FANUC

GE Fanuc Automation

Computer Numerical Control Products

Series 16 / 160 / 18 / 180 – TB for Lathe

Operator's Manual

GFZ-62444E/03 February 1995

GFL-001

Warnings, Cautions, and Notes as Used in this Publication

Warning

Warning notices are used in this publication to emphasize that hazardous voltages, currents, temperatures, or other conditions that could cause personal injury exist in this equipment or may be associated with its use.

In situations where inattention could cause either personal injury or damage to equipment, a Warning notice is used.

Caution

Caution notices are used where equipment might be damaged if care is not taken.

Note

Notes merely call attention to information that is especially significant to understanding and operating the equipment.

This document is based on information available at the time of its publication. While efforts have been made to be accurate, the information contained herein does not purport to cover all details or variations in hardware or software, nor to provide for every possible contingency in connection with installation, operation, or maintenance. Features may be described herein which are not present in all hardware and software systems. GE Fanuc Automation assumes no obligation of notice to holders of this document with respect to changes subsequently made.

GE Fanuc Automation makes no representation or warranty, expressed, implied, or statutory with respect to, and assumes no responsibility for the accuracy, completeness, sufficiency, or usefulness of the information contained herein. No warranties of merchantability or fitness for purpose shall apply.

Export Control Information

This document and the information contained in this document are classified with an Export Control Classification Number of 2E201. It is restricted and controlled under US export regulations. If you intend to export (or reexport), the document, directly or indirectly, or technical information relating thereto supplied hereunder or any portion thereof, it is your responsibility to ensure compliance with U.S. export control regulations and, if appropriate, to secure any required export licenses in your own name.

Table of Contents

I. GENERAL

II. PROGRAMMING

IV. MAINTENANCE

APPENDIX

I. GENERAL

1 GENERAL

This manual consists of the following parts:

About this manual

I. GENERAL
Describes chapter organization, applicable models, related manuals, and notes for reading this manual.

II. PROGRAMMING
Describes each function: Format used to program functions in the NC language, characteristics, and restrictions. When a program is created through conversational automatic programming function, refer to the manual for the conversational automatic programming function (Table1).

III. OPERATION
Describes the manual operation and automatic operation of a machine, procedures for inputting and outputting data, and procedures for editing a program.

IV. MAINTENANCE
Describes alarms, self–diagnosis, and procedures for replacing fuses and batteries.

V. APPENDIX
Lists tape codes, valid data ranges, and error codes.

Some functions described in this manual may not be applied to some products. For detail, refer to the DESCRIPTIONS manual.

This manual does not describe parameters in detail. For details on parameters mentioned in this manual, refer to the manual for parameters (B–62442E).

This manual describes all optional functions. Look up the options incorporated into your system in the manual written by the machine tool builder.
The models covered by this manual, and their abbreviations are:

Product name	Abbreviations	
FANUC Series 16–TB	16–TB	Series 16
FANUC Series 18–TB	18–TB	Series 18
FANUC Series 160–TB	160–TB	Series 160
FANUC Series 180–TB	180–TB	Series 180

Special symbols

This manual uses the following symbols:

IP_ : Indicates a combination of axes such as X__ Y__ Z (used in PROGRAMMING.).

; : Indicates the end of a block. It actually corresponds to the ISO code LF or EIA code CR.

Related manuals

The table below lists manuals related to MODEL B of Series 16, Series 18, Series 160 and Series 180.
In the table, this manual is marked with an asterisk (*).

Table 1 Related Manuals

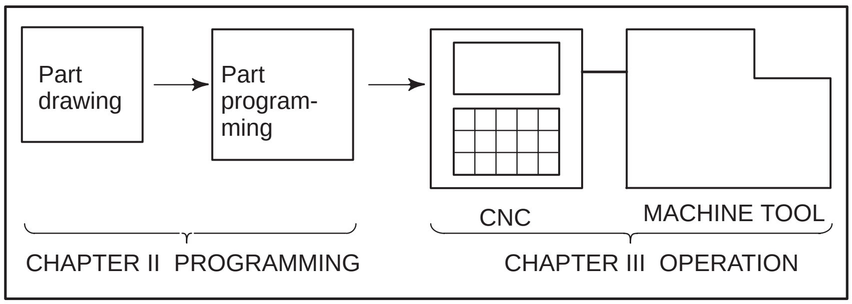
Manual name	Specification number	
DESCRIPTIONS	B–62442E	
CONNECTION MANUAL (Hardware)	B–62443E	
CONNECTION MANUAL (Function)	B–62443E–1	
OPERATOR’S MANUAL for Lathe	B–62444E	*
OPERATOR’S MANUAL for Machining center	B–62454E	
MAINTENANCE MANUAL	B–62445E	
PARAMETER MANUAL	B–62450E	
PROGRAMMING MANUAL (Macro Compiler / Macro Executer)	B–61803E–1	
FAPT MACRO COMPILER PROGRAMMING MANUAL	B–66102E	
FANUC Super CAP T OPERATOR’S MANUAL	B–62444E–1	
FANUC Super CAP M OPERATOR’S MANUAL	B–62154E	
FANUC Super CAP M PROGRAMMING MANUAL	B–62153E	
CONVERSATIONAL AUTOMATIC PROGRAMMING FUNCTION I for Lathe OPERATOR’S MANUAL	B–61804E–1	
CONVERSATIONAL AUTOMATIC PROGRAMMING FUNCTION for Lathe OPERATOR’S MANUAL (Series 15–MODEL B, Series 16 CAP II)	B–61804E–2	

1.1 GENERAL FLOW OF OPERATION OF CNC MACHINE TOOL

When machining the part using the CNC machine tool, first prepare the program, then operate the CNC machine by using the program.

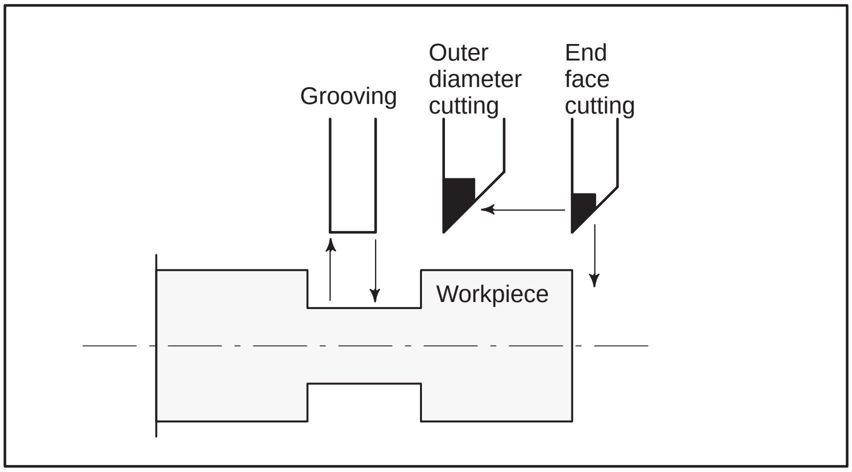
1) First, prepare the program from a part drawing to operate the CNC machine tool.
   How to prepare the program is described in the Chapter II. PROGRAMMING.

2) The program is to be read into the CNC system. Then, mount the workpieces and tools on the machine, and operate the tools according to the programming. Finally, execute the machining actually.
   How to operate the CNC system is described in the Chapter III. OPERATION.



Before the actual programming, make the machining plan for how to machine the part.
Machining plan
1. Determination of workpieces machining range
2. Method of mounting workpieces on the machine tool
3. Machining sequence in every cutting process
4. Cutting tools and cutting conditions
Decide the cutting method in every cutting process.

Cutting process / Cutting procedure	1	2	3
	End face cutting	Outer diameter cutting	Grooving
1. Cutting method : Rough Semi Finish			
2. Cutting tools			
3. Cutting conditions : Feedrate Cutting depth			
4. Tool path			



Prepare the program of the tool path and cutting condition according to the workpiece figure, for each cutting.

1.2 NOTES ON READING THIS MANUAL

1) The function of an CNC machine tool system depends not only on the CNC, but on the combination of the machine tool, its magnetic cabinet, the servo system, the CNC, the operator's panels, etc. It is too difficult to describe the function, programming, and operation relating to all combinations. This manual generally describes these from the stand–point of the CNC. So, for details on a particular CNC machine tool, refer to the manual issued by the machine tool builder, which should take precedence over this manual.

2) Headings are placed in the left margin so that the reader can easily access necessary information. When locating the necessary information, the reader can save time by searching though these headings.

> Machining programs, parameters, variables, etc. are stored in the CNC unit internal non–volatile memory. In general, these contents are not lost by the switching ON/OFF of the power. However, it is possible that a state can occur where precious data stored in the non–volatile memory has to be deleted, because of deletions from a maloperation, or by a failure restoration. In order to restore rapidly when this kind of mishap occurs, it is recommended that you create a copy of the various kinds of data beforehand.

> This manual describes as many reasonable variations in equipment usage as possible. It cannot address every combination of features, options and commands that should not be attempted.
> If a particular combination of operations is not described, it should not be attempted.

II. PROGRAMMING

1 GENERAL

1.1 TOOL MOVEMENT ALONG WORKPIECE PARTS FIGURE– INTERPOLATION

The tool moves along straight lines and arcs constituting the workpiece parts figure (See II–4).

Explanations

- **Tool movement along a straight line**

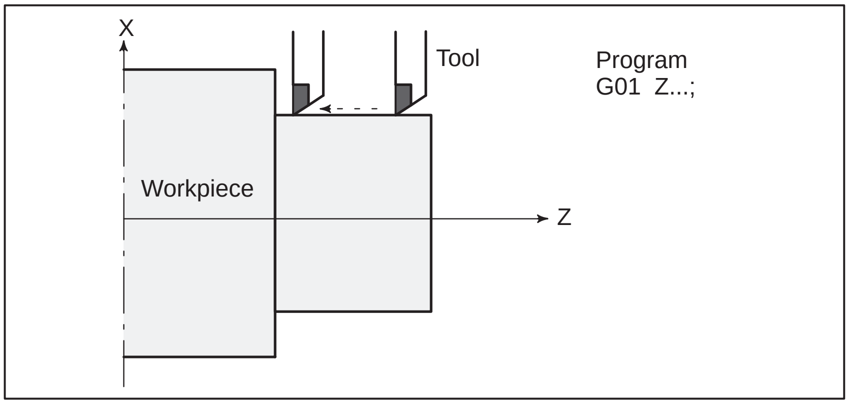


Fig.1.1 (a) Tool movement along the straight line which is parallel to Z–axis

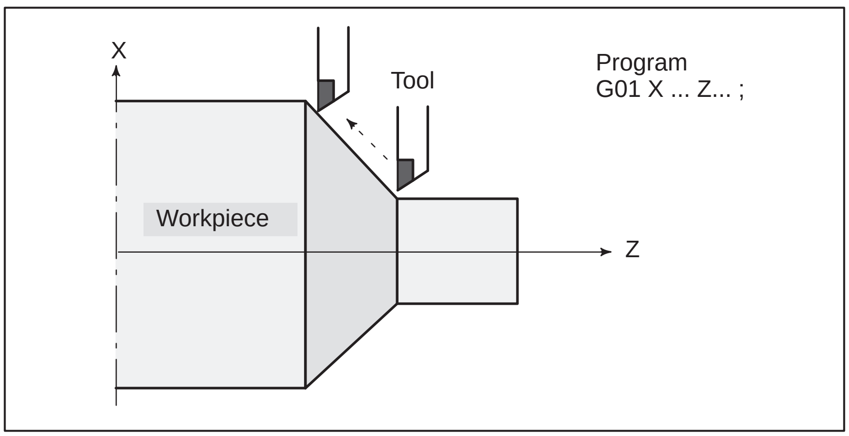


Fig.1.1 (b) Tool movement along the taper line

- **Tool movement along an arc**

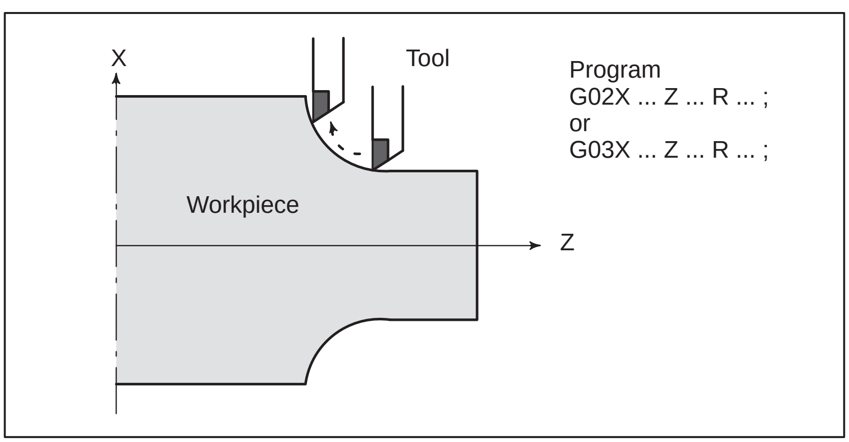


Fig. 1.1 (c) Tool movement along an arc

The term interpolation refers to an operation in which the tool moves along a straight line or arc in the way described above.
Symbols of the programmed commands G01, G02, ... are called the preparatory function and specify the type of interpolation conducted in the control unit.

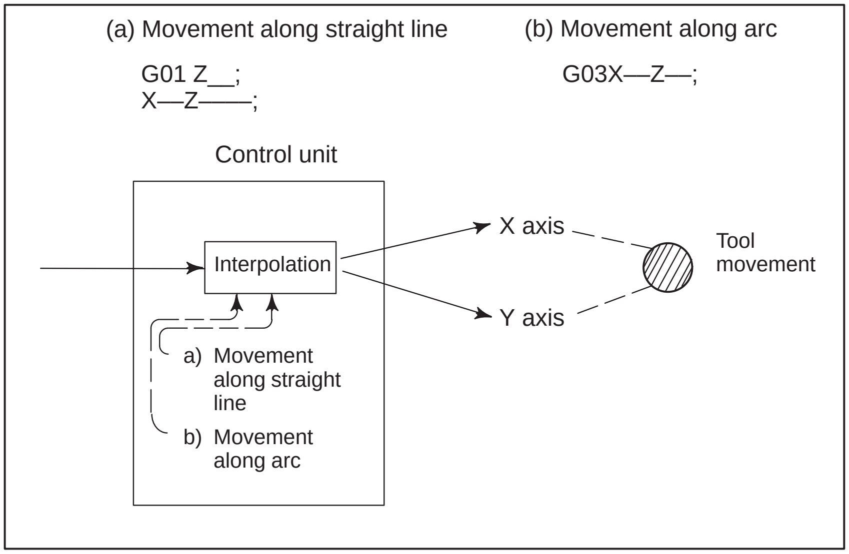


Fig. 1.1 (d) Interpolation function

> **Notes**
> Some machines move tables instead of tools but this manual assumes that tools are moved against workpieces.

- **Thread cutting**

Threads can be cut by moving the tool in synchronization with spindle rotation. In a program, specify the thread cutting function by G32.

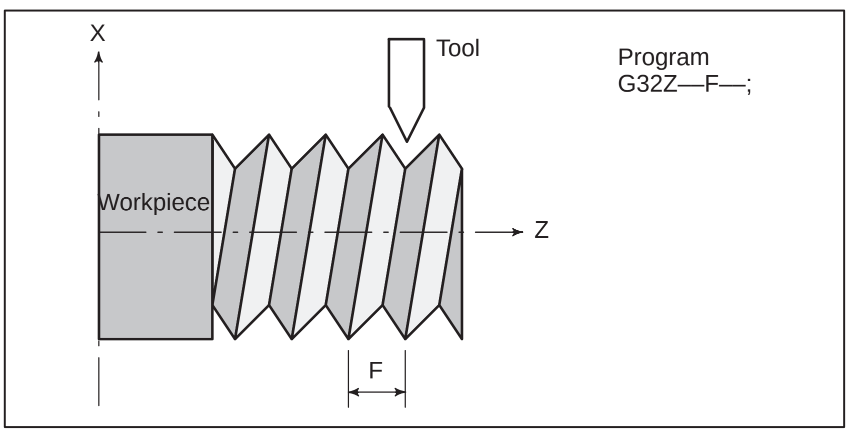


Fig. 1.1 (e) Straight thread cutting

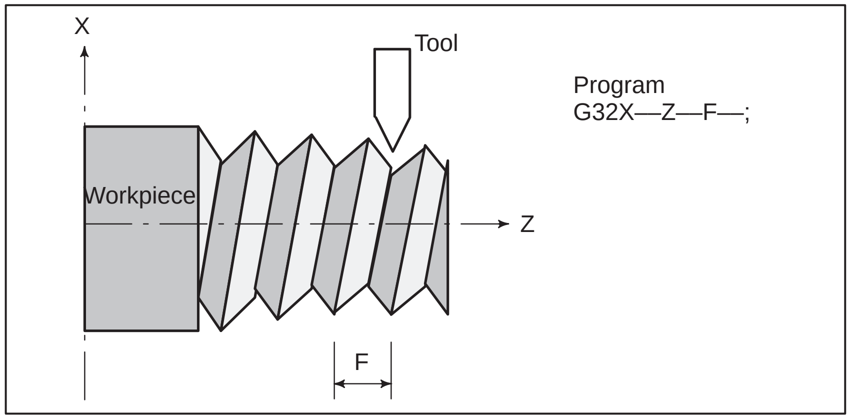


Fig. 1.1 (f) Taper thread cutting

1.2 FEED– FEED FUNCTION

Movement of the tool at a specified speed for cutting a workpiece is called the feed.

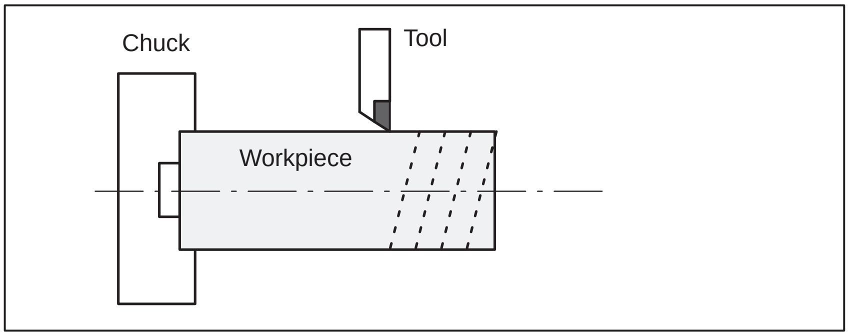


Fig. 1.2 (a) Feed function

Feedrates can be specified by using actual numerics.
For example, the following command can be used to feed the tool 2 mm while the workpiece makes one turn :

F2.0

The function of deciding the feed rate is called the feed function (See II–5).

1.3 PART DRAWING AND TOOL MOVEMENT

1.3.1 Reference Position (Machine–Specific Position)

A CNC machine tool is provided with a fixed position. Normally, tool change and programming of absolute zero point as described later are performed at this position. This position is called the reference position.

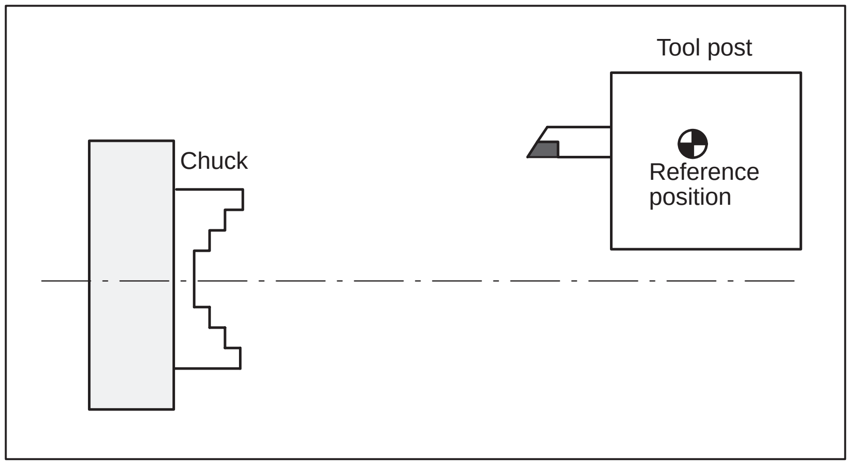


Fig. 1.3.1 (a) Reference position

Explanations

The tool can be moved to the reference position in two ways:

1. Manual reference position return (See III–3.1)
   Reference position return is performed by manual button operation.

2. Automatic reference position return (See II–6)
   In general, manual reference position return is performed first after the power is turned on. In order to move the tool to the reference position for tool change thereafter, the function of automatic reference position return is used.

1.3.2 Coordinate System on Part Drawing and Coordinate System Specified by CNC – Coordinate System

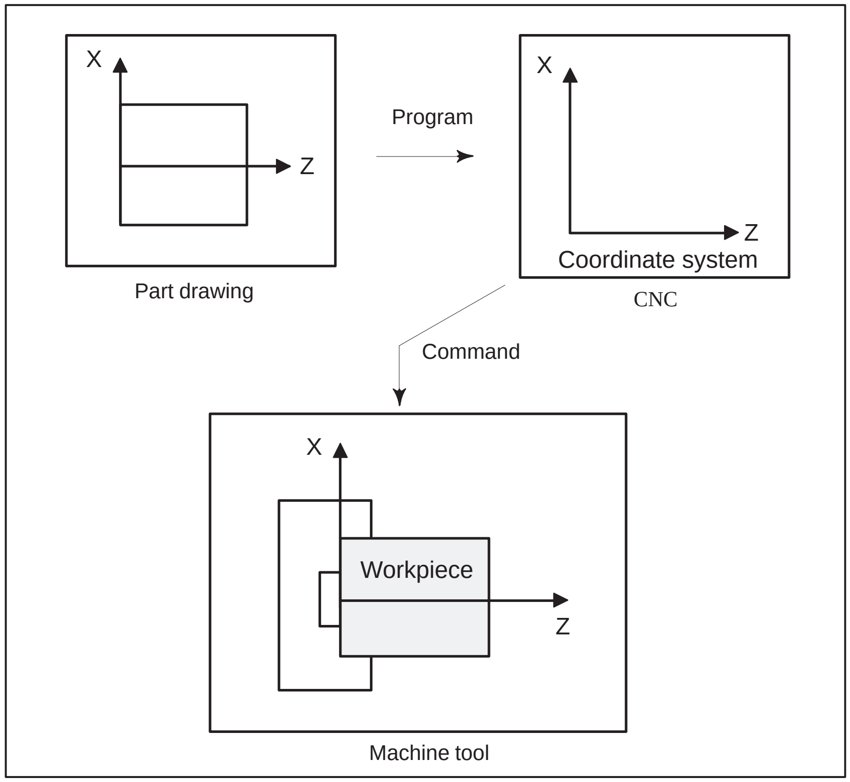


Fig. 1.3.2 (a) Coordinate system

Explanations

● Coordinate system

The following two coordinate systems are specified at different locations: (See II–8)

1. Coordinate system on part drawing
   The coordinate system is written on the part drawing. As the program data, the coordinate values on this coordinate system are used.

2. Coordinate system specified by the CNC
   The coordinate system is prepared on the actual machine tool. This can be achieved by programming the distance from the current position of the tool to the zero point of the coordinate system to be set.

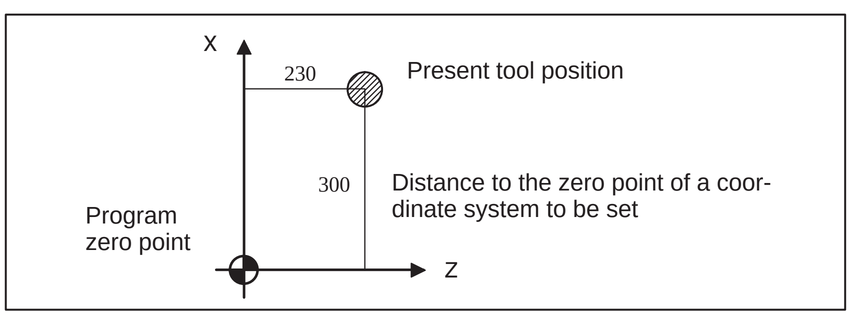


Fig. 1.3.2 (b) Coordinate system specified by the CNC

The tool moves on the coordinate system specified by the CNC in accordance with the command program generated with respect to the coordinate system on the part drawing, and cuts a workpiece into a shape on the drawing.
Therefore, in order to correctly cut the workpiece as specified on the drawing, the two coordinate systems must be set at the same position.

- **Methods of setting the two coordinate systems in the same position**

The following method is usually used to define two coordinate systems at the same location.

1. When coordinate zero point is set at chuck face

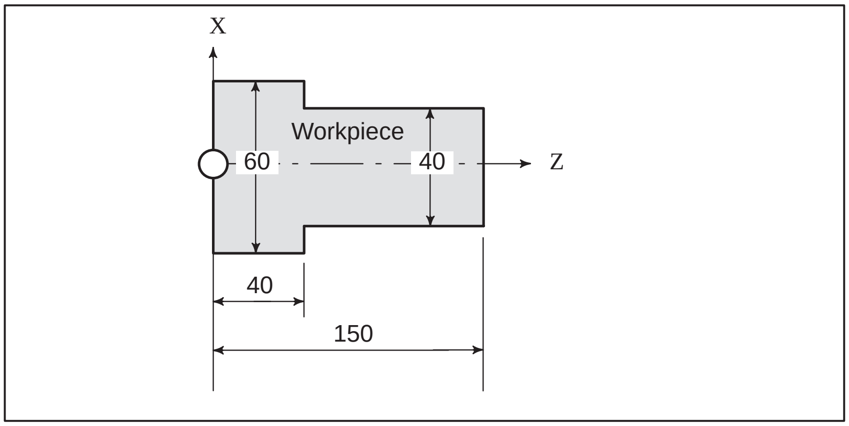


Fig. 1.3.2 (c)Coordinates and dimensions on part drawing

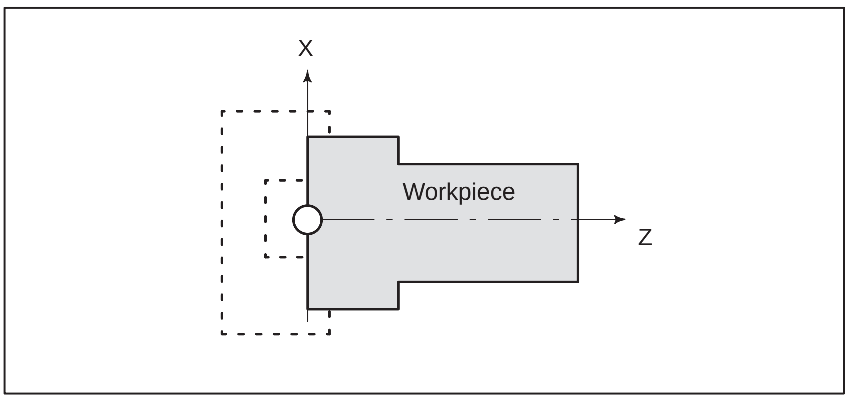


Fig. 1.3.2 (d)Coordinate system on lathe as specified by CNC
(made to coincide with the coordinate system on part drawing)

2. When coordinate zero point is set at work end face.

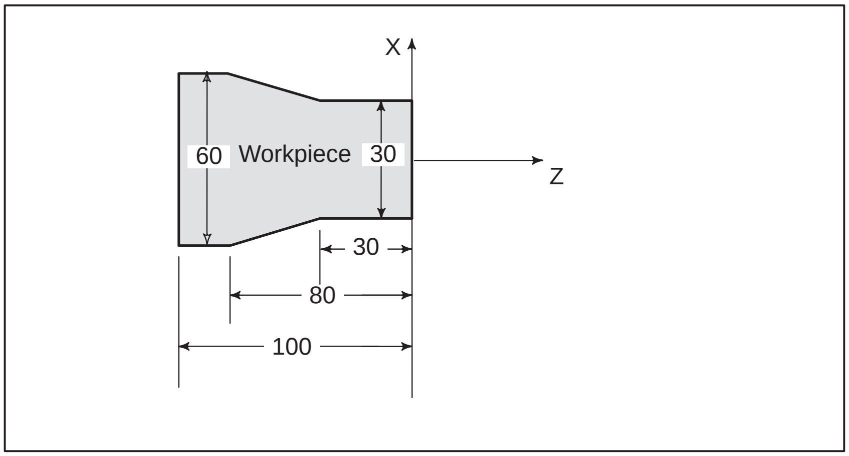


Fig. 1.3.2 (e) Coordinates and dimensions on part drawing

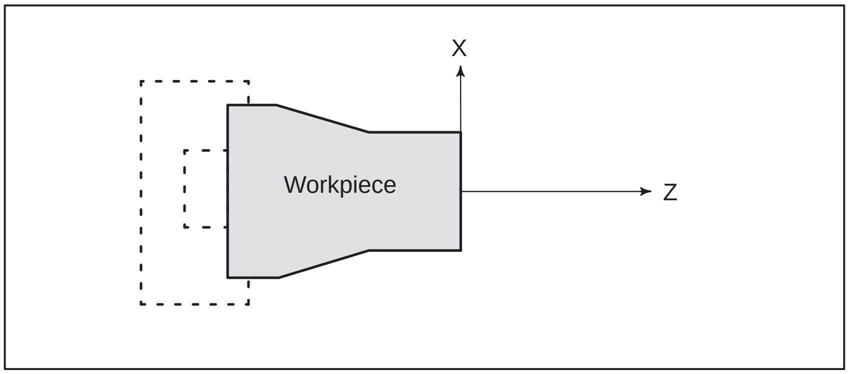


Fig. 1.3.2 (f) Coordinate system on lathe as specified by CNC
(made to coincide with the coordinate system on part drawing)

1.3.3 How to Indicate Command Dimensions for Moving the Tool – Absolute, Incremental Commands

Explanations

Methods of command for moving the tool can be indicated by absolute or incremental designation (See II–9.1).

- **Absolute command**

The tool moves to a point at ”the distance from zero point of the coordinate system” that is to the position of the coordinate values.

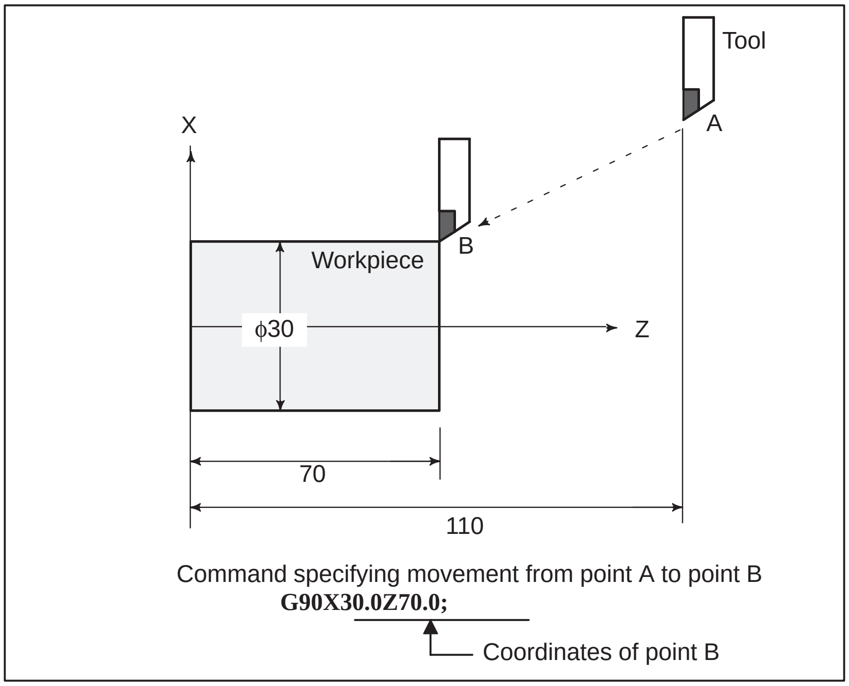


Fig. 1.3.3 (a) Absolute command

- **Incremental command**

Specify the distance from the previous tool position to the next tool position.

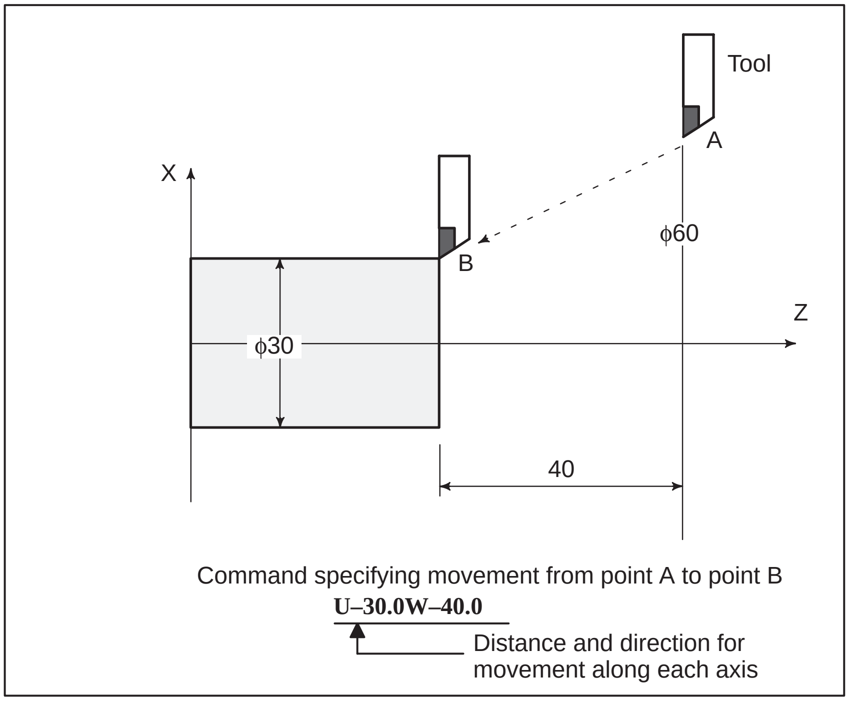


Fig. 1.3.3 (b) Incremental command

- **Diameter programming / radius programming**

Dimensions of the X axis can be set in diameter or in radius. Diameter programming or radius programming is employed independently in each machine.

1. Diameter programming
   In diameter programming, specify the diameter value indicated on the drawing as the value of the X axis.

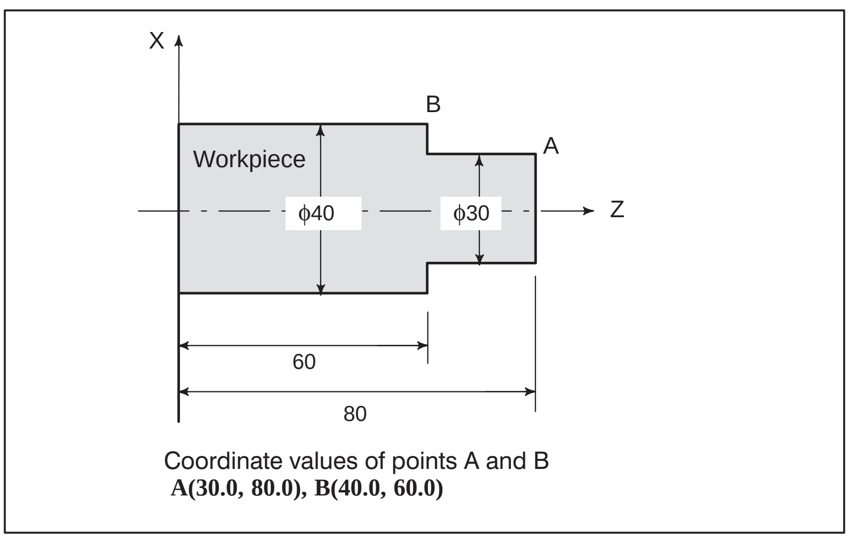
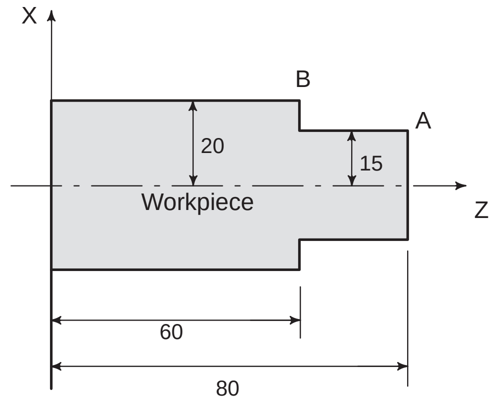


Fig. 1.3.3 (c) Diameter programming

2. Radius programming
   In radius programming, specify the distance from the center of the workpiece, i.e. the radius value as the value of the X axis.



Coordinate values of points A and B
A(15.0, 80.0), B(20.0, 60.0)

Fig. 1.3.3 (d) Radius programming

1.4 CUTTING SPEED – SPINDLE SPEED FUNCTION

The speed of the tool with respect to the workpiece when the workpiece is cut is called the cutting speed.
As for the CNC, the cutting speed can be specified by the spindle speed in rpm unit.

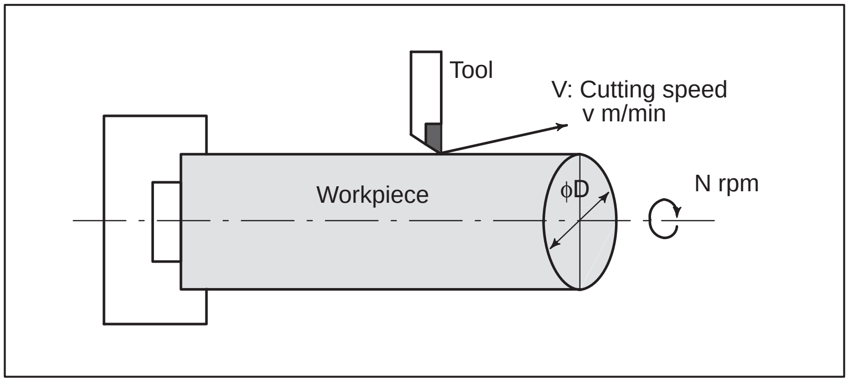


Fig. 1.4 (a) Cutting speed

Examples

<When a workpiece 200 mm in diameter should be machined at a cutting speed of 300 m/min. >

The spindle speed is approximately 478 rpm, which is obtained from $N=1000v/\pi D$. Hence the following command is required:

S478 ;

Commands related to the spindle speed are called the spindle speed function (See II–10).
The cutting speed v (m/min) can also be specified directly by the speed value. Even when the workpiece diameter is changed, the CNC changes the spindle speed so that the cutting speed remains constant.
This function is called the constant surface speed control function (See II–10.2).

1.5 SELECTION OF TOOL USED FOR VARIOUS MACHINING – TOOL FUNCTION

When drilling, tapping, boring, milling or the like, is performed, it is necessary to select a suitable tool. When a number is assigned to each tool and the number is specified in the program, the corresponding tool is selected.

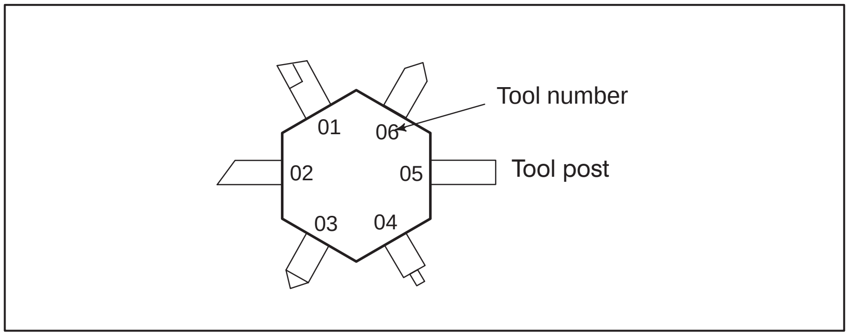


Fig. 1.5 (a) Tool used for various machining

Examples

<When No.01 is assigned to a roughing tool>

When the tool is stored at location 01 of the tool post, the tool can be selected by specifying **T0101**.
This is called the tool function (See II–11).

1.6 COMMAND FOR MACHINE OPERATIONS – MISCELLANEOUS FUNCTION

When machining is actually started, it is necessary to rotate the spindle, and feed coolant. For this purpose, on–off operations of spindle motor and coolant valve should be controlled (See II–12).

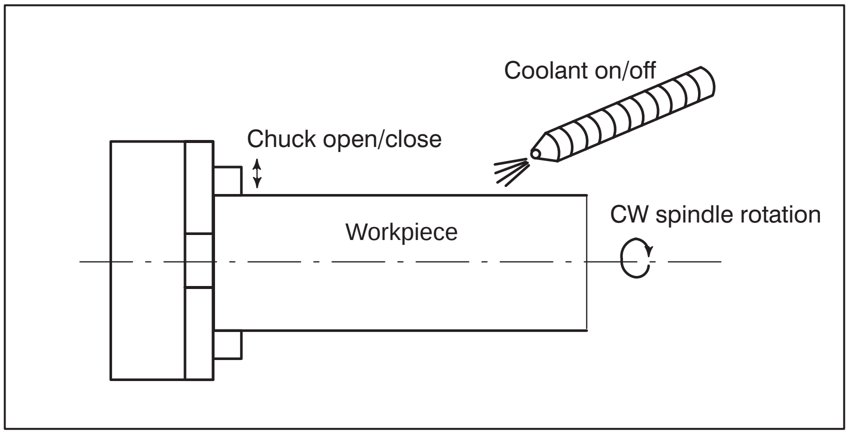


Fig. 1.6 (a) Command for machine operations

The function of specifying the on–off operations of the components of the machine is called the miscellaneous function. In general, the function is specified by an M code.
For example, when M03 is specified, the spindle is rotated clockwise at the specified spindle speed.

1.7 PROGRAM CONFIGURATION

A group of commands given to the CNC for operating the machine is called the program. By specifying the commands, the tool is moved along a straight line or an arc, or the spindle motor is turned on and off.
In the program, specify the commands in the sequence of actual tool movements.

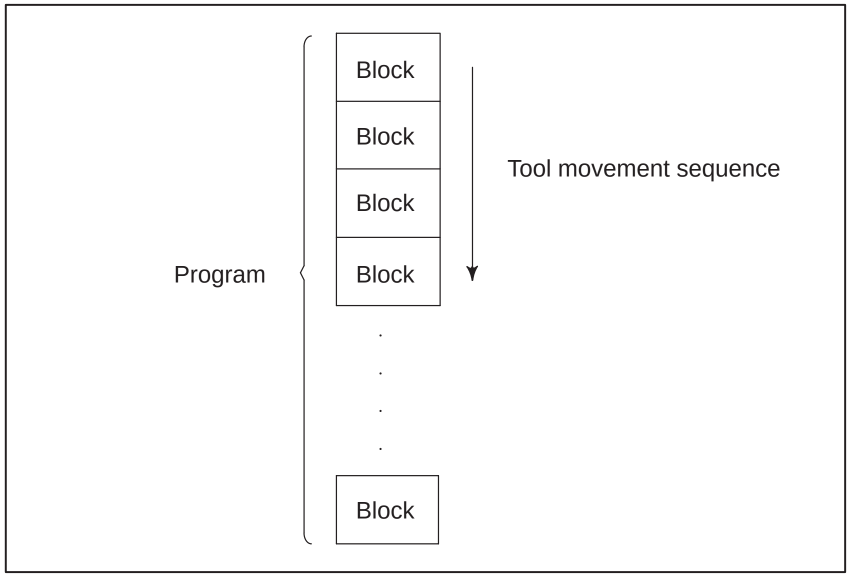


Fig. 1.7 (a) Program configuration

A group of commands at each step of the sequence is called the block. The program consists of a group of blocks for a series of machining. The number for discriminating each block is called the sequence number, and the number for discriminating each program is called the program number (See II–13).

Explanations

The block and the program have the following configurations.

• Block

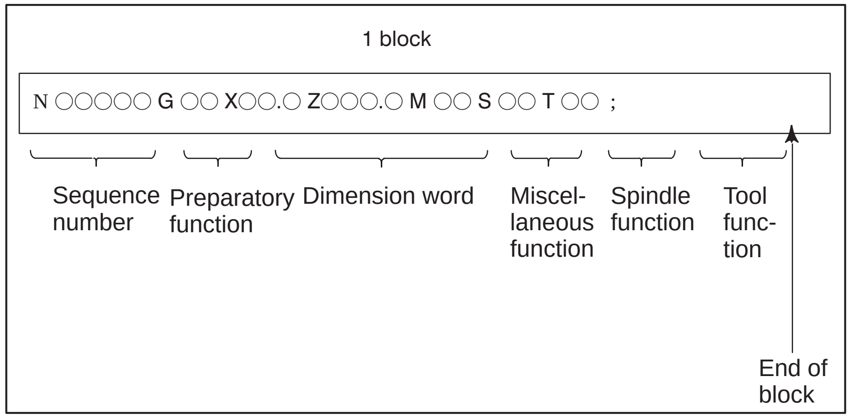


Fig. 1.7 (b) Block configuration

A block begins with a sequence number that identifies that block and ends with an end–of–block code.
This manual indicates the end–of–block code by ; (LF in the ISO code and CR in the EIA code).

• Program

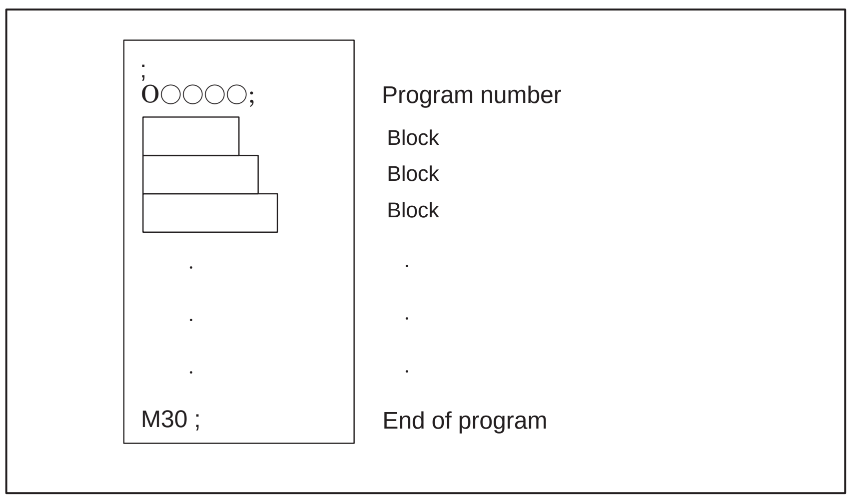
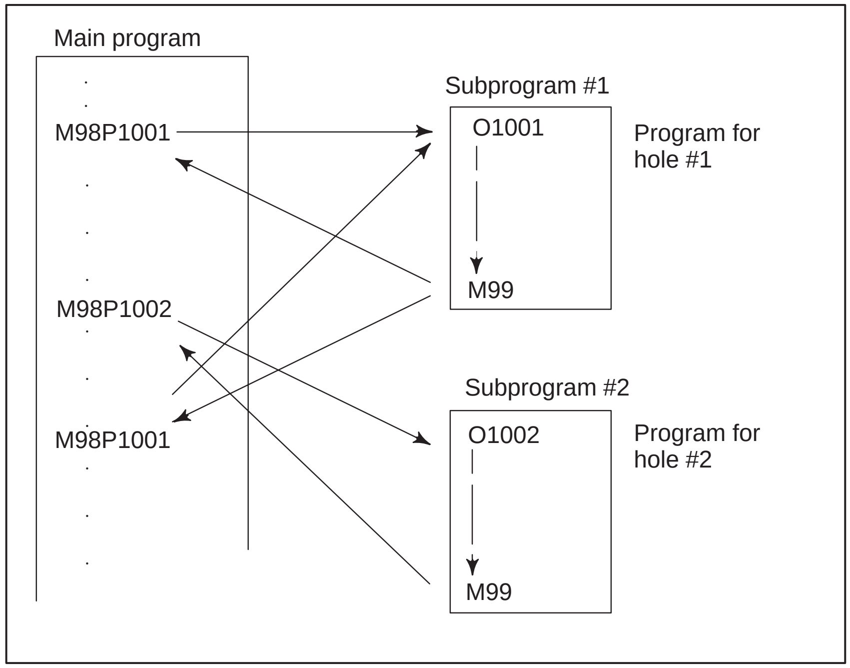


Fig. 1.7 (c) Program configuration

Normally, a program number is specified after the end–of–block (;) code at the beginning of the program, and a program end code (M02 or M30) is specified at the end of the program.

- **Main program and subprogram**

When machining of the same pattern appears at many portions of a program, a program for the pattern is created. This is called the subprogram. On the other hand, the original program is called the main program. When a subprogram execution command appears during execution of the main program, commands of the subprogram are executed. When execution of the subprogram is finished, the sequence returns to the main program.



1.8 TOOL FIGURE AND TOOL MOTION BY PROGRAM

Explanations

- **Machining using the end of cutter – Tool length compensation function (See II–15.1)**

Usually, several tools are used for machining one workpiece. The tools have different tool length. It is very troublesome to change the program in accordance with the tools.

Therefore, the length of each tool used should be measured in advance. By setting the difference between the length of the standard tool and the length of each tool in the CNC (data display and setting : see III–11), machining can be performed without altering the program even when the tool is changed. This function is called tool length compensation.

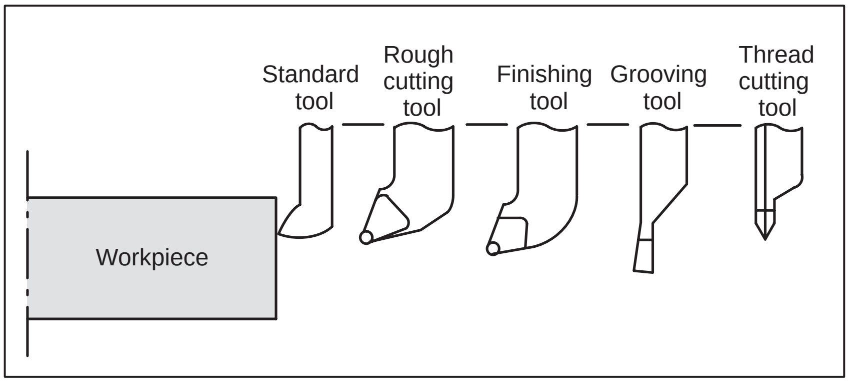
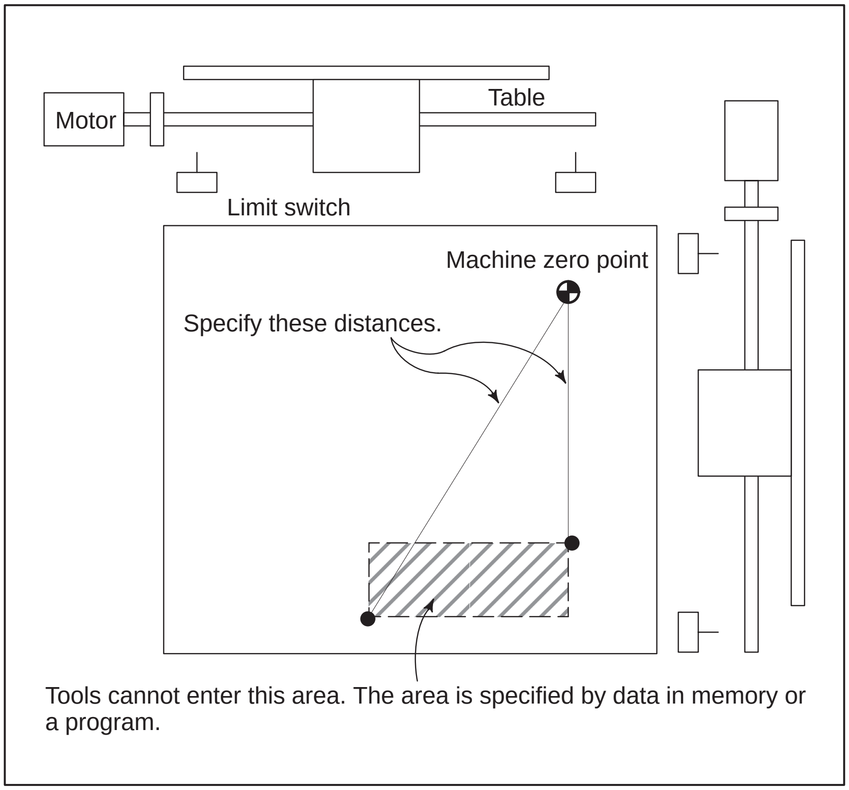


Fig. 1.8 (a) Tool offset

1.9 TOOL MOVEMENT RANGE – STROKE

Limit switches are installed at the ends of each axis on the machine to prevent tools from moving beyond the ends. The range in which tools can move is called the stroke. Besides the stroke limits, data in memory can be used to define an area which tools cannot enter.



Besides strokes defined with limit switches, the operator can define an area which the tool cannot enter using a program or data in memory (see Section III–11). This function is called stroke check.

2 CONTROLLED AXES

2.1 CONTROLLED AXES

Series 16
Series 160

Item	16–TB 160–TB	16–TB, 160–TB (two–path control)
Number of basic controlled axes	2 axes	2 axes for each tool post (4 axes in total)
Controlled axis expansion (total)	Max. 8 axes (Included in Cs axis)	Max. 6 axes for each tool post +Cs axis (Note)
Number of basic simulta-neously controlled axes	2 axes	2 axes for each tool post (4 axes in total)
Simultaneously controlled axis expansion (total)	Max. 6 axes	Max. 4 axes for each tool post

Note
A two–path control system with a 9–inch CRT has up to eight controlled axes.

> **Note**
> The number of simultaneously controllable axes for manual operation (jog feed, incremental feed, or manual handle feed) is 1 or 3 (1 when bit 0 (JAX) of parameter 1002 is set to 0 and 3 when it is set to 1).

Series 18
Series 180

Item	18–TB 180–TB	18–TB, 180–TB (two–path control)
Number of basic controlled axes	2 axes	2 axes for each tool post (4 axes in total)
Controlled axis expansion (total)	Max. 6 axes (Included in Cs axis)	Max. 4 axes for each tool post +Cs axis (Note)
Number of basic simulta-neously controlled axes	2 axes	2 axes for each tool post (4 axes in total)
Simultaneously controlled axis expansion (total)	Max. 4 axes	Max. 4 axes for each tool post

Note
A two–path control system with a 9–inch CRT has up to eight controlled axes.

> **Note**
> The number of simultaneously controllable axes for manual operation (jog feed, incremental feed, or manual handle feed) is 1 or 3 (1 when bit 0 (JAX) of parameter 1002 is set to 0 and 3 when it is set to 1).

2.2 NAMES OF AXES

Nine letters, A, B, C, U, V, W, X, Y, and Z, can be used as axis names. Each axis name is determined according to parameter No. 1020. If the parameter specifies 0 or anything other than the nine letters, the axis name defaults to a number from 1 to 8.
With two–path control, the names of two basic axes for one tool post are always X and Z; the names of additional axes can be optionally selected from A, B, C, U, V, W, and Y by using parameter No. 1020. For one tool post, the same axis name cannot be assigned to multiple axes, but the same axis name can be used
with the other tool post.

Limitations

- **Default axis name**

When a default axis name (1 to 8) is used, the system cannot operate in MEM or MDI mode.

- **Duplicate axis name**

If the parameter specifies an axis name more than once, only the first axis to be assigned that axis name becomes operable.

Notes

1 When G code system A is used, the letters U, V, and W cannot be used as an axis name (hence, the maximum of six controlled axes), because these letters are used as incremental commands for X, Y, and Z. To use the letters U, V, and W as axis names, the G code system must be B or C. Likewise, letter H is used as an incremental command for C, thus incremental commands cannot be used if A or B is used as an axis name.
2 With two–path control, when information (such as the current position) about each axis is displayed on the CRT screen, an axis name may be followed by a subscript to indicate a tool post number (e.g.,X1 and X2). This is axis name to help the user to easily understand which tool post an axis belongs to. When writing a program, the user must specify X, Y, Z, U, V, W, A, B, and C without attaching a subscript.
3 In G76 (multiple–thread cutting), the A address in a block specifies the tool nose angle instead of a command for axis A.
If C or A is used as an axis name, C or A cannot be used as an angle command for a straight line in chamfering or direct drawing dimension programming. Therefore, C and A should be used according to bit 4 (CCR) of parameter No. 3405.

2.3 INCREMENT SYSTEM

The increment system consists of the least input increment (for input) and least command increment (for output). The least input increment is the least increment for programming the travel distance. The least command increment is the least increment for moving the tool on the machine. Both increments are represented in mm, inches, or degrees. The increment system is classified into IS–B and IS–C (Tables 2.3(a) and 2.3(b)). Select IS–B or IS–C using bit 1 (ISC) of parameter 1004. When selecting IS–C, the option for the 1/10 increment system is necessary.

Table 2.3 (a) Increment system IS–B

		Least input increment	Least command increment
Metric system machine	**mm input**	0.001mm(Diameter)	0.0005mm
		0.001mm(Radius)	0.001mm
		0.001deg	0.001deg
	inch input	0.0001inch(Diameter)	0.0005mm
		0.0001inch(Radius)	0.001mm
		0.001deg	0.001deg
Inch machine system	**mm input**	0.001mm(Diameter)	0.00005inch
		0.001mm(Radius)	0.0001inch
		0.001deg	0.001deg
	inch input	0.0001inch(Diameter)	0.00005inch
		0.0001inch(Radius)	0.0001inch
		0.001deg	0.001deg

Table 2.3 (b) Increment system IS–C

		Least input increment	Least command increment
Metric system machine	**mm input**	0.0001mm(Diameter)	0.00005mm
		0.0001mm(Radius)	0.0001mm
		0.0001deg	0.0001deg
	inch input	0.00001inch(Diameter)	0.00005mm
		0.00001inch(Radius)	0.0001mm
		0.0001deg	0.0001deg
Inch machine system	**mm input**	0.0001mm(Diameter)	0.000005inch
		0.0001mm(Radius)	0.00001inch
		0.0001deg	0.0001deg
	inch input	0.00001inch(Diameter)	0.000005inch
		0.00001inch(Radius)	0.00001inch
		0.0001deg	0.0001deg

2.4 MAXIMUM STROKES

The maximum stroke controlled by this CNC is shown in the table below:
Maximum stroke=Least command increment±99999999

Table 2.4 (a) Maximum strokes

Increment system		Maximum strokes
IS–B	Metric machine system	±99999.999 mm ±99999.999 deg
	Inch machine system	±9999.9999 inch ±99999.999 deg
IS–C	Metric machine system	±9999.9999 mm ±9999.9999 deg
	Inch machine system	±999.99999 inch ±9999.9999 deg

Notes
1. The unit in the table is a diameter value with diameter programming and a radius value in radius programming.
2. A command exceeding the maximum stroke cannot be specified.
3. The actual stroke depends on the machine tool.

3 PREPARATORY FUNCTION (G FUNCTION)

A number following address G determines the meaning of the command for the concerned block.
G codes are divided into the following two types.

Type	Meaning
One–shot G code	The G code is effective only in the block in which it is specified
Modal G code	The G code is effective until another G code of the same group is specified.

(Example)
G01 and G00 are modal G codes.

```
G01X_;
   Z_;     G01 is effective in this range
   X_;
G00Z_;
```

There are three G code systems : A,B, and C (Table 3). Select a G code system using bits 6 (GSB) and 7 (GSC) of parameter 3401. Generally, this manual describes the use of G code system A, except when the described item can use only G code system B or C. ln such cases, the use of G code system B or C is described.

Explanations

1. If the CNC enters the clear state (see bit 6 (CLR) of parameter 3402) when the power is turned on or the CNC is reset, the modal G codes change as follows.
   (1) G codes marked with ◤ in Table 3 are enabled.
   (2) When the system is cleared due to power–on or reset, whichever specified, either G20 or G21, remains effective.
   (3) Bit 7 of parameter No. 3402 can be used to specify whether G22 or G23 is selected upon power–on. Resetting the CNC to the clear state does not affect the selection of G22 or G23.
   (4) Setting bit 0 (G01) of parameter 3402 determines which code, either G00 or G01, is effective.
   (5) Setting bit 3 (G91) of parameter 3402 determines which code, either G90 or G91, is effective.
2. G codes of group 00 except G10 and G11 are single–shot G codes.
3. P/S larm (No.010) is displayed when a G code not listed in the G code list is specified or a G code without a corresponding option is specified.
4. G codes of different groups can be specified in the same block. If G codes of the same group are specified in the same block, the G code specified last is valid.
5. If a G code of group 01 is specified in a canned cycle, the canned cycle is canceled in the same way as when a G80 command is specified. G codes of group 01 are not affected by G codes for specifying a canned cycle.
6. When G code system A is used for a canned cycle, only the initial level is provided at the return point.
7. G codes are displayed for each group number.

Table 3 G code list (1/3)

G code A	G code B	G code C	Group	Function
G00	G00	G00	01	Positioning (Rapid traverse)
G01	G01	G01		Linear interpolation (Cutting feed)
G02	G02	G02		Circular interpolation CW or Helical interpolation CW
G03	G03	G03		Circular interpolation CCW or Helical interpolation CCW
G04	G04	G04	00	Dwell
G05	G05	G05		High speed cycle cutting
G07.1 (G107)	G07.1 (G107)	G07.1 (G107)		Cylindrical interpolation
G10	G10	G10		Programmable data input
G10.6	G10.6	G10.6		Tool retract & recover
G11	G11	G11		Programmable data input cancel
G12.1 (G112)	G12.1 (G112)	G12.1 (G112)	21	Polar coordinate interpolation mode
G13.1 (G113)	G13.1 (G113)	G13.1 (G113)		Polar coordinate interpolation cancel mode
G17	G17	G17	16	XpYp plane selection
G18	G18	G18		ZpXp plane selection
G19	G19	G19		YpZp plane selection
G20	G20	G70	06	Input in inch
G21	G21	G71		Input in mm
G22	G22	G22	09	Stored stroke check function on
G23	G23	G23		Stored stroke check function off
G25	G25	G25	08	Spindle speed fluctuation detection off
G26	G26	G26		Spindle speed fluctuation detection on
G27	G27	G27	00	Reference position return check
G28	G28	G28		Return to reference position
G30	G30	G30		2nd, 3rd and 4th reference position return
G30.1	G30.1	G30.1		Floating reference point return
G31	G31	G31		Skip function
G32	G33	G33	01	Thread cutting
G34	G34	G34		Variable–lead thread cutting
G36	G36	G36	00	Automatic tool compensation X
G37	G37	G37		Automatic tool compensation Z
G39	G39	G39		Corner circular interpolation
G40	G40	G40	07	Tool nose radius compensation cancel
G41	G41	G41		Tool nose radius compensation left
G42	G42	G42		Tool nose radius compensation right
G50	G92	G92	00	Coordinate system setting or max. spindle speed setting
G50.3	G92.1	G92.1		Workpiece coordinate system preset

Table 3 G code list (2/3)

G code A	G code B	G code C	Group	Function
G50.2 (G250)	G50.2 (G250)	G50.2 (G250)	20	Polygonal turning cancel
G51.2 (G251)	G51.2 (G251)	G51.2 (G251)		Polygonal turning
G52	G52	G52	00	Local coordinate system setting
G53	G53	G53		Machine coordinate system setting
G54	G54	G54	14	Workpiece coordinate system 1 selection
G55	G55	G55		Workpiece coordinate system 2 selection
G56	G56	G56		Workpiece coordinate system 3 selection
G57	G57	G57		Workpiece coordinate system 4 selection
G58	G58	G58		Workpiece coordinate system 5 selection
G59	G59	G59		Workpiece coordinate system 6 selection
G65	G65	G65	00	Macro calling
G66	G66	G66	12	Macro modal call
G67	G67	G67		Macro modal call cancel
G68	G68	G68	04	Mirror image for double turrets ON or balance cut mode
G69	G69	G69		Mirror image for double turrets OFF or balance cut mode cancel
G70	G70	G72	00	Finishing cycle
G71	G71	G73		Stock removal in turning
G72	G72	G74		Stock removal in facing
G73	G73	G75		Pattern repeating
G74	G74	G76		End face peck drilling
G75	G75	G77		Outer diameter/internal diameter drilling
G76	G76	G78		Multiple threading cycle
G71	G71	G72	01	Traverse grinding cycle (for grinding machine)
G72	G72	G73		Traverse direct constant–dimension grinding cycle (for grinding machine)
G73	G73	G74		Oscilation grinding cycle (for grinding machine)
G74	G74	G75		Oscilation direct constant–dimension grinding cycle (for grinding machine)
G80	G80	G80	10	Canned cycle for drilling cancel
G83	G83	G83		Cycle for face drilling
G84	G84	G84		Cycle for face tapping
G86	G86	G86		Cycle for face boring
G87	G87	G87		Cycle for side drilling
G88	G88	G88		Cycle for side tapping
G89	G89	G89		Cycle for side boring
G90	G77	G20	01	Outer diameter/internal diameter cutting cycle
G92	G78	G21		Thread cutting cycle
G94	G79	G24		Endface turning cycle
G96	G96	G96	02	Constant surface speed control
G97	G97	G97		Constant surface speed control cancel

Table 3 G code list (3/3)

G code			Group	Function
A	**B**	**C**		
G98	G94	G94	05	Per minute feed
G99	G95	G95		Per revolution feed
–	G90	G90	03	Absolute programming
–	G91	G91		Incremental programming
–	G98	G98	11	Return to initial level (See **Explanations 6**)
–	G99	G99		Return to R point level (See **Explanations 6**)

4 INTERPOLATION FUNCTIONS

4.1 POSITIONING (G00)

The G00 command moves a tool to the position in the workpiece system specified with an absolute or an incremental command at a rapid traverse rate.
In the absolute command, coordinate value of the end point is programmed.
In the incremental command the distance the tool moves is programmed.

Format

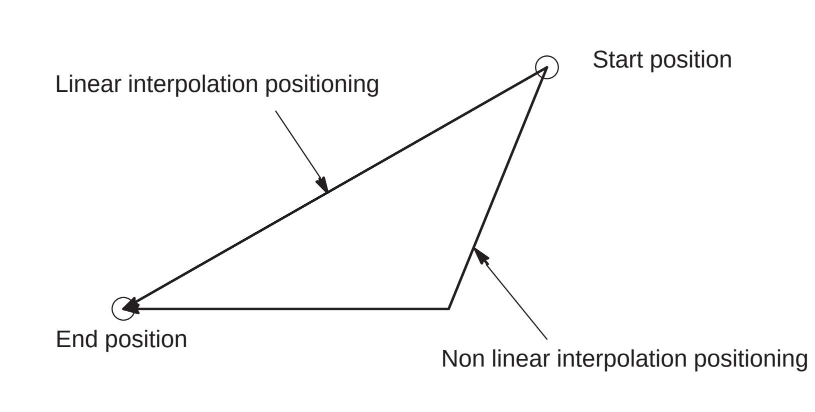
G00IP_;

IP_: For an absolute command, the coordinates of an end position, and for an incremental command, the distance the tool moves.

Explanations

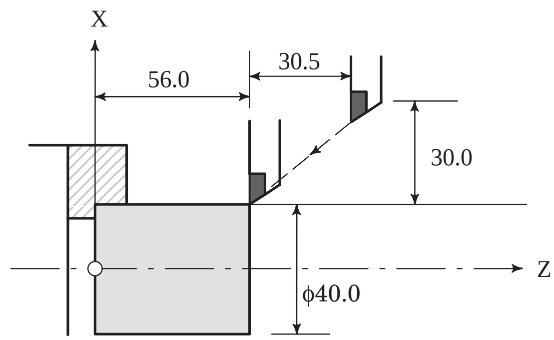
Either of the following tool paths can be selected according to bit 1 (LRP) of parameter No. 1401.

- **Nonlinear interpolation positioning**
  The tool is positioned with the rapid traverse rate for each axis separately. The tool path is normally straight.
- **Linear interpolation positioning**
  The tool path is the same as in linear interpolation (G01). The tool is positioned within the shortest possible time at a speed that is not more than the rapid traverse rate for each axis.



The rapid traverse rate in the G00 command is set to the parameter No.1420 for each axis independently by the machine tool builder. In the positioning mode actuated by G00, the tool is accelerated to a predetermined speed at the start of a block and is decelerated at the end of a block. Execution proceeds to the next block after confirming the in–position.
”In–position” means that the feed motor is within the specified range.
This range is determined by the machine tool builder by setting to parameter No.1826.

Examples



< Radius programming >
G00X40.0Z56.0 ; (Absolute command)
or
G00U–60.0W–30.5;(Incremental command)

Restrictions

The rapid traverse rate cannot be specified in the address F.
Even if linear interpolation positioning is specified, nonlinear interpolation positioning is used in the following cases. Therefore, be careful to ensure that the tool does not foul the workpiece.

- G28 specifying positioning between the reference and intermediate positions.
- G53

4.2 LINEAR INTERPOLATION (G01)

Tools can move along a line.

Format

G01 IP_F_;

IP_: **For an absolute command, the coordinates of an end point , and for an incremental command, the distance the tool moves.**

F_: **Speed of tool feed (Feedrate)**

Explanations

A tools move along a line to the specified position at the feedrate specified in F.
The feedrate specified in F is effective until a new value is specified. It need not be specified for each block.
The feedrate commanded by the F code is measured along the tool path. If the F code is not commanded, the feedrate is regarded as zero.
For feed–per–minute mode under 2–axis simultaneous control, the feedrate for a movement along each axis as follows :

G01$\alpha\underline{\alpha}\beta\underline{\beta}$ $F\underline{f}$ **;**

Feed rate of α axis direction : $F\alpha = \frac{\alpha}{L} \times f$

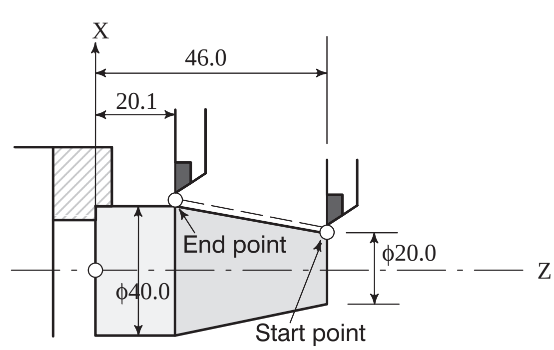
Feed rate of B axis direction : $F_\beta = \frac{\beta}{L} \times f$

$$L = \sqrt{\alpha^2 + \beta^2 + \gamma^2 + \zeta^2}$$

Examples

- **Linear interpolation**

< Diameter programming >
G01X40.0Z20.1F20 ; (Absolute command)
or
G01U20.0W–25.9F20 ; (Incremental command)



4.3 CIRCULAR INTERPOLATION (G02,G03)

The command below will move a tool along a circular arc.

Format

Arc in the XpYp plane

$$G17 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} Xp\_Yp\_ \begin{Bmatrix} I\_J\_ \\ R\_ \end{Bmatrix} F\_$$

Arc in the ZpXp plane

$$G18 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} Xp\_Zp\_ \begin{Bmatrix} I\_K\_ \\ R\_ \end{Bmatrix} F\_$$

Arc in the YpZp plane

$$G19 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} Yp\_Zp\_ \begin{Bmatrix} J\_K\_ \\ R\_ \end{Bmatrix} F\_$$

Table.4.3 Description of the Command Format

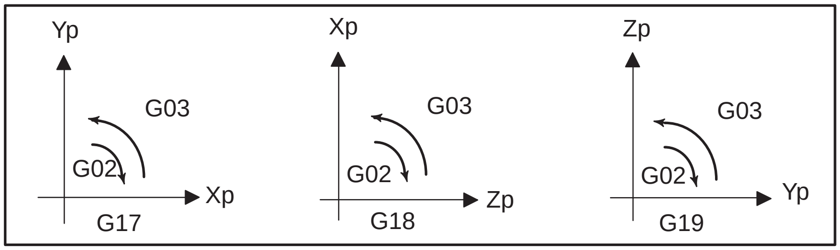
Command	Description
G17	Specification of arc on XpYp plane
G18	Specification of arc on ZpXp plane
G19	Specification of arc on YpZp plane
G02	Circular Interpolation Clockwise direction (CW)
G03	Circular Interpolation Counterclockwise direction (CCW)
$X_{p\_}$	Command values of X axis or its parallel axis (set by parameter No. 1022)
$Y_{p\_}$	Command values of Y axis or its parallel axis (set by parameter No. 1022)
$Z_{p\_}$	Command values of Z axis or its parallel axis (set by parameter No. 1022)
I_	X_p axis distance from the start point to the center of an arc with sign, radius value
J_	Y_p axis distance from the start point to the center of an arc with sign, radius value
k_	Z_p axis distance from the start point to the center of an arc with sign, radius value
R_	Arc radius with no sign (always with radius value)
F_	Feedrate along the arc

> **Note**
> The U–, V–, and W–axes (parallel with the basic axis) can be used with G–codes B and C.

Explanations

- **Direction of the circular interpolation**

"Clockwise"(G02) and "counterclockwise"(G03) on the X_pY_p plane (Z_pX_p plane or Y_pZ_p plane) are defined when the X_pY_p plane is viewed in the positive–to–negative direction of the Z_p axis (Y_p axis or X_p axis, respectively) in the Cartesian coordinate system. See the figure below.

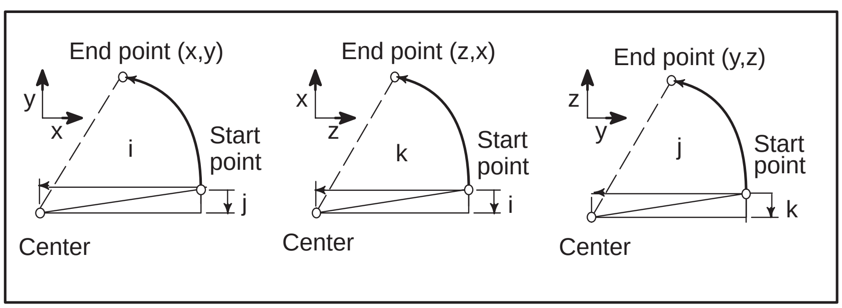


- **Distance moved on an arc**

The end point of an arc is specified by address Xp, Yp or Zp, and is expressed as an absolute or incremental value according to G90 or G91. For the incremental value, the distance of the end point which is viewed from the start point of the arc is specified.

- **Distance from the start point to the center of arc**

The arc center is specified by addresses I, J, and K for the Xp, Yp, and Zp axes, respectively. The numerical value following I, J, or K, however, is a vector component in which the arc center is seen from the start point, and is always specified as an incremental value irrespective of G90 and G91, as shown below.
I, J, and K must be signed according to the direction.



I0,J0, and K0 can be omitted.
If the difference between the radius at the start point and that at the end point exceeds the value in a parameter (No.3410), an P/S alarm (No.020) occurs.

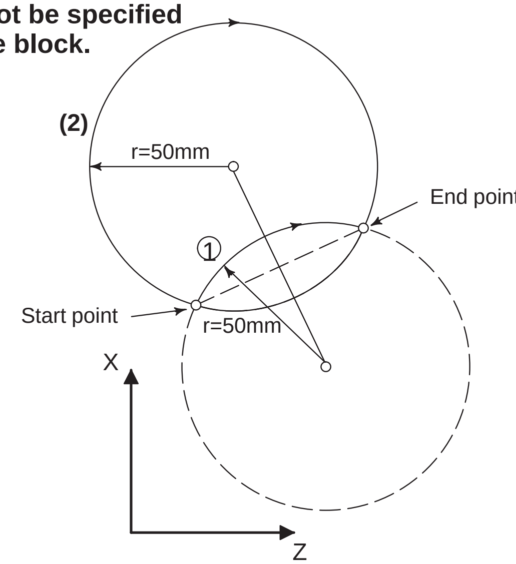
- **Full–circle programming**

When X_p, Y_p , and Z_p are omitted (the end point is the same as the start point) and the center is specified with I, J, and K, a 360° arc (circle) is specified.

- **Arc radius**

The distance between an arc and the center of a circle that contains the arc can be specified using the radius, R, of the circle instead of I, J, and K.
In this case, one arc is less than 180°, and the other is more than 180° are considered. An arc with a sector angle of 180°or wider cannot be specified. If Xp, Yp, and Zp are all omitted, if the end point is located at the same position as the start point and when R is used, an arc of 0°is programmed
G02R ; (The cutter does not move.)

For arc (1) (less than 180°)
G02 W60.0 U10.0 R50.0 F300.0 ;
For arc (2) (greater than 180°)
An arc with a sector angle of 180° or wider cannot be specified within a single block.



- **Feedrate**

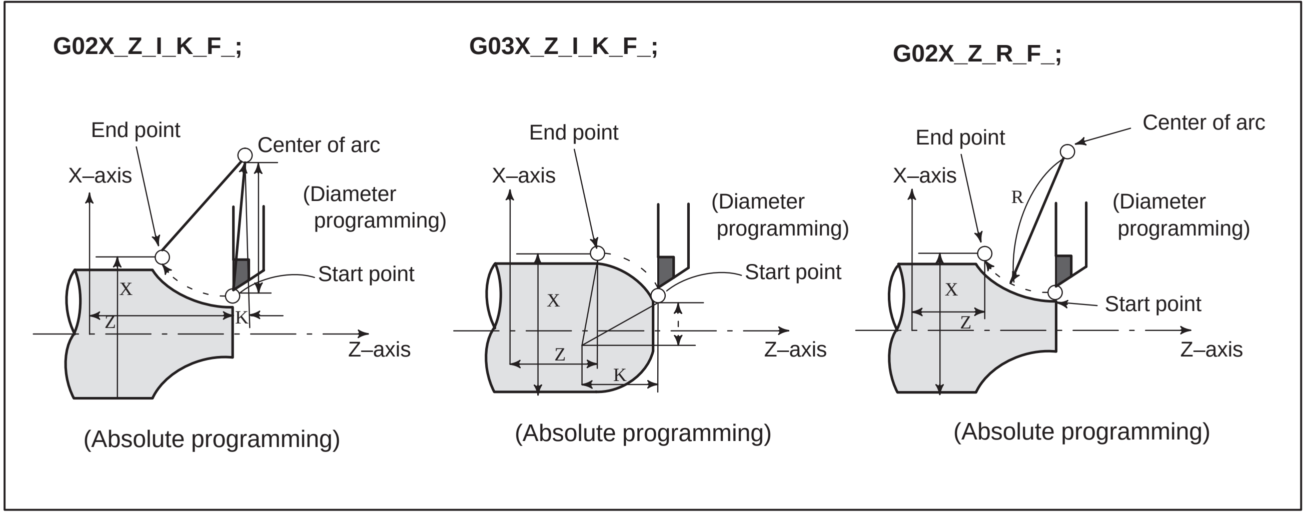
The feedrate in circular interpolation is equal to the feed rate specified by the F code, and the feedrate along the arc (the tangential feedrate of the arc) is controlled to be the specified feedrate.
The error between the specified feedrate and the actual tool feedrate is ±2% or less. However, this feed rate is measured along the arc after the tool nose radius compensation is applied

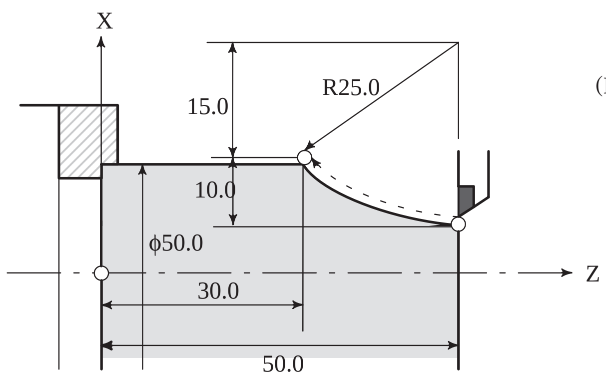
Restrictions

If I, J, K, and R addresses are specified simultaneously, the arc specified by address R takes precedence and the other are ignored.
If an axis not comprising the specified plane is commanded, an alarm is displayed.
For example, when a ZX plane is specified in G–code B or C, specifying the X–axis or U–axis (parallel to the X–axis) causes P/S alarm No. 028 to be generated.
If the difference in the radius between the start and end points of the arc exceeds the value specified in parameter No. 3410, P/S alarm No. 020 is generated.
If the end point is not on the arc, the tool moves in a straight line along one of the axes after reaching the end point.
If an arc having a central angle approaching 180 is specified with R, the calculation of the center coordinates may produce an error. In such a case, specify the center of the arc with I, J, and K.

Examples

- **Command of circular interpolation X, Z**





(Diameter programming)
G02X50.0Z30.0I25.0F0.3; or
G02U20.0W–020.0I25.0F0.3; or
G02X50.0Z30.0R25.0F0.3 or
G02U20.0W–20.0R25.F0.3;

4.4 HELICAL INTERPOLATION (G02,G03)

Helical interpolation which moved helically is enabled by specifying up to two other axes which move synchronously with the circular interpolation by circular commands.

Format

Synchronously with arc of XpYp plane

$$G17 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} Xp\_Yp\_ \begin{Bmatrix} I\_J\_ \\ R\_ \end{Bmatrix} A\_(B\_)F\_;$$

Synchronously with arc of ZpXp plane

$$G18 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} Xp\_Zp\_ \begin{Bmatrix} I\_K\_ \\ R\_ \end{Bmatrix} \alpha\_(\beta\_)F\_;$$

Synchronously with arc of YpZp plane

$$G19 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} Yp\_Zp\_ \begin{Bmatrix} J\_K\_ \\ R\_ \end{Bmatrix} A\_(B\_)F\_;$$

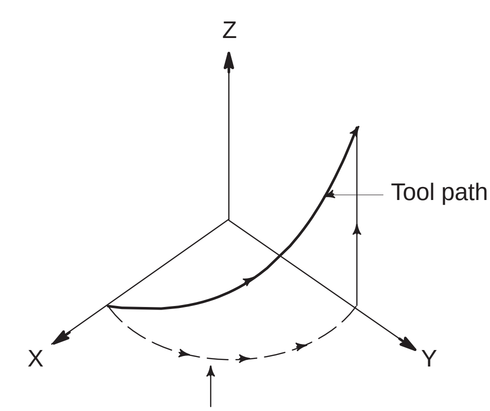
α,β:Any one axis where circular interpolation is not applied. Up to two other axes can be specified.

Explanations

The command method is to simply or secondary add a move command axis which is not circular interpolation axes. An F command specifies a feed rate along a circular arc. Therefore, the feed rate of the linear axis is as follows:

$$F \times \frac{\text{Length of linear axis}}{\text{Length of circular arc}}$$

Determine the feed rate so the linear axis feed rate does not exceed any of the various limit values. Bit 0 (HFC) of parameter No. 1404 can be used to prevent the linear axis feedrate from exceeding various limit values.



The feedrate along the circumference of two circular interpolated axes is the specified feedrate.

Limitations

- Cutter compensation is applied only for a circular arc.
- Tool offset and tool length compensation cannot be used in a block in which a helical interpolation is commanded.

4.5 POLAR COORDINATE INTERPOLATION (G12.1,G13.1)

Polar coordinate interpolation is a function that exercises contour control in converting a command programmed in a Cartesian coordinate system to the movement of a linear axis (movement of a tool) and the movement of a rotary axis (rotation of a workpiece). This method is useful in cutting a front surface and grinding a cam shaft on a lathe.

Format

- ·Specify G12.1 and G13.1 in Separate Blocks.

G12.1 ;	Starts polar coordinate interpolation mode (enables polar coordinate interpolation)
	Specify linear or circular interpolation using coordinates in a Cartesian coordinate system consisting of a linear axis and rotary axis (virtual axis).
G13.1 ;	Polar coordinate interpolation mode is cancelled (for not performing polar coordinate interpolation)
	G112 and G113 can be used in place of G12.1 and G13.1, respectively.

Explanations

- **Polar coordinate interpolation plane**

G12.1 starts the polar coordinate interpolation mode and selects a polar coordinate interpolation plane (Fig. 4.6 (a)). Polar coordinate interpolation is performed on this plane.

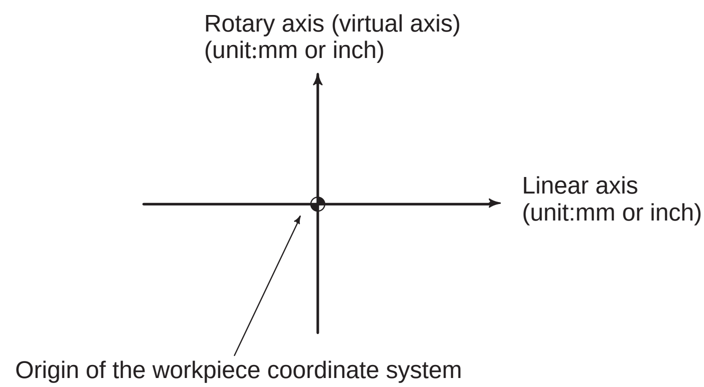


Fig4.5 (a) Polar coordinate interpolation plane.

When the power is turned on or the system is reset, polar coordinate interpolation is canceled (G13.1).
The linear and rotation axes for polar coordinate interpolation must be set in parameters (No. 5460 and 5461) beforehand.

Note

The plane used before G12.1 is specified (plane selected by G17, G18, or G19) is canceled. It is restored when G13.1 (canceling polar coordinate interpolation) is specified.
When the system is reset, polar coordinate interpolation is canceled and the plane specified by G17, G18, or G19 is used.

- **Distance moved and feedrate for polar coordinate interpolation**

  The unit for coordinates on the hypothetical axis is the same as the unit for the linear axis (mm/inch)

  The unit for the feedrate is mm/min or inch/min

In the polar coordinate interpolation mode, program commands are specified with Cartesian coordinates on the polar coordinate interpolation plane. The axis address for the rotation axis is used as the axis address for the second axis (virtual axis) in the plane. Whether a diameter or radius is specified for the first axis in the plane is the same as for the rotation axis regardless of the specification for the first axis in the plane. The virtual axis is at coordinate 0 immediately after G12.1 is specified. Polar interpolation is started assuming the angle of 0 for the position of the tool when G12.1 is specified.
Specify the feedrate as a speed (relative speed between the workpiece and tool) tangential to the polar coordinate interpolation plane (Cartesian coordinate system) using F.

- **G codes which can be specified in the polar coordinate interpolation mode**

G01 Linear interpolation
G02, G03 Circular interpolation
G04 Dwell
G40, G41, G42 Tool nose radius compensation
(Polar coordinate interpolation is applied to the path after cutter compensation.)
G65, G66, G67 Custom macro command
G98, G99 Feed per minute, feed per revolution

- **Circular interpolation in the polar coordinate plane**

The addresses for specifying the radius of an arc for circular interpolation (G02 or G03) in the polar coordinate interpolation plane depend on the first axis in the plane (linear axis).

· I and J in the Xp–Yp plane when the linear axis is the X–axis or an axis parallel to the X–axis.
· J and K in the Yp–Zp plane when the linear axis is Y–axis or an axis parallel to the Y–axis.
· K and I in the Zp–Xp plane when the linear axis is the Z–axis or an axis parallel to the Z–axis.

The radius of an arc can be specified also with an R command.

> **Note**
> The U–, V–, and W–axes (parallel with the basic axis) can be used with G–codes B and C.

- **Movement along axes not in the polar coordinate interpolation plane in the polar coordinate interpolation mode**

The tool moves along such axes normally, independent of polar coordinate interpolation.

- **Current position display in the polar coordinate interpolation mode**

Actual coordinates are displayed. However, the remaining distance to move in a block is displayed based on the coordinates in the polar coordinate interpolation plane (Cartesian coordinates).

Restrictions

- **Coordinate system for the polar coordinate interpolation**

Before G12.1 is specified, a workpiece coordinate system) where the center of the rotary axis is the origin of the coordinate system must be set. In the G12.1 mode, the coordinate system must not be changed (G92, G52, G53, relative coordinate reset, G54 through G59, etc.).

- **Tool nose radius compensation command**

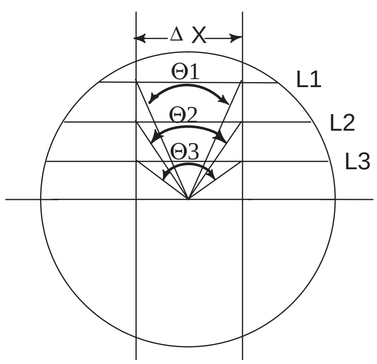
The polar coordinate interpolation mode cannot be started or terminated (G12.1 or G13.1) in the tool nose radius compensation mode (G41 or G42). G12.1 or G13.1 must be specified in the tool nose radius compensation canceled mode (G40).

- **Program restart**

For a block in the G12.1 mode, the program cannot be restarted.

- **Cutting feedrate for the rotation axis**

Polar coordinate interpolation converts the tool movement for a figure programmed in a Cartesian coordinate system to the tool movement in the rotation axis (C–axis) and the linear axis (X–axis). When the tool moves closer to the center of the workpiece, the C–axis component of the feedrate becomes larger and may exceed the maximum cutting feedrate for the C–axis (set in parameter (No. 1422)), causing an alarm (see the figure below). To prevent the C–axis component from exceeding the maximum cutting feedrate for the C–axis, reduce the feedrate specified with address F or create a program so that the tool (center of the tool when tool nose radius compensation is applied) does not move close to the center of the workpiece.



Consider lines L1, L2, and L3. ΔX is the distance the tool moves per time unit at the feedrate specified with address F in the Cartesian coordinate system. As the tool moves from L1 to L2 to L3, the angle at which the tool moves per time unit corresponding to ΔX in the Cartesian coordinate system increases from θ1 to θ 2 to θ3.
In other words, the C–axis component of the feedrate becomes larger as the tool moves closer to the center of the workpiece. The C component of the feedrate may exceed the maximum cutting feedrate for the C–axis because the tool movement in the Cartesian coordinate system has been converted to the tool movement for the C–axis and the X–axis.

L :Distance (in mm) between the tool center and workpiece center when the tool center is the nearest to the workpiece center
R :Maximum cutting feedrate (deg/min) of the C axis
Then, a speed specifiable with address F in polar coordinate interpolation can be given by the formula below. Specify a speed allowed by the formula. The formula provides a theoretical value; in practice, a value slightly smaller than a theoretical value may need to be used due to a calculation error.

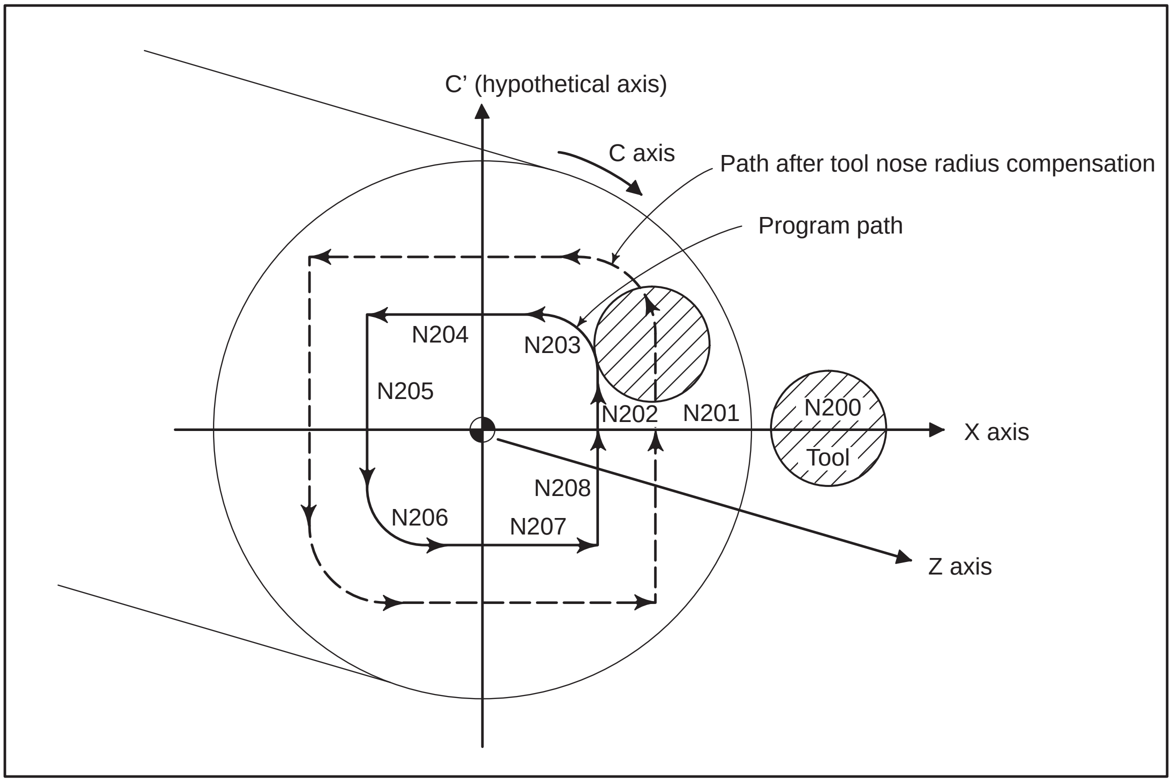
$$F < L \times R \times \frac{\pi}{180} \text{ (mm/min)}$$

- **Diameter and radius programming**

Even when diameter programming is used for the linear axis (X–axis), radius programming is applied to the rotary axis (C–axis).

Examples

Example of Polar Coordinate Interpolation Program Based on X Axis (Linear Axis) and C Axis (Rotary Axis)



X axis is by diameter programming, C axis is by radius programming.

```
O0001 ;
  :
N010 T0101
  :
N0100 G00 X120.0 C0 Z _ ;      Positioning to start position
N0200 G12.1 ;                   Start of polar coordinate interpolation
N0201 G42 G01 X40.0 F _ ;
N0202 C10.0 ;
N0203 G03 X20.0 C20.0 R10.0 ;
N0204 G01 X–40.0 ;              Geometry program
N0205 C–10.0 ;                  (program based on cartesian coordinates on
N0206 G03 X–20.0 C–20.0 I10.0 J0 ;   X–C' plane)
N0207 G01 X40.0 ;
N0208 C0 ;
N0209 G40 X120.0 ;
N0210 G13.1 ;                   Cancellation of polar coordinate interpolation
N0300 Z __ ;
N0400 X __C __ ;
  :
N0900M30 ;
```

4.6 CYLINDRICAL INTERPOLATION (G07.1)

The amount of travel of a rotary axis specified by an angle is once internally converted to a distance of a linear axis along the outer surface so that linear interpolation or circular interpolation can be performed with another axis. After interpolation, such a distance is converted back to the amount of travel of the rotary axis.
The cylindrical interpolation function allows the side of a cylinder to be developed for programming. So programs such as a program for cylindrical cam grooving can be created very easily.

Format

G07.1 IP r ; Starts the cylindrical interpolation mode (enables cylindrical interpolation).

:

G07.1 IP 0 ; The cylindrical interpolation mode is cancelled.

IP : An address for the rotation axis
r : The radius of the cylinder

Specify G07.1 IP r ; and G07.1 IP 0; in separate blocks.
G107 can be used instead of G07.1.

Explanations

• Plane selection (G17, G18, G19)

Use parameter No. 1002 to specify whether the rotation axis is the X–, Y–, or Z–axis, or an axis parallel to one of these axes. Specify the G code to select a plane for which the rotation axis is the specified linear axis.
For example, when the rotation axis is an axis parallel to the X–axis, G17 must specify an Xp–Yp plane, which is a plane defined by the rotation axis and the Y–axis or an axis parallel to the Y–axis.
Only one rotation axis can be set for cylindrical interpolation.

Note
The U–, V–, and W–axes (parallel with the basic axis) can be used with G–codes B and C.

• Feedrate

A feedrate specified in the cylindrical interpolation mode is a speed on the developed cylindrical surface.

- **Circular interpolation (G02,G03)**

In the cylindrical interpolation mode, circular interpolation is possible with the rotation axis and another linear axis. Radius R is used in commands in the same way as described in Section 4.4.
The unit for a radius is not degrees but millimeters (for metric input) or inches (for inch input).

< Example Circular interpolation between the Z axis and C axis >

For the C axis of parameter No. 1022, 5 (axis parallel with the X axis) is to be set. In this case, the command for circular interpolation is

G18 Z__C__;
G02 (G03) Z__C__R__;

For the C axis of parameter No. 1022, 6 (axis parallel with the Y axis) may be specified instead. In this case, however, the command for circular interpolation is

G19 C__Z__;
G02 (G03) Z__C__R__;

- **Cutter compensation**

To perform cutter compensation in the cylindrical interpolation mode, cancel any ongoing cutter compensation mode before entering the cylindrical interpolation mode. Then, start and terminate cutter compensation within the cylindrical interpolation mode.

- **Cylindrical interpolation accuracy**

In the cylindrical interpolation mode, the amount of travel of a rotary axis specified by an angle is once internally converted to a distance of a linear axis on the outer surface so that linear interpolation or circular interpolation can be performed with another axis. After interpolation, such a distance is converted back to an angle. For this conversion, the amount of travel is rounded to a least input increment.
So when the radius of a cylinder is small, the actual amount of travel can differ from a specified amount of travel. Note, however, that such an error is not accumulative.
If manual operation is performed in the cylindrical interpolation mode with manual absolute on, an error can occur for the reason described above.

$$\text{The actual amount of travel} = \left[\frac{\text{MOTION REV}}{2 \times 2\pi R} \left[\times \text{Specified value} \times \frac{2 \times 2\pi R}{\text{MOTION REV}} \right] \right]$$

MOTION REV : The amount of travel per rotation of the rotation axis (Setting value of parameter No. 1260)

R : Workpiece radius

[ ] :Rounded to the least input increment

Restrictions

- **Arc radius specification in the cylindrical interpolation mode**

In the cylindrical interpolation mode, an arc radius cannot be specified with word address I, J, or K.

- **Circular interpolation and tool nose radius compensation**

If the cylindrical interpolation mode is started when tool nose radius compensation is already applied, circular interpolation is not correctly performed in the cylindrical interpolation mode.

- **Positioning**

In the cylindrical interpolation mode, positioning operations (including those that produce rapid traverse cycles such as G28, G80 through G89) cannot be specified. Before positioning can be specified, the cylindrical interpolation mode must be cancelled. Cylindrical interpolation (G07.1) cannot be performed in the positioning mode (G00).

- **Coordinate system setting**

In the cylindrical interpolation mode, a workpiece coordinate system G50 cannot be specified.

- **Cylindrical interpolation mode setting**

In the cylindrical interpolation mode, the cylindrical interpolation mode cannot be reset. The cylindrical interpolation mode must be cancelled before the cylindrical interpolation mode can be reset.

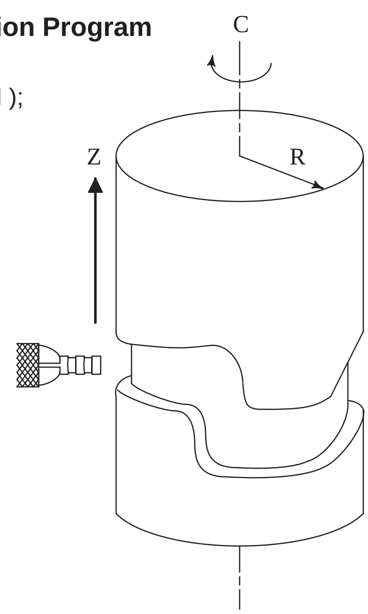
- **Canned cycle for drilling during cylindrical interpolation mode**

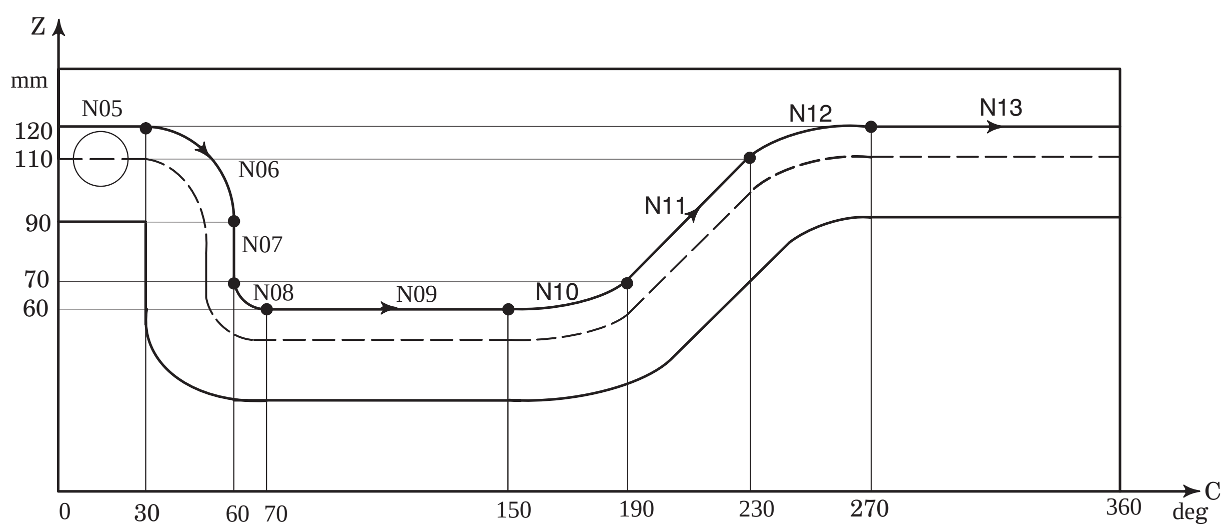
Canned cycles for drilling, G81 to G89, cannot be specified during cylindrical interpolation mode.

Examples

Example of a Cylindrical Interpolation Program

```
O0001 (CYLINDRICAL INTERPOLATION );
N01 G00 Z100.0 C0 ;
N02 G01 G18 W0 H0 ;
N03 G07.1 H57299 ;
N04 G01 G42 Z120.0 D01 F250 ;
N05 C30.0 ;
N06 G02 Z90.0 C60.0 R30.0 ;
N07 G01 Z70.0 ;
N08 G03 Z60.0 C70.0 R10.0 ;
N09 G01 C150.0 ;
N10 G03 Z70.0 C190.0 R75.0 ;
N11 G01 Z110.0 C230.0 ;
N12 G02 Z120.0 C270.0 R75.0 ;
N13 G01 C360.0 ;
N14 G40 Z100.0 ;
N15 G07.1 C0 ;
N16 M30 ;
```





4.7 CONSTANT LEAD THREADING (G32)

Tapered screws and scroll threads in addition to equal lead straight threads can be cut by using a G32 command.
The spindle speed is read from the position coder on the spindle in real time and converted to a cutting feedrate for feed–per minute mode, which is used to move the tool.

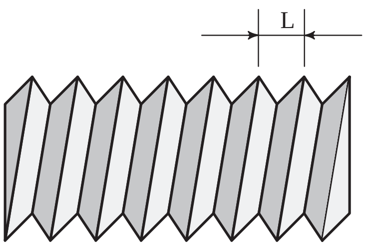


Fig. 4.7 (a) Straight Thread

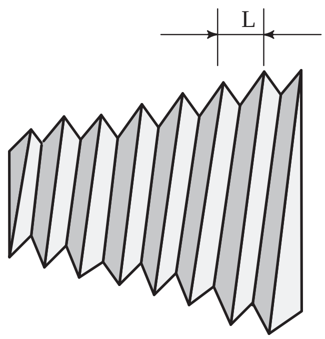


Fig. 4.7 (b) Tapered Screw

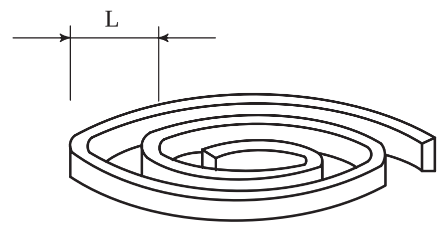


Fig. 4.7 (c) Scroll Thread

Format

G32IP_F_;

IP_: End point
F_: Lead of the long axis (always radius programming)

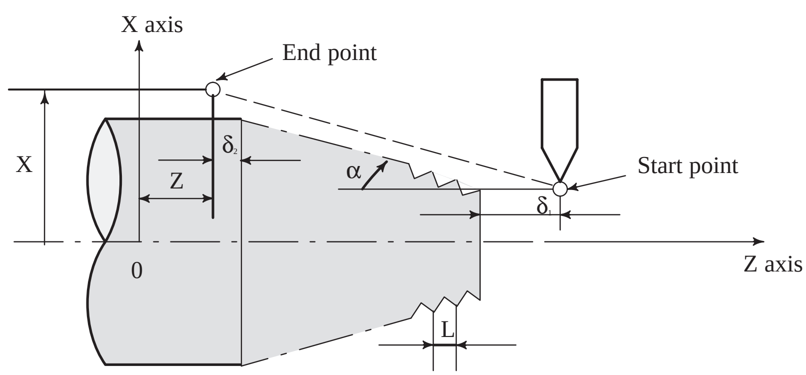


Fig. 4.7 (d) Example of Thread Cutting

Explanations

In general, thread cutting is repeated along the same tool path in rough cutting through finish cutting for a screw. Since thread cutting starts when the position coder mounted on the spindle outputs a 1–turn signal, threading is started at a fixed point and the tool path on the workpiece is unchanged for repeated thread cutting. Note that the spindle speed must remain constant from rough cutting through finish cutting. If not, incorrect thread lead will occur.

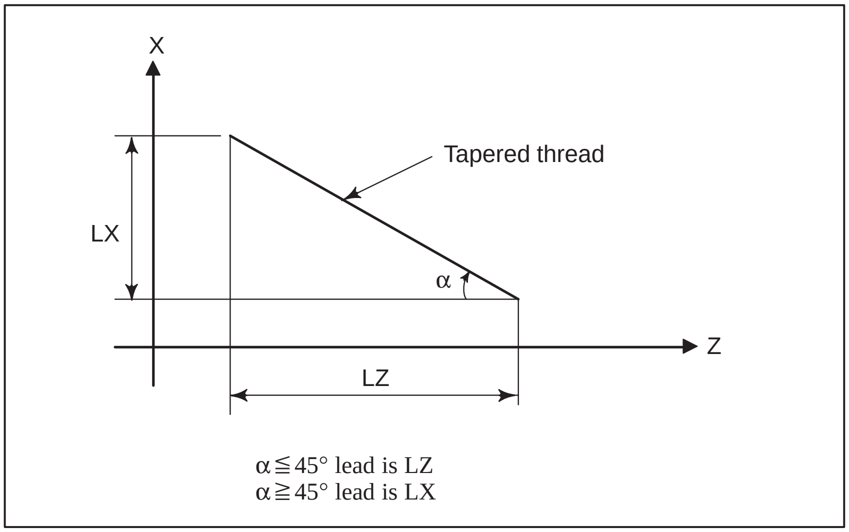


Fig. 4.7 (e) LZ and LX of a Tapered Thread

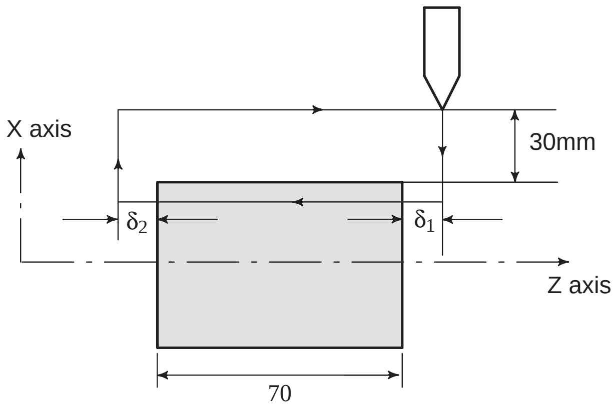
In general, the lag of the servo system, etc. will produce somewhat incorrect leads at the starting and ending points of a thread cut. To compensate for this, a threading length somewhat longer than required should be specified.
Table 4.7 (a) lists the ranges for specifying the thread lead.

Table. 4.7 (a) Ranges of lead sizes that can be specified

	Least command increment
mm input	0.0001 A500.0000mm
Inch input	0.000001 inchA9.999999inch

Explanations

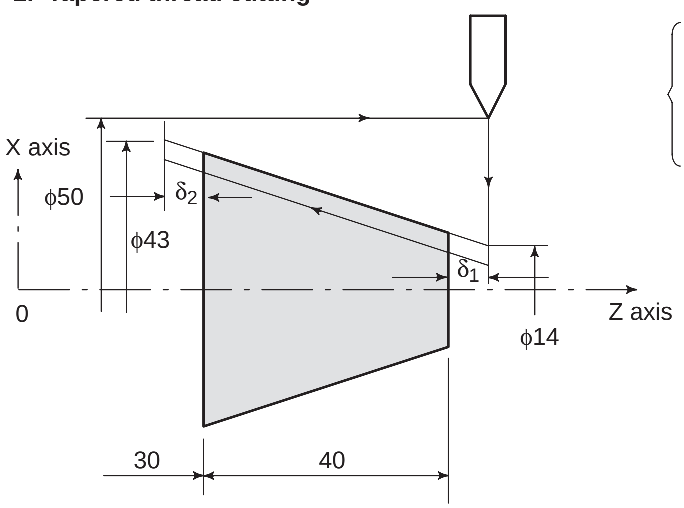
1. Straight thread cutting



The following values are used in programming :
Thread lead :4mm
δ_1=3mm
δ_2=1.5mm
Depth of cut :1mm (cut twice)
(Metric input, Diameter programming)

```
G00   U–62.0 ;
G32   W–74.5 F4.0 ;
G00   U62.0  ;
   W74.5  ;
   U–64.0 ;
   (For the second cut, cut 1mm more)
G32   W–74.5  ;
G00   U64.0  ;
   W74.5  ;
```

2. Tapered thread cutting



The following values are used in programming :
Thread lead : 3.5mm in the direction of the Z axis
δ_1=2mm
δ_2=1mm
Cutting depth in the X axis direction is 1mm
(Cut twice)
(Metric input, Diameter programming)

```
G00   X 12.0 Z72.0 ;
G32   X 41.0 Z29.0 F3.5 ;
G00   X 50.0 ;
   Z 72.0 ;
   X 10.0 ;
   (Cut 1mm more for the second cut)
G32   X 39.0 Z29.0  ;
G00   X 50.0  ;
   Z 72.0  ;
```

Notes

1. Feedrate override is effective (fixed at 100%) during thread cutting.
2. it is very dangerous to stop feeding the thread cutter without stopping the spindle. This will suddenly increase the cutting depth. Thus, the feed hold function is ineffective while thread cutting. If the feed hold!button is pressed during thread cutting, the tool will stop after a block not specifying thread cutting is!executed as if the SINGLE BLOCK button were pushed. However, the feed hold lamp (SPL lamp) lights!when the FEED HOLD button on the machine control panel is pushed. Then, when the tool stops, the!lamp is turned off (Single Block stop status).
3. When the FEED HOLD button is held down, or is pressed again in the first block that does not specify thread cutting immediately after a thread cutting block, the tool stops at the block that does not specify thread cutting.
4. When thread cutting is executed in the single block status, the tool stops after execution of the first block!not specifying thread cutting.
5. When the mode was changed from automatic operation to manual operation during thread cutting, the!tool stops at the first block not specifying thread cutting as when the feed hold button is pushed as !mentioned in Note 3.
   However, when the mode is changed from one automatic operation mode to another, the tool stops after!execution of the block not specifying thread cutting as for the single block mode in Note 4.
6. When the previous block was a thread cutting block, cutting will start immediately without waiting for!detection of the 1–turn signal even if the present block is a thread cutting block.

   G32Z _ F_ ;
   Z _; (A 1–turn signal is not detected before this block.)
   G32 ; (Regarded as threading block.)
   Z_ F_ ;(One turn signal is also not detected.)
7. Because the constant surface speed control is effective during scroll thread or tapered screw cutting and!the spindle speed changes, the correct thread lead may not be cut. Therefore, do not use the constant!surface speed control during thread cutting. Instead, use G97.
8. A movement block preceding the thread cutting block must not specify chamfering or corner R.
9. A thread cutting block must not specifying chamfering or corner R.
10. The spindle speed override function is disabled during thread cutting. The spindle speed is fixed at 100%.
11. Thread cycle retract function is ineffective to G32.

4.8 VARIABLE–LEAD THREAD CUTTING (G34)

Specifying an increment or a decrement value for a lead per screw revolution enables variable–lead thread cutting to be performed.

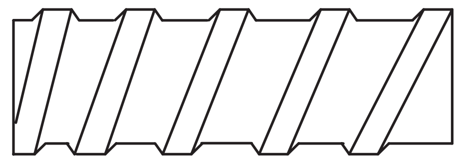


Fig. 4.8 (a) Variable–lead screw

Format

G34 IP_F_K_;

IP : End point
F : Lead in longitudinal axis direction at the start point
K : Increment and decrement of lead per spindle revolution

Explanations

Address other than K are the same as in straight/taper thread cutting with G32.
Table 4.8 (a) lists a range of values that can be specified as K.

Table 4.8 (a) Range of valid K values

Metric input	±0.0001 A ±500.0000 mm/rev
Inch input	±0.000001 A ±9.999999 inch/rev

P/S alarm (No. 14) is produced, for example, when K such that the value in Table 4.8 (a) is exceeded is directed, the maximum value of lead is exceeded as a result of increase or decrease by K or the lead has a negative value.

Note
The ”Thread Cutting Cycle Retract” is not effective for G34.

Examples

Lead at the start point: 8.0 mm
Lead increment: 0.3 mm/rev

G34 Z–72.0 F8.0 K0.3 ;

4.9 CONTINUOUS THREAD CUTTING

This function for continuous thread cutting is such that fractional pulses output to a joint between move blocks are overlapped with the next move for pulse processing and output (block overlap) .
Therefore, discontinuous machining sections caused by the interruption of move during continuously block machining are eliminated, thus making it possible to continuously direct the block for thread cutting instructions.

Explanations

Since the system is controlled in such a manner that the synchronism with the spindle does not deviate in the joint between blocks wherever possible, it is possible to performed special thread cutting operation in which the lead and shape change midway.

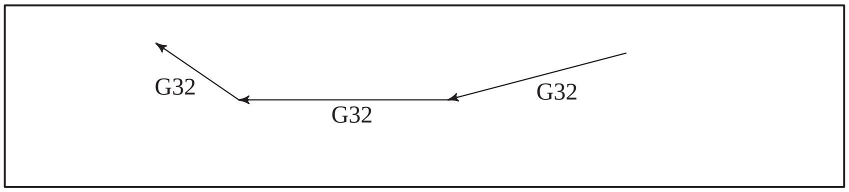


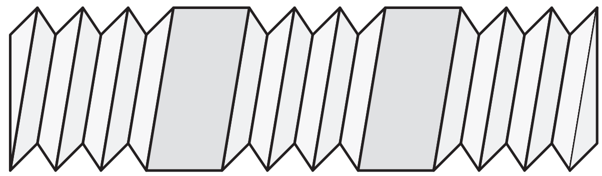
Fig. 4.9 (a) Continuous Thread Cutting

Even when the same section is repeated for thread cutting while changing the depth of cut, this system allows a correct machining without impairing the threads.

Notes
1. Block overlap is effective even for G01 command,!producing a more excellent finishing surface.
2. When extreme micro blocks continue, no block overlap!may function.

4.10 MULTIPLE–THREAD CUTTING

Using the Q address to specify an angle between the one–spindle–rotation signal and the start of threading shifts the threading start angle, making it possible to produce multiple–thread screws with ease.



Multiple–thread screws.

Format

(constant–lead threading)

G32 IP_ F_ Q_ ;
G32 IP_ Q_ ;

IP_ : End point
F_ : Lead in longitudinal direction
Q_ : Threading start angle

Explanations

- **Available thread cutting commands**

G32: Constant–lead thread cutting
G34: Variable–lead thread cutting
G76: Multiple–thread cutting cycle
G92: Thread cutting cycle

Limitations

- **Start angle**

The start angle is not a continuous–state (modal) value. It must be specified each time it is used. If a value is not specified, 0 is assumed.

- **Start angle increment**

The start angle (Q) increment is 0.001 degrees. Note that no decimal point can be specified.
Example:
For a shift angle of 180 degrees, specify Q180000.
Q180.000 cannot be specified, because it contains a decimal point.

- **Specifiable start angle range**

A start angle (Q) of between 0 and 360000 (in 0.001–degree units) can be specified. If a value greater than 360000 (360 degrees) is specified, it is rounded down to 360000 (360 degrees).

- **Multiple–thread cutting (G76)**

For the G76 multiple–thread cutting command, always use the FS15 tape format.

Examples

Program for producing double–threaded screws
(with start angles of 0 and 180 degrees)

```
G00   X40.0 ;
G32   W–38.0 F4.0 Q0 ;
G00   X72.0 ;
   W38.0 ;
   X40.0 ;
G32   W–38.0 F4.0 Q180000
;
G00   X72.0 ;
   W38.0 ;
```

4.11 SKIP FUNCTION (G31)

Linear interpolation can be commanded by specifying axial move following the G31 command, like G01. If an external skip signal is input during the execution of this command, execution of the command is interrupted and the next block is executed.
The skip function is used when the end of machining is not programmed but specified with a signal from the machine, for example, in grinding. It is used also for measuring the dimensions of a workpiece.
For details of how to use this function, refer to the manual supplied by the machine tool builder.

Format

G31 IP_ ;

G31: **One–shot G code (If is effective only in the block in which it is specified)**

Explanations

The coordinate values when the skip signal is turned on can be used in a custom macro because they are stored in the custom macro system variable #5061 to #5068, as follows:

#5061 X axis coordinate value
#5062 Z axis coordinate value
#5063 3rd axis coordinate value
:
:
#5068 8th axis coordinate value

Notes

1 If G31 command is issued while tool nose radius compensation is applied, an P/S alarm of No.035 is displayed. Cancel the cutter compensation with the G40 command before the G31 command is specified.
2 To increase the precision of the tool position when the skip signal is input, feedrate override, dry run, and automatic acceleration/deceleration is disabled for the skip function when the feedrate is specified as a feed per minute value. To enable these functions, set bit 7 (SKF) of parameter No. 6200 to 1. If the feedrate is specified as a feed per rotation value, feedrate override, dry run, and automatic acceleration/deceleration are enabled for the skip function, regardless of the setting of the SKF bit.
3 For the high–speed skip option, executing G31 during feed–per–rotation mode causes P/S alarm 211 to be generated.

Examples

- **The next block to G31 is an incremental command**

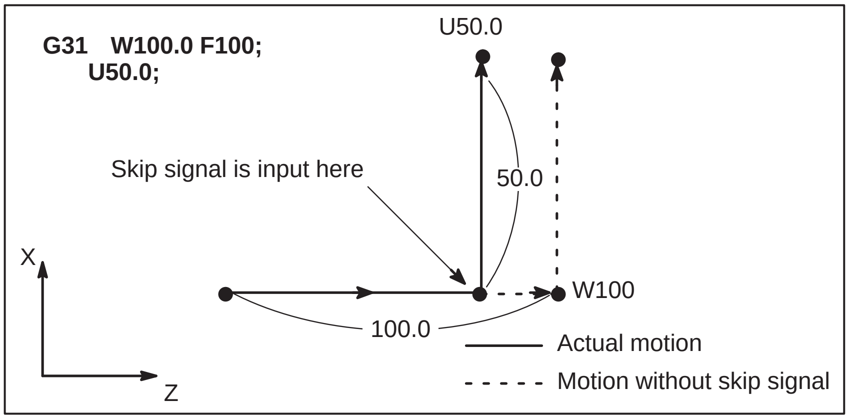


Fig.4.10(a) The next block is an incremental command

- **The next block to G31 is an absolute command for 1 axis**

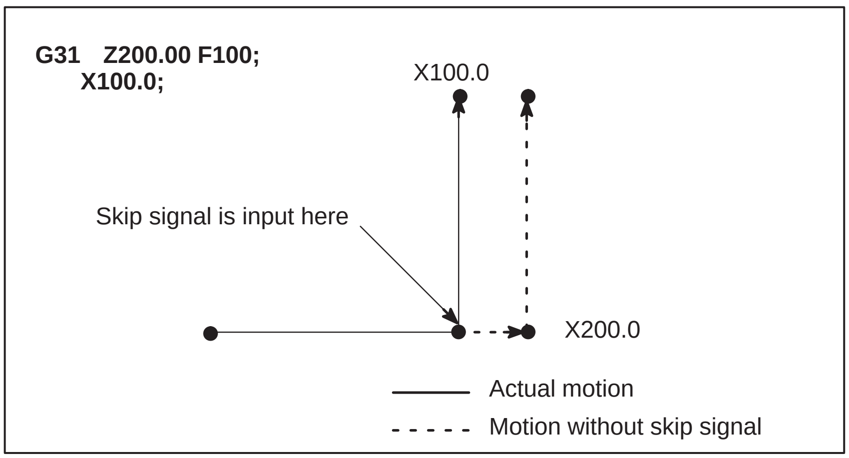


Fig.4.10(b) The next block is an absolute command for 1 axis

- **The next block to G31 is an absolute command for 2 axes**

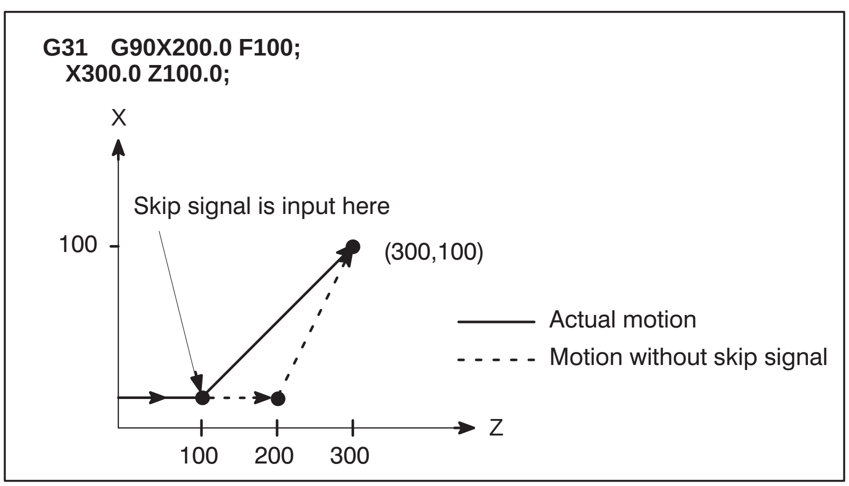


Fig 4.10(c) The next block is an absolute command for 2 axes

4.12 MULTISTAGE SKIP

In a block specifying P1 to P4 after G31, the multistage skip function stores coordinates in a custom macro variable when a skip signal (4–point or 8–point ; 8–point when a high–speed skip signal is used) is turned on. Parameters No. 6202 to No. 6205 can be used to select a 4–point or 8–point (when a high–speed skip signal is used) skip signal. One skip signal can be set to match multiple Pn or Qn (n=1,2,3,4) as well as to match a Pn or Qn on a one–to–one basis. Parameters DS1 to DS8 (No. 6206 #0A#7) can be used for dwell.
A skip signal from equipment such as a fixed–dimension size measuring instrument can be used to skip programs being executed.
In plunge grinding, for example, a series of operations from rough machining to spark–out can be performed automatically by applying a skip signal each time rough machining, semi–fine machining, fine–machining, or spark–out operation is completed.

Format

Move command

G31 IP __ F __ P __ ;

IP_ : End point
F_ : Feedrate
P_ : P1–P4

Dwell

G04 X (U, P)__ (Q__) ;

X(U, P)_ : Dwell time
Q_ : Q1 – Q4

Explanations

Multistage skip is caused by specifying P1, P2, P3, or P4 in a G31 block. For an explanation of selecting (P1, P2, P3, or P4), refer to the manual supplied by the machine tool builder.
Specifying Q1, Q2, Q3, or Q4 in G04 (dwell command) enables dwell skip in a similar way to specifying G31. A skip may occur even if Q is not specified. For an explanation of selecting (Q1, Q2, Q3, or Q4), refer to the manual supplied by the machine tool builder.

• Correspondence to skip signals

Parameter Nos. 6202 to 6205 can be used to specify whether the 4–point or 8–point skip signal is used (when a high–speed skip signal is used). Specification is not limited to one–to–one correspondence. It is possible to specify that one skip signal correspond to two or more Pn's or Qn's (n=1, 2, 3, 4). Also, bits 0 (DS1) to 7 (DS8) of parameter No. 6206 can be used to specify dwell.

Notes
Dwell is not skipped when Qn is not specified and parameters DS1–DS8 (No. 6206#0–#7) are not set.

4.13 TORQUE LIMIT SKIP (G31 P99)

With the motor torque limited (for example, by a torque limit command, issued through the PMC window), a move command following G31 P99 (or G31 P98) can cause the same type of cutting feed as with G01 (linear interpolation).
With the issue of a signal indicating a torque limit has been reached (because of pressure being applied or for some other reason), a skip occurs.
For details of how to use this function, refer to the manuals supplied by the machine tool builder.

Format

G31 P99 IP _ F_ ;

G31 P98 IP _ F_ ;

G31: One–shot G code (G code effective only in the block in which it is issued)

Explanations

- **G31 P99**

If the motor torque limit is reached, or a SKIP signal is received during execution of G31 P99, the current move command is aborted, and the next block is executed.

- **G31 P98**

If the motor torque limit is reached during execution of G31 P98, the current move command is aborted, and the next block is executed. The SKIP signal <X0004#7/Tool post 2 X0013#7> does not affect G31 P98. Entering a SKIP signal during the execution of G31 P98 does not cause a skip.

- **Torque limit command**

If a torque limit is not specified before the execution of G31 P99/98, the move command continues; no skip occurs even if a torque limit is reached.

- **Custom macro system variable**

When G31 P99/98 is specified, the custom macro variables hold the coordinates at the end of a skip. (See Section 4.9.)
If a SKIP signal causes a skip with G31 P99, the custom macro system variables hold the coordinates based on the machine coordinate system when it stops, rather than those when the SKIP signal is entered.

Limitations

- **Axis command**

Only one axis can be controlled in each block with G31 P98/99.
If two or more axes are specified to be controlled in such blocks, or no axis command is issued, P/S alarm No. 015 is generated.

- **Degree of servo error**

When a signal indicating that a torque limit has been reached is input during execution of G31 P99/98, and the degree of servo error exceeds 32767, P/S alarm No. 244 is generated.

- **High–speed skip**

With G31 P99, a SKIP signal can cause a skip, but not a high–speed skip.

- **Simplified synchronization and slanted axis control**

G31 P99/98 cannot be used for axes subject to simplified synchronization or the X–axis or Z–axis when under slanted axis control.

- **Speed control**

Bit 7 (SKF) of parameter No. 6200 must be set to disable dry run, override, and auto acceleration or deceleration for G31 skip commands.

- **Consecutive commands**

Do not use G31 P99/98 in consecutive blocks.

Notes

> **Notes**
>
> 1. Always specify a torque limit before a G31 P99/98 command. Otherwise, G31 P99/98 allows move commands to be executed without causing a skip.
> 2. If G31 is issued with tool nose radius compensation specified, P/S alarm No. 035 is generated. Therefore, before issuing G31, execute G40 to cancel tool nose radius compensation.

Examples

```
O0001 ;
  :
  :
  M■■ ;                  The PMC specifies the torque limit
                         through the window.
  :
  :
  G31 P99 X200. F100 ;   Torque limit skip command
  :
  G01 X100. F500 ;       Move command for which a torque
                         limit is applied
  :
  :
  M▲▲ ;
  :                      Torque limit canceled by the PMC
  :
  M30 ;
  :
  %
```

5 FEED FUNCTIONS

5.1 GENERAL

The feed functions control the feedrate of the tool. The following two feed functions are available:

- **Feed functions**

1. Rapid traverse
   When the positioning command (G00) is specified, the tool moves at!a rapid traverse feedrate set in the CNC (parameter No. 1420).
2. Cutting feed
   The tool moves at a programmed cutting feedrate.

- **Override**

Override can be applied to a rapid traverse rate or cutting feedrate using the switch on the machine operator's panel.

- **Automatic acceleration/ deceleration**

To prevent a mechanical shock, acceleration/deceleration is automatically applied when the tool starts and ends its movement (Fig. 5.1 (a)).

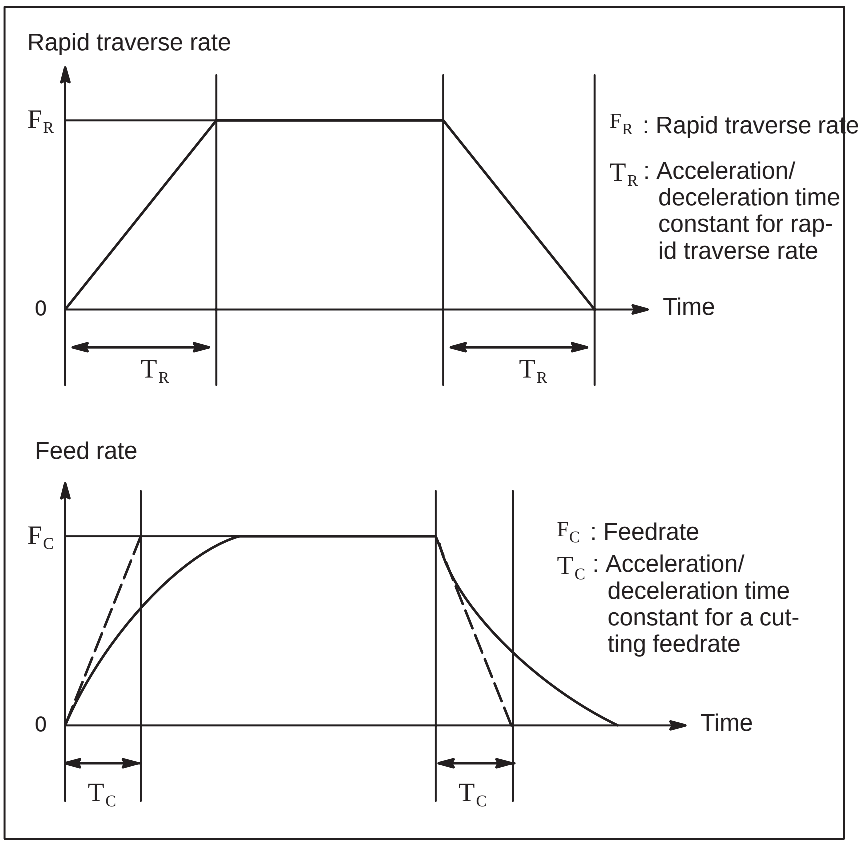


Fig. 5.1 (a) Automatic acceleration/deceleration (example)

- **Tool path in a cutting feed**

If the direction of movement changes between specified blocks during cutting feed, a rounded–corner path may result (Fig. 5.1 (b)).

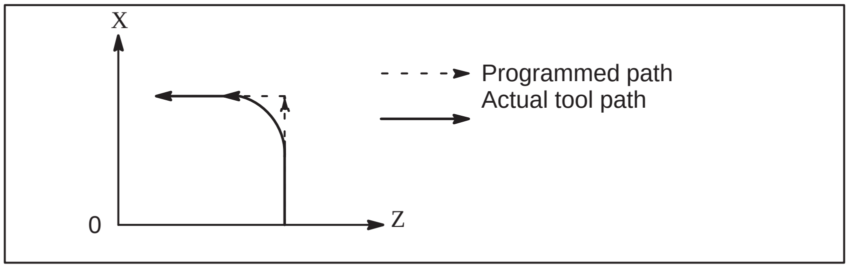


Fig. 5.1 (b) Example of Tool Path between Two Blocks

In circular interpolation, a radial error occurs (Fig. 5.1(c)).

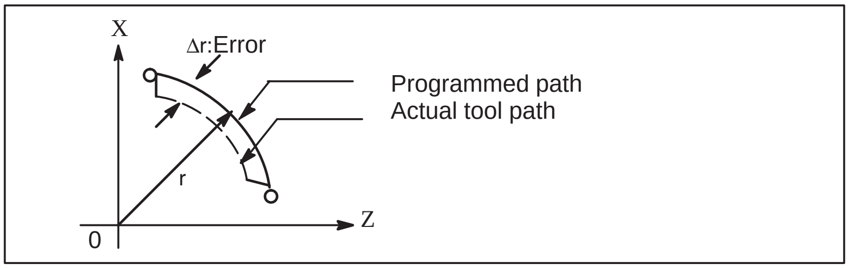


Fig. 5.1 (c) Example of Radial Error in Circular Interpolation

The rounded–corner path shown in Fig. 5.1(b) and the error shown in Fig. 5.1(c) depend on the feedrate. So, the feedrate needs to be controlled for the tool to move as programmed.

5.2 RAPID TRAVERSE

Format

G00 IP_ ;

G00 : G code (group 01) for positioning (rapid traverse)
IP_ ; Dimension word for the end point

Explanations

The positioning command (G00) positions the tool by rapid traverse. In rapid traverse, the next block is executed after the specified feedrate becomes 0 and the servo motor reaches a certain range set by the machine tool builder (in–position check).
A rapid traverse rate is set for each axis by parameter No. 1420, so no rapid traverse feedrate need be programmed.
The following overrides can be applied to a rapid traverse rate with the switch on the machine operator's panel:F0, 25, 50, 100%
F0: Allows a fixed feedrate to be set for each axis by parameter No. 1421.
For detailed information, refer to the appropriate manual of the machine tool builder.

5.3 CUTTING FEED

Feedrate of linear interpolation (G01), circular interpolation (G02, G03), etc. are commanded with numbers after the F code.
In cutting feed, the next block is executed so that the feedrate change from the previous block is minimized.
Two modes of specification are available:
1.Feed per minute (G98)
After F, specify the amount of feed of the tool per minute.
2.Feed per revolution (G99)
After F, specify the amount of feed of the tool per spindle revolution.
3.F1–digit feed
Specify a desired one–digit number after F. Then, the feedrate set with the CNC for that number is set.

Format

Feed per minute
G98 ; **G code (group 05) for feed per minute**
F_ ; **Feedrate command (mm/min or inch/min)**

Feed per revolution
G99 ; **G code (group 05) for feed per revolution**
F_ ; **Feedrate command (mm/rev or inch/rev)**

Explanations

- **Tangential speed constant control**

Cutting feed is controlled so that the tangential feedrate is always set at a specified feedrate.

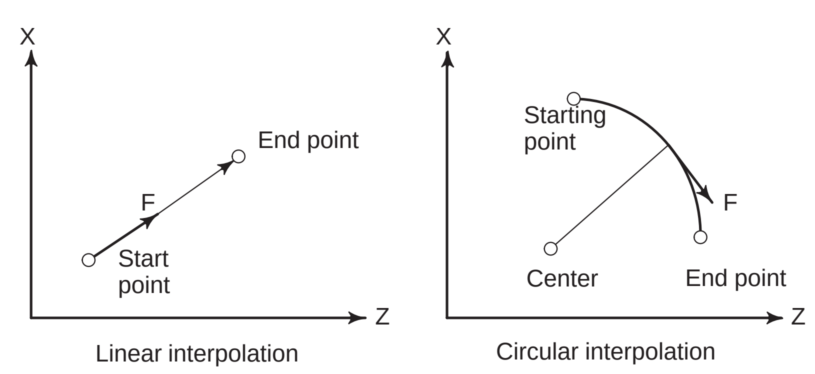


Fig. 5.3 (a) Tangential feedrate (F)

- **Feed per minute (G98)**

After specifying G98 (in the feed per minute mode), the amount of feed of the tool per minute is to be directly specified by setting a number after F. G98 is a modal code. Once a G98 is specified, it is valid until G99 (feed per revolution) is specified. At power–on, the feed per revolution mode is set.
An override from 0% to 254% (in 1% steps) can be applied to feed per minute with the switch on the machine operator's panel. For detailed information, see the appropriate manual of the machine tool builder.

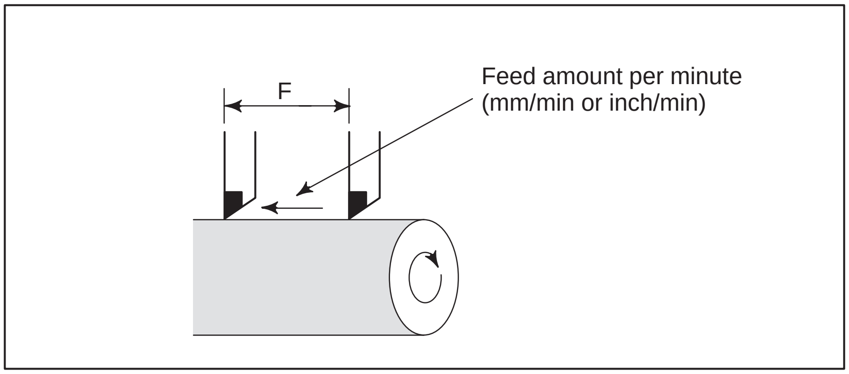


Fig. 5.3 (b) Feed per minute

Note
No override can be used for some commands such as for threading.

- **Feed per revolution (G99)**

After specifying G99 (in the feed per revolution mode), the amount of feed of the tool per spindle revolution is to be directly specified by setting a number after F. G99 is a modal code. Once a G99 is specified, it is valid until G98 (feed per minute) is specified.
An override from 0% to 254% (in 1% steps) can be applied to feed per revolution with the switch on the machine operator's panel. For detailed information, see the appropriate manual of the machine tool builder.

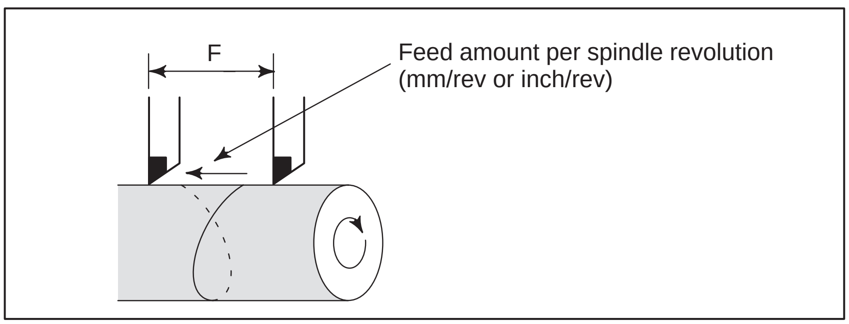


Fig. 5.3 (c) Feed per revolution

Note
When the speed of the spindle is low, feedrate fluctuation may occur. The slower the spindle rotates, the more frequently feedrate fluctuation occurs.

- **Cutting feedrate clamp**

A common upper limit can be set on the cutting feedrate along each axis with parameter No. 1422. If an actual cutting feedrate (with an override applied) exceeds a specified upper limit, it is clamped to the upper limit.

Note

An upper limit is set in mm/min or inch/min. CNC calculation may involve a feedrate error of ±2% with respect to a specified value. However, this is not true for acceleration/deceleration. To be more specific, this error is calculated with respect to a measurement on the time the tool takes to move 500 mm or more during the steady state:

- **Reference**

See Appendix C for a range of feedrates that can be specified.

5.4 DWELL (G04)

Format

Dwell G04 X_ ; or G04 U_ ; or G04 P_ ;

X_ : Specify a time (decimal point permitted)
U_ : Specify a time (decimal point permitted)
P_ : Specify a time (decimal point not permitted)

Explanations

By specifying a dwell, the execution of the next block is delayed by the specified time.
Bit 1 (DWL) of parameter No. 3405 can specify dwell for each rotation in feed per rotation mode (G99).

Table 5.4 (a)
Command value range of the dwell time (Command by X)

Increment system	Command value range	Dwell time unit
IS–B	0.001 to 99999.999	s or rev
IS–C	0.0001 to 9999.9999	

Table 5.4 (b)
Command value range of the dwell time (Command by P)

Increment system	Command value range	Dwell time unit
IS–B	1 to 99999999	0.001 s or rev
IS–C	1 to 99999999	0.0001 s or rev

6 REFERENCE POSITION

General

- **Reference position**

The reference position is a fixed position on a machine tool to which the tool can easily be moved by the reference position return function.
For example, the reference position is used as a position at which tools are automatically changed. Up to four reference positions can be specified by setting coordinates in the machine coordinate system in parameters (No. 1240 to 1243).

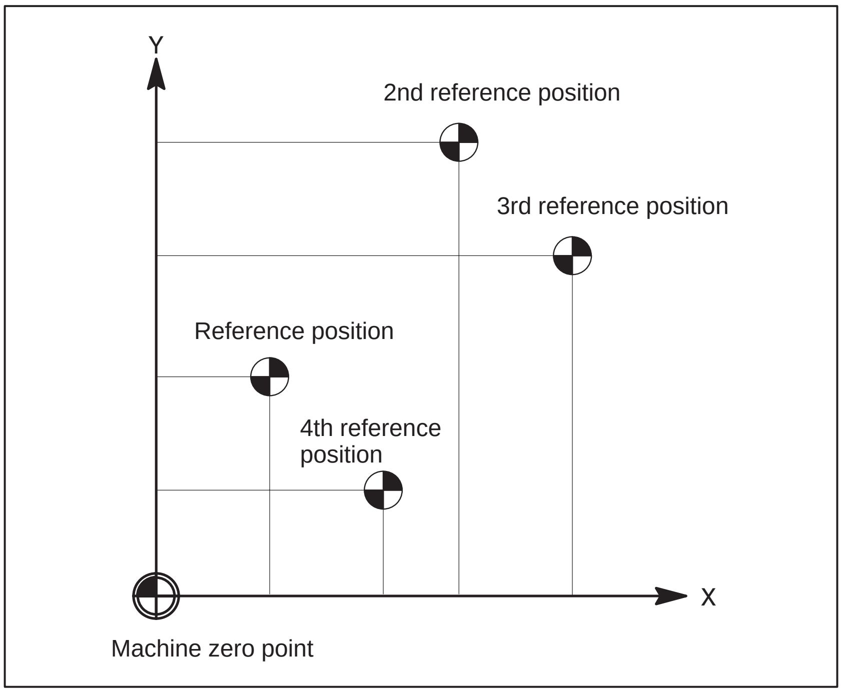


Fig. 6 (a) Machine zero point and reference positions

- **Reference position return**

Tools are automatically moved to the reference position via an intermediate position along a specified axis. When reference position return is completed, the lamp for indicating the completion of return goes on.

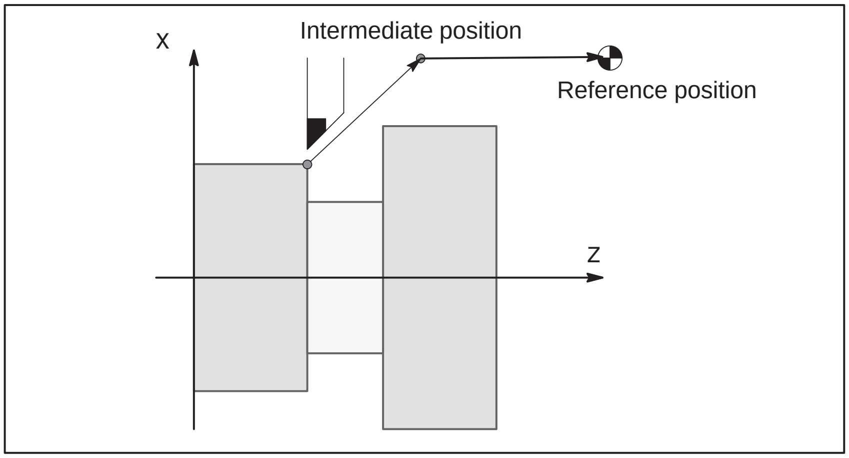


Fig. 6 (b) Reference position return

- **Reference position return check**

The reference position return check (G27) is the function which checks whether the tool has correctly returned to the reference position as specified in the program. If the tool has correctly returned to the reference position along a specified axis, the lamp for the axis goes on.

Format

- **Reference position return**

G28 IP_ ;	**Reference position return**	
G30 P2 IP_ ;	**2nd reference position return**	**(P2 can be omitted.)**
G30 P3 IP_ ;	**3rd reference position return**	
G30 P4 IP_ ;	**4th reference position return**	

IP : Command specifying the intermediate position
(Absolute/incremental command)

- **Reference position return check**

G27 IP_ ;

IP : Command specifying the reference position
(Absolute/incremental command)

Explanations

- **Reference position return (G28)**

Positioning to the intermediate or reference positions are performed at the rapid traverse rate of each axis.
Therefore, for safety, the tool nose radius compensation, and tool offset should be cancelled before executing this command.

- **2nd, 3rd, and 4th reference position return (G30)**

In a system without an absolute–position detector, the first, third, and fourth reference position return functions can be used only after the reference position return (G28) or manual reference position return (see III–3.1) is made. The G30 command is generally used when the automatic tool changer (ATC) position differs from the reference position.

- **Reference position return check (G27)**

G27 command positions the tool at rapid traverse rate. If the tool reaches the reference position, the reference position return lamp lights up.
However, if the position reached by the tool is not the reference position, an alarm (No. 092) is displayed.

Restrictions

- **Status the machine lock being turned on**

The lamp for indicating the completion of return does not go on when the machine lock is turned on, even when the tool has automatically returned to the reference position. In this case, it is not checked whether the tool has returned to the reference position even when a G27 command is specified.

- **First return to the reference position after the power has been turned on (without an absolute position detector)**

When the G28 command is specified when manual return to the reference position has not been performed after the power has been turned on, the movement from the intermediate point is the same as in manual return to the reference position.
In this case, the tool moves in the direction for reference position return specified in parameter ZMIx (bit 5 of No. 1006). Therefore the specified intermediate position must be a position to which reference position return is possible.

- **Reference position return check in an offset mode**

In an offset mode, the position to be reached by the tool with the G27 command is the position obtained by adding the offset value. Therefore, if the position with the offset value added is not the reference position, the lamp does not light up, but an alarm is displayed instead. Usually, cancel offsets before G27 is commanded.

- **Lighting the lamp when the programmed position does not coincide with the reference position**

When the machine tool is an inch system with metric input, the reference position return lamp may also light up even if the programmed position is shifted from the reference position by least input increment. This is because the least input increment of the machine is smaller than its least command increment.

Reference

- **Manual reference position return**

See III–3.1.

7 FLOATING REFERENCE POSITION RETURN (G30.1)

General

Tools ca be returned to the floating reference position.
A floating reference point is a position on a machine tool, and serves as a reference point for machine tool operation.
A floating reference point need not always be fixed, but can be moved as required.

Format

G30.1 IP ;

IP_ : Command of the intermediate position of the floating reference position
(Absolute command/incremental command)

Explanations

On some machine tools, the cutting tools can be replaced at any position unless they interfere with the workpiece or tail stock.
With these machines, the cutting tools should be replaced at a position as close to the workpiece as possible so as to minimize the machine cycle time. For this purpose, the tool change position is to be changed, depending on the figure of the workpiece. This operation can easily be performed using this function. That is, a tool change position suitable for the workpiece is memorized as a floating reference point. Then command G30. 1 can easily cause return to the tool change position.

• Floating reference position

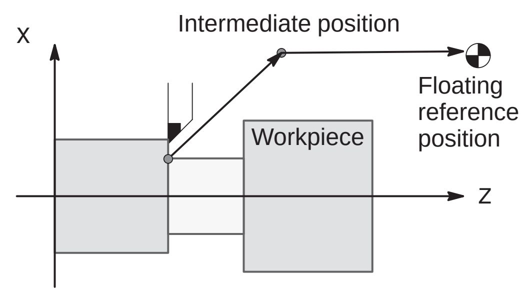
The G30.1 block first positions the tool at the intermediate point along the specified axes at rapid traverse rate, then further moves the tool from the intermediate point to the floating reference point at rapid traverse rate.
Before using G30.1, cancel cutter compensation and tool offset.

• Setting of floating reference position

A floating reference point becomes a machine coordinate position memorized by pressing the soft key **[SET FRP]** on the current positions display screen (see III–11.1.7).
A floating reference point is not lost even if power is turned off.

Examples

G30.1 X40.0 Z50.0 ;



8 COORDINATE SYSTEM

By teaching the CNC a desired tool position, the tool can be moved to the position. Such a tool position is represented by coordinates in a coordinate system. Coordinates are specified using program axes.
When two program axes, the X–axis and Z–axis, are used, coordinates are specified as follows:

X_Z_

This command is referred to as a dimension word.

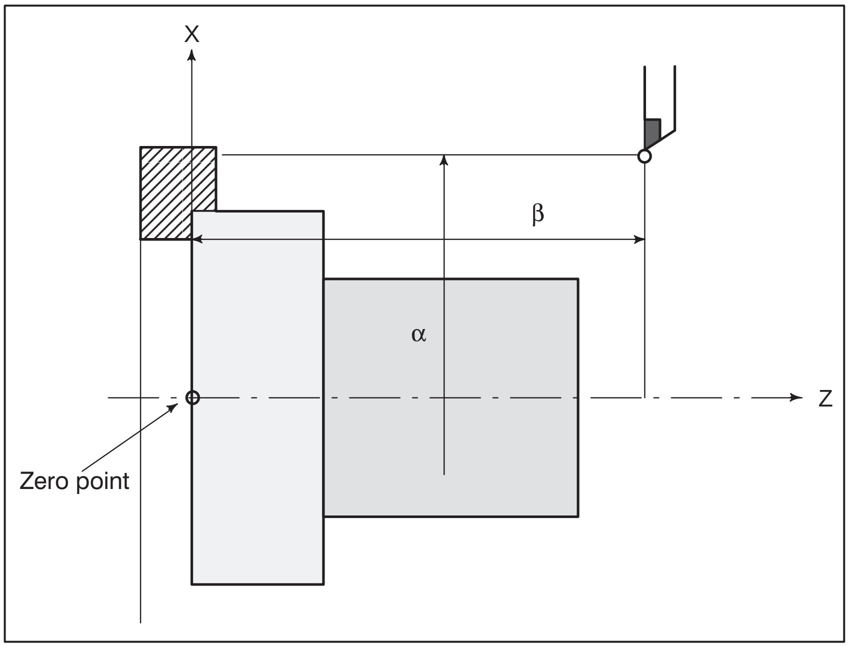


Fig. 8 Tool Position Specified by XαZβ

Coordinates are specified in one of following three coordinate systems:

(1) Machine coordinate system
(2) Workpiece coordinate system
(3) Local coordinate system

The number of the axes of a coordinate system varies from one machine to another. So, in this manual, a dimension word is represented as IP_.

8.1 MACHINE COORDINATE SYSTEM

The point that is specific to a machine and serves as the reference of the machine is referred to as the machine zero point. A machine tool builder sets a machine zero point for each machine.
A coordinate system with a machine zero point set as its origin is referred to as a machine coordinate system.
A machine coordinate system is set by performing manual reference position return after power–on (see III–3.1). A machine coordinate system, once set, remains unchanged until the power is turned off.

Format

G53 IP _ ;
IP _; Absolute dimension word

Explanations

- **Selecting a machine coordinate system (G53)**

When a position has been specified as a set of machine coordinates, the tool moves to that position by means of rapid traverse. G53, used for selecting the machine coordinate system, is a one–shot G code. Any commands based on the selected machine coordinate system are thus effective only in the block containing G53. The G53 command must be specified using absolute values. If incremental values are specified, the G53 command is ignored. When the tool is to be moved to a machine–specific position such as a tool change position, program the movement in a machine coordinate system based on G53.

Restrictions

- **Cancel of the compensation function**

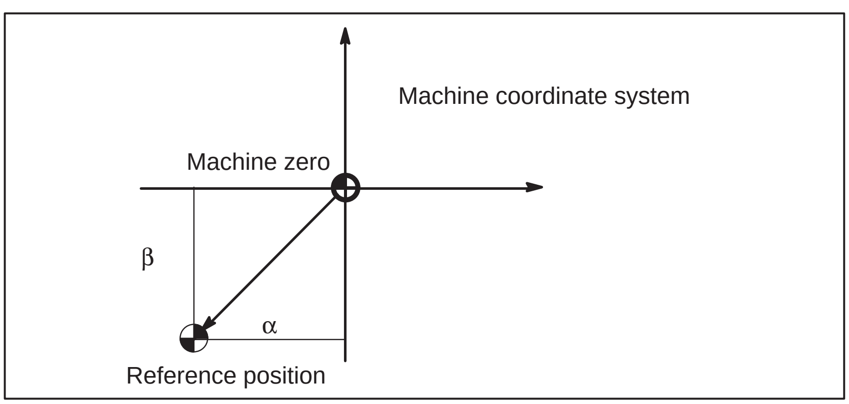
When the G53 command is specified, cancel the tool nose radius compensation and tool offset.

- **G53 specification immediately after power–on**

Since the machine coordinate system must be set before the G53 command is specified, at least one manual reference position return or automatic reference position return by the G28 command must be performed after the power is turned on. This is not necessary when an absolute–position detector is attached.

Reference

When manual reference position return is performed after power–on, a machine coordinate system is set so that the reference position is at the coordinate values of (α, β) set using parameter No.1240.



8.2 WORKPIECE COORDINATE SYSTEM

A coordinate system used for machining a workpiece is referred to as a workpiece coordinate system. A workpiece coordinate system is to be set with the NC beforehand (**setting a workpiece coordinate system**).
A machining program sets a workpiece coordinate system (**selecting a workpiece coordinate system**).
A set workpiece coordinate system can be changed by shifting its origin (**changing a workpiece coordinate system**).

8.2.1 Setting a Workpiece Coordinate System

A workpiece coordinate system can be set using one of three methods:

(1) Method using G50
A workpiece coordinate system is set by specifying a value after G50 in the program.

(2) Automatic setting
If bit 0 of parameter No. 1201 is set beforehand, a workpiece coordinate system is automatically set when manual reference position return is performed (see Part III–3.1.).

(3) Input using the CRT/MDI panel
Six workpiece coordinate systems can be set beforehand using the CRT/MDI panel (see Part III–3.1.).
When an absolute command is used, a workpiece coordinate system must be established in any of the ways described above.

Format

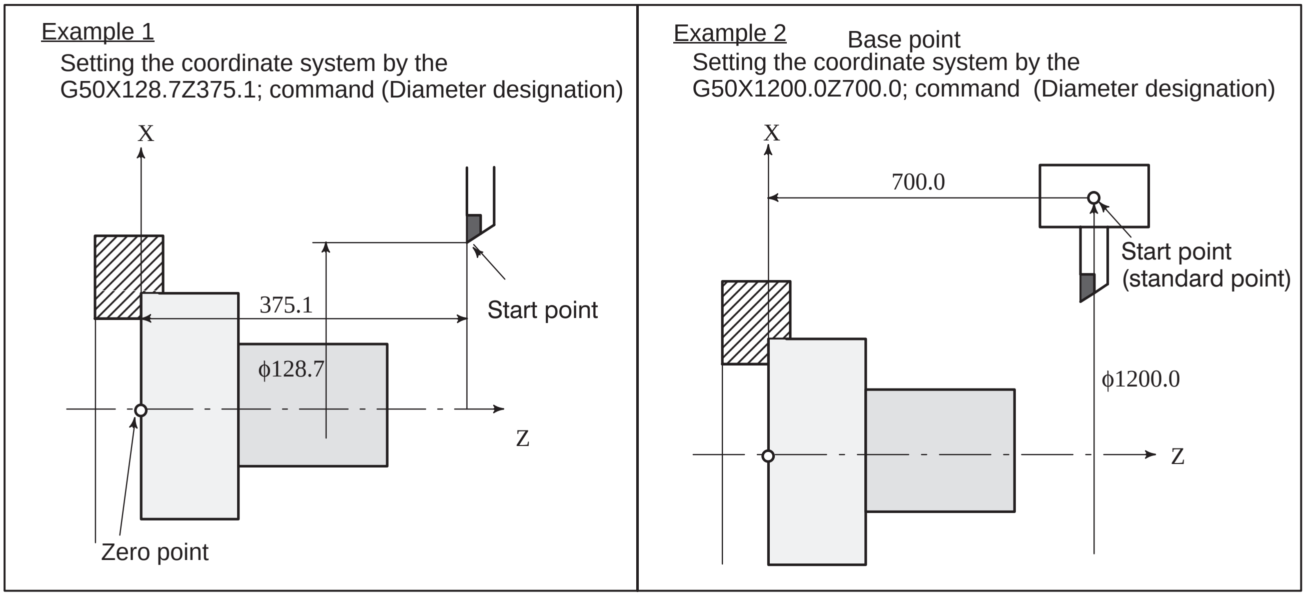
- **Setting a workpiece coordinate system by G50**

```
G50 IP_
```

Explanations

A workpiece coordinate system is set so that a point on the tool, such as the tool tip, is at specified coordinates. If IP is an incremental command value, the work coordinate system is defined so that the current tool position coincides with the result of adding the specified incremental value to the coordinates of the previous tool position. If a coordinate system is set using G50 during offset, a coordinate system in which the position before offset matches the position specified in G50 is set.

Examples



8.2.2 Selecting a Workpiece Coordinate System

The user can choose from set workpiece coordinate systems as described below. (For information about the methods of setting, see Subsec. II–8.2.1.)

(1) G50 or automatic workpiece coordinate system setting

Once a workpiece coordinate system is selected, absolute commands work with the workpiece coordinate system.

(2) Choosing from six workpiece coordinate systems set using the MDI

By specifying a G code from G54 to G59, one of the workpiece coordinate systems 1 to 6 can be selected.

G54 Workpiece coordinate system 1
G55 Workpiece coordinate system 2
G56 Workpiece coordinate system 3
G57 Workpiece coordinate system 4
G58 Workpiece coordinate system 5
G59 Workpiece coordinate system 6

Workpiece coordinate system 1 to 6 are established after reference position return after the power is turned on. When the power is turned on, G54 coordinate system is selected.
When bit 2 (G50) of parameter No. 1202 is set to 1, executing the G50 command results in the issue of P/S alarm No. 10. This is designed to prevent the user from confusing coordinate systems.

Examples

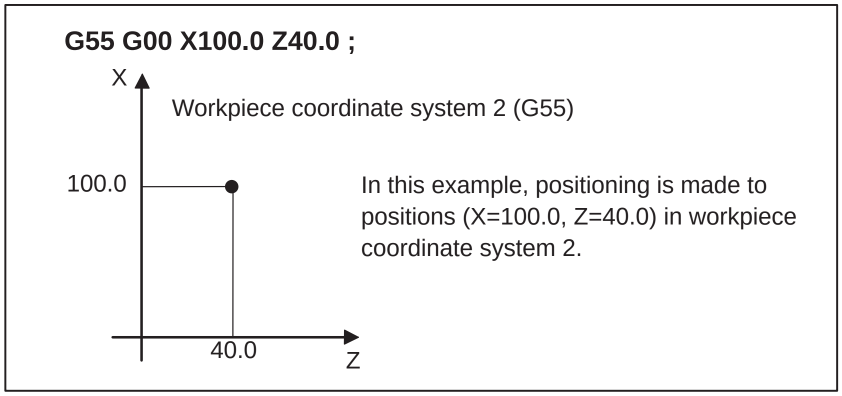


Fig. 8.2.2 (a)

8.2.3 Changing Workpiece Coordinate System

The six workpiece coordinate systems specified with G54 to G59 can be changed by changing an external workpiece zero point offset value or workpiece zero point offset value.

Three methods are available to change an external workpiece zero point offset value or workpiece zero point offset value.

