704-0111-301

Ultimax
NC Part Programming Manual

August, 2002 Revision A

NC Part Programming Manual

for
Hurco Machining Centers

Includes Industry Standard NC Programming

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.

For more information about Hurco products and services, contact:

Hurco Companies, Inc.
One Technology Way
P.O. Box 68180
Indianapolis, IN 46268-0180
Tel (317) 293-5309 (products)
(317) 298-2635 (service)
Fax (317) 328-2812 (service)

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

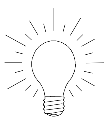
Using This Manual

Standard Text Icons

This manual may contain the following icons:

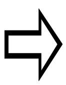
Caution

The machine may be damaged, or a part ruined, if the described procedure is not followed.



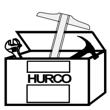
Hints and Tricks

Useful suggestions that show creative uses of the Ultimax features.



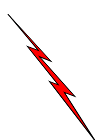
Important

Ensures proper operation of the machine and control.



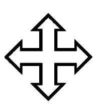
Troubleshooting

Steps that can be taken to solve potential problems.



Warning

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



Where can we go from here?

Lists several possible options the operator can take.

Sample Screens

Some sample screens in this manual were captured on a stand-alone Ultimax system. The screens on your system may vary slightly. The Input screen below illustrates softkeys and includes the software version (circled below).

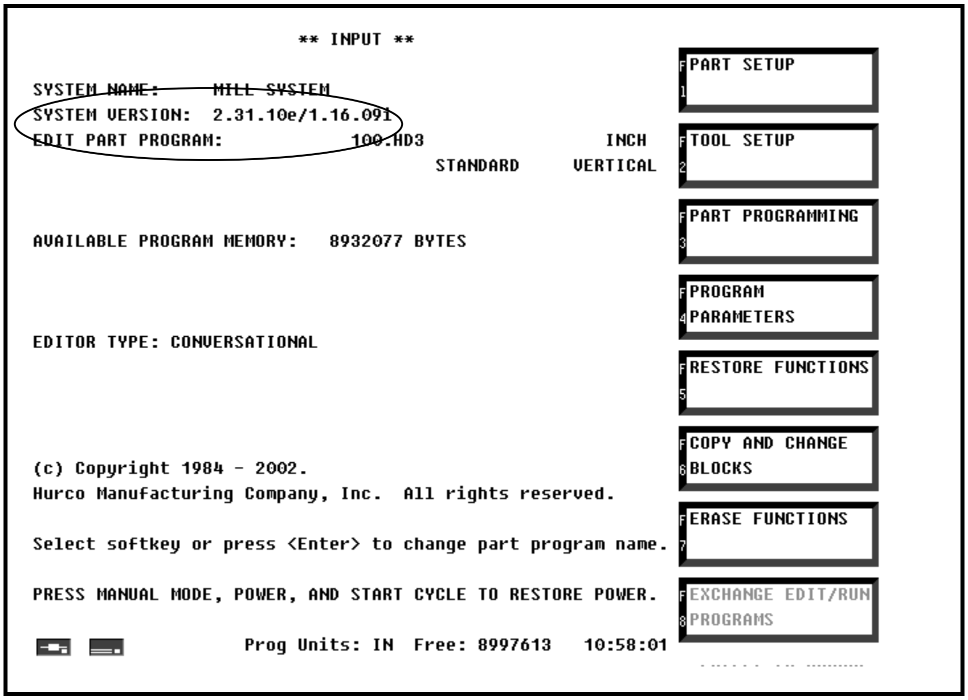


Figure 1. Input Screen

Ultimax screens have three areas of primary interest:

- Softkeys on the right side of the touch screen. Available softkeys may change even when the text and data entry area does not.

- Fields to the left of the softkeys. A field is an area that display or receives information entered by the operator.

- Prompt and error message area at the bottom of the screen. In the sample screen above, the message area reads, “Select softkey or press <Enter> to change part program name.”

Table of Contents

NC Part Programming

Index

Figure List

Table List

NC Part Programming

This manual describes the use of NC (Numerical Control) Part Programming, which includes the BNC (Basic Numerical Control) and the ISNC (Industry Standard Numerical Control) Editor portion of the CNC software as it is used on the machine tool console. This manual explains the following:

- Using the NC programming system
- Creating and editing NC programs on the control
- NC codes

Refer to the “Preparatory Functions—G Codes” section of this manual for information about the G codes and the “Miscellaneous Functions—M Codes” section for information about the M codes.

NC Part Programming Principles

NC part programming adheres to either the ANSI/EIA *RS–274–D* standard terminology for *BNC* mode, or the Fanuc 0™ programming standard for *ISNC* mode. In addition, the NC programming facilities were designed to use as much of the Ultimax Conversational system as possible. As a result, most of the screens are the same in both the NC and the conversational systems. This allows a smooth transition between the two.

The primary difference between conversational and NC programming is the program editors. The NC is programming uses standard G and M codes; whereas, conversational programming uses plain English or another supported programming language.

Important

The CNC software can read NC files from the serial port directly into dynamic memory or run NC files that are partially loaded into dynamic memory. NC files can be serially loaded to the hard disk.

NC Part Programming

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.

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. These components make up NC code:

<u>Program Start</u>

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.

<u>Sequence Number</u>

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.

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.

Note

If you request renumbering of part program sequence numbers, any sequence numbers in GOTO statements will not be updated. You must then press the (F1) Yes softkey before re-sequencing will take place. To cancel the renumbering, press the (F8) No softkey. In general, you will not want to renumber part programs that use GOTO statements.

Address Characters

An address character is the first character of a word in a program block. The Ignore Command signals the system to ignore the remainder of the block. The Comment Command characters are used to delimit comments. The following is a list of the address characters recognized by this system:

/	Ignore Command
( )	Comment Command
:	Subprogram Number (NCPP Option)
A	Rotary Dimension Around X-axis
B	Rotary Dimension Around Y-axis
D	Tool Diameter Offset
F	Feedrate
G	Preparatory Functions
H	Index into the tool length offset table
I	X-axis Arc Center/Offset, X scale factor, Canned Cycle Bore Shift
J	Y-axis Arc Center/Offset, Y scale factor, Canned Cycle Bore Shift
K	Z-axis Arc Center/Offset, Z scale factor, Canned Cycle Repeat
L	Tool Length Offset, Data Set Mode
M	Miscellaneous Functions
N	Sequence Number
O	Subprogram Number (NCPP option)
P	Subprogram Number, Dwell Time, Scaling Factor
Q	Canned Cycle Bore Shift, Peck Depth
R	Rotation Angle, Return Level, Circular Interpolation Radius
S	Spindle Speed Function
T	Tool Select
X	Primary X Motion Dimension, Dwell Time
Y	Primary Y Motion Dimension
Z	Primary Z Motion Dimension

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	**End of tape (EOT)** (optional for BNC and ISNC)—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]).

Note

[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 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 groups of words form a program block.

Block

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

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

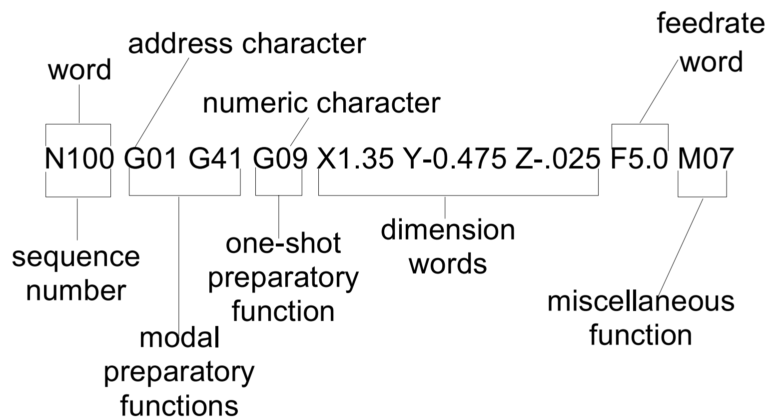


Figure 1. Typical NC Block

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 G Code Table in the "Preparatory Functions-G Codes" section.

Important

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 or the left Arrow 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.

Refer to the “Modifying an NC Part Program—Basic Programming Functions Main Menu” section for information about deleting blocks.

Absolute (G90) and Incremental (G91) machining modes determine whether the axis moves relative to part zero or in incremental distances from the previous block.

Refer to the “Absolute and Incremental (G90, G91)” section for information about these programming modes.

NC Editor

The NC Editor is used for creating or changing an NC part program. Refer to the "Switching Part Programming Editor Type" section for information about switching to the NC Editor. The NC Editor is similar to a text editor on a personal computer.

Editor Menus

The editor menus are arranged with the most commonly used features listed in the main menu (insert, delete, toggle, and jump). The submenus contain search and edit functions, which include graphics markers, syntax checking, and program execution features. The submenus also have file and program selection or deletion functions.

You can use the New File softkey from the File and Program Selection or Deletion editor menu to create a new NC file. For more information about the editor's softkeys, refer to the "Editing NC Part Programs" section.

Edit Screen Fields

The NC Editor screens have display and data entry areas and several softkey menus used for creating and editing NC part programs. After entering the tool and part setup information, press the Part Programming (F3) softkey on the Input screen.

The Input screen now appears with a Status Line and the Editing Region:

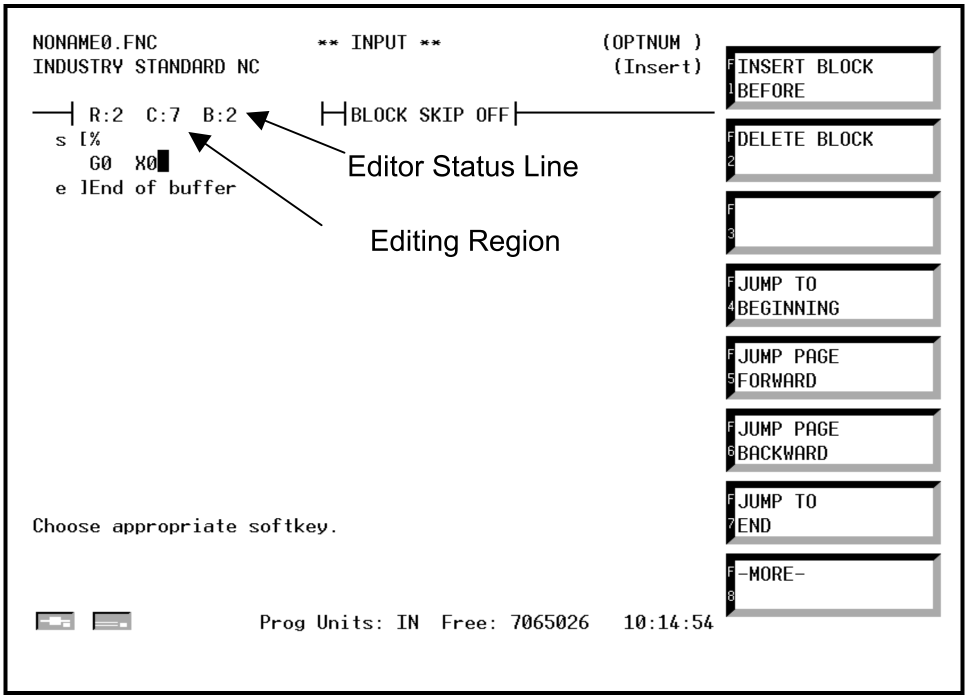


Figure 2. NC Editor Screen

The percent sign indicates the beginning of the program and the End of Buffer line indicates the end of the program.

The *Optnum/Autonum* status label in the upper right-hand corner of the display shows whether the program *numbering* is done manually by the operator or automatically by the system.

- In Optnum, either exclude the sequence numbers or type them at the beginning of each new programming block.
- If the field contains Autonum, the system automatically numbers the lines.

Using Autonum, the system assigns numbers to the new blocks created at the end of the program (inserted before the "End of Buffer"). If a block is inserted between two sequence numbered blocks, the editor splits the difference. For example, if the *numbering increment* is by tens, a block inserted between N30 and N40 is numbered N35. If blocks are inserted until the difference can no longer be divided, the editor automatically renumbers the program beginning with the new block and ending with the last block in program memory. Refer to the "Editing NC Part Programs" section for more information about the default increment for automatic numbering.

The *Insert/Over Indicator* is in the upper right-hand corner of the screen. It tells you whether the editor is in character insert (Insert) or character overwrite (Over) data entry mode.

- When using Insert mode, entered characters are inserted in front of the current character.
- In Overwrite mode, entered characters replace the currently highlighted character and move the cursor to the next character.

The *Editor Status Line* provides the following information:

- **Cursor Position**—**R** for cursor row, **C** for cursor column, and **B** for current Block Number. The Block number is the physical block including the initial percentage sign and not the sequence numbers entered by the operator.
- **Block Skip State**—whether blocks with the Ignore character (/) are interpreted and executed (Block Skip Off) or ignored (Block Skip On).
- **Free: ____%**—indicates the number of bytes available for NC program blocks.

Large Programs

If the program does not fit into the control's memory, the machine may delay because it runs out of part data to execute. If this situation occurs, the Z axis retracts from the part and a "Reloading Buffer…Tool Has Been Raised From Part Surface" message appears. After more data is loaded into the memory, the Z axis returns to the part surface and program execution continues. You can change the Depletion Retract Distance (the distance the Z axis retracts from the part while waiting for more data) using the General Parameters screen.

The block number reported in error checking is based on the number of blocks from a program's beginning. In a large part program, the block containing the error may no longer be in the control's memory. The part program will need to be reloaded to find the block referenced by the error.

Consider upgrading the memory capacity for the control if you use a lot of programs that span the control's current memory capacity. Upgrading memory is inexpensive and eliminates the need for program reloading.

Allocation

The available memory appears in the Free: xxxxxxx field at the bottom of each screen, where xxxxxxx represents the amount of available memory, including the reserved 64KB.

Each NC block that is loaded has some associated memory overhead (21 bytes per each line of NC, regardless of line length) and is included in the total bytes allowed. The number of bytes for an NC program is displayed on the Current Directory screen.

For every line in an NC part program (including the % and E lines) an additional 21 bytes of memory are used for formatting and displaying the program. Therefore, a 1,000 line program will consume an additional 21,000 bytes of program memory for overhead.

The amount of memory needed to load in a program is based on the sum of these values:

- Size of program
- Number of lines x 21
- 64KB

- **System Message Area**—displays program search responses and related messages. The pop-up message “String not found” in the screen below indicates that G53 is not in the program.

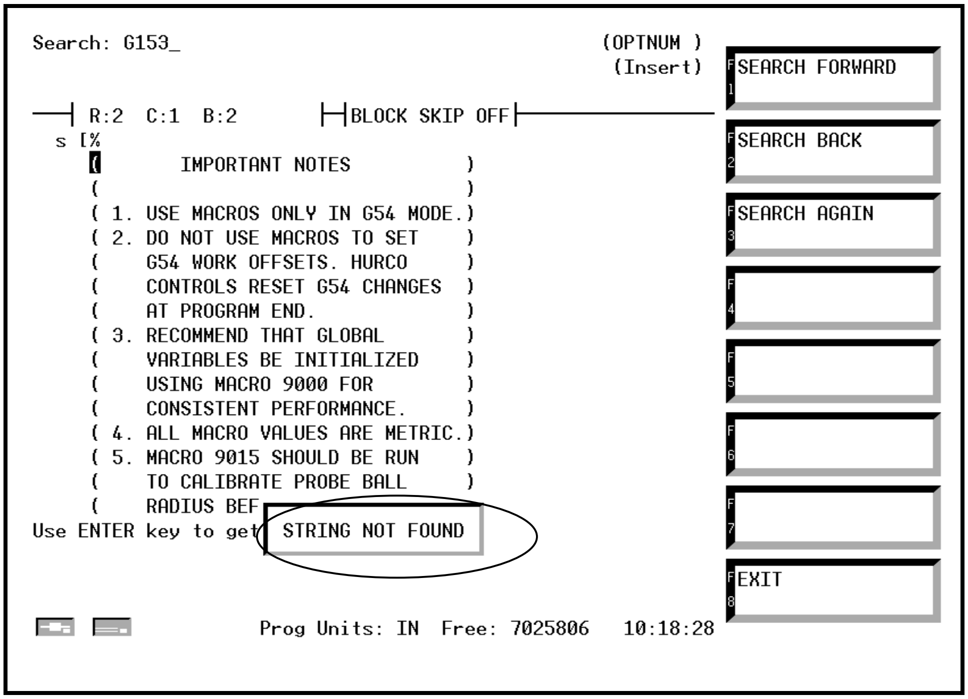


Figure 3. NC Editor’s System Message

The area in the center of the screen is the *NC Editing Region* where you enter the NC program. The four spaces to the left of the program block are reserved for program block indicators such as those listed below:

Program Block Indicator	**Definition**
/	Ignore character
s	Program start
e	End markers
[	Graphics start
]	Graphics stop
#	Graphics start and stop on same line
0 to 9	Tag number markers
*****	Tag range marker

The NC Editor provides *Syntax Checking* which ensures that the characters in a program are legal and in the proper order. This section describes this process.

The syntax checking facility searches for the following problems:

- Invalid or incomplete blocks.
- Invalid or incomplete address codes.
- Numeric errors.

In checking a block for errors, the system ensures that legal characters are entered for the currently active G codes. Blocks that violate the syntax rules appear on the screen with the “ERR” label to their left, as shown circled below:

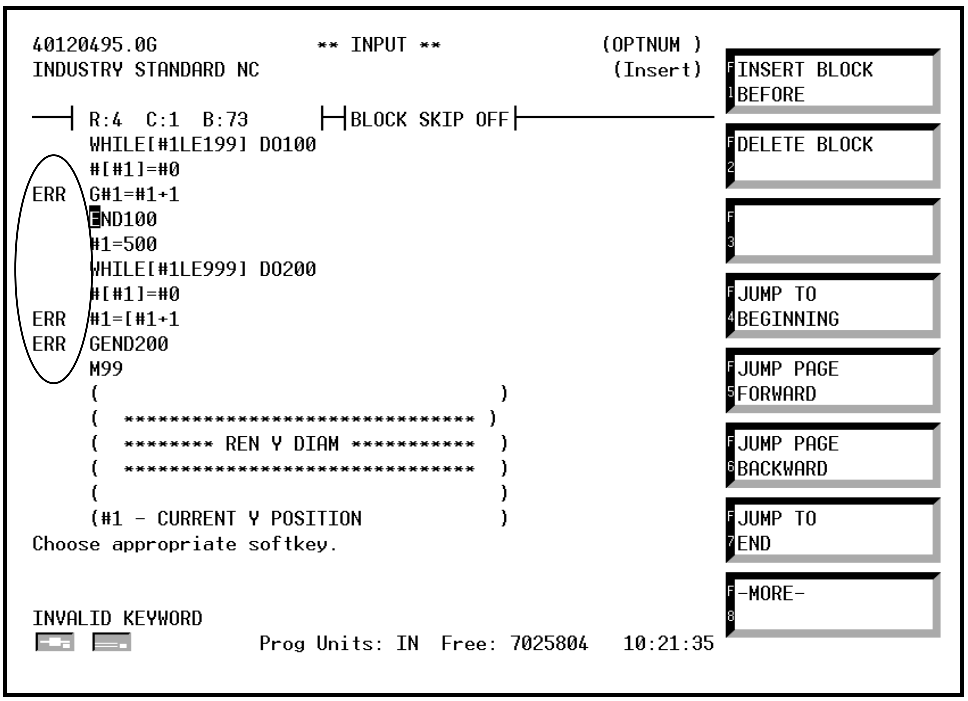


Figure 4. Syntax Error

Hints and Tricks

To avoid errors, set the units of measurement that will match those in the imported program. This is accomplished by selecting the Inch or Metric softkey when switching to the NC editor.

The NC software performs *Range Checking*. Ranges are specified after scaling. Several different levels of range checking are performed on values used in NC programs. Since all the alphabetic characters except G, M, N, O, and ":" can be used to pass parameters to subprograms, all the *address characters* are tested to verify that they are within the range -999999.99999 to +999999.99999. Sequence numbers from 0 to 9999999 are allowed.

The interpreter level, a more stringent range checking, is also performed on some address values because of their special meaning. For example, a more stringent limit is put on F (feedrate).

Additional range checking is performed at the navigation level to prevent the machine from exceeding physical limitations. More restrictive ranges may be implemented for various machine limitations.

This table lists the ranges for NC address characters:

Address Character	Definition	English Range	Metric Range
:	Program Number	0.0 to 9999	0.0 to 9999
F	Feedrate	0.0 to 999.9	0.0 to 99999.9
G	G Command	0.0 to 255.0	0.0 to 255.0
H	Tool Offset	0.0 to 200.0	0.0 to 200.0
K	Canned Cycle Repeat	0.0 to 6.0	0.0 to 6.0
M	M Command	0.0 to 255.0	0.0 to 255.0
N	Sequence Number	0.0 to 9999999	0.0 to 9999999
P	Program Number	0.0 to 9999	0.0 to 9999
R	Rotation Degrees	0.0 to 360.0	0.0 to 360.0
S	Spindle Speed	0.0 to 65535.0	0.0 to 65535.0
T	Tool Number	0.0 to 99.0	0.0 to 99.0

Table 1. English and Metric Ranges for NC Address Characters

Where can we go from here?

For information about editing an existing NC Part Program, go to the "Editing NC Part Programs" section in this manual.

Starting a New NC Program

To begin NC part programming, press the Input key. The following screen appears:

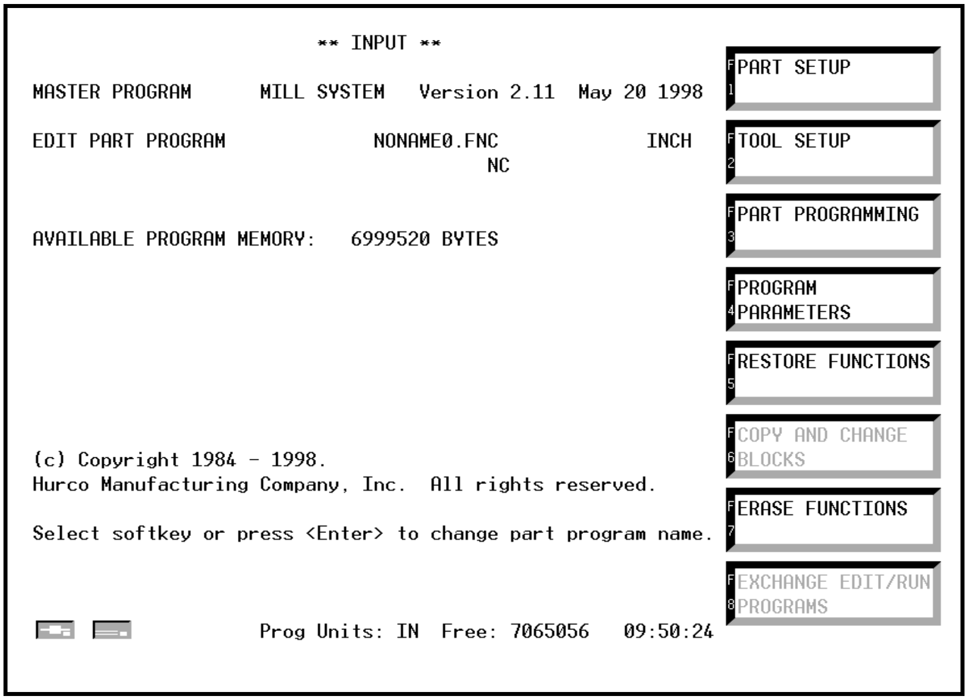


Figure 5. Input Screen Ready for NC Program Name

In the example above, the sample program has not yet been named by the operator, but it will be an NC program using inches as the units of measure.

Important

When there is no NC program loaded in memory, the system automatically assigns the name NONAME#, where the # represents a sequential number from 0 to 99. The file extension is HNC for BNC or FNC for ISNC unless the operator has changed the default extension.

Note

Refer to the "Changing the File Mask" section of the *Getting Started with Ultimax Manual* for information on how to change the default file extension for NC programs.

Follow these steps to *name the program*:

1. Move the cursor to the name field.
2. Press the console Enter key.
3. Enter a more descriptive name.
4. Press Enter again to complete.

Where can we go from here?

At this point, enter the program name, the *part setup information*, the *tool descriptions*, and/or the *program parameters*. These functions are described in the *Getting Started with Ultimax Manual*.

Caution

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

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. Refer to the "Erase Functions" section of the *Getting Started with Ultimax Manual* for instructions for removing this old information and entering descriptions for the new NC program.

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 spindle movements that will be necessary during cutting.

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)

Note

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 in the NC part program 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.

Modifying an NC Part Program

Hurco's NC system provides many levels of program editing as well as editing tools to simplify the task. This section describes making changes in NC part programs, editing groups of blocks including tagging, copying, and moving ranges of blocks, and using jump and search functions.

Basic Programming Functions Main Menu

While entering NC codes to create blocks, you may wish to display different sections of the part program on the screen, delete blocks, or insert new blocks into a section. These basic programming options are depicted in the following screen:

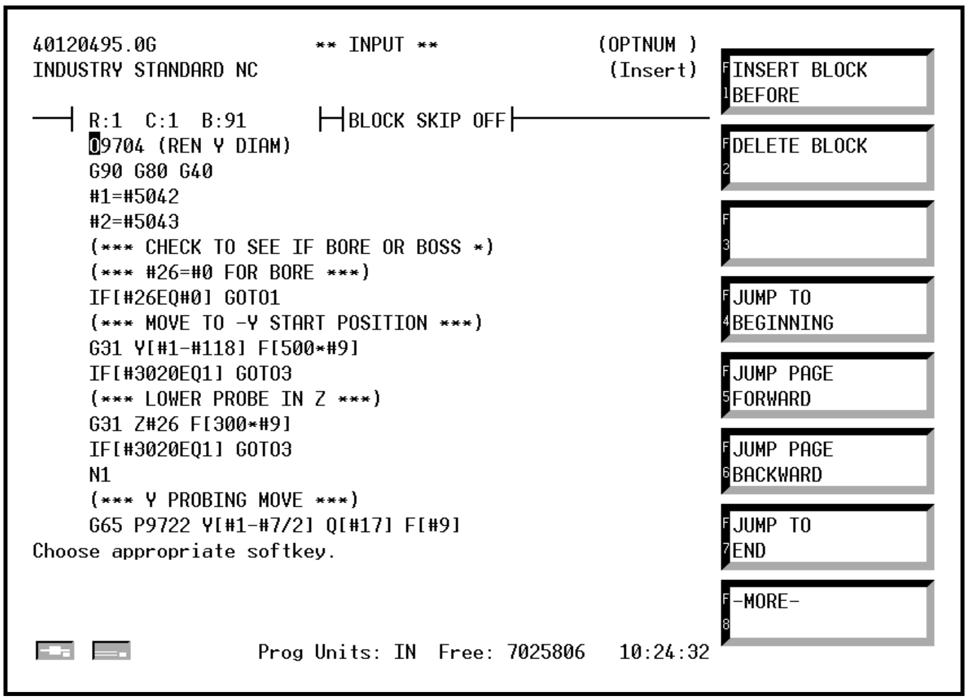


Figure 6. NC Editor's Main Menu

The softkeys provide these basic editing capabilities:

- **Insert Block Before (F1)** – makes a blank line before the block where the cursor is located. This permits addition of a new block of data.
- **Delete Block (F2)** – removes the block where the cursor is positioned.
- **Jump to Beginning (F4)** – moves the cursor to the beginning of the first program block in memory.
- **Jump Page Forward (F5)** – moves the cursor to the beginning of the last block on the screen. If the cursor is already on the last block of the screen, the next page of blocks is displayed with the cursor at the top of the page.
- **Jump Page Backward (F6)** – moves the cursor to the beginning of the first block on the screen.
- **Jump to End (F7)** – moves the cursor to the beginning of the last program block in memory.
- **More (F8)** – displays the next group of softkeys.

Search and Edit Functions

The softkeys in the second group provide additional searching and editing functions.

