supported.

M33 turns the live tooling spindle (S2) on in the clockwise direction. M3 will execute after Tool, Spindle, and E (Work Offset) codes execute, but before motion programmed in the block occurs.

An S code specifying the spindle speed must be programmed before the spindle will turn on. The advanced RPM Look Ahead feature that reduces part cycle time by ramping the spindle up to speed before the feed move begins improves the cycle time because the control does not dwell while the spindle ramps up to the programmed speed. When this feature is active, it is possible to create a part program containing an M33 and spindle speed command that does not turn on the spindle.

Execute after T and S codes, but before motion.

Format

S2:M33

Live Tool CCW (ISNC M34)

⚠ M34 is currently not supported.

M34 is the same as M0 except it commands the live tooling spindle (S2) to rotate in the counterclockwise direction. Execute after T and S codes, but before motion.

Format

S2:M34

Live Tool Stop (ISNC M35)

⚠ M35 is currently not supported.

M35 turns the live tooling spindle (S2) off. M5 executes at the end of the block after all motion occurs. Execute after tool change or motion (system M codes).

Format

S2:M35

Sub-spindle CW (ISNC M103)

M103 turns the sub-spindle (S3) on in the clockwise direction. M103 will execute after Tool, Spindle, and E (Work Offset) codes execute, but before motion programmed in the block occurs.

An S code specifying the spindle speed must be programmed before the spindle will turn on. The advanced RPM Look Ahead feature that reduces part cycle time by ramping the spindle up to speed before the feed move begins improves the cycle time because the control does not dwell while the spindle ramps up to the programmed speed. When this feature is active, it is possible to create a part program containing an M103 and spindle speed command that does not turn on the spindle.

Execute after T and S codes, but before motion.

Format

S3:M103

Sub-spindle CCW (ISNC M104)

M104 is the same as M0 except it commands the sub-spindle (S3) to rotate in the counterclockwise direction. Execute after T and S codes, but before motion.

Format

S3:M104

Sub-spindle Stop (ISNC M105)

M105 turns the sub-spindle (S3) off. M105 executes at the end of the block after all motion occurs. Execute after tool change or motion (system M codes).

Format

S3:M105

Secondary Coolant (ISNC M7)

M7 commands the secondary coolant pump to turn on, if so equipped. This command is cancelled by an M9 command. Execute before tool change or motion.

Primary Coolant On (ISNC M8)

M8 commands the primary coolant pump to turn on, typically flood coolant. This command is cancelled by an M9 command. Execute before tool change or motion.

Coolant Off (ISNC M9)

M9 command tells the control to turn the primary and secondary coolant pumps off. Execute after tool change or motion.

Chuck Pressure High (ISNC M67)

M67 command sets the chucking pressure to high.

Chuck Pressure Low (ISNC M66)

M66 command sets the chucking pressure to low.

Turret Index Reverse (ISNC M12)

M12 is used to reverse the turret.

Turret Index Forward (ISNC M13)

M13 is used to move the turret forward.

Chuck Open (ISNC M69)

M69 is used to open the chuck for use with the optional Bar Feeder.

Chuck Close (ISNC M68)

M68 is used to close the chuck for use with the optional Bar Feeder.

Main Spindle Orient (ISNC M19)

M19 orients the main spindle to 0°.

Chuck Open for Bar (ISNC M20)

M20 is used with the optional Bar Feeder.

Chuck Close for Bar (ISNC M21)

M21 is used with the optional Bar Feeder.

Start Bar Feeder (ISNC M22)

M22 is used with the optional Bar Feeder.

ISNC M Code Translation List - Page 2

Start Bar with Z (ISNC M23)

M23 is used with the optional Bar Feeder to open the chuck then cause the bar feeder with Z to guide the stock.

Part Conveyor On (ISNC M110)

M110 turns on the optional Part Conveyor.

Part Conveyor Off (ISNC M111)

M111 turns off the optional Part Conveyor.

Tailstock Quill Adv (ISNC M78)

M78 is used to advance the optional Tailstock Quill. If either the **Interrupt** or **Stop Cycle** console key is pressed before the quill is fully advanced, the quill will automatically retract. If in Interrupt mode and you return to Auto Mode and press Start cycle, the Tailstock will advance before the program continues.

Tailstock Quill Retr (ISNC M79)

M79 is used to retract the optional Tailstock Quill.