(1) Inputting from the MDI panel (see III–11.4.10)

(2) Programming by G10 or G50

(3) Using the external data input function
An external workpiece origin offset can be changed by using a signal input to the CNC. For details, refer to the relevant manual supplied by the machine tool builder.

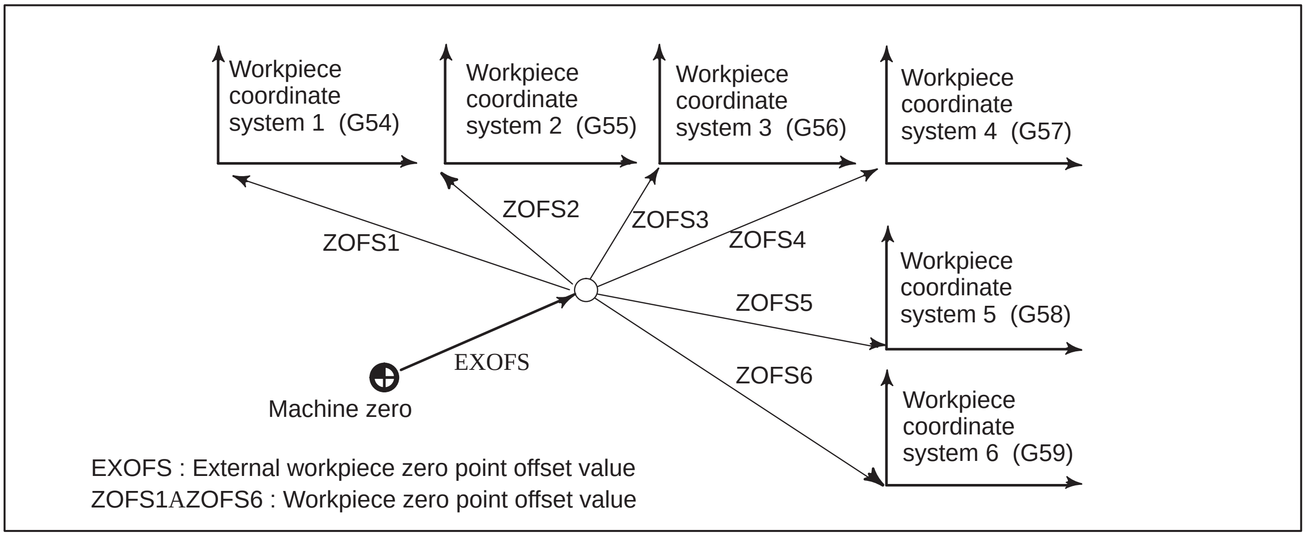


Fig. 8.2.3 (a) Changing an external workpiece zero point offset value or workpiece zero point offset value

Format

- **Changing by G10**

G10 L2 Pp IP _;

p=0 : External workpiece zero point offset value
p=1 to 6 : Workpiece zero point offset value correspond to workpiece coordinate system 1 to 6
IP : Workpiece zero point offset value of each axis

- **Changing by G50**

G50 IP _;

Explanations

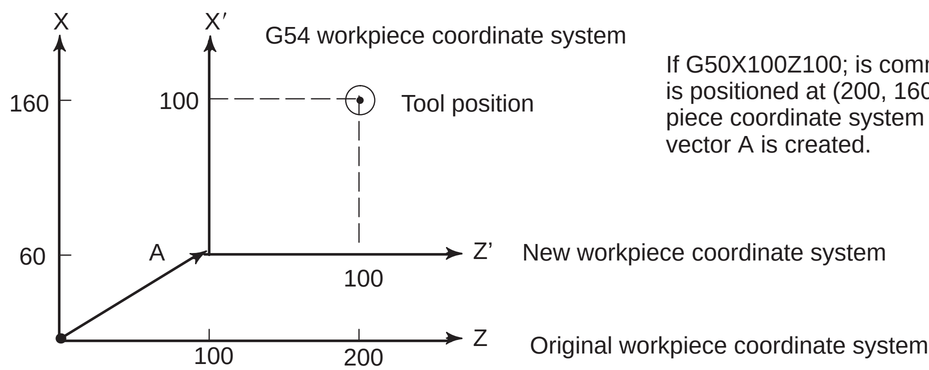
- **Changing by G10**

With the G10 command, each workpiece coordinate system can be changed separately.

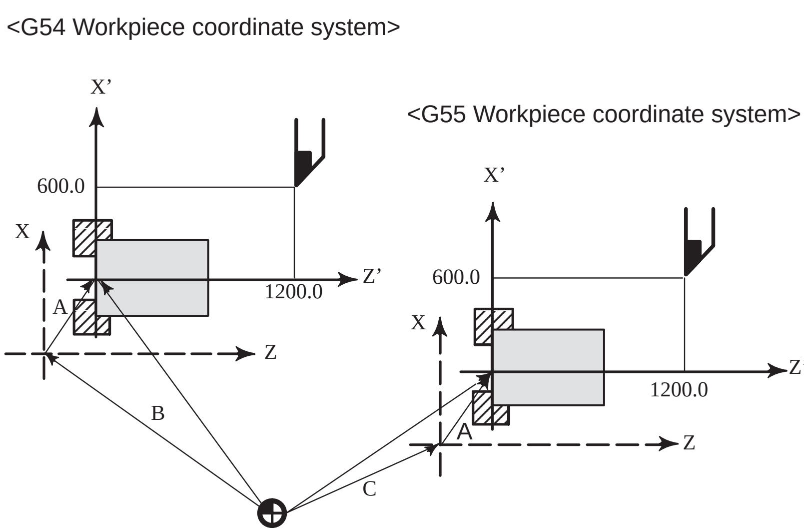
- **Changing by G50**

By specifying G50IP_;, a workpiece coordinate system (selected with a code from G54 to G59) is shifted to set a new workpiece coordinate system so that the current tool position matches the specified coordinates (IP_).
If IP is an incremental command value, the work coordinate system is defined so that the current tool position coincides with the result of adding the specified incremental value to the coordinates of the previous tool position.
Then, the amount of coordinate system shift is added to all the workpiece zero point offset values. This means that all the workpiece coordinate systems are shifted by the same amount.

Examples



If G50X100Z100; is commanded when the tool is positioned at (200, 160) in G54 mode, workpiece coordinate system 1 (X' – Z') shifted by vector A is created.



X' – Z' New workpiece coordinate system
X – Z - Original workpiece coordinate system
A : Offset value created by G50
B : Workpiece zero point offset value in G54
C : Workpiece zero point offset value in G55

Suppose that a G54 workpiece coordinate system is specified. Then, a G55 workpiece coordinate system where the black circle on the tool (figure at the left) is at (600.0,12000.0) can be set with the following command if the relative relationship between the G54 workpiece coordinate system and G55 workpiece coordinate system is set correctly:G50X600.0Z1200.0;Also, suppose that pallets are loaded at two different positions. If the relative relationship of the coordinate systems of the pallets at the two positions is correctly set by handling the coordinate systems as the G54 workpiece coordinate system and G55 workpiece coordinate system, a coordinate system shift with G50 in one pallet causes the same coordinate system shift in the other pallet. This means that workpieces on two pallets can be machined with the same program just by specifying G54 or G55.

8.2.4 Workpiece Coordinate System Preset (G92.1)

The workpiece coordinate system preset function presets a workpiece coordinate system shifted by manual intervention to the pre–shift workpiece coordinate system. The latter system is displaced from the machine zero point by a workpiece zero point offset value.

There are two methods for using the workpiece coordinate system preset function. One method uses a programmed command (G92.1). The other uses MDI operations on the absolute position display screen, relative position display screen, and overall position display screen (III – 11.1.4).

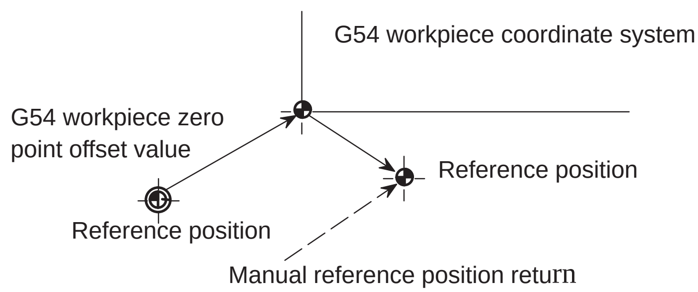
Format

G92.1 IP 0 ; (**G50.3 P0 ;** for G code system A)

IP 0 ; Specifies axis addresses subject to the workpiece coordinate system preset operation. Axes that are not specified are not subject to the preset operation.

Explanations

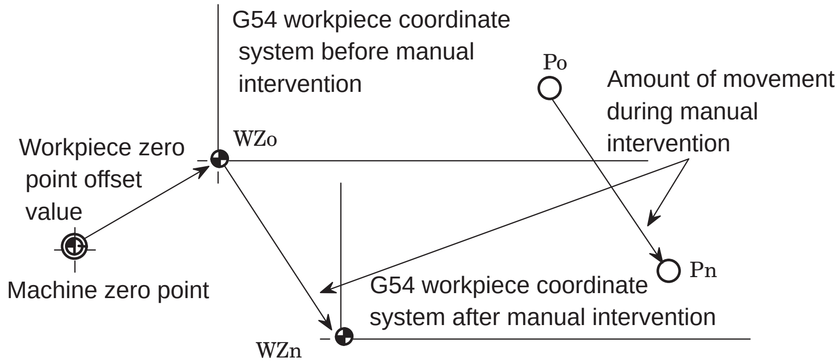
When manual reference position return operation is performed in the reset state, a workpiece coordinate system is shifted by the workpiece zero point offset value from the machine coordinate system zero point. Suppose that the manual reference position return operation is performed when a workpiece coordinate system is selected with G54. In this case, a workpiece coordinate system is automatically set which has its zero point displaced from the machine zero point by the G54 workpiece zero point offset value; the distance from the zero point of the workpiece coordinate system to the reference position represents the current position in the workpiece coordinate system.



If an absolute position detector is provided, the workpiece coordinate system automatically set at power–up has its zero point displaced from the machine zero point by the G54 workpiece zero point offset value. The machine position at the time of power–up is read from the absolute position detector and the current position in the workpiece coordinate system is set by subtracting the G54 workpiece zero point offset value from this machine position. The workpiece coordinate system set by these operations is shifted from the machine coordinate system using the commands and operations listed next page.

(a) Manual intervention performed when the manual absolute signal is off
(b) Move command executed in the machine lock state
(c) Movement by handle interrupt
(d) Operation using the mirror image function
(e) Setting the local coordinate system using G52, or shifting the workpiece coordinate system using G92

In the case of (a) above, the workpiece coordinate system is shifted by the amount of movement during manual intervention.



In the operation above, a workpiece coordinate system once shifted can be preset using G code specification or MDI operation to a workpiece coordinate system displaced by a workpiece zero point offset value from the machine zero point. This is the same as when manual reference position return operation is performed on a workpiece coordinate system that has been shifted. In this example, such G code specification or MDI operation has the effect of returning workpiece coordinate system zero point WZn to the original zero point WZo, and the distance from WZo to Pn is used to represent the current position in the workpiece coordinate system.

Bit 3 (PPD) of parameter No. 3104 specifies whether to preset relative coordinates (RELATIVE) as well as absolute coordinates.

When no workpiece coordinate system option (G54 to G59) is selected, the workpiece coordinate system is preset to the coordinate system set by automatic workpiece coordinate system setting. When automatic workpiece coordinate system setting is not selected, the workpiece coordinate system is preset with its zero point placed at the reference position.

Restrictions

- **Cutter compensation, tool length compensation, tool offset**

When using the workpiece coordinate system preset function, cancel compensation modes: cutter compensation, tool length compensation, and tool offset. If the function is executed without cancelling these modes, compensation vectors are temporarily cancelled.

- **Program restart**

The workpiece coordinate system preset function is not executed during program restart.

- **Prohibited modes**

Do not use the workpiece coordinate system preset function when the scaling, coordinate system rotation, programmable image, or drawing copy mode is set.

8.2.5 Workpiece Coordinate System shift

When the coordinate system actually set by the G50 command or the automatic system setting deviates from the programmed work system, the set coordinate system can be shifted (see III–3.1).
Set the desired shift amount in the work coordinate system shift memory.

Explanations

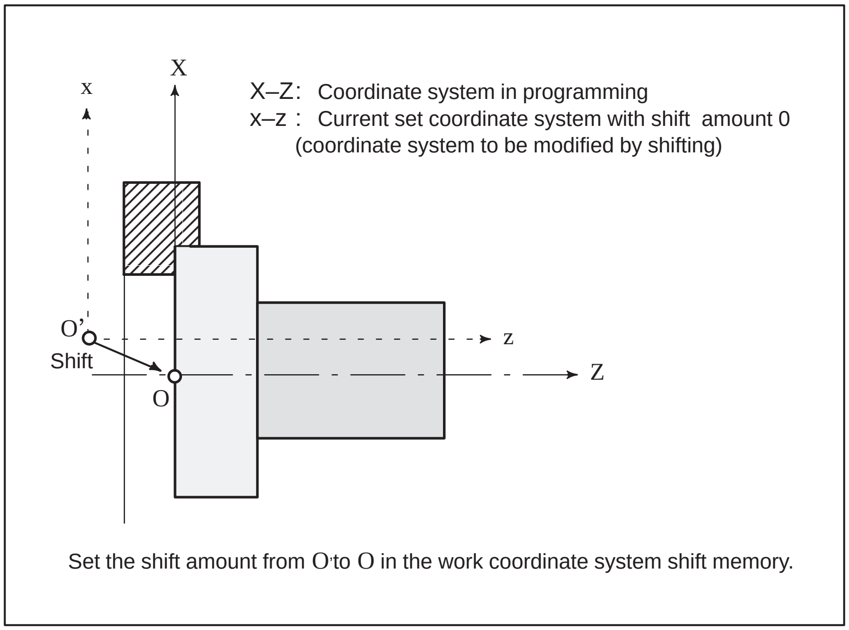


Fig. 8.2.5 (a) Workpiece Coordinate System shift

See Section 11.4.5 of Part III for how to specify the distance the work coordinate system is shifted.

8.3 LOCAL COORDINATE SYSTEM

When a program is created in a workpiece coordinate system, a child workpiece coordinate system may be set for easier programming. Such a child coordinate system is referred to as a local coordinate system.

Format

G52 IP _; Setting the local coordinate system

......

G52 IP 0 ; Canceling of the local coordinate system

IP _ : Origin of the local coordinate system

Explanations

By specifying G52IP_;, a local coordinate system can be set in all the workpiece coordinate systems (G54 to G59). The origin of each local coordinate system is set at the position specified by IP_ in the workpiece coordinate system.
Once a local coordinate system is established, the coordinates in the local coordinate system are used in an axis shift command. The local coordinate system can be changed by specifying the G52 command with the zero point of a new local coordinate system in the workpiece coordinate system.
To cancel the local coordinate system and specify the coordinate value in the workpiece coordinate system, match the zero point of the local coordinate system with that of the workpiece coordinate system.

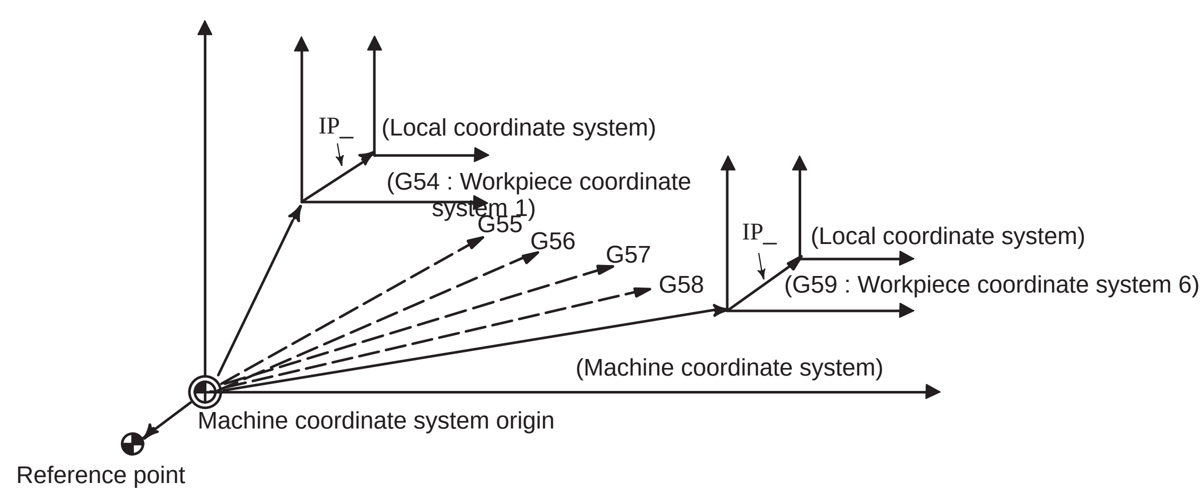


Fig. 8.3 Setting the local coordinate system

Notes

1 The local coordinate system setting does not change the workpiece and machine coordinate systems.

2 When G50 is used to define a work coordinate system, if coordinates are not specified for all axes of a local coordinate system, the local coordinate system remains unchanged.
If coordinates are specified for any axis of a local coordinate system, the local coordinate system is canceled.

3 G52 cancels the offset temporarily in tool nose radius compensation.

4 Command a move command immediately after the G52 block in the absolute mode.

5 Whether the local coordinate system is canceled upon reset depends on the specified parameters. The local coordinate system is canceled upon reset when bit 6 (CLR) of parameter No. 3402 or bit 3 (RLC) of parameter No. 1202 is set to 1.

8.4 PLANE SELECTION

Select the planes for circular interpolation, tool nose radius compensation, coordinate system rotation, and drilling by G–code.
The following table lists G–codes and the planes selected by them.

Explanations

Table 8.4 Plane selected by G code

G code	Selected plane	Xp	Yp	Zp
G17	Xp Yp plane	X–axis or an axis parallel to it	Y–axis or an axis parallel to it	Z–axis or an axis parallel to it
G18	Zp Xp plane			
G19	Yp Zp plane			

Xp, Yp, Zp are determined by the axis address appeared in the block in which G17, G18 or G19 is commanded.
When an axis address is omitted in G17, G18 or G19 block, it is assumed that the addresses of basic three axes are omitted.
Parameter No. 1022 specifies whether each axis is a basic axis (X–axis, Y–axis, or Z–axis) or an axis parallel to a basic axis.
The plane is unchanged in the block in which G17, G18 or G19 is not commanded.
When the power is turned on, G18 (ZX plane) is selected .
The movement instruction is irrelevant to the plane selection.

Notes

1 U–, V–, and W–axes (parallel to a basic axis) can be used with G–codes B and C.

2 Direct drawing dimension programming, chamfering, corner R, multiple repetitive canned cycle, and simple canned cycle are enabled only for the ZX plane.
Specifying these functions for other planes causes P/S alarm No. 212 to be generated.

Examples

Plane selection when the X–axis is parallel with the U–axis.

G17X_Y_; XY plane,
G17U_Y_; UY plane
G18X_Z_; ZX plane
X_Y_; Plane is unchanged (ZX plane)
G17 ; XY plane
G18 ; ZX plane
G17 U_ ; UY plane
G18Y_ ; ZX plane, Y axis moves regardless without any relation to the plane.

9 COORDINATE VALUE AND DIMENSION

This chapter contains the following topics.

9.1 ABSOLUTE AND INCREMENTAL PROGRAMMING (G90, G91)

There are two ways to command travels of the tool; the absolute command, and the incremental command. In the absolute command, coordinate value of the end position is programmed; in the incremental command, move distance of the position itself is programmed. G90 and G91 are used to command absolute or incremental command, respectively.
Absolute programming or incremental programming is used depending on the command used. See following tables.

G code system	A	B or C
Command method	Address word	G90, G91

Format

- **G code system A**

	Absolute command	Incremental command
X axis move command Z axis move command Y axis move command C axis move command	X Z Y C	U W V H

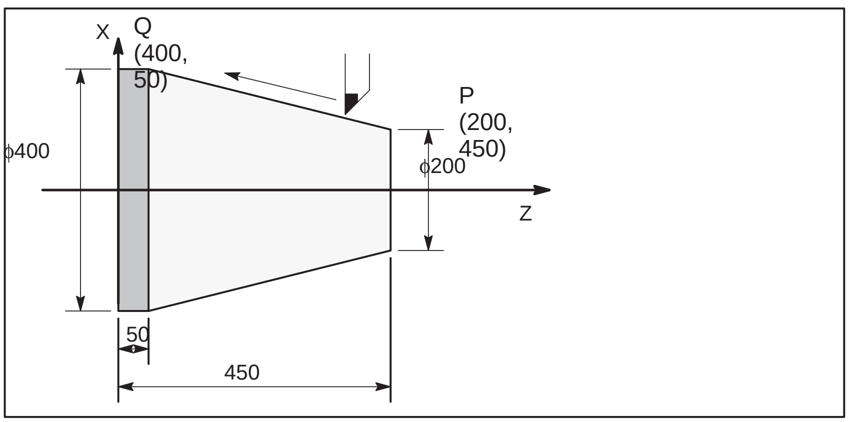
- **G code system B or C**

Absolute command **G90 IP;**
Incremental command **G91 IP;**

Examples

- **Tool movement from point P to point Q (diameter programming is used for the X–axis)**

	G code system A	G code system B or C
Absolute command	X400.0 Z50.0 ;	G90 X400.0 Z50.0 ;
Incremental command	U200.0 W–400.0 ;	G91 X200.0 Z–400.0 ;



Notes
1. Absolute and incremental commands can be used together in a block.
   In the above example, the following command can be specified :
   X400.0 W–400.0 ;
2. When both X and U or W and Z are used together in a block, the one specified later is effective.
3. Incremental commands cannot be used when names of the axes are A and B during G code system A is selected.

9.2 INCH/METRIC CONVERSION(G20,G21)

Either inch or metric input can be selected by G code.

Format

G20 ;	**Inch input**
G21 ;	**mm input**

This G code must be specified in an independent block before setting the coordinate system at the beginning of the program. After the G code for inch/metric conversion is specified, the unit of input data is switched to the least inch or metric input increment of increment system IS–B or IS–C (Section II–2.3). The unit of data input for degrees remains unchanged. The unit systems for the following values are changed after inch/metric conversion:

– Feedrate commanded by F code
– Positional command
– Work zero point offset value
– Tool compensation value
– Unit of scale for manual pulse generator
– Movement distance in incremental feed
– Some parameters

When the power is turned on, the G code is the same as that held before the power was turned off.

Notes

1. G20 and G21 must not be switched during a program.
2. When the least input increment and the least command increment systems are different, the maximum error is half of the least command increment. This error is not accumulated.
3. When switching inch input (G20) to metric input (G21) and vice versa, the tool compensation value must be re–set according to the least input increment.
   However, when bit 0 (OIM) of parameter 5006 is 1, tool compensation values are automatically converted and need not be re–set.
4. Reference position return is performed at a low speed for the first G28 command after the inch input is switched to the metric input or vice versa.
5. The inch and metric input can also be switched using setting parameter INI (No.0002#2).

9.3 DECIMAL POINT PROGRAMMING

Numerical values can be entered with a decimal point. A decimal point can be used when entering a distance, time, or speed. Decimal points can be specified with the following addresses:
X, Y, Z, U, V, W, A, B, C, I, J, K, R, and F.

Explanations

There are two types of decimal point notation: calculator–type notation and standard notation.
When calculator–type decimal notation is used, a value without decimal point is considered to be specified in millimeters. When standard decimal notation is used, such a value is considered to be specified in least input increments. Select either calculator–type or standard decimal notation by using the DPI bit (bit 0 of parameter 3401).Values can be specified both with and without decimal point in a single program.

Examples

Program command	Pocket calculator type decimal point programming	Standard type decimal point programming
X1000 Command value with-out decimal point	1000mm Unit : mm	1mm Unit : Least input increment (0.001 mm)
X1000.0 Command value with decimal point	1000mm Unit : mm	1000mm Unit : mm

Notes

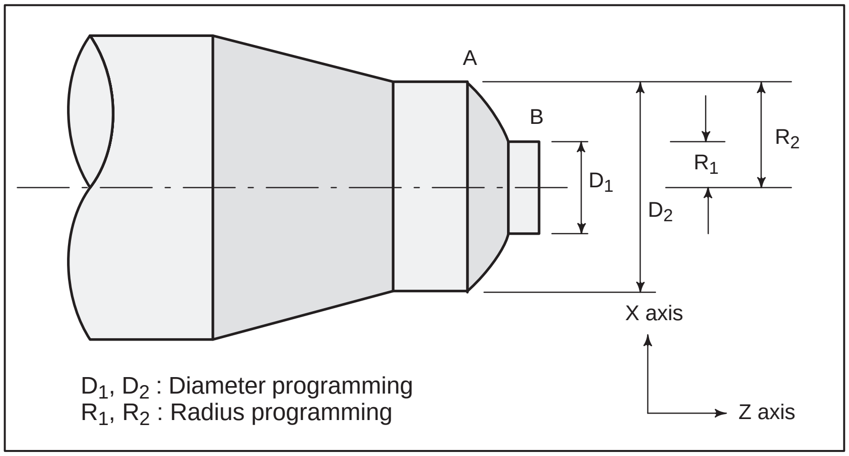
1. In a single block, specify a G code before entering a value. The position of decimal point may depend on the command.
   Examples:
   G20; Input in inches
   X1.0 G04; X1.0 is considered to be a distance and processed as X10000. This command is equivalent to G04 X10000. The tool dwells for 10 seconds.
   G04 X1.0; Equivalent to G04 X1000. The tool dwells for one second.
2. Fractions less than the least input increment are truncated.
   Examples:
   X1.2345; Truncated to X1.234 when the least input increment is 0.001 mm.
   Processed as X1.2345 when the least input increment is 0.0001 inch.
3. When more than eight digits are specified, an alarm occurs. If a value is entered with a decimal point, the number of digits is also checked after the value is converted to an integer according to the least input increment.
   Examples:
   X1.23456789; P/S alarm 003 occurs because more than eight digits are specified.
   X123456.7; If the least input increment is 0.001 mm, the value is converted to integer 123456700. Because the integer has more than eight digits, an P/S alarm 003 occurs.

9.4 DIAMETER AND RADIUS PROGRAMMING

Since the work cross section is usually circular in CNC lathe control programming, its dimensions can be specified in two ways :

Diameter and Radius

When the diameter is specified, it is called diameter programming and when the radius is specified, it is called radius programming.



Explanations

- **Notes on diameter programming/radius programming for each command**

Radius programming or diameter programming can be specified by parameter DIA (No.1006#3). When using diameter programming, note the conditions listed in the table9.4(a).

Table 9.4(a) Notes on specifying diameter value

Item	Notes
X axis command	Specified with a diameter value
Incremental command	Specified with a diameter value In the above figure, specifies D2 minus D1 for tool path B to A.
Coordinate system setting (G50)	Specifies a coordinate value with a diameter value
Component of tool offset value	Parameter (No.5004#1) determines either diameter or radius value
Parameters in canned cycle, such as cutting depth along X axis. (R)	Specifies a radius value
Radius designation in circular interpolation (R, I, K, and etc.)	Specifies a radius value
Feedrate along axis	Specifies change of radius/rev. or change of radius/min.
Display of axis position	Displayed as diameter value

10 SPINDLE SPEED FUNCTION

The spindle speed can be controlled by specifying a value following address S.
In addition, the spindle can be rotated by a specified angle.
This chapter contains the following topics.

10.1 SPECIFYING THE SPINDLE SPEED WITH A BINARY CODE

Specifying a value following address S sends code and strobe signals to the machine. On the machine, the signals are used to control the spindle speed. A block can contain only one S code. Refer to the appropriate manual provided by the machine tool builder for details such as the number of digits in an S code or the execution order when a move command and an S code command are in the same block.

10.2 SPECIFYING THE SPINDLE SPEED VALUE DIRECTLY (S5–DIGIT COMMAND)

The spindle speed can be specified directly by address S followed by a five–digit value (rpm). The unit for specifying the spindle speed may vary depending on the machine tool builder. Refer to the appropriate manual provided by the machine tool builder for details.

10.3 CONSTANT SURFACE SPEED CONTROL (G96, G97)

Specify the surface speed (relative speed between the tool and workpiece) following S. The spindle is rotated so that the surface speed is constant regardless of the position of the tool.

Format

- **Constant surface speed control command**

G96 S○○○○○ ;
↑Surface speed (m/min or feet/min)

Note : This surface speed unit may change according to machine tool builder's specification.

- **Constant surface speed control cancel command**

G97 S○○○○○ ;
↑Spindle speed (rpm)

Note : This surface speed unit may change according to machine tool builder's specification.

- **Clamp of maximum spindle speed**

G50 S_ ; The maximum spindle speed (rpm) follows S.

Explanations

- **Constant surface speed control command (G96)**

G96 (constant surface speed control command) is a modal G code. After a G96 command is specified, the program enters the constant surface speed control mode (G96 mode) and specified S values are assumed as a surface speed. A G96 command must specify the axis along which constant surface speed control is applied. A G97 command cancels the G96 mode.When constant surface speed control is applied, a spindle speed higher than the value specified in G50S_; (maximum spindle speed) is clamped at the maximum spindle speed. When the power is turned on, the maximum spindle speed is not yet set and the speed is not clamped.S (surface speed) commands in the G96 mode are assumed as S = 0 (the surface speed is 0) until M03 (rotating the spindle in the positive direction) or M04 (rotating the spindle in the negative direction) appears in the program.

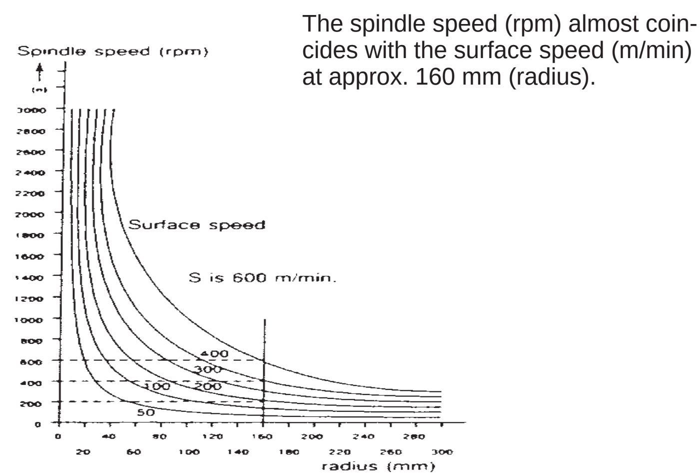


Fig. 10.3 (a) Relation between workpiece radius, spindle speed and surface speed

- **Setting the workpiece coordinate system for constant surface speed control**

To execute the constant surface speed control, it is necessary to set the work coordinate system , Z axis, (axis to which the constant surface speed control applies) becomes zero.

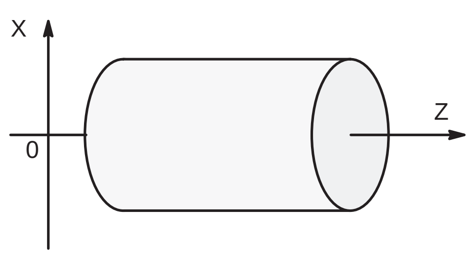
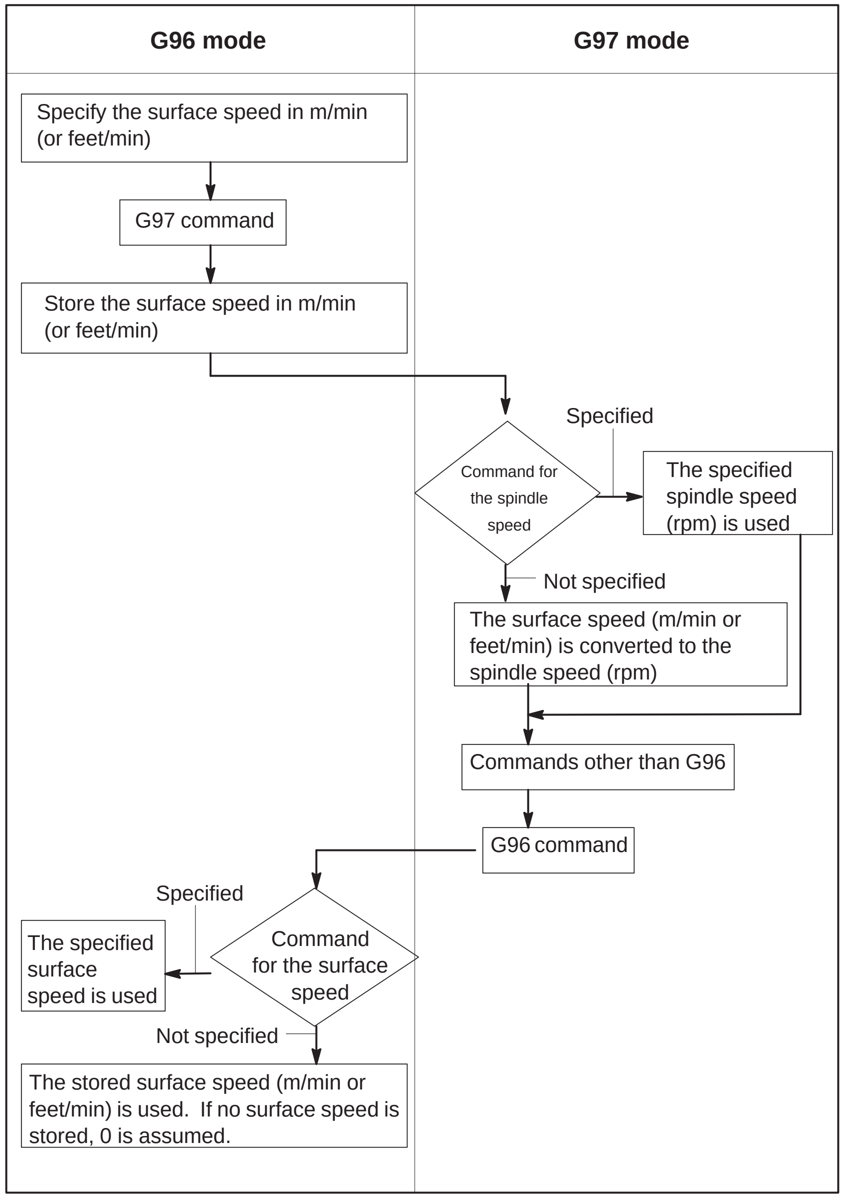


Fig. 10.3 (b) Example of the Workpiece Coordinate System for Constant Surface Speed Control

- **Surface speed specified in the G96 mode**



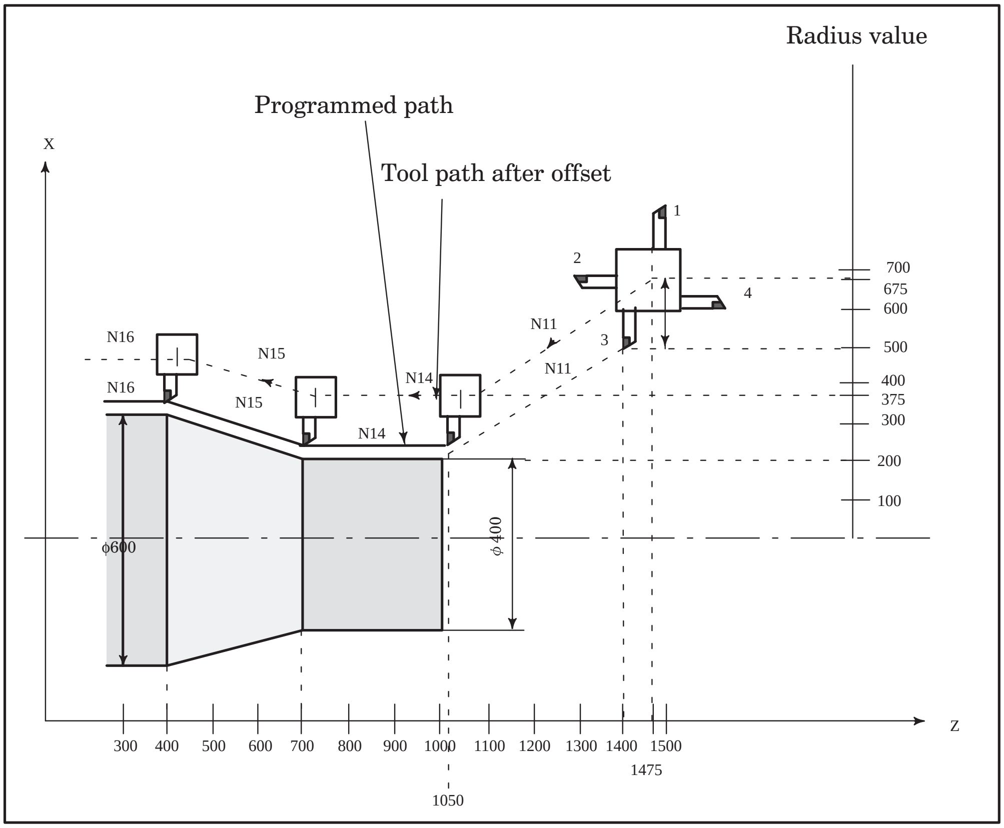
Restrictions

- **Constant surface speed control for threading**

The constant surface speed control is also effective during threading. Accordingly, it is recommended that the constant surface speed control be invalidated with G97 command before starting the scroll threading and taper threading, because the response problem in the servo system may not be considered when the spindle speed changes.

- **Constant surface speed control for rapid traverse (G00)**

In a rapid traverse block specified by G00, the constant surface speed control is not made by calculating the surface speed to a transient change of the tool position, but is made by calculating the surface speed based on the position at the end point of the rapid traverse block, on the condition that cutting is not executed at rapid traverse.



Example

N8 G00 X1000.0Z1400.0 ;
N9 T33;
N11 X400.0Z1050.0;
N12 G50S3000 ; (Designation of max. spindle speed)
N13 G96S200 ; (Surface speed 200m/min)
N14 G01 Z 700.0F1000 ;
N15 X600.0Z 400.0;
N16 Z … ;

The CNC calculates the spindle speed which is proportional to the specified surface speed at the position of the programmed coordinate value on the X axis. This is not the value calculated according to the X axis coordinate after offset when offset is valid. At the end point N15 in the example above, the speed at 600 dia. (Which is not the turret center but the tool nose) is 200 m/min. If X axis coordinate value is negative, the CNC uses the absolute value.

10.4 SPINDLE SPEED FLUCTUATION DETECTION FUNCTION (G25, G26)

With this function, an overheat alarm (No. 704) is raised when the spindle speed deviates from the specified speed due to machine conditions.
This function is useful, for example, for preventing the seizure of the guide bushing.

Format

G26 enables spindle speed fluctuation detection.
G25 disables spindle speed fluctuation detection.

G26 Pp Qq Rr ;	**Spindle fluctuation detection on**
G25 ;	**Spindle fluctuation detection off**

p : Time (in ms) from the issue of a new spindle rotation command (S command) to the start of checking whether the actual spindle speed is so fast that an overheat can occur.
When a specified speed is reached within the time period of P, spindle speed is checked at that time.

q : Tolerance (%) of a specified spindle speed

$$q = \frac{1\text{–}actual\ spindle\ speed}{specified\ spindle\ speed} \times 100$$

If a specified spindle speed lies within this range, it is regarded as having reached the specified value. Then, an actual spindle speed is checked.

r : Spindle speed fluctuation (%) at which the actual spindle speed is so fast that an overheat can occur

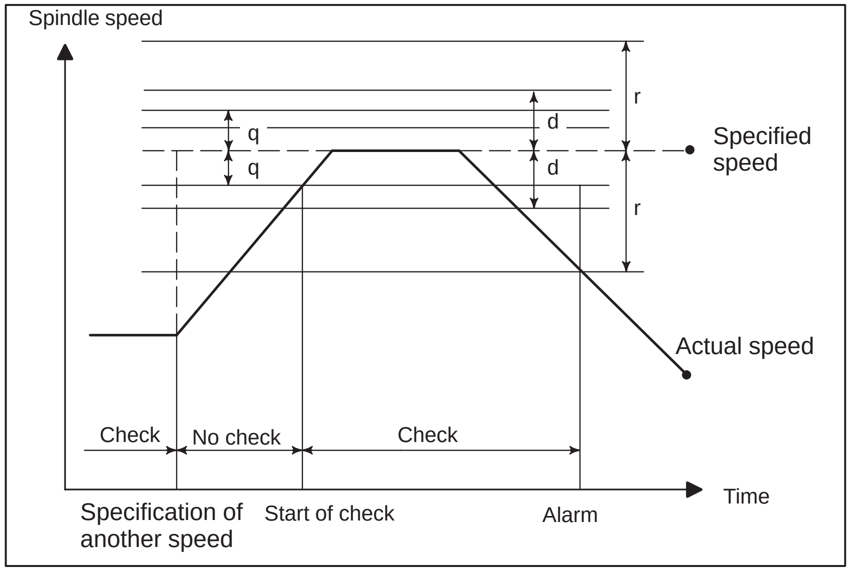
$$r = \frac{1\text{–}speed\ that\ can\ cause\ overheat}{specified\ spindle\ speed} \times 100$$

G26 enables the spindle speed fluctuation detection function, and G25 disables the spindle speed fluctuation detection.
Even if G25 is specified, p, q, and r are not cleared.

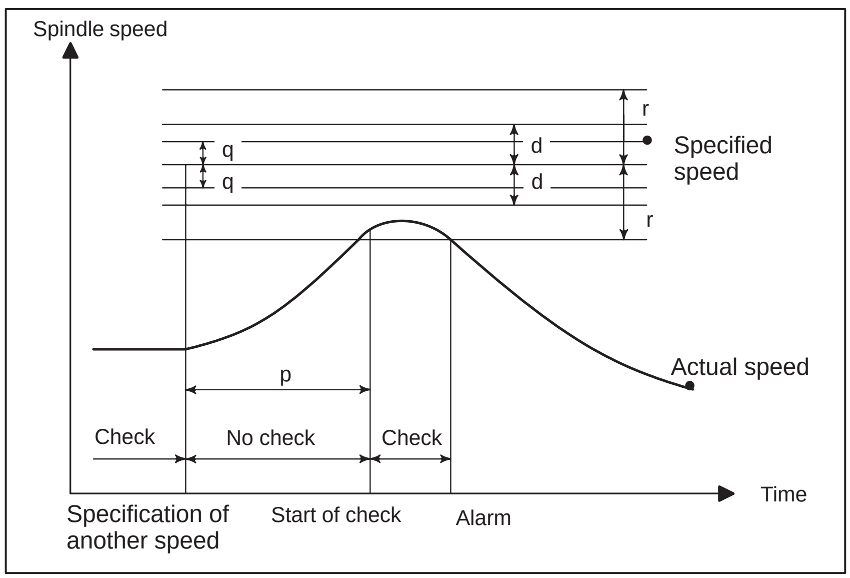
Explanations

The fluctuation of the spindle speed is detected as follows:

1. When an alarm is issued after a specified spindle speed is reached



2. When an alarm is issued before a specified spindle speed is reached



Specified speed :
(Speed specified by address S and five–digit value) × (spindle override)

Actual speed : Speed detected with a position coder

p : Time elapses since the specified speed changes until a check starts.

q : (Percentage tolerance for a check to start) × (specified speed)

r : (Percentage fluctuation detected as an alarm condition) × (specified speed)

d : Fluctuation detected as an alarm (specified in parameter 4913)

An alarm is issued when the difference between the specified speed and the actual speed exceeds both r and d.

Notes

1. When an alarm is issued in automatic operation, a single block stop occurs. The spindle overheat alarm is indicated on the CRT screen, and the alarm signal ”SPAL” is output (set to 1 for the presence of an alarm). This signal is cleared by resetting.
2. Even when reset operation is performed after an alarm occurs, the alarm is issued again unless the cause of the alarm is corrected.
3. No check is made during spindle stop state (*SSTP = 0).
4. By setting the parameter (No. 4913), an allowable range of speed fluctuations can be set which suppresses the occurrence of an alarm. However, an alarm is issued one second later if the actual speed is found to be 0 rpm.

10.5 SPINDLE POSITIONING FUNCTION

In turning, the spindle connected to the spindle motor is rotated at a certain speed to rotate the workpiece mounted on the spindle. The spindle positioning function turns the spindle connected to the spindle motor by a certain angle to position the workpiece mounted on the spindle at a certain angle. The spindle is positioned about the C–axis.
The spindle positioning function involves the following three operations :

1. Canceling the spindle rotation mode and entering the spindle!positioning mode (spindle orientation)
2. Positioning the spindle in the spindle positioning mode
3. Canceling the spindle positioning mode, and entering the spindle!rotation mode

10.5.1 Spindle orientation

When spindle positioning is first performed after the spindle motor is used for normal spindle operation, or when spindle positioning is interrupted, the spindle orientation is required.
Orientation permits the spindle to stop at a predetermined position.
Orientation is directed by the M code set in parameter No. 4960. The direction of orientation can be set with a parameter. For the analog spindle, the direction is set in ZMIx (bit 5 of parameter 1006).
For the serial spindle, it is set in RETRN (bit 5 of parameter 4005).

10.5.2 Spindle positioning

The spindle can be positioned with an arbitrary angle or semi–fixed angle.

- **Positioning with a semi–fixed angle specified by an M code**

Address M is followed by a 2–digit numeric. The specifiable value may be one of the six values from Mα to M(α+5). Value α must be set in parameter No. 4962 beforehand. The positioning angles corresponding to Mα to M(α+5) are listed below. Value β must be set in parameter No. 4963 beforehand.

M–code	Positioning angle	(Ex.)β=30,
Mα	β	30,
M(α+1)	2β	60,
M(α+2)	3β	90,
M(α+3)	4β	120,
M(α+4)	5β	150,
M(α+5)	6β	180,

Specify the command with incremental values. The direction of rotation can be specified in parameter IDM (bit 1 of parameter 4950).

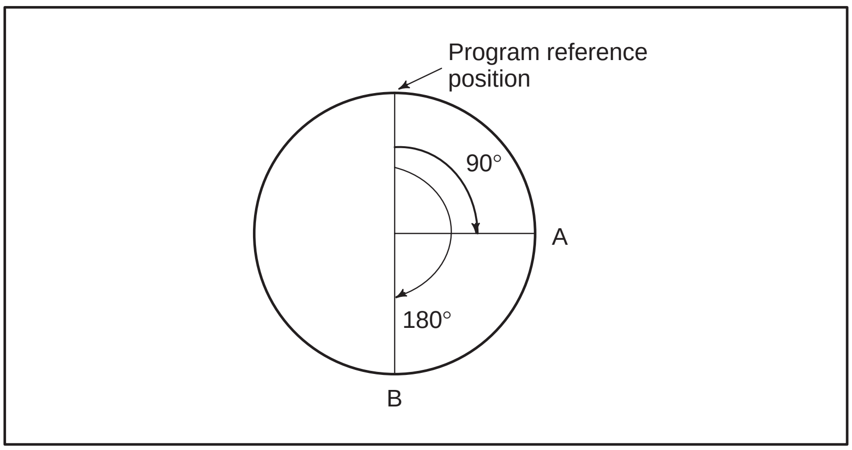
- **Positioning with a given angle specified by address C or H**

Specify the position using address C or H followed by a signed numeric value or numeric values. Addresses C and H must be specified in the G00 mode.
(Example) C–1000
H4500
The end point must be specified with a distance from the program reference position (in absolute mode) using address C. Alternatively, the end point must also be specified with a distance from the start point to the end point (in incremental mode) using address H.
A numeric with the decimal point can be entered.
The value must be specified in degrees.
(Example) C35.0=C35 degrees

- **Program reference position**

The position to which the spindle is oriented is assumed as the program reference position. The program reference position can be changed by setting of a coordinate system (G50) or automatic setting of a coordinate system (#OZPR of parameter 1202).

- **Feedrate for positioning**



Command format		G code A		G code B and C	
		Address used	Command A–B in the above figure	Address used and G code	Command A–B in the above figure
Absolute command	Specify the end point with a distance from the program reference position.	C	C180.0 ;	G90,C	G90C180.0;
Incremental command	Specify a distance from the start point to the end point.	H	H90.0 ;	G91,C	G90C90.0 ;

- **Feedrate during positioning**

The feedrate during positioning equals the rapid traverse speed specified in parameter No. 1420. Linear acceleration/deceleration is performed. For the specified speed, an override of 100%,50%,25%,and F0 (parameter No. 1421) can be applied.

- **Speed during orientation**

The tool moves at the rapid traverse speed set in parameter No.1420 until a sufficient speed for orientation is attained. After the speed for orientation has been attained, orientation is performed at the speed set in parameter No.1425.

10.5.3 Canceling spindle positioning

When modes are to be switched from spindle positioning to normal spindle rotation, the M code set in parameter No. 4961 is specified.

Notes

1. Specify spindle positioning alone in a block. A move command for the X or Z axis cannot be specified within the same block.
2. When emergency stop is applied during spindle positioning, spindle positioning stops. To resume it, restart with the orientation step.
3. Feed hold, dry run, machine lock, and auxiliary function lock cannot be performed during spindle positioning.
4. The serial spindle Cs–axis contour control function and the spindle positioning function cannot be used at a time. If both options are specified, the spindle positioning function has priority.
5. Parameter No. 4962 must always be set even when positioning with a semi–fixed angle specified in an M–code is not performed. If the parameter is not set, M–codes from the M00 to M05 do not function properly.
6. The spindle positioning axis is indicated in pulses in the machine coordinate system.

11 TOOL FUNCTION (T FUNCTION)

Two tool functions are available. One is the tool selection function, and the other is the tool life management function.

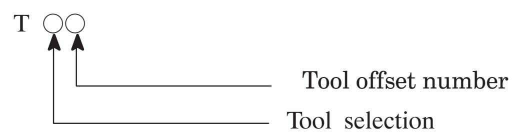
11.1 TOOL SELECTION

By specifying a 2–digit/4–digit numerical value following address T, a code signal and a strobe signal are transmitted to the machine tool.
This is mainly used to select tools on the machine.
One T code can be commanded in a block. Refer to the machine tool builder's manual for the number of digits commandable with address T and the correspondence between the T codes and machine operations.
When a move command and a T code are specified in the same block, the commands are executed in one of the following two ways:

1. Simultaneous execution of the move command and T function commands.
2. Executing T function commands upon completion of move command execution.

The selection of either sequence depends on the machine tool builder's specifications. Refer to the machine tool builder's manual for details.

1.Last one digit of T–code designates the offset number.



2.Last two digits of T–code designate the offset number.

Explanations

The value after the T code indicates the desired tool. Part of the values is also used as the offset number indicating the compensation amount for tool offset.
Refer to the machine tool builder's manual for correspondence between the T–code and the tool and the number of digit to specify tool selection.
Example(T2+2)
N1G00X1000Z1400
N2T0313; (Select Tool No. 3 and Offset value No.13)
N3X400Z1050;
Some machines use a 1–digit value to specify tool selection.

11.2 TOOL LIFE MANAGEMENT

Tools are classified into some groups. For each group, a tool life (time or frequency of use) is specified. Each time a tool is used, the time for which the tool is used is accumulated. When the tool life has been reached, the next tool previously determined in the same group is used. This function is called the tool life management function.
With 2–path control, tool life management is performed for each tool post separately. So tool life management data is also set for each tool post.

11.2.1 Program of Tool Life Data

Format

Tools used sequentially in each group and their tool life are registered in the CNC as following program format of table 11.2.1(a).

Table11.2.1(a) Program format of life management

Tape format	Meaning
O_ _ _ _ ;	Program number
G10L3;	Start of setting tool life data
P_ _ _ L_ _ _ _ ;	P___ :Group number (1 to 128) L___ :Tool life (1 to 9999)
T_ _ _ _ ; T_ _ _ _ ;	(1) T:____ Tool number (2) Tools are selected from (n) (1)to (2) to ... to (n).
P_ _ _ L_ _ _ _ ; T_ _ _ _ ; T_ _ _ _ ;	Data for the next group
G11;	End of setting tool life data
M02(M30);	End of program

For the method of registering tool life data in CNC ,refer to Subsec. III–11.4.14.

Explanations

A tool life is specified either as the time of use (in minutes) or the frequency of use, which depends on the parameter setting parameter No. 6800#2(LTM) .
Up to 4300 minutes in time or 9999 times in frequency can be specified for a tool life.
The number of groups to be registered and the number of tools registered per group can be combined in three ways. One of the three combinations is set by a parameter No.6800#0,#1(Each GS1 and GS2).

	GS2	GS1	Number of groups	Number of tools
(1)	0	0	16	16
(2)	0	1	32	8
(3)	1	0	64	4

In any combination, up to 256 tools in total can be registered.
If there are not more than 16 groups and each group contain not more than 16 tools, for example, select combination (2)(1).Select combination if there are not more than 32 groups and each group contain not more than eight tools. To change the combination, change the parameter, then set program is executed with the old tool group combination set in the NC. Whenever the parameter is changed, be sure to reexecute the group setting program.
The same tool number may appear anywhere any times in the program of tool life data.

Example

```
O0001 ;
G10L3 ;
P001L0150 ;
T0011 ;        } Data of group 1
T0132 ;
T0068 ;
P002L1400 ;
T0061;
T0241 ;        } Data of group 2
T0134;
T0074;
P003L0700 ;
T0012;         } Data of group 3
T0202 ;
G11 ;
M02 ;
```

Explanations

The group numbers specified in P need not be serial. They need not be assigned to all groups, either. When using two or more offset numbers for the same tool in the same process, set as follows;

Tape format	Meaning
P004L0500; T0101; T0105; T0108; T0206; T0203; T0202; T0209; T0304; T0309; P005L1200; T0405;	The tools in group 4 are used from (1) to (2) to (3). (1) Each tool is used 500 times (or for 500 minutes). (2) When this group is specified three times in one process, the offset numbers are selected in the following orders: (3) Tools (1): 01→05→08 Tools (2): 06→03→02→09 Tools (3): 04→09

11.2.2 COUNTING A TOOL LIFE

Explanation

- **When a tool life is specified as the time of use (in minutes)**

Between TΔΔ99(ΔΔ=Tool group number)and TΔΔ88 in a machining program, the time for which the tool is used in the cutting mode is counted at intervals of 4 seconds. The time taken for single–block stoppage, feed hold, rapid traverse, dwelling, and FIN wait is ignored.
Up to 4300 minutes can be specified for a life.

- **When a tool life is specified as the frequency of use**

Counting is performed for each process that is initiated by the cycle start of a machining program and ended when the NC is reset by the M02 or M03 command. The counters for tool groups used in a process are incremented by one. Even when the same group is specified more than once in one process, the counter is incremented only by one. Up to 9999 can be set for a tool life.
Counting of a tool life is performed for each group. The life counter contents are not erased even when the power of CNC is cut off.
When a life is specified as the frequency of use, apply an external reset (ERS) signal to the CNC when M02 or M30 is executed.

11.2.3 Specifying a tool group in a machining program

In machining programs, T codes are used to specify tool groups as follows:

Tape format	Meaning
:	
T△△99;	Ends the tool used by now, and starts to use the tool of the ΔΔgroup. ”99” distinguishes this specification from ordinary specification.
:	
T△△88;	Cancels the offset of the tool of the group. ”88” distin–guishes this specification from ordinary specification.
:	
M02(M300);	Ends the machining program.

Explanations

Tape format	Meaning
T0199;	Ends the previous tool, and starts to use the tool of the 01 group.
:	
T0188;	Cancels the offset of the tool of the 01 group.
:	
T0508;	Ends the tool of the 01 group. Selects tool number 05 and offset number 08.
:	
T0500;	Cancels the offset of tool number 05.
:	
T0299;	Ends tool number 05, and starts to use the tool of the 02 group.
:	
T0199;	Ends the tool of the 02 group, and starts to use the tool of the 01 group. If more than one offset number is specified for the tool, the second offset number is selected. Otherwise, the previous offset number is used.
:	

12 AUXILIARY FUNCTION

There are two types of auxiliary functions ; miscellaneous function (M code) for specifying spindle start, spindle stop program end, and so on, and secondary auxiliary function (B code) .

When a move command and miscellaneous function are specified in the same block, the commands are executed in one of the following two ways:

i) Simultaneous execution of the move command and miscellaneous function commands.

ii) Executing miscellaneous function commands upon completion of move command execution.

The selection of either sequence depends on the machine tool builder's specification. Refer to the manual issued by the machine tool builder for details.

12.1 AUXILIARY FUNCTION (M FUNCTION)

When address M followed by a number is specified, a code signal and strobe signal are transmitted. These signals are used for turning on/off the power to the machine.
In general, only one M code is valid in a block but up to three M codes can be specified in a block (although some machines may not allow that).
The correspondence between M codes and functions is up to the machine tool builder.
All M codes are processed in the machine except for M98, M99,M198, M codes for calling a subprogram (parameters Nos. 6071 to 6079), and M codes for calling a custom macro (parameters Nos. 6080 to 6089). Refer to the appropriate manual issued by the machine tool builder.

Explanations

The following M codes have special meanings.

- **M02,M03 (End of program)**

This indicates the end of the main program
Automatic operation is stopped and the CNC unit is reset. This differs with the machine tool builder. After a block specifying the end of the program is executed,control returns to the start of the program. Bit 5 of parameter No.3404 (M02) or bit 4 of parameter No.30404 (M03)can be used to disable M02 from returning control to the start of the program.

- **M00 (Program stop)**

Automatic operation is stopped after a block containing M00 is executed. When the program is stopped, all existing modal information remains unchanged. The automatic operation can be restarted by actuating the cycle operation. This differs with the machine tool builder.

- **M01 (Optional stop)**

Similarly to M00, automatic operation is stopped after a block containing M01 is executed. This code is only effective when the Optional Stop switch on the machine operator’s panel has been pressed.

- **M98 (Calling of sub–program)**

This code is used to call a subprogram. The code and strobe signals are not sent. See the subprogram section II–13.3 for details .

- **M99 (End of subprogram)**

This code indicates the end of a subprogram.
M99 execution returns control to the main program. No code or strobe signal is sent. See the subprogram section II–13.3 for details.

Notes
A block immediately after an M00, M01, M02, or M03 block is not buffered. Similarly, ten M codes which do not buffer can be set by parameters (Nos. 3411 to 3421). Refer to the machine tool builder’s instruction manual for these M codes.

12.2 MULTIPLE M COMMANDS IN A SINGLE BLOCK

So far, one block has been able to contain only one M code. Up to three M codes can be specified in a single block when bit 7 (M3B) of parameter No. 3404 is set to 1.
Up to three M codes specified in a block are simultaneously output to the machine. This means that compared with the conventional method of a single M command in a single block, a shorter cycle time can be realized in machining.

Explanations

CNC allows up to three M codes to be specified in one block. However, some M codes cannot be specified at the same time due to mechanical operation restrictions. For detailed information about the mechanical operation restrictions on simultaneous specification of multiple M codes in one block, refer to the manual of each machine tool builder.
M00, M01, M02, M30, M98, M99, or M198 must not be specified together with another M code.
Some M codes other than M00, M01, M02, M30, M98, M99, and M198 cannot be specified together with other M codes; each of those M codes must be specified in a single block.
Such M codes include these which direct the CNC to perform internal operations in addition to sending the M codes themselves to the machine. To be specified, such M codes are M codes for calling program numbers 9001 to 9009 and M codes for disabling advance reading (buffering) of subsequent blocks. Meanwhile, multiple of M codes that direct the CNC only to send the M codes themselves (without performing internal operations) can be specified in a single block.

Examples

One M command in a single block	Multiple M commands in a single block
M40 ; M50 ; M60 ; G28G91X0Z0 ; : : :	M40M50M60 ; G28G91X0Z0 ; : : : : :

12.3 M CODE GROUP CHECK FUNCTION

The M code group check function checks if a combination of multiple M codes (up to three M codes) contained in a block is correct.
This function has two purposes. One is to detect if any of the multiple M codes specified in a block include an M code that must be specified alone. The other purpose is to detect if any of the multiple M codes specified in a block include M codes that belong to the same group. In either of these cases, P/S alarm No. 5016 is issued.
For details on group data setting, refer to the manual available from the machine tool builder.

Explanations

- **M code setting**

Up to 500 M codes can be specified. In general, M0 to M99 are always specified. M codes from M100 and up are optional.

- **Group numbers**

Group numbers can be set from 0 to 127. Note, however, that 0 and 1 have special meanings. Group number 0 represents M codes that need not be checked. Group number 1 represents M codes that must be specified alone.

12.4 THE SECOND AUXILIARY FUNCTIONS (B CODES)

Indexing of the table is performed by address B and a following 8–digit number. The relationship between B codes and the corresponding indexing differs between machine tool builders.
Refer to the manual issued by the machine tool builder for details.

Explanations

- **Command range**

0 to 99999999

- **Command method**

1. The decimal point can be used for input.
   Command Output value
   B10. 10000
   B10 10
2. It is possible to change over the scale factor of B output, 1000 or 1 when the decimal point input is omitted, using the parameter DPI (No.3401#0).
   Command Output value
   When DPI is 1: B1 1000
   When DPI is 0: B1 1
3. It is possible to change over the scale factor of B output 1000 or 10000 when the decimal point input is omitted in the inch input system, using the parameter AUX (No.3405#0) When DPI=1.
   Command Output value
   When AUX is 1: B1 10000
   When AUX is 0: B1 1000

Restrictions

When this functions is used, the B address specifying an axis movement disabled.

13 PROGRAM CONFIGURATION

General

- **Main program and subprogram**

There are two program types, main program and subprogram. Normally, the CNC operates according to the main program. However, when a command calling a subprogram is encountered in the main program, control is passed to the subprogram. When a command specifying a return to the main program is encountered in a subprogram, control is returned to the main program.

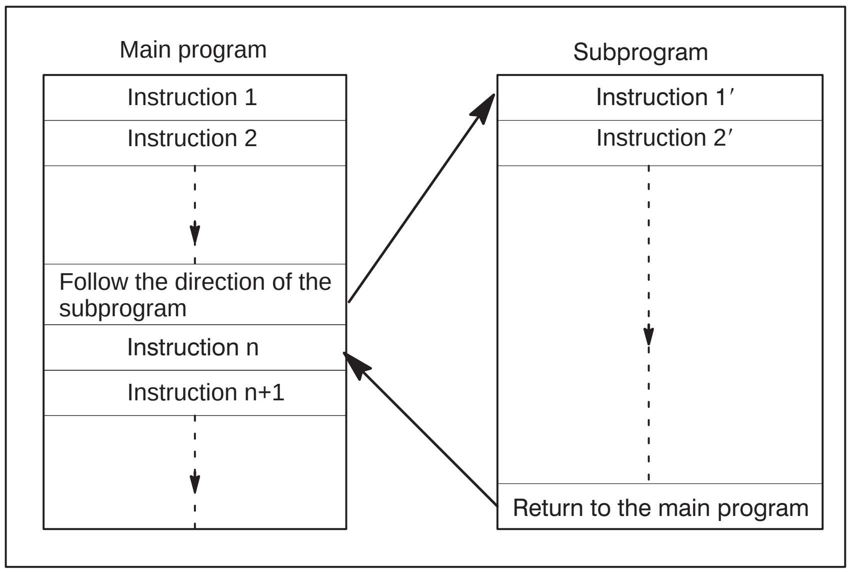


Fig. 13 (a) Main program and Subprogram

The CNC memory can hold up to 400 main programs and subprograms (63 as standard). A main program can be selected from the stored main programs to operate the machine. See Chapter III–10 for the methods of registering and selecting programs.

- **Program components**

A program consists of the following components:

Table 13(a) Program components

Components	Descriptions
Tape start	Symbol indicating the start of a program file
Leader section	Used for the title of a program file, etc.
Program start	Symbol indicating the start of a program
Program section	Commands for machining
Comment section	Comments or directions for the operator
Tape end	Symbol indicating the end of a program file

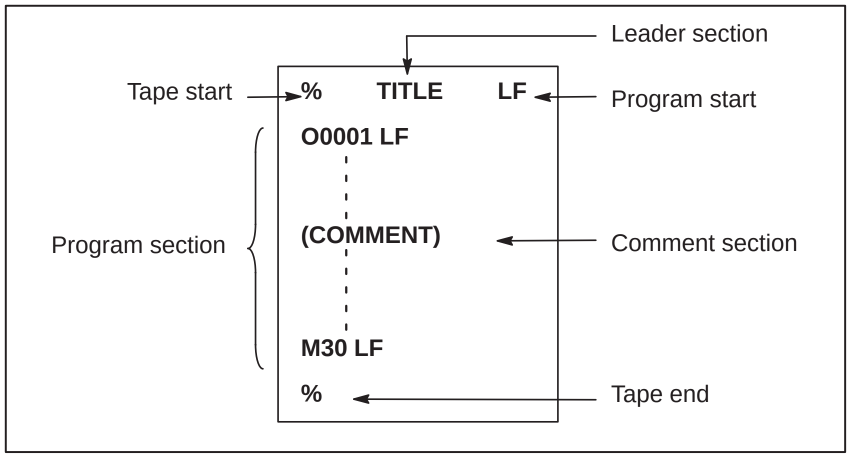


Fig. 13(b) Program configuration (Example of using ISO code)

- **Program section configuration**

A program section consists of several blocks. A program section starts with a program number and ends with a program end code.

Program section configuration	**Program section (Example of using ISO code)**
Program number	O0001 LF
Block 1	N1 G91 G00 X120.0 Y80.0 LF
Block 2	N2 G43 Z–32.0 H01 LF
:	:
Block n	Nn Z0 LF
Program end	M30 LF

A block contains information necessary for machining, such as a move command or coolant on/off command.Specifying a value following a slash (/) at the start of a block disables the execution of some blocks (see ”optional block skip” in Section II–13.2).

13.1 PROGRAM COMPONENTS OTHER THAN PROGRAM SECTIONS

This section describes program components other than program sections. See Section II–13.2 for a program section.

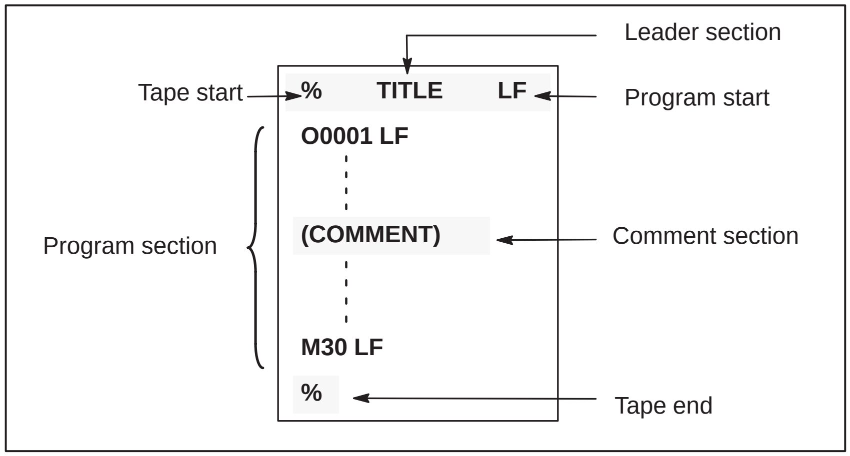


Fig. 13.1(a) Program configuration (Example of using ISO code)

Explanations

- **Tape start**

The tape start indicates the start of a file that contains CNC programs.
The mark is not required when programs are entered using SYSTEM P or ordinary personal computers. The mark is not displayed on the CRT display screen. However, if the file is output,the mark is automatically output at the start of the file.

Table 13.1(a) Code of a tape start

Name	ISO code	EIA code	Notation in this manual
Tape start	%	ER	%

- **Leader section**

Data entered before the programs in a file constitutes a leader section.
When machining is started, the label skip state is usually set by turning on the power or resetting the system. In the label skip state, all information is ignored until the first end–of–block code is read. When a file is read into the CNC unit from an I/O device, leader sections are skipped by the label skip function.
A leader section generally contains information such as a file header. When a leader section is skipped, even a TV parity check is not made. So a leader section can contain any codes except the EOB code.

- **Program start**

The program start code is to be entered immediately after a leader section, that is, immediately before a program section. This code indicates the start of a program, and is always required to disable the label skip function.
With SYSTEM P or ordinary personal computers, this code can be entered by pressing the return key.

Table 13.1(b) Code of a program start

Name	ISO code	EIA code	Notation in this manual
Program start	LF	CR	;

Notes
If one file contains multiple programs, the EOB code for label skip operation must not appear before a second or subsequent program number. However, an program start is required at the start of a program if the preceding program ends with %.

- **Comment section**

Any information enclosed by the control–out and control–in codes is regarded as a comment and skipped by the CNC.The user can enter a header, comments, directions to the operator, etc. in a comment section using the EOB code or any other code.There is no limit on the length of a comment section.

Table 13.1(c) Codes of a control–in and a control–out

Name	ISO code	EIA code	Notation in this manual	Meaning
Control–out	(	2–4–5	(	Start of comment section
Control–in	)	2–4–7	)	End of comment section

When a program is read into memory for memory operation, comment sections, if any, are not ignored but are also read into memory. Note, however, that codes other than those listed in the code table in Appendix F are ignored, and thus are not read into memory. When the program in this memory is output to an external input/output device (see Section III–8), any comments are also output.
When a program is displayed on the screen, its comment sections are also displayed. However, those codes that were ignored when read into memory are not outputted or displayed.
During memory operation or DNC operation, all comment sections are ignored.
The TV check function can be used for a comment section by setting parameter CTV (bit 1 of No. 0100).

Notes
1. If a long comment section appears in the middle of a program section, a move along an axis may be suspended for a long time because of such a comment section. So a comment section should be placed where movement suspension may occur or no movement is involved.
2. If only a control–in code is read with no matching control–out code, the read control–in code is ignored.

• Tape end

A tape end is to be placed at the end of a file containing NC programs. If programs are entered using the automatic programming system, the mark need not be entered. The mark is not displayed on the CRT display screen. However, when a file is output, the mark is automatically output at the end of the file.
If an attempt is made to execute % when M02 or M03 is not placed at the end of the program, the P/S alarm (No. 5010) is occurred.

Table 13.1(d) Code of a tape end

Name	ISO code	EIA code	Notation in this manual
Tape end	%	ER	%

13.2 PROGRAM SECTION CONFIGURATION

This section describes elements of a program section. See Section II–13.1 for program components other than program sections.

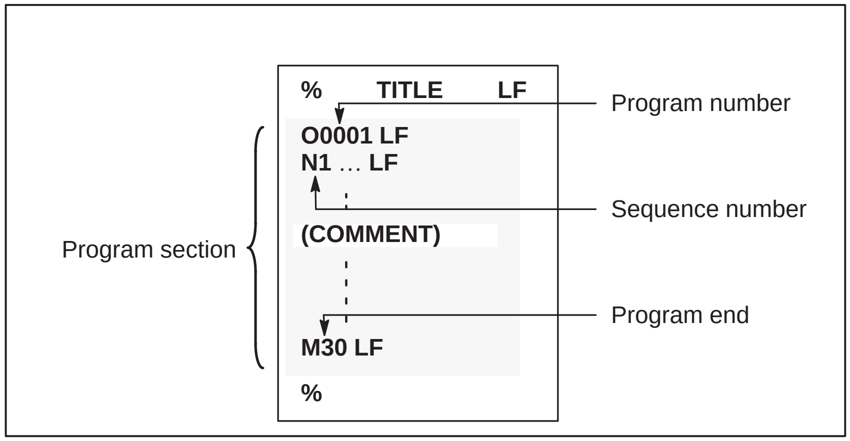


Fig. 13.2(a) Program configuration (Example of using ISO code)

- **Program number**

A program number consisting of address O followed by a four–digit number is assigned to each program at the beginning registered in memory to identify the program.
In ISO code, the colon (:) can be used instead of O.
When no program number is specified at the start of a program, the sequence number (N....) at the start of the program is regarded as its program number. If a five–digit sequence number is used, the lower four digits are registered as a program number. If the lower four digits are all 0, the program number registered immediately before added to 1 is registered as a program number. Note, however, that N0 cannot be used for a program number.
If there is no program number or sequence number at the start of a program, a program number must be specified using the CRT/MDI panel when the program is stored in memory(See Section 9.3 in Part III.).

Notes
Program numbers 8000 to 9999 may be used by machine tool builders, and the user may not be able to use these numbers.

- **Sequence number and block**

A program consists of several commands. One command unit is called a block. One block is separated from another with an EOB of end of block code.

Table 13.2(a) EOB code

Name	ISO code	EIA code	Notation in this manual
End of block (EOB)	LF	CR	;

At the head of a block, a sequence number consisting of address N followed by a number not longer than five digits (1 to 99999) can be placed. Sequence numbers can be specified in a random order, and any numbers can be skipped. Sequence numbers may be specified for all blocks or only for desired blocks of the program. In general, however, it is convenient to assign sequence numbers in ascending order in phase with the machining steps (for example, when a new tool is used by tool replacement, and machining proceeds to a new surface with table indexing.)

```
N300 X200.0 Z300.0 ;   A sequence number is underlined.
```

Fig. 13.2(b) Sequence number and block (example)

Notes

N0 must not be used for the reason of file compatibility with other CNC systems.

Program number 0 cannot be used. So 0 must not be used for a sequence number regarded as a program number.

- **TV check (Vertical parity check along tape)**

A parity check is made for a block on input tape vertically. If the number of characters in one block (starting with the code immediately after an EOB and ending with the next EOB) is odd, an P/S alarm (No.002) is output. No TV check is made only for those parts that are skipped by the label skip function. A comment section enclosed in parentheses is also subject to TV check to count the number of characters. The TV check function can be enabled or disabled by setting on the MDI unit (See subsec. 11.4.7 in Part III.).

- **Block configuration (word and address)**

A block consists of one or more words. A word consists of an address followed by a number some digits long. (The plus sign (+) or minus sign (–) may be prefixed to a number.)

Word = Address + number (Example : X–1000)

For an address, one of the letters (A to Z) is used ; an address defines the meaning of a number that follows the address. Table 13.2 (b) indicates the usable addresses and their meanings.

The same address may have different meanings, depending on the preparatory function specification.

Table 13.2(b) Major functions and addresses

Function	Address	Meaning
Program number	O (1)	Program number
Sequence number	N	Sequence number
Preparatory function	G	Specifies a motion mode (linear, arc, etc.)
Dimension word	X, Y, Z, U, V, W, A, B, C	Coordinate axis move command
	I, J, K	Coordinate of the arc center
	R	Arc radius
Feed function	F	Rate of feed per minute, Rate of feed per revolution
Spindle speed function	S	Spindle speed
Tool function	T	Tool number
Auxiliary function	M	On/off control on the machine tool
	B	Table indexing, etc.
Dwell	P, X, U	Dwell time
Program number designation	P	Subprogram number
Number of repetitions	P	Number of subprogram repetitions
Parameter	P, Q	Canned cycle parameter

Notes

1. In ISO code, the colon (:) can also be used as the address of a program number.

N_	G_	X_ Z_	F_	S_	T_	M_	;
Sequence number	Preparatory function	Dimension word	Feed–function	Spindle speed function	Tool function	Miscellaneous function	

Fig. 13.2 (c) 1 block (example)

- **Major addresses and ranges of command values**

Major addresses and the ranges of values specified for the addresses are shown below. Note that these figures represent limits on the CNC side, which are totally different from limits on the machine tool side. For example, the CNC allows a tool to traverse up to about 100 m (in millimeter input) along the X axis.
However, an actual stroke along the X axis may be limited to 2 m for a specific machine tool.
Similarly, the CNC may be able to control a cutting federate of up to 240 m/min, but the machine tool may not allow more than 3 m/min. When developing a program, the user should carefully read the manuals of the machine tool as well as this manual to be familiar with the restrictions on programming.

Table 13.2(c) Major addresses and ranges of command values

Function		Address	Input in mm	Input in inch
Program number		O (1)	1–9999	1–9999
Sequence number		N	1–99999	1–99999
Preparatory function		G	0–99	0–99
Dimen-sion word	Increment system IS–B	X, Y, Z, U, V, W, A, B, C, I, J, K, R,	–99999.999 – +99999.999	–9999.9999– +9999.9999
	Increment system IS–C		–9999.9999– +9999.9999	–999.99999– +999.99999
Feed per minute	Increment system IS–B	F	1–240000mm/min	0.01–9600.00 inch/min
	Increment system IS–C		1–100000mm/min	0.01–4000.00 inch/min
Feed per revolution		F	0.01–500.00 mm/rev	0.0001–9.9999 inch/rev
Spindle speed function		S	0–20000	0–20000
Tool function		T	0–99999999	0–99999999
Auxiliary function		M	0–99999999	0–99999999
		B	0–99999999	0–99999999
Dwell	Increment system IS–B	P, X, U	0–99999.999s	0–99999.999s
	Increment system IS–C		0–9999.9999s	0–9999.9999s
Designation of a pro-gram number		P	1–9999	1–9999
Number of repetitions		P	1–9999	1–9999

Note
1. In ISO code, the colon (:) can also be used as the address of a program number.

- **Optional block skip**

When a slash followed by a number (/n (n=1 to 9)) is specified at the head of a block, and optional block skip switch n on the machine operator panel is set to on, the information contained in the block for which /n corresponding to switch number n is specified is ignored in tape operation or memory operation.

When optional block skip switch n is set to off, the information contained in the block for which /n is specified is valid. This means that the operator can determine whether to skip the block containing /n.

Number 1 for /1 can be omitted. However, when two or more optional block skip switches are used for one block, number 1 for /1 cannot be omitted.

Example)

```
(Incorrect)     (Correct)
//3 G00X10.0;   /1/3 G00X10.0;
```

This function is ignored when programs are loaded into memory. Blocks containing /n are also stored in memory, regardless of how the optional block skip switch is set.

Programs held in memory can be output, regardless of how the optional block skip switches are set.

Optional block skip is effective even during sequence number search operation.

Depending on the machine tool, all optional block skip switches (1 to 9) may not be usable. Refer to manuals of the machine tool builder to find which switches are usable.

Notes

1. ***Position of a slash***

A slash (/) must be specified at the head of a block. If a slash is placed elsewhere, the information from the slash to immediately before the EOB code is ignored.

2. ***TV and TH check***

When an optional block skip switch is on. TH and TV checks are made for the skipped portions in the same way as when the optional block skip switch is off.

3. ***Disabling an optional block skip switch***

Optional block skip operation is processed when blocks are read from memory or tape into a buffer. Even if a switch is set to on after blocks are read into a buffer, the blocks already read are not ignored.

Program end

The end of a program is indicated by punching one of the following codes at the end of the program:

Table 13.2(d) Code of a program end

Code	Meaning usage
M02	For main program
M30	
M99	For subprogram

If one of the program end codes is executed in program execution, the CNC terminates the execution of the program, and the reset state is set. When the subprogram end code is executed, control returns to the program that called the subprogram.

Notes

A block containing an optional block skip code such as /M02 ; , /M30 ; , or /M99 ; is not regarded as the end of a program, if the optional block skip switch on the machine operator's panel is set to on.
(See Section 13.2 for optional block skip.)

13.3 SUBPROGRAM

If a program contains a fixed sequence or frequently repeated pattern, such a sequence or pattern can be stored as a subprogram in memory to simplify the program.
A subprogram can be called from the main program.
A called subprogram can also call another subprogram.

Format

- **Subprogram configuration**

One subprogram

```
O □□□□ ;      Subprogram number
                (or the colon (:) optionally in the case of ISO)
  :
  :
M99 ;          Program end
```

M99 need not constitute a separate block as indicated below.
Example) **X100.0 Y100.0 M99 ;**

- **Subprogram call**

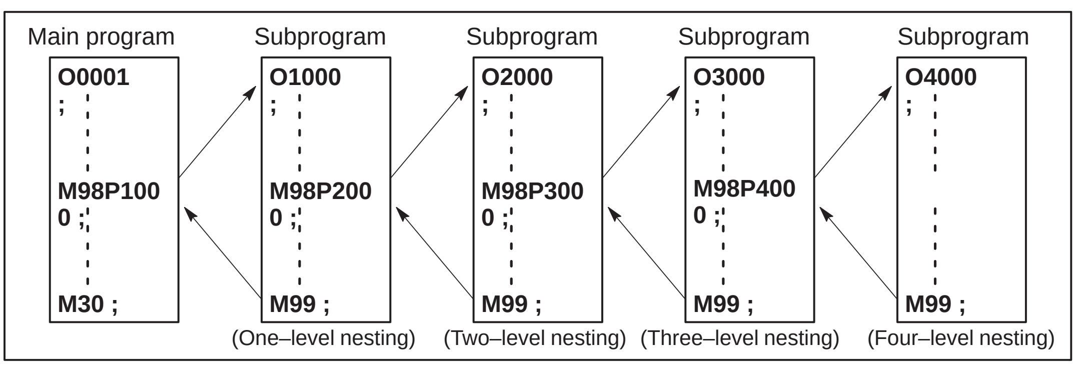
```
M98 P ○○○○ ○○○○ ;
      ↑          ↑
      Number of times the subprogram is called repeatedly
                 Subprogram number
```

Number of times the subprogram is called repeatedly / **Subprogram number**

When no repetition data is specified, the subprogram is called just once.

Explanations

When the main program calls a subprogram, it is regarded as a one–level subprogram call. Thus, subprogram calls can be nested up to four levels as shown below.



A single call command can repeatedly call a subprogram up to 9999 times.
For compatibility with automatic programming systems, in the first block, Nxxxx can be used instead of a subprogram number that follows O (or :). A sequence number after N is registered as a subprogram number.

- **Reference**

See Chapter 10 in Part III for the method of registering a subprogram.

Notes

> **Notes**
> 1, The M98 and M99 signals are not output to the machine tool.
> 2. If the subprogram number specified by address P cannot be found, an alarm (No. 078) is output.

Examples

☆ **M98 P51002 ;**

This command specifies ”Call the subprogram (number 1002) five times in succession.” A subprogram call command (M98P_) can be specified in the same block as a move command.

☆ **X1000.0 M98 P1200 ;**

This example calls the subprogram (number 1200) after an X movement.

☆ Execution sequence of subprograms called from a main program

Main program	1 2 3	Subprogram
N0010 ;		**O1010 ;**
N0020 ;		**N1020 ;**
N0030 M98 P21010 ;		**N1030 ;**
N0040 ;		**N1040 ;**
N0050 M98 P1010 ;		**N1050 ;**
N0060 ;		**N1060 M99 ;**

A subprogram can call another subprogram in the same way as a main program calls a subprogram.

Special Usage

- **Specifying the sequence number for the return destination in the main program**

If P is used to specify a sequence number when a subprogram is terminated, control does not return to the block after the calling block, but returns to the block with the sequence number specified by P. Note, however, that P is ignored if the main program is operating in a mode other than memory operation mode.
This method consumes a much longer time than the normal return method to return to the main program.

Main program	Subprogram
N0010 … ;	**O0010** … ;
N0020 … ;	**N1020** … ;
N0030 M98 P1010 ;	**N1030** … ;
N0040 … ;	**N1040** … ;
N0050 … ;	**N1050** … ;
N0060 … ;	**N1060 M99 P0060 ;**

• Using M99 in the main program

If M99 is executed in a main program, control returns to the start of the main program. For example, M99 can be executed by placing /M99 ; at an appropriate location of the main program and setting the optional block skip function to off when executing the main program. When M99 is executed, control returns to the start of the main program, then execution is repeated starting at the head of the main program.

Execution is repeated while the optional block skip function is set to off. If the optional block skip function is set to on, the /M99 ; block is skipped ; control is passed to the next block for continued execution.

If/M99Pn ; is specified, control returns not to the start of the main program, but to sequence number n. In this case, a longer time is required to return to sequence number n.

```
                        N0010 … ;
                        N0020 … ;
                        N0030 … ;
                        N0040 … ;
Optional block skip     N0050 … ;
OFF                   / N0060 M99 P0030 ;      Optional block
                        N0070 … ;              skip
                        N0080 M02 ;            ON
```

• Using a subprogram only

A subprogram can be executed just like a main program by searching for the start of the subprogram with the MDI.

(See Section 9.4 in Part III for information about search operation.)

In this case, if a block containing M99 is executed, control returns to the start of the subprogram for repeated execution. If a block containing M99Pn is executed, control returns to the block with sequence number n in the subprogram for repeated execution. To terminate this program, a block containing /M02 ; or /M30 ; must be placed at an appropriate location, and the optional block switch must be set to off ; this switch is to be set to on first.

```
  N1010 … ;
  N1020 … ;
  N1030 … ;
  N1040 M02 ;
/ N1050 M99 P1020 ;      Optional block skip
                         ON
```

14 FUNCTIONS TO SIMPLIFY PROGRAMMING

General

This chapter explains the following items:

14.1 CANNED CYCLE
14.2 MULTIPLE REPETITIVE CYCLE (G70AG76)
14.3 CANNED CYCLE FOR DRILLING (G80AG89)
14.4 CANNED GRINDING CYCLE (FOR 16/18–GCA ONLY)
14.5 CHAMFERING AND CORNER R
14.6 MIRROR IMAGE FOR DOUBLE TURRET (G68, G69)
14.7 DIRECT DRAWING DIMENSIONS PROGRAMMING

Notes

Explanatory diagrams in this chapter uses diameter programming in X axis.
In radius programming, changes U/2 with U and X/2 with X.

14.1 CANNED CYCLE (G90, G92, G94)

There are three canned cycles : the outer diameter/internal diameter cutting canned cycle (G90), the thread cutting canned cycle (G92), and the end face turning canned cycle (G94).

14.1.1 Outer diameter / internal diameter cutting cycle (G90)

- **Straight cutting cycle**

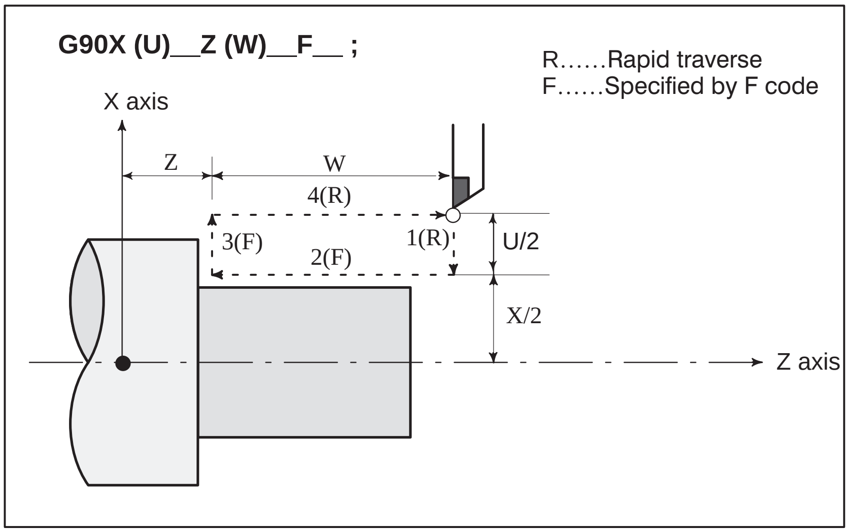


Fig.14.1.1 (a) Straight Cutting Cycle

In incremental programming, the sign of the numbers following address U and W depends on the direction of paths 1 and 2. In the cycle of 14. 1 1 (a), the signs of U and W are negative.
In single block mode, operations 1, 2, 3 and 4 are performed by pressing the cycle start button once.

- **Taper cutting cycle**

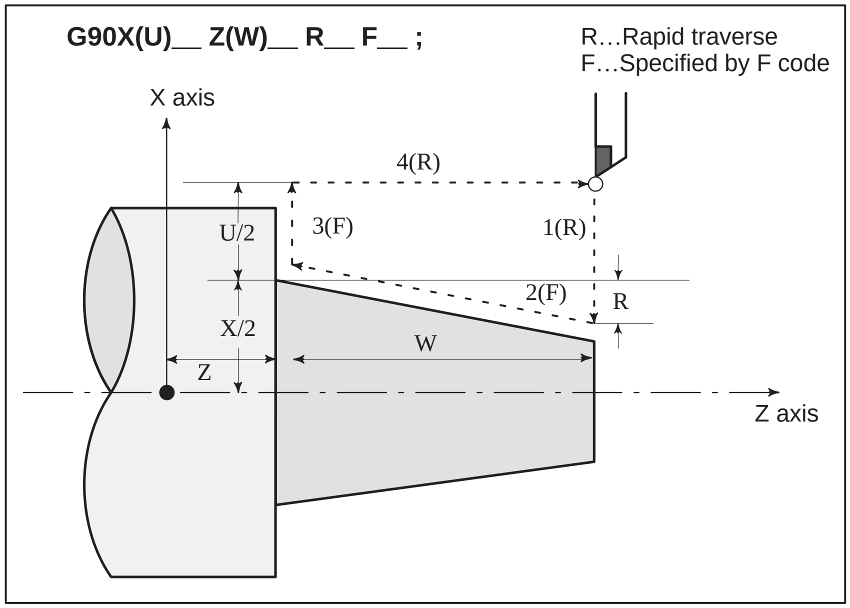
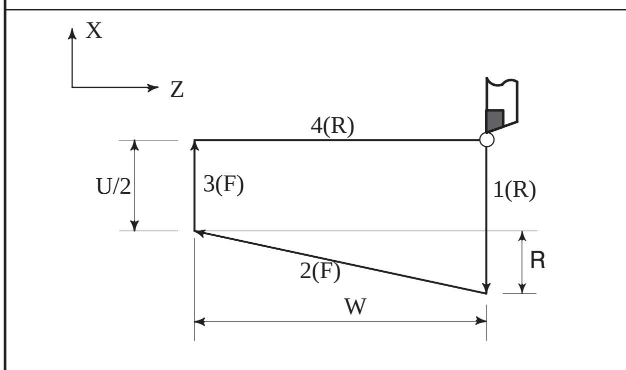
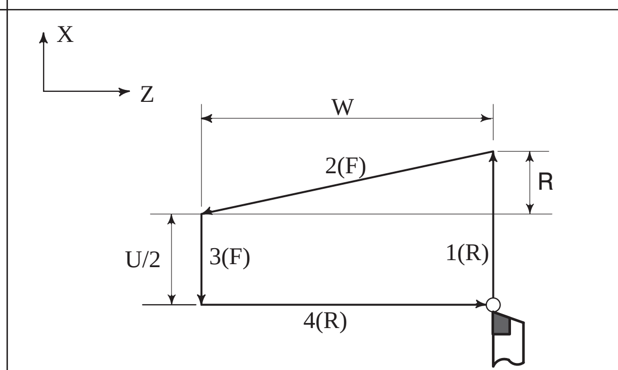
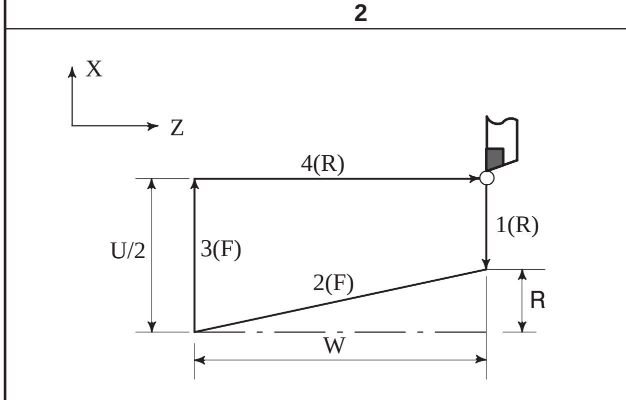
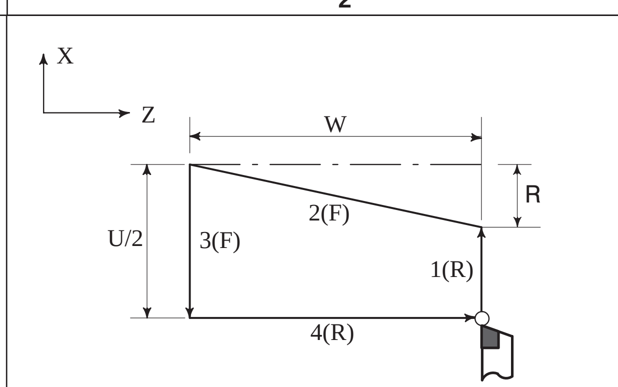


Fig. 14.1.1 (b) Taper Cutting Cycle

- **Signs of numbers specified in the taper cutting cycle**

In incremental programming, the relationship between the signs of the numbers following address U, W, and R, and the tool paths are as follows:

1. U < 0, W < 0, R < 0	2. U > 0, W < 0, R > 0
	
3. U < 0, W < 0, R > 0 at CRC≦C C $\frac{U}{2}$	**4. U > 0, W < 0, R<0 at CRC≦C C $\frac{U}{2}$**
	

14.1.2 Thread cutting cycle (G92)

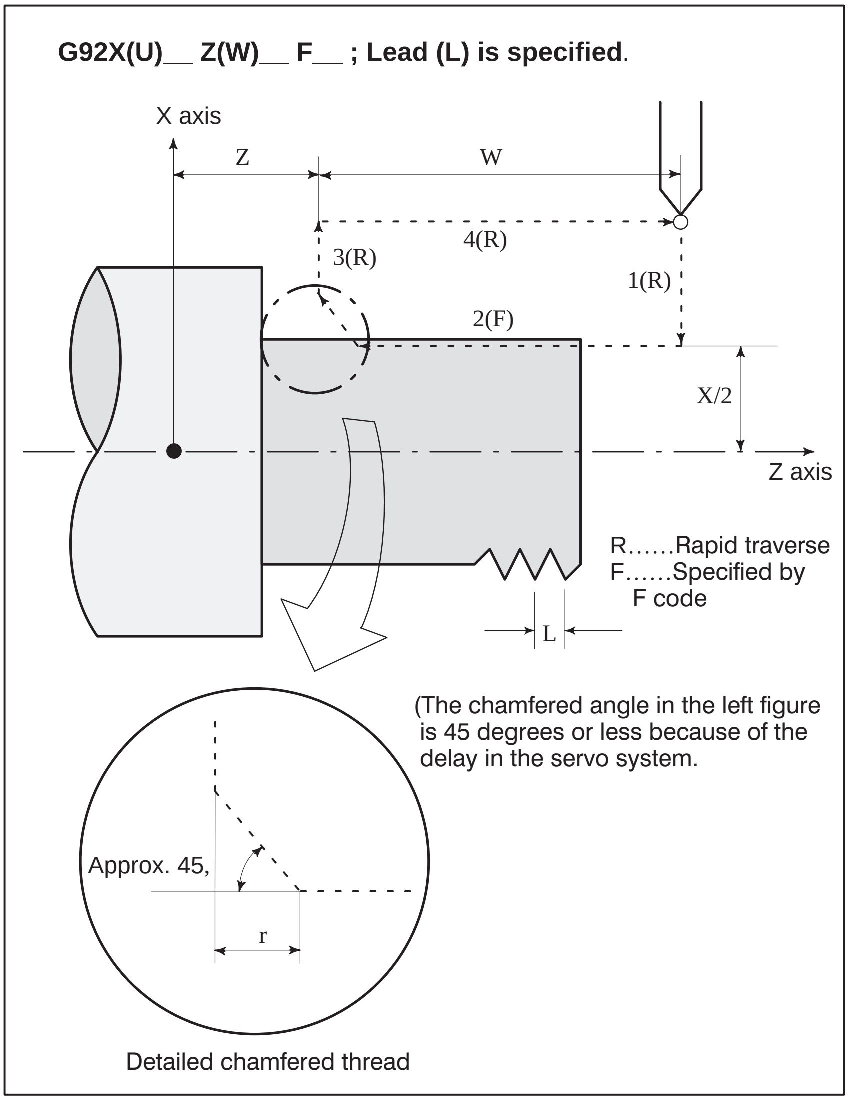
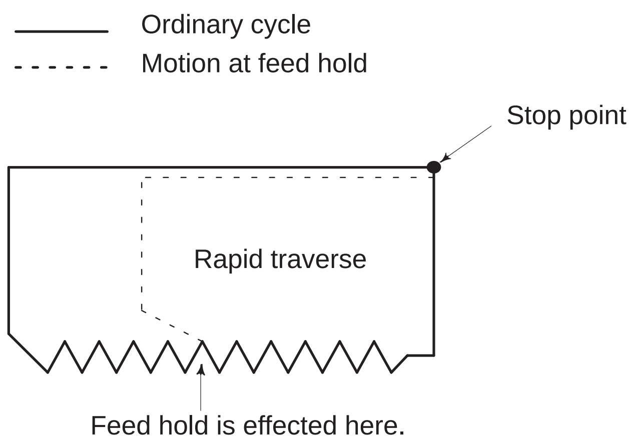


Fig. 14.1.2 (a) Straight Thread Cutting

In incremental programming, the sign of numbers following addresses U and W depends on the direction of paths 1 and 2. That is , if the direction of path 1 is the negative along the X axis, the value of U is negative.
The range of thread leads, limitation of spindle speed, etc. are the same as in G32 (thread cutting). Thread chamfering can be performed in this thread cutting cycle. A signal from the machine tool, initiates thread chamfering. The chamfering distance is specified in a range from 0.1L to 12.7L in 0.1L increments by parameter (No. 5130). (In the above expression, L is the thread lead.)
In the single block mode, operations 1, 2, 3, and 4 are performed by pressing cycle start button once.

Notes

1. Notes on this thread cutting are the same as in thread cutting in G32. However, a stop by feed hold is as follows ; Stop after completion of path 3 of thread cutting cycle.
2. The tool retreats while chamfering and returns to the start point on the X axis then the Z axis, as soon as the feed hold status is entered during thread cutting (motion 2) when the ”Thread Cutting Cycle retract” option is used.



Another feed hold cannot be made during retreat. The chamfered amount is the same as that at the end point.

- **Taper thread cutting cycle**

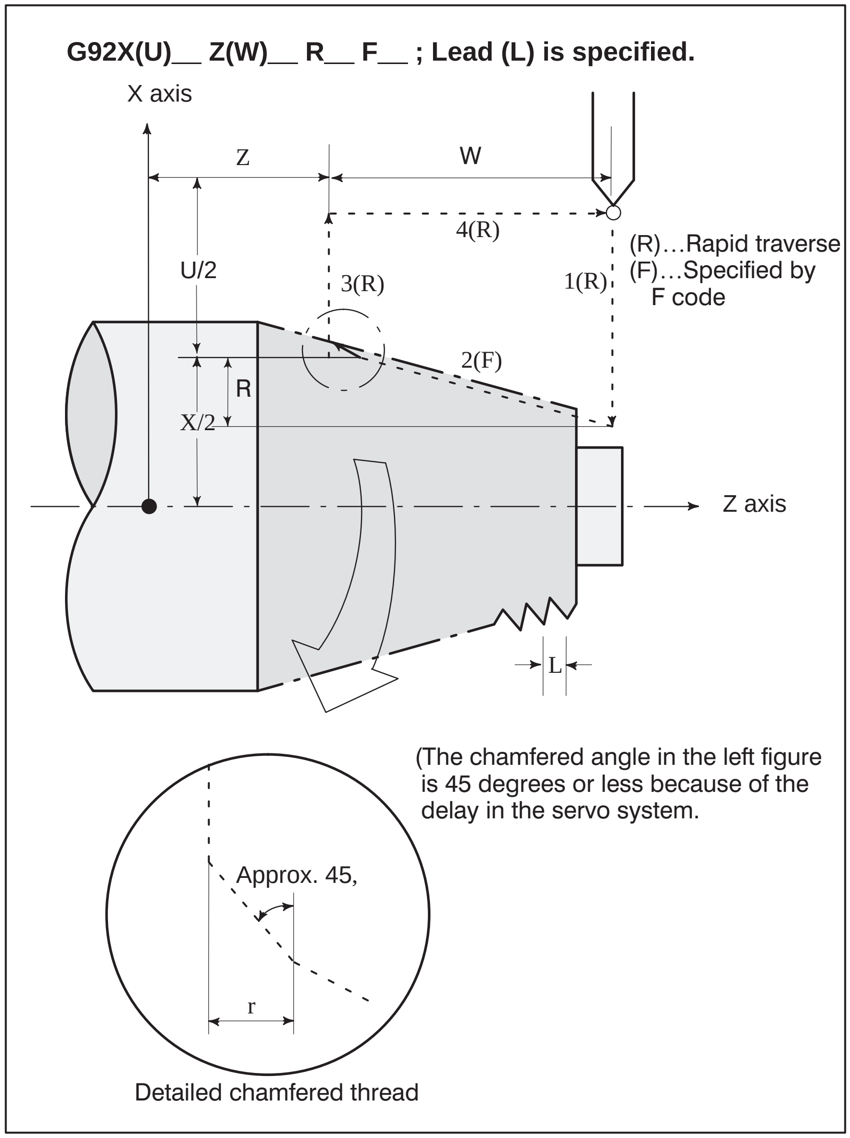


Fig. 14.1.2 (b) Taper thread cutting cycle

14.1.3 End face turning cycle (G94)

- **Face cutting cycle**

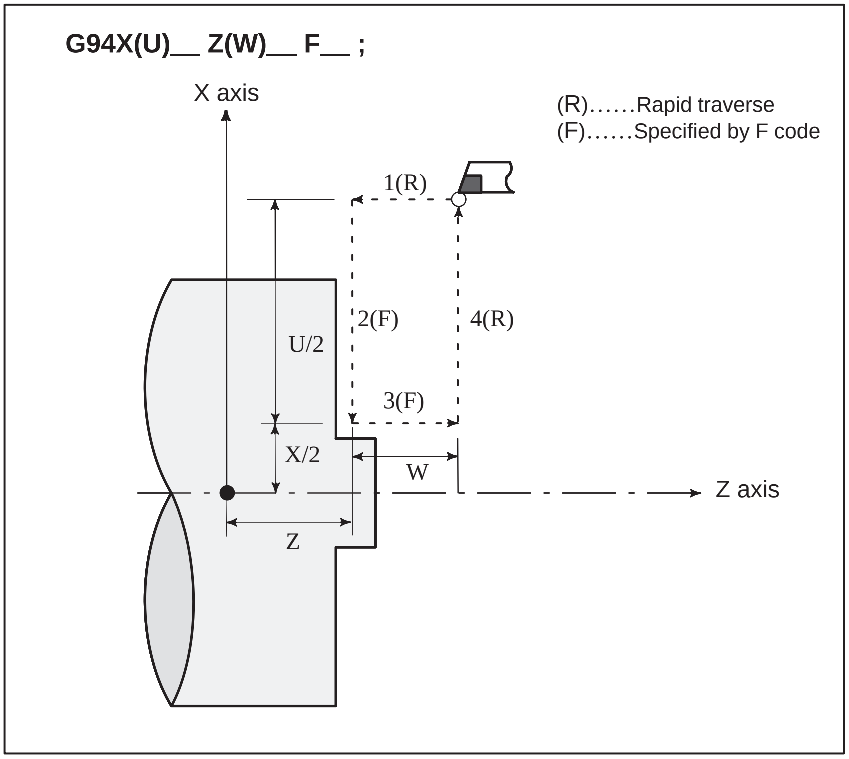


Fig. 14.1.3 (a) Face Cutting Cycle

In incremental programming, the sign of numbers following addresses U and W depends on the direction of paths 1 and 2. That is, if the direction of the path is in the negative direction of the Z axis, the value of W is negative.
In single block mode, operations 1, 2, 3, and 4 are performed by pressing the cycle start button once.

- **Taper face cutting cycle**

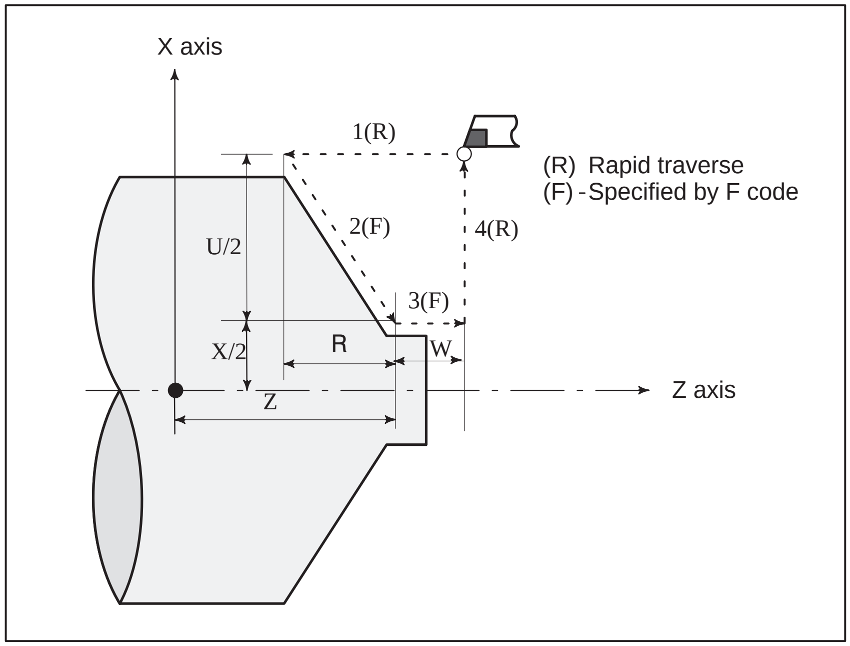
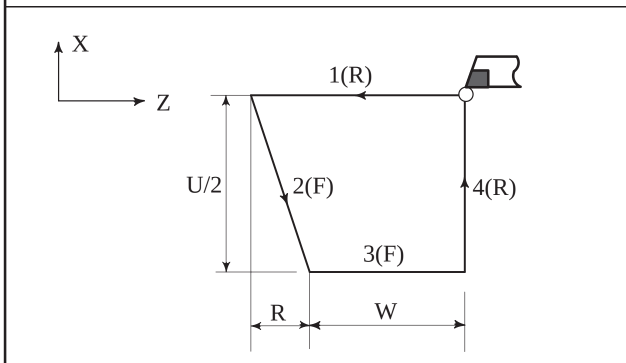
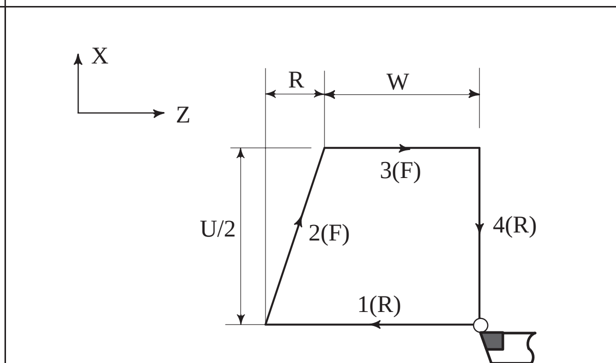
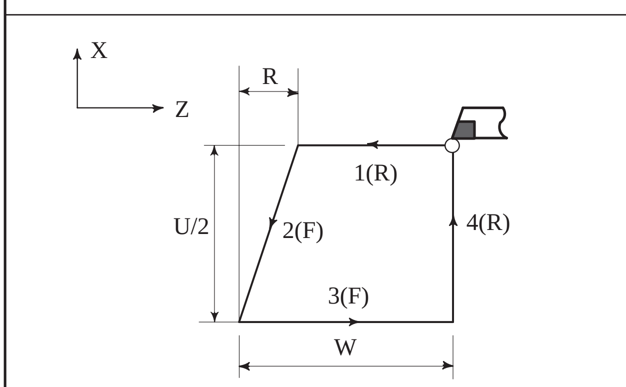
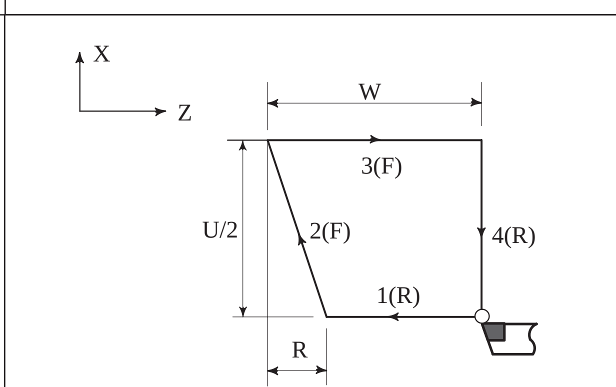


Fig. 14.1.3 (b)

- **Signs of numbers specified in the taper cutting cycle**

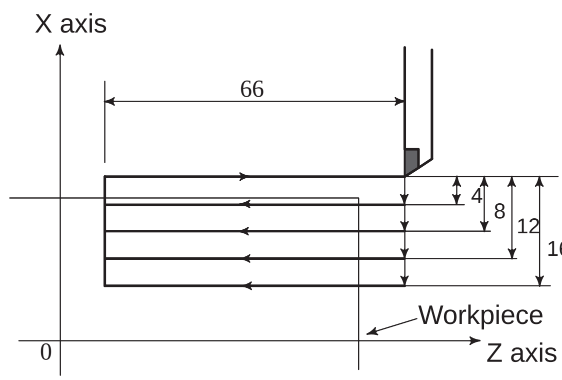
In incremental programming, the relationship between the signs of the numbers following address U, W, and R, and the tool paths are as follows:

1. U < 0, W < 0, R < 0	**2. U > 0, W < 0, R < 0**
	
3. U < 0, W < 0, R > 0 at CRC≦CWC	**4. U > 0, W < 0, R<0 at CRC≦CWC**
	

Notes

1 Since data values of X (U), Z (W) and R during canned cycle are modal, if X (U), Z (W), or R is not newly commanded, the previously specified data is effective. Thus, when the Z axis movement amount does not vary as in the example below, a canned cycle can be repeated only by specifying the movement commands for the X–axis.
However, these data are cleared, if a one–shot G code expect for G04 (dwell) or a G code in the group 01 except for G90, G92, G94 is commanded.

(Example)



The cycle in the above figure is executed by the following program.

```
N030    G90 U–8.0 W–66.0 F0.4 ;
N031    U–16.0 ;
N032    U–24.0 ;
N033    U–32.0 ;
```

2 The following three applications can be performed.

(1) If an EOB or zero movement commands are specified for the block following that specified with!a canned cycle, the same canned cycle is repeated.

(2) By specifying a canned cycle in the MDI mode, and pushing the cycle start button after the!block terminates, the same canned cycle as the previous one will be performed.

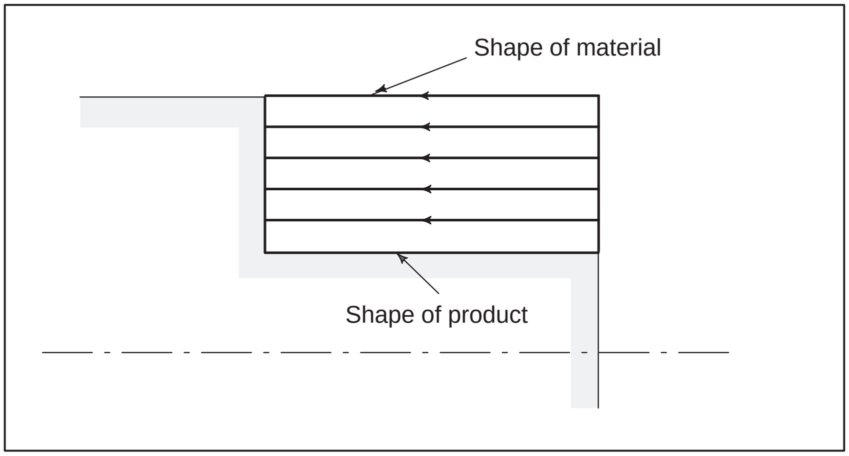
(3) If the M, S, T function is commanded during the canned cycle mode, both the canned cycle and!M, S, or T function can be performed simultaneously. If this is inconvenient, cancel the canned!cycle once as in the program examples below (specify G00 or G01) and execute the M, S, or T command. After the execution of M, S, or T terminates, command the canned cycle again.
(Example)

```
N003 T0101 ;
  :
  :
N010    G90 X20.0 Z10.0 F0.2 ;
N011    G00 T0202 ;
N012    G90 X20.5 Z10.0 ;
```

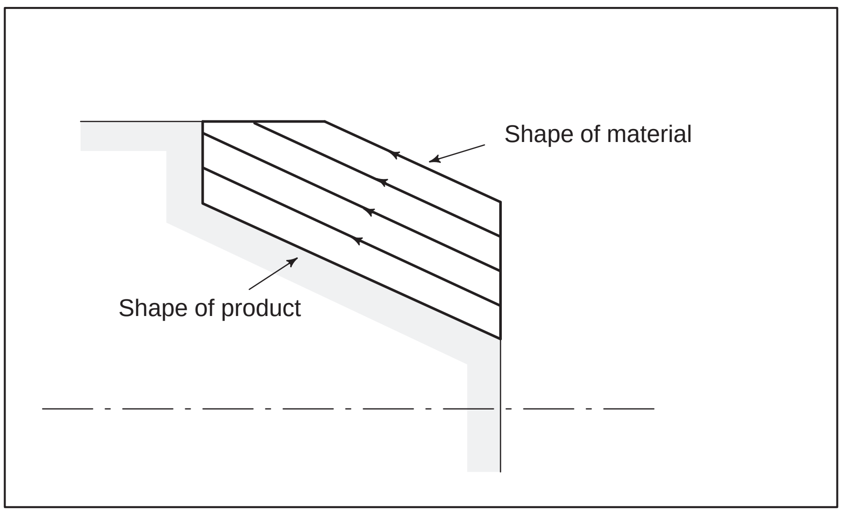
14.1.4 How to use canned cycles (G90, G92, G94)

An appropriate canned cycle is selected according to the shape of the material and the shape of the product.

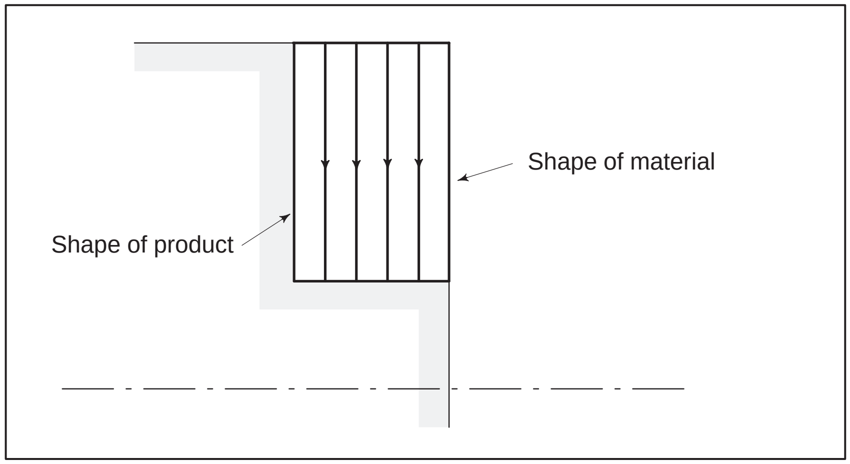
- **Straight cutting cycle (G90)**



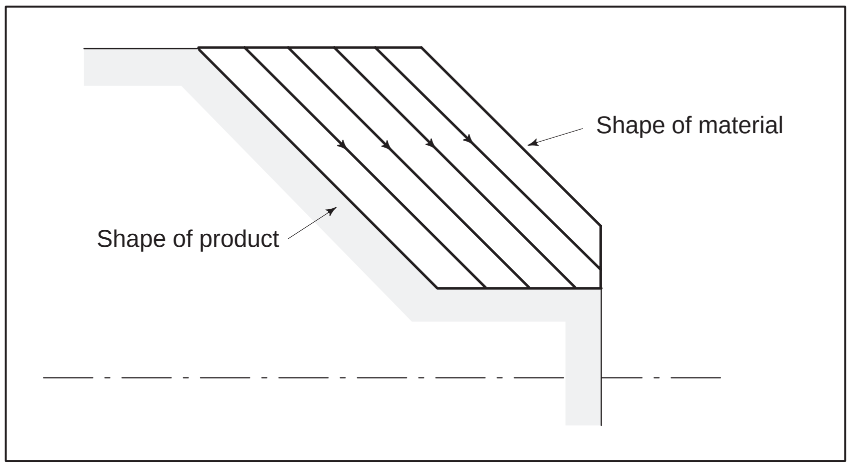
- **Taper cutting cycle (G90)**



- **Face cutting cycle (G94)**



- **Face taper cutting cycle (G94)**



14.2 MULTIPLE REPETITIVE CYCLE (G70–G76)

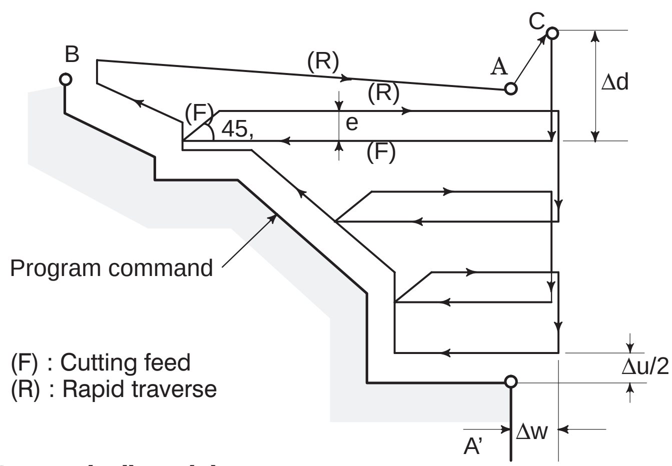
This option canned cycles to make CNC programming easy. For instance, the data of the finish work shape describes the tool path for rough machining. And also, a canned cycles for the thread cutting is available.

14.2.1 Stock Removal in Turning (G71)

There are two types of stock removals in turning : Type I and II.

- **Type I**

If a finished shape of A to A' to B is given by a program as in the figure below, the specified area is removed by Δd (depth of cut), with finishing allowance Δu/2 and Δw left.



G71 U (Δd) R (e) ;

G71 P (ns) Q (nf) U (Δu) W (Δw) F (f) S (s) T (t)

N (ns)...........
....
F____
S____
T____
N (nf)..........;

The move command between A and B is specified in the blocks from sequence number ns to nf.

Δd : Depth of cut (radius designation)
Designate without sign. The cutting direction depends on the direction AA'. This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter (No. 5132), and the parameter is changed by the program command.

e : Escaping amount
This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter (No. 5133), and the parameter is changed by the program command.

ns: Sequence number of the first block for the program of finishing shape.

nf : Sequence number of the last block for the program of finishing shape.

Δu : Distance and direction of finishing allowance in X direction (diameter / radius designation).

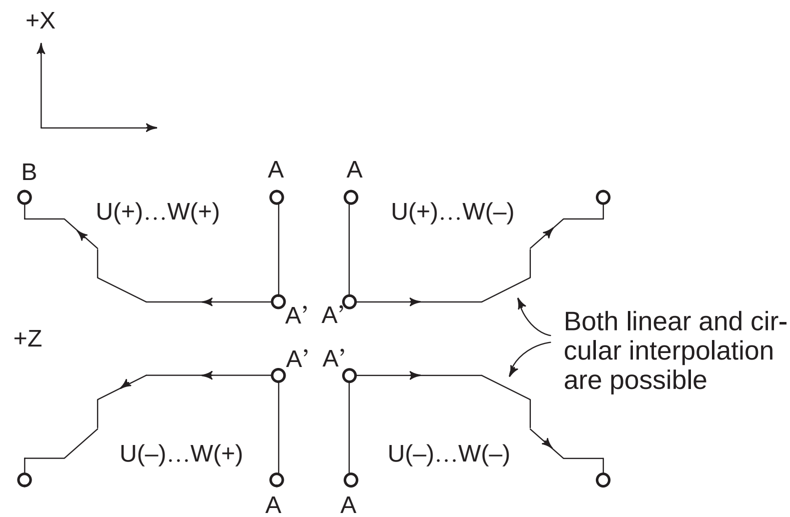
Δw : Distance and direction of finishing allowance in Z direction.

f,s,t : Any F , S, or T function contained in blocks ns to nf in the cycle is ignored, and the F, S, or T function in this G71 block is effective.

Fig. 14.2.1 (a) Cutting Path in Stock Removal in Turning (Type I)

Notes

1. While both Δd and Δu, are specified by address U, the meanings of them are determined!by the presence of addresses P and Q.
2. The cycle machining is performed by G71 command with P and Q specification.
   F, S, and T functions which are specified in the move command between points A and B!are ineffective and those specified in G71 block or the previous block are effective.
   When an option of constant surface speed control is selected, G96 or G97 command!specified in the move command between points A and B are ineffective, and that specified!in G71 block or the previous block is effective.
   The following four cutting patterns are considered. All of these cutting cycles are made!paralleled to Z axis and the sign of Δu and Δw are as follows:



   The tool path between A and A’ is specified in the block with sequence number ”ns”!including G00 or G01, and in this block, a move command in the Z axis cannot be!specified. The tool path between A’ and B must be steadily increasing or decreasing!pattern in both X and Z axis. When the tool path between A and A’ is programmed by!G00/G01, cutting along AA’ is performed in G00/G01 mode respectively.
3. The subprogram cannot be called from the block between sequence number ”ns” and ”nf”.

- **Type II**

Type II differs from type I in the following : The profile need not show monotone increase or decrease along the X axis, and it may have up to 10 concaves (pockets).

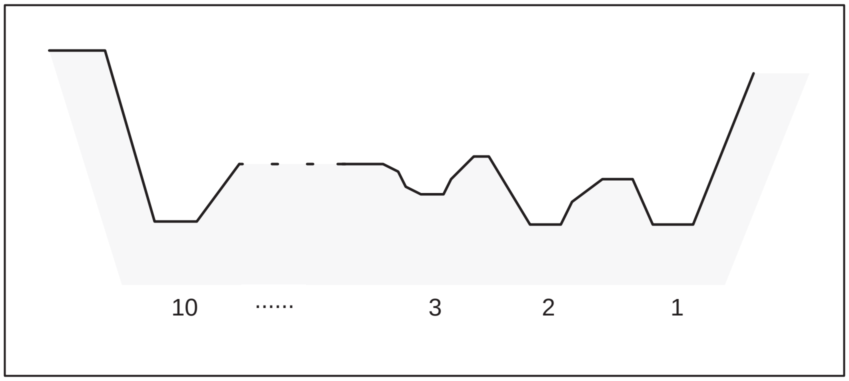


Fig. 14.2.1 (b) Number of Pockets in Stock Removal in Turning (Type II)

Note that, however, the profile must have monotone decrease or increase along the Z axis. The following profile cannot be machined:

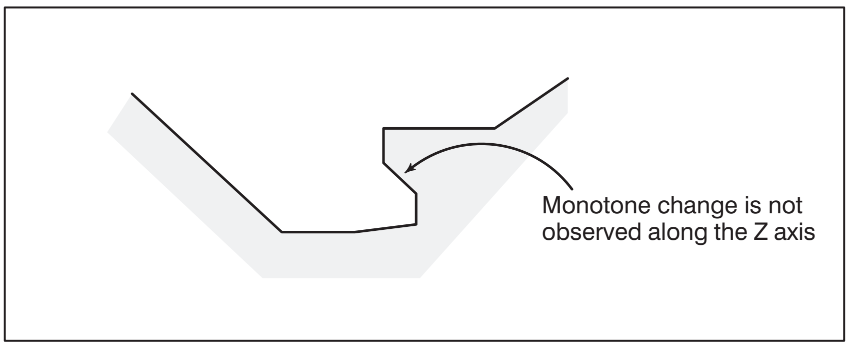


Fig. 14.2.1 (c) Figure Which Cannot Be Machined in Stock Removal in Turning (Type II)

The first cut portion need not be vertical ; any profile is permitted if monotone change is shown along the Z axis.

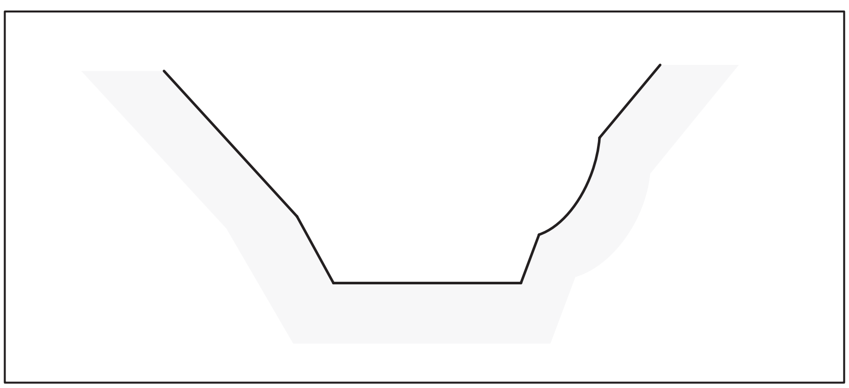


Fig. 14.2.1 (d) Figure Which Can Be Machined (Monotonic change) in Stock Removal in Turning (Type II)

After turning, a clearance is provided by cutting along the workpiece profile.

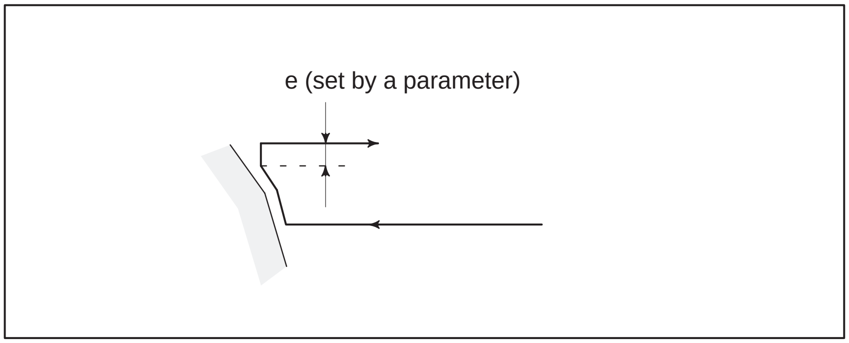


Fig. 14.2.1 (e) Chamfering in Stock Removal in Turning (Type II)

The clearance e (specified in R) to be provided after cutting can also be set in parameter No. 5133.
A sample cutting path is given below:

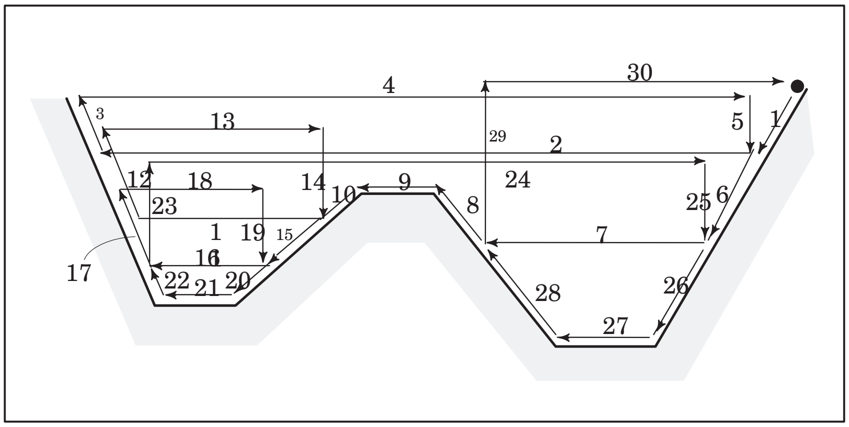


Fig. 14.2.1 (f) Cutting Path in Stock Removal in Facing

The offset of the tool tip radius is not added to finishing allowances Δu and Δw. In turning, the offset of the tool tip radius is assumed to be zero.
W=0 must be specified ; otherwise, the tool tip may cut into one wall side.
For the first block of a repetitive portion, two axes X(U) and Z (W) must be specified. When Z motion is not performed, W0 is also specified.

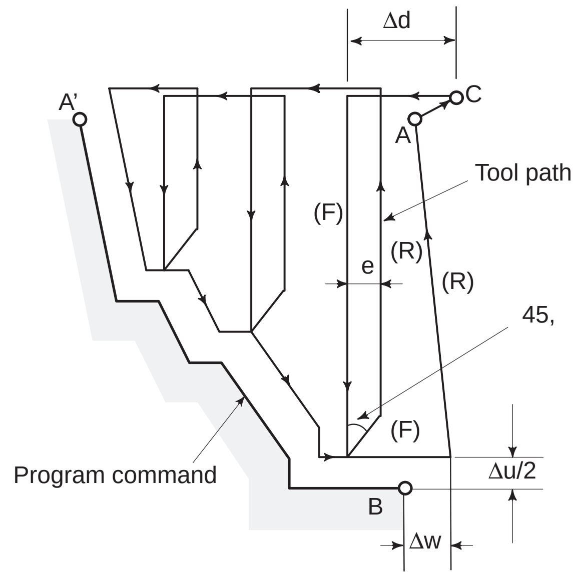
- **Distinction between type I and type II**

When only one axis is specified in the first block of a repetitive portion
-- Type I
When two axes are specified in the first block of a repetitive portion
-- Type II
When the first block does not include Z motion and type IIis to be used, W0 must be specified.
(Example)

```
  TYPEI     TYPEII
G71 V10.0 R5.0 ; G71 V10.0 R5.0 ;
G71 P100 Q200....;  G71 P100 Q200........;
N100X (U)___; N100X (U)___ Z(W)___;
   :    :
   :    :
N200..............;  N200.........................;
```

14.2.2 Stock removal in facing (G72)

As shown in the figure below, this cycle is the same as G71 except that cutting is made by a operation parallel to X axis.



G72 W(<u>Δd</u>) R(<u>e</u>) ;
G72 P(<u>ns</u>) Q(<u>nf</u>) U(<u>Δu</u>) W(<u>Δw</u>) F(<u>f</u>) S(<u>s</u>) T(<u>t</u>) ;
The meanings of Δd, e, ns, nf, Δu,Δw, f, s, and t are the same as those in G71.

Fig. 14.2.2 (a) Cutting Path in Stock Removal in Facing

- **Signs of specified numbers**

The following four cutting patterns are considered. All of these cutting cycles are made parallel to X axis and the sign of Δu and Δw are as follows :

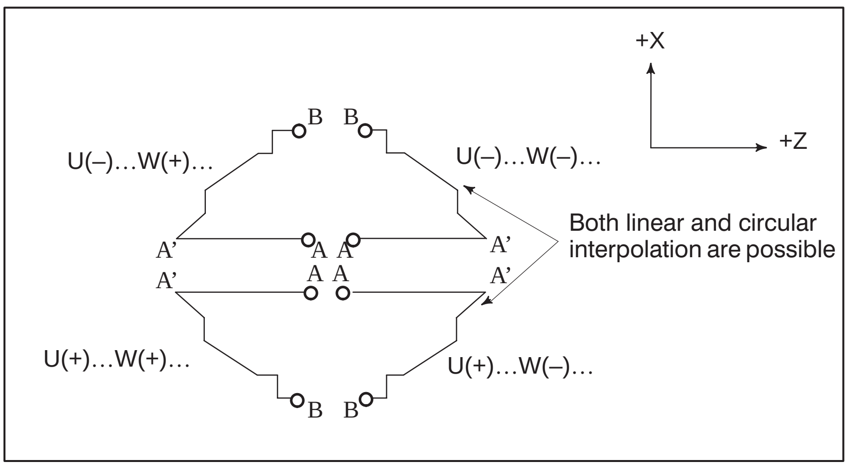
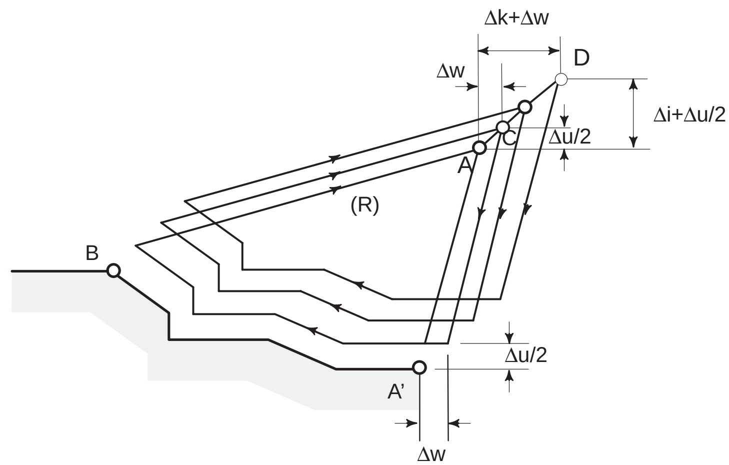


Fig. 14.2.2 (b)
Signs of Numbers Specified with U and W in Stock Removal in Facing

The tool path between A and A' is specified in the block with sequence number ”ns” including G00 or G01, and in this block, a move command in the X axis cannot be specified. The tool path between A' and B must be steadily increasing and decreasing pattern in both X and Z axes. Whether the cutting along AA' is G00 or G01 mode is determined by the command between A and A', as described in item 14.2.1.

14.2.3 Pattern repeating (G73)

This function permits cutting a fixed pattern repeatedly, with a pattern being displaced bit by bit. By this cutting cycle, it is possible to efficiently cut work whose rough shape has already been made by a rough machining, forging or casting method, etc.



The pattern commanded in the program should be as follows.

A→A'→B

G73 U (Δi) W (Δk) R (d) ;

G73 P (ns) Q (nf) U (Δu) W (Δw) F (f) S (s) T (t) ;

N (ns)··········
················
F____
S____
T____
N (nf)··········;

The move command between A and B is specified in the blocks from sequence number ns to nf.

Δi : Distance and direction of relief in the X axis direction (Radius designation). This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter No. 5135, and the parameter is changed by the program command.

Δk : Distance and direction of relief in the Z axis direction. This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter No. 5136, and the parameter is changed by the program command.

d : The number of division
This value is the same as the repetitive count for rough cutting. This designation is modal and is not changed until the other value is designated. Also, this value can be specified by the parameter No. 5137, and the parameter is changed by the program command.

ns: Sequence number of the first block for the program of finishing shape.

nf : Sequence number of the last block for the program of finishing shape.

Δu : Distance and direction of finishing allowance in X direction (diameter/radius designation)

Δw : Distance and direction of finishing allowance in Z direction

f,s,t : Any F, S, and T function contained in the blocks between sequence number "ns" and "nf" are ignored, and the F, S, and T functions in this G73 block are effective.

Fig. 14.2.3 (a) Cutting path in Pattern Repeating

Notes

1. While the values Δi and Δk, or Δu and Δw are specified by address U and W!respectively, the meanings of them are determined by the presence of addresses P and Q!in G73 block. When P and Q are not specified in a same block, addresses U and W!indicates Δi and Δk respectively. When P and Q are specified in a same block, addreses!U and W indicates Δu and Δw respectively.
2. The cycle machining is performed by G73 command with P and Q specification.
   The four cutting patterns are considered. Take care of the sign of Δu, Δw, Δk, and Δi.
   When the machining cycle is terminated, the tool returns to point A.

14.2.4 Finishing cycle (G70)

After rough cutting by G71, G72 or G73, the following command permits finishing.

Format

G70P (ns) Q (nf) ;

(ns) : Sequence number of the first block for the program of finishing shape.

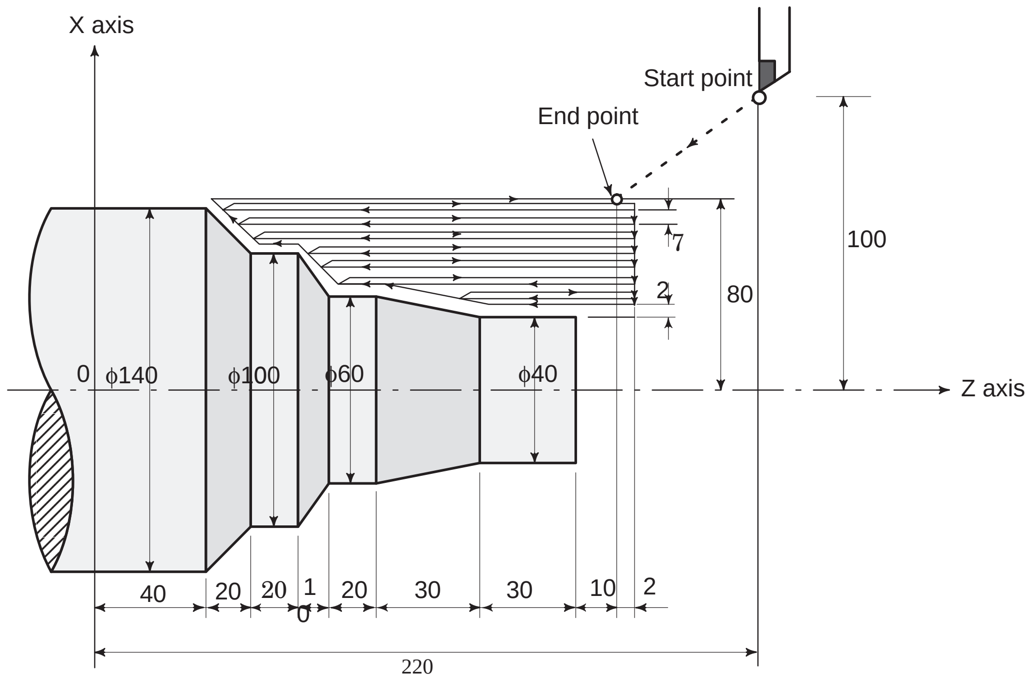
(nf) : Sequence number of the last block for the program of finishing shape.

Notes

1. F, S, and T functions specified in the block G71, G72, G73 are not effective but those!specified between sequence numbers ”ns” and ”nf” are effective in G70.
2. When the cycle machining by G70 is terminated, the tool is returned to the start point and!the next block is read.
   3.In blocks between ”ns” and ”nf” referred in G70 through G73, the subprogram cannot be!called.

Examples

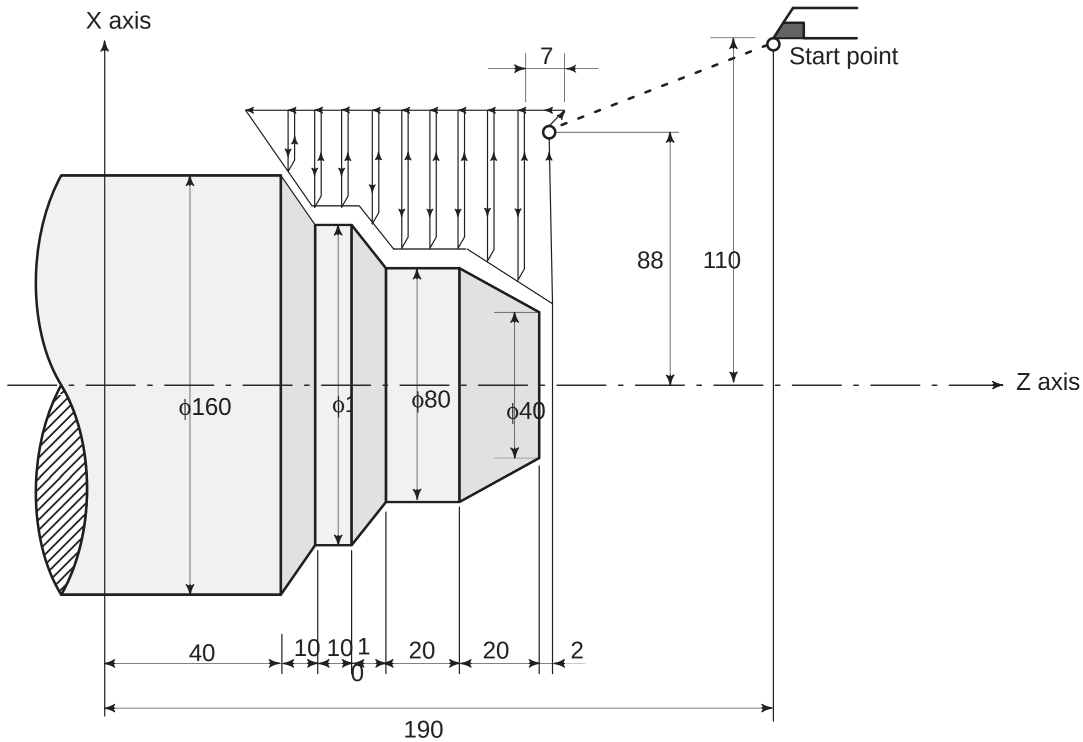
Stock Removal in Turning (G71) (Type I)



(Diameter designation, metric input)

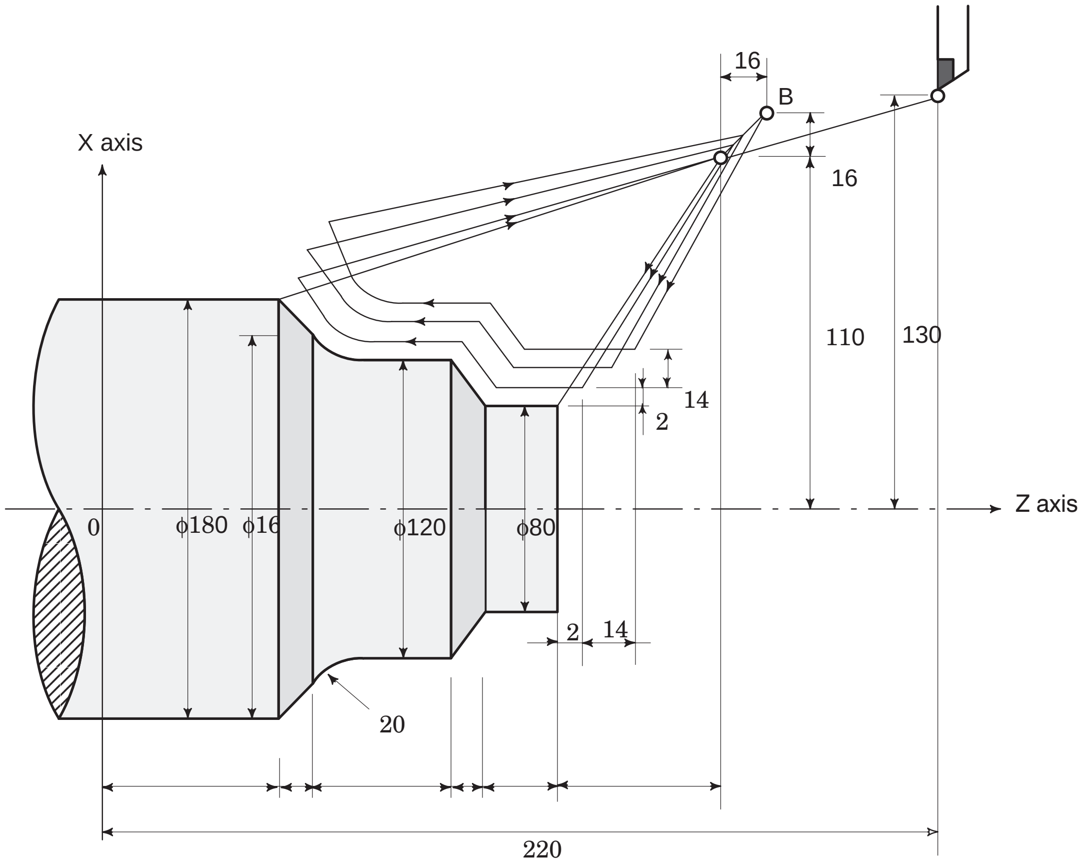
```
N010  G50 X200.0 Z220.0 ;
N011  G00 X160.0 Z180.0 ;
N012  G71 U7.0 R1.0 ;
N013  G71 P014 Q020 U4.0 W2.0 F0.3 S550 ;
N014  G00 X40.0 F0.15 S700 ;
N015  G01 W–40.0;
N016      X60.0 W–30.0 ;
N017      W–20.0 ;
N018      X100.0 W–10.0 ;
N019      W–20.0 ;
N020      X140.0 W–20.0 ;
N021  G70 P014 Q020 ;
```

Stock Removal In Facing (G72)




```
(Diameter designation, metric input)
   N010  G50 X220.0 Z190.0 ;
   N011  G00 X176.0 Z132.0 ;
   N012  G72 W7.0 R1.0 ;
   N013  G72 P014 Q019 U4.0 W2.0 F0.3 S550 ;
   N014  G00 Z58.0 S700 ;
   N015  G01 X120.0 W12.0 F0.15 ;
   N016      W10.0 ;
   N017      X80.0 W10.0 ;
   N018      W20.0 ;
   N019      X36.0 W22.0 ;
   N020  G70 P014 Q019 ;
```

Pattern Repeating (G73)

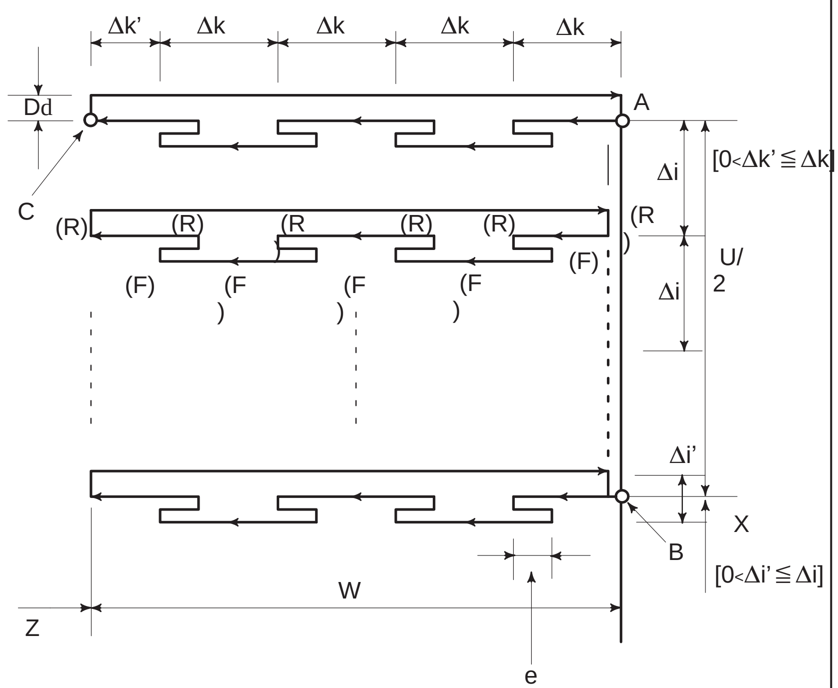


(Diameter designation, metric input)

```
N010  G50 X260.0 Z220.0 ;
N011  G00 X220.0 Z160.0 ;
N012  G73 U14.0 W14.0 R3 ;
N013  G73 P014 Q019 U4.0 W2.0 F0.3 S0180 ;
N014  G00 X80.0 W–40.0 ;
N015  G01 W–20.0 F0.15 S0600 ;
N017  W–20.0 S0400 ;
N018  G02 X160.0 W–20.0 R20.0 ;
N019  G01 X180.0 W–10.0 S0280 ;
N020  G70 P014 Q019 ;
```

14.2.5 End face peck drilling cycle (G74)

The following program generates the cutting path shown in Fig. 14.2.5 (a). Chip breaking is possible in this cycle as shown below. If X (U) and Pare omitted, operation only in the Z axis results, to be used for drilling.



G74R (e) ;

G74X(U)_ Z(W)_ P(Δi) Q(Δk) R(Δd) F (f) ;

e : Return amount
This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter No. 5139, and the parameter is changed by the program command.
X : X component of point B
U : Incremental amount from A to B
Z : Z component of point C
W : Increment amount from A to C
Δi : Movement amount in X direction (without sign)
Δk : Depth of cut in Z direction (without sign)
Δd : Relief amount of the tool at the cutting bottom. The sign of Δd is always plus (+). However, if address X (U) and Δi are omitted, the relief direction can be specified by the desired sign.
f : Feed rate

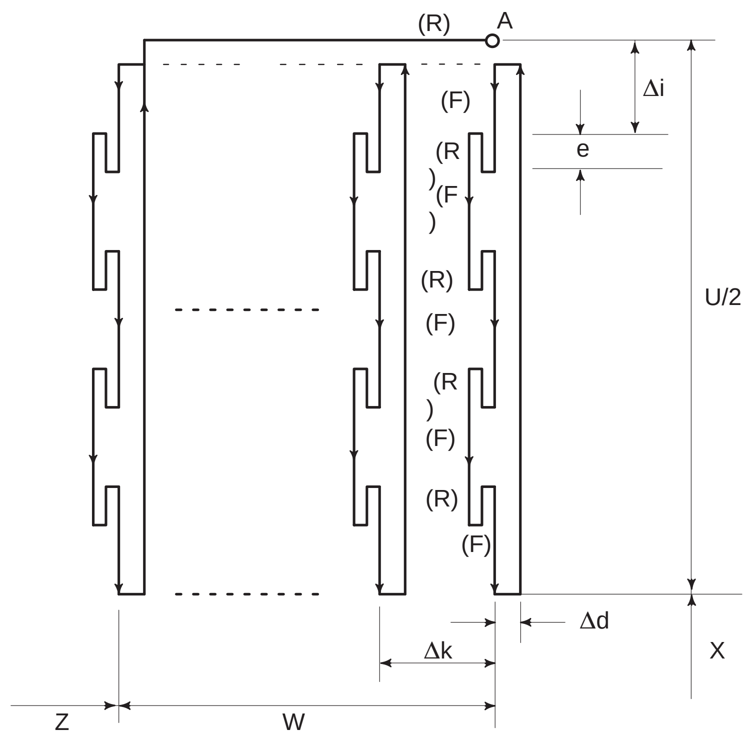
Fig. 14.2.5 (a) Cutting Path in End Face Peek Drilling Cycle

Notes

1. While both e and Δd are specified by address R, the meanings of them are determined by!the present of address X (U). When X(U) is specified, Δd is used.
2. The cycle machining is performed by G74 command with X (U) specification.

14.2.6 Outer diameter / internal diameter drilling cycle (G75)

The following program generates the cutting path shown in Fig. 14.2.6 (a). This is equivalent to G74 except that X is replaced by Z. Chip breaking is possible in this cycle, and grooving in X axis and peck drilling in X axis (in this case, Z, W, and Q are omitted) are possible.



G75R **(e)** **;**

G75X(U)_ Z(W)_ P(Δi) Q(Δk) R(Δd) F(f) ;

Fig. 14.2.6 (a)
Cutting Path in Outer Diameter / Internal Diameter Drilling Cycle

Both G74 and G75 are used for grooving and drilling, and permit the tool to relief automatically. Four symmetrical patterns are considered, respectively.

14.2.7 Multiple thread cutting cycle (G76)

The thread cutting cycle as shown in Fig.14.2.7 (a) is programmed by the G76 command.

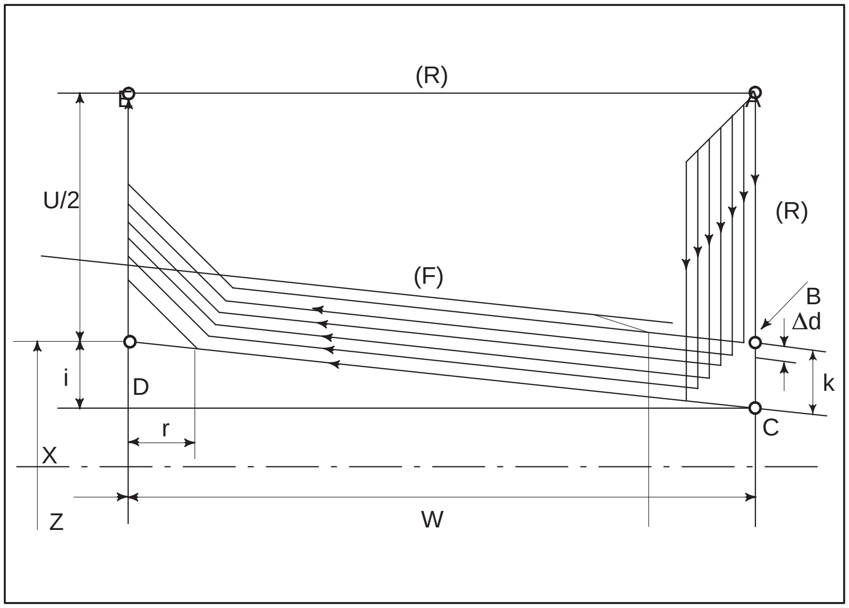
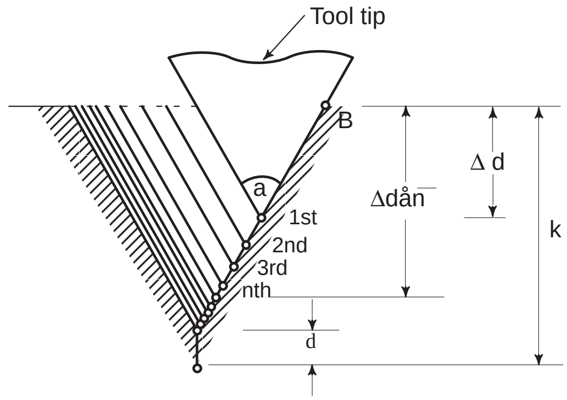


Fig. 14.2.7 (a) Cutting Path in Multiple thread cutting cycle



G76P (m) (r) (a) Q (Δd min) R(d);
G76X (u) _ Z(W) _ R(i) P(k) Q(Δd) F(L) ;

m ; Repetitive count in finishing (1 to 99)
This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter No. 5142, and the parameter is changed by the program command.

r : Chamfering amount
When the thread lead is expressed by L, the value of L can be set from 0.0L to 9.9L in 0.1L increment (2–digit number from 00 to 90). This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter No. 5130, and the parameter is changed by the program command.

a : Angle of tool tip
One of six kinds of angle, 80°, 60°, 55°, 30°, 29°, and 0°, can be selected, and specified by 2–digit number.
This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter No. 5143, and the parameter is changed by the program command.

m, r, and a are specified by address P at the same time.

(Example)
When m=2, r=1.2L, a=60°, specify as shown below (L is lead of thread).

$$P\ \underset{m}{\underline{02}}\ \underset{r}{\underline{12}}\ \underset{a}{\underline{60}}$$

Δdmin: Minimum cutting depth (specified by the radius value)
When the cutting depth of one cycle operation (Δd – Δd –1) becomes smaller than this limit, the cutting depth is clamped at this value. This designation is modal and is not changed until the other value is designated. Also this value can be specified by parameter No. 5140, and the parameter is changed by the program command.

d : Finishing allowance
This designation is modal and is not changed until the other value is designated. Also this value can be specified by parameter No. 5141, and the parameter is changed by the program command.

i : Difference of thread radius If i = 0, ordinary straight thread cutting can be made.

k : Height of thread
This value is specified by the radius value.

Δd : Depth of cut in 1st cut (radius value)

L : Lead of thread (same as G32).

Fig. 14.2.7 (b) Detail of cutting

- **Thread cutting cycle retract**

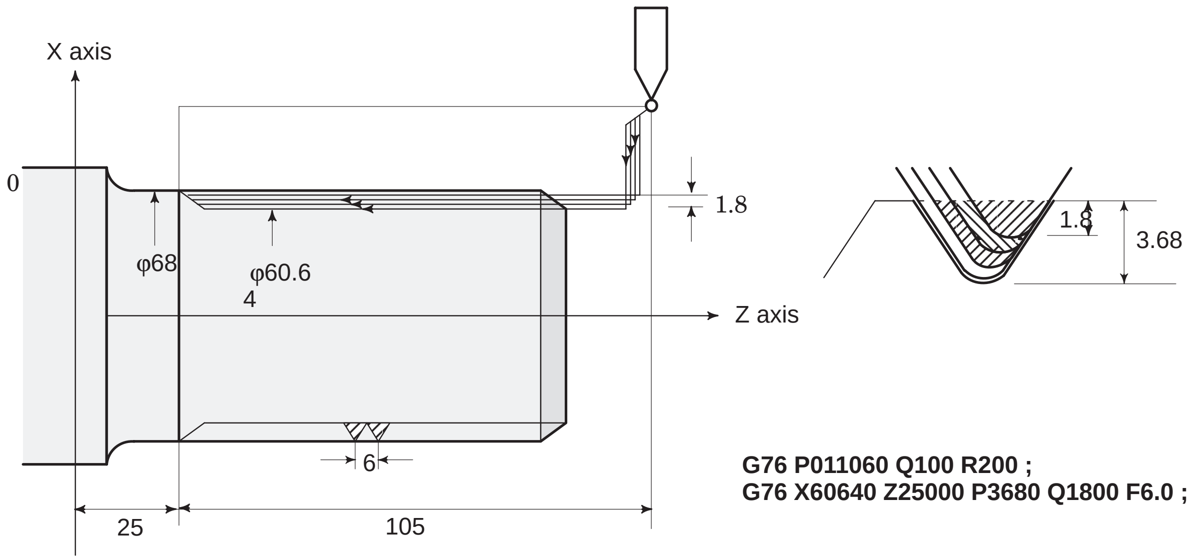
When feed hold is applied during threading in the multiple thread cutting cycle (G76), the tool quickly retracts in the same way as in chamfering performed at the end of the thread cutting cycle. The tool goes back to the start point of the cycle. When cycle start is triggered, the multiple thread cutting cycle resumes.
Without this retraction function, when feed hold is applied during threading, the tool goes back to the start point of the cycle after threading is completed.
See notes in 14.1.2.

Notes

1. The meanings of the data specified by address P, Q, and R determined by the presence of!X (U) and X (W).
2. The cycle machining is performed by G76 command with X (U) and Z (W) specification.
   By using this cycle , one edge cutting is performed and the load on the tool tip is reduced.
   Making the cutting depth △d for the first path, and △dn for the nth path, cutting amount!per one cycle is held constant.
   Four symmetrical patterns are considered corresponding to the sign of each address.
   The internal thread cutting is available. In the above figure, the feed rate between C and D!is specified by address F, and in the other path, at rapid traverse. The sign of incremental!dimensions for the above figure is as follows:
   U, W : minus (determined by the direction of the tool path AC and CD.)
   R : minus (determined by the direction of the tool path AC.)
   P : plus (always)
   Q : plus (always)
3. Notes on thread cutting are the same as those on G32 thread cutting and G92 thread!cutting cycle.
4. The designation of chamfering is also effective for G92 thread cutting cycle.
5. The tool returns to the cycle start point at that time (cutting depth △dn) as soon as the!feed hold status is entered during thread cutting when the ”Thread Cutting Cycle retract”!option is used.

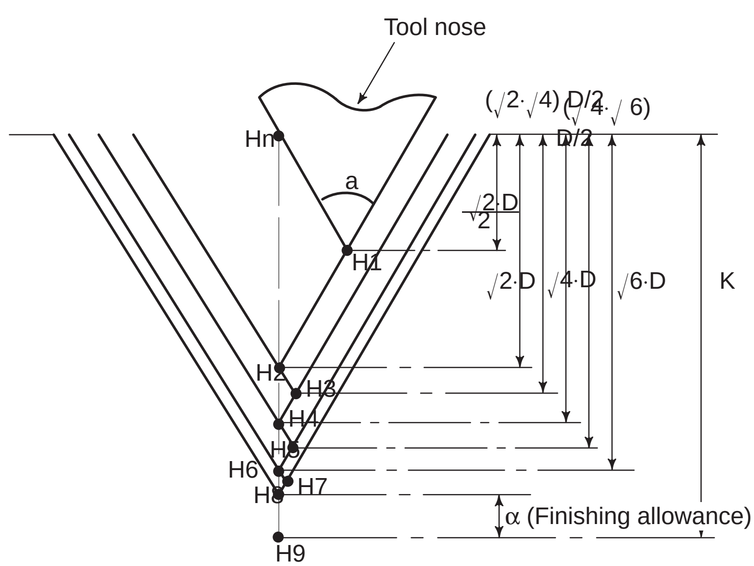
Examples

Multiple repetitive cycle (G76)



- **Staggered thread cutting**

Specifying P2 can perform staggered thread cutting with a constant depth of cut.
Example: G76 X60640 Z25000 K3680 D1800 F6.0 A60 P2;
For staggered thread cutting, always use the FS15 tape format (see Section 18.5).
If the depth of cut in one cycle is less than dmin (specified in parameter No. 5140), the depth of cut is clamped at Δdmin.