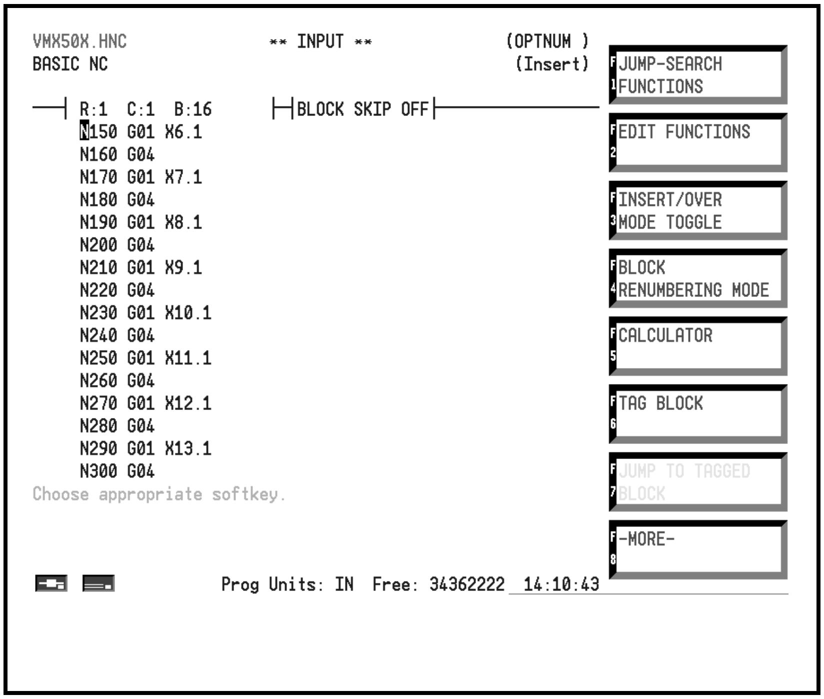


Figure 7. NC Editor's Menu

When More (F8) is pressed from the first menu, additional softkeys provide these capabilities:

- **Jump-Search Functions (F1)** – displays a submenu used to locate specific codes and blocks in a part program.
- **Edit Functions (F2)** – displays a submenu used to change blocks in a part program.
- **Insert/Over Mode Toggle (F3)** – switches the data entry style between insert and overwrite.
- **Block Renumbering Mode (F4)** – displays the numbering submenu used to locate and change block renumbering.

- **Calculator (F5)** – performs calculations on the numeric portion of a word. This feature is often used to recalculate an axis position.

  First, position the cursor on the numerical portion of the word that must be changed and press this softkey. A calculation area (Calculator:) appears at the top of the screen. Then type the complete equation using a plus (+), minus (-), multiplication symbol (*), or division line (/) to indicate the type of calculation. Finally, press the Enter key to perform the calculation.

- **Tag Block (F6)** – tags up to 10 blocks (0 to 9). The editor displays the tag number to the left of the current block. To tag a block, place the cursor in the field you wish to tag and press the Tag Block (F6) softkey. Once a block is tagged, it remains tagged until the tag number is reused. Tag numbers are reused after the last number (9) has been assigned. The screen below shows four tagged blocks:

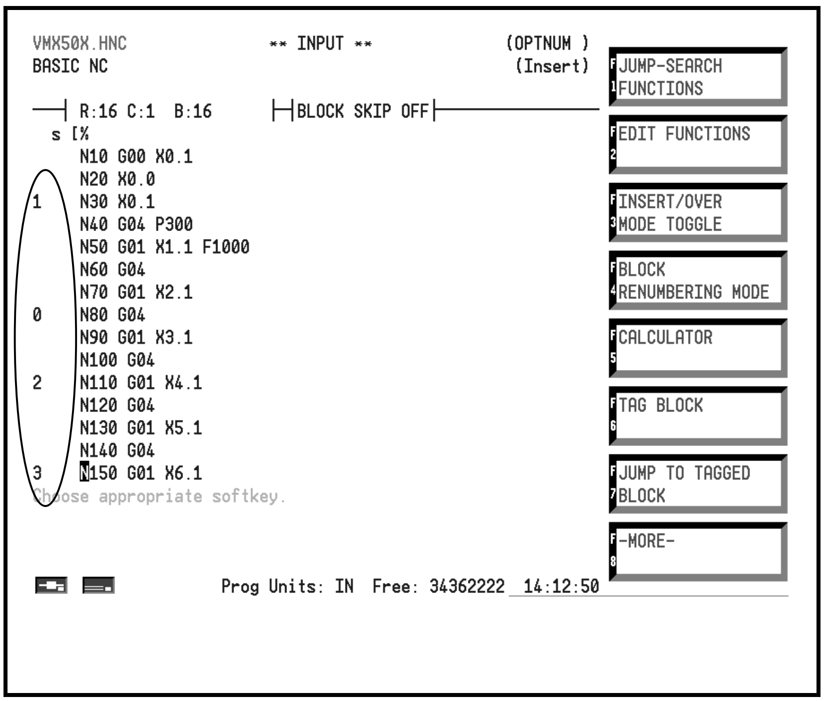


Figure 8. Tagged Blocks

- **Jump to Tagged Block (F7)** – allows the tag number to be entered. When Enter is pressed, the system positions the cursor at the selected tagged block.

- **More (F8)** – displays the next group of softkeys.

The sub-menu Jump-Search Functions (F1), Edit Functions (F2), and Block Numbering Mode (F4) softkeys on the Search and Edit Functions screen contain additional softkeys. They provide additional programming capabilities.

The *Jump and Search* functions provide 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.

Pressing the Jump-Search Functions (F1) softkey changes the softkey selection and allows access to the Search Functions softkeys shown below:

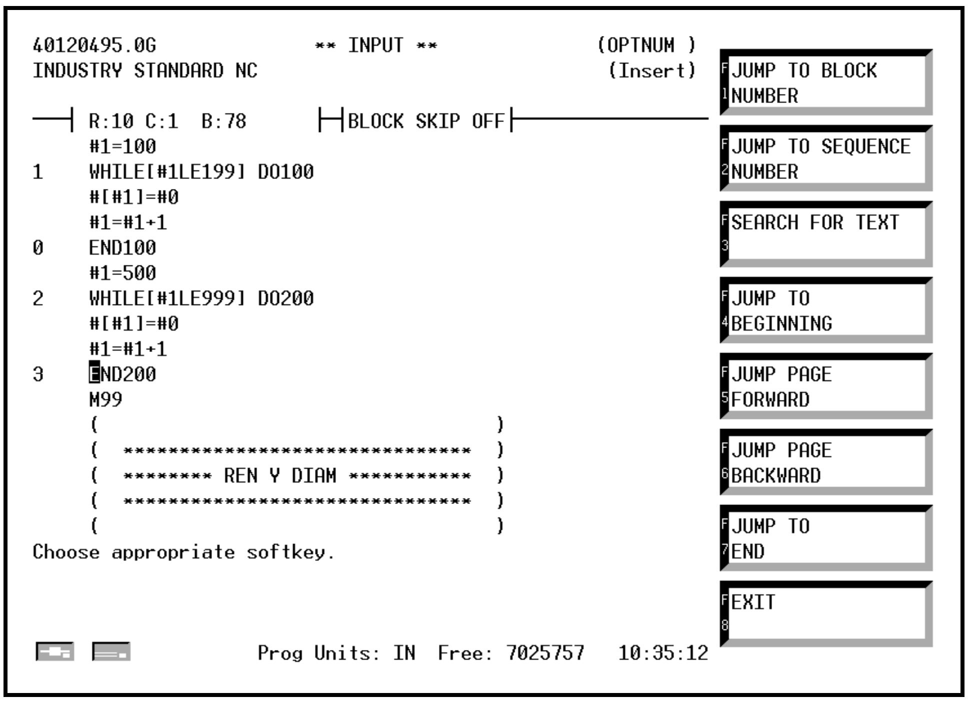


Figure 9. Search Functions Softkeys

The following are the Jump and Search softkeys and functions:

- **Jump to Block Number (F1)** – positions the cursor on blocks according to their places in NC memory (not according to their sequence numbers). After this softkey is pressed, the system displays an area at the top of the screen where the number of the line in the program can be entered.

  Pressing the plus key (+) before typing the number tells the editor to jump forward in the program by the specified number. Pressing the minus key (-) before typing the number tells the system to jump backward in the program by the specified number.

  Entering an unsigned number jumps the cursor to that block number. For example, if a 5 is entered, the cursor jumps to the fifth block number.

  Specifying a block number that is larger than the number of blocks in a program causes the cursor to jump to the last block of the program.

- **Jump to Sequence Number (F2)** – positions the cursor on the N code line. After pressing this softkey, the system displays an area at the top of the screen to type in the sequence number of the line in the program you wish the cursor to jump to.
- **Search for Text (F3)** – changes the Jump-Search softkeys to the Search Functions softkeys described at the end of this section.
- **Jump to Beginning (F4)** – positions the cursor at the beginning of the part program.
- **Jump Page Forward (F5)** – displays another screen full of NC blocks after the currently displayed screen full of blocks or places the cursor at the bottom of the NC editing region.
- **Jump Page Backward (F6)** – displays the NC blocks before the currently displayed screen full of blocks or places the cursor at the top of the NC editing region.
- **Jump to End (F7)** – positions the cursor at the end of the part program.
- **Exit (F8)** – re-displays the previous group of softkey options.

Pressing the Search for Text (F3) softkey on the Jump and Search submenu changes the Jump-Search softkeys to the *Search Functions softkeys* depicted below:

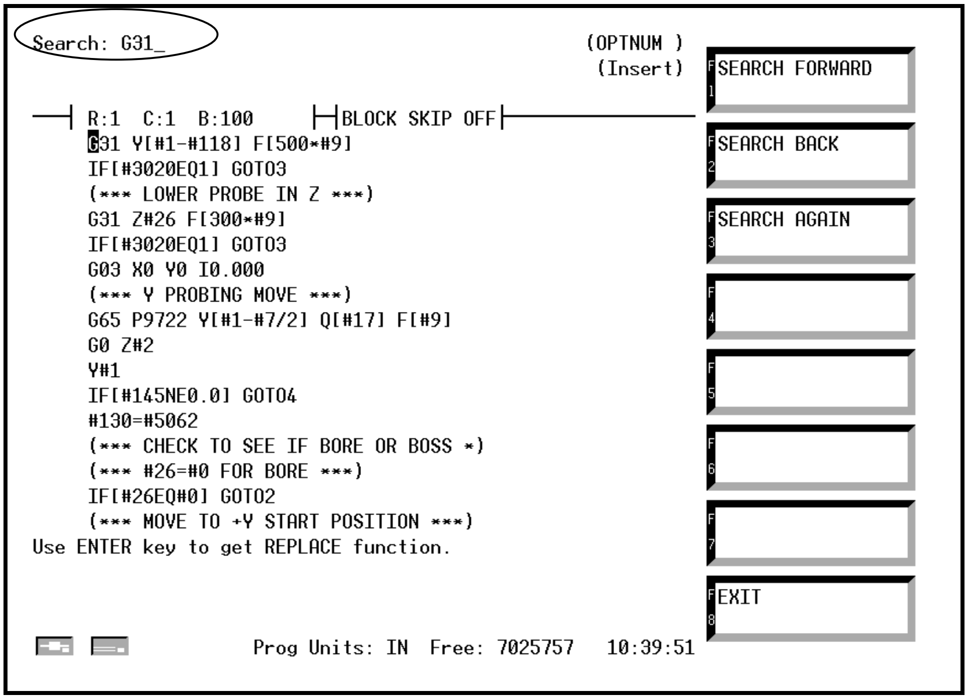


Figure 10. Search Softkeys

To perform *search* operations, follow these steps:

1. If a character is in the Search field from a previous search, press the console Clear (C) key. Type the address letter.
2. Enter a numeric value at this point, or press Enter to type in a replacement string.
3. Select the Search Forward (F1) softkey or the Search Back (F2) softkey.
   - To continue searching select the Search Again (F3) softkey.
   - To change the direction of the search, select the appropriate softkey—Search Back followed by Search Forward or vice versa).
4. Continue to press the softkey to search until the block is found or until the “No more occurrences” message appears.

To *replace* the search text throughout the program, follow these steps:

1. Type in the search text at the top of the screen.
2. Press the Enter key.
3. Verify the replace operation by pressing the Replace Yes softkey. This operation may be repeated.
4. Press the Replace No softkey to stop the replace operation.
5. Press the Exit (F8) softkey to return to the original Jump and Search softkeys.

To perform *editing operations* on groups of blocks within the NC part program, follow these steps:

1. Press More (F8) on the main menu screen to display the Edit Functions (F2) softkey.
2. Press F2, and the following softkeys appear:

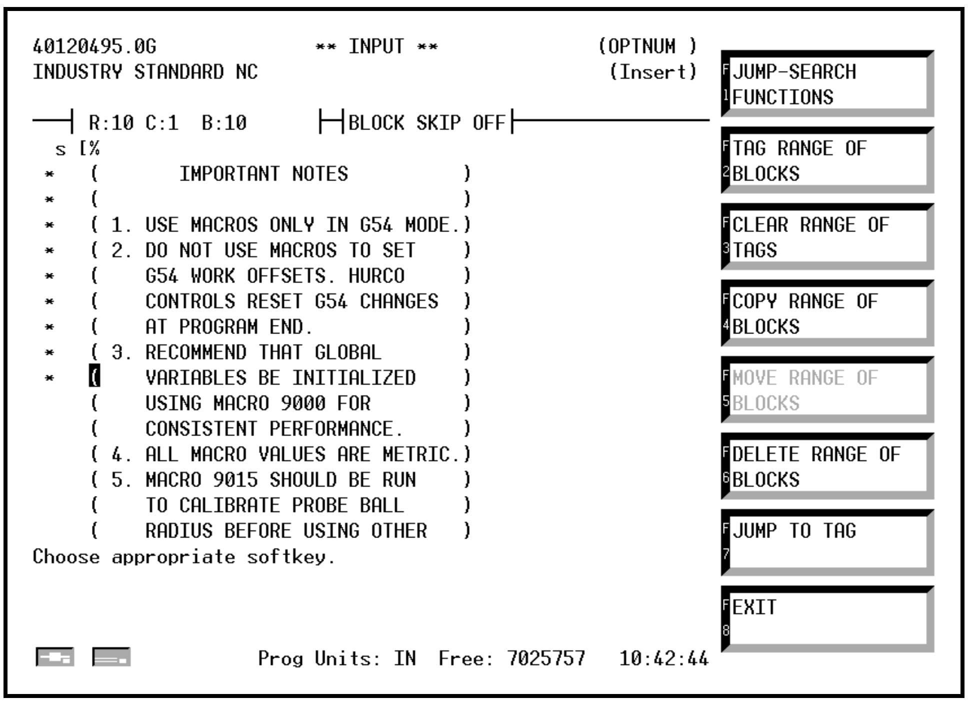


Figure 11. Edit Functions Screen

Important

Before performing any of the edit functions, identify the blocks to move, copy, or delete by tagging a range of blocks. Use the Tag Range of Blocks (F2) softkey to group the blocks for editing functions.

To *tag one or more blocks*, follow these steps:

1. Position the cursor on the first block to be tagged.
2. Press the Tag Range of Blocks (F2) softkey to insert an asterisk (*) in the left-hand column.
3. Use the Arrow keys to move the cursor to a different block within the program. All blocks between the first and last tagged blocks are marked with asterisks as shown below:

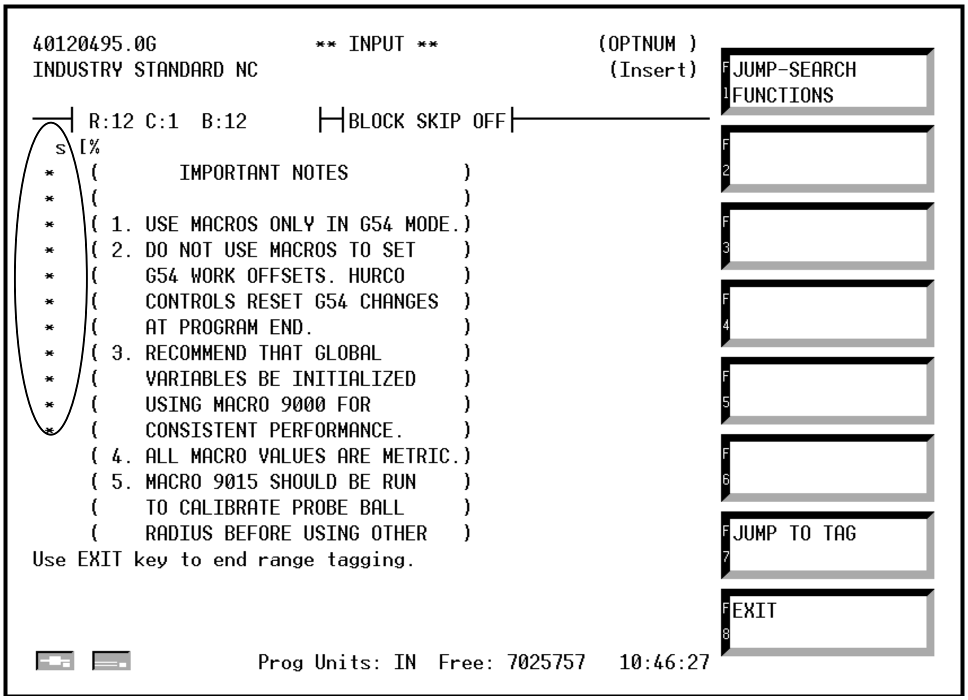


Figure 12. Tagged Range of Blocks

4. Press the Exit (F8) softkey to complete the block tagging operation.

Note

Refer to the “Clearing a Tag Range” section for information about erasing tags.

To *copy a block or a range of blocks* to another location in the program, follow these steps:

1. Place the cursor where the tagged blocks are to be copied.
2. Press the Copy Range of Blocks (F4) softkey.
    - One or more blocks must be tagged with an asterisk (*) before the copying operation can be performed.
    - The system asks if the block should be copied before the currently selected block. Press the Yes (F1) softkey to copy the blocks or the No (F8) softkey to cancel the operation.

To *move a range of blocks* to another location, follow these steps:

1. Place the cursor where the tagged blocks are to be moved.
2. Press the Move Range of Blocks (F5) softkey.
    - One or more blocks must be tagged with an asterisk (*) before the move operation can be performed.
    - The system asks if the block should be moved before the currently selected block. Press the Yes (F1) softkey to move the blocks or the No (F8) softkey to cancel the operation.

Important

The *sequence numbers* of the copied or moved range of blocks are not changed, so there will be duplicate sequence numbers in the program unless the program is re-numbered. However, this will not affect program execution since the system ignores the sequence numbers while running the program (except when GOTOs or M99 jump statements are used in the program with the NCPP option).

Also, check the program to be certain that the copied G codes do not cancel active G codes and create errors in the program.

To *delete a range of blocks* in the program, follow these steps:

1. Place the cursor on the group of blocks to be erased.
2. Press the Delete Range of Blocks (F5) softkey. The system asks for confirmation to delete the tagged blocks. Press the Yes (F1) softkey to delete the blocks or the No (F8) softkey to cancel the operation.

Important

The program *sequence numbers* will have a gap in the numeric order unless the program is re-numbered. This does not affect program execution.

Be sure to check the program so that the deleted G codes do not create errors in the program.

To *erase the tag marks*, follow these steps:

1. Press the Clear Range of Tags (F3) softkey.
2. Select either the Yes or No softkey to confirm the operation.

Important

Clearing the tag range does not delete the blocks.

To change the *Numbering Increment* for automatic numbering, follow these steps:

1. From the main menu screen, press the More (F8) softkey.
2. Press the *Block Renumbering Mode* (F4) softkey. The softkey options change as shown below:

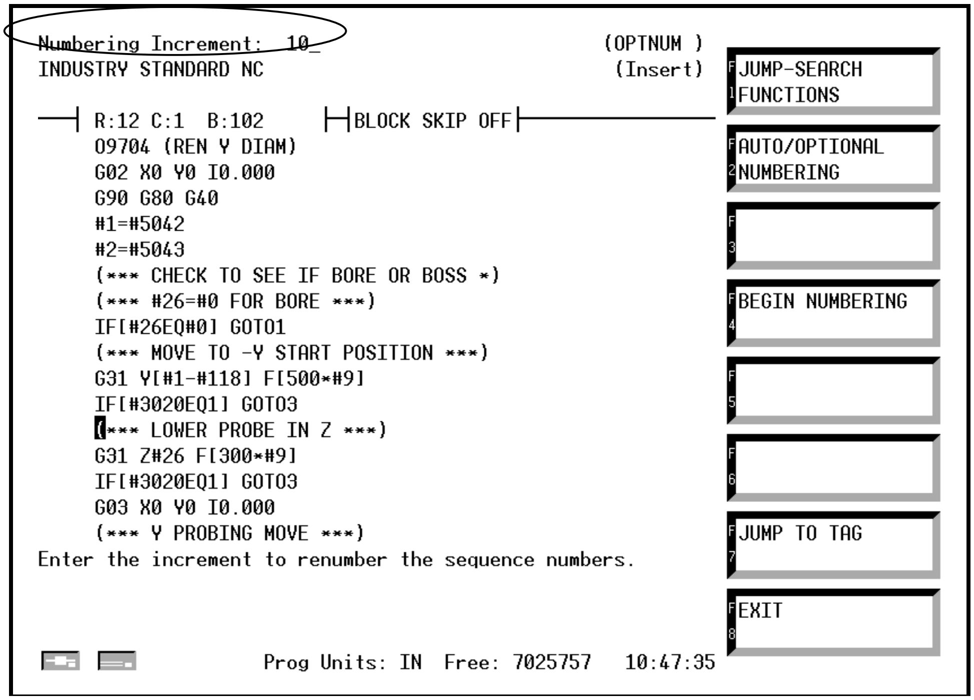


Figure 13. Numbering Submenu

The default automatic numbering increment appears in the *Numbering Increment* field in the top left corner of the screen. To change this setting, follow these steps:

1. Press the left Arrow key.
2. Type the new number; most operators use increments of 5 to 20. Up to four digits may be used.
3. Press the Exit (F8) softkey to access more softkey functions.

The softkeys on the Numbering submenu have these functions:

- **Jump-Search Functions (F1)** – provides search and replace codes as well as jumping to different locations in a part program.
- **Auto/Optional Numbering (F2)** – switches the line numbering method between Autonum (system automatically numbers) and Optnum (operator enters the numbers).
- **Begin Numbering (F4)** – renumbers all of the blocks that have sequence numbers according to the selected increment. Blocks that do not have sequence numbers will not be re-numbered.
- **Jump to Tag (F7)** – positions a previously tagged block at the top of the editing region.
- **Exit (F8)** – re-displays the previous sub-menu softkeys.

Graphics Markers and Syntax Errors

The third group of softkey options allows a portion of a part program to be selected by setting start and end markers on the Graphics screen. It also finds and positions the cursor on a syntax error in a part program.

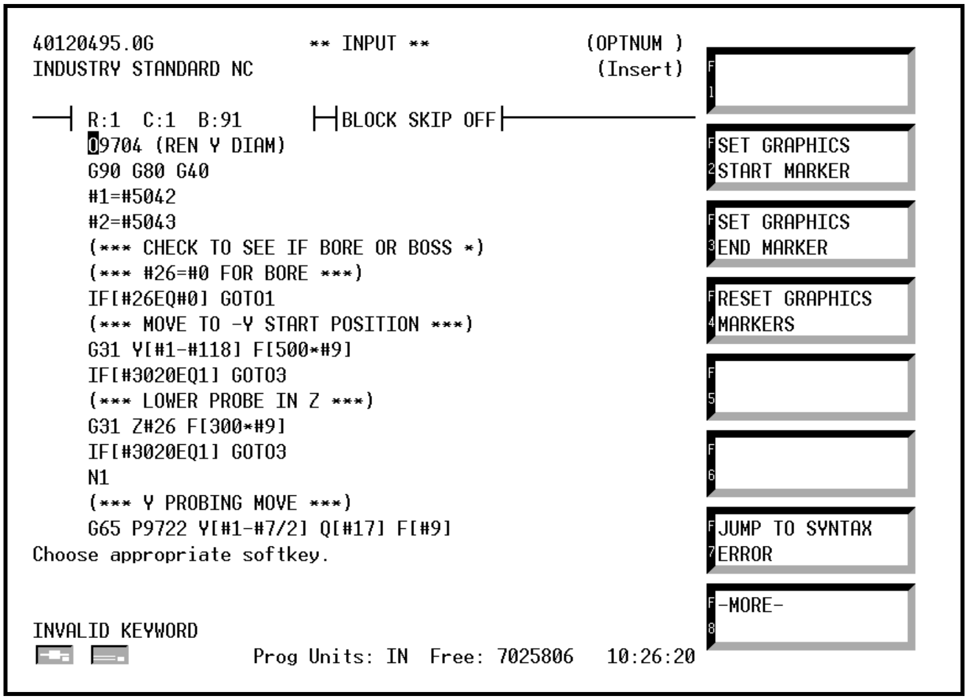


Figure 14. NC Editor Menu

This group of softkeys provides these functions:

- **Set Graphics Start Marker (F2)** – marks the current block as the beginning point for the graphics display by inserting a left bracket "[" to the left of the block. If the start and end blocks are set as the same block, a pound sign "#" appears before the block.
- **Set Graphics End Marker (F3)** – marks the current block as the ending point for the graphics display by inserting a right bracket "]" to the left of the block. If the start and end blocks are set as the same block, a pound sign "#" appears before the block.
- **Reset Graphics Markers (F4)** – returns the start and end graphics markers to their default settings. The defaults are the beginning and end of the program.
- **Jump to Syntax Error (F7)** – searches for the next invalid block. When it reaches the end of the program, the system begins searching from the beginning of the program. When the system finds a block with errors, the cursor stops at the block, and the cursor is positioned on the incorrect word. An error message also appears at the bottom of the screen to explain the error.
- **More (F8)** – displays the next group of softkeys.

Program Execution

Use the fourth group of software keys from the main menu to prepare for program execution or to restrict the program execution to a specific section. To access this group of options, press More (F7) on the main menu until the following softkey group appears:

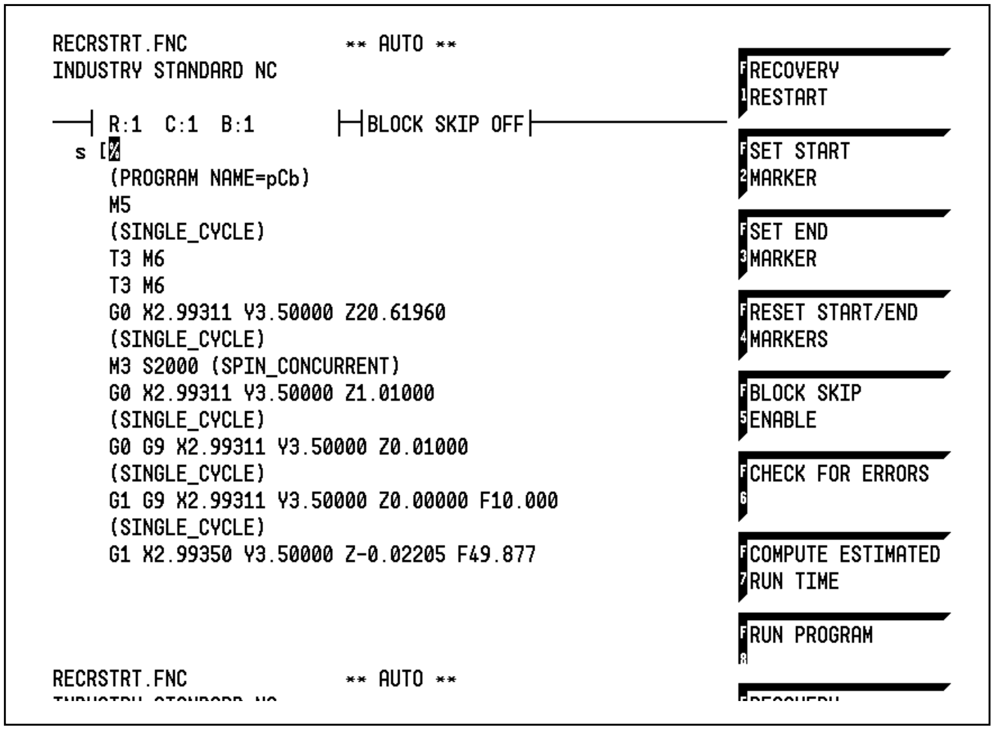


Figure 15. NC Editor Menu

These softkeys provide the following preparation facilities for program execution:

- **Recovery Restart** – restart an NC program after it was aborted either by the machine or operator.
    - Set Restart Marker – manually set the restart marker. An ‘r’ will appear to the left of the block selected, and will disappear if the program successfully runs.
    - Auto Set Restart Marker – Ultimax automatically marks the last block being executed when the program was stopped, or where an error occurred during error checking.
    - Reset Restart Marker – clears the Restart Marker and cancels the Recovery Restart operation.This marker can be used on a block that follows a G41 or G42.
- **Set Start Marker** – indicates the block that the system should use to start program execution when running the program from the Auto screen.
- **Set End Marker** – indicates the block that the system should use to end program execution.
- **Reset Start/End Markers** – restores the start and end markers to their defaults. The defaults are the beginning and end of the program.
- **Block Skip Enable** – switches on the block skipping facility during program execution. With this facility enabled, the system skips the blocks that were previously marked using the Toggle Block Skip softkey on the main menu.
- **Check for Errors** – searches for errors in blocks. If no errors are found, the system displays the estimated run time. If the system finds an error, the softkey labels are removed, and an error message appears at the bottom of the screen. Press the Enter key to continue, and the system positions the cursor before the incorrect block on the editor screen. Then correct the error.
- **Compute Estimated Run Time** – determines the approximate length of time it will take to run the program. This estimate does <u>not</u> include the time required for communication, tool changes, or program stops.
- **Run Program** – initiate program execution and display monitoring information. If the machine is not calibrated, the Manual screen immediately displays.