End of Program (ISNC M30)

M30 stops program execution and moves the part program pointer to the top of the part program. M30 commands the spindle to turn off. Execute after tool change or motion (system M codes).

Incr Cycle Count (ISNC M31)

M31 increments the cycle counter.

Incr Setup Cycle (ISNC M32)

M32 increments the setup cycle.

Feedrate Override (ISNC M48)

M48 uses the value specified by the Feedrate Override dial (FPM %). Execute before tool change or motion.

Ignore Feed Override (ISNC M49)

M49 ignores the value specified by the Feedrate Override (FPR %) and uses the programmed feedrate.

Conveyor On (ISNC M24)

M24 is used to turn on the optional Conveyor.

Conveyor Off (ISNC M25)

M25 is used to turn off the optional Conveyor.

Part Catcher w/Eject (ISNC M57)

M57 is used to advance the optional Part Catcher with bar end eject.

Part Catcher Advance (ISNC M58)

M58 is used to advance the optional Part Catcher.

Part Catcher Retract (ISNC M59)

M59 is used to retract the optional Part Catcher.

Sel External Chuck (ISNC M60)

M60 is used to select external chucking.

Sel Internal Chuck (ISNC M61)

M61 is used to select internal chucking.

Tool Setter Retract (ISNC M71)

M71 is used to retract the optional Tool Setter.

Tool Setter Adv (ISNC M72)

M72 is used to advance the optional Tool Setter.

Auto Door Open (ISNC M85)

M85 is used to open the optional Auto Door.

Auto Door Close (ISNC M86)

M86 is used to close the optional Auto Door.

Single Block Off (ISNC M91)

M91 is used for inactivating a single block.

Single Block On (ISNC M92)

M92 is used for activating a single block.

Sub-spindle Chuck Open (ISNC M168)

M168 is used for opening the sub-spindle chuck.

ISNC M Code Translation List - Page 3

Sub-spindle Chuck Close (ISNC M169)

M169 is used for closing the sub-spindle chuck.

Turning Mode (ISNC M130)

M130 sets the control to turning mode.

Milling Mode (ISNC M131)

M131 sets the control to milling mode.

C Axis Clamp (ISNC M41)

M41 enables the C axis clamp.

C Axis Hold (ISNC M133)

M133 enables the C axis hold. This code is valid for TMM machines only.

C Axis Unclamp (ISNC M40)

M40 releases both C axis clamps.

Sub-spndl Ext Chuck (ISNC M160)

M160 selects external chucking for sub-spindle.

Sub-spndl Int Chuck (ISNC M161)

M161 selects internal chucking for sub-spindle.

W Torque Mon On (ISNC M186)

M186 activates torque monitoring for the W-axis. Torque monitoring is available when picking up or transferring parts from either the Main spindle to the Sub-spindle or from the Sub-spindle to the Main spindle. The force, or load, of the material on the W-axis is monitored when M186 is used.

If the motor load is greater than 50% of the rated Full load for the W-axis, the program stops and an error message appears. The 50% value cannot and does not change. Use M186 in conjunction with G06 (Probe/Block Skip Function). With G06 active, the system stops the motion and deletes the remaining distance to go. G06 is a one-shot command and stays in effect until the end of the block.

M186 functions when Feed motions are used (G01, G02, G03). M186 does not function with Rapid motions (G00).

M186 Example

```
M241(Allow Sub-spindle Chuck to Open when Spindle Running)
M114(Open Sub-spindle Chuck)
G0 W200(Rapid W to Approach position)
M186(Activate W Axis Torque monitoring)
G1 G6 W180 F20(Feed W Axis forward to Grab Part and activate Probe/Block
skip)
M187 (Stop W Axis Torque Monitoring)
G0 W200 (Rapid W back to Approach position)
```

W Torque Mon Off (ISNC M187)

M187 deactivates torque monitoring for the W-axis.

Block Skip Sync (ISNC M200)

M200 allows you to turn on block skip while the part program is executing. Using this code ensures that blocks marked with block delete codes are skipped after block skip has been turned on, without requiring you to stop and restart the program. Execute before tool change or motion. Refer to *Block Skip Code, on page 1 - 6* for information about Block Skip.

Sync Spindles Forward (ISNC M203)

M203 synchronizes the main spindle and sub-spindle forward.