Staggered thread cutting with a constant depth of cut

14.2.8 Notes on multiple repetitive cycle (G70–G76)

1. In the blocks where the multiple repetitive cycle are commanded, the!addresses P, Q, X, Z, U, W, and R should be specified correctly for each!block.
2. In the block which is specified by address P of G71, G72 or G73, G00 or G01 group should be commanded. If it is not commanded, P/S alarm No.65 is generated.
3. In MDI mode, G70, G71, G72, or G73 cannot be commanded. If it is commanded, P/S alarm No. 67 is generated. G74, G75, and G76 can be commanded in MDI mode.
4. In the blocks in which G70, G71, G72, or G73 are commanded and between the sequence number specified by P and Q, M98 (subprogram call) and M99 (subprogram end) cannot be commanded.
5. In the blocks between the sequence number specified by P and Q, the following commands cannot be specified.
   ·One shot G code except for G04 (dwell)
   ·01 group G code except for G00, G01, G02, and G03
   ·06 group G code
   ·M98 / M99
6. While a multiple repetitive cycle (G70AG76) is being executed, it is possible to stop the cycle and to perform manual operation. But, when the cycle operation is restarted, the tool should be returned to the position where the cycle operation is stopped.
   If the cycle operation is restarted without returning to the stop position, the movement in manual operation is added to the absolute value, and the tool path is shifted by the movement amount in manual operation.
7. When G70, G71, G72, or G73 is executed, the sequence number specified by address P and Q should not be specified twice or more in the same program.
8. Do not program so that the final movement command of the finishing shape block group designated with P and Q for G70, G71, G72, and G73 finishes with chamfering or corner rounding. If it is specified,P/S alarm No. 69 is generated.

14.3 CANNED CYCLE FOR DRILLING (G80–G89)

The canned cycle for drilling simplifies the program normally by directing the machining operation commanded with a few blocks, using one block including G function.
This canned cycle conforms to JIS B 6314.
Following is the canned cycle table.

Table 14.3(a) Canned Cycles

G code	Drilling axis	Hole machining operation (– direction)	Operation in the bottom hole position	Retraction operation (+ direction)	Applications
G80	——	——	——	——	Cancel
G83	Z axis	Cutting feed / intermittent	Dwell	Rapid traverse	Front drilling cycle
G84	Z axis	Cutting feed	Dwell→spindle CCW	Cutting feed	Front tapping cycle
G85	Z axis	Cutting feed	——	Cutting feed	Front boring cycle
G87	X axis	Cutting feed / intermittent	Dwell	Rapid traverse	Side drilling cycle
G88	X axis	Cutting feed	Dwell→Spindle CCW	Cutting feed	Side tapping cycle
G89	X axis	Cutting feed	Dwell	Cutting feed	Side boring cycle

Explanations

In general, the drilling cycle consists of the following six operation sequences.

Operation 1 Positioning of X (Z) and C axis
Operation 2 Rapid traverse up to point R level
Operation 3 Hole machining
Operation 4 Operation at the bottom of a hole
Operation 5 Retraction to point R level
Operation 6 Rapid traverse up to the initial point

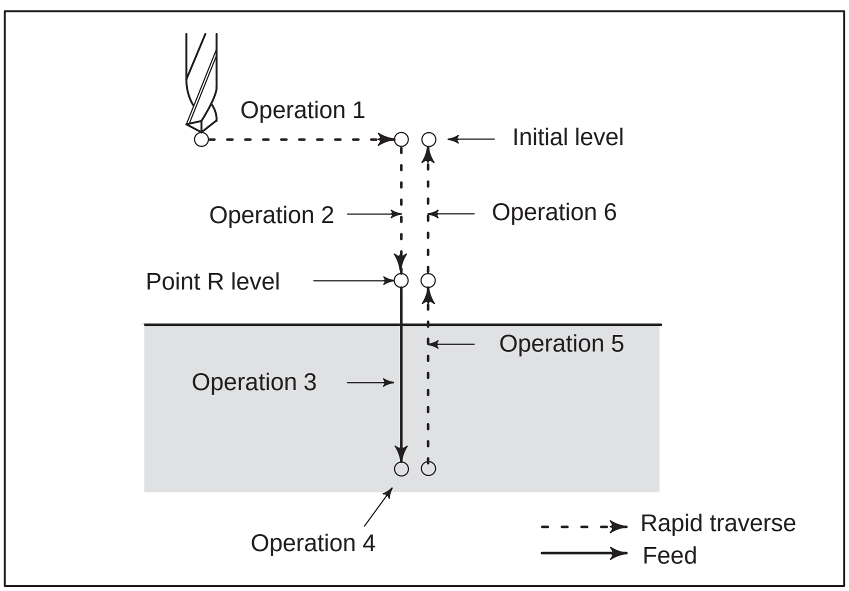


Fig. 14. 3 (a) Drilling Cycle Operation Sequence

- **Positioning axis and drilling axis**

A drilling G code specifies positioning axes and a drilling axis as shown below. The C–axis and X– or Z–axis are used as positioning axes. The X– or Z–axis, which is not used as a positioning axis, is used as a drilling axis.
Although canned cycles include tapping and boring cycles as well as drilling cycles, in this chapter, only the term drilling will be used to refer to operations implemented with canned cycles.

Table14.1(b) Positioning axis and drilling axis

G code	Positioning plane	Drilling axis
G83, G84, G85	X axis, C axis	Z axis
G87, G88, G89	Z axis, C axis	X axis

G83 and G87, G84 and G88, and G85 and G89 have the same function respectively except for axes specified as positioning axes and a drilling axis.

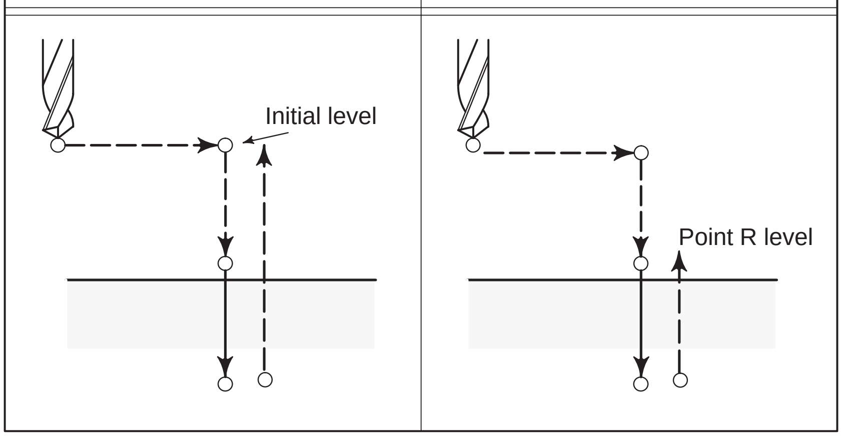
- **Drilling mode**

G83AG85 / G87A89 are modal G codes and remain in effect until canceled. When in effect, the current state is the drilling mode.
Once drilling data is specified in the drilling mode, the data is retained until modified or canceled.
Specify all necessary drilling data at the beginning of canned cycles; when canned cycles are being performed, specify data modifications only.

- **Return point level G98/G99**

In G code system A, the tool returns to the initial level from the bottom of a hole. In G code system B or C, specifying G98 returns the tool to the initial level from the bottom of a hole and specifying G99 returns the tool to the point–R level from the bottom of a hole.
The following illustrates how the tool moves when G98 or G99 is specified. Generally, G99 is used for the first drilling operation and G98 is used for the last drilling operation.
The initial level does not change even when drilling is performed in the G99 mode.

G98(**Return to initial level**)	**G99**(**Return to point R level**)



- **Repeat**

To repeat drilling for equally–spaced holes, specify the number of repeats in K_.
K is effective only within the block where it is specified.
Specify the first hole position in incremental mode.
If it is specified in absolute mode, drilling is repeated at the same position.

Number of repeats K	The maximum command value = 9999

If K0 is specified, drilling data is stored, but drilling is not performed.

- **M code used for C–axis clamp/unclamp**

When an M code specified in parameter No.5110 for C–axis clamp / unclamp is coded in a program, the CNC issues the M code for C–axis clamp after the tool is positioned and before the tool is fed in rapid traverse to the point–R level. The CNC also issues the M code for C–axis unclamp after the tool retracts to the point–R level. The tool dwells for the time specified in parameter No. 5112.

- **Cancel**

To cancel a canned cycle, use G80 or a group 01 G code.

<u>**Group 01 G codes**</u>

G00 : Positioning (rapid traverse)
G01 : Linear interpolation
G02 : Circular interpolation (CW)
G03 : Circular interpolation (CCW)

- **Symbols in figures**

Subsequent sections explain the individual canned cycles. Figures in these explanations use the following symbols:

– – →	Positioning (rapid traverse G00)
——→	Cutting feed (linear interpolation G01)
∼∼→	Manual feed
OSS	Oriented spindle stop (The spindle stops at a fixed rotation position)
⇨	Shift (rapid traverse G00)
P1	Dwell specified in the program
P1	Dwell specified in parameter No.5111
Mα	Issuing the M code for C–axis clamp
Mβ	Issuing the M code for C–axis unclamp

14.3.1 Front drilling cycle (G83) / side drilling cycle (G87)

The peck drilling cycle or high–speed peck drilling cycle is used depending on the setting in RTR, bit 2 of parameter No. 5101. If depth of cut for each drilling is not specified, the normal drilling cycle is used.

- **High–speed peck drilling cycle (G83, G87) (parameter RTR (No. 5101#2) =0)**

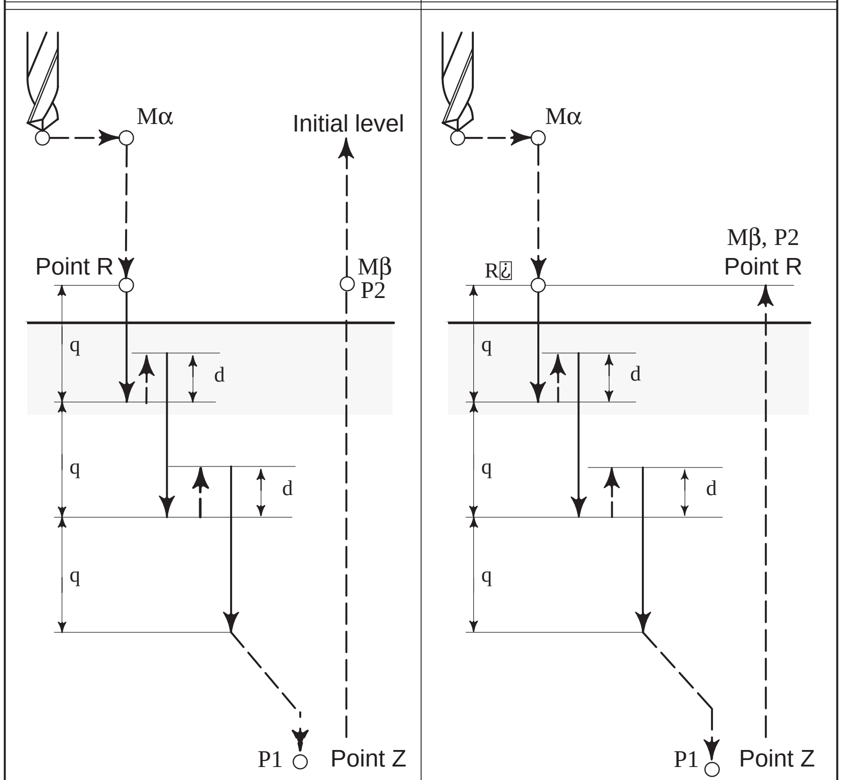
This cycle performs high–speed peck drilling. The drill repeats the cycle of drilling at the cutting feedrate and retracting the specified retraction distance intermittently to the bottom of a hole. The drill draws cutting chips out of the hole when it retracts.

Format

G83 X(U)_ C(H)_ Z(W)_ R_ Q_ P_ F_ M_ ;
or
G87 Z(W)_ C(H)_ X(U)_ R_ Q_ P_ F_ M_ ;

X_ C_ or Z_ C_ : Hole position data
Z_ or X_ : The distance from point R to the bottom of the hole
R_ : The distance from the initial level to point R level
Q_ : Depth of cut for each cutting feed
P_ : Dwell time at the bottom of a hole
F_ : Cutting feedrate
K_ : Number of repeats (When it is needed)
M_ : M code for C–axis clamp (When it is needed.)

G83 or G87 (G98 mode)	G83 or G87 (G99 mode)



Mα : M code for C–axis clamp
Mβ : M code for C–axis unclamp
P1 : Dwell specified in the program
P2 : Dwell specified in parameter No. 5112
d : Retraction distance specified in parameter No. 5114

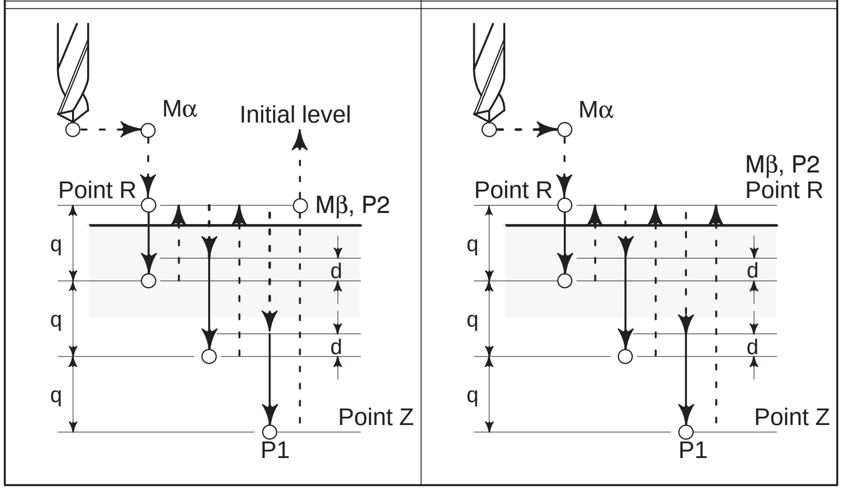
- **Peck drilling cycle (G83, G87) (parameter No. 5101#2 =1)**

Format

G83 X(U)_ C(H)_ Z(W)_ R_ Q_ P_ F_ M_ K_ ;
or
G87 Z(W)_ C(H)_ X(U)_ R_ Q_ P_ F_ M_ K_ ;

X_ C_ or Z_ C_ : Hole position data
Z_ or X_ : The distance from point R to the bottom of the hole
R_ : The distance from the initial level to point R level
Q_ : Depth of cut for each cutting feed
P_ : Dwell time at the bottom of a hole
F_ : Cutting feedrate
K_ : Number of repeats (When it is needed.)
M_ : M code for C–axis clamp (When it is needed.)

G83 or G87 (G98 mode)	G83 or G87 (G99 mode)



Mα : M code for C–axis clamp
Mβ : M code for C–axis unclamp
P1 : Dwell specified in the program
P2 : Dwell specified in parameter No.5112
d : Retraction distance specified in parameter No. 5114

Examples

M51 ; **Setting C–axis index mode ON**
M3 S2000 ; **Rotating the drill**
G00 X50.0 C0.0 ; **Positioning the drill along the X– and C–axes**
G83 Z–40.0 R–5.0 Q5000 F5.0 M31 ; **Drilling hole 1**
C90.0 M31 ; **Drilling hole 2**
C180.0 M31 ; **Drilling hole 3**
C270.0 M31 ; **Drilling hole 4**
G80 M05 ; **Canceling the drilling cycle and stopping drill rotation**
M50 ; **Setting C–axis index mode off**

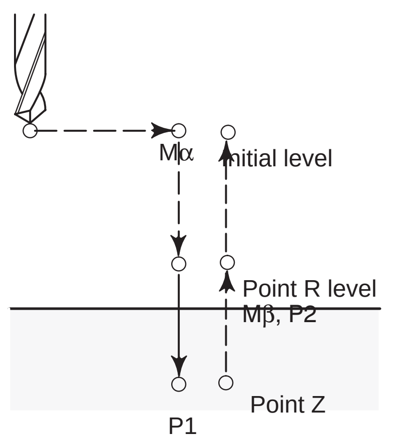
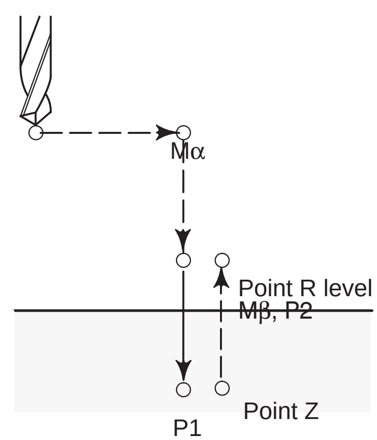
- **Drilling cycle (G83 or G87)**

If depth of cut is not specified for each drilling, the normal drilling cycle is used. The tool is then retracted from the bottom of the hole in rapid traverse.

Format

G83 X(U)_ C(H)_ Z(W)_ R_ P_ F_ M_ K_ ;
or
G87 Z(W)_ C(H)_ X(U)_ R_ P_ F_ M_ K_ ;

X_ C_ or Z_ C_ : Hole position data
Z_ or X_ : The distance from point R to the bottom of the hole
R_ : The distance from the initial level to point R level
P_ : Dwell time at the bottom of a hole
F_ : Cutting feedrate
K_ : Number of repeats (When it is needed.)
M_ : M code for C–axis clamp (When it is needed.)

G83 or G87 (G98 mode)	G83 or G87 (G99 mode)
	

Mα : M code for C–axis clamp
Mβ : M code for C–axis unclamp
P1 : Dwell specified in the program
P2 : Dwell specified in parameter No. 5112

Examples

M51 ; Setting C–axis index mode ON
M3 S2000 ; Rotating the drill
G00 X50.0 C0.0 ; Positioning the drill along the X–and C–axes
G83 Z–40.0 R–5.0 P500 F5.0 M31 ; Drilling hole 1
C90.0 M31 ; Drilling hole 2
C180.0 M31 ; Drilling hole 3
C270.0 M31 ; Drilling hole 4
G80 M05 ; Canceling the drilling cycle and stopping drill rotation
M50 ; Setting C–axis index mode off

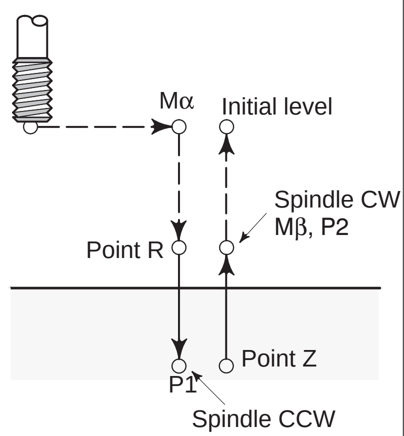
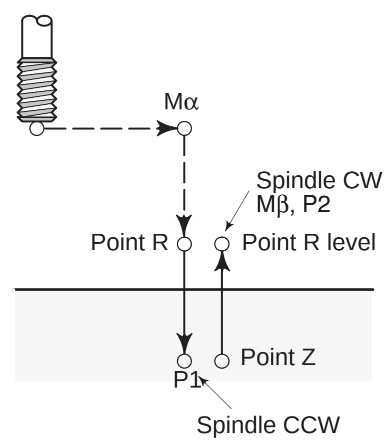
14.3.2 Front Tapping Cycle (G84) / Side Tapping Cycle (G88)

This cycle performs tapping.
In this tapping cycle, when the bottom of the hole has been reached, the spindle is rotated in the reverse direction.

Format

G84 X(U)_ C(H)_ Z(W)_ R_ P_ F_ M_ K_ ;
or
G88 Z(W)_ C(H)_ X(U)_ R_ P_ F_ M_ K_ ;

X_ C_ or Z_ C_ : Hole position data
Z_ or X_ : The distance from point R to the bottom of the hole
R_ : The distance from the initial level to point R level
P_ : Dwell time at the bottom of a hole
F_ : Cutting feedrate
K_ : Number of repeats (When it is needed.)
M_ : M code for C–axis clamp (when it is needed.)

G84 or G88 (G98 mode)	G84 or G88 (G99 mode)
	

Explanations

Tapping is performed by rotating the spindle clockwise. When the bottom of the hole has been reached, the spindle is rotated in the reverse direction for retraction. This operation creates threads.
Feedrate overrides are ignored during tapping. A feed hold does not stop the machine until the return operation is completed.

Notes

Bit 6 of parameter No. 5101 specifies whether the spindle stop command (M05) is issued before the direction in which the spindle rotates is specified with M03 or M04. For details, refer to the operator's manual created by the machine tool builder.

Examples

M51 ; Setting C–axis index mode ON
M3 S2000 ; Rotating the drill
G00 X50.0 C0.0 ; Positioning the drill along the X– and C– axes
G83 Z–40.0 R–5.0 P500 F5.0 M31 ; Drilling hole 1
C90.0 M31 ; Drilling hole 2
C180.0 M31 ; Drilling hole 3
C270.0 M31 ; Drilling hole 4
G80 M05 ; Canceling the drilling cycle and stopping drill rotation
M50 ; Setting C–axis index mode off

14.3.3
Front Boring Cycle (G85) / Side Boring Cycle (G89)

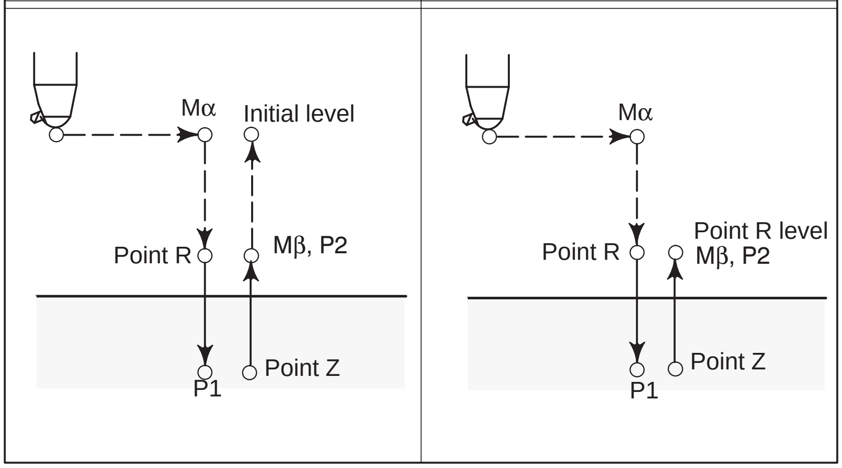
This cycle is used to bore a hole.

Format

G85 X(U)_ C(H)_ Z(W)_ R_ P_ F_ K_ M_ ;
or
G89 Z(W)_ C(H)_ X(U)_ R_ P_ F_ K_ M_ ;

X_ C_ or Z_ C_ : Hole position data
Z_ or X_ : The distance from point R to the bottom of the hole
R_ : The distance from the initial level to point R level
P_ : Dwell time at the bottom of a hole
F_ : Cutting feedrate
K_ : Number of repeats (When it is needed.)
M_ : M code for C–axis clamp (When it is needed.)

G85 or G89 (G98 mode)	G85 or G89 (G99 mode)



Explanations

After positioning, rapid traverse is performed to point R.
Drilling is performed from point R to point Z.
After the tool reaches point Z, it returns to point R at a feedrate twice the cutting feedrate.

Examples

M51 ; Setting C–axis index mode ON
M3 S2000 ; Rotating the drill
G00 X50.0 C0.0 ; Positioning the drill along the X– and C–axes
G83 Z–40.0 R–5.0 P500 F5.0 M31 ; Drilling hole 1
C90.0 M31 ; Drilling hole 2
C180.0 M31 ; Drilling hole 3
C270.0 M31 ; Drilling hole 4
G80 M05 ; Canceling the drilling cycle and stopping drill rotation
M50 ; Setting C–axis index mode off

14.3.4 Canned Cycle for Drilling Cancel (G80)

G80 cancels canned cycle.

Format

```
G80 ;
```

Explanations

Canned cycle for drilling is canceled to perform normal operation.
Point R and point Z are cleared. Other drilling data is also canceled (cleared).

Examples

M51 ; Setting C–axis index mode ON
M3 S2000 ; Rotating the drill
G00 X50.0 C0.0 ; Positioning the drill along the X– and C–axes.
G83 Z–40.0 R–5.0 P500 F5.0 M31 ; Drilling hole 1
C90.0 M31 ; Drilling hole 2
C180.0 M31 ; Drilling hole 3
C270.0 M31 ; Drilling hole 4
G80 M05 ; Canceling the drilling cycle and stopping drill rotation
M50 ; Setting C–axis index mode off

14.3.5 Precautions to be taken by operator

- **Reset and emergency stop**

Even when the controller is stopped by resetting or emergency stop in the course of drilling cycle, the drilling mode and drilling data are saved ; with this mind, therefore, restart operation.

- **Single block**

When drilling cycle is performed with a single block, the operation stops at the end points of operations 1, 2, 6 in Fig. 14.3 (a). Consequently, it follows that operation is started up 3 times to drill one hole. The operation stops at the end points of operations 1, 2 with the feed hold lamp ON. The operation stops in the feed hold conditions at the end point of operation 6 if the repeat remains, and it stops in stop conditions in other cases.

- **Feed hold**

When ”Feed Hold” is applied between operations 3 and 5 by G84/G88, the feed hold lamp lights up immediately if the feed hold is applied again to operation 6.

- **Override**

During operation with G84 and G88, the feedrate override is 100%.

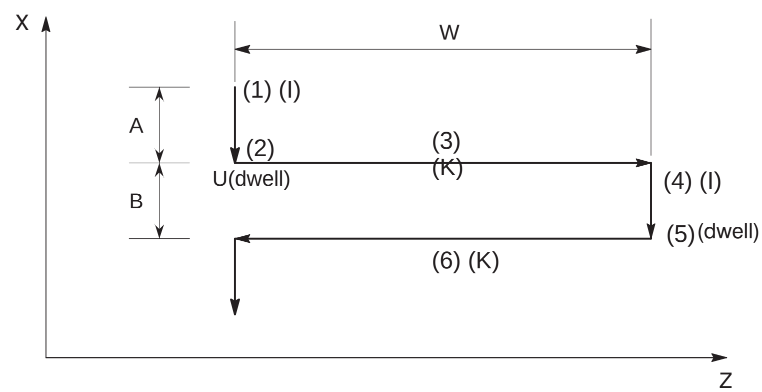
14.4 CANNED GRINDING CYCLE (FOR GRINDING MACHINE)

There are four grinding canned cycles : the traverse grinding cycle (G71), traverse direct fixed–dimension grinding cycle, oscillation grinding cycle, and oscillation direct fixed–dimension grinding cycle.
With a machine tool that allows canned cycles for grinding to be used, the multiple repetitive canned cycle for turning cannot be used.

14.4.1 Traverse grinding cycle (G71)

Format

G71 A_ B_ W_ U_ I_ K_ H_ ;



A : First depth of cut
B : Second depth of cut
W : Grinding range
U : Dwell time Maximum specification time : 99999.999 seconds
I : Feedrate of A and B
K : Feedrate of W
H : Number of repetitions Setting value : 1 to 9999

Explanations

The specification ranges and units of the canned cycle for grinding are described below.

Move command Range : ±8 digits
Units : 1 μ/0.0001 inch
0.1 μ/0.00001 inch

Feedrate Range
Feed per minute : 0.001 to 240000 mm/min
0.0001 to 9600 inch/min
(for 1 μ/0.0001 inch)
Feed per revolution : 0.00001 to 500 mm/rev
0.00001 to 9 inch/rev

A, B, and W are to be specified in an incremental mode.
In the case of a single block, the operations 1, 2, 3, 4, 5, and 6 are performed with one cycle start operation.
A=B=0 results in a spark–out.

14.4.2 Traverse direct fixed–dimension grinding cycle (G72)

Format

G72 P_ A_ B_ W_ U_ I_ K_ H_ ;

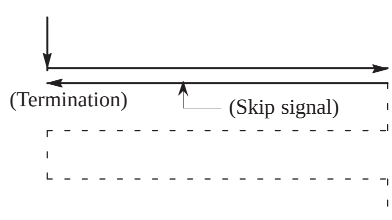
P : Gauge number (1 to 4)
A : First depth of cut
B : Second depth of cut
W : Grinding range
U : Dwell time   Maximum specification time : 99999.999seconds
I : Feedrate of A and B
K : Feedrate of W
H : Number of repetitions   Setting value : 1 to 9999

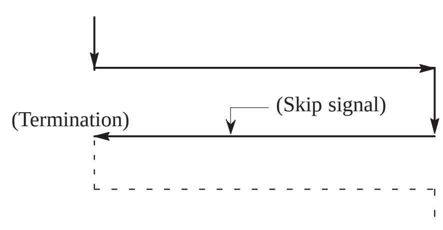
Explanations

When the multistage skip operation is used, a gauge number can be specified. The method of gauge number specification is the same as the method of multistage skip function. When the multistage skip operation is not used, the conventional skip signal is valid.
The same specifications as G71 apply except for gauge number specification.

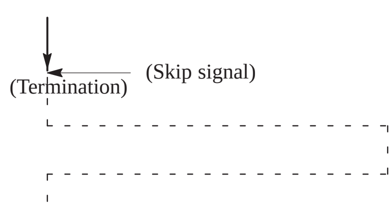
- **Operation at the time of skip signal input**

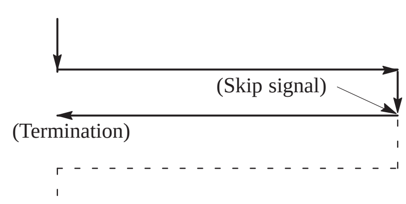
1. When the tool moves along the Z–axis to grind a workpiece, if a skip signal is input, the tool returns to the Z coordinate where the cycle started after the tool reaches the end of the specified grinding area.





2. When the tool cuts a workpiece along the X–axis, if a skip signal is input, the tool stops cutting immediately and returns to the Z coordinate where the cycle started.



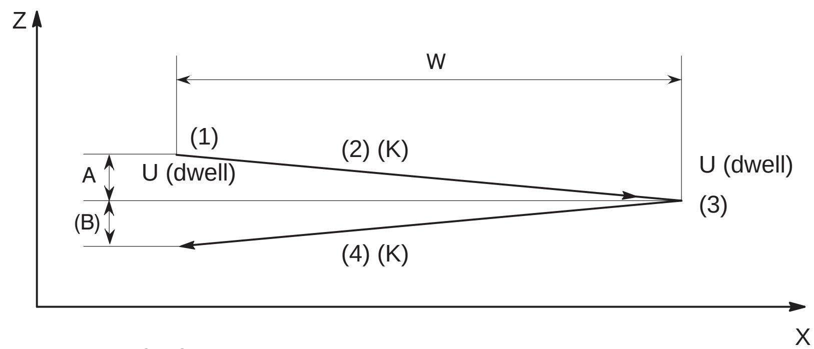


3. The skip signal is valid during dwell, without being affected by parameters DS1 to DS8 (No. 6206#0 to #7). Dwell is immediately stopped for return to the Z axis coordinate where the cycle started.

14.4.3 Oscillation Grinding Cycle (G73)

Format

G73 A_ (B_) W_ U_ K_ H_ ;



A : Depth of cut
B : Depth of cut
W : Grinding range
U : Dwell time
K : Feedrate
H : Number of repetitions Setting value : 1A9999

Explanations

A, B, and W are to be specified in an incremental mode.
In the case of a single block, the operations 1, 2, 3, and 4 are performed with one cycle start operation.
The specification of B is valid only for a specified block. This is not associated with B of the G71 or G72 cycle.

14.4.4 Oscillation Direct Fixed–Dimension Grinding Cycle

Format

G74 P_ A_ (B_) W_ U_ K_ H_ ;

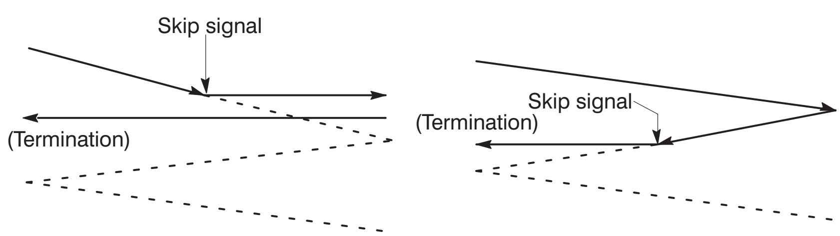
P : Gauge number (1 to 4)
A : Depth of cut
B : Depth of cut
W : Grinding range
U : Dwell time
K : Feedrate of W
H : Number of repetitions Setting value : 1 to 9999

Explanations

When the multistage skip operation is used, a gauge number can be specified. The method of gauge number specification is the same as the method of multistage skip function. When the multistage skip operation is not used, the conventional skip signal is valid.
The same specifications as G73 apply to the other items.

- **Operation at the time of skip signal input**

1. When the tool moves along the Z–axis to grind a workpiece, if a skip signal is input, the tool returns to the Z coordinate where the cycle started after the tool reaches the end of the specified grinding area.



2. The skip signal is valid during dwell, without being affected by parameters DS1 to DS8 (No. 6206#0 to #7). Dwell is immediately stopped for return to the Z axis coordinate where the cycle started.

Notes
1. The data items A, B, W, I, and K in a canned cycle are modal values common to G71 through G74. The data items A, B, W, U, I and K are cleared when a one–shot G code other than G04 or a 01 group G code other than G71 to G74 is specified.
2. No B code can be specified in the canned cycle mode.

14.5 CHAMFERING AND CORNER R

A chamfer or corner can be inserted between two blocks which intersect at a right angle as follows :

- **Chamfering Z → X**

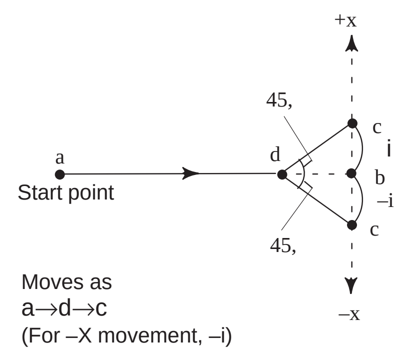
Format	Tool movement
G01 Z(W) I (C) <u>±i</u> ; Specifies movement to point b with an absolute or incremental command in the figure on the right.	

Fig. 14.5 (a) Chamfering (Z→X)

- **Chamfering X → Z**

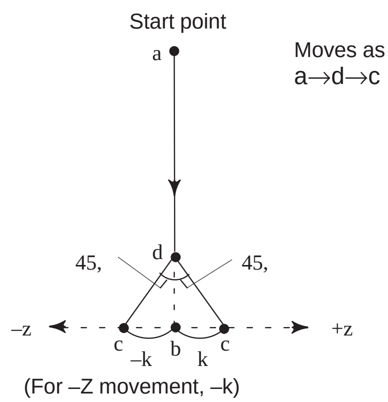
Format	Tool movement
G01 X(U) K (C) <u>±k</u> ; Specifies movement to point b with an absolute or incremental command in the figure on the right.	

Fig. 14.5 (b) Chamfering (X→Z)

- **Corner R Z → X**

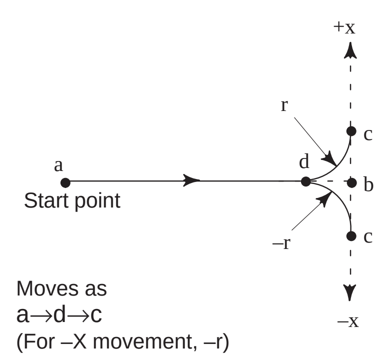
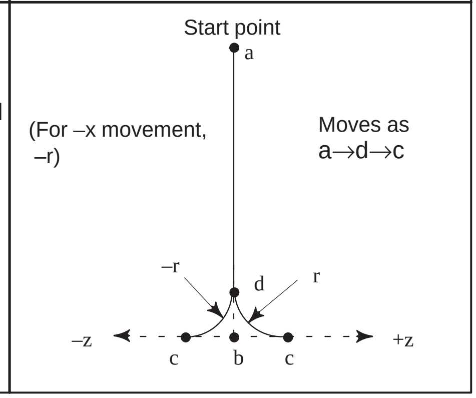
Format	Tool movement
G01 Z(W) R <u>±r</u> ; Specifies movement to point b with an absolute or incremental command in the figure on the right.	

Fig. 14.5 (c) Corner R (Z→X)

- **Corner R X → Z**

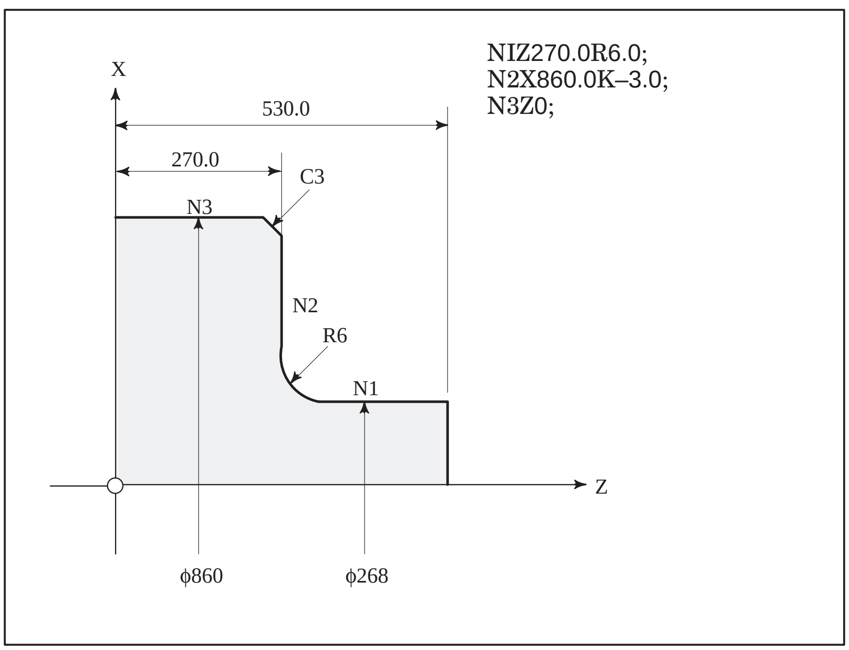
Format	Tool movement
G01 X(U) R ±r ; Specifies movement to point b with an absolute or incremental command in the figure on the right.	

14.5 (d) Corner R (X→Z)

Explanations

The movement for chamfering or corner R must be a single movement along the X or Z axis in G01 mode. The next block must be a single movement along the X or Z axis perpendicular to the former block.
I or K, and R always specify a radius value.
Note that the start point for a command specified in a block following a chamfering or corner–R block is not point c but point b shown in Figs. 14.5 (a) to (d). In incremental programming, specify a distance from point b.

Examples



Notes

1 The following commands cause an alarm.
   1) One of I, K, or R is commanded when X and Z axes are specified by G01. (P/S alarm No. 054)
   2) Move amount of X or Z is less than chamfering value and corner R value in the block where chamfering and corner R are specified. (P/S alarm No. 055)
   3) Next block to the block where chamfering and corner R were specified, has not G01 command. (P/S alarm No. 051, 052)
   4) If more than one of I, K, and R are specified in G01, P/S alarm No. 053 is issued.

2 A single block stops at point c of Fig. 15.5 (a) A 15.5 (d), not at point d.

3 Chamfering and corner R cannot be applied to a thread cutting block.

4 C can be used instead of I or K as an address for chamfering on the system which does not use C as an axis name. To use C for an address for chamfering, fix parameter CCR No. 3405#4 to 1.

5 If both C and R are specified with G01 in a block, the address specified last is valid.

6 Neither chamfering nor corner–R machining can be specified in direct drawing dimension programming.

14.6 MIRROR IMAGE FOR DOUBLE TURRET (G68, G69)

Format

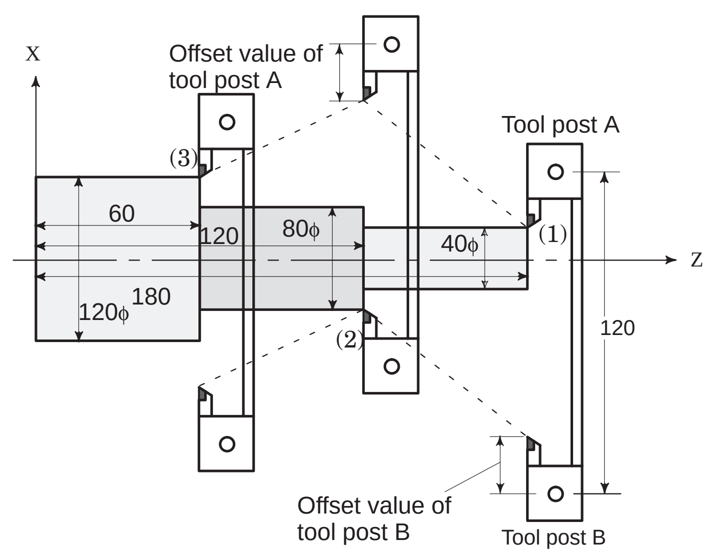
G68 : Double turret mirror image on
G69 : Mirror image cancel

Explanations

Mirror image can be applied to X–axis with G code.
When G68 is designated, the coordinate system is shifted to the mating turret side, and the X–axis sign is reversed from the programmed command to perform symmetrical cutting. To use this function, set the distance between the two turrets to a parameter (No. 1290).

Examples

- **Double turret programming**



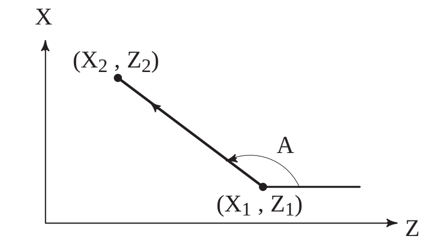
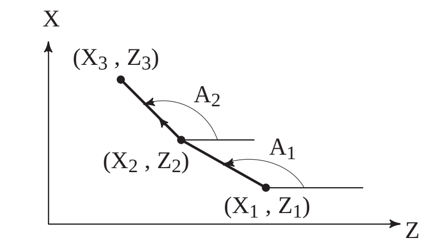
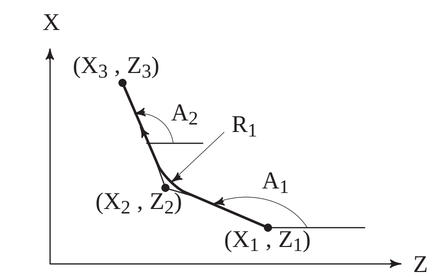
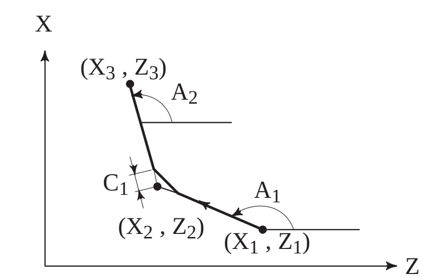
X40.0 Z180.0 T0101 ;	Position turret A at (1)
G68 ;	Shift the coordinate system by the distance A to B (120mm), and turn mirror image on.
X80.0 Z120.0 T0202 ;	Position turret B at (2)
G69 ;	Shift the coordinate system by the distance B to A, and turn mirror image cancel.
X120.0 Z60.0 T0101 ;	Position turret A at (3)

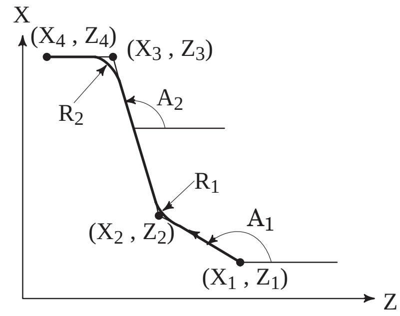
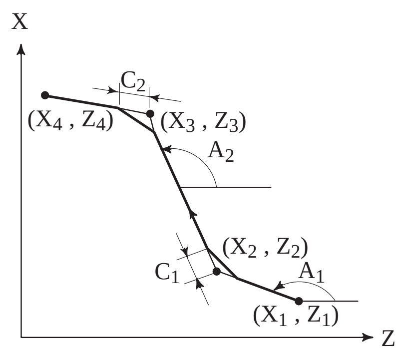
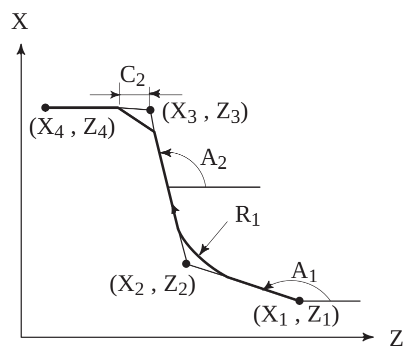
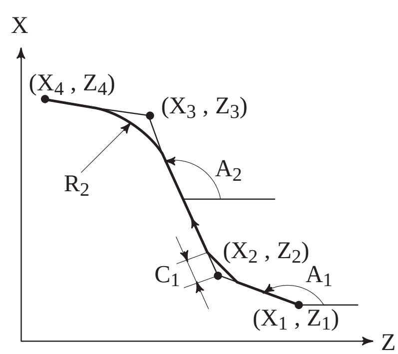
14.7 DIRECT DRAWING DIMENSIONS PROGRAMMING

Angles of straight lines, chamfering value, corner rounding values, and other dimensional values on machining drawings can be programmed by directly inputting these values. In addition, the chamfering and corner rounding can be inserted between straight lines having an optional angle. This programming is only valid in memory operation mode.

Format

Table 14.7 (a) Commands table

	Commands	Movement of tool
1	X_2_ (Z_2_), A_ ;	
2	,A_1_ ; X_3_ Z_3_, A_2_ ;	
3	X_2_ Z_2_, R_1_ ; X_3_ Z_3_ ; or ,A_1_, R_1_ ; X_3_ Z_3_, A_2_ ;	
4	X_2_ Z_2_, C_1_ ; X_3_ Z_3_ ; or ,A_1_, C_1_ ; X_3_ Z_3_, A_2_ ;	

	Commands	Movement of tool
5	X_2_ Z_2_ , R_1_ ; X_3_ Z_3_ , R_2_ ; X_4_ Z_4_ ; or ,A_1_, R_1_ ; X_3_ Z_3_, A_2_, R_2_ ; X_4_ Z_4_ ;	
6	X_2_ Z_2_ , C_1_ ; X_3_ Z_3_ , C_2_ ; X_4_ Z_4_ ; or ,A_1_, C_1_ ; X_3_ Z_3_, A_2_, C_2_ ; X_4_ Z_4_ ;	
7	X_2_ Z_2_ , R_1_ ; X_3_ Z_3_ , C_2_ ; X_4_ Z_4_ ; or ,A_1_, R_1_ ; X_3_ Z_3_, A_2_, C_2_ ; X_4_ Z_4_ ;	
8	X_2_ Z_2_ , C_1_ ; X_3_ Z_3_ , R_2_ ; X_4_ Z_4_ ; or ,A_1_, C_1_ ; X_3_ Z_3_, A_2_, R_2_ ; X_4_ Z_4_ ;	

Explanations

A program for machining along the curve shown in Fig. 14.7 (a) is as follows :

```
X (x2) Z (z2) , C (c1) ;
X (x3) Z (z3) , R (r2) ;
X (x4) Z (z4) ;

  or

,A (a1) , C (c1) ;
X (x3) Z (z3) , A (a2) , R (r2) ;
X (x4) Z (z4) ;
```

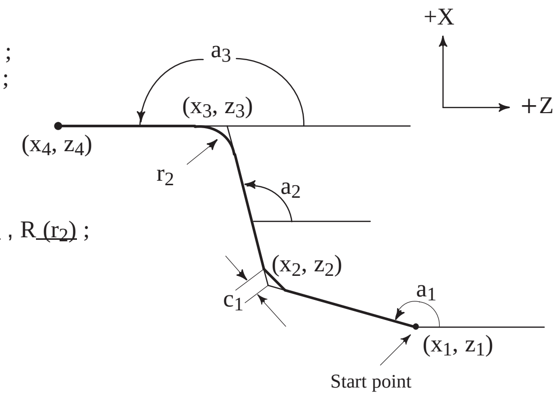


Fig. 14.7 (a) Machining Drawing (example)

For command a straight line, specify one or two out of X, Z, and A.
If only one is specified, the straight line must be primarily defined by a command in the next block.
To command the degree of a straight line or the value of chamfering or corner R, command with a comma (,) as follows :

, A_
, C_
, R_

By specifying 1 to parameter CCR No. 3405#4 on the system which does not use A or C as an axis name, the degree of a straight line or the value of chamfering or corner R can be commanded without a comma (,) as follows :

A_
C_
R_

Notes

1 The following G codes are not applicable to the same block as commanded by direct input of drawing dimensions or between blocks of direct input of drawing dimensions which define sequential figures.
   1) G codes (other than G04) in group 00.
   2) G02, G03, G90, G92, and G94 in group 01.

2 Corner rounding cannot be inserted into a threading block.

3 Neither chamfering and corner rounding commands specified in 14.5 nor those of direct input of drawing dimensions can be used concurrently.

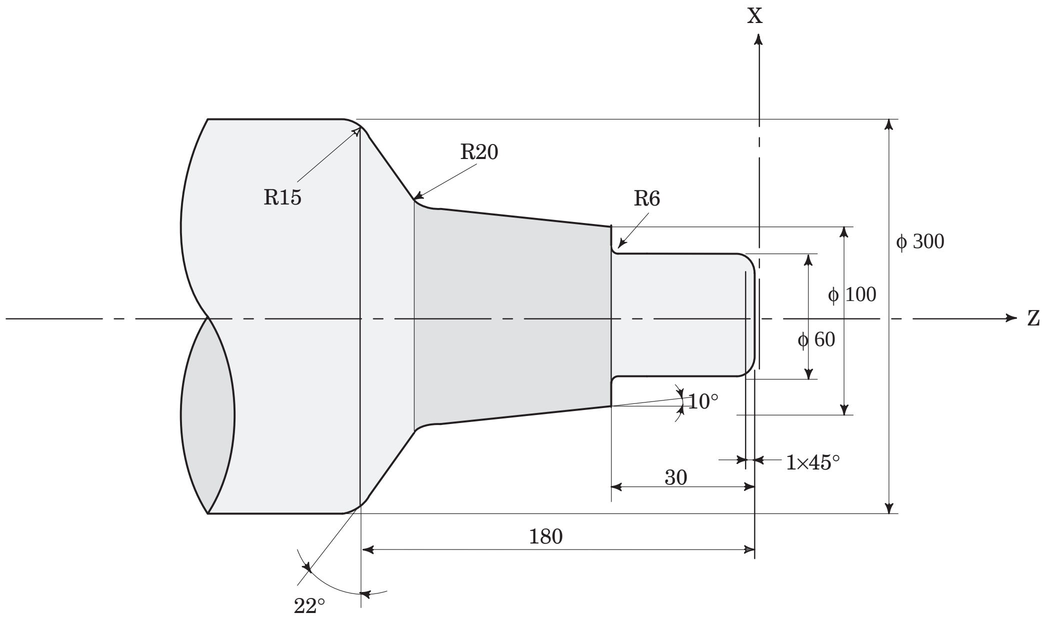
4 When the end point of the previous block is determined in the next block according to sequential commands of direct input of drawing dimensions, the single block stop is not done, but the feed hold stop is done at the end point of the previous block.

5 The angle allowance in calculating the point of intersection in the program below is ±1°. (Because the travel distance to be obtained in this calculation is too large.)
   1) X_ , A_ ; (If a value within 0°±1°or 180°±1°is specified for the angle instruction, the P/S alarm No.057 occurs.)
   2) Z_ , A_ ; (If a value within 90°±1°or 270°±1°is specified for the angle instruction, the P/S alarm No. 057 occurs.)

6 An alarm occurs if the angle made by the 2 lines is within ±1°when calculating the point of intersection.

7 Chamfering or corner % is ignored if the angle made by the 2 lines is within ±1°.

8 Both a dimensional command (absolute programming) and angle instruction must be specified in the block following a block in which only the angle instruction is specified.
   (Example)
   N1 X_, A_, R_ ;
   N2, A_ ;
   N3 X_ Z_, A_ ;
   (In addition to the dimentional command, angle instruction must be specified in block No. 3.)



(Diameter specification, metric input)

```
N001  G50  X0.0  Z0.0 ;
N002  G01  X60.0,  A90.0,  C1.0  F80 ;
N003  Z–30.0,  A180.0,  R6.0 ;
N004  X100.0,  A90.0 ;
N005  ,A170.0,  R20.0 ;
N006  X300.0  Z–180.0,  A112.0, R15.0 ;
N007  Z–230.0,  A180.0 ;
      :
      :
```

14.8 RIGID TAPPING

Front face tapping cycles (G84) and side face tapping cycles (G88) can be performed either in conventional mode or rigid mode.
In conventional mode, the spindle is rotated or stopped, in synchronization with the motion along the tapping axis according to miscellaneous functions M03 (spindle CW rotation), M04 (spindle CCW rotation), and M05 (spindle stop).
In rigid mode, the spindle motor is controlled in the same way as a control motor, by the application of compensation to both motion along the tapping axis and that of the spindle.
For rigid tapping, each turn of the spindle corresponds to a certain amount of feed (screw lead) along the spindle axis. This also applies to acceleration/deceleration. This means that rigid tapping does not demand the use of float tappers as in the case of conventional tapping, thus enabling high–speed, high–precision tapping.
When the system is equipped with the optional multispindle control function, the second spindle can be used for rigid tapping.

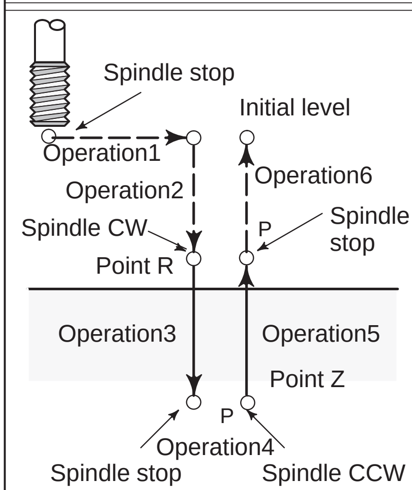
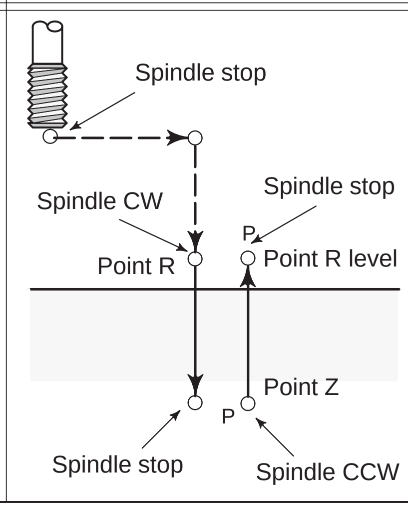
14.8.1 Front Face Rigid Tapping Cycle (G84)/Side Face Rigid Tapping Cycle (G88)

Controlling the spindle motor in the same way as a servo motor in rigid mode enables high–speed tapping.

Format

G84 X(U)_ C(H)_ Z(W)_ R_ P_ F_ M_ K_ ;
or
G88 Z(W)_ C(H)_ X(U)_ R_ P_ F_ M_ K_ ;

X_ C_ or Z_ C_ : Hole position data
Z_ or X_ : The distance from point R to the bottom of the hole
R_ : The distance from the initial level to point R level
P_ : Dwell time at the bottom of a hole
F_ : Cutting feedrate
K_ : Number of repeats (When it is needed.)
M_ : M code for C–axis clamp (when it is needed.)

G84 or G88 (G98 mode)	G84 or G88 (G99 mode)
	

Explanations

Once positioning for the X–axis (G84) or Z–axis (G88) has been completed, the spindle is moved, by rapid traverse, to point R. Tapping is performed from point R to point Z, after which the spindle stops and observes a dwell time. Then, the spindle starts reverse rotation, retracts to point R, stops rotating, then moves to the initial level by rapid traverse. During tapping, the feedrate override and spindle override are assumed to be 100%. For retraction (operation 5), however, a fixed override of up to 200% can be applied by setting parameter No. 5211 (RGOVR) and bit 4 (DOV) of parameter No. 5200.

- **Rigid mode**

Rigid mode can be specified by applying any of the following methods:

- Specifying M29S***** before a tapping block
- Specifying M29S***** within a tapping block
- Handling G84 or G88 as a G code for rigid tapping (Set bit 0 (G84) of parameter No. 5200 to 1.)

- **Screw lead**

In feed per minute mode, the feedrate divided by the spindle speed is equal to the screw lead. In feed per rotation mode, the feedrate is equal to the screw lead.

Limitations

- **S commands**

When a value exceeding the maximum rotation speed for the gear being used is specified, P/S alarm No. 200 is issued. For an analog spindle, when a command is specified such that more than 4095 pulses are generated during 8 ms (detection unit), P/S alarm No. 202 is issued. For a serial spindle, when a command is specified such that more than 32767 pulses are generated during 8 ms (detection unit), P/S alarm No. 202 is issued.

<Example>

For a built–in motor equipped with a detector having a resolution of 4095 pulses per rotation, the maximum spindle speed during rigid tapping is as follows:

For an analog spindle

$(4095\times1000\div8\times60)\div4095 = 7500$ (rpm)

For a serial spindle

$(32767\times1000\div8\times60)\div4095 = 60012$ (rpm) [Note: Ideal value]

- **F commands**

Specifying a value larger than the upper limit for cutting feed will cause P/S alarm No. 201 to be issued.

- **M29**

Specifying an S command or axis movement between M29 and M84 will cause P/S alarm No. 203 to be issued. Specifying M29 during a tapping cycle will cause P/S alarm No. 204 to be issued.

- **Rigid tapping command M code**

The M code used to specify rigid tapping mode is usually set in parameter No. 5210. To set a value of more than 255, however, use parameter No. 5212.

- **Maximum position deviation during movement along the tapping axis**

The maximum position deviation during movement along the tapping axis in rigid tapping mode is usually set in parameter No. 5310. Use parameter No. 5314, however, when setting a value of more than 32767, for example, according to the resolution of the detector being used.

- **R**

The value of R must be specified in a block which performs drilling. If the value is specified in a block which does not perform drilling, it is not stored as modal data.

- **Cancellation**

G00 to G03 (G codes in group 01) must not be specified in a block containing G84 or G88. If specified, G84 or G88 in that block is canceled.

- **Tool position offset**

Any tool position offset is ignored in canned cycle mode.

- **Units for F**

	Metric input	Inch input	Remark
G98	1 mm/min	0.01inch/min	Decimal point allowed
G99	0.01mm/rev	0.0001inch/rev	Decimal point allowed

Examples

Tapping axis feedrate: 1000 mm/min
Spindle speed: 1000 rpm
Screw lead: 1.0 mm

<Programming for feed per minute>

G98 ;	Command for feed per minute
G00 X100.0 ;	Positioning
M29 S1000 ;	Command for specifying rigid mode
G84 Z–100.0 R–20.0 F1000 ;	Rigid tapping

<Programming for feed per rotation>

G99 ;	Command for feed per rotation
G00 X100.0 ;	Positioning
M29 S1000 ;	Command for specifying rigid mode
G84 Z–100.0 R–20.0 F1.0 ;	Rigid tapping

15 COMPENSATION FUNCTION

This chapter describes the following compensation functions:

15.1 TOOL OFFSET

Tool offset is used to compensate for the difference when the tool actually used differs from the imagined tool used in programming (usually, standard tool).

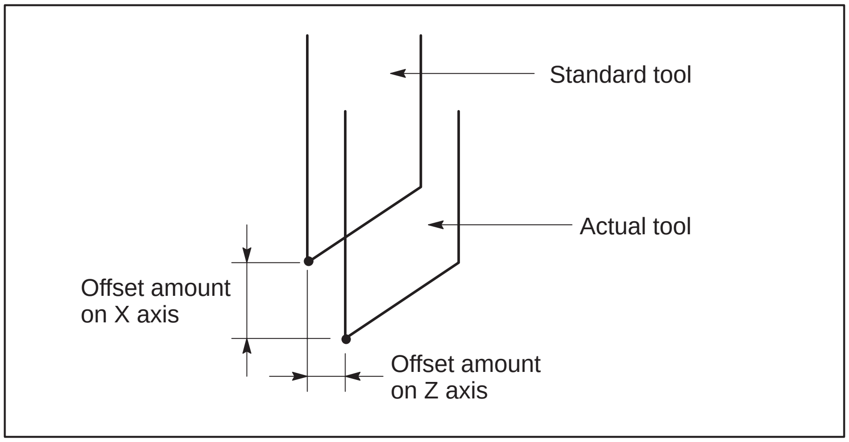


Fig.15.1(a) Tool offset

In this unit, there is no G code to specify tool offset.
The tool offset is specified by T code.

15.1.1 Tool geometry offset and tool wear offset

Tool geometry offset and tool wear offset are possible to divide the tool offset to the tool geometry offset for compensating the tool shape or tool mounting position and the tool wear offset for compensating the tool nose wear.
Total value of tool geometry offset value and tool wear offset value is set as the tool wear offset value without option.

Notes
Tool geometry offset and tool wear offset are optioned.

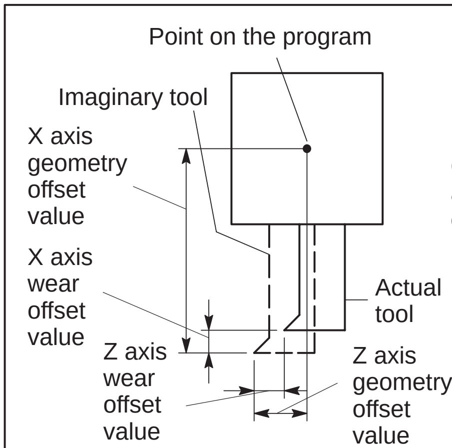


Fig. 15.1.1(a) Difference the tool geometry offset from tool wear offset

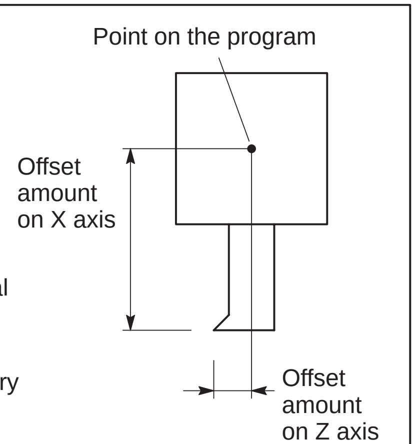


Fig. 15.1.1(b) Not difference the tool geometry offset from tool wear offset

15.1.2 T code for Tool Offset

There are two methods for specifying a T code as shown in Table 15.1.2(a) and Table 15.1.2(b).

Format

- **Lower digit of T code specifies geometry and wear offset number**

Table 15.1.2(a)

Kind of T code	Meaning of T code	Parameter setting for specifying of offset No.	
2–digit command	T ○ ○ Tool wear and tool geometry offset number Tool selection	When LD1, bit 0 of parameter No.5002, is set to 1, a tool wear offset number is specified with the last digit of a T code.	When LGN, bit 1 of parameter No.5002, is set to 0, the tool geometry offset number and tool wear offset number specified for a certain tool are the same.
4–digit command	T ○○ ○○ Tool wear and tool geometry offset number Tool selection	When LD1, bit 0 of parameter No.5002, is set to 0, a tool wear offset number is specified with the last two digits of a T code.	

- **Lower digit of T code specifies wear offset number and higher digit number specifies tool selection number and geometry offset number**

Table 15.1.2(b)

Kind of T code	Meaning of T code	Parameter setting for specifying of offset No.	
2–digit command	T ○ ○ Tool wear offset number Tool selection and tool geometry offset number	When LD1, bit 0 of parameter No.5002, is set to 1, a tool wear offset number is specified with the last digit of a T code.	When LGN, bit 1 of parameter No.5002, is set to 0, the tool geometry offset number and tool wear offset number specified for a certain tool are the same.
4–digit command	T ○○ ○○ Tool wear offset number Tool selection and tool geometry offset number	When LD1, bit 0 of parameter No.5002, is set to 0, a tool wear offset number is specified with the last two digits of a T code.	

15.1.3 Tool Selection

Tool selection is made by specifying the T code corresponding to the tool number. Refer to the machine tool builder's manual for the relationship between the tool selection number and the tool.

15.1.4 Offset Number

Tool offset number has two meanings.

It is specifies the offset distance corresponding to the number that is selected to begin the offset function. A tool offset number of 0 or 00 indicates that the offset amount is 0 and the offset is cancelled.

15.1.5 Offset

There are two types of offset. One is tool wear offset and the other is tool geometry offset.

Explanations

- **Tool wear offset**

The tool path is offset by the X, Y, and Z wear offset values for the programmed path. The offset distance corresponding to the number specified by the T code is added to or subtracted from the and position of each programmed block.

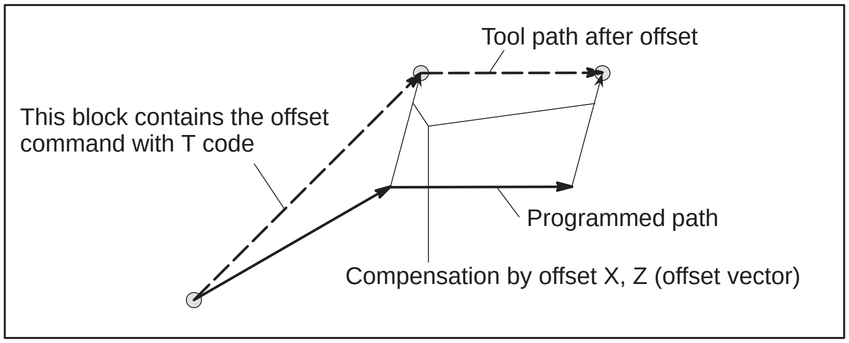


Fig.15.1.5(a) Movement of offset (1)

- **Offset vector**

In Fig.15.1.5(a), the vector with offset X, Y, and Z is called the offset vector. Compensation is the same as the offset vector.

- **Offset cancel**

Offset is cancelled when T code offset number 0 or 00 is selected. At the end of the cancelled block, the offset vector becomes 0.

N1 X50.0 Z100.0 T0202 ; Creates the offset vector corresponding to offset number 02
N2 X200.0 ;
N3 X100.0 Z250.0 T0200 ; Specifying offset number 00 deletes the offset vector.

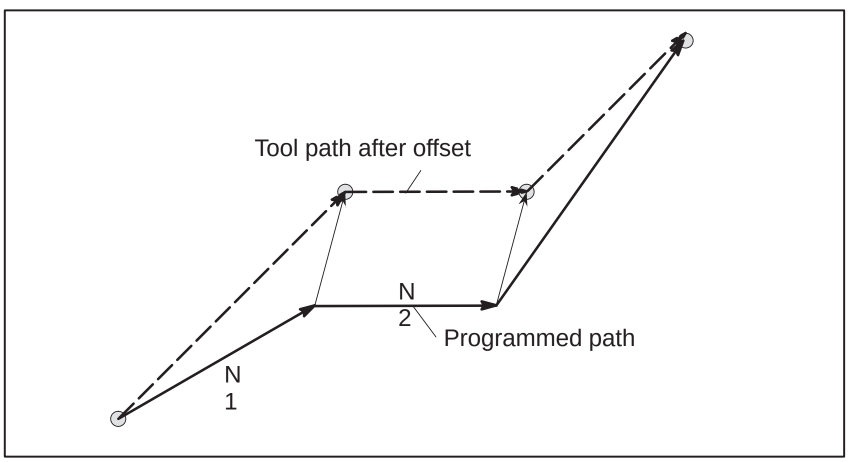


Fig.15.1.5(b) Movement of offset (2)

When the power is first turned on , and the reset key on the MDI units is pushed or the reset signal is input to the CNC from the machine tool, the offset is cancelled.
Parameter LVK (No.5003#6) can be set so that offset will not be cancelled by pressing the reset key or by reset input.

- **Only T code**

When only a T code is specified in a block, the tool is moved by the wear offset value without a move command. The movement is performed at rapid traverse rate in the G00 mode . It is performed at feedrate in other modes.
When a T code with offset number 0 or 00is specified by itself, movement is performed to cancel the offset.

> **Notes**
> When G50 X_Z_T_ ; is specified
> Tool is not moved.
> The coordinate system in which the coordinate value of the tool position is (X,Z) is set. The tool position is obtained by subtracting the wear offset value corresponding to the offset number specified in the T code.

- **Tool geometry offset**

With the tool geometry offset, the work coordinate system is shifted by the X, Y, and Z geometry offset amounts. Namely, the offset amount corresponding to the number designated with the code is added to or subtracted from the current position.

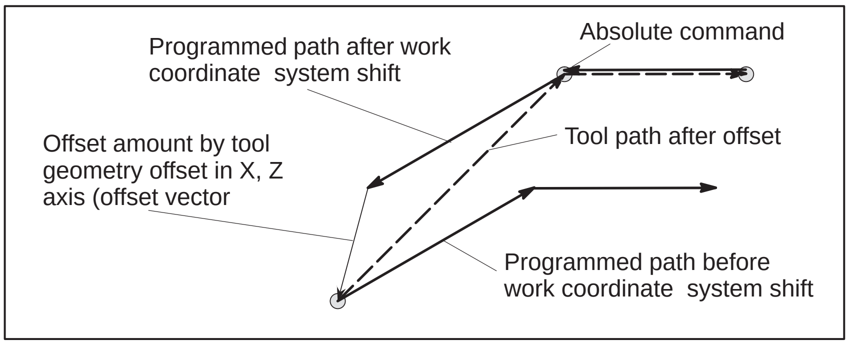


Fig.15.1.5(c) Movement of tool geometry offset

> **Notes**
> As well as wear offset, the tool can be compensated by parameter setting LGT(No.5002#4) to add or subtract the programmed end point of each block.

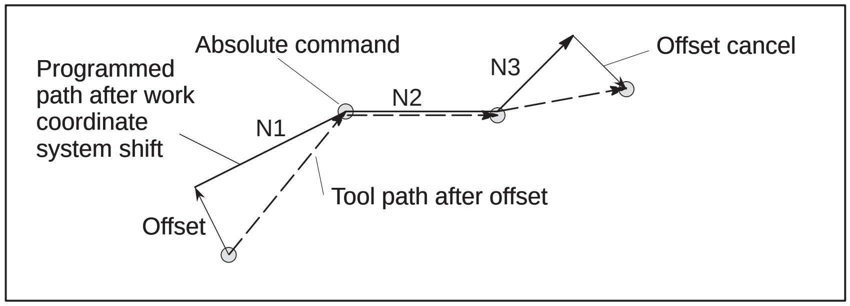
- **Offset cancel**

Specifying offset number 0, 00, or 0000 cancels offset.

> **Notes**
> When LGC, bit 5 of parameter No.5002, is set to 0, specifying offset number 0 or 00 does not cancel offset.

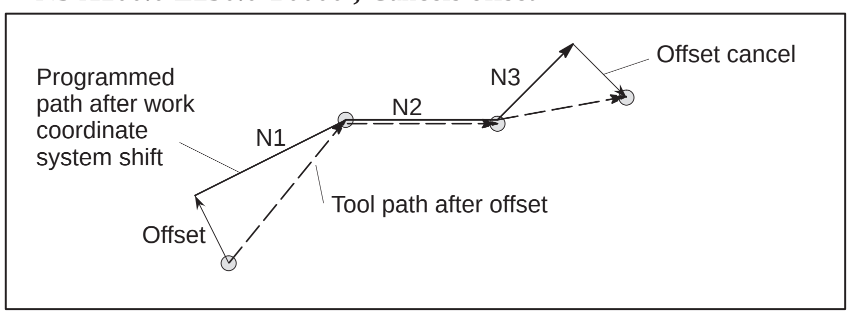
Examples

1. When a tool geometry offset number and tool wear offset number are specified with the last two digits of a T code
 (when LGN, bit 1 of parameter No.5002, is set 0),
 N1 X50.0 Z100.0 T0202 ; Specifies offset number 02
 N2 Z200.0 ;
 N3 X100.0 Z250.0 T0200 ; Cancels offset



Notes
When LGC, bit 5 of parameter No.5002, is set to 0, specifying offset number 0 does not cancel tool geometry offset.

2. Assume that geometry offset is not cancelled with offset No.0
 (Set the parameter (No.5002#1).)
 N1 X50.0 Z100.0 T0202 ; Tool selection number (specified tool geometry offset number 02)
 N2 Z200.0 ;
 N3 X100.0 Z250.0 T0000 ; Cancels offset



15.1.6 G53, G28, G30, and G30.1 Commands When Tool Position Offset is Applied

This section describes the following operations when tool position offset is applied: G53, G28, G30, and G30.1 commands, manual reference position return, and the canceling of tool position offset with a T00 command.

Explanations

- **Reference position return (G28) and G53 command when tool position offset is applied**

Executing reference position return (G28) or a G53 command when tool position offset is applied does not cancel the tool position offset vector. The absolute position display is as follows, however, according to the setting of bit 4 (LGT) of parameter No. 5002.

LGT = 0 (Tool geometry compensation is based on shift of the coordinate system.)

		Tool position offset (without option)	Tool geometry compensation	Tool wear compensation
Absolute position coordinate display	Block for reference position return or G53 command	The vector is not reflected. The coordinates are displayed as if the offset had been temporarily canceled.	The shift is reflected. Coordinates shifted according to the tool geometry compensation are displayed.	The vector is not reflected. The coordinates are displayed as if the offset had been temporarily canceled.
	Next block	The vector is reflected.	Coordinates shifted according to the tool geometry compensation are displayed.	The vector is reflected.

LGT = 1 (Tool geometry compensation is based on tool movement.)

		Tool position offset (without option)	Tool geometry compensation	Tool wear compensation
Absolute position coordinate display	Block for reference position return or G53 command	The vector is not reflected. The coordinates are displayed as if the offset had been temporarily canceled.	The vector is not reflected. The coordinates are displayed as if the offset had been temporarily canceled.	The vector is not reflected. The coordinates are displayed as if the offset had been temporarily canceled.
	Next block	The vector is reflected.	The vector is reflected.	The vector is reflected.

Notes

Bit 6 (DAL) of parameter No. 3104 is set to 0 (the actual positions to which the tool position offset is applied are displayed in the absolute position display).

- **Manual reference position return when tool position offset is applied**

Executing manual reference position return when tool position offset is applied does not cancel the tool position offset vector. The absolute position display is as follows, however, according to the setting of bit 4 (LGT) of parameter No. 5002.

LGT = 0 (Tool geometry compensation is based on shift of the coordinate system.)

		Tool position offset (without option)	Tool geometry compensation	Tool wear compensation
Absolute position coordinate display	Upon manual reference position return	The vector is not reflected. The coordinates are displayed as if the offset had been temporarily canceled.	The shift is reflected. Coordinates shifted according to tool geometry compensation are displayed.	The vector is not reflected. The coordinates are displayed as if the offset had been temporarily canceled.
	Next block	The vector is reflected.	Coordinates shifted according to tool geometry compensation are displayed.	The vector is reflected.

LGT = 1 (Tool geometry compensation is based on tool movement.)

		Tool position offset (without option)	Tool geometry compensation	Tool wear compensation
Absolute position coordinate display	Upon manual reference position return	The vector is not reflected. The coordinates are displayed as if the offset had been temporarily canceled.	The vector is not reflected. The coordinates are displayed as if the offset had been temporarily canceled.	The vector is not reflected. The coordinates are displayed as if the offset had been temporarily canceled.
	Next block	The vector is reflected.	The vector is reflected.	The vector is reflected.

Notes

Bit 6 (DAL) of parameter No. 3104 is set to 0 (the actual positions to which the tool position offset is applied are displayed in the absolute position display).

- **Canceling tool position offset with T00**

Whether specifying T00 alone, while tool position offset is applied, cancels the offset depends on the settings of the following parameters:

When the tool geometry/wear compensation option is selected

LGN = 0

LGN (No.5002#1)	LGT (No.5002#4)	LGC (No.5002#5)	
The geometry offset number is: 0: Same as the wear offset number 1: Same as the tool selection number	Geometry compensation is applied: 0: Based on shift of the coordinate system 1: Based on movement of the tool	The geometry offset is: 0: Not canceled with T00 1: Canceled with T00	Result
LGT=0	LGT=0	LGC=0 LGC=1	Not canceled Canceled
		LWM (No.5002#6)	
		Tool position offset is applied: 0: By means of T code 1: By means of movement along axis	
	LGT=1	LWM=0 LWM=1	Canceled Not canceled

Notes
1 When LGT=0, LWM is unrelated.
2 When LGT=1, LGC is unrelated, even when LGN = 0.

LGN = 1

LGN (No.5002#1)	LGT (No.5002#4)	LGC (No.5002#5)	
The geometry offset number is: 0: Same as the wear offset number 1: Same as the tool selection number	Geometry compensation is applied: 0: Based on shift of the coordinate system 1: Based on movement of the tool	The geometry offset is: 0: Not canceled with T00 1: Canceled with T00	Result
LGT=0	LGT=0	LGC is unrelated.	Canceled
		LWM (No.5002#6)	
		Tool position offset is applied: 0: By means of T code 1: By means of movement along axis	
	LGT=1	LWM=0 LWM=1	Canceled Not canceled

Notes
1 When LGT = 0, LWM is unrelated.
2 When LGT = 1, LGC is unrelated.

When the tool geometry/wear compensation option is not selected

<table>
<tr><th>LGN (No.5002#1)</th><th>LGT (No.5002#4)</th><th>LGC (No.5002#5)</th><th></th></tr>
<tr><td>The geometry offset number is:
0: Same as the wear offset number
1: Same as the tool selection number</td><td>Geometry compensation is applied:
0: Based on shift of the coordinate system
1: Based on movement of the tool</td><td>The geometry offset is:
0: Not canceled with T00
1: Canceled with T00</td><td>Result</td></tr>
<tr><td rowspan="4">LGN is unrelated.

The tool position offset number always uses the low–order digits.</td><td rowspan="4">LGT is unrelated.

Tool position offset is always applied based on the movement of the tool.</td><td>LGC is unrelated.</td><td rowspan="3"></td></tr>
<tr><td>LWM (No.5002#6)</td></tr>
<tr><td>Tool position offset is applied:
0: By means of T code
1: By means of movement along axis</td></tr>
<tr><td>LWM=0
LWM=1</td><td>Canceled
Not canceled</td></tr>
</table>

15.2 OVERVIEW OF TOOL NOSE RADIUS COMPENSATION

It is difficult to produce the compensation necessary to form accurate parts when using only the tool offset function due to tool nose roundness in taper cutting or circular cutting. The tool nose radius compensation function compensates automatically for the above errors.

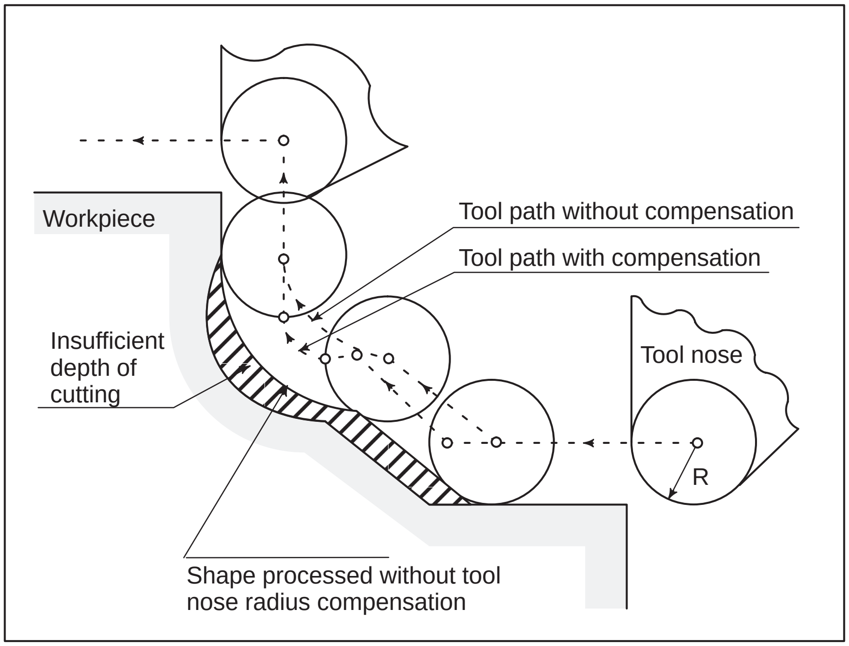


Fig15.2 Tool path of tool nose radius compensation

15.2.1 Imaginary Tool Nose

The tool nose at position A in following figure does not actually exist.
The imaginary tool nose is required because it is usually more difficult to set the actual tool nose radius center to the start position than the imaginary tool nose (Note).
Also when imaginary tool nose is used, the tool nose radius need not be considered in programming.
The position relationship when the tool is set to the start position is shown in the following figure.

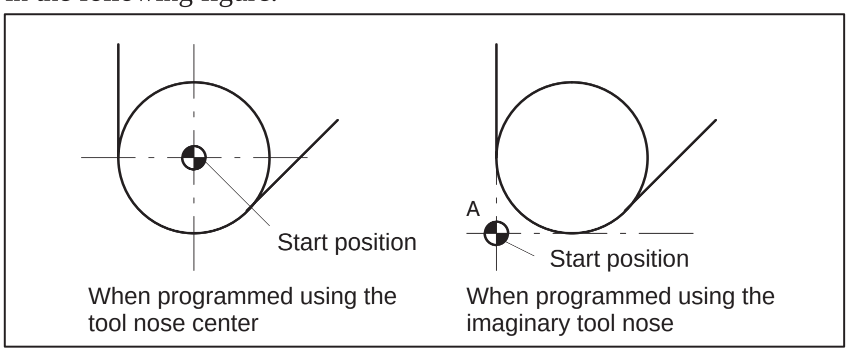


Fig.15.2.1(a) Tool nose radius center and imaginary tool nose

Notes

In a machine with reference positions, a standard position like the turret center can be placed over the start position. The distance from this standard position to the nose radius center or the imaginary tool nose is set as the tool offset value.

Setting the distance from the standard position to the tool nose radius center as the offset value is the same as placing the tool nose radius center over the start position, while setting the distance from the standard position to the imaginary tool nose is the same as placing the imaginary tool nose over the standard position. To set the offset value, it is usually easier to measure the distance from the standard position to the imaginary tool nose than from the standard position to the tool nose radius center.

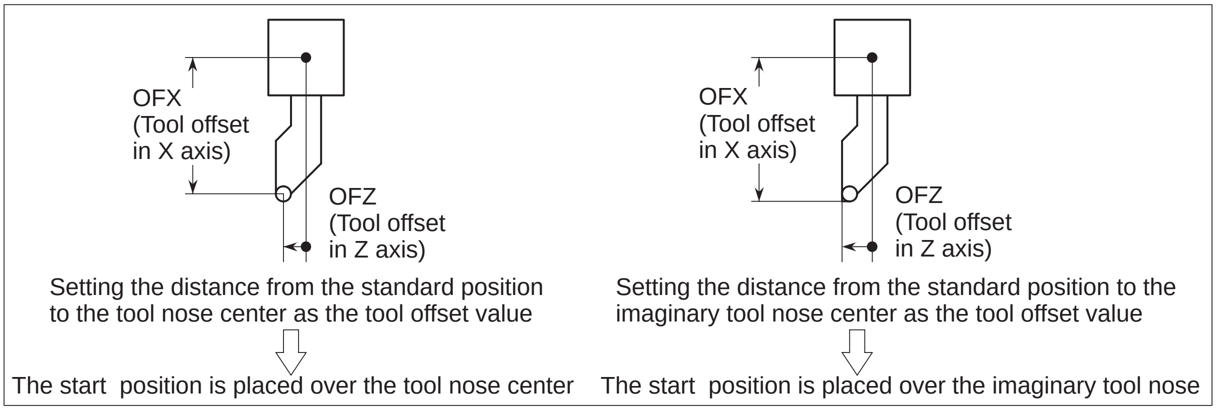


Fig15.2.1(b) Tool offset value when the turret center is placed over the start position

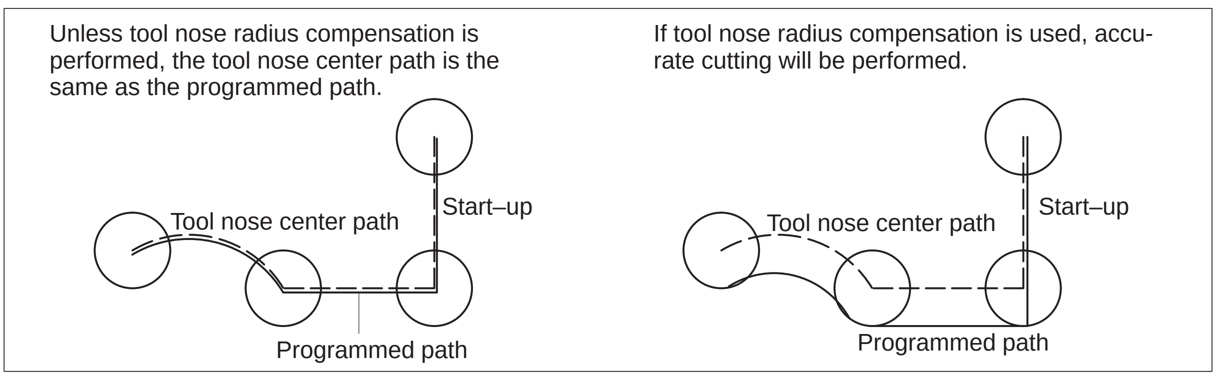


Fig15.2.1(c) Tool path when programming using the tool nose center

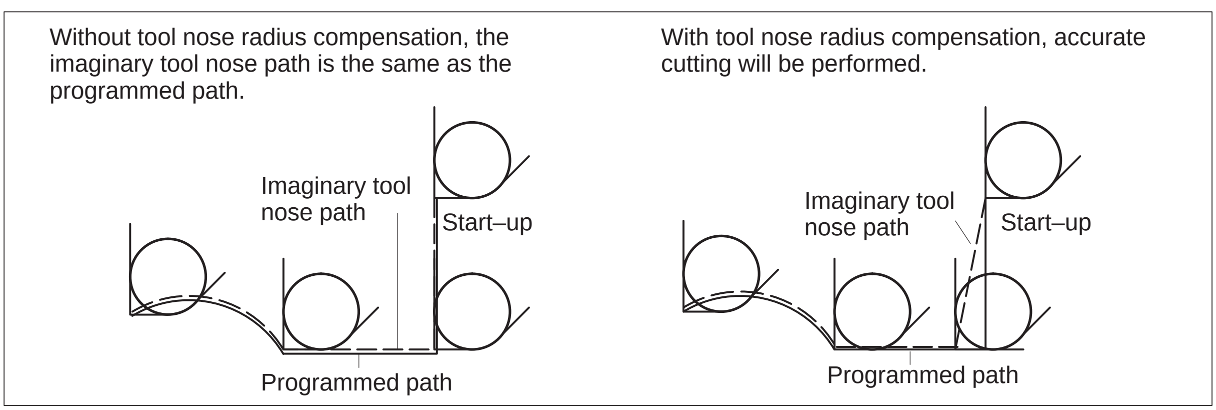


Fig15.2.1(d) Tool path when programming using the imaginary tool nose

15.2.2 Direction of Imaginary Tool Nose

The direction of the imaginary tool nose viewed from the tool nose center is determined by the direction of the tool during cutting, so it must be set in advance as well as offset values.

The direction of the imaginary tool nose can be selected from the eight specifications shown in the Fig.15.2.2 below together with their corresponding codes.

This Fig15.2.2 illustrates the relation between the tool and the start position. The following apply when the tool geometry offset and tool wear offset option are selected.

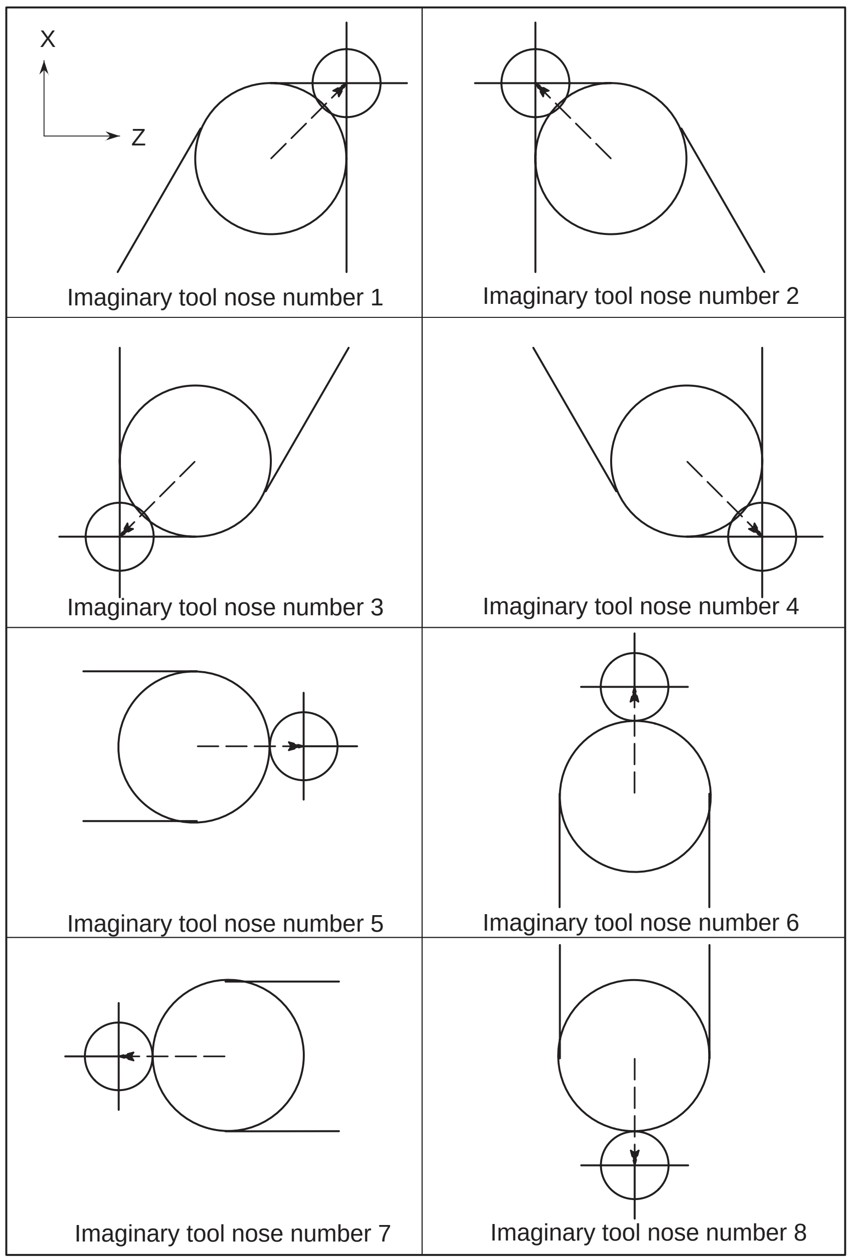
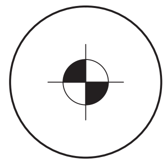


Fig.15.2.2 Direction of imaginary tool nose

Imaginary tool nose numbers 0 and 9 are used when the tool nose center coincides with the start position. Set imaginary tool nose number to address OFT for each offset number.
Bit 7 (WNP) of parameter No. 5002 is used to determine whether the tool geometry offset number or the tool wear offset number specifies the direction of the virtual tool nose for tool nose radius compensation.



Imaginary tool nose number 0 to 9

Limitations

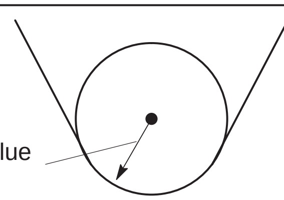
- **Plane selection**

Virtual tool nose directions 1 to 8 can be used only in the G18 (Z–X) plane. For virtual tool nose 0 or 9, compensation is applied in both the G17 and G19 planes.

15.2.3 Offset number and offset value

Explanations

- **Offset number and offset value**



This value is set from the MDI according to the offset number.
When the options of tool geometry compensation and tool wear compensation are selected, offset values become as follows :

Table 15.2.3(a) Offset number and offset value

Offset number	OFX (Offset value on X axis)	OFZ (Offset value on Z axis)	OFR (Tool nose radius compensation value)	OFT (Direction of imaginary tool nose)	OFY (Offset value on Y axis)
01	0.040	0.020	0.20	1	0.030
02	0.060	0.030	0.25	2	0.040
:	:	:	:	:	:
98	0.050	0.015	0.12	6	0.025
99	0.030	0.025	0.24	3	0.035

When the options of tool geometry compensation and tool wear compensation are selected, the offset values become as follows :

Table 15.2.3(b) Tool geometry offset

Geometry offset number	OFGX (X–axis geometry offset amount)	OFGZ (Z–axis geometry offset amount)	OFGR (Tool nose radius geometry offset value)	OFT (Imaginary tool nose direction)	OFGY (Y–axis geometry offset amount)
G01	10.040	50.020	0	1	70.020
G02	20.060	30.030	0	2	90.030
G03	0	0	0.20	6	0
G04	:	:	:	:	:
G05	:	:	:	:	:
:	:	:	:	:	:

Table 15.2.3(c) Tool wear offset

Wear offset number	OFGX (X–axis wear offset amount)	OFGZ (Z–axis wear offset amount)	OFGR (Tool nose radius wear offset value)	OFT (Imaginary tool nose direction)	OFGY (Y–axis wear offset amount)
W01	0.040	0.020	0	1	0.010
W02	0.060	0.030	0	2	0.020
W03	0	0	0.20	6	0
W04	:	:	:	:	:
W05	:	:	:	:	:
:	:	:	:	:	:

- **Tool nose radius compensation**

In this case, the tool nose radius compensation value is the sum of the geometry or the wear offset value.

OFR=OFGR+OFWR

- **Imaginary tool nose direction**

The imaginary tool nose direction may be set for either the geometry offset or the wear offset.
However, the last designated direction later is effective.

- **Command of offset value**

A offset number is specified with the same T code as that used for tool offset. For details, see Subsec. II–15.1.2.

Notes

When the geometry offset number is made common to the tool selection by the parameter LGT(No.5002#1)**setting and a T code for which the geometry offset and wear offset number differ from each other is designated, the imaginary tool nose direction specified by the geometry offset number is valid.**

Example) **T0102**
$OFR=RFGR_{01}+OFWR_{02}$
$OFT=OFT_{01}$

- **Setting range of offset value**

The range of the offset value is an follows :

Increment system	metric system	Inch system
IS–B	0 to ± 999.999 mm	0 to ± 99.9999 inch
IS–C	0 to ± 999.9999 mm	0 to ± 99.99999 inch

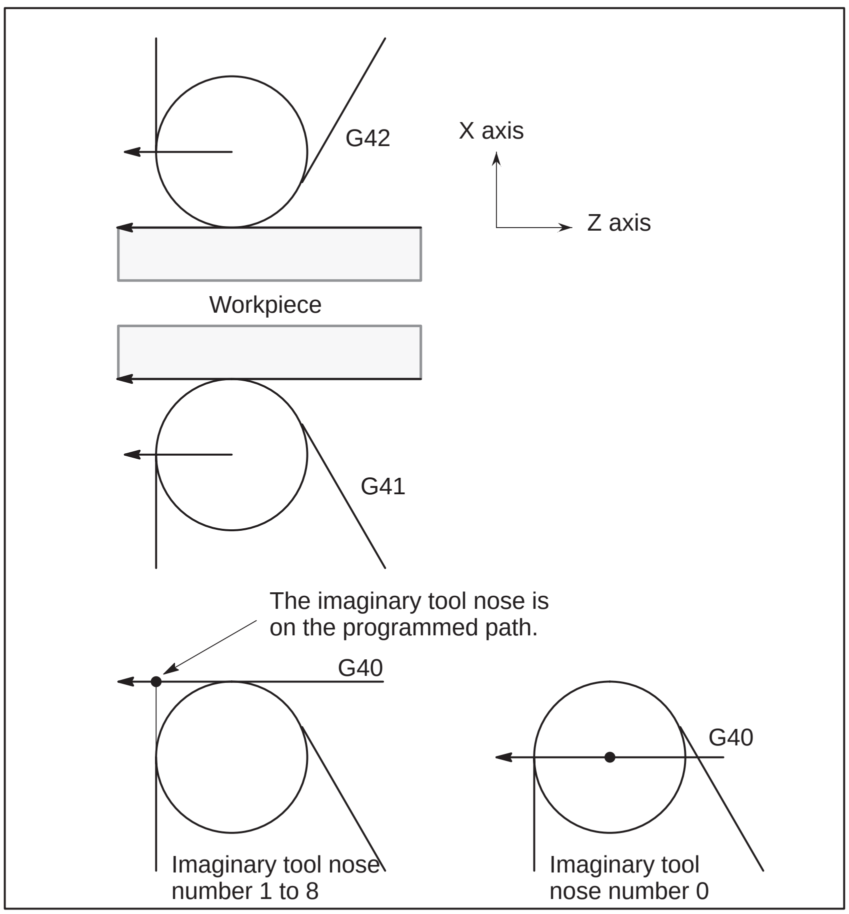
The offset value corresponding to the offset number 0 is always 0.
No offset value can be set to offset number 0.

15.2.4 Work Position and Move Command

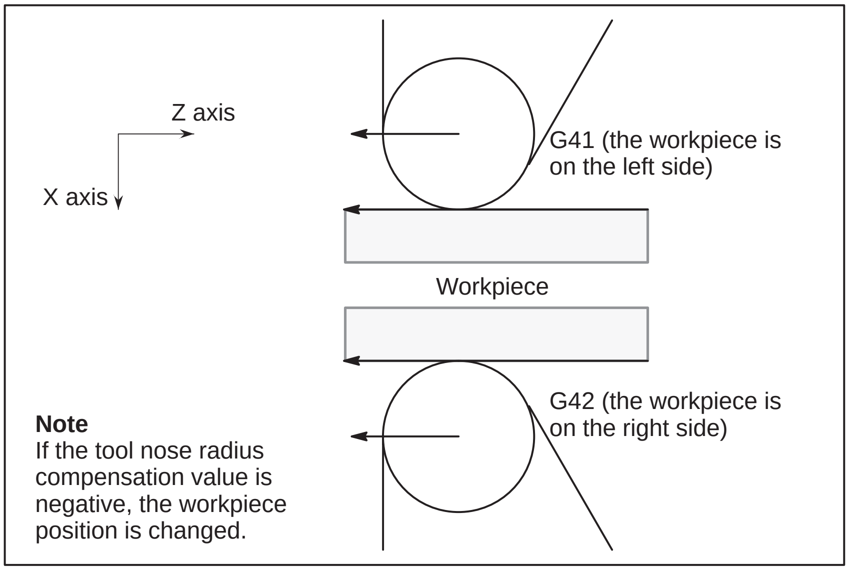
In tool nose radius compensation, the position of the workpiece with respect to the tool must be specified.

G code	Workpiece position	Tool path
G40	(Cancel)	Moving along the programmed path
G41	Right side	Moving on the left side the programmed path
G42	Left side	Moving on the right side the programmed path

The tool is offset to the opposite side of the workpiece.



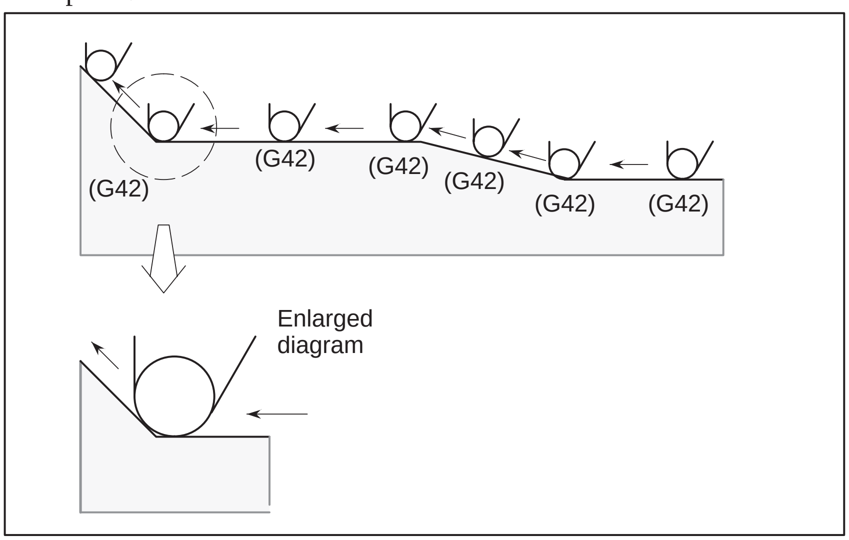
The workpiece position can be changed by setting the coordinate system as shown below.



G40, G41, and, G42 are modal.
Don't specify G41 while in the G41 mode. If you do, compensation will not work properly.
Don't specify G42 while in the G42 mode for the same reason.
G41 or G42 mode blocks in which G41 or G42 are not specified are expressed by (G41) or (G42) respectively.

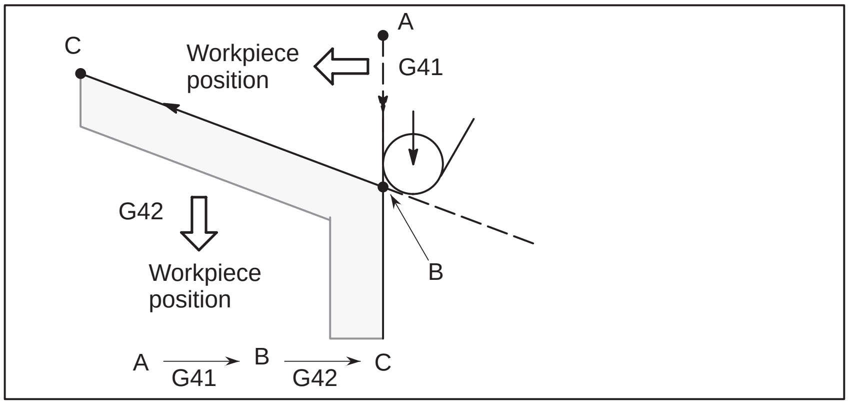
- **Tool movement when the workpiece position does not change**

When the tool is moving, the tool nose maintains contact with the workpiece.



- **Tool movement when the workpiece position changes**

The workpiece position against the toll changes at the corner of the programmed path as shown in the following figure.



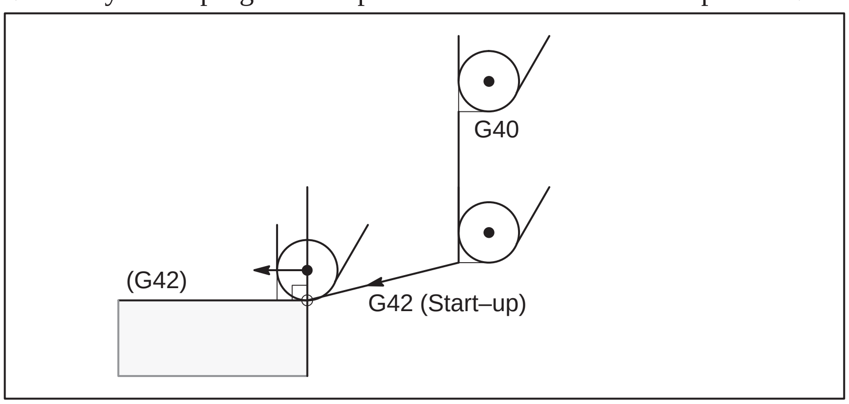
Although the workpiece does not exist on the right side of the programmed path in the above case. the existence of the workpiece is assumed in the movement from A to B. The workpiece position must not be changed in the block next to the start–up block. In the above example, if the block specifying motion from A to B were the start–up block, the tool path would not be the same as the one shown.

- **Start–up**

The block in which the mode changes to G41 or G42 from G40 is called the start–up block.

G40 _ ;
G41 _ ; (Start–up block)

Transient tool movements for offset are performed in the start–up block. In the block after the start–up block, the tool nose center is positioned Vertically to the programmed path of that block at the start position.

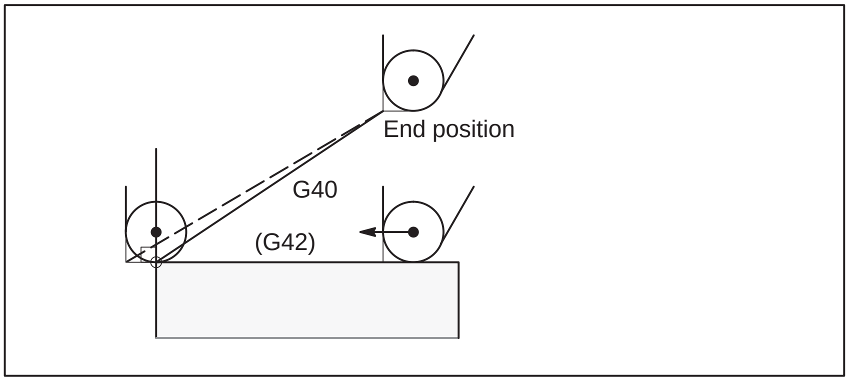


- **Offset cancel**

The block in which the mode changes to G40 from G41 or G42 is called the offset cancel block.

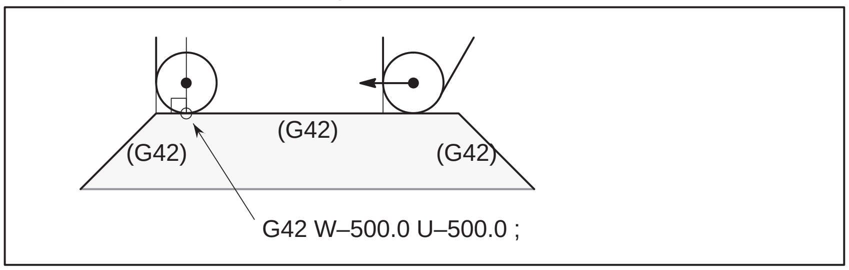
G41 _ ;
G40 _ ; (Offset cancel block)

The tool nose center moves to a position vertical to the programmed path in the block before the cancel block. The tool is positioned at the end position in the offset cancel block (G40) as shown below.



- **Specification of G41/G42 in G41/G42 mode**

When is specified again in G41/G42 mode , the tool nose center is positioned vertical to the programmed path of the preceding block at the end position of the preceding block.

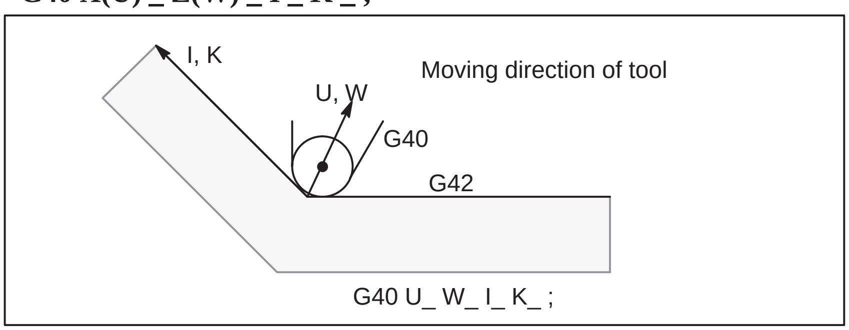


In the block that first specifies G41/G42, the above positioning of the tool nose center is not performed.

- **Tool movement when the moving direction of the tool in a block which includes a G40 command is different from the direction of the workpiece**

When you wish to retract the tool in the direction specified by X(U) and Z(W) cancelling the tool nose radius compensation at the end of machining the first block in the figure below, specify the following :

G40 X(U) _ Z(W) _ I _ K _ ;

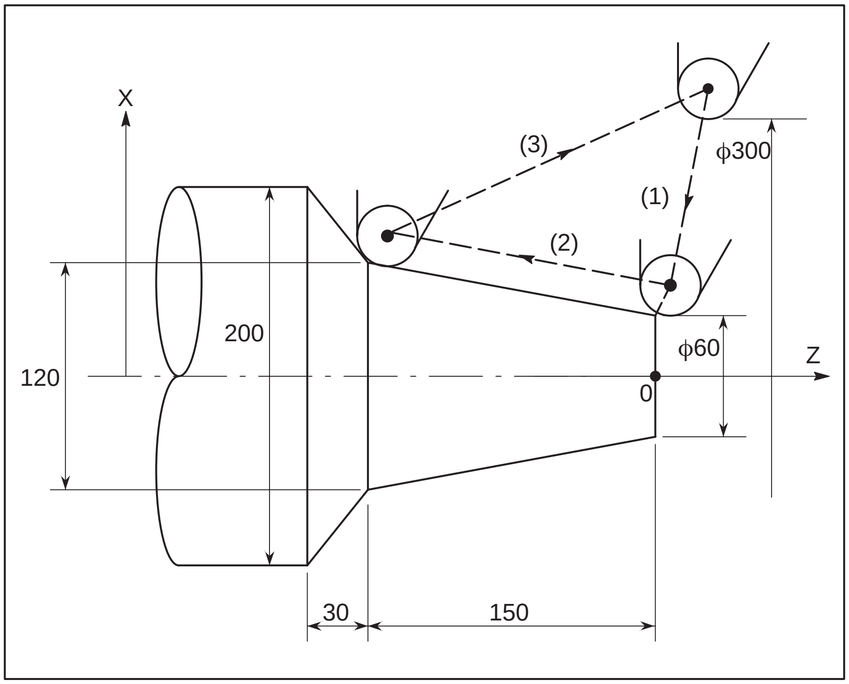


The workpiece position specified by addresses I and K is the same as that in the preceding block. If I and/or K is specified with G40 in the cancel mode, the I and/or K is ignored.

G40 X_ Z_ I_ K_ ;	Tool nose radius compensation
G40 G02 X_ Z_ I_ K_ ;	Circular interpolation

G40 G01 X_ Z_ ;
G40 G01 X_ Z_ I_ K_ ; Offset cancel mode (I and k are ineffective.)
The numeral s followed I and K should always be specified as radius values.

Examples



(G40 mode)
1.G42 G00 X60.0 ;
2.G01 X120.0 W–150.0 F10 ;
3.G40 G00 X300.0 W150.0 I40.0 K–30.0 ;

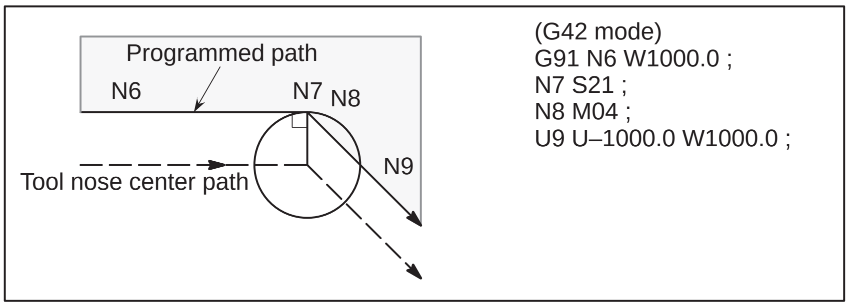
15.2.5 Notes on tool nose radius compensation

Explanations

● Tool movement when two or more blocks without a move command should not be programmed consecutively

1.M05 ;	M code output
2.S210 ;	S code output
3.G04 X1000 ;	Dwell
4.G01 U0 ;	Feed distance of zero
5.G98 ;	G code only
6.G10 P01 X10.0 Z20.0 R0.5 Q2 ;	Offset change

If two or more of the above blocks are specified consecutively, the tool nose center comes to a position vertical to the programmed path of the preceding block at the end of the preceding block. However, if the no movement commands is **4** above, the above tool motion is attained only with one block.

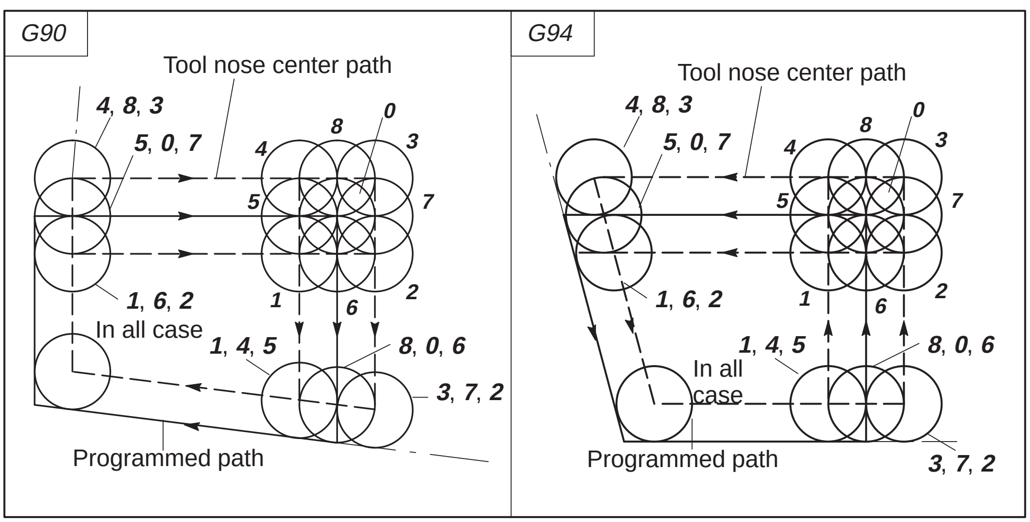


● Tool nose radius compensation with G90 or G94

Tool nose radius compensation with G90 (outer diameter/internal diameter cutting cycle) or G94 (end face turning cycle) is as follows, :

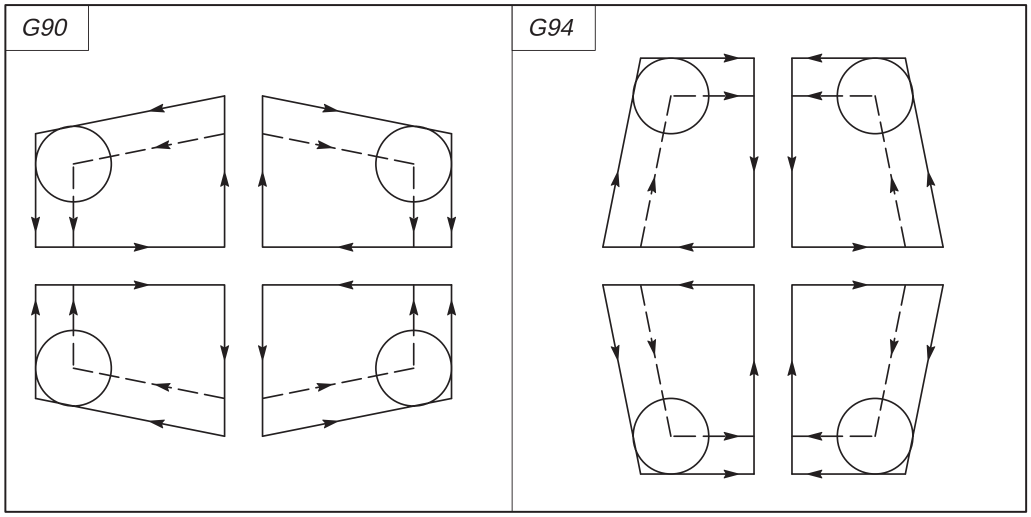
1.Motion for imaginary tool nose numbers

For each path in the cycle, the tool nose center path is generally parallel to the programmed path.



2.Direction of the offset

The offset direction is indicated in the figure below regardless of the G41/G42 mode.



- **Tool nose radius compensation with G71 to G76 or G78**

When one of following cycles is specified, the cycle deviates by a tool nose radius compensation vector. During the cycle, no intersection calculation is performed.

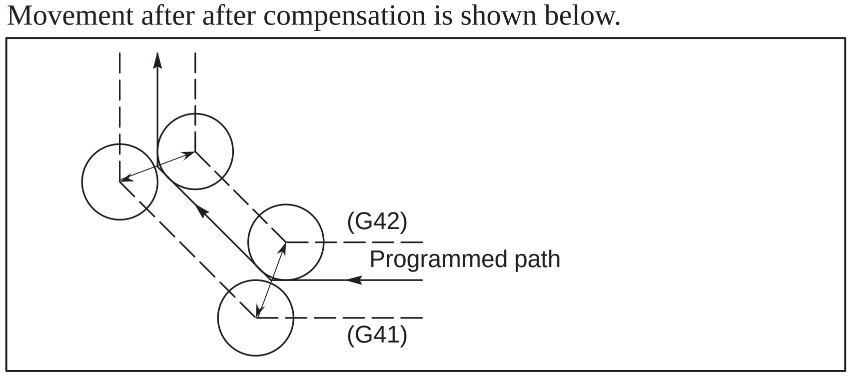
G71 (Stock removal in turning or traverse grinding cycle)
G72 (Stock removal in facing or traverse direct constant–dimension grinding cycle)
G73 (Pattern repeating or Oscillation grinding cycle)

When one of following cycles is specified, the tool nose radius compensation is not performed.

G74 (End face peck drilling)
G75 (Outer diameter/internal diameter drilling)
G76 (Multiple threading cycle)
G78 (Threading cycle)

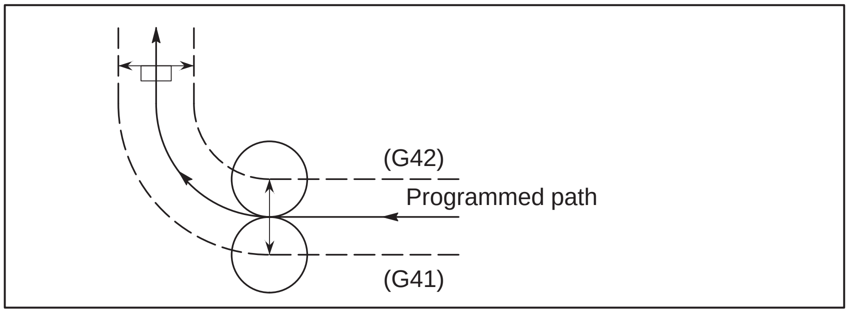
- **Tool nose radius compensation when chamfering is performed**

Movement after after compensation is shown below.



- **Tool nose radius compensation when a corner arc is inserted**

Movement after compensation is shown below.



- **Tool nose radius compensation when the block is specified from the MDI**

In this case, tool nose radius compensation is not performed.

15.3 DETAILS OF TOOL NOSE RADIUS COMPENSATION

This section provides a detailed explanation of the movement of the tool for tool nose radius compensation outlined in Section 15.2.
This section consists of the following subsections:

15.3.1 General

- **Tool nose radius center offset vector**

The tool nose radius center offset vector is a two dimensional vector equal to the offset value specified in a T code, and the is calculated in the CNC. Its dimension changes block by block according to tool movement.
This offset vector (simply called vector herein after) is internally crated by the control unit as required for proper offsetting and to calculate a tool path with exact offset (by tool nose radius) from the programmed path.
This vector is deleted by resetting.
The vector always accompanies the tool as the tool advances.
Proper understanding of vector is essential to accurate programming.
Read the description below on how vectors are created carefully.

- **G40, G41, G42**

G40, G41 or G42 is used to delete or generate vectors.
These codes are used together with G00, G01, G02, G03 or G33 to specify a mode for tool motion (Offsetting).

G code	Function	Workpiece position
G40	Tool nose radius compensation cancel	Neither
G41	Left offset along tool path	Right
G42	Right offset along tool path	Left

G41 and G42 specify an off mode, while G40 specifies cancellation of the offset.

- **Cancel mode**

The system enters the cancel mode immediately after the power is turned on, when the RESET button on the MDI is pushed or a program is forced to end by executing M02 or M30. (the system may not enter the cancel mode depending on the machine tool.) In the cancel mode, the vector is set to zero, and the path of the center of tool nose coincides with the programmed, path. A program must end in cancel mode. If it ends in the offset mode, the tool cannot be positioned at the end point, and the tool stops at a location the vector length away from the end point.

- **Start–up**

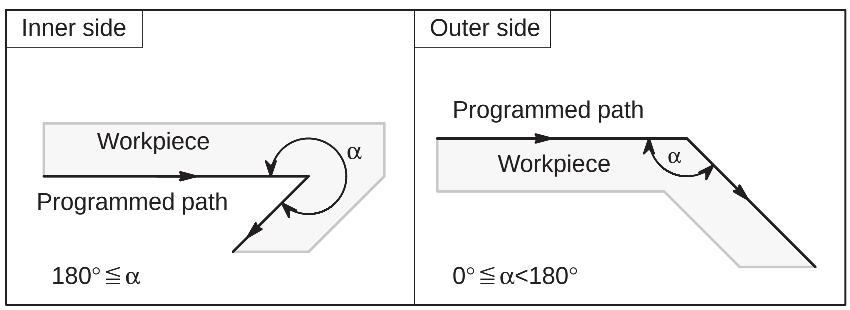
When a block which satisfies all the following conditions is executed in cancel mode, the system enters the offset mode. Control during this operation is called start–up.

- G41 or G42 is contained in the block, or has been specified to set the system enters the offset mode. Control during this operation is called start–up.
- The offset number for tool nose radius compensation is not 00.
- X or Z moves is specified in the block and the move distance is not zero.

A circular command (G02 or G03) is not allowed in start–up.
If specified, P/S alarm (PS34) will occur. Two blocks are read in during start– up. The first block is executed, and the second block is entered into the tool nose radius compensation buffer. In the single block mode, two blocks are read and the first block is executed, then the machine stops.
In subsequent operations, two blocks are read in advance, so the CNC has the block currently being executed, and the next two blocks.

- **Inner side and outer side**

When an angle of intersection created by tool paths specified with move commands for two blocks is over 180°, it is referred to as “inner side.” When the angle is between 0° and 180°, it is referred to as “outer side.”



- **Meaning of symbols**

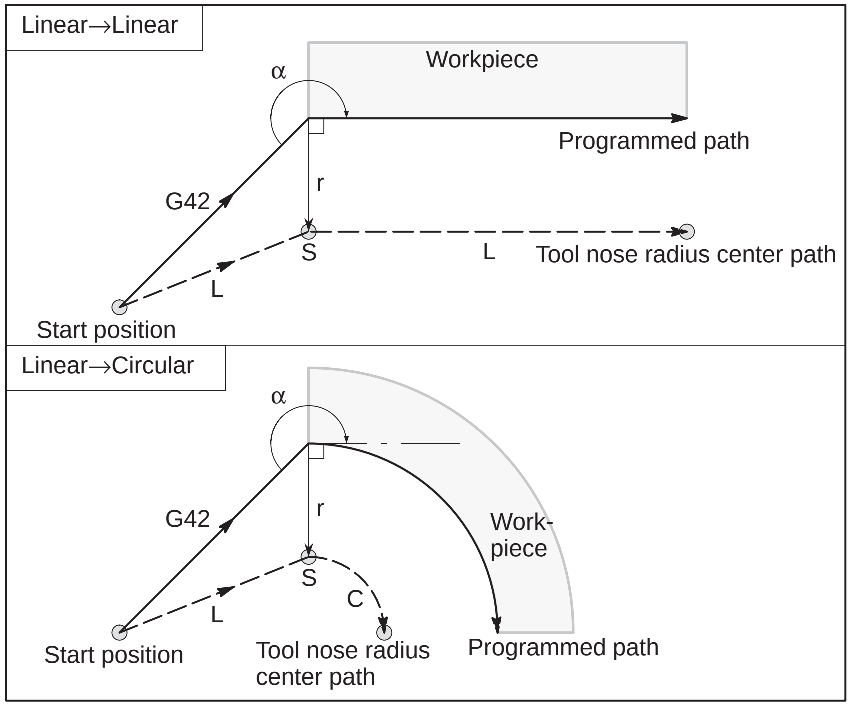
The following symbols are used in subsequent figures:
– ***S*** indicates a position at which a single block is executed once.
– ***SS*** indicates a position at which a single block is executed twice.
– ***SSS*** indicates a position at which a single block is executed three times.
– ***L*** indicates that the tool moves along a straight line.
– ***C*** indicates that the tool moves along an arc.
– ***r*** indicates the tool nose radius compensation value.
– An intersection is a position at which the programmed paths of two blocks intersect with each other after they are shifted by ***r***.
– indicates ○ the center of the tool nose radius.

15.3.2 Tool Movement in Start–up

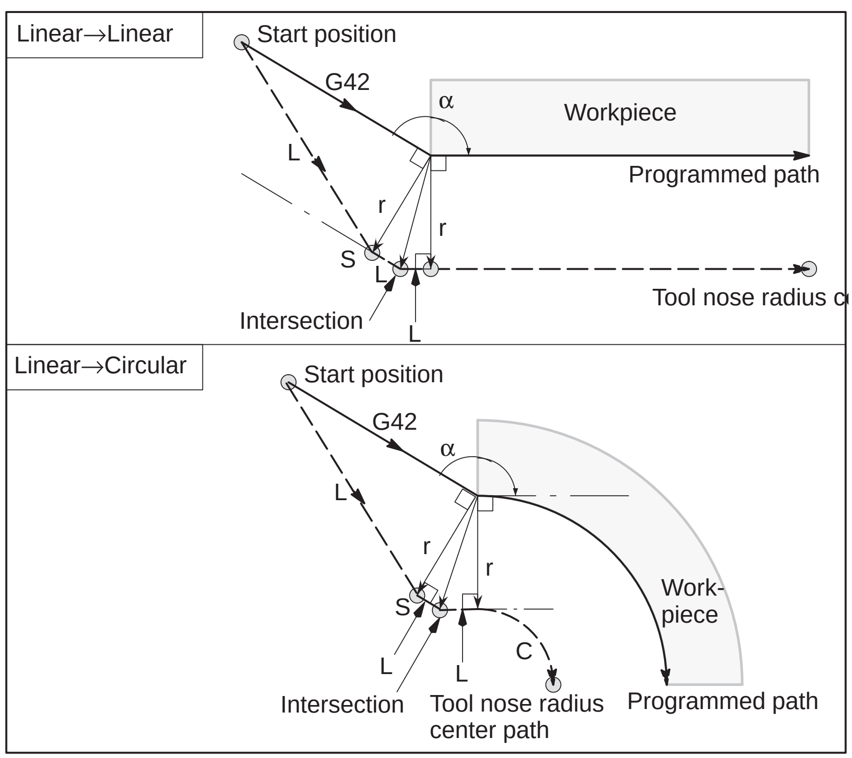
When the offset cancel mode is changed to offset mode, the tool moves as illustrated below (start–up):

Explanations

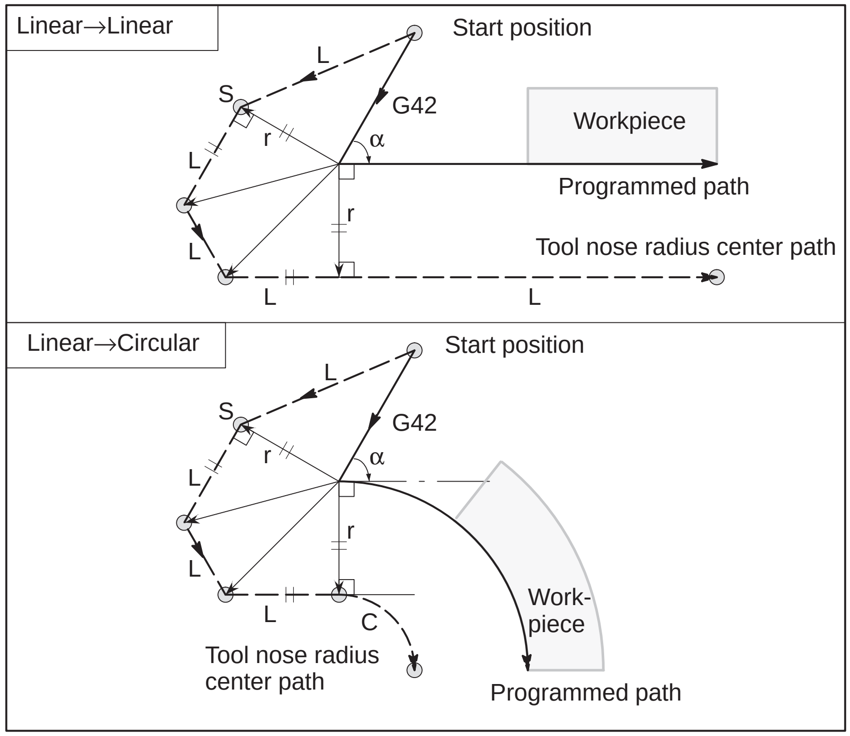
- **Tool movement around an inner side of a corner (180°≦α)**



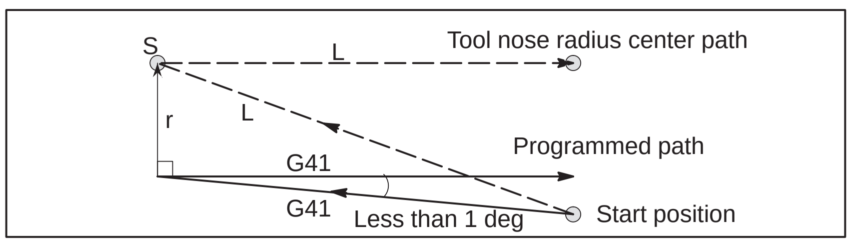
- **Tool movement around the outside of a corner at an obtuse angle (90°≦α<180°)**



- **Tool movement around the outside of an acute angle (α<90°)**

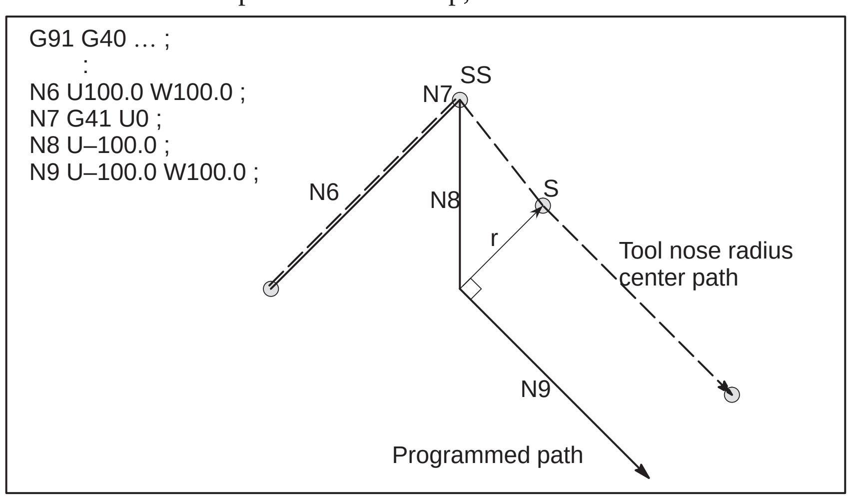


- **Tool movement around the outside linear→linear at an acute angle less than 1 degree (α<1°)**



- **A block without tool movement specified at start–up**

If the command is specified at start–up, the offset vector is not created.



Notes

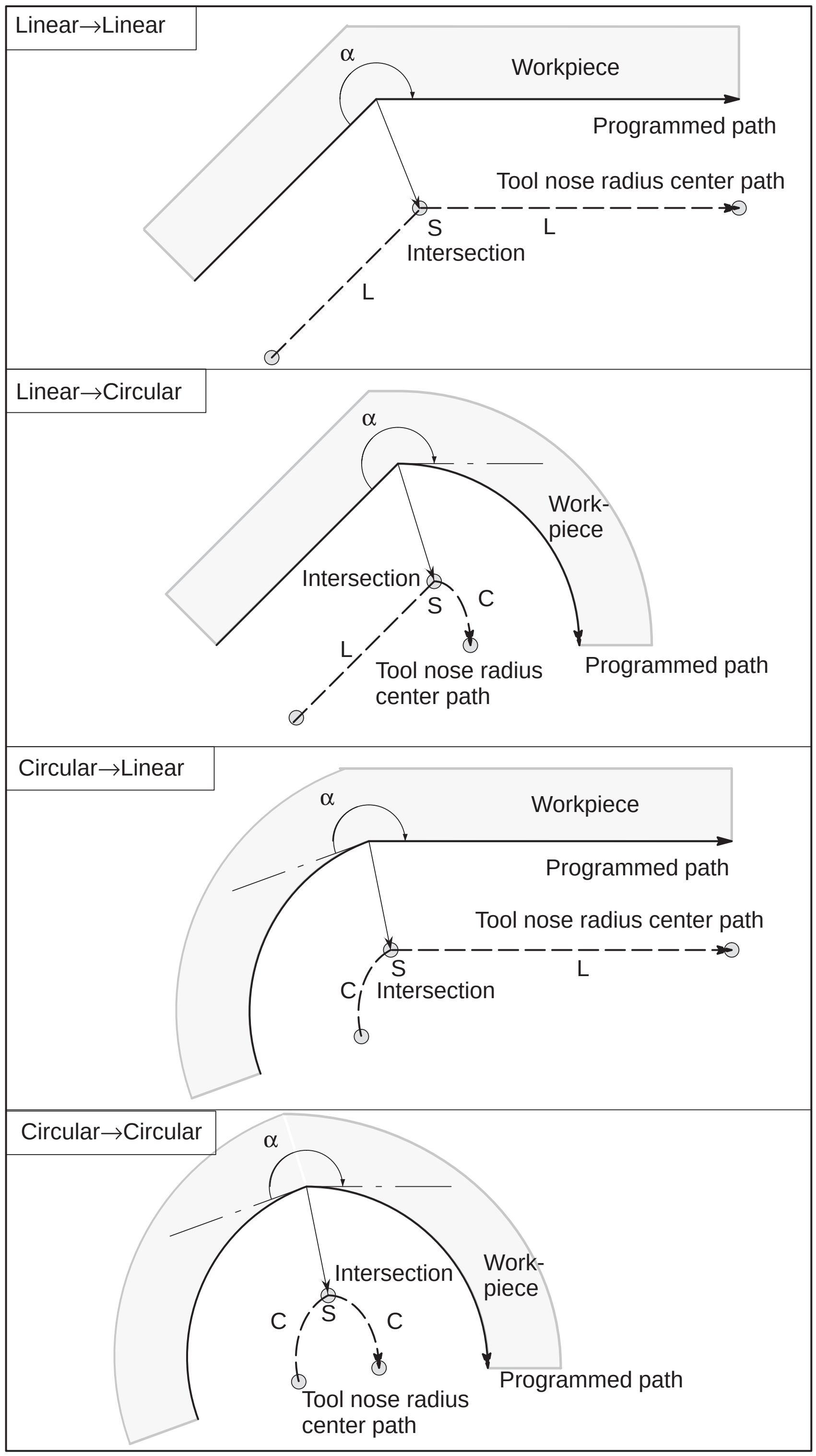
For the definition of blocks that do not move the tool, see Subsec. II–15.3.3.

15.3.3 Tool Movement in Offset Mode

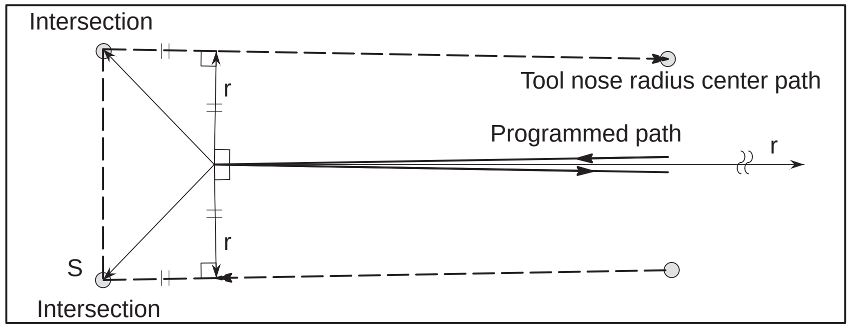
In the offset mode, the tool moves as illustrated below:

Explanations

- **Tool movement around the inside of a corner (180°≦α)**

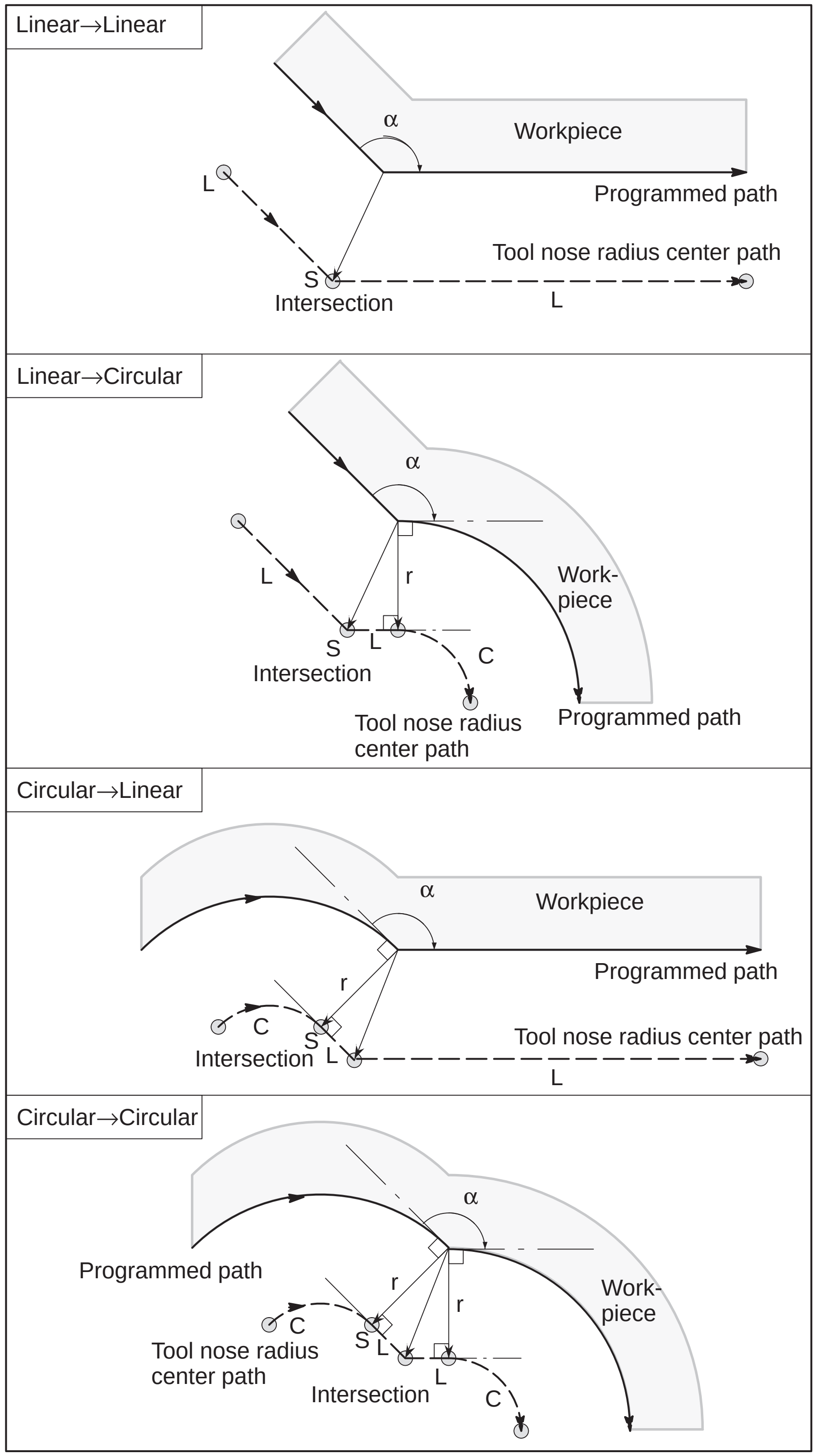


- **Tool movement around the inside (α<1°) with an abnormally long vector, linear → linear**

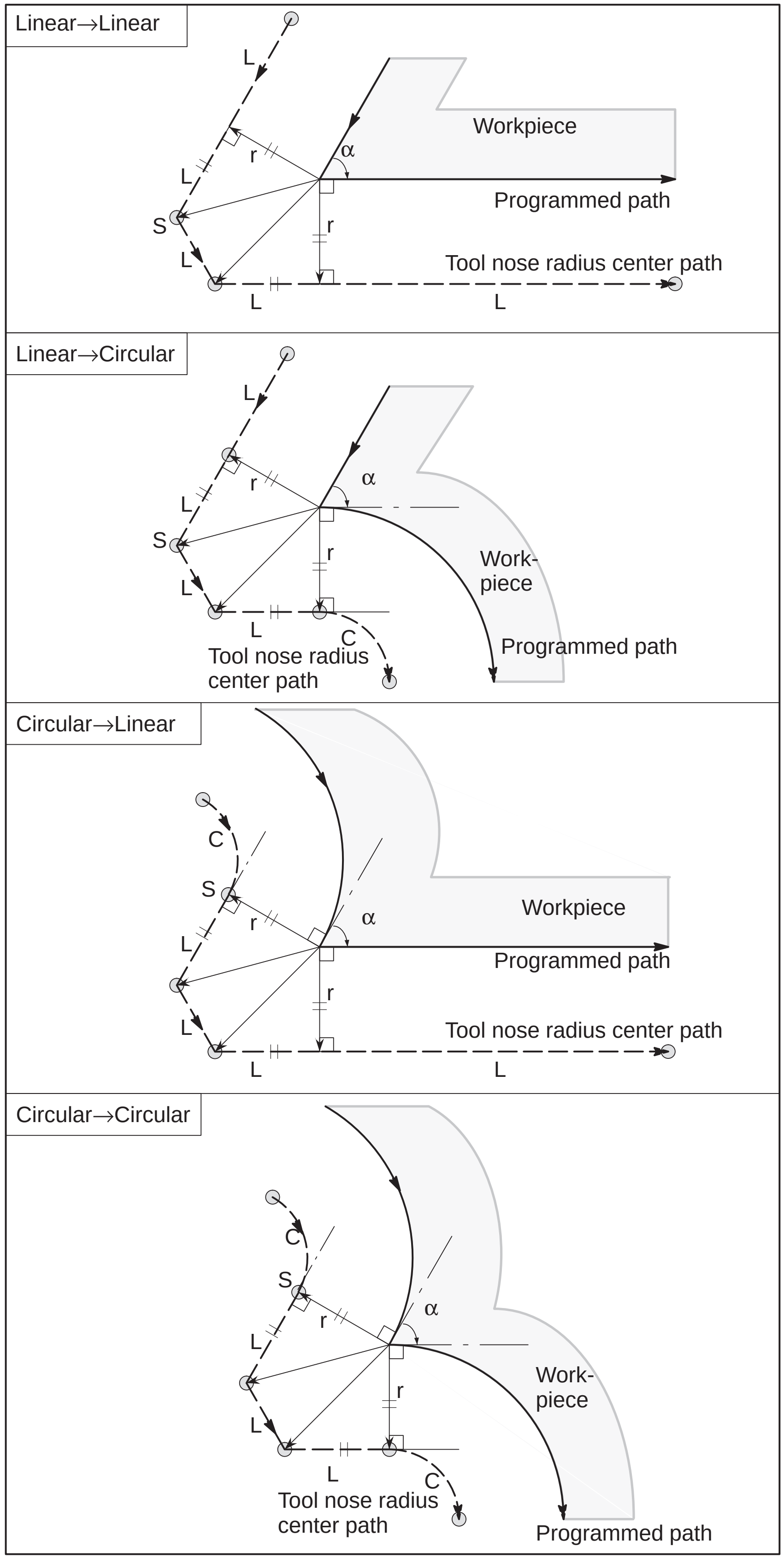


Also in case of arc to straight line, straight line to arc and arc to arc, the reader should infer in the same procedure.

- **Tool movement around the outside corner at an obtuse angle (90°≦α<180°)**



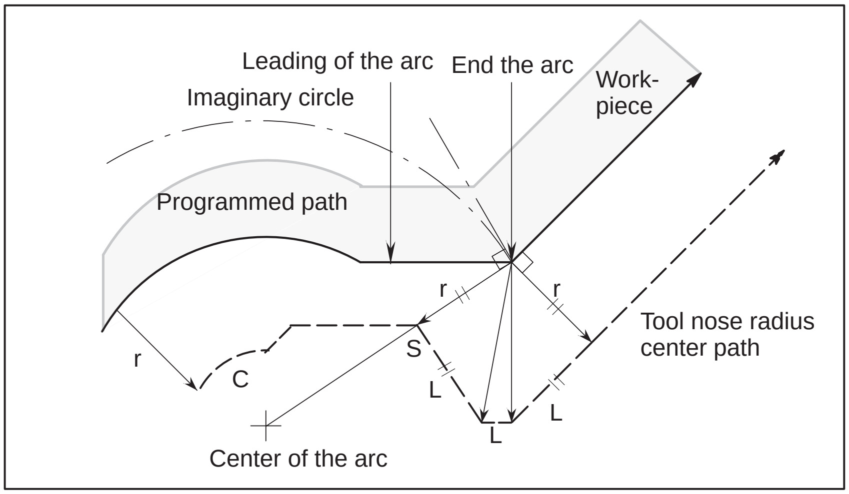
- **Tool movement around the outside corner at an acute angle (α<90°)**



When it is exceptional

End position for the arc is not on the arc

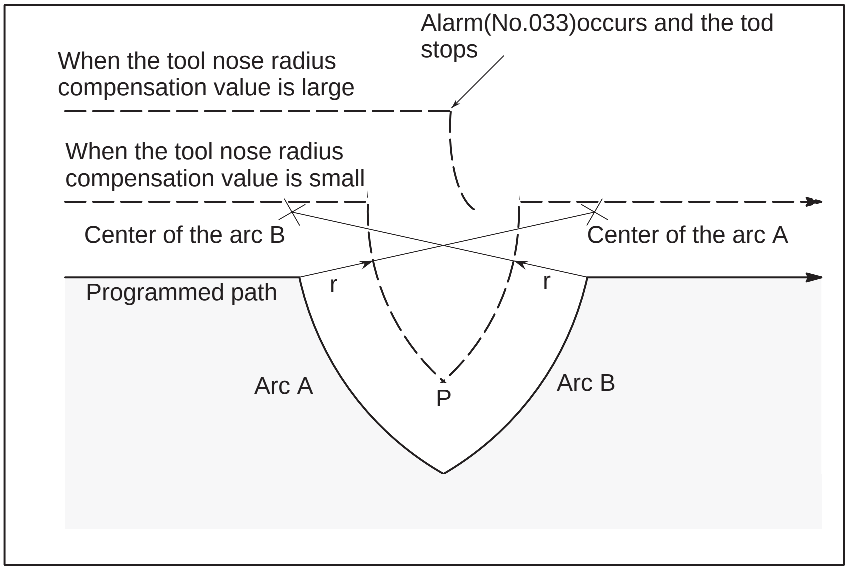
If the end of a line leading to an arc is programmed as the end of the arc by mistake as illustrated below, the system assumes that tool nose radius compensation has been executed with respect to an imaginary circle that has the same center as the arc and passes the specified end position. Based on this assumption, the system creates a vector and carries out compensation. The resulting tool nose radius center path is different from that created by applying tool nose radius compensation to the programmed path in which the line leading to the arc is considered straight.



The same description applies to tool movement between two circular paths.

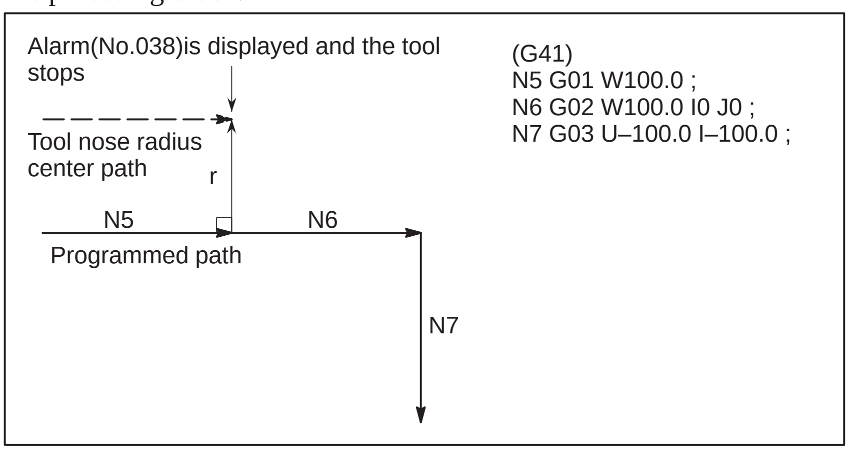
- There is no inner intersection

If the tool nose radius compensation value is sufficiently small, the two circular Tool nose radius center paths made after compensation intersect at a position (P). Intersection P may not occur if an excessively large value is specified for tool nose radius compensation. When this is predicted, P/S alarm (No.33) occurs at the end of the previous block and the tool is stopped. In the example shown below, Tool nose radius center paths along arcs A and B intersect at P when a sufficiently small value is specified for tool nose radius compensation. If an excessively large value is specified, this intersection does not occur.



- The center of the arc is identical with the start position or the end position

If the center of the arc is identical with the start position or end point, P/S alarm (No. 038) is displayed, and the tool will stop at the end position of the preceding block.



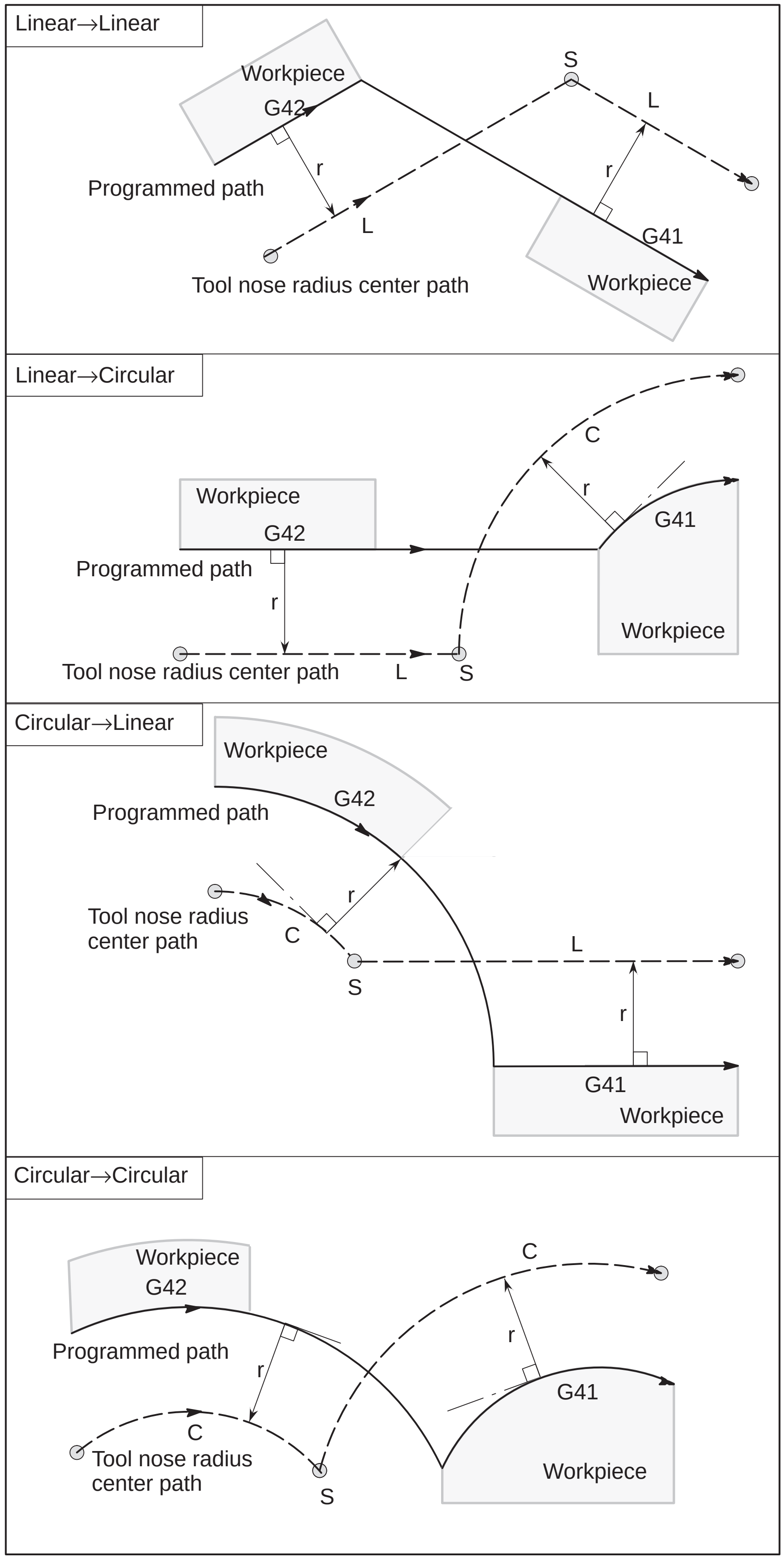
- **Change in the offset direction in the offset mode**

The offset direction is decided by G codes (G41 and G42) for tool nose radius and the sign of tool nose radius compensation value as follows.

G code \ Sign of offset value	+	–
G41	Left side offset	Right side offset
G42	Right side offset	Left side offset

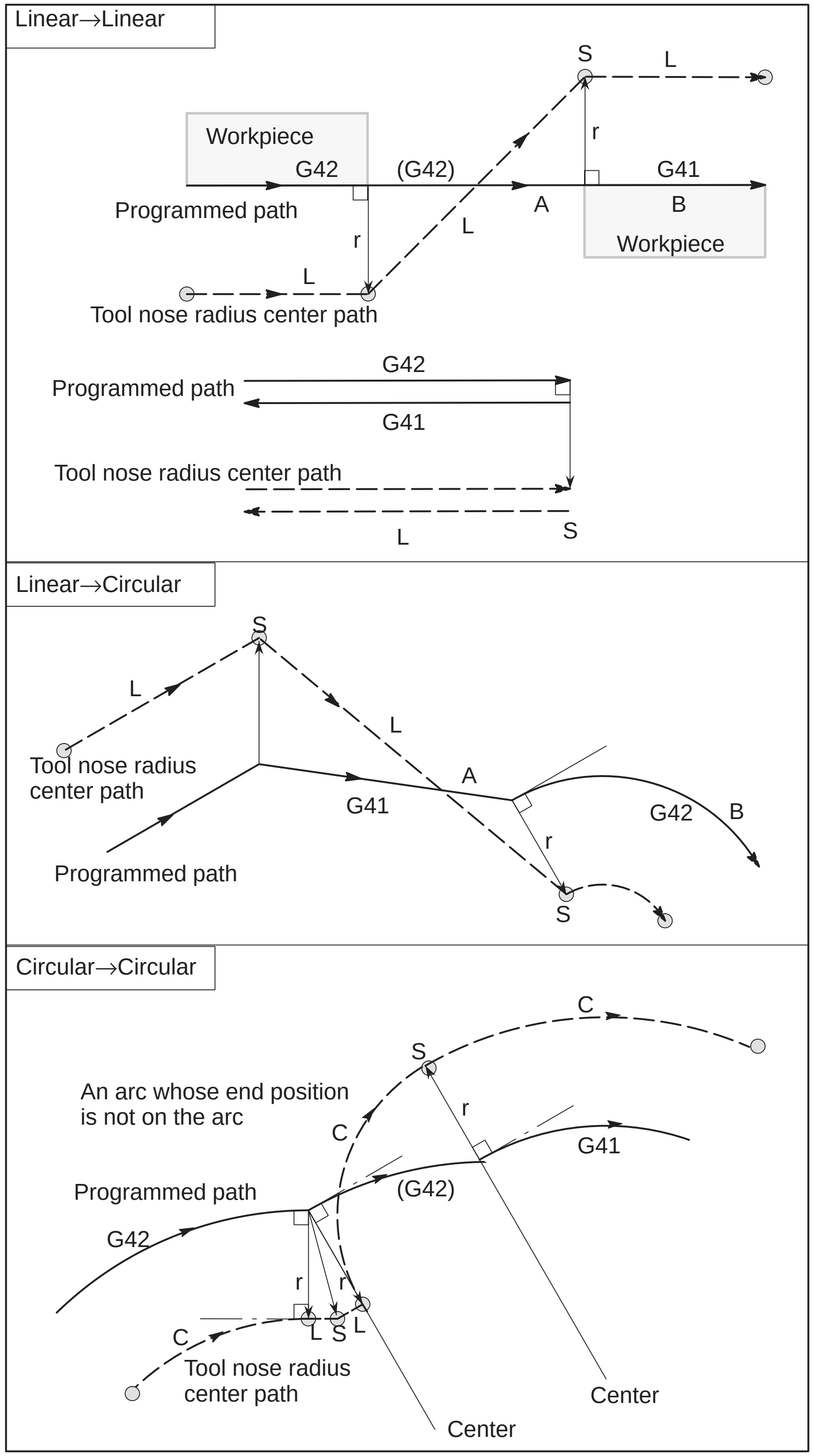
The offset direction can be changed in the offset mode. If the offset direction is changed in a block, a vector is generated at the intersection of the tool nose radius center path of that block and the tool nose radius center path of a preceding block. However, the change is not available in the start–up block and the block following it.

- Tool nose radius center path with an intersection



- Tool nose radius center path without an intersection

When changing the offset direction in block A to block B using G41 and G42, if intersection with the offset path is not required, the vector normal to block B is created at the start point of block B.

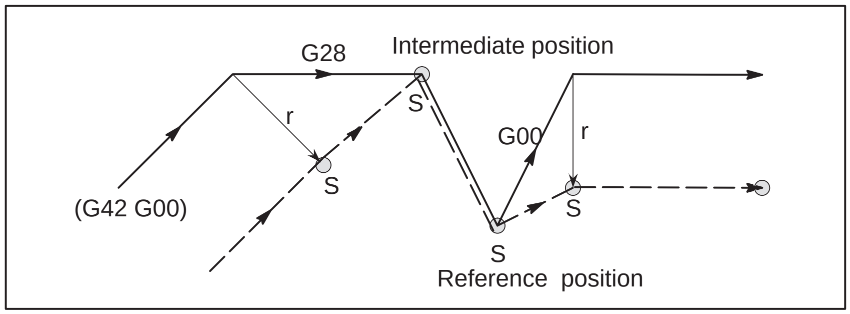


- **Temporary tool nose radius compensation cancel**

If the following command is specified in the offset mode, the offset mode is temporarily canceled then automatically restored. The offset mode can be canceled and started as described in Subsections II–15.3.2 and II–15.3.4.