File and Program Selection or Deletion

The fifth group of softkeys for the NC menu is used to create new files, access files, locate programs, and delete files.

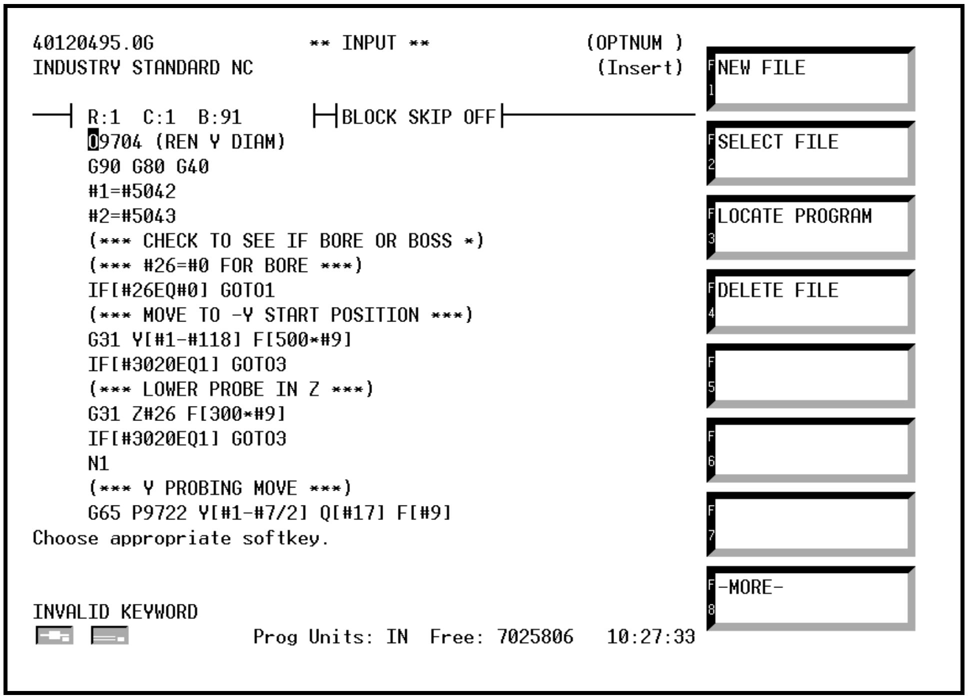


Figure 16. *NC Editor Softkeys*

These softkeys change as described on the following pages. To return to the menu for this group after selecting a softkey, use the Exit (F8) softkey, which appears on each submenu.

These softkeys provide the following file and program features:

- **New File (F1)** – creates a new file from a blank program template.
- **Select File (F2)** – selects a previously loaded file for editing or program execution. A list of files currently loaded in memory appears on the screen. The current NC file has an arrow to the left of the filename.

  Select a file by highlighting it and pressing the Select (F1) softkey. The graph is automatically re-scaled when the program is run to ensure that the drawing is scaled properly.

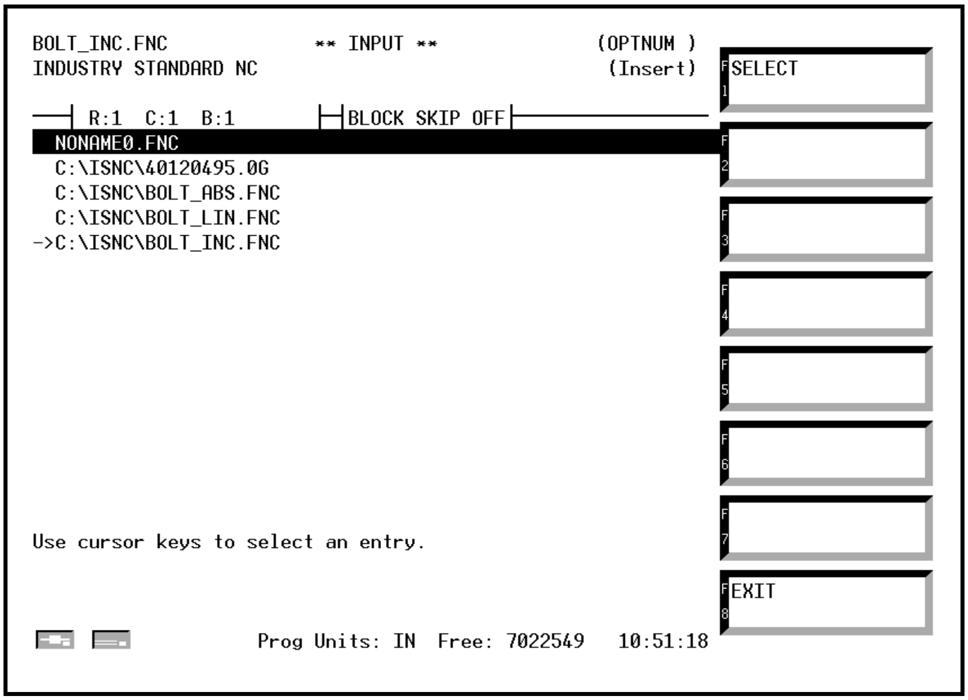


Figure 17. Select File Screen

Important

When disk operations are used to load a number of files into memory, the last file loaded is selected by default. That file is available for editing. Program execution begins with the first block of that file.

To reduce the number of Duplicate Program error messages, the software checks to see if each file is currently loaded into memory. If the file is already in memory, a prompt appears asking if the file should be overwritten. Select Yes (overwrite) or No (abort) through the softkeys.

- **Locate Program (F3)** – selects a previously loaded program number for editing or program execution. A list of program numbers currently loaded in memory appears on the screen.

  Select a program number by highlighting it and pressing the Select (F1) softkey. The file appears with the cursor at the beginning of the selected subprogram. If a program number appears in more than one NC block, the NC block with the second listing of the program number is flagged with an error.

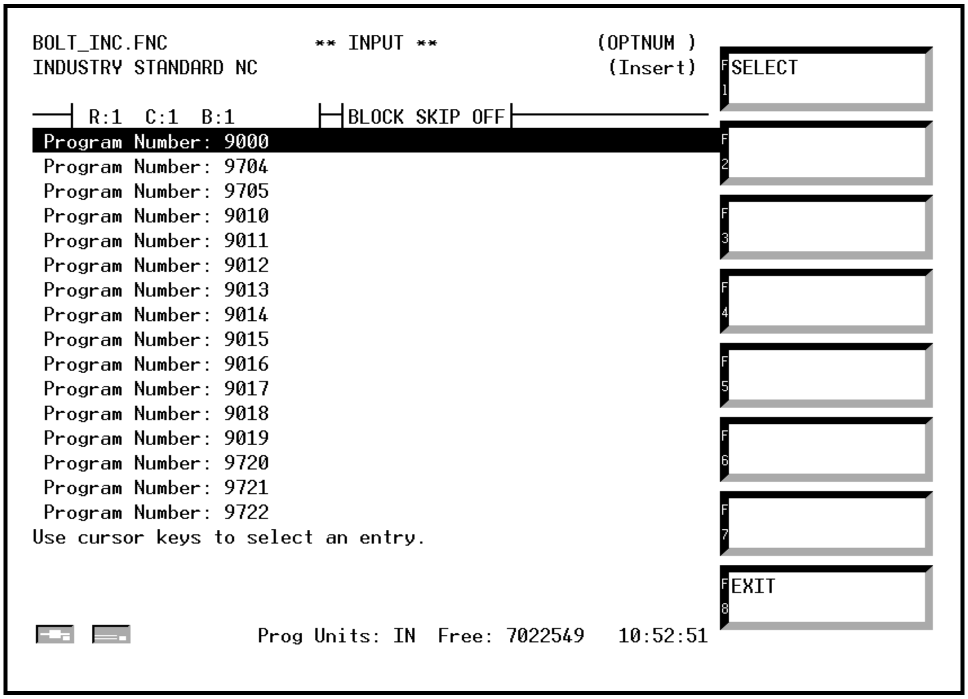


Figure 18. Locate Program Screen

- **Delete File (F4)** – erases the currently displayed NC program from temporary memoryThe deletion operation must be confirmed by pressing the Yes (F1) softkey before the program is erased. Pressing the No (F8) softkey cancels the deletion operation.

  This operation does not erase the program from permanent memory on the hard drive if it was previously saved to the hard drive.

- **More (F8)** – displays the original group of NC editing softkeys.

Where can we go from here?

Press the down arrow, and the system provides a new program block line beginning with an “N” to type in a sequence number if Optnum is being used. The system automatically inserts a complete sequence number if Autonum is being used.

Refer to the sections that follow for detailed information about entering NC codes. Then begin entering NC codes to create the block.

Distance to Go

The Distance to Go (DTG) displays real-time tool location for a data block. DTG is calculated as a straight line from the current tool position to its final destination. DTG values are displayed for rapid and machine moves, and are not calculated for ATC moves (zeros are displayed). DTG can be used in both Conversational and NC programming.

Calculating Distance to Go

For arcs and circular moves, DTG is calculated as the linear distance to the end point of the arc. The DTG for the following figure is 1, 2, 0.

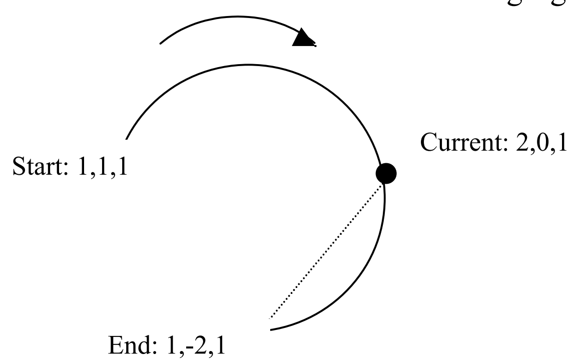


Figure 19. DTG for Arcs and Circular Moves

For canned cycles with pecking, DTG is the distance to the bottom of the peck (not the bottom of the hole). The DTG for a mill contour is the distance from the current position to the endpoint of each segment that makes up the contour. In the following figure, the ⊗ indicates a segment end point used to compute DTG.

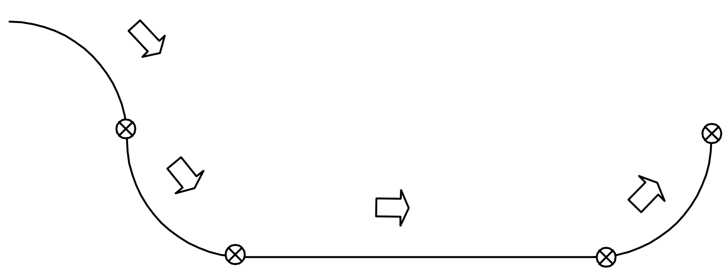


Figure 20. DTG for a Mill Contour

The DTG for a mill frame with radius is calculated as the distance to the endpoint of each segment, rather than the ends of the frame.

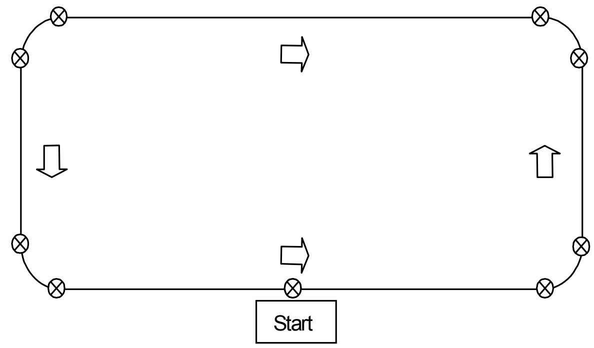


Figure 21. DTG for a Mill Frame

Using the Distance To Go Feature

From the Auto Cycle, Single Cycle or Test Run mode screen, press the Select DRO softkey.

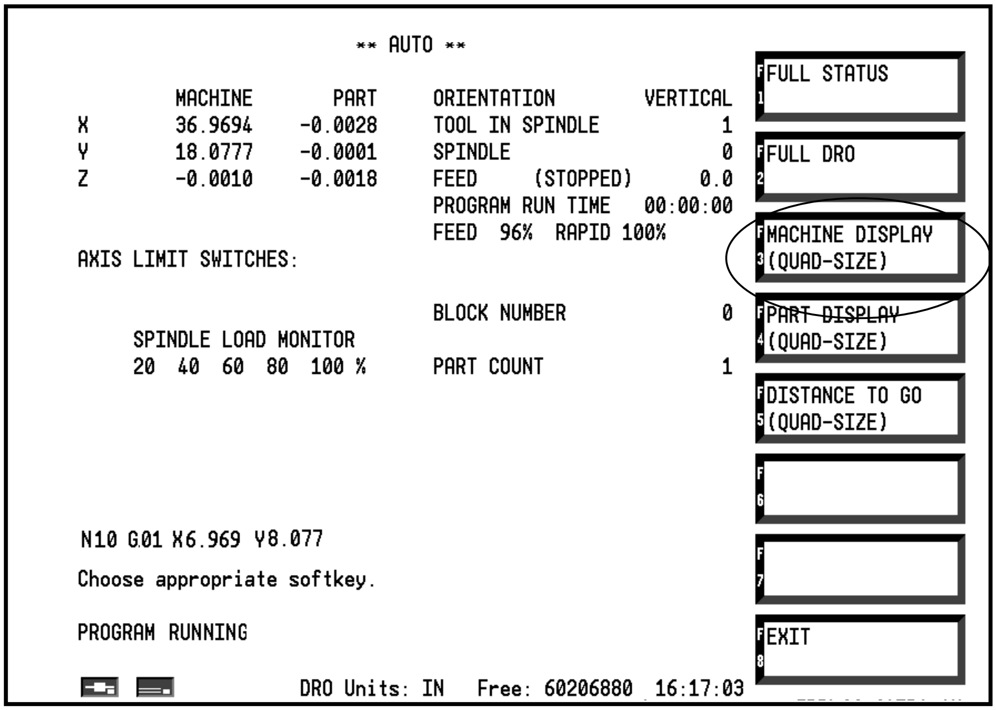


Figure 22. NC Full Status (Select DRO) Screen

Note

The Machine DRO and Machine Display (Quad-Size) functions are not available in Conversational programming.

Touch the Full Status or Exit softkeys to return to Auto Cycle, Single Cycle or Test Run mode screen.

Full DRO

Touch the Full DRO softkey to display the real-time location of Machine (NC programming only), Part and Distance to Go X, Y, and Z coordinates as shown on the following screen. The area delineated on the following figure will be different in Conversational mode.

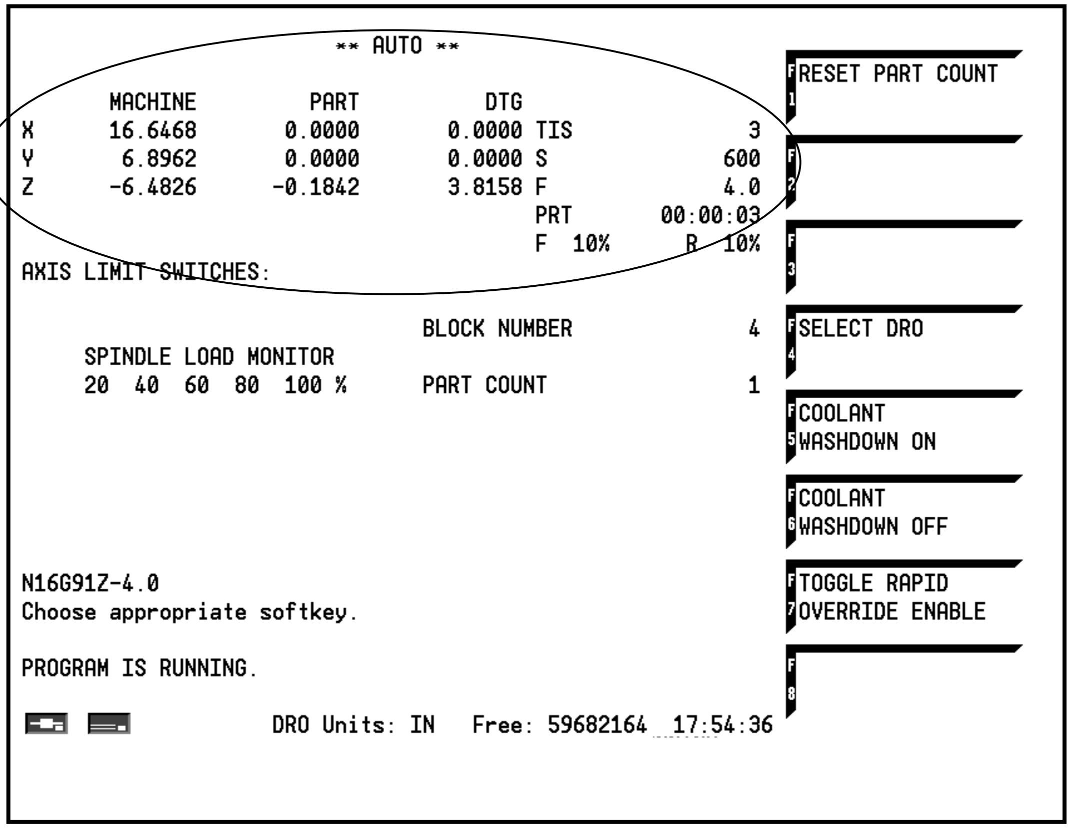


Figure 23. NC Full DRO Screen

Note

The DTG values in Single Mode will be zero until the Start Cycle softkey is pressed. The distance to the endpoint for that block will then be shown. As the tool comes closer to its destination, the DTG will count down to zero. When the endpoint is reached, the DTG will be displayed as zeros until the Start Cycle softkey is pressed and the process begins again.

Touch the Select DRO softkey to return to the DRO Select screen.

Machine Display (Quad Size)

Touch the Machine Display (Quad-Size) softkey to display an enlarged view of the Machine coordinates X_m, Y_m, and Z_m. This feature is available in NC Programming only. Touch the Select DRO softkey to return to the DRO Select screen.

Part Display (Quad-Size)

Touch the Part Display (Quad-Size) softkey to display an enlarged view of the Part coordinates X_p, Y_p, and Z_p. Touch the Select DRO softkey to return to the DRO Select screen.

Distance to Go (Quad-Size)

Touch the Distance to Go (Quad-Size) softkey to display an enlarged view of the Distance to Go coordinates X_{dtg}, Y_{dtg}, and Z_{dtg}. Touch the Select DRO softkey to return to the DRO Select screen.

Preparatory Functions - G Codes

This section defines G codes and their functions. This information is often needed when using an off-line CAM or CAD/CAM system to create NC part programs.

G Code Groups

The G codes are grouped by functions.

Group	Function	Group	Function
00	One-Shot	10	Return from Canned Cycles
01	Interpolation	11	Scaling
02	Plane Selection	12	Macro/Subprogram
03	Dimension	14	Coordinate System Selection
05	Feed	15	Precision Cornering
06	Measurement	16	Rotation
07	Cutter Compensation	17	Polar Coordinates
08	Tool Length Compensation	18	Mirroring
09	Canned Cycles	19	Program Parameters for Surface Finish/Data Smoothing

Table 1. G Code Groups

Notes

The system displays the number 010 as an alarm if an invalid G code (one not listed in the following table) is entered.

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.

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

G Code Table

This table lists the G codes, identifies the defaults (in the shaded areas), lists Modal (M) or Non-modal (N) types, identifies groups, and describes the G codes' functions.

Some G codes are strictly BNC or strictly ISNC, and are identified as such in this manual. Otherwise, the G codes apply to either dialect.

G Code	Type	Group	Function
G00	M	01	Positioning (Rapid Traverse)
G01	M		Linear Interpolation (Cutting Feed)
G02	M		Circular Interpolation/Helical CW
G02.4	M		3D Circular Interpolation CW
G03	M		Circular Interpolation/Helical CCW
G03.4	M		3D Circular Interpolation CCW
G04	N	00	Dwell, Exact Stop
G05.1	M	19	Surface Finish Parameters
G05.2	M	19	Data Smoothing
G09	N	00	Decelerate Axis to Zero
G10	N		Data Setting
G11	N		Data Setting Mode Cancel
G15	M	17	Polar Coordinates Cancel
G16	M		Polar Coordinates
G17	M	02	XY Plane Selection
G18	M		ZX Plane Selection
G19	M		YZ Plane Selection
ISNC G20	M	06	Input in Inch
ISNC G21	M		Input in mm
G28	N	00	Return to Reference Point
G29	N		Return from Reference Point
G31	N		Skip Function
G40	M	07	Cutter Compensation Cancel
G41	M		Cutter Compensation Left
G42	M		Cutter Compensation Right
G43	M	08	Tool Length Compensation + Direction
G44	M		Tool Length Compensation - Direction

Continued on next page.

Continued from previous page.

G Code	Type	Group	Function
G45	N	00	Tool Offset Increase
G46	N		Tool Offset Decrease
G47	N		Tool Offset Double Increase
G48	N		Tool Offset Double Decrease
G49	M	08	Tool Length Offset Compensation Cancel
G50	M	11	Scaling Cancel
G51	M		Scaling
G50.1	M	18	Mirroring Cancel
G51.1	M		Mirroring
G52	N	00	Local Coordinate System Setting
G53	N		Machine Coordinate System Selection
G54	M	14	Work Coordinate System 1 Selection
G55	M		Work Coordinate System 2 Selection
G56	M		Work Coordinate System 3 Selection
G57	M		Work Coordinate System 4 Selection
G58	M		Work Coordinate System 5 Selection
G59	M		Work Coordinate System 6 Selection
G61	M	15	Decelerates to Zero–Precision Cornering
G64	M		Cancels Precision Cornering
G65	N	12	Macro Command, Subprogram Call
G66	M		Modal Subprogram Call
G67	M		Modal Subprogram Call Cancel
G68	M	16	Coordinate Rotation
G69	M		Coordinate Rotation Cancel
BNC G70	M	06	Input in Inch
BNC G71	M		Input in mm
G73	M	09	Peck Drilling Cycle
ISNC G74	M		Left-handed Tapping Cycle
ISNC G74 with M29	M		Rigid Tapping
BNC G74	M	01	Single-quadrant Circular Interpolation
BNC G75	M		Multi-quadrant Circular Interpolation

Continued on next page.

Continued from previous page.

G Code	Type	Group	Function
G76	M	09	Bore Orient Cycle
G80	M		Canned Cycle Cancel
G81	M		Drilling Cycle, Spot Boring
G82	M		Drilling Cycle, Counter Boring
G83	M		Peck Drilling Cycle
G84	M		Tapping Cycle
ISNC G84.2	M		Rigid Tapping Cycle
ISNC G84.3	M		Rigid Tapping Cycle
ISNC G84 with M29	M		Rigid Tapping Cycle
G85	M		Boring Cycle
BNC G86	M		Bore Orient Cycle
ISNC G86	M		Bore Rapid Out Cycle
BNC G87	M		Chip Breaker Cycle
ISNC G87	M		Back Boring Cycle
BNC G88	M		Rigid Tapping Cycle
ISNC G88	M		Boring Cycle Manual Feed Out, Dwell
G89	M		Boring Cycle Bore and Dwell
G90	M	03	Absolute Command
G91	M		Incremental Command
G92	N	00	Programming of Absolute Zero Point
G94	M	05	Feed per Minute
G98	M	10	Return to Initial Point in Canned Cycle
G99	M		Return to R Point in Canned Cycle

Table 2. G Codes

Group	G Codes	Type	Function
00	G04	N	Dwell, Exact Stop
	G09	N	Decelerate Axis to Zero
	G10	N	Data Setting
	G11	N	Data Setting Mode Cancel
	G28	N	Return to Reference Point
	G29	N	Return from Reference Point
	G31	N	Skip Function
	G45	N	Tool Offset Increase
	G46	N	Tool Offset Decrease
	G47	N	Tool Offset Double Increase
	G48	N	Tool Offset Double Decrease
	G92	N	Programming of Absolute Zero Point
01	G00	M	Positioning (Rapid Traverse)
	G01	M	Linear Interpolation (Cutting Speed)
	G02	M	Circular Interpolation/Helical CW
	G02.4	M	3D Circular Interpolation CW
	G03	M	Circular Interpolation/Helical CCW
	G03.4	M	3D Circular Interpolation/Helical CCW
	BNC G74	M	Single-quadrant Circular Interpolation
	BNC G75	M	Multi-quadrant Circular Interpolation
02	G17	M	XY Plane Selection
	G18	M	ZX Plane Selection
	G19	M	YZ Plane Selection
03	G90	M	Absolute Command
	G91	M	Incremental Command
05	G94	M	Feed per Minute
06	BNC G70	M	Input in Inch
06	BNC G71	M	Input in mm
07	G40	M	Cutter Compensation Cancel
	G41	M	Cutter Compensation Left
	G42	M	Cutter Compensation Right

Continued on next page.

Continued from previous page.

Group	G Codes	Type	Function
08	G43	M	Total Length Compensation + Direction
	G44	M	Total Length Compensation – Direction
	G49	M	Tool Length Offset Compensation Cancel
09	G73	M	Peck Drilling Cycle
	ISNC G74	M	Left-handed Tapping Cycle
	ISNC G74 with M29	M	Rapid Tapping
	G76	M	Bore Orient Cycle
	G80	M	Canned Cycle Cancel
	G81	M	Drilling Cycle, Spot Boring
	G82	M	Drilling Cycle, Counter Boring
	G83	M	Peck Drilling Cycle
	G84	M	Tapping Cycle
	ISNC G84.2	M	Rigid Tapping Cycle
	ISNC G84.3	M	Rigid Tapping Cycle
	ISNC G84 with M29	M	Rigid Tapping Cycle
	G85	M	Boring Cycle
	BNC G86	M	Bore Orient Cycle
	ISNC G86	M	Bore Rapid Out Cycle
	BNC G87	M	Chip Breaker Cycle
	ISNC G87	M	Back Boring Cycle
	BNC G88	M	Rigid Tapping Cycle
	ISNC G88	M	Boring Cycle Manual Feed Out, Dwell
	G89	M	Boring Cycle, Bore and Dwell
10	G98	M	Return to Initial Point in Canned Cycle
	G99	M	Return to R Point in Canned Cycle
11	G50	M	Scaling Cancel
	G51	M	Scaling

Continued on next page.

Continued from previous page.

Group	G Codes	Type	
12	G65	N	Macro Command, Subprogram Call
	G66	M	Modal Subprogram Call
	G67	M	Modal Subprogram Call Cancel
14	G54	M	Work Coordinate System 1 Selection
	G55	M	Work Coordinate System 2 Selection
	G56	M	Work Coordinate System 3 Selection
	G57	M	Work Coordinate System 41 Selection
	G58	M	Work Coordinate System 5 Selection
	G59	M	Work Coordinate System 6 Selection
15	G61	M	Decelerates to Zero-Precision Cornering
	G64	M	Cancels Precision Cornering
16	G68	M	Coordinate Rotation
	G69	M	Coordinate Rotation Cancel
17	G15	M	Polar Coordinate Cancel
	G16	M	Polar Coordinates
19	G05.1	M	Surface Finish Parameters
	G05.2	M	Data Smoothing

Table 3. G Codes in order of Groups

Rapid Traverse (G00)

Rapid Traverse mode (G00) is the default and moves the axes to a specified location at the rapid feedrate programmed in the Program Parameters screen. Up to five axes (X, Y, Z, A, B) of coordinated rapid motion can be specified while in this mode.