Sync Spindles Reverse (ISNC M204)

M204 synchronizes the main spindle and sub-spindle in reverse.

Clear Spindle Sync (ISNC M205)

M205 clears the main spindle and sub-spindle synchronization.

Bypass Chuck Intlk (ISNC M231)

M231 bypasses the main spindle chuck open interlock.

C3 Axis Clamp (ISNC M232)

M232 enables the C3 axis clamp.

C3 Axis Unclamp (ISNC M234)

M234 releases the C3 axis clamp.

Spindle CW/Coolant On (ISNC M63)

M63 performs 2 functions at once:

- M63 turns the spindle on in the clockwise direction. M63 will execute after Tool, Spindle, and E (Work Offset) codes execute, but before motion programmed in the block occurs.
- M63 commands the primary coolant pump to turn on, typically flood coolant. Execute before tool change or motion.

Format

S1 or S2:M63

Spindle CCW/Coolant Off (ISNC M64)

M64 performs 2 functions at once:

- M64 turns the spindle on in the counterclockwise direction. M64 will execute after Tool, Spindle, and E (Work Offset) codes execute, but before motion programmed in the block occurs.
- M64 commands the primary coolant pump to turn off. Execute before tool change or motion.

Format

S1 or S2:M64

Spindle Off/Coolant Off (ISNC M65)

M65 performs 2 functions at once:

- M65 turns the spindle off. M64 will execute after Tool, Spindle, and E (Work Offset) codes execute, but before motion programmed in the block occurs.
- M65 commands the primary coolant pump to turn off. Execute after tool change or motion.

Format

S1 or S2:M65

Sub-Spdl CW/CLNT On (ISNC M163)

M163 performs 2 functions at once:

- M163 turns the sub-spindle on in the clockwise direction. M63 will execute after Tool, Spindle, and E (Work Offset) codes execute, but before motion programmed in the block occurs.
- M163 commands the primary coolant pump to turn on, typically flood coolant. Execute before tool change or motion.

Format

S3:M163

Sub-Spdl CCW/CLNT Off (ISNC M164)

M164 performs 2 functions at once:

- M164 turns the sub-spindle on in the counterclockwise direction. M64 will execute after Tool, Spindle, and E (Work Offset) codes execute, but before motion programmed in the block occurs.
- M164 commands the primary coolant pump to turn off. Execute before tool change or motion.

Format

S3:M164

Sub-Spdl Off/CLNT Off (ISNC M165)

M165 performs 2 functions at once:

- M165 turns the sub-spindle off. M64 will execute after Tool, Spindle, and E (Work Offset) codes execute, but before motion programmed in the block occurs.
- M165 commands the primary coolant pump to turn off. Execute after tool change or motion.

Format

S3:M165

Rigid Tap (ISNC M29)

M29 is used to activate rigid tapping. M29 must precede the tapping G code and must be on a block by itself.

M29 Example

```
M29
G84 Z-0.5 F0.05
```

ISNC M Code Translation List - Page 4

Auxiliary Output 1 On (ISNC M52)

M52 is used to turn on auxiliary equipment or a unique machine function.

Auxiliary Output 2 On (ISNC M53)

M53 is used to turn on auxiliary equipment or a unique machine function.

Auxiliary Output 3 On (ISNC M54)

M54 is used to turn on auxiliary equipment or a unique machine function.

Auxiliary Output 4 On (ISNC M55)

M55 is used to turn on auxiliary equipment or a unique machine function.

Auxiliary Output 1 Off (ISNC M152)

M152 is used to turn off auxiliary equipment or a unique machine function.

Auxiliary Output 2 Off (ISNC M153)

M153 is used to turn off auxiliary equipment or a unique machine function.

Auxiliary Output 3 Off (ISNC M154)

M154 is used to turn off auxiliary equipment or a unique machine function.

Auxiliary Output 4 Off (ISNC M155)

M155 is used to turn off auxiliary equipment or a unique machine function.

Chuck Air On (ISNC M16)

M16 turns on the chuck air blow.

Chuck Air Off (ISNC M17)

M17 turns off the chuck air blow.

Set Max Rapid (ISNC M94)

M94 is used to set the maximum rapid rate for the program. M94 requires an argument to specify the maximum rapid rate and must be expressed as MMPM.

M94 Example:

```
M94:5080
```

This block sets the Maximum Rapid Rate to 5080 MMPM (200 IPM).