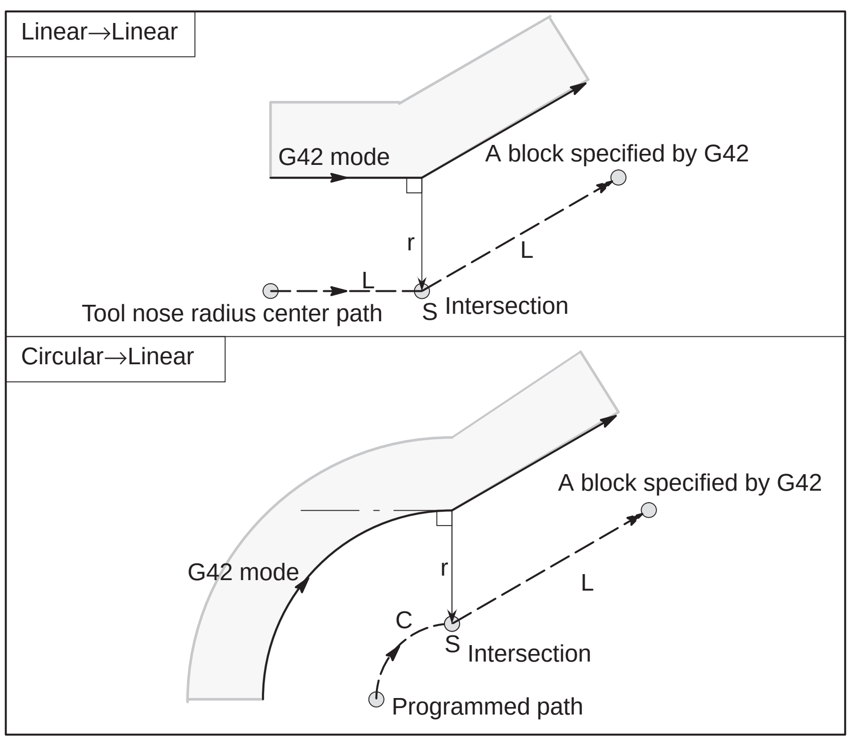
- Specifying G28 (automatic return to the reference position) in the offset mode

If G28 is specified in the offset mode, the offset mode is canceled at an intermediate position. If the vector still remains after the tool is returned to the reference position, the components of the vector are reset to zero with respect to each axis along which reference position return has been made.



- **Tool nose radius compensation G code in the offset mode**

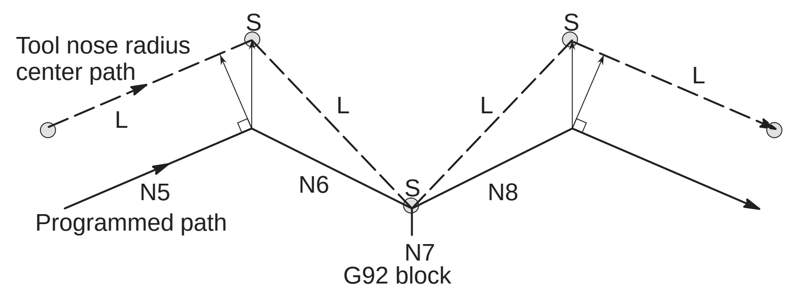
The offset vector can be set to form a right angle to the moving direction in the previous block, irrespective of machining inner or outer side, by commanding the tool nose radius compensation G code (G41, G42) in the offset mode, independently. If this code is specified in a circular command, correct circular motion will not be obtained.
When the direction of offset is expected to be changed by the command of tool nose radius compensation G code (G41, G42), refer to ”Change in the offset direction in the offset mode” in Subsec.15.3.3.



- **Command cancelling the offset vector temporality**

During offset mode, if G50 is commanded,the offset vector is temporarily cancelled and thereafter offset mode is automatically restored.
In this case, without movement of offset cancel, the tool moves directly from the intersecting point to the commanded point where offset vector is canceled. Also when restored to offset mode, the tool moves directly to the intersecting point.

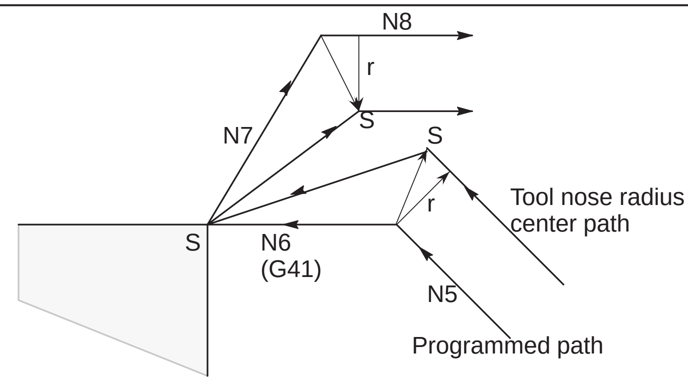
- **Workpiece coordinate system setting (G50)**



(G41)
N5 G91 G01 U700.0 W300.0 ;
N6 U600.0 W–300.0 ;
N7 G50 X200.0 Z100.0 ;
N8 G01 X800.0 Z400.0 ;

- **Canned cycles (G90, G92, G94) and Multiple repetitive cycles (G71 to G76)**

See Sections II–15.1 (G90, G92, G94) and II–15.2 (G70 to G76) for the tool nose radius compensation is related canned cycles.



(G42)
N5 G01 U500.0 W600.0 ;
N6 W–800.0 ;
N7 G90 U–600.0 Z–800.0 I–300.0 ;
N8 U1200.0 W500.0 ;

- **A block without tool movement**

The following blocks have no tool movement. In these blocks, the tool will not move even if tool nose radius compensation is effected.

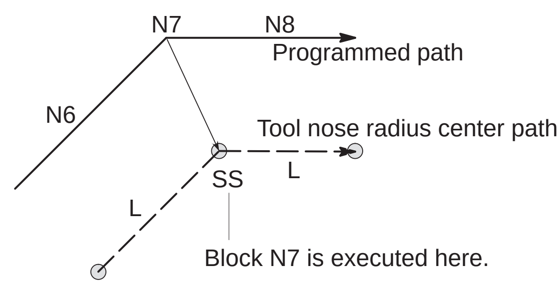
1. M05 ; M code output
2. S21 ; S code output
3. G04 X10.0 ; Dwell
4. G10 P01 X10 Z20 R10.0 ; tool nose radius compen– sation value setting
5. (G17) Z200.0 ; Move command not included in the offset plane.
6. G98 ; G code only
7. X0 ; Move distance is zero.

Commands **1** to **6** are of no movement.

- A block without tool movement specified in offset mode

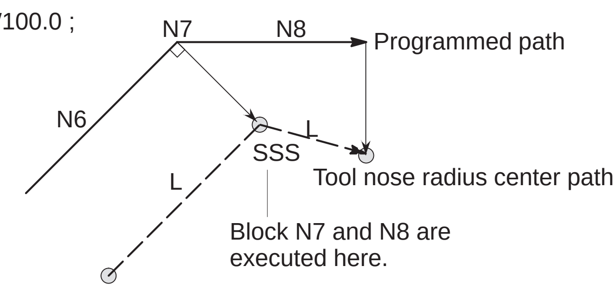
When a single block without tool movement is commanded in the offset mode, the vector and Tool nose radius center path are the same as those when the block is not commanded. This block is executed at the single block stop point.

N6 U100.0 W100.0 ;
N7 G04 Z100.0 ;
N8 U100.0 ;



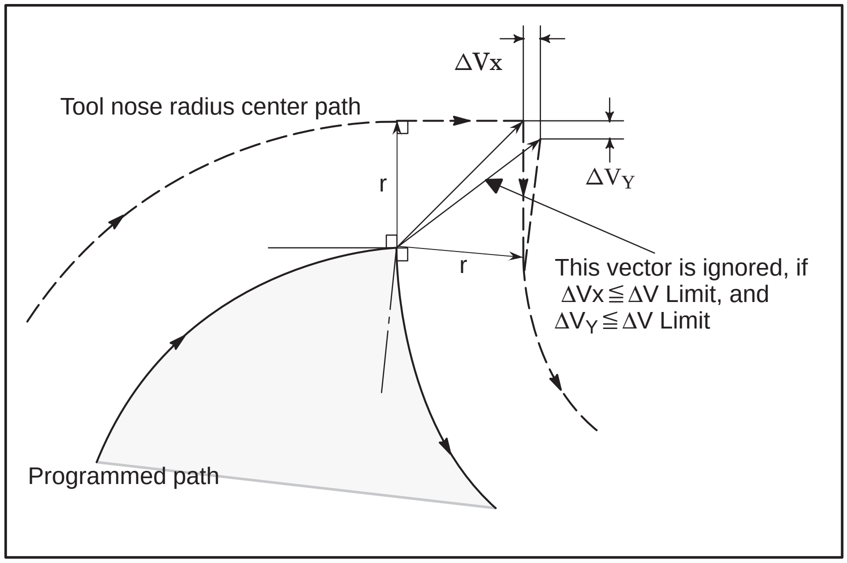
However, when the move distance is zero, even if the block is commanded singly, tool motion becomes the same as that when more than one block of without tool movement are commanded, which will be described subsequently.

N6 G91 U100.0 W100.0 ;
N7 S21 ;
N8 G04 X10.0 ;
N9 W100.0 ;

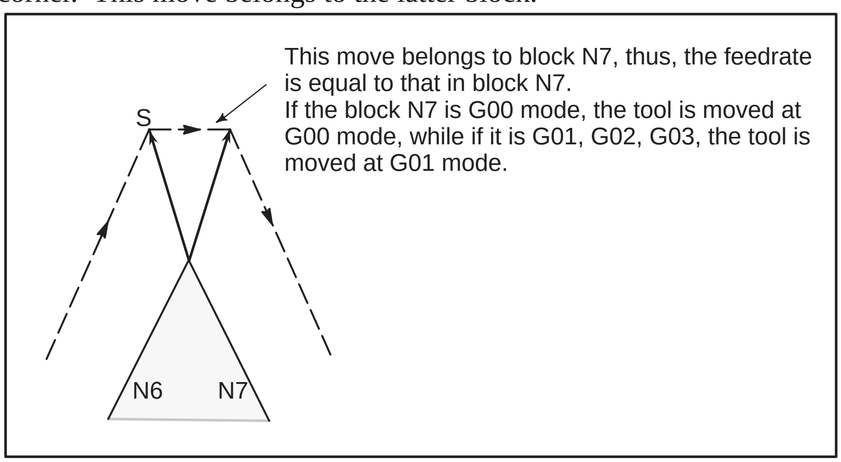


- **Corner movement**

When two or more vectors are produced at the end of a block, the tool moves linearly from one vector to another. This movement is called the corner movement.
If these vectors almost coincide with each other, the corner movement isn't performed and the latter vector is ignored.



If $\Delta Vx \leqq \Delta Vlimit$ and $\Delta Vy \leqq \Delta Vlimit$, the latter vector is ignored. The ΔVlimit is set in advance by parameter (No. 5010).
If these vectors do not coincide, a move is generated to turn around the corner. This move belongs to the latter block.



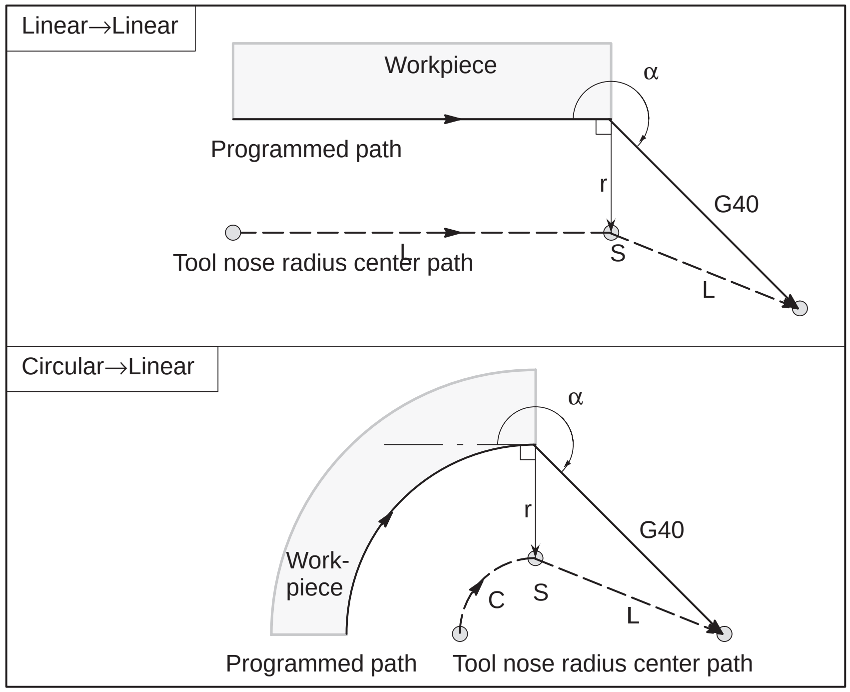
- **Interruption of manual operation**

For manual operation during the tool nose radius compensation, refer to Section III–3.5, " Manual Absolute ON and OFF."

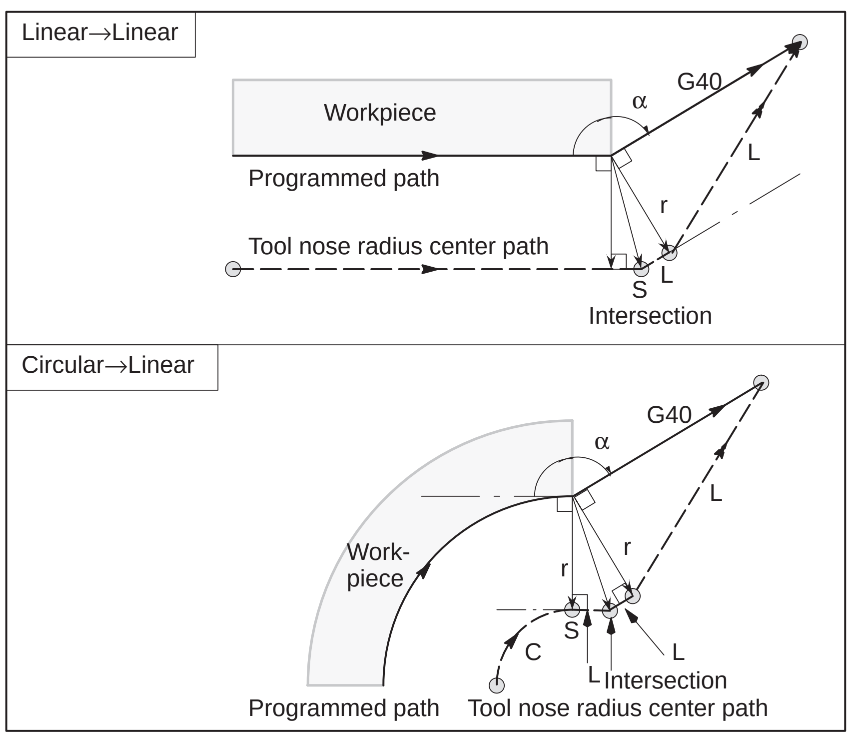
15.3.4 Tool Movement in Offset Mode Cancel

Explanations

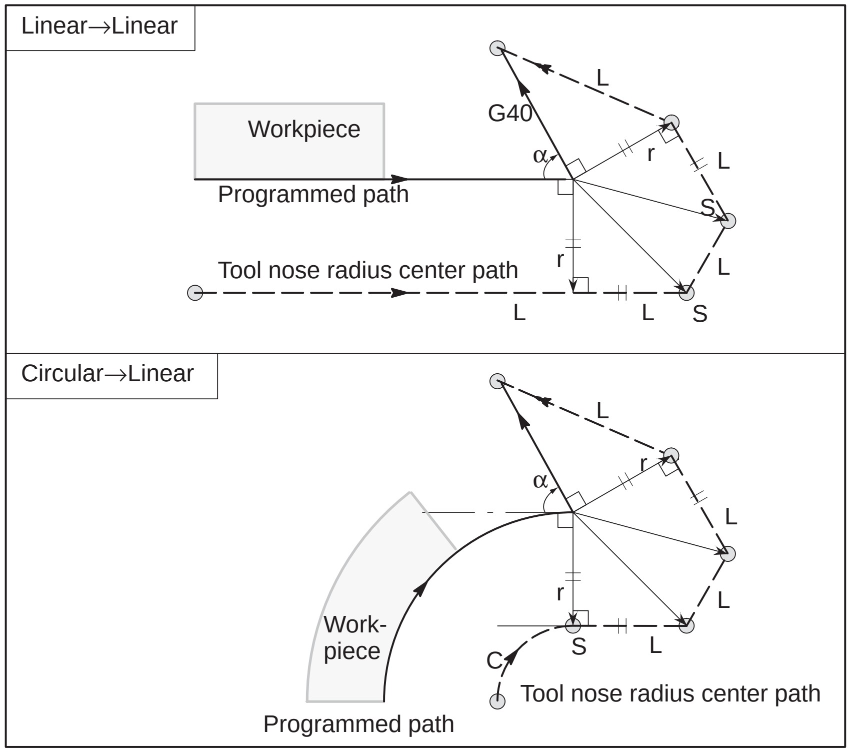
- **Tool movement around an inside corner (180°≦α)**



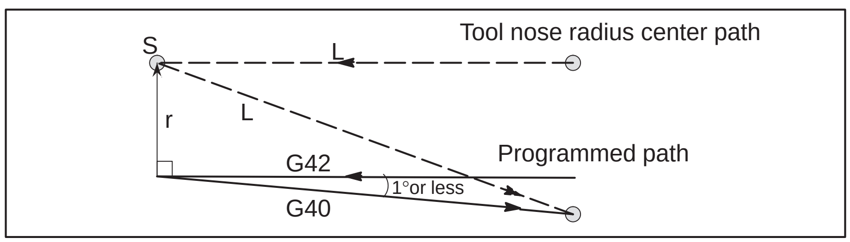
- **Tool movement around an outside corner at an obtuse angle (90°≦α<180°)**



- **Tool movement around an outside corner at an acute angle (α<90°)**

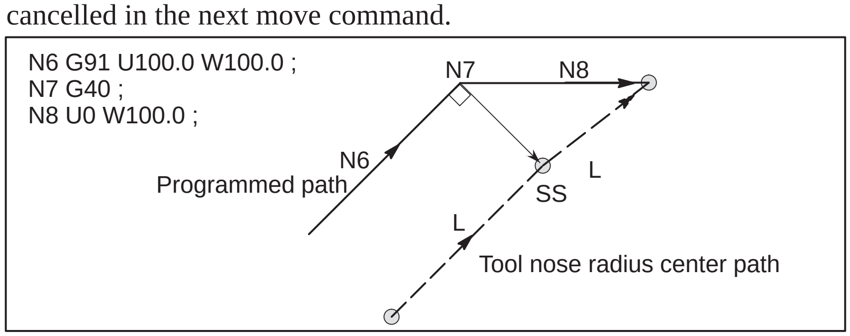


- **Tool movement around the outside linear→linear at an acute angle less than 1 degree (α<1°)**



- **A block without tool movement specified together with offset cancel**

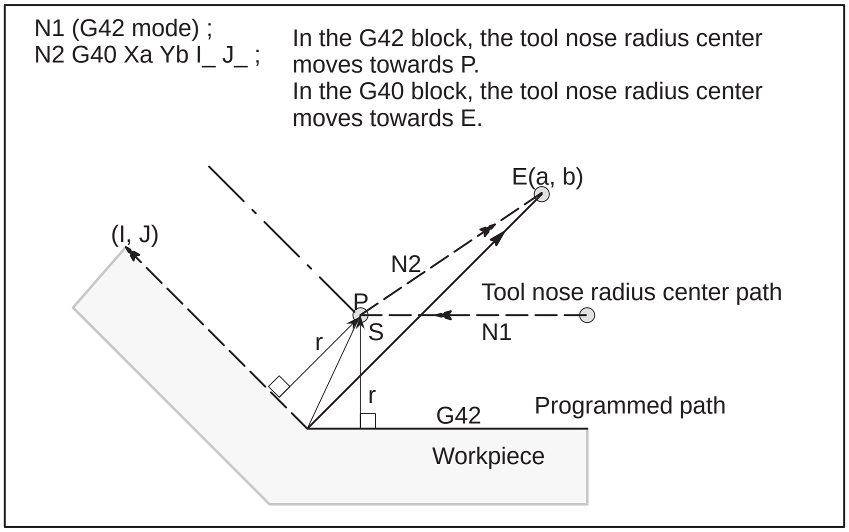
When a block without tool movement is commanded together with an offset cancel, a vector whose length is equal to the offset value is produced in a normal direction to tool motion in the earlier block, the vector is cancelled in the next move command.



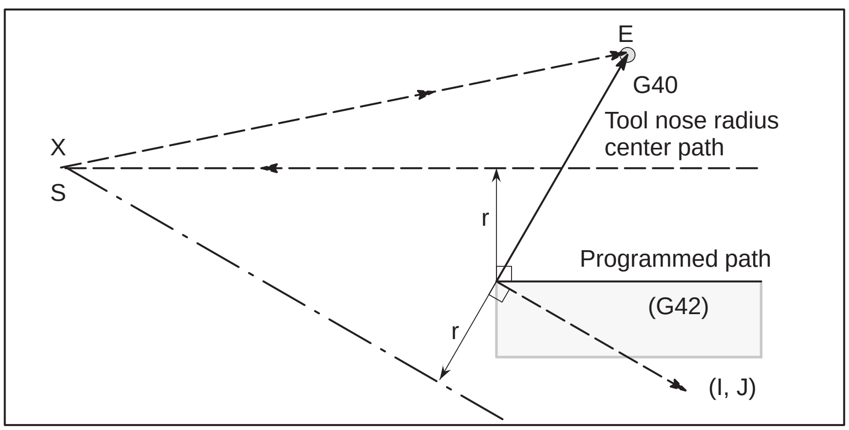
- **Block containing G40 and I_J_K_**

  - The previous block contains G41 or G42

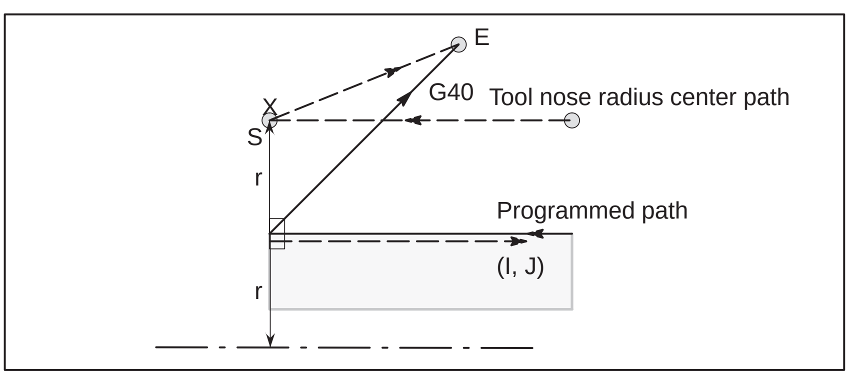
If a G41 or G42 block precedes a block in which G40 and I_, J_, K_ are specified, the system assumes that the path is programmed as a path from the end position determined by the former block to a vector determined by (I,J), (I,K), or (J,K). The direction of compensation in the former block is inherited.



In this case, note that the CNC obtains an intersection of the tool path irrespective of whether inner or outer side machining is specified



When an intersection is not obtainable, the tool comes to the normal position to the previous block at the end of the previous block.



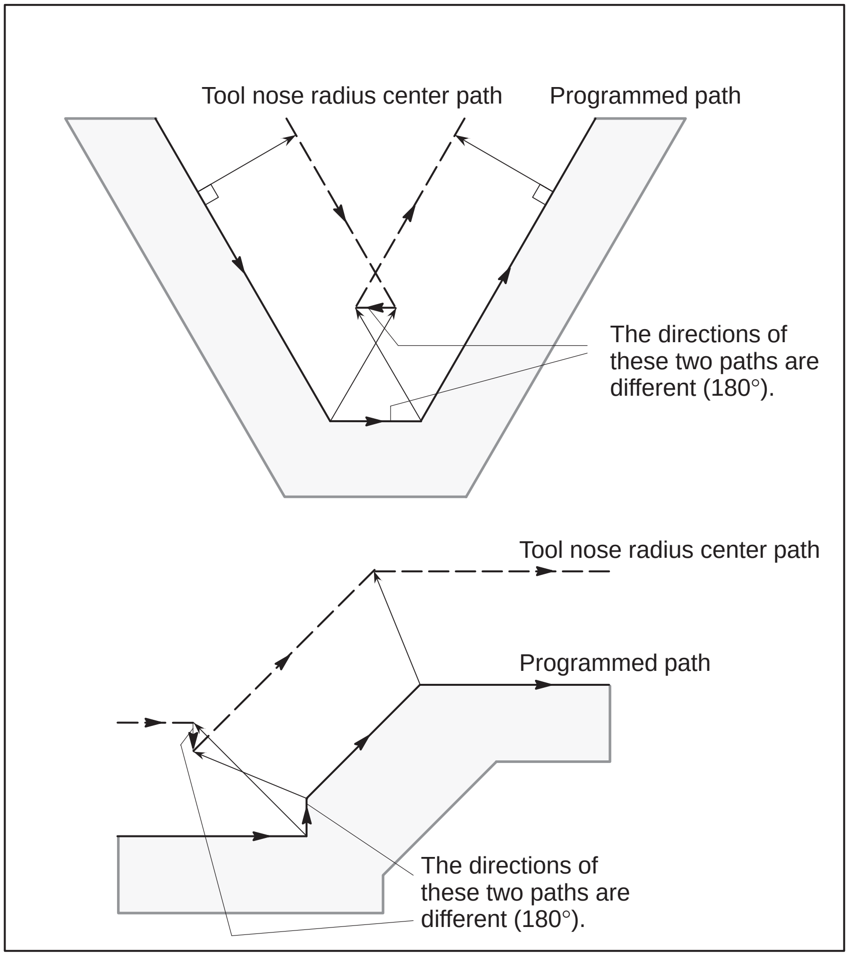
15.3.5 Interference Check

Tool overcutting is called interference. The interference check function checks for tool overcutting in advance. However, all interference cannot be checked by this function. The interference check is performed even if overcutting does not occur.

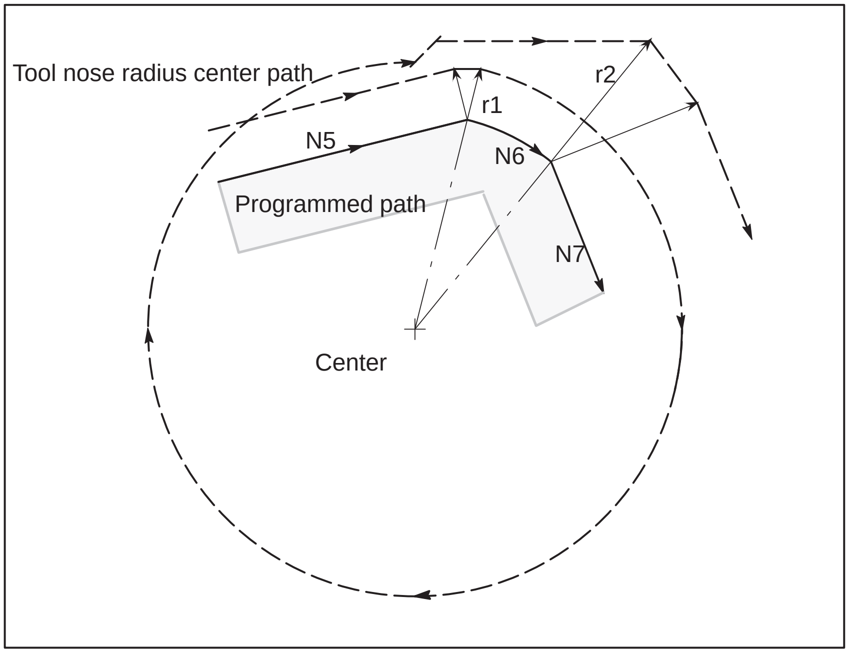
Explanations

- **Criteria for detecting interference**

(1) The direction of the tool nose radius path is different from that of the programmed path (from 90 degrees to 270 degrees between these paths).



(2) In addition to the condition (1), the angle between the start point and end point on the Tool nose radius center path is quite different from that between the start point and end point on the programmed path in circular machining(more than 180 degrees).



(G41)
N5 G01 U200.0 W800.0 T1 ;
N6 G02 U–160.0 W320.0 I–800.0 K–200.0 T2 ;
N7 G01 U–500.0 W200.0 ;
(Tool compensation value corresponding to T1 : r_1 = 200.0)
(Tool compensation value corresponding to T2 : r_2 = 600.0)

In the above example, the arc in block N6 is placed in the one quadrant. But after tool nose radius compensation, the arc is placed in the four quadrants.

- **Correction of interference in advance**

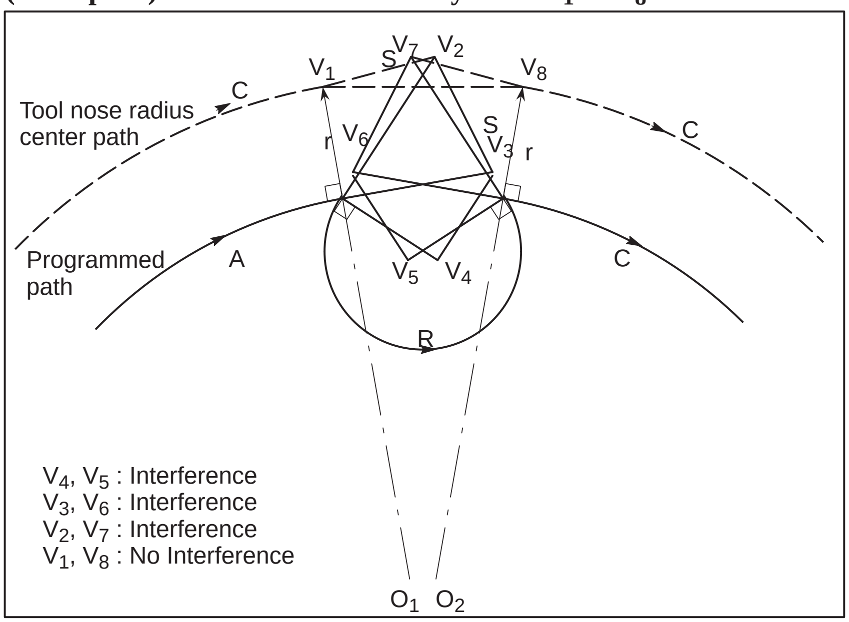
(1) Removal of the vector causing the interference

When tool nose radius compensation is performed for blocks A, B and C and vectors V_1, V_2, V_3 and V_4 between blocks A and B, and V_5, V_6, V_7 and V_8 between B and C are produced, the nearest vectors are checked first. If interference occurs, they are ignored. But if the vectors to be ignored due to interference are the last vectors at the corner, they cannot be ignored.

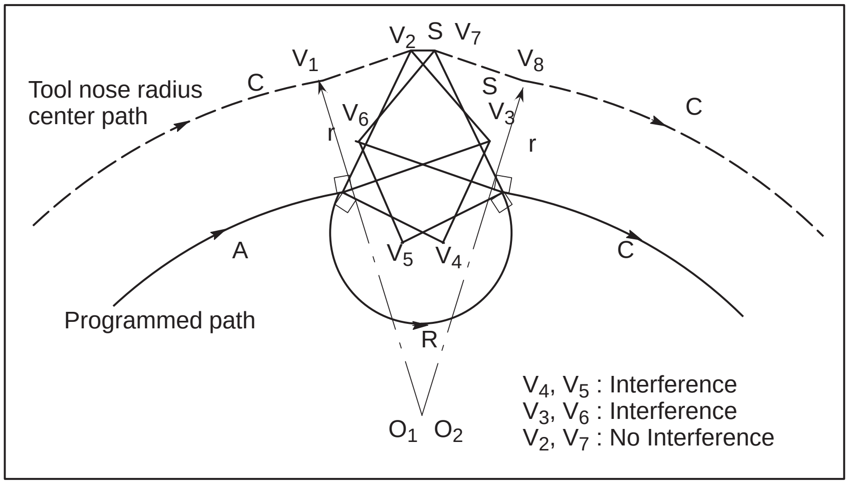
Check between vectors V_4 and V_5
  Interference V_4 and V_5 are ignored.
Check between V_3 and V_6
  Interference V_3 and V_6 are ignored
Check between V_2 and V_7
  Interference V_2 and V_7 are Ignored
Check between V_1 and V_8
  Interference V_1 and V_8 are cannot be ignored

If while checking, a vector without interference is detected, subsequent vectors are not checked. If block B is a circular movement, a linear movement is produced if the vectors are interfered.

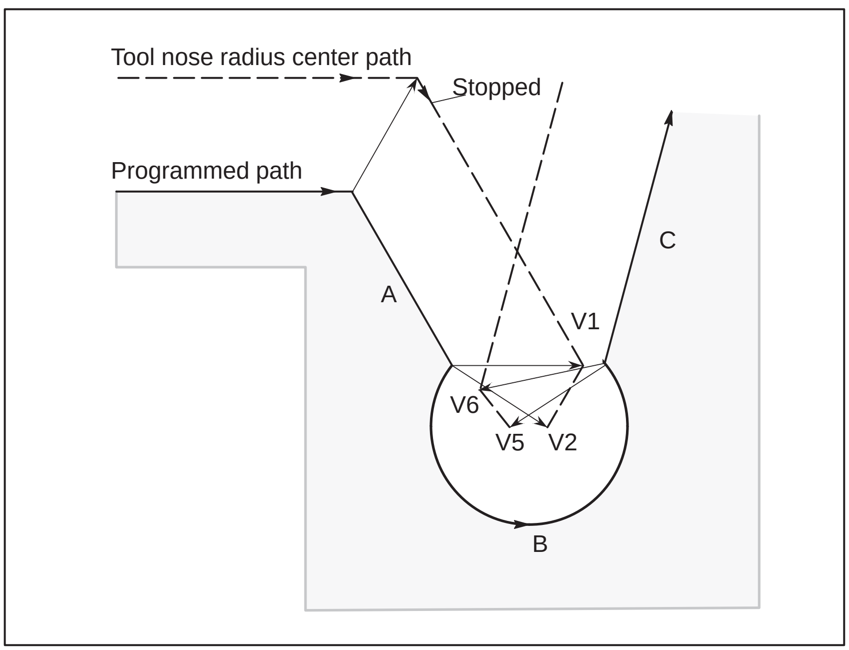
(Example 1) The tool moves linearly from V_1 to V_8



(Example 2) The tool moves linearly from V_1, V_2, V_7, to V_8



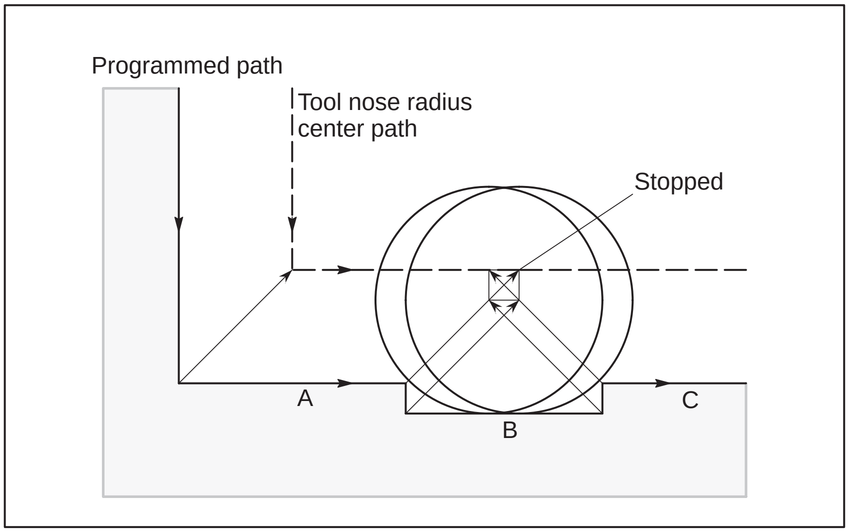
(2) If the interference occurs after correction (1), the tool is stopped with an alarm.
If the interference occurs after correction (1) or if there are only one pair of vectors from the beginning of checking and the vectors interfere, the P/S alarm (No.41) is displayed and the tool is stopped immediately after execution of the preceding block. If the block is executed by the single block operation, the tool is stopped at the end of the block.



After ignoring vectors V_2 and V_5 because of interference, interference also occurs between vectors V_1 and V_6. The alarm is displayed and the tool is stopped.

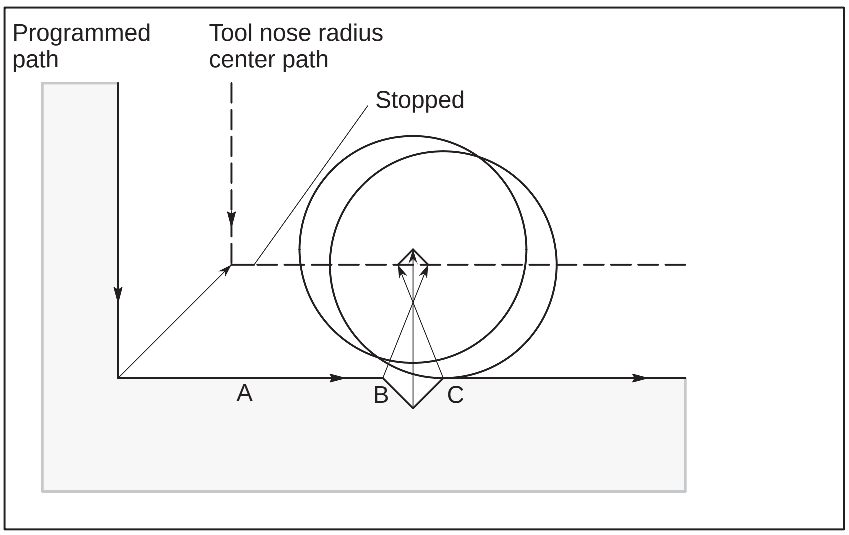
- **When interference is assumed although actual interference does not occur**

(1)**Depression which is smaller than the tool nose radius compensation value**



There is no actual interference, but since the direction programmed in block B is opposite to that of the path after tool nose radius compensation the tool stops and an P/S alarm(No.041) is displayed.

(2)**Groove which is smaller than the tool nose radius compensation value**



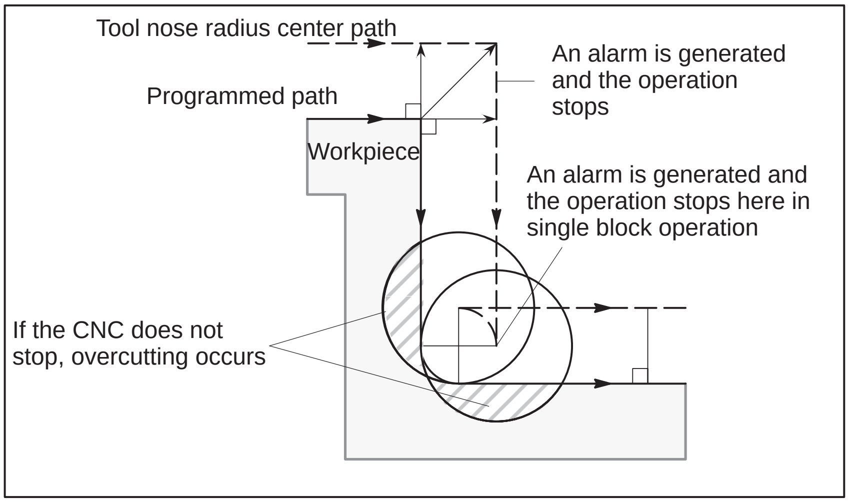
Like (1) , the direction is reverse in block B.

15.3.6 Overcutting by tool nose radius compensation

Explanations

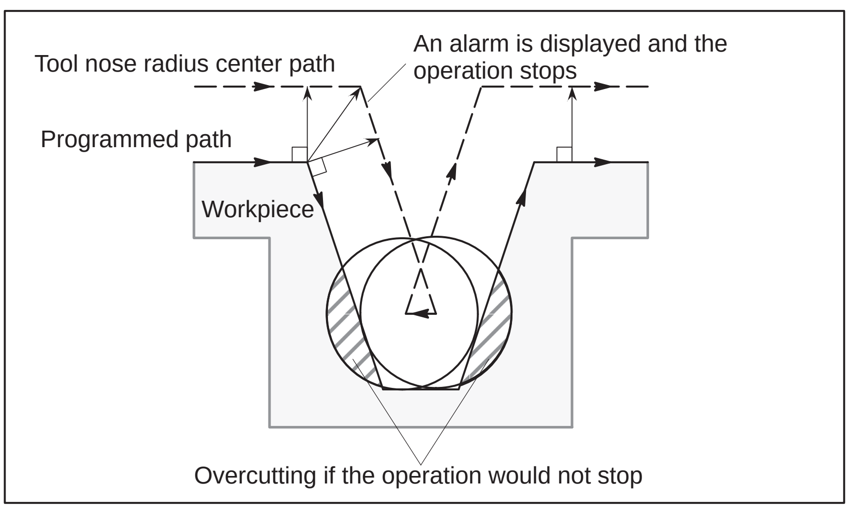
- **Machining an inside corner at a radius smaller than the tool nose radius**

When the radius of a corner is smaller than the cutter radius, because the inner offsetting of the cutter will result in overcuttings, an alarm is displayed and the CNC stops at the start of the block. In single block operation, the overcutting is generated because the tool is stopped after the block execution.



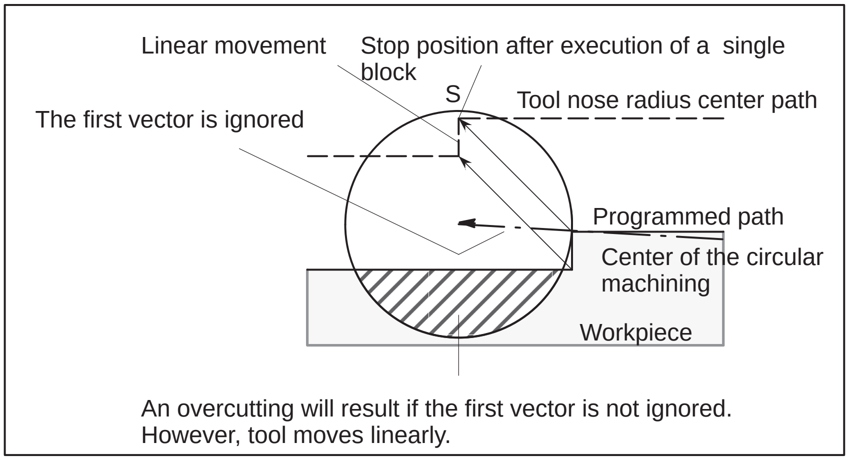
- **Machining a groove smaller than the tool nose radius**

Since the tool nose radius compensation forces the path of the center of the tool to move in the reverse of the programmed direction, overcutting will result. In this case an alarm is displayed and the CNC stops at the start of the block.



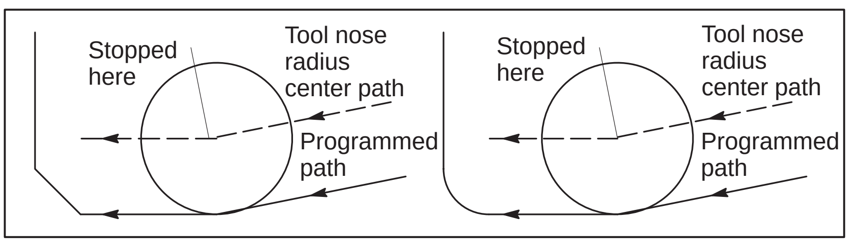
- **Machining a step smaller than the tool nose radius**

When machining of the step is commanded by circular machining in the case of a program containing a step smaller than the tool nose radius, the path of the center of tool with the ordinary offset becomes reverse to the programmed direction. In this case, the first vector is ignored, and the tool moves linearly to the second vector position. The single block operation is stopped at this point. If the machining is not in the single block mode, the cycle operation is continued. If the step is of linear, no alarm will be generated and cut correctly. However uncut part will remain.

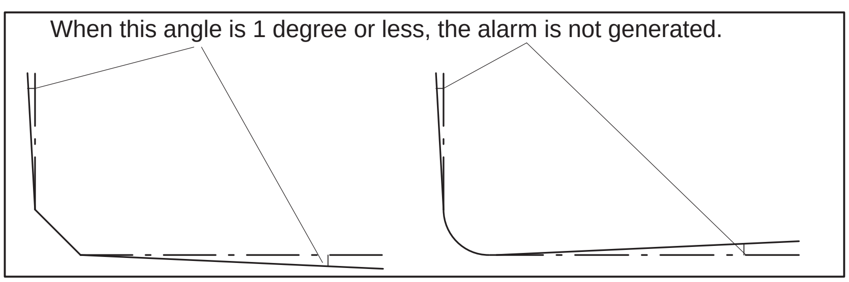


15.3.7 Correction in Chamfering and Corner Arcs

In chamfering or corner arcs, tool nose radius compensation only be performed when an ordinary intersection exists at the corner.
In offset cancel mode, a start–up block or when exchanging the offset direction, compensation cannot be performed, an P/S alarm (No.39) is displayed and the tool is stopped.
In inner chamfering or inner corner arcs, if the chamfering value or corner arc value is smaller than the tool nose radius value, the tool is stopped with an P/S alarm (No.39) since overcutting will occur.

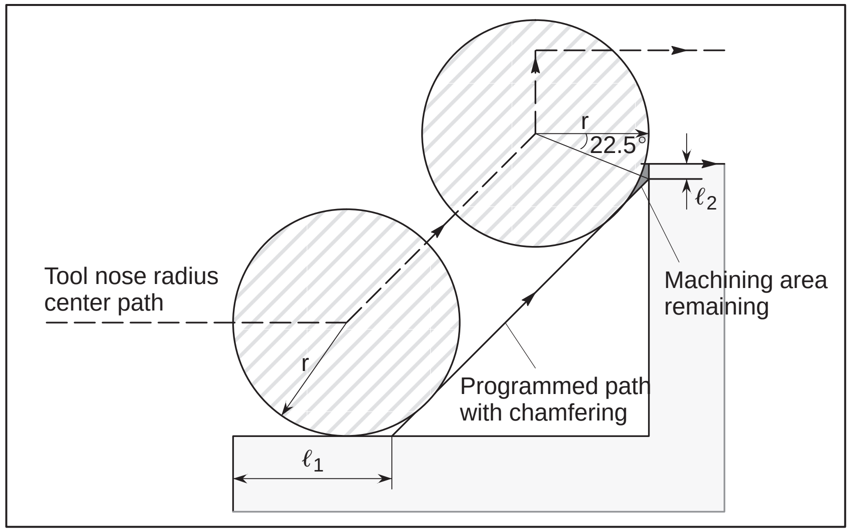


The valid inclination angle of the programmed path in the blocks before and after the corner is 1 degree or less so that the P/S alarm (No.52, 54) generated by the calculating error of tool nose radius compensation does not occur.

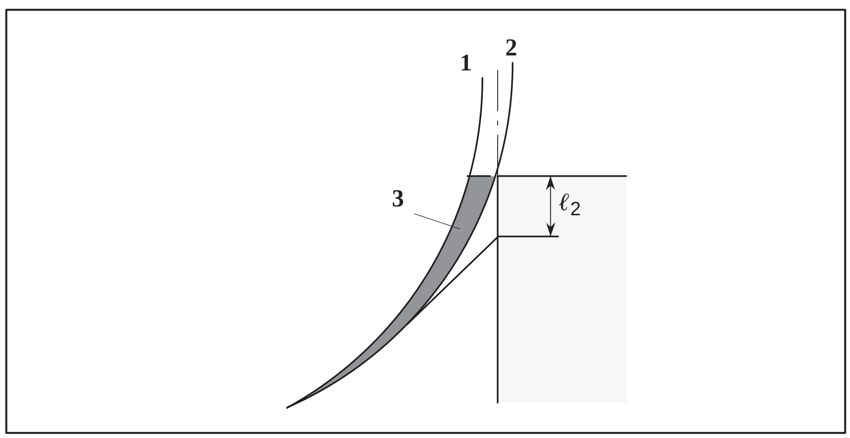


- **When machining area remains or an alarm is generated**

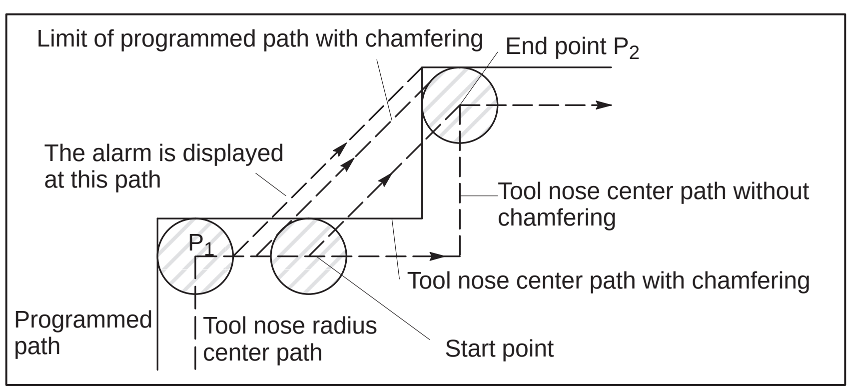
The following example shows a machining area which cannot be cut sufficiently.



In inner chamfering, if the portion of the programmed path that is not a part of the chamfering (in the above figure ℓ_1 or ℓ_2) is in following range, insufficiently cut are will exist.
$0 \leqq \ell_1$ or $\ell_2 < r \cdot \tan 22.5°$ (r : too nose radius)
Enlarged view on the remaining machining area



Although the tool should be positioned at **2** in the above figure, the tool is positioned at **1** (the tool nose is tangent to line L).
Thus, area **3** is not machined.
P/S alarm No.52 or 55 is displayed in the following cases :



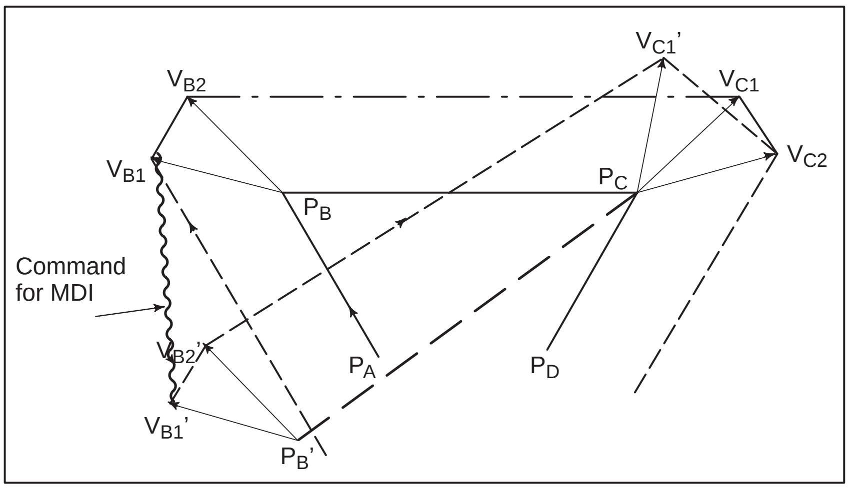
In outer chamfering with an offset, a limit is imposed on the programmed path. The path during chamfering coincides with the intersection points P_1 or P_2 without chamfering, therefore, outer chamfering is limited. In the figure above, the end point of the tool center path with chamfering coincides with the intersection point (P2) of the next block without chamfering. If the chamfering value is more than the limit value specified, P/S alarm No.52 or 55 will be displayed.

15.3.8 Input Command from MDI

Tool nose radius compensation is not performed for commands input from the MDI.

However, when automatic operation using absolute commands is temporarily stopped by the single block function, MDI operation is performed, then automatic operation starts again, the tool path is as follows :

In this case, the vectors at the start position of the next block are translated and the other vectors are produced by the next two blocks. Therefore, from next block but one, tool nose radius compensation is accurately performed.



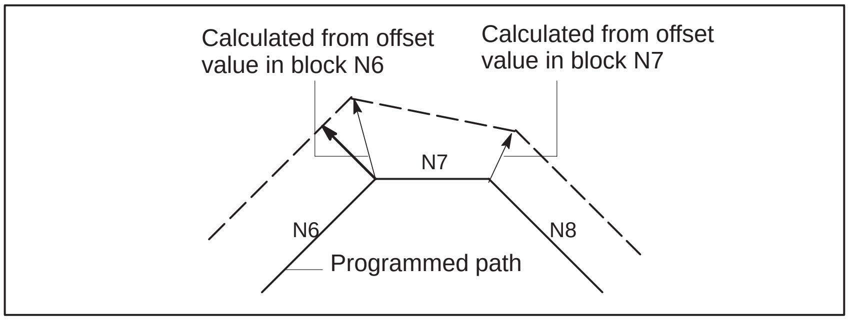
When position P_A, P_B, and P_C are programmed in an absolute command, tool is stopped by the single block function after executing the block from P_A to P_B and the tool is moved by MDI operation. Vectors V_{B1} and V_{B2} are translated to V_{B1}' and V_{B2}' and offset vectors are recalculated for the vectors V_{C1} and V_{C2} between block P_B–P_C and P_C–P_D.

However, since vector V_{B2} is not calculated again, compensation is accurately performed from position P_C.

15.3.9 General Precautions for Offset Operations

- **Changing the offset value**

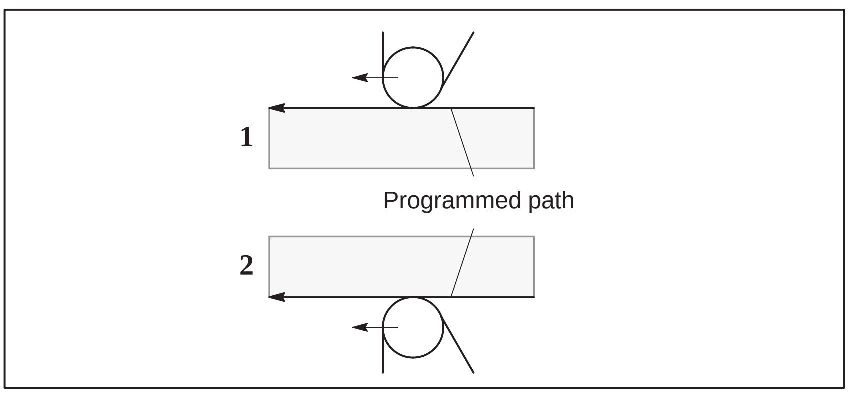
In general, the offset value is changed in cancel mode, or when changing tools. If the offset value is changed in offset mode, the vector at the end point of the block is calculated for the new offset value.



When some vectors are produced between blocks N6 and N7, the vector at the end point of the present blocks is calculated using the offset value of the block N6.

- **The polarity of the offset amount and the tool nose center path**

When a negative offset value is specified, the program is executed for the figure which is created by exchanging G41 for G42 or G42 for G41 in the process sheet.
A tool machining an inner profile will machine the occur profile, and tool machining the outer profile will machine the inner profile.
An example is shown below. In general, CNC machining is programmed assuming a positive offset value. When a program specifies a tool path as shown in **1**, the tool will move as shown in **2** if a negative offset is specified. The tool in **2** will move as shown in **1** when the sign of the offset value is reserved.



Notes
When the sign of the offset value is reversed, the offset vector of the tool nose is reversed but the imaginary tool nose direction does not change.
Therefore, do not reverse the sign of the offset value when starting the machining meeting the imaginary tool nose to the start point.

15.3.10 G53, G28, G30, and G30.1 commands in tool–tip radius compensation mode

- When a G53 command is executed in tool–tip radius compensation mode, the tool–tip radius compensation vector is automatically canceled before positioning, that vector being automatically restored by a subsequent move command. The format for restoring the tool–tip radius compensation vector is the FS16 type when bit 2 (CCN) of parameter No. 5003 is set to 0, or the FS15 type when the bit is set to 1.
- When a G28, G30, or G30.1 command is executed in tool–tip radius compensation mode, the tool–tip radius compensation vector is automatically canceled before automatic reference position return, that vector being automatically restored by a subsequent move command. The timing and format for canceling and restoring the tool–tip radius compensation vector are the FS15 type when bit 2 (CCN) of parameter No. 5003 is set to 1, or the FS16 type when the bit is set to 0.

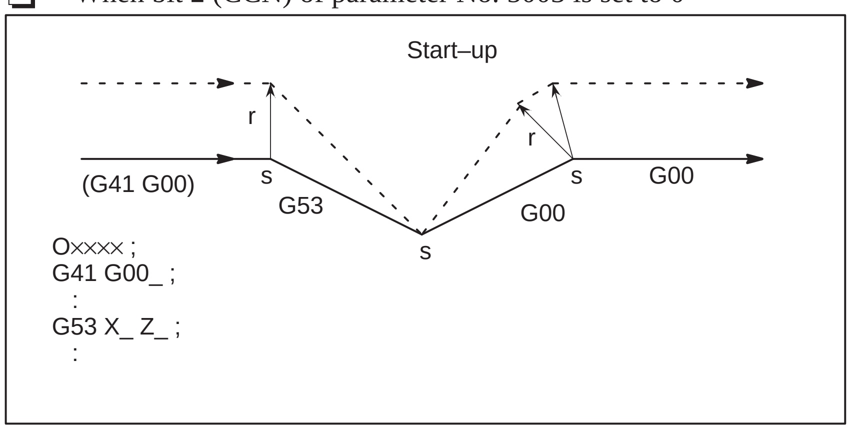
Explanations

- **G53 command in tool–tip radius compensation mode**

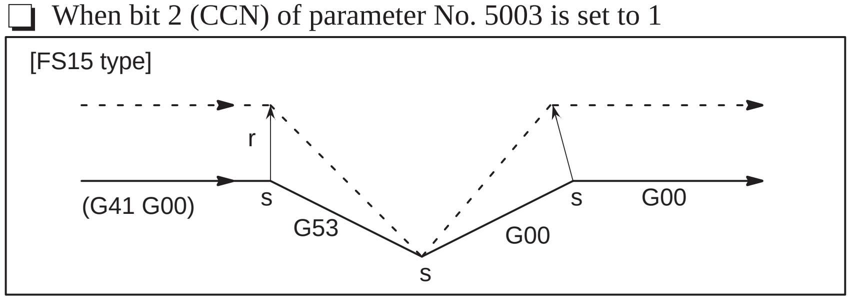
When a G53 command is executed in tool–tip radius compensation mode, a vector having a length equal to the offset is created, at the end of the preceding block, perpendicular to the direction in which the tool moves. When the tool moves to a specified position according to the G53 command, the offset vector is canceled. When the tool moves according to the next command, the offset vector is automatically restored.
The format for restoring the tool–tip radius compensation vector is the start–up type when bit 2 (CCN) of parameter No. 5003 is set to 0, or the intersection vector type (FS15 type) when the bit is set to 1.

- G53 command in offset mode

❑ When bit 2 (CCN) of parameter No. 5003 is set to 0

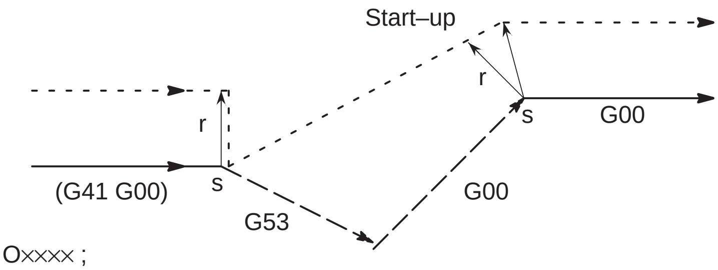


❑ When bit 2 (CCN) of parameter No. 5003 is set to 1



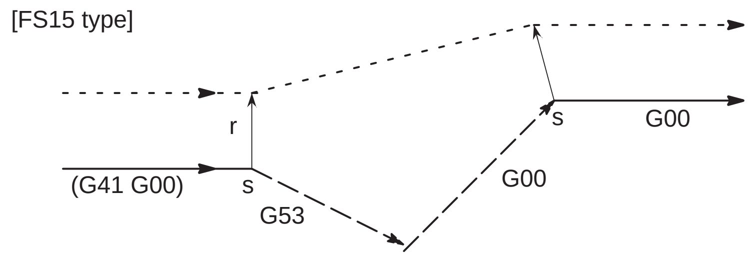
- Incremental G53 command in offset mode

❑ When bit 2 (CCN) of parameter No. 5003 is set to 0



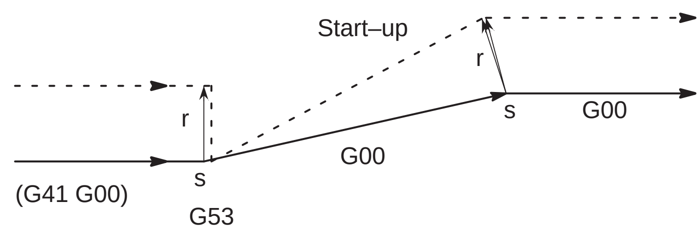

```
O××××ￓ ;
G41 G00_ ;
  :
G53 U_ W_ ;
  :
```

❑ When bit 2 (CCN) of parameter No. 5003 is set to 1



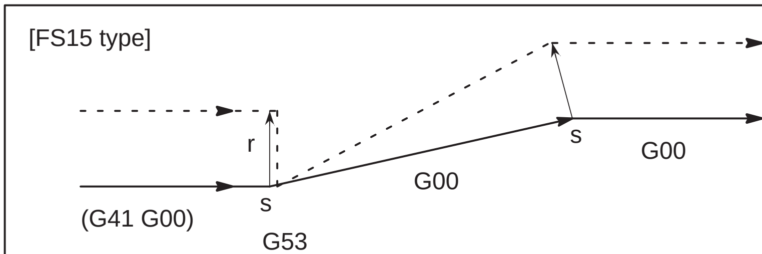
- G53 command specifying no movement in offset mode

❑ When bit 2 (CCN) of parameter No. 5003 is set to 0




```
O×××× ;
G90 G41_ ;
  :
G00 X20. Y20. ;
G53 X20. Y20. ;
  :
```

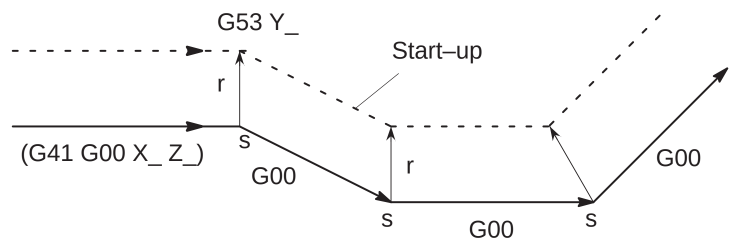
❑ When bit 2 (CCN) of parameter No. 5003 is set to 1



Notes

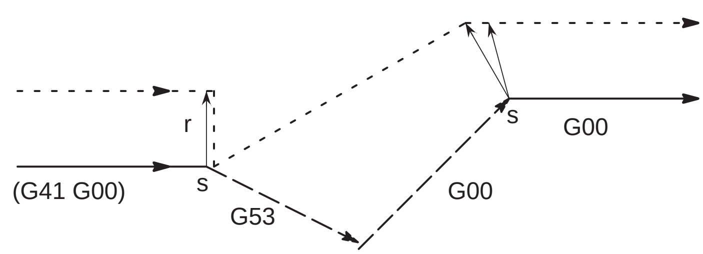
1 When an axis not included in the tool–tip radius compensation plane is specified in a G53 command, a vector perpendicular to the direction in which the tool moves is created at the end of the preceding block and the tool does not move. Offset mode is automatically resumed from the next block (in the same way as when two or more blocks specifying no movement are consecutively executed).

Example)
When bit 2 (CCN) of parameter No. 5003 is set to 0



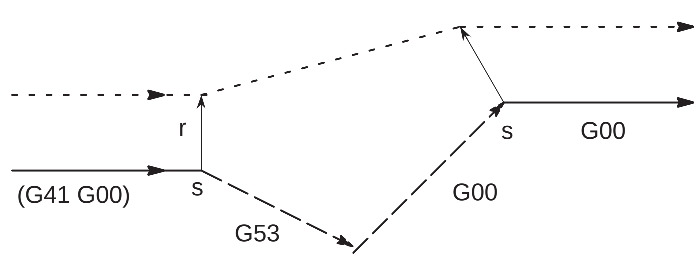
2 When a G53 command is executed in tool–tip radius compensation mode when all–axis machine lock is applied, positioning is not performed for those axes to which machine lock is applied and the offset vector is not canceled. When bit 2 (CCN) of parameter No. 5003 is set to 0 or each–axis machine lock is applied, the offset vector is canceled.

Example 1)
When bit 2 (CCN) of parameter No. 5003 is set to 0 and all–axis machine lock is applied



Example 2)
When bit 2 (CCN) of parameter No. 5003 is set to 1 and all–axis machine lock is applied

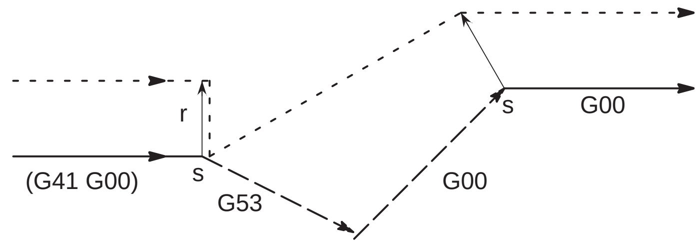
[FS15 type]



Notes

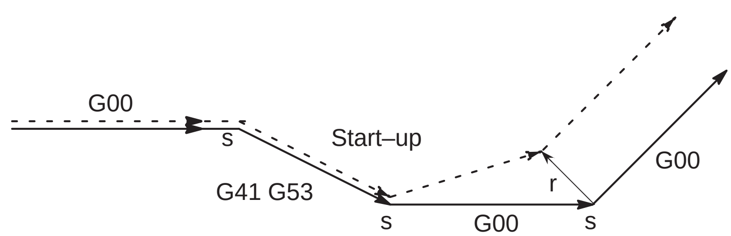
Example 3)
When bit 2 (CCN) of parameter No. 5003 is set to 1 and each–axis machine lock is applied

[FS15 type]



3 When a G53 command is specified as a start–up block, the next block actually becomes the start–up block. When bit 2 (CCN) of parameter No. 5003 is set to 1, however, the next block creates an intersection vector.

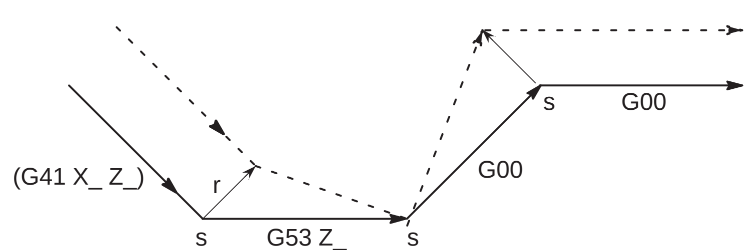
Example)
When bit 2 (CCN) of parameter No. 5003 is set to 0



4 When a compensation axis is specified in a G53 command in tool–tip radius compensation mode, the vectors for other compensation axes are also canceled. This also applies when bit 2 (CCN) of parameter No. 5003 is set to 1. (The FS15 cancels only the vector for the specified axis. Note that the FS15 type cancellation differs from the actual FS15 specification in this point.)

Example)
When bit 2 (CCN) of parameter No. 5003 is set to 0

[FS15 type]

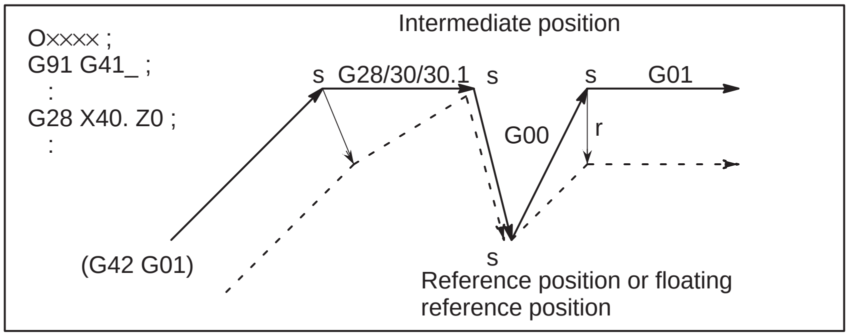


- **G28, G30, G30.1 command in tool–tip radius compensation mode**

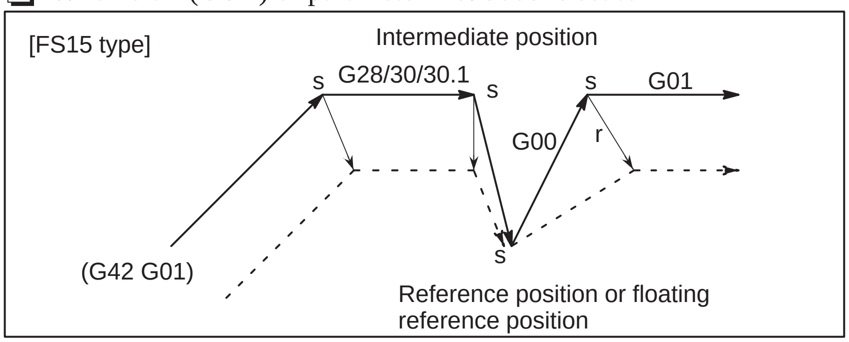
When a G28, G30, or G30.1 command is executed in tool–tip radius compensation mode, the operation specified in the command is performed according to the FS15 format if bit 2 (CCN) of parameter No. 5003 is set to 1. An intersection vector is created at the end of the preceding block and a perpendicular vector is created at the intermediate position. The offset vector is canceled when the tool moves from the intermediate position to the reference position. The offset vector is restored as an intersection vector by the next block.

- G28, G30, or G30.1 command in offset mode (with movement to both an intermediate position and reference position performed)

❑ When bit 2 (CCN) of parameter No. 5003 is set to 0

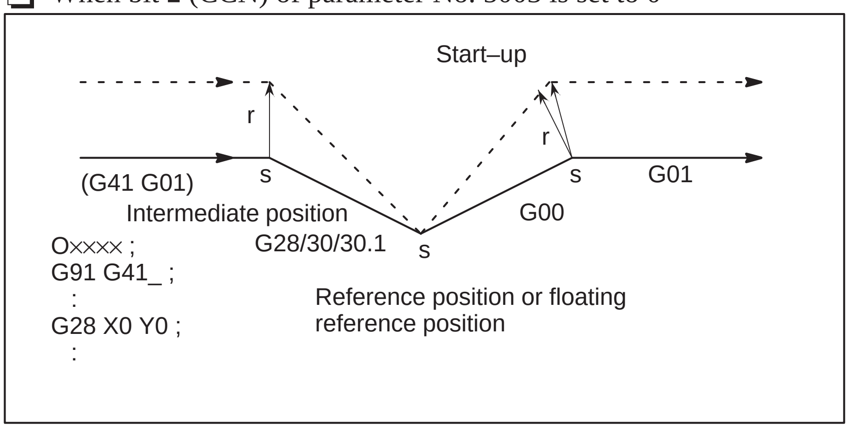


❑ When bit 2 (CCN) of parameter No. 5003 is set to 1

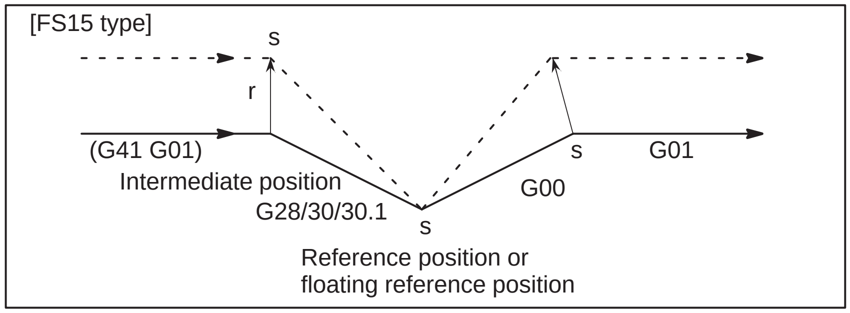


- G28, G30, or G30.1 command in offset mode (with movement to an intermediate position not performed)

❑ When bit 2 (CCN) of parameter No. 5003 is set to 0

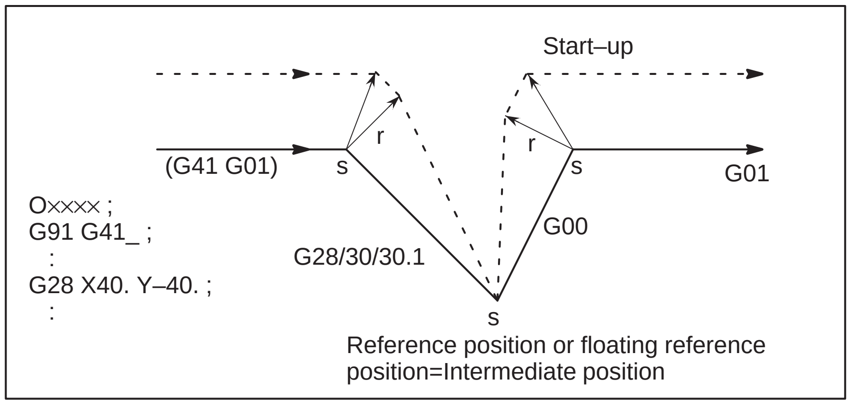


❑ When bit 2 (CCN) of parameter No. 5003 is set to 1

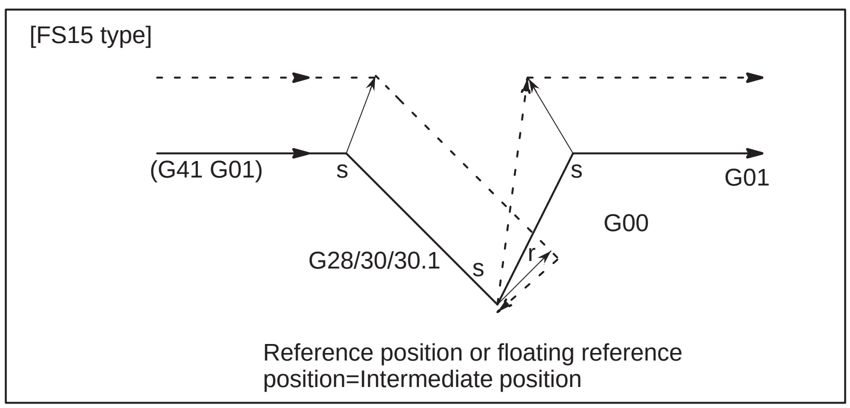


- G28, G30, or G30.1 command in offset mode (with movement to a reference position not performed)

❑ When bit 2 (CCN) of parameter No. 5003 is set to 0

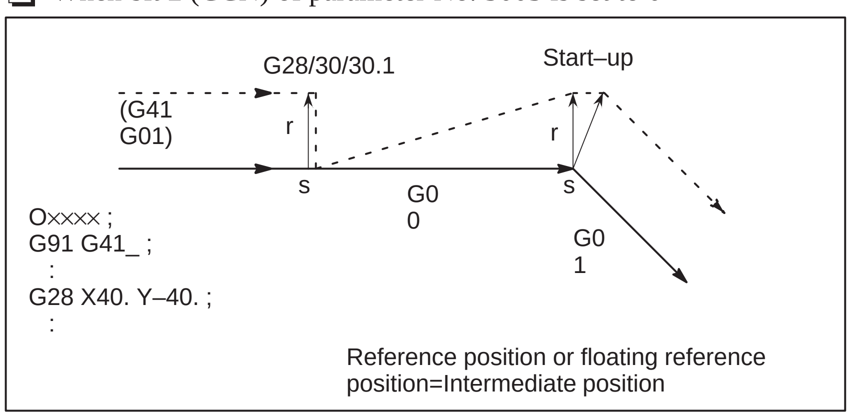


❑ When bit 2 (CCN) of parameter No. 5003 is set to 1



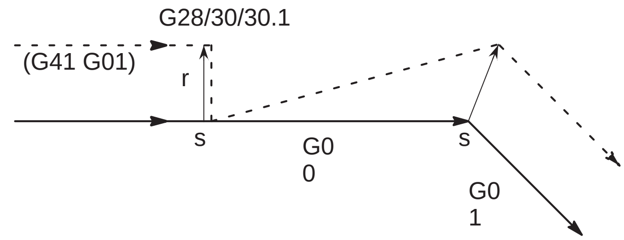
- G28, G30, or G30.1 command in offset mode (with no movement)

❑ When bit 2 (CCN) of parameter No. 5003 is set to 0



❑ When bit 2 (CCN) of parameter No. 5003 is set to 1

[FS15 type]



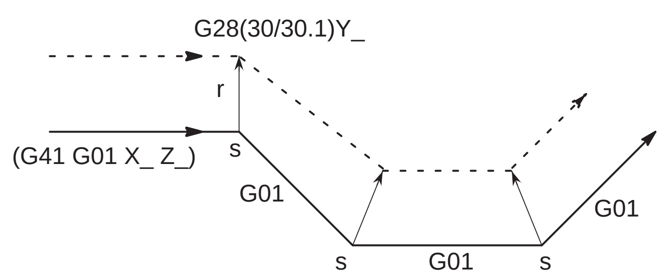
Reference position or floating reference position=Intermediate position

Notes

1 When an axis not included in the tool–tip radius compensation plane is specified in a G28, G30, or G30.1 command, a vector perpendicular to the direction in which the tool moves is created at the end of the preceding block and the tool does not move. Offset mode is automatically resumed from the next block (in the same way as when two or more blocks specifying no movement are consecutively executed).

Example)
When bit 2 (CCN) of parameter No. 5003 is set to 1.

[FS15 type]

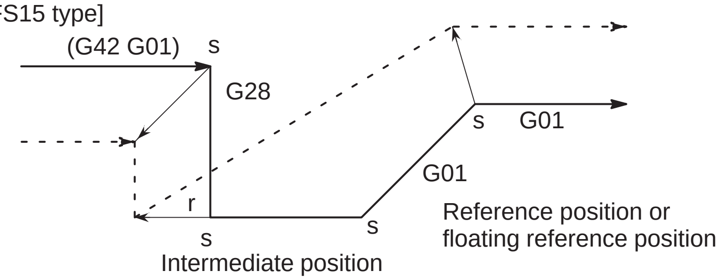


Notes

2 When a G28, G30, or G30.1 command is executed when all–axis machine lock is applied, a vector perpendicular to the direction in which the tool moves is created at the intermediate position. In this case, the tool does not move to the reference position and the offset vector is not canceled. When bit 2 (CCN) of parameter No. 5003 is set to 0 or each–axis machine lock is applied, the offset vector is canceled.

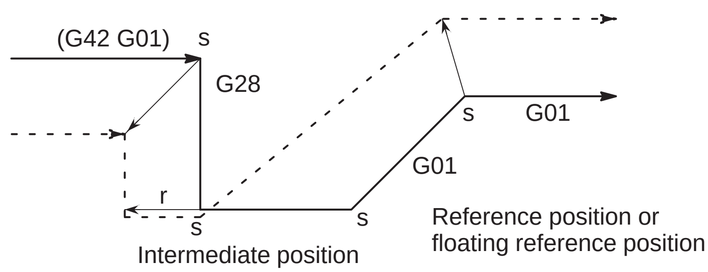
Example 1)
When bit 2 (CCN) of parameter No. 5003 is set to 1.

[FS15 type]



Example 2)
When bit 2 (CCN) of parameter No. 5003 is set to 0 and all–axis machine lock is applied

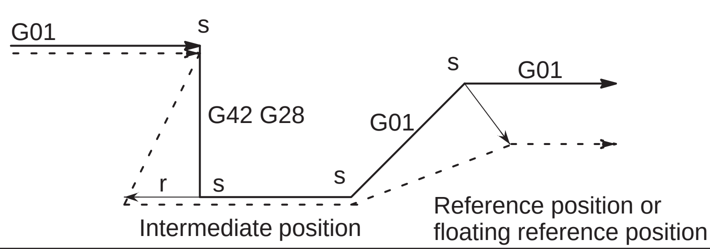
[FS15 type]



3 When a G28, G30, or G30.1 command is specified as a start–up block, a vector perpendicular to the direction in which the tool moves is created at the intermediate position. The vector is then canceled at the reference position. The next block creates an intersection vector.

Example 1)
When bit 2 (CCN) of parameter No. 5003 is set to 1.

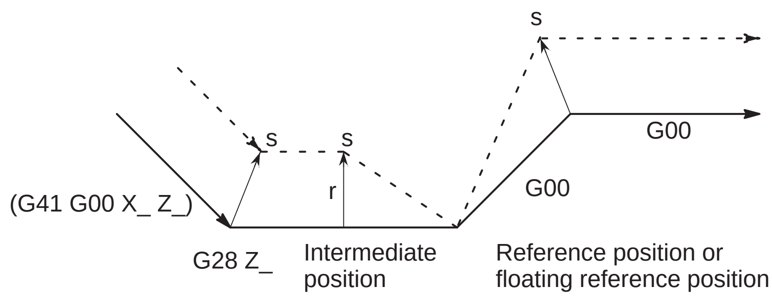
[FS15 type]



Notes

4 When a compensation axis is specified in a G28, G30, or G30.1 command in tool–tip radius compensation mode, the vectors for other compensation axes are also canceled. This also applies when bit 2 (CCN) of parameter No. 5003 is set to 1. (The FS15 cancels only the vector for the specified axis. Note that the FS15 type cancellation differs from the actual FS15 specification in this point.)

[FS15 type]



15.4 CORNER CIRCULAR INTERPOLATION FUNCTION (G39)

During radius compensation for the tool tip, corner circular–interpolation, with the specified compensation value used as the radius, can be performed by specifying G39 in offset mode.

Format

In offset mode, specify
G39;
or
G39 I_J_K_;.

Explanations

- **Corner circular–interpolation**

Corner circular–interpolation, with the specified compensation value used as a radius, can be performed by specifying the operation as shown above. Whether the tool moves clockwise or counterclockwise depends on whether the last–specified direction code is G41 or G42. G39 is a single–shot G code.

- **G39 without I, J, and K**

Specifying G39; creates a corner arc for which the end vector is perpendicular to the start point of the next block.

- **G39 with I, J, and K**

Specifying G39 I_J_K_; creates a corner arc for which the end vector is perpendicular to the vector specified with I, J, and K.

Limitations

- **Move command**

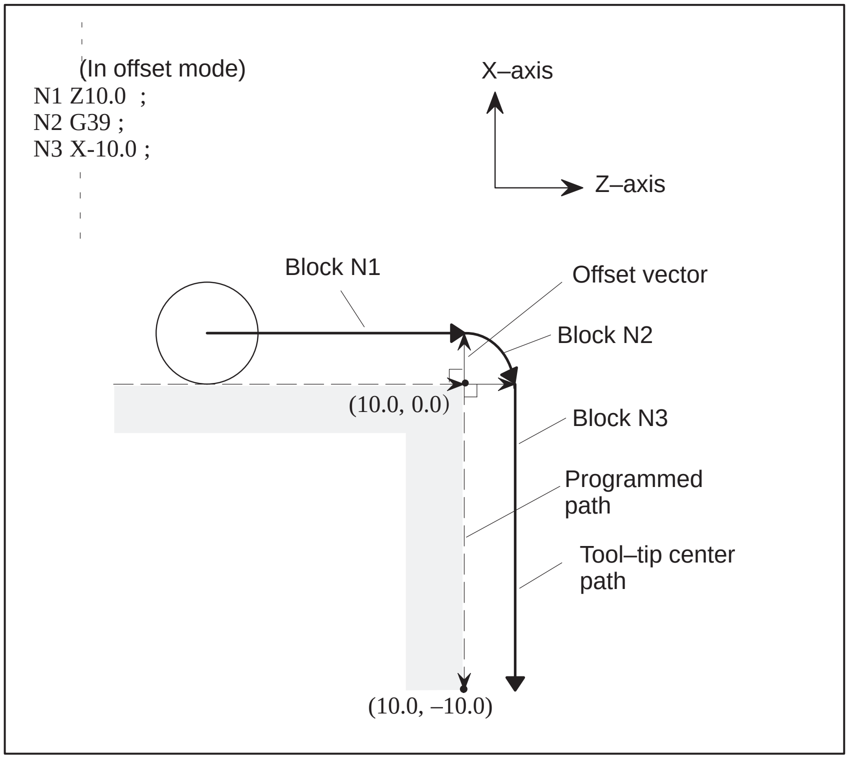
A move operation cannot be specified in a block in which G39 is specified.

- **Non–move command**

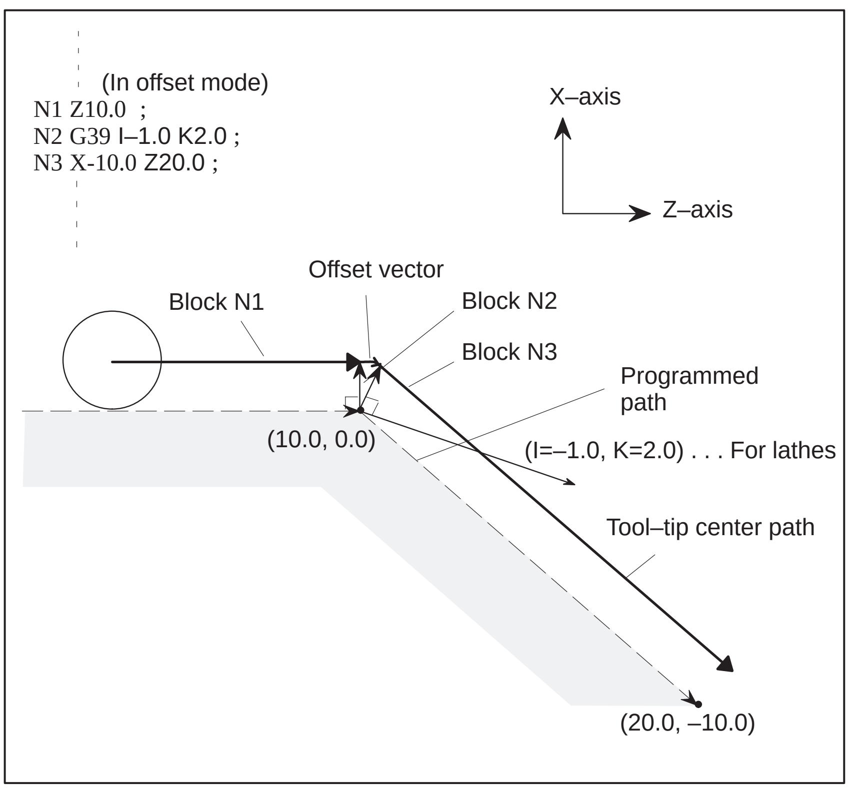
Two or more contiguous blocks with no move operations can not be specified immediately after a block in which G39, without I, J, and K, is specified. (If a move command is specified in a block with a move distance of 0, it is assumed to be two or more contiguous blocks with no more operations.) If those blocks are specified, the offset vector momentarily disappears and the system automatically returns to offset mode.

Examples

- **G39 without I, J, and K**



- **G39 with I, J, and K**



15.5 TOOL COMPENSA– TION VALUES, NUMBER OF COMPENSATION VALUES, AND ENTERING VALUES FROM THE PROGRAM (G10)

Tool compensation values include tool geometry compensation values and tool wear compensation (Fig. 15.5 (a)).
Tool compensation can be specified without differentiating compensation for tool geometry from that for tool wear.

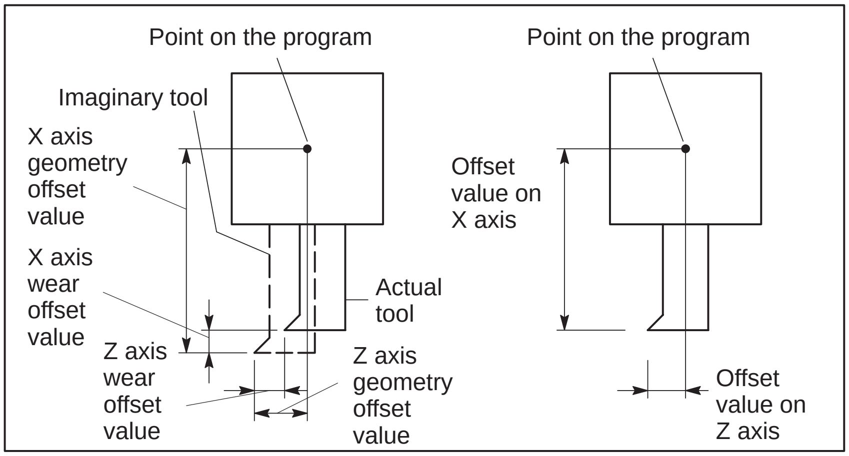


Fig. 15.5(a) Difference the tool geometry offset from tool wear offset

Fig. 15.5(b) Not difference the tool geometry offset from tool wear offset

Tool compensation values can be entered into CNC memory from the CRT/MDI panel (see section III–9.1) or from a program.
A tool compensation value is selected from the CNC memory when the corresponding code is specified after address T in a program.
The value is used for tool offset or tool nose radius compensation.
See subsec. II–15.1.2 for details.

15.5.1 Tool Compensation and Number of Tool Compensation

- **Valid range of tool compensation values**

Table 15.4.1 shows the valid input range of tool compensation values.

Table 15.5.1 Valid range of tool compensation values

Increment system	Tool compensation value	
	Metric input (mm)	Inch input (inch)
IS–B	–999.999 to +999.999 mm	–99.9999 to +99.9999 inch
IS–C	–999.9999 to +999.9999 mm	–99.99999 to +99.99999 inch

The maximum tool wear compensation can be changed by setting parameter No.5013.

- **Number of tool compensation**

The memory can hold 16, 32, 64, or 99 tool compensation values.

Notes
With the two–path control, the number of specified tool compensation values equals the number of tool compensations for each tool post.

15.5.2 Changing of Tool Offset value (Programmable Data Input) (G10)

Offset values can be input by a program using the following command :

Format

G10 P_ X_ Y_ Z_ R_ Q_ ;

or

G10 P_ U_ V_ W_ C_ Q_ ;

P : Offset number
- 0 : Command of work coordinate system shift value
- 1–64 : Command of tool wear offset value
  Command value is offset number
- 10000+(1–64) : Command of tool geometry offset value
  (1–64) : Offset number

X : Offset value on X axis (absolute)
Y : Offset value on Y axis (absolute)
Z : Offset value on Z axis (absolute)
U : Offset value on X axis (incremental)
V : Offset value on Y axis (incremental)
W: Offset value on Z axis (incremental)
R : Tool nose radius offset value (absolute)
R : Tool nose radius offset value (incremental)
Q : Imaginary tool nose number

In an absolute command, the values specified in addresses X, Y , Z, and R are set as the offset value corresponding to the offset number specified by address P. In an incremental command, the value specified in addresses U, V, W, and C is added to the current offset value corresponding to the offset number.

Notes

Notes

1. Addresses X, Y, Z, U, V, and W can be specified in the same block.
2. Use of this command in a program allows the tool to advance little by little. This command can also be used input offset values one at a time from a program by specifying this command successively instead of inputting these values one at a time from the MDI unit.

15.6 AUTOMATIC TOOL OFFSET (G36, G37)

When a tool is moved to the measurement position by execution of a command given to the CNC, the CNC automatically measures the difference between the current coordinate value and the coordinate value of the command measurement position and uses it as the offset value for the tool. When the tool has been already offset, it is moved to the measurement position with that offset value. If the CNC judges that further offset is needed after calculating the difference between the coordinate values of the measurement position and the commanded coordinate values, the current offset value is further offset.
Refer to the instruction manuals of the machine tool builder for details.

Explanations

- **Coordinate system**

When moving the tool to a position for measurement, the coordinate system must be set in advance. (The work coordinate system for programming is used in common.)

- **Movement to measurement position**

A movement to a measurement position is performed by specifying as follows in the MDI, or MEM mode :

G36 Xx_a ; or **G37 Zz_a ;**

In this case, the measurement position should be x_a or z_a (absolute command).
Execution of this command moves the tool at the rapid traverse rate toward the measurement position, lowers the feedrate halfway, then continues to move it until the approach end signal from the measuring instrument is issued. When the tool tip reaches the measurement position, the measuring instrument outputs the measurement position reach signal to the CNC which stops the tool.

- **Offset**

The current tool offset value is further offset by the difference between the coordinate value (α or β) when the tool has reached the measurement position and the value of x_a or z_a specified in G36Xx_a or G37Zz_a.

Offset value x = Current offset value x+(α–x_a)
Offset value z = Current offset value z+(β–z_a)

x_a : Programmed X–axis measurement point
z_a : Programmed Z–axis measurement point

These offset values can also be altered from the MDI keyboard.

- **Feedrate and alarm**

The tool, when moving from the stating position toward the measurement position predicted by x_a or z_a in G36 or G37, is fed at the rapid traverse rate across area **A**. Then the tool stops at point T (x_a–γ_x or z_a–γ_z) and moves at the measurement feedrate set by parameter (No.6241) across areas **B**, **C**, and **D**. If the approach end signal turns on during movement across area B, alarm is generated. If the approach end signal does not turn on before point V, and tool stops at point V and P/S alarm (No.080) is generated.

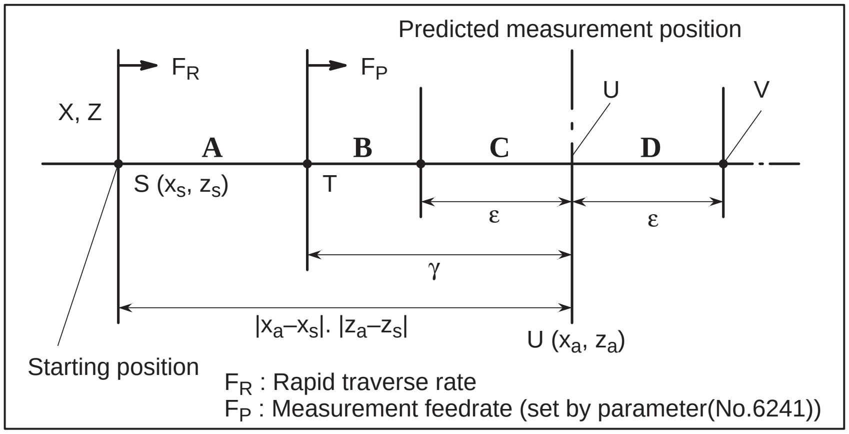
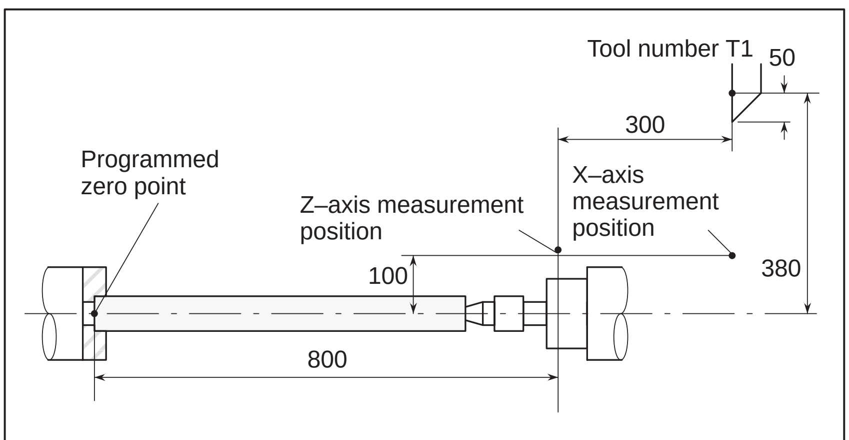


Fig.15.6(a) Feedrate and Alarm

Examples

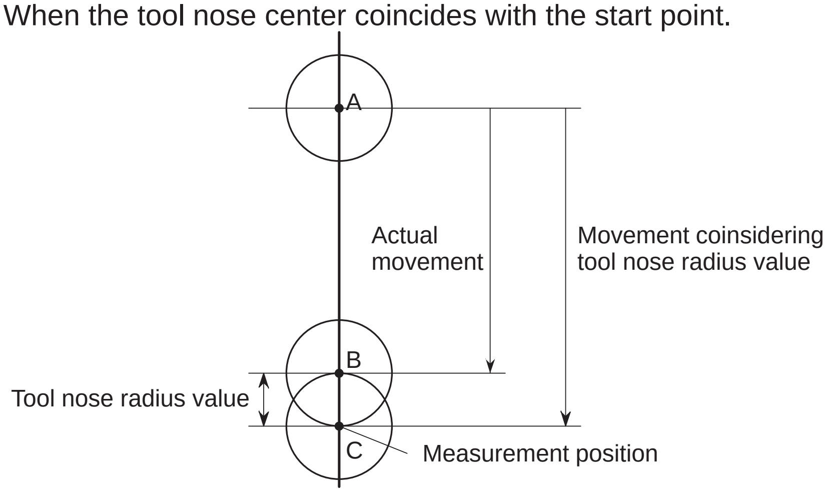


G50 X760.0 Z1100.0 ;	Programming of absolute zero point (Coordinate system setting)
S01 M03 T0101 ;	Specifies tool T1, offset number 1, and spindle revolution
G36 X200.0 ;	Moves to the measurement position If the tool has reached the measurement position at X198.0 ; since the correct measurement position is 200 mm, the offset value is altered by 198.0–200.0=–2.0mm.
G00 X204.0 ;	Refracts a little along the X axis.

G37 Z800.0 ;	Moves to the Z–axis measurement position. If the tool has reached the measurement position at X804.0, the offset value is altered by 804.0–800.0=4.0mm.
T0101 ;	Further offsets by the difference. The new offset value becomes valid when the T code is specified again.

Notes

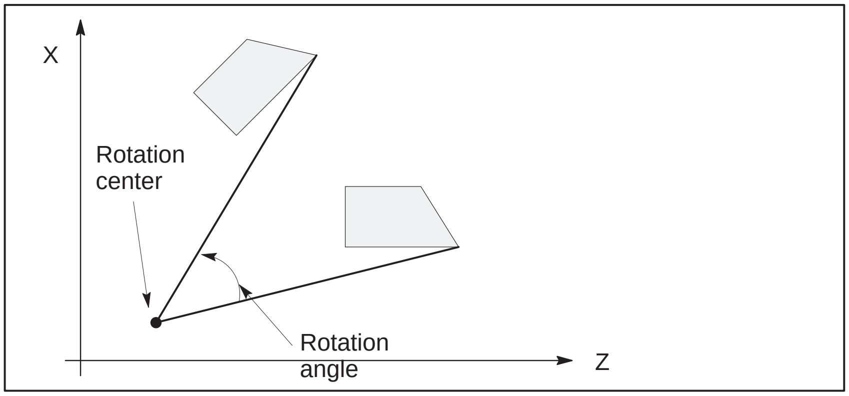
1 When there is no T code command before G36 or G37, P/S alarm No.81 is generated.
2 When a T code is specified in the same block as G36 or G37, P/S alarm No.82 is generated.
3 Measurement speed(Fp), γ, and ε are set as parameters (Fp : No.6241, γ : No.6251, ε : No.6254) by machine tool builder. ε must be positive numbers so that $\gamma > \varepsilon$.
4 Cancel the tool nose radius compensation before G36, G37.
5 When a manual movement is inserted into a movement at a measurement feedrate, return the tool to the position before the inserted manual movement for restart.
6 When using the optional tool nose radius compensation function, the tool offset amount is determined considering the value of tool nose R. Make sure that tool nose radius value is set correctly.
Example) When the tool nose center coincides with the start point.



The tool actually moves from point A to point B, but the tool offset value is determined assuming that the tool moves to point C considering the tool nose radius value.

15.7 COORDINATE ROTATION (G68.1, G69.1)

With the coordinate rotation function, it is possible to rotate a figure specified in a program. For example, a program that produces patterns of a figure rotated at increasingly larger angles can be created as a pair of subprograms, one of which defines a figure, the other of which calls the figure definition subprogram by specifying rotation. This method is useful for reducing the program development time and the size of the program.



Format

G17 / G18 / G19 G68.1 α_ β_ R_ ; —— Starts rotating the coordinates

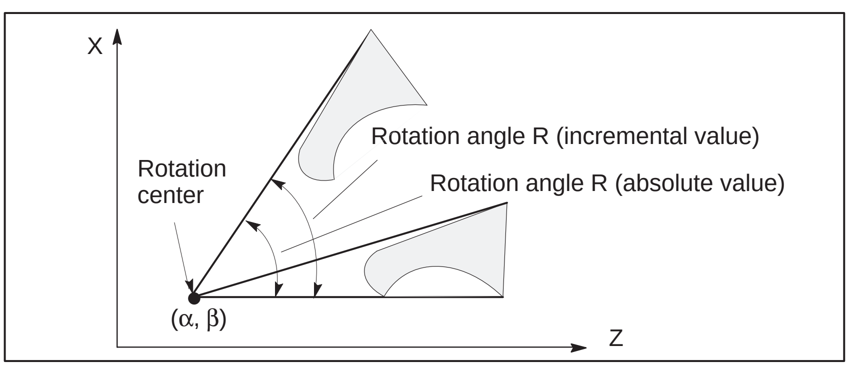
⋮ —— Coordinate rotation mode (the coordinates are rotated)

G69.1 ; —— Cancels coordinate rotation

G17 (G18 or G19) :
Selects a plane where the figure to be rotated is

α, β :
Specify two coordinates (from among X, Y, and Z) of the rotation center that match G17, G18, and G19. The values specified as the coordinates of the rotation center must always be absolute values.

R:
Specifies the rotation angle as an absolute value. Counterclockwise rotation is assumed to be positive. However, setting bit 0 (RIN) of parameter No. 5400 enables the use of an incremental value.
Incremental units of the angle: 0.001 degrees
Specifiable range: –360,000 to +360,000



Explanations

- **Plane selection G code, G17, G18, or G19**

Plane selection G code (G17, G18, or G19) can be specified in a block ahead of the coordinate rotation G code (G68.1). Do not specify G17, G18, or G19 in coordinate rotation mode.

- **Rotation center**

If the rotation center (α_, β_) is not specified, the location of the tool when G68.1 is issued is assumed as the rotation center.

- **Rotation angle command**

If the rotation angle command (R_) is not specified, the value specified in parameter No. 5410 is used as the rotation angle.

- **Coordinate rotation cancel**

The coordinate rotation cancel G code (G69.1) can be specified in the same block as other commands.

- **Tool compensation**

Tool compensation, such as tool offset or tool nose radius compensation, is processed after coordinate rotation is performed for a program defining a figure.
G68.1 can be used in either G00 or G01 mode.

Limitations

- **Reference position return**

A reference position return command G27, G28, G29, or G30 can be issued only in G69.1 mode.

- **Changes to coordinates**

Do not attempt to change coordinates in G68.1 mode (commands such as G50, G54 to G59, and the tool offset command).

- **Canned cycles**

Coordinate rotation cannot be used in simple canned cycles, multiple repetitive canned cycles, or canned drilling cycles.

- **Incremental command**

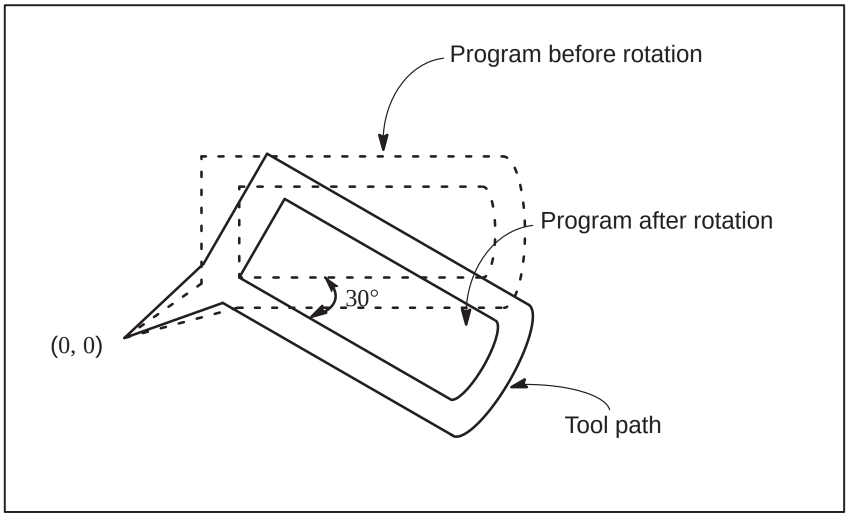
Always use absolute values in a move command that immediately follows the coordinate rotation command (G68.1) or coordinate rotation cancel command (G69.1). Specifying an incremental value results in the move command failing to operate normally.

Examples

- **Tool nose radius and coordinate rotation**

G68.1 and G69.1 can be specified during tool nose radius compensation, provided that the coordinate rotation plane coincides with the tool nose radius compensation plane.

N1 G50 X0 Z0 G69.1 G01 ;
N2 G42 X1000 Z1000 F1000 T0101 ;
N3 G68 R–30000 ;
N4 Z3000 ;
N5 G03 U1000 R1000 ;
N6 G01 Z1000 ;
N7 U–1000 ;
N8 G69.1 G40 X0 Z0 ;



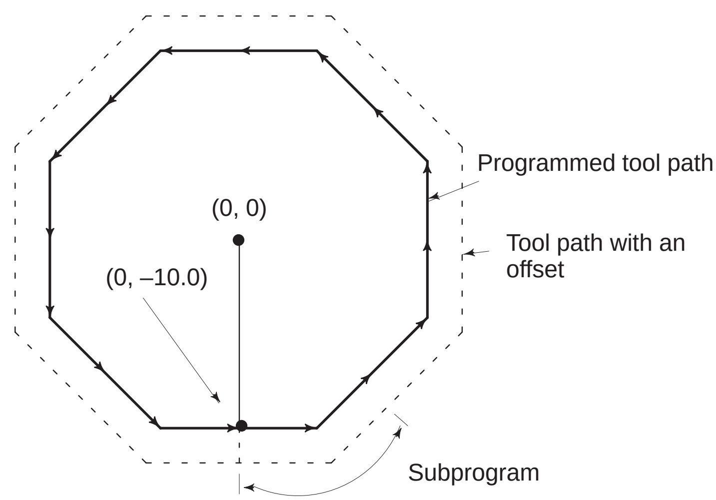
- **Repetitive coordinate rotation**

Coordinate rotation can be repeated by calling a registered subprogram more than once, but with increasingly greater rotation angles.

Set bit 0 (RIN) of parameter No. 5400 to 1 to specify the rotation angle as being incremental. (G code A, radius programming along the X–axis)

G50 X0 Z0 G18 ;
G01 F200 T0101 ;
M98 P2100 ;
M98 P2200 L7 ;
G00 X0 Z0 M30 ;

O2200 ;
G68.1 X0 Z0 R45.0 ;
G90 M98 P2100 ;
M99 ;
O2100 ;
G01 G42 X–10.0 Z0 ;
X–10.0 Z4.142 ;
X–7.071 Z7.071 ;
G40 M99 ;



16 CUSTOM MACRO

Although subprograms are useful for repeating the same operation, the custom macro function also allows use of variables, arithmetic and logic operations, and conditional branches for easy development of general programs such as pocketing and user–defined canned cycles. A machining program can call a custom macro with a simple command, just like a subprogram.

Machining program

```
O0001 ;
   :
   :
   :
G65 P9010 R50.0 L2 ;
   :
   :
M30 ;
```

Custom macro

```
O9010 ;
#1=#18/2 ;
G01 X#1 Z#1 F0.3 ;
G02 X#1 Z–#1 R#1 ;
   :
   :
   :
M99 ;
```

16.1 VARIABLES

An ordinary machining program specifies a G code and the travel distance directly with a numeric value; examples are G100 and X100.0.
With a custom macro, numeric values can be specified directly or using a variable number. When a variable number is used, the variable value can be changed by a program or using operations on the MDI panel.

```
#1=#2+100 ;

G01 X#1 F0.3 ;
```

Explanation

- **Variable representation**

When specifying a variable, specify a number sign (#) followed by a variable number. Personal computers allow a name to be assigned to a variable, but this capability is not available for custom macros.

Example: #1

An expression can be used to specify a variable number. In such a case, the expression must be enclosed in brackets.

Example: #[#1+#2–12]

- **Types of variables**

Variables are classified into four types by variable number.

Table 16.1 Types of variables

Variable number	Type of variable	Function
#0	Always null	This variable is always null. No value can be assigned to this variable.
#1 – #33	Local variables	Local variables can only be used within a macro to hold data such as the results of operations. When the power is turned off, local variables are initialized to null. When a macro is called, arguments are assigned to local variables.
#100 – #149 (#199) #500 – #531 (#999)	Common variables	Common variables can be shared among different macro programs. When the power is turned off, variables #100 to #149 are initialized to null. Variables #500 to #531 hold data even when the power is turned off. As an option, common variables #150 to #199 and #532 to #999 are also available. However, when these values are using, the length of the tape that can be used for storage decreases by 8.5 m.
#1000 –	System variables	System variables are used to read and write a variety of NC data items such as the current position and tool compensation values.

Notes
Common variables #150 to #199 and #532 to #999 are optional.

• Range of variable values

Local and common variables can have value 0 or a value in the following ranges :
-10^{47} to -10^{-29}
0
$+10^{-29}$ to $+10^{47}$
If the result of calculation turns out to be invalid, an P/S alarm No. 111 is issued.

• Omission of the decimal point

When a variable value is defined in a program, the decimal point can be omitted.

Example:
When #1=123; is defined, the actual value of variable #1 is 123.000.

• Referencing variables

To reference the value of a variable in a program, specify a word address followed by the variable number. When an expression is used to specify a variable, enclose the expression in brackets.

Example: G01X[#1+#2]F#3;

A referenced variable value is automatically rounded according to the least input increment of the address.

Example:
When G00X#1; is executed on a 1/1000–mm CNC with 12.3456 assigned to variable #1, the actual command is interpreted as G00X12.346;.

To reverse the sign of a referenced variable value, prefix a minus sign (–) to #.

Example: G00X–#1;

When an undefined variable is referenced, the variable is ignored up to an address word.

Example:
When the value of variable #1 is 0, and the value of variable #2 is null, execution of G00X#1Z#2; results in G00X0;.

• Undefined variable

When the value of a variable is not defined, such a variable is referred to as a ”null” variable. Variable #0 is always a null variable. It cannot be written to, but it can be read.

(a) Quotation

When an undefined variable is quotated, the address itself is also ignored.

When #1 = < vacant >	When #1 = 0
G90 X100 Y#1 ↓ G90 X100	G90 X100 Y#1 ↓ G90 X100 Y0

(b) Operation

< vacant > is the same as 0 except when replaced by < vacant>

When #1 = < vacant >	When #1 = 0
#2 = #1 ↓ #2 = < vacant >	#2 = #1 ↓ #2 = 0
#2 = #1*5 ↓ #2 = 0	#2 = #1*5 ↓ #2 = 0
#2 = #1+#1 ↓ #2 = 0	#2 = #1 + #1 ↓ #2 = 0

(c) Conditional expressions

< vacant > differs from 0 only for EQ and NE.

When #1 = < vacant >	When #1 = 0
#1 EQ #0 ↓ Established	#1 EQ #0 ↓ Not established
#1 NE 0 ↓ Established	#1 NE 0 ↓ Not established
#1 GE #0 ↓ Established	#1 GE #0 ↓ Established
#1 GT 0 ↓ Not established	#1 GT 0 ↓ Not established

- **Custom macro variables common to tool posts (two–path control)**

With the two–path control, macro variables are provided for each tool post. Specifying parameter Nos.6036 and 6037 allows some of the common variables to be used for all tool posts.

- **Displaying variable values**

```
VARIABLE                                O1234 N12345
  NO.        DATA         NO.        DATA
  100        123.456      108
  101          0.000      109
  102                     110
  103                     111
  104                     112
  105                     113
  106                     114
  107                     115

ACTUAL POSITION (RELATIVE)
    X        0.000          Y       0.000
    Z        0.000          B       0.000

MEM **** *** ***         18:42:15

[ MACRO ] [ MENU ] [  OPR  ] [      ] [ (OPRT) ]
```

· When the value of a variable is blank, the variable is null.

· The mark ******** indicates an overflow (when the absolute value of a variable is greater than 99999999) or an underflow (when the absolute value of a variable is less than 0.0000001).

Limitations

Program numbers, sequence numbers, and optional block skip numbers cannot be referenced using variables.

Example:

Variables cannot be used in the following ways:
O#1;
/#2G00X100.0;
N#3Z200.0;

16.2 SYSTEM VARIABLES

System variables can be used to read and write internal NC data such as tool compensation values and current position data. Note, however, that some system variables can only be read. System variables are essential for automation and general–purpose program development.

Explanations

- **Interface signals**

Signals can be exchanged between the programmable machine controller (PMC) and custom macros.

Table 16.2(a) System variables for interface signals

Variable number	Function
#1000–#1015 #1032	A 16–bit signal can be sent from the PMC to a custom macro. Variables #1000 to #1015 are used to read a signal bit by bit. Variable #1032 is used to read all 16 bits of a signal at one time.
#1100–#1115 #1132	A 16–bit signal can be sent from a custom macro to the PMC. Variables #1100 to #1115 are used to write a signal bit by bit. Variable #1132 is used to write all 16 bits of a signal at one time.
#1133	Variable #1133 is used to write all 32 bits of a signal at one time from a custom macro to the PMC. Note, that values from –99999999 to +99999999 can be used for #1133.

For detailed information, refer to the connection manual (B–62443E–1).

- **Tool compensation values**

When the system does not differentiate tool geometry compensation from tool wear compensation, use variable numbers for wear compensation.

Table 16.2(b) System variables for tool compensation memory C

Compensation number	X axis compensation value		Z axis compensation value		Tool nose radius compensation value		Imaginary tool nose position T	Y axis compensation value	
	Wear	Geometry	Wear	Geometry	Wear	Geometry		Wear	Geometry
1 : 49 : 64	#2001 : : : #2064	#2701 : #2749	#2101 : : : #2164	#2801 : #2849	#2201 : : : #2264	#2901 : : : #2964	#2301 : : : #2364	#2401 : #2449	#2451 : #2499

Table 16.2(c) System variables for 99 tool compensation values

Compensation number	X axis compensation value		Z axis compensation value		Tool nose radius compensation value		Imaginary tool nose position T	Y axis compensation value	
	Wear	Geometry	Wear	Geometry	Wear	Geometry		Wear	Geometry
1 : : 99	#10001 : : #10099	#15001 : : #15099	#11001 : : #11099	#12001 : : #12099	#12001 : : #12099	#17001 : : #17099	#13001 : : #13099	#14001 : : #14099	#19001 : : #19099

Note
System variables #2001 to #2964 can also be used to determine Y–axis wear or geometry compensation values No. 1 to 49, X–axis or Z–axis geometry compensation values No. 1 to 49, and other compensation values No. 1 to 64.

- **Macro alarms**

Table 16.2(d) System variable for macro alarms

Variable number	Function
#3000	When a value from 0 to 200 is assigned to variable #3000, the CNC stops with an alarm. After an expression, an alarm message not longer than 26 characters can be described. The CRT screen displays alarm numbers by adding 3000 to the value in variable #3000 along with an alarm message.

Example:
#3000=1(TOOL NOT FOUND);
→ The alarm screen displays "3001 TOOL NOT FOUND."

- **Time information**

Time information can be read and written.

Table 16.2(e) System variables for time information

Variable number	Function
#3001	This variable functions as a timer that counts in 1–millisecond increments at all times. When the power is turned on, the value of this variable is reset to 0. When 2147483648 milliseconds is reached, the value of this timer returns to 0.
#3002	This variable functions as a timer that counts in 1–hour increments when the cycle start lamp is on. This timer preserves its value even when the power is turned off. When 9544.371767 hours is reached, the value of this timer returns to 0.
#3011	This variable can be used to read the current date (year/month/day). Year/month/day information is converted to an apparent decimal number. For example, March 28, 1993 is represented as 19930328.
#3012	This variable can be used to read the current time (hours/minutes/seconds). Hours/minutes/seconds information is converted to an apparent decimal number. For example, 34 minutes and 56 seconds after 3 p.m. is represented as 153456.

Automatic operation control

The control state of automatic operation can be changed.

Table 16.2(e) System variable (#3003) for automatic operation control

#3003	Single block	Completion of an auxiliary function
0	Enabled	To be awaited
1	Disabled	To be awaited
2	Enabled	Not to be awaited
3	Disabled	Not to be awaited

· When the power is turned on, the value of this variable is 0.

· When single block stop is disabled, single block stop operation is not performed even if the single block switch is set to ON.

· When a wait for the completion of auxiliary functions (M, S, and T functions) is not specified, program execution proceeds to the next block before completion of auxiliary functions. Also, distribution completion signal DEN is not output.

Table 16.2(f) System variable (#3004) for automatic operation control

#3004	Feed hold	Feedrate Override	Exact stop
0	Enabled	Enabled	Enabled
1	Disabled	Enabled	Enabled
2	Enabled	Disabled	Enabled
3	Disabled	Disabled	Enabled
4	Enabled	Enabled	Disabled
5	Disabled	Enabled	Disabled
6	Enabled	Disabled	Disabled
7	Disabled	Disabled	Disabled

· When the power is turned on, the value of this variable is 0.

· When feed hold is disabled:

(1) When the feed hold button is held down, the machine stops in the single block stop mode. However, single block stop operation is not performed when the single block mode is disabled with variable #3003.

(2) When the feed hold button is pressed then released, the feed hold lamp comes on, but the machine does not stop; program execution continues and the machine stops at the first block where feed hold is enabled.

· When feedrate override is disabled, an override of 100% is always applied regardless of the setting of the feedrate override switch on the machine operator’s panel.

· When exact stop check is disabled, no exact stop check (position check) is made even in blocks including those which do not perform cutting.

- **Settings**

Settings can be read and written. Binary values are converted to decimals.

#3005

	#15	#14	#13	#12	#11	#10	#9	#8
Setting							FCV	

	#7	#6	#5	#4	#3	#2	#1	#0
Setting			SEQ			INI	ISO	TVC

#9 (FCV) : Whether to use the FS15 tape format conversion capability
#5 (SEQ) : Whether to automatically insert sequence numbers
#2 (INI) : Millimeter input or inch input
#1 (ISO) : Whether to use EIA or ISO as the output code
#0 (TVC) : Whether to make a TV check

- **Mirror image**

The mirror–image status for each axis set using an external switch or setting operation can be read through the output signal (mirror–image check signal). The mirror–image status present at that time can be checked. (See Section 4.7 in III.)
The value obtained in binary is converted into decimal notation.

#3007

	#7	#6	#5	#4	#3	#2	#1	#0
Setting	8th axis	7th axis	6th axis	5th axis	4th axis	3th axis	2th axis	1th axis

For each bit, 0 (mirror–image function is disabled) or 1 (mirror–image function is enabled) is indicated.

Example : If #3007 is 3, the mirror–image function is enabled for the first and second axes.

· When the mirror–image function is set for a certain axis by both the mirror–image signal and setting, the signal value and setting value are ORed and then output.

· When mirror–image signals for axes other than the controlled axes are turned on, they are still read into system variable #3007.

· System variable #3007 is a write–protected system variable. If an attempt is made to write data in the variable, P/S 116 alarm ”WRITE PROTECTED VARIABLE” is issued.

- **Number of machined parts**

The number (target number) of parts required and the number (completion number) of machined parts can be read and written.

Table 16.2(g) System variables for the number of parts required and the number of machined parts

Variable number	Function
#3901	Number of machined parts (completion number)
#3902	Number of required parts (target number)

Notes
Do not substitute a negative value.

- **Modal information**

Modal information specified in blocks up to the immediately preceding block can be read.

Table 16.2(h) System variables for modal information

Variable number	Function	
#4001	G00, G01, G02, G03, G33, G34	(Group 01)
#4002	G96, G97	(Group 02)
#4003		(Group 03)
#4004	G68, G69	(Group 04)
#4005	G98, G99	(Group 05)
#4006	G20, G21	(Group 06)
#4007	G40, G41, G42	(Group 07)
#4008	G25, G26	(Group 08)
#4009	G22, G23	(Group 09)
#4010	G80 – G89	(Group 10)
#4011		(Group 11)
#4012	G66, G67	(Group 12)
#4014	G54–G59	(Group 14)
#4015		(Group 15)
#4016	G17 – G19	(Group 16)
:	:	:
#4022		(Group 22)
#4109	F code	
#4113	M code	
#4114	Sequence number	
#4115	Program number	
#4119	S code	
#4120	T code	

Example:
When #1=#4001; is executed, the resulting value in #1 is 0, 1, 2, 3, or 33.

When a modal information reading system variable corresponding to a G code group which cannot be used is specified, a P/S alarm is issued.

- **Current position**

Position information cannot be written but can be read.

Table 16.2(i) System variables for position information

Variable number	Position information	Coordinate system	Tool compensation value	Read operation during movement
#5001–#5008	Block end point	Workpiece coordinate system	Not included	Enabled
#5021–#5028	Current position	Machine coordinate system	Included	Disabled
#5041–#5048	Current position	Workpiece coordinate system		
#5061–#5068	Skip signal position			Enabled

Table 16.2(i) System variables for position information

Variable number	Position information	Coordinate system	Tool compensation value	Read operation during movement
#5081, #5082	Tool offset value			Disabled
#5101–#5108	Deviated servo position			

· The first digit (from 1 to 8) represents an axis number.

· The tool offset value currently used for execution rather than the immediately preceding tool offset value is held in variables #5081 to 5088.

· The tool position where the skip signal is turned on in a G31 (skip function) block is held in variables #5061 to #5068. When the skip signal is not turned on in a G31 block, the end point of the specified block is held in these variables.

· When read during movement is ”disabled,” this means that expected values cannot be read due to the buffering (preread) function.

- **Workpiece coordinate system compensation values (workpiece zero point offset values)**

Workpiece zero point offset values can be read and written.

Table 16.2(j) System variables for workpiece zero point offset values

Variable number	Function
#5201 : #5208	First–axis external workpiece zero point offset value : Eighth–axis external workpiece zero point offset value
#5221 : #5228	First–axis G54 workpiece zero point offset value : Eighth–axis G54 workpiece zero point offset value
#5241 : #5248	First–axis G55 workpiece zero point offset value : Eighth–axis G55 workpiece zero point offset value
#5261 : #5268	First–axis G56 workpiece zero point offset value : Eighth–axis G56 workpiece zero point offset value
#5281 : #5288	First–axis G57 workpiece zero point offset value : Eighth–axis G57 workpiece zero point offset value
#5301 : #5308	First–axis G58 workpiece zero point offset value : Eighth–axis G58 workpiece zero point offset value
#5321 : #5328	First–axis G59 workpiece zero point offset value : Eighth–axis G59 workpiece zero point offset value

Notes

To use variables #5201 to #5328, the workpiece coordinate system option is necessary.

16.3 ARITHMETIC AND LOGIC OPERATION

The operations listed in Table 16.3(a) can be performed on variables. The expression to the right of the operator can contain constants and/or variables combined by a function or operator. Variables #j and #K in an expression can be replaced with a constant. Variables on the left can also be replaced with an expression.

Table 16.3(a) Arithmetic and logic operation

Function	Format	Remarks
Definition	#i=#j	
Sum Difference Product Quotient	#i=#j+#k; #i=#j–#k; #i=#j*#k; #i=#j/#k;	
Sine Cosine Tangent Arctangent	#i=SIN[#j]; #i=COS[#j]; #i=TAN[#j]; #i=ATAN[#j]/[#k];	An angle is specified in degrees. 90 degrees and 30 minutes is represented as 90.5 degrees.
Square root Absolute value Rounding off Rounding down Rounding up	#i=SQRT[#j]; #i=ABS[#j]; #i=ROUND[#j]; #i=FIX[#j]; #i=FUP[#j];	
OR XOR AND	#i=#j OR #k; #i=#j XOR #k; #i=#j AND #k;	A logical operation is performed on binary numbers bit by bit.
Conversion from BCD to BIN Conversion from BIN to BCD	#i=BIN[#j]; #i=BCD[#j];	Used for signal exchange to and from the PMC

Explanations

• Angle units

The units of angles used with the SIN, COS, TAN, and ATAN functions are degrees. For example, 90 degrees and 30 minutes is represented as 90.5 degrees.

• ATAN function

After the ATAN function, specify the lengths of two sides separated by a slash. A result is found where $0 \leqq \text{result} < 360$.

Example :
When #1=ATAN[1]/[–1], the value of #1 is 135.0

• ROUND function

· When the ROUND function is included in an arithmetic or logic operation command, IF statement, or WHILE statement, the ROUND function rounds off at the first decimal place.

Example:
When #1=ROUND[#2]; is executed where #2 holds 1.2345, the value of variable #1 is 1.0.

·When the ROUND function is used in NC statement addresses, the ROUND function rounds off the specified value according to the least input increment of the address.

Example:

Creation of a drilling program that cuts according to the values of variables #1 and #2, then returns to the original position
Suppose that the increment system is 1/1000 mm, variable #1 holds 1.2345, and variable #2 holds 2.3456. Then,
G00 G91 X–#1; Moves 1.235 mm.
G01 X–#2 F300; Moves 2.346 mm.
G00 X[#1+#2];
Since 1.2345 + 2.3456 = 3.5801, the travel distance is 3.580, which
does not return the tool to the original position.
This difference comes from whether addition is performed before or after rounding off. G00X–[ROUND[#1]+ROUND[#2]] must be specified to return the tool to the original position.

- **Rounding up and down to an integer**

With CNC, when the absolute value of the integer produced by an operation on a number is greater than the absolute value of the original number, such an operation is referred to as rounding up to an integer. Conversely, when the absolute value of the integer produced by an operation on a number is less than the absolute value of the original number, such an operation is referred to as rounding down to an integer. Be particularly careful when handling negative numbers.

Example:

Suppose that #1=1.2 and #2=–1.2.
When #3=FUP[#1] is executed, 2.0 is assigned to #3.
When #3=FIX[#1] is executed, 1.0 is assigned to #3.
When #3=FUP[#2] is executed, –2.0 is assigned to #3.
When #3=FIX[#2] is executed, –1.0 is assigned to #3.

- **Abbreviations of arithmetic and logic operation commands**

When a function is specified in a program, the first two characters of the function name can be used to specify the function.

Example:

ROUND → RO
FIX → FI

- **Priority of operations**

(1) Functions
(2) Operations such as multiplication and division (*, /, AND, MOD)
(3) Operations such as addition and subtraction (+, –, OR, XOR)

Example) $\#1=\underbrace{\#2+\underbrace{\#3*\underbrace{SIN[\#4]}_{(1)}}_{(2)}}_{(3)};$

(1), (2), and (3) indicate the order of operations.

- **Bracket nesting**

Brackets are used to change the order of operations. Brackets can be used to a depth of five levels including the brackets used to enclose a function. When a depth of five levels is exceeded, alarm No. 118 occurs.

```
Example) #1=SIN [ [ [#2+#3] *#4 +#5] *#6] ;
                    (1)
                      (2)
                       (3)
                          (4)
                        (5)
                        (1) to (5) indicate the order of operations.
```

Limitations

- **Brackets**

Brackets ([,]) are used to enclose an expression. Note that parentheses are used for comments.

- **Operation error**

Errors may occur when operations are performed.

Table 16.3(b) Errors involved in operations

Operation	Average error	Maximum error	Type of error
a = b*c	1.55×10^{-10}	4.66×10^{-10}	Relative error(*1) $\left\lvert \frac{\epsilon}{b} \right\rvert$
a = b / c	4.66×10^{-10}	1.88×10^{-9}	
a = $\sqrt{b}$	1.24×10^{-9}	3.73×10^{-9}	
a = b + c a = b – c	2.33×10^{-10}	5.32×10^{-10}	(*2) Min $\left\lvert \frac{\epsilon}{b} \right\rvert \left\lvert \frac{\epsilon}{c} \right\rvert$
a = SIN [b] a = COS [b]	5.0×10^{-9}	1.0×10^{-8}	Absolute error(*3) $\lvert \varepsilon \rvert$ degrees
a = ATAN [b] / [c] (*4)	1.8×10^{-6}	3.6×10^{-6}	

Notes

1. The relative error depends on the result of the operation.
2. Smaller of the two types of errors is used.
3. The absolute error is constant, regardless of the result of the operation.
4. Function TAN performs SIN/COS.

- The precision of variable values is about 8 decimal digits. When very large numbers are handled in an addition or subtraction, the expected results may not be obtained.

Example:

When an attempt is made to assign the following values to variables #1 and #2:
#1=9876543210123.456
#2=9876543277777.777
the values of the variables become:
#1=9876543200000.000
#2=9876543300000.000
In this case, when #3=#2–#1; is calculated, #3=100000.000 results. (The actual result of this calculation is slightly different because it is performed in binary.)

- Also be aware of errors that can result from conditional expressions using EQ, NE, GE, GT, LE, and LT.

Example:

IF[#1 EQ #2] is effected by errors in both #1 and #2, possibly resulting in an incorrect decision.
Therefore, instead find the difference between the two variables with IF[ABS[#1–#2]LT0.001].
Then, assume that the values of the two variables are equal when the difference does not exceed an allowable limit (0.001 in this case).

- Also, be careful when rounding down a value.

Example:

When #2=#1*1000; is calculated where #1=0.002;, the resulting value of variable #2 is not exactly 2 but 1.99999997.
Here, when #3=FIX[#2]; is specified, the resulting value of variable #1 is not 2.0 but 1.0. In this case, round down the value after correcting the error so that the result is greater than the expected number, or round it off as follows:
#3=FIX[#2+0.001]
#3=ROUND[#2]

● Divisor

When a divisor of zero is specified in a division or TAN[90], alarm No. 112 occurs.

16.4 MACRO STATEMENTS AND NC STATEMENTS

The following blocks are referred to as macro statements:

- **Blocks containing an arithmetic or logic operation (=)**
- **Blocks containing a control statement (such as GOTO, DO, END)**
- **Blocks containing a macro call command (such as macro calls by G65, G66, G67, or other G codes, or by M codes)**

Any block other than a macro statement is referred to as an NC statement.

Explanations

- **Differences from NC statements**

- Even when single block mode is on, the machine does not stop. Note, however, that the machine stops in the single block mode when bit 5 (SBM) of parameter 6000 is 1.
- Macro blocks are not regarded as blocks that involve no movement in the tool nose radius compensation mode (see Section II–16.7).

- **NC statements that have the same property as macro statements**

- If a block contains a subprogram call command (M98, a subprogram call using an M code, or a subprogram call using a T code) and does not contain any command address other than O, N, P, or L, that block is equivalent to a macro statement.
- If a block contains M99 and does not contain any command address other than O, N, P, or L, that block is equivalent to a macro statement.

16.5 BRANCH AND REPETITION

In a program, the flow of control can be changed using the GOTO statement and IF statement. Three types of branch and repetition operations are used:

Branch and repetition
- GOTO statement (unconditional branch)
- IF statement (conditional branch: if ..., then...)
- WHILE statement (repetition while ...)

16.5.1 Unconditional Branch (GOTO Statement)

A branch to sequence number n occurs. When a sequence number outside of the range 1 to 99999 is specified, P/S alarm No. 128 occurs. A sequence number can also be specified using an expression.

GOTO n ; n: Sequence number (1 to 99999)

Example:

GOTO1;

GOTO#10;

16.5.2 Conditional Branch (IF Statement)

Specify a conditional expression after IF. If the specified conditional expression is satisfied, a branch to sequence number n occurs. If the specified condition is not satisfied, the next block is executed.

If the value of variable #1 is greater than 10, a branch to sequence number N2 occurs.

```
If the condition      IF [#1 GT 10] GOTO 2 ;
is not satisfied
                         Processing          If the condition is satisfied
                  N2 G00 G91 X10.0 ;
                         :
```

Explanations

- **Conditional expression**

A conditional expression must include an operator inserted between two variables or between a variable and constant, and must be enclosed in brackets ([,]). An expression can be used instead of a variable.

• Operators

Operators each consist of two letters and are used to compare two values to determine whether they are equal or one value is smaller or greater than the other value. Note that the inequality sign cannot be used.

Table 16.5.2 Operators

Operator	Meaning
EQ	Equal to(=)
NE	Not equal to(≠)
GT	Greater than(>)
GE	Greater than or equal to(≧)
LT	Less than(<)
LE	Less than or equal to(≦)

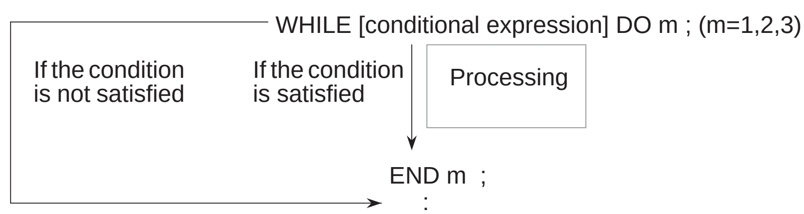
Sample program

The sample program below finds the total of numbers 1 to 10.

```
O9500;
 #1=0;Initial value of the variable to hold the sum
 #2=1;Initial value of the variable as an addend
N1 IF[#2 GT 10] GOTO 2;   Branch to N2 when the addend is greater than
10
 #1=#1+#2;  Calculation to find the sum
 #2=#2+1; Next addend
 GOTO 1;  Branch to N1
N2 M30;End of program
```

16.5.3 Repetition (While Statement)

Specify a conditional expression after WHILE. While the specified condition is satisfied, the program from DO to END is executed. If the specified condition is not satisfied, program execution proceeds to the block after END.

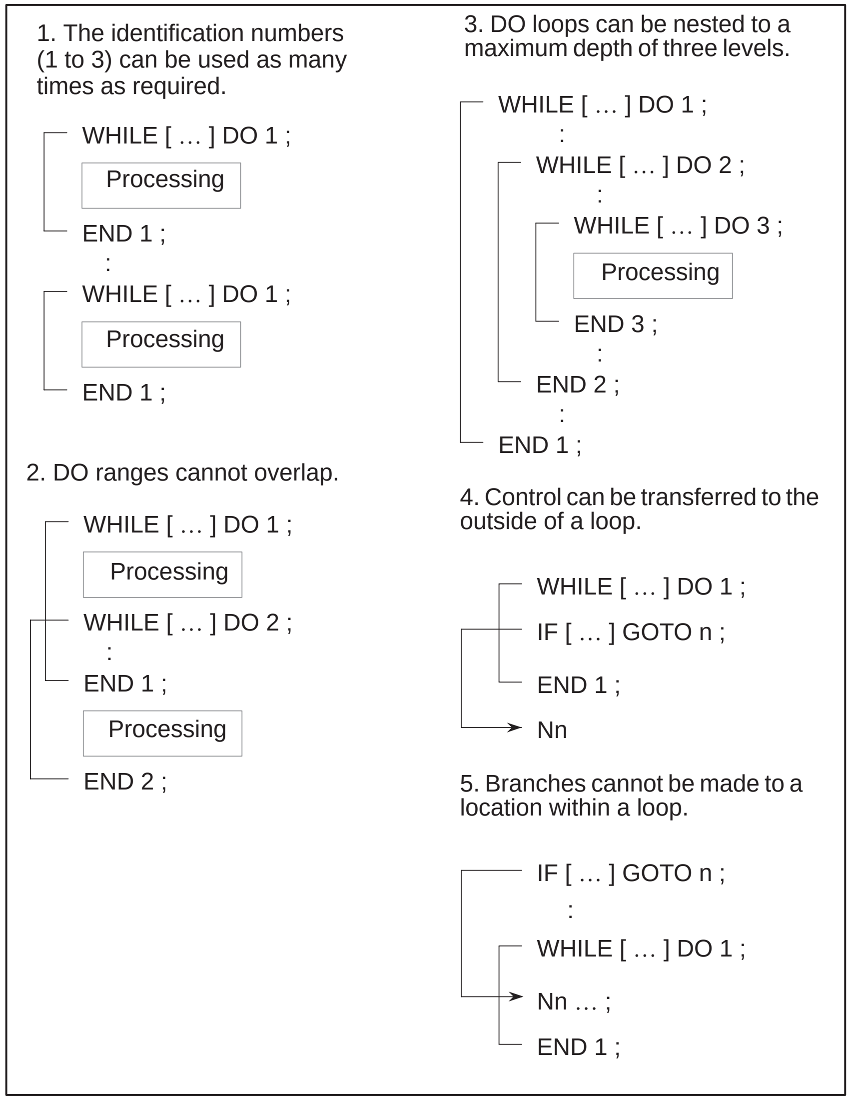


Explanations

While the specified condition is satisfied, the program from DO to END after WHILE is executed. If the specified condition is not satisfied, program execution proceeds to the block after END. The same format as for the IF statement applies. A number after DO and a number after END are identification numbers for specifying the range of execution. The numbers 1, 2, and 3 can be used. When a number other than 1, 2, and 3 is used, P/S alarm No. 126 occurs.

- **Nesting**

The identification numbers (1 to 3) in a DO–END loop can be used as many times as desired. Note, however, when a program includes crossing repetition loops (overlapped DO ranges), P/S alarm No. 124 occurs.



Limitations

- **Infinite loops**

When DO m is specified without specifying the WHILE statement, an infinite loop ranging from DO to END is produced.

- **Processing time**

When a branch to the sequence number specified in a GOTO statement occurs, the sequence number is searched for. For this reason, processing in the reverse direction takes a longer time than processing in the forward direction. Using the WHILE statement for repetition reduces processing time.

- **Undefined variable**

In a conditional expression that uses EQ or NE, a null value and zero have different effects. In other types of conditional expressions, a null value is regarded as zero.

Sample program

The sample program below finds the total of numbers 1 to 10.

```
O0001;
#1=0;
#2=1;
WHILE[#2 LE 10]DO
1;
#1=#1+#2;
#2=#2+1;
END 1;
M30;
```

16.6 MACRO CALL

A macro program can be called using the following methods:

Macro call
- Simple call ((G65)
- modal call (G66, G67)
- Macro call with G code
- Macro call with M code
- Subprogram call with M code
- Subprogram call with T code

Restrictions

- **Differences between macro calls and subprogram calls**

Macro call (G65) differs from subprogram call (M98) as described below.

· With G65, an argument (data passed to a macro) can be specified. M98 does not have this capability.

· When an M98 block contains another NC command (for example, G01 X100.0 M98Pp), the subprogram is called after the command is executed. On the other hand, G65 unconditionally calls a macro.

· When an M98 block contains another NC command (for example, G01 X100.0 M98Pp), the machine stops in the single block mode. On the other hand, G65 does not stops the machine.

· With G65, the level of local variables changes. With M98, the level of local variables does not change.

16.6.1 Simple Call (G65)

When G65 is specified, the custom macro specified at address P is called. Data (argument) can be passed to the custom macro program.

G65 P p L ℓ <argument–specification> ;

P	:	Number of the program to call
ℓ	:	Repetition count (1 by default)
Argument	:	Data passed to the macro

```
O0001 ;
   :
G65 P9010 L2 A1.0 B2.0 ;
   :
M30 ;
```

```
O9010 ;
#3=#1+#2 ;
IF [#3 GT 360] GOTO 9 ;
G00 X#3 ;
N9 M99 ;
```

Explanations

- **Call**

· After G65, specify at address P the program number of the custom macro to call.

· When a number of repetitions is required, specify a number from 1 to 9999 after address L. When L is omitted, 1 is assumed.

· By using argument specification, values are assigned to corresponding local variables.

- **Argument specification**

Two types of argument specification are available. Argument specification I uses letters other than G, L, O, N, and P once each. Argument specification II uses A, B, and C once each and also uses I, J, and K up to ten times. The type of argument specification is determined automatically according to the letters used.

Argument specification I

Address	Variable number
A	#1
B	#2
C	#3
D	#7
E	#8
F	#9
H	#11

Address	Variable number
I	#4
J	#5
K	#6
M	#13
Q	#17
R	#18
S	#19

Address	Variable number
T	#20
U	#21
V	#22
W	#23
X	#24
Y	#25
Z	#26

· Addresses G, L, N, O, and P cannot be used in arguments.

· Addresses that need not be specified can be omitted. Local variables corresponding to an omitted address are set to null.

Argument specification II

Argument specification II uses A, B, and C once each and uses I, J, and K up to ten times. Argument specification II is used to pass values such as three–dimensional coordinates as arguments.

Address	Variable number
A	#1
B	#2
C	#3
I_1	#4
J_1	#5
K_1	#6
I_2	#7
J_2	#8
K_2	#9
I_3	#10
J_3	#11

Address	Variable number
K_3	#12
I_4	#13
J_4	#14
K_4	#15
I_5	#16
J_5	#17
K_5	#18
I_6	#19
J_6	#20
K_6	#21
I_7	#22

Address	Variable number
J_7	#23
K_7	#24
I_8	#25
J_8	#26
K_8	#27
I_9	#28
J_9	#29
K_9	#30
I_{10}	#31
J_{10}	#32
K_{10}	#33

· Subscripts of I, J, and K for indicating the order of argument specification are not written in the actual program.

Restrictions

- **Format**

G65 must be specified before any argument.

- **Mixture of argument specifications I and II**

The CNC internally identifies argument specification I and argument specification II. If a mixture of argument specification I and argument specification II is specified, the type of argument specification specified later takes precedence.

- **Position of the decimal point**

The units used for argument data passed without a decimal point correspond to the least input increment of each address. The value of an argument passed without a decimal point may vary according to the system configuration of the machine. It is good practice to use decimal points in macro call arguments to maintain program compatibility.

- **Call nesting**

Calls can be nested to a depth of four levels including simple calls (G65) and modal calls (G66). This does not include subprogram calls (M98).

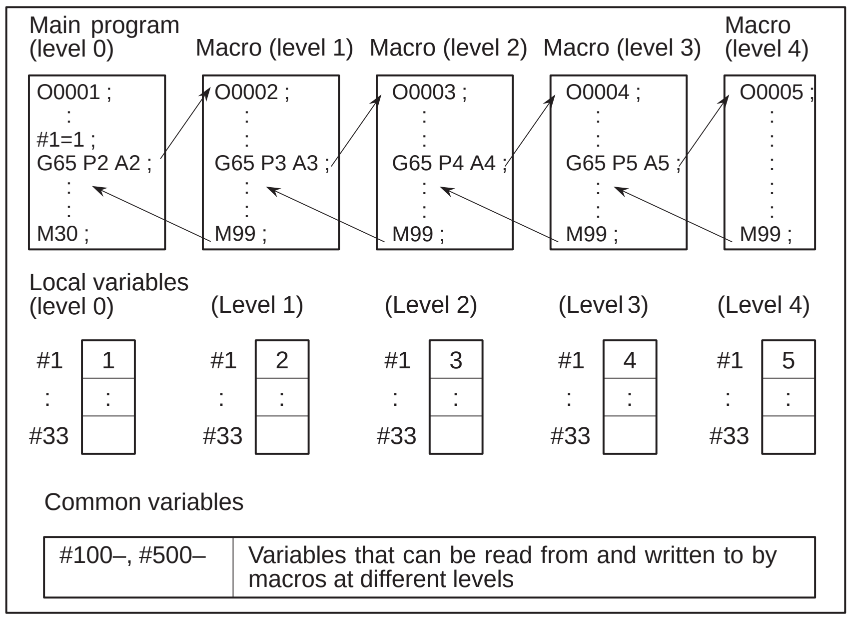
- **Local variable levels**

· Local variables from level 0 to 4 are provided for nesting.

· The level of the main program is 0.

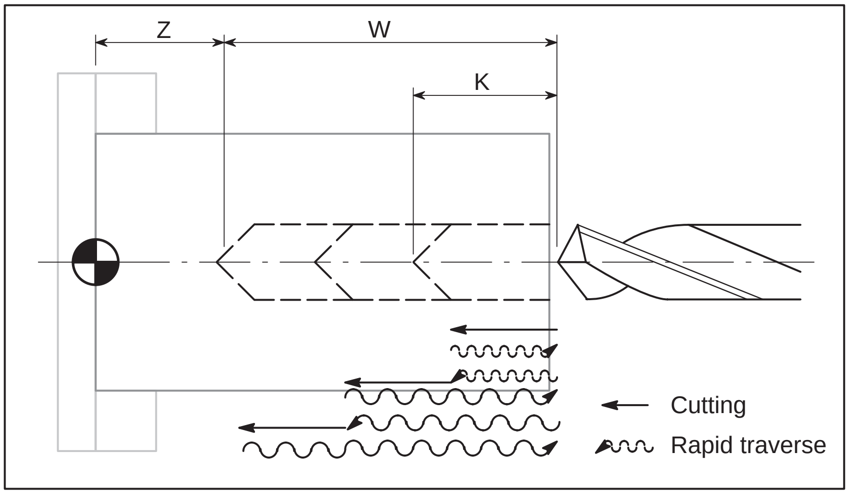
· Each time a macro is called (with G65 or G66), the local variable level is incremented by one. The values of the local variables at the previous level are saved in the CNC.

· When M99 is executed in a macro program, control returns to the calling program. At that time, the local variable level is decremented by one; the values of the local variables saved when the macro was called are restored.



Sample program (Drill cycle)

Move the tool beforehand along the X– and Z–axes to the position where a drilling cycle starts. Specify Z or W for the depth of a hole, K for the depth of a cut, and F for the cutting feedrate to drill the hole.



- **Calling format**

$$\text{G65 P9100}\left\{\begin{matrix}\text{Zz}\\\text{Ww}\end{matrix}\right\}\text{Kk Ff ;}$$

Z : Hole depth (absolute specification)
U: Hole depth (incremental specification)
K: Cutting amount per cycle
F : Cutting feedrate

- **Program calling a macro program**

```
O0002;
G50 X100.0 Z200.0 ;
G00 X0 Z102.0 S1000 M03 ;
G65 P9100 Z50.0 K20.0 F0.3 ;
G00 X100.0 Z200.0 M05 ;
M30 ;
```

- **Macro program (called program)**

```
  O9100;
  #1=0 ; ....................... Clear the data for the depth of the current hole.
  #2=0 ; ....................... Clear the data for the depth of the preceding
hole.
  IF [#23 NE #0] GOTO 1 ; ..... If incremental programming, specifies the
                                jump to N1.
  IF [#26 EQ #0] GOTO 8 ; ..... If neither Z nor W is specified, an error
occurs.
  #23=#5002−#26 ; ........... Calculates the depth of a hole.
N1  #1=#1+#6 ; ............... Calculates the depth of the current hole.
  IF [#1 LE #23] GOTO 2 ; ...... Determines whether the hole to be cut is
    too deep.?
  #1=#23 ; .................... Clamps at the depth of the current hole.
N2  G00 W−#2 ; ............... Moves the tool to the depth of the preceding
                                hole at the cutting feedrate.
  G01 W− [#1−#2] F#9 ; ....... Drills the hole.
  G00 W#1 ; ................... Moves the tool to the drilling start point.
  IF [#1 GE #23] GOTO 9 ; ..... Checks whether drilling is completed.
  #2=#1 ; ..................... Stores the depth of the current hole.
  GOTO 1 ;
N9  M99 ;
N8  #3000=1 (NOT Z OR U COMMAND)
```

16.6.2 Modal Call (G66)

Once G66 is issued to specify a modal call a macro is called after a block specifying movement along axes is executed. This continues until G67 is issued to cancel a modal call.

```
G66  P p  L  ℓ  <argument–specification> ;
   P          : Number of the program to call
   ℓ          : Repetition count (1 by default)
   Argument   : Data passed to the macro
```

```
O0001 ;
   :
G66 P9100 L2 A1.0 B2.0 ;
G00 G90 X100.0 ;
X125.0 ;
X150.0 ;
G67 ;
   :
M30 ;
```

```
O9100 ;
   :
G00 Z–#1 ;
G01 Z–#2 F0.3 ;
   :
   :
   :
   :
M99 ;
```

Explanations

● Call

- After G66, specify at address P a program number subject to a modal call.
- When a number of repetitions is required, a number from 1 to 9999 can be specified at address L.
- As with a simple call (G65), data passed to a macro program is specified in arguments.

● Cancellation

When a G67 code is specified, modal macro calls are no longer performed in subsequent blocks.

● Call nesting

Calls can be nested to a depth of four levels including simple calls (G65) and modal calls (G66). This does not include subprogram calls (M98).

● Modal call nesting

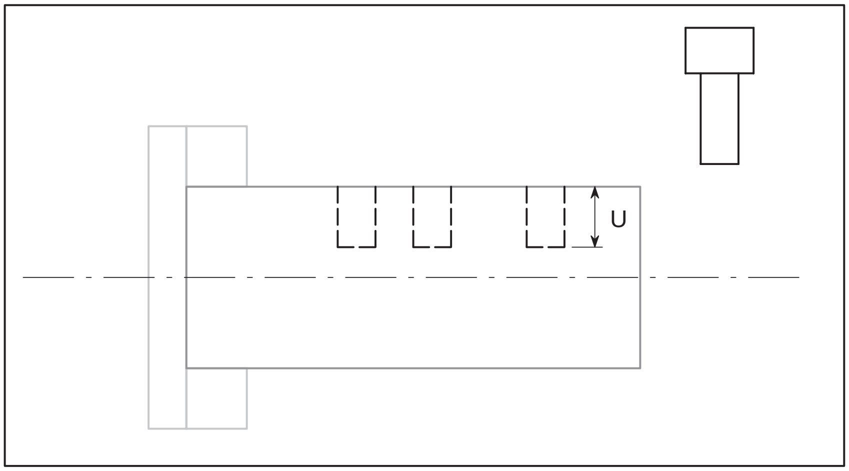
Modal calls can be nested by specifying another G66 code during a modal call.

Restrictions

- In a G66 block, no macros can be called.
- G66 needs to be specified before any arguments.
- No macros can be called in a block which contains a code such as a miscellaneous function that does not involve movement along an axis.
- Local variables (arguments) can only be set in G66 blocks. Note that local variables are not set each time a modal call is performed.

Sample program

This program makes a groove at a specified position.



- **Calling format**

```
G66 P9110 Uu Ff ;
```

U: Groove depth (incremental specification)
F : Cutting feed of grooving

- **Program that calls a macro program**

O0003 ;
G50 X100.0 Z200.0 ;
S1000 M03 ;
G66 P9110 U5.0 F0.5 ;
G00 X60.0 Z80.0 ;
Z50.0 ;
Z30.0 ;
G67 ;
G00 X00.0 Z200.0 M05 ;
M30;

- **Macro program (program called)**

O9110 ;
G01 U–#21 F#9 ; . . . . Cuts the workpiece.
G00 U#21 ; . . . . Retracts the tool.
M99 ;

16.6.3 Macro Call Using G Code

By setting a G code number used to call a macro program in a parameter, the macro program can be called in the same way as for a simple call (G65).

```
O0001 ;                        O9010 ;
    :                             :
G81 X10.0 Z–10.0 ;                :
    :                             :
M30 ;                          N9 M99 ;
Parameter No.6050 = 81
```

Explanations

By setting a G code number from 1 to 9999 used to call a custom macro program (9010 to 9019) in the corresponding parameter (Nos. 6050 to 6059), the macro program can be called in the same way as with G65.
For example, when a parameter is set so that macro program O9010 can be called with G81, a user–specific cycle created using a custom macro can be called without modifying the machining program.

- **Correspondence between parameter numbers and program numbers**

Program number	Parameter number
O9010	6050
O9011	6051
O9012	6052
O9013	6053
O9014	6054
O9015	6055
O9016	6056
O9017	6057
O9018	6058
O9019	6059

- **Repetition**

As with a simple call, a number of repetitions from 1 to 9999 can be specified at address L.

- **Argument specification**

As with a simple call, two types of argument specification are available: Argument specification I and argument specification II. The type of argument specification is determined automatically according to the addresses used.

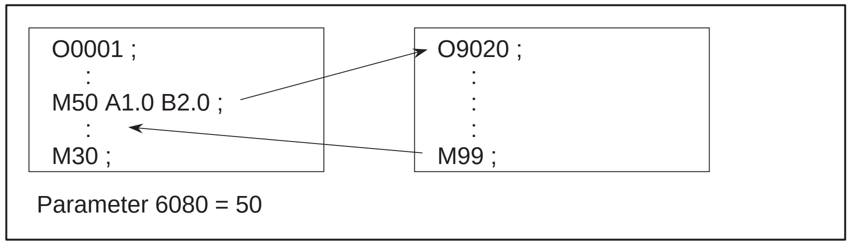
Restrictions

- **Nesting of calls using G codes**

In a program called with a G code, no macros can be called using a G code. A G code in such a program is treated as an ordinary G code. In a program called as a subprogram with an M or T code, no macros can be called using a G code. A G code in such a program is also treated as an ordinary G code.

16.6.4 Macro Call Using an M Code

By setting an M code number used to call a macro program in a parameter, the macro program can be called in the same way as with a simple call (G65).



Explanations

By setting an M code number from 1 to 99999999 used to call a custom macro program (O9020 to O9029) in the corresponding parameter (Nos. 6080 to 6089), the macro program can be called in the same way as with G65.

- **Correspondence between parameter numbers and program numbers**

Program number	Parameter number
O9020	6080
O9021	6081
O9022	6082
O9023	6083
O9024	6084
O9025	6085
O9026	6086
O9027	6087
O9028	6088
O9029	6089

- **Repetition**

As with a simple call, a number of repetitions from 1 to 9999 can be specified at address L.

- **Argument specification**

As with a simple call, two types of argument specification are available: Argument specification I and argument specification II. The type of argument specification is determined automatically according to the addresses used.

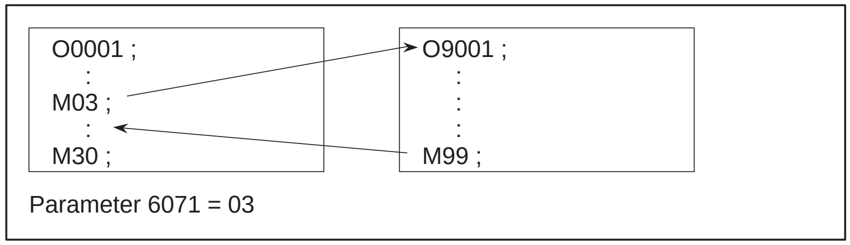
Restrictions

– An M code used to call a macro program must be specified at the start of a block.

– In a macro called with a G code or in a program called as a subprogram with an M or T code, no macros can be called using an M code. An M code in such a macro or program is treated as an ordinary M code.

16.6.5 Subprogram Call Using an M Code

By setting an M code number used to call a subprogram (macro program) in a parameter, the macro program can be called in the same way as with a subprogram call (M98).



Explanations

By setting an M code number from 1 to 99999999 used to call a subprogram in a parameter (Nos. 6071 to 6076), the corresponding custom macro program (O9001 to O9006) can be called in the same way as with M98.

- **Correspondence between parameter numbers and program numbers**

Program number	Parameter number
O9001	6071
O9002	6072
O9003	6073
O9004	6074
O9005	6075
O9006	6076
O9007	6077
O9008	6078
O9009	6079

- **Repetition**

As with a simple call, a number of repetitions from 1 to 9999 can be specified at address L.

- **Argument specification**

Argument specification is not allowed.

- **M code**

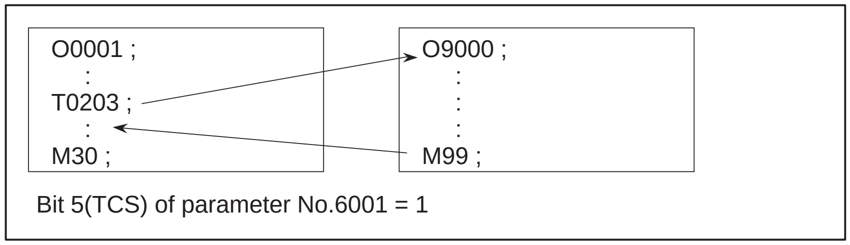
An M code in a macro program that has been called is treated as an ordinary M code.

Limitations

In a macro called with a G code or in a program called with an M or T code, no subprograms can be called using an M code. An M code in such a macro or program is treated as an ordinary M code.

16.6.6 Subprogram Calls Using a T Code

By enabling subprograms (macro program) to be called with a T code in a parameter, a macro program can be called each time the T code is specified in the machining program.



Explanations

- **Call**

By setting bit 5 (TCS) of parameter No.6001 to 1, the macro program O9000 can be called when a T code is specified in the machining program. A T code specified in a machining program is assigned to common variable #149.

Limitations

In a macro called with a G code or in a program called with an M or T code, no subprograms can be called using a T code. A T code in such a macro or program is treated as an ordinary T code.

16.6.7 Sample Program

By using the subprogram call function that uses M codes, the cumulative usage time of each tool is measured.

Conditions

- The cumulative usage time of each of tool numbers 1 to 5 is measured. The time is not measured for tools whose number is 6 or more.
- The following variables are used to store the tool numbers and measured times:

#501	Cumulative usage time of tool number 1
#502	Cumulative usage time of tool number 2
#503	Cumulative usage time of tool number 3
#504	Cumulative usage time of tool number 4
#505	Cumulative usage time of tool number 5

- Usage time starts being counted when the M03 command is specified and stops when M05 is specified. System variable #3002 is used to measure the time during which the cycle start lamp is on. The time during which the machine is stopped by feed hold and single block stop operation is not counted, but the time used to change tools and pallets is included.

Operation check

- **Parameter setting**

Set 3 in parameter No.6071, and set 05 in parameter No.6072.

- **Variable value setting**

Set 0 in variables #501 to #505.

- **Program that calls a macro program**

```
O0001;
T0100 M06;
M03;
  :
M05; . . . . . . . . . . . . . . . . . . . . . . . . . Changes #501.
T0200 M06;
M03;
  :
M05; . . . . . . . . . . . . . . . . . . . . . . . . . Changes #502.
T0300 M06;
M03;
  :
M05; . . . . . . . . . . . . . . . . . . . . . . . . . Changes #503.
T0400 M06;
M03;
  :
M05; . . . . . . . . . . . . . . . . . . . . . . . . . Changes #504.
T0500 M06;
M03;
  :
M05; . . . . . . . . . . . . . . . . . . . . . . . . . Changes #505.
M30;
```

Macro program (program called)

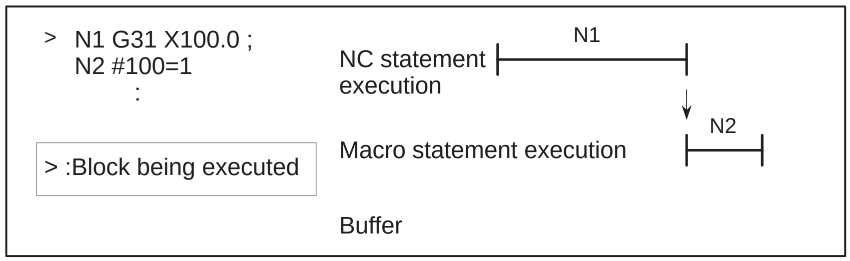
```
O9001(M03); ........................... Macro to start counting
  M01;
  IF[FIX[#4120/100] EQ 0]GOTO 9; ............. No tool specified
  IF[FIX[#4120/100] GT 5]GOTO 9; ...... Out–of–range tool number
  #3002=0; ................................. Clears the timer.
N9 M03; .............. Rotates the spindle in the forward direction.
  M99;

O9002(M05); ........................... Macro to end counting
  M01;
  IF[FIX[#4120/100] EQ 0]GOTO 9; ............. No tool specified
  IF[FIX[#4120/100] GT 5]GOTO 9; ...... Out–of–range tool number
  #[500+FIX[#4120/100]]=#3002+#[500+FIX[#4120/100]];
 .................................. Calculates cumulative time.

N9 M05; ................................... Stops the spindle.
  M99;
```

16.7 PROCESSING MACRO STATEMENTS

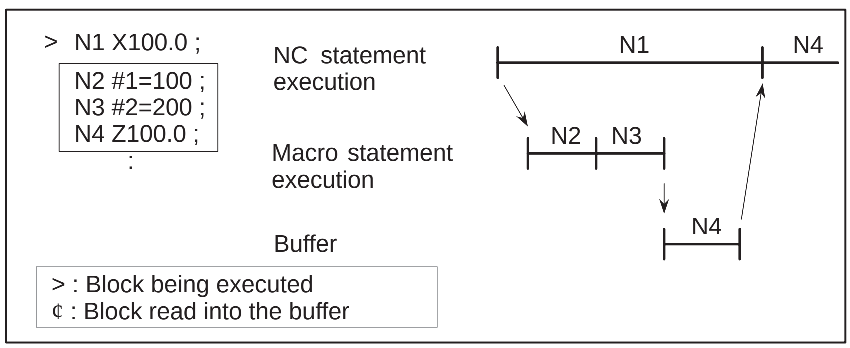
For smooth machining, the CNC prereads the CNC statement to be performed next. This operation is referred to as buffering. In tool nose radius compensation mode (G41, G42), the NC prereads NC statements two or three blocks ahead to find intersections. Macro statements for arithmetic expressions and conditional branches are processed as soon as they are read into the buffer. Blocks containing M00, M01, M02, or M30, blocks containing M codes for which buffering is suppressed by setting parameter(Nos.3411 to 3420), and blocks containing G31 are not preread.