Set either a linear or non-linear tool path on the NC Parameters screen. The linear tool path is the default. ISNC and BNC have different linear tool path modes. In the ISNC linear mode the tool motion is in all three axes (X, Y, Z) simultaneously. In the BNC linear mode the motion is divided into separate X, Y, and Z moves. The motion in the XY plane is a straight line.

The ISNC and BNC non-linear modes are the same. In the non-linear tool path mode, the XY plane motion is broken down into a 45° move and a straight line move parallel to either the X or Y axis. The determination of whether the 45° move or the straight line move is made depends first on the distances from the current position to the end position along the X and Y axes.

If it is desired that the tool move to a position which is compensated, G41 or G42 needs to be specified along with the offset before any axis coordinates are given. The rapid traverse rate is set on the General Parameters screen.

Notes

The G00 mode is canceled by using the G01, G02, G03, or canned cycle (G73, G76, G81–G89) commands.

G17, G18, or G19 determine plane of offset.

G90 specifies absolute dimensioning and G91 specifies incremental dimensioning.

Another work coordinate system can be selected by using commands G54 through G59.

Format

The format of the rapid traverse command is as follows:

G00 X_____Y_____Z_____A_____B_____

Example

If one axis of movement is specified in a G00 block, that axis moves at the rapid traverse feedrate. When two axes of movement are specified in a G00 block, the rapid traverse feedrate is assigned to the longest vector component. The resulting feed that appears on the screen may actually exceed the rapid traverse feedrate parameter.

If a block containing a G00 word also contains a Z word that causes the Z-axis to move away from the part, the Z-axis moves first. The other specified axes then move in linear or non-linear mode at the rapid feedrate to their specified end points. If Z is to move toward the part, all axes except Z move in linear or non-linear mode at rapid feedrate to their specified end points; then Z moves down to its end point. If no Z is programmed, all axes move at rapid feedrate coordinated to the specified end point. G00 is a member of the tool positioning code group and is *canceled* by G01, G02, G03, and G81–G89.

Caution

This code is used for positioning only and should never be used for cutting material.

The diagram below shows the two different rapid traverse modes:

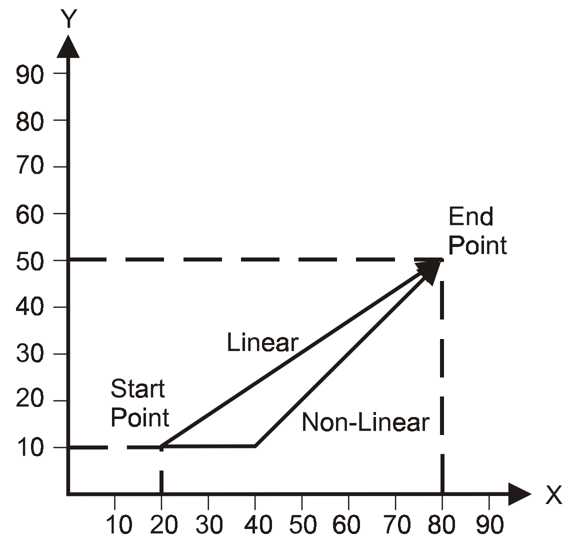


Figure 24. G00 Axis Movement

Linear Interpolation (G01)

The Linear Interpolation code (G01) moves the axes to a specified location at the programmed feedrate. Up to five axes (X, Y, Z, A, B) of coordinated motion can be specified while in this mode. The programmed feedrate can be changed by adding an F word to any NC block while in this mode. X, Y, Z, A, B, and F dimensions need to be supplied only if they change.

Notes

G01 is a member of the tool positioning code group and is *canceled* by G00, G02, G03 and the canned cycle (G73, G76, G81–G89) commands.

G90 specifies absolute dimensioning and G91 specifies incremental dimensioning.

G41 or G42 may be optionally selected if a *cutter offset* is desired.

Important

This code is used when the tool is in contact with the work piece to cut a line parallel to an axis or at an angle to an axis.

<u>Format</u>

The format of the linear interpolation command is as follows:

G01 X_____ Y_____ Z_____ A_____ B_____ F_____

F specifies the associated feedrate along the tool path. If rotary axis parameters (A and B) are used, the feedrate units are in degrees/minute.

Example

The diagram below illustrates the linear interpolation axis movement:

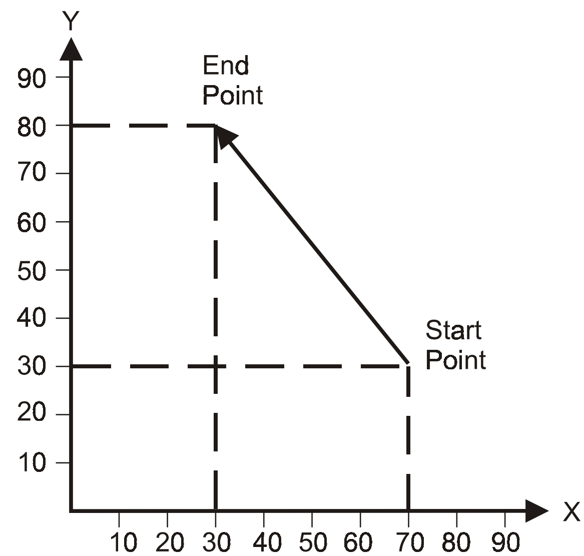


Figure 25. G01 Axis Movement

Circular and Helical Interpolation (G02 and G03)

These two codes are members of the tool positioning code group. The Clockwise Circular or Helical Interpolation code (G02) causes the axes to generate an arc or helix in a clockwise direction. The previous block's end point defines the start point of the arc.

The Counterclockwise Circular or Helical Interpolation code (G03) causes the axes to generate an arc or helix in a counterclockwise direction. The previous block's end point defines the start point of the arc.

Important

Calculate the linear feedrate to verify that it does not exceed various limit values.

Notes

Both G02 and G03 codes are *canceled* by G00, G01, the canned cycle (G73, G76, G81–G89) commands, or by each other.

The programmed feedrate can be changed by adding an F word to any NC block when this code is active.

G17, G18, or G19 specify plane of interpolation.

G41 or G42 may be optionally selected if a cutter offset is desired.

G40 is used to *cancel cutter offset*.

G02 or G03 cannot be used in a start up block in offset mode.

(X,Y) for G17, (X,Z) for G18, and (Y,Z) for G19 specify the end location on the selected plane.

Notes (continued)

R or the incremental coordinates ((I,J) or (I,K) or (J,K)) specify the arc center location. The R is modal and stays in effect until another R value is specified or (I,J) is used. With the R (radius) parameter, you specify the radius. You do not need to calculate the center point.

- A positive R produces an arc of less than or equal to 180°.
- A negative R produces an arc of greater than or equal to 180°.
- The R takes precedence over an I, J, or K in the same block.

For BNC, I, J, K, and R are modal for G02 and G03. The I, J, and K center point location is incremental from start point in G91 mode and absolute coordinates in G90 mode.

For ISNC, when G02 or G03 are specified, the I, J, and K are reset to 0.0. They remain modal until another G02 or G03 is encountered. R is not reset to 0.0. For ISNC, the I, J, and K are incremental from the start point in both G90 and G91 mode.

You can specify an R value for arcs when the arc is in the G17 XY plane or the G19 YZ plane.

Arcs use the right-hand coordinate system for all planes, except when using Basic NC for the G18 XZ plane. Arcs use a left-hand coordinate system when using the Basic NC for the G18 XZ plane.

F specifies the feedrate in degrees/minute along the arc in the circular plane.

Format

The formats of the Circular Interpolation commands are as follows:

Circular interpolation (Z = 0)
Helical interpolation (Z ≠ 0)

G02/G03 (for G17) X_____ Y_____ {R_____ or [I_____ and J_____]}
Z_____ F_____

G02/G03 (for G18) X_____ Z_____ {R_____ or [I_____ and K_____]}
Z_____ F_____

G02/G03 (for G19) Y_____ Z_____ {R_____ or [J_____ and K_____]}
Z_____ F_____

This diagram illustrates circular and helical interpolation:

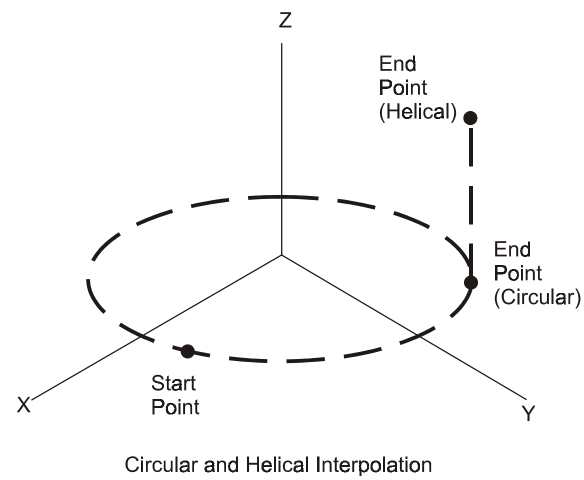


Figure 26. Circular and Helical Interpolation

G02 Example

Consider the following section of an NC program using a G02 code in absolute mode using R to specify the modal turn radius:

NC Part Program 1 Inch
G02.FNC

```
%
G00 G90
M25
T1 M06
Z5.05
X2.0 Y0.0
S2000 M03
Z0.05
G01 Z-0.5 F10.
G01 X2.0 Y0.0
G01 X0.5
G02 X0.0 Y0.5 R0.5    ⇐ R needs to be specified only once.
G01 Y2.5
G02 X0.5 Y3.0
G01 X3.5
G02 X4.0 Y2.5
G01 Y0.5
G02 X3.5 Y0.0
G01 X2.0
M25
M05
M02
```

Press the console Draw key and this screen appears:

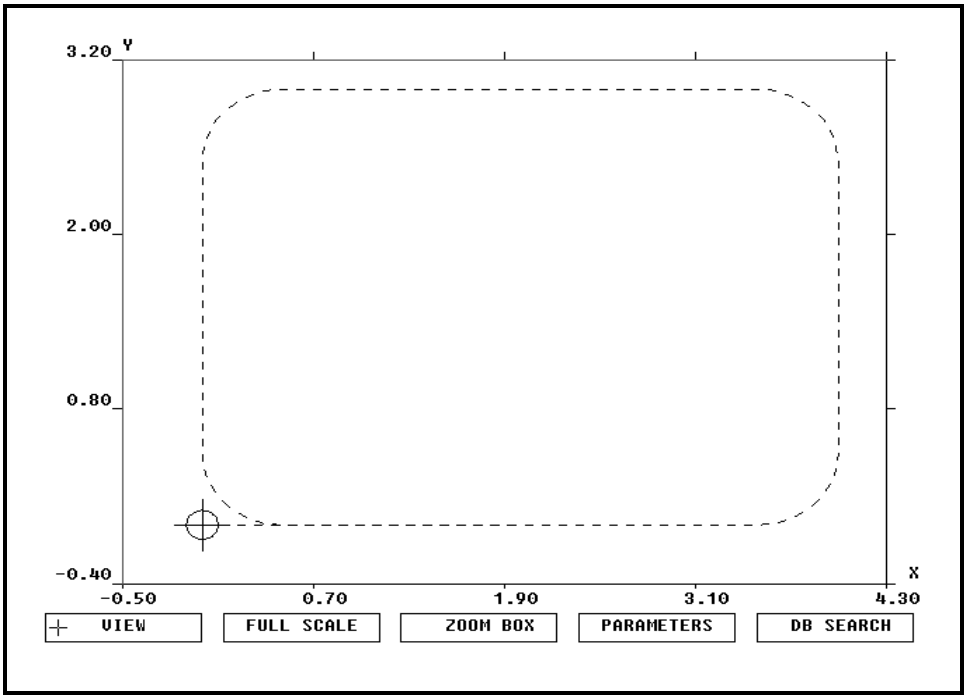


Figure 27. Display of Clockwise Circular or Helical Interpolation Code (G02)

BNC G03 Example

Consider the following section of a BNC program using a G03 code in absolute mode:

NC Part Program 1 Inch
G03ABS.HNC

```
%
N110 Z0 G91
N115 T01 M06
N116 X0. Y0. Z0.
N120 F40 S1000 M3
N130 G00 X3. Y4.
N140 G01 X3. Y2. F10
N150 G03 X4. Y1.5858 I4. J3.
N160 X7.4142 Y5. J5.
N170 G01 Y7.
N220 M02
E
```

This is an example of the same geometry using the incremental mode:

NC Part Program 1 Inch
G03INC.HNC

```
%
N110 Z0
N115 T01 M06
N116 X0. Y0. Z0.
N120 F40 S1000 M3
N130 G00 X3. Y4.
N140 G01 Y-2. F10
N150 G03 X1. Y-.4142 I1. J1.
N160 X3.4142  Y3. I0. J3.4142
N170 G01 Y2.
N220 M02
E
```

A and B words are not allowed in circular interpolation mode. The programmed feedrate can be changed by adding an F word to any block while in this mode.

The following is that portion of the program as it appears graphically:

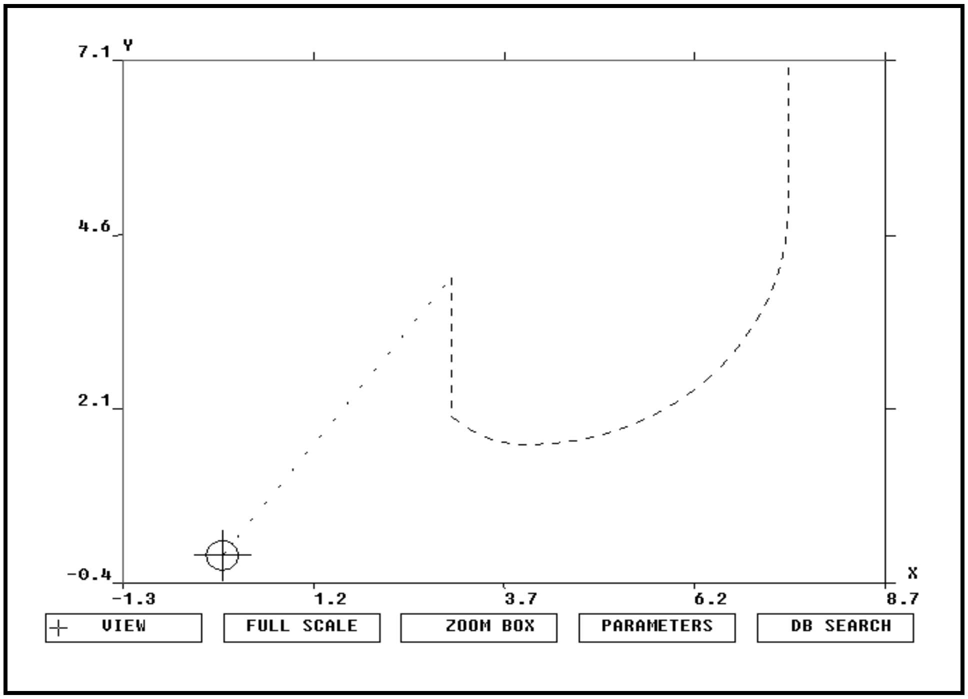


Figure 28. Display of BNC G03 Sample

X, Y, and Z define the end point of the arc and I, J, and K define the center point of the arc. I represents the X center point; J represents the Y center point; and K represents the Z center point. The X, Y, Z, and F words do not need to be programmed when you are initially setting the circular interpolation mode if they have not changed from the previous block.

For BNC, the I, J, and K dimensions must be specified when initially setting the circular interpolation mode (when a G02 or G03 is in the block) to establish a center point.

For ISNC, I, J, or K are set to 0.0 if they are not initially specified.

Once the circular interpolation mode is set, the X, Y, Z, I, J, K, and F dimensions need to be supplied only if they change. A block with missing dimensions uses the last specified locations.

Note

A circle or circular helix may be programmed by either using the same end and start point, or by not programming the end points. Ensure that the specified end point is mathematically on the arc.

If the programmed end point is not on the arc or helix, an end point is calculated using the start point, center point, and programmed end point. The start point and center point determine the radius of the arc and thus the distance of the calculated end point from the center point. The center point and programmed end point determine the line on which the calculated end point results.

Note

Arcs in this system are approximations that are comprised of small line segments or arc chords.

The chord error of arcs and helices may be controlled through the *chord error* parameter in the Program Parameters screen. The default chord error is 0.0001 inches (.003 mm). This creates very smooth arcs, but may limit the maximum feedrate for the arc or helix. Larger chord errors allow higher feedrate arcs or helices, but may be less accurate.

3D Circular Interpolation (G02.4 and G03.4)

The 3D Circular Interpolation (G02.4 and G03.4) codes are part of the tool positioning code group. These codes require two lines of NC code:

- The first line contains a set of X, Y, and Z values which represent the Intermediate Point.
- The second line contains a set of X, Y, and Z values which represent the End Point.

The Radius, Direction (CW or CCW), and Center Point are calculated based on the current location, the Intermediate Point, and the End Point. G02.4 and G03.4 can be used interchangeably to represent the same arc. The actual direction is calculated by the software.

Notes

Both G02.4 and G03.4 codes are canceled by G00, G01, the canned cycle commands (G73, G76, G81-G89), or by each other.

The programmed feedrate can be changed by adding an F word to any NC block when this code is active.

G17, G18, G19 are irrelevant for these G codes.

G41 and G42 may not be used with these G codes.

Example

Below is a program example using G03.4:

```
%
T1 M6 S500 M3
G0 X0 Y0 Z6
G1 X0 Y00.0 F5.
Z0
G3.4 X5.0 Y2.5 Z1.0   (Intermediate Point)
X10.0 Y0.0 Z0.0       (End Point)
G0Z6
M2
E
```

Dwell Mode (G04)

The Dwell Mode code (G04) causes the machine to delay the shift to the next block in the program by the amount specified by parameter P or X for a specified amount of time. When an integer is used with the G04 command, the value is multiplied by 0.01 for BNC and .001 or .0001 for ISNC depending on the Least Dwell Units programmed on the NC Parameters screen.

The BNC format for the dwell time is as follows:

Real Number: .3 second = G04 X.3 or G04 P.3

Integer Number: .3 second = G04 X30 or G04 P30

This is the ISNC format for the dwell time programmed with a real number:

Real Number: .3 second = G04 X.3 or G04 P.3

When .001 is programmed for the Least Dwell Units field on the NC Parameters screen, the ISNC format for the dwell time programmed with an integer is this:

Integer Number: .3 second = G04 X300 or G04 P300

When .0001 is programmed for the Least Dwell Units field on the NC Parameters screen, the ISNC format for the dwell time programmed with an integer is this:

Integer Number: .3 second = G04 X3000 or G04 P3000

The Dwell Mode code is only active in the programmed block, but the dwell time is modal and it affects most of the canned cycles.

Format

G04 P_____ or X_____

The P or X parameters specify seconds. The range of values is 0.001–9999.999 seconds.

Surface Finish (G05.1)

The code determines the type of finish quality, Pn. When n = 1, the Surface Finish Quality is Finish. When n = 2 the Surface Finish Quality is Semi-Finish. When n = 3 the Surface Finish Quality is Rough. The parameter Qm sets the chord segment for the finish, where m is the acceptable error value.

Data Smoothing (G05.2)

This code has two components.

Pn enables or disables the process. When n = 0, NC Block Smoothing is Off. When n = 1, NC Block Smoothing is On.

Qm sets the Tool Path Tolerance. The deviation from the tool path the system will tolerate, m, must be between 0 to 0.0005 inches, inclusive (0 to 0.012 mm).

Precision Cornering (G09)

The Precision Cornering code (G09) decelerates the axes to zero velocity at the end of the block in which it is programmed. After stopping, the axes accelerate to the programmed feedrate in the next block. This causes a sharp corner to be cut regardless of the programmed feedrate. The G09 code is not part of a G code group, so it affects only the axis movement of the block in which it is specified.

Setting Work Coordinate Systems with G10

This code is used for setting tool offsets, entering tool wear data, and changing work coordinate systems. The G11 command cancels the data setting mode. It is only required with the G10 L3 command. (Refer to the "Setting Tool Offsets with G10" section for information about using G10 to set tool offsets.)

Setting External Work Zero Offsets (G10 with L2)

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 parameter = 0), or one of the work coordinate systems (P parameter = 1 to 6), and X, Y, Z, A, B is the work zero point offset value of each axis. G90 specifies absolute dimensioning and G91 specifies incremental dimensioning.

Format

The command format for setting external work zero offsets is as follows:

G10 L2 P____ X____Y____Z____ A ____ B ____

Setting Tool Offsets with G10

This code is used for setting tool offsets, entering tool wear data, and changing work coordinate systems. The G11 command cancels the data setting mode. It is only required with the G10 L3 command. (Refer to the "Coordinate System Setting" section for information about changing work coordinate system.)

Initializing Tool Length Offsets (G10 with P, R)

G10 is used with the P and R parameters. P is the offset number 01 through 200, and R is the offset amount which may be absolute or incremental depending on G90 or G91.

<u>Format</u>

This is the command format for initializing tool length offsets:

G10 P____ R____

Initializing Tool Offsets (G10 with T, H, D)

G10 is used with the T, H and D parameters. The T parameter is the offset number 01 through 200, H is the tool length offset and D is the tool radius offset.

<u>Format</u>

This is the command format for initializing tool offsets:

G10 T___ H___ D____

Assigning Tool Offsets (G10 with L3)

The following example shows how to assign tool offsets. T is the tool number, H is the tool length offset number, and D is the cutter compensation (offset) number.

Format

The command format for assigning tool offsets is as follows:

G10 L3 T_____H_____D_____

Example

The following tool offset initialization example shows how to set up a program to assign offsets to the tools.

G10L3

T0001 H_____ D_____ (for tool 1)

•

•

•

T000n H_____ D_____ (for tool n)

G11 (to cancel)

Note

The tool number does not require leading zeroes.

Polar Coordinates Command (G16)

This command allows coordinates in the current block to be input in polar coordinates (radius and angle). The first coordinate in the currently selected plane is the radius coordinate in mm, and the second coordinate in the plane is the angle in degrees. The first coordinate in the currently-selected plane is the radius coordinate, and the second coordinate in the plane is the angle in degrees. For the XY plane, the X value represents radius and the Y value represents the angle.

G16 is canceled by G15 (Polar Coordinates Cancel).

<u>Format</u>

The command format for Polar Coordinates is as follows:

G16 X____ Y_____ Z_____

<u>Example</u>

Select Metric mode for the following sample program using the Polar Coordinates command:

NC Part Program 1 Metric
PIE.FNC

```
%
T1 M06
M03 G00 G90 X0 Y0 Z0 S1800
G01 Z-.25 F20.
G01 G16 X50. Y60.
G03 X50. Y120. R50.
G15
G01 X0 Y0
M02
```

Press the console Draw key to view the following screen:

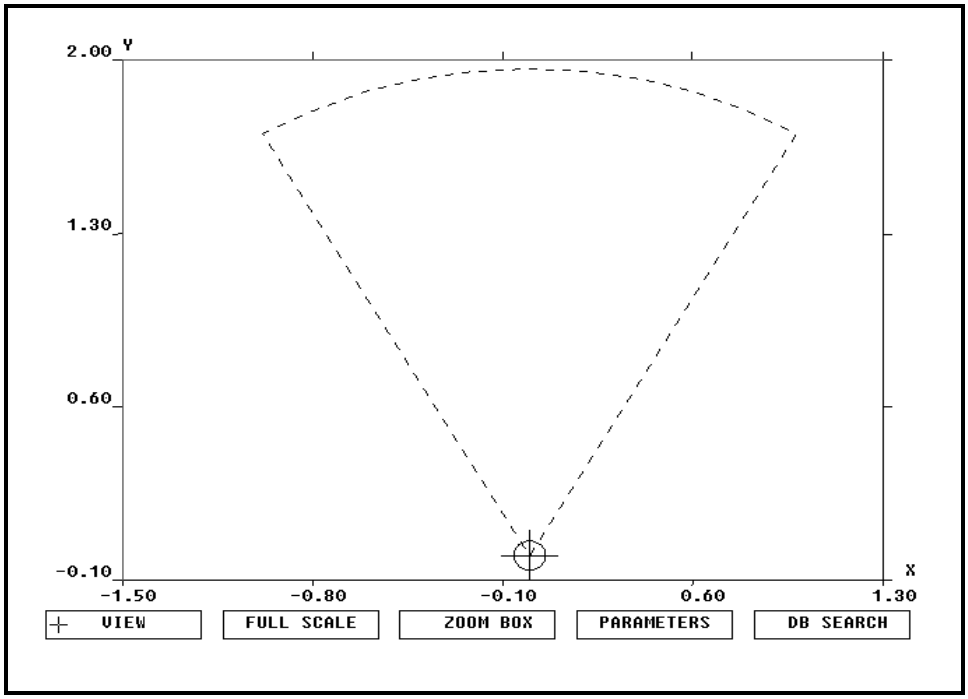


Figure 29. Display of Polar Coordinates Example

Plane Selection

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

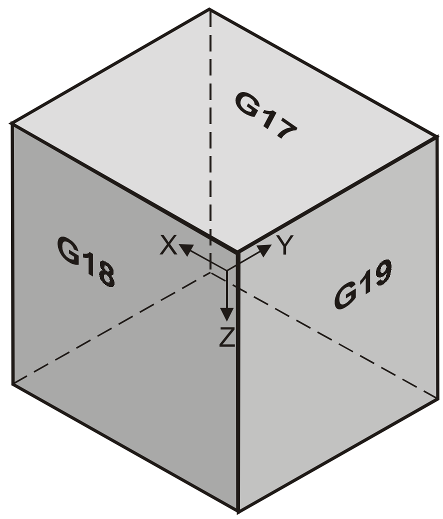


Figure 30. Plane Selection Group Codes

XY Plane Selection (G17)

The XY Plane Selection code (G17) is the default and sets the plane for circular interpolation modes G02 and G03 to the XY plane. 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.

G17 is canceled by G18 and G19.

Format

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

G17 X____ Y____

Example

The diagram below illustrates XY plane selection:

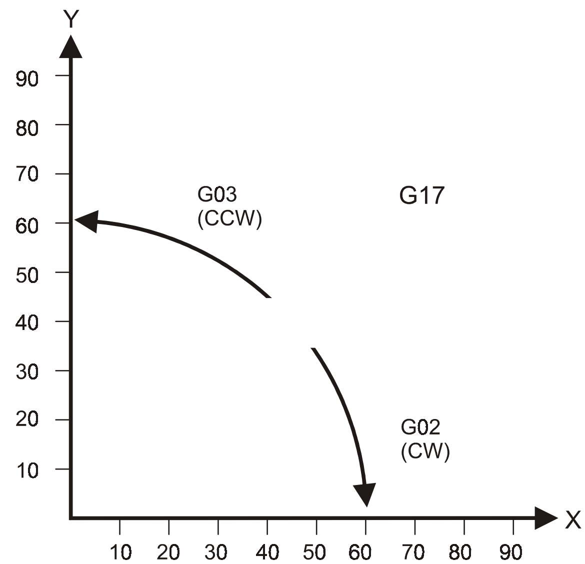


Figure 31. XY Plane Selection (G17)

XZ Plane Selection (G18)

The XZ Plane Selection code (G18) sets the plane for the circular interpolation codes G02 and G03 to the XZ plane. 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.

G18 is canceled by G17 and G19.

Format

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

G18 Z____ X____

Example

The diagrams below illustrate XZ plane selection in Basic NC and in ISNC:

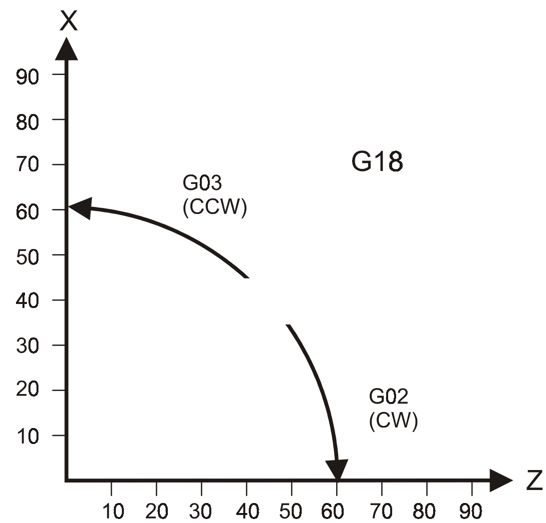


Figure 32. Basic NC XZ Plane Selection (G18)

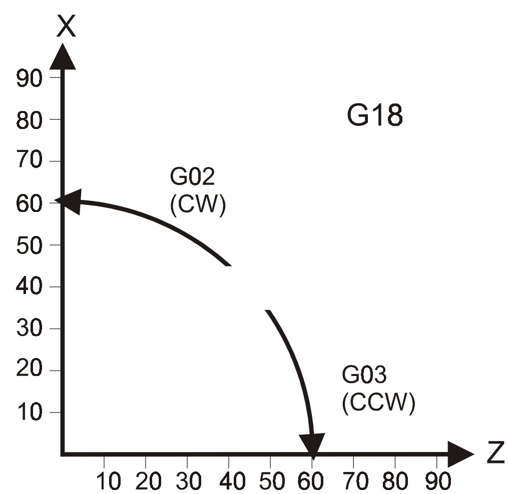


Figure 33. ISNC NC XZ Plane Selection (G18)

YZ Plane Selection (G19)

The YZ Plane Selection code (G19) sets the plane for circular interpolation codes G02 and G03 to the YZ plane. 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.