Part Transfer Sub-spindle (ISNC 241)

M241 allows the sub-spindle chuck to open while the sub-spindle is running. This interlock bypass is cancelled auto0matically when M115 (Close Sub-spindle Chuck) is executed.

End of Program (no rewind) (ISNC M2)

M2 stops program execution and leaves the part program pointer on the block that follows the M2 command. M2 is required at the end of all programs. It commands the spindle to turn off. Execute at the end of the block after all motion occurs.

Steady Rest Clamp (ISNC M38)

The Steady Rest device can be manually hitched to the Z-Axis to assist with movement of the Steady Rest to the proper location. The position of the Steady Rest device is tracked in order to avoid colliding with the tailstock.

⚠ The position of the Steady Rest can only be tracked when moving the Steady Rest with the Z-Axis. If the Steady Rest is moved manually without the use of the Z-Axis, then the control will not be aware of this new position for the Steady Rest which will either allow the tailstock to possibly collide with it or improperly limit the motion of the Z-Axis when hitched to the Tailstock.

A safety switch located on a bracket under the way cover also protects against the Steady Rest and Tailstock colliding. If the safety switch is tripped, the machine will enter the Emergency Stop mode.

This optional device is used with turning centers to hold long pieces of stock in the center while cutting.

Steady Rest Unclamp (ISNC M39)

After the spindle is stopped, M39 unclamps the optional Steady Rest device during automatic cycle operation. The spindle MUST BE stopped when this M-Code executes or an error will appear and the cycle will be halted.

⚠ The spindle must be stopped prior to executing the M39 Steady Rest Unclamp.

The Steady Rest device can be manually hitched to the Z-Axis to assist with movement of the Steady Rest to the proper location. The position of the Steady Rest device is tracked in order to avoid colliding with the tailstock.

⚠ The position of the Steady Rest can only be tracked when moving the Steady Rest with the Z-Axis. If the Steady Rest is moved manually without the use of the Z-Axis, then the control will not be aware of this new position for the Steady Rest which will either allow the tailstock to possibly collide with it or improperly limit the motion of the Z-Axis when hitched to the Tailstock.

A safety switch located on a bracket under the way cover also protects against the Steady Rest and Tailstock colliding. If the safety switch is tripped, the machine will enter the Emergency Stop mode.

This optional device is used with turning centers to hold long pieces of stock in the center while cutting.

Spindle Gear 1 (ISNC M41)

M41 sets the spindle to Gear 1, or the low gear range.

Low Gear Speed Range for TM18:
Minimum RPM = 5
Maximum RPM = 600

Spindle Gear 2 (ISNC M42)

M42 sets the spindle to Gear 2, or the high gear range.

High Gear Speed Range for TM18:
Minimum RPM = 15
Maximum RPM = 1600

Z-axis Tailstock Hitch (ISNC M128)

M128 is used to move the Z-axis to the Tailstock and subsequently to hitch to the Tailstock. The Tailstock can then move with the Z-axis to the specified Z location.

Z-axis Tailstock Unhitch (ISNC M129)

M129 is used to unhitch the Z-axis from the Tailstock. The Z-axis can then move freely from the tailstock.bypass is cancelled automatically when M115 (Close Sub-spindle Chuck) is executed.

T CODES

This section describes how to program a tool change and activate the tool's offsets.

Tool Change Sequence

You must precede a tool change sequence with the commands required to move the axes to the machine's tool change position.

A typical tool change sequence is shown below.

```
T0303
M03 S1200
```

⚠ The T code will cause the turret to index into position. It is the responsibility of the part programmer or machine operator to ensure that the turret is in a safe position to perform an index.

The format for a T code is:

T __ nn

where:

__ is the tool number—The first digits in the T code identify the tool number that must be indexed into position.

nn is the tool offset number—The last two digits identify the tool table offsets to use for the current tool.

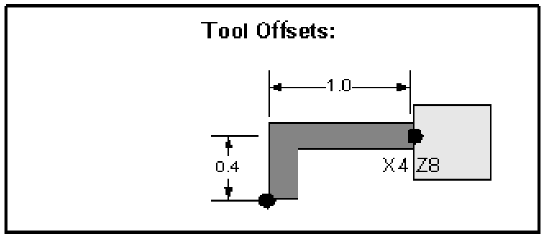
A **T00** cancels any tool offsets.