Explanations

- **When the next block is not buffered (M codes that are not buffered, G31, etc.)**



- **Buffering the next block in other than tool nose radius compensation mode (G41, G42) (normally prereading one block)**

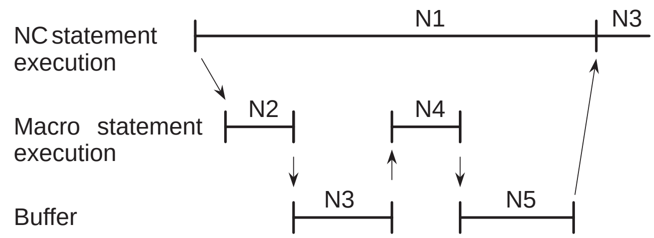


When N1 is being executed, the next NC statement (N4) is read into the buffer. The macro statements (N2, N3) between N1 and N4 are processed during execution of N1.

- **Buffering the next block in tool nose radius compensation mode (G41, G42)**

```
> N1 G01 G41 G91 Z100.0 F100 T0101 ;
  N2 #1=100 ;
  N3 X100.0 ;
  N4 #2=200 ;
  N5 Z50.0 ;
     :
```

> : Block being executed
¢ : Blocks read into the buffer

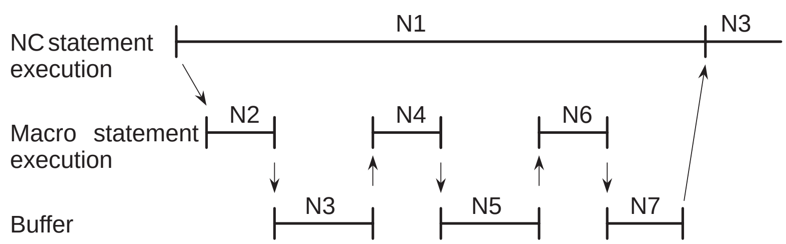


When N1 is being executed, the NC statements in the next two blocks (up to N5) are read into the buffer. The macro statements (N2, N4) between N1 and N5 are processed during execution of N1.

- **When the next block involves no movement in tool nose radius compensation C (G41, G42) mode**

```
> N1 G01 G41 X100.0 G100 T0101 ;
  N2 #1=100 ;
  N3 Z50.0 ;
  N4 #2=200 ;
  N5 M08 ;
  N6 #3=300 ;
  N7 X200.0 ;
     :
```

> : Block being executed
¢ : Blocks read into the buffer



When the NC1 block is being executed, the NC statements in the next two blocks (up to N5) are read into the buffer. Since N5 is a block that involves no movement, an intersection cannot be calculated. In this case, the NC statements in the next three blocks (up to N7) are read. The macro statements (N2, N4, and N6) between N1 and N7 are processed during execution of N1.

16.8 REGISTERING CUSTOM MACRO PROGRAMS

Custom macro programs are similar to subprograms. They can be registered and edited in the same way as subprograms. The storage capacity is determined by the total length of tape used to store both custom macros and subprograms.

16.9 LIMITATIONS

- **MDI operation**

The macro call command can be specified in MDI mode too. During automatic operation, however, it is impossible to switch to the MDI mode for a macro program call.

- **Sequence number search**

A custom macro program cannot be searched for a sequence number.

- **Single block**

Even while a macro program is being executed, blocks can be stopped in the single block mode (except blocks containing macro call commands, arithmetic operation commands, and control commands).
A block containing a macro call command (G65, G66, or G67) does not stop even when the single block mode is on. Blocks containing arithmetic operation commands and control commands can be stopped in single block mode by setting SBKM (bit 5 of parameter 6000) to 1.
Single block stop operation is used for testing custom macro programs. When SBKM (bit 5 of parameter 6000) is set to 1, a single block stop takes place at every macro statement.
Note that when a single block stop occurs at a macro statement in tool nose radius compensation mode, the statement is assumed to be a block that does not involve movement, and proper compensation cannot be performed in some cases. (Strictly speaking, the block is regarded as specifying a movement with a travel distance 0.)

- **Optional block skip**

A / appearing in the middle of an <expression> (enclosed in brackets [ ] on the right–hand side of an arithmetic expression) is regarded as a division operator; it is not regarded as the specifier for an optional block skip code.

- **Operation in EDIT mode**

Registered custom macro programs and subprograms should be protected from being destroyed by accident. By setting NE8 (bit 0 of parameter 3202) and NE9 (bit 4 of parameter 3202) to 1, deletion and editing are disabled for custom macro programs and subprograms with program numbers 8000 to 8999 and 9000 to 9999. When the entire memory is cleared (by pressing the RESET and DELETE keys at the same time to turn on the power), the contents of memory such as custom macro programs are deleted.

- **Reset**

With a reset operation, local variables and common variables #100 to #149 are cleared to null values. They can be prevented from being cleared by setting, CLV and CCV (bits 7 and 6 of parameter 6001). System variables #1000 to #1133 are not cleared.
A reset operation clears any called states of custom macro programs and subprograms, and any DO states, and returns control to the main program.

- **Display of the PROGRAM RESTART screen**

As with M98, the M and T codes used for subprogram calls are not displayed.

- **Feed hold**

When a feed hold is enabled during execution of a macro statement, the machine stops after execution of the macro statement. The machine also stops when a reset or alarm occurs.

- **Constant values that can be used in <expression>**

+0.0000001 to +99999999
–99999999 to –0.0000001
The number of significant digits is 8 (decimal). If this range is exceeded, P/S alarm No. 003 occurs.

16.10 EXTERNAL OUTPUT COMMANDS

In addition to the standard custom macro commands, the following macro commands are available. They are referred to as external output commands.

- **BPRNT**
- **DPRNT**
- **POPEN**
- **PCLOS**

These commands are provided to output variable values and characters through the reader/punch interface.

Explanations

Specify these commands in the following order:

Open command: POPEN
Before specifying a sequence of data output commands, specify this command to establish a connection to an external input/output device.

Data output command: BPRNT or DPRNT
Specify necessary data output.

Close command: PCLOS
When all data output commands have completed, specify PCLOS to release a connection to an external input/output device.

• Open command POPEN

POPEN
POPEN establishes a connection to an external input/output device. It must be specified before a sequence of data output commands. The CNC outputs a DC2 control code.

• Data output command BPRNT

BPRNT [a #b [c] …]

Number of significant decimal places
Variable
Character

The BPRNT command outputs characters and variable values in binary.

(i) Specified characters are converted to corresponding ISO codes according to the setting data (ISO) that is output at that time.
Specifiable characters are as follows:
- **Letters (A to Z)**
- **Numbers**
- **Special characters (*, /, +, –, etc.)**

An asterisk (*) is output by a space code.

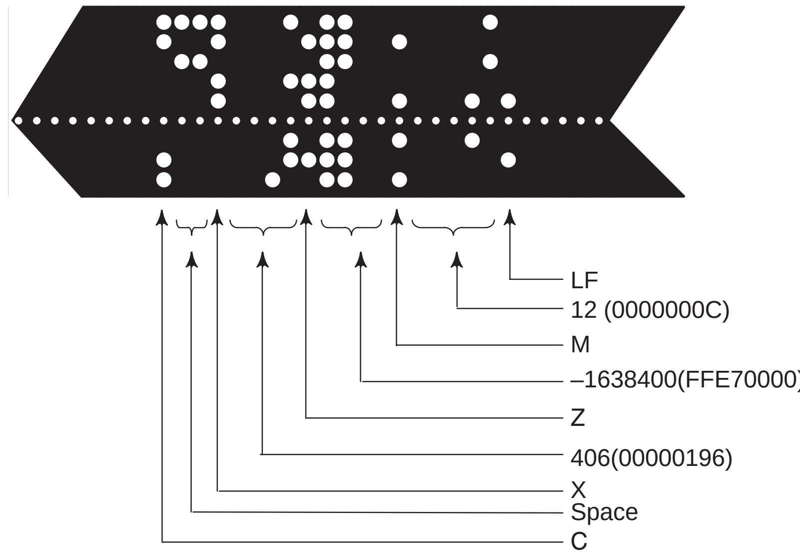
(ii) All variables are stored with a decimal point. Specify a variable followed by the number of significant decimal places enclosed in brackets. A variable value is treated as 2–word (32–bit) data, including the decimal digits. It is output as binary data starting from the highest byte.

(iii) When specified data has been output, an EOB code is output according to the ISO code settings.

(iv) Null variables are regarded as 0.

Example)

BPRINT [C X#100 [3] Z#101 [3] M#10 [0]]**
Variable value
#100=0.40596
#101=–1638.4
#10=12.34



- **Data output command DPRNT**

DPRNT [a #b [c d] …]

- a: Character
- #b: Variable
- c: Number of significant digits in the integer part
- d: Number of significant decimal places

The DPRNT command outputs characters and each digit in the value of a variable according to the code set in the settings (ISO).

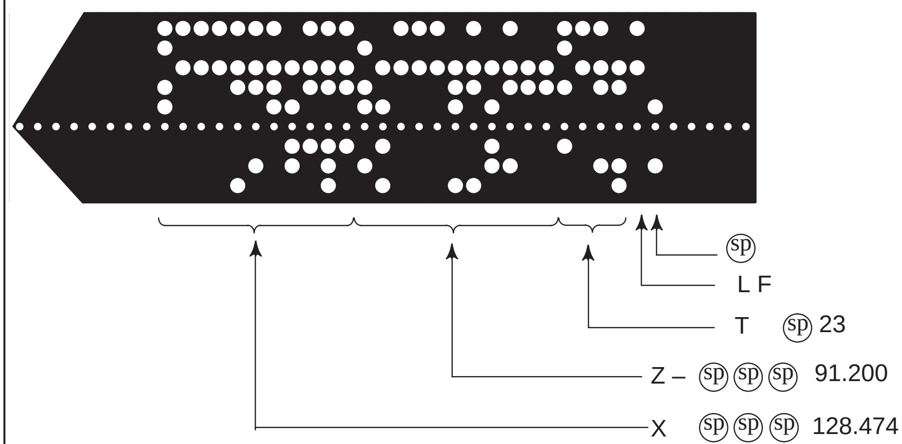
(i) For an explanation of the DPRNT command, see Items (i), (iii), and (iv) for the BPRNT command.

(ii) When outputting a variable, specify # followed by the variable number, then specify the number of digits in the integer part and the number of decimal places enclosed in brackets.
One code is output for each of the specified number of digits, starting with the highest digit. For each digit, a code is output according to the settings (ISO). The decimal point is also output using a code set in the settings (ISO).
Each variable must be a numeric value consisting of up to eight digits. When high–order digits are zeros, these zeros are not output if PRT (bit1 of parameter 6001) is 1. If PRT(bit 1 of parameter 6001) is 0, a space code is output each time a zero is encountered.
When the number of decimal places is not zero, digits in the decimal part are always output. If the number of decimal places is zero, no decimal point is output. When PRT (bit 1 of parameter 6001) is 0, a space code is output to indicate a positive number instead of +; if PRT(bit 1 of parameter 6001) is 1, no code is output.

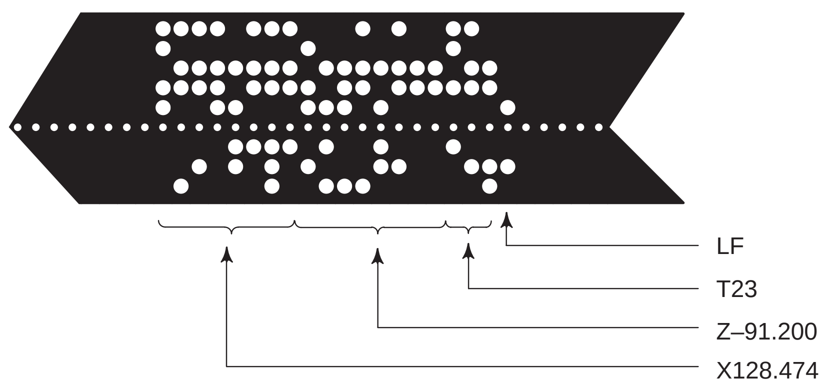
Example)

DPRNT [X#2 [53] Z#5 [53] T#30 [20]]
Variable value
#2=128.47398
#5=–91.2
#30=123.456

(1) Parameter PRT(No.6001#1)=0



(2) Parameter PRT(No.6001#1)=1



- **Close command PCLOS**

PCLOS ;
The PCLOS command releases a connection to an external input/output device. Specify this command when all data output commands have terminated. DC4 control code is output from the CNC.

- **Required setting**

Specify the channel use for parameter 020. According to the specification of this parameter, set data items (such as the baud rate) for the reader/punch interface.
I/O channel 0 : Parameters 101, 102 and 103
I/O channel 1 : Parameters 111, 112 and 113
I/O channel 2 : Parameters 121, 122 and 123
Never specify output to the Fanuc Cassette or floppy disks.)
When specifying a DPRNT command to output data, specify whether leading zeros are output as spaces (by setting PRT (bit 1 of parameter 6001) to 1 or 0). To indicate the end of a line of data in ISO code, specify whether to use only an LF (NCR, of bit 3 of parameter 0103 is 0) or an LF and CR (NCR is 1).

Notes

Notes

1 It is not necessary to always specify the open command (POPEN), data output command (BPRNT, DPRNT), and close command (PCLOS) together. Once an open command is specified at the beginning of a program, it does not need to be specified again except after a close command was specified.

2 Be sure to specify open commands and close commands in pairs. Specify the close command at the end of the program. However, do not specify a close command if no open command has been specified.

3 When a reset operation is performed while commands are being output by a data output command, output is stopped and subsequent data is erased. Therefore, when a reset operation is performed by a code such as M30 at the end of a program that performs data output, specify a close command at the end of the program so that processing such as M30 is not performed until all data is output.

4 Abbreviated macro words enclosed in brackets [ ] remains unchanged. However, note that when the characters in brackets are divided and input several times, the second and subsequent abbreviations are converted and input.

5 O can be specified in brackets [ ]. Note that when the characters in brackets [ ] are divided and input several times, O is omitted in the second and subsequent inputs.

16.11 INTERRUPTION TYPE CUSTOM MACRO

When a program is being executed, another program can be called by inputting an interrupt signal (UINT) from the machine. This function is referred to as an interruption type custom macro function. Program an interrupt command in the following format:

Format

M96 P○○○○ ;	Enables custom macro interrupt
M97 ;	Disables custom macro interrupt

Explanations

Use of the interruption type custom macro function allows the user to call a program during execution of an arbitrary block of another program. This allows programs to be operated to match situations which vary from time to time.

(1) When a tool abnormality is detected, processing to handle the abnormality is started by an external signal.

(2) A sequence of machining operations is interrupted by another machining operation without the cancellation of the current operation.

(3) At regular intervals, information on current machining is read.
Listed above are examples like adaptive control applications of the interruption type custom macro function.

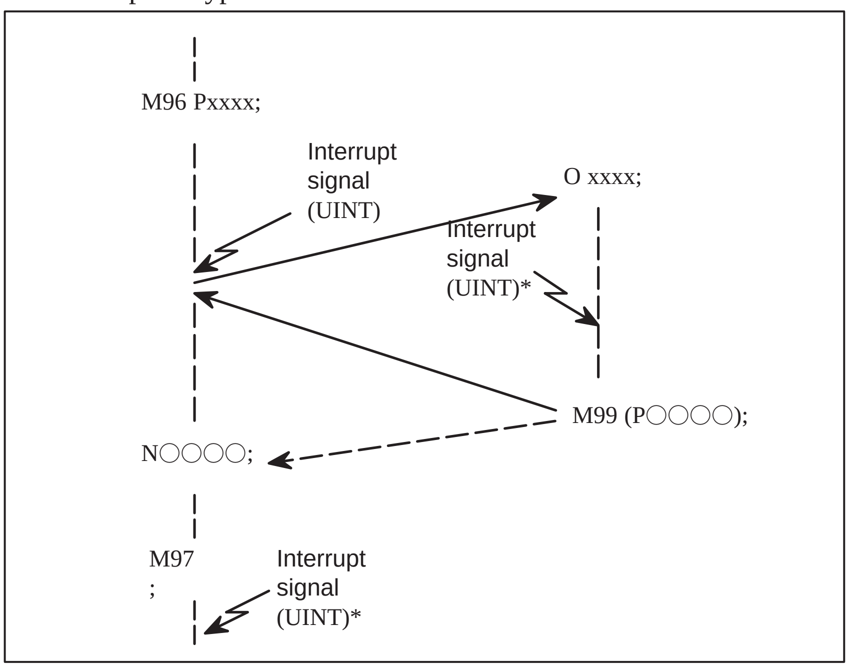


Fig 16.11 Interruption type custom macro function

When M96Pxxxx is specified in a program, subsequent program operation can be interrupted by an interrupt signal (UINT) input to execute the program specified by Pxxxx.
When the interrupt signal (UINT, marked by * in Fig. 16.11 is input during execution of the interrupt program or after M97 is specified, it is ignored.

16.11.1 Specification Method

Explanations

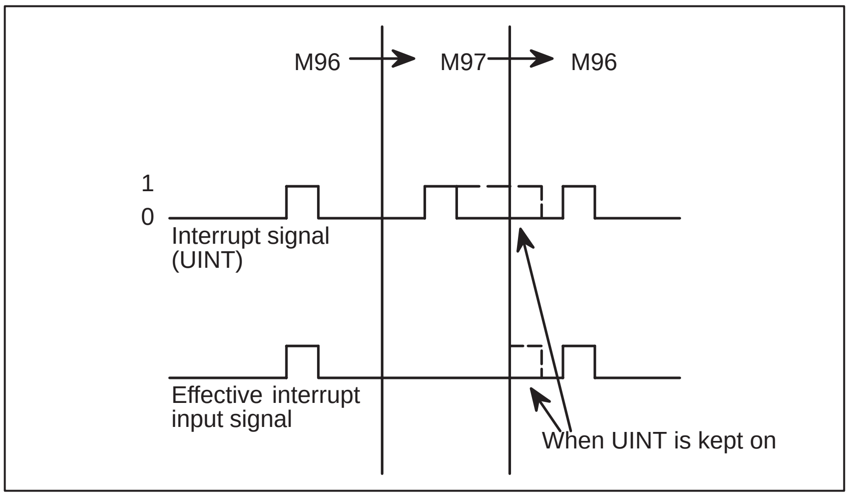
- **Interrupt conditions**

A custom macro interrupt is available only during program execution. It is enabled under the following conditions

– **When memory operation or MDI operation is selected**
– **When STL (start lamp) is on**
– **When a custom macro interrupt is not currently being processed**

- **Specification**

Generally, the custom macro interrupt function is used by specifying M96 to enable the interrupt signal (UINT) and M97 to disable the signal.
Once M96 is specified, a custom macro interrupt can be initiated by the input of the interrupt signal (UINT) until M97 is specified or the NC is reset. After M97 is specified or the NC is reset, no custom macro interrupts are initiated even when the interrupt signal (UINT) is input. The interrupt signal (UINT) is ignored until another M96 command is specified.



The interrupt signal (UINT) becomes valid after M96 is specified. Even when the signal is input in M97 mode, it is ignored. When the signal input in M97 mode is kept on until M96 is specified, a custom macro interrupt is initiated as soon as M96 is specified (only when the status–triggered scheme is employed); when the edge–triggered scheme is employed, the custom macro interrupt is not initiated even when M96 is specified.

Notes

Notes
For the status–triggered and edge–triggered schemes, see Item"Custom macro interrupt signal (UINT)" of Subsec. 16.11.2.

16.11.2 Details of Functions

Explanations

• ubprogram–type interrupt and macro–type interrupt

There are two types of custom macro interrupts: Subprogram–type interrupts and macro–type interrupts. The interrupt type used is selected by MSB (bit 5 of parameter 6003).

(a) Subprogram–type interrupt
An interrupt program is called as a subprogram. This means that the levels of local variables remain unchanged before and after the interrupt. This interrupt is not included in the nesting level of subprogram calls.

(b) Macro–type interrupt
An interrupt program is called as a custom macro. This means that the levels of local variables change before and after the interrupt. The interrupt is not included in the nesting level of custom macro calls. When a subprogram call or a custom macro call is performed within the interrupt program, this call is included in the nesting level of subprogram calls or custom macro calls. Arguments cannot be passed from the current program even when the custom macro interrupt is a macro–type interrupt.

• M codes for custom macro interrupt control

In general, custom macro interrupts are controlled by M96 and M97. However, these M codes, may already being used for other purposes (such as an M function or macro M code call) by some machine tool builders. For this reason, MPR (bit 4 of parameter 6003) is provided to set M codes for custom macro interrupt control.
When specifying this parameter to use the custom macro interrupt control M codes set by parameters, set parameters 6033 and 6034 as follows:
Set the M code to enable custom macro interrupts in parameter 6033, and set the M code to disable custom macro interrupts in parameter 6034.
When specifying that parameter–set M codes are not used, M96 and M97 are used as the custom macro control M codes regardless of the settings of parameters 6033 and 6034.
The M codes used for custom macro interrupt control are processed internally (they are not output to external units). However, in terms of program compatibility, it is undesirable to use M codes other than M96 and M97 to control custom macro interrupts.

• Custom macro interrupts and NC statements

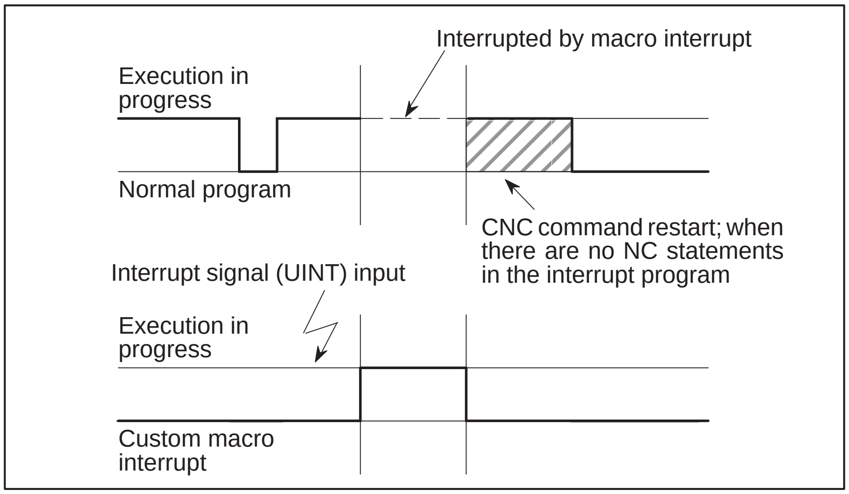
When performing a custom macro interrupt, the user may want to interrupt the NC statement being executed, or the user may not want to perform the interrupt until the execution of the current block is completed. MIN (bit 2 of parameter 6003)is used to select whether to perform interrupts even in the middle of a block or to wait until the end of the block.

- **Type I (when an interrupt is performed even in the middle of the block)**

(i) When the interrupt signal (UINT) is input, any movement or dwell being performed is stopped immediately and the interrupt program is executed.

(ii) If there are NC statements in the interrupt program, the command in the interrupted block is lost and the NC statement in the interrupt program is executed. When control is returned to the interrupted program, the program is restarted from the next block after the interrupted block.

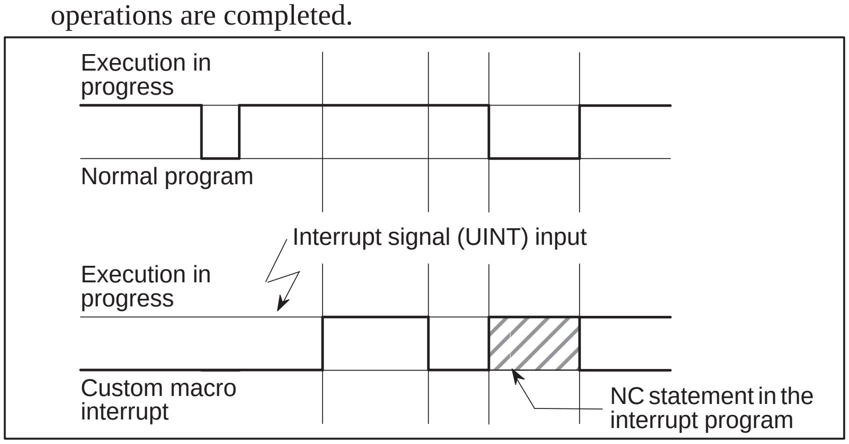
(iii)If there are no NC statements in the interrupt program, control is returned to the interrupted program by M99, then the program is restarted from the command in the interrupted block.



- **Type II (when an interrupt is performed at the end of the block)**

(i)If the block being executed is not a block that consists of several cycle operations such as a drilling canned cycle and automatic reference position return (G28), an interrupt is performed as follows:
When an interrupt signal (UINT) is input, macro statements in the interrupt program are executed immediately unless an NC statement is encountered in the interrupt program. NC statements are not executed until the current block is completed.

(ii) If the block being executed consists of several cycle operations, an interrupt is performed as follows:
When the last movement in the cycle operations is started, macro statements in the interrupt program are executed unless an NC statement is encountered. NC statements are executed after all cycle operations are completed.



- **Conditions for enabling and disabling the custom macro interrupt signal**

The interrupt signal becomes valid after execution starts of a block that contains M96 for enabling custom macro interrupts. The signal becomes invalid when execution starts of a block that contains M97.

While an interrupt program is being executed, the interrupt signal becomes invalid. The signal become valid when the execution of the block that immediately follows the interrupted block in the main program is started after control returns from the interrupt program. In type I, if the interrupt program consists of only macro statements, the interrupt signal becomes valid when execution of the interrupted block is started after control returns from the interrupt program.

- **Custom macro interrupt during execution of a block that involves cycle operation**

- **Type IFor type I**

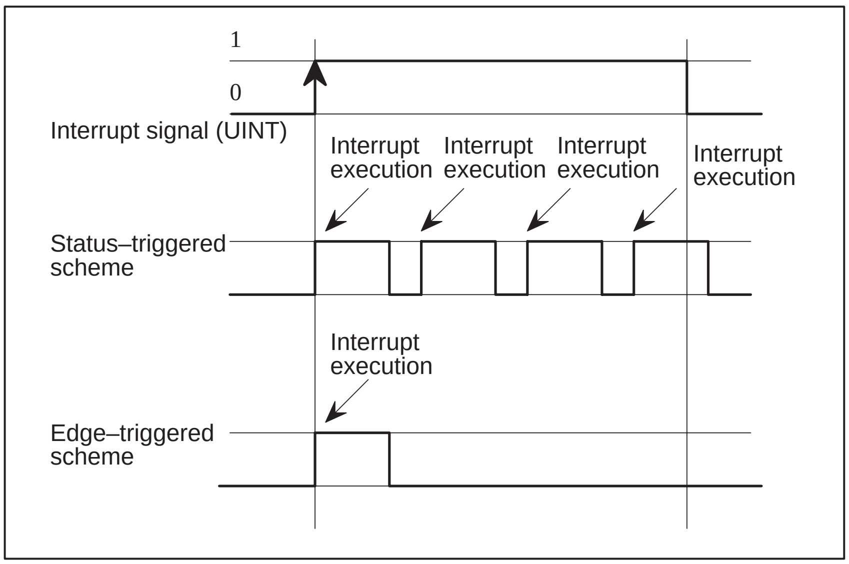
Even when cycle operation is in progress, movement is interrupted, and the interrupt program is executed. If the interrupt program contains no NC statements, the cycle operation is restarted after control is returned to the interrupted program. If there are NC statements, the remaining operations in the interrupted cycle are discarded, and the next block is executed.

- **Type IFor type II**

When the last movement of the cycle operation is started, macro statements in the interrupt program are executed unless an NC statement is encountered. NC statements are executed after cycle operation is completed.

- **Custom macro interrupt signal (UINT)**

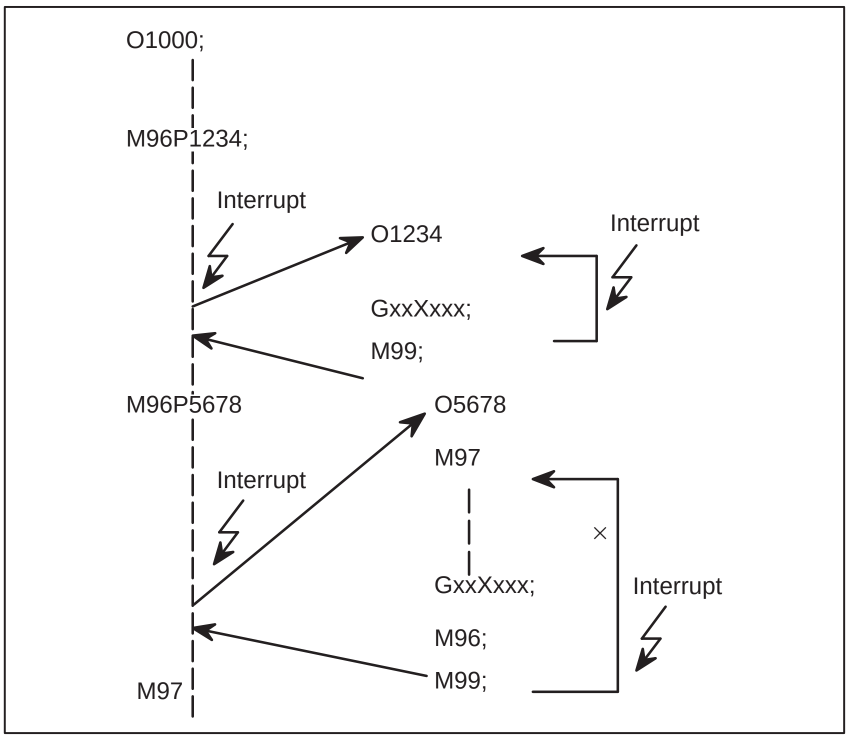
There are two schemes for custom macro interrupt signal (UINT) input: The status–triggered scheme and edge– triggered scheme. When the status–triggered scheme is used, the signal is valid when it is on. When the edge triggered scheme is used, the signal becomes valid on the rising edge when it switches from off to on status.
One of the two schemes is selected with TSE (bit 3 of parameter 6003). When the status–triggered scheme is selected by this parameter, a custom macro interrupt is generated if the interrupt signal (UINT) is on at the time the signal becomes valid. By keeping the interrupt signal (UINT) on, the interrupt program can be executed repeatedly.
When the edge–triggered scheme is selected, the interrupt signal (UINT) becomes valid only on its rising edge. Therefore, the interrupt program is executed only momentarily (in cases when the program consists of only macro statements). When the status–triggered scheme is inappropriate, or when a custom macro interrupt is to be performed just once for the entire program (in this case, the interrupt signal may be kept on), the edge–triggered scheme is useful.
Except for the specific applications mentioned above, use of either scheme results in the same effects. The time from signal input until a custom macro interrupt is executed does not vary between the two schemes.



In the above example, an interrupt is executed four times when the status triggered scheme is used; when the edge– triggered scheme is used, the interrupt is executed just once.

- **Return from a custom macro interrupt**

To return control from a custom macro interrupt to the interrupted program, specify M99. A sequence number in the interrupted program can also be specified using address P. If this is specified, the program is searched from the beginning for the specified sequence number. Control is returned to the first sequence number found.
When a custom macro interrupt program is being executed, no interrupts are generated. To enable another interrupt, execute M99. When M99 is specified alone, it is executed before the preceding commands terminate. Therefore, a custom macro interrupt is enabled for the last command of the interrupt program. If this is inconvenient, custom macro interrupts should be controlled by specifying M96 and M97 in the program.
When a custom macro interrupt is being executed, no other custom macro interrupts are generated; when an interrupt is generated, additional interrupts are inhibited automatically. Executing M99 makes it possible for another custom macro interrupt to occur. M99 specified alone in a block is executed before the previous block terminates. In the following example, an interrupt is enabled for the Gxx block of O1234. When the signal is input, O1234 is executed again. O5678 is controlled by M96 and M97. In this case, an interrupt is not enabled for O5678 (enabled after control is returned to O1000).



Notes

When an M99 block consists only of address O, N, P, L, or M, this block is regarded as belonging to the previous block in the program. Therefore, a single–block stop does not occur for this block. In terms of programming, the following (1) and (2) are basically the same. (The difference is whether G○○ is executed before M99 is recognized.)

(1) G○○ X○○○ ;
M99 ;

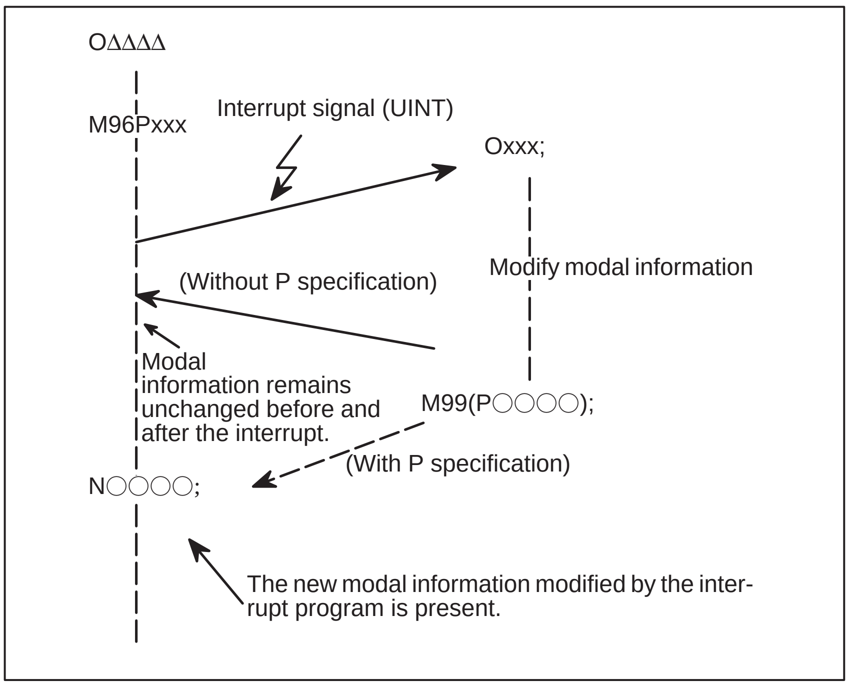
(2) G○○ X○○○ M99 ;

● Custom macro interrupt and modal information

A custom macro interrupt is different from a normal program call. It is initiated by an interrupt signal (UINT) during program execution. In general, any modifications of modal information made by the interrupt program should not affect the interrupted program.
For this reason, even when modal information is modified by the interrupt program, the modal information before the interrupt is restored when control is returned to the interrupted program by M99.
When control is returned from the interrupt program to the interrupted program by M99 Pxxxx, modal information can again be controlled by the program. In this case, the new continuous information modified by the interrupt program is passed to the interrupted program. Restoration of the old modal information present before the interrupt is not desirable. This is because after control is returned, some programs may operate differently depending on the modal information present before the interrupt. In this case, the following measures are applicable:

(1) The interrupt program provides modal information to be used after control is returned to the interrupted program.

(2) After control is returned to the interrupted program, modal information is specified again as necessary.



• Type I Modal information when control is returned by M99

The modal information present before the interrupt becomes valid. The new modal information modified by the interrupt program is made invalid.

• Type I Modal information when control is returned by M99 P○○○○

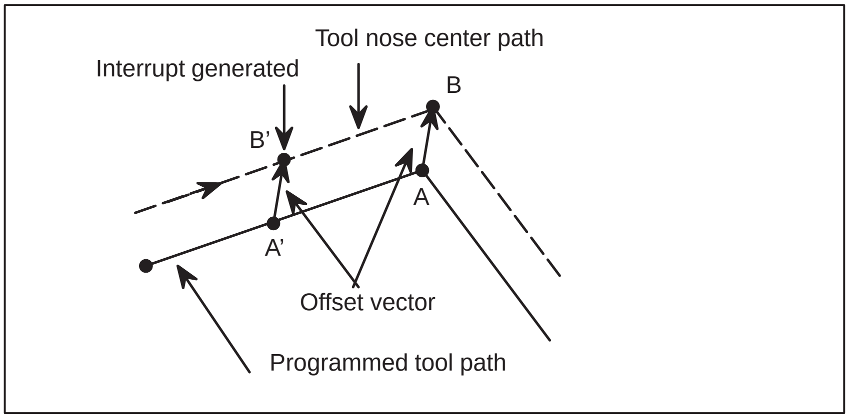
The new modal information modified by the interrupt program remains valid even after control is returned. The old modal information which was valid in the interrupted block can be read using custom macro system variables #4001 to #4120.
Note that when modal information is modified by the interrupt program, system variables #4001 to #4120 are not changed.

● System variables (position information values) for the interrupt program

· The coordinates of point A can be read using system variables #5001 and up until the first NC statement is encountered.

· The coordinates of point A' can be read after an NC statement with no move specifications appears.

· The machine coordinates and workpiece coordinates of point B' can be read using system variables #5021 and up and #5041 and up.



● Custom macro interrupt and custom macro modal call

When the interrupt signal (UINT) is input and an interrupt program is called, the custom macro modal call is canceled (G67). However, when G66 is specified in the interrupt program, the custom macro modal call becomes valid. When control is returned from the interrupt program by M99, the modal call is restored to the state it was in before the interrupt was generated. When control is returned by M99Pxxxx;, the modal call in the interrupt program remains valid.

● Custom macro interrupt and program restart

When the interrupt signal (UINT) is input while a return operation is being performed in the dry run mode after the search operation for program restart, the interrupt program is called after restart operation terminates for all axes. This means that interrupt type II is used regardless of the parameter setting.

17 PROGRAMMABLE PARAMETER ENTRY (G10)

General

The values of parameters can be entered in a program. This function is used for setting pitch error compensation data when attachments are changed or the maximum cutting feedrate or cutting time constants are changed to meet changing machining conditions.

Format

Format

G10L50; Parameter entry mode setting
N_R_; For parameters other than the axis type
N_P_R_; For axis type parameters
:
G11; Parameter entry mode cancel

Meaning of command

N_: Parameter No. (4digits) or compensation position No.(0 to 1023) for pitch errors compensation +10,000 (5digit)
R_: Parameter setting value (Leading zeros can be omitted.)
P_: Axis No. 1 to 8 (Used for entering axis type parameters)

Explanations

• Parameter setting value (R_)

Do not use a decimal point in a value set in a parameter (R_).
a decimal point cannot be used in a custom macro variable for R_ either.

• Axis No.(P_)

Specify an axis number (P_) from 1 to 8 (up to eight axes) for an axis type parameter. The control axes are numbered in the order in which they are displayed on the CNC display.
For example, specify P2 for the control axis which is displayed second.

Notes

Do not fail to perform reference point return manually after changing the pitch error compensation data or backlash compensation data. Without this, the machine position can deviate from the correct position.
Other NC statements cannot be specified while in parameter input mode.
The canned–cycle mode must be cancelled before entering of parameters. When not cancelled, the drilling motion will be activated.

Examples

1. Set bit 2 (SPB) of bit type parameter No. 3404

```
G10L50 ;                    Parameter entry mode
N3404 R 00000100 ;          SBP setting
G11 ;              cancel parameter entry mode
```

2. Change the values for the Z–axis and C–axis in axis type parameter No.1322 (the coordinates of stored stroke limit 2 in the positive direction for each axis).

```
G10L50 ;                    Parameter entry mode
N1322P3R4500 ;              Modify Z axis
N1322P4R12000 ;             Modify C axis
G11 ;              cancel parameter entry mode
```

18 MEMORY OPERATION by FS15 TAPE FORMAT

Programs in the FS15 tape format can be registered in memory for memory operation by setting bit 1 of parameter No. 0001. Registration to memory and memory operation are possible for the functions which use the same tape format as that for the FS15 as well as for the following functions which use a different tape format:

- **Equal–lead threading**
- **Subprogram calling**
- **Canned cycle**
- **Multiple repetitive canned cycle**
- **Canned drilling cycle**

Notes

Registration to memory and memory operation are possible only for the functions available in this CNC.

18.1 ADDRESSES AND SPECIFIABLE VALUE RANGE FOR Series 15 TAPE FORMAT

Some addresses which cannot be used for the this CNC can be used in the Series 15 tape format. The specifiable value range for the Series 15 tape format is basically the same as that for the this CNC. Sections II–18.2 to II–18.5 describe the addresses with a different specifiable value range. If a value out of the specifiable value range is specified, an alarm is issued.

18.2 EQUAL–LEAD THREADING

Format

G32IP_F_Q_;
or
G32IP_E_Q_;

IP:Combination of axis addresses
F:Lead along the longitudinal axis
E:Lead along the longitudinal axis
Q:Sight of the threading start angle (ignored if specified)

Explanations

- **Address**

Although the Series 15 allows the operator to specify the number of threads per inch with address E, the Series 15 tape format does not. Addresses E and F are used in the same way for specifying the lead along the longitudinal axis. The thread lead specified with address E is therefore also assumed as a continuous–state value for address F. Address Q for specifying the shift of the threading start angle can be specified but is ignored.

- **Specifiable value range for the thread lead**

Address for thread lead		mm input	inch input
E		0.0001 to 500.0000mm	0.000001 to 9.999999inch
F	Command with a decimal point	0.0001 to 500.0000mm	0.000001 to 9.999999inch
	Command without a decimal point	0.01 to 500.00mm	0.0001 to 9.9999inch

- **Specifiable value range for the feedrate**

Address for feedrate			mm input	inch input
F	Feed per minute	Increment system (IS–B)	1 to 240000 mm/min	0.01 to 9600.00 inch/min
		Increment system (IS–C)	1 to 100000 mm/min	0.01 to 4800.00 inch/min
	Feed per rotation		0.01 to 500.00 mm/rev	0.0001 to 9.9999 inch/rev

Notes

Specify the feedrate one more time when switching between feed per minute and feed per rotation.

18.3 SUBPROGRAM CALLING

Format

M98P○○○○L○○○○;

P:Subprogram number
L:Repetition count

Explanation

- **Address**

Address L cannot be used in this CNC tape format but can be used in the Series 15 tape format.

- **Subprogram number**

The specifiable value range is the same as that for this CNC (1 to 9999) . If a value of more than four digits is specified, the last four digits are assumed as the subprogram number.

- **Repetition count**

The repetition count L can be specified in the range from 1 to 9999. If no repetition count is specified, 1 is assumed.

18.4 CANNED CYCLE

Format

Outer / inner surface turning cycle (straight cutting cycle)
G90X_Z_F_;

Outer / inner surface turning cycle (taper cutting cycle)
G90X_Z_I_F_;
I:Length of the taper section along the X–axis (radius)

Threading cycle (straight threading cycle)
G92X_Z_F_Q_;
F:Thread lead
Q:Shift of the threading start angle (ignored if specified)

Threading cycle (taper threading cycle)
G92X_Z_I_F_;
I:Length of the taper section along the X–axis (radius)

End surface turning cycle (front taper cutting cycle)
G94X_Z_F_;

End surface turning cycle (front taper cutting cycle)
G94X_Z_K_F_;
K:Length of the taper section along the Z–axis

- **Address**

Addresses I and K cannot be used for a canned cycle in this CNC tape format but can be used in the Series 15 tape format. Address Q can be specified in the Series 15 format but is ignored.

- **Specifiable value range for the feedrate**

Same as that for equal–lead threading in section II–18.2. See section II–**18.2**.

18.5 MULTIPLE REPETITIVE CANNED TURNING CYCLE

Format

Outer / inner surface turning cycle

G71P_Q_U_W_I_K_D_F_S_T_;

I :Length and direction of cutting allowance for finishing the rough machining cycle along the X–axis (ignored if specified)
K :Length and direction of cutting allowance for finishing the rough machining cycle along the Z–axis (ignored if specified)
D :Depth of cut

End surface rough machining cycle

G72P_Q_U_W_I_K_D_F_S_T_;

I :Length and direction of cutting allowance for finishing the rough machining cycle along the X–axis (ignored if specified)
K :Length and direction of cutting allowance for finishing the rough machining cycle along the Z–axis (ignored if specified)
D :Depth of cut

Closed–loop turning cycle

G73P_Q_U_W_I_K_D_F_S_T_;

I :Length and direction of clearance along the X–axis (radius)
K :Length and direction of clearance along the Z–axis
D :Number of divisions

End surface cutting–off cycle

G74X_Z_I_K_F_D_;

or

G74U_W_I_K_F_D_;

I :Distance to be traveled along the X–axis
K :Depth of cut along the Z–axis
D :Clearance of the tool at the end of the cutting path

Outer / inner surface cutting–off cycle

G75X_Z_I_K_F_D_;

or

G75U_W_I_K_F_D_;

I :Distance to be traveled along the X–axis
K :Depth of cut along the Z–axis
D :Clearance of the tool at the end of the cutting path

Multiple repetitive threading cycle

G76X_Z_I_K_D_F_A_P_Q_;

I :Difference of radiuses at threads
K :Height of thread crest (radius)
D :Depth of the first cut (radius)
A :Angle of the tool tip (angle of ridges)
P :Method of cutting

- **Addresses and specifiable value range**

If the following addresses are specified in the Series 15 tape format, they are ignored.
·I and K for the outer/inner surface rough machining cycle (G71)
·I and K for the end surface rough machining cycle (G72)
Address P for specifying the method of cutting for the multiple repetitive threading cycle (G76) is always P1 (constant depth of cut with a single edge) regardless of the command value for P. A value of between 0 and 120 degrees can be specified for tool tip angle A. If other values are specified, P/S alarm 062 is issued.
Address D (cutting depth and retraction distance) can be specified with a value between –99999999 and 99999999, in the minimum input increment, even when calculator–like decimal point input is specified (when bit 0 (DPI) of parameter No. 3401 is set to 1). When address D contains a decimal point, P/S alarm No. 007 is issued.

The specifiable value range for the feedrate is the same as that for equal–lead threading. See section II–18.2.

18.6 CANNED DRILLING CYCLE FORMATS

Format

Drilling cycle
G81X_C_Z_F_L_ ; or G82X_C_Z_R_F_L_ ;
R : Distance from the initial level to the R position
P : Dwell time at the bottom of the hole
F : Cutting feedrate
L : Number of repetitions

Peck drilling cycle
G81X_C_Z_R_Q_P_F_L_ ;
R : Distance from the initial level to the R position
Q : Depth of cut in each cycle
P : Dwell time at the bottom of the hole
F : Cutting feedrate
L : Number of repetitions

High–speed peck drilling cycle
G83.1X_C_Z_R_Q_P_F_L_ ;
R : Distance from the initial level to the R position
Q : Depth of cut in each cycle
P : Dwell time at the bottom of the hole
F : Cutting feedrate
L : Number of repetitions

Tapping
G84X_C_Z_R_P_F_L_ ;
R : Distance from the initial level to the R position
P : Dwell time at the bottom of the hole
F : Cutting feedrate
L : Number of repetitions

Rigid tapping
G84.2X_C_Z_R_P_F_L_S_ ;
R : Distance from the initial level to the R position
P : Dwell time at the bottom of the hole
F : Cutting feedrate
L : Number of repetitions
S : Spindle speed

Boring cycle
G85X_C_Z_R_F_L_ ; or G89X_C_Z_R_P_F_L_ ;
R : Distance from the initial level to the R position
P : Dwell time at the bottom of the hole
F : Cutting feedrate
L : Number of repetitions

Cancel
G80 ;

Explanations

● ● Address

For this CNC tape format, the address used to specify the number of repetitions is K. For the FS 15 tape format, it is L.

- **G code**

Some G codes are valid only for this CNC tape format or FS15 tape format. Specifying an invalid G code results in P/S alarm No. 10 being generated.

G codes valid only for the FS15 tape format	G81, G82, G83.1, G84.2
G codes valid only for the Series 16/18/160/180 tape format	G87, G88

- **Positioning plane and drilling axis**

For this CNC tape format, the positioning plane and drilling axis are determined according to the G code for the canned cycle used.
For the FS15 tape format, the positioning plane and drilling axis are determined according to G17/G19.
The drilling axis is the basic axis (Z–axis or X–axis) that does not lie in the positioning plane.

G code	Positioning plane	Drilling axis
G17	XY plane	Z–axis
G19	YZ plane	X–axis

Resetting bit 1 (FXY) of parameter No. 5101 enables fixing of the drilling axis to the Z–axis.

- **Details of data specifying machining**

Data for the canned cycle is specified as follows:

G¢¢ X C Z R Q P F L ;

Drilling mode
Drilling data
Number of repetitions
Hole position data

Setting	Address	Explanation
Drilling mode	G□□	Canned drilling cycle G code
Hole position data	X/U (Z/W) C/H	Incremental or absolute value used to specify the hole position
Drilling mode	Z/W (X/U)	Incremental or absolute value used to specify the distance from the R position to the bottom of the hole
	R	Incremental value used to specify the distance from the initial level to the R position, or absolute value used to specify the R position. Which to use depends on bit 6 of parameter No. 5102 and the G code system being used.
	Q	Incremental value used to specify the depth of cut in each G83 or G83.1 cycle with radius programming.
	P	Dwell time at the bottom of the hole. The relationship between the dwell time and the specified value is the same as that for G04.
	F	Cutting feedrate
Number of repetitions	L	Number of repetitions for a sequence of cutting operations. If L is not specified, it is assumed to be 1.

- **Specifying the R position**

The R position is specified as an incremental value for the distance between the initial level to the R position. For the FS15 tape format, the parameter and the G code system used determine whether an incremental or absolute value is to be used to specify the distance between the initial level and the R position.
If bit 6(RAB) of parameter No. 5102 is 0, an incremental value is always used. If it is 1, the type of value used depends on the G code system used. When G code system A is used, an absolute value is used. When G code system B or C is used, an absolute value is used in G90 mode, and an incremental value is used in G91 mode.

<table>
<tr><th colspan="4">FS15 tape format</th><th>Series 16/18/160/180 tape format</th></tr>
<tr><td colspan="3">Bit 6 of parameter No. 5102 = 1</td><td>Bit 6 of parameter No. 5102 = 0</td><td rowspan="5">Incremental</td></tr>
<tr><td colspan="3">G code system</td><td rowspan="4">Incremental</td></tr>
<tr><td>A</td><td colspan="2">B, C</td></tr>
<tr><td rowspan="2">Absolute</td><td>G90</td><td>G91</td></tr>
<tr><td>Absolute</td><td>Incremental</td></tr>
</table>

- **Details of the canned cycle**

The correspondence between the G codes and this CNC tape format or FS15 tape format is listed below. This list also provides notes on dwell during a canned cycle.

No. G□□ (Use) This CNC command format

1. **G81 (Drilling cycle) G83 (G87) P0 <Q not specified>**
   No dwelling
2. **G82 (Drilling cycle) G83 (G87) P <Q not specified>**
   The tool always dwells at the bottom of the hole.
3. **G83 (Peck drilling cycle) G83 (G87) <Type B>**
   If the block contains a P command, the tool dwells at the bottom of the hole.
4. **G83.1 (Peck drilling cycle) G83 (G87) <Type A>**
   If the block contains a P command, the tool dwells at the bottom of the hole.
   Note) Either type A or B is selected according to bit 2 (RTR) of parameter No. 5101.
5. **G84 (Tapping) G84 (G88)I**
   f the block contains a P command, the tool dwells after it reaches the bottom of the hole and after it is retracted to the R position.
6. **G84.2 (Rigid tapping) M29 S_ G84 (G88)**
   If the block contains a P command, the tool dwells before the spindle starts rotating in reverse at the bottom of the hole and before it starts rotating in the normal direction at the R position.
7. **G85 (Boring cycle) G85 (G89) P0**
   No dwelling
8. **G89 (Boring cycle) G85 (G89) P_**
   The tool always dwells at the bottom of the hole.

- ● **Clearance d for G83 and G83.1**

Parameter No. 5114 determines clearance d for G83 and G83.1.

- **Dwell with G83 and G83.1**

For Series 15–T, G83 or G83.1 does not cause the tool to dwell. For the FS15 tape format, the tool dwells at the bottom of the hole only if the block contains a P address.

- **Dwelling with G84 and G84.2**

In Series 15–T, G84/G84.2 causes the tool to dwell before the spindle starts rotating in either the normal or reverse direction, according to the corresponding parameter setting. For the FS15 tape format, when the block contains a P address, the tool dwells at the bottom of the hole and at the R position before the spindle starts rotating either in the normal or reverse direction.

- **Rigid tapping**

For the FS15 tape format, rigid tapping can be specified by using the methods listed below:

<table>
<tr><th>Format</th><th>Condition (parameter), comment</th></tr>
<tr><td>G84.2 X_ Z_ R_ ...S**** ;</td><td rowspan="2">Setting (F10/F11) = 1</td></tr>
<tr><td>S**** ;
G84.2 X_ Z_ R_ .... ;</td></tr>
<tr><td>M29 S**** ;
G84 X_ Z_ R_ .... ;</td><td rowspan="2">* Common to Series 16 format</td></tr>
<tr><td>M29 S**** G84 X_ Z_ R_ .... ;</td></tr>
<tr><td>G84 X_ Z_ R_ .... S**** ;</td><td rowspan="2">G84 is made a G code for rigid tapping.
Bit 0 (G84) of parameter No. 5200 = 1
* Common to Series 16 format</td></tr>
<tr><td>S**** ;
G84 X_ Z_ R_ .... ;</td></tr>
</table>

- **Diameter or radius programming**

Specifying 1 for bit 7 (RDI) of parameter No. 5102 causes the canned cycle R command diameter or radius programming mode in the FS15 tape format to match the diameter or radius programming mode for the drilling axis.

- **Disabling the FS15 format**

Specifying bit 3 (F16) of parameter No. 5102 disables the FS15 tape format. This applies only to the canned drilling cycle. However, the number of repetitions must be specified by using the L address.

Notes

Setting bit 3 (F16) of parameter No. 5102 to 1 overrides bits 6 (RAB) and 7 (RDI) of parameter No. 5102; both settings are assumed to be 0.

Limitations

- **C–axis as the drilling axis**

It is impossible to use the C–axis (the third axis) as a drilling axis. So, specifying G18 (ZX plane) generates P/S alarm No. 28 (plane selection command error).

- **Clamping the C–axis**

For the FS15 tape format, it is impossible to specify an M code for clamping the C–axis.

19 FUNCTIONS FOR HIGH SPEED CUTTING

19.1 HIGH SPEED CYCLE CUTTING

This function can convert the machining profile to a data group that can be distributed as pulses at high–speed by the macro compiler and macro executor. The function can also call and execute the data group as a machining cycle using the CNC command (G05 command).
This function is applied to 1–path lathe control.

Format

G05 P10○○○ L○○○ ;

P10○○○ is number of the machining cycle to be called first:
P10001 to P10999
L○○○ is repetition count of the machining cycle
(L1 applies when this parameter is omitted.) :
L1 to L999

Call and execute the data for the high speed cutting cycle specified by the macro compiler and macro executor using the above command.
Cycle data can be prepared for up to 999 cycles. Select the machining cycle by address P. More than one cycle can be called and executed in series using the cycle connection data in the header.
Specify the repetition count of the called machining cycle by address L. The repetition count in the header can be specified for each cycle.
The connection of cycles and their repetition count are explained below with an example.

Example) Assume the following:
Cycle 1 Cycle connection data 2 Repetition count 1
Cycle 2 Cycle connection data 3 Repetition count 3
Cycle 3 Cycle connection data 0 Repetition count 1
G05 P10001 L2 ;
The following cycles are executed in sequence:
Cycles 1, 2, 2, 2, 3, 1, 2, 2, 2, and3

Notes

Notes

1. An alarm is issued if the function is executed in the G41/G42 mode.

2. Single block stop, dry run/feedrate override, automatic acceleration/deceleration and handle interruption are disabled during high–speed cycle machining.

Alarms

Alarm number	Descriptions
115	The contents of the header are invalid. This alarm is issued in the following cases. 1. The header corresponding to the number of the specified call machining cycle was not found. 2. A cycle connection data value is not in the valid range (0 to 999). 3. The number of data items in the header is not in the valid range (1 to 32767). 4. The first variable No. for storing data in the executable format is not in the valid range (#20000 to #85535). 5. The last variable No. for storing data in the executable format exceeds the limit (#85535). 6. The first variable No. for start data in the executable format overlaps with a variable No. used in the header.
178	High–speed cycle machining was specified in the G41/G42 mode.
179	The number of control axes specified in parameter 7510 exceeds the maximum number.

19.2 DISTRIBUTION PROCESSING TERMINATION MONITORING FUNCTION FOR THE HIGH–SPEED MACHINING COMMAND (G05)

During high–speed machining, the distribution processing status is monitored. When distribution processing terminates, P/S alarm No. 000 and P/S alarm No. 179 are issued upon completion of the high–speed machining command (according to the setting of ITPDL (bit 7 of parameter No. 7501)).

These P/S alarms can be canceled only by turning off the CNC power.

Explanations

- **High–speed machining command**

High–speed machining using the high–speed remote buffer A function, high–speed remote buffer B function, and high–speed cycle function based on the G05 command

- **Distribution processing termination**

Failure to perform normal distribution processing because distribution processing required for high–speed machining exceeded the CNC processing capacity, or because distribution data sent from the host was delayed for some reason while the high–speed remote buffer A or G function was being used

<table>
<tr><th>Number</th><th>Message</th><th>Contents</th></tr>
<tr><td>000</td><td>PLEASE TURN OFF POWER</td><td rowspan="2">During high–speed machining, distribution processing was terminated.
Related parameters:
Remote buffer transfer baud rate (parameter No. 133)
Number of controlled axes in high–speed machining (parameter No. 7150)
High–speed axis selection during high–speed machining (bit 0 of parameter No. 7510)</td></tr>
<tr><td>179</td><td>PARAM. (NO. 7510) SETTING ERROR</td></tr>
</table>

20 AXIS CONTROL FUNCTION

20.1 POLIGONAL TURNING

Polygonal turning means machining a polygonal figure by rotating the workpiece and tool at a certain ratio.

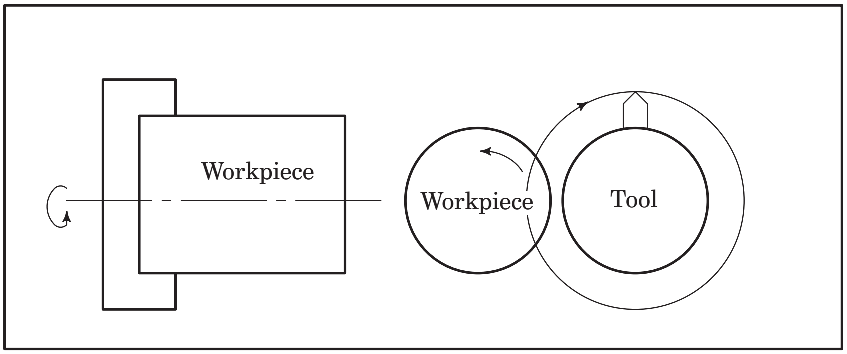


Fig.20.1 (a)Polygonal turning

By changing conditions which are rotation ratio of workpiece and tool and number of cutters,the machining figure can be changed to a square or hexagon. The Machining time can be reduced as compared with polygonal figure machining using C and X axes of the polar coordinate. The machined figure however, is not exactly polygonal. Generally, polygonal turning is used for the heads of square and/or hexagon bolts or hexagon nuts.

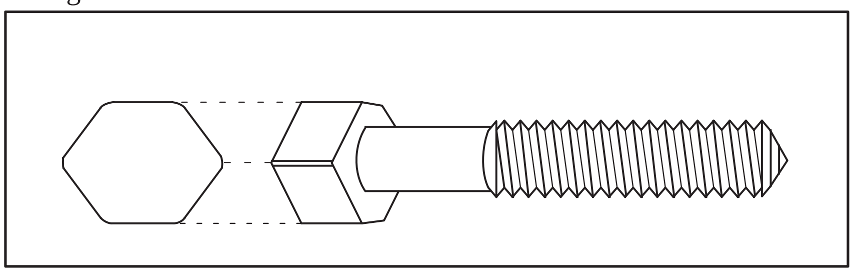


Fig.20.1 (b)Hexagon bolt

Format

G51.2(G251)	**P_Q_;**	
	P,Q:	**Rotation ratio of spindle and Y axis** **Specify range**:**Intefer 1 to 9 for both P and Q** **When Q is a positive value, Y axis makes positive rotation.** **When Q is a negative value, Y axis makes negative rotation.**

Explanations

Tool rotation for polygonal turning is controlled by CNC controlled axis. This rotary axis of tool is called Y axis in the following description.
The Y axis is controlled by G51.2 command, so that the rotation speeds of the workpiece mounted on the spindle (previously specified by S–command) and the tool becomt the specified ratio.
(Example) Rotation ratio of workpiece (spindle) to Y axis is 1:2, and the Y axis makes positive rotation.

G51.2P1Q2;

When simultaneous start is specified by G51.2, the one–rotation signal sent from the position codes set on the spindle is detected. After this detection. the Y axis rotation is controoled according to the rotation ratio (P:Q) while synchronizing with the spindle speed. Namely, the Y axis rotation is controlled so that the spindle and Y axis stand in a relation of P:Q. This relation will be maintained until the plygonal turning cancel command is executed (G50.2 or reset operation). The direction of Y axis rotation is determined by the code
Q and not affected by the direction of the lposition coder rotation.
Synchronixation of the spindle and Y axis is canceled by the following commnad:

G50.2(G250);

When G50.2 is specified, synchronization of the spindle and Y axis is canceled and the Y axis stops.
This synchronization is also canceledd in the following casset:

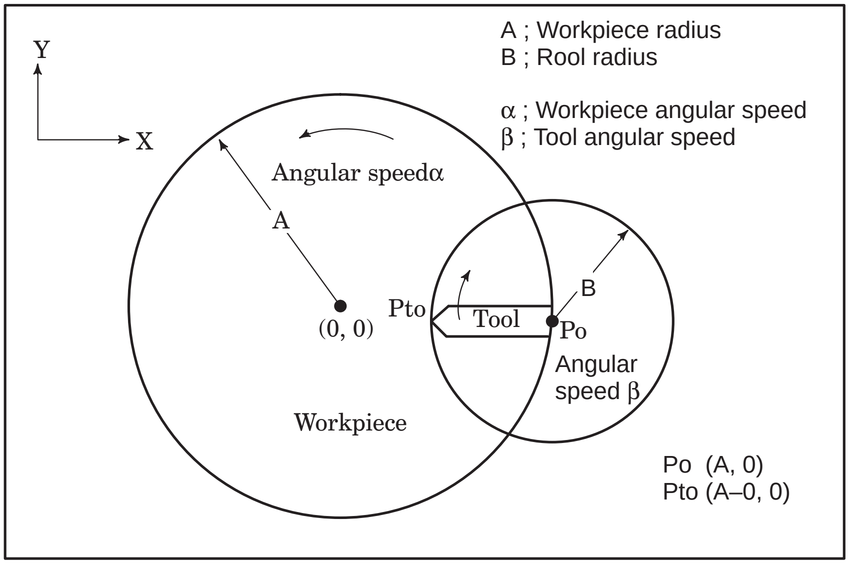
i) Power off
ii) Emergency stop
iii) Servo alarm
iv) Reset (external reset signal ERS, reset/rewind ignal RRW, and RESET key on the CRT/MDI panel)
v) Occurrence of P/S alarm Nos. 217 to 221

Example

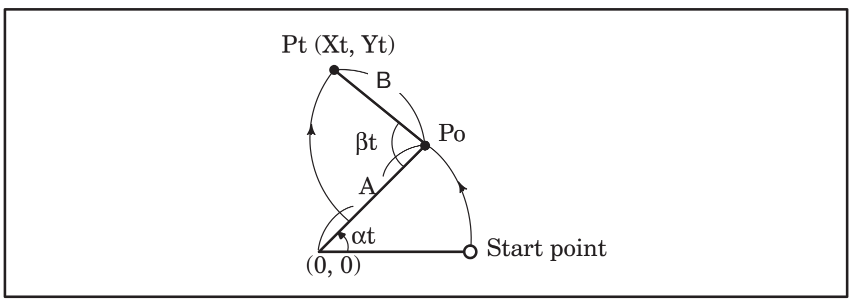
G00X100. 0Z20.0 S1000.0M03 ; Workpiece rotation speed 1000rmp
G51.2P1 Q2 ; Tool rotation start (tool rotation speed 2000rpm)
G01X80.0 F10.0 ; X axis infeed
G04X2. ;
G00X100.0 ; X axis escape
G50.2 ; Tool rotation stop
M05 ; Spindle stop
Specify **G50.2** and **G51.2** always in a single block.

- **Principle of Polygonal Turning**

The principle of polygonal turning is explained below. In the figure below the radius of tool and workpiece are A and B, and the angular speeds of tool and workpiece are $\alpha$and β. The origin of XY cartesian coordinates is assumed to be the center of the workpiece.
Simplifying the explanation, consider that the tool center exists at the position
Po (A,0) on the workpiece periphery, and the tool nose starts from position Pto(A−B, 0).

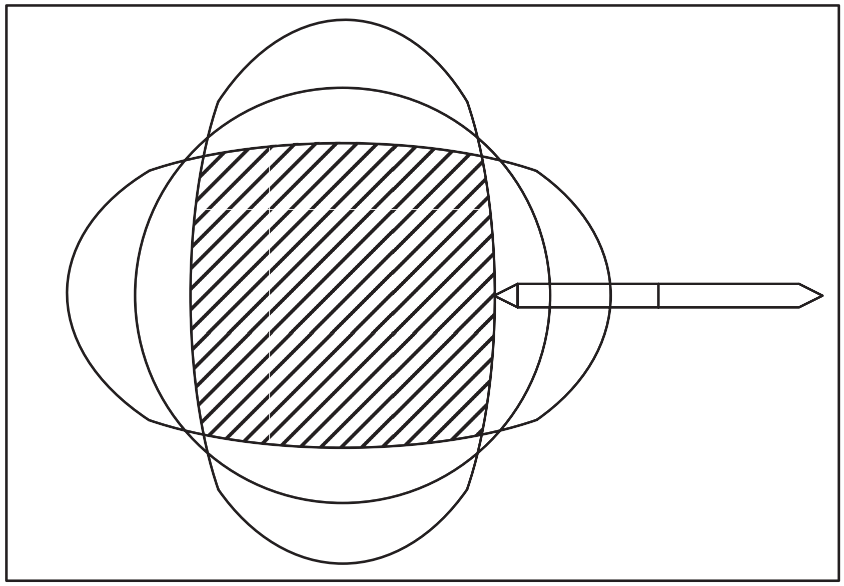


In this case, the tool nose position Pt (Xt,Yt) after time t is expressed by equation 1:

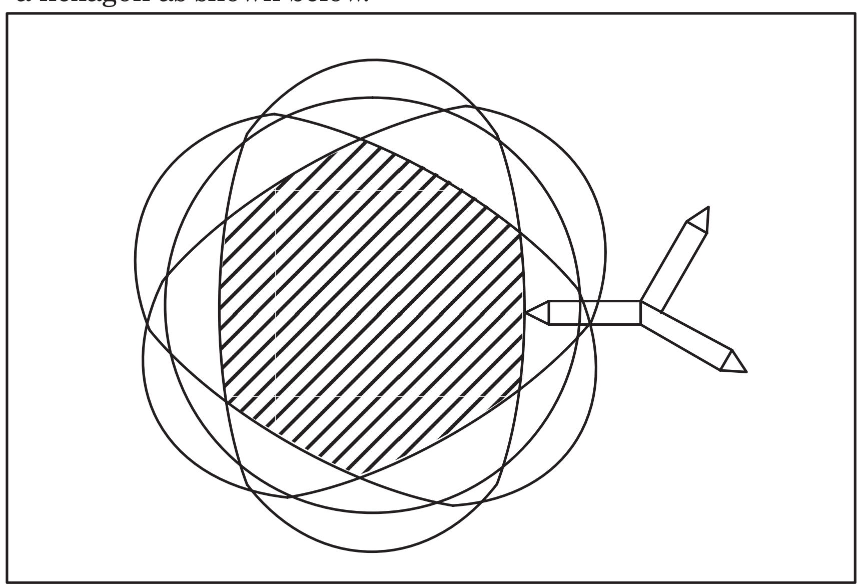


$Xt=A\cos \alpha t-B\cos(\beta-\alpha)t$
(Equation 1)
$Yt=A\sin \alpha t+B\sin(\beta-\alpha)t$
Assuming that the rotation ration of workpiece to tool is 1:2, namely, $\beta=2\alpha$,
equation 1 is modified as follows

$Xt=A\cos\alpha t-B\cos\alpha t=(A-B)\cos\alpha t$
(Equation 2)
$Xt=A\sin\alpha t+B\sin\alpha t=(A+B)\sin\alpha t$
Equation 2 indicates that the tool nose path draws an ellipse with longer diameter A+B and shorter diameter A−B.
Then consider the case when one tool is set at 180° symmetrical positions, for atotal of two. It is seen that a square can be machined with these tools as shown below.



If three tools are set at every 120°, the machining figure will be a hexagon as shown below.



Notes

1. For the maximum rotation speed of tool, refer to the manual published by the MTB. Do not specify a spindle speed or ratio to spindle speed exceeding the maximum rotation speed of the tool.
2. The Y axis, unlike the other controlled axes, cannot be specifiied a move command as Y––. That is, an axis move command is unnecessary for the Y axis. Because, when G51.2 (polygonal turning mode) is specified, it is only necessary to control the Y axis so that the tool rotates at a certain ratio to the spindle rotation speed.
   However, only the reference point return command (G28V0;) can be specified since the Y axis rotation is stopped at the unstable position when G50.2 (polygonal turning mode cancel command) is specified. If the tool rotation start position is unstable , a problem may occur, for example, when the same figure is machined with a finishing tool after once machined with a roughing tool.
   Specification of G28V0; for Y axis is equal to the orientation command for the spindle. In the other axes, unlike the manual reference point return, G28 usually makes reference point return without detecting the deceleration limit. However, with G28V0; , for the Y axis, reference point return is executed by detecting the deceleration limit, like manual reference point return.
   To machine a workpiece into the same figure as the previous one, the tool and the spindle must be in the same position as the previous time when the tool starts rotating. The tool is set start rotation when the one–rotation signal of the position coder set on the spindle is detected.
3. Unlike the other controlled axes, the least command increment of the Y axis is not 0.001 degree since an axis move command is unnecessary for the Y axis. The least command increment of the Y axis is related to parameters such as feedrate.
   Therefore, pay attention to these parameters upon machine adjustment.
   - L : Move distance (deg) per motor rotation
   - Q : Number of pulses per pulse coder rotation
   - CMR : Command multiply (Set parameter No. 1820)
   - DMR : Detection multiply (Set parameter No.1816#4,#5 and #6 However,when flexible feed gear is used,set the value of numerator and the value of denominator of DMR)

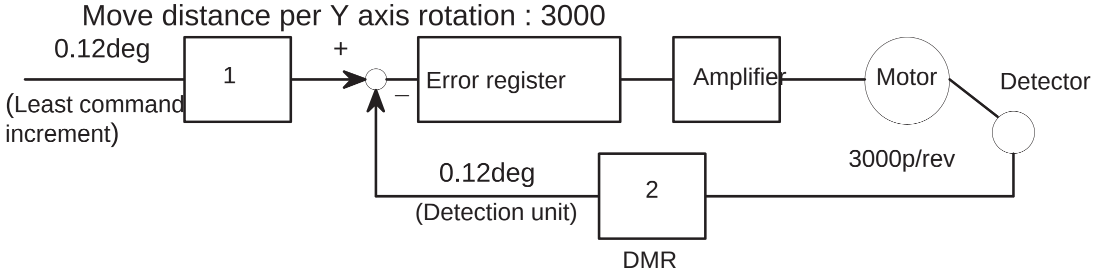
Least command increment $= \dfrac{L \times CMR}{Q \times DMR}$

Detection unit $= \dfrac{Least\ \ command\ \ increment}{CMR} = \dfrac{L}{Q \times DMR}$

Move distance per Y axis rotation $= \dfrac{360}{Least\ \ command\ \ increment}$(Parameter No.7620)

For example, in case of L=720 deg, Q=3000 pulses, CMR=2,the move distance is as follows;

Least command increment : 0.12deg
Detection unit : 0.12deg
Move distance per Y axis rotation : 3000



When the least command increment of the Y axis is 0.12 deg, the parameters concerning the feedrate of the Y axis should be set in unit of 120 deg/min (Thousandfold of least command increment), not in 1 deg/min.

4. The Y axis used to control tool rotation for polygonal turning uses the 4th axis. However, by setting parameters, (No.7610) the 3rd axis may also be used. In this case, that axis must be named C axis.
5. Among the position display of the Y axis, the display for the machine coordinate value (MECHINE) will change from a range of 0 to the parameter setting (the amount of movement per revolution) as the Y axis moves.
   Absolute or relative coordinate values are not renewed.
6. An absolute position detector cannot be set on the Y axis.
7. The following signals become either valid or invalid in relation to the Y axis in synchronous operation.
   Valid signals in relation to Y axis:
      machine lock
      servo off
   Invalid signals in relation to Y axis:
      feed hold
      interlock
      ovrride
      dry run
      (During a dry run, however, there is no wait for a revolution signal in the G51.2 block.)
8. Manual continuous feed or handle feed is invalid when the Y axis is in synchronous operation.
9. The starting point of the threading process becomes inconsistent when performed during synchronous operation.
   Cancel the synchronizing by executing G50.2 when threading.
10. The Y axis in synchronous operation is not included in the number of axis controlled simultaneously.

20.2 ROTARY AXIS ROLL–OVER

The roll–over function prevents coordinates for the rotation axis from overflowing. The roll–over function is enabled by setting bit 0 of parameter 1008 to 1.

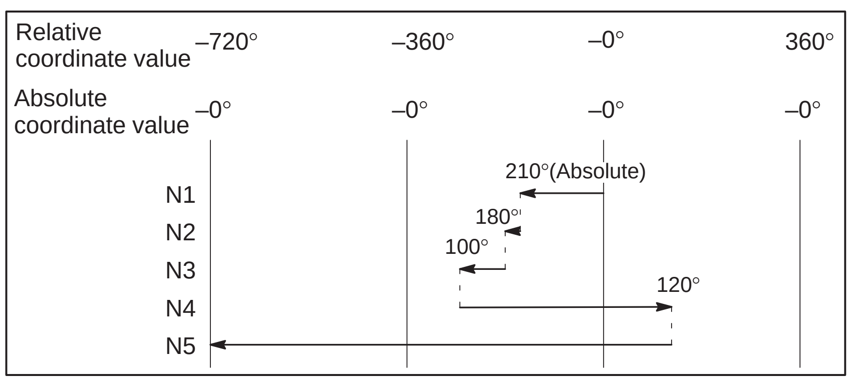
Explanations

For an incremental command, the tool moves the angle specified in the command. For an absolute command, the coordinates after the tool has moved are values set in parameter No. 1260, and rounded by the angle corresponding to one rotation. The tool moves in the direction in which the final coordinates are closest when bit 1 (ROAx)of parameter No. 1008 is set to 0. Displayed values for relative coordinates are also rounded by the angle corresponding to one rotation when bit 2 (ROAx) of parameter No. 1008 is set to 1

Examples

Assume that axis C is the rotating axis and that the amount of movement per rotation is 360.000 (parameter No. 1260 = 360000). When the following program is executed using the roll–over function of the rotating axis, the axis moves as shown below.

C0 ;	**Sequence number**	**Actual movement value**	**Absolute coordinate value after movement end**
N1 C–150.0 ;	N1	–150	210
N2 C540.0 ;	N2	–30	180
N3 C–620.0 ;	N3	–80	100
N4 H380.0 ;	N4	+380	120
N5 H–840.0 ;	N5	–840	0



20.3 SIMPLE SYNCHRONIZATION CONTROL

The simple synchronization control function allows synchronous and normal operations on two specified axes to be switched, according to an input signal from the machine.
For a machine with two tool posts that can be independently driven with different controlled axes, this function enables the operations described below.
This section describes the operations of a machine having two tool posts, both of which can be independently operated along the X–axis and Y–axis. If your machine uses other axes for the same purpose, substitute the corresponding axis names for X and Y.

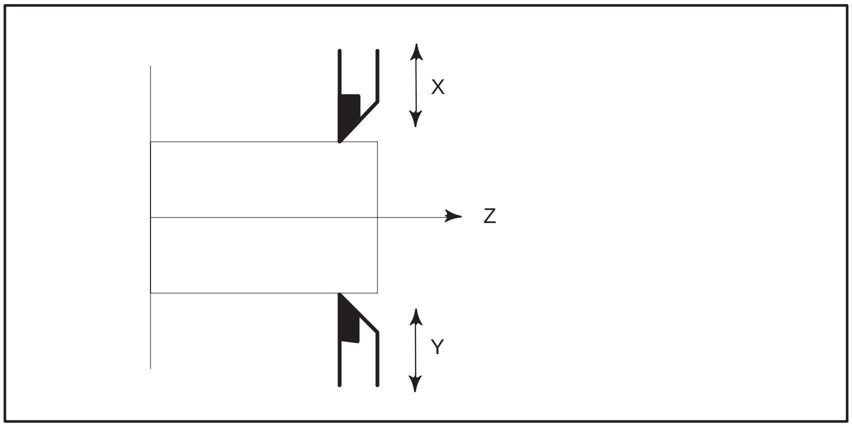


Fig. 20.3 (a) Sample Axis Configuration of a Machine on which the Simple Synchronization Control Function is Executed

Explanations

- **Synchronous operation**

Synchronous operation is possible on a machine having two tool posts. In synchronous operation mode, movement on one axis can be synchronized with movement specified for another axis. The move command can be specified for one of the two axes, which is referred to as the master axis. The other axis, for referred synchronization with the master axis is maintained, is referred to as the slave axis. If the master axis is X and the slave axis is Y, synchronous operation on the X–axis (master axis) and Y–axis (slave axis) are performed according to Xxxxx commands issued for the master axis.
In synchronous operation mode, a move command specified for the master axis results in simultaneous operation of the servo motors of the master and slave axes.
In this mode, synchronization error compensation is not performed. That is, any positioning error between the two servo motors is not monitored, nor is the servo motor of the slave axis adjusted to minimize any error. No synchronization error alarm is output. Automatic operations can be synchronized, but manual operations cannot.

- **Normal operation**

Normal operation is performed when different workpieces are machined on different tables. As with normal CNC control, move commands for the master and slave axes are specified with the addresses of those axes (X and Y). Move commands for the two axes can be specified in an identical block.

1 According to the Xxxxx command programmed for the master axis, movement is performed along the X–axis, as in normal mode.

2 According to the Yyyyy command programmed for the slave axis, movement is performed along the Y–axis, as in normal mode.

3 According to the Xxxxx Yyyyy command, simultaneous movements are performed along both the X–axis and Y–axis, as in normal mode. Both automatic and manual operations can be controlled, as in normal CNC control.

- **Switching synchronous and normal operations**

For details of how to switch the synchronous and normal operations, refer to the manual supplied by the machine tool builder.

- **Automatic reference position return**

If a command for automatic reference position return (G28), or return to the second, third, or fourth reference position (G30), is issued in synchronous operation mode, a reference position return is performed for the X–axis, and an identical movement is performed for the Y–axis. If this Y–axis movement agrees with a return to the reference position on the Y–axis, a lamp indicating that reference position return has been completed for the Y–axis also lights.
It is recommended, however, that G28 and G30 be specified in normal operation mode.

- **Checking automatic reference position return**

If a command for checking automatic reference position return (G27) is issued in synchronous operation mode, identical movements are performed for the X–axis and Y–axis.
If these X–axis and Y–axis movements correspond to returns to the reference positions on the X–axis and Y–axis, the lamps indicating that reference position return has been completed for the X–axis and Y–axis light. If not, an alarm is output.
It is recommended, however, that G27 be specified in normal operation mode.

- **Slave axis command**

If a move command is specified for the slave axis in synchronous operation mode, P/S alarm 213 is output.

- **Master and slave axes**

The master axis is defined in parameter 8311. The slave axis is specified by an external signal.

Limitations

- **Coordinate system setting and tool compensation**

If coordinate system setting or tool compensation causing a shift in the coordinate system is performed in synchronous operation mode, P/S alarm 214 is output.

- **External deceleration, interlock, machine lock**

In synchronous operation mode, the signal for external deceleration, interlock, or machine lock of the master axis only is valid. The corresponding slave axis signal is ignored.

- **Pitch error compensation**

Pitch error compensation and backlash compensation are performed separately for the master and slave axes.

- **Manual absolute switch**

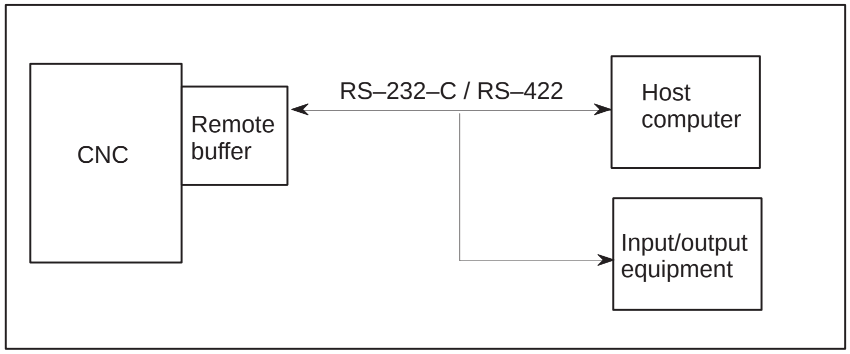
In synchronous operation mode, the manual absolute switch must be set to on (ABS must be set to 1). If the switch is set to off, the correct slave axis movement may not be made.

- **Manual operation**

Manual operations cannot be synchronized.

20.4 HIGH–SPEED REMOTE BUFFER

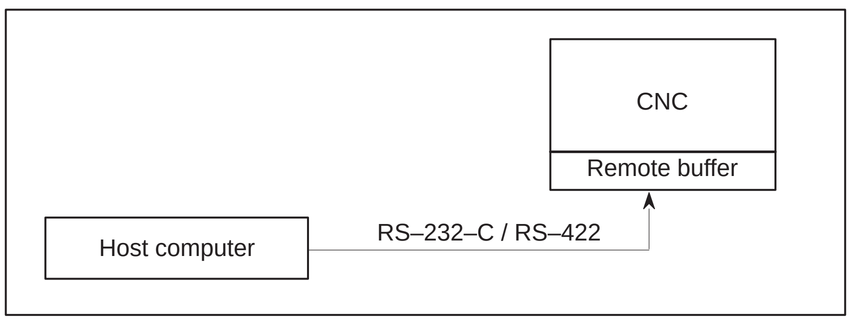
A remote buffer can continuously supply a large amount of data to the CNC at high speeds when connected to the host computer or input/output equipment via a serial interface.



When the remote buffer is connected online to the host computer, fast and reliable DNC operation is possible.
The remote buffer function includes high–speed remote buffer A and high–speed remote buffer B for high–speed machining. High–speed remote buffer A uses binary data. For details on remote buffer specifications, refer to the ”Remote Buffer Supplement” (B–61802–1).

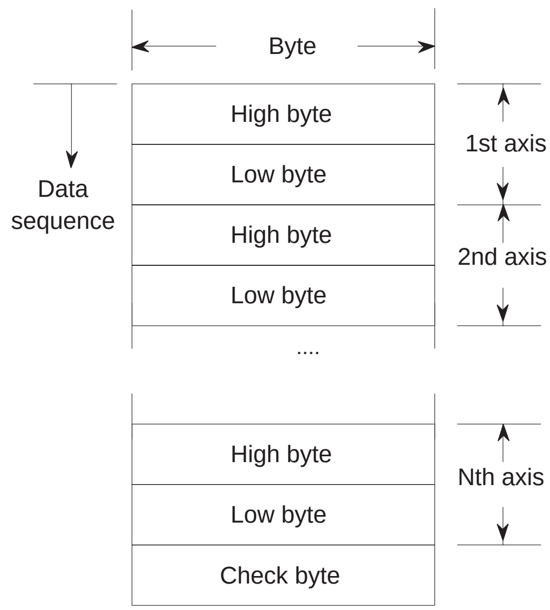
20.4.1 High–speed remote buffer A (G05)

Specify G05 only in a block using normal NC command format. Then specify move data in the special format explained below. When zero is specified as the travel distance along all axes, normal NC command format can be used again for subsequent command specification.



Format

·Binary input operation enabled : G05;
·Binary input operation disabled : The travel distance along all axes are set to zero.
·Data format for binary input operation



In the data format for binary input operation, the travel distance along each axis (2 bytes) per unit time is specified. The travel distances along all axes are placed sequentially from the first axis, then a check byte is added. (The data length for one block is [2 x N + 1] bytes). All data must be specified in binary.

Explanations

• Selecting the unit time

The unit time (in ms) can be selected by setting bits 4, 5, and 6 of parameter No. 7501.

• Travel distance data

The following unit is used for specifying the travel distance along each axis. (A negative travel distance is indicated in 2's complement.)

Increment system	IS–B	IS–C	Unit
Millimeter machine	0.001	0.0001	mm
Inch machine	0.0001	0.00001	inch

The data format of the travel distance is as follows. The bits marked * are used to specify a travel distance per unit time.

15	14	13	12	11	10	9	8	7	6	5	4	3	2	1	0
*	*	*	*	*	*	*	0	*	*	*	*	*	*	*	0

Example: When the travel distance is 700 μm per unit time (millimeter machine with increment system IS–B)

15	14	13	12	11	10	9	8	7	6	5	4	3	2	1	0
0	0	0	0	1	0	1	0	0	1	1	1	1	0	0	0

- **Check Byte**

All bytes of the block except for the check byte ([2*N] bytes) are summed up, and any bits above 8th bit are discarded.

- **Transfer speed**

The CNC reads (2 x N + 1)–byte data (where N is the number of axes) for every unit time that is set in the parameter. To allow the CNC to continue machining without interruption, the following minimum baud rate is required for data transfer between the host and remote buffer:

$$(2 \times N+1) \times \frac{11}{T} \times 1000 \text{ baud (T : Unit time)}$$

- **Cutter compensation**

If G05 is specified in cutter compensation mode, the P/S 178 alarm is issued.

- **Feed hold and interlock**

Feed hold and interlock are effective.

- **Mirror image**

The mirror image function (programmable mirror image and setting mirror image) cannot be turned on or off in the G05 mode.

- **Acceleration / deceleration type**

In binary input operation mode, when tool movement starts and stops in cutting feed mode, exponential acceleration/deceleration is performed (the acceleration/deceleration time constant set in parameter No. 1622 is used).

Limitations

- **Modal command**

In binary input operation mode, only linear interpolation as specified in the defined data format is executed (equivalent to the incremental command for linear interpolation).

- **Invalid functions**

The single block, feedrate override, and maximum cutting feedrate clamp functions have no effect. The program restart, block restart, and high–speed machining functions cannot be used. In addition, miscella–neous functions cannot be executed in binary operation.

- **Memory registration**

No data can be stored in memory.

20.5 SYNCHRONIZATION CONTROL

The synchronization control function enables the synchronization of movements on two axes. If a move command is programmed for one of those two axes (master axis), the function automatically issues the same command to the other axis (slave axis), thus establishing synchronization between the two axes. The parking state can be selected to suppress movement of the slave axis, even if a move command is specified for the master axis. If the parking state is used with the synchronization control function, the operation can be controlled as follows:

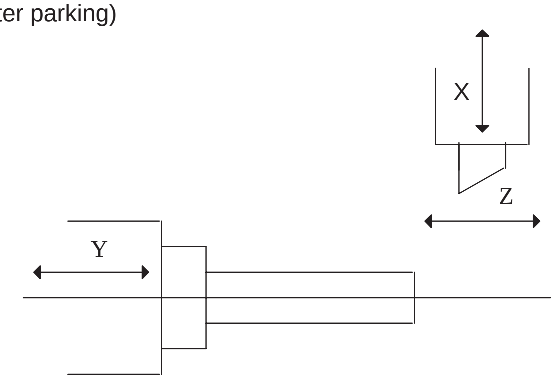
1 Synchronizes the movement on the slave axis with that of the master axis.

2 Performs slave axis movement according to the move command programmed for the master axis. However, the movement specified by the command is not made for the master axis itself (master parking).

3 Updates the slave axis coordinates according to the distance travelled along the master axis. However, no movement is made for the slave axis (slave parking).

When method 2 above is used, the following operation can be performed:

(Example) Synchronizing movements on the Z–axis and Y–axis (master parking)

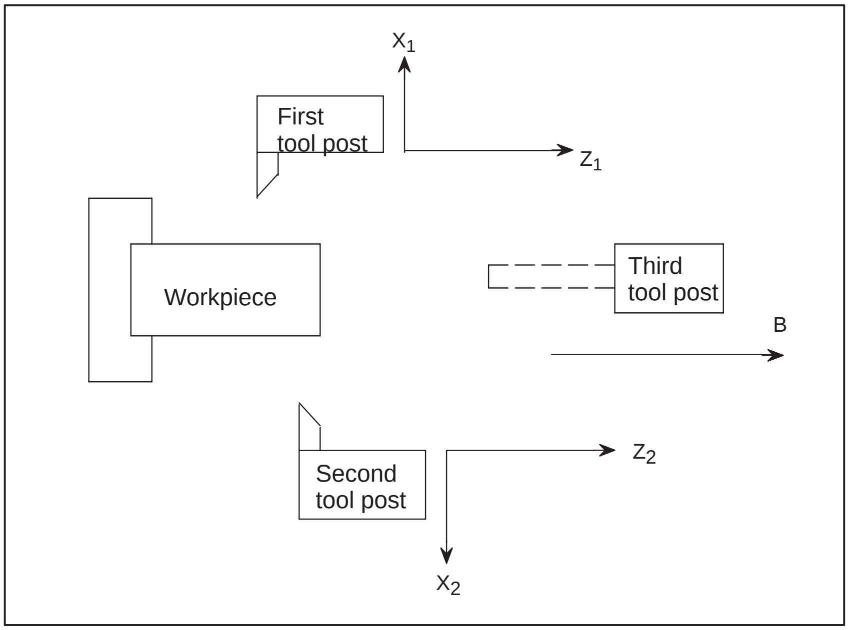


Movement is performed for the X–axis and Y–axis according to commands issued for the X–axis and Z–axis. (The Y–axis movement is synchronized with that of the Z–axis.) If the Z–axis is set to the parking state, the coordinates on the Z–axis and Y–axis are updated.
As the coordinates on the Z–axis and Y–axis are always updated, the coordinate system need not be reset when the synchronization status is changed. A move command can be executed immediately after the status is changed.

Notes
1. In the synchronization control described above, an identical move command is simultaneously output for two servo processing systems. Positional error between the two servo motors is not monitored, nor is either servo motor adjusted to minimize the error. That is, synchronization error compensation is not carried out.
2. The method used to specify the synchronization control function varies with the machine tool builder. For details, refer to the manual supplied by the machine tool builder.

20.6 B–AXIS CONTROL (G100, G101, G102, G103, G110)

This function sets an axis (B–axis) independent of the basic controlled axes X_1, Z_1, X_2, and Z_2 and allows drilling, boring, or other machining along the B–axis, in parallel with the operations for the basic controlled axes. The X_2 and Z_2 axes can be used in two–path control mode.



Format

- **Registering operation programs**

G101–G100 : Starts registering the first program.
G102–G100 : Starts registering the second program.
G103–G100 : Starts registering the third program.
G100 : Ends registering of the programs.

Three operations (programs) on the B–axis can be registered. (In two–path control mode, three programs can be registered for each tool post.) The B–axis operation program must be specified in the blocks between G101, G102, or G103 and G100, allowing it to be discriminated from the normal NC program.
The registered operation is started upon executing the corresponding M code, described below.

```
O1234 ;
      ...
         Normal NC program
G101 ;- - - - - - - - - - - - -   Starts registering of a B–axis
      ...                         operation program.
         B–axis operation program
G100 ;  - - - - - - - - - - - -   Ends registering of the B–axis
      ...                         operation program.
         Normal NC program
M30 ;
```

Note) In the block of G101, G102, G103, or G100, specify no other codes.

• Command used to start the operation

To start an operation, the miscellaneous functions (M**) specified in parameters 8251 to 8253 are used.

Parameter 8251:
M code used to start operation of the first program
Parameter 8252:
M code used to start operation of the second program
Parameter 8253:
M code used to start operation of the third program

O1234 ;	
...	
M** ;	Starts executing the registered B–axis operation. In subsequent blocks, the normal NC program and the B–axis operation program are executed in parallel. (** is specified in parameters 8251 to 8253.)
...	
M30 ;	

Example

01234 ;	
G50 X100. Z200. ;	
G101 ;	$\boxed{1}$ Starts registering of an operation program.
G00 B10. ;	$\boxed{2}$ Blocks of the B–axis operation program
M03 ;	
G04 P2500 ;	
G81 B20. R15. F500 ;	
G28 ;	
G100 ;	$\boxed{3}$ Ends registering of the operation program.
G00 X80. Z50. ;	
G01 X45. F1000 ;	
G00 X10. ;	
M** ;	$\boxed{4}$ Command used to start the programmed operation
G01 Z30. F300 ;	
M30 ;	

$\boxed{1}$ to $\boxed{3}$: Specify the B–axis operation program in blocks between G101, G102, or G103 and G100. The program is registered in program memory.

$\boxed{4}$: Starts executing the B–axis operation registered with $\boxed{1}$ to $\boxed{3}$ above. In subsequent blocks, the normal NC operation and the B–axis operation are executed in parallel. An M code of the miscellaneous function is used to start the B–axis operation. The M code, used to start the operation, is specified in parameters 8251 to 8253.

• Single–motion operation

G110 [operation command];

A single–motion operation for the B–axis can be specified and executed as shown above. Such an operation need not be registered as a special (first to third) program. Nor does it need to be by a special command, as described above.

Explanations

- **Specifying two–path control mode**

One of the following three two–path control modes can be selected:

1 B–axis control is executed for either tool post 1 or 2.

2 B–axis control is executed separately for tool posts 1 and 2.

3 Identical B–axis control is executed for tool posts 1 and 2.

The mode is selected according to the value specified for parameter 8250 for each tool post.

- **Codes that can be used in a B–axis operation program**

The following 13 G codes, and the M, S, and T codes of the miscellaneous functions, can be used in a B–axis operation program:

Code	Description
G00	Positioning (rapid traverse)
G01	Linear interpolation (cutting feed)
G04	Dwell
G28	Reference position return, automatic coordinate system setting
G80	Canned cycle, cancel
G81	Drilling cycle, spot drilling
G82	Drilling cycle, counterboring
G83	Peck drilling cycle
G84	Tapping cycle
G85	Boring cycle
G86	Boring cycle
G98	Feed per minute
G99	Feed per rotation
M**	Auxiliary function
S**	Auxiliary function
T**	Auxiliary function, tool offset

G28 (reference position return)
Unlike the normal G28 cycle, the G28 cycle for a B–axis operation does not include intermediate point processing. For example, the following cannot be specified:

G28 B99.9;

G80 to G86 (canned drilling cycle)
Of the canned drilling cycles supported by the FANUC Series 16 or Series 18 for machining centers, those cycles equivalent to G80 to G86 can be executed.
Data can be specified in the same way as for the FANUC Series 16 or Series 18 for machining centers, except for the following points:

1. The drilling position is not specified with X and Y.
2. The distance from point R to the bottom of the hole is specified with B.

3. All operations are executed in the initial level return mode.
4. The repetition count (K) cannot be specified.
5. In canned cycle mode, point R must be specified. (If point R is omitted, P/S alarm No. 5036 is output.)
6. The drilling start point (d) for the G83 (peck drilling) cycle is specified with parameter 8258.

G98, G99 (feed per minute, feed per rotation)

The MDF bit (bit 2 of parameter 8241) specifies an initial continuous–state G code for G110, or the G code to start registration of the operation program (G101, G102, G103).
When the MDF bit is set to 0, the initial continuous–state code is G98.
When the MDF bit is set to 1, the initial continuous–state code is G99.

Example)
When MDF is set to 0
G110 B100. F1000. ; 1000 mm/min
G110 G99 B100. F1 ; 1 mm/rev

Note
In two–path control mode, the system uses the actual spindle speed, calculated from the feedback signal output by the position coder connected to the tool post to which the controlled axis belongs.

M, S, and T codes (auxiliary functions)

According to a numeric value subsequent to address M, S, or T, the binary code and strobe signal are sent to the machine. The codes and signals for addresses M, S, and T are all output to an identical interface and can be used to control power–on or power–off of the machine. For this purpose, the axis control interface of the PMC is used, which differs from that used for the miscellaneous functions for the normal NC program. The following M codes, used to control the spindle, are automatically output during the G84 (tapping) or G86 (boring) cycle:
M03: Forward spindle rotation
M04: Reverse spindle rotation
M05: Spindle stop
T** to T(** + 9), where ** is the number specified in parameter 8257, are used as the codes of the auxiliary functions to adjust the tool offset.

Example)
T50 to T59 if parameter 8257 is set to 50

1. An M, S, or T code must not be specified in a block containing another move command. The M, S, and T codes must not be specified in an identical block.
2. Usually, normal NC operation and B–axis operation are independent of each other. Synchronization between operations can be established by coordinating the miscellaneous functions of the normal NC program and B–axis operation program.

(Normal NC operation) (Registered B–axis operation)
: :
M11 ; G00 B111 ;
G01 X999 : G01 B222 ;
G28 Z777 ; G28 ;
M50 ; M50 ;
G00 X666 ; G81 B444 R111 F222 ;
: :

Upon receiving M50 of both the normal NC program and the B–axis program, the PMC ladder outputs the completion signals (FIN) for the two miscellaneous functions. G00 X666 of the normal NC program and G81 B444 R111 F222 of the B–axis program are executed simultaneously.

Custom macro
Custom macro variables (local variables, common variables, system variables #****) can be used in an operation program between G101, G102, or G103 and G100.

1. The value of the macro variable is calculated not from the data existing upon execution of the B–axis operation, but from the data existing at registration of the operation program.
2. An instruction that causes a branch to a location beyond the range of G101, G102, or G103 to G100 is processed without being checked.
3. In the two–path control mode, tool posts 1 and 2 use different macro variables.

- **Operation program**

When a new operation program is registered, the previous operation program is automatically deleted.
If an error is detected in an operation program to be registered, the program is initialized but is not registered.

- **Modal**

In the same way as a normal NC program, the B–axis operation program can use the following as modal data: modal G codes, F codes, and P, Q, and F codes in the canned cycle. These codes do not affect the modal information of the normal NC program. When a B–axis operation program is started (by G101, G102, or G103), the initial modal data is set for the program. It is not affected by the previous modal information.

Example)
:
G01 X10. F1000 ;
G101 (G102, G103) ; 2
B10. ; 3
G01 B–10. F500 ; 4
G100 ; 5
X–10. ; 6
:

Irrespective of the modal information for normal operation (G01 specified in block), block 3 specifies G00 if the MDG bit (bit 1 of parameter 8241) is set to 0, or G01 if the MDG bit is set to 1.
Block 6 causes movement with F1000, specified in block 1.

- **Operation start command**

The MST bit (bit 7 of parameter 8240) specifies the method used to start the B–axis operation as described below:
If the MST bit is set to 1, the B–axis operation is started when the M code to start the operation is executed.
If the MST bit is set to 0, the B–axis operation is started when the M code used to start the operation is executed and the PMC outputs the miscellaneous function completion signal (FIN).
Up to five M codes for starting the programs can be stored. The programs corresponding to these M codes are executed in succession. (In two–path control mode, up to five codes can be stored for each tool post.)

Example)

When the first, second, and third programs are started by M40, M41, and M42, respectively

```
O1234. ;
    :
    :
M40 ;   M code for starting the first program
M41 ;   M code for starting the second program
M42 ;   M code for starting the third program
M40 ;   M code for starting the first program
M41 ;   M code for starting the second program
    :
    :
M30 ;
```

As M41 is specified while the program started by M40 is being executed, the second program is automatically started upon termination of the first program.
M42, M40, and M41, specified during execution of the first program, are stored such that the corresponding programs are executed in the same order as that in which the M codes are specified.
If six or more M codes for starting the programs are specified while a program is being executed, P/S alarm 5038 is output.
In two–path control mode, the M code specified for tool post 1 starts the B–axis program registered for tool post 1. The M code specified for tool post 2 starts the B–axis program registered for tool post 2.

- **Specifying absolute or incremental mode**

The amount of travel along the B–axis can be specified in either absolute or incremental mode. In absolute mode, the end point of travel along the B–axis is programmed. In incremental mode, the amount of travel along the B–axis is programmed directly.
The ABS bit (bit 6 of parameter 8240) is used to set absolute or incremental mode. When the ABS bit is set to 1, absolute mode is selected. When the ABS bit is set to 0, incremental mode is selected. The mode is specified with this parameter when the program is registered.

- **Specifying a tool offset**

The T**; command shifts the end point of the specified B–axis travel, in either the positive or negative direction, by the amount specified with the B–axis offset screen. If this function is used to set the difference between the programmed tool position and actual tool position in machining, the program need not be modified to correct the tool position.
The value specified with parameter 8257 is assigned to the auxiliary function to cancel the offset. The subsequent nine numbers are assigned to the tool offset functions. These auxiliary function numbers are displayed on the B–axis offset screen. For details, see ”OPERATION.”

- **Single–motion operation**

If a G110 block is specified, a single–motion operation along the B–axis can be specified and executed. In single–motion operation mode, a single block results in a single operation. The single–motion operation is executed immediately provided if it is specified before the B–axis operation is started. If the operation is specified while a registered program is being executed, the operation is executed once that program has terminated.
After the specified single–motion operation has been executed, the next block is executed.

```
  :
G110 G01 B100. F200 ;    Block for single–motion
                         operation along B–axis
G00 X100. Z20. ;
  :
```

- **Program memory**

An operation program is registered in program memory as a series of different blocks of the move, dwell, auxiliary, and other functions. Program memory can hold a desired number of blocks, up to a maximum of 65535 blocks for each program. If the program memory contains no free space when an attempt is made to register a B–axis program, P/S alarm 5033 is output. Six blocks require 80 characters of program memory. A canned cycle (G81 to G86) is also registered as a series of blocks, such as travel and dwell.
The entire program memory is backed up by battery. The programs registered in program memory are thus retained even after the system power is turned off. After turning the system power on, the operation can be started simply by specifying the M code for starting the program.

Example)

```
  :
G101 ;
G00 B10. ; . . . . . . . . . . . . . . . . . . One block
G04 P1500 ; . . . . . . . . . . . . . . . . . One block
G81 B20. R50. F600 ; . . . . . . . . Three blocks
G28 ; . . . . . . . . . . . . . . . . . . . . . . One block
M15 ; . . . . . . . . . . . . . . . . . . . . . One block
G100 ;
  :                                (Total 7 blocks)
```

- **Reset**

When the NC is reset by pressing the MDI reset key or by the issue of an external reset signal, reset and rewind signal, or emergency stop, B–axis control is also reset. The PMC interface signal can reset only B–axis control. For details, refer to the manual supplied by the machine tool manufacturer.

- **PMC–controlled axis**

A B–axis operation can be executed only when the B–axis can be controlled by the PMC. For details, refer to the manual supplied by the machine tool builder.

Limitations

- **Single–motion operation**

1. Only a single–motion operation can be specified with G110.

   G110 G00 B100. ; . . . . OK
   G110 G28 ; . . . . OK
   G110 G81 B100. R150.0 F100 ; . . . P/S alarm No.5034

2. A canned cycle (G81 to G86), and other operations containing multiple motions, cannot be specified with G110.
   If an inhibited operation is specified, P/S alarm No.5034 is output.
3. modal information specified with G110 does not affect the subsequent blocks. In the G110 block, the initial modal value specified at the start of the operation becomes valid, irrespective of the modal information specified the previous blocks.

   Example)

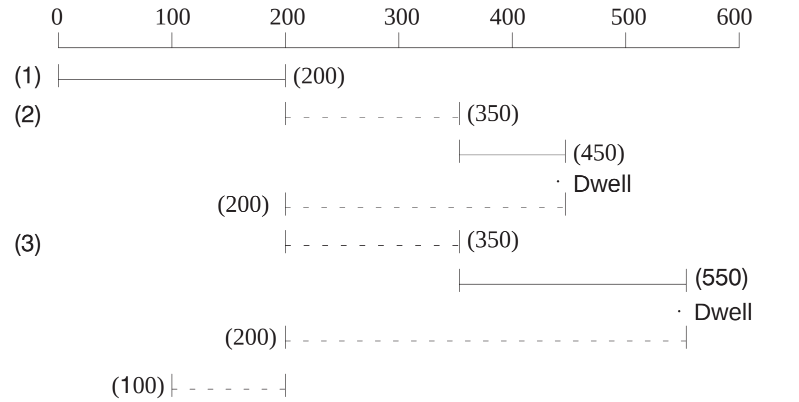
   When the MDG bit (bit 1 of parameter 8241) is set to 1 and the MDF bit (bit 2 of parameter 8241) is set to 1
   G98 G00 X100. F1000 ; . . . . (1)
   G110 B200. F2 ; . . . . (2)
   X200. ; . . . . (3)
   G01 X200. ; . . . . (4)

   Block (2) instigates cutting feed (G01) at 2.0 mm/rev (G99).
   Block (3) instigates rapid traverse (G00).
   Block (4) instigates cutting feed (G01) at 1000 mm/min (G98).
4. During tool–tip radius compensation, two or more G110 blocks cannot be specified in succession. If such blocks are specified in succession, P/S alarm No. 504 is output. To specify two or more G110 blocks in succession for a B–axis operation, register the blocks as a program with G101, G102, or G103 and G100.

Examples

- **Absolute or incremental mode**

Absolute or incremental mode



(- - - Rapid traverse —— Cutting feed ·Dwell (***) Absolute value)

Incremental mode

```
  G101 (G012, 103) ;
(1) G01 B200. F100 ;
(2) G82 B100. R150. P5000 F200 ;
(3)     B200. R150. P5000 ;
(4) G00 B–100. ;
   G100 ;
   M** ;
   M30 ;
```

Absolute mode

```
  G101 (G012, G103) ;
(1) G01 B200. F100 ;
(2) G82 B450. R350. P5000 F200 ;
(3)     B550. R350. P5000 ;
(4) G00 B100. ;
   G100 ;
   M** ;
   M30 ;
```

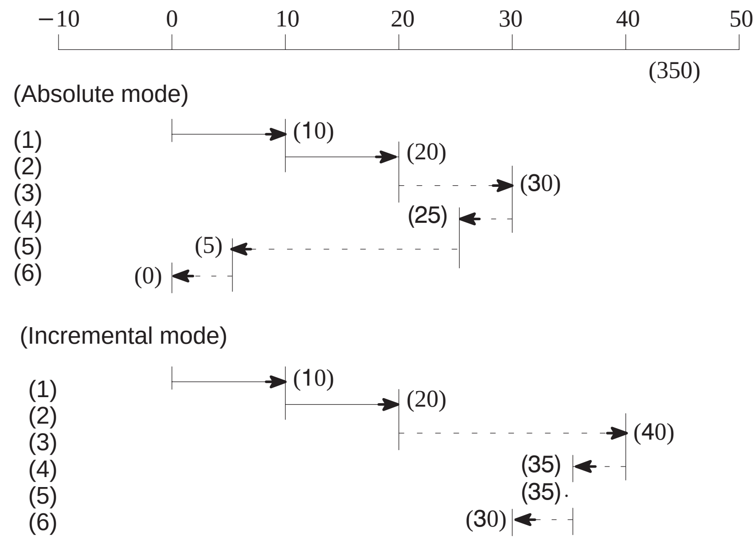
- **Tool posts 1 and 2**

If a single axis is used as the common B–axis of the two tool posts in two–path control, tool posts 1 and 2 share the B coordinate.
For example, after program 1 for tool post 1 and program 2 for tool post 2 are executed in that order, the total travel along the B–axis appears to be +100.

```
<Program 1>
  G101 ;
     :
  G00 B200. ; (Absolute mode)
  G100 ;
     :
  M30 ;

<Program 2>
  G101 ;
  G00 B300. ; (Absolute mode)
     :
  G100 ;
     :
  M30 ;
```

• Tool offset

Example)
When parameter 8257 is set to 50
Auxiliary function used to cancel the offset: T50
Auxiliary functions used to adjust a tool offset: T51 to T59



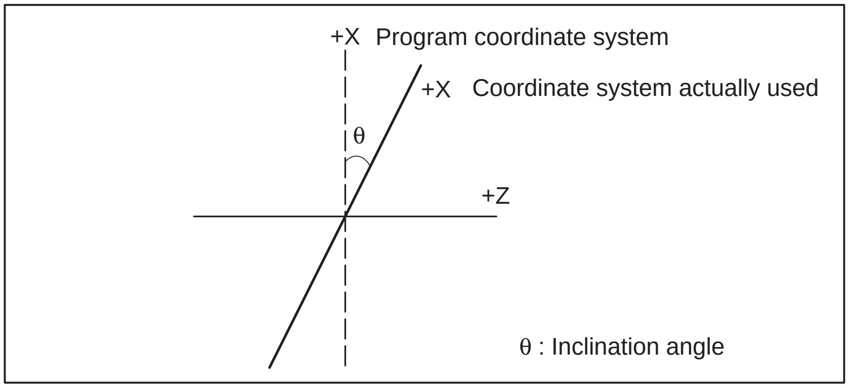
Program

```
    G101 (G012, G103) ;
(1) G01 B10. F100 ;
(2) T51 ;
(3) G00 B20. ;
(4) T52 ;
(5) B0. ;
(6) T50 ;
    G100 ;

    M**;
```

Where the offset of T51 is 10.0 and the offset of T52 is 5.0

20.7 ANGULAR AXIS CONTROL

When the X–axis makes an angle other than 90° with the Z–axis, the inclined axis control function controls the distance traveled along each axis according to the inclination angle. A program, when created, assumes that the X–axis and Z–axis intersect at right angles. However, the actual distance traveled is controlled according to an inclination angle.



Explanations

The distance traveled along each axis is controlled according to the formulas described below.
The distance to be traveled along the X–axis is determined by the following formula when it is specified with a diameter :

$$X = \frac{Xp}{2\cos\theta}$$

The distance traveled along the Z–axis is corrected by the inclination of the X–axis, and is determined by the following formula:

$$Za = Zp - \frac{1}{2} Xp \tan\theta$$

The feedrate is determined as described below. The speed component along the X–axis is determined by the following formula:

$$Fa = \frac{Fp}{\cos\theta}$$

Xa, Za, Fa: Actual distance and speed
Xp, Zp, Fp: Programmed distance and speed

• Method of use

Parameter AAC (No. 8200#0) enables or disables the inclined axis control function. If the function is enabled, the distance traveled along each axis is controlled according to an inclination angle (No. 8210).
Parameter AZR (No. 8200#2) enables X–axis manual reference point return only with a distance along the X–axis.
Bit 5 (NOZAGC) of signal G0063 can enable slanted axis control only for the X–axis are converted to those along the slanted axis without affecting commands along the Z–axis.

• Absolute and relative position display

An absolute and a relative position are indicated in the programmed Cartesian coordinate system. Machine position display

• Machine position display

A machine position indication is provided in the machine coordinate system where an actual movement is taking place according to an inclination angle. However, when inch/metric conversion is performed, a position is indicated which incorporates inch/metric conversion applied to the results of inclination angle operation.

Notes

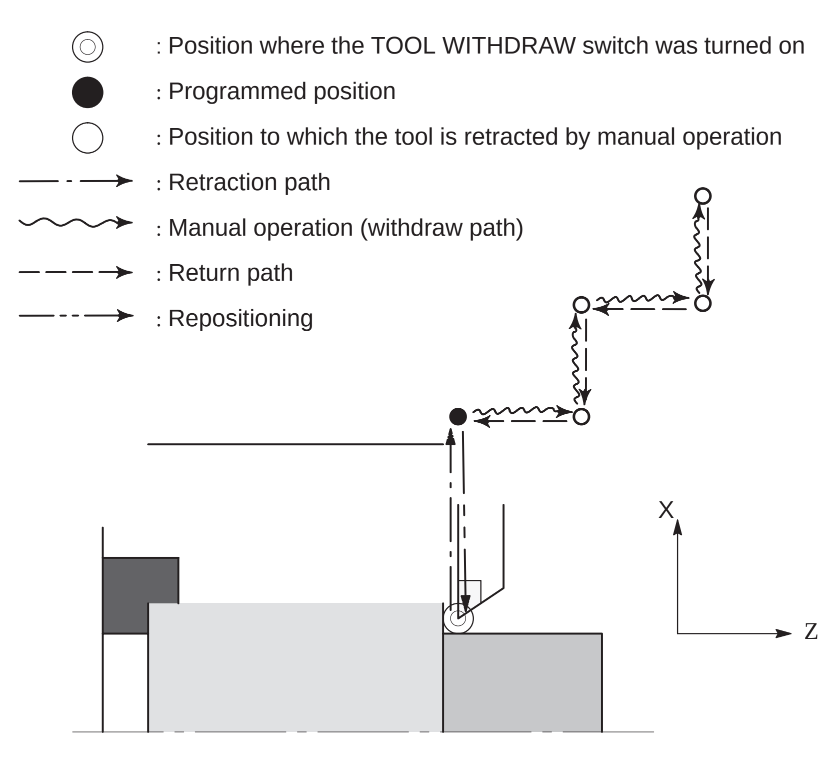
1. After inclined axis control parameter setting, be sure to perform manual reference point return operation.
2. If a movement along the Z–axis occurs in X–axis manual reference point return operation, be sure to perform reference point return operation starting with the X–axis.
3. If an inclination angle close to 0° or ±90° is set, an error can occur. A range from ±20° to ±60° should be used.
4. Before a Z–axis reference point return check (G37) can be made, X–axis reference point return operation must be completed.
5. After moving the tool along the X–axis with bit 5 (NOZAGC) of signal G0063 set to 1, be sure to perform manual reference position return.

20.8 TOOL WITHDRAWAL AND RETURN (G10.6)

To replace the tool damaged during machining or to check the status of machining, the tool can be withdrawn from a workpiece. The tool can then be advanced again to restart machining efficiently.
The tool withdrawal and return operation consists of the following four steps:

- Retract
  The tool is retracted to a predefined position using the TOOL WITHDRAW switch.
- Withdrawal
  The tool is moved to the tool–change position manually.
- Return
  The tool returns to the retract position.
- Repositioning
  The tool returns to the interrupted position.

For the tool withdrawal and return operations, see Section 4.8 in ”Operation.”



Format

Specify a retraction axis and distance in the following format:

G10.6 IP_ ;
IP_ : In incremental mode, retraction distance from the position where the retract signal is turned on
In the absolute mode, retraction distance to an absolute position

Explanations

• Retraction

When the TOOL WITHDRAW switch on the machine operator's panel is turned on during automatic operation or in the automatic operation stop or hold state, the tool is retracted the length of the programmed retraction distance. This operation is called retraction. The position at which retraction is completed is called the retraction position. Upon completion of retraction, the RETRACT POSITION LED on the machine operator's panel goes on.

When the TOOL WITHDRAW switch is turned on during execution of a block in automatic operation, execution of the block is interrupted immediately and the tool is retracted. After retraction is completed, the system enters the automatic operation hold state.

If the retraction distance and direction are not programmed, retraction is not performed. In this state, the tool can be withdrawn and returned.

When the TOOL WITHDRAW switch is turned on in the automatic operation stop or hold state, the tool is retracted, then the automatic operation stop or hold state is entered again.

When the TOOL WITHDRAW switch is turned on, the tool withdraw mode is set. When the tool withdraw mode is set, the TOOL BEING WITHDRAWN LED on the machine operator's panel goes on.

• Withdrawal

When the manual mode is set, the tool can be moved manually (manual continuous feed or manual handle feed) to replace the tool or measure a machined workpiece. This operation is called a withdrawal. The tool withdrawal path is automatically memorized by the CNC.

• Return

When the mode is returned to automatic operation mode and the TOOL RETURN switch on the machine operator's panel is turned off, the CNC automatically moves the tool to the retraction position by tracing the manually–moved tool path backwards. This operation is called a return. Upon completion of a return to the retraction position, the RETRACTIONS POSITION LED comes on.

• Repositioning

When the cycle start button is pressed while the tool is in the retraction position, the tool moves to the position where the TOOL WITHDRAW switch was turned on. This operation is called repositioning. Upon completion of repositioning, the TOOL BEING WITHDRAWN LED is turned off, indicating that the tool withdrawal mode has terminated. Operation after completion of repositioning depends on the automatic operation state when the tool withdrawal mode is set.

(1) When the tool withdrawal mode is set during automatic operation, operation is resumed after completion of repositioning.

(2) When the tool withdrawal mode is set when automatic operation is held or stopped, the original automatic operation hold or stop state is set after completion of repositioning. When the cycle start button is pressed again, automatic operation is resumed.

Limitations

- **offset**

If the origin, presetting, or workpiece offset is changed after retraction is specified with G10.6 in absolute mode, the change is not reflected in the retraction position. After such changes are made, the retraction position must be respecified with G10.6.
When the tool is damaged, automatic operation can be interrupted with a tool withdrawal and return operation in order to replace the tool. Note that if the offset value is changed after tool replacement, the change is ignored when automatic operation is resumed from the start point or other point in the interrupted block.

- **Machine lock, mirror image, and scaling**

When withdrawing the tool manually in the tool withdrawal mode, never use the machine lock, mirror–image, or scaling function.

- **Threading**

Tool withdrawal and return operation cannot be performed during threading.

- **Drilling canned cycle**

Tool withdrawal and return operation cannot be performed during a drilling canned cycle.

- **Reset**

Upon reset, the retraction data specified in G10.6 is cleared. Retraction data needs to be specified again.

- **Retraction command**

The tool withdrawal and return function is enabled even when the retraction command is not specified. In this case, retraction and repositioning are not performed.

Notes

> **Notes**
> The retraction axis and retraction distance specified in G10.6 need to be changed in an appropriate block according to the figure being machined. Be very careful when specifying the retraction distance; an incorrect retraction distance may damage the workpiece, machine, or tool.

21 TWO–PATH CONTROL FUNCTION

21.1 GENERAL

Two–path control can be used with a lathe that supports simultaneous cutting by its two independently operating tool posts.

- **Application to latheswith one spindle and two tool posts**

Series 16–TB (two–path control) can be used for a lathe that machines one workpiece attached to one spindle with two tool posts simultaneously. For example, while one tool post is performing outer surface machining, the other tool post can perform inner surface machining, thus reducing machining time dramatically.

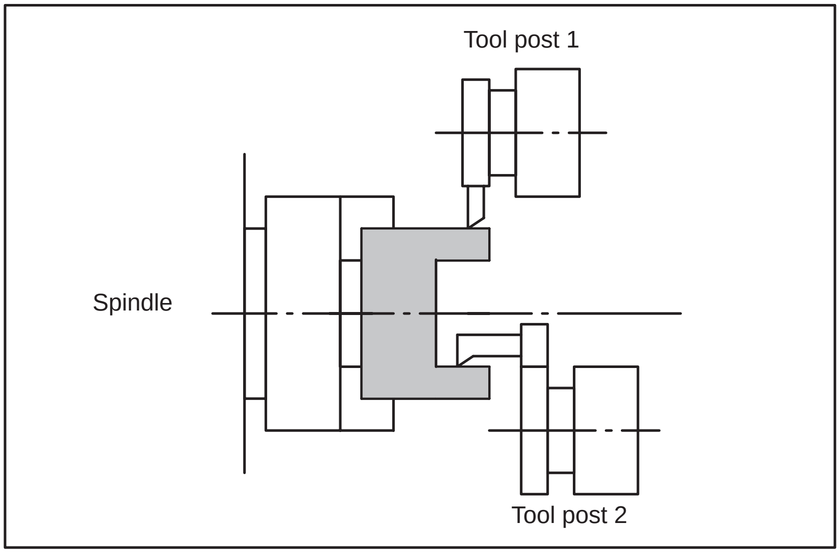


Fig.21.1(a) Application to lathes with one spindle and tow tool posts

- **Application to lathes with two spindles and two tool posts**

Series 16–TB (two–path control) can be used for a lathe that machines a workpiece attached to each of two spindles with two tool posts simultaneously. In this case, each tool post operates independently of each other as if two lathes were used, thus improving productivity.

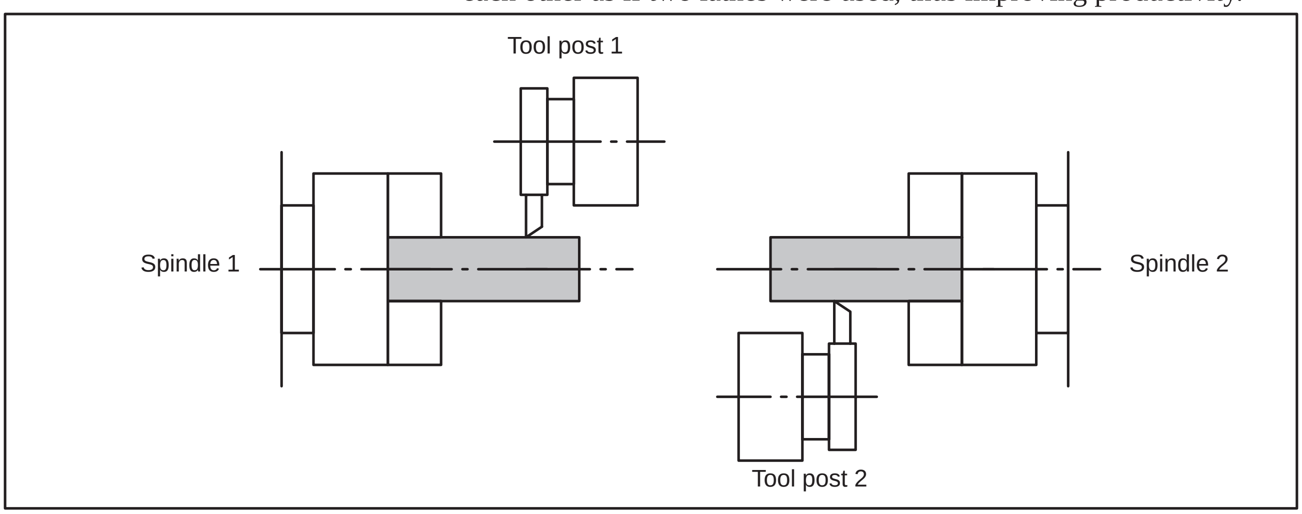


Fig. 21.1(b) Application to lathes with two spindles and two tool posts

- **Controlling two tool posts independently at the same time**

The operations of two tool posts are programmed independently of each other, and each program is stored in program memory for each tool post. When automatic operation is to be performed, each tool post is activated after selecting a program for machining with tool post 1 and a program for machining with tool post 2 from the programs stored in program memory for each tool post. Then the programs selected for the tool posts are executed independently at the same time. When tool post 1 and tool post 2 need to wait for each other during machining, the waiting function is available (Section **21.2**)

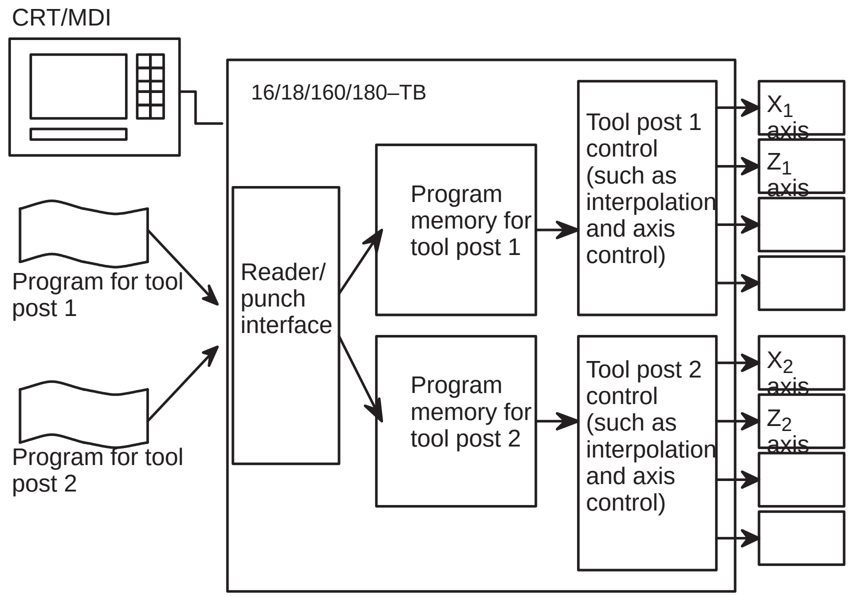


Fig. 21.1(c) Controlling two tool posts independently at the same time

Just one CRT/MDI is provided for the two tool posts. Before operation and display on the CRT/MDI, the tool post selection signal is used to switch between the two tool posts.

Notes

> **Notes**
> Simultaneous operation of the two tool posts or the operation of only a single tool post can be selected by pressing a key on the machine operator's panel. For details, refer to the manual supplied by the machine tool builder.

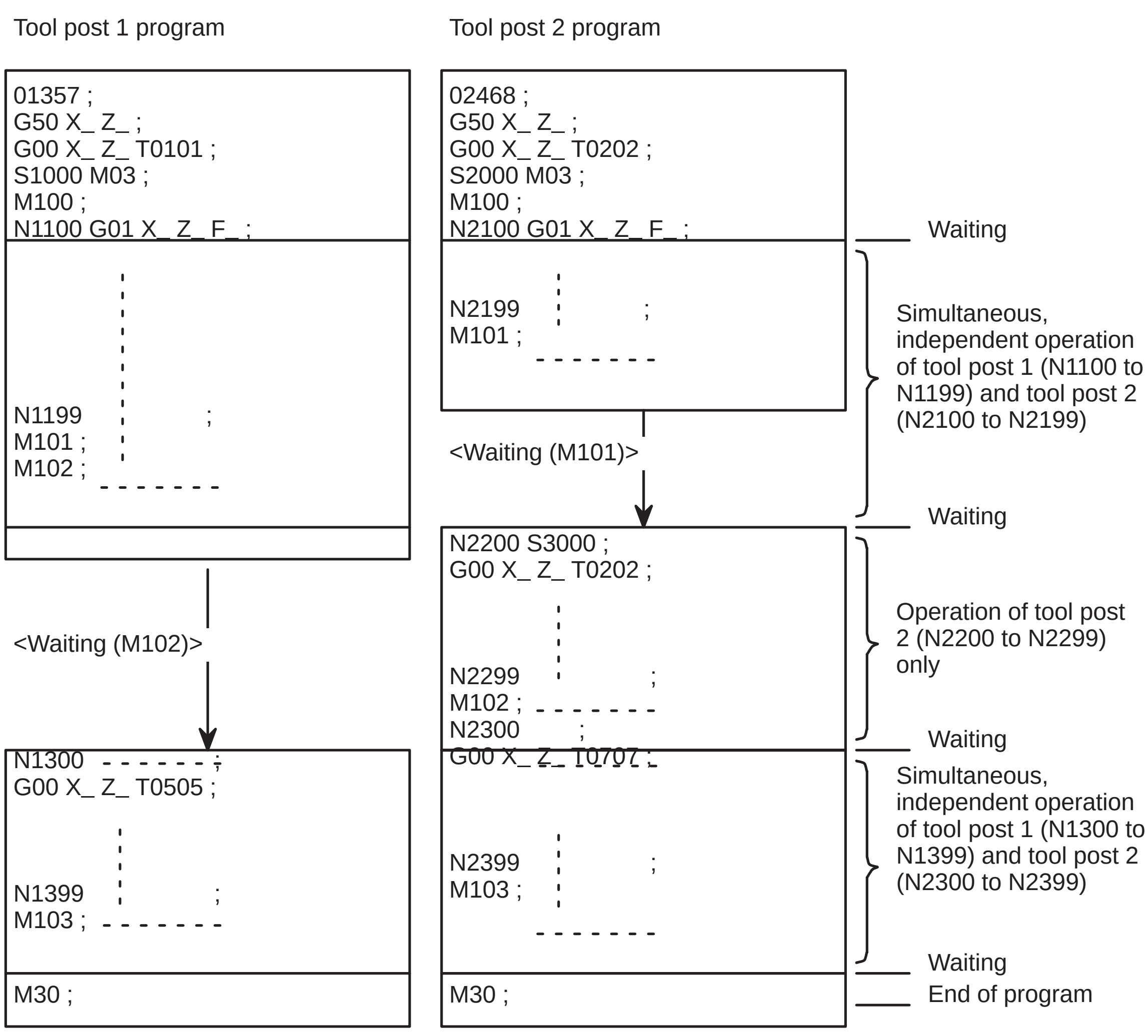
21.2 WAITING FOR TOOL POSTS

Explanations

Control based on M codes is used to cause one tool post to wait for the other during machining. By specifying an M code in a machining program for each tool post, the two tool posts can wait for each other at a specified block. When an M code for waiting is specified in a block for one tool post during automatic operation, the other tool post waits for the same M code to be specified before staring the execution of the next block. This function is called the tool post waiting function.
A range of M codes used as M codes for waiting is to be set in the parameters (Nos. 8110 and 8111) before hand.

Example

M100 to M103 are used as M codes for waiting.
Parameter setting:No. 8110=100(Minimum M code for waiting: M100)
No. 8111=103(Maximum M code for waiting: M103)



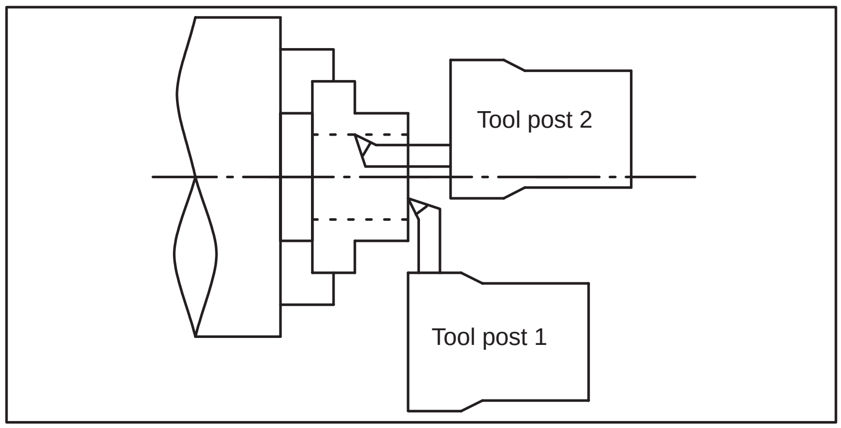
Notes

1. An M code for waiting must always be specified in a single block.
2. If one tool post is waiting because of an M code for waiting specified, and a different M code for waiting is specified with the other tool post, an P/S alarm (No. 160) is raised, In this case, both tool posts stop operation.
3. PMC–CNC interface
   Unlike other M codes, the M code for waiting is not output to the PMC.
4. Operation of a single tool post
   If the operation of a single tool post is required, the M code for waiting need not be deleted. By using the NOWT signal to specify that waiting be ignored (G0063, #1), the M code for waiting in a machining program can be ignored. For details, refer to the manual supplied by the machine tool builder.

21.3 Tool Post Interface Check

21.3.1 General

When two tool posts machine the same workpiece simultaneously, the tool posts can approach each other very closely. If the two tool posts interfere with each other due to a program error or any other setting error, a serious damage such as a tool or machine destruction can occur.
The function ” tool post interference check” is available which can decelerate and stop the two tool posts before the tool posts interfere with each other due to an incorrect command.



The contours of the two tool posts are checked to determine whether or not an interference occurs.

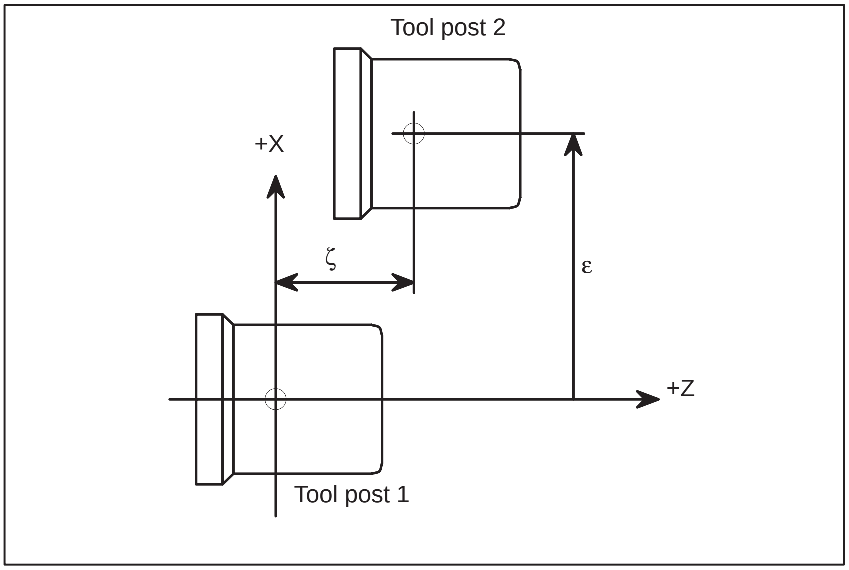
21.3.2 Data Setting for the Tool Post Interference Check Function

To make a tool post interference check, data including the relationships between the two tool posts and interference forbidden areas (that is, tool shapes) needs to be set. The method of such data setting is described below.
With the tool post interference check function, whether or not the two tool posts interfere with each other is determined by checking if the interference forbidden areas (based on the interference forbidden areas of the currently selected tools) of the tool posts overlap each other after the movement of the tool posts.

Explanations

- **Position setting for reference points of two tool posts**

When reference point return operation is completed with all axes (X1,Z1, X2, Z2), the reference point of tool post 1 is set at the origin of the ZX plane coordinate system. At this time, the position of the reference point of tool post 2 is set in a parameter. The next item describes the reference points.



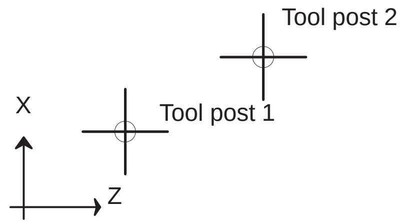
In the ZX plane coordinate system at the origin of which the reference point of tool post 1 is set, set the X coordinate (ε) of the reference point of tool post 2 in parameter No. 8151, and its Z coordinate (ζ) in parameter No.8152 .
The unit of setting is the least command increment. For an axis subject to diameter specification, a diameter value is to be specified.
Measure (ε) and(ζ) when reference point return operation is completed with the four axes (X1, Z1, X2, Z2). When the relative coordinate parameters (Nos.8151 and 8152) of the two tool posts are to be updated, reference point return operation must always be completed with the four axes beforehand. Otherwise, the internally memorized relational positions of the tool posts are not updated to new parameter values.

- **Set the relationship between the coordinate systems of the two tool posts in parameter No.8140**

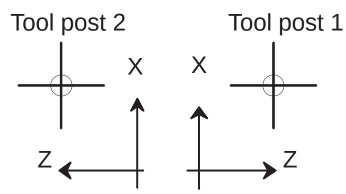
	#7	#6	#5	#4	#3	#2	#1	#0
8140							TY1	TY0

TY0, TY1:Set the relationship between the coordinate systems of the two tool posts, with tool post 1 used as the reference.

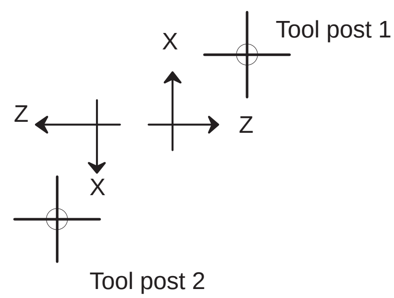
(1) When TY1=0 and TY0=0



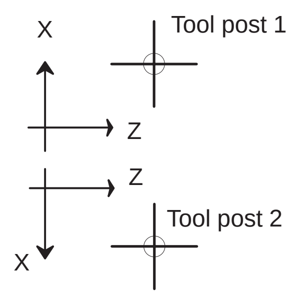
(2) When TY1=0 and TY0=1



(3) When TY1=1and TY0=0



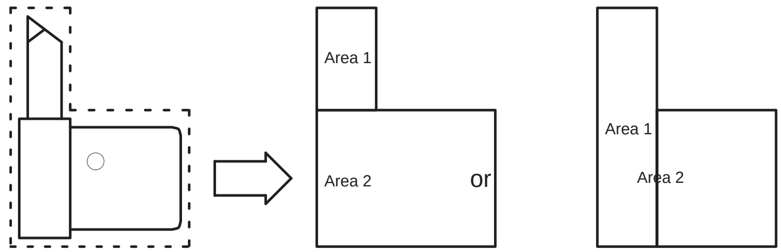
(4) When TY1=1 and TY0=1



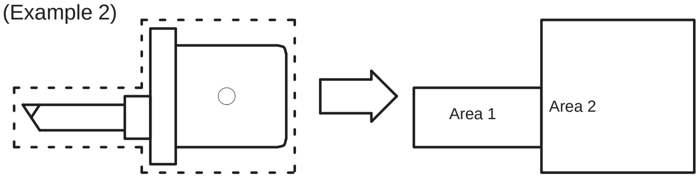
- **Setting of interference forbidden area**

An interference forbidden area is set using a combination of two rectangular areas. Some examples are shown below. The dashed lines indicate interference forbidden areas.

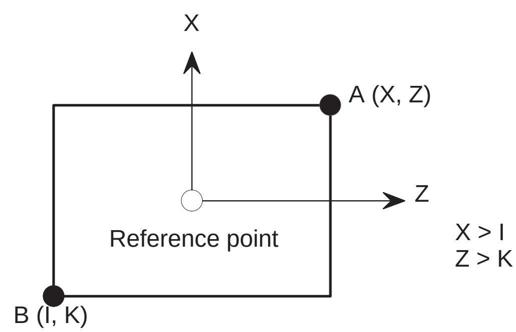
(Example 1)



(Example 2)



The coordinates of the upper and lower ends (points A and B shown below) of each of two rectangles are set, with the reference point of the tool post set as the origin.



See Section **21.3.3** for information about the coordinate setting procedure.

21.3.3 Setting and Display of Interference Forbidden Areas for Tool Post Interference Checking

Explanations

Display and set tool shape data (interference forbidden areas) according to the procedure below.

(1) Press function [OFFSET SETTING] key.

(2) Press the chapter selection soft key **[TOOLFM]**.

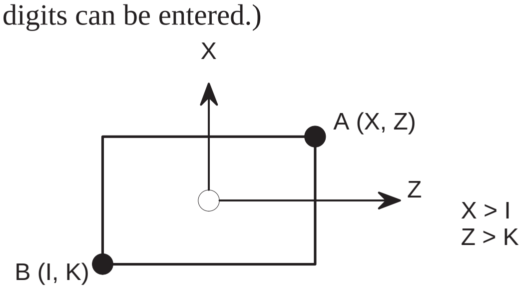
(3) With the tool post selection signal, select a tool post for which interference forbidden areas for tool post interference checking are to be displayed and set.

(4) Display the screen including a tool number for which data is to be set.
Method 1:Select the screen by using the page keys and cursor keys.
Method 2:Enter a desired tool number, then press the soft key **[NO.SRH]**

```
TOOL FORM DATA                        O0001  N00001
  OFFSET NO. = 01
       AREA1                          AREA 2
    X =      20.000            X =  40.000
    Z =    70.000              Z =  70.000
    I =    –10.000             I =  20.000
    K =    –50.000             K =  30.000
  OFFSET NO  = 02
       AREA1                          AREA 2
    X =    80.000              X =  –100.000
    Z =    170.000             Z =    –60.000
    I =    –100.000            I =  –140.000
    K =    –120.000            K =  –120.000


 >_                                    S  0 T0000
 MEM    ****  ***  ***     12 : 02 : 08  HEAD1
[ NO.SRH  ][      ][      ][ +INPUT ][ INPUT ]
```

(5) Move the cursor to a data item to be set, with the cursor move keys.
(When data for point A is to be set, move the cursor to X and Z.
When data for point B is to be set, move the cursor to I and K.)

(6) With the numeric keys, enter the coordinates of point A or B.
(Fraction digits can be entered.)



(7) By pressing the soft key **[INPUT]**, the entered coordinates are set.
(Press the soft key **[+INPUT]** when an entered numeric value is to set after it is added to data already set.)

Notes Tool number
The tool geometry data must be set for each tool number. The tool number here refers to the offset number. When both tool geometry offset and tool wear offset are used, the tool number corresponds to the wear offset number. To use two or more offset numbers for the same tool, the same data for the tool must be set two or more times in the tool geometry data.

21.3.4 Conditions for Making a Tool Post Interference Check

A tool post interference check is made when all conditions listed below are satisfied.

(1) Parameter IFE (No.8140#4) for enabling the tool post interference check function is set to 0.

(2) After power is turned on, reference point return operation is completed with all axes (X1,Z1, X2, Z2).(When an absolute–position detector is used, the matching between a machine position and absolute–position detector position must be completed.)

(3) Offset numbers other than 0 are specified using T codes for two tool posts.

(4) When manual mode is used,parameter IFM(No.8140#3) for enabling the tool post interference check function in manual mode is set to 1.
When all conditions for making a tool post interference check are satisfied, the tool–post–interference–check–in–progress signal is output to the PMC.

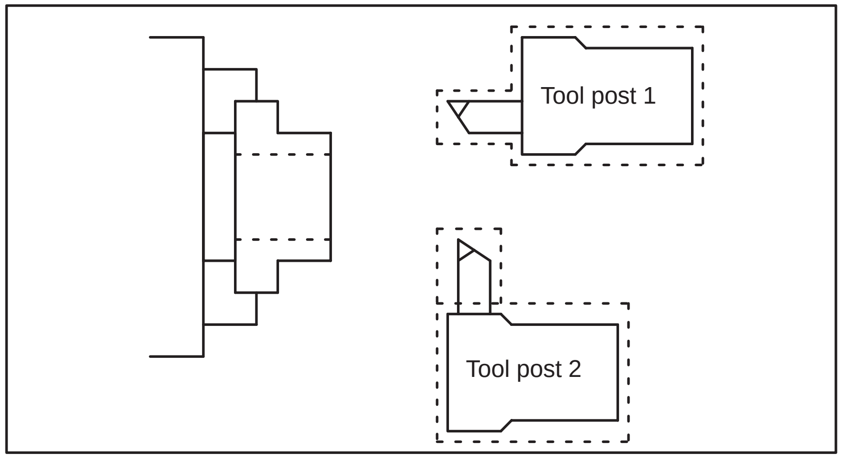
Notes

Notes
The tool post interference check function can be executed only when the number of the tool actually selected agrees with the programmed tool number.
The function cannot be executed correctly if the tool is selected by a manual operation or if no tool selection command is specified after power–on.

21.3.5 Execution of Tool Post Interference Checking

when all conditions described in Section **21.3.4** are satisfied, a tool post interference check is started. When a tool post interference check is made, an interference forbidden area is set for the two tool posts by using the tool shape data corresponding to the currently selected tool numbers.
Then whether the areas interfere with each other is checked.

Explanations



When interface forbidden areas (tool shapes) as indicated by dashed lines are set for tool posts 1 and 2 as shown above, a check is made by determining whether the two interference forbidden areas indicated by dashed lines overlap each other after the movement of the tool posts.
If the two areas interfere with each other an P/S alarm (No. 508 or No. 509) is raised;the two tool posts are decelerated and stopped.
If an interference alarm is raised, a tool post interference alarm signal is output to the PMC.
If an interference alarm is raised by the interference of the two tool posts during automatic operation, switch to manual mode to move the tool posts out of the interference state. Then release the alarm by a reset.
The interference check function can be enabled even in manual mode by setting the parameter (No. 8140#3) to 1. This allows the tool posts interfering with each other to be moved along the axes only in such directions that clear the interference. With this capability, the two tool posts interfering with each other in automatic operation cannot be manually moved by mistake further into the interference forbidden areas after the mode is switched to manual mode to clear the interference, thus providing safety.

Notes

Notes

1. When an alarm is raised, the CNC system and machine system stop with some delay in time.
   So an actual stop position can be closer to the other tool post beyond an interference forbidden position specified using tool shape data. So, for safety, tool shape data a little larger than the actual shape should be set. The extra distance, L, required for this purpose is calculated form a rapid traverse feedrate as follows

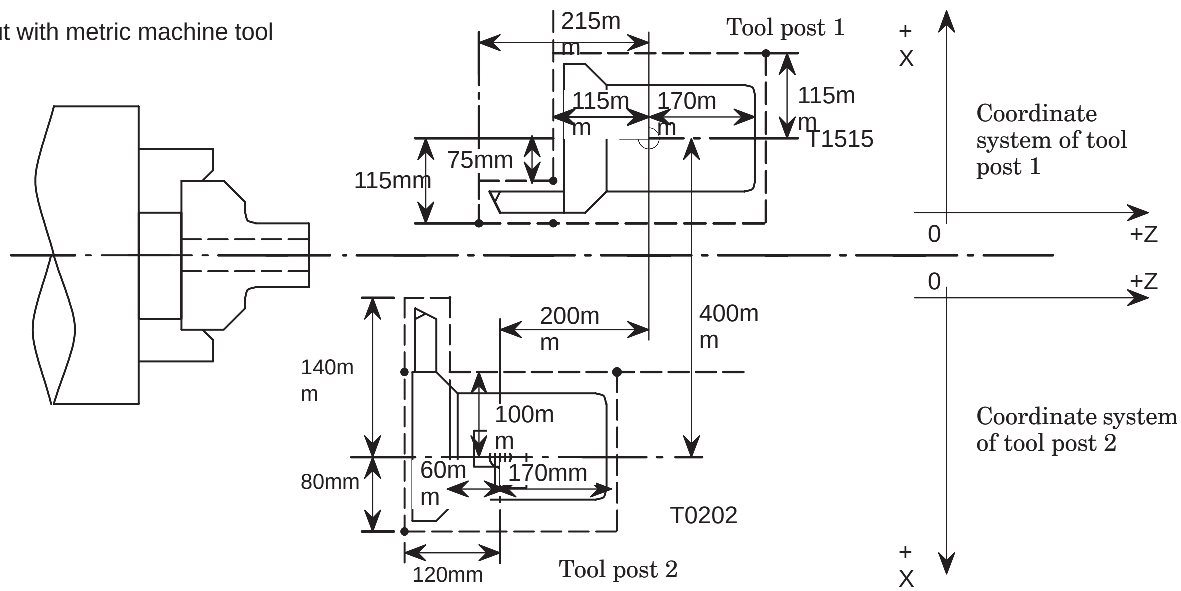
   $$L = (\text{Rapid traverse feedrate}) \times \frac{1}{7500}$$

   For example, when a rapid traverse feedrate of 15 m/min is used, L=2mm.
2. When parameters and interface forbidden areas (Section 23.3.2) are set to use the interference check function, be sure to check that correct interference forbidden areas are set. For this purpose, set manual mode, and cause the tool posts to interfere with each other in various directions.

21.3.6 Example of Making a Tool Post Interference Check

Explanations

Metric input with metric machine tool



The coordinate systems shown on the right of the figure above are the ZX plane coordinate systems of tool posts 1 and 2. For clarity, the coordinate systems are shifted;actually, the origins of the coordinate systems must match the machine zero points.

Assume the machine configuration shown above. Assume also that offset number 02 is assigned to tool post 1, and offset number 15 is assigned to tool post 2.

Suppose that the figure represents the state of reference point return operation completed with all axes (X1,Z1, X2, Z2). Then set –800 mm(diameter) and –200 mm in parameter Nos. 8151 and 8152, respectively.

The positional relationship of the two tool posts matches type (4) indicated in Section **21.3.2(b)**. So set parameters TY0 and TY1(No.8140#0,#1) as follows:

Parameter TY1 (No.8140#1)=1

Parameter TY0 (No.8140#0)=1

Then set tool shape data (interference forbidden area) for each tool post.

The figures below show the setting of data for tool number 02 assigned to tool post 1 and for tool number 15 assigned to tool post 2.

```
TOOL FORM DATA                     O0001  N00001
  OFFSET NO.   = 01
     AREA 1                          AREA 2
   X=        20.000                X=   40.000
   Z=       70.000                 Z=   70.000
   I=   -10.000                    I=   20.000
   K=   -50.000                    K=   30.000
  OFFSET NO.    = 02
     AREA 1                          AREA 2
   X=      80.000                  X=   -100.000
   Z=     170.000                  Z=     -60.000
   I=   -100.000                   J=   -140.000
   K=   -120.000                   K=   -120.000

 >_                                  S  0 T0000
 MEM     ****  ***  ***          12:02:08   HEAD 1
[ NO.SRH ][      ][      ][ +INPUT ][ INPUT ]
```

```
TOOL FORM DATA                     O0001  N00001
  OFFSET NO.   = 15
     AREA 1                          AREA 2
   X=        115.000               X=     -75.000
   Z=     170.000                  Z=   -115.000
   I=   -115.000                   I=   -115.000
   K=   -115.000                   K=   -215.000
  OFFSET NO.   = 16
     AREA 1                          AREA 2
   X=      0.000                   X=      0.000
   Z=      0.000                   Z=      0.000
   I=      0.000                   I=      0.000
   K=      0.000                   K=      0.000

 >_                                  S  0 T0000
 MEM     ****  ***  ***          12:02:36   HEAD 2
[ NO.SRH ][      ][      ][ +INPUT ][ INPUT ]
```

Set data for other tools similarly. A preparation for an interference check is completed when data has been set for all tools. Turn on power. Then, an interference check is started when a T code is specified with each tool post
after reference point return operation is completed with all of the four axes (X1, Z1, X2, Z2).

21.4 BALANCE CUT (G68,G69)

When a thin workpiece is to be machined as shown below, a precision machining can be achieved by machining each side of the workpiece with a tool simultaneously;this function can prevent the workpiece from warpage that can result when only one side is machined at a time. When both sides are machined at the same time, the movement of one tool must be in phase with that of the other tool. Otherwise, the workpiece can vibrate, resulting in poor machining. With this function, the movement of one tool post can be easily synchronized with that of the other tool post.

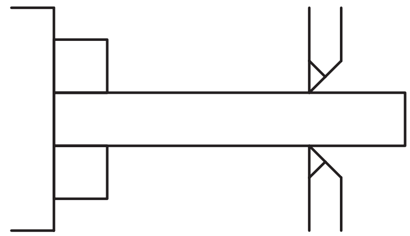


Fig. 21.4(a) Balance cut

Explanations

When G68 is specified in the programs for both tool post 1 and tool post 2, the pulse distribution of tool post 1 is synchronized with that of tool post 2 to start balance cutting. Thus the two tool posts can move exactly at the same time to allow balance cutting.

G code	Meaning
G68	Balance cut mode
G69	Balance cut mode cancel

In the balance cut mode, balance cutting is performed only when a move command is specified for both tool posts. Balance cutting is performed even when different axes are specified for each tool post or an offset move command is specified. G68 or G69 must be specified in a single block. (Otherwise, a P/S alarm (No. 163) is raised. When G68 or G69 is specified with one tool post, the tool post does not move until the execution for the other tool post proceeds to G68 or G69. And if cutting is specified with one tool post in the balance cut mode, the tool post does not move until the execution of or the other tool post proceeds to a cutting command.

Notes
Balance cut only starts cutting feed on both tool posts at the same time; it does not maintain synchronization thereafter. To synchronize all the movements of both tool posts, the setting for both tool posts, such as the travel distance and feedrate, must be the same.

Example

Tool post 1 program	Tool post 2 program	
G68 ;	G68 ;	Balance cut mode
G01Z100.0 ;	G01Z100.0 ;	Balance cut
Z0 ;	Z0 ;	Balance cut
G69 ;	G69 ;	Balance cut mode cancel

Notes

1 Time delay before the pulse distribution of both tool posts is started is 2 msec or shorter.

2 In the balance cut mode, synchronization is established at the start of a move block, so movement can momentarily stop.

3 If feed hold operation is performed during balance cutting using both tool posts, balance cut processing is not performed at restart time, it is performed when the next move command is specified for both tool posts.

4 Balance cutting is not performed in dry run or machine lock state.

5 When rapid traverse operation is specified, balance cut processing is not performed.

6 A workpiece for which thread cutting has been performed in the balance cut mode cannot be subjected to thread cutting in the cancel mode. Thread cutting starts at a different position.

7 The cancel mode (G69) is set by a reset.

8 When the option ”mirror image for double turrets” is selected, the balance cut function cannot be used.

21.5 MEMORY COMMON TO TOOL POSTS

A machine with two tool posts has different custom macro common variables and tool compensation memory areas for tool posts 1 and 2. Tool posts 1 and 2 can share the custom macro common variables and tool compensation memory areas provided certain parameters are specified accordingly.

Explanations

- **Custom macro common variables**

Tool posts 1 and 2 can share all or part of custom macro common variables #100 to #149 and #500 to #531, provided parameters 6036 and 6037 are specified accordingly. (The data for the shared variables can be written or read from either tool post.) See Section 16.1 of Part II.

- **Tool compensation memory**

Tool post 2 can reference or specify the data in the tool compensation memory area of tool post 1, provided the CMF bit (bit 5 of parameter 8100) is specified accordingly. This can be executed only when tool posts 1 and 2 have identical data for tool compensation (number of groups, number of columns, unit system, etc.).

21.6 SPINDLE CONTROL IN TWO–PATH CONTROL

The two–path control function supports two spindle interfaces. Thus, 16–TB can control a lathe that simultaneously machines a workpiece attached to one spindle with two tool posts, or can control a lathe that simultaneously machines a workpiece attached to each of two spindles with two tool posts.
The former spindle control is referred to as 1–spindle control, and the latter is referred to as 2–spindle control.
Parameter 2SP (No.3703#0) is used to select 1–spindle control or 2–spindle control.

Explanations

- **1–spindle control**

One spindle is controlled by programmed commands for tool post 1 or tool post 2. Programmed commands(Note 1) for the spindle can be specified from either tool post. However, a spindle speed output selection signal (Note 2) determines which commands from the two tool posts are valid. The spindle is controlled according to the commands from a tool post selected by the signal.
A feedback pulse signal from the position coder attache to the spindle is applied to both tool posts. Such a feedback pulse signal is used for processing such as thread cutting and feed per rotation with each tool post.

- **2–spindle control**

Two spindles, spindle 1 and spindle 2 (Note3), are controlled independently of each other according to programmed commands (Note 1) for each tool post. Usually, programmed commands for tool post 1 are used to control spindle 1, and programmed commands for tool post 2 are used to control spindle 2. Feedback pulse signals from the position coders attached to spindle 1 and spindle 2 are applied to tool post 1 and tool post 2, respectively.
The spindle speed output selection signal (Note 2) can be used to specify which spindle must be controlled by programmed commands for which tool post. In addition, a spindle feedback input selection signal (Note 2) can be used to specify which spindle must be controlled by programmed commands for which tool post. In addition, a spindle feedback input selection signal (Note 2) can be used to specify which tool post must receieve a feedback signal from which spindle. Thus, tool post 1 can control spindle 2, and tool post 2 can control spindle 1.

Notes

1. The programmed commands for spindles include the following.
   · S code to specify a spindle speed
   · M03 (forward spindle rotation), M04 (reverse spindle rotation)
   · Commands for constant surface speed control (G96, G97, S code to specify surface speeds, commands to specify maximum spindle speeds)
2. Refer to the ”FANUC Series 16/18/160/180–MODEL B CONNECTION MANUAL (FUNCTION)” for detailed information about the spindle speed output selection signal and spindle feedback input selection signal. Control over these signals varies from one machine tool builder to another. So be sure to read the relevant manual prepared by the machine tool builder to be familiar with the commands for the spindles.
3. The spindle connected to spindle interface 1 (main CPU board) is defined as spindle 1, and the spindle connected to spindle interface 2 (optional board 2) is defined as spindle 2. For detail refer to FANUC Series 16/18/160/180–MODEL B CONNECTION MANUAL (FUNCTION).

21.7 SYNCHRONIZATION CONTROL AND COMPOSITE CONTROL

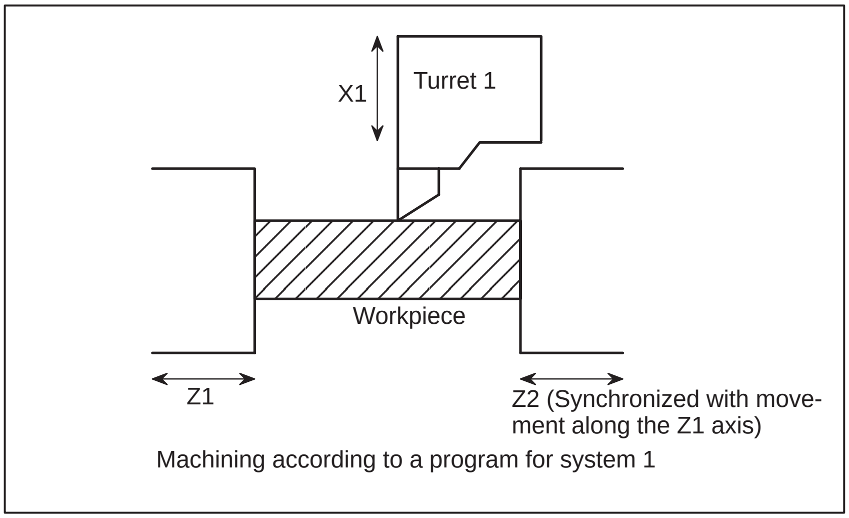
In 2–paths control, the synchronization control function and composite control function enable synchronization control in a single system or between two systems, composite control of two systems, and superposition control of two systems.