G19 is canceled by G17 and G18.

<u>Format</u>

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

G18 Y____ Z____

<u>Example</u>

The diagram below illustrates YZ plane selection:

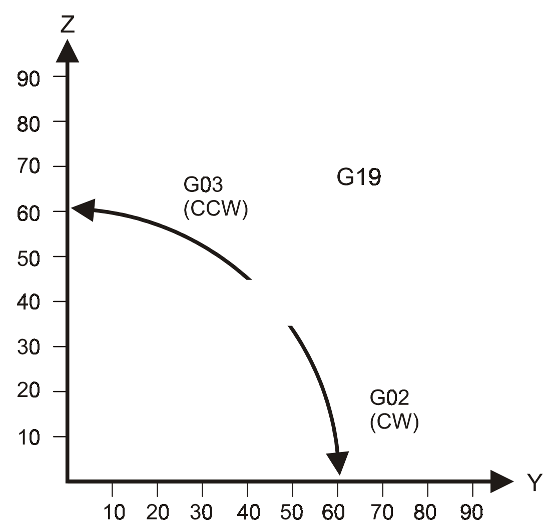


Figure 34. YZ Plane Selection (G19)

Units of Measure ISNC G20, G21

Before setting the coordinate system at the beginning of the program, the units of measure must be specified in an independent block. A part program may switch between English and Metric modes as long as the format of the dimensions is correct for the chosen mode.

The Imperial Units of Measure code ISNC G20) signals the system that the dimensions are in inches.

ISNC G20 is canceled by G21.

The Metric Units of Measure code ISNC G21) signals the system that the dimensions are metric units.

ISNC G21 is canceled by G20.

Format

These are the command formats for the inch/metric conversion commands:

ISNC:

G20: Inch command

G21: Metric command

Important

The ISNC G20 and G21 codes do not affect the units of measure used in the graphics and machine status display screens. The displays are controlled by the units selected when entering NC editing.

Automatic Return To and From Reference Point (G28 and G29)

Any point within the machine coordinate system can be selected as the reference point. The return to reference point is often used to move the part forward so you can remove chips from the part and inspect the part. Select the reference point on the NC Parameters screen.

The *Automatic Return To Reference Point* command (G28) 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.

The *Automatic Return From Reference Point* command (G29) specifies an automatic return from the reference point through the intermediate point, if specified by a previous G28, and to the end point designated by the X, Y, and Z parameters. If no intermediate point coordinates were specified during the current program execution, the machine will return directly from the reference point to the specified end point.

<u>G28 Format</u>

The format for the automatic return to reference point command is:

G28 X____ or Y____ or Z____

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 axes which follow 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.

G29 Format

The format for the Automatic Return to Reference Point command is:

G29 X____ Y____ Z____

When the G29 command is given, the system returns to the most recently used working coordinate system. These parameters specify the absolute or incremental location of the end point in coordinates relative to the current coordinate system in effect when the G28 command was processed.

Example

This sample program uses G28 and G29 to return the spindle to and from the reference point. Set Part Zero to X12 Y9 before running the program.

```
NC Part Program                 1                        Inch
PLAIN_28.FNC
%
G10 L2 P1 X12 Y9 Z-5
G00 X0 Y0 Z0
G28 X-7 Y-8
G29 X3 Y-4
M02
```

Press the console Draw key. The screen below appears:

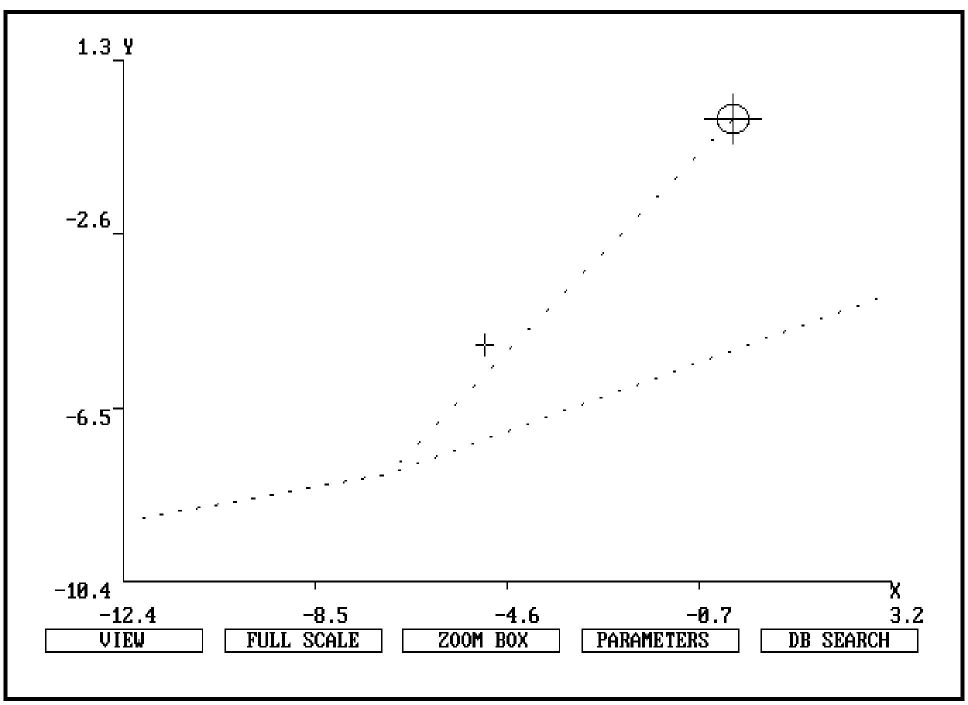


Figure 35.   Display of Automatic Return To and From Reference Point Example

Skip (Probing) Function (G31)

The Skip Function command is used to perform probing within an NC program, allowing you to specify a target position. The machine axis will stop when the target position is reached, or if the probe comes in contact with another surface. The Skip Function command can be programmed on a PC, but it can only be run properly on the control. On a PC, the command works similarly to the Linear Interpolation (Cutting Feed) (G01) command. The Skip Function command is a one-shot command and is only effective in the current block.

Two-touch and single-touch probing is supported. These modes are selected with the M41 and M42 codes. Two-touch probing is the default probing mode.

When performing two-touch probing (M42 specified), the probe moves in the specified direction until it touches the part, backs up away from the part, and then moves forward again at a feedrate of F/10. When the probe touches the part again, the trigger point is stored in variable #5061 (X axis), #5062 (Y axis), or #5063 (Z axis) with the NCPP option. (Refer to the *NC Productivity Package Option Manual* for more information about the variables and subprograms.) The NCPP option allows you to create macro subroutines and use conditional statements and math functions.

When performing one-touch probing (M41 specified), the probe does not back up after the first touch.

Values may be written to tool offset variables so they can be viewed after running the program on the Tool Offset screen. If the system does not have the NCPP option, the values need to be recorded manually. A Programmed Stop (M00) command can follow the G31 block to stop the machine so you can record the machine's location.

The *values* which are stored are referenced to the current coordinate system (working, local, or machine). If the probe does not touch the part before the end of the movement, the end coordinate value is stored in #5061, #5062, or #5063.

The current positions of the XY and Z axes can be retrieved using the #5041, #5042, and #5043 registers. The values can then be stored into part setup using the G10 code. For example, to set the X value for work offset G54, use the following G10 command:

```
G10 L2 P1 X[#5041]
```

Format

The format for the Skip (Probing) Function is as follows:

G31 X____ Y____ Z____ and/or F____

Note

This command cannot be performed with *cutter compensation* (G41, G42, G43 {with G18 or G19}, G45, G46, G47, and G48).

Example

The following program example finds the center point within a box when run on the control.

NC Part Program 1 Inch
G31_TEXT.FNC

```
%
(GO TO INITIAL PART ZERO)
G01 X0 Y0 F15.
G31 X7 F15.
#2001 = #5061
G01 X0 Y0 F25.
G31 X-7 F15.
#2002 = #5061
#2003 = [#2002+#2001]/Z
G01X#2005Y#2006F25.
N100 M00
G31 Y5 F15.
#2004 = #5062
G01 X#2003 Y0F25.
G31 Y-5 F15.
#2005 = #5062
#2006 = [#2004+#2005]/Z
(THE SPINDLE NOW MOVES TO THE CENTER OF THE BOX)
G01 X#2003 Y#2006
```

Parallel sides are assumed to be aligned with the X and Y axes. Additional programming steps will be required to determine the angle between the sides and the X and Y axes (skew angle) if the sides are not aligned. The initial part zero is set somewhere within the box.

The probe moves in the +X and then the -X direction to determine the center point between the sides in the X axis.

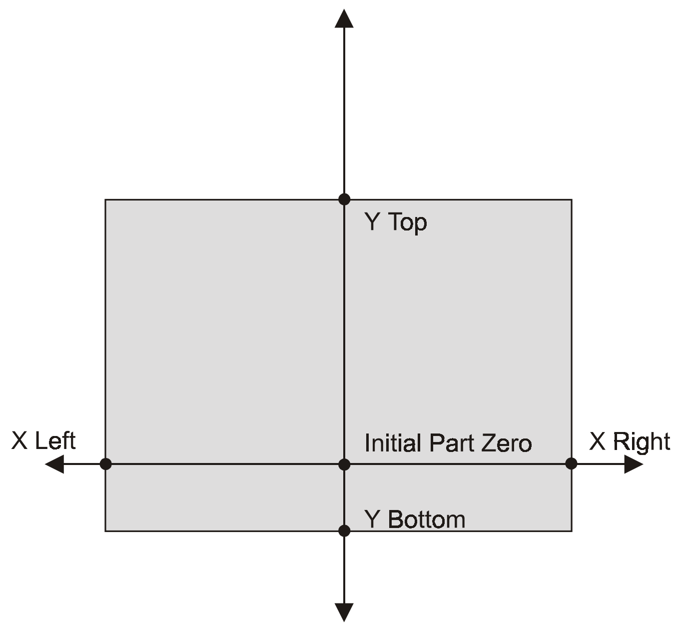


Figure 36.   ISNC Skip (Probing) Function

Tool Offsets (G40–G49)

Tool Offsets include Cutter Compensation and the Tool Length and Tool Radius Offsets. Cutter Compensation G codes G40–G42 are used to control tool movement. The Tool Length Offset Table contains the tool length offset (G43, G44), and is accessed from the Tool Setup screen. The Tool Radius Offset Table holds signed values for cutter compensation (G40–G42) and diameter compensation (G43–G48), and is accessd from the Tool Length Offsets screen.

The measurement units for the offsets in the Tool Offset Table depend on the programmed units. If -9.5 is entered for tool offset 15, that tool offset is -9.5 inches (or -9.5 mm, depending on the unit of measurement).

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 as shown in the illustration below.

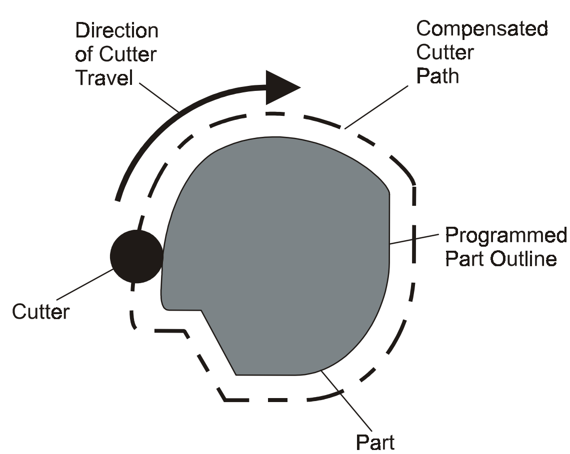


Figure 37. Cutter Compensation

Cutter Compensation – ISNC and Basic NC Programming Differences

You may program cutter compensation using ISNC or Basic NC. There are differences between the two. For example, programming cutter compensation using ISNC requires that you use D codes. This is not the case with Basic NC. Other differences include how ISNC and Basic NC interpret D code values.

<u>Tool Radius Offset</u>

ISNC

To program cutter compensation using ISNC, you must use a D code. The D code specifies an index into the Tool Offset Table or an actual offset value. For example, in the command G41 D5.0, the index value is an actual offset value of 5.0.

> **Note**
>
> You may access the Tool Radius Offset Table from the Tool Length Offset screen. The Tool Radius Offset page contains 200 registers for storing radius offsets.

Basic NC

To program cutter compensation using Basic NC, you may choose whether or not to use a D code. If you do not use a D code, Basic NC will use the value in the Diameter Comp field from the Tool Setup screen.

If you use a D code, Basic NC interprets the D code value based on whether you are calling out a tool change or commanding a G41 or G42.

If you use a D code when calling out a tool change, enter the actual tool diameter. Basic NC divides this value by two to calculate the tool diameter offset.

If you are programming a G41 or G42 code, Basic NC interprets the D code based on whether the D code value contains a decimal point:

- Contains a decimal point – Basic NC interprets the D value as the tool diameter offset
- Does not contain a decimal point – Basic reads the D value as an index value for the Tool Offset table

Tool Length Offset

You may use a G43 code to program tool length offsets. This is true for ISNC and Basic NC. Use an H code to specify an index into the Tool Offset Table. For example, in the command G43 H01, the index value is "01".

The value in the Tool Offset Table is a negative value that represents the distance from the Z home position to the top of the part with the tool tip touching the top of the part.

If you are using G43 codes in your part program, do not store a value in the Zero Calibration field in the Tool Setup screen. Any value in that field for the tool will be added to the value in the Tool Offsets Table. Otherwise, your distance will be too large and the length value will be lower than the actual part zero.

Hints and Tricks

Another way to program a tool length offset would be to use the Zero Calibration field in the Tool Setup screen, and not use the G43 H code. This is recommended, especially if you are using the Tool Probing software.

Cutter Compensation Off (G40)

The Cutter Compensation Off code (G40) is the default. It cancels cutter compensation by erasing all the data in the system's cutter compensation look-ahead buffers and moving to the current uncompensated endpoint at the programmed feedrate.

G00 or G01 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.

Format

The command format for Cutter Compensation Off is as follows:

G40 X____ Y____

If the parameters are omitted, the tool moves the old vector amount in the opposite direction which effectively cancels the offset.

> **Note**
>
> It is possible to switch from left to right cutter compensation (and vice versa) without first canceling cutter compensation.

Cutter Compensation Left (G41)

The Cutter Compensation Left code (G41) switches on cutter compensation. It establishes a new tool path left and parallel to the programmed path. The distance between the new tool path and the programmed path is equal to the cutter compensation value for the programmed tool.

G41 is canceled by G40.

The offset executes only in the G17 offset plane. In simultaneous three-axis control, the tool path projected on the offset plane is compensated.

G00, G01, G02, or G03 must be specified.

<u>Format</u>

The command format for cutter compensation left is as follows:

G41 X _____ Y_____ D_____

If the offset number for cutter compensation is D00, the system will not go into G41 mode.

Cutter Compensation Right (G42)

The Cutter Compensation Right code (G42) switches on cutter compensation and establishes a new tool path right of and parallel to the programmed path. The distance between the new tool path and the programmed path is equal to the cutter compensation value for the programmed tool.

G42 is canceled by G40.

This command is an offset method similar to G41 except that the offset is to the right of the programmed path looking in the direction in which the tool is advancing. The offset is performed only in the G17 offset plane. Only the coordinate values of an axis in the offset plane are affected by the offset. In simultaneous three-axis control, the tool path projected on the offset plane is compensated.

G00, G01, G02, or G03 must be specified.

<u>Format</u>

The command format for cutter compensation right is as follows:

G42 X _____ Y_____ D_____

If the offset number is D00, the system will not go into G42 mode.

Cutter Compensation Programming

Follow these steps to use cutter compensation:

1. Enter the part surface description according to the final dimensions of the part.
2. Enter the full cutter diameter as a positive number in the Diameter Compensation field in Tool Setup, or supply a D word when changing tools (Basic NC only).
3. Activate cutter compensation in the desired direction (left or right of part surface with respect to tool path direction).
4. Supply an entry move from somewhere outside the part to the start point of the part surface, i.e., somewhere outside of the compensated path. The part surface appears as a blue line on the graphics display.
5. Following all of the blocks to be compensated, provide an *exit move* to somewhere outside the compensated path and turn off cutter compensation (G40). When a G40 is programmed, cutter compensation extracts any remaining information from its look-ahead buffer and moves to the last programmed end point. The tool moves from the compensated end point of the previous move to the end point of the exit move.
6. Be certain that the exit move is outside the compensated path. Otherwise, turning off cutter compensation may cut into the part surface. To check the exit move, use graphics to verify the tool path movements.

The tool moves from the start point of the entry move and ends at the compensated start point for the part surface as shown in the graph below:

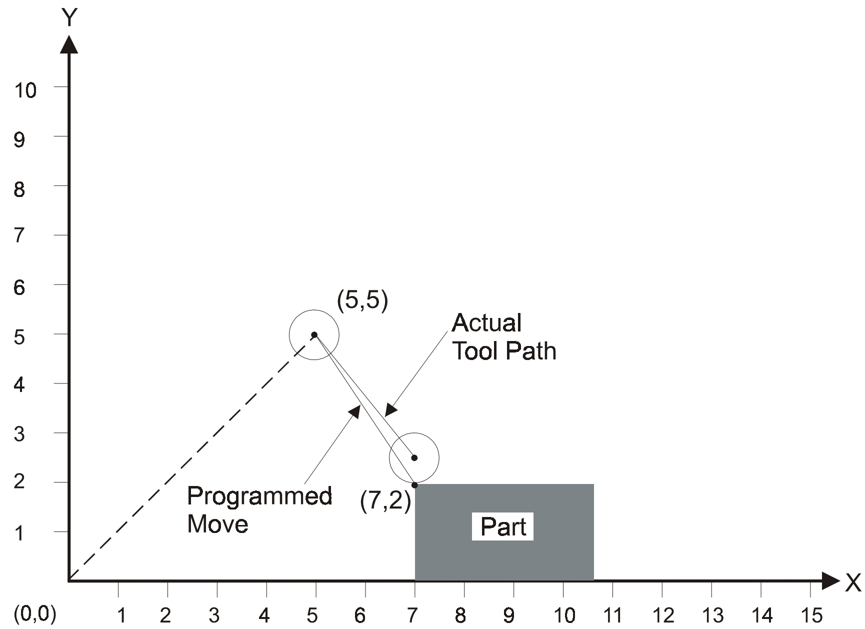


Figure 38. Cutter Compensated Tool Movement

In the previous illustration, the value in the Diameter Compensation field is 1.00, and these codes were used to control tool movement:

- G00 G40 X5. Y5.
- G41 X7. Y2.

Hints and Tricks

Z movements may be used for the entry and exit moves. For example, turn on cutter compensation when moving to a Z Start plane before plunging. Turn off cutter compensation after retracting the tool from the part.

Caution

Turn off cutter compensation using a G40 code before ending a program or all programmed blocks may not be cut.

Tool Length Offset (G43, G44, G49)

Tool offsets, G43 and G44 tool length compensation codes, are used to compensate tool length without altering the NC program. G43 is for positive tool length compensation. G44 is for negative tool length compensation. Either the G49 command, or an H00 command, immediately cancels the offset.

The tool offset specified by a G43 or G44 overrides the tool length offset from the Zero Calibration field on the Tool Setup screen. The Zero Calibration field on the Tool Setup screen is always treated as a negative Z offset. For example, if a value of 3.0 is put in the Zero Calibration field, a Z offset value of -3.0 is stored. If the command G43 H1 is then used where the value -2.2 is stored in the H1 offset register, Ultimax uses a tool offset of –2.2. The table below illustrates tool offsets:

Tool Setup Screen Zero Calibration	Tool Length Offset Mode	H1	Total Tool Offset
3.0	G43	-2.2	-2.2
3.0	G44	-2.2	+2.2
3.0	G49	-2.2	-3.0
3.0	G49	+2.2	-3.0
3.0	G43	+2.2	+2.2
3.0	G44	+2.2	-2.2

Table 4. Tool Offsets

Note

The values in the Tool Setup screens always remain in the units selected when going into the NC Editor.

For Basic NC (BNC)

An H address may specify an index into the tool length offset table without specifying a G43 or G44. In such a case, the value in the tool length offset table is used as the tool offset. This is equivalent to the *Zero Calibration* field on the *Tool Setup screen*.

For BNC and ISNC, if the system is in the G43 and G44 mode already, an H code can be used by itself to replace the existing tool length already in effect.

Notes

If no G43 or G44 is programmed and an H offset is not specified, tool lengths are taken from the Tool Setup screen.

An offset in the X axis can be specified with the G19.

An offset in the Y axis can be specified with the G18.

If a G17 is provided, or none of the plane selection commands (G17, G18, and G19) is present, specify an offset in the Z axis.

The G17, G18, and G19 used in this block are only used to specify the axis of the tool offset and will not affect the specified plane.

An offset in the X or Y axis cannot be specified when cutter compensation is active or commands G45–G48 are being used.

Commands G45–48 support existing X or Y axis tool offset programs; however, to save time, automatic Cutter Compensation (G40–G42) should be used instead.

Commanding an H00 cancels an offset.

Either G43 or G44 is in effect until a G49 or H00 is used.

Format

The H is the Offset Code with a range of H00 to H200. G17 is optional when a Z axis offset is desired.

[G17 _____] or [G18 _____] or [G19 _____] H_____.

The following four examples illustrate tool length offset H codes with the G43 and G44 codes. Tool 1 had a value of 5.0 for the Zero Calibration field on the Tool Setup screen. The Tool Length Offset 1 value is –6.0 and the Tool Length Offset 2 value is –7.5..

Example 1

```
T01 M06
```

With these offsets, the calibrated tool length will be 5.0. That length is taken from the Tool Setup screen)

Example 2

```
T01 M06
H02
```

At this point, the calibrated tool length will still be 5.0 because no G43 or G44 has been entered.

Example 3

```
T01 L4.0 M06
G43 H01
```

At this point, the calibrated tool length is 6.0 because the L value was superceded by the offset value of 6.0.

Example 4

`T01 M06` - sets the tool length to be 5.0
`H02` - tool length remains 5.0
`G43` - tool length remains 5.0
`H02` - sets the tool length to be 7.5

Tool Radius Offset (G45–G48)

Caution

ISNC and Basic NC – the tool used for cutter compensation must be smaller than or equal to the arc that you have programmed. If the tool radius is greater than or equal to the arc radius, the compensated tool path will sweep in the opposite direction of the programmed arc.

The tool position offset commands increase or decrease the amount of axis movement. Offset values within the following ranges can be selected for the tool radius offset commands:

	mm input	inch input
Offset Value	0~±999.999 mm	0~±99.999 in.
	0~± 999.999°	0~±999.999°

Note

G45–48 support existing X or Y axis tool offset programs. However, to save time, use the automatic Cutter Compensation (G40-G42) instead.

Tool Radius Offset Increase (G45)

This command increases the specified block’s tool radius offset amount by the value stored in the offset value memory.

Tool Radius Offset Decrease (G46)

This command decreases the specified block’s movement amount by the value stored in the offset value memory.

Tool Radius Offset Double Increase (G47)

This command increases the specified block’s movement amount by twice the value stored in the offset value memory.

Tool Radius Offset Double Decrease (G48)

This command decreases the specified block’s movement amount by twice the value stored in the offset value memory.

Format

The command format for the tool position offsets is as follows:

GXX X_____Y_____Z_____A_____B____ D____

GXX is an optional Interpolation (Group 1) move command, and D is the offset command. The number which follows D is an index into the tool offsets table. The offset value is modal and needs to be specified only once. The offset is applied to all axes specified in the parameters.

Example

Set tool offset 1 to the desired offset before running the following program using the Tool Radius Offset commands (G45 through G48):

Industry-Standard NC Part Program 1 Inch
G45_G48.FNC

```
%
N10 G10 P1 R0.5
N20 G00
N30 G90
N40 M25
N50 T1 M06
N60 Z5.0 X0. Y0.
N70 S2000 M03
N80 Z0.05
N90 G00 Z-0.5 F10.
N200(INNER OUTLINE WITHOUT USING OFFSETS)
N210 G91 X4 Y4
N220 G01 X3
N230 Y1.5
N240 X4
N250 G45 Y-1.5
N260 X3
N130 G03 X1 Y1 I0. J1
N140 G01 Y4
N150 X0
N160 G02 X-2 Y2 I0. J2
N170 G01 Y0
N180 X-3
N190 Y-2.5
N200 X-3
N210 Y2.5
N220 X-3
```

NC Part Program **2** **Inch**
G45_G48.FNC

```
N230 G03 X-1 Y-1 I0. J-1
N240 G01 Y-2
N250 X1
N260 Y-4
N265 Z5.05
N270 G00 X-4 Y-4
N275 G90 X0 Y0
N280(OUTER OUTLINE USING G45, G46, G47, AND G48)
N290 G91 G46 X4 Y4 D1
N300 G47 G01 X3 F20.
N305 Y1.5
N307 G48 X4
N308 Y-1.5
N309 G45 X3
N310 G45 G03 X1 Y1 I0. J1
N320 G45 G01 Y4
N330 G46 X0
N340 G46 G02 X-2 Y2 I0. J2
N350 G45 G01 Y0.
N360 G47 X-3
N370 Y-2.5
N380 G48 X-3
N390 Y2.5
N400 G45 X-3
N410 G45 G03 X-1 Y-1 I0. J-1
N420 G45 G01 Y-2
N430 X1
N440 Y-4
N450 G00 G46 X-4 Y-4
N460 G00 Z5.05
N470 M25
N480 M05
N490 M02
```

Scaling (G50 and G51)

The G51 code is used to scale subsequent move commands by a programmable scale factor and must be in an independent block.

Scaling is not applicable to the following movement in case of canned Z axis movements. If scaling results are rounded and units less than 5 are ignored, the move amount may become zero and may affect cutter movement. Whether the scaling function is effective or not, it can be set by a parameter for each axis. The scaling function always becomes effective for the circular radius command R in the G51 mode, regardless of these parameters.

One or more axes' scaling can be disabled on the NC Parameters screen. The methods for specifying the scaling center point and the scaling factor are different with BNC and ISNC.

For BNC, X, Y, and Z are the scaling center points in absolute coordinates. The I, J, and K codes specify the scale factor for the X, Y, and Z axes. If only I is specified, all axes will be scaled by that factor. Scaling G51 codes may not be nested.

For ISNC, these two methods can be used to specify scaling parameters:

Method 1: **X, Y, or Z present.** X, Y, Z define the scaling center point. If I, J, or K are present, they define the scaling factors. If they are absent and P is present, the P value defines the scaling factor for all three axes. If P is an integer (no decimal point) the value is multiplied by the least scaling factor parameter on the NC Parameters screen; otherwise, the exact P value is used.

Method 2: **X, Y, or Z absent.** I, J, K define the scaling center point. P provides the scaling factor if provided for all three axes. If P is an integer (no decimal point) the value is multiplied by the least scaling factor parameter on the NC Parameters screen; otherwise, the exact P value is used.

Format

The format of the BNC scaling code is as follows:

G51 X___ Y___ Z___ I___ J___ K___

The format of the ISNC scaling code is as follows:

Method 1: G51 X___ Y___ Z___ (I____ J_____ K____ or P____)

Method 2: G51 I____ J_____ K____ P____

Notes

The smallest unit of scaling is either 0.001 or 0.00001 when an integer P value is provided. The Least Scaling Factor field on the Configuration Setup screen is used for setting the smallest unit of scaling.

If scaling factors are not specified, the default scaling factor 1.0 is used.

If the scaling center point is not specified, the G51 command point is used for the scaling center.

Scaling can be enabled/disabled for a particular axis on the Configuration Setup screen.