The coordinates for each tool offset are stored in the Tool Offset table. All dimensions in the tool offset table represent dimensions measured from machine coordinate zero to the desired tool offset position. A tool offset value is provided for both axes.

Execution of a tool offset T code does not cause tool motion unless axis motion is programmed in the block that contains the T code, or until motion is programmed in a following block. This applies to Work Offset E codes as well.

Tool Offsets

The Tool Offset implements the Work Offset E codes. The drawings that follow show the tool movement.



- **Immediate Action**—the axes will move to a position relative to the new tool offset as soon as any axis is programmed.

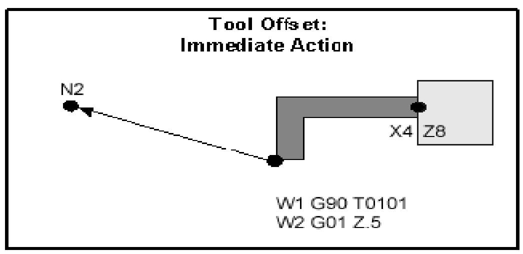


Figure 4–1. Tool Offset Behaviors

Record of Changes

v11.01, December 2018

Revised by: H. Arle

Approved by: D. Skrzypczak

Changes
Removed Basic NC.

704-0115-309, April 2013

Revised by: K. Gross

Approved by: J.Small, D. Skrzypczak, G.Traicoff April 2013

Changes
704-0115-309 Updates based on changes through v9.02.34 software. Added G101-G109 for BNC and ISNC. Updated some examples.

704-0115-308, January 2013

Revised by: K. Gross

Approved by: D. Skrzypczak January 2013

Changes
704-0115-308 Updates based on changes through v9.02.11 software. Added information for Disable NC Editor Undo and Redo function.

704-0115-307, October 2012

Revised by: K. Gross

Approved by: D. Skrzypczak, K.Szabo, K. Van Blaircum October 2012

Changes
704-0115-307 Updates based on changes through v9.00.43 software. Re-organized ISNC M Code presentation to follow the M Code Translation screens.

704-0115-306, March 2012

Revised by: K. Gross

Approved by: D. Skrzypczak March 2012

Changes
704-0115-306 Updates based on changes through v8.1.3.48 software.

704-0115-305, December 2011

Revised by: K. Gross

Approved by: D. Skrzypczak, J. Mulkey, G.Traicoff, K. Van Blaircum December 2011

Changes
704-0115-305 Updates based on changes through v8.1.2.30 software,
Set cross-references between books.
Added ScreenPath condition and paragraph tag.

704-0115-304, April 2011

Revised by: K. Gross

Approved by: D. Skrzypczak, J. Mulkey, G.Traicoff, K. Van Blaircum April 2011

Changes
• 704-0115-304 Updates based on changes through v8.1.2 software, including updated NC Editor screen and the addition of optional ISNC.

704-0115-303, June 2010, ECN 16538

Revised by: K. Gross

Approved by: D. Skrzypczak, J. Mulkey, C.Thale, G.Traicoff, K. Van Blaircum June 2010

Changes
• 704-0115-303 Updates based on changes through v8.1.1 software.
• Updated graphics for G02, G73, G74, G81, G82, G84, G98, and G99.
• Added descriptions for M10, M11, M16, M17, M20, M23, M24, M25, M31, M32, M74, M85, M86, M94, M114, M115, M132, M133, M134, M57, M160, M161, M186, M187, M203, M204, M205, M231, M232, M234, and M241.
• Removed descriptions for M36 and M37.

704-0115-302, January 2009, ECN 16508

Revised by: K. Gross

Approved by: D. Skrzypczak, January 2009

Changes
• 704-0115-302 rC Revised cover. Made minor formatting changes for Help development. January 2009. • 704-0115-302 rB Updates based on v2.02.02 software. Added information to front matter about On-screen Help and accessing the On-screen Help in PDF format, November 2008. • 704-0115-302 rA Updates based on v1.01, v1.02, and v2.0 software and the introduction of the Live Tooling (TMM Series) machine. October 2007.

704-0115-301, 14 July 2005, ECN 15866

Revised by: K. Gross

Approved by: G. Traicoff, C. Thale, D. Skrzypczak, July 2005

Changes
New manual release.

INDEX

D

E

F

G

H

I

N

O

P