Explanations

- **Synchronization control**

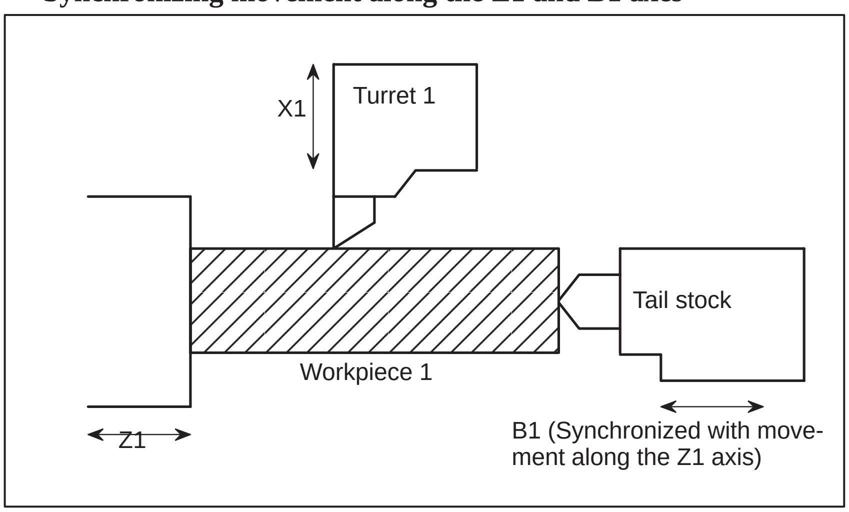
Synchronizes movement along an axis of one system with that along an axis of the other system.

Example)
Synchronizing movement along the Z1 and Z2 axes



Synchronizes movement along an axis of one system with that along another axis of the same system.

Example)
Synchronizing movement along the Z1 and B1 axes



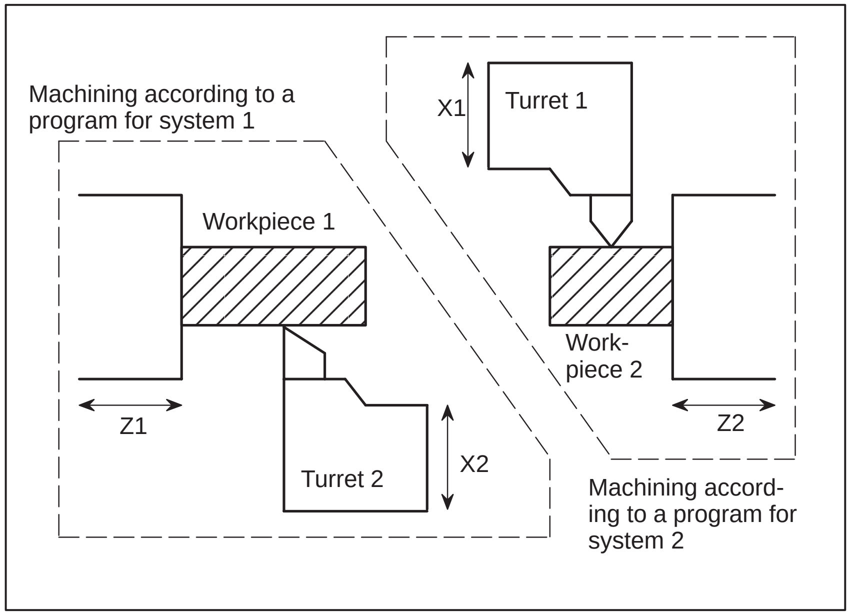
- **Composite control**

Exchanges the move commands for different axes of different systems.

Example)

Exchanging the commands for the X1 and X2 axes

–> Upon the execution of a command programmed for system 1, movement is performed along the X2 and Z1 axes.
Upon the execution of a command programmed for system 2, movement is performed along the X1 and Z2 axes.

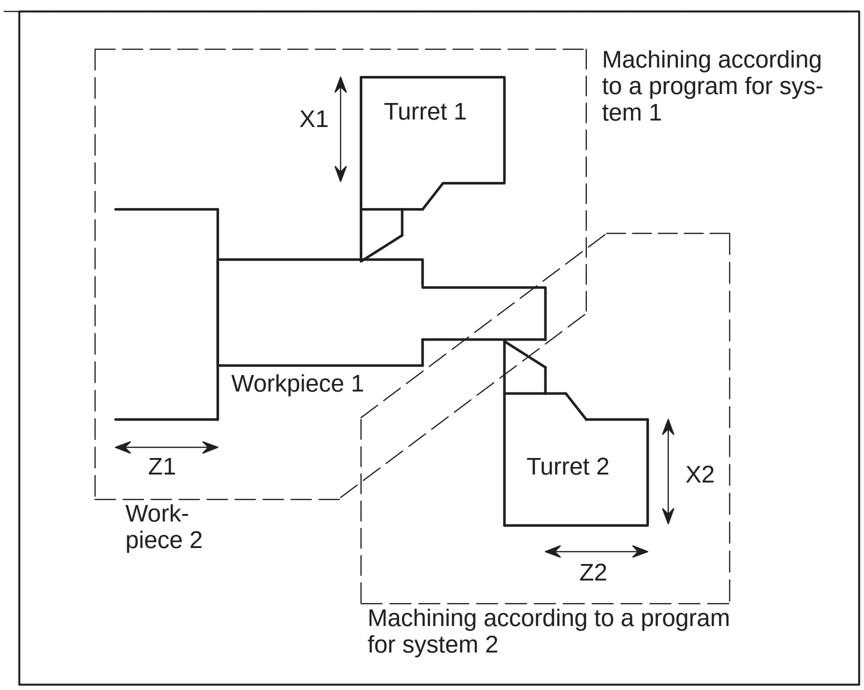


- **Superposition control**

Provides a move command of an axis for a different axis in another system.

Example)

Providing the Z2 axis with a move command specified for the Z1 axis



Notes

Notes

The method used to specify synchronization or composite control varies with the machine tool builder. For details, refer to the manual supplied by the machine tool builder.

22 PATTERN DATA INPUT FUNCTION

This function enables users to perform programming simply by extracting numeric data (pattern data) from a drawing and specifying the numerical values from the CRT/MDI panel.
This eliminates the need for programming using an existing NC language.

With the aid of this function, a machine tool builder can prepare the program of a hole machining cycle (such as a boring cycle or tapping cycle) using the custom macro function, and can store it into the program memory.
This cycle is assigned pattern names, such as BOR1, TAP3, and DRL2.

An operator can select a pattern from the menu of pattern names displayed on the screen.
Data (pattern data) which is to be specified by the operator should be created in advance with variables in a drilling cycle.
The operator can identify these variables using names such as DEPTH, RETURN RELIEF, FEED, MATERIAL or other pattern data names. The operator assigns values (pattern data) to these names.

22.1 DISPLAYING THE PATTERN MENU

Pressing the OFFSET SETTING key and ▷ **[MENU]** is displayed on the following pattern menu screen.

```
 MENU : HOLE PATTERN            O0000 N00000
      1.  TAPPING
      2.  DRILLING
      3.  BORING
      4.  POCKET
      5.  BOLT HOLE
      6.  LINE ANGLE
      7.  GRID
      8.  PECK
      9.  TEST PATRN
     10.  BACK


> _
MDI **** *** ***      16:05:59
[ MACRO ] [ MENU ] [  OPR  ] [      ] [(OPRT)]
```

HOLE PATTERN :
This is the menu title. An arbitrary character string consisting of up to 12 characters can be specified.

BOLT HOLE :
This is the pattern name. An arbitrary character string consisting of up to 10 characters can be specified, including katakana.

The machine tool builder should specify the character strings for the menu title and pattern name using the custom macro, and load the character strings into program memory as a subprogram of program No. 9500.

- **Macro commands specifying the menu title**

Menu title : C_1 C_2 C_3 C_4 C_5 C_6 C_7 C_8 $C_9 C_{10}$ C_{11} C_{12}

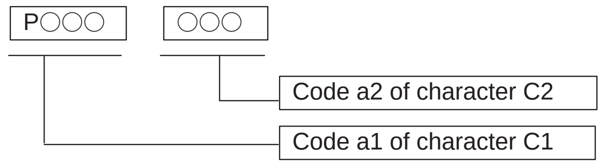
C_1,C_2, ,C_{12} : Characters in the menu title (12 characters)

Macro instruction

G65 H90 P_p Q_q R_r I_i J_j K_k ;

H90:Specifies the menu title

p : Assume a1 and a2 to be the codes of characters C1 and C2. Then,



q : Assume a_3 and a_4 to be the codes of characters C_3 and C_4. Then,
$q=a_3\,10^3+a_4$

r : Assume a_5 and a_6 to be the codes of characters C_5 and C_6. Then,
$r=a_5\,10^3+a_6$

i : Assume a_7 and a_8 to be the codes of characters C_7 and C_8. Then,
$i=a_7\,10^3+a_8$

j : Assume a_9 and a_{10} to be the codes of characters C_9 and C_{10}. Then,
$j=a_9\,10^3+a_{10}$

k : Assume a_{11} and a_{12} to be the codes of characters C_{11} and C_{12}.Then,
$k=a_{11}\,10^3+a_{12}$

Example)

If the title of the menu is”HOLE PATTERN” then the macro instruction is as follows:

G65 H90 P072079 Q076069 R032080
HO LE ␣P
I065084 J084069 K082078;
AT TE RN

For codes corresponding to these characters, refer to the table in II–22.3.

- **Macro instruction describing the pattern name**

Pattern name: $C_1\ C_2\ C_3\ C_4\ C_5\ C_6\ C_7\ C_8\ C_9 C_{10}$
C_1, C_2, ,C_{10}: Characters in the pattern name (10 characters)
Macro instruction
G65 H91 P_n Q_q R_r I_i J_j K_k ;
H91: Specifies the menu title
n : Specifies the menu No. of the pattern name
__ n=1 to 10
q : Assume a_1 and a_2 to be the codes of characters C_1 and C_2. Then,
$q=a_1 \times 10^3+a_2$
r : Assume a_3 and a_4 to be the codes of characters C_3 and C_4. Then,
$r=a_3 \times 10^3+a_4$
i : Assume a_5 and a_6 to be the codes of characters C_5 and C_6. Then,
$i=a_5 \times 10^3+a_6$
j : Assume a_7 and a_8 to be the codes of characters C_7 and C_8. Then,
$j=a_7 \times 10^3+a_8$
k : Assume a_9 and a_{10} to be the codes of characters C_9 and C_{10}. Then,
$k=a_9 \times 10^3+a_{10}$

Example)
If the pattern name of menu No. 1 is "BOLT HOLE" then the macro instruction is as follows.
G65 H91 P1 Q<u>066079</u> R<u>076084</u> I<u>032072</u> J<u>079076</u> K<u>069032</u> ;
BO LT ␣H OL E␣

- **Pattern No. selection**

To select a pattern from the pattern menu screen, enter the corresponding pattern No. The following is an example.

[1] [INPUT]

The selected pattern No. is assigned to system variable #5900. The custom macro of the selected pattern can be started by starting a fixed program (external program No. search) with an external signal then referring to the system variable #5900 in the program.

> **Notes**
> If each characters of P, Q, R, I, J, and K are not specified in a macro instruction, two spaces are assigned to each omitted character.

Example

Custom macros for the menu title and hole pattern names.

```
 MENU : HOLE PATTERN            O0000 N00000
      1.  TAPPING
      2.  DRILLING
      3.  BORING
      4.  POCKET
      5.  BOLT HOLE
      6.  LINE ANGLE
      7.  GRID
      8.  PECK
      9.  TEST PATRN
     10.  BACK


> _
MDI **** *** ***      16:05:59
[ MACRO ] [ MENU ] [ OPR ] [     ] [ (OPRT) ]
```

O9500 ;

N1G65 H90 P072 079 Q076 069 R032 080 I 065 084 J 084 069 K082 078 ;	HOLE PATTERN
N2G65 H91 P1 Q066 079 R076 084 I 032 072 J 079 076 K069 032 ;	1.BOLT HOLE
N3G65 H91 P2 Q071 082 R073 068 ;	2.GRID
N4G65 H91 P3 Q076 073 R078 069 I 032 065 J 078071 K076069 ;	3.LINE ANGLE
N5G65 H91 P4 Q084 065 R080 080 I 073 078 J 071 032 ;	4.TAPPING
N6G65 H91 P5 Q068 082 R073 076 I 076 073 J 078 071 ;	5.DRILLING
N7G65 H91 P6 Q066079 R082073 I 078 071 ;	6.BORING
N8G65 H91 P7 Q080 079 R067 075 I 069 084 ;	7.POCKET
N9G65 H91 P8 Q080069 R067075 ;	8.PECK
N10G65 H91 P9 Q084 069 R083 084 I032 080 J065 084 K082 078 ;	9.TEST PATRN
N11G65 H91 P10 Q066 065 R067 0750 ;	10.BACK
N12M99 ;	

22.2 PATTERN DATA DISPLAY

When a pattern menu is selected, the necessary pattern data is displayed.

```
VAR. : BOLT HOLE                O0001 N00000
NO.  NAME          DATA   COMMENT
500  TOOL          0.000
501  STANDARD X    0.000 *BOLT HOLE
502  STANDARD Y    0.000 CIRCLE*
503  RADIUS        0.000 SET PATTERN
504  S. ANGL       0.000 DATA TO VAR.
505  HOLES NO      0.000 NO.500–505.
506                0.000
507                0.000

ACTUAL POSITION (RELATIVE)
   X    0.000      Y   0.000
   Z    0.000
> _
MDI **** *** ***       16:05:59
[ MACRO ] [ MENU ] [  OPR  ] [      ] [(OPRT)]
```

BOLT HOLE :
This is the pattern data title. A character string consisting of up to 12 characters can be set.

TOOL :
This is the variable name. A character string consisting of up to 10 characters can be set.

BOLT HOLE CIRCLE :
This is a comment statement. A character string can be displayed consisting of up to 8 lines, 12 characters per line.

(It is permissible to use katakana in a character string or line.)
The machine tool builder should program the character strings of pattern data title, pattern name, and variable name using the custom macro, and load them into the program memory as a subprogram whose No. is 9500 plus the pattern No. (O9501 to O9510).

● Macro instruction specifying the pattern data title (the menu title)

Menu title : C_1 C_2 C_3 C_4 C_5 C_6 C_7 C_8 $C_9 C_{10} C_{11} C_{12}$
C_1 ,C_2,..., C_{12} : Characters in the menu title (12 characters)
Macro instruction
G65 H92 P_n Q_q R_r I_i J_j K_k ;
H92 : Specifies the pattern name
p : Assume a_1 and a_2 to be the codes of characters C_1 and C_2. Then,
$p=a_1 \times 10^3+a_2$
See 17.3 for character codes.
q : Assume a_3 and a_4 to be the codes of characters C_3 and C_4. Then,
$q=a_3 \times 10^3+a_4$
r : Assume a_5 and a_6 to be the codes of characters C_5 and C_6. Then,
$r=a_5 \times 10^3+a_6$
i : Assume a_7 and a_8 to be the codes of characters C_7 and C_8. Then,
$i=a_7 \times 10^3+a_8$
j : Assume a_9 and a_{10} to be the codes of characters C_9 and C_{10}. Then,
$j=a_9 \times 10^3+a_{10}$
k : Assume a_{11} and a_{12} to be the codes of characters C_{11} and C_{12}. Then,
$k=a_{11} \times 10^3+a_{12}$

Example)
Assume that the pattern data title is ”BOLT HOLE.”The macro instruction is given as follows:

$$\text{G65 H92 P}\underbrace{066079}_{\text{BO}}\ \text{Q}\underbrace{076084}_{\text{LT}}\ \text{R}\underbrace{032072}_{\sqcup\text{H}}\ \text{I}\underbrace{079076}_{\text{OL}}\ \text{J}\underbrace{069032}_{\text{E}};$$

● Macro instruction specifying the variable name

Variable name : C_1 C_2 C_3 C_4 C_5 C_6 C_7 C_8 $C_9 C_{10}$
C_1, C_2,..., C_{10} : Characters in the variable name (10 characters)
Macro instruction
G65 H93 P_n Q_q R_r I_i J_j K_k ;
H93 : Specifies the variable name
n : Specifies the menu No. of the variable name
n=1 to 10
q : Assume a_1 and a_2 to be the codes of characters C_1 and C_2. Then,
$q=a_1 \times 10^3+a_2$
r : Assume a_3 and a_4 to be the codes of characters C_3 and C_4. Then,
$r=a_3 \times 10^3+a_4$
i : Assume a_5 and a_6 to be the codes of characters C_5 and C_6. Then,
$i=a_5 \times 10^3+a_6$
j : Assume a_7 and a_8 to be the codes of characters C_7 and C_8. Then,
$j=a_7 \times 10^3+a_8$
k : Assume a_9 and a_{10} to be the codes of characters C_9 and C_{10}. Then,
$k=a_9 \times 10^3 a+a_{10}$

Example)
Assume that the variable name of the variable No. 503 is ”RADIUS.”
The macro instruction is given as follows:

$$\text{G65 H93 P503 Q}\underbrace{082065}_{\text{RA}}\ \text{R}\underbrace{068073}_{\text{DI}}\ \text{I}\underbrace{085083}_{\text{US}}\ ;$$

> Variable names can be assigned to 32 common variables #500 to #531, which are not cleared when the power is turned off.

- **Macro instruction to describe a comment**

One comment line: C_1 C_2 C_3 C_4 C_5 C_6 C_7 C_8 C_9 C_{10} C_{11} C_{12}
C_1, C_2,..., C_{12} : Character string in one comment line (12 characters)
Macro instruction
G65 H94 P_n Q_q R_r I_i J_j K_k ;
H94 : Specifies the comment
p : Assume a_1 and a_2 to be the codes of characters C_1 and C_2. Then,
$p=a_1 \times 10^3+a_2$
See 17.7 for character codes.
q : Assume a_3 and a_4 to be the codes of characters C_3 and C_4. Then,
$q=a_3 \times 10^3+a_4$
r : Assume a_5 and a_6 to be the codes of characters C_5 and C_6. Then,
$r=a_5 \times 10^3+a_6$
i : Assume a_7 and a_8 to be the codes of characters C_7 and C_8. Then,
$i=a_7 \times 10^3+a_8$
j : Assume a_9 and a_{10} to be the codes of characters C_9 and C_{10}. Then,
$j=a_9 \times 10^3+a_{10}$
k : Assume a_{11} and a_{12} to be the codes of characters C_{11} and C_{12}. Then,
$k=a_{11} \times 10^3+a_{12}$

A comment can be displayed in up to eight lines. The comment consists of the first line to the eighth line in the programmed sequence of G65 H94 for each line.

Example)

Assume that the comment is ”BOLT HOLE.” The macro instruction is given as follows:

G65 H94 P<u>042066</u> Q<u>079076</u> R<u>084032</u> I<u>072079</u> J<u>076069</u>;
*B OL T␣ HO LE

Examples

Macro instruction to describe a parameter title , the variable name, and a comment.

```
VAR. : BOLT HOLE                     O0001 N00000
NO.  NAME           DATA   COMMENT
500  TOOL           0.000
501  STANDARD X     0.000 *BOLT HOLE
502  STANDARD Y     0.000 CIRCLE*
503  RADIUS         0.000 SET PATTERN
504  S. ANGL        0.000 DATA TO VAR.
505  HOLES NO       0.000 NO.500–505.
506                 0.000
507                 0.000

ACTUAL POSITION (RELATIVE)
   X    0.000      Y   0.000
   Z    0.000
> _
MDI **** *** ***      16:05:59
[ MACRO ] [ MENU ] [  OPR  ] [      ] [(OPRT)]
```

O9501 ;

N1G65 H92 P066 079 Q076 084 R032 072 I 079 076 J069 032 ;	VAR : BOLT HOLE
N2G65 H93 P500 Q084 079 R079076 ;	#500 TOOL
N3G65 H93 P501 Q075 073 R074 085 I078 032 J088 032 ;	#501 KIJUN X
N4G65 H93 P502 Q075 073 R074 085 I 078 032 J089 032 ;	#502 KIJUN Y
N5G65 H93 P503 Q082 065 R068 073 I 085 083 ;	#503 RADIUS
N6G65 H93 P504 Q083 046 R032 065 I 078 071 J 076 032 ;	#504 S.ANGL
N7G65 H93 P505 Q072 079 R076 069 I 083 032 J078 079 K046 032 ;	#505 HOLES NO
N8G65 H94 ;	Comment
N9G65 H94 P042 066 Q079 076 R084 032 I072 079 J076 069 ;	*BOLT HOLE
N10G65 H94 R032 067 I073 082 J067 076 K069 042 ;	CIRCLE*
N11G65 H94 P083 069 Q084 032 080 065 I084 084 J069 082 K078 032 ;	SET PATTERN
N12G65 H94 P068 065 Q084 065 R032 084 I079 032 J086 065 K082046 ;	DATA NO VAR.
N13G65 H94 P078 079 Q046 053 R048 048 I045 053 J048 053 K046 032 ;	No.500–505
N14M99 ;	

22.3 CHARACTERS AND CODES TO BE USED FOR THE PATTERN DATA INPUT FUNCTION

Table.22.3(a) Characters and codes to be used for the pattern data input function

Char-acter	Code	Comment	Char-acter	Code	Comment
A	065		6	054	
B	066		7	055	
C	067		8	056	
D	068		9	057	
E	069			032	Space
F	070		*	033	Exclama– tion mark
G	071		I	034	Quotation mark
H	072		t	035	Hash sign
I	073		p	036	Dollar sign
J	074		%	037	Percent
K	075		&	038	Ampersand
L	076		’	039	Apostrophe
M	077		(	040	Left parenthesis
N	078		)	041	Right parenthesis
O	079		*	042	Asterisk
P	080		+	043	Plus sign
Q	081		,	044	Comma
R	082		–	045	Minus sign
S	083		.	046	Period
T	084		/	047	Slash
U	085		:	058	Colon
V	086		;	059	Semicolon
W	087		<	060	Left angle bracket
X	088		=	061	Equal sign
Y	089		>	062	Right angle bracket
Z	090		?	063	Question mark
0	048		w	064	HAt”mark
1	049		[	091	Left square bracket
2	050		0	092	
3	051		o	093	Yen sign
4	052		]	094	Right squar bracket
5	053		_	095	Underscore

Table 22.3 (b)Numbers of subprograms employed in the pattern data input function

Subprogram No.	Function
O9500	Specifies character strings displayed on the pattern data menu.
O9501	Specifies a character string of the pattern data corresponding to pattern No.1
O9502	Specifies a character string of the pattern data corresponding to pattern No.2
O9503	Specifies a character string of the pattern data corresponding to pattern No.3
O9504	Specifies a character string of the pattern data corresponding to pattern No.4
O9505	Specifies a character string of the pattern data corresponding to pattern No.5
O9506	Specifies a character string of the pattern data corresponding to pattern No.6
O9507	Specifies a character string of the pattern data corresponding to pattern No.7
O9508	Specifies a character string of the pattern data corresponding to pattern No.8
O9509	Specifies a character string of the pattern data corresponding to pattern No.9
O9510	Specifies a character string of the pattern data corresponding to pattern No.10

Table. 22.3 (c)Macro instructions used in the pattern data input function

G code	H code	Function
G65	H90	Specifies the menu title.
G65	H91	Specifies the pattern name.
G65	H92	Specifies the pattern data title.
G65	G93	Specifies the variable name.
G65	H94	Specifies the comment.

Table. 22.3 (d)System variables employed in the pattern data input function

System variable	Function
t5900	Pattern No. selected by user.

III. OPERATION

1 GENERAL

1.1 MANUAL OPERATION

Explanations

- **Manual reference position return (See Section III–3.1)**

The CNC machine tool has a position used to determine the machine position.
This position is called the reference position, where the tool is replaced or the coordinate are set. Ordinarily, after the power is turned on, the tool is moved to the reference position.
Manual reference position return is to move the tool to the reference position using switches and pushbuttons located on the operator's panel.

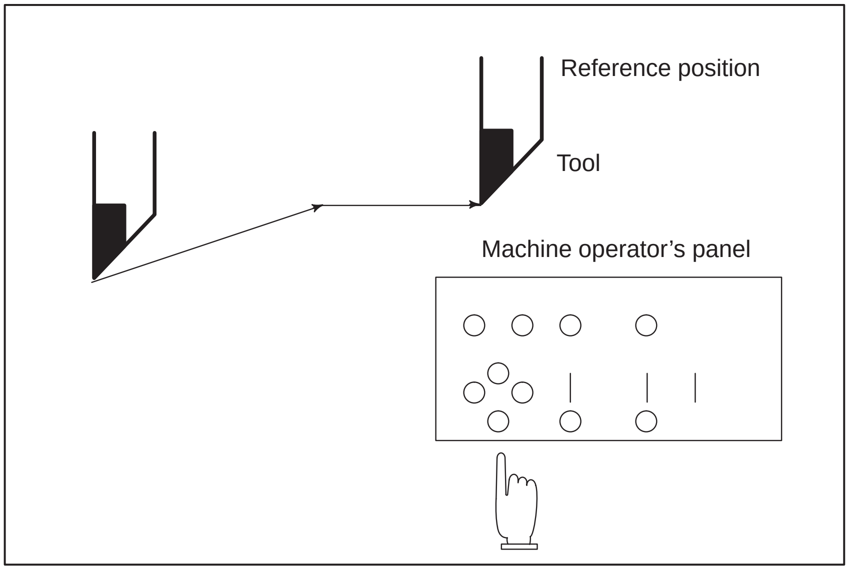


Fig.1.1 (a) Manual reference position return

The tool can be moved to the reference position also with program commands.
This operation is called automatic reference position return (See Section II–6).

- **The tool movement by manual operation**

Using machine operator's panel switches, push buttons, or the manual handle, the tool can be moved along each axis.

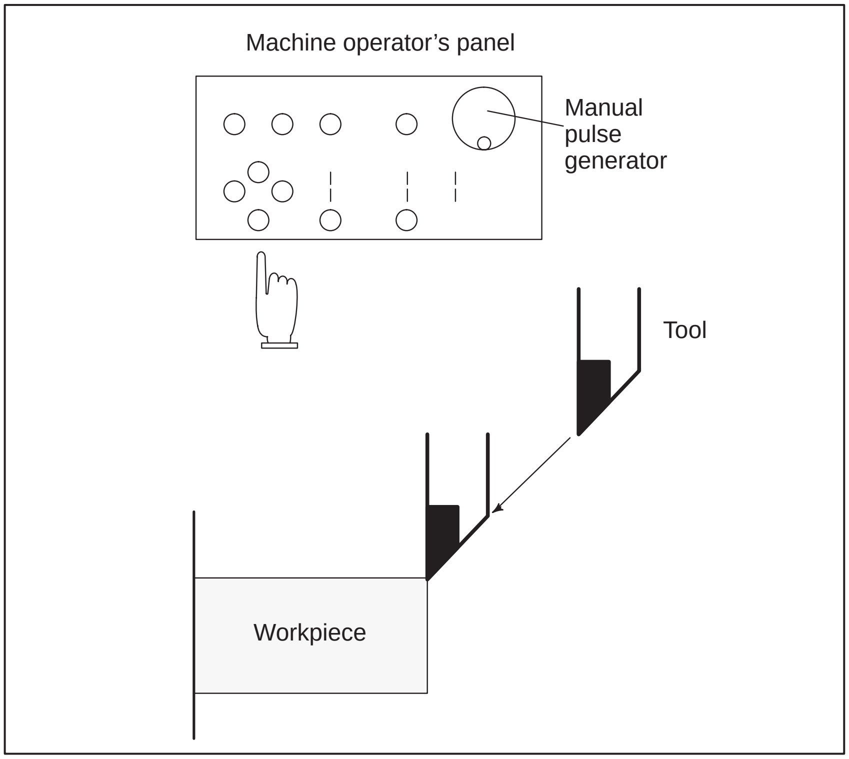


Fig.1.1 (b) The tool movement by manual operation

The tool can be moved in the following ways:

(i) Manual continuous feed (jog feed) (See Section III–3.2)
The tool moves continuously while a push button remains pressed.

(ii) Incremental feed (See Section III–3.3)
The tool moves by the predetermined distance each time a button is pressed.

(iii)Manual handle feed (See Section III–3.4)
By rotating the manual handle,the tool moves by the distance corresponding to the degree of handle rotation.

1.2 TOOL MOVEMENT BY PROGRAMING – AUTOMATIC OPERATION

Automatic operation is to operate the machine according to the created program. It includes memory, MDI, and DNC operations. (See Section III–4).

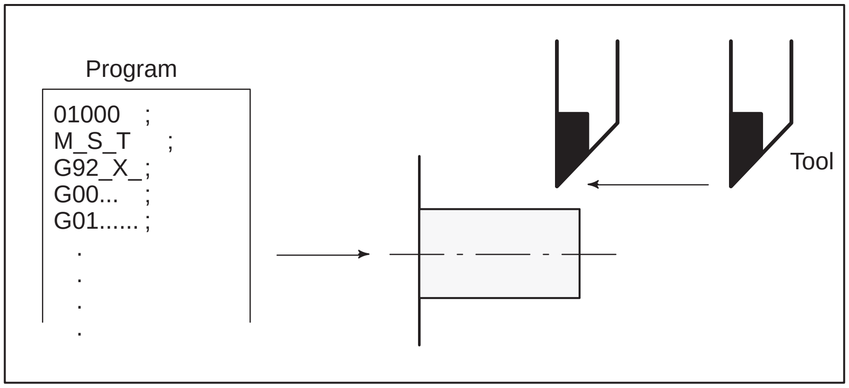


Fig.1.2 (a) Tool Movement by Programming

Explanations

● Memory operation

After the program is once registered in memory of CNC, the machine can be run according to the program instructions. This operation is called memory operation.

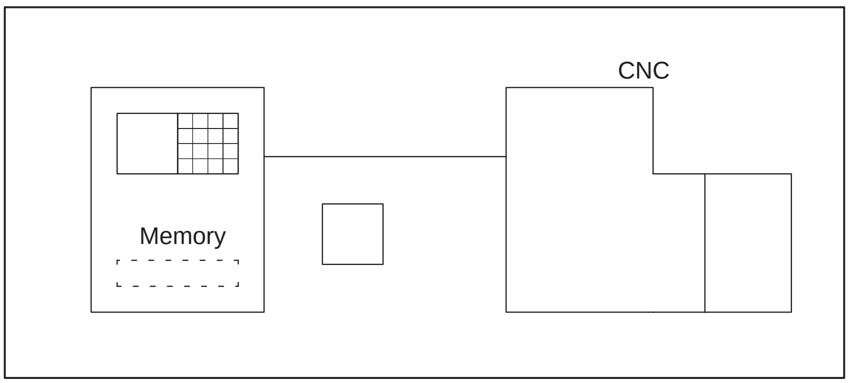


Fig.1.2 (b) Memory Operation

● MDI operation

After the program is entered, as an command group, from the MDI keyboard, the machine can be run according to the program. This operation is called MDI operation.

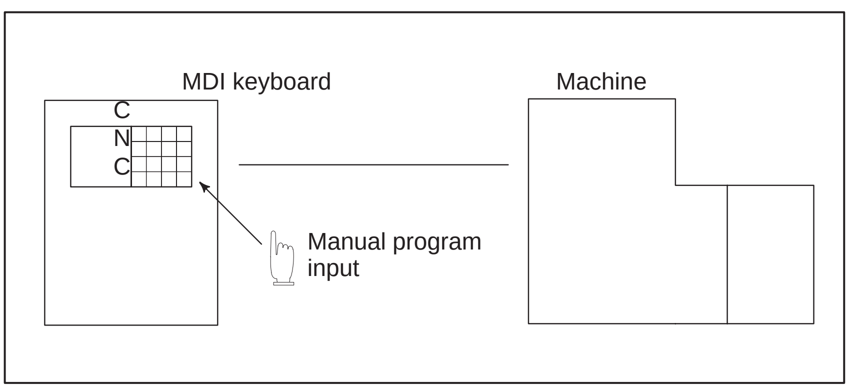


Fig.1.2 (c) MDI operation

● DNC operation

The machine can be operated by reading a program directly from an external input/output device, without having to register the program in CNC memory. This is called DNC operation.

1.3 AUTOMATIC OPERATION

Explanations

• Program selection

Select the program used for the workpiece. Ordinarily, one program is prepared for one workpiece. If two or more programs are in memory, select the program to be used, by searching the program number (Section III–9.3).

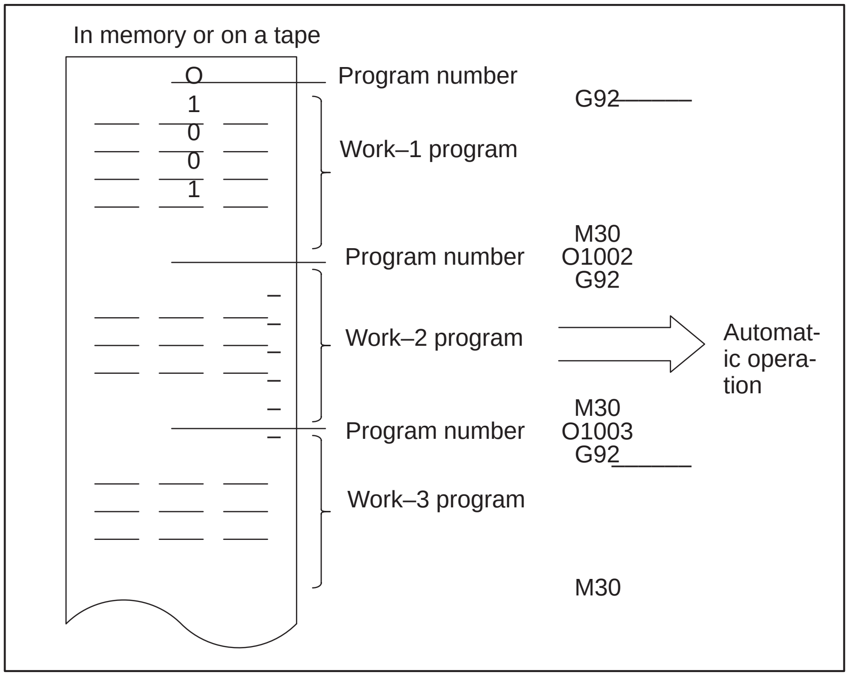


Fig.1.3 (a) Program Selection for Automatic Operation

• Start and stop (See Section III–4)

Pressing the cycle start push button causes automatic operation to start. By pressing the feed hold or reset push button, automatic operation pauses or stops. By specifying the program stop or program termination command in the program, the running will stop during automatic operation. When one process machining is completed, automatic operation stops.

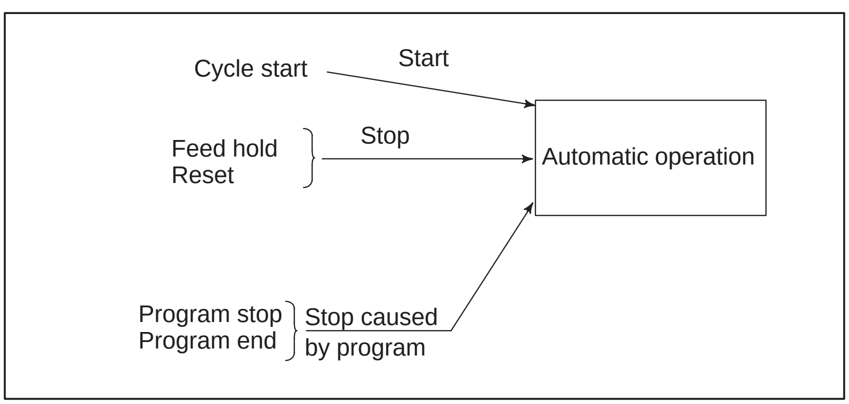


Fig.1.3 (b) Start and Stop for Automatic Operation

- **Handle interruption (See Section III–4.6)**

While automatic operation is being executed, tool movement can overlap automatic operation by rotating the manual handle.

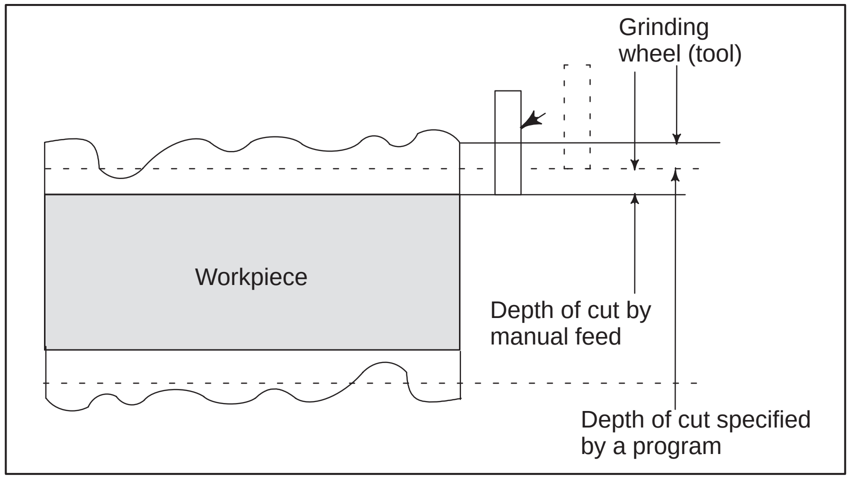


Fig.1.3 (c) Handle Interruption for Automatic Operation

1.4 TESTING A PROGRAM

Before machining is started, the automatic running check can be executed. It checks whether the created program can operate the machine as desired. This check can be accomplished by running the machine actually or viewing the position display change (without running the machine) (See Section III–5).

1.4.1 Check by Running the Machine

Explanations

- **Dry run (See Section III–5.4)**

Remove the workpiece, check only movement of the tool. Select the tool movement rate using the dial on the operator's panel.

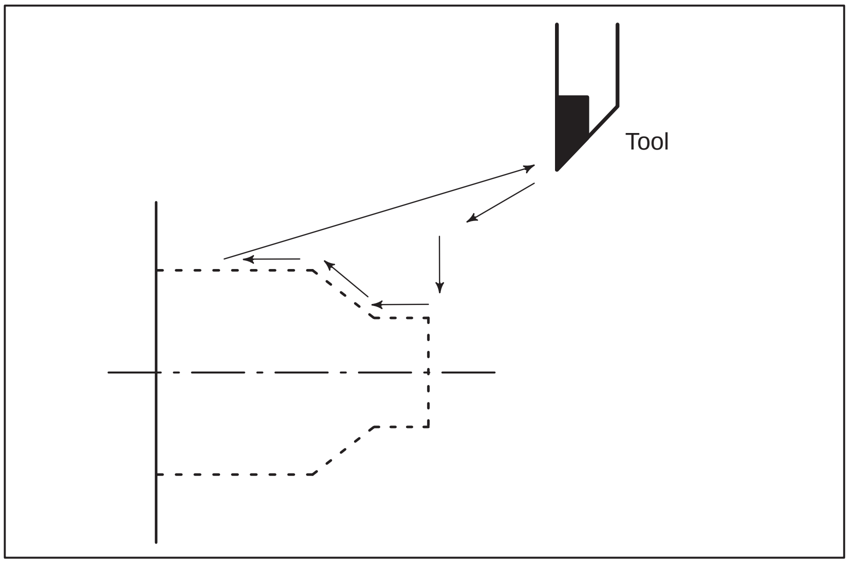


Fig.1.4 (a) Dry run

- **Feedrate override (See Section III–5.2)**

Check the program by changing the rate specified in the program.

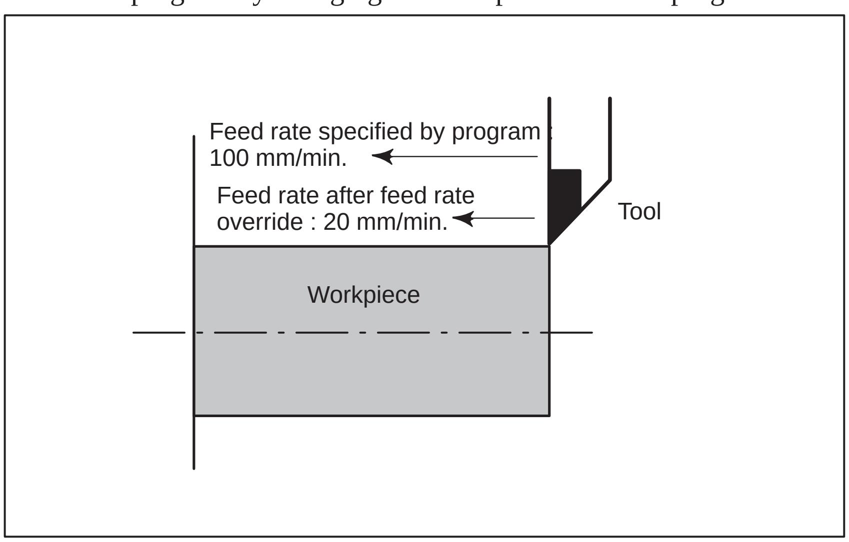


Fig1.4 (b) Feedrate Override

- **Single block (See Section III–5.5)**

When the cycle start push button is pressed, the tool executes one operation then stops. By pressing the cycle start again, the tool executes the next operation then stops. The program is checked in this manner.

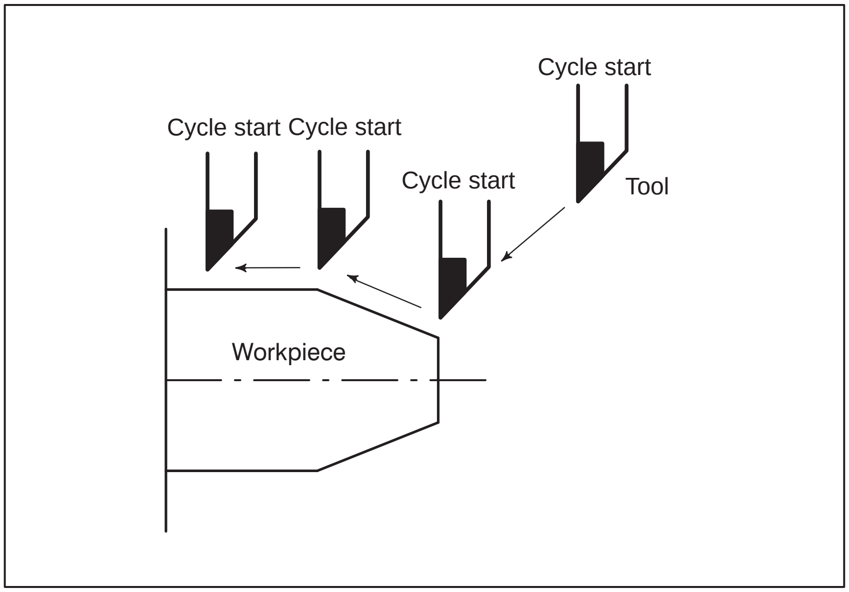


Fig.1.4 (c) Single Block

1.4.2 How to View the Position Display Change without Running the Machine

Explanations

- **Machine lock (See Sections III–5.1)**

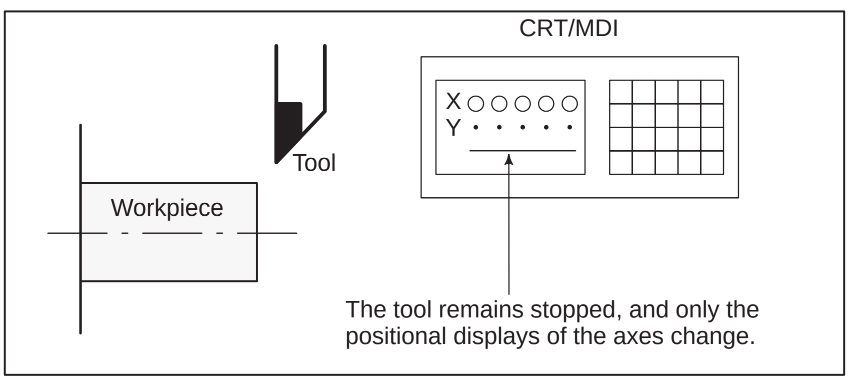


Fig1.4 (d) Machine Lock

- **Auxiliary function lock (See Section III–5.1)**

When automatic running is placed into the auxiliary function lock mode during the machine lock mode, all auxiliary functions (spindle rotation, tool replacement, coolant on/off, etc.) are disabled.

1.5 EDITING A PART PROGRAM

After a created program is once registered in memory, it can be corrected or modified from the CRT/MDI panel (See Section III–9).
This operation can be executed using the part program storage/edit function.

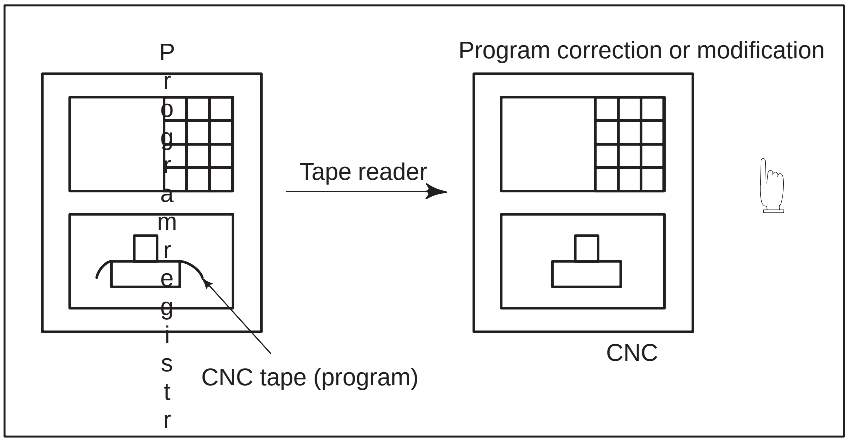


Fig.1.5 (a) Part Program Editing

1.6 DISPLAYING AND SETTING DATA

The operator can display or change a value stored in CNC internal memory by key operation on the CRT/MDI screen (See III–11).

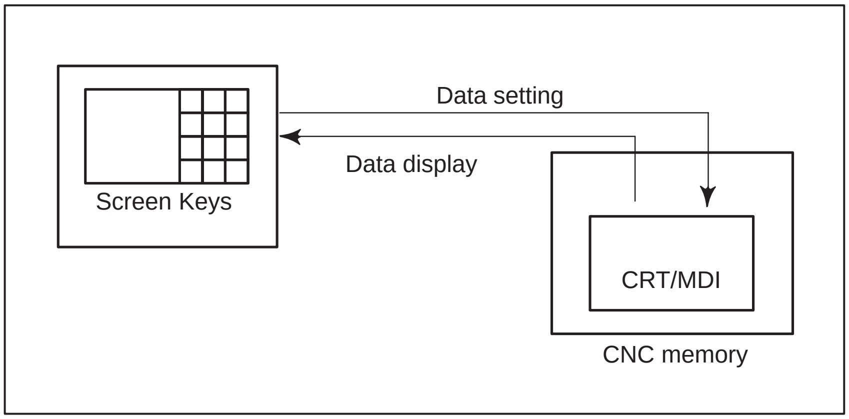


Fig.1.6 (a) Displaying and Setting Data

Explanations

- **Offset value**

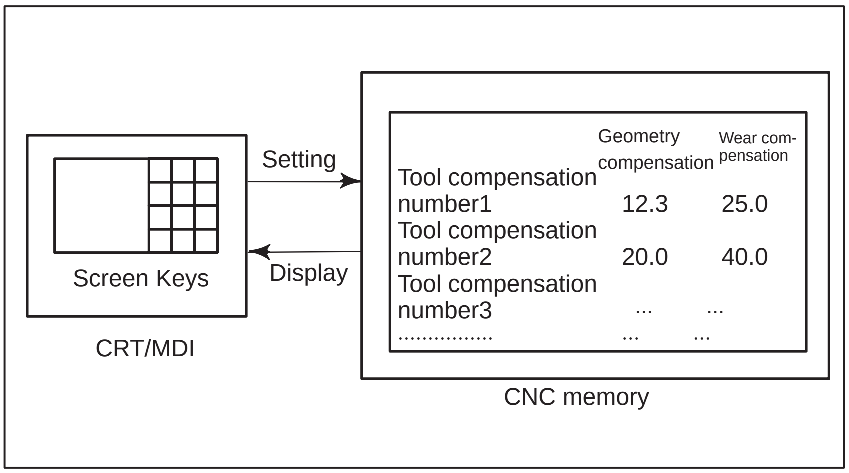


Fig.1.6 (b) Displaying and Setting Offset Values

The tool has the tool dimension (length, diameter). When a workpiece is machined, the tool movement value depends on the tool dimensions.
By setting tool dimension data in CNC memory beforehand, automatically generates tool routes that permit any tool to cut the workpiece specified by the program. Tool dimension data is called the offset value (See Section III–11.4.1).

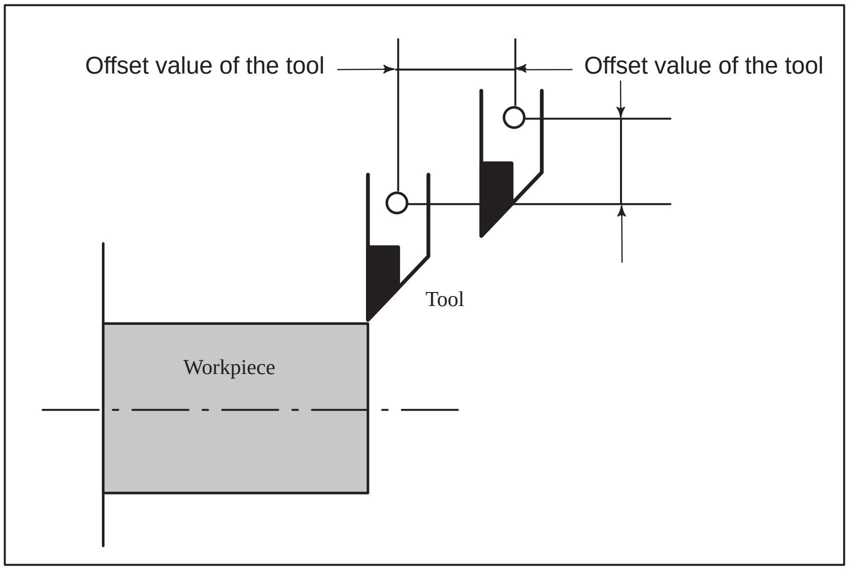


Fig.1.6 (c) Offset Value

- **Displaying and setting operator's setting data**

Apart from parameters, there is data that is set by the operator in operation. This data causes machine characteristics to change.
For example, the following data can be set:
·Inch/Metric switching
·I/O devices selection
·Mirror image cutting on/off
The above data is called setting data (See Section III–11.4.3).

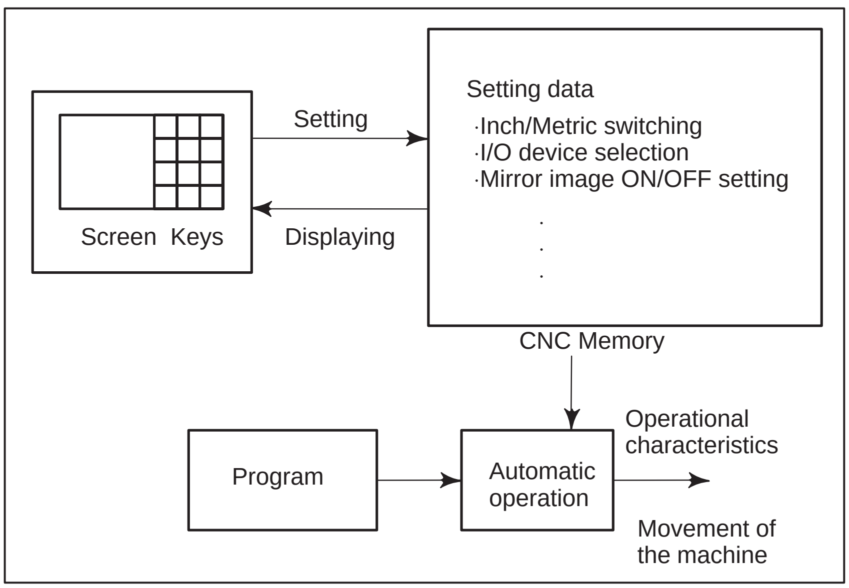


Fig.1.6 (d) Displaying and Setting Operator's setting data

- **Displaying and setting parameters**

The CNC functions have versatility in order to take action in characteristics of various machines.

For example, CNC can specify the following:

·Rapid traverse rate of each axis

·Whether increment system is based on metric system or inch system.

·How to set command multiply/detect multiply (CMR/DMR)

Data to make the above specification is called parameters (See Section III–11.5.1).

Parameters differ depending on machine tool.

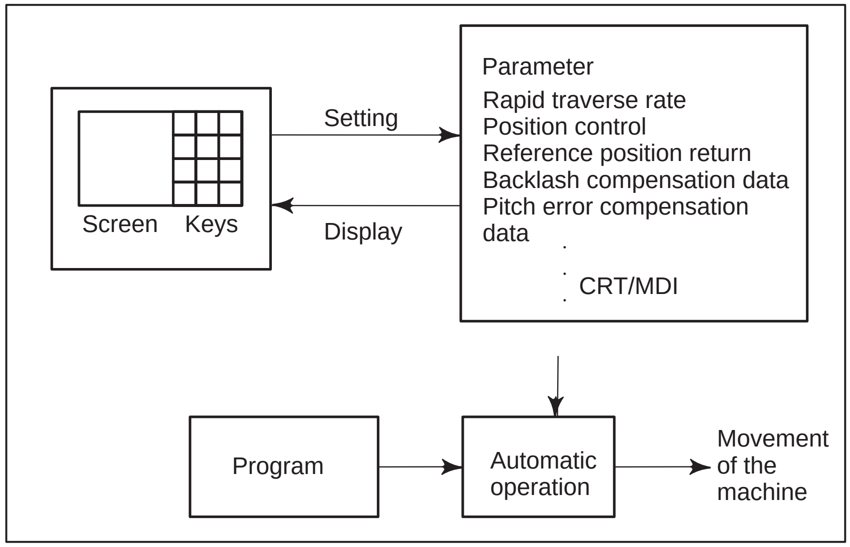


Fig.1.6 (e) Displaying and setting parameters

- **Data protection key**

A key called the data protection key can be defined. It is used to prevent part programs, offset values, parameters, and setting data from being registered, modified, or deleted erroneously (See Section III–11).

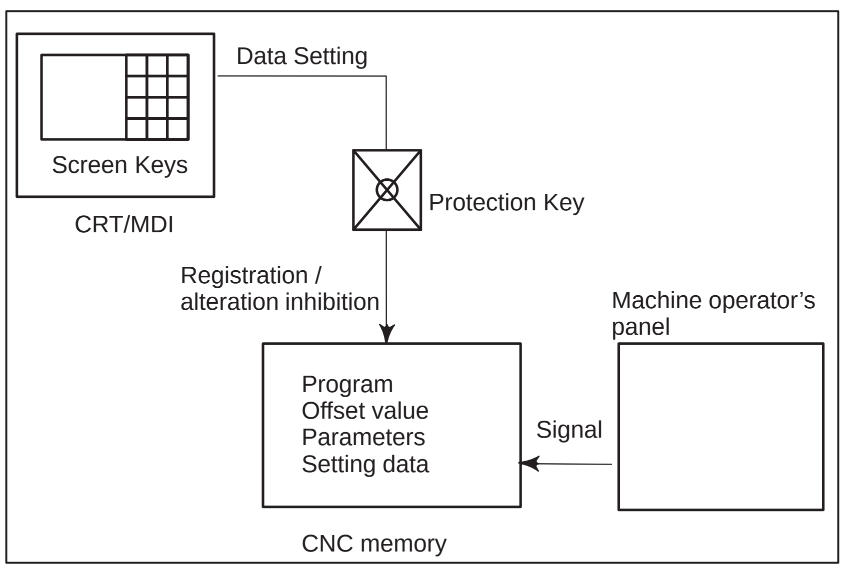
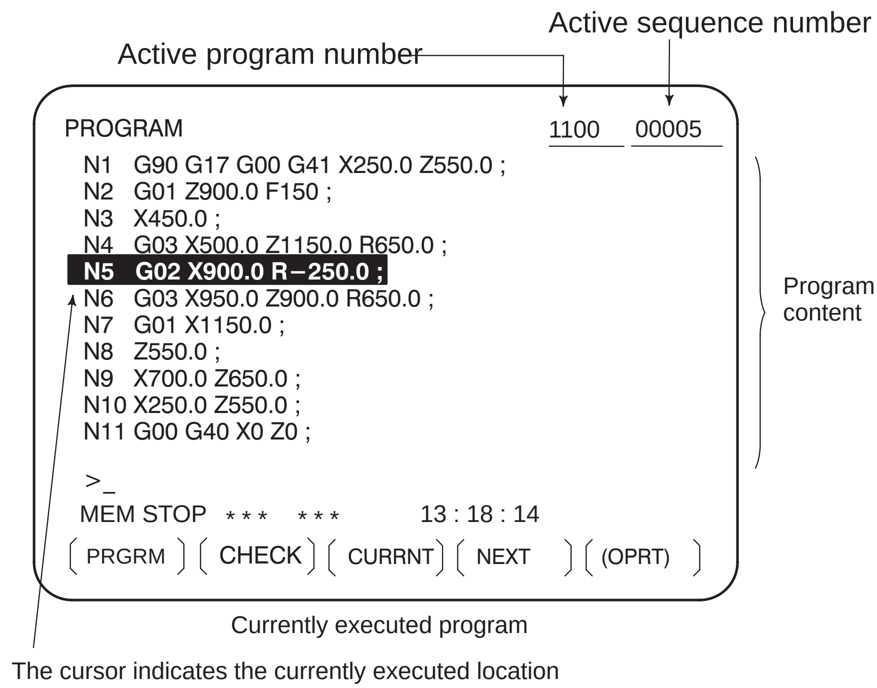


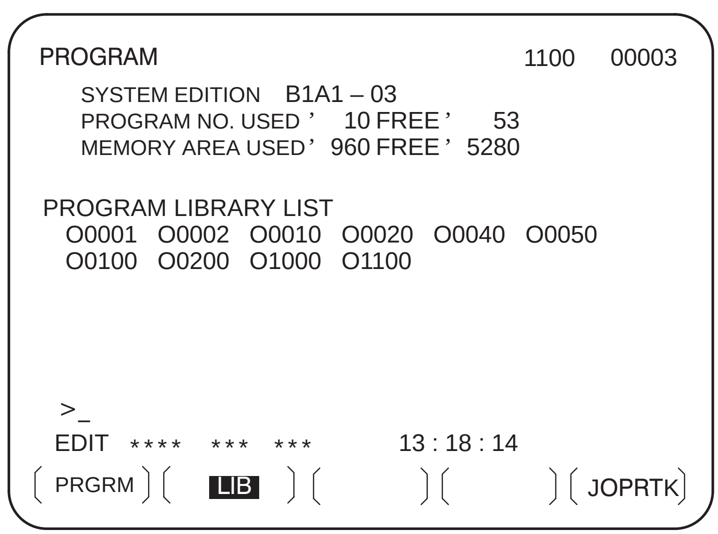
Fig.1.6 (f) Data Protection Key

1.7 DISPLAY

1.7.1 Program Display

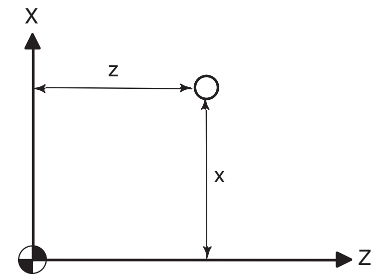
The contents of the currently active program are displayed. In addition, the programs scheduled next and the program list are displayed.
(See Section III–11.2.1)



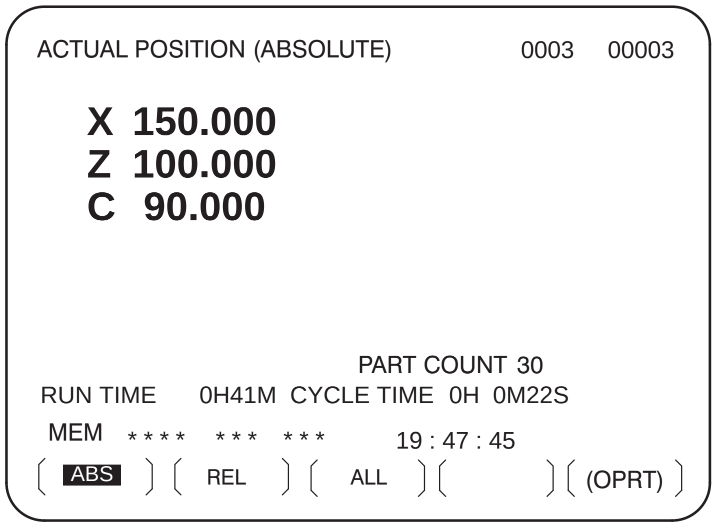


1.7.2 Current Position Display

The current position of the tool is displayed with the coordinate values. The distance from the current position to the target position can also be displayed. (See Section III–11.1 to 11.1.3)

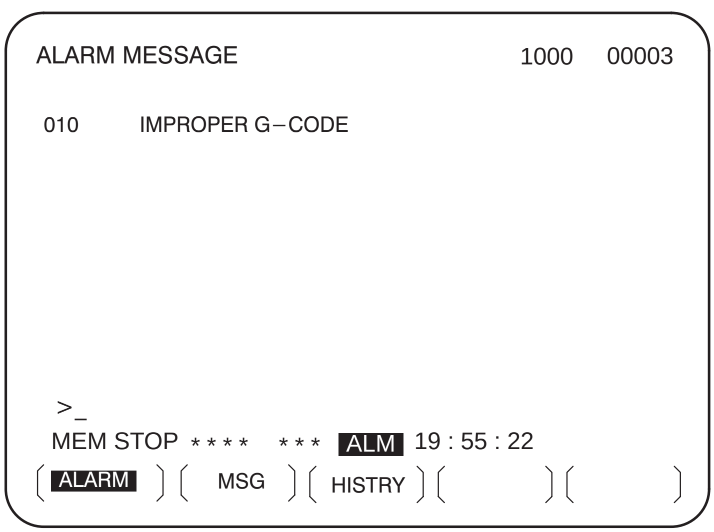


Workpiece coordinate system



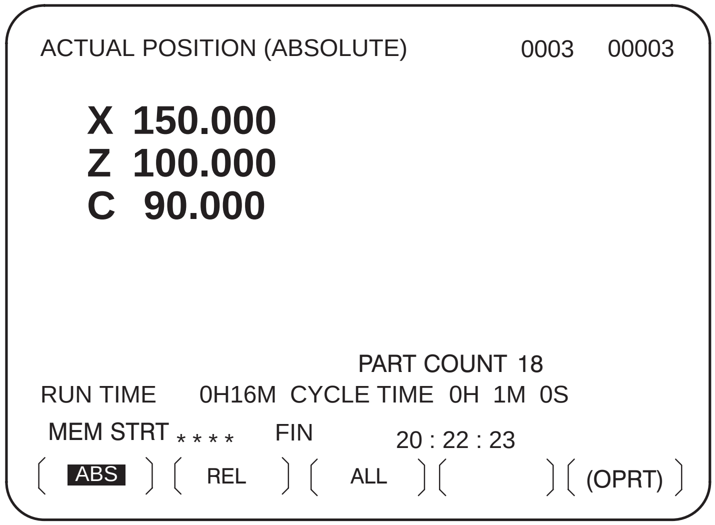
1.7.3 Alarm Display

When a trouble occurs during operation, error code and alarm message are displayed on CRT screen. See APPENDIX G for the list of error codes and their meanings. (See Section III–7.1)



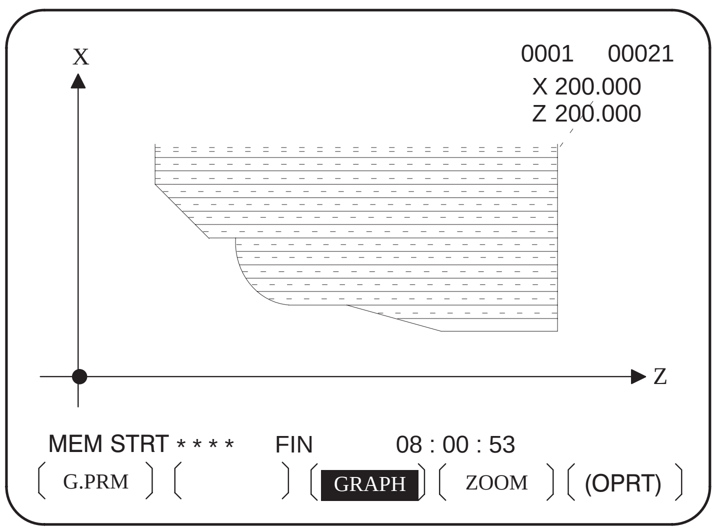
1.7.4 Parts Count Display, Run Time Display

When this option is selected, two types of run time and number of parts are displayed on the screen.(See Section lll–11.4.9)

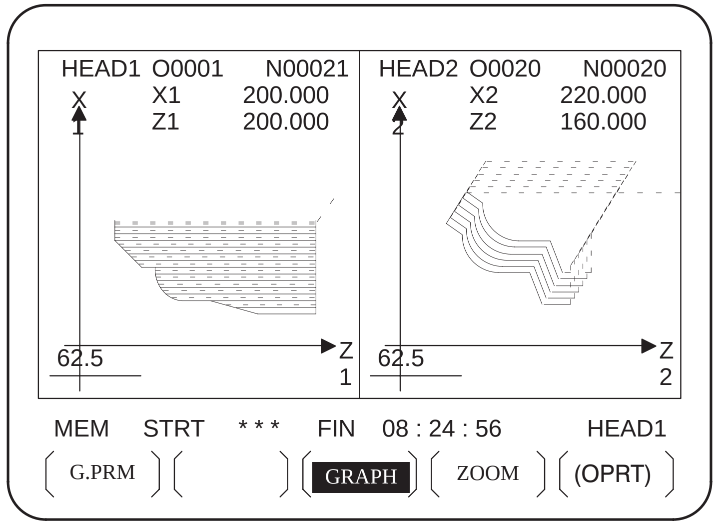


1.7.5 Graphic Display (See Section III–12)

The graphic can be used to draw a tool path for automatic operation and manual operation, thereby indicating the progress of cutting and the position of the tool. (See Section III–12)



1–path control



2–path control

1.8 DATA OUTPUT

Programs, offset values, parameters, etc. input in CNC memory can be output to paper tape, cassette, or a floppy disk for saving. After once output to a medium, the data can be input into CNC memory.

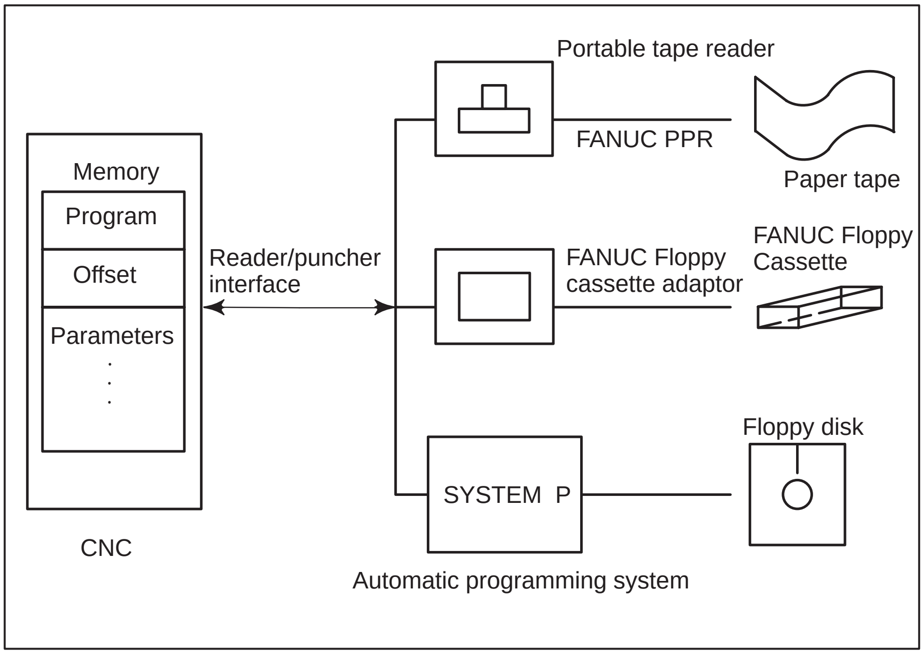


Fig.1.8 (a) Data Output

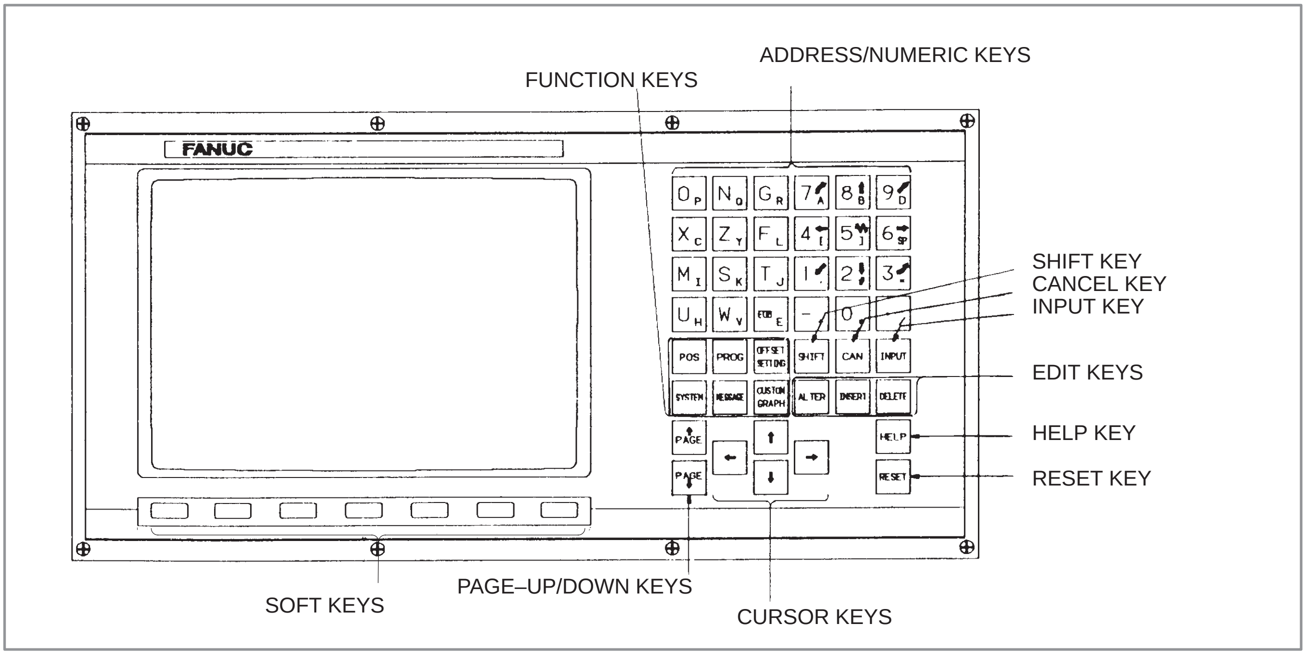
2 OPERATIONAL DEVICES

The peripheral devices available include the CRT/MDI panel (or LCD/MDI panel) attached to the CNC, machine operator's panel and external input/output devices such as tape reader, PPR, floppy cassette, and FA card.

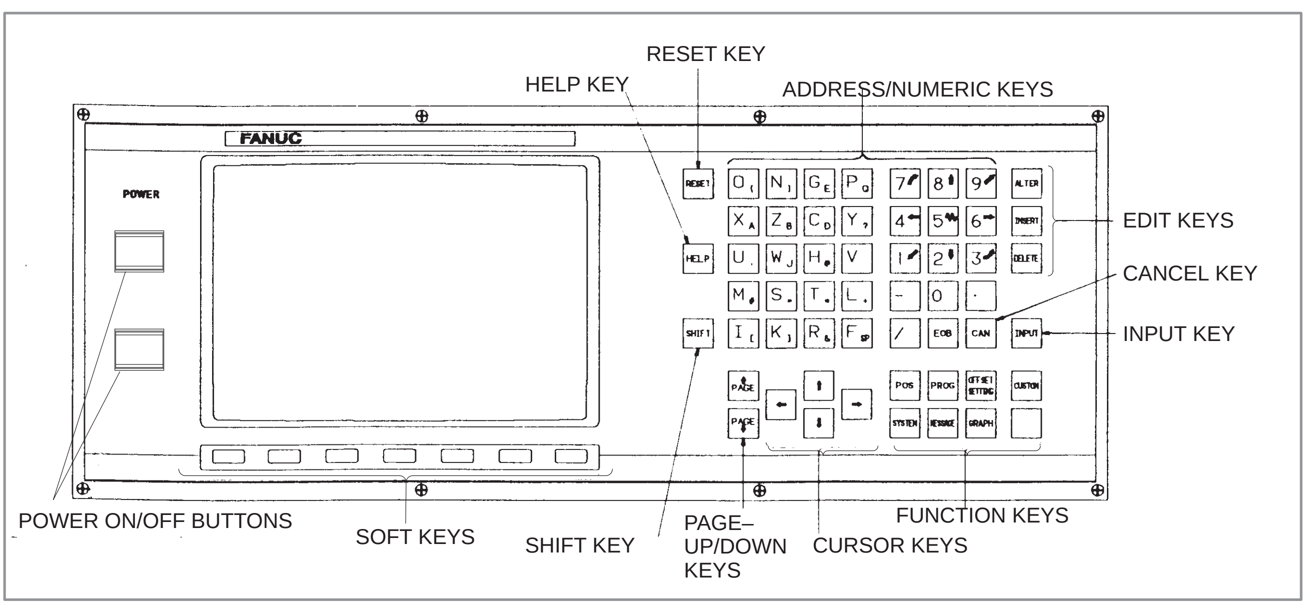
2.1 SETTING AND DISPLAY UNIT

The following Setting and Display units are available.

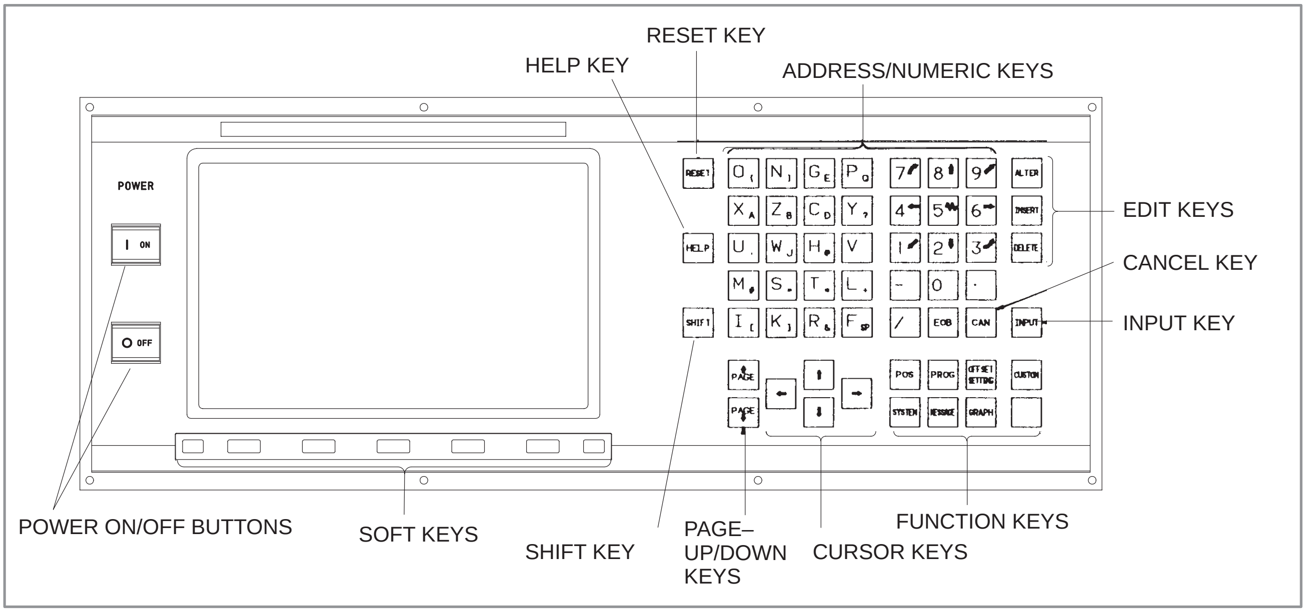
2.1.1 9-inch Monochrome/Color CRT/MDI Panel (Small Type)



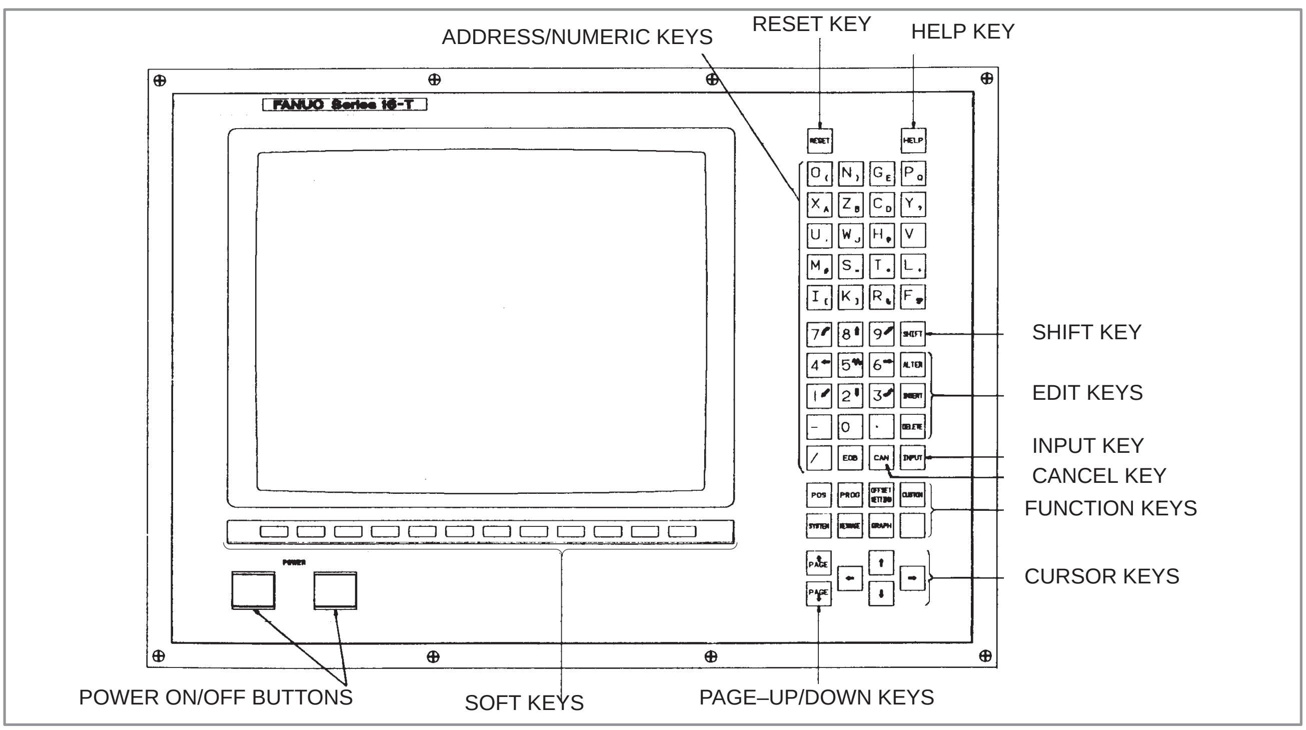
2.1.2 9-inch Monochrome/Color CRT/MDI Panel (Standard Type)



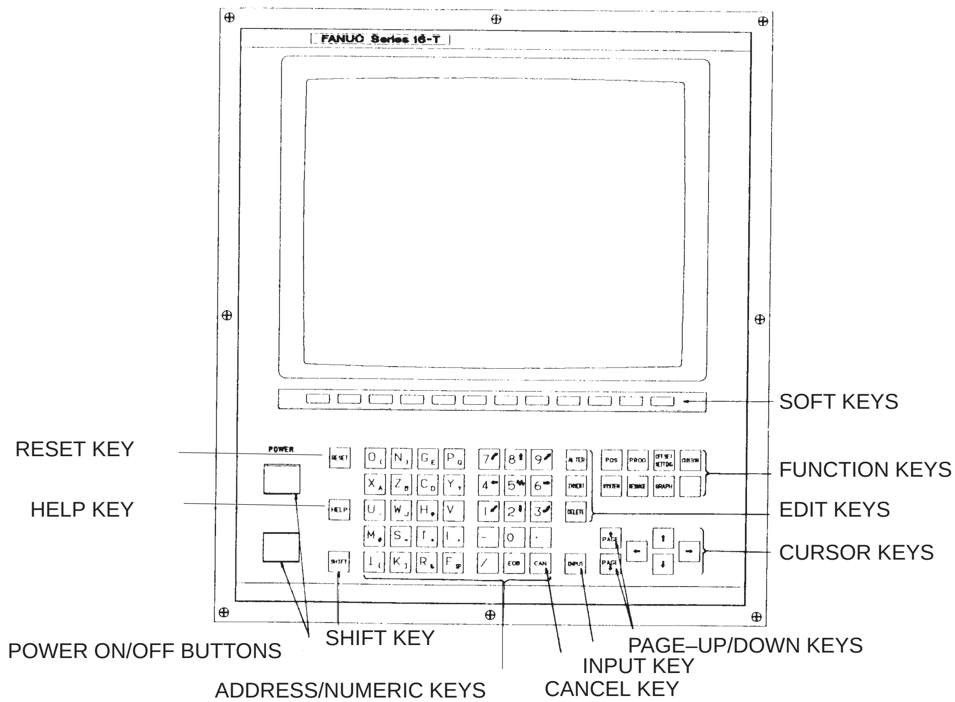
2.1.3 9-inch Monochrome PDP/MDI (Standard Type)



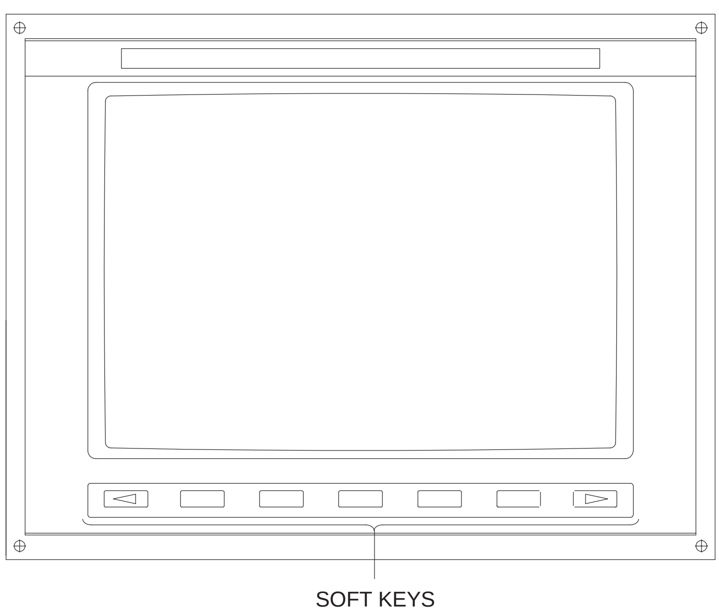
2.1.4 14-inch Color CRT/MDI (Horizontal Type)



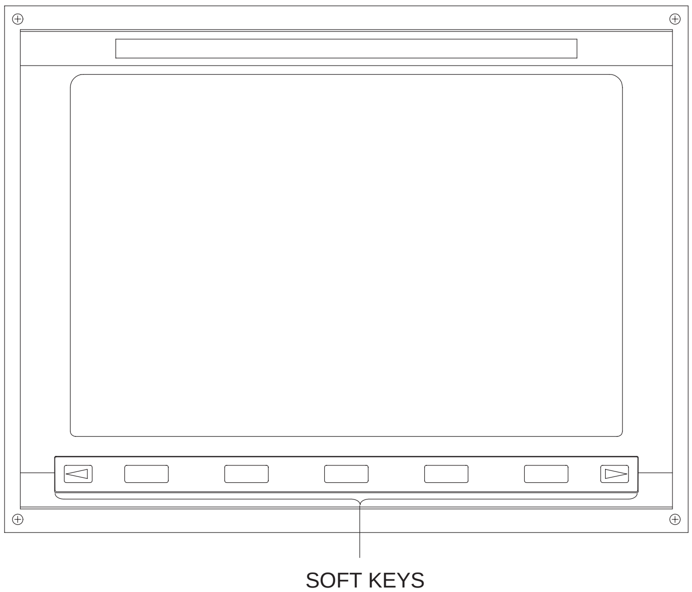
2.1.5 14-inch Color CRT/MDI (Vertical Type)



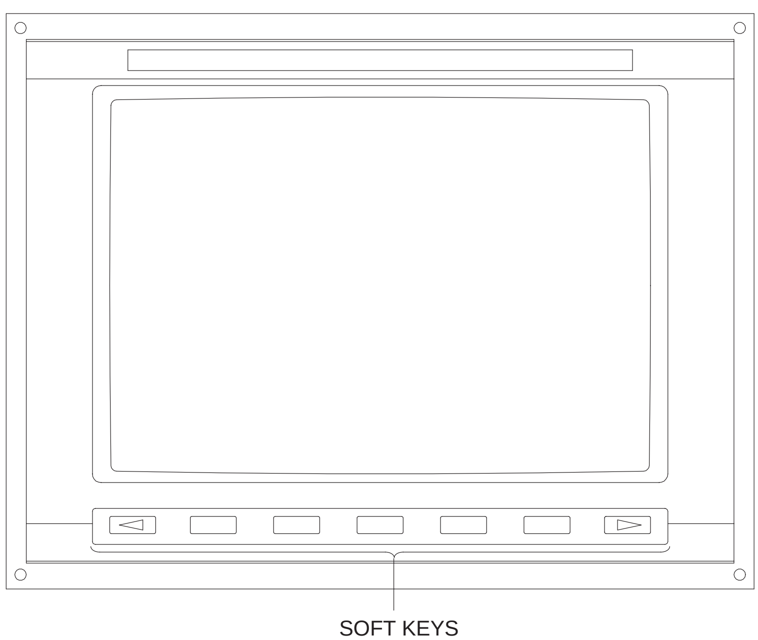
2.1.6 9-inch Monochrome/Color CRT (Separate Type)



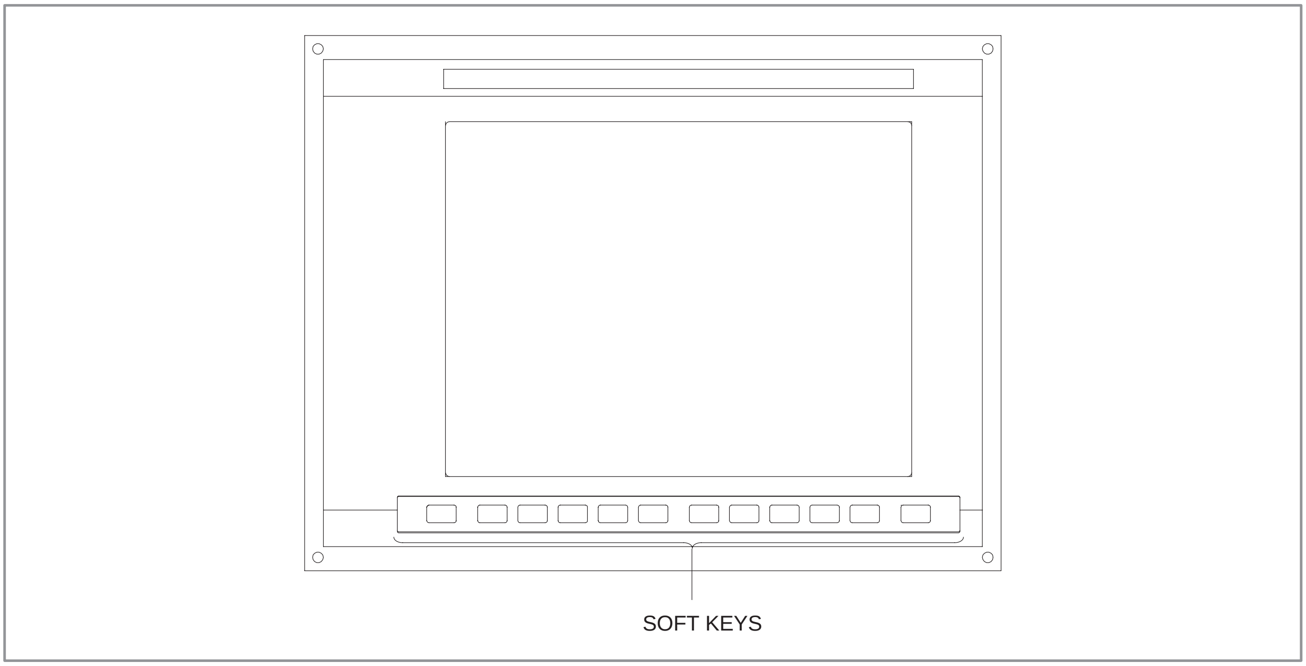
2.1.7
9–inch Monochrome PDP (Separate Type)



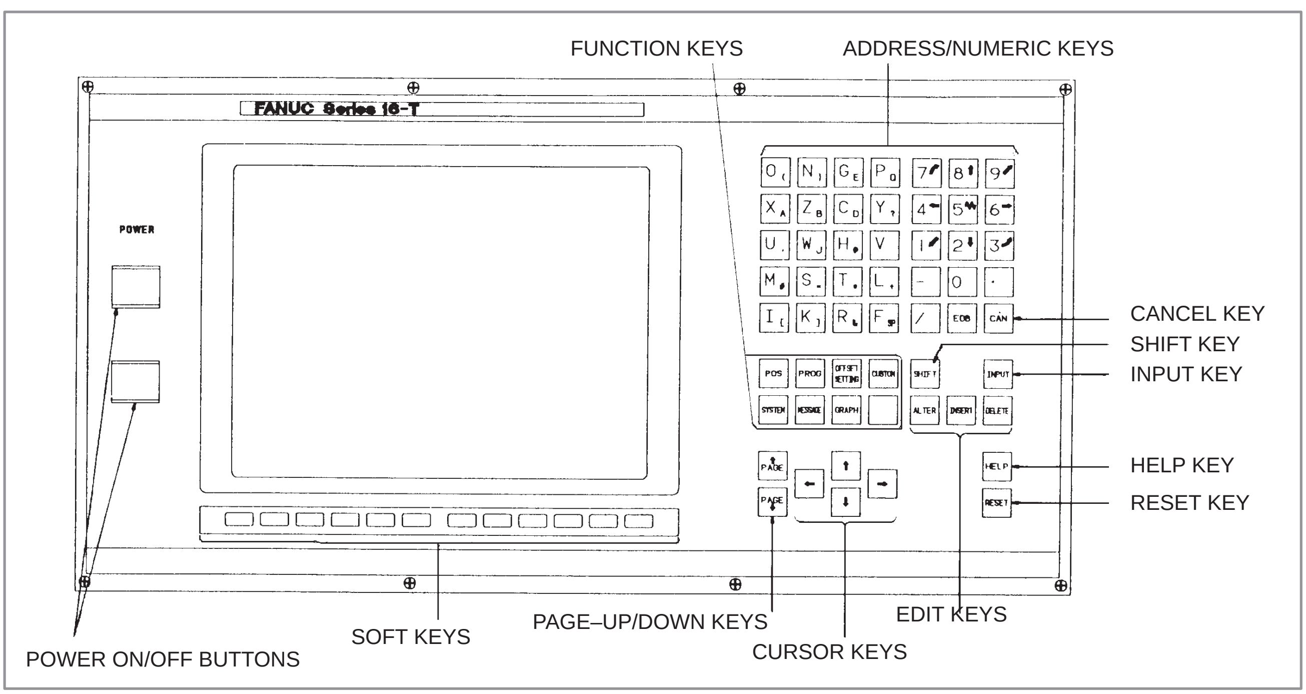
2.1.8
7.2–inch Monochrome LCD (Separate Type)



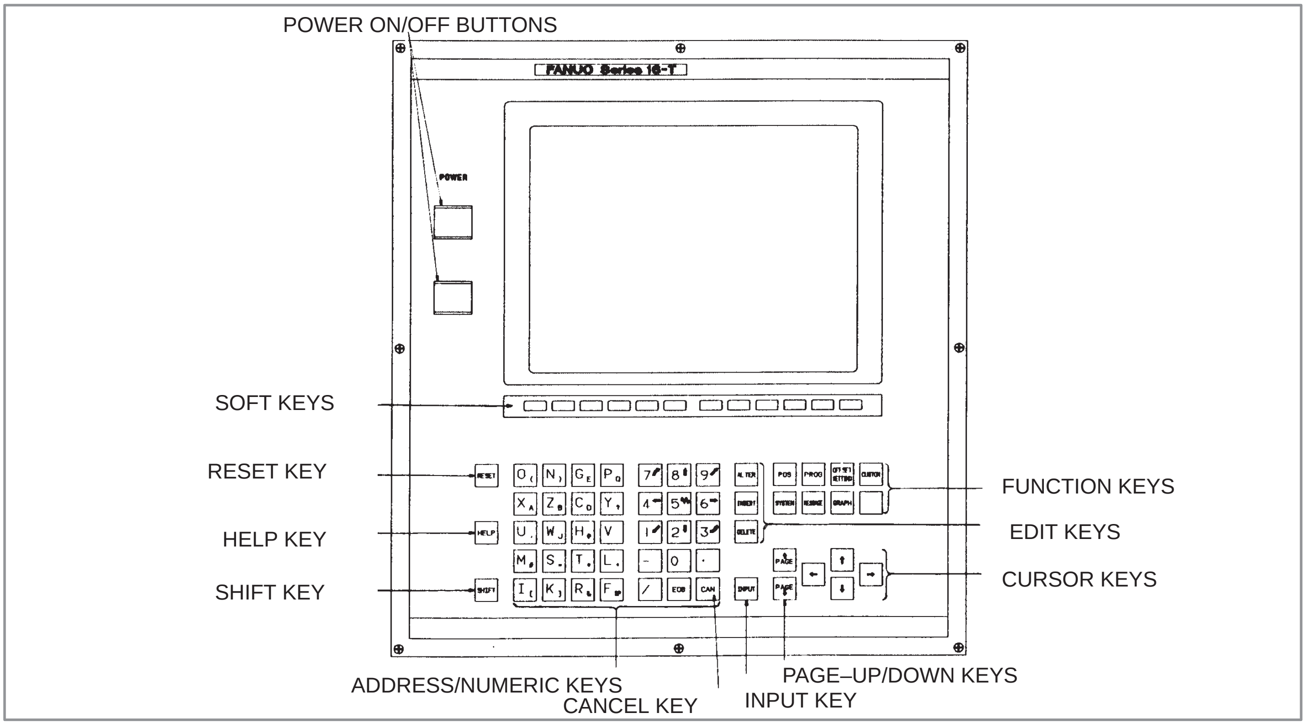
2.1.9 8.4–inch Color LCD (Separate Type)



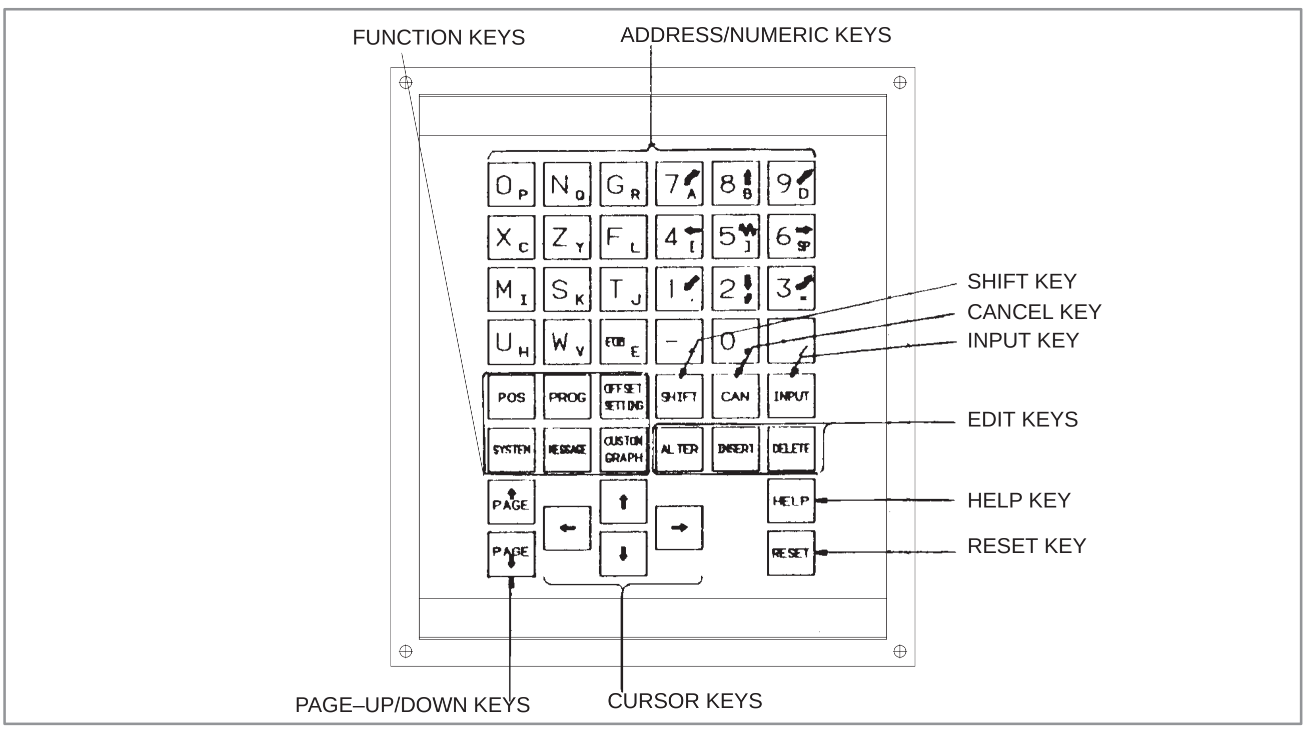
2.1.10 9.5-inch Color LCD/MDI (Horizontal Type)



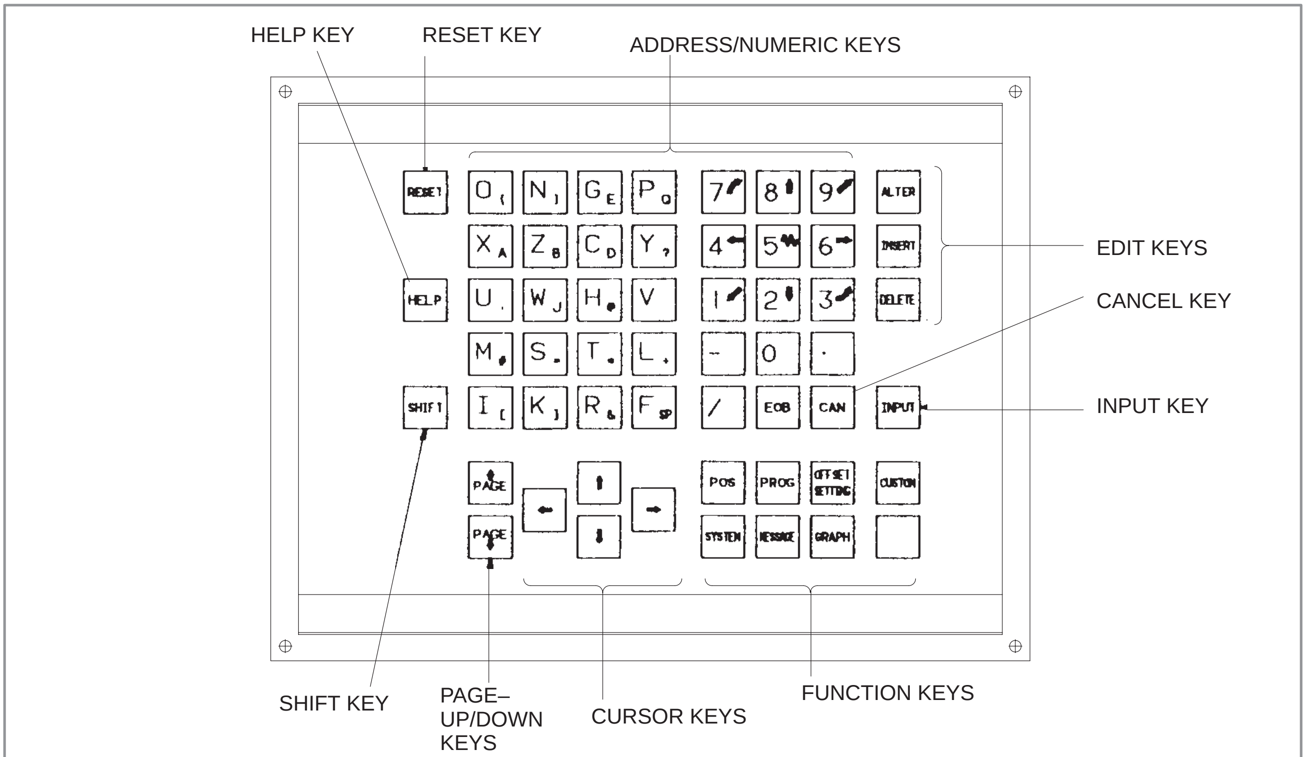
2.1.11
9.5-inch Color LCD/MDI
(Vertical Type)



2.1.12
Separate Type MDI
(Small Type)



2.1.13 Separate Type MDI (Standard Type)



Explanation of the keyboard

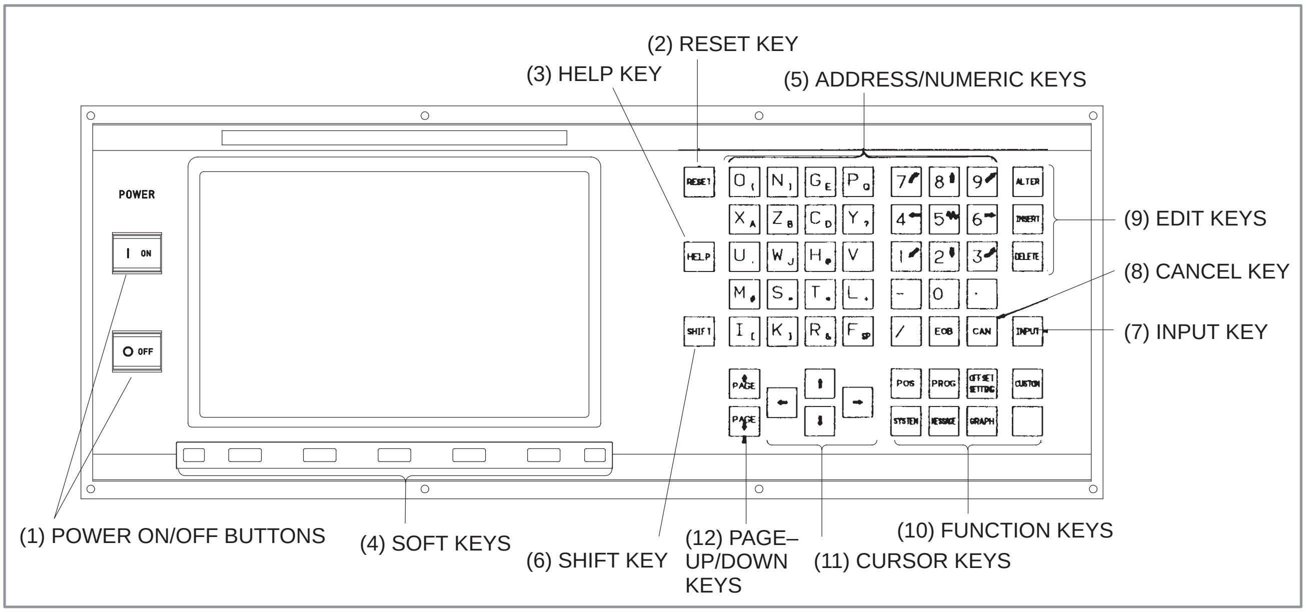


Fig2.1(g) 9” monochrome or color CRT/MDI panel (horizontal type)

Table2.1 Explanation of the MDI keyboard

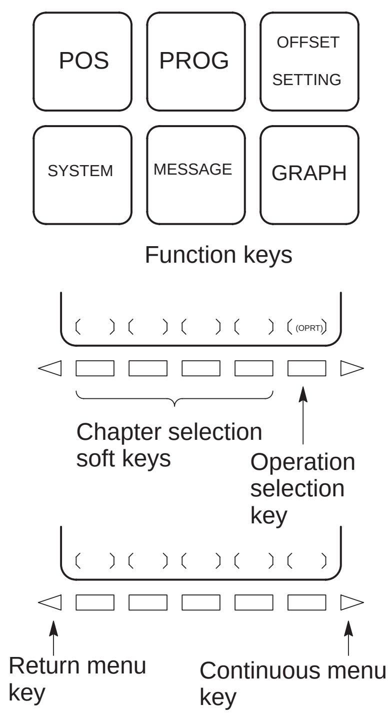
Number	Name	Explanation
1	Power ON and OFF buttons ○ OFF | ON	Press theses buttons to turn CNC power ON and OFF.
2	RESET key RESET	Press this key to reset the CNC, to cancel an alarm, etc.
3	HELP key HELP	Press this key to display how to operate the machine tool, such as MDI key operation, or the details of an alarm which occurred in the CNC (Help function).
4	Soft keys	The soft keys have various functions, according to the Applications. The soft key functions are displayed at the bottom of the CRT screen.
5	Address and numeric keys N) 4 ← …	Press these keys to input alphabetic, numeric, and other characters.
6	SHIFT key SHIFT	Some keys have two characters on their keytop. Pressing the <SHIFT> key switches the characters. Special character Ê is displayed on the screen when a character indicated at the bottom right corner on the keytop can be entered.
7	INPUT key INPUT	When an address or a numerical key is pressed, the data is input to the buffer, and it is displayed on the CRT screen. To copy the data in the key input buffer to the offset register, etc., press the <INPUT> key. This key is equivalent to the [INPUT] key of the soft keys, and either can be pressed to produce the same result.

Table2.1 Explanation of the MDI keyboard

Number	Name	Explanation
8	Cancel key [CAN]	Press this key to delete the last character or symbol input to the key input buffer. When the key input buffer displays >N001X100Z_ and the cancel [CAN] key is pressed, Z is canceled and >N001X100_ is displayed.
9	Program edit keys [ALTER] [INSERT] [DELETE]	Press these keys when editing the program. [ALTER] : Alteration [INSERT] : Insertion [DELETE] : Deletion
10	Function keys [POS] [PROG] …	Press theses keys to switch display screens for each function. See sec. 2.2 for details of the function keys.
11	Cursor move keys [←] [↑] [↓] [→]	There are four different cursor move keys. [→] : This key is used to move the cursor to the right or in the forward direction. The cursor is moved in short units in the forward direction. [←] : This key is used to move the cursor to the left or in the reverse direction. The cursor is moved in short units in the reverse direction. [↓] : This key is used to move the cursor in a downward or forward direction. The cursor is moved in large units in the forward direction. [↑] : This key is used to move the cursor in an upward or reverse direction. The cursor is moved in large units in the reverse direction.
11	Page change keys [↑ PAGE] [PAGE ↓]	Two kinds of page change keys are described below. [PAGE ↓] : This key is used to changeover the page on the CRT screen in the forward direction. [↑ PAGE] : This key is used to changeover the page on the CRT screen in the reverse direction.

2.2 FUNCTION KEYS AND SOFT KEYS

2.2.1 General Screen Operations



1 Press a function key on the CRT/MDI (or LCD/MDI) panel. The chapter selection soft keys that belong to the selected function appear.

2 Press one of the chapter selection soft keys. The screen for the selected chapter appears. If the soft key for a target chapter is not displayed, press the continuous menu key (next–menu key).
In some cases, additional chapters can be selected within a chapter.

3 When the target chapter screen is displayed, press the operation selection key to display data to be manipulated.

4 To redisplay the chapter selection soft keys, press the return menu key.

The general screen display procedure is explained above. However, the actual display procedure varies from one screen to another. For details, see the description of individual operations.

2.2.2 Function Keys

Function keys are provided to select the type of screen to be displayed. The following function keys are provided on the CRT/MDI and LCD/MDI panels:

POS	Press this key to display the **position screen**.
PROG	Press this key to display the **program screen**.
OFFSET SETTING	Press this key to display the **offset/setting screen**.
SYSTEM	Press this key to display the **system screen**.
MESSAGE	Press this key to display the **message screen.**
HELP	Press this key to display the **help screen.**
GRAPH	Press this key to display the **graphics screen**.

2.2.3 Soft Keys

To display a more detailed screen, press a function key followed by a soft key. Soft keys are also used for actual operations.
The following illustrates how soft key displays are changed by pressing each function key.

The symbols in the following figures mean as shown below :

: Indicates screens

: Indicates a screen that can be displayed by pressing a function key(*1)

[ ] : Indicates a soft key(*2)

( ) : Indicates input from the MDI panel.

[__] : Indicates a soft key displayed in green (or highlighted).

: Indicates the continuous menu key (rightmost soft key)(*3).

*1 Press function keys to switch between screens that are used frequently.

*2 Some soft keys are not displayed depending on the option configuration.

*3 In some cases, the continuous menu key is omitted when the 14” CRT display or 10” LCD is used.

POSITION SCREEN

Soft key transition triggered by the function key POS

- POS
  - **Absolute coordinate display**
    - [ABS] — [(OPRT)]
      - [PTSPRE] — [EXEC]
      - [RUNPRE] — [EXEC]
  - **Relative coordinate display**
    - [REL] — [(OPRT)]
      - (Axis or numeral) — [PRESET]
      - [ORIGIN]
        - [ALLEXE]
        - (Axis name) — [EXEC]
      - [PTSPRE] — [EXEC]
      - [RUNPRE] — [EXEC]
  - **Current position display**
    - [ALL] — [(OPRT)]
      - (Axis or numeral) — [PRESET]
      - [ORIGIN]
        - [ALLEXE]
        - (Axis name) — [EXEC]
      - [PTSPRE] — [EXEC]
      - [RUNPRE] — [EXEC]
  - **Handle interruption**
    - [HNDL] — [(OPRT)]
      - [PTSPRE] — [EXEC]
      - [RUNPRE] — [EXEC]
  - ▷
    - **Monitor screen**
      - [MONI] — [(OPRT)]
        - [PTSPRE] — [EXEC]
        - [RUNPRE] — [EXEC]

PROGRAM SCREEN

Soft key transition triggered by the function key PROG in the MEM mode

1/2

PROG

Program display screen

- [PRGRM] —— [(OPRT)] (1)
  - [BG–EDT] ⇨ See"When the soft key [BG–EDT] is pressed"
  - (O number) —— [O SRH]
  - (N number) —— [N SRH]
  - [REWIND]
  - ▷
  - [P TYPE]
  - [Q TYPE]
  - ▷
  - [F SRH] — [CAN] / [EXEC]

Program check display screen

- [CHECK] —— [ABS] / [REL] —— [(OPRT)]
  - [BG–EDT] ⇨ See"When the soft key [BG–EDT] is pressed"
  - (O number) —— [O SRH]
  - (N number) —— [N SRH]
  - [REWIND]
  - ▷
  - [P TYPE]
  - [Q TYPE]
  - ▷
  - [F SRH] — [CAN] / [EXEC]

Current block display screen

- [CURRNT] —— [(OPRT)] —— [BG–EDT] ⇨ See"When the soft key [BG–EDT] is pressed"

Next block display screen

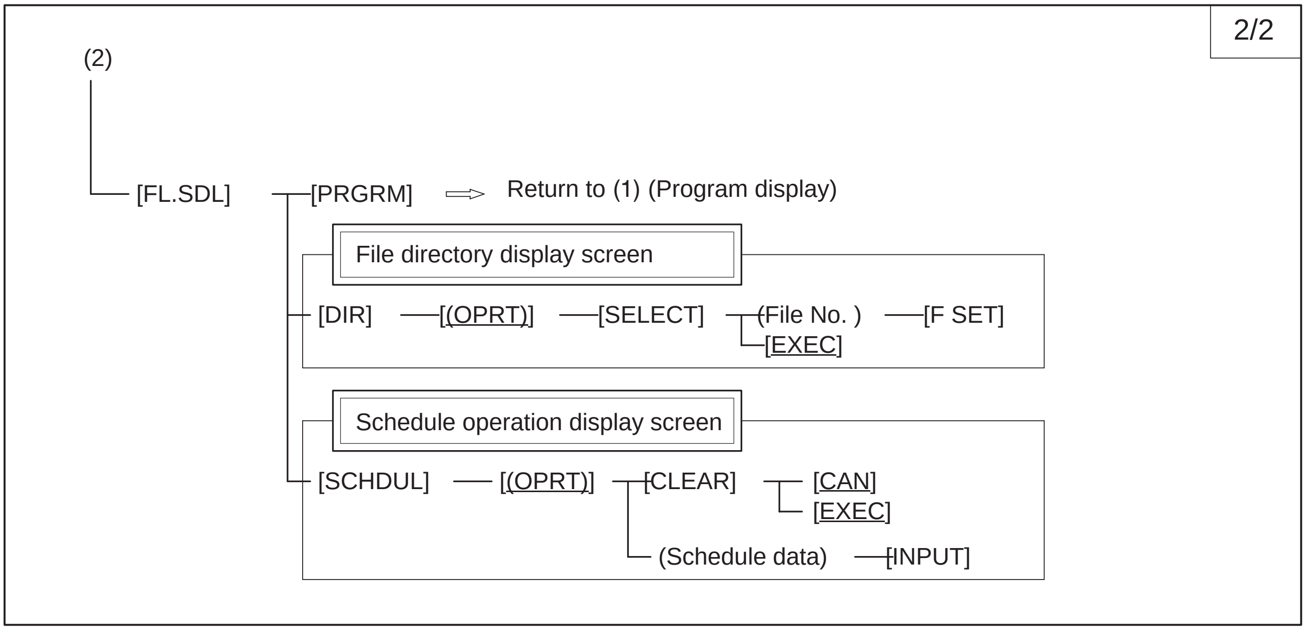
- [NEXT] —— [(OPRT)] —— [BG–EDT] ⇨ See"When the soft key [BG–EDT] is pressed"

▷

Program restart display screen

- [RSTR] —— [(OPRT)] —— [BG–EDT] ⇨ See"When the soft key [BG–EDT] is pressed"

(2)(Continued on the next page)


2/2
(2)
[FL.SDL]
[PRGRM]
Return to (1) (Program display)
File directory display screen
[DIR]
[(OPRT)]
[SELECT]
(File No.)
[F SET]
[EXEC]
Schedule operation display screen
[SCHDUL]
[(OPRT)]
[CLEAR]
[CAN]
[EXEC]
(Schedule data)
[INPUT]

PROGRAM SCREEN

Soft key transition triggered by the function key PROG in the EDIT mode

1/2

PROG

Program display

- [PRGRM] — [(OPRT)]
  - [BG–EDT] ⇒ See"When the soft key [BG–EDT] is pressed"
  - (O number) — [O SRH]
  - (Address) — [SRH↓]
  - (Address) — [SRH↑]
  - [REWIND]
  - ▷
  - [F SRH]
    - [CAN]
    - (N number) — [EXEC]
  - [READ]
    - [CHAIN] (The cursor moves to the end of a program.)
    - [STOP]
    - [CAN]
    - (O number) — [EXEC]
  - [PUNCH]
    - [STOP]
    - [CAN]
    - (O number) — [EXEC]
  - [DELETE]
    - [CAN]
    - (N number) — [EXEC]
  - [EX–EDT]
    - [COPY]
      - [CRSRA] / [ACRSR] / [ABTTM] / [ALL] — (O number) — [EXEC]
    - [MOVE]
      - [CRSRA] / [ACRSR] / [ABTTM] / [ALL] — (O number) — [EXEC]
    - [MERGE]
      - [ACRSR] / [ABTTM] — (O number) — [EXEC]
    - [CHANGE] — (Address) — [BEFORE] — (Address) — [AFTER]
      - [SKIP]
      - [1–EXEC]
      - [EXEC]

(1)(Continued on the next page)

2/2

(1)

Program directory display

- [LIB] — [(OPRT)]
  - [BG–EDT] ⇨ See”When the soft key [BG–EDT] is pressed”
  - (O number) — [O SRH] ⇨ Return to the program
  - ▷
  - [READ]
    - [CHAIN]
    - [STOP]
    - [CAN]
    - (O number) — [EXEC]
  - [PUNCH]
    - [STOP]
    - [CAN]
    - (O number) — [EXEC]

Graphic Conversational Programming

- [C.A.P.]
  - [PRGRM] ⇨ Return to the program
  - [G.MENU]
  - (G number) — [BLOCK] — (Data)
    - [LINE]
    - [CHAMF]
    - [CNR.R]
    - [INPUT]

  When a G number is omitted, the standard screen appears.

▷

Floppy directory display

- [FLOPPY]
  - [PRGRM] ⇨ Return to the program
  - [DIR] — [(OPRT)]
    - [F SRH]
      - (Numeral) — [F SET]
      - [CAN]
      - [EXEC]
    - [READ]
      - (Numeral) — [F SET]
      - (Numeral) — [O SET]
      - [STOP]
      - [CAN]
      - [EXEC]
    - [PUNCH]
      - (Numeral) — [F SET]
      - (Numeral) — [O SET]
      - [STOP]
      - [CAN]
      - [EXEC]
    - [DELETE]
      - (Numeral) — [F SET]
      - [CAN]
      - [EXEC]

PROGRAM SCREEN

Soft key transition triggered by the function key PROG in the MDI mode

PROG

Program display

[PRGRM] —— [(OPRT)] —— [BG–EDT] ⇒ See”When the soft key [BG–EDT] is pressed”

Program input screen

[MDI] —— [(OPRT)] —— [BG–EDT] ⇒ See”When the soft key [BG–EDT] is pressed”
- (Address) —— [SRH↓]
- (Address) —— [SRH↑]
- [REWIND]

Current block display screen

[CURRNT] —— [(OPRT)] —— [BG–EDT] ⇒ See”When the soft key [BG–EDT] is pressed”

Next block display screen

[NEXT] —— [(OPRT)] —— [BG–EDT] ⇒ See”When the soft key [BG–EDT] is pressed”

▷

Program restart display screen

[RSTR] —— [(OPRT)] —— [BG–EDT] ⇒ See”When the soft key [BG–EDT] is pressed”

PROGRAM SCREEN

Soft key transition triggered by the function key PROG in the HNDL, JOG, or REF mode

PROG

- Program display
  [PRGRM] —[(OPRT)] —[BG–EDT] ⇒ See"When the soft key [BG–EDT] is pressed"
- Current block display screen
  [CURRNT] —[(OPRT)] —[BG–EDT] ⇒ See"When the soft key [BG–EDT] is pressed"
- Next block display screen
  [NEXT] —[(OPRT)] —[BG–EDT] ⇒ See "When the soft key [BG–EDT] is pressed"

▷

- Program restart display screen
  [RSTR] — [(OPRT)] —[BG–EDT] ⇒ See"When the soft key [BG–EDT] is pressed"

PROGRAM SCREEN

Soft key transition triggered by the function key PROG in the TJOG or THDL mode

PROG

- Program display
  [PRGRM] —[(OPRT)] —[BG–EDT] ⇒ See "When the soft key [BG–EDT] is pressed"
  - (O number) —[O SRH] ⇒ Return to the program
  - (Address) —[SRH↓]
  - (Address) —[SRH↑]
  - [REWIND]
- Program directory display
  [LIB] —[(OPRT)] —[BG–EDT] ⇒ See "When the soft key [BG–EDT] is pressed"
  - (O number) —[O SRH] ⇒ Return to the program

PROGRAM SCREEN

Soft key transition triggered by the function key PROG
(When the soft key [BG–EDT] is pressed in all modes)

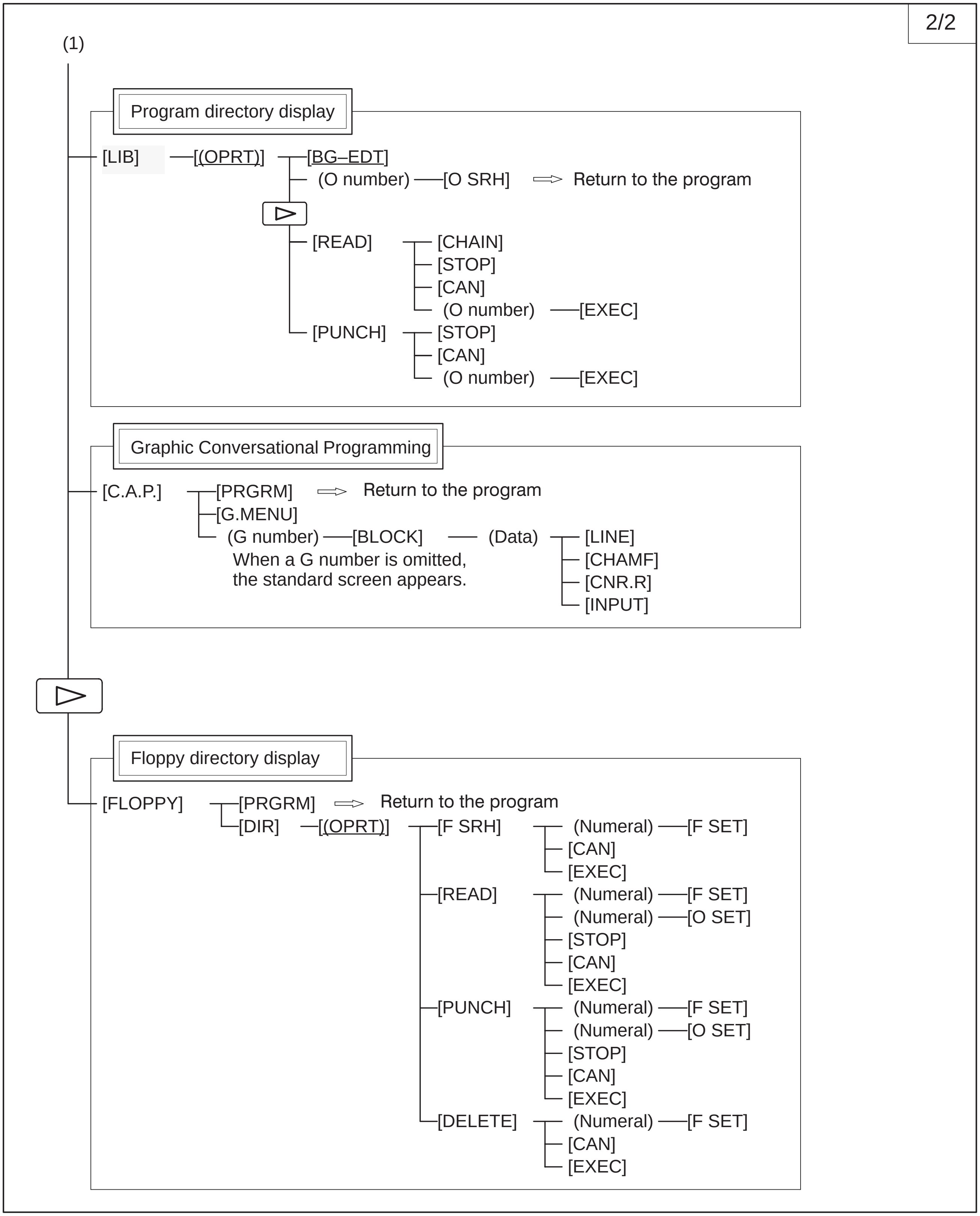
1/2

PROG

Program display

- [PRGRM] — [(OPRT)]
  - [BG–END]
  - (O number) — [O SRH]
  - (Address) — [SRH↓]
  - (Address) — [SRH↑]
  - [REWIND]
  - ▷
  - [F SRH]
    - [CAN]
    - (N number) — [EXEC]
  - [READ]
    - [CHAIN] (The cursor moves to the end of a program.)
    - [STOP]
    - [CAN]
    - (O number) — [EXEC]
  - [PUNCH]
    - [STOP]
    - [CAN]
    - (O number) — [EXEC]
  - [DELETE]
    - [CAN]
    - (N number) — [EXEC]
  - [EX–EDT]
    - [COPY]
      - [CRSRA] / [ACRSR] / [ABTTM] / [ALL] — (O number) — [EXEC]
    - [MOVE]
      - [CRSRA] / [ACRSR] / [ABTTM] / [ALL] — (O number) — [EXEC]
    - [MERGE]
      - [ACRSR] / [ABTTM] — (O number) — [EXEC]
    - [CHANGE] — (Address) — [BEFORE] — (Address) — [AFTER]
      - [SKIP]
      - [1–EXEC]
      - [EXEC]

(1)(Continued on the next page)


2/2
(1)
Program directory display
[LIB]
[(OPRT)]
[BG–EDT]
(O number)
[O SRH]
Return to the program
[READ]
[CHAIN]
[STOP]
[CAN]
(O number)
[EXEC]
[PUNCH]
[STOP]
[CAN]
(O number)
[EXEC]
Graphic Conversational Programming
[C.A.P.]
[PRGRM]
Return to the program
[G.MENU]
(G number)
[BLOCK]
(Data)
[LINE]
[CHAMF]
[CNR.R]
[INPUT]
When a G number is omitted, the standard screen appears.
Floppy directory display
[FLOPPY]
[PRGRM]
Return to the program
[DIR]
[(OPRT)]
[F SRH]
(Numeral)
[F SET]
[CAN]
[EXEC]
[READ]
(Numeral)
[F SET]
(Numeral)
[O SET]
[STOP]
[CAN]
[EXEC]
[PUNCH]
(Numeral)
[F SET]
(Numeral)
[O SET]
[STOP]
[CAN]
[EXEC]
[DELETE]
(Numeral)
[F SET]
[CAN]
[EXEC]

OFFSET/SETTING SCREEN

Soft key transition triggered by the function key OFFSET SETTING

1/2

OFFSET SETTING

Tool offset screen

- [OFFSET]
  - [WEAR] / [GEOM]
    - [(OPRT)]
      - (Number) — [NO SRH]
      - (Axis name and numeral) — [MEASUR]
      - (Axis name) — [INP.C.]
      - (Numeral) — [+INPUT]
      - (Numeral) — [INPUT]
      - ▷
      - [CLEAR] — [ALL] / [WEAR] / [GEOM]
      - [READ] — [CAN] / [EXEC]
      - [PUNCH] — [CAN] / [EXEC]

Setting screen

- [SETING]
  - [(OPRT)]
    - (Number) — [NO SRH]
    - [ON:1]
    - [OFF:0]
    - (Numeral) — [+INPUT]
    - (Numeral) — [INPUT]

Work coordinate system setting screen

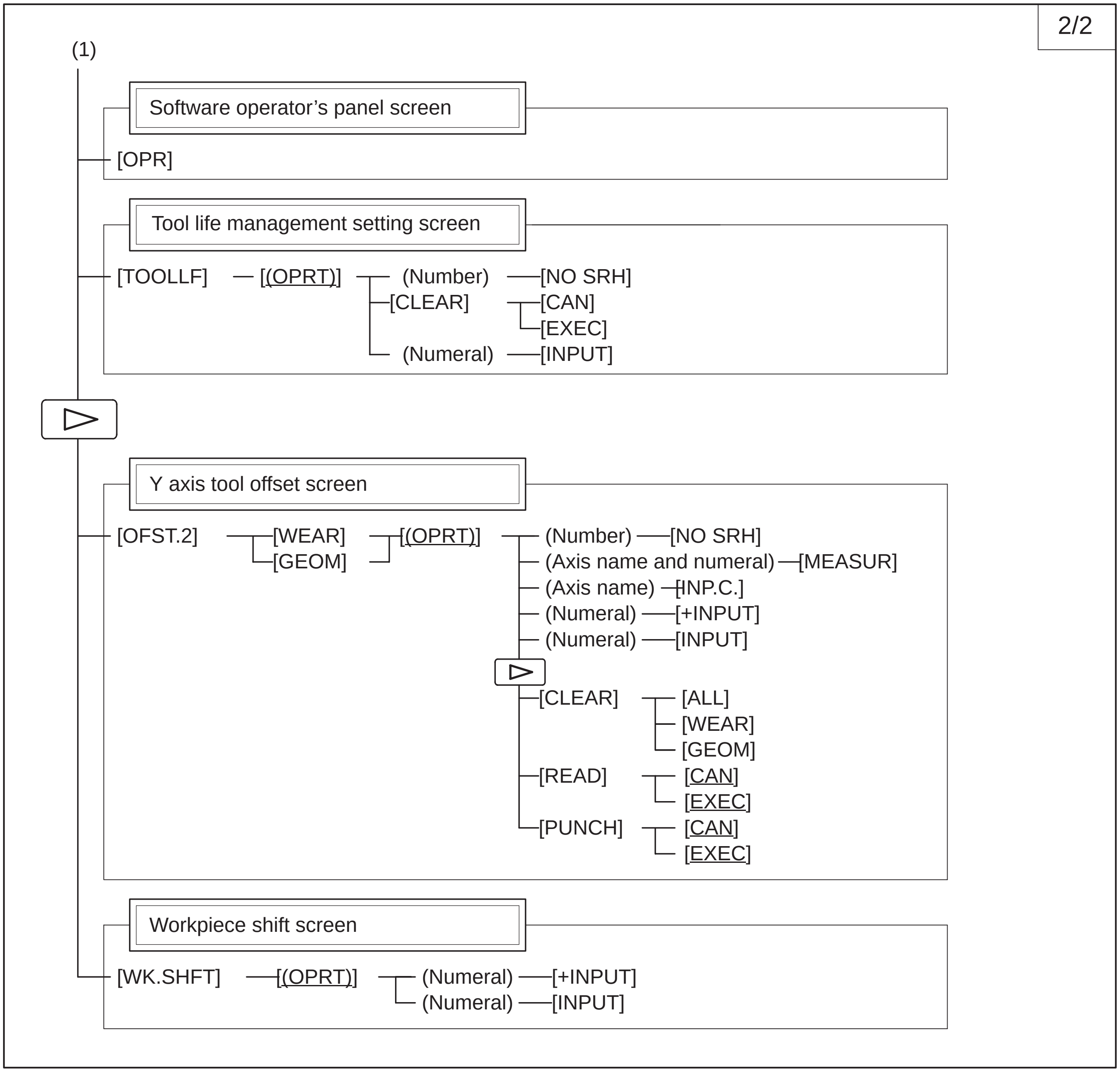
- [WORK]
  - [(OPRT)]
    - (Number) — [NO SRH]
    - (Axis name and numeral) — [MEASUR]
    - (Numeral) — [+INPUT]
    - (Numeral) — [INPUT]

▷

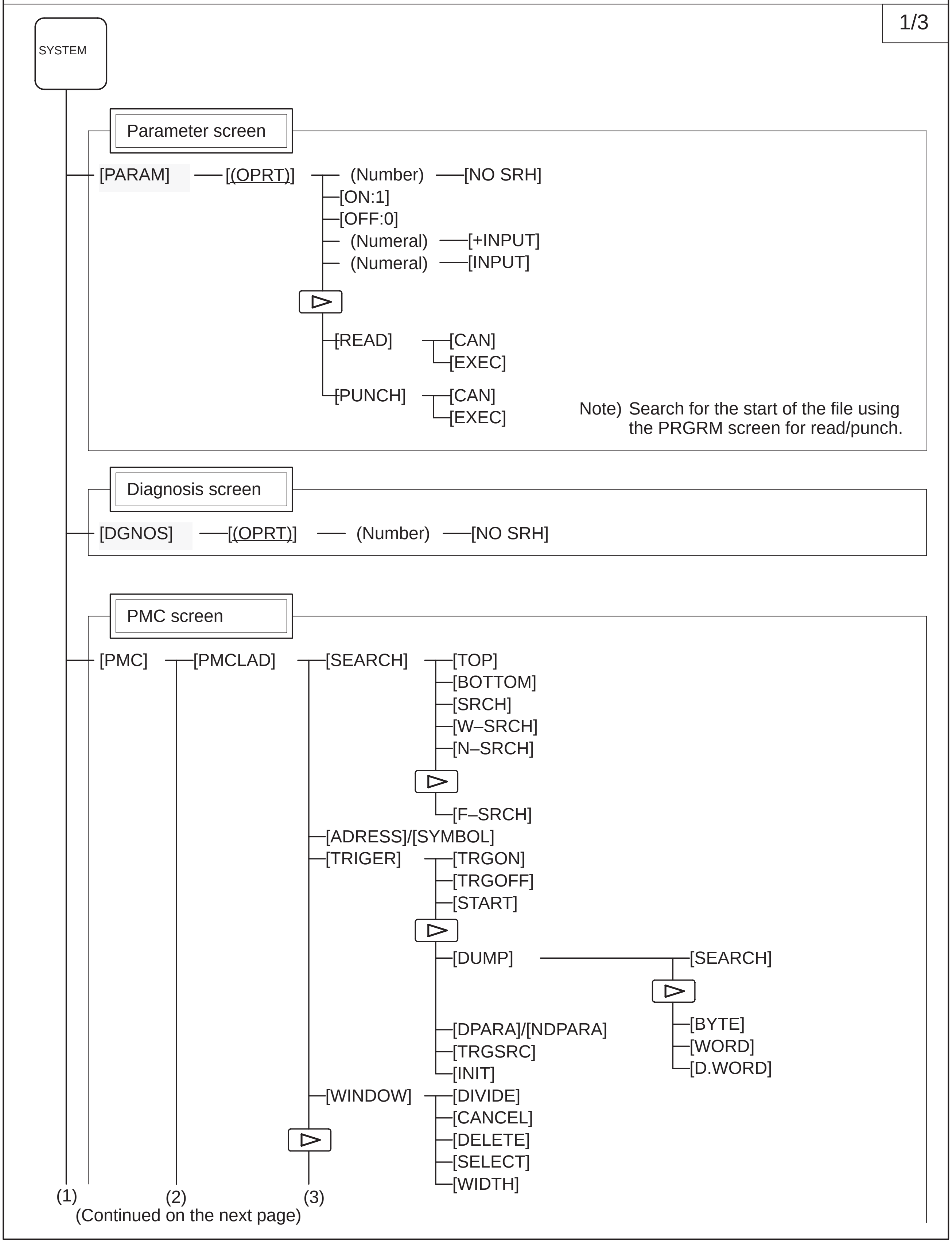
Macro variables display screen

- [MACRO]
  - [(OPRT)]
    - (Number) — [NO SRH]
    - (Axis name) — [INP.C.]
    - (Numeral) — [INPUT]
    - ▷
    - [PUNCH] — [CAN] / [EXEC]

(1)(Continued on the next page)


2/2
(1)
Software operator’s panel screen
[OPR]
Tool life management setting screen
[TOOLLF]
[(OPRT)]
(Number)
[NO SRH]
[CLEAR]
[CAN]
[EXEC]
(Numeral)
[INPUT]
Y axis tool offset screen
[OFST.2]
[WEAR]
[GEOM]
[(OPRT)]
(Number)
[NO SRH]
(Axis name and numeral)
[MEASUR]
(Axis name)
[INP.C.]
(Numeral)
[+INPUT]
(Numeral)
[INPUT]
[CLEAR]
[ALL]
[WEAR]
[GEOM]
[READ]
[CAN]
[EXEC]
[PUNCH]
[CAN]
[EXEC]
Workpiece shift screen
[WK.SHFT]
[(OPRT)]
(Numeral)
[+INPUT]
(Numeral)
[INPUT]

SYSTEM SCREEN Soft key transition triggered by the function key SYSTEM



2/3

(1) (2) (3)

- (1)
  - (2)
    - (3)
      - [DUMP]
        - [SEARCH]
        - ▷
        - [BYTE]
        - [WORD]
        - [D.WORD]
      - [DRARA]/[NDPARA]
    - [PMCDGN]
      - [TITLE]
      - [STATUS]
        - [SEARCH]
      - [ALARM]
      - [TRACE]
        - [T.DISP]/[TRCPRM]
        - [EXEC]
      - With PMC–RC only:
        - [M.SRCH]
          - [SEARCH]
          - [INPUT]
        - [ANALYS]
          - [SCOPE]
            - [SGNPRM]
            - [START]/[STOP]
            - [T–SRCH]
            - [ADRESS]/[SYMBOL]
            - [EXCHG]
              - [SELECT]
                - [TO]
                  - [EXEC]
              - [CANCEL]
          - [DELETE]
          - [INIT]
          - [ADRESS]/[SYMBOL]
    - [PMCPRM]
      - [TIMER]
      - [COUNTR]
      - [KEEPRL]
      - [DATA]
        - [G.DATA]
          - [C.DATA]
          - [G–SRCH]
          - [SEARCH]
        - [G.CONT]
        - [NO.SRH]
        - [INIT]
      - [SETTING]
        - [YES]/[MANUAL]/[ROM]
        - [NO]/[AUTO]/[RAM]
    - ▷
    - [STOP]/[RUN]
    - [I/O]
      - [EXEC]
      - [CANCEL]
      - (No.)
      - ▷
      - [SPEED]
        - [INPUT]
        - [INIT]
      - [MDI]/[ROM]
  - System configuration screen
    - [SYSTEM]
  - ▷
  - (4)

(Continued on the next page)

3/3

(4)

Pitch error compensation screen

- [PITCH] — [(OPRT)]
  - (No.) — [NO SRH]
  - [ON:1]
  - [OFF:0]
  - (Numeral) — [+INPUT]
  - (Numeral) — [INPUT]
  - ▷
    - [READ]
      - [CAN]
      - [EXEC]
    - [PUNCH]
      - [CAN]
      - [EXEC]

Note) Search for the start of the file using the PRGRM screen for read/punch.

Servo parameter screen

- [SV.PRM]
  - [SV.SET] / [SV.TUN] — [(OPRT)]
    - [ON:1]
    - [OFF:0]
    - (Numeral) — [INPUT]
  - [SV.TRC] — [(OPRT)]
    - [TRACE]
    - [TRNSF]

Spindle parameter screen

- [SP.PRM]
  - [SP.SET] / [SP.TUN] / [SP.MON] — [(OPRT)]
    - [ON:1]
    - [OFF:0]
    - [INPUT]

▷

Waveform diagnosis screen

- [W.DGNS]
  - [W.PRM]
  - [W.GRPH]
    - [STSRT]
    - [TIME→]
    - [←TIME]
    - [H–DOBL]
    - [H–HALF]
    - ▷
      - [STSRT]
      - [CH–1↑]
      - [CH–1↓]
      - [V–DOBL]
      - [V–HALF]
      - ▷
        - [STSRT]
        - [CH–2↑]
        - [CH–2↓]
        - [V–DOBL]
        - [V–HALF]

MESSAGE SCREEN

Soft key transition triggered by the function key MESSAGE

MESSAGE

Alarm display screen
- [ALARM]

Message display screen
- [MSG]

Alarm history screen
- [HISTRY] —— [(OPRT)] —— [CLEAR]

HELP SCREEN

Soft key transition triggered by the function key HELP

HELP

Alarm detail screen
- [1 ALAM] —— [(OPRT)] —— [SELECT]

Operation method screen
- [2 OPR] —— [(OPRT)] —— [SELECT]

Parameter table screen
- [3 PARA]

GRAPHIC SCREEN

Soft key transition triggered by the function key GRAPH

Tool path graphics

GRAPH

Tool path graphics

- [G.PRM] —— [(OPRT)] —— [NORMAL]
- [GRAPH] —— [(OPRT)]
  - [HEAD]
  - [ERASE]
  - [PROCES]
  - [EXEC]
  - [STOP]
- [ZOOM] —— [(OPRT)]
  - [ACT]
  - [HI/LO]

2.2.4 Key Input and Input Buffer

When an address and a numerical key are pressed, the character corresponding to that key is input once into the key input buffer. The contents of the key input buffer is displayed at the bottom of the CRT screen.

In order to indicate that it is key input data, a ”>” symbol is displayed immediately in front of it. A ”_” is displayed at the end of the key input data indicating the input position of the next character.

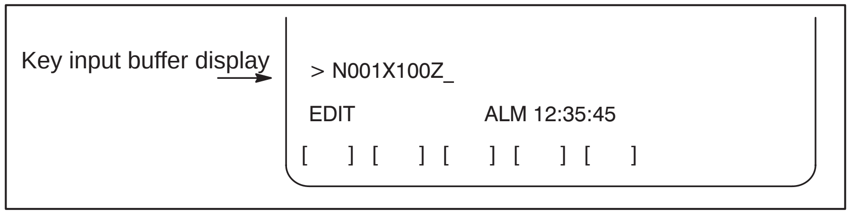


Fig. 2.2.4 Key input buffer display

To input the lower character of the keys that have two characters inscribed on them, first press the SHIFT key and then the key in question.

When the SHIFT key is pressed, ”_” indicating the next character input position changes to ” ∧ ”. Now lowercase characters can be entered (shift state).

When a character is input in shift status the shift status is canceled. Furthermore, if the SHIFT key is pressed in shift status, the shift status is canceled.

It is possible to input up to 32 characters at a time in the key input buffer.

Press the CAN key to cancel a character or symbol input in the key input buffer.

(Example)

When the key input buffer displays
>N001X100Z_
and the cancel CAN key is pressed, Z is canceled and
>N001X100_
is displayed.

2.2.5 Warning Messages

After a character or number has been input from the MDI panel, a data check is executed when INPUT key or a soft key is pressed. In the case of incorrect input data or the wrong operation a flashing warning message will be displayed on the status display line.

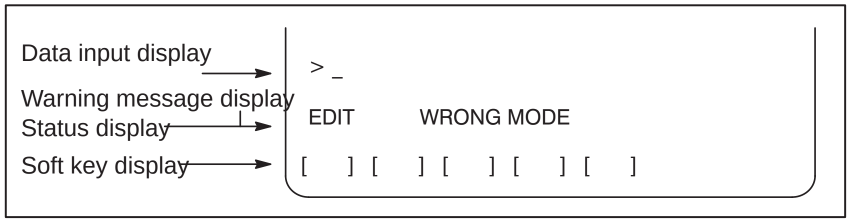


Fig. 2.2.5 Warning message display

Table2.2.5 Warning Messages

Warning message	Content
FORMAT ERROR	The format is incorrect.
WRITE PROTECT	Key input is invalid because of data protection key or the parameter is not write enabled.
DATA IS OUT OF RANGE	The input value exceeds the permitted range.
TOO MANY DIGITS	The input value exceeds the permitted number of digits.
WRONG MODE	Parameter input is not possible in any mode other than MDI mode.
EDIT REJECTED	It is not possible to edit in the current CNC status.

2.2.6 14” CRT, 9.5” LCD, and 8.4” LCD Soft Key Configuration

There are 12 soft keys in the 14” CRT/MDI, 9.5” LCD/MDI, or 8.4” LCD/MDI panel. As illustrated below, the 5 soft keys on the right and those on the right and left edges operate in the same way as the 9” CRT, whereas the 5 keys on the left hand side are expansion keys dedicated to the 14” CRT, 9.5” LCD, or 8.4” LCD.

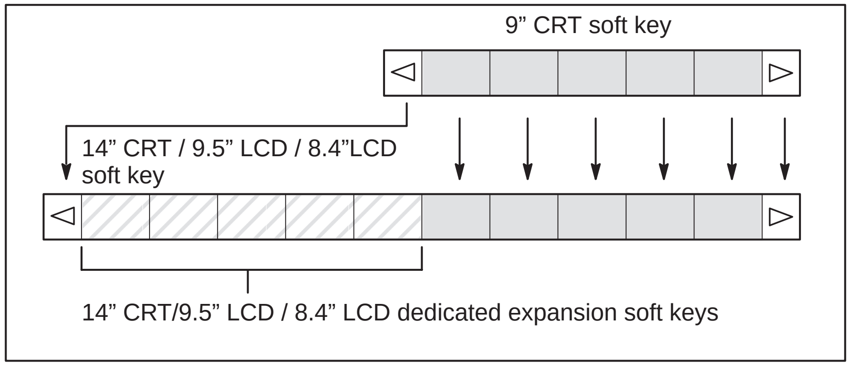
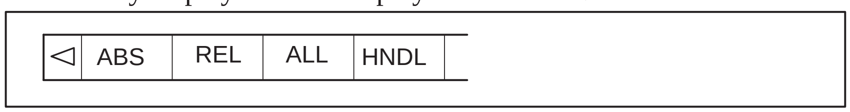


Fig. 2.2.6(a) CRT and LCD soft key configuration

Whenever a position display appears in the left half of the screen after a function key other than POS is pressed, the soft keys on the left half of the soft key display area are displayed as follows:



The soft key corresponding to the position display is indicated in reverse video.

2.3 EXTERNAL I/O DEVICES

Five types of external input/output devices are available. This section outlines each device. For details on these devices, refer to the corresponding manuals listed below.

Table 2.3(a) External I/O device

Device name	Usage	Max. storage capacity	Reference manual
FANUC Handy File	Easy–to–use, multi function input/output device. It is designed for FA equipment and uses floppy disks.	3600m	B–61834E
FANUC Floppy Cassette	Input/output device. Uses floppy disks.	2500m	B–66040E
FANUC FA Card	Compact input/output device. Uses FA cards.	160m	B–61274E
FANUC PPR	Input/output device consisting of a paper tape reader, tape punch, and printer.	275m	B–58584E
Portable Tape Reader	Input device for reading paper tape.	______	Appendix H

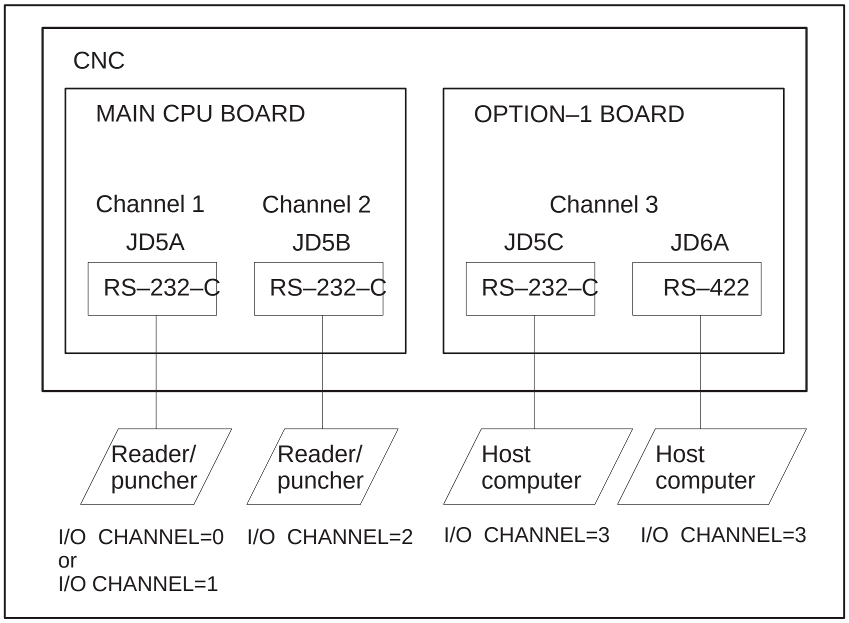
The following data can be input/output to or from external input/output devices:

1. Programs
2. Offset data
3. Parameters
4. Custom macro common variables
5. Pitch error compensation data

For how data is input and output, see Chapter III–8.

Parameter

Before an external input/output device can be used, parameters must be set as follows.

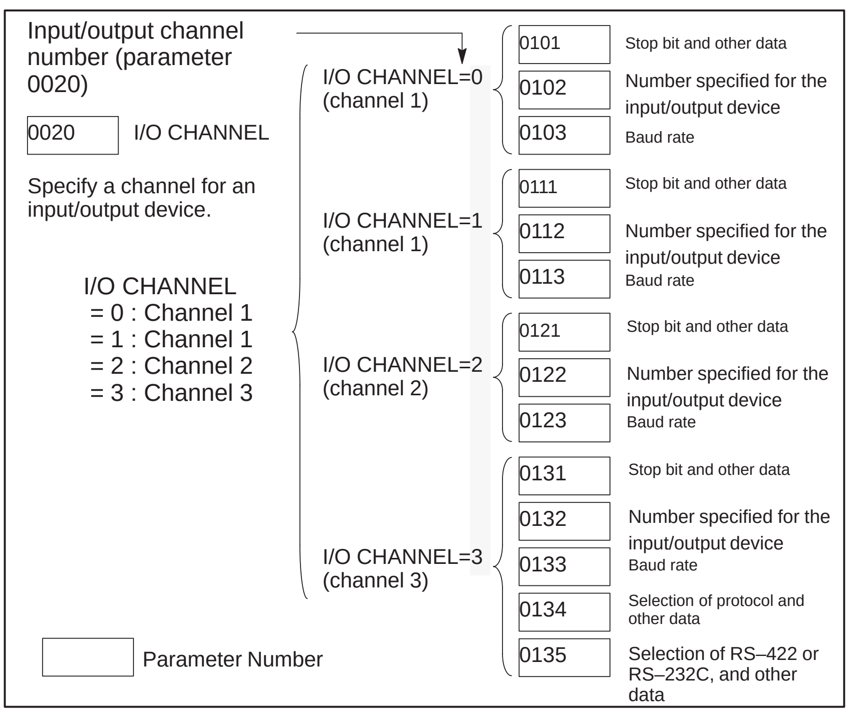


This CNC has three channels of reader/punch interfaces. The input/output device to be used is specified by setting the channel connected to that device in setting parameter I/O CHANNEL.

The specified data, such as a baud rate and the number of stop bits, of an input/output device connected to a specific channel must be set in parameters for that channel in advance.

For channel 1, two combinations of parameters to specify the input/output device data are provided.

The following shows the interrelation between the reader/punch interface parameters for the channels.

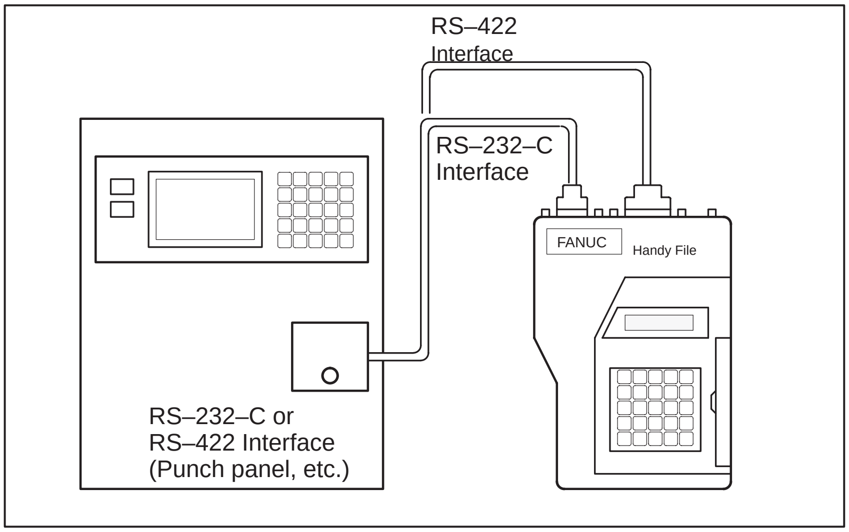


2.3.1 FANUC Handy File

The Handy File is an easy–to–use, multi function floppy disk input/output device designed for FA equipment. By operating the Handy File directly or remotely from a unit connected to the Handy File, programs can be transferred and edited.

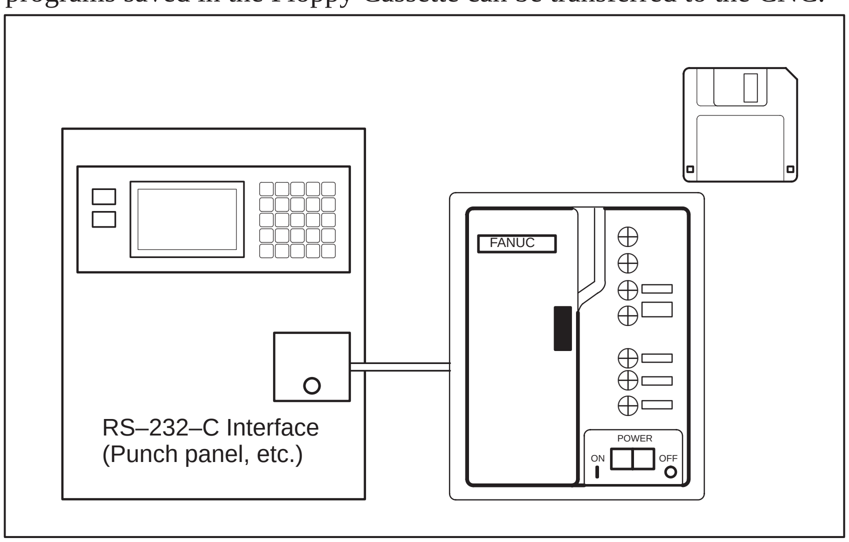
The Handy File uses 3.5–inch floppy disks, which do not have the problems of paper tape (i.e., noisy during input/output, easily broken, and bulky).

One or more programs (up to 1.44M bytes, which is equivalent to the memory capacity of 3600–m paper tape) can be stored on one floppy disk.



2.3.2 FANUC Floppy Cassette

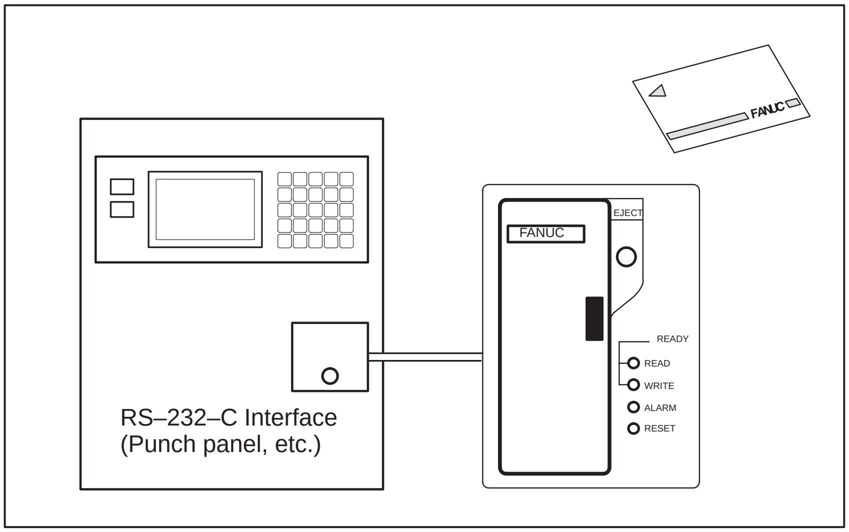
When the Floppy Cassette is connected to the CNC, machining programs stored in the CNC can be saved on a Floppy Cassette, and machining programs saved in the Floppy Cassette can be transferred to the CNC.



2.3.3 FANUC FA Card

An FA Card is a memory card used as an input medium in the FA field. It is compact, but has a large memory capacity with high reliability, and requires no special maintenance.

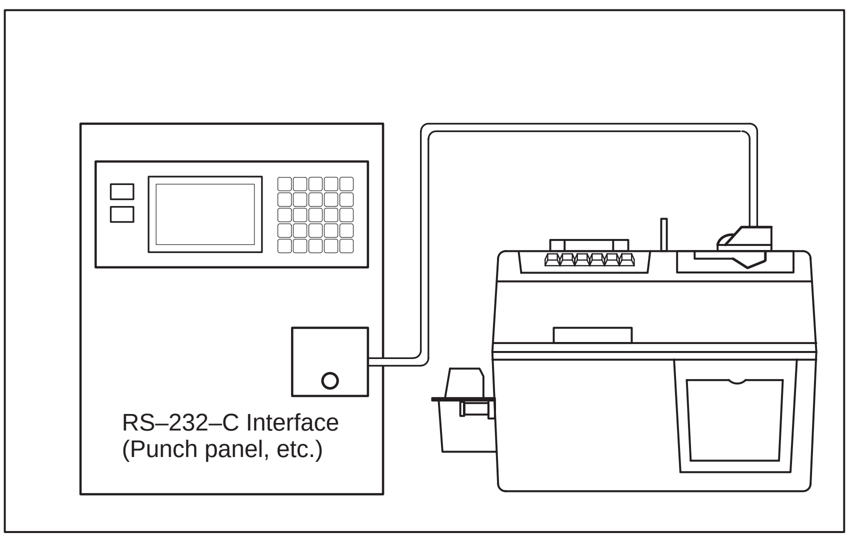
When an FA Card is connected to the CNC via the card adapter, machining programs stored in the CNC can be transferred to and saved in an FA Card. Machining programs stored on an FA Card can also be transferred to the CNC.



2.3.4 FANUC PPR

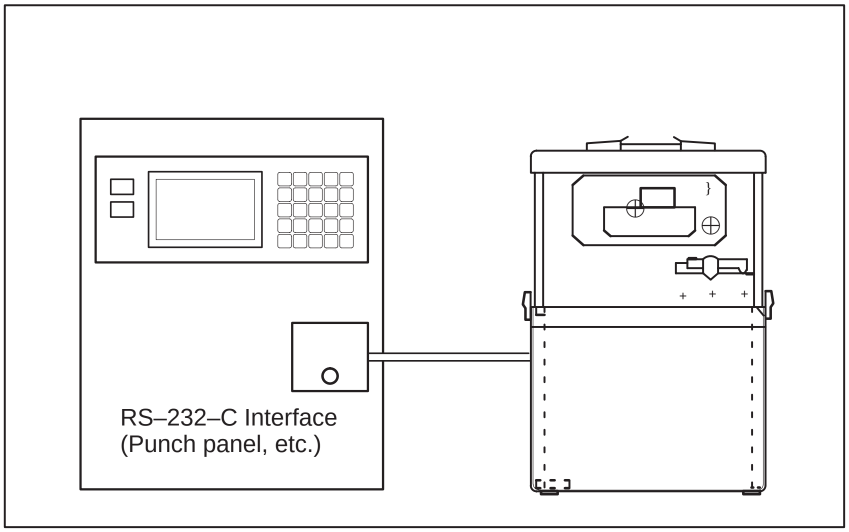
The FANUC PPR consists of three units: A printer, paper tape punch, and paper tape reader.

When the PPR is used alone, data can be read from the tape reader and printed or punched out. It is also possible to perform TH and TV checks on data that was read.



2.3.5 Portable Tape Reader

The portable tape reader is used to input data from paper tape.



2.4 POWER ON/OFF

2.4.1 Turning on the Power

Procedure of turning on the power

1 Check that the appearance of the CNC machine tool is normal. (For example, check that front door and rear door are closed.)