<u>Example</u>

Here is a BNC sample using the scaling codes:

```
Using G91 —    G00 X20. Y20.
               G51 X40. Y40. I.5
               G01 X40.
               Y40.
               X-40.
               Y-40.
               G50
```

The following diagram illustrates the previous code sample:

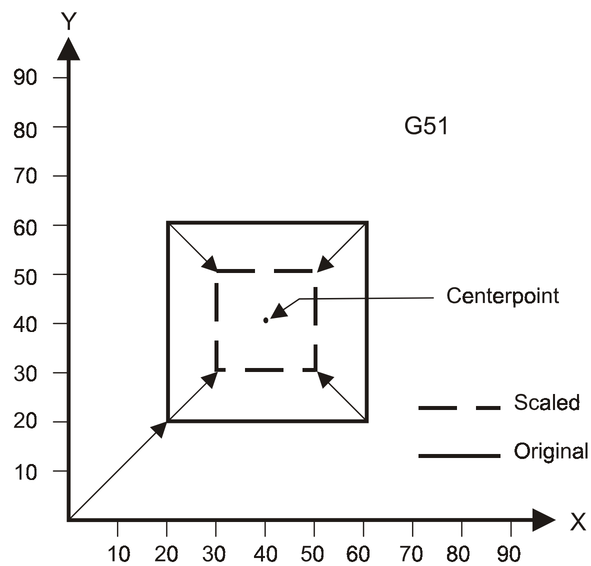


Figure 39.   G51 Scaling Code

Caution

The Scaling (G51) command must always be canceled with a Cancel Scaling (G50) command.

Mirror Image (G50.1 and G51.1)

Mirroring (G51.1) and Mirroring Cancel (G50.1) commands are used when the shape of the workpiece is symmetric to an axis. The whole part can be prepared by programming a subprogram and using programmable mirror imaging. Ordinary mirror image comes after the programmed mirror image. The first movement command must be absolute when in this mode. The following actions occur when the mirror image is applied to only one axis composing a plane:

- Circular Command: CW and CCW are reversed.
- Cutter Compensation: Right and Left Offset are reversed.
- Coordinate Rotation: Rotation angle becomes reversed.

Format

The formats of the mirroring codes are as follows:

G51.1 X___ or Y___ or Z___

G50.1 [X0] or [Y0] or [Z0]

Specifying a G50.1 with no X, Y, or Z parameter cancels the mirroring code in the X, Y, and Z axes. The coordinates about which the tool path will be mirrored are in absolute values. The mirroring codes create the following special conditions:

- For circular commands CW and CCW are reversed.
- Cutter compensation for Right and Left are reversed.
- Mirroring G51.1 codes may not be nested.

G51.1 is used to mirror a tool path about the X, Y, or Z axis while G50.1 is used to cancel mirroring for the X, Y, or Z axis.

Important

This mode is canceled by G50.1. The first movement command after a G50.1 command must be an absolute command. This mode must not be specified in the G68 or G50 mode.

Example

In the illustration below, part #1 (in the upper right corner) is mirrored three times into part #2, #3 and #4. Note that the direction of the tool path (shown as directional arrows numbered 1, 2, and 3) on each part changes with each mirroring operation:

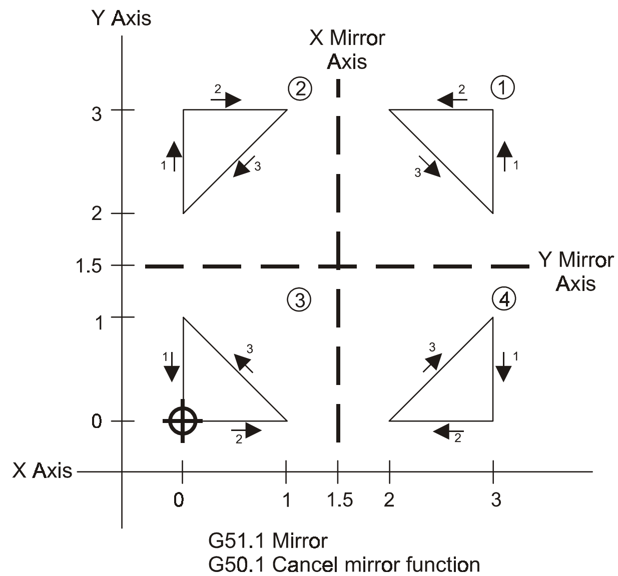


Figure 40. ***BNC G50.1 and G51.1 Mirroring Codes***

The following program example mirrors the part as shown in the example from the previous page:

```
NC Part Program                  1                inch
MIRROR.FNC
%
N10 (FIG 7-94 MIRRORING EXAMPLE)
N42 (  )
N44( MAIN PROGRAM )
N46 (  )
N50 M98 P8888
N60 (2-PART#1 MIRRORING IN X)
N70 G51.1 X1.5
N80 M98 P8888
N90 (3-MIRRORING CONTINUED IN Y)
N100 G51.1 Y1.5
N110 M98 P8888
N120 (CANCEL INITIAL X & Y MIRROR)
N130 G50.1
N140 (4-PART#1 MIRRORED IN Y)
N150 G51.1 Y1.5
N160 M98 P8888
N170 M02
N172 (END OF MAIN PROGRAM)
N180 (  )
N190 (SUB-PROGRAM 8888)
N200 (1-PART#1 UPPER RIGHT)
N210 (TRIANGLE 3,2 3,3 2,3)
N215 (  )
N220 O8888
N230 G00 G90 T01 M06
N240 X0 Y0 Z.05 M03 S800
N250 G00 X3 Y2 Z0
N260 G1 Y3 F50
N270 X2
N280 X3 Y2
N290 M99
E
```

Local Coordinate System Setting (G52)

While programming in a work coordinate system, it is sometimes more convenient to use a local coordinate system within the current work coordinate system. The zero point of each local coordinate system is equal to the X, Y, Z, position of the current work coordinate system.

To cancel the local coordinate system, the zero point of the local coordinate system should be matched with the zero point of the work coordinate system by using the G52 X0 Y0 Z0 command. The local coordinate system can also be canceled by switching to another work coordinate system (including the current work coordinate system) or to the machine coordinate system (G53).

The G52 command is most useful in conjunction with *subprogram commands*. A subprogram could be used to cut a part and the local coordinate system can be shifted to cut a number of similar parts.

<u>Format</u>

The format of the local coordinate system command is as follows:

G52 X_____Y_____Z_____

<u>Example</u>

This illustration shows setting a local coordinate system using the G52 command:

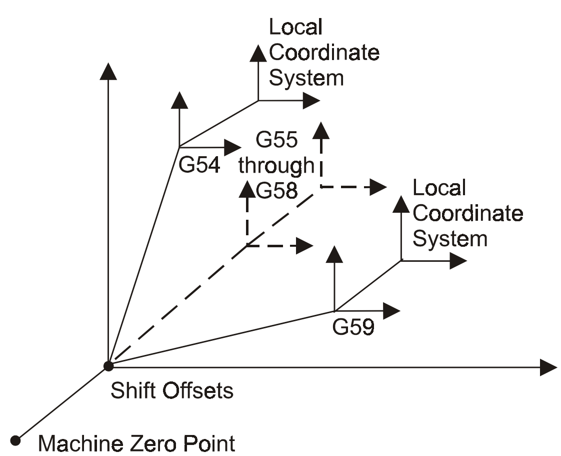


Figure 41. Setting Local Coordinate System Using G52

The following is a sample program, which uses G52 to set local coordinates:

NC Part Program 1 Inch
LOC_COOR.FNC

```
%
N10 G00 G90
N40 M25
N45 X0 Y0
N50 T1 M06
N60 Z5.
N90 S2000 M03
N100 Z0.05
N110 M98 P2121
(USE LOCAL COORD SYSTEM)
N240 G52 X-1.5 Y-1.5
N320 G65 P2121
N380 G52 X1.5 Y-1.5
N390 M98 P2121
N430 G52 X0 Y-3
N440 M98 P2121
N430 Z5.
N1170 M25
N1190 M05
N1200 M02
O2121
N500 X1
N510 Y1
N520 X0
N530 Y0
M99
```

Press the console Draw key and this screen appears:

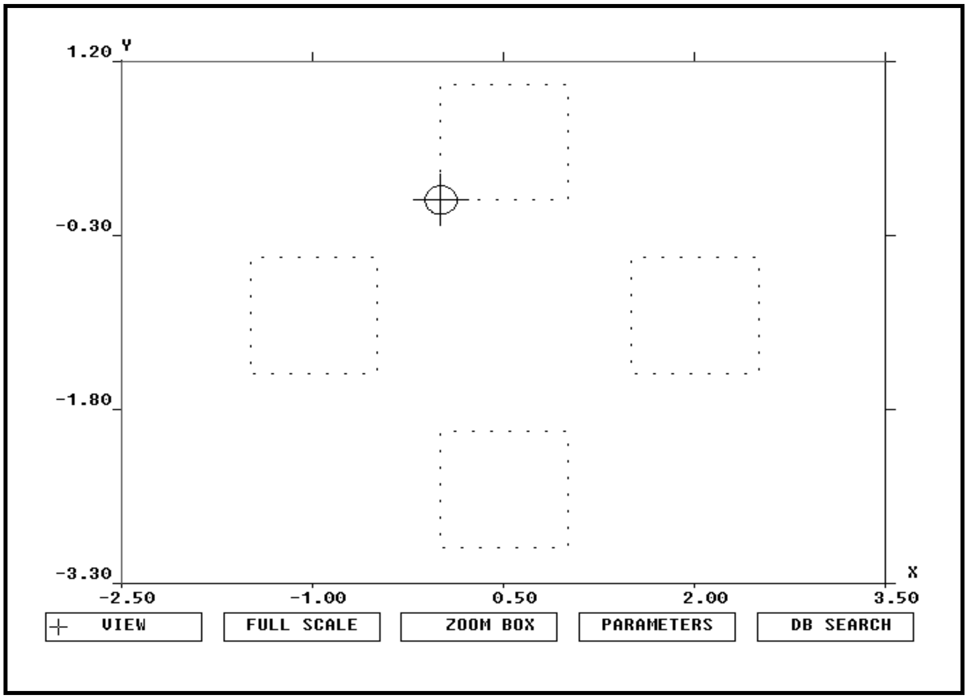


Figure 42. Display of Local Coordinates Example

Machine Coordinates (G53)

This Machine Coordinates (G53) command moves the tool to the X,Y,Z, A, B machine coordinate position at rapid traverse. This command is only effective in the block in which it is specified and in Absolute mode (G90). The system reverts to the last commanded work coordinate system.

If a local coordinate (G52) is used before a machine coordinate (G53) is commanded, the local coordinate is canceled when the system goes back to the last commanded coordinate system. Reinstate the local coordinate system with another G52.

Format

The format of the machine coordinates command is as follows:

G53 X____Y____Z____ A ____ B ____

Example

Before running this sample program, set the shift offsets to X0 Y0 Z0 and set part zero to X2.0 Y3.0 and Z1.0.

```
NC Part Program                 1                         Inch
MACHCOOR.FNC
%
G00 G90
M25
X0 Y0
T1 M06
Z5.
S2000 M03
Z0.05
G01 X1 F30.
Y1
X0
Y0
(USE MACHINE COORD SYSTEM)
G01 G53 X0
G53 X1 F30.
G53 Y1
G53 X0
G53 Y0
G53 G00 Z5.
M25 M05
M02
E
```

Press the console Draw key and the following screen appears:

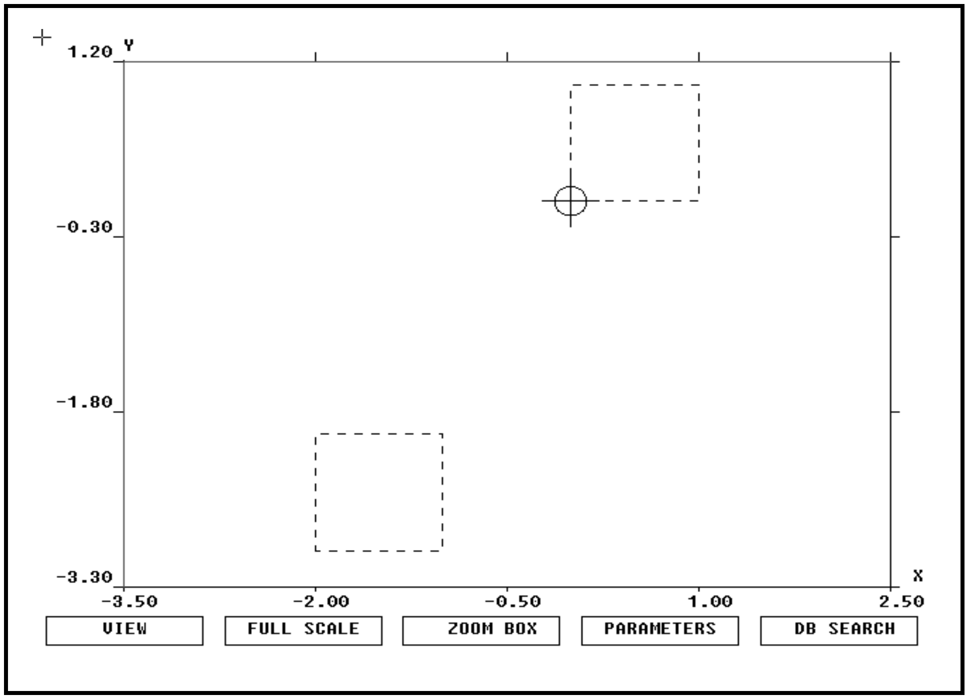
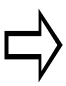


Figure 43. Display of Machine Coordinates Example



Important

When running a program on the control, do not use negative shift offsets with G28 or G53. An error message will occur since the negative machine positions cannot be implemented.

Multiple Work Coordinate Systems (G54–G59)

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 G92 (Set Part Zero) 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.

Use the G10 command to set tool offsets, enter tool wear data, and change work coordinate systems, and use the G92 command to set part zero. All six work coordinate systems can be moved an equal distance and direction by using the 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 the NC mode, the Part Setup screen has the Work Offsets (F1) softkey to display up to six work coordinates (G54–G59) and a set of shift offset values. As shown below, these codes are used to set multiple part zeroes for *multiple parts* fixtured to the table and milled consecutively using the same part program.

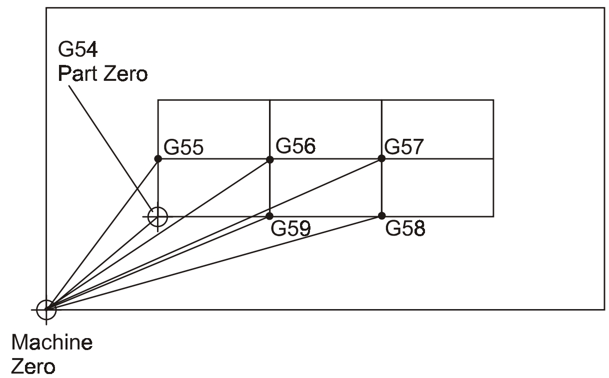


Figure 44. Work Offset G Codes for Multiple Parts

The coordinates defining G54 are the part zero coordinates for the original part defined on the Part Setup screen. Set the X, Y, and Z 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, Y, and offset Z. Editing G54 work offsets for multiple coordinate systems updates the part setup for X, Y, and Z on the Part Setup screen.

Precision Cornering On (G61) and Off (G64)

Precision cornering allows non-tangent blocks to be milled with precise corners, regardless of programmed feedrate.

Note

Precision cornering works differently on machines that have the UltiPro II option installed. Please use the tables below to determine how precision cornering will operate on your machine.

The NC Precision Cornering codes work in the following manner in standard Hurco machines. If you have the UltiPro II option installed, refer to the Precision Cornering with UltiPro II Option table.

Code	Action
G61	Causes the axes to decelerate to zero velocity at the end of a block, if the blocks are not tangent. Tangency is defined as an angle of 5° or less between two consecutive blocks. If the angle is greater than 5°, the system stops and then accelerates to the programmed feedrate in the next block.
G64 (default)	Causes the axes to decelerate to zero velocity at the end of a block, if the blocks are not tangent. Tangency is defined as an angle of 44° or less between two consecutive blocks. If the angle is greater than 44°, the system stops and then accelerates to the programmed feedrate in the next block. The first line is marked as a stop when complete.

Table 5. Standard Precision Cornering

The NC Precision Cornering codes work in the following manner in Hurco machines that have the UltiPro II option installed. If you do not have the UltiPro II option installed, please refer to the Standard Precision Cornering table

Code	Action
G61	Causes the axes to decelerate to zero velocity at the end of a block, if the blocks are not tangent. Tangency is defined as an angle of 44° or less between two consecutive blocks. If the angle is greater than 44°, the system stops and then accelerates to the programmed feedrate in the next block. The first line is marked as a stop when complete.
G64 (default)	Causes the axes to traverse all blocks at a constant feedrate and blends for constant surface finish with no regard to tangency. The first line is not marked as a stop when complete.

Table 6. Precision Cornering with UltiPro II Option

Special Program Support

Rotation (G68 and G69)

The Coordinate Rotation (G68) command turns on coordinate system rotation, and the Coordinate Rotation Cancel (G69) command turns off coordinate system rotation.

Format

The G68 code uses this format to command rotation:

G68 (XY___ or XZ ___ or YZ___) R___

When the G17 plane is used, X and Y addresses are used in the format to describe the center point. When G18 is used, X and Z describe the center point. If the plane is defined using G19, the Y and Z addresses define the center point.

R specifies the *angle of rotation*. A positive R value indicates a CCW direction, and a negative R value indicates a CW direction. When the coordinate values of rotation center are omitted, the current position is used as the center point.

The range of R depends on whether BNC or ISNC is selected and whether an integer or decimal value is specified. Here are the R ranges for each NC type:

BNC: R has a range of -360.0 to +360.0, whether an integer or real number is used.

ISNC: Units of R have a value of 0.001° when R is an integer.

R has a range of -360,000 ≤ R ≤ 360,000 when R is an integer value.

R has a range of -360.0 to +360.0 when R is a real number.

Rotation is canceled with a G69. Do not use G17, G18, or G19 while in the G68 mode. Use G69 to disable the G68 mode, change the plane, and then go back to the G68 mode.

⇨

Important

G68 codes may not be nested.

<u>Example</u>

This program uses the rotation codes:

```
ISNC Part Program                1                            Inch
G68.FNC
%
M25
(USING REAL NUMBER WITH G68)
G68 X0 Y0 R-75.0
T1 M06
Z5.05
G01 Z-0.5 F10.
G91 X1.0
Y2.0
X-1.0
Y-2.0
(CANCEL ROTATION)
G69
(USING INTEGER NUMBER WITH G68)
G68 X0 Y0 R45000
X1.0
Y2.0
X-1.0
Y-2.0
(CANCEL ROTATION)
G69
M25
M05
M02
```

Press the console Draw key and this screen appears:

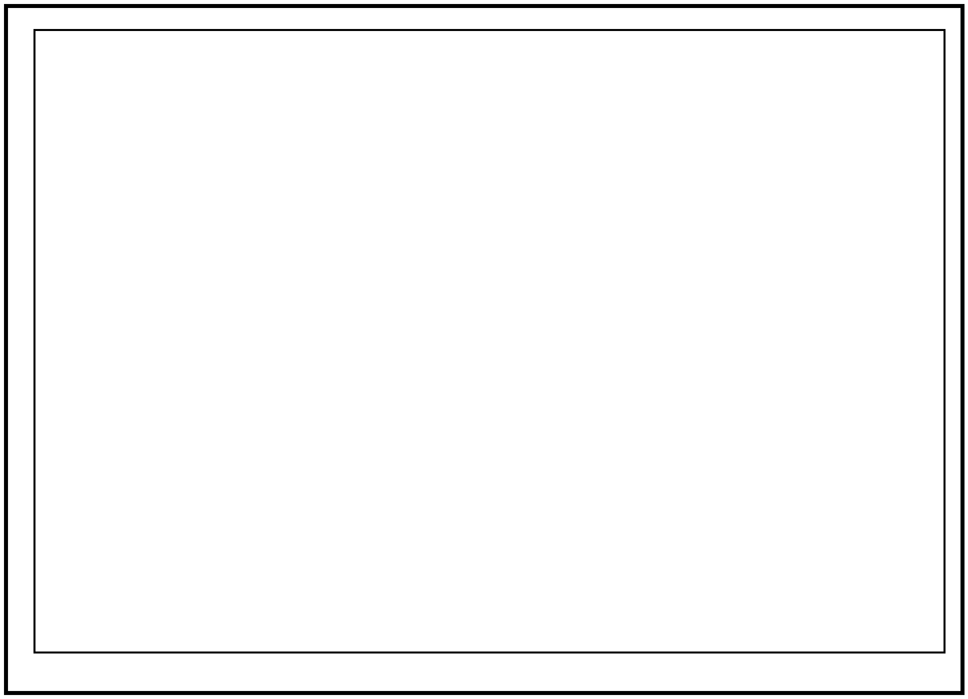


Figure 45. Graphical Representation of Rotation (G68) Code Example

Units of Measure (BNC G70, G71

Before setting the coordinate system at the beginning of the program, the units of measure must be specified in an independent block. A part program may switch between English and Metric modes as long as the format of the dimensions is correct for the chosen mode.

The Imperial Units of Measure code (BNC G70 signals the system that the dimensions are in inches.

BNC G70 is canceled by G71.

The Metric Units of Measure code (BNC G71signals the system that the dimensions are metric units.

BNC G71 is canceled by G70.

Format

These are the command formats for the inch/metric conversion commands:

BNC:

G70: Inch command

G71: Metric command

Important

The BNC G70 and G71 codes do not affect the units of measure used in the graphics and machine status display screens. The displays are controlled by the units selected when entering NC editing.

Peck Drilling (G73)

For Peck Drilling, the spindle moves down in incremental steps and retracts to a position set on the Holes Parameter screen. After each peck, the drill is retracted by the Peck Clearance Plane value set on the General Parameters screen. These screens are described earlier in this manual.

Spindle positioning is performed on the XY plane and hole machining is performed on the Z axis. These parameters are stored as modal values; therefore, if a parameter value does not change for subsequent drilling commands, those commands do not have to contain the parameter.

<u>Format</u>

The command format for the Peck Drilling canned cycle is as follows:

G73 X____, Y____, Z____, R____, Q____, F____, [K____ or L____]

<u>Example</u>

The diagram below illustrates tool movement for the G73 command:

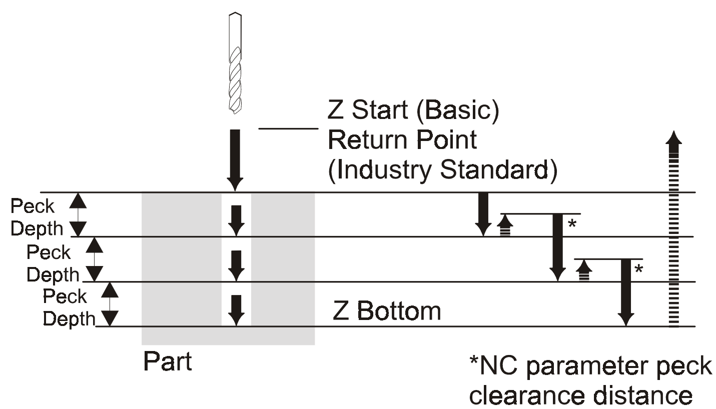


Figure 46. Tool Movement for the Peck Drilling Cycle (G73)

Left-Handed Tapping Cycle (ISNC G74)

During the Left-Handed Tapping Cycle the spindle rotates CCW to the bottom of the hole. Then the spindle stops, an optional dwell is performed, the spindle rotates CW, and left-handed tapping is performed.

The positioning for this cycle is performed on the XY plane and hole machining is performed on the Z axis. During left-handed tapping, the feedrate override is ignored and the cycle does not stop until the end of the return operation, even if a feed hold is applied.

If a Start Spindle Clockwise (M3) code is in effect, the spindle direction will be reversed prior to executing a G74 cycle. Rigid Tapping is performed when an Enable Rigid Tapping (ISNC M29) code is used in a block previous to the G74 block.

<u>Format</u>

The command format for the Left-Handed Tapping cycle is as follows*:*

G74 X____, Y____, Z____, R____, P____, F____, [Q____,]
[K____, or L____]

Note

Z is the distance from the R Point (in Return to R Point in Canned Cycle [G99] mode) to the Z Bottom or the distance from the Initial Point (in Return to Initial Point in Canned Cycle [G98] mode) to the Z Bottom.

Q is the optional peck depth. If Q equals 0.0, pecking is not performed. Q used for G74 with M29 applies only to rigid tapping.

Single-Quadrant Circular Interpolation (BNC G74)

The Single-Quadrant Circular Interpolation Mode (G74) causes the system to interpolate arcs and helices in a single quadrant only. The arc or helix must remain within the quadrant in which it started (the arc or helix cannot be larger than 90°). Since the arc cannot cross quadrants, the center point is determined by looking toward the center of the arc from the start point. I, J, and K are unsigned incremental distances from the arc start point to the center of the arc.

G74 is canceled by G75.

Multi-Quadrant Circular Interpolation (BNC G75)

The Multi-Quadrant Circular Interpolation Mode (G75) is the default and causes the system to interpolate an arc or helix across all quadrants. The arc or helix can start and end in any quadrant. An arc or helix may be up to 360° in this mode. The center point data can be represented in two different ways based on the current machine dimension mode (G90 and G91).

G75 is canceled by G74.

Bore Orient (G76)

The Bore Orient cycle provides a feed-in, stop-feed, orient spindle, move tool away from part surface, rapid-out, and spindle restart sequence suitable for boring operations when the tool needs to be moved away from the part surface before retracting out of the hole. If the default Bore Orient Retract vector is not suitable, I and J words may be used to specify a new retract position.

A value needs to be entered in the Bore Orient Retract field on the Holes Parameters screen (described earlier in this manual). That value specifies the distance the X and Y axes travel to retract the tool from the part surface during the Bore Orient cycle.

A spindle oriented stop is performed at the bottom of the hole and the spindle retracts after shifting in the direction opposite to the cutter direction. High precision and efficient boring is performed without scratching the workpiece surface.

> **Note**
>
> The Bore Orient G86 mode continues to be supported to provide compatibility with existing BNC programs.

The bore orient cycle moves the axes in this manner:

1. The spindle should already be switched on.
2. The spindle positions the tool at the rapid speed to the XY location, if necessary.
3. The spindle moves down at the specified feedrate to the Z value.
4. The spindle stops and orients.
5. The spindle moves from the XY location to the IJ position or to the Bore Orient Retract distance.
6. The system rapidly moves Z to the initial Z location.

Important

This cycle applies only to machines that have an electronic or mechanical orient feature (refer to the machine tool owner's manual).

Format

The format of the Bore Orient cycle is as follows:

G76 X____, Y____, Z____, [I_____, J_____, or Q____] R____, P____, F____, [K____, or L____]

Notes

I and J may also be used instead of Q to specify an incremental bore shift value and direction. If Q is used, the Q value must be a positive number; otherwise, an error message will occur.

For BNC, Q and I, J are optional to maintain compatibility with older programs. A default Z value of 1.0 will be used for BNC if a Q word is not contained in the same block with the G86 command. Q is not modal for BNC.

Caution

The Q value is modal. Since Q is used as the cut-in value for G73 and G83, use care when specifying Q.

Example

The diagram below illustrates tool movement for the Bore Orient cycle:

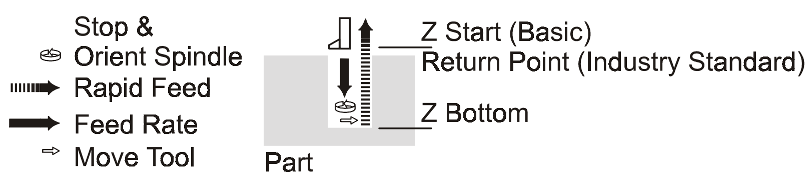


Figure 47. Tool Movement for the Bore Orient Cycle (G76)

Canned Cycle Cancel (G80)

Canned Cycle Cancel is a machine default mode and cancels all canned cycles. When a cycle is canceled using a G80, program execution returns to the One-Shot (G00, G01, G02, or G03) mode that was in effect before the canned cycle was executed. Use either G00, G01, G02, or G03 to cancel a canned cycle.

The G80 cycle also cancels the R and Z Points. That is, R = 0 and Z = 0 for the incremental command. Other drilling data are also canceled.