2 Turn on the power according to the manual issued by the machine tool builder.

3 After the power is turned on, check that the position screen is displayed. An alarm screen is displayed if an alarm occurs upon power–on. If the screen shown in Section III–2.4.2 is displayed, a system failure may have occurred.
If the machine tool is in the emergency stop state, the software configuration screen, shown in III–2.4.2, appears.
Press the POS function key on the CRT/MDI or LCD/MDI, or release the machine from emergency stop.
The position screen then appears. If the position screen is not displayed, a system failure may have occurred.

```
ACTUAL POSITION(ABSOLUTE)      O1000 N00010

   X        123.456
   Z          0.000

              PART COUNT       5
  RUN TIME    0H15M     CYCLE TIME   0H 0M38S
  ACT.F     3000 MM/M         S   0 T0000

MEM **** *** ***          09:06:35
[  ABS  ] [  REL  ] [  ALL  ] [  HNDL ] [ OPRT ]
```

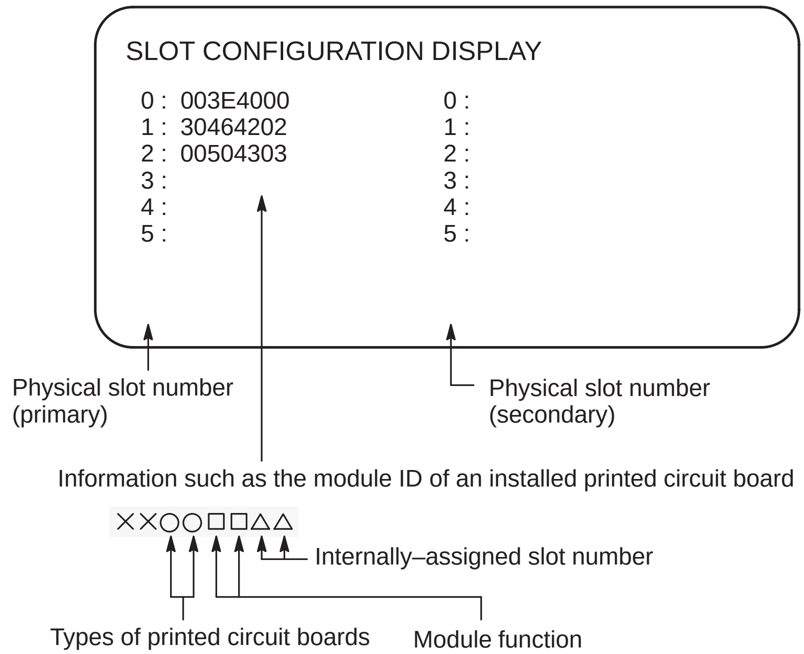
4 Check that the fan motor is rotating.

> **Note**
> Until the positional or alarm screen is displayed at the power on, do not touch them. Some keys are used for the maintenance or special operation purpose. When they are pressed, unexpected operation may be caused.

2.4.2 Screen Displayed at Power–on

If a hardware failure or installation error occurs, the system displays one of the following three types of screens then stops.
Information such as the type of printed circuit board installed in each slot is indicated. This information and the LED states are useful for failure recovery.

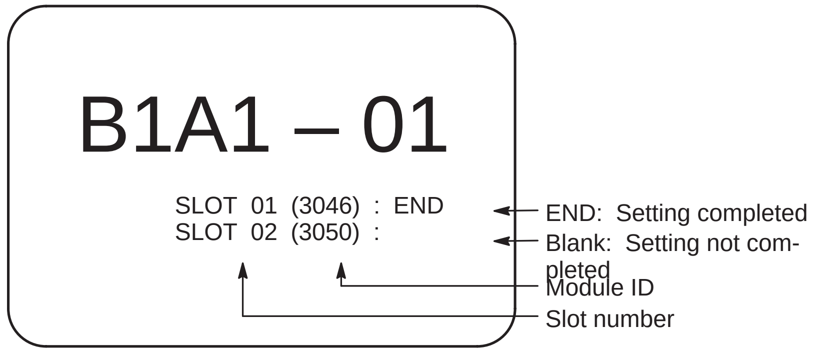
Slot status display



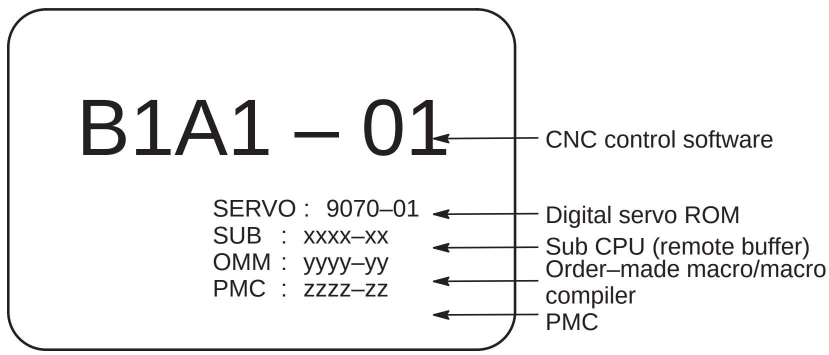
Module ID
3E : Main CPU
3F : Remote buffer
41 : PMC–RC
45 : Graphics
46 :
50 : SUB CPU
53 : C.A.P.II

Software ID
40 : CNC
41 : PMC
42 :
43 : SUB
45 : Graphics (C.A.P.I)
46 : Graphics (C.A.P.II)
49 : C.A.P.II
4A : Remote buffer
4F : PMC–RC

Screen indicating module setting status



Display of software configuration



2.4.3 Power Disconnection

Procedure for Poser Disconnection

1 Check that the LED indicating the cycle start is off on the operator's panel.

2 Check that all movable parts of the CNC machine tool is stopping.

3 If an external input/output device such as the Handy File is connected to the CNC, turn off the external input/output device.

4 Continue to press the POWER OFF pushbutton for about 5 seconds.

5 Refer to the machine tool builder's manual for turning off the power to the machine.

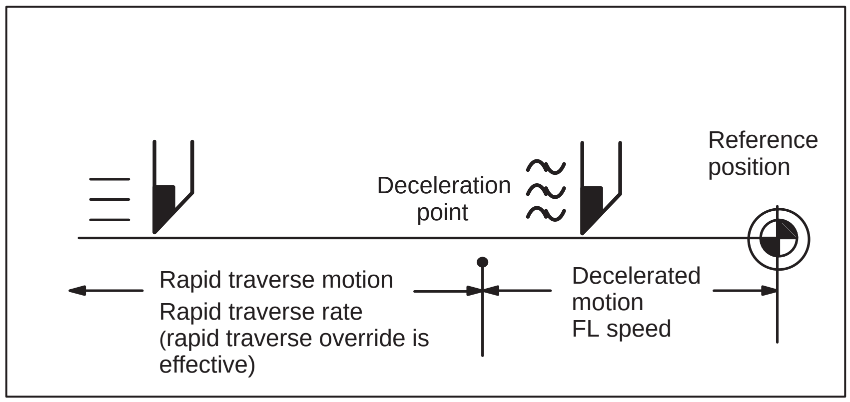
3 MANUAL OPERATION

MANUAL OPERATION are four kinds as follows :

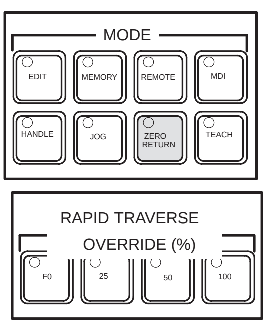
3.1 Manual reference position return
3.2 Manual continuous feed
3.3 Incremental feed
3.4 Manual handle feed
3.5 Manual absolute on/off

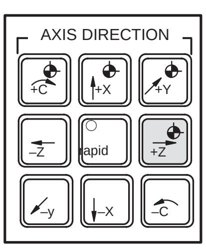
3.1 MANUAL REFERENCE POSITION RETURN

The tool is returned to the reference position as follows :
The tool is moved in the direction specified in parameter ZMI (bit 5 of No. 1006) for each axis with the reference position return switch on the machine operator's panel. The tool moves to the deceleration point at the rapid traverse rate, then moves to the reference position at the FL speed. The rapid traverse rate and FL speed are specified in parameters (No. 1420,1421, and 1425).
Four step rapid traverse override is effective during rapid traverse.
When the tool has returned to the reference position, the reference position return completion LED goes on. The tool generally moves along only a single axis, but can move along three axes simultaneously when specified so in parameter JAX(bit 0 of No.1002).



Procedure for Manual Reference Position Return



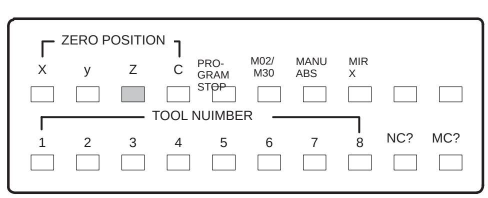


1 Press the reference position return switch, one of the mode selection switches.

2 To decrease the feedrate, press a rapid traverse override switch.

3 Press the feed axis and direction selection switch corresponding to the axis and direction for reference position return. Continue pressing the switch until the tool returns to the reference position. The tool can be moved along three axes simultaneously when specified so in an appropriate parameter setting. The tool moves to the deceleration point at the rapid traverse rate, then moves to the reference position at the FL speed set in a parameter.
When the tool has returned to the reference position, the reference position return completion LED goes on.

4 Perform the same operations for other axes, if necessary.
The above is an example. Refer to the appropriate manual provided by the machine tool builder for the actual operations.



Explanation

- **Automatically setting the coordinate system**

Bit 0 (ZPR) of parameter No. 1201 is used for automatically setting the coordinate system. When ZPR is set, the coordinate system is automatically determined when manual reference position return is performed.
When α and γ are set in parameter 1250, the workpiece coordinate system is determined so that the reference point on the tool holder or the position of the tip of the reference tool is X=α,Z=γ when reference position return is performed. This has the same effect as specifying the following command for reference position return:
G92X$\underline{\alpha}$Zγ;

Restrictions

- **Moving the tool again**

Once the REFERENCE POSITION RETURN COMPLETION LED lights at the completion of reference position return, the tool does not move unless the REFERENCE POSITION RETURN switch is turned off.

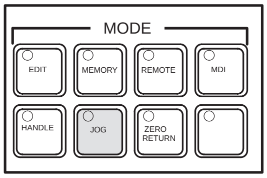
- **Reference position return completion LED**

The REFERENCE POSITION RETURN COMPLETION LED is extinguished by either of the following operations:
– Moving from the reference position.
– Entering an emergency stop state.

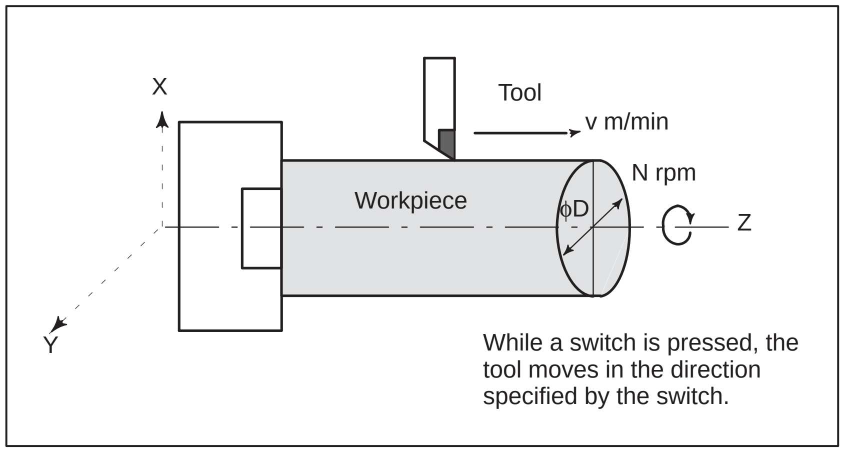
- **The distance to return to reference position**

For the distance (Not in the deceleration condition) to return the tool to the reference position, refer to the manual issued by the machine tool builder.

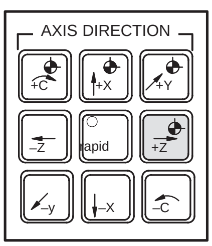
3.2 MANUAL CONTINUOUS FEED



In the manual continuous mode, pressing a feed axis and direction selection switch on the machine operator's panel continuously moves the tool along the selected axis in the selected direction.
The manual continuous feedrate is specified in a parameter (No.1423)
The manual continuous feedrate can be adjusted with the manual continuous feedrate override dial.
Pressing the rapid traverse switch moves the tool at the rapid traverse feedrate regardless of the position of the manual continuous feedrate override dial.
Manual operation is allowed for one axis at a time. 3 axes can be selected at a time by parameter JAX (No.1002#0).



Procedure for Manual Continuous Feed



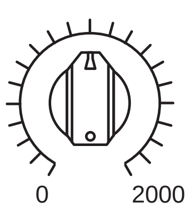
1 Press the manual continuous switch, one of the mode selection switches.

2 Press the feed axis and direction selection switch corresponding to the axis and direction the tool is to be moved. While the switch is pressed, the tool moves at the feedrate specified in a parameter (No. 1423). The tool stops when the switch is released.

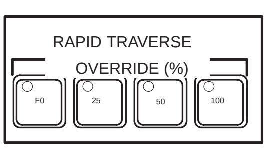
3 The manual continuous feedrate can be adjusted with the manual continuous feedrate override dial.

4 Pressing the rapid traverse switch while pressing a feed axis and direction selection switch moves the tool at the rapid traverse rate while the rapid traverse switch is pressed. Rapid traverse override by the rapid traverse override switches is effective during rapid traverse.

The above is an example. Refer to the appropriate manual provided by the machine tool builder for the actual operations.



JOG FEED RATE OVERRIDE



Explanations

- **Manual synchronous feed**

To enable manual synchronous feed, set bit 4 (JRV) of parameter No. 1402 to 1.
During manual synchronous feed, the tool is jogged at the following feedrate:
Feed distance per rotation of the spindle (mm/rev) (specified with parameter No. 1423) x manual continuous feedrate override x actual spindle speed (rev/min).

Restrictions

- **Acceleration/decelera–tion for rapid traverse**

Feedrate, time constant and method of automatic acceleration/ deceleration for manual rapid traverse are the same as G00 in programmed command.

- **Change of modes**

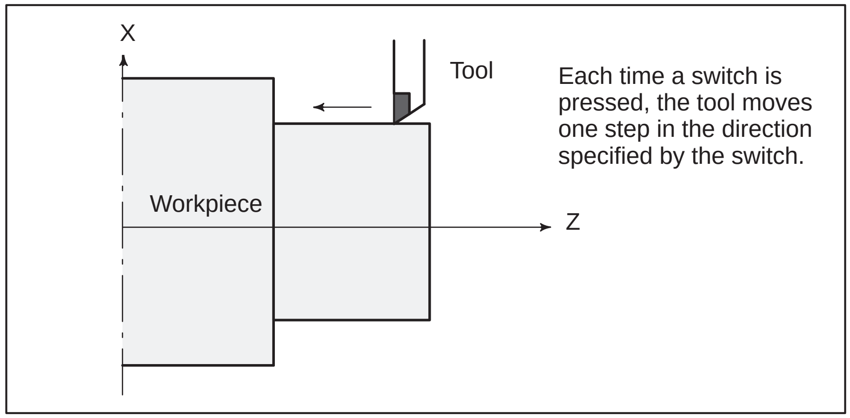
Changing the mode to the manual continuous feed (JOG) mode while pressing a feed axis and direction selection switch does not enable manual continuous feed. To enable manual continuous feed, enter the manual continuous feed (JOG) mode first, then press a feed axis and direction selection switch.

- **Rapid traverse prior to reference position return**

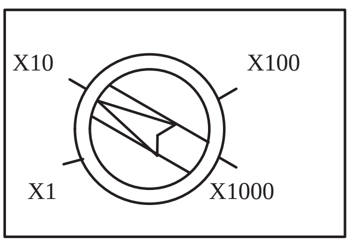
If reference position return is not performed after power–on, pushing RAPID TRAVERSE button does not actuate the rapid traverse but the remains at the manual continuous feedrate. This function can be disabled by setting parameter RPD (No.1401#01).

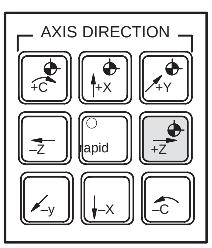
3.3 INCREMENTAL FEED

In the incremental (INC) mode, pressing a feed axis and direction selection switch on the machine operator's panel moves the tool one step along the selected axis in the selected direction. The minimum distance the tool is moved is the least input increment. Each step can be 10, 100, or 1000 times the least input increment.
This mode is effective when a manual pulse generator is not connected.



Procedure for Incremental Feed





1 Press the INC switch, one of the mode selection switches.

2 Select the distance to be moved for each step with the magnification dial.

3 Press the feed axis and direction selection switch corresponding to the axis and direction the tool is to be moved. Each time a switch is pressed, the tool moves one step. The feedrate is the same as the jog feedrate.

4 Pressing the rapid traverse switch while pressing a feed axis and direction selection switch moves the tool at the rapid traverse rate. Rapid traverse override by the rapid traverse override switch is effective during rapid traverse.

The above is an example. Refer to the appropriate manual provided by the machine tool builder for the actual operations.

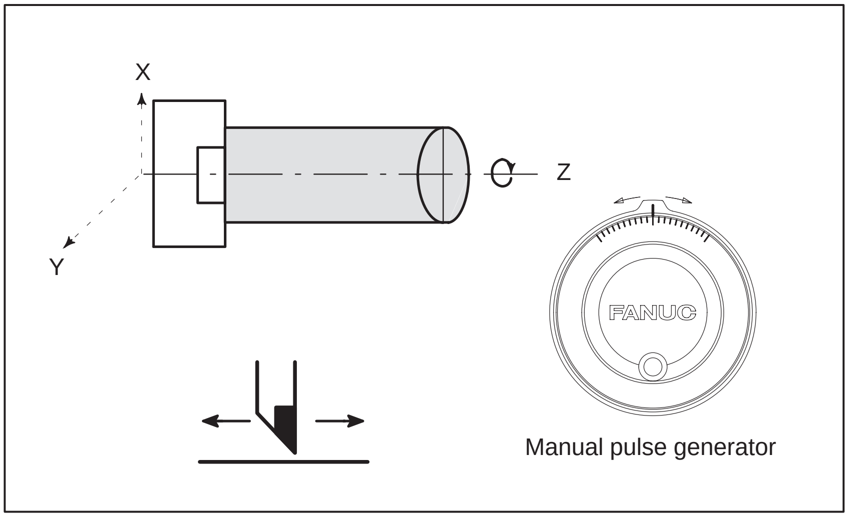
Explanation

- **Travel distance specified with a diameter**

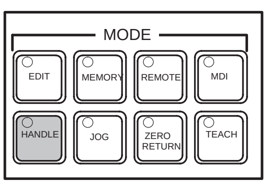
The distance the tool travels along the X–axis can be specified with a diameter.

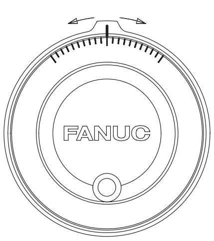
3.4 MANUAL HANDLE FEED

In the handle mode, the tool can be minutely moved by rotating the manual pulse generator on the machine operator's panel. Select the axis along which the tool is to be moved with the handle feed axis selection switches.
The minimum distance the tool is moved when the manual pulse generator is rotated by one graduation is equal to the least input increment. Or the distance the tool is moved when the manual pulse generator is rotated by one graduation can be magnified by 10 times or by one of the two magnifications specified by parameters (No. 7113 and 7114).



Procedure for Manual Handle Feed





Manual pulse generator

1 Press the HANDLE switch, one of the mode selection switches.

2 Select the axis along which the tool is to be moved by pressing a handle feed axis selection switch.

3 Select the magnification for the distance the tool is to be moved by pressing a handle feed magnification switch. The minimum distance the tool is moved when the manual pulse generator is rotated by one graduation is equal to the least input increment.

4 Move the tool along the selected axis by rotating the handle. Rotating the handle 360 degrees moves the tool the distance equivalent to 100 graduations.
The above is an example. Refer to the appropriate manual provided by the machine tool builder for the actual operations.

Explanation

- **Availability of manual pulse generator in Jog mode (JHD)**

Parameter JHD (bit 0 of No. 7100) enables or disables the manual pulse generator in the JOG mode.
When the parameter JHD(bit 0 of No. 7100) is set 1,both manual handle feed and incremental feed are enabled.

- **Availability of manual pulse generator in TEACH IN JOG mode (THD)**

Parameter THD (bit 1 of No. 7100) enables or disables the manual pulse generator in the TEACH IN JOG mode.

- **A command to the MPG exceeding rapid traverse rate**

Parameter HPF (bit 4 of No. 7100)
Parameter HPF (bit 4 of No. 7100) or (No. 7117) specifies as follows:

SET VALUE 0 : The feedrate is clamped at the rapid traverse rate and generated pulses exceeding the rapid traverse rate are ignored.(The distance the tool is moved may not match the graduations on the manual pulse generator.)

SET VALUE 1 : The feedrate is clamped at the rapid traverse rate and generated pulses exceeding the rapid traverse rate are not ignored but accumulated in the CNC.
(No longer rotating the handle does not immediately stop the tool. The tool is moved by the pulses accumulated in the CNC before it stops.)

- **Movement direction of an axis to the rotation of MPG (HNGx)**

Parameter HNGx (bit 0 of No. 7102) switches the direction in which the tool moves along an axis, corresponding to the direction in which the handle of the manual pulse generator is rotated.

Restrictions

- **Number of MPGs**

Up to three manual pulse generators can be connected, one for each axis.
The three manual pulse generators can be simultaneously operated.

Notes

1. Rotate the manual pulse generator at a rate of five rotations per second or lower. If the manual pulse generator is rotated at a rate higher than five rotations per second, the tool may not stop immediately after the handle is no longer rotated or the distance the tool moves may not match the graduations on the manual pulse generator.
2 Rotating the handle quickly with a large magnification such as x100 moves the tool too fast. The feedrate is clamped at the rapid traverse feedrate.

3.5 MANUAL ABSOLUTE ON AND OFF

Whether the distance the tool is moved by manual operation is added to the coordinates can be selected by turning the manual absolute switch on or off on the machine operator's panel. When the switch is turned on, the distance the tool is moved by manual operation is added to the coordinates. When the switch is turned off, the distance the tool is moved by manual operation is not added to the coordinates.

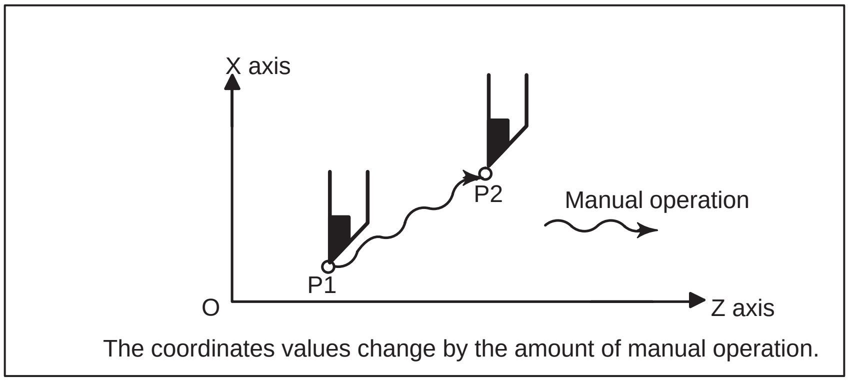


Fig. 3.5(a) Coordinates with the switch ON

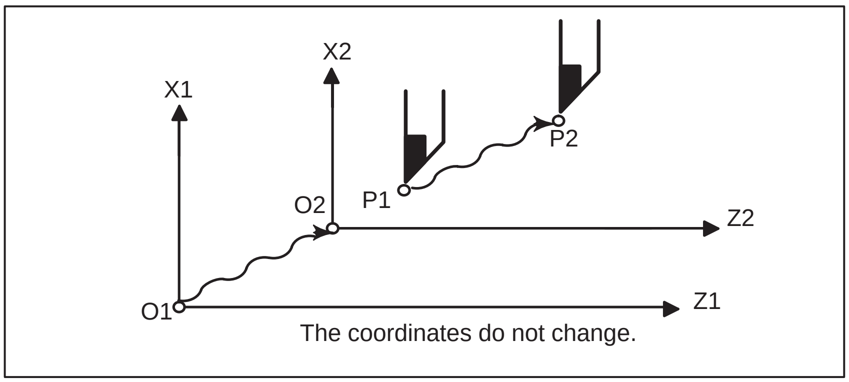


Fig. 3.5(b) Coordinates with the switch OFF

Explanation

The following describes the relation between manual operation and coordinates when the manual absolute switch is turned on or off, using a program example.

```
G01G90  X100.0Z100.0F010 ;        (1)
        X200.0Z150.0     ;        (2)
        X300.0Z200.0     ;        (3)
```

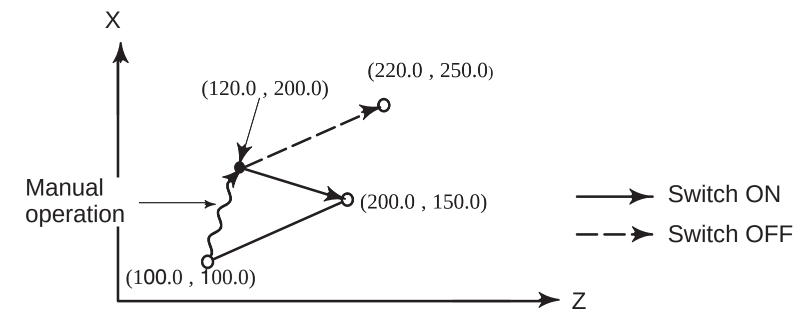
The subsequent figures use the following notation:

⟶ Movement of the tool when the switch is on
– – ➝ Movement of the tool when the switch is off

The coordinates after manual operation include the distance the tool is moved by the manual operation. When the switch is off, therefore, subtract the distance the tool is moved by the manual operation.

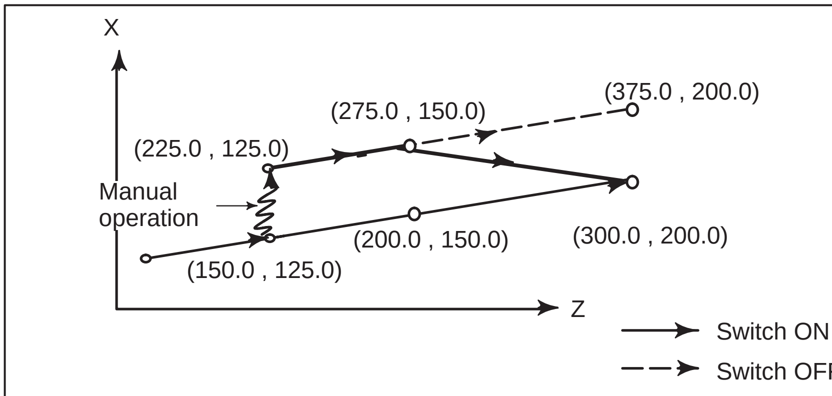
- **Manual operation after the end of block**

Coordinates when block (2) has been executed after manual operation (X–axis +20.0, Z–axis +100.0) at the end of movement of block (1).



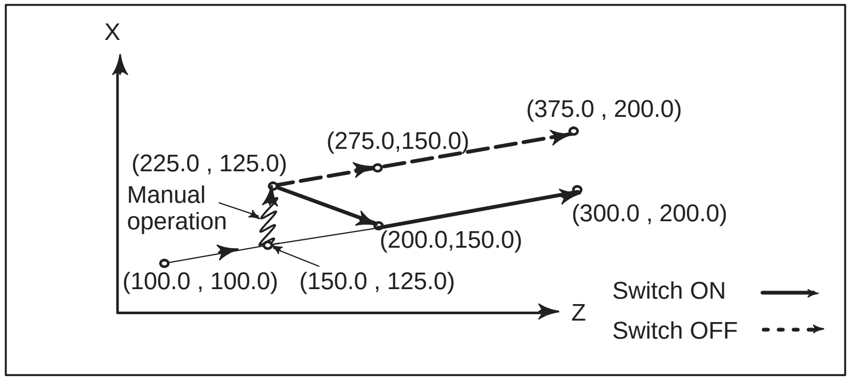
- **Manual operation after a feed hold**

Coordinates when the feed hold button is pressed while block (2) is being executed, manual operation (X–axis + 75.0) is performed, and the cycle start button is pressed and released



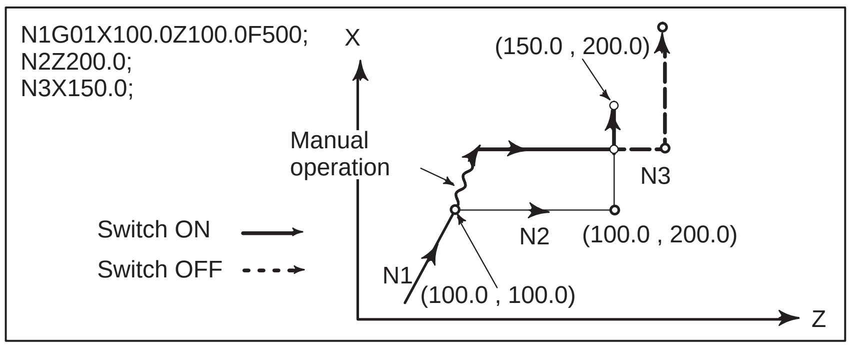
- **When reset after a manual operation following a feed hold**

Coordinates when the feed hold button is pressed while block (2) is being executed, manual operation (Y–axis +75.0) is performed, the control unit is reset with the RESET button, and block (2) is read again



- **When a movement command in the next block is only one axis**

When there is only one axis in the following command, only the commanded axis returns.



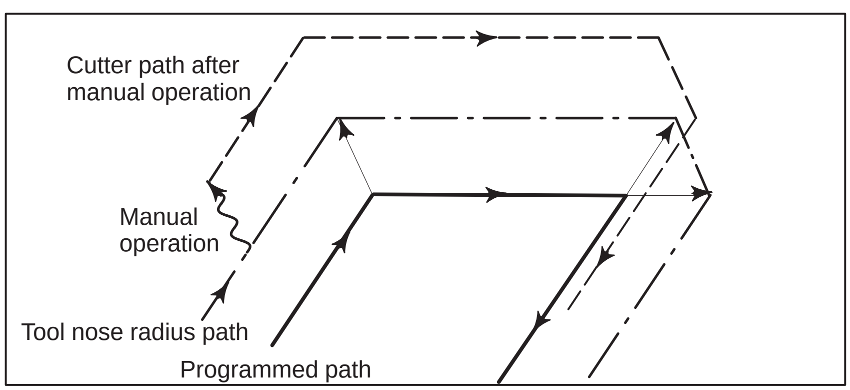
- **When the next move block is an incremental**

When the following commands are incremental commands, operation is the same as when the switch is OFF.

- **Manual operation during tool nose radius compensation**

When the switch is OFF

After manual operation is performed with the switch OFF during tool nose radius compensation, automatic operation is restarted then the tool moves parallel to the movement that would have been performed if manual movement had not been performed. The amount of separation equals to the amount that was performed manually.

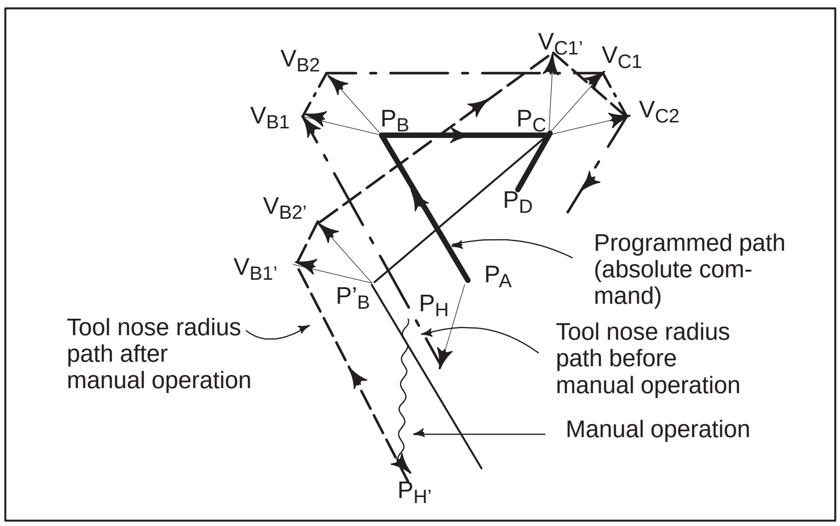


When the switch is ON during tool nose radius compensation

Operation of the machine upon return to automatic operation after manual intervention with the switch is ON during execution with an absolute command program in the tool nose radius compensation mode will be described. The vector created from the remaining part of the current block and the beginning of the next block is shifted in parallel. A new vector is created based on the next block, the block following the next block and the amount of manual movement. This also applies when manual operation is performed during cornering.

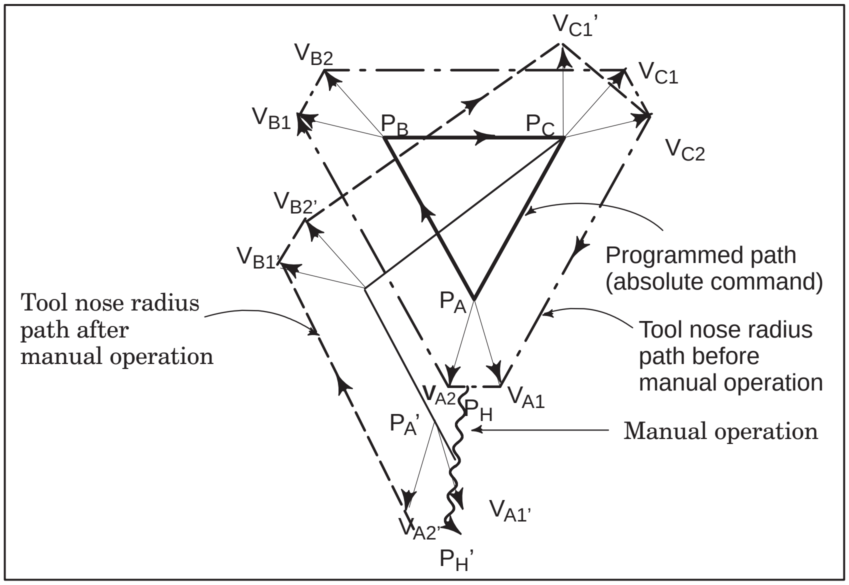
Manual operation performed in other than cornering

Assume that the feed hold was applied at point P_H while moving from P_A to P_B of programmed path P_A, P_B, and P_C and that the tool was manually moved to $P_{H'}$. The block end point P_B moves to the point $P_{B'}$ by the amount of manual movement, and vectors V_{B1} and V_{B2} at P_B also move to $V_{B1'}$ and $V_{B2'}$. Vectors V_{C1} and V_{C2} between the next two blocks P_B – P_C and P_C – P_D are discarded and new vectors $V_{C1'}$ and $V_{C2'}$ ($V_{C2'}$ = V_{C2} in this example) are produced from the relation between $P_{B'}$ – P_C and P_C – P_D. However, since $V_{B2'}$ is not a newly calculated vector, correct offset is not performed at block $P_{B'}$ – P_C. Offset is correctly performed after P_C.



Manual operation during cornering

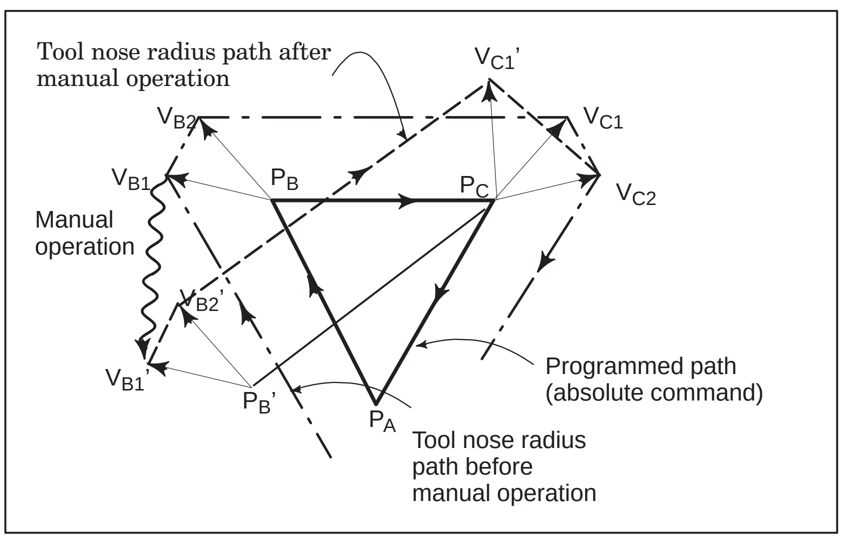
This is an example when manual operation is performed during cornering. $V_{A2'}$, $V_{B1'}$, and $V_{B2'}$ are vectors moved in parallel with V_{A2}, V_{B1} and V_{B2} by the amount of manual movement. The new vectors are calculated from V_{C1} and V_{C2}. Then correct tool nose radius compensation is performed for the blocks following Pc.



Manual operation after single block stop

Manual operation was performed when execution of a block was terminated by single block stop.

Vectors V_{B1} and V_{B2} are shifted by the amount of manual operation. Sub–sequent processing is the same as case a described above. An MDI operation can also be interveneted as well as manual operation. The movement is the same as that by manual operation.



4 AUTOMATIC OPERATION

Programmed operation of a CNC machine tool is referred to as automatic operation.
This chapter explains the following types of automatic operation:

- **MEMORY OPERATION**
  Operation by executing a program registered in CNC memory
- **MDI OPERATION**
  Operation by executing a program entered from the MDI panel
- **DNC operation**
  Operation while reading a program from an input/output device
- **PROGRAM RESTART**
  Restarting a program for automatic operation from an intermediate point
- **SCHEDULING FUNCTION**
  Scheduled operation by executing programs (files) registered in an external input/output device (Handy File, Floppy Cassette, or FA Card)
- **SUBPROGRAM CALL FUNCTION**
  Function for calling and executing subprograms (files) registered in an external input/output device (Handy File, Floppy Cassette, or FA Card) during memory operation
- **MANUAL HANDLE INTERRUPTION**
  Function for performing manual feed during movement executed by automatic operation
- **MIRROR IMAGE**
  Function for enabling mirror–image movement along an axis during automatic operation
- **MANUAL INTERVENTION AND RETURN**
  Function restarting automatic operation by returning the tool the position where manual intervention was started during automatic operation.

4.1 MEMORY OPERATION

Programs are registered in memory in advance. When one of these programs is selected and the cycle start switch on the machine operator's panel is pressed, automatic operation starts, and the cycle start LED goes on.
When the feed hold switch on the machine operator's panel is pressed during automatic operation, automatic operation is stopped temporarily. When the cycle start switch is pressed again, automatic operation is restarted.
When the reset switch on the CRT/MDI (or LCD/MDI) panel is pressed, automatic operation terminates and the reset state is entered.
For the two–path control, the programs for the two tool posts can be executed simultaneously so the two tool posts can operate independently at the same time.
The following procedure is given as an example. For actual operation, refer to the manual supplied by the machine tool builder.

Procedure for Memory Operation

1 Press the **MEMORY** mode selection switch.

2 Select a program from the registered programs. To do this, follow the steps below.

2–1 Press PROG to display the program screen.

2–2 Press address O .

2–3 Enter a program number using the numeric keys.

2–4 Press the **[O SRH]** soft key.
For the two–path control, select the program for the tool post to be operated. When operating the two tool posts at the same time, select a program for each tool post.

3 For the two–path control, select the tool post to be operated with the tool post selection switch on the machine operator's panel.

4 Press the cycle start switch on the machine operator's panel. Automatic operation starts, and the cycle start LED goes on. When automatic operation terminates, the cycle start LED goes off.

5 To stop or cancel memory operation midway through, follow the steps below.

a. Stopping memory operation
Press the feed hold switch on the machine operator's panel. The feed hold LED goes on and the cycle start LED goes off. The machine responds as follows:
(i) When the machine was moving, feed operation decelerates and stops.
(ii) When dwell was being performed, dwell is stopped.
(iii) When M, S, or T was being executed, the operation is stopped after M, S, or T is finished.

When the cycle start switch on the machine operator's panel is pressed while the feed hold LED is on, machine operation restarts.

b. Terminating memory operation

Press the RESET key on the CRT/MDI (or LCD/MDI) panel.

Automatic operation is terminated and the reset state is entered. When a reset is applied during movement, movement decelerates then stops.

Explanation

Memory operation

After memory operation is started, the following are executed:

(1) A one–block command is read from the specified program.

(2) The block command is decoded.

(3) The command execution is started.

(4) The command in the next block is read.

(5) Buffering is executed. That is, the command is decoded to allow immediate execution.

(6) Immediately after the preceding block is executed, execution of the next block can be started. This is because buffering has been executed.

(7) Hereafter, memory operation can be executed by repeating the steps (4) to (6).

Stopping and terminating memory operation

Memory operation can be stopped using one of two methods: Specify a stop command, or press a key on the machine operator's panel.

– The stop commands include M00 (program stop), M01 (optional stop), and M02 and M30 (program end).

– There are two keys to stop memory operation: The feed hold key and reset key.

• Program stop (M00)

Memory operation is stopped after a block containing M00 is executed. When the program is stopped, all existing modal information remains unchanged as in single block operation. The memory operation can be restarted by pressing the cycle start button. Operation may vary depending on the machine tool builder. Refer to the manual supplied by the machine tool builder.

• Optional stop (M01)

Similarly to M00, memory operation is stopped after a block containing M01 is executed. This code is only effective when the Optional Stop switch on the machine operator's panel is set to ON. Operation may vary depending on the machine tool builder. Refer to the manual supplied by the machine tool builder.

• Program end (M02, M30)

When M02 or M30 (specified at the end of the main program) is read, memory operation is terminated and the reset state is entered.
In some machines, M30 returns control to the top of the program. For details, refer to the manual supplied by the machine tool builder.

• Feed hold

When Feed Hold button on the operator's panel is pressed during memory operation, the tool decelerates to a stop at a time.

- **Reset**

Automatic operation can be stopped and the system can be made to the reset state by using RESET key on the CRT/MDI or external reset signal. When reset operation is applied to the system during a tool moving status, the motion is slowed down then stops.

- **Optional block skip**

When the optional block skip switch on the machine operator's panel is turned on, blocks containing a slash (/) are ignored.

- **Cycle start for the two–path control**

For the two–path control, a cycle start switch is provided for each tool post. This allows the operator to activate a single tool posts to operate them at the same time in memory operation or MDI operation. In general, select the tool post to be operated with the tool post selection switch on the machine operator's panel and then press the cycle start button to activate the selected tool post. (The procedure may vary with the machine tool builder. Refer to the appropriate manual issued by the machine tool builder.)

Calling a subprogram stored in an external input/output device

A file (subprogram) in an external input/output device such as a Floppy Cassette can be called and executed during memory operation. For details, see Section **4.5**.

4.2 MDI OPERATION

In the **MDI** mode, a program consisting of up to 10 lines can be created in the same format as normal programs and executed from the MDI panel. MDI operation is used for simple test operations.
The following procedure is given as an example. For actual operation, refer to the manual supplied by the machine tool builder.

Procedure for MDI Operation

1 Press the **MDI** mode selection switch.
For the two–path control, select the tool post for which a program is to be created with the tool post selection switch. Create a separate program for each tool post.

2 Press the PROG function key on the CRT/MDI (or LCD/MDI) panel to select the program screen. The following screen appears:

```
PROGRAM ( MDI )                          0010  00002

 O0000;



 G00  G90  G94  G40  G80  G50  G54  G69
 G17  G22  G21  G49  G98  G67  G64  G15
        B    H M
   T         D
   F      S

 >_
  MDI   ****   ***   ***          20 : 40 : 05
[ PRGRM ][  MDI  ][ CURRNT ][ NEXT ][ (OPRT) ]
```

Program number O0000 is entered automatically.

3 Prepare a program to be executed by an operation similar to normal program editing. M99 specified in the last block can return control to the beginning of the program after operation ends. Word insertion, modification, deletion, word search, address search, and program search are available for programs created in the MDI mode. For program editing, see Chapter III–9.

4 To entirely erase a program created in MDI mode,use one of the following methods:

a. Enter address O, then press the DELETE key on the MDI panel.

b. Alternatively, press the RESET key. In this case, set bit 7 of parameter 3203 to 1 in advance.

5 To execute a program, set the cursor on the head of the program. (Start from an intermediate point is possible.) Push Cycle Start button on the operator's panel. By this action, the prepared program will start. (For the two–path control, select the tool post to be operated with the tool post selection switch on the machine operator's panel beforehand.)
When the program end (M02, M30) or ER(%) is executed, the prepared program will be automatically erased and the operation will end.
By command of M99, control returns to the head of the prepared program.

```
PROGRAM ( MDI )                                   O0001 N00003
O0000  G00  X100.0  Z200. ;
 M03 ;
 G01  Z120.0  F500 ;
 M93  P9010 ;
 G00  Z0.0 ;
 %
 G00   G90   G94   G40   G80   G50   G54   G69
 G17   G22   G21   G49   G98   G67   G64   G15
       B    H M
   T        D
   F      S
 >_
 MDI   ****   ***   ***            12 : 42 : 39
( PRGRM )( MDI )( CURRNT )( NEXT )( (OPRT) )
```

6 To stop or terminate MDI operation in midway through, follow the steps below.

a. Stopping MDI operation
Press the feed hold switch on the machine operator's panel. The feed hold LED goes on and the cycle start LED goes off. The machine responds as follows:
(i) When the machine was moving, feed operation decelerates and stops.
(ii)When dwell was being performed, dwell is stopped.
(iii) When M, S, or T was being executed, the operation is stopped after M, S, or T is finished.
When the cycle start switch on the machine operator's panel is pressed, machine operation restarts.

b. Terminating MDI operation
Press the RESET key on the CRT/MDI (or LCD/MDI) panel.
Automatic operation is terminated and the reset state is entered. When a reset is applied during movement, movement decelerates then stops.

Explanation

The previous explanation of how to execute and stop memory operation also applies to MDI operation, except that in MDI operation, M30 does not return control to the beginning of the program (M99 performs this function).

• Erasing the program

Programs prepared in the **MDI** mode will be erased in the following cases:

- In MDI operation, if M02, M30 or ER(%) is executed.
  (If bit 6 (MER) of parameter No. 3203 is set to 1, however, the program is erased when execution of the last block of the program is completed by single–block operation.)
- In **MEMORY** mode, if memory operation is performed.
- In **EDIT** mode, if any editing is performed.
- Background editing is performed.
- Upon reset when bit 7 (MCL) of parameter No. 3203 is set to 1

• Restart

After the editing operation during the stop of MDI operation was done, operation starts from the current cursor position.

• Editing a program during MDI operation

A program can be edited during MDI operation. The editing of a program, however, is disabled until the CNC is reset, when bit 5 (MIE) of parameter No. 3203 is set accordingly.

Limitation

• Program registration

Programs created in MDI mode cannot be registered.

• Number of lines in a program

A program can have as many lines as can fit on one page of the CRT screen.
A program consisting of up to six lines can be created. When parameter MDL (No. 3107 #7) is set to 0 to specify a mode that suppresses the display of continuous–state information, a program of up to 10 lines can be created.
If the created program exceeds the specified number of lines, % (ER) is deleted (prevents insertion and modification).

• Subprogram nesting

Calls to subprograms (M98) can be specified in a program created in the MDI mode. This means that a program registered in memory can be called and executed during MDI operation. In addition to the main program executed by automatic operation, up to two levels of subprogram nesting are allowed (when the custom macro option is provided, up to four levels are allowed).

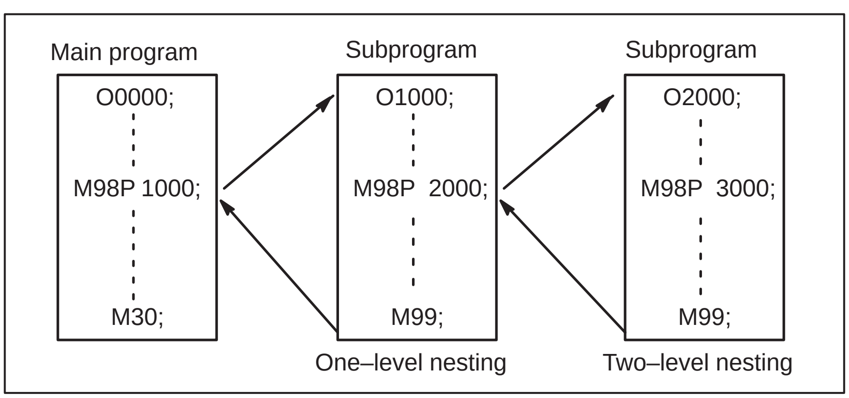


Fig. 4.2(a)Nesting Level of Subprograms Called from the MDI Program

- **Macro call**

When the custom macro option is provided, macro programs can also be created, called, and executed in the **MDI** mode. However, macro call commands cannot be executed when the mode is changed to **MDI** mode after memory operation is stopped during execution of a subprogram.

- **Memory area**

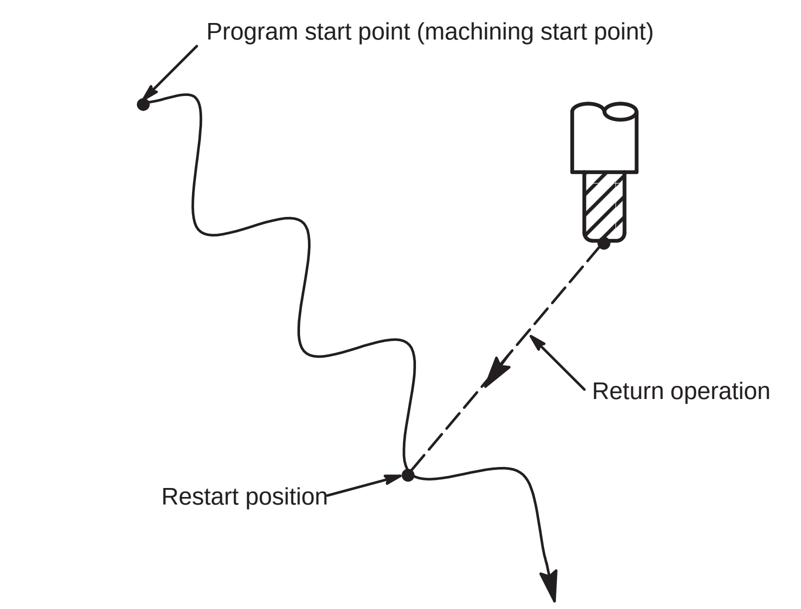
When a program is created in the **MDI** mode, an empty area in program memory is used. If program memory is full, no programs can be created in the **MDI** mode.

4.3 PROGRAM RESTART

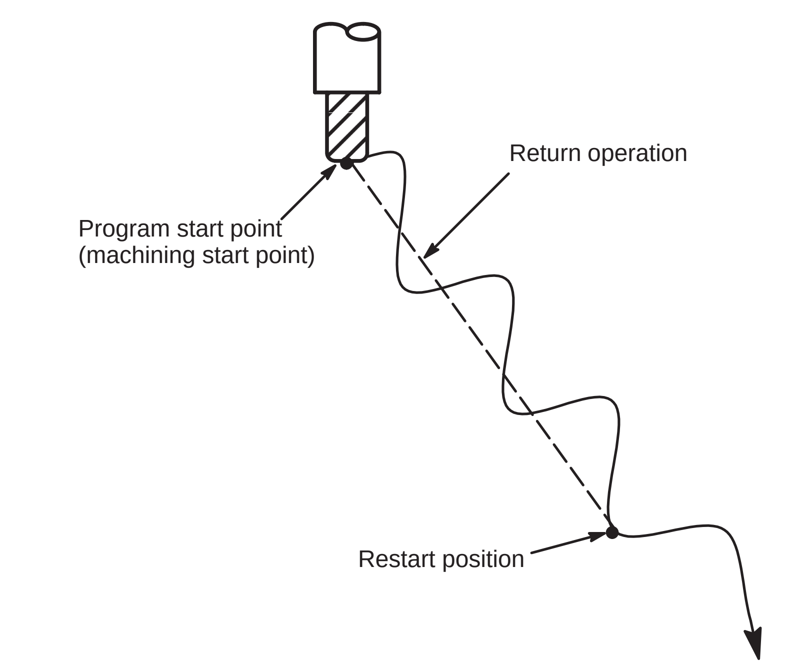
This function specifies Sequence No. or Block No. of a block to be restarted when a tool is broken down or when it is desired to restart machining operation after a day off, and restarts the machining operation from that block. It can also be used as a high–speed program check function.

There are two restart methods: the P–type method and Q–type method.

P TYPE	Operation can be restarted anywhere. This restart method is used when operation is stopped because of a broken tool.



Q TYPE	Before operation can be restarted, the machine must be moved to the programmed start point (machining start point)



Procedure for Program restart by Specifying a sequence number

Procedure 1

[P TYPE]

1 Retract the tool and replace it with a new one. When necessary, change the offset. (Go to step 2.)

[Q TYPE]

1 When power is turned ON or emergency stop is released, perform all necessary operations at that time, including the reference position return.

2 Move the machine manually to the program starting point (machining start point), and keep the modal data and coordinate system in the same conditions as at the machining start.

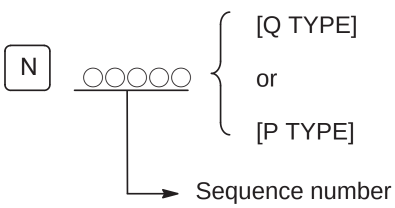
3 If necessary, modify the offset amount.

Procedure 2

[COMMON TO P TYPE / Q TYPE]

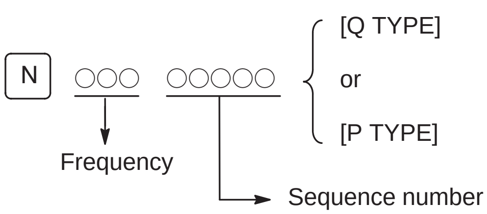
1 Turn the program restart switch on the machine operator's panel ON.

2 Press PROG on the CRT/MDI panel to display the desired program.



3 Find the program head.

4 Enter the sequence number of the block to be restarted, then press the **[P TYPE]** or **[Q TYPE]** soft key.



If the same sequence number appears more than once, the location of the target block must be specified. Specify a frequency and a sequence number.

5 The sequence number is searched for, and the program restart screen appears on the CRT display.

```
PROGRAM RESTART                              O0002  N00100

   DESTINATION              M1 2
     X 57. 096                1 2
     Z 56. 943                1 2
                              1 2
                              1 2
                              1 ********
   DISTANCE TO GO             ******** ********
   1 X 1. 459
   2 Z 7. 320               T ******** ********
                            S *****

                                        S    0   T0000
  MEM  ****  ***  ***            10 : 10 : 40
[ RSTR ][        ][ FL.SDL ][        ][ (OPRT) ]
```

DESTINATION shows the position at which machining is to restart. DISTANCE TO GO shows the distance from the current tool position to the position where machining is to restart. A number to the left of each axis name indicates the order of axes (determined by parameter setting) along which the tool moves to the restart position.
The coordinates and amount of travel for restarting the program can be displayed for up to five axes. If your system supports six or more axes, pressing the **[RSTR]** soft key again displays the data for the sixth and subsequent axes. (The program restart screen displays only the data for CNC–controlled axes.)
M: Fourteen most recently specified M codes
T: Two most recently specified T codes
S: Most recently specified S code
Codes are displayed in the order in which they are specified. All codes are cleared by a program restart command or cycle start in the reset state.

6 Turn the program re–start switch OFF. At this time, the figure at the left side of axis name DISTANCE TO GO blinks.

7 Check the screen for the M, S, and T codes to be executed. If they are found, enter the **MDI** mode, then execute the M, S and T functions. After execution, restore the previous mode.
These codes are not displayed on the program restart screen.

8 Check that the distance indicated under DISTANCE TO GO is correct. Also check whether there is the possibility that the tool might hit a workpiece or other objects when it moves to the machining restart position. If such a possibility exists, move the tool manually to a position from which the tool can move to the machining restart position without encountering any obstacles.

9 Press the cycle start button. The tool moves to the machining restart position at the dry run feedrate sequentially along axes in the order specified by parameter settings (No. 7310). Machining is then restarted.

Procedure for Program Restart by Specifying a Block Number

Procedure 1

[P TYPE]

1 Retract the tool and replace it with a new one. When necessary, change the offset. (Go to step 2.)

[Q TYPE]

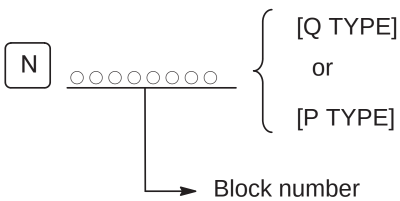
1 When power is turned ON or emergency stop is released, perform all necessary operations at that time, including the reference position return.

2 Move the machine manually to the program starting point (machining start point), and keep the modal data and coordinate system in the same conditions as at the machining start.

3 If necessary, modify the offset amount.

Procedure 2

[COMMON TO P TYPE / Q TYPE]



1 Turn the program restart switch on the machine operator's panel ON.

2 Press PROG on the CRT/MDI panel to display the desired program.

3 Find the program head. Press function RESET key.

4 Enter the number of the block to be restarted then press the **[P TYPE]** or **[Q TYPE]** soft key. The block number cannot exceed eight digits.

5 The block number is searched for, and the program restart screen appears on the CRT display.

```
PROGRAM RESTART                          O0002  N01000

  DESTINATION              M1 2
    X 57. 096                1 2
    Z 56. 943                1 2
                             1 2
                             1 2
                             1 ********
  DISTANCE TO GO             ******** ********
    X 1. 459
    Z 7. 320               T ******** ********
                           S *****

                                         S     0   T0000
 MEM  ****   ***   ***           10 : 10 : 40
[ RSTR ][        ][ FL.SDL ][        ][ (OPRT) ]
```

DESTINATION shows the position at which machining is to restart. DISTANCE TO GO shows the distance from the current tool position to the position where machining is to restart. A number to the left of each axis name indicates the order of axes (determined by parameter setting) along which the tool moves to the restart position.

The coordinates and amount of travel for restarting the program can

be displayed for up to five axes. If your system supports six or more axes, pressing the **[RSTR]** soft key again displays the data for the sixth and subsequent axes. (The program restart screen displays only the data for CNC–controlled axes.)
M: Fourteen most recently specified M codes
T: Two most recently specified T codes
S: Most recently specified S code
B: Most recently specified B code
Codes are displayed in the order in which they are specified. All codes are cleared by a program restart command or cycle start in the reset state.

6 Turn the program re–start switch OFF. At this time, the figure at the left side of axis name DISTANCE TO GO blinks.

7 Check the screen for the M, S, T, and B codes to be executed. If they are found, enter the **MDI** mode, then execute the M, S, T, and B functions. After execution, restore the previous mode.
These codes are not displayed on the program restart screen.

8 Check that the distance indicated under DISTANCE TO GO is correct. Also check whether there is the possibility that the tool might hit a workpiece or other objects when it moves to the machining restart position. If such a possibility exists, move the tool manually to a position from which the tool can move to the machining restart position without encountering any obstacles.

9 Press the cycle start button. The tool moves to the machining restart position at the dry run feedrate sequentially along axes in the order specified by parameter settings (No. 7310). Machining is then restarted.

Explanations

• Block number

When the CNC is stopped, the number of executed blocks is displayed on the program screen or program restart screen. The operator can specify the number of the block from which the program is to be restarted, by referencing the number displayed on the CRT. The displayed number indicates the number of the block that was executed most recently. For example, to restart the program from the block at which execution stopped, specify the displayed number, plus one.

The number of blocks is counted from the start of machining, assuming one NC line of a CNC program to be one block.

< Example 1 >

CNC Program	Number of blocks
O 0001 ;	1
G90 G92 X0 Y0 Z0 ;	2
G01 X100. F100 ;	3
G03 X01 –50. F50 ;	4
M30 ;	5

< Example 2 >

CNC Program	Number of blocks
O 0001 ;	1
G90 G92 X0 Y0 Z0 ;	2
G90 G00 Z100. ;	3
G81 X100. Y0. Z–120. R–80. F50. ;	4
#1 = #1 + 1 ;	4
#2 = #2 + 1 ;	4
#3 = #3 + 1 ;	4
G00 X0 Z0 ;	5
M30 ;	6

Macro statements are not counted as blocks.

- **Storing / clearing the block number**

The block number is held in memory while no power is supplied. The number can be cleared by cycle start in the reset state.

- **Block number when a program is halted or stopped**

The program screen usually displays the number of the block currently being executed. When the execution of a block is completed, the CNC is reset, or the program is executed in single–block stop mode, the program screen displays the number of the program that was executed most recently.

When a CNC program is halted or stopped by feed hold, reset, or single–block stop, the following block numbers are displayed:

Feed hold : Block being executed
Reset : Block executed most recently
Single–block stop : Block executed most recently

For example, when the CNC is reset during the execution of block 10, the displayed block number changes from 10 to 9.

- **MDI intervention**

When MDI intervention is performed while the program is stopped by single–block stop, the CNC commands used for intervention are not counted as a block.

- **Block number exceeding eight digits**

When the block number displayed on the program screen exceeds eight digits, the block number is reset to 0 and counting continues.

Limitation

- **P–type restart**

Under any of the following conditions, P–type restart cannot be performed:

· When automatic operation has not been performed since the power was turned on

· When automatic operation has not been performed since an emergency stop was released

· When automatic operation has not been performed since the coordinate system was changed or shifted (change in an external offset from the workpiece reference point)

- **Restart block**

The block to be restarted need not be the block which was interrupted; operation can restart with any block. When P–type restart is performed, the restart block must use the same coordinate system as when operation was interrupted.

- **Single block**

When single block operation is ON during movement to the restart position, operation stops every time the tool completes movement along an axis. When operation is stopped in the single block mode, MDI intervention cannot be performed.

- **Manual intervention**

During movement to the restart position, manual intervention can be used to perform a return operation for an axis if it has not yet been performed for the axis. A return operation cannot be done further on axes for which a return has already been completed.

- **Reset**

Never reset during the time from the start of a search at restart until machining is restarted. Otherwise, restart must be performed again from the first step.

- **Manual absolute**

Regardless of whether machining has started or not, manual operation must be performed when the manual absolute mode is on.

- **Reference position return**

If no absolute–position detector (absolute pulse coder) is provided, be sure to perform reference position return after turning on the power and before performing restart.

Alarm

Alarm No.	Contents
071	The specified block number for restarting the program is not found.
094	After interruption, a coordinate system was set, then P–type restart was specified.
095	After interruption, the coordinate system shift was changed, then P–type restart was specified.
096	After interruption, the coordinate system was changed, then P–type restart was specified.
097	When automatic operation has not been performed since the power was turned on, emergency stop was released, or P/S alarm (Nos. 094 to 097) was reset, P–type restart was specified.
098	After the power was turned on, restart operation was performed without reference position return, but a G28 command was found in the program.
099	A move command was specified from the MDI panel during a restart operation.
5020	An erroneous parameter was specififed for restarting a program.

Notes

As a rule, the tool cannot be returned to a correct position under the following conditions.

- Special care must be taken in the following cases since none of them cause an alarm:
- Manual operation is performed when the manual absolute mode is OFF.
- Manual operation is performed when the machine is locked.
- When the mirror image is used.
- When manual operation is performed in the course of axis movement for returning operation.
- When the program restart is commanded for a block between the block for skip cutting and subsequent absolute command block.
- When program restart specified for an intermediate block for a multiple repetitive canned cycle

4.4 SCHEDULING FUNCTION

The schedule function allows the operator to select files (programs) registered on a floppy–disk in an external input/output device (Handy File, Floppy Cassette, or FA Card) and specify the execution order and number of repetitions (scheduling) for performing automatic operation.

It is also possible to select only one file from the files in the external input/output device and execute it during automatic operation.

```
FILE DIRECTORY
FILE NO.  FILE NAME

0001  O0010
0002  O0020
0003  O0030
0004  O0040
```

List of files in an external input/output device

Set file number and number of repetitions.

```
ORDER FILE NO   REPETITION

01    0002      2
02    0003      1
03    0004      3
04    0001      2
```

Scheduling screen

Executing automatic operation

Procedure for Scheduling Function

·Procedure for executing one file

1 Press the **MEMORY** switch on the machine operator's panel, then press the PROG function key on the MDI panel.

2 Press the rightmost soft key (continuous menu key), then press the **[FL. SDL]** soft key. A list of files registered in the Floppy Cassette is displayed on screen No. 1. To display more files that are not displayed on this screen, press the page key on the MDI panel. Files registered in the Floppy Cassette can also be displayed successively.

```
FILE DIRECTORY                                O0001  N00000
 CURRENT SELECTED : SCHEDULE
  NO. FILE NAME  (METER)  VOL
  0000  SCHEDULE
  0001  PARAMETER   58.5
  0002  ALL PROGRAM  11.0
  0003  O0001       1.9
  0004  O0002       1.9
  0005  O0010       1.9
  0006  O0020       1.9
  0007  O0040       1.9
  0008  O0050       1.9

 MEM  ****  ***  ***           19 : 14 : 47
( PRGRM )(        )(  DIR  )( SCHDUL )( (OPRT) )
```

Screen No.1

3 Press the **[(OPRT)]** and **[SELECT]** soft keys to display "SELECT FILE NO." (on screen No. 2). Enter a file number, then press the **[F SET]** and **[EXEC]** soft keys. The file for the entered file number is selected, and the file name is indicated after "CURRENT SELECTED:".

```
FILE DIRECTORY                                O0001  N00000
 CURRENT SELECTED:O0040
  NO. FILE NAME  (METER)  VOL
  0000  SCHEDULE
  0001  PARAMETER   58.5
  0002  ALL PROGRAM  11.0
  0003  O0001       1.9
  0004  O0002       1.9
  0005  O0010       1.9
  0006  O0020       1.9
  0007  O0040       1.9
  0008  O0050       1.9
  SELECT FILE NO.=7
 >_
 MEM  ****  ***  ***           19 : 17 : 10
( F SET )(        )(        )(        )(  EXEC  )
```

Screen No.2

4 Press the **REMOTE** switch on the machine operator's panel to enter the **RMT** mode, then press the cycle start switch. The selected file is executed. For details on the **REMOTE** switch, refer to the manual supplied by the machine tool builder. The selected file number is indicated at the upper right corner of the screen as an F number (instead of an O number).

```
FILE DIRECTORY                                F0007  N00000
 CURRENT SELECTED:O0040












RMT  ****  ***  ***              13 : 27 : 54
[ PRGRM ][        ][ DIR ][ SCHDUL ][ (OPRT) ]
```

Screen No.3

- **Procedure for executing the scheduling function**

1 Display the list of files registered in the Floppy Cassette. The display procedure is the same as in steps 1 and 2 for executing one file.

2 On screen No. 2, press the **[(OPRT)]** and **[SELECT]** soft keys to display ”SELECT FILE NO.”

3 Enter file number 0, and press the **[F SET]**, and **[EXEC]** soft keys. ”SCHEDULE” is indicated after ”CURRENT SELECTED:”.

4 Press the left most soft key (return menu key) and the **[SCHDUL]** soft key. Screen No. 4 appears.

```
FILE DIRECTORY                                F0000  N02000
   ORDER FILE NO.  REQ.REP CUR.REP
     01
     02
     03
     04
     05
     06
     07
     08
     09
     10

>_
MEM  ****  ***  ***              22 : 07 : 00
[ PRGRM ][        ][ DIR ][ SCHDUL ][ (OPRT) ]
```

Screen No.4

Move the cursor and enter the file numbers and number of repetitions in the order in which to execute the files. At this time, the current number of repetitions ”CUR.REP” is 0.

5 Press the **REMOTE** switch on the machine operator’s panel to enter the **RMT** mode, then press the start switch. The files are executed in the specified order. When a file is being executed, the cursor is positioned at the number of that file.
The current number of repetitions CUR.REP is increased when M02

or M30 is executed in the program being run.

```
FILE DIRECTORY                              O0000  N02000

ORDER  FILE NO.  REQ.REP CUR.REP
   01       000      5        5
   02       7                23       23
   03       0004        9999    156
   04       0005     LOOP       0
   05
   06
   07
   08
   09
   10

 RMT  ****  ***  ***         10 : 10 : 40

[ PRGRM ][        ][ DIR    ][SCHDUL ][(OPRT) ]
```

Screen No.5

Explanations

- **Specifying no file number**

  If no file number is specified on screen No. 4 (the file number field is left blank), program execution is stopped at that point. To leave the file number field blank, press numeric key 0 then INPUT.

- **Endless repetition**

  If a negative value is set as the number of repetitions, <**LOOP**> is displayed, and the file is repeated indefinitely.

- **Clear**

  When the **[(OPRT)]**, **[CLEAR]**, and **[EXEC]** soft keys are pressed on screen No. 4, all data is cleared. However, these keys do not function while a file is being executed.

- **Return to the program screen**

  When the **[PRGRM]** soft key is pressed on screen No. 1, 2, 3, 4, or 5, the program screen is displayed.

Limitation

- **Number of repetitions**

  Up to 9999 can be specified as the number of repetitions. If 0 is set for a file, the file becomes invalid and is not executed.

- **Number of files registered**

  By pressing the page key on screen No. 4, up to 20 files can be registered.

- **M code**

  When M codes other than M02 and M30 are executed in a program, the current number of repetitions is not increased.

- **Displaying the floppy disk directory during file execution**

  During the execution of file, the floppy directory display of background editing cannot be referenced.

- **Restarting automatic operation**

  To resume automatic operation after it is suspended for scheduled operation, press the reset button.

- **Scheduling function for the two–path control**

  The scheduling function can be used only for a single tool post.

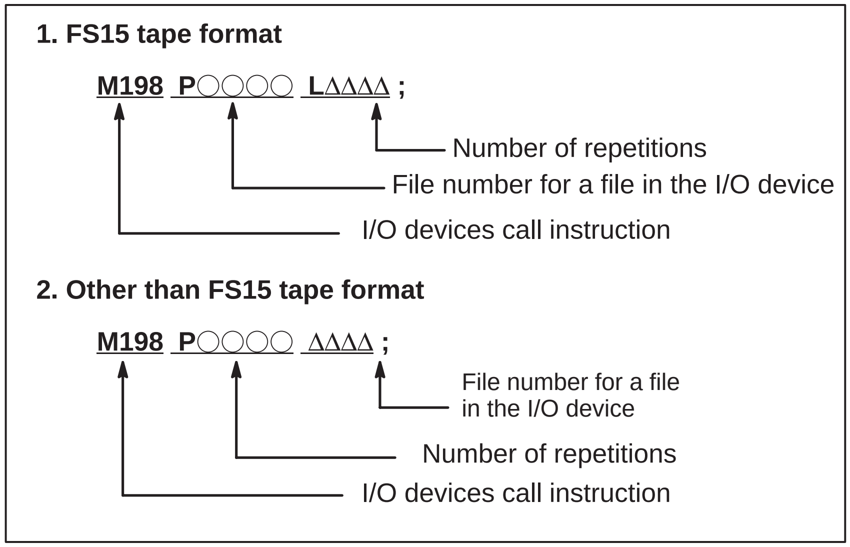
Alarm

Alarm No.	Description
086	An attempt was made to execute a file that was not registered in the floppy disk.
210	M198 and M099 were executed during scheduled operation, or M198 was executed during DNC operation.

4.5 SUBPROGRAM CALL FUNCTION

The subprogram call function is provided to call and execute subprogram files stored in an external input/output device(Handy File, FLOPPY CASSETTE, FA Card)during memory operation.
When the following block in a program in CNC memory is executed, a subprogram file in the external input/output device is called:

Format



Explanation

The subprogram call function is enabled when parameter No.0102 for the input/output device is set to 3. When the custom macro option is provided, either format 1 or 2 can be used. A different M code can be used for a subprogram call depending on the setting of parameter No.6030. In this case, M198 is executed as a normal M code. The file number is specified at address P. If the SBP bit (bit 2) of parameter No.3404 is set to 1, a program number can be specified. When a file number is specified at address P, Fxxxx is indicated instead of Oxxxx.

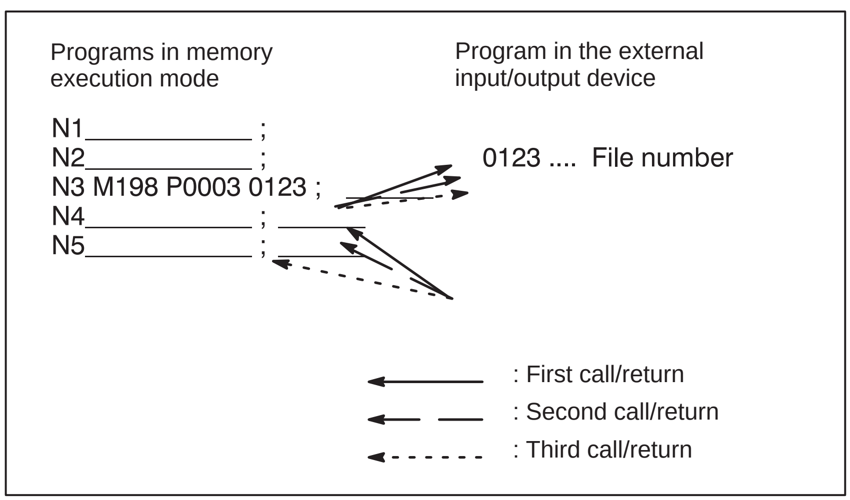


Fig. 4.5 (a) Program Flow When M198 is Specified

Restrictions

For the two–path control, subprograms in a floppy cassette cannot be called for the two tool posts at the same time.

Notes

1. When M198 in the program of the file saved in a floppy cassette is executed, a P/S alarm (No.210) is given. When a program in the memory of CNC is called and M198 is executed during execution of a program of the file saved in a floppy cassette, M198 is changed to an ordinary M–code.
2. When MDI is intervened and M198 is executed after M198 is commanded in the memory mode, M198 is changed to an ordinary M–code. When the reset operation is done in the MDI mode after M198 is commanded in the MEMORY mode, it does not influence on the memory operation and the operation is continued by restarting it in the MEMORY mode.

4.6 MANUAL HANDLE INTERRUPTION

The movement by manual handle operation can be done by overlapping it with the movement by automatic operation in the automatic operation mode.

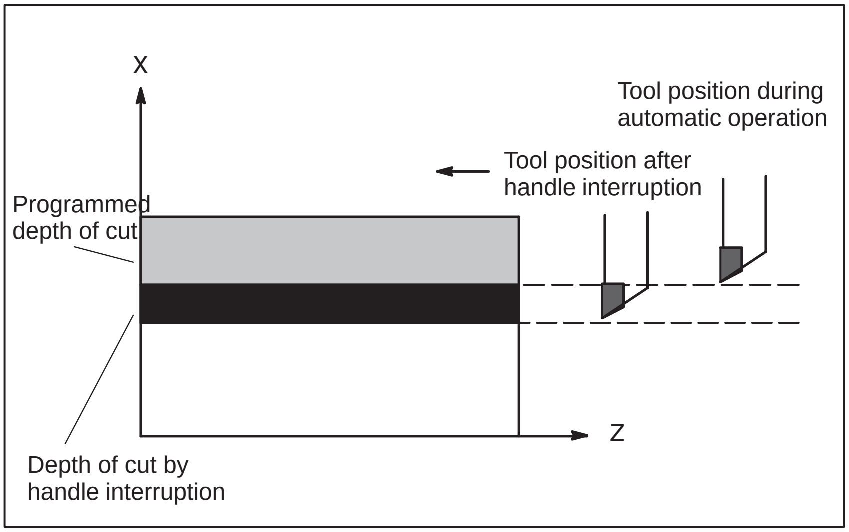


Fig. 4.6 (a) Manual Handle Interruption

· Handle interruption axis selection signals
For the handle interruption axis selection signals, refer to the manual supplied by the machine tool builder.

During automatic operation, handle interruption is enabled for an axis if the handle interruption axis selection signal for that axis is on. Handle interruption is performed by turning the handle of the manual pulse generator.

Notes

1. The travel distance by handle interruption is determined according to the amount by which the manual pulse generator is turned and the handle feed magnification (x1, x10, xM, xN).
   Since this movement is not accelerated or decelerated, it is very dangerous to use a large magnification value for handle interruption.
   The move amount per scale at x1 magnification is 0.001 mm (metric output) or 0.0001 inch (inch output).
2. Handle interruption is disabled when the machine is locked during automatic operation.

Explanations

• Relation with other functions

The following table indicates the relation between other functions and the movement by handle interrupt.

Display	Relation
Machine lock	Machine lock is effective. The tool does not move even when this signal turns on.
Interlock	Interlock is effective. The tool does not move even when this signal turns on.
Mirror image	Mirror image is not effective. Interrupt functions on the plus direction by plus direction command, even if this signal turns on.

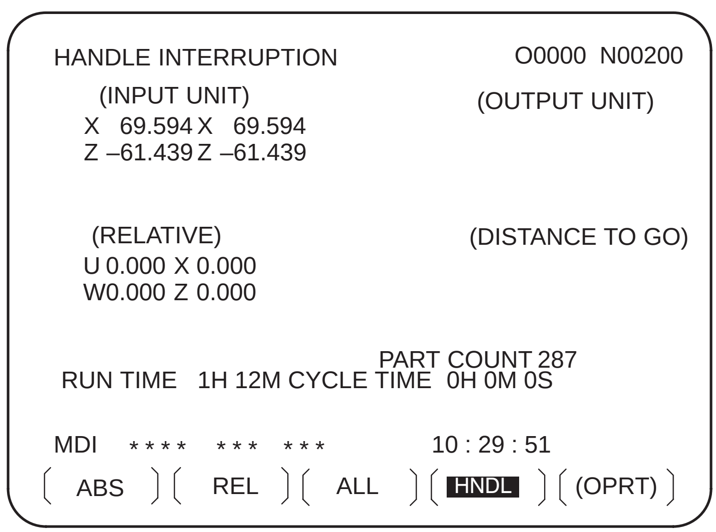
• Position display

The following table shows the relation between various position display data and the movement by handle interrupt.

Display	Relation
Absolute coordinate value	Handle interruption does not change absolute coordinates.
Relative coordinate value	Handle interruption does not change relative coordinates.
Machine coordinate value	Machine coordinates are changed by the travel distance specified by handle interruption.

• Travel distance display

Press the function key POS , then press the chapter selection soft key **[HNDL]**.
The move amount by the handle interrupt is displayed. The following 4 kinds of data are displayed concurrently.



(a) INPUT UNIT :
Handle interrupt move amount in input unit system
Indicates the travel distance specified by handle interruption according to the least input increment.

(b) OUTPUT UNIT :
Handle interrupt move amount in output unit system
Indicates the travel distance specified by handlei nterruption according to the least command increment.

(c) RELATIVE :
Position in relative coordinate system
These values have no effect on the travel distance specified by handle interruption.

(d) DISTANCE TO GO :
The remaining travel distance in the current block has no effect on the travel distance specified by handle interruption.

The handle interrupt move amount is cleared when the manual reference position return ends every axis.

4.7 MIRROR IMAGE

During automatic operation, the mirror image function can be used for movement along an axis. To use this function, set the mirror image switch to ON on the machine operator's panel, or set the mirror image setting to ON from the CRT/MDI (or LCD/MDI).

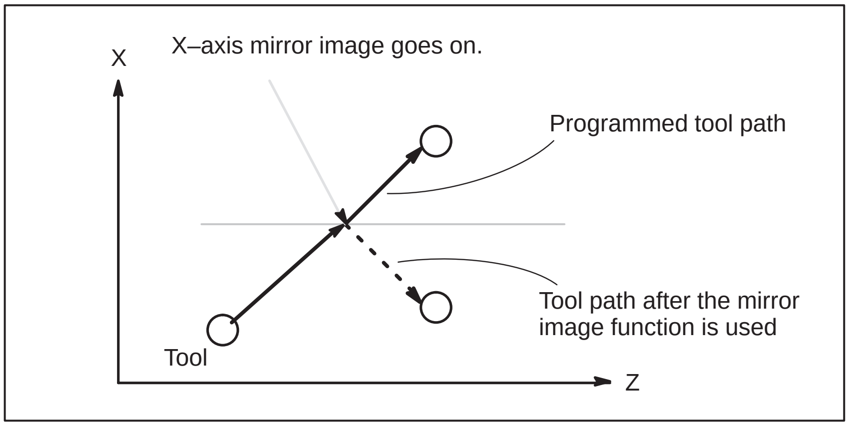


Fig. 4.7 (a) Mirror Image

Procedure

The following procedure is given as an example. For actual operation, refer to the manual supplied by the machine tool builder.

1 Press the single block switch to stop automatic operation. When the mirror image function is used from the beginning of operation, this step is omitted.

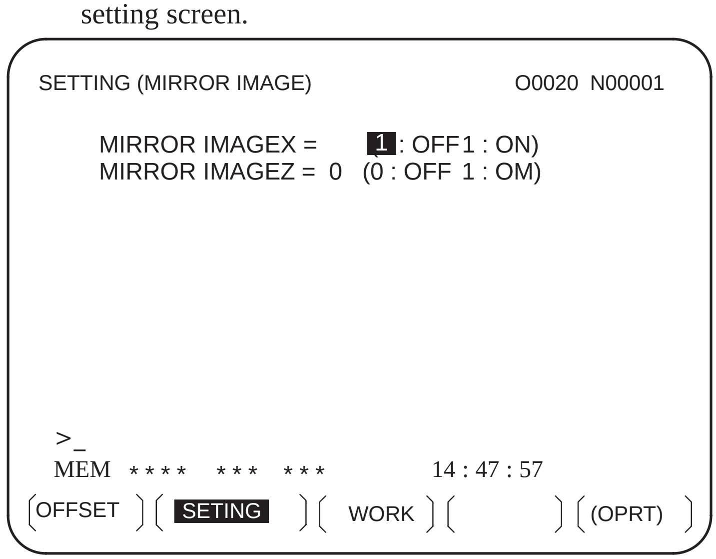
2 Press the mirror image switch for the target axis on the machine operator's panel.
Alternatively, turn on the mirror image setting by following the steps below:

2–1 Set the **MDI** mode.

2–2 Press the 

function key.

2–3 Press the **[SETING]** soft key for chapter selection to display the setting screen.



2–4 Move the cursor to the mirror image setting position, then set the target axis to 1.

3 Enter an automatic operation mode (memory mode or MDI mode), then press the cycle start button to start automatic operation.

Explanations

- The mirror image function can also be turned on and off by setting bit 0 (MIRx) of parameter (No.0012) to 1 or 0.
- For the mirror image switches, refer to the manual supplied by the machine tool builder.

Restrictions

The direction of movement during manual operation, the direction of movement from an intermediate point to the reference position during automatic reference position return (G28).

4.8 MANUAL INTERVENTION AND RETURN

In cases such as when tool movement along an axis is stopped by feed hold during automatic operation so that manual intervention can be used to replace the tool: When automatic operation is restarted, this function returns the tool to the position where manual intervention was started.
To use the conventional program restart function and tool withdrawal and return function, the switches on the operator's panel must be used in conjunction with the MDI keys. This function does not require such operations.

Explanations

- **Manual absolute on/off**

In manual absolute off mode, the tool does not return to the stop position, but instead operates according to the manual absolute on/off function.

- **Override**

For the return operation, the dry run feedrate is used, and the jog feedrate override function is enabled.

- **Return operation**

Return operation is performed according to positioning based on nonlinear interpolation.

- **Single block**

If the single block stop switch is on during return operation, the tool stops at the stop position and restarts movement when the cycle start switch is pressed.

- **Cancellation**

If a reset occurs or an alarm is issued during manual intervention or the return operation, this function is cancelled.

- **MDI mode**

This function can be used in the MDI mode as well.

Restrictions

- **Enabling and disabling manual intervention and return**

This function is enabled only when the automatic operation hold LED is on. When there is no travel distance remaining, this function has no effect even if a feed hold stop is performed with the automatic operation hold signal *SP (bit 5 of G008).

- **Offset**

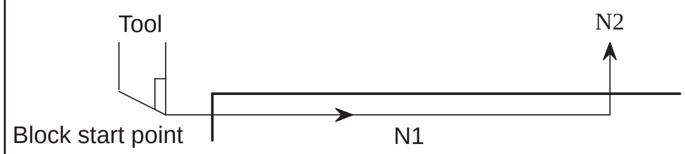
When the tool is replaced using manual intervention for a reason such as damage, the tool movement cannot be restarted by a changed offset in the middle of the interrupted block.

- **Machine lock, mirror image, and scaling**

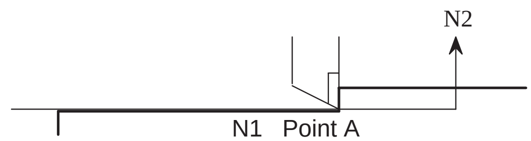
When performing manual intervention, never use the machine lock, mirror image, or scaling functions.

Example

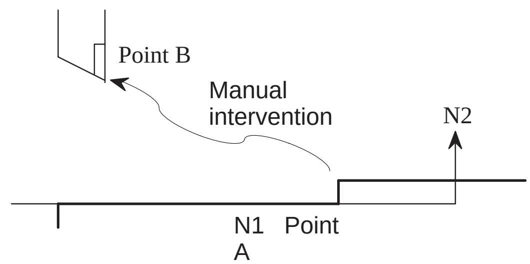
1. The N1 block cuts a workpiece



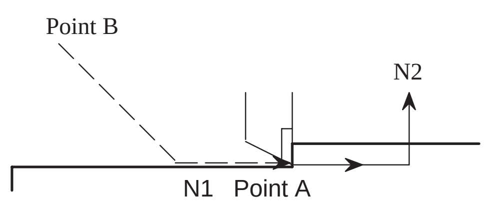
2. The tool is stopped by pressing the feed hold switch in the middle of the N1 block (point A).



3. After retracting the tool manually to point B, tool movement is restarted.



4. After automatic return to point A at the dry run feedrate, the remaining move command of the N1 block is executed.



Notes

When performing manual intervention, pay particular attention of machining and the shape of the workpiece so that the machine and tool are not damaged.

4.9 DNC OPERATION

By activating automatic operation during the DNC operation mode (RMT), it is possible to perform machining (DNC operation) while a program is being read in via reader/puncher interface, or remote buffer.
If the floppy cassette directory display option is available, it is possible to select files (programs) saved in an external input/output unit of a floppy format (Handy File, Floppy Cassettes, or FA card) and specify (schedule) the sequence and frequency of execution for automatic operation.
To use the DNC operation function, it is necessary to set the parameters related to the reader/punch interface, and remote buffer in advance.

DNC OPERATION

Procedure

1 Search for the program (file) to be executed.

2 Press the REMOTE switch on the machine operator's panel to set RMT mode, then press the cycle start switch. The selected file is executed. For details of the use of the REMOTE switch, refer to the relevant manual supplied by the machine tool builder.

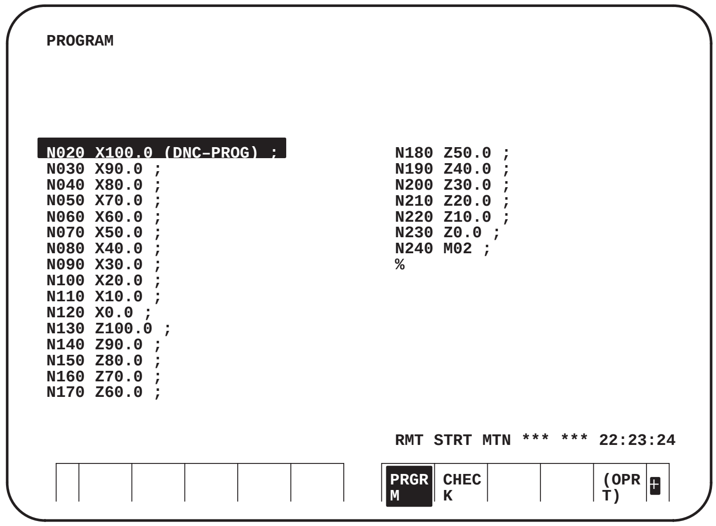
- **Program check screen (9” CRT)**

```
PROGRAM CHECK                        O0001 N00020

N020 X100.0 Z100.0 (DNC–PROG) ;
N030 X200.0 Z200.0 ;
N050 X400.0 Z400.0 ;

 (RELATIVE)  (DIST TO GO) G00  G17  G90
 X   100.000 X     0.000  G22  G94  G21
 Y   100.000 Y     0.000  G41  G49  G80
 Z     0.000 Z     0.000  G98  G50  G67
 A     0.000 A     0.000       B
 C     0.000 C     0.000  H    M
HD.T        NX.T          D    M
     F                 S       M
 ACT.F            SACT         REPEAT
 RMT STRT MTN *** ***     21:20:05
[  ABS  ][  REL  ][        ][      ][ (OPRT) ]
```

- **Program screen (9” CRT)**

```
PROGRAM                              O0001 N00020

N020 X100.0 Z100.0 (DNC–PROG) ;
N030 X200.0 Z200.0 ;
N040 X300.0 Z300.0 ;
N050 X400.0 Z400.0 ;
N060 X500.0 Z500.0 ;
N070 X600.0 Z600.0 ;
N080 X700.0 Z400.0 ;
N090 X800.0 Z400.0 ;
N100 x900.0 z400.0 ;
N110 x1000.0 z1000.0 ;
N120 x800.0 z800.0 ;


 RMT STRT MTN *** ***     21:20:05
[ PRGRM ][ CHECK ][        ][      ][ (OPRT) ]
```

- **Program screen (9.5”/8.4”/14” CRT)**



During DNC operation, the program currently being executed is displayed on the program check screen and program screen.
The number of displayed program blocks depends on the program being executed.
Any comment enclosed between a control–out mark (() and control–in mark ()) within a block is also displayed.

Explanations

- During DNC operation, programs stored in memory can be called.
- During DNC operation, macro programs stored in memory can be called.

Limitations

- **Limit on number of characters**

In program display, no more than 256 characters can be displayed. Accordingly, character display may be truncated in the middle of a block.

- **M198 (command for calling a program from within an external input/output unit)**

In DNC operation, M198 cannot be executed. If M198 is executed, P/S alarm No. 210 is issued.

- **Custom macro**

In DNC operation, custom macros can be specified, but no repeat instruction and branch instruction can be programmed. If a repeat instruction or branch instruction is executed, P/S alarm No. 123 is issued.
When reserved words (such as IF, WHILE, COS, and NE) used with custom macros in DNC operation are displayed during program display, a blank is inserted between adjacent characters.

Example

		[During DNC operation]
#102=SIN[#100] ;	→	#102 = S I N[#100] ;
IF[#100NE0]GOTO5 ;	→	I F[#100NE0] G O T O 5 ;

- **M99**

When control is returned from a subprogram or macro program to the calling program during DNC operation, it becomes impossible to use a return command (M99P****) for which a sequence number is specified.

Alarm

Number	Message	Contents
086	DR SIGNAL OFF	When entering data in the memory by using Reader / Puncher interface, the ready signal (DR) of reader / puncher was turned off. Power supply of I/O unit is off or cable is not connected or a P.C.B. is defective.
123	CAN NOT USE MACRO COMMAND IN DNC	Macro control command is used during DNC operation. Modify the program.
210	CAN NOT COMAND M198/M199	Or M198 is executed in the DNC operation. Modify the program.

5 TEST OPERATION

The following functions are used to check before actual machining whether the machine operates as specified by the created program.

1. Machine Lock and Auxiliary Function Lock
2. Feedrate Override
3. Rapid Traverse Override
4. Dry Run
5. Single Block5

5.1 MACHINE LOCK AND AUXILIARY FUNCTION LOCK

To display the change in the position without moving the tool, use machine lock.
There are two types of machine lock, all–axis machine lock, which stops the movement along all axes, and specified–axis machine lock, which stops the movement along specified axes only. In addition, auxiliary function lock, which disables M, S, and T commands, is available for checking a program together with machine lock.

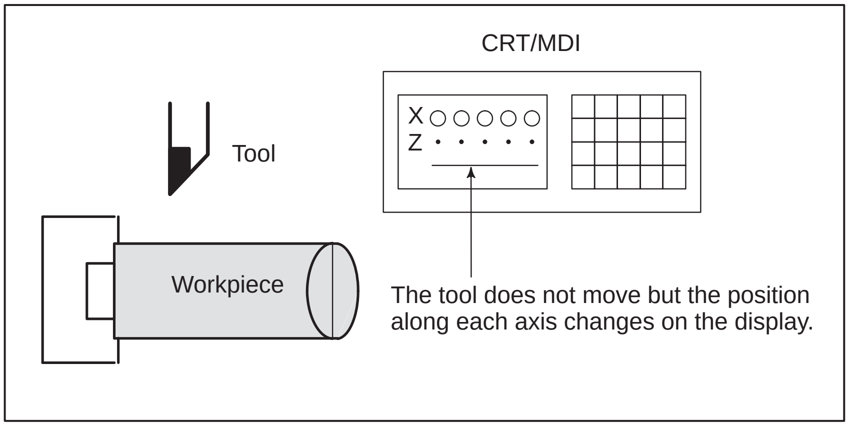


Fig. 5.1 Machine lock

Procedure for Machine Lock and Auxiliary Function Lock

- **Machine Lock**

Press the machine lock switch on the operator's panel. The tool does not move but the position along each axis changes on the display as if the tool were moving.
Some machines have a machine lock switch for each axis. On such machines, press the machine lock switches for the axes along which the tool is to be stopped. Refer to the appropriate manual provided by the machine tool builder for machine lock.
The positional relationship between the workpiece coordinates and machine coordinates may differ before and after automatic operation using machine lock. In such a case, specify the workpiece coordinate system by using a coordinate setting command or by performing manual reference position return.

- **Auxiliary Function Lock**

Press the auxiliary function lock switch on the operator's panel. M, S, and T codes are disabled and not executed. Refer to the appropriate manual provided by the machine tool builder for auxiliary function lock.

Restrictions

- **M, S, T command by only machine lock**

M, S, and T commands are executed in the machine lock state.

- **Reference position return under Machine Lock**

When a G27, G28, or G30 command is issued in the machine lock state, the command is accepted but the tool does not move to the reference position and the reference position return LED does not go on.

- **M codes not locked by auxiliary function lock**

M00, M01, M02, M30, M98, and M99 commands are executed even in the auxiliary function lock state.

5.2 FEEDRATE OVERRIDE

A programmed feedrate can be reduced or increased by a percentage (%) selected by the override dial. This feature is used to check a program. For example, when a feedrate of 100 mm/min is specified in the program, setting the override dial to 50% moves the tool at 50 mm/min.

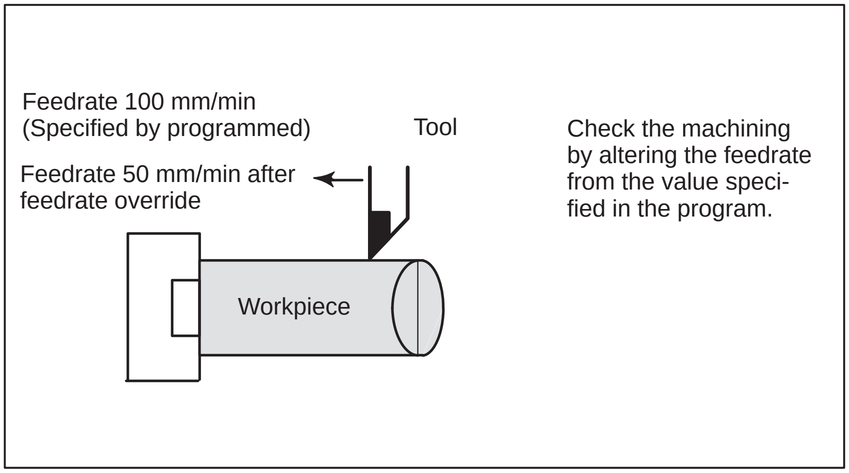
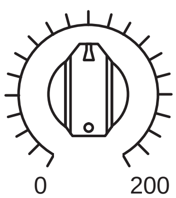


Fig. 5.2 Feedrate override

Procedure for Feedrate Override



JOG FEED RATE OVERRIDE

Set the feedrate override dial to the desired percentage (%) on the machine operator's panel, before or during automatic operation. On some machines, the same dial is used for the feedrate override dial and manual continuous feedrate dial. Refer to the appropriate manual provided by the machine tool builder for feedrate override.

Restrictions

- **Override Range**

The override that can be specified ranges from 0 to 254%. For individual machines, the range depends on the specifications of the machine tool builder.

- **Override during thread**

During threading, the override is ignored and the feedrate remains as specified by program.

5.3 RAPID TRAVERSE OVERRIDE

An override of four steps (F0, 25%, 50%, and 100%) can be applied to the rapid traverse rate. F0 is set by a parameter (No. 1421).

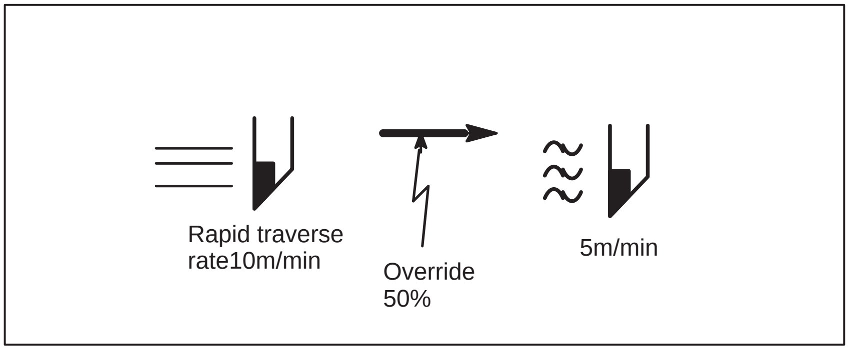
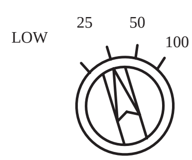


Fig. 5.3 Rapid traverse override

Procedure for Rapid Traverse Override



Rapid traverse override

Select one of the four feedrates with the rapid traverse override switch during rapid traverse. Refer to the appropriate manual provided by the machine tool builder for rapid traverse override.

Explanation

The following types of rapid traverse are available. Rapid traverse override can be applied for each of them.

1) Rapid traverse by G00.
2) Rapid traverse during a canned cycle.
3) Rapid traverse in G27, G28 and G30.
4) Manual rapid traverse.
5) Rapid traverse of manual reference position return

5.4 DRY RUN

The tool is moved at the feedrate specified by a parameter regardless of the feedrate specified in the program. This function is used for checking the movement of the tool under the state that the workpiece is removed from the table.

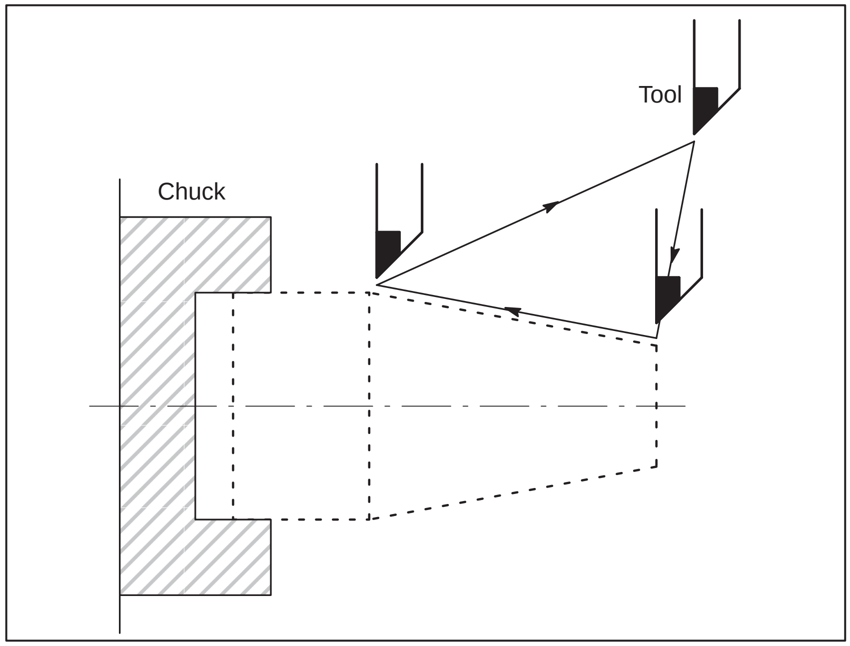


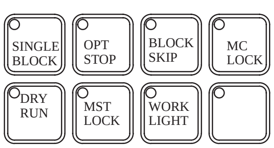
Fig. 5.4 Dry run

Procedure for Dry Run

Press the dry run switch on the machine operator's panel during automatic operation. The tool moves at the feedrate specified in a parameter. The rapid traverse switch can also be used for changing the feedrate. Refer to the appropriate manual provided by the machine tool builder for dry run.

Explanation

- **Dry run feedrate**



The dry run feedrate changes as shown in the table below according to the rapid traverse switch and parameters.

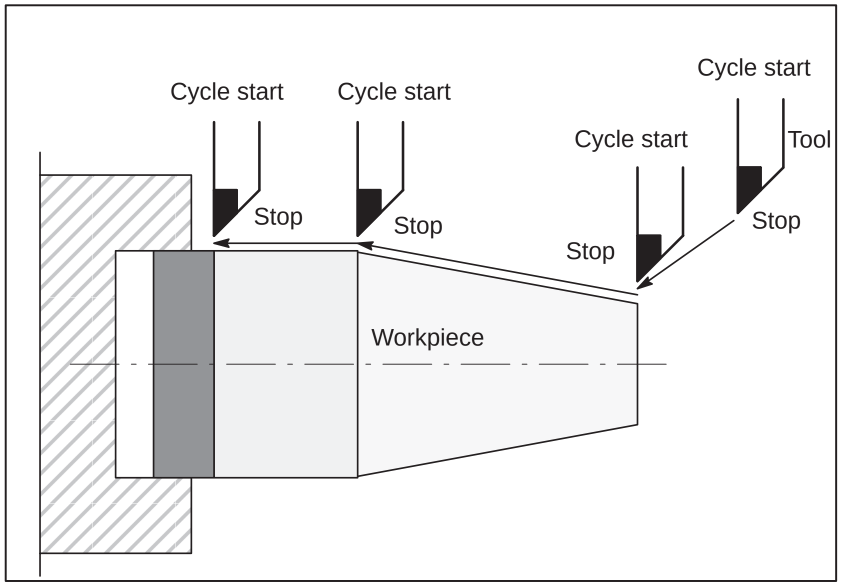
Rapid traverse button	Program command	
	Rapid traverse	Feed
ON	Rapid traverse rate	Dry run feedrate×JVmax *2)
OFF	Dry run speed × JV,or rapid traverse rate *1)	Dry run feedrate × JV

Max. cutting feedrate .... Setting by parameter No.1422
Rapid traverse rate .... Setting by parameter No.1420
Dry run feedrate .... Setting by parameter No.1410

JV: Jog feedrate override
*1) Dry run feedrate x JV when parameter RDR (bit 6 of No. 1401) is
1. Rapid traverse rate when parameter RDR is 0.
*2) Clamped to the maximum cutting feedrate
JVmax: Maximum value of jog feedrate override

5.5 SINGLE BLOCK

Pressing the single block switch starts the single block mode. When the cycle start button is pressed in the single block mode, the tool stops after a single block in the program is executed. Check the program in the single block mode by executing the program block by block.



Single block

Procedure for Single Block

1 Press the single block switch on the machine operator's panel. The execution of the program is stopped after the current block is executed.

2 Press the cycle start button to execute the next block. The tool stops after the block is executed.

Refer to the appropriate manual provided by the machine tool builder for single block execution.

Explanation

- **Reference position return and single block**

If G28 to G30 are issued, the single block function is effective at the intermediate point.

- **Single block during a canned cycle**

In a canned cycle, the single block stop points are as follows.

— — → Rapid traverse
S : Single block ——→ Cutting feed

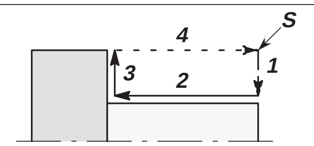
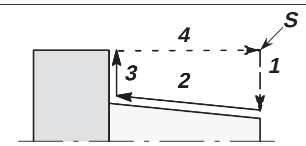
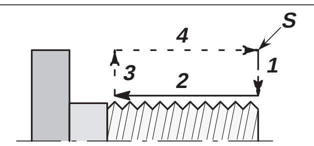
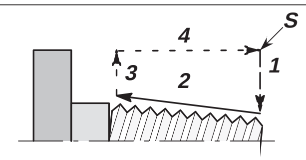
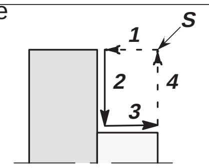
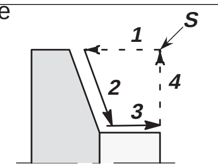
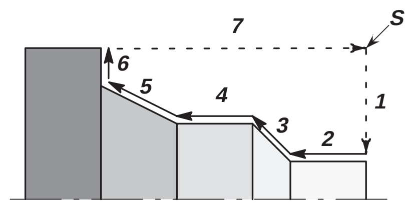
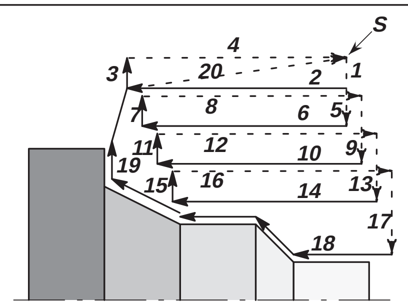
	Tool path		Explanation
	Straight cutting cycle	Taper cutting cycle	
☆G90 (Outer/inner turning cycle)			Tool path 1 to 4 is assumed as one cycle. After 4 is finished, a stop is made.
	Straight threading cycle	Taper threading cycle	
☆G92 (Threading cycle)			Tool path 1 to 4 is assumed as one cycle. After 4 is finished, a stop is made.
	Straight end surface cutting cycle	Taper end surface cutting cycle	
☆G94 (End surface turning cycle)			Tool path 1 to 4 is assumed as one cycle. After 4 is finished, a stop is made.
☆G70 (Finishing cycle)			Tool path 1 to 7 is assumed as one cycle. After 7 is finished, a stop is made.
☆G71 (Outer surface rough machining cycle) G72 (End surface rough machining cycle)	 This figure shows the case for G71. G72 is the same.		Each tool path 1 to 4,5 to 8,9 to 12, 13 to 16 and 17 to 20 is assumed as one cycle. After each cycle is finished, a stop is made.

Fig. 5.5(a) Single block during canned cycle (1/2)

S : Single–block stop — — ➤ Rapid traverse ——➤ Cutting feed

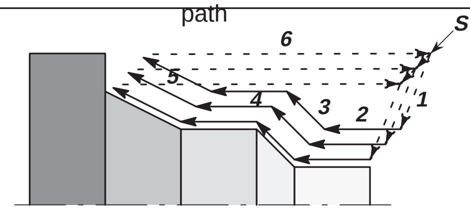
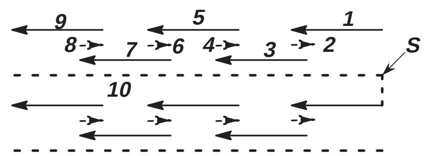
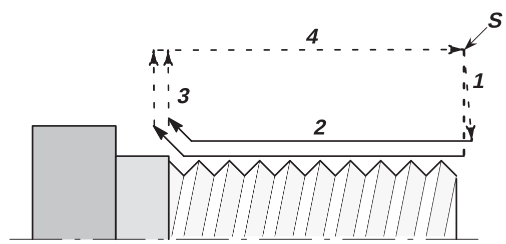
	Tool path	Explanation
☆G73 (Closed–loop cutting cycle)		Tool path 1 to 6 is assumed as one cycle. After 10 is finished, a stop is made.
☆G74 (End surface cutting–off cycle) G75 (Outer/inner surface cutting–off cycle)	 This figure shows the case for G74. G75 is the same.	Tool path 1 to 10 is assumed as one cycle. After 10 is finished, a stop is made.
☆G76 (Multiple repetitive threading cycle)		Tool path 1 to 4 is assume as one cycle. After 4 is finished, a stop is made.

Fig. 5.5(b) Single block during canned cycle (2/2)

- **Subprogram call and single block**

Single block stop is not performed in a block containing M98P_;. M99; or G65.
However, single block stop is even performed in a block with M98P_ or M99 command, if the block contains an address other than O, N or P.

- **Special single–block control**

Two–path control supports a single–block command signal for each of tool posts 1 and 2. Single–block stop can thus be specified for the automatic operation program for each tool post. Note, however, that when the single–block command signals for both tool posts 1 and 2 are turned on, the tools may stop at different positions according to the command programs.

The special single–block control function eliminates such a difference by applying feed hold to a tool post when the other tool post enters single–block stop mode.

The special single–block control function is enabled when bit 6 (DSB) of parameter No. 8100 is set to 1.

The single–block command signals for tool posts 1 and 2 are effective even when the special single–block control function is used.

When tool post 1 or 2 is placed in the single–block mask state or feed–hold mask state by a threading or custom macro program, the tool is not stopped until the mask state is terminated.

The tool posts are not synchronized. Therefore, if the following programs are executed, feed hold is applied to tool post 2 upon the completion of X10.0 for tool post 1, but the tool of tool post 2 is not stopped exactly at X10.0.

Tool post 1	Tool post 2
O0001 ;	O0002 ;
G50 X0 ;	G50 X0 ;
G01 X10. F100 ;	G01 X20. F100 ;
G01 X20. ;	

6 SAFETY FUNCTIONS

To immediately stop the machine for safety, press the Emergency stop button. To prevent the tool from exceeding the stroke ends, Overtravel check and Stroke check are available. This chapter describes emergency stop, overtravel check, and stroke check.

6.1 EMERGENCY STOP

If you press Emergency Stop button on the machine operator's panel, the machine movement stops in a moment.

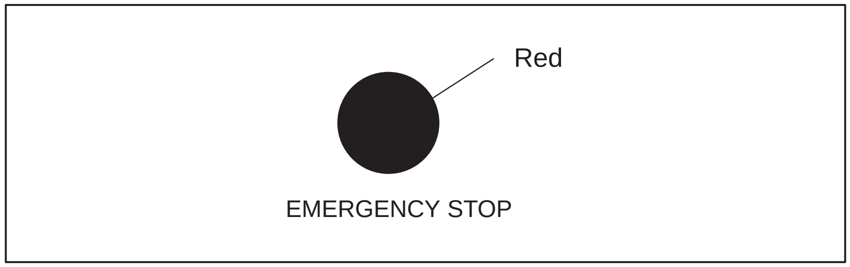


Fig. 6.1 Emergency stop

This button is locked when it is pressed. Although it varies with the machine tool builder, the button can usually be unlocked by twisting it.

Explanation

EMERGENCY STOP interrupts the current to the motor.
Causes of trouble must be removed before the button is released.

6.2 OVERTRAVEL

When the tool tries to move beyond the stroke end set by the machine tool limit switch, the tool decelerates and stops because of working the limit switch and an OVER TRAVEL is displayed.

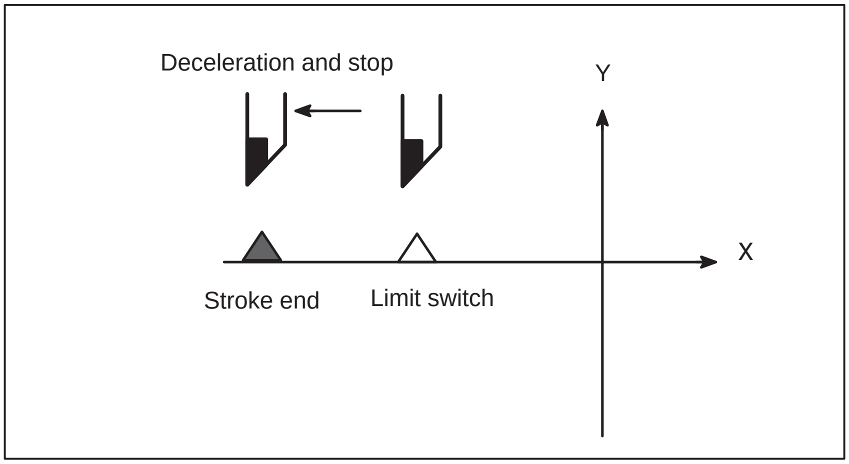


Fig. 6.2 Overtravel

Explanation

- **Overtravel during automatic operation**

When the tool touches a limit switch along an axis during automatic operation, the tool is decelerated and stopped along all axes and an overtravel alarm is displayed.

- **Overtravel during manual operation**

In manual operation, the tool is decelerated and stopped only along the axis for which the tool has touched a limit switch. The tool still moves along the other axes.

- **Releasing overtravel**

Press the reset button to reset the alarm after moving the tool to the safety direction by manual operation. For details on operation, refer to the operator's manual of the machine tool builder.

- **Alarm**

No.	Message	Description
506	Overtravel: +n	The tool has exceeded the hardware–specified overtravel limit along the positive nth axis (n: 1 to 8).
507	Overtravel: –n	The tool has exceeded the hardware–specified overtravel limit along the negative nth axis (n: 1 to 8).

6.3 STROKE CHECK

There areas which the tool cannot enter can be specified with stored stroke limit 1, stored stroke limit 2,and stored stroke limit 3.

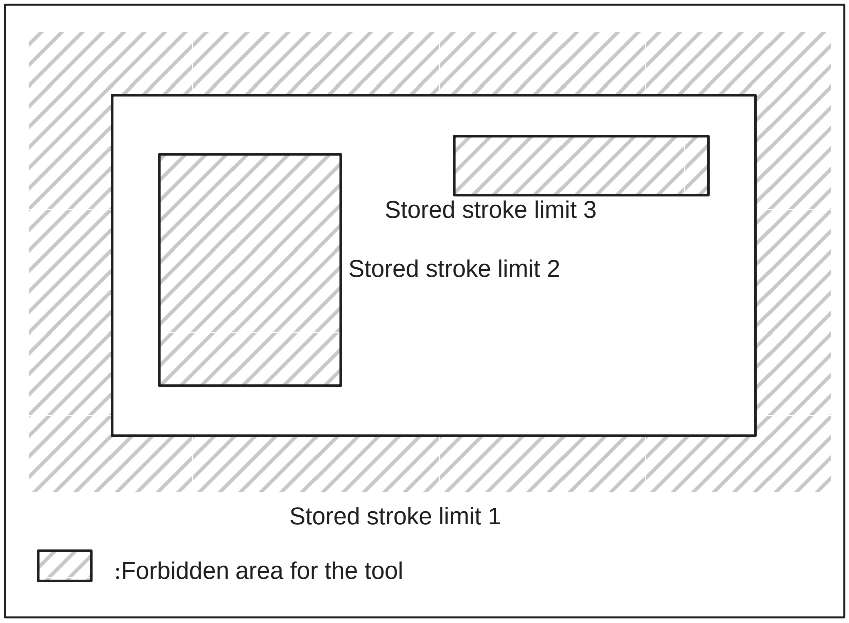


Fig. 6.3(a) Stroke check

When the tool exceeds a stored stroke limit, an alarm is displayed and the tool is decelerated and stopped.
When the tool enters a forbidden area and an alarm is generated, the tool can be moved in the reverse direction from which the tool came.

Explanation

- **Stored stroke limit 1**

Parameters (Nos. 1320, 1321 or Nos. 1326, 1327) set boundary. Outside the area of the set limits is a forbidden area. The machine tool builder usually sets this area as the maximum stroke.

- **Stored stroke limit 2 (G22, G23)**

Parameters (Nos. 1322, 1323) or commands set these boundaries. Inside or outside the area of the limit can be set as the forbidden area. Parameter OUT (No. 1300#0) selects either inside or outside as the forbidden area.

In case of program command a G22 command forbids the tool to enter the forbidden area, and a G23 command permits the tool to enter the forbidden area. Each of G22; and G23; should be commanded independently of another commands in a block.
The command below creates or changes the forbidden area:

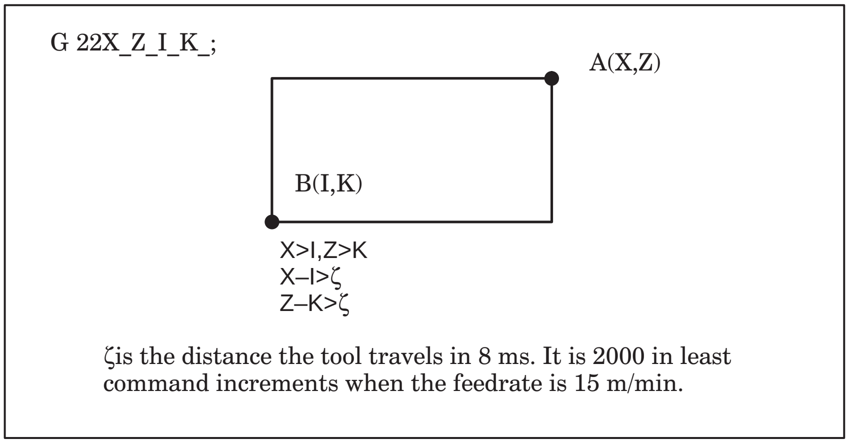


Fig. 6.3(b) Creating or changing the forbidden area using a program

When setting the area by parameters, points A and B in the figure below must be set.

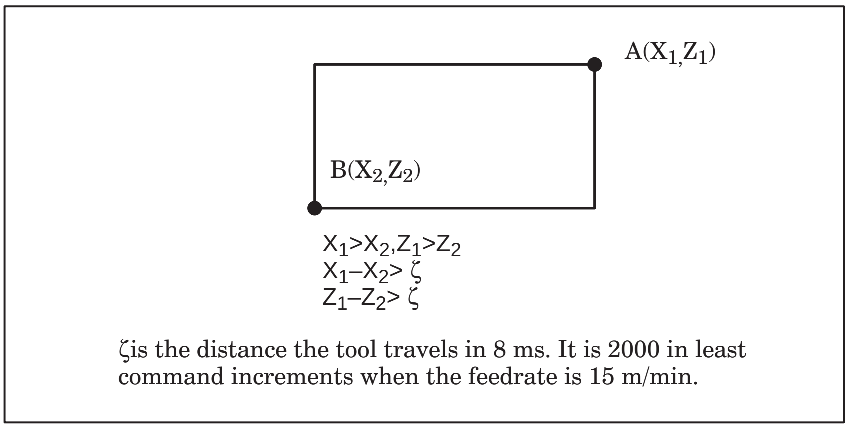


Fig. 6.3(c) Creating or changing the forbidden area using a parameters

In limit 2, even if you mistake the order of the coordinate value of the two points, a rectangular, with the two points being the apexes, will be set as the area.
When you set the forbidden area X_1,Z_1,X_2,and Z_2 through parameters (Nos. 1322, 1323), the data should be specified by the distance from the reference position in the least command increment. (Output increment)

If set the forbidden area XZIK by a G22 command, specify the data by the distance from the reference position in the least input increment (Input increment.) The programmed data are then converted into the numerical values in the least command increment, and the values are set as the parameters.

- **Stored stroke limit 3**

Set the boundary with parameters No. 1324 and 1325. The area inside the boundary becomes the forbidden area.

- **Checkpoint for the forbidden area**

The parameter setting or programmed value (XZIK) depends on which part of the tool or tool holder is checked for entering the forbidden area. Confirm the checking position (the top of the tool or the tool chuck) before programming the forbidden area.
If point C (The top of the tool) is checked in Fig. 6.3 (d) , the distance “c” should be set as the data for the stored stroke limit function. If point D (The tool chuck) is checked, the distance “d” must be set.

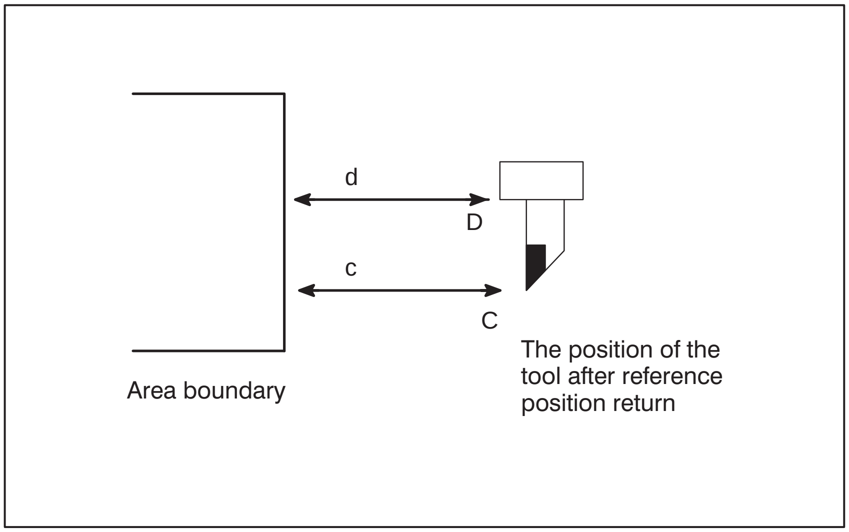


Fig. 6.3(d) Setting the forbidden area

- **Forbidden area over–lapping**

Area can be set in piles.

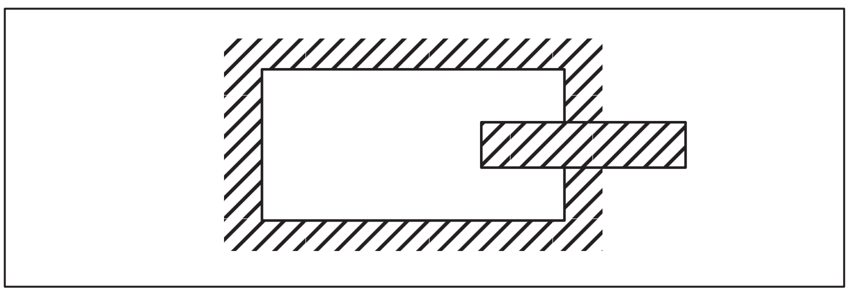


Fig. 6.3(e) Setting the forbidden area over lapping

Unnecessary limits should be set beyond the machine stroke.

- **Effective time for a forbidden area**

Each limit becomes effective after the power is turned on and manual reference position return or automatic reference position return by G28 has been performed.
After the power is turned on, if the reference position is in the forbidden area of each limit, an alarm is generated immediately. (Only in G22 mode for stored stroke limit 2).

- **Releasing the alarms**

When the tool has become unmovable in the forbidden area, push the emergency stop button to release the forbidden condition and move the tool out of the forbidden area in the G23 mode; then, if the setting is wrong, correct it and perform the reference position return again.

- **Change from G23 to G22 in a forbidden area**

When G23 is switched to G22 in the forbidden area, the following results.
(1)When the forbidden area is inside, an alarm is informed in the next move.
(2)When the forbidden area is outside, an alarm is informed immediately.

- **Setting the forbidden area for the two–path control**

For the two–path control, set a forbidden area for each tool post.

Notes
In setting a forbidden area, if the two points to be set are the same, the area is as follows:
(1)When the forbidden area is limit 1, all areas are forbidden areas.
(2)When the forbidden area is limit 2 or limit 3 all areas are movable areas.

- **Timing for displaying an alarm**

Parameter BFA (bit 7 of No. 1300) selects whether an alarm is displayed immediately before the tool enters the forbidden area or immediately after the tool has entered the forbidden area.

ALram

Number	Message	Contents
500	OVER TRAVEL: +n	Exceeded the n–th axis (1–8) + side stored stroke limit 1.
501	OVER TRAVEL: –n	Exceeded the n–th axis (1–8) – side stored stroke limit 1.
502	OVER TRAVEL: +n	Exceeded the n–th axis (1–8) + side stored stroke limit 2.
503	OVER TRAVEL: –n	Exceeded the n–th axis (1–8) – side stored stroke limit 2.
504	OVER TRAVEL: +n	Exceeded the n–th axis (1–8) + side stored stroke limit 3.
505	OVER TRAVEL: –n	Exceeded the n–th axis (1–8) – side stored stroke limit 3.

6.4 CHUCK AND TAILSTOCK BARRIERS

The chuck–tailstock barrier function prevents damage to the machine by checking whether the tool tip fouls either the chuck or tailstock.
Specify an area into which the tool may not enter (entry–inhibition area). This is done using the special setting screen, according to the shapes of the chuck and tailstock. If the tool tip should enter the set area during a machining operation, this function stops the tool and outputs an alarm message.
The tool can be cleared from the area only by retracting it in the direction opposite to that in which the tool entered the area.

Setting the chuck and tailstock barriers

- **Setting the shapes of the chuck and tailstock**

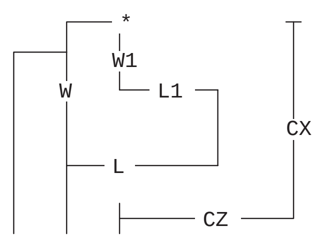
1 Press the OFFSET SETTING function key.

2 Press the ▷ continuous menu key. Then, press the **[BARIER]** chapter selection soft key.

3 Pressing the page key toggles the display between the chuck barrier setting screen and tailstock barrier setting screen.

Chuck barrier setting screen

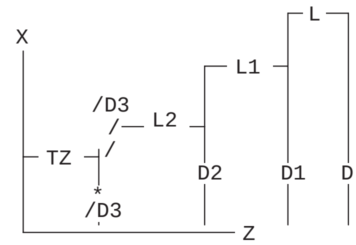
```
BARRIER (CHUCK)                    O0000 N00000
```




```
TY=0(0:IN,1:OUT)
L =    50.000
W =    60.000
L1=    25.000
W1=    30.000

CX=   200.000
CZ=  –100.000

ACTUAL POSITION (ABSOLUITE)
    X   200.000           Z    50.000

>_
MDI **** *** ***    14:46:09
[     ][ W.SHFT ][     ][ BARIER ][ (OPRT) ]
```

Tailstock barrier setting screen

```
BARRIER (TAILSTOCK)                          O0000 N00000
```




```
L =   100.000
D =   200.000
L1=    50.000
D1=   100.000
L2=    50.000
D2=    50.000
D3=    30.000

TZ=   100.000

ACTUAL POSITION (ABSOLUTE)
    X   200.000              Z    50.000

>_
MDI **** *** ***     14:46:09
[ INPUT ][ +INPUT ][   SET   ][      ][      ]
```

4 Position the cursor to each item defining the shape of the chuck or tailstock, enter the corresponding value, then press the **[INPUT]** soft key. The value is set. Pressing the **[+INPUT]** soft key after a value has been entered adds the entered value to the current value, the new setting being the sum of the two values.
Items CX and CZ, both on the chuck barrier setting screen, and item TZ on the tailstock barrier setting screen can also be set in another way. Manually move the tool to the desired position, then press the **[SET]** soft key to set the coordinate(s) of the tool in the workpiece coordinate system. If a tool having an offset other than 0 is manually moved to the desired position with no compensation applied, compensate for the tool offset in the set coordinate system.
Items other than CX, CZ, and TZ cannot be set by using the **[SET]** soft key.

Example)

When the tool tip enters the entry–inhibition area during machining, the function stops the movement of the tool and displays an alarm message. Since the machine system can stop only a slight delay after the CNC stops, the tool will actually stop moving at a point within the specified boundary. For safety, therefore, set an area a little larger than the determined area. The distance between the boundaries of these two areas, L, is calculated from the following equation, based on the rapid traverse rate.

$$L = \textit{(Rapid traverse Rate)} \times \frac{1}{7500}$$

When the rapid traverse rate is 15 m/min, for example, set an area having a boundary 2 mm outside that of the determined area.
The shapes of the chuck and tailstock can be set using parameters No. 1330 to 1345.

Note)

Set G23 mode before attempting to specify the shapes of the chuck and tailstock.

- **Reference position return**

1 Return the tool to the reference position along the X– and Z–axes. The chuck–tailstock barrier function becomes effective only once reference position return has been completed after power on.
When an absolute position detector is provided, reference position return need not always be performed. The positional relationship between the machine and the absolute position detector, however, must be determined.

- **Reference position return**

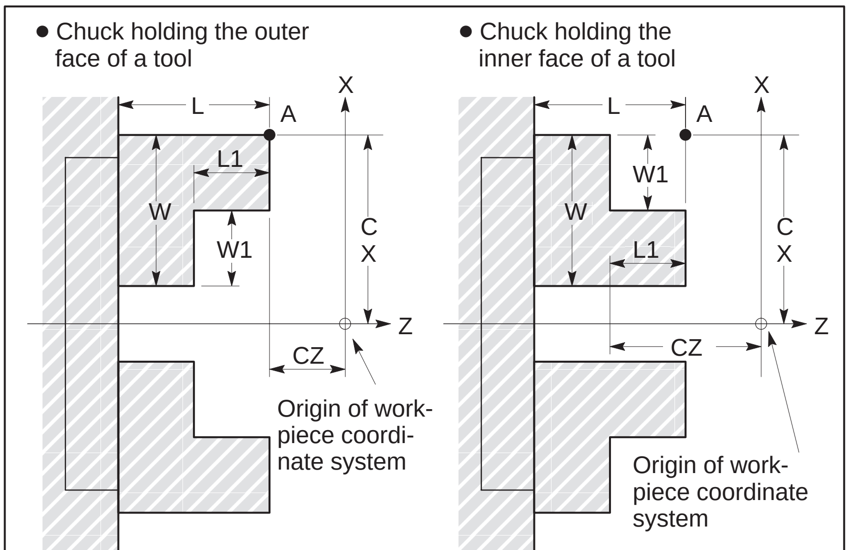
1 After reference position return, specifying G22 (stored stroke limit on) makes the entry–inhibition areas for the chuck and tailstock effective. Specifying G23 (stored stroke limit off) disables the function.
Even if G22 is specified, the entry–inhibition area for the tailstock can be disabled by issuing a tailstock barrier signal.
When the tailstock is pushed up against a workpiece or separated from the workpiece by using the miscellaneous functions, PMC signals are used to enable or disable the tailstock setting area.

G code	Tailstock barrier signal	Chuck barrier	Tailstock barrier
G22	0	Effective	Effective
	1	Effective	Ineffective
G23	No relation	Ineffective	Ineffective

G22 is usually selected when the power is turned on. Using G23, bit 7 of parameter No. 3402, however, it can be changed to G23.

Explanations

- **Setting the shape of the chuck barrier**



Note) The hatched areas indicate entry–inhibition areas.

Symbol	Description
TY	Chuck–shape selection (0: Holding the inner face of a tool, 1: Holding the outer face of a tool)
CX	Chuck position (along X–axis)
CZ	Chuck position (along Z–axis)
L	Length of chuck jaws
W	Depth of chuck jaws (radius)
L1	Holding length of chuck jaws
W1	Holding depth of chuck jaws (radius)

TY :

Selects a chuck type, based on its shape. Specifying 0 selects a chuck that holds the inner face of a tool. Specifying 1 selects a chuck that holds the outer face of a tool. A chuck is assumed to be symmetrical about its Z–axis.

CX, CZ:

Specify the coordinates of a chuck position, point A, in the workpiece coordinate system. These coordinates are not the same as those in the machine coordinate system. Table 1 lists the units used to specify the data.

Note)

Whether diameter programming or radius programming is used for the axis determines the programming system. When diameter programming is used for the axis, use diameter programming to enter data for the axis.

Table 1 Units

Increment system	Data unit		Valid data range
	IS–A	IS–B	
Metric input	0.001 mm	0.0001 mm	−99999999 to +99999999
Inch input	0.0001 inch	0.00001 inch	−99999999 to +99999999

L, L1, W, W1:

Define the shape of a chuck. Table 2 lists the units used to specify the data.

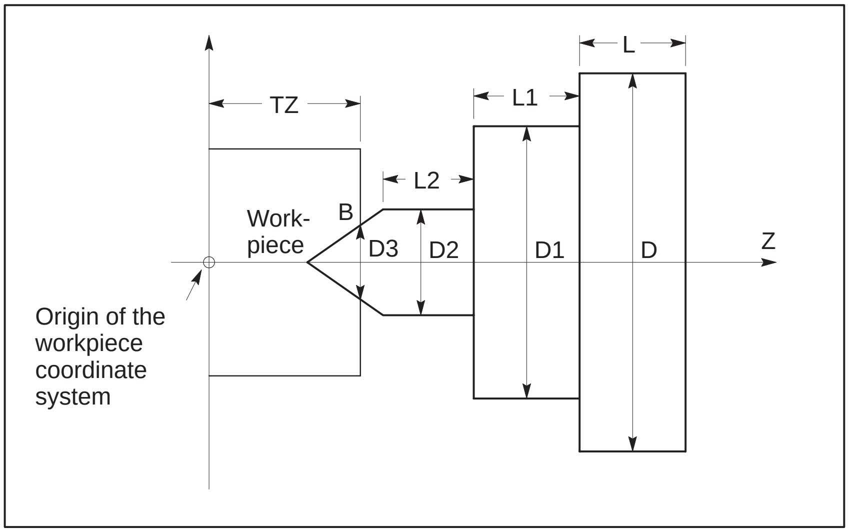
Note)

Always specify W and W1 in radius. When radius programming is used for the Z–axis, specify L and L1 in radius.

Table 2 Units

Increment system	Data unit		Valid data range
	IS–A	IS–B	
Metric input	0.001 mm	0.0001 mm	−99999999 to +99999999
Inch input	0.0001 inch	0.00001 inch	−99999999 to +99999999

- **Setting the shape of a tailstock barrier**



Symbol	Description
TZ	Tailstock position (along the Z–axis)
L	Tailstock length
D	Tailstock diameter
L1	Tailstock length (1)
D1	Tailstock diameter (1)
L2	Tailstock length (2)
D2	Tailstock diameter (2)
D3	Tailstock diameter (3)

TZ :

Specifies the Z coordinate of the chuck position, point B, in the workpiece coordinate system. These coordinates are not the same as those in the machine coordinate system. Table 3 lists the units used to specify the data. A tailstock is assumed to be symmetrical about its Z–axis.

Note)

Whether diameter programming or radius programming is used for the Z–axis determines the programming system.

Table 3 Units

Increment system	Data unit		Valid data range
	IS–A	IS–B	
Metric input	0.001 mm	0.0001 mm	−99999999 to +99999999
Inch input	0.0001 inch	0.00001 inch	−99999999 to +99999999

L, L1, L2, D, D1, D2, D3:

Define the shape of a tailstock. Table 4 lists the units used to specify the data.

Note)

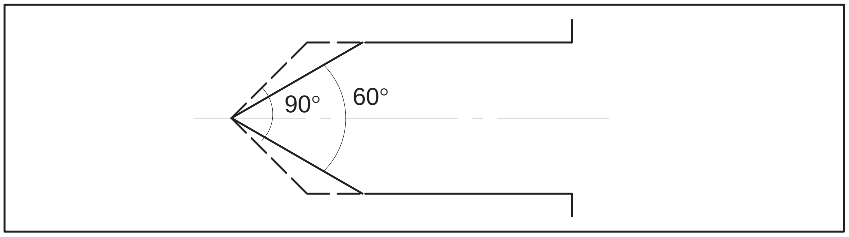
Always specify D, D1, D2, and D3 in diameter programming. When radius programming is used for the Z–axis, specify L, L1, and L2 in radius.

Table 4 Units

Increment system	Data unit		Valid data range
	IS–A	IS–B	
Metric input	0.001 mm	0.0001 mm	−99999999 to +99999999
Inch input	0.0001 inch	0.00001 inch	−99999999 to +99999999

- **Setting the entry–inhibition area for the tailstock tip**

The tip angle of the tailstock is 60 degrees. The entry–inhibition area is set around the tip, assuming the angle to be 90 degrees, as shown below.



Limitations

- **Correct setting of an entry–inhibition area**

If an entry–inhibition area is incorrectly set, it may not be possible to make the area effective. Avoid making the following settings:

- L < L1 or W < W1 in the chuck–shape settings.
- D2 < D3 in the tailstock–shape settings.
- A chuck setting overlapping that of the tailstock.

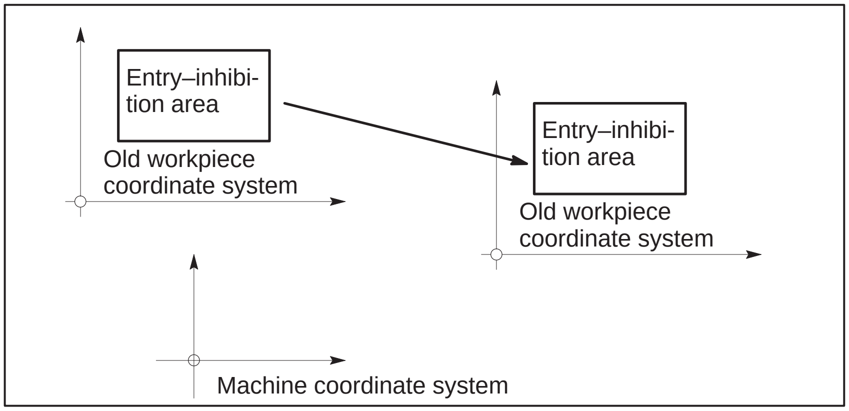
- **Retraction from the entry–inhibition area**

If the tool enters the entry–inhibition area and an alarm is issued, switch to manual mode, retract the tool manually, then reset the system to release the alarm. In manual mode, the tool can be moved only in the opposite direction to that in which the tool entered the area. The tool cannot be moved in the same direction (further into the area) as it was travelling when the tool entered the area.
When the entry–inhibition areas for the chuck and tailstock are enabled, and the tool is already positioned within those areas, an alarm is issued when the tool moves. When the tool cannot be retracted, change the setting of the entry–inhibition areas, such that the tool is outside the areas, reset the system to release the alarm, then retract the tool. Finally, reinstall the original settings.

- **Coordinate system**

An entry–inhibition area is defined using the workpiece coordinate system. Note the following.

1 When the workpiece coordinate system is shifted by means of a command or operation, the entry–inhibition area is also shifted by the same amount.



Use of the following commands and operations will shift the workpiece coordinate system.

Commands:

G54 to G59, G52, G50 (G92 in G code system B or C)

Operations:

Manual handle interrupt, change in offset relative to the workpiece reference point, change in tool offset (tool geometry compensation), operation with machine lock, manual operation with machine absolute signal off

2 When the tool enters an entry–inhibition area during automatic operation, set the manual absolute signal, *ABSM, to 0 (on), then manually retract the tool from the area. If this signal is 1, the distance the tool moves in manual operation is not counted in the tool coordinates in the workpiece coordinate system. This results in the state where the tool can never be retracted from the entry–inhibition area.

- **Stored stroke limit 2**

When both stored stroke limit 2 and the chuck–tailstock barrier function are provided, the barrier takes priority over the stroke limit. Stored stroke limit 2 is ignored.

Alarms

Number	Message	Contents
502	OVER TRAVEL: +X	The tool has entered the entry–inhibition area during positive–direction movement along the X–axis.
	OVER TRAVEL: +Z	The tool has entered the entry–inhibition area during positive–direction movement along the Z–axis.
503	OVER TRAVEL: –X	The tool has entered the entry–inhibition area during negative–direction movement along the X–axis.
	OVER TRAVEL: –Z	The tool has entered the entry–inhibition area during negative–direction movement along the Z–axis.

6.5 STROKE LIMIT CHECK PRIOR TO PERFORMING MOVEMENT

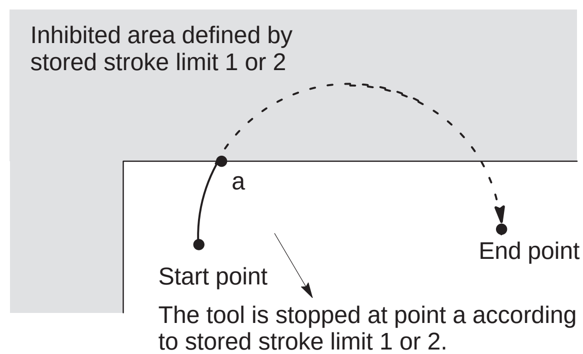
During automatic operation, before the movement specified by a given block is started, whether the tool enters the inhibited area defined by stored stroke limit 1, 2, or 3 is checked by determining the position of the end point from the current position of the machine and a specified amount of travel. If the tool is found to enter the inhibited area defined by a stored stroke limit, the tool is stopped immediately upon the start of movement for that block, and an alarm is displayed.

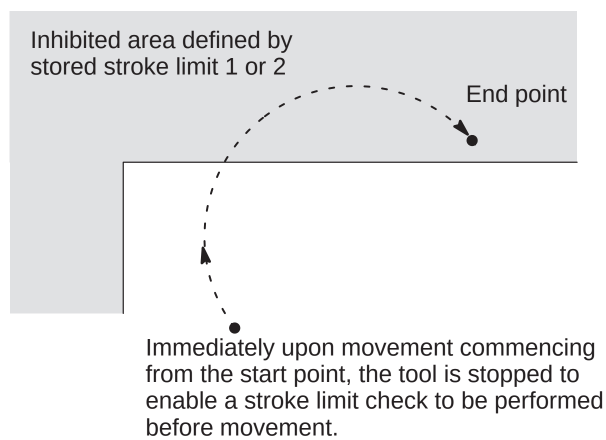
Notes

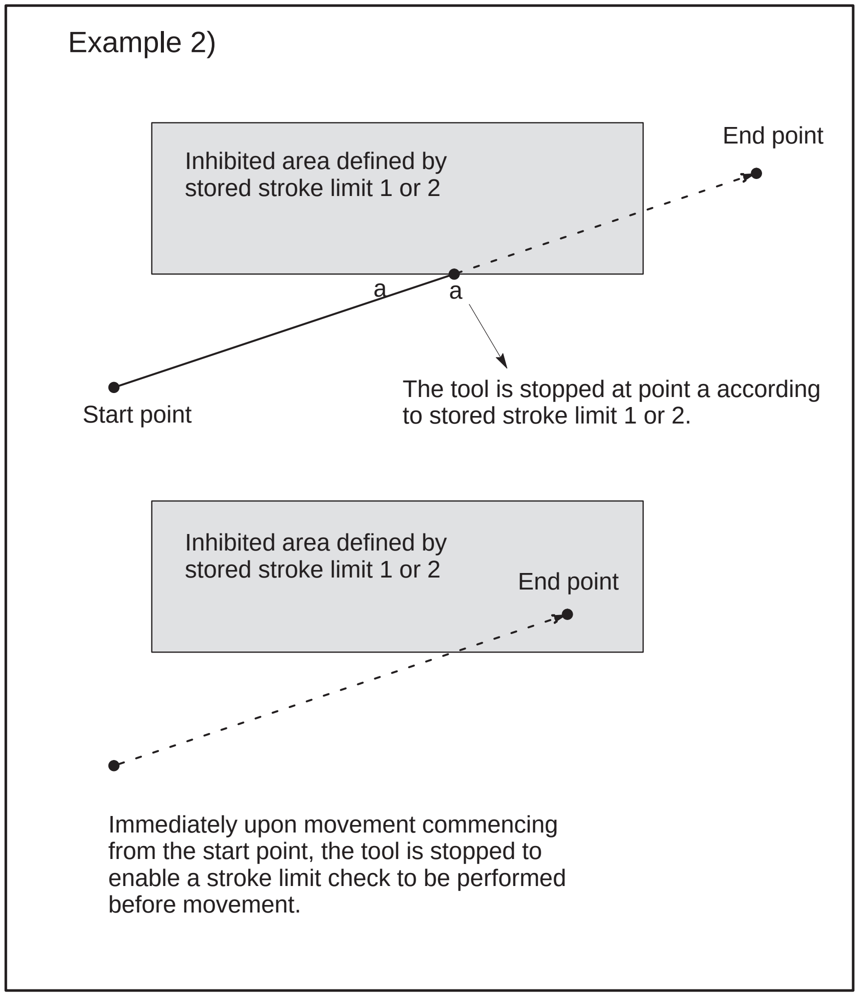
Notes

1 Whether the coordinates of the end point, reached as a result of traversing the distance specified in each block, are in a inhibited area is checked. In this case, the path followed by a move command is not checked. However, if the tool enters the inhibited area defined by stored stroke limit 1, 2, or 3, an alarm is issued. (See the examples below.)

Example 1)







Explanations

When a stroke limit check prior to movement is performed, whether to check the movement performed by a G31 (skip) block and G37 (automatic tool length measurement) block can be determined using NPC (bit 2 of parameter No. 1301).

Limitations

- **Machine lock**

If machine lock is applied at the start of movement, no stroke limit check made before movement is performed.

- **G23**

When stored stroke limit 2 is disabled (G23 mode), no check is made to determine whether the tool enters the inhibited area defined by stored stroke limit 2.

- **Program restart**

When a program is restarted, an alarm is issued if the restart position is within a inhibited area.

- **Manual intervention following a feed hold stop**

When the execution of a block is restarted after manual intervention following a feed hold stop, no alarm is issued even if the end point is within a inhibited area.

- **A block consisting of multiple operations**

If a block consisting of multiple operations (such as a canned cycle and exponential interpolation) is executed, an alarm is issued at the start point of any operation whose end point falls within a inhibited area.

- **Cyrindrical interpolation mode**

  In cylindrical interpolation mode, no check is made.

- **Polar coordinate interpolation mode**

  In polar coordinate interpolation mode, no check is made.

- **Slanted axis control**

  When the slanted axis control option is selected, no check is made.

- **Simple synchronous control**

  In simple synchronous control, only the master axis is checked; no slave axes are checked.

- **Drawing**

  During drawing, no check is made.

- **PMC axis control**

  No check is made for a movement based on PMC axis control.

- **Chuck/tailstock barrier**

  No check is made for a chuck/tailstock barrier area (lathe system).

- **Synchronous mixed mode**

  No check is made for an axis placed in synchronous mixed mode (two–path lathe control).

Alarm

Number	Message	Contents
506	OVER TRAVEL : +n	The stroke limit check, made before movements is performed, reveals that the tool would enter a inhibited area in the + direction.
507	OVER TRAVEL : –n	The stroke limit check, made before movements is performed, reveals that the tool would enter a inhibited area in the – direction.

7 ALARM AND SELF–DIAGNOSIS FUNCTIONS

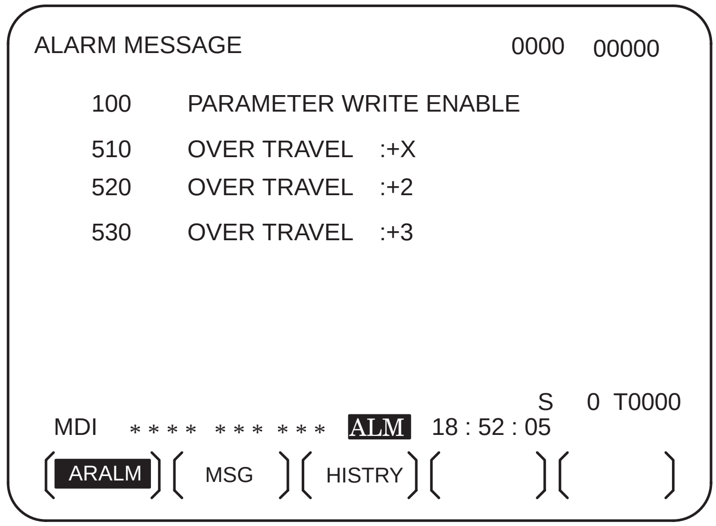
When an alarm occurs, the corresponding alarm screen appears to indicate the cause of the alarm. The causes of alarms are classified by error codes. Up to 25 previous alarms can be stored and displayed on the screen (alarm history display).
The system may sometimes seem to be at a halt, although no alarm is displayed. In this case, the system may be performing some processing. The state of the system can be checked using the self–diagnostic function.

7.1 ALARM DISPLAY

Explanations

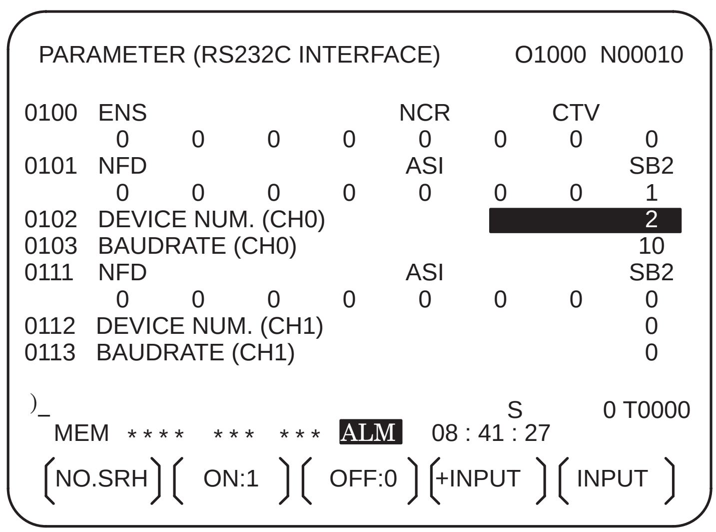
- **Alarm screen**

When an alarm occurs, the alarm screen appears.



- **Another method for alarm displays**

In some cases, the alarm screen does not appear, but an ALM is displayed at the bottom of the screen.



In this case, display the alarm screen as follows:

1 Press the function key MESSAGE .

2 Press the chapter selection soft key **[ALARM]**.

- **Reset of the alarm**

Error codes and messages indicate the cause of an alarm. To recover from an alarm, eliminate the cause and press the reset key.

- **Error codes**

The error codes are classified as follows:
No. 000 to 232: Program errors(*)
No. 300 to 308: Absolute pulse coder (APC) alarms
No. 350 and 351: Serial pulse coder (SPC) alarms
No. 400 to 417: Servo alarms
No. 500 to 507: Overtravel alarms
No. 700 to 704: Overheat alarms
No. 750 to 762: Spindle alarms
No. 900 to 973: System alarms

*For an alarm (No. 000 to 232) that occurs in association with background operation, the indication ”xxxBP/S alarm” is provided (where xxx is an alarm number). Only a BP/S alarm is provided for No. 140.
See the error code list in the appendix for details of the error codes.

7.2 ALARM HISTORY DISPLAY

Up to 25 of the most recent CNC alarms are stored and displayed on the screen.
Display the alarm history as follows:

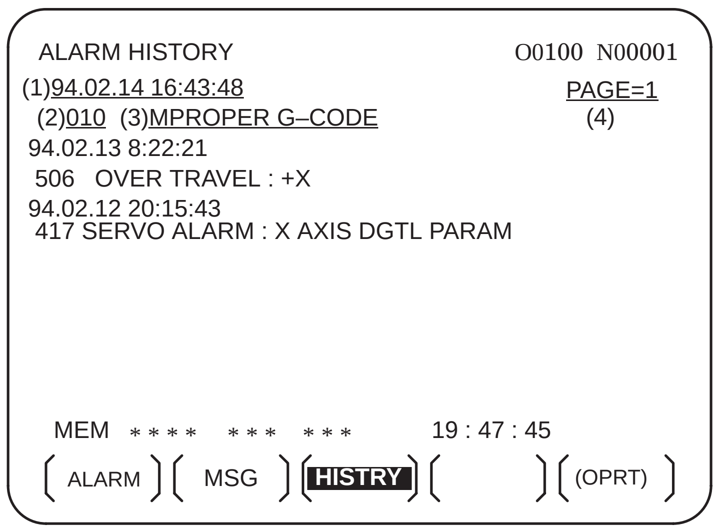
Procedure for Alarm History Display

1 Press the function key MESSAGE .

2 Press the chapter selection soft key **[HISTRY]**.
The alarm history appears.
The following information items are displayed.
(1)The date the alarm was issued
(2)Alarm No.
(3)Alarm message (some contains no message)

3

4 To delete the recorded information, press the softkey **[(OPRT)]** then the **[DELETE]** key.



(1)The date the alarm was issued
(2)Alarm No.
(3)Alarm message (some contains no message)
(4)Page number

7.3 CHECKING BY SELF–DIAGNOSTIC SCREEN

The system may sometimes seem to be at a halt, although no alarm has occurred. In this case, the system may be performing some processing. The state of the system can be checked by displaying the self–diagnostic screen.

Procedure for Diagnostic

1 Press the function key SYSTEM .

2 Press the chapter select key **[DGNOS]**.

3 The diagnostic screen has more than 1 pages. Select the screen by the following operation.

(1) Change the page by the 1–page change key.

(2) Method by soft key
– Key input the number of the diagnostic data to be displayed.
– Press **[N SRCH]**.

```
DIAGNOSTIC (GENERAL)                          O0000  N00000


000  WAITING FOR FIN SIGNAL                          :0
001  MOTION                                          :0
002  DWELL                                           :0
003  IN–POSITION  CHECK                              :0
004  FEEDRATE  OVERRIDE  0%                          :0
005  INTERLOCK/START–LOCK                            :0
006 SPINDLE  SPEED  ARRIVAL  CHECK                   :0

 )_
 EDIT  ****   ***   ***          14 : 51 : 55
[ PARAM ][ DGNOS ][  PMC  ][ SYSTEM ][ (OPRT) ]
```

Explanations

- **Self diagnostic screen at 2–path control**

For the two–path control, the diagnostic screen for the tool post selected with the tool post selection switch is displayed. When displaying the diagnostic screen for the other tool post, specify the tool post with the tool post selection switch.

Explanations

Diagnostic numbers 000 to 015 indicate states when a command is being specified but appears as if it were not being executed. The table below lists the internal states when 1 is displayed at the right end of each line on the screen.

Table 7.3 (a) Alarm displays when a command is specified but appears as if it were not being executed

No.	Display	Internal status when 1 is displayed
000	WAITING FOR FIN SIGNAL	M, S. T function being executed
001	MOTION	Move command in automatic operation being executed
002	DWELL	Dwell being executed
003	IN–POSITION CHECK	In–position check being executed
004	FEEDRATE OVERRIDE 0%	Cutting feed override 0%
005	INTERLOCK/START–LOCK	Interlock ON
006	SPINDLE SPEED ARRIVAL CHECK	Waiting for spindle speed arrival signal to turn on
010	PUNCHING	Data being output via reader puncher interface
011	READING	Data being input via reader puncher interface
012	WAITING FOR (UN) CLAMP	Waiting for index table clamp/unclamp before B axis index table indexing start/after B axis index table indexing end to complete
013	JOG FEEDRATE OVERRIDE 0%	Jog override 0%
014	WAITING FOR RESET.ESP.RRW.OFF	Emergency stop, external reset, reset & rewind, or MDI panel reset key on
015	EXTERNAL PROGRAM NUMBER SEARCH	External program number searching

Table 7.3 (b) Alarm displays when an automatic operation is stopped or paused.

No.	Display	Internal status when 1 is displayed
020	CUT SPEED UP/DOWN	Set when emergency stop turns on or when servo alarm occurs
021	RESET BUTTON ON	Set when reset key turns on
022	RESET AND REWIND ON	Reset and rewind turned on
023	EMERGENCY STOP ON	Set when emergency stop turns on
024	RESET ON	Set when external reset, emergency stop, reset, or reset & rewind key turns on
025	STOP MOTION OR DWELL	A flag which stops pulse distribution. It is set in the following cases. (1)External reset turned on. (2)Reset & rewind turned on. (3)Emergency stop turned on. (4)Feed hold turned on. (5)The MDI panel reset key turned on. (6)Switched to the manual mode(JOG /HANDLE/INC). (7)Other alarm occurred. (There is also alarm which is not set.)

Diagnostic numbers 020 to 025 indicate the states when automatic operation is stopped or paused.

The table below shows the signals and states which are enabled when each diagnostic data item is 1. Each combination of the values of the diagnostic data indicates a unique state.

020	CUT SPEED UP/DOWN	1	0	0	0	1	0	0
021	RESET BUTTON ON	0	0	1	0	0	0	0
022	RESET AND REWIND ON	0	0	0	1	0	0	0
023	EMERGENCY STOP ON	1	0	0	0	0	0	0
024	RESET ON	1	1	1	1	0	0	0
025	STOP MOTION OR DWELL	1	1	1	1	1	1	0

Emergency stop signal input
External reset signal input
MDI reset button turned on
Reset & rewind input
Servo alarm generation
Changed to another mode or feed hold
Single block stop

Diagnostic numbers 030 and 031 indicate TH alarm states.

No.	Display	Meaning of data
030	CHARACTER NUMBER TH DATA	The position of the character which caused TH alarm is displayed by the number of characters from the beginning of the block at TH alarm
031	TH DATA	Read code of character which caused TH alarm

8 DATA INPUT/OUTPUT

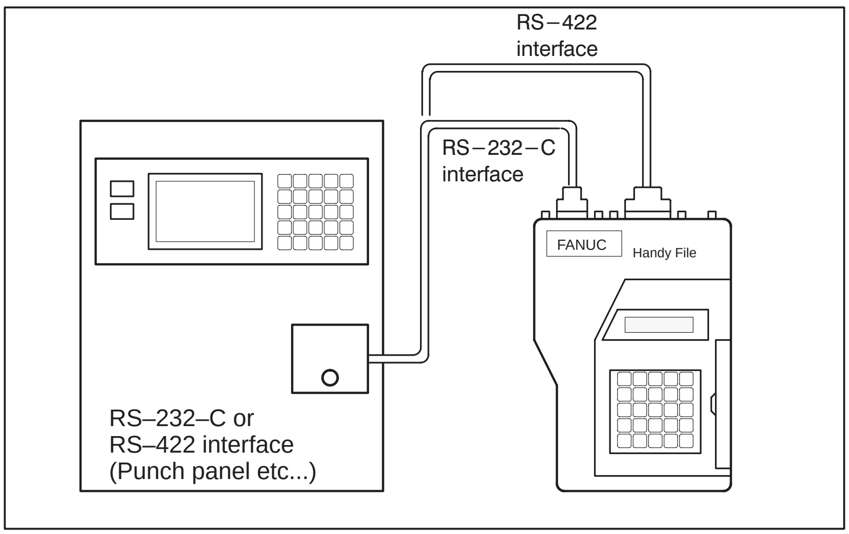
NC data is transferred between the CNC and external input/output devices such as the Handy File.

The following types of data can be entered and output :

1.Program
2.Offset data
3.Parameter
4.Pitch error compensation data
5.Custom macro common variable

Before an input/output device can be used, the input/output related parameters must be set.

For how to set parameters, see Chapter III–2.



8.1 FILES

Of the external input/output devices, the FANUC Handy File and FANUC Floppy Cassette use floppy disks as their input/output medium, and the FANUC FA Card uses an FA card as its input/output medium.
In this manual, an input/output medium is generally referred to as a floppy. However, when the description of one input/output medium varies from the description of another, the name of the input/output medium is used. In the text below, a floppy represents a floppy disk or FA card.
Unlike an NC tape, a floppy allows the user to freely choose from several types of data stored on one medium on a file–by–file basis.
Input/output is possible with data extending over more than one floppy disk.

Explanations

● What is a File

The unit of data, which is input/output between the floppy and the CNC by one input/output operation (pressing the VREADW or VPUNCHW key), is called a HfileI. When inputting CNC programs from, or outputting them to the floppy, for example, one or all programs within the CNC memory are handled as one file.
Files are assigned automatically file numbers 1,2,3,4 and so on, with the lead file as 1.