Drill, Spot Boring (G81)

The Drill, Spot Boring cycle is a feed-in, rapid-out sequence. The axes move in this manner with the spindle switched On:

1. Ensure that the initial Z location is above Z bottom and above any obstructions.
2. The tool is positioned at the Initial Z location and moves at the rapid speed to XY if it is in the block.
3. The spindle drills down to Z Bottom at the specified feedrate.
4. The spindle moves up to Z Start at the rapid speed.

<u>Format</u>

The command format for Drill cycle is as follows:

G81 X____, Y____, Z____, R____, F____, [K____, or L____]

<u>Example</u>

This is a sample BNC Drilling cycle:

G81 Z1.0000 (inches) G90 or G91

Here is a sample ISNC Drilling cycle and a tool movement diagram:

G81 Z-1.0000 (inches) in G91 mode

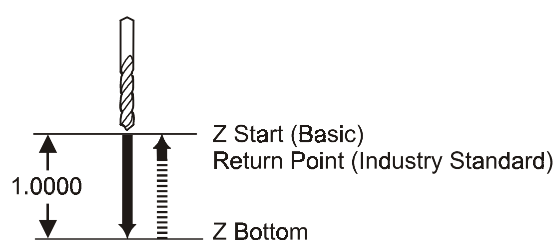


Figure 48. Tool Movement for the Spot Boring Cycle (G81)

Drill with Dwell, Counter Boring (G82)

The Drill with Dwell, Counter Boring cycle provides a feed-in, dwell, and rapid-out sequence.

Format

The command format for the Drill with Dwell cycle, or Counter Boring, is as follows:

G82 X____, Y____, Z____, R____, P____, F____, [K____, or L____]

Example

This diagram illustrates tool movement for the Counter Boring cycle:

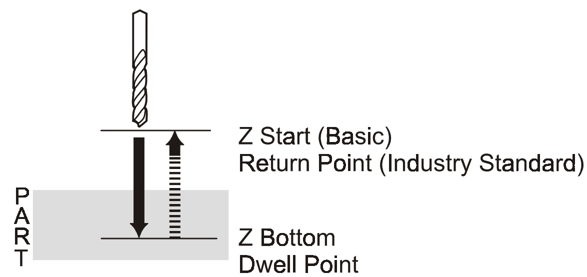


Figure 49. Tool Movement for the Counter Boring Cycle (G82)

Deep Hole Drilling (G83)

The Deep Hole Drilling cycle provides a sequence of feed-in and rapid-out movements until the specified hole depth is reached.

For BNC, each feed-in moves the distance of the peck depth. The tool will rapid back to the Z Start position.

For ISNC the tool will rapid out to the Return point.

Next the tool will rapid down until it reaches the starting point for the next peck (for either BNC or ISNC). The starting point is an incremental distance above the last peck, defined on the Holes Parameter screen as the Peck Clearance Distance.

BNC has three Z values: Z1, Z2, and Z3. They may be programmed in this canned cycle and are unsigned incremental distances. There is a rapid traverse back to R at the end of each pecking cycle and then the tool feed begins above where the tool stopped during the last pecking cycle.

Notes

Z1 is the total depth for the hole.

Z2 is the depth of the first peck.

Z3 is the depth for each of the remaining pecks.

Z2 and Z3 must be smaller than Z1.

If Z2 and Z3 are not programmed, this canned cycle functions like G81.

If Z3 is not programmed, Z2 is the depth for each peck. The last peck for the hole is the programmed peck depth or the remaining distance from the last peck to the bottom of the hole, whichever is smaller.

If Z1, Z2, and Z3 do not change between G83 blocks, they need not be reprogrammed. Use the Precision Cornering codes (G61 and G64) to control the Z axis deceleration between pecks.

Format

The command formats for the Deep Hole Drilling cycle are as follows:

BNC: G83 X____,Y____, Z____, [Z____,] [Z____,] F____,
[K____, or L____]

Notes

For BNC, the first Z is the distance from Z Start to Z Bottom. The second Z is the first cut-in depth. The optional third Z is the depth of the remaining pecks. The Zs are always positive. All of the peck depths will be the same if the third Z is left out.

For BNC, R is always positive and is an incremental distance from the initial point to point R.

ISNC: G83 X____,Y____, Z____, R____, Q____, F____,
[K____, or L____]

Note

ISNC has one Z parameter which represents the location of Z Bottom.

Example

The diagram below illustrates tool movement for the G83 code:

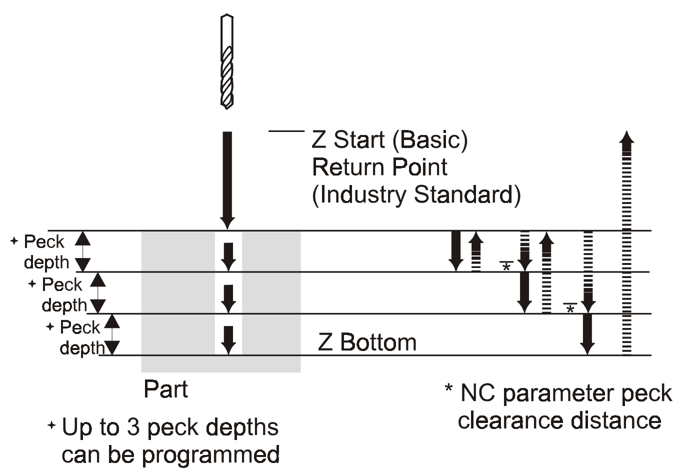


Figure 50. Tool Movement for the Deep Hole Drilling Cycle (G83)

Tapping (G84)

The Tapping cycle provides a tap sequence. The current feedrate (F) and spindle speed (S) are used. The spindle accelerates to the defined speed and the Z axis plunges at the defined feedrate. At the bottom of the hole, the spindle and Z axis decelerate in coordination to a stop. They then reverse directions and accelerate in coordination to the programmed feed and speed. Once back to the original Z level, the spindle shuts off and reverses back to the original direction in preparation for the next operation.

For BNC, G84 is used for right- and left-handed tapping. Start Spindle Clockwise (M3) or Start Spindle Counterclockwise (M4) commands determine whether right- or left-handed tapping is performed.

For ISNC, G84 performs right-handed tapping only. A Start Spindle Counterclockwise (M4) command causes the system to reverse the spindle direction at the start of the cycle to ensure that right-handed tapping is performed.

Use the following formula to calculate the correct feed and speed for the tap cycle:

Feedrate:

$$\text{Feed in inches or mm per minute} = \frac{\text{Spindle RPM}}{\text{threads per inches or mm}}$$

Spindle RPM:

Spindle RPM = Feed in inches (mm) per minute × threads per inch (mm)

Important

When an M3/M4 command is detected in a program and the current tool in the spindle is defined as a tapping tool in tool setup, the system looks 10 blocks ahead for another tap cycle, a G01/G02/G03 code, or a canned cycle other than a tap. If any cutting move (G01, G02, G03, or any canned cycle other than a tap) is found within 10 moves or 10 rapid moves are found, the spindle is turned on as usual. If a G84 is found and all moves from the M3/M4 are rapid moves, the spindle is not turned on, and the rapid moves will be executed with the spindle off.

The spindle rotates clockwise to the bottom of the hole. At the bottom of the hole, the spindle is reversed and rotates counterclockwise and tapping is performed. During the tapping, the feedrate override is ignored and the cycle does not stop until the end of the return operation, even if a feed hold is applied.

For ISNC, a Rigid Tap Enable (M29) command initiates rigid tapping instead of regular tapping. Rigid Tap is disabled with a G00, G01, G02, G03, or G80 command. The programmed feedrate can be overridden for rigid tapping.

Format

The command format for the Tapping cycle is as follows:

G84 X____, Y____, Z ____, R ____, P____, F ____, [Q____,] [K____, or L____]

Note

P is used only with ISNC for the Tapping cycle. P specifies a dwell period at the bottom of the hole and after leaving the hole.

Q, the optional peck depth, is only used with ISNC for the Tapping cycle. If Q equals 0.0, pecking is not performed. Q used for G74 with M29 applies only to rigid tapping.

Example

The diagram below illustrates tool movement for the Tapping cycle (G84):

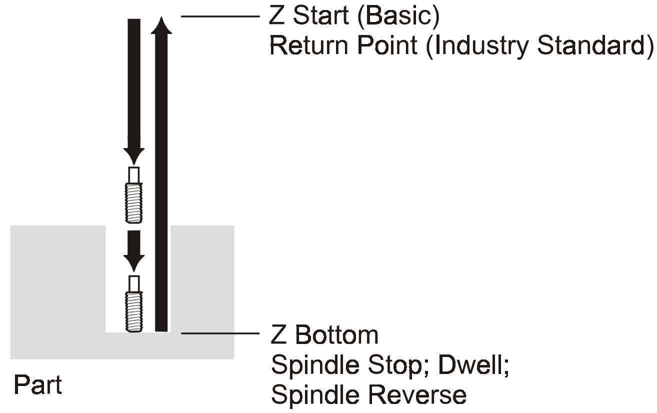


Figure 51. Tool Movement for the Tapping Cycle (G84)

Boring (G85)

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

The 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.

Note

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

Format

The command format of the Boring cycle is as follows:

G85 X____, Y____, Z____, R____, F____, [K____, or L____]

Example

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

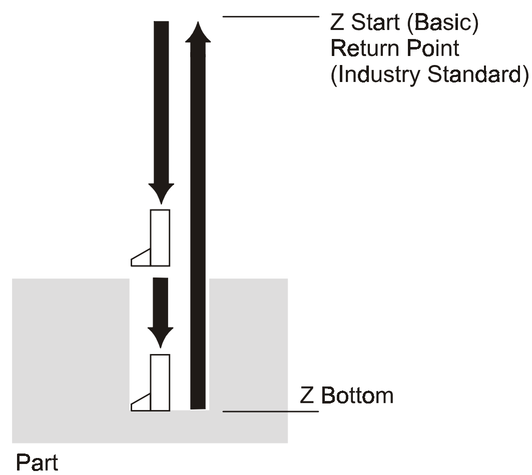


Figure 52. Tool Movement for the Boring Cycle (G85)

Bore Rapid Out Cycle (ISNC G86)

The ISNC 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.

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.

<u>Format</u>

The command format for the Bore Rapid Out cycle is as follows:

G86 X____, Y____, Z____, R____, F____, [K____, or L____]

<u>Example</u>

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

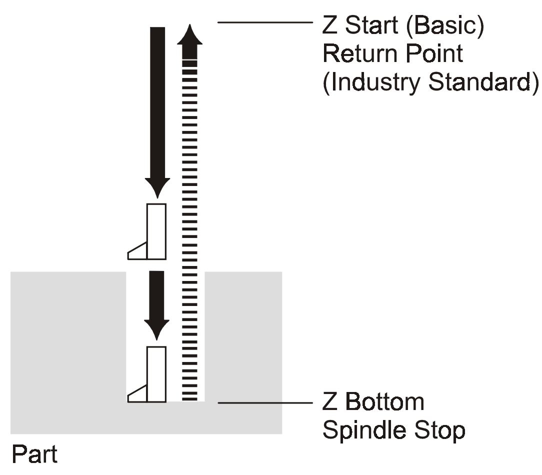


Figure 53. Tool Movement for the Bore Rapid Out Cycle (G86)

Chip Breaker (BNC G87)

The Chip Breaker cycle provides drilling with a dwell every 0.050" (1.27 mm) to break off the chip. The dwell time is automatically calculated so the spindle revolves two times to break the chip. After the dwell, the system feeds another 0.050" (1.27 mm) and again breaks the chip until the bottom of the hole is reached. This cycle breaks the chip without retracting the tool entirely from the hole as with the Deep Hole Drilling cycle (G83). Use the Precision Cornering codes (G61 and G64) to control the Z axis deceleration between dwells.

The Chip Breaker cycle moves the axes in this manner with the spindle switched on:

1. The tool is positioned at the rapid speed to XY if necessary.
2. The spindle moves down 0.05" at the feedrate.
3. The spindle dwells at that location for two rotations.
4. The spindle moves down another 0.05" at the feedrate.
5. This is repeated until the Z depth is reached.
6. The spindle moves at the rapid speed to the initial Z location.

<u>Format</u>

The format of the Chip Breaker cycle is as follows:

G87 X___ Y___ Z___ F____, [K____, or L____]

Back Boring (ISNC G87)

The Back Boring cycle provides a boring sequence in the positive Z direction. Boring is performed from the specified R level to the Z level. Positioning is performed on the XY plane and hole machining is performed on the Z axis.

Format

The command format for the back boring cycle is as follows:

G87 X____, Y____, Z____, R____, Q____, I_____, J_____, P____, F____, [K____, or L____]

Notes

R is used to specify the depth to which the bore moves before shifting over Q or IJ and moving up to the Z level.

Q is used to store an incremental bore shift value. I and J may also be used instead of Q to specify an incremental bore shift value. I and J can be used to specify a distance and direction. Q can only specify distance; the direction is pre-defined by machine parameters.

ISNC G87 Example

The drawing below illustrates tool movement for the Back Boring cycle (ISNC G87):

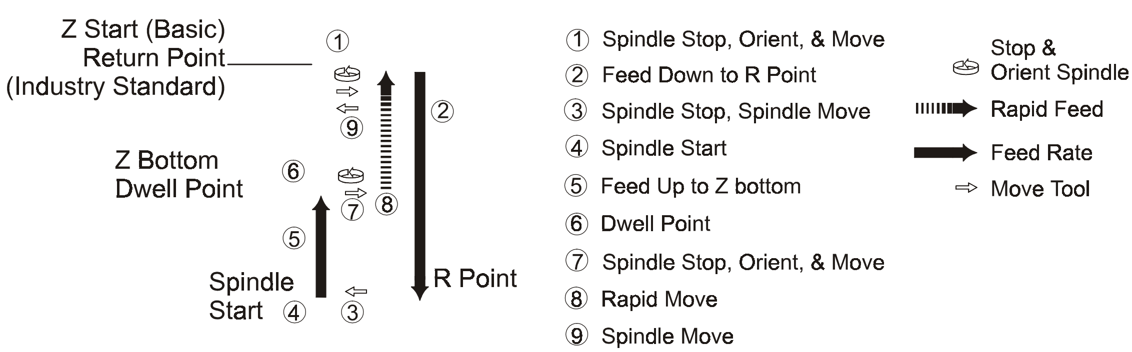


Figure 54. Tool Movement for the Back Boring Cycle (ISNC G87)

Rigid Tapping (BNC G88; ISNC G84.2; ISNC G84.3)

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. ISNC G84.2 is used for right-handed tapping, and ISNC G84.3 is used for left-handed tapping.

The format of the rigid tapping cycle is as follows:

G88 X____, Y____, Z____, Z____, R____, F____, P____
[K____, or L____]

Note

The second Z parameter defines the peck depth.

Canned Boring with Manual Feed Out and Dwell (ISNC G88)

With this canned cycle, a dwell is performed at the bottom of the hole and the system goes into Interrupt mode. The spindle can then be retracted manually using the jog controls. When the desired manual position is reached, follow these steps:

1. Press the console Auto button (in Machine Mode group).
2. The Start button starts flashing and the "Press Start Button" message displays.
3. Press the Start button.
4. The program finishes the canned cycle and then continues with the rest of the program.

Format

The command format for the Boring With Manual Feed Out and Dwell canned cycle is as follows:

G88 X____, Y____, Z____, R____, I_____, J_____, P____, F____,
[K____, or L____]

Example

The drawing below illustrates tool movement for the Canned Boring with Manual Feed Out and Dwell cycle (ISNC G88):

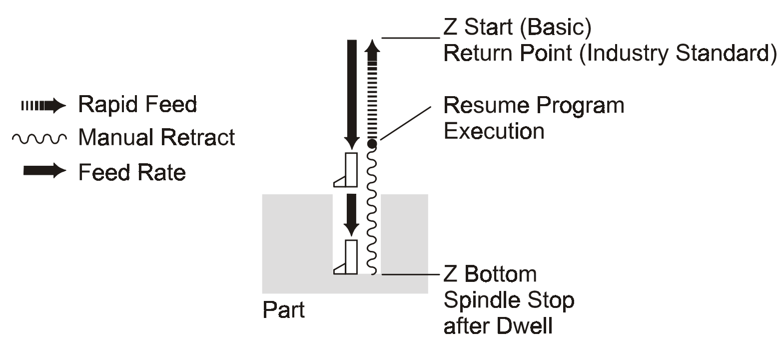


Figure 55. Tool Movement for ISNC G88 Cycle

Bore with Dwell (G89)

The Bore with Dwell cycle provides a feed-in, dwell, and feed-out sequence.

The Bore with Dwell cycle moves the axes in this manner with the spindle switched on:

1. The tool positions at the rapid speed to XY position, if necessary.
2. The spindle moves down at the feedrate to Z Bottom.
3. The spindle stays at the Z Bottom position for the specified dwell time.
4. The spindle moves Z up to Z Start at the rapid speed.

<u>Format</u>

The command format for the Bore with Dwell cycle is as follows:

G89 X____, Y____, Z____, R____, P____, F____, [K____, or L____]

<u>Example</u>

The drawing below illustrates tool movement for the Bore with Dwell cycle (G89):

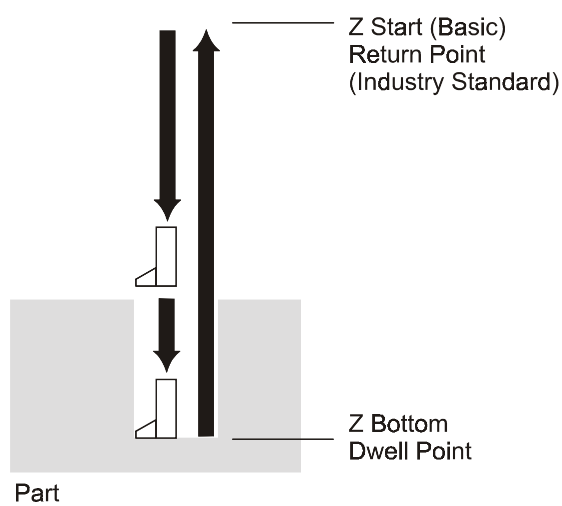


Figure 56. Tool Movement for the Bore with Dwell Cycle (G89)

Absolute and Incremental (G90, G91)

The Absolute Machining Mode (G90) is the default and signals the system that the programmed dimensions are relative to part zero. Once programmed, this default stays in effect until canceled with a G91.

The Incremental Machining Mode (G91) signals the system that all programmed dimensions are incremental distances from the position in the previous block. Once programmed, this mode stays in effect until canceled with a G90.

If Absolute Machining Mode (G90) is activated, the center points I, J, and K are absolute Cartesian (rectangular) coordinates from part zero.

If Incremental Machining Mode (G91) is activated, the center points I, J, and K are signed incremental distances from the arc start point.

<u>Format</u>

This is the command format for each position command:

Absolute command :

G90 X_____Y_____Z_____

Incremental command :

G91 X_____Y_____Z_____

Example

A machine is resting at the programmed part zero location, and the following blocks are executed in inches:

N2 G01 X1.0 Y1.0 F10.0

N4 X1.0 Y1.5

If the system is in Absolute Machining mode (G90), the N2 block causes the axes to travel at a 45° angle to the 1.0" position in X and 1.0" in Y. As a result of the N4 block, the machine remains at the 1.0" position in X and Y moves to the 1.5" position.

If the system is in Incremental Machining mode (G91), the N2 block causes the axes to travel at a 45° angle to the 1.0" position in X and the 1.0" position in Y—just as before. But, as a result of the N4 block, X continues to move 1.0" to the 2.0" position; Y moves 1.5" to the 2.5" position. The diagram below illustrates absolute and incremental axis moves.

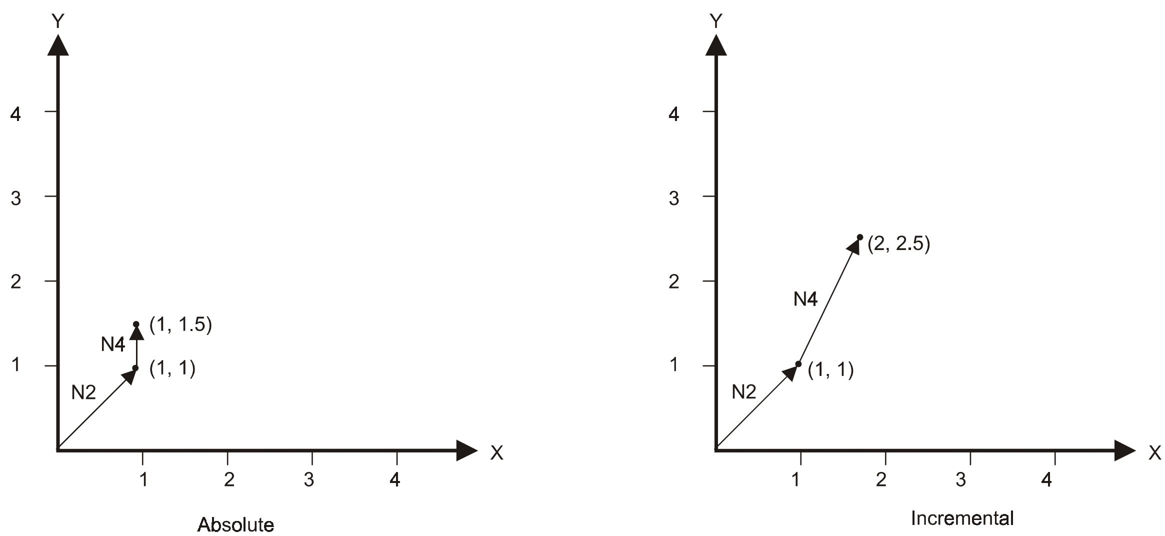


Figure 57. Differences Between Absolute and Incremental

Coordinate System Setting

This section explains the commands used for these coordinate system settings: part zero, machine coordinates, multiple work coordinates, local coordinates, polar coordinates, and automatic return to and from reference point.

Part Zero Setting (G92)

This command establishes the work coordinate system so that a certain point of the tool, for example the tool tip, becomes X, Y, Z, A, B in the established work coordinate system. The distance shifted with this command is added to all subsequent work coordinate system zero point offset values; all work coordinate systems move by the same distance. The G92 command can be used in any work coordinate system (G54–G59).

Important

Cancel Scaling (G50) must be active before selecting G92.

A G92 command makes the dimensions included in the block the new part relative position for the current machine location. The new part zero location is calculated from the current location of the axes and the dimensions included in the G92 block.

The part zero location is only altered for dimensions programmed in the G92 block. This makes it possible to alter the part zero locations for certain axes without affecting the others.

G92 is invalid while *cutter compensation* is on.

Format

This is the format of the setting part zero command:

G92 X_____Y_____Z_____A_____B_____

Example

Set Part Zero (G92) establishes new part relative coordinates at the current axis positions. For example, if the machine is positioned at part relative X2.0 and Y2.0, the block G92 X0.0 Y0.0 would make the current X and Y axis part relative positions equal 0.0. The machine axes will not move, but the status screen changes to reflect the new part zero reference point(s). Any programmed coordinates after the G92 block are referenced to the new part zero location(s).

Use the G92 code for repeating parts of a program at another location. The following is a sample of the codes used in incremental mode. Refer to the diagram below for an illustration of these codes.

```
NC Part Program                 1                          Inch
PARTZERO.FNC
%
N10 G0 X20. Y20.
N12 X40.0
N14 Y40.0
N16 X20.0
N18 Y20.0
N20 X70.
N22 G92 X0.          ⇐ Set new part zero
N24 X20.0
N26 Y40.0
N28 X0.0
N30 Y20.0
M02
```

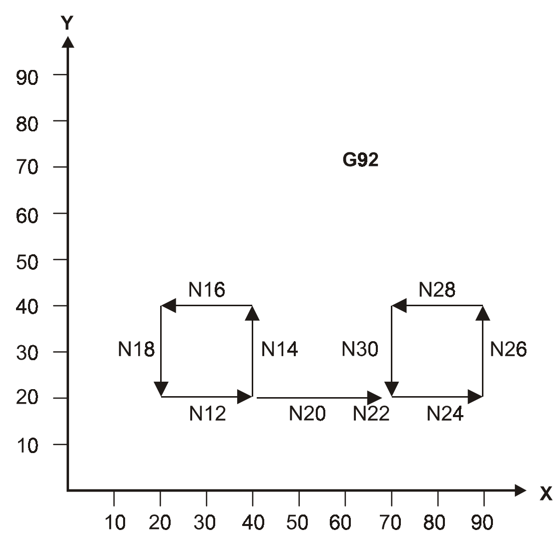


Figure 58. Set Part Zero (G92)

Feed Functions

The Feedrate (F words) value establishes the non-rapid move feedrate. It remains active for all non-rapid moves until another Feedrate code is entered.

For Basic NC, two formats are accepted for both inch and metric units of measurement. The first is in integer form and is interpreted as “tenths of an inch per minute” (to get inches per minute, divide the value by 10). In the second format, the value has a decimal point and is interpreted as “inches per minute.” In the metric mode, the decimal point does not make a difference.

BNC Feedrate Measurement Formats			
Format	**Units**	**Programmed Value**	**Actual BNC Feedrate**
1	English	F30	3.0 inches/min
2	English	F30.0	30 inches/min
1	Metric	F75	75 mm/min
2	Metric	F75.0	75 mm/min

Table 7. BNC Feedrate Measurement Formats

The Feedrate code is active before the other commands in the program block are executed.

Feed Per Minute (G94)

This default mode puts the machine in feed per minute mode. Since the G95 (Feed per Rotation) command is not supported, the G94 is the default command, and no action may be required for the command.

Canned Cycle Descriptions

Canned cycle descriptions, formats, and examples follow.

Return to Initial Point in Canned Cycles (G98)

Position the Z axis to the initial level. The Z axis rapids or feeds to the Z Retract Clearance level, based on the canned cycle being performed. Z Start in the canned cycle description is then equal to the initial point.

Format

G98 (no parameters follow)

Example

Tool movement for the Return to Initial Point in Canned cycles (BNC G98) command:

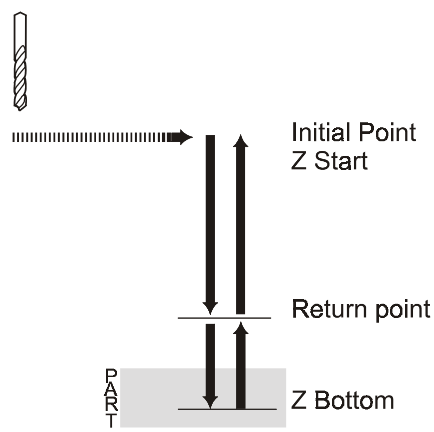


Figure 59. Tool Movement for the BNC G98 Cycle

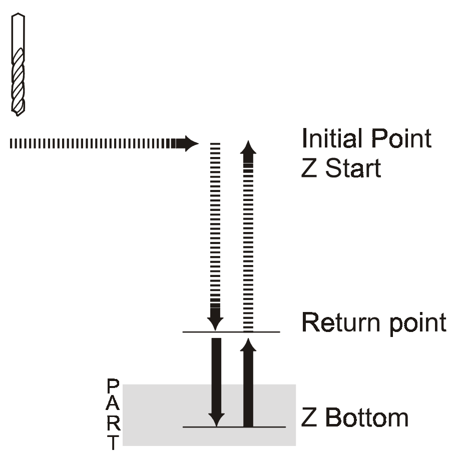


Figure 60. Tool Movement for the ISNC G98 Cycle

Return to R Level in Canned Cycles (G99)

The Return to R Level in Canned Cycles command positions the Z axis to a return (R) level. The Z axis rapids or feeds to the return level between locations during canned cycles. Z Start in the canned cycle descriptions is then equal to the Return Point.

For BNC, specify an R with the G99.

For ISNC, the modal value of R is used.

Format

The format of this code is as follows:

G99 R___

Notes

For BNC, the R parameter is an incremental distance from the initial Z level. Use this code to reduce the returned distance between locations during canned cycles.

For ISNC, the R parameter is an absolute Z level in G90 mode and an incremental negative Z distance in G91 mode.

Example

The drawing below illustrates tool movement for the Return to R Level in Canned cycles (G99) command:

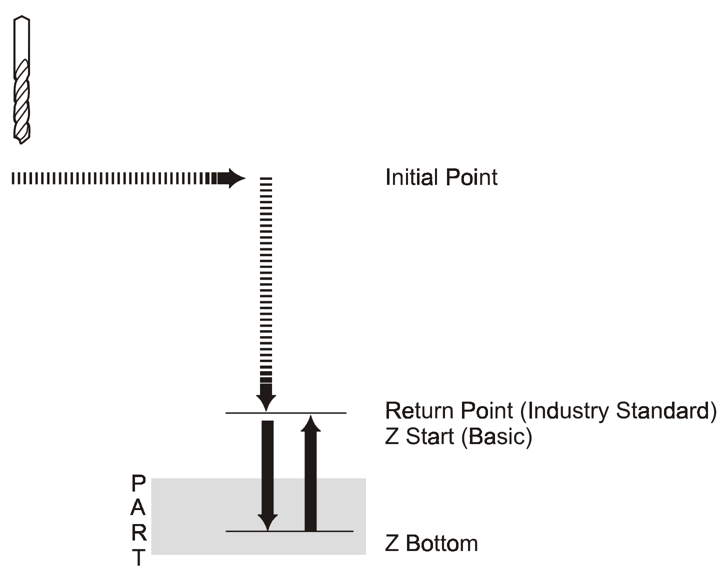


Figure 61. Tool Movement for the G99 Cycle

Canned Cycles

Canned cycles use a one-block G code to provide drilling, boring, and tapping operations. Using one G code instead of several helps simplify writing NC programs. Various *parameters* are used in common with all or most of the canned cycles. For instance, Z is used to specify the canned cycle's depth, P is used to specify dwell time, and F is used to specify the feedrate. For BNC, if there is no spindle speed and direction specified in the program, these values are retrieved from the tool page.

The table below contains canned cycles, G codes, and spindle operation while moving in the negative Z direction, being at Z Bottom, and moving in the positive Z direction.

	G Codes		Spindle Operation		
Canned Cycle	**BNC**	**ISNC**	**In -Z Direction**	**At Z Bottom**	**In +Z Direction**
Peck Drilling	G73	G73	Peck Feed	None	Rapid Traverse
Left Handed Tapping	G84 with M04	G74	Feed	Spindle Stop, Dwell, Spindle CW	Feed
Bore Orient	G76 G86	G76	Feed	Oriented Spindle Stop	Rapid Traverse
Canned Cycle Cancel	G80	G80	None	None	None
Drill, Spot Boring	G81	G81	Feed	None	Rapid Traverse
Drill with Dwell, Counter Boring	G82	G82	Feed	Dwell	Rapid Traverse
Deep Hole Drilling	G83	G83	Peck Feed	None	Rapid Traverse
Tapping	G84 with M03	G84	Feed	Spindle Stop, Dwell, ISNC Spindle CCW or BNC Spindle CW	Feed
Boring	G85	G85	Feed	None	Feed

Continued on next page.

Continued from previous page.

Canned Cycle	**G Codes**		**Spindle Operation**		
	BNC	**ISNC**	**In -Z Direction**	**At Z Bottom**	**In +Z Direction**
Bore Orient Cycle	G86	-			
Bore Rapid Out	-	G86	Feed	Spindle Stop	Rapid Traverse
Back Boring	–	G87	Feed	Spindle Stop, Spindle Move, Spindle Start	Rapid Traverse
Chip Breaker	G87	–	Peck Feed with Dwell	None	Rapid Traverse
Boring with Manual Feed Out	–	G88	Feed	Dwell	Manual Move, Rapid Traverse
Rigid Tapping	G88	G74 with M29; G 84 with M29; 84.2; or 84.3	Feed	Spindle Stop, Dwell, Spindle Reverse	Feed
Bore with Dwell	G89	G89	Feed	Dwell	Rapid Traverse

Table 8. Canned Cycles, G Codes and Z Spindle Operations

These canned cycles are different for BNC than ISNC:

BNC-Specific Canned Cycles	ISNC-Specific Canned Cycles
G84 with M04 Left-Handed Tapping	G74 Left-Handed Tapping
G84 with M03 Tapping	G84 Tapping
G86 Bore Orient Cycle	G86 Bore Rapid Out
G87 Chip Breaker	G87 Back Boring
G88 Rigid Tapping	G74 and G84 with M29 Rigid Tapping
	G88 Boring with manual Feed Out

Table 9. BNC and ISNC Specific Canned Cycles

Canned Cycle Parameters

These parameters are used for programming the various canned cycles. They determine the spindle movement. In the pages that follow, the canned cycles are described and the parameters for each one are identified.

Parameter	Description
F	Feedrate
I	Signed, incremental distance from start point to center of spindle shift position (X axis).
J	Signed, incremental distance from start point to center of spindle shift position (Y axis).
K	Number of repeats for a series of operations in a specified block. Range = 1 through 6; Default = 1. If K = 0, drilling data is stored and no drilling is performed. The incremental distance and direction between canned cycles is determined by the previous block's position from the first canned cycle's position. K and L parameters function the same.
L	Number of repeats for a series of operations in a specified block. Range = 1 through 6; Default = 1. If L = 0 drilling data is stored and no drilling is performed. The incremental distance and direction between canned cycles is determined by the previous block's position from the first canned cycle's position. K and L parameters function the same.
P	Dwell time at the bottom of the hole.
Q	Incremental peck depth value or spindle shift distance.
R	BNC: Incremental, positive distance from the Initial Point to Point R. Only used in G99 mode for BNC. ISNC: Represents absolute Z level at which machining begins in either G98 or G99. Must be specified for all ISNC canned cycles.
X	X axis hole position data.
Y	Y axis hole position data.
Z	Defines Z Bottom location. BNC: Always a positive value. In G98 mode: incremental distance down from initial point. In G99 mode: incremental distance down from the R level. ISNC: In G90 mode: absolute Z level. In G91 mode: negative incremental value measured from the R level.

Table 10. Canned Cycle Parameters

Depth (Z Parameter)

Z is used to specify the canned cycle's depth. All canned cycles require a Z word. Z Start is the Z level where the negative Z (-Z) axis movement begins. This dimension is the same as the Return to Initial Point in Canned Cycle (G98) and the Return to R Point in Canned Cycle (G99) codes. The Z Bottom parameter is the point of maximum Z down (except for ISNC G88) and the dimension where the -Z axis movement ends.

A rapid move at the Z Start level is automatically used to move from one canned cycle block to another. Make sure the current Z Start level is high enough to clear all fixtures and obstacles.

Note the differences in the definitions for BNC and ISNC Z parameters in the previous table.

- For BNC, the current Z level should be established before invoking the canned cycle (via G00 or G01). Once a Z distance is established, it does not need to be reprogrammed until the canned cycle mode is canceled or changed.
- For ISNC, the Z word represents a negative or positive absolute Z drilling level in G90 mode which must be below the current Z level, or an incremental negative distance from the current R level in G91 mode.

Dwell (P Parameter)

Many of the canned cycles have dwell capability. The scaling factors used with the canned cycle dwell *parameter* P are the same as Dwell, Exact Stop (G04). The length of dwell time is modal and can be specified using one of these methods:

- G04 with a P or X value
- P value with a canned cycle command
- Dwell parameters on the Holes Parameters screen

Note

Taps use the Bore Dwell parameter.

If you use the default dwell parameters on the Holes Parameters screen, G04 P0.0 or a P0.0 is required with the canned cycle command to cancel any previously commanded dwell time.

Feedrate (F Parameter)

The current feedrate is used for feed moves and may be reprogrammed in any canned cycle block by including an F word. The feedrate parameter applies only to the Z direction during canned cycles.

Note

- For BNC files, if no decimal point is included, the system automatically divides the feedrate by 10.
- For ISNC files, if no decimal point is included and the Assume Feedrate .1 Increment field on the NC Parameters—Configuration Parameters screen is set to Yes, the system automatically divides the feedrate by 10.

Canceling or Replacing Canned Cycles

All canned cycles are *canceled* by G00, G01, G02, G03, (the One-Shot Group 00 G codes) or G80 (Canned Cycle Cancel).

Current canned cycles can be *replaced* with another canned cycle without first canceling the canned cycle.

If a G00, G01, G02, G03, or G80 occurs in the same block with a canned cycle command (for example G00 G85), the G00 is ignored and the canned cycle command (G85 in this case) is executed. If a G00, G01, G02, or G03 command follows a canned cycle command, the X, Y, Z parameters are used to perform the interpolation or rapid positioning, and the remaining canned cycle parameters in the block are ignored.

All canned cycle data are modal. When a canned cycle is canceled using G00 or G80, the R point, canned cycle repetition value K, and the Q (cut-in, bore shift) are canceled.

Important

Except for tap cycles, canned cycles do not activate the spindle. The program must have a Start Spindle Clockwise (M03) or Start Spindle Counterclockwise (M04) to turn on the spindle prior to executing a canned cycle. For tap cycles, both the spindle speed and direction are retrieved from the tool library if not specified in the program. If a spindle speed is not provided with the M3 or M4, the spindle speed from the tool library is used.

Canned cycles, which turn off the spindle during the cycle, automatically restore the spindle to the original speed and direction before completing the cycle. If a canned cycle requires a certain spindle direction and the opposite spindle direction is currently in effect, the system reverses the spindle direction automatically.

Spindle Speed - S Codes

The Spindle Speed code (S) specifies the spindle rotation speed. The spindle does not rotate until a Start Spindle Clockwise (M03) or a Start Spindle Counterclockwise (M04) is programmed. The software retrieves the spindle speed from the tool library if an S code is not provided.

If the S is present with an M03 or an M04 in the same program block, it is active before the other codes in the program block are executed. If an S is not specified prior to the first M03 or M04, the speed specified in the Tool Setup data is used. As soon as an S appears in the program, its value is used for the M03s and M04s that follow until a new S value is encountered.

For ISNC, if the spindle has already been turned on, the S code is sufficient for changing spindle speed. If the spindle is already turned on and an S code occurs either in a tool change block or in a block following a tool change block, the spindle ramps up to the new spindle speed after the tool change.

Tool Functions

These codes control tool selection: T, L, and D. The L and D codes are for BNC only. To activate these codes, an M06 code must be contained in the same block. To activate the L and D words, an M06 must be used with a T word. The NC Parameters screen contains two fields for controlling tool changes: the Default Tool Number and the M6 Initiates Tool Change.

D Codes (BNC)

The Tool Diameter Offset codes (D values) are used in BNC programs and cause the specified dimension to be loaded into the tool diameter register. Otherwise, the Diameter value in the appropriate Tool Setup data is used. This dimension is used for cutter compensation.

Negative values are not permitted.

L Codes(BNC)

The Tool Length Offset (L) codes cause the specified dimension to be loaded into the tool offset register. Otherwise, the Zero Calibration value in the appropriate Tool Setup data is used.

Negative values are not permitted.

T Codes

The Tool Select (T) codes specify the tool number. The value is composed of up to two digits. Placing the T word in a block does NOT cause a tool change to occur.

If the M6 Initiates Tool Change field is set to yes, the M06 code must be used to initiate the tool change.

Miscellaneous Functions - M Codes

Miscellaneous Functions (M codes) cause machine-related action (e.g., coolant control and tool changes). Each Miscellaneous Function is explained below. Multiple M codes can be used within an NC block.

M Code Table

M Code	Definition
M00	Cancels the spindle and coolant functions; stops part program execution
M01	Program stop often used when the operator wants to refixture the part
M02	Marks the end of the program; stops the spindle, coolant, and axes feed
M03	Starts clockwise rotation of the spindle
M04	Starts counterclockwise rotation of the spindle
M05	Switches the spindle off
M06	Requests an automatic tool change
M07	Switches on secondary coolant systems
M08	Switches on primary coolant system
M09	Switches off both the primary and secondary coolant
M10	Switches on both the primary and secondary coolant
M20	Advances the indexer one position
M25	Retracts the Z axis to the home position (tool change height)
ISNC M29	Enables rigid tapping
M30	Indicates the end of the main program
M32	Clamps the rotary A axis
M33	Unclamps the rotary A axis
M34	Clamps the rotary B axis
M35	Unclamps the rotary B axis
M36	Switches off the servos
M38	Reads and places the state of the laser OK signal.
M39	Reads and places the state of the laser static signal.
M40	Reads and places the state of the laser dynamic signal.

Table 11. M Codes

M Code Table (Continued)—Laser Operation

M Code	Definition
M41	Deactivates two-touch probing when using the G31 command
M42	Enables automatic two-touch probing with the G31 command. If the part probe touches during a G31 move, the probe will automatically back up and then attempt a second touch at a reduced feedrate.
M43	Increases the barrier air.
M44	Reduces barrier air.
M45	Opens the shutter.
M46	Closes the shutter.
M47	Turns the laser emitter on.
M48	Turns the laser emitter off.
M49	Turns the laser receiver on.
M50	Turns the laser receiver off.
M52	Enables auxiliary output 1
M53	Enables auxiliary output 2
M54	Enables auxiliary output 3
M55	Enables auxiliary output 4
M62	Disables auxiliary output 1
M63	Disables auxiliary output 2
M64	Disables auxiliary output 3
M65	Disables auxiliary output 4
M76	Normal A Axis operation (default)
M77	Reverses A Axis operation
M78	Normal B Axis operation (default)
M79	Reverses B Axis operation
M80	C Axis is right-handed (default)
M81	C Axis is left-handed
M98	Subprogram call
M99	Jump; Return from subprogram

Table 12. M Codes

Program Functions

The Program Functions (M00, M01, M02, and M30) stop the execution of the part programs.

Program Stop (M00)

The Program Stop (M00) cancels the spindle and coolant functions and terminates further program execution after completion of other commands in the same program block. When the program is stopped, existing modal information remains unchanged as in single block operation. The Start Cycle button on the control flashes and this prompt message appears:

Cycle complete; press start to continue.

Pressing the Start Cycle button resumes the spindle and coolant operation and continues the program execution.

This M code should not be set simultaneously with other M codes. M00 is executed following execution of the rest of the address words on the block. Here is an example using the M00 code:

```
N10 G01 X2. Y1. F10. M00
```

In this example, the machine moves to the X2/Y1 location before it shuts down.

Important

Program blocks should be included that retract the tool to a safe position before a block containing an M00 is programmed. If these program blocks are not included, the spindle stops while cutting the part.

Planned Stop (M01)

The Planned Stop Code (M01) pauses the program and shuts off the spindle. M01 is ignored unless previously validated in the parameter page. If you want to open the CE Safety enclosure doors after M01 executes, press the Machine Mode Interrupt console key, then press the Start Cycle button. The enclosure doors can be opened and the axes jogged. To continue with the program, close the enclosure doors and press the Start Cycle button.

M01 is only effective when the Optional Program Stop field on the General Parameters screen is Enabled.

Note

Include a data block to retract the tool to a safe position before a block containing an M01 is programmed. If the retract tool data block is not included, the spindle will stop while cutting the part.

End of Program (M02)

The End Of Program code (M02) indicates the end of the main program (the completion of the part), and is necessary for the registration of CNC commands from tape to memory. M02 stops the spindle, the coolant, and the axis feed after completing all of the commands in the program. M02 is active after the block is executed.

Note

The M02 does NOT stop the NC program loader if the program is loading from a serial link. An E character must be transmitted to signal the loader that the entire program has been sent to the remote device.

Important

This M code should not be set simultaneously with other M codes unless it is the last M code in the block.

Start Spindle Clockwise (M03)

The Start Spindle Clockwise code starts a clockwise spindle rotation (as viewed from the headstock). The spindle reaches the programmed speed before X, Y, and Z (also A and B if present) axis feed starts. If the spindle speed has not been defined, the Tool Setup screen’s spindle speed is used.

M03 is active before the other commands in the block are executed.

Start Spindle Counterclockwise (M04)

The Start Spindle Counterclockwise code starts spindle rotation in a counterclockwise direction (as viewed from the headstock). The spindle reaches the programmed speed before X, Y, Z (A or B) feed starts. If the spindle speed has not been defined, the Tool Setup screen's spindle speed is used.

M04 is active before the other commands in the block are executed.

Spindle Off (M05)

The Spindle Off code is the default and causes the spindle to stop in a normal manner. If the machine is equipped with a brake, it is applied. The coolant is also turned Off.

M05 is active after the other commands in the block are executed.

M6 Initiates Tool Change

Use this field on the NC Parameters screen to indicate whether tool changes are initiated with the M6 or with the T code. Set this field to No and the M6 is ignored and tool changes are initiated whenever a T code is found in the program (not when T is used for user-defined subprogram or subprogram parameter).

If this field is set to Yes, the M6 is required for tool changes.

The following is the Change NC Parameters screen containing the M6 Initiates Tool Change field:

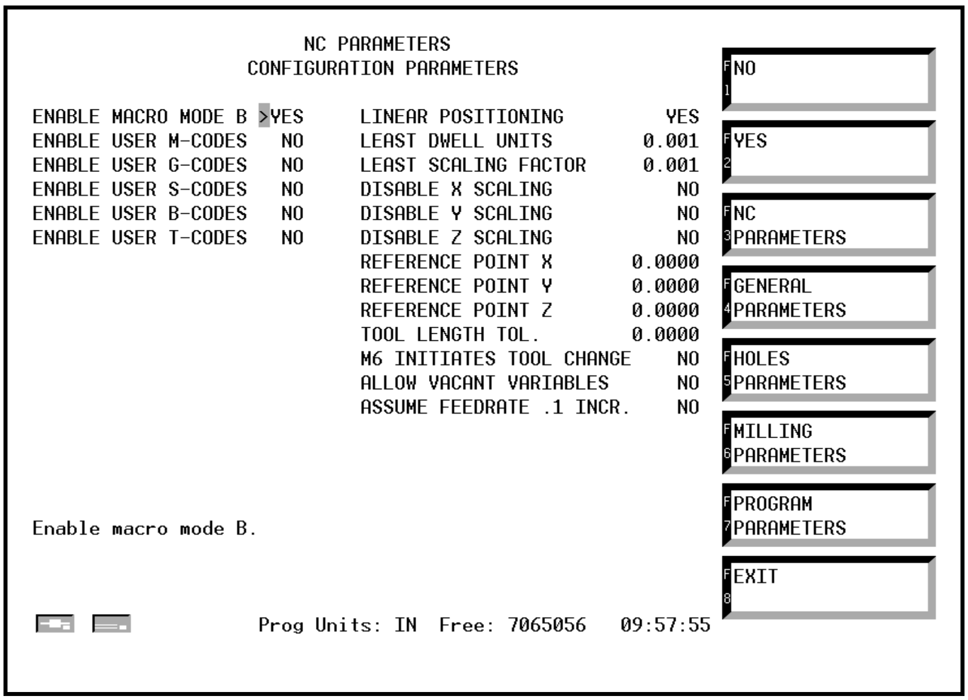


Figure 62. NC Parameters—Configuration Parameters Screen

Change Tool (M06)

The Change Tool code requests that the machine perform a tool change. These tool changes should be performed in rapid traverse mode. The following sequence occurs if an automatic tool changer is present and in the Auto Tool Change mode:

1. The Z axis retracts to tool change position.
2. The machine moves the X and Y axes to the Tool Change position if the tool change position parameter is set to Yes.
3. The spindle orients and stops.
4. The “old” tool is returned to the tool changer.
5. The “new” tool is placed in the spindle.
6. New tool offsets from the Tool Offset screen are loaded into the appropriate registers. The Tool Length Offsets from G43 and G44 remain in effect.
7. The program continues.

Important

The M06 is optional if the M6 Initiates Tool Change field on the NC Parameters screen is set to Yes; otherwise, tool changes are performed with the T code.

This sequence occurs for manual tool changes:

1. Z axis retracts to its tool change position.
2. The machine moves the X and Y axes to the Tool Change position if the tool change position parameter is set to Yes.
3. The spindle stops and orients.
4. The screen prompts for a tool change.
5. Change the tool and press the Start Cycle button on the control to allow the program to continue.
6. New tool offsets are loaded into the appropriate registers.
7. The program continues.

Note

The first Z dimension after a tool change must be absolute. Any Z dimension programmed in a tool change block is ignored.

Secondary Coolant On (M07)

The Secondary Coolant On code switches on the mist coolant, if available. M07 is active before the other commands in the block are executed.

Primary Coolant On (M08)

The Primary Coolant On code switches on the flood coolant, if available. M08 is active before the other commands in the block are executed.

Both Coolant Systems Off (M09)

The Coolant Off code is the default and switches off the coolant if it has been activated by Secondary Coolant On (M07) or Primary Coolant On (M08). M09 is active after the other commands in the block are executed.

Both Coolant Systems On (M10)

The Both Coolant Systems On code switches on the coolant if it has been activated by Both Coolant Systems Off (M09).

Clamp C-axis (M12)

The Clamp C axis code clamps the C axis. For C axis moves after M12, the C axis is automatically unclamped for the move and clamped again after the move is complete.

M12 is active before the other commands in the block are executed and is canceled by an Unclamp C axis (M13) command.

Unclamp C-axis (M13)

The Unclamp C axis code unclamps the C axis until an M12 is programmed.

M13 is active before the other commands in the block are executed and is canceled by a Clamp C axis (M12) command.

Oriented Spindle Stop (M19)

The Oriented Spindle Stop code causes the spindle to stop in the oriented position. A brake, if available, will be applied. The coolant is also turned off. This function only applies to machines which have an orient feature. On machines without the orient feature, this function works like the Spindle Off (M05) command.

M19 is active after the other commands in the block are executed.

Pulse Indexer One Increment (M20)

The Pulse Indexer One Increment code advances the *indexer* one position. A reply signal is sent back from the indexer to indicate when it is in position. When the signal is received, the program continues. For multiple indexes, separate M20 blocks must be programmed. (Refer to the indexer's manual and the *Hurco Maintenance Manual* for information on attaching an indexer to the machine.)

M20 is active after the other commands in the block are executed.

Z Axis to Home Position (M25) - Basic NC Programming only

The Z Axis to Home Position code retracts the Z axis to the home position (tool change height) at the rapid traverse rate selected in the Program Parameters screen. The first Z value after an M25 must be absolute.

M25 is active before the other commands in the block are executed.

Enable Rigid Tapping (ISNC M29)

When Enable Rigid Tapping (M29) is used before a Left-Handed Tapping Cycle (ISNC G74) or Taping Cycle (G84) command, rigid tapping is performed. M29 stays in effect until a One-Shot (G00, G01, G02, G03) code or Canned Cycle Cancel (G80) command is used.

End Program (M30)

This command indicates the end of the main program and is necessary for the registration of CNC commands from *tape* to memory. When the end of program command is executed, the CNC enters a *reset state* and the program returns to the beginning. Cycle operation may be stopped, and the CNC unit may reset depending on the machine tool. The CNC tape is rewound to the start of the program in both memory and tape operation. However, when using a tape reader without reels, the tape is not rewound. When using a tape reader with reels, the tape returns to the ER (%) code at the start of the tape even if several programs exist. Some machines indicate tape rewind with the End of Program (M02) command.

Important

This M code should not be set simultaneously with other M codes unless it is the last M code in the block.

Clamp A-axis (M32)

The Clamp A-axis code clamps the A axis. For A axis moves after M32, the A axis is automatically unclamped for the move and clamped again after the move is complete.

M32 is active before the other commands in the block are executed and is canceled by an Unclamp A axis (M33) command.

Unclamp A-axis (M33)

The Unclamp A axis code unclamps the A axis until an M32 is programmed.

M33 is active before the other commands in the block are executed and is canceled by a Clamp A axis (M32) command.

Clamp B-axis (M34)

The Clamp B axis code clamps the B axis. For B axis moves after M34, the B axis is automatically unclamped for the move and clamped again after the move is complete.

M34 is active before the other commands in the block are executed and is canceled by an Unclamp B axis (M35) command.

Unclamp B-axis (M35)

The Unclamp B axis code unclamps the B axis until an M34 is programmed.

M35 is active before the other commands in the block are executed and is canceled by a Clamp B axis (M34) command.

Servo Off Code (M36)

The servos may be turned off using the Servo Off (M36) command.

Control power to the machine will be turned off. The control will still be powered on. This is similar to an emergency stop.

Laser Input Update (M38-M40)

These codes read the state of the three laser inputs (M38: OK signal; M39: static signal; and M40: dynamic signal).

Single-Touch Probing (M41)

For a G31 probing move, perform one touch.

Double-Touch Probing (M42)

For a G31 probing move, perform two touches. This is the default mode.

Barrier Air Control (M43 and M44)

Barrier air is used to prevent chips and debris from getting into the laser emitter and receiver. M43 causes the air flow at the probe to increase M44 reduces the airflow. During operation of the probe, the barrier air should be increased whenever the probe shutter is open. It should remain at the high flow rate except during the actual tool measurement. When the shutter is closed, the flow rate may be reduced.

Shutter Probe Control (M45 and M46)

A pneumatic shutter protects the probe. During a measurement, the barrier air should be increased and the shutter opened. After the probe cycle is completed, the shutter should be closed and the barrier air reduced. M45 causes a brief puff of air that helps clear chips and debris from the probe. M46 closes the shutter.

Laser Emitter On/Off Control (M47 and M48)

M47 turns the laser emitter on. M48 turns the laser off. It is recommended to turn the laser emitter off when not in use.

Laser Receiver On/Off (M49 and M50)

M47 turns the laser receiver on. M48 turns the laser receiver off. It is recommended to turn the laser receiver off when not in use.

Enable Auxiliary Output 1 through 4 (M52 – M55)

M52 through M55 are used to individually enable auxiliary equipment or a unique machine function from within a part program. Enter performance time for the machine-specific M code in the M Code Table. When M52 through M55 are active, the corresponding auxiliary equipment or machine function is turned on, and any performance time is added to estimated run time.

M52 enables Auxiliary Output 1, M53 enables Auxiliary Output 2, M54 enables Auxiliary Output 3, M55 enables Auxiliary Output 4.

Disable Auxiliary Output 1 through 4 (M62 – M65)

M62 through M65 turn off auxiliary equipment or machine functions enabled with M codes M52 through M55.

M62 disables Auxiliary Output 1 (M52), M63 disables Auxiliary Output 2 (M53), M64 disables Auxiliary Output 3 (M54), and M64 disables Auxiliary Output 4 (M55).

Right Handed C Axis (M80)

When this M code is active and a command is given to the C axis to go in a positive direction, the axis will rotate counter clockwise.

Left Handed C Axis (M81)

When is M code is active and a command is given to the C axis to go in a negative direction, the axis will rotate clockwise.

Subprogram Call (M98)

One way of specifying the number of iterations for a subprogram to perform is with M98 subprogram calls.

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.

In Example 1 below, M98 P60050 must be used instead of M98 P650 to run program 50 with 6 iterations because the subprogram number (50) is less than 1000.

In Example 2, the M98 P23013000 subprogram example, the four digits to the left (2301) specify the number of iterations, and the four digits to the right (3000) specify the subprogram number.

Example 1:

M98 P60050

Subprogram number

Number of iterations

Example 2:

M98 P23013000

Subprogram number

Number of iterations

As other examples, M98 P1 runs program 1 with no iterations, and M98 P100001 runs program 1 ten times.

Jump; Return from Subprogram (M99)

Each subprogram ends with an *M99* Jump statement.

NC Example Program Filenames

This table identifies example programs used in this manual.

Filename	Section
PARTZERO.FNC	Part Zero Setting (G92)
MACHCOOR.FNC	Machine Coordinates (G53)
LOC_COOR.FNC	Local Coordinate System Setting (G52)
PIE.FNC	Polar Coordinate Command (One Shot) (G16)
PLAIN_28.FNC	Return To and From Reference Point (G28 and G29)
G02.FNC	Circular and Helical Interpolation (G02)
G03ABS.HNC	Basic NC G03
G03INC.HNC	Basic NC G03
G10 (Partial Prog.)	Assigning Tool Offsets (L3) with G10
G43 (Partial Prog.)	Tool Length Offset (G43, G44, G49)
G45_G48.FNC	Tool Radius Offset (G45-G48)
G68.FNC	Rotation (G68 and G69)
G31_TEXT.FNC	Skip (Probing) Function (G31)
TRU_CRC.FNC	Variable Example
BOLT_LN.FNC	G65 Subprogram Call
G65INST.FNC	G65 Macro Instruction Listing
G66.FNC	G66 Modal Subprogram Call
G86_TRAN.FNC	User Defined G Code Example
USERMAC.FNC	User Defined M and G Codes Example
BST.FNC	User Defined S, B, and T Codes
PART.FNC	NC Part Programming Example

Table 13. NC Example Programs

Index

C (continued)

D

D (continued)

E

F

G

G (continued)

G (continued)

H

I

J

K

L

M

M (continued)

N

O

P

P (continued)

Q

R

S