File 1	File 2	File 3		File n	Blank

● Request for floppy replacement

When one file has been entered over two floppies, LEDs on the adaptor flash alternately on completion of data input/output between the first floppy and the CNC, prompting floppy replacement. In this case, take the first floppy out of the adaptor and insert a second floppy in its place. Then, data input/output will continue automatically.
Floppy replacement is prompted when the second floppy and later is required during file search–out, data input/output between the CNC and the floppy, or file deletion.

Floppy 1

File 1	File 2	File 3		File (k–1)	File k

Floppy 2

Continua-tion of file k	File (k+1)		File n	Blank

Since floppy replacement is processed by the input/output device, no special operation is required. The CNC will interrupt data input/output operation until the next floppy is inserted into the adaptor.
When reset operation is applied to the CNC during a request for floppy replacement, the CNC is not reset at once, but reset after the floppy has been replaced.

- **Protect switch**

The floppy is provided with the write protect switch. Set the switch to the write enable state. Then, start output operation.

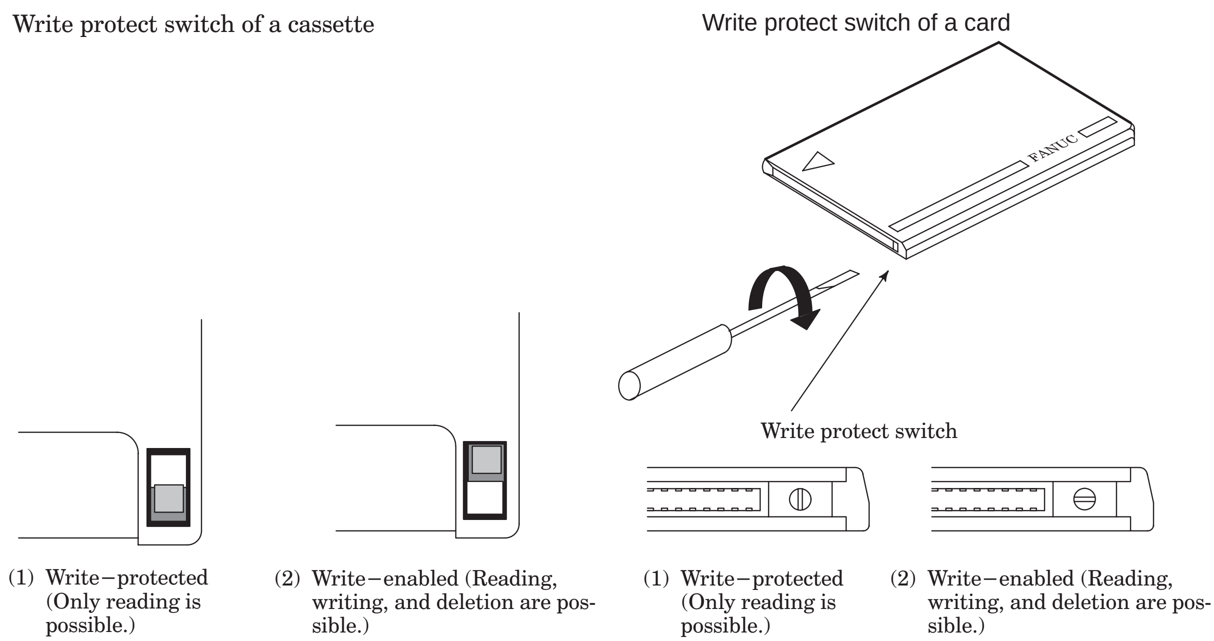


Fig. 8.1 Protect switch

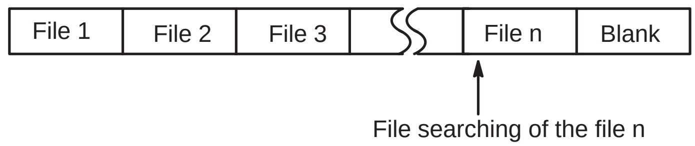
- **Writing memo**

Once written in the cassette or card, data can subsequently be read out by correspondence between the data contents and file numbers. This correspondence cannot be verified, unless the data contents and file numbers are output to the CNC and displayed. The data contents can be displayed with display function for directory of floppy disk (See Section III–8.8).
To display the contents,write the file numbers and the contents on the memo column which is the back of floppy.

(Entry example on MEMO)
File 1 NC parameters
File 2 Offset data
File 3 NC program O0100
. .
. .
. .
File (n–1) NC program O0500
File n NC program O0600

8.2 FILE SEARCH

When the program is input from the floppy, the file to be input first must be searched.
For this purpose, proceed as follows:



Procedure for File Heading

1 Press the EDIT or MEMORY switch on the machine operator's panel.

2 Press function key PROG .

3 Press soft key **[(OPRT)]**

4 Press the rightmost soft key (next–menu key).

5 Enter address N.

6 Enter the number of the file to search for.

· N0
The beginning of the cassette or card is searched.

· One of N1 to N9999
Of the file Nos. 1 to 9999, a designated file is searched.

· N–9999
The file next to that accessed just before is searched.

· N–9998
When N–9998 is designated, N–9999 is automatically inserted each time a file is input or output. This condition is reset by the designation of N1,N1 to 9999, or N－9999 or reset.

7 Press soft keys **[FSRH]** and **[EXEC]**
The specified file is searched for.

Explanation

- **File search by N–9999**

The same result is obtained both by sequentially searching the files by specifying Nos. N1 to N9999 and by first searching one of N1 to N9999 and then using the N–9999 searching method. The searching time is shorter in the latter case.

Alarm

No.	Description
86	The ready signal (DR) of an input/output device is off. An alarm is not immediately indicated in the CNC even when an alarm occurs during head searching (when a file is not found, or the like). An alarm is given when the input/output operation is performed after that. This alarm is also raised when N1 is specified for writing data to an empty floppy. (In this case, specify N0.)

8.3 FILE DELETION

Files stored on a floppy can be deleted file by file as required.

Procedure for File Deletion

1 Insert the floppy into the input/output device so that it is ready for writing.

2 Press the EDIT switch on the machine operator's panel.

3 Press function key PROG

4 Press soft key **[(OPRT)]**

5 Press the rightmost soft key ▷ (next–menu key).

6 Enter address N.

7 Enter the number (from 1 to 9999) of the file to delete.

8 Press soft key **[DELETE]**, then press soft key **[EXEC]**.
The file specified in step **7** is deleted.

Explanations

- **File number after the file is deleted**

When a file is deleted, the file numbers after the deleted file are each decremented by one. Suppose that a file numbered k was deleted. In this case, files are renumbered as follows:

Before deletion after deletion
1A(k>1) 1A(k>1)
k Deleted
(k+1)An kA(n>1)

- **Protect switch**

Set the write protect switch to the write enable state to delete the files.

8.4 PROGRAM INPUT/OUTPUT

8.4.1 Inputting a Program

This section describes how to load a program into the CNC from a floppy or NC tape.

Procedure for Inputting a Program

1 Make sure the input device is ready for reading.
For the two–path control, select the tool post for which a program to be input is used with the tool post selection switch.

2 Press the EDIT switch on the machine operator's panel.

3 When using a floppy, search for the required file according to the procedure in Section **III–8.2**.

4 Press function key PROG

5 Press soft key **[(OPRT)]**

6 Press the rightmost soft key ▷ (next–menu key).

7 After entering address O, specify a program number to be assigned to the program. When no program number is specified here, the program number used on the floppy or NC tape is assigned.

8 Press soft keys **[READ]** and **[EXEC]**
The program is input and the program number specified in step **7** is assigned to the program.

Explanations

- **Collation**

If a program is input while the data protect key on the machine operator's panel turns ON, the program loaded into the memory is verified against the contents of the floppy or NC tape.
If a mismatch is found during collation, the collation is terminated with an P/S alarm (No. 79).
If the operation above is performed with the data protection key turns OFF, collation is not performed, but programs are registered in memory.

- **Inputting multiple programs from an NC tape**

When a tape holds multiple programs, the tape is read up to ER (or %).

- **Program numbers on a NC tape**

- When a program is entered without specifying a program number.
- The O–number of the program on the NC tape is assigned to the program. If the program has no O–number, the N–number in the first block is assigned to the program.•When the program has neither an O–number nor N–number, the previous program number is incremented by one and the result is assigned to the program.
- When the program does not have an O–number but has a five–digit sequence number at the start of the program, the lower four digits of the sequence number are used as the program number. If the lower four digits are zeros, the previously registered program number is incremented by one and the result is assigned to the program.
- When a program is entered with a program number
  The O–number on the NC tape is ignored and the specified number is assigned to the program. When the program is followed by additional programs, the first additional program is given the program number. Additional program numbers are calculated by adding one to the last program.

- **Program registration in the background**

The method of registration operation is the same as the method of foreground operation. However, this operation registers a program in the background editing area. As with edit operation, the operations described below are required at the end to register a program in foreground program memory.

[(OPRT)] [BG–END]

Alarm

No.	Description
70	The size of memory is not sufficient to store the input programs
73	An attempt was made to store a program with an existing program number.
79	The verification operation found a mismatch between a program loaded into memory and the contents of the program on the floppy or NC tape.

8.4.2 Outputting a Program

A program stored in the memory of the CNC unit is output to a floppy or NC tape.

Procedure for Outputting a Program

1 Make sure the output device is ready for output.
For the two–path control, select the tool post for which a program to be output is used with the tool post selection switch.

2 To output to an NC tape, specify the punch code system (ISO or EIA) using a parameter.

3 Press the EDIT switch on the machine operator's panel.

4 Press function key PROG .

5 Press soft key **[(OPRT)]**.

6 Press the rightmost soft key ▷ (next–menu key).

7 Enter address O.

8 Enter a program number. If –9999 is entered, all programs stored in memory are output.
To output multiple programs at one time, enter a range as follows :
OΔΔΔΔ,O□□□□
Programs No.ΔΔΔΔ to No.□□□□ are output.
The program library screen displays program numbers in ascending order when bit 4 (SOR) of parameter No. 3107 is set to 1.

9 Press soft keys **[PUNCH]** and **[EXEC]**
The specified program or programs are output.

Explanations
(Output to a floppy)

- **File output location**

When output is conducted to the floppy, the program is output as the new file after the files existing in the floppy. New files are to be written from the beginning with making the old files invalid, use the above output operation after the N0 head searching.

- **An alarm while a program is output**

When P/S alarm (No.086) occurs during program output, the floppy is restored to the condition before the output.

- **Outputting a program after file heading**

When program output is conducted after N1 to N9999 head searching, the new file is output as the designated n–th position. In this case, 1 to n–1 files are effective, but the files after the old n–th one are deleted. If an alarm occurs during output, only the 1 to n–1 files are restored.

- **Efficient use of memory**

To efficiently use the memory in the cassette or card, output the program by setting parameter NFD (No. 0101#7,No. 0111#7 or 0121#7) to 1. This parameter makes the feed is not output, utilizing the memory efficiently.

- **On the memo record**

Head searching with a file No. is necessary when a file output from the CNC to the floppy is again input to the CNC memory or compared with the content of the CNC memory. Therefore, immediately after a file is output from the CNC to the floppy, record the file No. on the memo.

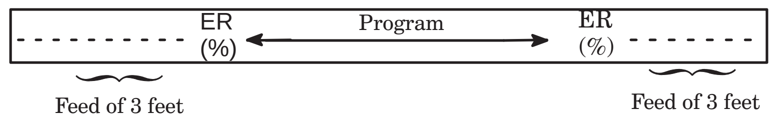
● Punching programs in the background

Punch operation can be performed in the same way as in the foreground. This function alone can punch out a program selected for foreground operation.
<O> (Program No.) **[PUNCH] [EXEC]**: Punches out a specified program.
<O> H–9999I **[PUNCH] [EXEC]**: Punches out all programs.

Explanations (Output to an NC tape)

● Format

A program is output to paper tape in the following format:



If three–feet feeding is too long, press the CAN key during feed punching to cancel the subsequent feed punching.

● TV check

A space code for TV check is automatically punched.

● ISO code

When a program is punched in ISO code, two CR codes are punched after an LF code.

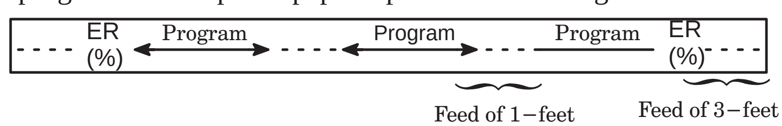
- - - - - - - - - - - - - - - LF CR CR

● Stopping the punch

Press the RESET key to stop punch operation.

● Punching all programs

All programs are output to paper tape in the following format.



The sequence of the programs punched is undefined.

8.5 OFFSET DATA INPUT AND OUTPUT

8.5.1 Inputting Offset Data

Offset data is loaded into the memory of the CNC from a floppy or NC tape. The input format is the same as for offset value output. See section **III–8.5.2.** When an offset value is loaded which has the same offset number as an offset number already registered in the memory, the loaded offset data replaces existing data.

Procedure for Inputting Offset Data

1 Make sure the input device is ready for reading
For the two–path control, select the tool post for which offset data to be input is used with the tool post selection switch.

2 Press the EDIT switch on the machine operator's panel.

3 When using a floppy, search for the required file according to the procedure in Section III–8.2.

4 Press function key OFFSET SETTING to display tool offset screeen.

5 Press soft keys **[(OPRT)]**.

6 Press rightmost soft key ▷ (next menu key).

7 Press soft keys **[READ]** and **[EXEC]**.

8 The input offset data will be displayed on the screen after completion of input operation.

8.5.2 Outputting Offset Data

All offset data is output in a output format from the memory of the CNC to a floppy or NC tape.

Procedure for Outputting Offset Data

1 Make sure the output device is ready for output.
For the two–path control, select the tool post for which offset data to be output is used with the tool post selection switch.

2 Specify the punch code system (ISO or EIA) using a parameter.

3 Press the EDIT switch on the machine operator's panel.

4 Press function key OFFSET SETTING to display tool offset screen.

5 Press soft key **[(OPRT)]**.

6 Press the rightmost soft key ▷ (next–menu key)

7 Press soft keys **[PUNCH]** and **[EXEC]**.
Offset data is output in the output format described below.

Explanations

- **Output format**

Output format is as follows:

Format

G10P_X_Y_Z_R_Q_;
P: Offset number
. . . . Work sheet
:P=0
. . . . For wear offset amount
:P=Wear offset number
. . . . For geometry offset amount
:p=10000+geometry offset number
X:Offset value on X axis
Y: Offset value on Y axis
Z:Offset value on Z axis
Q:Imaginary tool nose number
R:Tool nose radius offset value

The L1 command may be used instead of L11 for format compatibility of the conventional CNC.

- **Output file name**

When the floppy disk directory display function is used, the name of the output file is OFFSET.

8.6 INPUTTING AND OUTPUTTING PARAMETERS AND PITCH ERROR COMPENSATION DATA

Parameters and pitch error compensation data are input and output from different screens, respectively. This chapter describes how to enter them.

8.6.1 Inputting Parameters

Parameters are loaded into the memory of the CNC unit from a floppy or NC tape. The input format is the same as the output format. See Subsec. **III–8.6.2**. When a parameter is loaded which has the same data number as a parameter already registered in the memory, the loaded parameter replaces the existing parameter.

Procedure for Inputting Parameters

1 Make sure the input device is ready for reading.
For the two–path control, select the tool post for which parameters to be input are used with the tool post selection switch.

2 When using a floppy, search for the required file according to the procedure in Section **III–8.2**.

3 Press the EMERGENCY STOP button on the machine operator's panel.

4 Press function key OFFSET SETTING .

5 Press the soft key **[SETING]** for chapter selection.

6 Enter 1 in response to the prompt for "PARAMETER WRITE (PWE)" in setting data. P/S alarm (No.100(indicating that parameters can be written)) appears.

7 Press soft key SYSTEM .

8 Press chapter selection soft key **[PARAM]**.

9 Press soft key **[(OPRT)]**.

10 Press the rightmost soft key ▷ (next–menu key).

11 Press soft keys **[READ]** and **[EXEC]**.
Parameters are read into memory. Upon completion of input, the "INPUT" indicator at the lower–right corner of the screen disappears.

12 Press function key OFFSET SETTING .

13 Press soft key **[SETING]** for chapter selection.

14 Enter 0 in response to the prompt for "PARAMETER WRITE (PWE)" in setting data.

15 Turn the power to the NC back on.

16 Release the EMERGENCY STOP button on the machine operator's panel.

8.6.2 Outputting Parameters

All parameters are output in the defined format from the memory of the CNC to a floppy or NC tape.

Procedure for Outputting Parameters

1 Make sure the output device is ready for output.
For the two–path control, select the tool post for which parameters to be input are used with the tool post selection switch.

2 Specify the punch code system (ISO or EIA) using a parameter.

3 Press the EDIT switch on the machine operator's panel.

4 Press function key SYSTEM .

5 Press chapter selection soft key **[PARAM]** to display the parameter screen.

6 Press soft key **[(OPRT)]**.

7 Press rightmost soft key ▷ (next–menu key).

8 Press soft keys **[PUNCH]** and **[EXEC]**.
All parameters are output in the defined format.

Explanations

- **Output format**

Output format is as follows:
N . P . . ;
N . A1P . . . A2P . . . AnP . . . ;
N . P . . ;

N:Parameter No.
A:Axis No.(n is the number of control axis)
P:Parameter setting value .

- **Output file name**

When the floppy disk directory display function is used, the name of the output file is PARAMETER.

8.6.3 Inputting Pitch Error Compensation Data

Pitch error compensation data are loaded into the memory of the CNC from a floppy or NC tape. The input format is the same as the output format. See Section **8.6.4.** When a pitch error compensation data is loaded which has the corresponding data number as a pitch error compensation data already registered in the memory, the loaded data replaces the existing data.

Procedure for Pitch Error Compensation Data

1 Make sure the input device is ready for reading.
For the two–path control, select the tool post for which pitch error compensation data to be input is used with the tool post selection switch.

2 When using a floppy, search for the required file according to the procedure in Section **III–8.2**.

3 Press the EMERGENCY STOP button on the machine operator's panel.

4 Press function key OFFSET SETTING .

5 Press the soft key **[SETING]** for chapter selection.

6 Enter 1 in response to the prompt for "PARAMETER WRITE (PWE)" in setting data. P/S alarm (No. 100 (indicating that parameters can be written)) appears.

7 Press soft key SYSTEM .

8 Press the rightmost soft key ▷ (next–menu key)and press chapter selection soft key **[PITCH]**.

9 Press soft key **[(OPRT)]**.

10 Press the rightmost soft key ▷ (next–menu key).

11 Press soft keys **[READ]** and **[EXEC]**.
Pitch error compensation data are read into memory. Upon completion of input, the "INPUT" indicator at the lower–right corner of the screen disappears.

12 Press function key OFFSET SETTING .

13 Press soft key **[SETING]** for chapter selection.

14 Enter 0 in response to the prompt for "PARAMETER WRITE (PWE)" in setting data.

15 Turn the power to the NC back on.

16 Release the EMERGENCY STOP button on the machine operator's panel.

Explanations

- **Pitch error compensation**

Parameters 3620 to 3624 and pitch error compensation data must be set correctly to apply pitch error compensation correctly
(See subsec. **III–11.5.2**)

8.6.4 Outputting Pitch Error Compensation Data

All pitch error compensation data are output in the defined format from the memory of the CNC to a floppy or NC tape.

Procedure for Outputting Pitch Error Compensation Data

1 Make sure the output device is ready for output.
For the two–path control, select the tool post for which pitch error compensation data to be output is used with the tool post selection switch.

2 Specify the punch code system (ISO or EIA) using a parameter.

3 Press the EDIT switch on the machine operator's panel.

4 Press function key SYSTEM .

5 Press the rightmost soft key ▷ (next–menu key) and press chapter selection soft key **[PITCH]**.

6 Press soft key **[(OPRT)]**.

7 Press rightmost soft key ▷ (next–menu key).

8 Press soft keys **[PUNCH]** and **[EXEC]**.
All pitch error compensation data are output in the defined format.

Explanations

- **Output format**

Output format is as follows:
N 10000 P ;
N 11023 P ;
N:Pitch error compensation point No. +10000
P:Pitch error compensation data

- **Output file name**

When the floppy disk directory display function is used, the name of the output file is "**PITCH ERROR**".

8.7 INPUTTING/OUTPUTT–ING CUSTOM MACRO COMMON VARIABLES

8.7.1 Inputting Custom Macro Common Variables

The value of a custom macro common variable (#500 to #999) is loaded into the memory of the CNC from a floppy or NC tape. The same format used to output custom macro common variables is used for input. See Subsec. **8.7.2.** For a custom macro common variable to be valid, the input data must be executed by pressing the cycle start button after data is input. When the value of a common variable is loaded into memory, this value replaces the value of the same common variable already existing (if any) in memory.

Procedure for Inputting Custom Macro Common Variables

1 Register the program which has been output, as described in Section III–8.7.2, in memory according to the program input procedure described in Section III–8.4.2.

2 Press the MEMORY switch on the machine operator's panel upon completing input.

3 Press the cycle start button to execute the loaded program.

4 Display the macro variable screen to check whether the values of the common variables have been set correctly.

Display of the macro variable screen

·Press function key OFFSET SETTING .

· Press the rightmost soft key (next–menu key).

·Press soft key **[MACRO]**.

·Select a variable with the page keys or numeric keys and soft key **[NO.SRH]**.

Explanations

- **Common variables**

The common variables (#500 to #531) can be input and output.
When the option for adding a common variable is specified, values from #500 to #999 can be input and output.
#100 to #199 can be input and output when bit 3 (PU5) of parameter No. 6001 is set to 1.

8.7.2 Outputting Custom Macro Common Variable

Custom macro common variables (#500 to #999) stored in the memory of the CNC can be output in the defined format to a floppy or NC tape.

Procedure for Outputting Custom Macro Common Variable

1 Make sure the output device is ready for output.

2 Specify the punch code system (ISO or EIA) using a parameter.

3 Press the EDIT switch on the machine operator's panel.

4 Press function key OFFSET SETTING .

5 Press the rightmost soft key ▷ (next–menu key), then press soft key **[MACRO]**.

6 Press soft key **[(OPRT)]**.

7 Press the rightmost soft key ▷ (next–menu key).

8 Press soft keys **[PUNCH]** and **[EXEC]**.
Common variables are output in the defined format.

Explanations

- **Output format**

The output format is as follows:

```
%
;
#500=[25283*65536+65536]/134217728   (1)
#501=#0;  (2)
#502=0; (3)
#503=;
.   ;
.   ;
#531=;
M02;
%
```

(1) The precision of a variable is maintained by outputting the value of the variable as <expression>.

(2) Undefined variable

(3) When the value of a variable is 0

- **Output file name**

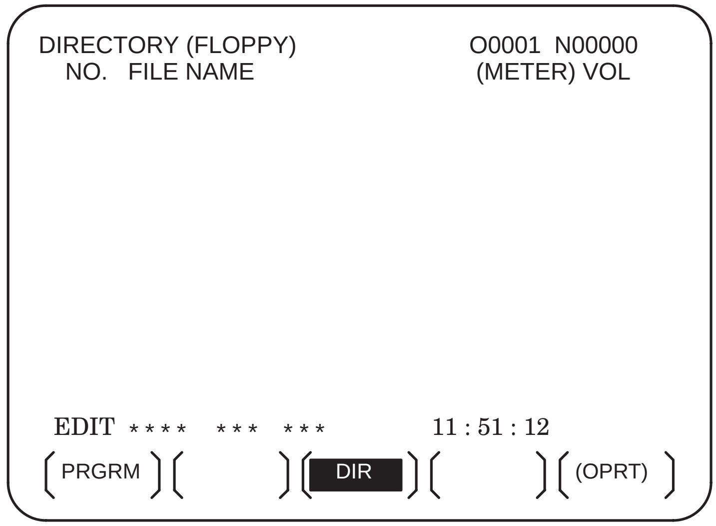
When the floppy disk directory display function is used, the name of the output file is **"MACRO VAR"**.

- **Common variable**

The common variables (#500 to #531) can be input and output.
When the option for adding a common variable is specified, values from #500 to #999 can be input and output.
#100 to #199 can be input and output when bit 3 (PU5) of parameter No. 6001 is set to 1.

8.8 DISPLAYING DIRECTORY OF FLOPPY DISK

On the floppy directory display screen, a directory of the FANUC Handy File, FANUC Floppy Cassette, or FANUC FA Card files can be displayed. In addition, those files can be loaded, output, and deleted.



8.8.1 Displaying the Directory

Displaying the Directory of Floppy Disk Files

Procedure 1

Use the following procedure to display a directory of all the files stored in a floppy:

1 Press the EDIT switch on the machine operator’s panel.

2 Press function key PROG .

3 Press the rightmost soft key ▷ (next–menu key).

4 Press soft key **[FROPPY]**.

5 Press page key PAGE ↑ or PAGE ↓ .

6 The screen below appears.

```
DIRECTORY (FLOPPY)                              O0001  N00000
  NO. FILE NAME                                  (METER) VOL
  0001  PARAMETER                                    58.5
  0002  O0001                                         1.9
  0003  O0002                                         1.9
  0004  O0010                                         1.3
  0005  O0040                                         1.3
  0006  O0050                                         1.9
  0007  O0100                                         1.9
  0008  O1000                                         1.9
  0009  O9500                                         1.6



  EDIT  ****   ***   ***                 11 : 53 : 04
( F SRH )( READ )( PUNCH )( DELETE )(        )
```

Fig.8.8.1 (a)

7 Press a page key again to display another page of the directory.

Procedure 2

Use the following procedure to display a directory of files starting with a specified file number :

1 Press the EDIT switch on the machine operator's panel.

2 Press function key PROG .

3 Press the rightmost soft key ▷ (next–menu key).

4 Press soft key **[FLOPPY]**.

5 Press soft key **[(OPRT)].**

6 Press soft key **[F SRH].**

7 Enter a file number.

8 Press soft keys **[F SET]** and **[EXEC]**.

9 Press a page key to display another page of the directory.

10 Press soft key **[CAN]** to return to the soft key display shown in the screen of **Fig. 8.8.1(a).**

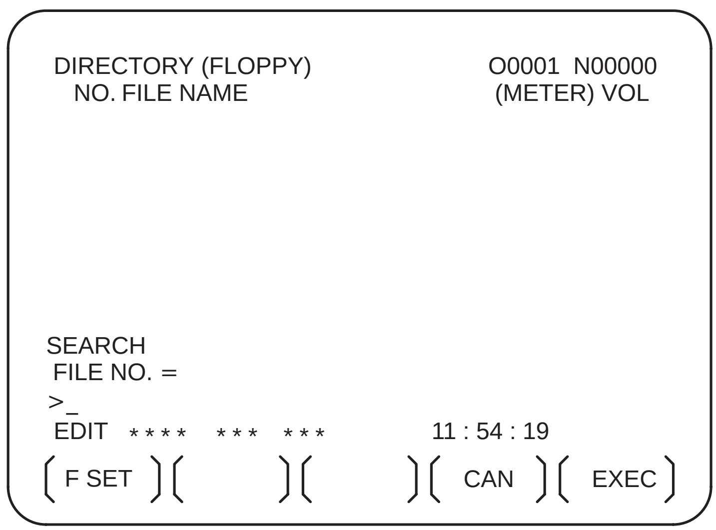


Fig.8.8.1 (a)

Explanations

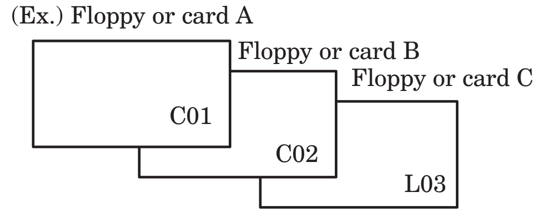
- **Screen fields and their meanings**

NO :Displays the file number

FILE NAME :Displays the file name.

(METER) :Converts and prints out the file capacity to paper tape length. You can also produce H (FEET)I by setting the INPUT UNIT to INCH of the setting data.

VOL. :When the file is multi–volume, that state is displayed.



C(number)means CONTINUE
L(number)means LAST
number number of floppies or cards

8.8.2 Reading Files

The contents of the specified file number are read to the memory of NC.

Procedure for Reading Files

1 Press the EDIT switch on the machine operator's panel.
For the two–path control, select the tool post for which a file is to be input in memory with the tool post selection switch.

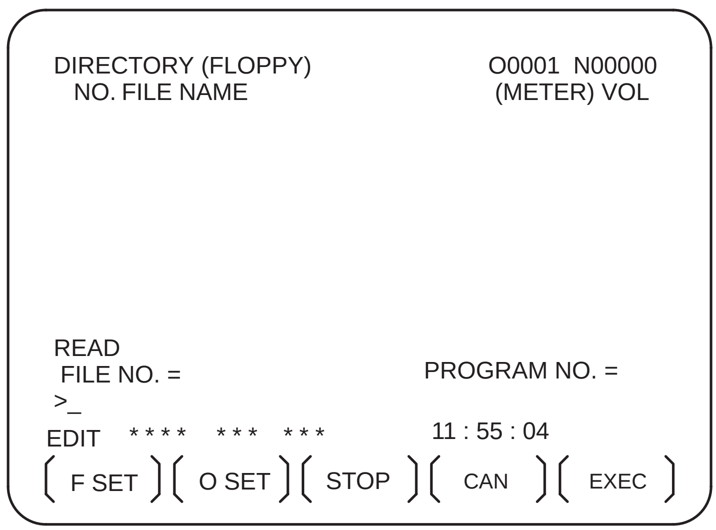
2 Press function key PROG .

3 Press the rightmost soft key ▷ (next–menu key).

4 Press soft key **[FLOPPY]**.

5 Press soft key **[(OPRT)]**.

6 Press soft key **[READ]**.



7 Enter a file number.

8 Press soft key **[F SET]**.

9 To modify the program number, enter the program number, then press soft key **[O SET]**.

10 Press soft key **[EXEC]**. The file number indicated in the lower–left corner of the screen is automatically incremented by one.

11 Press soft key **[CAN]** to return to the soft key display shown in the screen of **Fig. 8.8.1.(a)**.

8.8.3 Outputting Programs

Any program in the memory of the CNC unit can be output to a floppy as a file.

Procedure for Outputting Programs

1 Press the EDIT switch on the machine operator's panel.
For the two–path control, select the tool post for which a program is to be output from floppy with the tool post selection switch.

2 Press function key PROG .

3 Press the rightmost soft key ▷ (next–menu key).

4 Press soft key **[FLOPPY].**

5 Press soft key **[(OPRT)].**

6 Press soft key **[PUNCH].**

```
DIRECTORY (FLOPPY)                          O0002  N01000
  NO. FILE NAME                              (METER) VOL









PUNCH
 FILE NO. =                          PROGRAM NO. =
>_
EDIT  ****   ***   ***              11 : 55 : 26
( F SET )( O SET )( STOP )( CAN )( EXEC )
```

7 Enter a program number. To write all programs into a single file, enter –9999 in the program number field. In this case, the file name **"ALL.PROGRAM"** is registered.

8 Press soft key **[O SET]**.

9 Press soft key **[EXEC]**. The program or programs specified in step **7** are written after the last file on the floppy. To output the program after deleting files starting with an existing file number, key in the file number, then press soft key **[F SET]** followed by soft key **[EXEC]**.

10 Press soft key **[CAN]** to return to the soft key display shown in the screen of **Fig.8.8.1(a)**.

8.8.4 Deleting Files

The file with the specified file number is deleted.

Procedure for Deleting Files

1 Press the EDIT switch on the machine operator's panel.

2 Press function key PROG .

3 Press the rightmost soft key ▷ (next–menu key).

4 Press soft key **[FLOPPY].**

5 Press soft key **[(OPRT)].**

6 Press soft key **[DELETE].**

```
DIRECTORY (FLOPPY)                        O0001  N00000
  NO. FILE NAME                            (METER) VOL






DELETE
 FILE NO. =        NAME=
>_
EDIT  ****  ***  ***              11 : 55 : 51
( F SET  )( F NAME )(        )(  CAN  )(  EXEC  )
```

7 Specify the file to be deleted.
When specifying the file with a file number, type the number and press soft key **[F SET]**.When specifying the file with a file name, type the name and press soft key **[F NAME]**.

8 Press soft key **[EXEC]**.
The file specified in the file number field is deleted. When a file is deleted, the file numbers after the deleted file are each decremented by one.

9 Press soft key **[CAN]** to return to the soft key display shown in the screen of **Fig. 8.8.1(a)**.

Limitations

- **Inputting file numbers and program numbers with keys**

If **[F SET]** or **[O SET]** is pressed without key inputting file number and program number, file number or program number shows blank. When 0 is entered for file numbers or program numbers, 1 is displayed.

- **I/O devices**

To use channel 0 ,set a device number in parameter 102.
Set the I/O device number to parameter No. 0112 when channel 1 is used. Set it to No. 0122 when channel 2 is used.

- **Significant digits**

For the numeral input in the data input area with FILE NO. and PROGRAM NO., only lower 4 digits become valid.

- **Collation**

When the data protection key on the machine operator's panel is ON, no programs are read from the floppy. They are verified against the contents of the memory of the CNC instead.

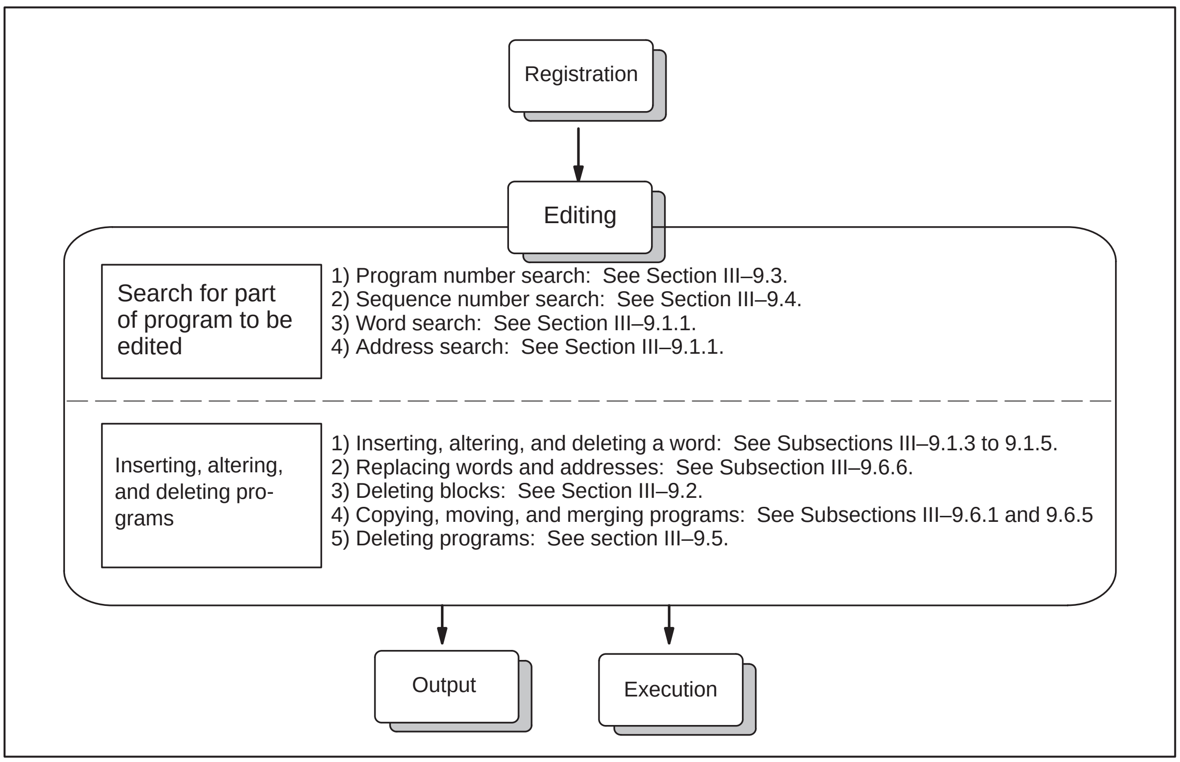
ALARM

No.	Contents
71	An invalid file number or program number was entered. (Specified program number is not found.)
79	Verification operation found a mismatch between a program loaded into memory and the contents of the floppy
86	The datas et–ready signal (DR) for the input/output device is turned off. (The no file error or duplicate file error occurred on the input/output device because an invalid file number, program number, or file name was entered.

9 EDITING PROGRAMS

General

This chapter describes how to edit programs registered in the CNC. Editing includes the insertion, modification, deletion, and replacement of words. Editing also includes deletion of the entire program and automatic insertion of sequence numbers. The extended part program editing function can copy, move, and merge programs. This chapter also describes program number search, sequence number search, word search, and address search, which are performed before editing the program.



9.1 INSERTING ,ALTERING AND DELETING A WORD

This section outlines the procedure for inserting, modifying, and deleting a word in a program registered in memory.

Procedure for inserting, altering and deleting a word

1 Select **EDIT** mode.

2 Press PROG .

3 Select a program to be edited.
If a program to be edited is selected, perform the operation 4.
If a program to be edited is not selected, search for the program number.

4 Search for a word to be modified.
·Scan method
·Word search method

5 Perform an operation such as altering, inserting, or deleting a word.

Explanation

- **Concept of word and editing unit**

A word is an address followed by a number. With a custom macro, the concept of word is ambiguous.
So the editing unit is considered here.
The editing unit is a unit subject to alteration or deletion in one operation.
In one scan operation, the cursor indicates the start of an editing unit.
An insertion is made after an editing unit.
Definition of editing unit
(i) Program portion from an address to immediately before the next address
(ii) An address is an alphabet, **IF**, **WHILE**, **GOTO**, **END**, **DO=**,or **; (EOB)**.
According to this definition, a word is an editing unit.
The word "word," when used in the description of editing, means an editing unit according to the precise definition.

Notes

The user cannot continue program execution after altering, inserting, or deleting data of the program by suspending machining in progress by means of an operation such as a single block stop or feed hold operation during program execution. If such a modification is made, the program may not be executed exactly according to the contents of the program displayed on the screen after machining is resumed. So, when the contents of memory are to be modified by part program editing, be sure to enter the reset state or reset the system upon completion of editing before executing the program.

9.1.1 Word Search

A word can be searched for by merely moving the cursor through the text (scanning), by word search, or by address search.

Procedure for scanning a program

1 Press the cursor key [→]

The cursor moves forward word by word on the screen; the cursor is displayed at a selected word.

2 Press the cursor key [←]

The cursor moves backward word by word on the screen; the cursor is displayed at a selected word.

Example) When Z1250.0 is scanned

```
Program                                   O0050  N01234
 O0050  ;
 N01234  X100.0 [redacted]  ;
 S12  ;
 N56789  M03  ;
 M02  ;
 %
```

3 Holding down the cursor key [→] or [←] scans words continuously.

4 The first word of the next block is searched for when the cursor key [↓] is pressed.

5 The first word of the previous block is searched for when the cursor key [↑] is pressed.

6 Holding down the cursor key [↓] or [↑] moves the cursor to the head of a block continuously.

7 Pressing the page key [PAGE ↓] displays the next page and searches for the first word of the page.

8 Pressing the page key [↑ PAGE] displays the previous page and searches for the first word of the page.

9 Holding down the page key [PAGE ↓] or [↑ PAGE] displays one page after another.

Procedure for searching a word

Example) of Searching for S12

```
PROGRAM                    O0050  N01234
 O0050 ;
 N01234  X100.0  Z1250.0 ;
 S12 ;
 N56789  M03 ;
 M02 ;
 %
```

N01234 is being searched for/ scanned currently.

S12 is searched for.

1 Key in address S .

2 Key in 1 2 .

- S12 cannot be searched for if only S1 is keyed in.
- S09 cannot be searched for by keying in only S9.
  To search for S09, be sure to key in S09.

3 Pressing the **[SRH↓]** key starts search operation.
Upon completion of search operation, the cursor is displayed at S12.
Pressing the **[SRH↑]** key rather than the **[SRH↓]** key performs search operation in the reverse direction.

Procedure for searching an address

Example) of Searching for M03

```
PROGRAM                    O0050  N01234
 O0050 ;
 N01234  X100.0  Z1250.0 ;
 S12 ;
 N56789  M03 ;
 M02 ;
 %
```

N01234 is being searched for/ scanned currently.

M03 is searched for.

1 Key in address M .

2 Press the **[SRH↓]** key.
Upon completion of search operation, the cursor is displayed at M03.
Pressing the **[SRH↑]** key rather than the **[SRH↓]** key performs search operation in the reverse direction.

Alarm

Alarm number	Description
71	The word or address being searched for was not found.

9.1.2 Heading a Program

The cursor can be jumped to the top of a program. This function is called heading the program pointer. This section describes the three methods for heading the program pointer.

Procedure for Heading a Program

Method 1

1 Press RESET when the program screen is selected in EDIT mode. When the cursor has returned to the start of the program, the contents of the program are displayed from its start on the screen.

Method 2

Search for the program number.

1 Press address O ,when a program screen is selected in the **MEMORY** or **EDIT** mode.

2 Input a program number.

3 Press the soft key **[O SRH]**.

Method 3

1 Select **MEMORY** or **EDIT** mode.

2 Press PROG .

3 Press the **[(OPRT)]** key.

4 Press the **[REWIND]** key.

9.1.3 Inserting a Word

Procedure for inserting a word

1. Search for or scan the word immediately before a word to be inserted.
2. Key in an address to be inserted.
3. Key in data.
4. Press the INSERT key.

Example of Inserting T15

Procedure

1. Search for or scan Z1250.

```
Program                                  O0050  N01234
 O0050  ;
 N01234  X100.0  Z1250.0  ;
 S12  ;
 N56789  M03  ;
 M02  ;
 %
```

Z1250.0 is searched for/ scanned.

2. Key in T 1 5 .
3. Press the INSERT key.

```
Program                                  O0050  N01234
 O0050  ;
 N01234  X100.0  Z1250.0  T15  ;
 S12  ;
 N56789  M03  ;
 M02  ;
 %
```

T15 is inserted.

9.1.4 Altering a Word

Procedure for altering a word

1. Search for or scan a word to be altered.
2. Key in an address to be inserted.
3. Key in data.
4. Press the ALTER key.

Example of changing T15 to M15

Procedure

1. Search for or scan T15.

```
Program                                  O0050  N01234
 O0050  ;
 N01234  X100.0  Z1250.0 T15  ;   <----  T15 is searched
 S12  ;                                  for/scanned.
 N56789  M03  ;
 M02  ;
 %
```

2. Key in M 1 5 .

3. Press the ALTER key.

```
Program                                  O0050  N01234
 O0050  ;
 N1234  X100.0  Z1250.0  M15  ;   <----  T15 is changed to
 S12  ;                                  M15.
 N5678  M03  ;
 M02  ;
 %
```

9.1.5 Deleting a Word

Procedure for deleting a word

1 Search for or scan a word to be deleted.

2 Press the DELETE key.

Example of deleting X100.0

Procedure

1 Search for or scan X100.0.

```
Program                        O0050  N01234
 O0050  ;
 N01234  X100.0   Z1250.0  M15  ;
 S12  ;
 N56789  M03  ;
 M02  ;
 %
```

X100.0 is searched for/ scanned.

2 Press the DELETE key.

```
Program                        O0050  N01234
 O0050  ;
 N01234  Z1250.0  M15  ;
 S12  ;
 N56789  M03  ;
 M02  ;
 %
```

X100.0 is deleted.

9.2 DELETING BLOCKS

A block or blocks can be deleted in a program.

9.2.1 Deleting a Block

The procedure below deletes a block up to its EOB code; the cursor advances to the address of the next word.

Procedure for deleting a block

1 Search for or scan address N for a block to be deleted.

2 Key in [EOB].

3 Press the [DELETE].

Example of deleting a block of No.1234

Procedure

1 Search for or scan N01234.

```
Program                              O0050  N01234
 O0050  ;
 N01234  Z1250.0  M15  ;
 S12  ;
 N56789  M03  ;
 M02  ;
 %
```

N01234 is searched for/ scanned.

2 Key in [EOB].

3 Press the [DELETE] key.

```
Program                              O0050  N01234
 O0050  ;
 S12  ;
 N56789  M03  ;
 M02  ;
 %
```

Block containing N01234 has been deleted.

9.2.2 Deleting Multiple Blocks

The blocks from the currently displayed word to the block with a specified sequence number can be deleted.

Procedure for deleting multiple blocks

1 Search for or scan a word in the first block of a portion to be deleted.

2 Key in address N .

3 Key in the sequence number for the last block of the portion to be deleted.

4 Press the DELETE key.

Example of deleting blocks from a block containing N01234 to a block containing N56789

Procedure

1 Search for or scan N01234.

```
Program                                   O0050  N01234
 O0050  ;
 N01234  Z1250.0  M15  ;
 S12  ;
 N56789  M03  ;
 M02  ;
 %
```

N01234 is searched for/ scanned.

2 . Key in N 5 6 7 8 9 .

```
Program                                   O0050  N01234
 O0050  ;
 N01234  Z1250.0  M15  ;
 S12  ;
 N56789  M03  ;
 M02  ;
 %
```

Underlined part is deleted.

3 Press the DELETE key.

```
Program                                   O0050  N01234
 O0050  ;
 M02  ;
 %
```

Blocks from block containing N01234 to block containing N56789 have been deleted.

9.3 PROGRAM NUMBER SEARCH

When memory holds multiple programs, a program can be searched for. There are three methods as follows.

Procedure for program number search

Method 1

1 Select **EDIT** or **MEMORY** mode.

2 Press PROG to display the program screen.

3 Key in address O .

4 Key in a program number to be searched for.

5 Press the **[O SRH]** key.

6 Upon completion of search operation, the program number searched for is displayed in the upper–right corner of the CRT screen
If the program is not found , P/S alarm No. 71 occurs.

Method 2

1 Select **EDIT** or **MEMORY** mode.

2 Press PROG to display the program screen.

3 Press the **[O SRH]** key.
In this case, the next program in the directory is searched for .

Method 3

This method searches for the program number (0001 to 0015) corresponding to a signal on the machine tool side to start automatic operation. Refer to the relevant manual prepared by the machine tool builder for detailed information on operation.

1 Select **MEMORY** mode.

2 Set the reset state(*1)
·The reset state is the state where the LED for indicating that automatic operation is in progress is off. (Refer to the relevant manual of the machine tool builder.)

3 Set the program number selection signal on the machine tool side to a number from 01 to 15.
·If the program corresponding to a signal on the machine tool side is not registered, P/S alarm (No. 59) is raised.

4 Press the cycle start button.
·When the signal on the machine tool side represents 00, program number search operation is not performed.

Alarm

Alarm No.	Contents
59	The program with the selected number cannot be searched during external program number search.
71	The specified program number was not found during program number search.

9.4 SEQUENCE NUMBER SEARCH

Sequence number search operation is usually used to search for a sequence number in the middle of a program so that execution can be started or restarted at the block of the sequence number.

Example) Sequence number 02346 in a program (O0002) is searched for.

```
                           Program
                           O0001 ;
                           N01234 X100.0 Z100.0 ;
                           S12 ;
                             :
Selected program  →        O0002 ;                    This section is
                           N02345 X20.0 Z20.0 ;       searched starting at
Target sequence   →        N02346 X10.0 Z10.0 ;       the beginning.
number is found.             :                        (Search operation is
                           O0003 ;                    performed only within a
                             :                        program.)
```

Procedure for sequence number search

1 Select **MEMORY** mode.

2 Press PROG .

3 ·If the program contains a sequence number to be searched for, perform the operations 4 to 7 below.
·If the program does not contain a sequence number to be searched for,select the program number of the program that contains the sequence number to be searched for.

4 Key in address N .

5 Key in a sequence number to be searched for.

6 Press the **[N SRH]** key.

7 Upon completion of search operation, the sequence number searched for is displayed in the upper–right corner of the CRT screen.
If the specified sequence number is not found in the program currently selected, P/S alarm (No. 60) occurs.

Explanations

• Operation during Search

Those blocks that are skipped do not affect the CNC. This means that the data in the skipped blocks such as coordinates and M, S, and T codes does not alter the CNC coordinates and modal values.
So, in the first block where execution is to be started or restarted by using a sequence number search command, be sure to enter required M, S, and T codes and coordinates. A block searched for by sequence number search usually represents a point of shifting from one process to another. When a block in the middle of a process must be searched for to restart execution at the block, specify M, S, and T codes, G codes, coordinates, and so forth as required from the MDI after closely checking the machine tool and CNC states at that point.

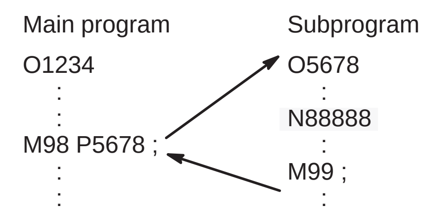
• Checking during search

During search operation, the following checks are made:
·Optional block skip
·P/S alarm (No. 003 to 010)

Restrictions

• Searching in sub–program

During sequence number search operation, M98Pxxxx (subprogram call) is not executed. So an P/S alarm (No.060) is raised if an attempt is made to search for a sequence number in a subprogram called by the program currently selected.



If an attempt is made to search for N8888 in the example above, an alarm is raised.

Alarm

Alarm No.	Contents
60	Command sequence number was not found in the sequence number search.

9.5 DELETING PROGRAMS

Programs registered in memory can be deleted,either one program by one program or all at once. Also, More than one program can be deleted by specifying a range.

9.5.1 Deleting One Program

A program registered in memory can be deleted.

Procedure for deleting one program

1 Select the **EDIT** mode.

2 Press PROG to display the program screen.

3 Key in address O .

4 Key in a desired program number.

5 Press the DELETE key.
The program with the entered program number is deleted.

9.5.2 Deleting All Programs

All programs registered in memory can be deleted.

Procedure for deleting all programs

1 Select the **EDIT** mode.

2 Press PROG to display the program screen.

3 Key in address O .

4 Key in –9999.

5 Press edit key DELETE to delete all programs.

9.5.3 Deleting More Than One Program by Specifying a Range

Programs within a specified range in memory are deleted.

Procedure for deleting more than one program by specifying a range

1 Select the **EDIT** mode.

2 Press PROG to display the program screen.

3 Enter the range of program numbers to be deleted with address and numeric keys in the following format:
OXXXX,OYYYY
where XXXX is the starting number of the programs to be deleted and YYYY is the ending number of the programs to be deleted.

4 Press edit key DELETE to delete programs No. XXXX to No. YYYY.

9.6 EXTENDED PART PROGRAM EDITING FUNCTION

With the extended part program editing function, the operations described below can be performed using soft keys for programs that have been registered in memory.

Following editing operations are available :

·All or part of a program can be copied or moved to another program.

·One program can be merged at free position into other programs.

·A specified word or address in a program can be replaced with another word or address.

9.6.1 Copying an Entire Program

A new program can be created by copying a program.

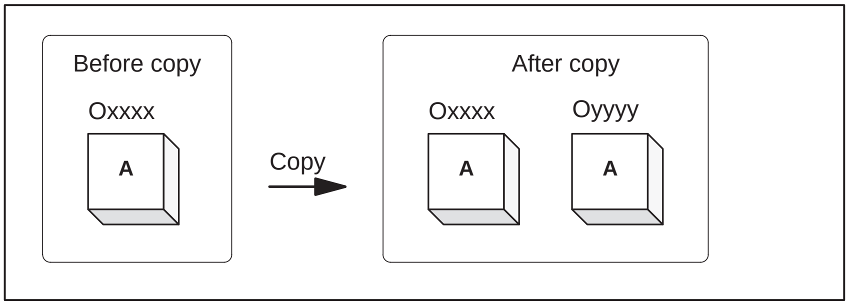


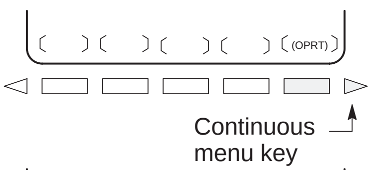
Fig. 9.6.1 Copying an Entire Program

In Fig. 9.6.1, the program with program number xxxx is copied to a newly created program with program number yyyy. The program created by copy operation is the same as the original program except the program number.

Procedure of copying an entire program

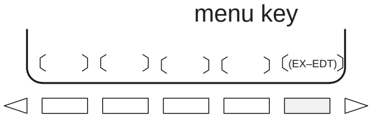
1 Enter the **EDIT** mode.

2 Press function key PROG .

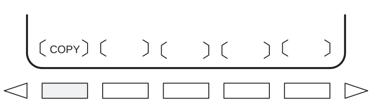


3 Press soft key **[(OPRT)]**.

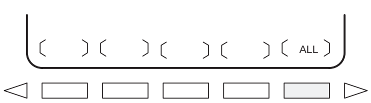
4 Press the continuous menu key.



5 Press soft key **[EX–EDT]**.



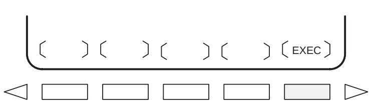
6 Check that the screen for the program to be copied is selected and press soft key **[COPY]**.



7 Press soft key **[ALL]**.

Numeric keys 0 to 9

8 Enter the number of the new program (with only numeric keys) and press the INPUT key.



9 Press soft key **[EXEC]**.

9.6.2 Copying Part of a Program

A new program can be created by copying part of a program.

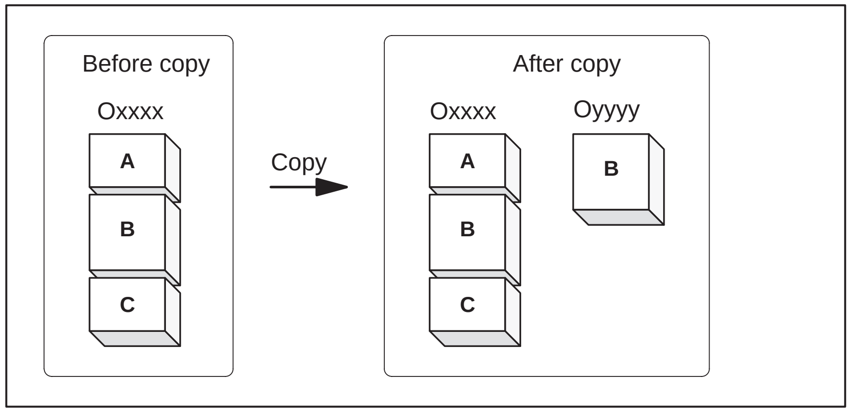
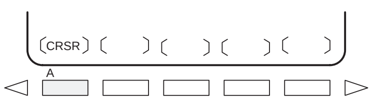


Fig. 9.6.2 Copying Part of a Program

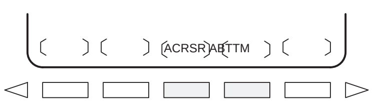
In Fig. 9.6.2, part B of the program with program number xxxx is copied to a newly created program with program number yyyy. The program for which an editing range is specified remains unchanged after copy operation.

Procedure for copying part of a program

1 Perform steps **1** to **6** in subsection III–9.6.1.



2 Move the cursor to the start of the range to be copied and press soft key **[CRSR~]**.

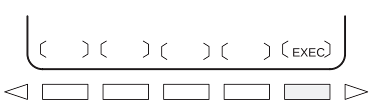


3 Move the cursor to the end of the range to be copied and press soft key **[~CRSR]** or **[~BTTM]** (in the latter case, the range to the end of the program is copied regardless of the position of the cursor).

Numeric keys 0 to 9

4 Enter the number of the new program (with only numeric keys) and press the INPUT key.

5 Press soft key **[EXEC]**.



9.6.3 Moving Part of a Program

A new program can be created by moving part of a program.

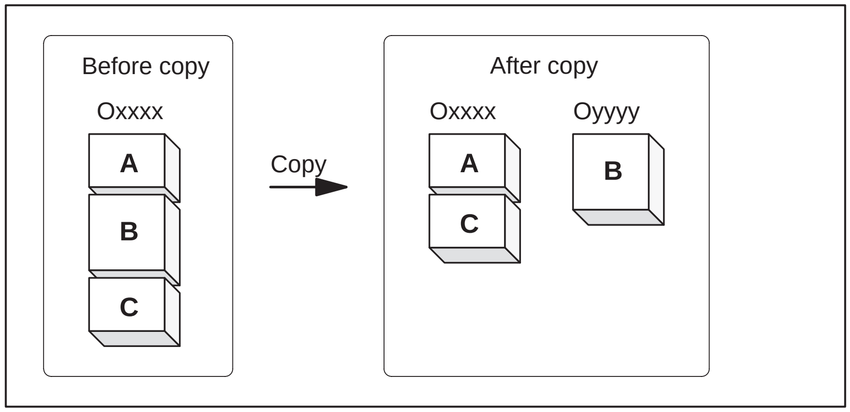
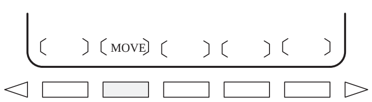


Fig. 9.6.3 Moving Part of a Program

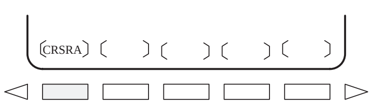
In **Fig. 9.6.3**, part B of the program with program number xxxx is moved to a newly created program with program number yyyy; part B is deleted from the program with program number xxxx.

Procedure for moving part of a program

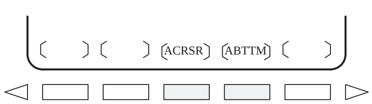
1 Perform steps 1 to 5 in subsection III–9.6.1.



2 Check that the screen for the program to be moved is selected and press soft key **[MOVE]**.



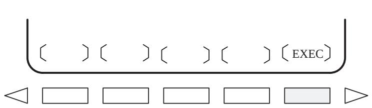
3 Move the cursor to the start of the range to be moved and press soft key **[CRSR~]**.



4 Move the cursor to the end of the range to be moved and press soft key **[~CRSR]** or **[~BTTM]**(in the latter case, the range to the end of the program is copied regardless of the position of the cursor).

Numeric keys 0 to 9

5 Enter the number of the new program (with only numeric keys) and press the INPUT key.



6 Press soft key **[EXEC]**.

9.6.4 Merging a Program

Another program can be inserted at an arbitrary position in the current program.

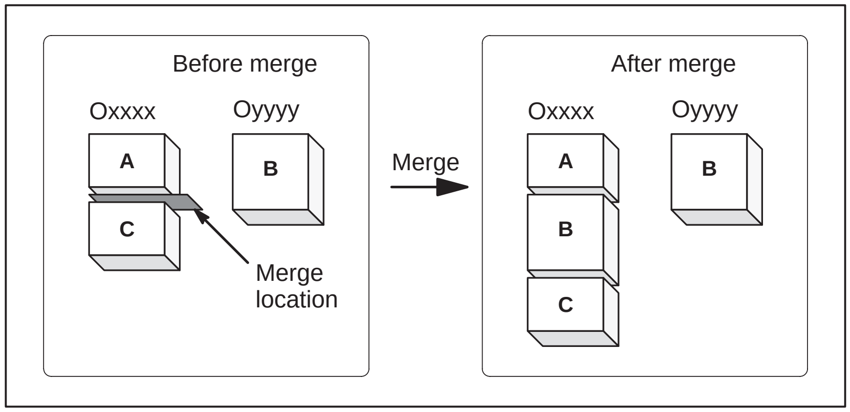
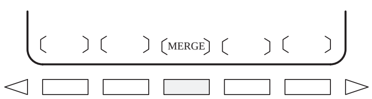


Fig. 9.6.4 Merging a program at a specified location

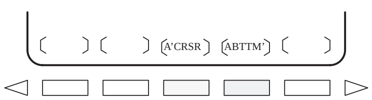
In **Fig. 9.6.4,** the program with program number XXXX is merged with the program with program number YYYY. The OYYYY program remains unchanged after merge operation.

Procedure for merging a program

1 Perform steps **1** to **5** in subsection III–9.6.1.



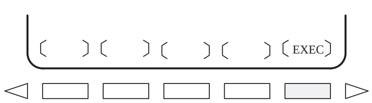
2 Check that the screen for the program to be edited is selected and press soft key **[MERGE]**.



3 Move the cursor to the position at which another program is to be inserted and press soft key **[~'CRSR]** or **[~BTTM']**(in the latter case, the end of the current program is displayed).

Numeric keys 0 to 9

4 Enter the number of the program to be inserted (with only numeric keys) and press the INPUT key.



5 Press soft key **[EXEC]**.
The program with the number specified in step 4 is inserted before the cursor positioned in step 3.

9.6.5 Supplementary Explanation for Copying,Moving and Merging

Explanations

● Setting an editing range

The setting of an editing range start point with **[CRSR~]** can be changed freely until an editing range end point is set with **[~CRSR]** or **[~BTTM]**. If an editing range start point is set after an editing range end point, the editing range must be reset starting with a start point.
The setting of an editing range start point and end point remains valid until an operation is performed to invalidate the setting.
One of the following operations invalidates a setting:
·An edit operation other than address search, word search/scan, and search for the start of a program is performed after a start point or end point is set.
·Processing is returned to operation selection after a start point or end point is set.

● Without specifying a program number

In copying program and moving program, if **[EXEC]** is pressed without specifying a program number after an editing range end point is set, a program with program number O0000 is registered as a work program. This O0000 program has the following features:

· The program can be edited in the same way as a general program. (Do not run the program.)
· If a copy or move operation is newly performed, the previous information is deleted at execution time, and newly set information (all or part of the program) is re–registered. (In merge operation, the previous information is not deleted.) However, the program, when selected for foreground operation, cannot be re–registered in the background. (A BP/S140 alarm is raised.) When the program is re–registered, a free area is produced. Delete such a free area with the RESET key.
· When the program becomes unnecessary, delete the program by a normal editing operation.

● Editing when the system waiting for a program number to be entered

When the system is waiting for a program number to be entered, no edit operation can be performed.

Restrictions

● Number of digits for program number

If a program number is specified by 5 or more digits, a format error is generated.

Alarm

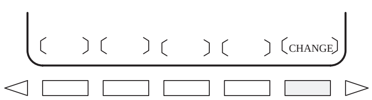
Alarm No.	Contents
70	Memory became insufficient while copying or inserting a program. Copy or insertion is terminated.
101	The power was interrupted during copying, moving, or inserting a program and memory used for editing must be cleared. When this alarm occurs, press the key RESET while pressing function key PROG . Only the program being edited is deleted.

9.6.6 Replacement of Words and Addresses

Replace one or more specified words.
Replacement can be applied to all occurrences or just one occurrence of specified words or addresses in the program.

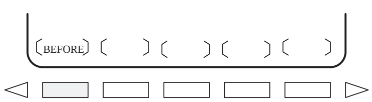
Procedure for change of words or addresses

1 Perform steps 1 to 5 in subsection 9.6.1.



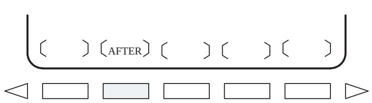
2 Press soft key **[CHANGE]**.

3 Enter the word or address to be replaced.

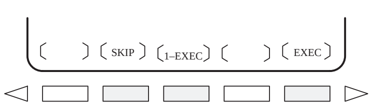


4 Press soft key **[BEFORE]**.

5 Enter the new word or address.



6 Press soft key **[AFTER]**.



7 Press soft key **[EXEC]** to replace all the specified words or addresses after the cursor.
Press soft key **[1–EXEC]** to search for and replace the first occurrence of the specified word or address after the cursor.
Press soft key **[SKIP]** to only search for the first occurrence of the specified word or address after the cursor.

EXAMPLES

- **Replace X100 with Z200**

[CHANGE] X 1 0 0 **[BEFORE]** Z 2 0 0 **[AFTER] [EXEC]**

- **Replace X100Z200 with X30**

[CHANGE] X 1 0 0 Z 2 0 0 **[BEFORE]** X 3 0 **[AFTER] [EXEC]**

- **Replace IF with WHILE**

[CHANGE] I F **[BEFORE]** W H I L E **[AFTER] [EXEC]**

- **Replace X with ,C10**

[CHANGE] X **[BEFOR]** , C 1 0 **[AFTER] [EXEC]**

Explanation

- **Replacing custom macros**

The following custom macro words are replaceable:
IF, WHILE, GOTO, END, DO, BPRNT, DPRINT, POPEN, PCLOS
The abbreviations of custom macro words can be specified.
When abbreviations are used, however, the screen displays the abbreviations as they are key input, even after soft key **[BEFORE]** and **[AFTER]** are pressed.

Restrictions

- **The number of characters for replacement**

Up to 15 characters can be specified for words before or after replacement. (Sixteen or more characters cannot be specified.)

- **The characters for replacement**

Words before or after replacement must start with a character representing an address.(A format error occurs.)

9.7 EDITING OF CUSTOM MACROS

Unlike ordinary programs, custom macro programs are modified, inserted, or deleted based on editing units.
Custom macro words can be entered in abbreviated form.
Comments can be entered in a program.
Refer to the section 10.1 for the comments of a program.

Explanations

• Editing unit

When editing a custom macro already entered, the user can move the cursor to each editing unit that starts with any of the following characters and symbols:

(a) Address

(b) # located at the start of the left side of a substitution statement

(c) /, (,=, and ;

(d) First character of IF, WHILE, GOTO, END, DO, POPEN, BPRNT, DPRNT and PCLOS
On the CRT screen, a blank is placed before each of the above characters and symbols.

(Example) Head positions where the cursor is placed
N001 X–#100 ;
#1 =123 ;
N002 /2 X[12/#3] ;
N003 X–SQRT[#3/3*[#4+1]] ;
N004 X–#2 Z#1 ;
N005 #5 =1+2–#10 ;
IF[#1NE0] GOTO10 ;
WHILE[#2LE5] DO1 ;
#[200+#2] =#2*10 ;
#2 =#2+1 ;
END1 ;

• Abbreviations of custom macro word

When a custom macro word is altered or inserted, the first two characters or more can replace the entire word.
Namely,

WHILE → WH	**GOTO → GO**	**XOR → XO**	**AND → AN**
SIN → SI	**COS → CO**	**TAN → TA**	**ATAN → AT**
SQRT → SQ	**ABS → AB**	**BCD → BC**	**BIN → BI**
FIX → FI	**FUP → FU**	**ROUND → RO**	**END → EN**
POPEN → PO	**BPRNT → BP**	**DPRNT → DP**	**PCLOS→PC**

(Example) Keying in
WH [AB [#2] LE RO [#3]]
has the same effect as
WHILE [ABS [#2] LE ROUND [#3]]
The program is also displayed in this way.

9.8 BACKGROUND EDITING

Editing a program while executing another program is called background editing. The method of editing is the same as for ordinary editing (foreground editing).
A program edited in the background should be registered in foreground program memory by performing the following operation:
During background editing, all programs cannot be deleted at once.

Procedure for background editing

1 Enter **EDIT** or **MEMORY** mode.
Memory mode is allowed even while the program is being executed.

2 Press function key PROG .

3 Press soft key **[(OPRT)]**, then press soft key **[BG–EDT]**.
The background editing screen is displayed (PROGRAM (BG–EDIT) is displayed at the top left of the screen).

4 Edit a program on the background editing screen in the same way as for ordinary program editing.

5 After editing is completed, press soft key **[(OPRT)]**, then press soft key **[BG–EDT].** The edited program is registered in foreground program memory.

Explanation

- **Alarms during background editing**

Alarms that may occur during background editing do not affect foreground operation. Conversely, alarms that may occur during foreground operation do not affect background editing. In background editing, if an attempt is made to edit a program selected for foreground operation, a BP/S alarm (No. 140) is raised. On the other hand, if an attempt is made to select a program subjected to background editing during foreground operation (by means of subprogram calling or program number search operation using an external signal), a P/S alarm (Nos. 059, 078) is raised in foreground operation. As with foreground program editing, P/S alarms occur in background editing. However, to distinguish these alarms from foreground alarms, BP/S is displayed in the data input line on the background editing screen.

9.9 PASSWORD FUNCTION

The password function (bit 4 (NE9) of parameter No. 3202) can be locked using parameter No. 3210 (PASSWD) and parameter No. 3211 (KEYWD) to protect program Nos. O9000 to O9999. In the locked state, parameter NE9 cannot be set to 0. In this state, program Nos. O9000 to O9999 cannot be modified unless the correct keyword is set.
A locked state means that the value set in the parameter PASSWD differs from the value set in the parameter KEYWD. The values set in these parameters are not displayed. The locked state is released when the value already set in the parameter PASSWD is also set in parameter KEYWD. When 0 is displayed in parameter PASSWD, parameter PASSWD is not set.

Procedure for locking and unlocking

Locking

1 Set the MDI mode.

2 Enable parameter writing. At this time, P/S alarm No. 100 is issued on the CNC.

3 Set parameter No. 3210 (PASSWD). At this time, the locked state is set.

4 Disable parameter writing.

5 Press the RESET key to release the alarm state.

Unlocking

1 Set the MDI mode.

2 Enable parameter writing. At this time, P/S alarm No. 100 is issued on the CNC.

3 In parameter No. 3211 (KEYWD), set the same value as set in parameter No. 3210 (PASSWD) for locking. At this time, the locked state is released.

4 Set bit 4 (NE9) of parameter No. 3202 to 0.

5 Disable parameter writing.

6 Press the RESET key to release the alarm state.

7 Subprograms from program Nos. 9000 to 9999 can now be edited.

Explanations

- **Setting parameter PASSWD**

The locked state is set when a value is set in the parameter PASSWD. However, note that parameter PASSWD can be set only when the locked state is not set (when PASSWD = 0, or PASSWD = KEYWD). If an attempt is made to set parameter PASSWD in other cases, a warning is given to indicate that writing is disabled. When the locked state is set (when PASSWD = 0 and PASSWD = KEYWD), parameter NE9 is automatically set to 1. If an attempt is made to set NE9 to 0, a warning is given to indicate that writing is disabled.

- **Changing parameter PASSWD**

Parameter PASSWD can be changed when the locked state is released (when PASSWD = 0, or PASSWD = KEYWD). After step 3 in the procedure for unlocking, a new value can be set in the parameter PASSWD. From that time on, this new value must be set in parameter KEYWD to release the locked state.

- **Setting 0 in parameter PASSWD**

When 0 is set in the parameter PASSWD, the number 0 is displayed, and the password function is disabled. In other words, the password function can be disabled by either not setting parameter PASSWD at all, or by setting 0 in parameter PASSWD after step 3 of the procedure for unlocking. To ensure that the locked state is not entered, care must be taken not to set a value other than 0 in parameter PASSWD.

- **Re–locking**

After the locked state has been released, it can be set again by setting a different value in parameter PASSWD, or by turning the power to the NC off then on again to reset parameter KEYWD.

Notes

Once the locked state is set, parameter NE9 cannot be set to 0 and parameter PASSWD cannot be changed until the locked state is released or the memory all–clear operation is performed. Special care must be taken in setting parameter PASSWD.

10 CREATING PROGRAMS

Programs can be created using any of the following methods:

- MDI keyboard
- PROGRAMMING IN TEACH IN MODE
- CONVERSATIONAL PROGRAMMING INPUT WITH GRAPHIC FUNCTION
- CONVERSATIONAL AUTOMATIC PROGRAMMING FUNCTION
- AUTOMATIC PROGRAM PREPARATION DEVICE (FANUC SYSTEM P)

This chapter describes creating programs using the MDI panel, TEACH IN mode, and conversational programming with graphic function. This chapter also describes the automatic insertion of sequence numbers.

10.1 CREATING PROGRAMS USING THE MDI PANEL

Programs can be created in the **EDIT** mode using the program editing functions described in Chapter III–9.

Procedure for Creating Programs Using the MDI Panel

Procedure

1. Enter the **EDIT** mode.
2. Press the PROG key.
3. Press address key O and enter the program number.
4. Press the INSERT key.
5. Create a program using the program editing functions described in Chapter 9.

Explanation

- **Comments in a program**

Comments can be written in a program using the control in/out codes.

Example)O0001 (FANUC SERIES 16) ;
M08 (COOLANT ON) ;

· When the INSERT key is pressed after the control–out code ”(”,comments, and control–in code ”)” have been typed, the typed comments are registered.

· When the INSERT key is pressed midway through comments, to enter the rest of comments later, the data typed before the INSERT key is pressed may not be correctly registered (not entered, modified, or lost) because the data is subject to an entry check which is performed in normal editing.

Note the following to enter a comment:

· Control–in code ”)” cannot be registered by itself.

· Comments entered after the INSERT key is pressed must not begin with a number, space, or address O.

· If an abbreviation for a macro is entered, the abbreviation is converted into a macro word and registered (see Section 9.7).

· Address O and subsequent numbers, or a space can be entered but are omitted when registered.

10.2 AUTOMATIC INSERTION OF SEQUENCE NUMBERS

Sequence numbers can be automatically inserted in each block when a program is created using the MDI keys in the EDIT mode.
Set the increment for sequence numbers in parameter 3216.

Procedure for automatic insertion of sequence numbers

Procedure

1 Set 1 for SEQUENCE NO. (see subsection III–11.4.3).

2 Enter the **EDIT** mode.

3 Press PROG to display the program screen.

4 Search for or register the number of a program to be edited and move the cursor to the EOB (;) of the block after which automatic insertion of sequence numbers is started.
When a program number is registered and an EOB (;) is entered with the INSERT key, sequence numbers are automatically inserted starting with 0. Change the initial value, if required, according to step 10, then skip to step 7.

5 Press address key N and enter the initial value of N.

6 Press INSERT key.

7 Enter each word of a block.

8 Press EOB key.

9 Press INSERT . The EOB is registered in memory and sequence numbers are automatically inserted. For example, if the initial value of N is 10 and the parameter for the increment is set to 2, N12 inserted and displayed below the line where a new block is specified.

```
PROGRAM                                        O0040  N00012
 O0040 ;
 N10 G92 X0 Y0 Z0 ;
 N12
 %

>_
 EDIT  ****  ***  ***          13 : 18 : 08
[ PRGRM ][  LIB  ][       ][ C.A.P ][ (OPRT) ]
```

10

- In the example above, if N12 is not necessary in the next block, pressing the DELETE key after N12 is displayed deletes N12.
- To insert N100 in the next block instead of N12, enter N100 and press ALTER after N12 is displayed. N100 is registered and initial value is changed to 100.

10.3 CREATING PROGRAMS IN TEACH IN MODE

When the playback option is selected, the **TEACH IN JOG** mode and **TEACH IN HANDLE** mode are added. In these modes, a machine position along the X, Z, and Y axes obtained by manual operation is stored in memory as a program position to create a program.
The words other than X, Z, and Y, which include O, N, G, R, F, C, M, S, T, P, Q, and EOB, can be stored in memory in the same way as in **EDIT** mode.

Procedure for Creating Programs in TEACH IN Mode

The procedure described below can be used to store a machine position along the X, Z, and Y axes.

1 Select the **TEACH IN JOG** mode or **TEACH IN HANDLE** mode.

2 Move the tool to the desired position with jog or handle.

3 Press PROG key to display the program screen. Search for or register the number of a program to be edited and move the cursor to the position where the machine position along each axis is to be registered (inserted).

4 Key in address X .

5 Press the INSERT key. Then a machine position along the X axis is stored in memory.

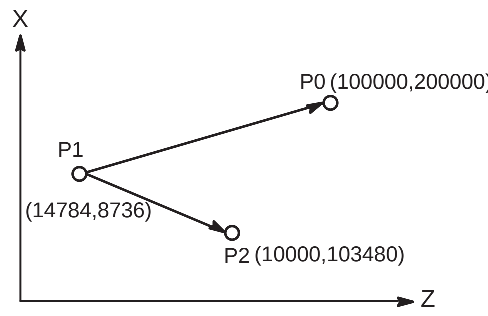
(Example) X10.521 Absolute position (for mm input)
X10521 Data stored in memory

6 Similarly, key in Z , then press the INSERT key. Then a machine position along the Z axis is stored in memory. Further, key in Y , then press the INSERT key. Then a machine position along the Y axis is stored in memory.

All coordinates stored using this method are absolute coordinates.

Examples

```
O1234 ;
N1 G50 X100000 Z200000 ;
N2 G00 X14784 Z8736 ;
N3 G01 Z103480 F300 ;
N4 M02 ;
```



1 Set the setting data SEQUENCE NO. to 1 (on). (The incremental value parameter (No. 3212) is assumed to be “1”.)

2 Select the **TEACH IN HANDLE** mode.

3 Make positioning at position P0 by the manual pulse generator.

4 Select the program screen.

5 Enter program number O1234 as follows:

O 1 2 3 4 INSERT

This operation registers program number O1234 in memory.
Next, press the following keys:

EOB INSERT

An EOB (;) is entered after program number O1234. Because no number is specified after N, sequence numbers are automatically inserted for N0 and the first block (N1) is registered in memory.

6 Enter the P0 machine position for data of the first block as follows:

G 5 0 INSERT X INSERT Z INSERT EOB INSERT

This operation registers G50 X100000 Z200000 ; in memory. The automatic sequence number insertion function registers N2 of the second block in memory.

7 Position the tool at P1 with the manual pulse generator.

8 Enter the P1 machine position for data of the second block as follows:

G 0 0 INSERT X INSERT Z INSERT EOB INSERT

This operation registers G00 X14784 Z8736; in memory. The automatic sequence number insertion function registers N3 of the third block in memory.

9 Position the tool at P2 with the manual pulse generator.

10 Enter the P2 machine position for data of the third block as follows:

G 0 1 INSERT Z INSERT F 3 0 0

INSERT EOB INSERT

This operation registers G01 Z103480 F300; in memory.
The automatic sequence number insertion function registers N4 of the fourth block in memory.

11 Register M02; in memory as follows:

M 0 2 INSERT EOB INSERT

N5 indicating the fifth block is stored in memory using the automatic sequence number insertion function. Press the DELETE key to delete it.

This completes the registration of the sample program.

Explanations

- **Checking contents of the memory**

The contents of memory can be checked in the **TEACH IN** mode by using the same procedure as in **EDIT** mode.

```
PROGRAM                                         O1234  N00004
     (RELATIVE) (ABSOLUTE)
     U      –85.216  X      14.784
     W    –191.264  Z        8.736

     O1234  ;
     N1 G50 X100000 Y0 Z20000  ;
     N2 G00 X14784 Z8736  ;
     N3 G01 Z103480 F300  ;
     N4 M02 ;
     %

  >_
  THND      ****   ***   ***                   14 : 17 : 27
 [PRGRM] [  LIB  ] [       ] [       ] [ (OPRT) ]
```

- **Registering a position with compensation**

When a value is keyed in after keying in address X, Z, or Y, then the INSERT key is pressed, the value keyed in for a machine position is added for registration. This operation is useful to correct a machine position by key–in operation.

- **Registering commands other than position commands**

Commands to be entered before and after a machine position must be entered before and after the machine position is registered, by using the same operation as program editing in **EDIT** mode.

10.4 CONVERSATIONAL PROGRAMMING WITH GRAPHIC FUNCTION

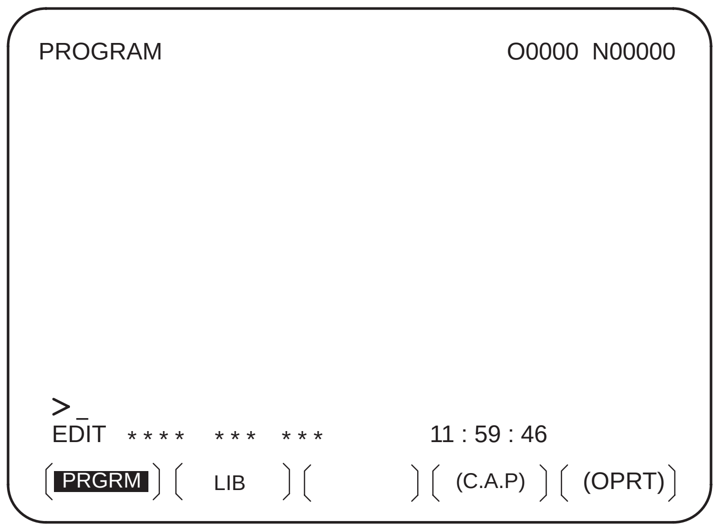
Programs can be created block after block on the conversational screen while displaying the G code menu.
Blocks in a program can be modified, inserted, or deleted using the G code menu and conversational screen.

Procedure for Conversational Programming with Graphic Function

Procedure 1 Creating a program

1 Enter the **EDIT** mode.

2 Press PROG . If no program is registered, the following screen is displayed. If a program is registered, the program currently selected is displayed.



3 Key in the program number of a program to be registered after keying in address O, then press INSERT . For example, when a program with program number 10 is to be registered, key in O 1 0 , then press INSERT . This registers a new program O0010.

4 Press the **[C.A.P]** soft key. The following G code menu is displayed on the screen.
If soft keys different from those shown in step 2 are displayed, press the menu return key to display the correct soft keys.

```
PROGRAM                                        O1234  N00004
  G00  :  POSITIONING
  G01  :  LINEAR IPL
  G02  :  CIRCULAR IPL. CW
  G03  :  CIRCULAR IPL. CCW
  G04  :  DWELL
  G10  :  OFFSET VALUE SETTING (0)
  G20  :  INCH
  G21  :  METRIC
  G22  :  STORED STROKE CHECK ON(0)
  G23  :  STORED STROKE CHECK OFF  (0)
  G25  :  SPINDLE SPEED DETECT OFF
  G26  :  SPINDLE SPEED DETECT ON
  >_
  EDIT  ****   ***   ***             14 : 26 : 15
[PRGRM] [        ] [G.MENU] [ BLOCK ] [        ]
```

5 Key in the G code corresponding to a function to be programmed. When the positioning function is desired, for example, the G code menu lists the function with the G code G00. So key in G00. If the screen does not indicate a function to be programmed, press the page key to display the next G code menu screen. Repeat this operation until a desired function appears. If a desired function is not a G code, key in no data.

6 Press the soft key **[BLOCK]** to display a detailed screen for a keyed in G code. The figure below shows an example of detailed screen for G00.

```
PROGRAM                                        O1234  N00000
G00  :  POSITIONING

     G00   G G G X
     X
     Z   W     (X, Z)
     M
     S
     T
     :
                                                  U

                                              W        Z
  EDIT  ****   ***   ***             14 : 32 : 57
[PRGRM] [        ] [G.MENU] [BLOCK] [ (OPRT) ]
```

When no keys are pressed, the standard details screen is displayed.

```
PROGRAM                                  O0010  N00000

  G        G        G        G
  X                 U
  Z                 W
  A                 C
  F                 H
  I                 K
  P                 Q
  R                 M
  S                 T
   :

 EDIT  ****  ***  ***          14 : 41 : 10
[ PRGRM ][        ][ G.MENU ][ BLOCK ][ (OPRT) ]
```

7 Move the cursor to the block to be modified on the program screen. At this time, a data address with the cursor blinks.

8 Enter numeric data by pressing the numeric keys and press the **[INPUT]** soft key or INPUT key. This completes the input of one data item.

9 Repeat this operation until all data required for the entered G code is entered.

10 Press the INSERT key. This completes the registration of data of one block in program memory. On the screen, the G code menu screen is displayed, allowing the user to enter data for another block. Repeat the procedure starting with **5** as required.

11 After registering all programs, press the **[PRGRM]** soft key. The registered programs are converted to the conversational format and displayed.

12 Press the RESET key to return to the program head.

Procedure2 Modifying a block

1. Move the cursor to the block to be modified on the program screen and press the **[C.A.P]** soft key. Or, press the **[C.A.P]** soft key first to display the conversational screen, then press the ↓ or ↑ page key until the block to be modified is displayed.
2. When data other than a G code is to be altered, just move the cursor to the data and key in a desired value, then press the **[INPUT]** soft key or INPUT key.
3. When a G code is to be altered, press the menu return key ◁ and the soft key **[G.MENU]**. Then the G code menu appears. Select a desired G code, then key in the value. For example, to specify a cutting feed, since the G code menu indicates G01, key in G01. Then press the soft key **[BLOCK]**. The detailed screen of the G code is displayed, so enter the data.
4. After data is changed completely, press the ALTER key. This operation replaces an entire block of a program.

Procedure3 Inserting a block

1. On the conversational screen, display the block immediately before a new block is to be inserted, by using the page keys. On the program screen, move the cursor with the page keys and cursor keys to immediately before the point where a new block is to be inserted.
2. Press the soft key **[G.MENU]** to display the G code menu. Then enter new block data.
3. When input of one block of data is completed in step **2**, press the INSERT key. This operation inserts a block of data.

Procedure4 Deleting a block

1. On the conversational screen, display the contents of a block to be deleted, then press the DELETE key.
2. The contents of the block displayed are deleted from program memory. Then the contents of the next block are displayed on the conversational screen.

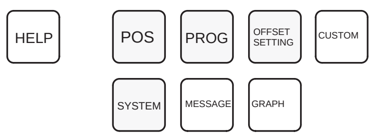
11 SETTING AND DISPLAYING DATA

General

To operate a CNC machine tool, various data must be set on the CRT/MDI or LCD/MDI for the CNC. The operator can monitor the state of operation with data displayed during operation.
This chapter describes how to display and set data for each function.

Explanations

● Screen transition chart



MDI function keys
(Shaded keys ( ) are described in this chapter.)

The screen transition for when each function key on the MDI panel is pressed is shown below. The subsections referenced for each screen are also shown. See the appropriate subsection for details of each screen and the setting procedure on the screen. See other chapters for screens not described in this chapter.

See Chapter III–7 for the screen that appears when function key MESSAGE is pressed. See Chapter III–12 for the screen that appears when function key GRAPH is pressed. See Chapter III–13 for the screen that appears when function key HELP is pressed. In general, function key CUSTOM is prepared by the machine tool builder and used for macros. Refer to the manual issued by the machine tool builder for the screen that appears when function key CUSTOM is pressed.

● Data protection key

The machine may have a data protection key to protect part programs, tool compensation values, setting data, and custom macro variables. Refer to the manual issued by the machine tool builder for where the data protection key is located and how to use it.

POSITION DISPLAY SCREEN

Screen transition triggered by the function key POS

POS

↓

Current position screen

[ABS] [REL] [ALL] [HNDL] [(OPRT)] [▷]

ABS	REL	ALL	HNDL
Position display of work coordinate system ⇒See III–11.1.1.	Position displays relative coordinate system ⇒See III–11.1.2.	Total position display of each coordinate system ⇒See III–11.1.3.	Manual handle interruption ⇒See III–4.6.
Display of part count and run time ⇒See III–11.1.6.	Display of part count and run time ⇒See III–11.1.6.	Display of part count and run time ⇒See III–11.1.6.	
Display of actual speed ⇒See III–11.1.5.	Display of actual speed ⇒See III–11.1.5.	Display of actual speed ⇒See III–11.1.5.	
Setting of floating reference position ⇒See III–11.1.7.	Setting of floating reference position ⇒See III–11.1.7.	Setting of floating reference position ⇒See III–11.1.7.	
	Setting of relative coordinate values ⇒See III–11.1.2.	Setting of relative coordinate values ⇒See III–11.1.2.	

Current position screen

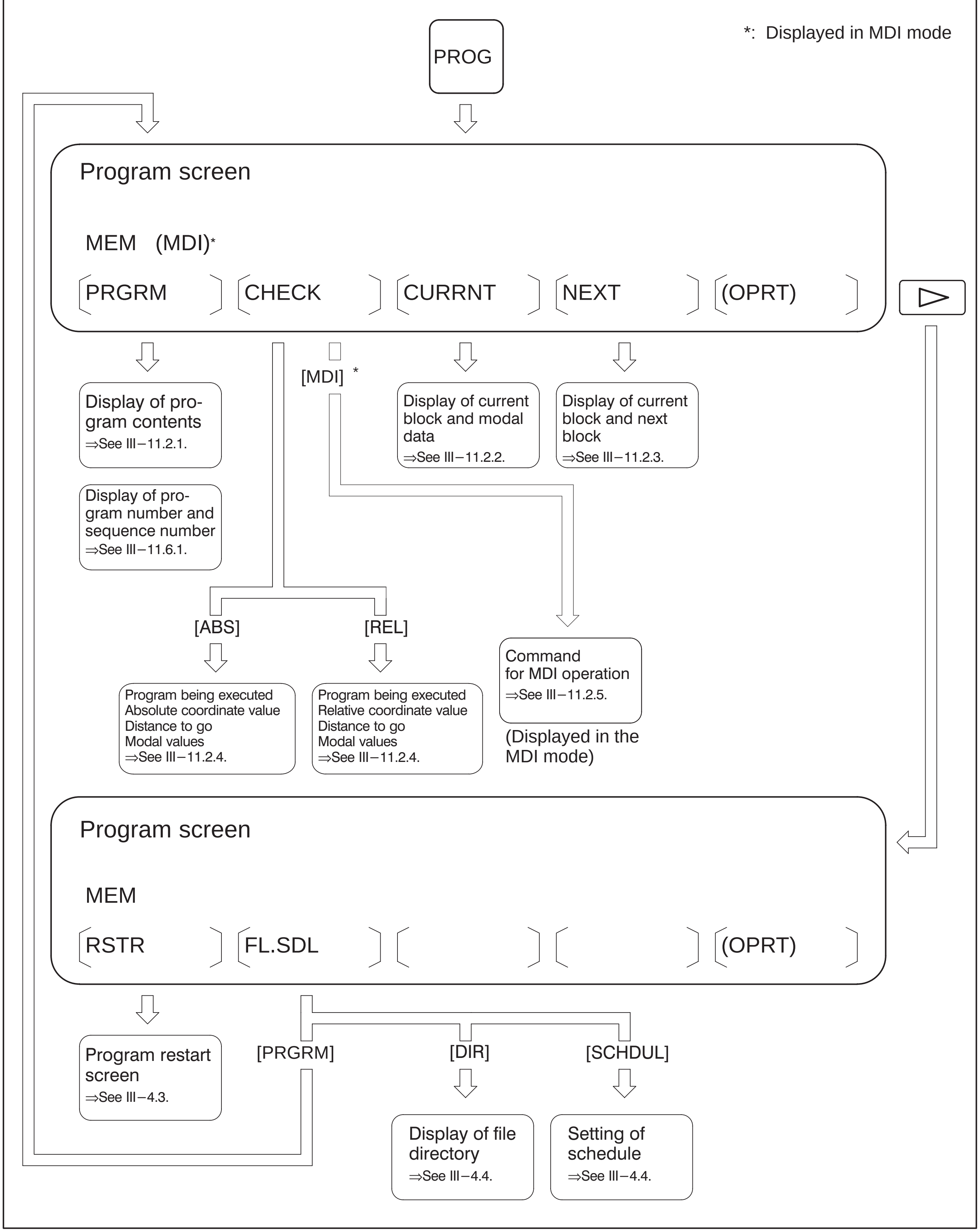
[MONI] [ ] [ ] [ ] [(OPRT)]

↓

Display of operating monitor ⇒See III–11.1.8.

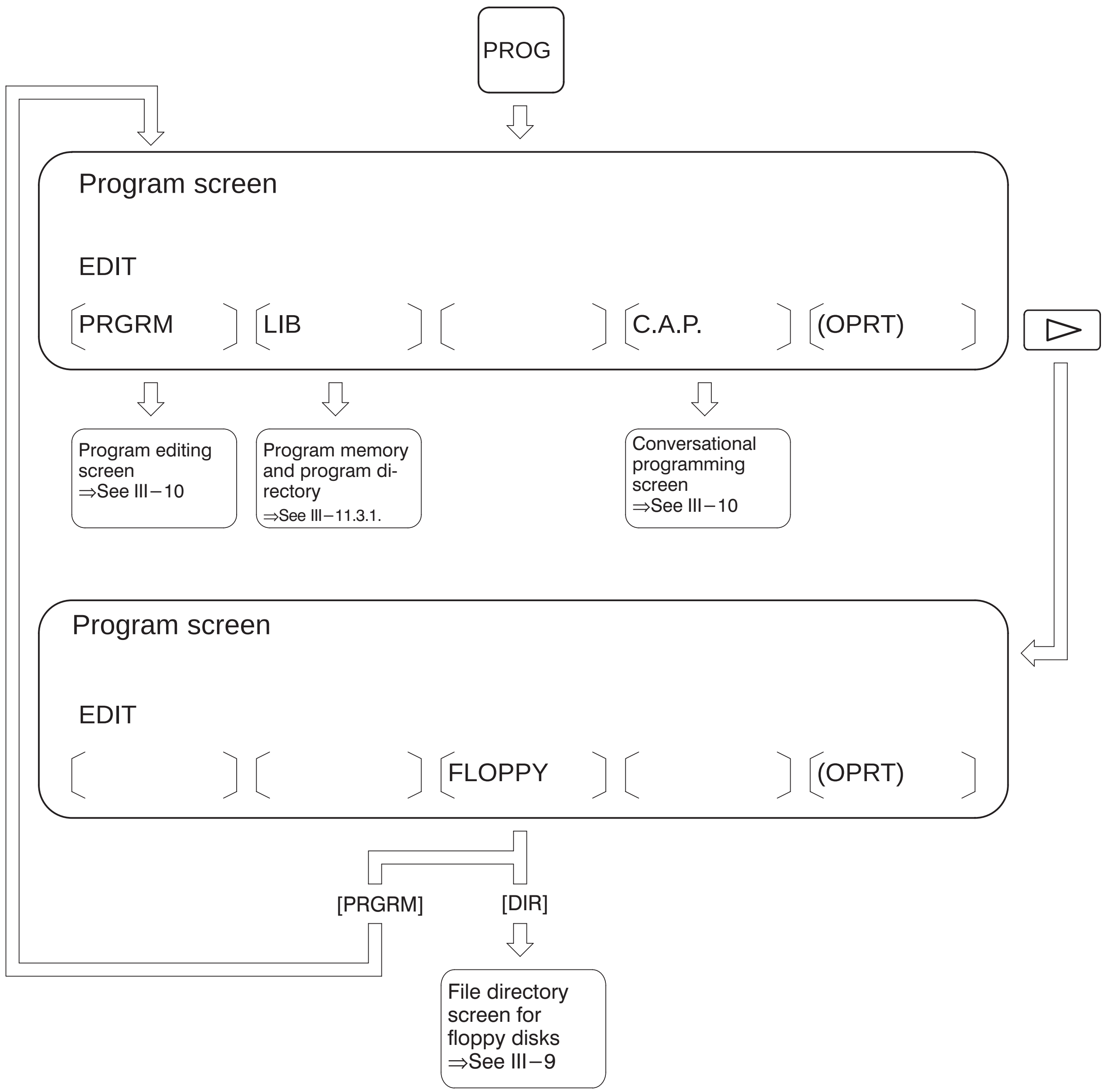
PROGRAM SCREEN

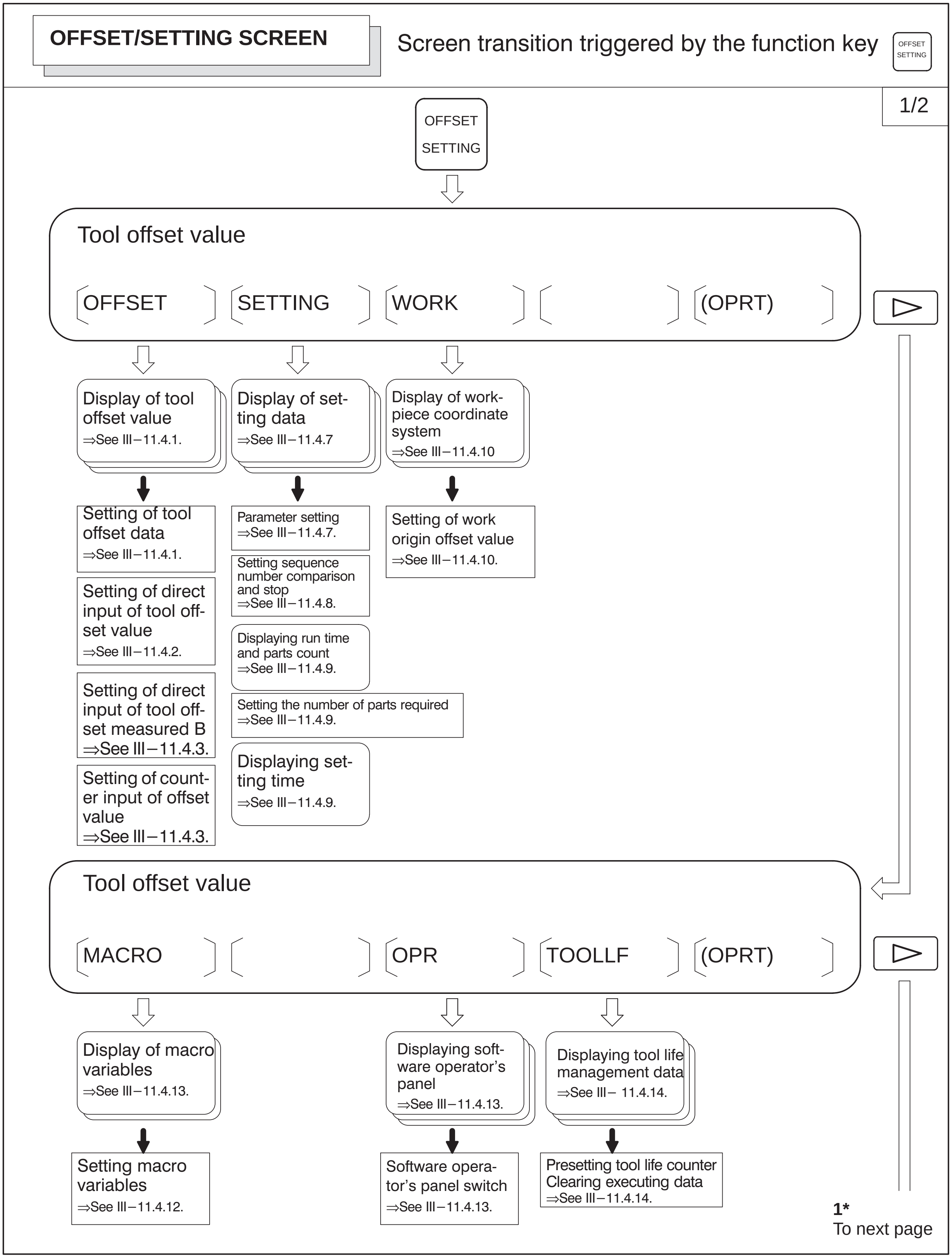
Screen transition triggered by the function key PROG in the MEMORY or MDI mode



PROGRAM SCREEN

Screen transition triggered by the function key PROG in the EDIT mode




OFFSET/SETTING SCREEN
Screen transition triggered by the function key
OFFSET SETTING
1/2
OFFSET SETTING
Tool offset value
OFFSET
SETTING
WORK
(OPRT)
Display of tool offset value ⇒See III–11.4.1.
Display of setting data ⇒See III–11.4.7
Display of workpiece coordinate system ⇒See III–11.4.10
Setting of tool offset data ⇒See III–11.4.1.
Setting of direct input of tool offset value ⇒See III–11.4.2.
Setting of direct input of tool offset measured B ⇒See III–11.4.3.
Setting of counter input of offset value ⇒See III–11.4.3.
Parameter setting ⇒See III–11.4.7.
Setting sequence number comparison and stop ⇒See III–11.4.8.
Displaying run time and parts count ⇒See III–11.4.9.
Setting the number of parts required ⇒See III–11.4.9.
Displaying setting time ⇒See III–11.4.9.
Setting of work origin offset value ⇒See III–11.4.10.
Tool offset value
MACRO
OPR
TOOLLF
(OPRT)
Display of macro variables ⇒See III–11.4.13.
Setting macro variables ⇒See III–11.4.12.
Displaying software operator's panel ⇒See III–11.4.13.
Software operator's panel switch ⇒See III–11.4.13.
Displaying tool life management data ⇒See III– 11.4.14.
Presetting tool life counter Clearing executing data ⇒See III–11.4.14.
1*
To next page

2/2

1*

Tool offset value

[OFST.2] [W.SHFT] [ ] [ ] [(OPRT)]

Display of Y axis offset value
⇒See III–11.4.6.

Setting of Y axis offset data
⇒See III–11.4.6.

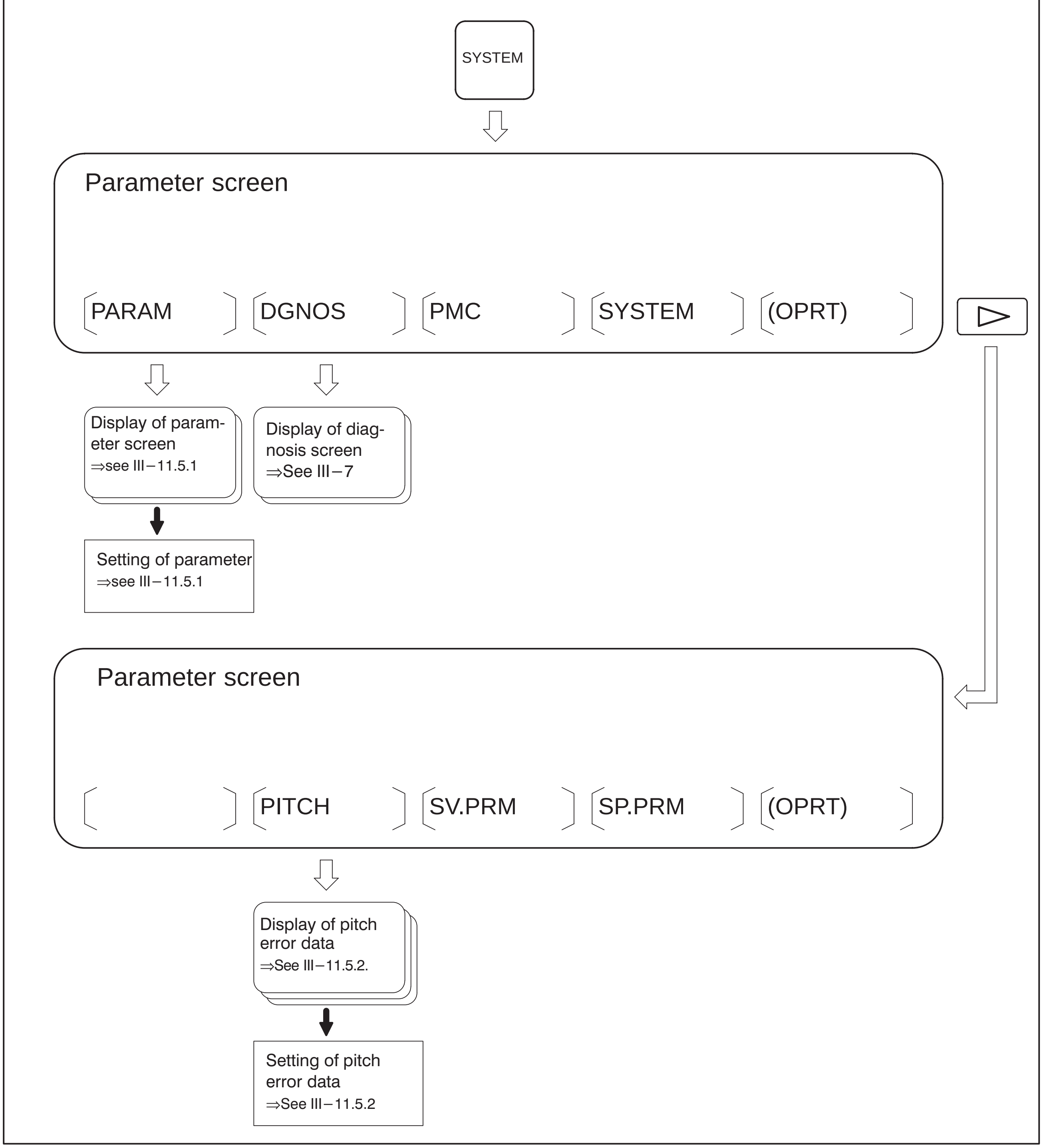
Display of work coordinate system value
⇒See III–11.4.5

Setting of work coordinate system shift value
⇒See III–11.4.5

Setting of work-piece coordinate shift value by direct input function B for tool offset measured 2.
⇒See III–11.4.3.

SYSTEM SCREEN

Screen transition triggered by the function key SYSTEM



- **Setting screens**

The table below lists the data set on each screen.

Table.11. Setting screens and data on them

No.	Setting screen	Contents of setting	Reference item
1	Tool offset value	Tool offset value Tool nose radius compensation value	Subsec. 11.4.1
		Direct input of tool offset value	Subsec. 11.4.2
		Direct input of tool offset value measured B	Subsec. 11.4.3
		Counter input of offset value	Subsec. 11.4.4
		Y axis offset	Subsec. 11.4.6
2	Workpiece coordinate system setting	Workpiece coordinate system shift value	Subsec. 11.4.5
		Workpiece origin offset value	Subsec. 11.4.10
3	Setting data (handy)	Parameter write TV check Punch code (EIA/ISO) Input unit (mm/inch) I/O channel Automatic insert of Sequence No. Conversion of tape format (F15)	Subsec. 11.4.7
		Sequence number comparison and stop	Subsec. 11.4.8
4	Setting data (mirror image)	Mirror image	Subsec. 11.4.7
5	Setting data (timer)	Parts required	Subsec. 11.4.9
6	Macro variables	Custom macro common variables (#100A#149) or (#100A#199) (#500A#531) or (#500A#599)	Subsec. 11.4.12
7	Parameter	Parameter	Subsec. 11.5.1
8	Pitch error	Pitch error compensation data	Subsec. 11.5.2
9	software operator's panel	Mode selection Jog feed axis selection Jog rapid traverse Axis selection for Manual pulse generator Multiplication for manual pulse generator Jog feedrate Feedrate override Rapid traverse override Optional block skip Single block Machine lock Dry run Protect key Feed hold	Subsec. 11.4.13
10	Tool life data (Tool life management)	Life count	Subsec. 11.4.14
11	Current position display screen	Floating reference position	Subsec. 11.1.7

11.1 SCREENS DISPLAYED BY FUNCTION KEY POS

Press function key POS to display the current position of the tool.

The following three screens are used to display the current position of the tool:

·**Position display screen for the work coordinate system.**
·**Position display screen for the relative coordinate system.**
·**Overall position display screen.**

The above screens can also display the feedrate, run time, and the number of parts. In addition, a floating reference position can be set on these screens.

Function key POS can also be used to display the load on the servo motor and spindle motor and the rotation speed of the spindle motor (operating monitor display).

Function key POS can also be used to display the screen for displaying the distance moved by handle interruption. See Section 4.6 for details on this screen.

11.1.1 Position Display in the Workpiece Coordinate System

Displays the current position of the tool in the workpiece coordinate system. The current position changes as the tool moves. The least input increment is used as the unit for numeric values. The title at the top of the screen indicates that absolute coordinates are used.

Display procedure for the current position screen in the workpiece coordinate system

1 Press function key POS .

2 Press soft key **[ABS]**.

3 On a 9” CRT, press the **[ABS]** soft key one more time to display the coordinates along axes other than the six standard axes.

- **Display with one–path control**

```
ACTUAL POSITION(ABSOLUTE)      O1000 N00010

   X        123.456
   Z        456.789

             PART COUNT      5
 RUN TIME    0H15M    CYCLE TIME   0H 0M38S
 ACT.F     3000 MM/M         S   0 T0000

MEM STRT MTN ***         09:06:35
[ ABS  ][  REL  ][  ALL  ][  HNDL ][(OPRT)]
```

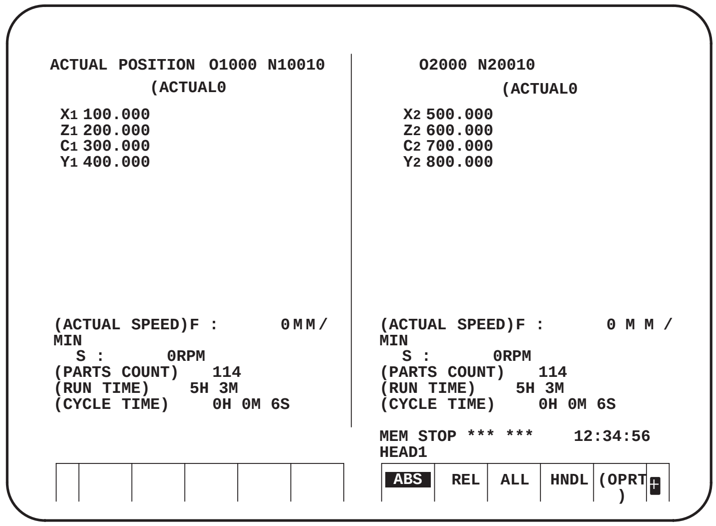
- **Display with two–path control (9”CRT)**

```
ACTUAL POSITION(ABSOLUTE)      O1000 N00010

   X1       123.456
   Z1       456.789
   X2       123.456
   Z2       456.789
             PART COUNT      5
 RUN TIME    0H15M    CYCLE TIME   0H 0M38S
 ACT.F     3000 MM/M         S   0 T0000

MEM STRT MTN ***         09:06:35      HEAD1
[ ABS  ][  REL  ][  ALL  ][  HNDL ][(OPRT)]
```

- **Two–path lathe control / 9” CRT**

Note For the two–path control, the display may not be as shown above. In some cases, only the coordinates along the axes on tool post 1 are displayed due to the number of axes. In that case, press the **[ABS]** soft key one more time to display the coordinates along the axes on tool post 2.

- **Display with two–path control (14”CRT)**



Explanations

- **Display including compensation values**

Bits 6 and 7 of parameter 3104 can be used to select whether the displayed values include tool offset value and tool nose radius compensation.

- **Displaying the sixth and subsequent axes**

On the 9” CRT or the shared screen of the 14” CRT, only the coordinates for the first to fifth axes are displayed initially whenever there are six or more controlled axes. Pressing the **[ABS]** soft key displays the coordinates for the sixth and subsequent axes. When six or more controlled axes are used under two–path control, the coordinates for path 1 are displayed initially on the 9” CRT. Pressing the **[ABS]** soft key displays the coordinates for path 2. On the shared screen of the 14” CRT, the tool post selection signal is used to select the display for path 1 or 2.

11.1.2 Position Display in the Relative Coordinate System

Displays the current position of the tool in a relative coordinate system based on the coordinates set by the operator. The current position changes as the tool moves. The increment system is used as the unit for numeric values. The title at the top of the screen indicates that relative coordinates are used.

Display procedure for the current position screen with the relative coordinate system

1 Press function key POS .

2 Press soft key **[REL]**.

3 On a 9” CRT, press the **[REL]** soft key one more time to display the coordinates along axes other than the six standard axes.

- **Display with one–path control**

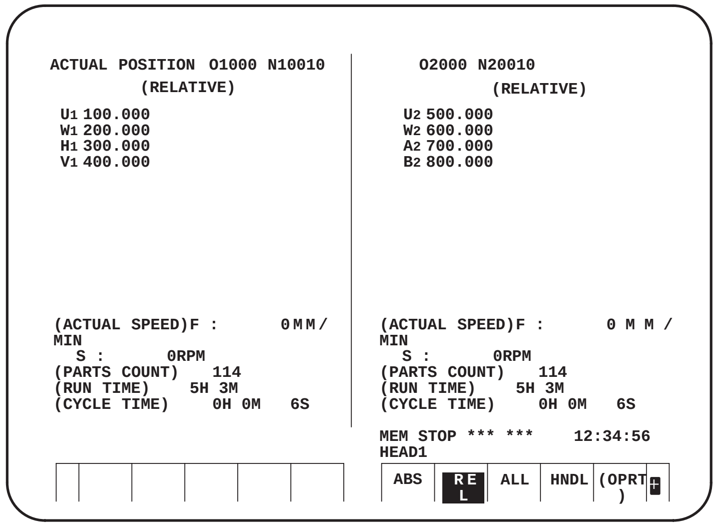
```
ACTUAL POSITION(RELATIVE)     O1000 N00010

  U      123.456
  W      456.789

              PART COUNT      5
 RUN TIME   0H15M    CYCLE TIME   0H 0M38S
 ACT.F    3000 MM/M        S   0 T0000

MEM STRT M[redacted]          09:06:35
[  ABS  ] [       [redacted]LL  ] [  HNDL ] [(OPRT)]
```

- **Display with two–path control (9”CRT)**

```
ACTUAL POSITION(RELATIVE)     O1000 N00010

  U1     100.000
  W1     200.000
  U2     300.000
  W2     400.000
              PART COUNT      5
 RUN TIME   0H15M    CYCLE TIME   0H 0M38S
 ACT.F    3000 MM/M        S   0 T0000

MEM STRT M[redacted]          09:06:35      HEAD1
[  ABS  ] [       [redacted]LL  ] [  HNDL ] [(OPRT)]
```

Note For the two–path lathe control (9”CRT), the display may not be as shown above. In some cases, only the coordinates along the axes on tool post 1 are displayed due to the number of axes. In that case, press the **[REL]** soft key one more time to display the coordinates along the axes on tool post 2.

- **Display with two–path control) (14”CRT)**

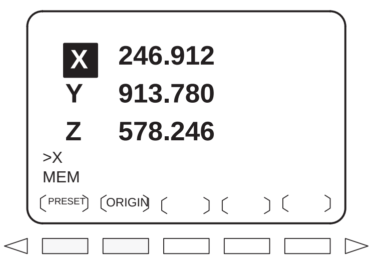


Explanations

- **Setting the relative coordinates**

The current position of the tool in the relative coordinate system can be reset to 0 or preset to a specified value as follows:

Procedure to set the axis coordinate to a specified value

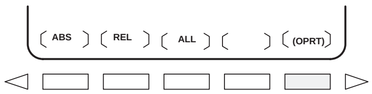


1 Enter an axis address (such as X or Z) on the screen for the relative coordinates. The indication for the specified axis blinks and the soft keys change as shown on the left.

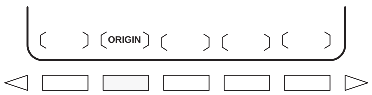
2 · To reset the coordinate to 0, press soft key **[ORIGIN]**. The relative coordinate for the blinking axis is reset to 0.

· To preset the coordinate to a specified value, enter the value and press soft key **[PRESET]**. The relative coordinate for the blinking axis is set to the entered value.

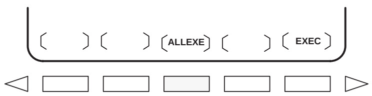
Procedure to reset all axes



1 Press soft key **[(OPRT)]**.



2 Press soft key **[ORIGIN]**.



3 Press soft key **[ALLEXE]**.
The relative coordinates for all axes are reset to 0.

- **Display including compensation values**

Bits 4 (DRL) and 5 (DRC) of parameter 3104 can be used to select whether the displayed values include tool offset and tool nose radius compensation.

- **Presetting by setting a coordinate system**

Bit 3 of parameter 3104 is used to specify whether the displayed positions in the relative coordinate system are preset to the same values as in the workpiece coordinate system when a coordinate system is set by a G50 (G code system A) or a G92 (G code system B or C) command or when the manual reference position return is made.

- **Displaying the sixth and subsequent axes**

On the 9” CRT or the shared screen of the 14” CRT, only the coordinates for the first to fifth axes are displayed initially whenever there are six or more controlled axes. Pressing the **[REL]** soft key displays the coordinates for the sixth and subsequent axes. When six or more controlled axes are used under two–path control, the coordinates for path 1 are displayed initially on the 9” CRT. Pressing the **[REL]** soft key displays the coordinates for path 2. On the shared screen of the 14” CRT, the tool post selection signal is used to select the display for path 1 or 2.

11.1.3 Overall Position Display

Displays the following positions on a screen : Current positions of the tool in the workpiece coordinate system, relative coordinate system, and machine coordinate system, and the remaining distance. The relative coordinates can also be set on this screen. See subsection III–11.1.2 for the procedure.

Procedure for displaying overall position display screen

1 Press function key POS .

2 Press soft key **[ALL]**.

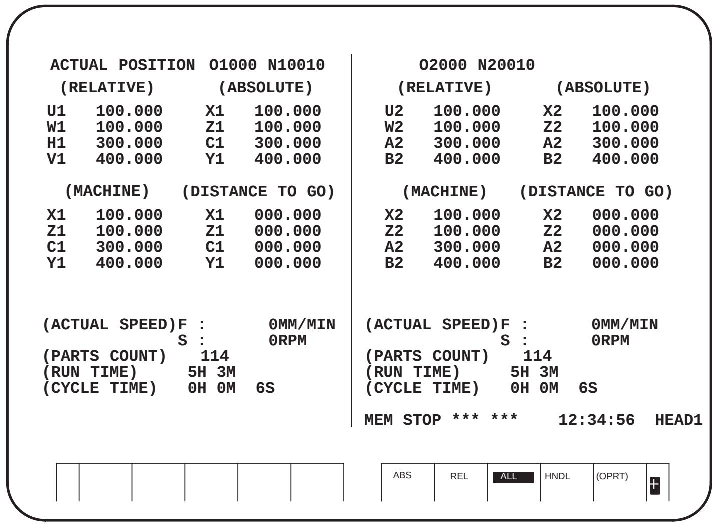
- **Display with one–path control**

```
ACTUAL POSITION          O1000 N00010
  (RELATIVE)               (ABSOLUTE)
 U  246.912               X  123.456
 W  913.780               Z  456.890

  (MACHINE)               (DISTANCE TO
 X    0.000               GO)
 Z    0.000               X    0.000
                          Z    0.000
                PART COUNT      5
 RUN TIME   0H15M    CYCLE TIME  0H 0M38S
 ACT.F    3000 MM/M       S  0 T0000
MEM **** *** ***          09:06:35
[  ABS  ] [  REL  ] [  ALL  ] [  HNDL ] [(OPRT)]
```

- **Display with two–path control (9"CRT)**

```
ACTUAL POSITION          O1000 N00010
  (RELATIVE)               (ABSOLUTE)
 U1  100.000              X1  100.000
 W1  200.000              Z1  200.000
 U2  300.000              X2  300.000
 W2  400.000              Z2  400.000
  (MACHINE)               (DISTANCE TO
 X1  100.000              GO)
 Z1  200.000              X1  000.000
 X2  300.000              Z1  000.000
 Z2  400.000              X2  000.000
                          Z2  000.000
                PART COUNT      5
 RUN TIME   0H15M    CYCLE TIME  0H 0M38S
 ACT.F    3000 MM/M       S  0 T0000
MEM **** *** ***          09:06:35        HEAD1
[  ABS  ] [  REL  ] [  ALL  ] [  HNDL ] [(OPRT)]
```

- **Display with two–path control (14”CRT)**



Explanations

- **Coordinate display**

The current positions of the tool in the following coordinate systems are displayed at the same time:

· Current position in the relative coordinate system (relative coordinate)

· Current position in the work coordinate system (absolute coordinate)

· Current position in the machine coordinate system (machine coordinate)

· Distance to go (distance to go)

- **Distance to go**

The distance remaining is displayed in the MEMORY or MDI mode. The distance the tool is yet to be moved in the current block is displayed.

- **Machine coordinate system**

The least command increment is used as the unit for values displayed in the machine coordinate system. However, the least input increment can be used by setting bit 0 (MCN) of parameter 3104.

- **Resetting relative coordinates**

On the overall position display screen, relative coordinates can be reset to 0 or preset to specified values. The procedure is the same as that for resetting the relative coordinates described in III–11.1.2.

- **Displaying the sixth and subsequent axes**

On the shared screen of the 14” CRT, only the coordinates for the first to fifth axes are displayed initially whenever there are six or more controlled axes. Pressing the **[ALL]** soft key displays the coordinates for the sixth and subsequent axes. On the shared screen of the 14” CRT, the tool post selection signal is used to select the display for path 1 or 2.

- **Displaying the fifth and subsequent axes**

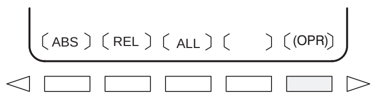
On the 9” CRT, relative coordinates cannot be displayed together with absolute coordinates when there are five or more controlled axes (when the total number of controlled axes is five or more, for two–path control). Pressing the **[ALL]** soft key toggles the display between absolute and relative coordinates.

11.1.4 Presetting the Workpiece Coordinate System

A workpiece coordinate system shifted by an operation such as manual intervention can be preset using MDI operations to a pre–shift workpiece coordinate system. The latter coordinate system is displaced from the machine zero point by a workpiece zero point offset value.
A command (G92.1) can be programmed to preset a workpiece coordinate system. (See Subsec. III–8.2.4.)

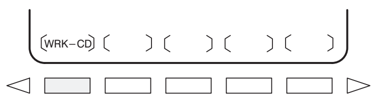
Procedure for Presetting the Workpiece Coordinate System

1 Press function key POS .

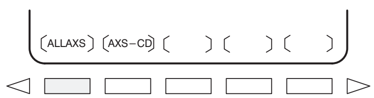


2 Press soft key **[(OPRT)]**.

3 When **[WRK–CD]** is not displayed, press the continuous menu key ▷.



4 Press soft key **[WRK–CD]**.



5 Press soft key **[ALLAXS]** to preset all axes.

6 To preset a particular axis in step 5, enter the axis name (X , Y , ...) and 0 , then press soft key **[AXS–CD]**.

Explanations

- **Operation mode**

This function can be executed when the reset state or automatic operation stop state is entered, regardless of the operation mode.

- **Presetting relative coordinates**

As with absolute coordinates, bit 3 (PPD) of parameter No. 3104 is used to specify whether to preset relative coordinates (RELATIVE).

11.1.5 Actual Feedrate Display

The actual feedrate on the machine (per minute) can be displayed on a current position display screen or program check screen by setting bit 0 (DPF) of parameter 3015. On a 14–inch CRT, the actual feedrate is always displayed.

Display procedure for the actual feedrate on the current position display screen

1 Press function key [POS] to display a current position display screen.

```
  ACTUAL POSITION(ABSOLUTE)     O1000 N00010

     X      123.456
     Z      456.789

                       PART COUNT       5
   RUN TIME    0H15M  CYCLE TIME   0H 0M38S
   ACT.F     3000 MM/M        S   0 T0000

MEM STRT MTN ***          09:06:35
[ ABS  ] [  REL  ] [  ALL  ] [  HNDL ] [ OPRT ]
```

Actual feedrate is displayed after ACT.F.

The actual feedrate is displayed in units of millimeter/min or inch/min (depending on the specified least input increment) under the display of the current position.

Explanations

- **Actual feedrate value**

The actual rate is calculated by the following expression:

$$Fact = \sqrt{\sum_{i=1}^{n}(fi)^2}$$

where

n : Number of axes

fi : Cutting feed rate in the tangential direction of each axis or rapid traverse rate

Fact : Actual feedrate displayed

The display unit: mm/min (metric input).
inch/min (Inch input, Two digits below the decimal point are displayed.)

The feedrate along the PMC axis can be omitted by setting bit 1 (PCF) of parameter 3105.

- **Actual feedrate display of feed per revolution**

In the case of feed per revolution and thread cutting, the actual feedrate displayed is the feed per minute rather than feed per revolution.

- **Actual feedrate display of rotary axis**

In the case of movement of rotary axis, the speed is displayed in units of deg/min but is displayed on the screen in units of input system at that time. For example, when the rotary axis moves at 50 deg/min, the following is displayed: 0.50 INCH/M

- **Actual feedrate display on the other screen**

The program check screen also displays the actual feedrate.

11.1.6 Display of Run Time and Parts Count

The run time, cycle time, and the number of machined parts are displayed on the current position display screens.

Procedure for displaying run time and parts count on the current position display screen

1 Press function key POS to display a current position display screen.

```
ACTUAL POSITION(RELATIVE)     O1000 N00010

    X      123.456
    Z      456.789


                       PART COUNT       5
 RUN TIME    0H15M  CYCLE TIME   0H 0M38S
 ACT.F     3000 MM/M        S   0 T0000

MEM STRT MTN ***         09:06:35
[  ABS  ] [  REL  ] [  ALL  ] [  HNDL ] [ OPRT ]
```

The number of machined parts (PART COUNT), run time (RUN TIME), and cycle time (CYCLE TIME) are displayed under the current position.

Explanations

- **PART COUNT**

Indicates the number of machined parts. The number is incremented each time M02, M30, or an M code specified by parameter 6710 is executed.

- **RUN TIME**

Indicates the total run time during automatic operation, excluding the stop and feed hold time.

- **CYCLE TIME**

Indicates the run time of one automatic operation, excluding the stop and feed hold time. This is automatically preset to 0 when a cycle start is performed at reset state. It is preset to 0 even when power is removed.

- **Display on the other screen**

Details of the run time and the number of machined parts are displayed on the setting screen. See subsection III–11.4.9.

- **Parameter setting**

The number of machined parts and run time cannot be set on current position display screens. They can be set by parameters 6711, 6751, and 6752 or on the setting screen.

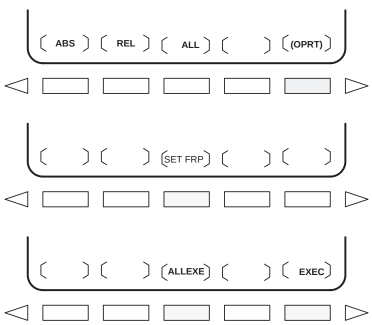
- **Incrementing the number of machined parts**

Bit 0 (PCM) of parameter 6700 is used to specify whether the number of machined parts is incremented each time M02, M30, or an M code specified by parameter 6710 is executed, or only each time an M code specified by parameter 6710 is executed.

11.1.7 Setting the Floating Reference Position

To perform floating reference position return with a G30.1 command, the floating reference position must be set beforehand.

Procedure for setting the floating reference position



1 Press function key POS to display a screen used for displaying the current position. Any of the following three screens may be selected: The screen for displaying the current position in the relative coordinate system, screen for displaying the current position in the workpiece coordinate system, and screen for displaying the current positions in four different coordinate systems.

2 Move the tool to the floating reference position by jogging.

3 Press soft key **[(OPRT)]**.

4 Press soft key **[SET FRP]**.

5 To register the floating reference positions for all axes, press soft key **[ALLEXE]**.
To register the floating reference position of a specific axis, enter the name of the axis (X , etc.), then press soft key **[EXEC]**. Two or more names can be entered consecutively (e.g., X Y Z **[EXEC]**).
The above operation stores the floating reference position. It can be checked with parameter (no. 1244).

6 In step **4**, the floating reference position along a specified axis can also be stored by entering the axis name (such as X) and pressing soft key **[SET FRP]**.

Explanations

- **Presetting the relative coordinate system**

By parameter FPC (bit 3 of parameter 1201), the relative position can be preset to 0 when a floating reference position is registered.

11.1.8 Operating Monitor Display

The reading on the load meter can be displayed for each servo axis and the serial spindle by setting bit 5 (OPM) of parameter 3111 to 1. The reading on the speedometer can also be displayed for the serial spindle.

Procedure for displaying the operating monitor

1 Press function key POS to display a current position display screen.

2 Press the continuous–menu key ▷.

3 Press soft key **[MONI]**.

```
OPERATING MONITOR        O0001 N00001
 (LOAD METER)

X :        *   *   *  80%  S1               201%

Z :   *   *   *   *   *   0%     (SPEED METER RPM)

C :   *   *   *   *   *   0%  S1 :       *   *   *
1500
                           PART COUNT       5
   RUN TIME    0H15M   CYCLE TIME    0H 0M38S
   ACT.F     3000 MM/M

 MEM STRT MTN ***            09:06:35
 [ MONI ][  REL  ][  ALL  ][  HNDL ][ OPRT ]
```

Explanations

- **Display of the servo axes**

The reading on the load meter can be displayed for up to three servo axes by setting parameters 3151 to 3158.
When all these parameters are set to 0, data is displayed only for the basic axes.

- **Display of the spindle axes**

When serial spindles are used, the reading on the load meter and speedometer can be displayed only for the main serial spindle.

- **Unit of graph**

The bar graph for the load meter shows load up to 200% (only a value is displayed for load exceeding 200%). The bar graph for the speedometer shows the ratio of the current spindle speed to the maximum spindle speed (100%).

- **Load meter**

The reading on the load meter depends on servo parameter 2086 and spindle parameter 4127.
The spindle speed to be displayed during operation monitoring is calculated from the speed of the spindle motor (see the formula below). The spindle speed can therefore be displayed, during operation monitoring, even when no position coder is used. To display the correct spindle speed, however, the maximum spindle speed for each gear (spindle speed at each gear ratio when the spindle motor rotates at the maximum speed) must be set in parameters No. 3741 to 3744.

The input of the clutch and gear signals for the first serial spindle is used to determine the gear which is currently selected. Control the input of the CTH1A and CTH2A signals according to the gear selection, by referring to the table below.

(Formula for calculating the spindle speed to be displayed)

$$\text{Spindle speed displayed during operation monitoring} = \frac{\text{Speed of spindle motor}}{\text{Maximum speed of spindle motor}} \times \text{Maximum spindle speed with the gear being used}$$

The following table lists the correspondence between clutch and gear selection signals CTH1A and CTH2A <G070#3, #2>, used to determine the gear being used, and parameters:

CTH1A	CTH2A	Parameter
0	0	=No.3741 (Maximum spindle speed with gear 1)
0	1	=No.3742 (Maximum spindle speed with gear 2)
1	0	=No.3743 (Maximum spindle speed with gear 3)
1	1	=No.3744 (Maximum spindle speed with gear 4)

The speed of the spindle motor and spindle can be displayed, during operation monitoring, only for the first serial spindle and the spindle switching axis for the first serial spindle. It cannot be displayed for the second spindle.

- **Speed meter**

Although the speedometer normally indicates the speed of the spindle motor, it can also be used to indicate the speed of the spindle by setting bit 6 (OPS) of parameter 3111 to 1.

- **Color of graph**

On a color CRT, if the value of a load meter exceeds 100%, the bar graph turns purple.

11.2 SCREENS DISPLAYED BY FUNCTION KEY PROG (IN MEMORY MODE OR MDI MODE)

This section describes the screens displayed by pressing function key PROG in MEMORY or MDI mode.The first four of the following screens display the execution state for the program currently being executed in MEMORY or MDI mode and the last screen displays the command values for MDI operation in the MDI mode:

11.2.1 Program contents display screen
11.2.2 Current block display screen
11.2.3 Next block display screen
11.2.4 Program check screen
11.2.5 Program screen for MDI operation
11.2.6 Operation time Stamp
11.2.7 Displaying the B–axis operation state

Function key PROG can also be pressed in MEMORY mode to display the program restart screen and scheduling screen.
See Section III–4.3 for the program restart screen.
See Section III–4.4 for the scheduling screen.

11.2.1 Program Contents Display

Displays the program currently being executed in MEMORY or MDI mode.

Procedure for displaying the program contents

1 Press function key PROG to display a program screen.

2 Press chapter selection soft key **[PRGRM]**.
The cursor is positioned at the block currently being executed.

```
PROGRAM                              O2000 N00130
O2000 ;
N100 G50 X0 Z0. ;
N110 G91 G00 X–70. ;
N120 Z–70. ;
N130 G01 X–60 ;
N140 G41 G03 X–17.5 Z17.5 R17.5 ;
N150 G01 X–25. ;
N160 G02 X27.5 Z27.5 R27.5
N170 G01 X20. ;
N180 G02 X45. Z45. R45. ;

> _                               S   0  T0000
MEM STRT    ***        16:05:59
[ PRGRM ][ CHECK ][ CURRNT ][ NEXT ][ (OPRT) ]
```

Explanations

- **14–inch CRT 9.5/8.4–inch LCD**

On a 14–inch CRT, 9.5–ibch, or 8.4–inch LCD, the contents of the program are displayed on the right half of the screen or on the entire screen (switched each time soft key **[PRGRM]** is pressed).

```
                                      O0006  N00000
PROGRAM

O0003 ;                               G65 H01 P#100 O#3901 ;
G65 H01 P#2001 O0 ;                   G65 H01 P#101 O#3902 ;
G65 H01 P#2014 O0 ;                   G65 H01 P#3901 O#102 ;
G65 H01 P#2110 O0 ;                   G65 H01 P#3902 O#103 ;
G04 P2000 ;                           G04 P5000 ;
G04 P2000 ;                           G04 P5000 ;
G04 P2000 ;                           G04 ;
G65 H01 P#2001 O50000 ;               G65 H01 P#100 O#4001 ;
G65 H01 P#2014 O60000 ;               G65 H01 P#101 O#4002 ;
G65 H01 P#2110 O30000 ;               / G65 H01 P#102 O#4003 ;
G04 P2000 ;                           G65 H01 P#103 O#4004 ;
G04 P2000 ;                           G65 H01 P#104 O#4005 ;
G04 P2000 ;                           G65 H01 P#105 O#4006 ;
G65 H02 P#2001 O#2001 R3 ;            G65 H01 P#106 O#4007 ;
G65 H03 P#2014 O15000 R#2014          G65 H01 P#107 O#4008 ;
;                                     G65 H01 P#108 O#4009 ;
G65 H04 P#2110 O3 R#2110 ;

                                      MEM **** *** ***       07:12:55

|  |  |  |  |  |  |        |  | O SRH | SRH↑ | SRH↓ | REWIND | + |
```

11.2.2 Current Block Display Screen

Displays the block currently being executed and modal data in the MEMORY or MDI mode.

Procedure for displaying the current block display screen

1 Press function key PROG .

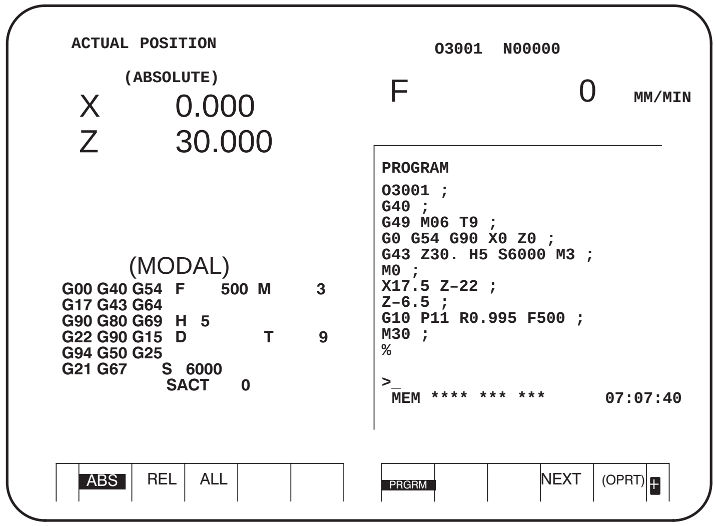
2 Press chapter selection soft key **[CURRNT]**.
The block currently being executed and modal data are displayed.
The screen displays up to 22 modal G codes and up to 11 G codes specified in the current block.

```
 PROGRAM                            O2000 N00130
        (CURRNT)     (MODAL)
    G01 ·X   100.500   G18  G00   F
        ·F    50.000   G50.2G97
                       G13.1G69
                            G99
                            G21   T
                            G40   S
                            G25
                            G22
                            G80
                            G67   SACT     0
                            G54
> _                                S   0  T0000
MEM STRT    ***       16:05:59
[ PRGRM ][ CHECK ][ CURRNT ][ NEXT ][ (OPRT) ]
```

Explanations

- **14 inch CRT screen**

The current block display screen is not provided for 14–inch CRTs. Press soft key **[PRGRM]** to display the contents of the program on the right half of the screen. The block currently being executed is indicated by the cursor. Modal data is displayed on the left half of the screen.
The screen displays up to 18 modal G codes.



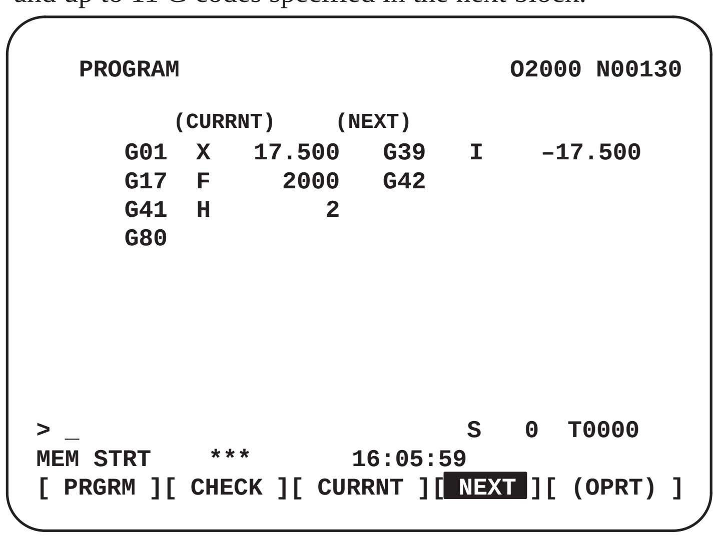
11.2.3 Next Block Display Screen

Displays the block currently being executed and the block to be executed next in the MEMORY or MDI mode.

Procedure for displaying the next block display screen

1 Press function key PROG .

2 Press chapter selection soft key **[NEXT]**.
The block currently being executed and the block to be executed next are displayed.
The screen displays up to 11 G codes specified in the current block and up to 11 G codes specified in the next block.



11.2.4 Program Check Screen

Displays the program currently being executed, current position of the tool, and modal data in the MEMORY mode.

Procedure for displaying the program check screen

1 Press function key PROG .

2 Press chapter selection soft key **[CHECK]**.
The program currently being executed, current position of the tool, and modal data are displayed.

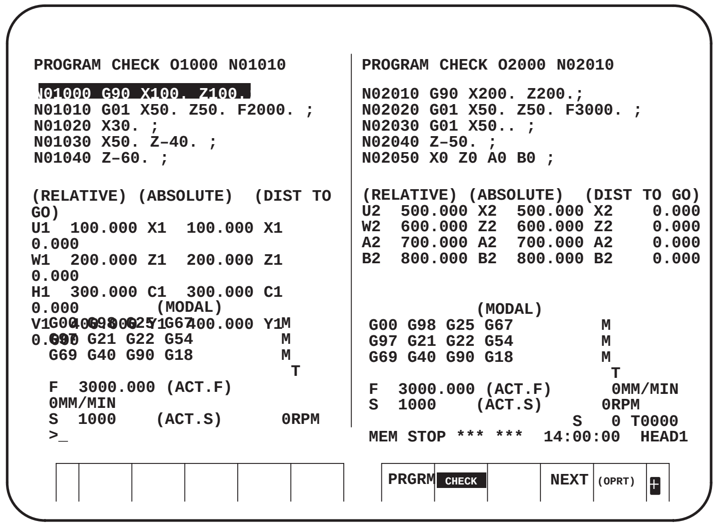
- **Display with one–path control**

```
 PROGRAM                              O2000 N00130
  O0010
  G92 G90 X100. Z50. ;
  G00 X0 Z0 ;
  G01 Z250. F1000 ;
  (ABSOLUTE)(DIST TO GO)  G00  G94  G80
  X    0.000 X    0.000  G17  G21  G98
  Z    0.000 Z    0.000  G90  G40  G50
                         G22       G67
                              B
                         H    M
     T                   D
     F              S
> _                                S   0  T0000
MEM STRT    ***        16:05:59
[ PRGRM ][ CHECK ][ CURRNT ][ NEXT ][ (OPRT) ]
```

- **Display with two–path control (9”CRT)**

```
 PROGRAM                              O2000 N00130
  O0010
  G92 G90 X100. Z50. ;
  G00 X0 Z0 ;
  G01 Z250. F1000 ;
  (ABSOLUTE)(DIST TO GO)  G00  G94  G80
  X    0.000 X    0.000  G17  G21  G98
  Z    0.000 Z    0.000  G90  G40  G50
                         G22       G67
                              B
                         H    M
     T                   D
     F              S
> _                                S   0  T0000
MEM STRT    ***        16:05:59    HEAD1
[ PRGRM ][ CHECK ][ CURRNT ][ NEXT ][ (OPRT) ]
```

- **Display with two path control (14”CRT)**



Explanations

- **Program display**

The screen displays up to four blocks (five blocks on the 14” CRT when two–path control is being used) of the current program, starting from the block currently being executed. The block currently being executed is displayed in reverse video. During DNC operation, however, only three blocks can be displayed.

- **Current position display**

The position in the workpiece coordinate system or relative coordinate system and the remaining distance are displayed. The absolute positions and relative positions are switched by soft keys **[ABS]** and **[REL]**.
On the 9” CRT when there are six or more controlled axes, pressing the **[ABS]** soft key toggles the display between the absolute coordinates for the first to fifth axes and those for the sixth to eighth axes. Pressing the **[REL]** soft key toggles the relative coordinate display in the same way.

- **Modal G codes**

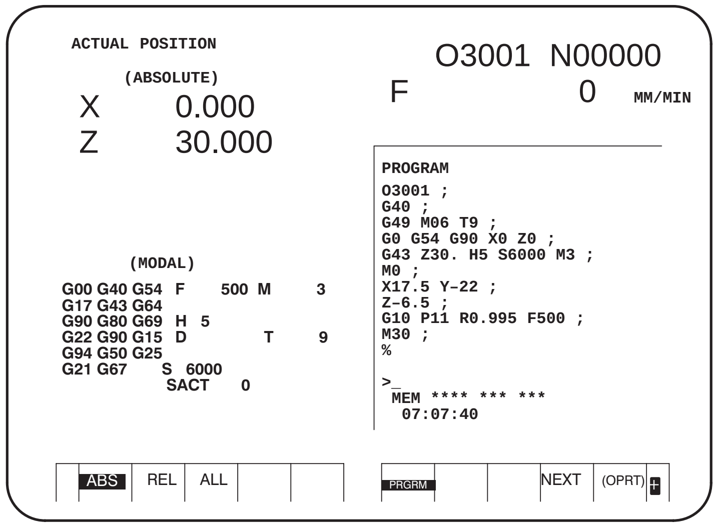
Up to 12 modal G codes are displayed.
(12 G codes for each path, on the 14” CRT when two–path control is being used)

- **Displaying during automatic operation**

During automatic operation, the actual speed, SCAT, and repeat count are displayed. The key input prompt (>_) is displayed otherwise.

- **14 inch CRT with one–path control**

The program check screen is not provided for 14–inch CRTs with one–path control. Press soft key **[PRGRM]** to display the contents of the program on the right half of the screen. The block currently being executed is indicated by the cursor. The current position of the tool and modal data are displayed on the left half of the screen.
Up to 18 modal G codes are displayed.



11.2.5 Program Screen for MDI Operation

Displays the program input from the MDI and modal data in the **MDI** mode.

Procedure for displaying the program screen for MDI operation

1 Press function key PROG .

2 Press chapter selection soft key **[MDI]**.
The program input from the MDI and modal data are displayed.

Program

Modal information

```
  PROGRAM  (MDI)                 O2000 N00130
   O0000 G00 X100.0 Z200.0 ;
   M03 ;
   G01 Z120.0 F500 ;
   M98 P9010 ;
   G00 Z0.0 ;
   %

   G00  G90  G94  G40  G80  G50  G54  G69
   G17  G22  G21  G49  G98  G67  G64  G15
                            H    M
        T                   D
        F                S
> _                           S   0  T0000
MDI **** *** ***      16:05:59
[ PRGRM ][  MDI  ][ CURRNT ][ NEXT ][ (OPRT) ]
```

Explanations

- **MDI operation**

See Section II–4.2 for MDI operation.

- **Modal information**

The modal data is displayed when bit 7 (MDL) of parameter 3107 is set to 1. Up to 16 modal G codes are displayed.
On a 14–inch CRT, however, the contents of the program are displayed on the right half of the screen and the modal data is displayed on the left half of the screen, regardless of this parameter.

- **Displaying during automatic operation**

During automatic operation, the actual speed, SCAT, and repeat count are displayed. The key input prompt (>_) is displayed otherwise.

11.2.6 Stamping the Machining Time

When a machining program is executed, the machining time of the main program is displayed on the program machining time display screen. The machining times of up to ten main programs are displayed in hours/minutes/seconds. When more than ten programs are executed, data for the oldest programs is discarded.

Procedure for Stamping Machining Time

Procedure 1 Machining time calculation and display

1 Select the memory operation mode, then press the RESET key.

2 Select the program screen, then select a program whose machining time is to be calculated.

3 Execute the program to perform actual machining.

4 When the RESET key is pressed, or M02 or M30 is executed, the machining time count operation stops. When the machining time display screen is selected, the program number of the stopped main program and its machining time are displayed.
To display the machining time display screen, use the procedure below. (Machining time data can be displayed in any mode and during background editing.)
· Press the function key PROG .
· Press the rightmost soft key once or twice to display soft key **[TIME]**.
· Press soft key **[TIME]**. The machining time display screen appears.

Machining time display screen

```
PROGRAM (TIME)                  O0010 N0002

   No.
  NO.TIME           TIME
  O0020  12H48M02S









> _
EDIT **** *** ***         16:52:13
[  TIME  ][      ][      ][      ][ (OPRT) ]
```

5 To calculate the machining times of additional programs, repeat the above procedure. The machining time display screen displays the executed main program numbers and their machining times sequentially.
Note, that machining time data cannot be displayed for more than ten main programs. When more than ten programs are executed, data for the oldest programs is discarded. The screens below show how the screen display changes from the initial state where the machining times of ten main programs (O0020, O0040, ..., and O0200) are displayed to the state where the machining time of the main program O0220 is calculated.

```
 PROGRAM (TIME)                          O0000 N0000
      NO.           TIME
     O0020        12H48M01S
     O0040         0H48M01S
     O0060         4H16M01S
     O0080         0H16M01S
     O0100         1H20M01S
     O0120         2H08M02S
     O0140         2H32M01S
     O0160         0H51M01S
     O0180        15H04M01S
     O0200         0H56M01S
>_
 EDIT **** *** ***      16:52:13
[  TIME  ][       ][       ][       ][ (OPRT) ]
```

```
 PROGRAM (TIME)                          O0000 N0000
      NO.           TIME
     O0040         0H48M01S
     O0060         4H16M01S
     O0080         0H16M01S
     O0100         1H20M01S
     O0120         2H08M02S
     O0140         2H32M01S
     O0160         0H51M01S
     O0180        15H04M01S
     O0200         0H56M01S
     O0220         0H03M01S
>_
 EDIT **** *** ***      16:52:20
[  TIME  ][       ][       ][       ][ (OPRT) ]
```

Procedure 2 Stamping machining time

1 To insert the calculated machining time of a program in a program as a comment, the machining time of the program must be displayed on the machining time display screen. Before stamping the machining time of the program, check that the machining time display screen shows the program number

2 Set the part program storage and edit mode or background edit state and select the program screen. Then select the program whose machining time is to be inserted.

3 Suppose that the machining time of O0100 is displayed on the machining time display screen.Press soft key **[(OPRT)]** to display the operation soft keys. Then, hold down the rightmost soft key until soft key **[INS–TM]** appears. When soft key **[INS–TM]** is pressed, the cursor moves to the start of the program, and the machining time of the program is inserted after the program number.

```
PROGRAM                                O0100 N0000

 O0100 ;
 N10 G92 X100. Z10. ;
 N20 S1500 M03 ;
 N30 G00 X20.5 Z5. T0101 ;
 N40 G01 X–10. F25. ;
 N50 G02 X–16.5 Z–12. R2. ;
 N60 G01 X40. ;
 N70 X42. Z–13. ;
 N80 Z–50. ;
 N90 X44. Z–51. ;
 N100 X80. ;


EDIT   *** *** *** ***     16:05:59
[  INS–TM  ][        ][        ][        ][        ]
```

```
PROGRAM                                O0100 N0000

 O0100 (001H20M01S) ;
 N10 G92 X100. Z10. ;
 N20 S1500 M03 ;
 N30 G00 X20.5 Z5. T0101 ;
 N40 G01 Z–10. F25. ;
 N50 G02 X16.5 Z–12. R2. ;
 N60 G01 X40. ;
 N70 X42. Z–13. ;
 N80 Z–50. ;
 N90 X44. Z–51. ;
 N100 X80. ;


EDIT   *** *** *** ***     16:05:59
[  INS–TM  ][        ][        ][        ][        ]
```

4 If a comment already exists in the block containing the program number of a program whose machining time is to be inserted, the machining time is inserted after the existing comment.

```
PROGRAM                                  O0100 N0000
 O0100 (SHAFT XSF001) ;
 N10 G92 X100. Z10. ;
 N20 S1500 M03 ;
 N30 G00 X20.5 Z5. T0101 ;
 N40 G01 X–10. F25. ;
 N50 G02 X16.5 Z–12. R2. ;
 N60 G01 X40. ;
     X42. Z–13. ;
 N80 Z–50. ;
 N90 X44. Z–51. ;
 N100 X80. ;

EDIT   *** *** *** ***     16:52:13
[  INS–TM  ][       ][       ][       ][       ]
```

⇩

```
PROGRAM                                  O0100 N0000

 O0100 (SHAFT XSF001)(001H20M01S) ;
 N10 G92 X100. Z10. ;
 N20 S1500 M03 ;
 N30 G00 X20.5 Z5. T0101 ;
 N40 G01 Z–10. F25. ;
 N50 G02 X16.5 Z–12. R2. ;
 N60 G01 X40. ;
 N70 X42. Z–13. ;
 N80 Z–50. ;
 N90 X44. Z–51. ;
 N100 X80. ;

EDIT   *** *** *** ***     16:52:13
[  INS–TM  ][       ][       ][       ][       ]
```

5 The machining time of a program inserted as a comment can be displayed after an existing program comment on the program directory screen.

```
PROGRAM                                O0000 N0000

   SYSTEM EDITION     B001 – 05
  PROGRAM NO. USED :   8 FREE :   55
 MEMORY AREA USED :4320 FREE : 5760
PROGRAM LIBRARY LIST
  O0020 (GEAR XGR001 ):(012H48M01S)
  O0002 (GEAR XGR002 ):(000H48M01S)
  O0010 (BOLT YBT001 ):(004H16M01S)
  O0020 (BOLT YBT002 ):(000H16M01S)
  O0040 (SHAFT XSF001  ):(001H20M01S)
  O0050 (SHAFT XSF002  ):(002H08M01S)
  O0100 (SHAFT XSF011  ):(002H32M02S)
>_O0200 (PLATE XPL100  ):(000H51M01S)
EDIT **** *** ***     14:46:09
[ PRGRM ][  LIB  ][     ][     ][ (OPRT) ]
```

Explanations

- **Machining time**

Machining time is counted from the initial start after a reset in memory operation mode to the next reset. If a reset does not occur during operation, machining time is counted from the start to M03 (or M30). However, note that the time during which operation is held is not counted, but the time used to wait for completion of M, S, T, and/or B functions is counted.

- **Stamping the machining time**

The displayed machining time can be inserted (stamped) as a comment in a program stored in memory. Machining time is inserted as a comment after the program number.

- **Program directory**

The machining time inserted after a program number can be displayed on the program directory screen by setting bit 0 (NAM) of parameter No. 3107 to 1. This lets the user know the machining time of each program. This information is useful as reference data when planning processing.

Restrictions

- **Alarm**

When program execution is terminated by an alarm during the machining time count, the machining time until the alarm is released is counted.

- **M02**

If the user specifies that M02 does not reset the CNC but returns completion signal FIN to the CNC to restart the program from the beginning successively (with bit 5 (M02) of parameter No. 3404 set to 0), the machining time count stops when M02 returns completion signal FIN.

- **Stamping the machining time**

When the machining time of a program to be stamped is not displayed on the machining time display screen, the machining time cannot be inserted into the program even if soft key **[INS–TM]** is pressed.

- **Program directory**

When the machining time inserted into a program is displayed on the program directory screen and the comment after the program number consists of only machining time data, the machining time is displayed in both the program name display field and machining time display field.
If machining time data is inserted into a program as shown below, the program directory screen does not display the data or displays only part of the data.

Example 1:Program directory screen when a program name longer than 16 characters

```
PROGRAM                                   O0100 N0000

 O0240 (SHAFT XSF301 MATERIAL=FC25)
  (001H20M01S);
 N10 G92 X100. Z10. ;
 N20 S1500 M03 ;
 N30 G00 X20.5 Z5. T0101 ;
 N40 G01 Z–10. F25. ;
 N50 G02 X16.5 Z–12. R2. ;
 N60 G01 X40. ;
 N70 X42. Z–13. ;
 N80 Z–50. ;
 N90 X44. Z–51. ;


EDIT   *** *** *** ***     16:52:13
[  INS–TM  ][       ][       ][       ][       ]
```

⇩

All characters after the first 16 characters of the program comment are discarded and the machining time display field is left blank.

```
PROGRAM                                O0000 N0000
SYSTEM EDITION     B1A1 – 05
PROGRAM NO. USED :   8 FREE :   55
MEMORY AREA USED :2480 FREE : 5760
PROGRAM LIBRARY LIST
O0240  (SHAFT XSF301   ):(          )







>_
EDIT **** *** ***     16:52:13
[ PRGRM ][  LIB  ][      ][      ][ (OPRT) ]
```

Example 2:Program directory screen when two or more machining times are stamped.

```
PROGRAM                                    O0260 N0000

  O0260 (SHAFT XSF302) (001H15M59S)
   (001H20M01S) ;
  N10 G92 X100. Z10. ;
  N20 S1500 M03 ;
  N30 G00 X20.5 Z5. T0101 ;
  N40 G01 Z–10. F25. ;
  N50 G02 X16.5 Z–12. R2. ;
  N60 G01 X40. ;
  N70 X42. Z–13. ;
  N80 Z–50. ;
  N90 X44. Z–51. ;

EDIT   *** *** *** ***     16:52:13
[  INS–TM  ][       ][       ][       ][       ]
```

Only the first machining time is displayed.

```
PROGRAM                            O0260 N0000

  SYSTEM EDITION     B1A1 – 05
  PROGRAM NO. USED :   8 FREE :   55
  MEMORY AREA USED :2480 FREE : 5760
PROGRAM LIBRARY LIST
  O0260  (SHAFT XSF302   ):(001H15M59S)

>_
EDIT **** *** ***     16:52:13
[ PRGRM ][  LIB  ][     ][     ][ (OPRT) ]
```

Example 3:Program directory screen when inserted machining time data does not conform to the format hhhHmmMssS (3–digit number followed by H, 2–digit number followed by M, and 2–digit number followed by S, in this order)

```
PROGRAM                                O0280 N0000

  O0280 (SHAFT XSF303) (1H10M59S)
  N10 G92 X100. Z10. ;
  N20 S1500 M03 ;
  N30 G00 X20.5 Z5. T0101 ;
  N40 G01 Z–10. F25. ;
  N50 G02 X16.5 Z–12. R2. ;
  N60 G01 X40. ;
  N70 X42. Z–13. ;
  N80 Z–50. ;
  N90 X44. Z–51. ;
  N100 X80. ;


EDIT   *** *** *** ***     16:52:13
[  INS–TM  ][       ][       ][       ][       ]
```

The machining time display field is blank.

```
PROGRAM                              O0280 N0000

   SYSTEM EDITION     B1A1 – 05
  PROGRAM NO. USED :   8 FREE :   55
  MEMORY AREA USED :2480 FREE : 5760
PROGRAM LIBRARY LIST
  O0260 (SHAFT XSF302   ):(001H15M59S)
  O0280 (SHAFT XSF303   ):(    )




>_
EDIT **** *** ***     16:52:13
[ PRGRM ][  LIB  ][     ][     ][ (OPRT) ]
```

- **Correcting the machining time**

If an incorrect machining time is calculated (such as when a reset occurs during program execution), reexecute the program to calculate the correct machining time. If the machining time display screen displays multiple programs with the same program number, select the machining time of the latest program number for insertion into the program.

11.2.7 Displaying the B–axis Operation State

Displaying the B–axis operation state

1 Press the PROG function key.

2 Press the **[CHECK]** chapter selection soft key.

3 Press the **[B–DSP]** chapter selection soft key. Then, the B–axis operation state is displayed on the program check screen. The command currently being executed and the next command are displayed.

```
PROGRAM CHECK                O0001 N00001
  M102 ;
  G00 X10. Z20. ;
  G01 X20. Z30. F1000 ;
  G04 P1000 ;
  (ABSOLUTE)  (B–AXIS) G00   G95   G22
  X   40.000  G01(CURR) G97   G21   G80
  Z   40.000  B –200.000G90   G40   G50
  Y    0.000  F 0.1500 G69   G25   G67
  B –125.994  G00(NEXT)
              B 250.000        M        102
     T
     F     0.1000    S
  ACT.F       0   SCAT     0S    0 T0000
MEM STRT *** FIN     21:20:05
[  ABS  ][  REL  ][  B.DSP  ][       ][ (OPRT) ]
```

11.3 SCREENS DISPLAYED BY FUNCTION KEY PROG (IN THE EDIT MODE)

This section describes the screens displayed by pressing function key PROG in the EDIT mode. Function key PROG in the EDIT mode can display the program editing screen and the program display screen (displays memory used and a list of programs). Pressing function key PROG in the EDIT mode can also display the conversational graphics programming screen and the floppy file directory screen. See Chapter 10 for the program editing screen and conversational graphics programming screen. See Chapter 9 for the floppy file directory screen.

11.3.1 Displaying Memory Used and a List of Programs

Displays the number of registered programs, memory used, and a list of registered programs.

Procedure for displaying memory used and a list of programs

1 Select the **EDIT** mode.
For the two–path control, select the tool post for which a program is to be displayed with the tool post selection switch.

2 Press function key PROG .

3 Press chapter selection soft key **[LIB]**.

```
PROGRAM                 O2000 N00130

   SYSTEM EDITION    B1A1 – 05
  PROGRAM NO.  USED :    11FREE :    52
  MEMORY AREA  USED :  1200FREE :  4320
PROGRAM LIBRARY LIST
 O0010 O0001 O0003 O0002 O0555 O0999
 O0062 O0004 O0005 O1111 O0969 O6666
 O0021 O1234 O0588 O0020 O0040




> _                  S  0 T0000
MDI **** *** ***    16:05:59
[ PRGRM ][ LIB ][      ][ C.A.P. ][ (OPRT) ]
```

Explanations

- **Details of memory used**

PROGRAM NO. USED

PROGRAM NO. USED : The number of the programs registered (including the subprograms)

FREE : The number of programs which can be registered additionally.

MEMORY AREA USED

MEMORY AREA USED : The capacity of the program memory in which data is registered (indicated by the number of characters).

FREE : The capacity of the program memory which can be used additionally (indicated by the number of characters).

- **Program library list**

Program Nos. registered are indicated.
Also, the program name can be displayed in the program table by setting parameter NAM (No. 3107#0) to 1.

```
 PROGRAM                        O2000 N00130
     SYSTEM EDITION    B1A1 – 05
   PROGRAM NO.  USED :    11FREE :    52
   MEMORY AREA  USED :  1200FREE :  4320
 PROGRAM LIBRARY LIST
   O0001 (MACRO–GCODE.MAIN)
   O0002 (MACRO–GCODE.SUB1)
   O0010 (TEST–PROGRAM.ARTHMETIC NO.1)
   O0020 (TEST–PROGRAM.F10–MACRO)
   O0040 (TEST–PROGRAM.OFFSET)
   O0050
   O0100 (INCH/MM CONVERT CHECK NO.1)
   O0200 (MACRO–MCODE.MAIN)
> _
EDIT **** *** ***     16:05:59
[ PRGRM ][  LIB  ][      ][ C.A.P. ][ (OPRT) ]
```

- **Program name**

Always enter a program name between the control out and control in codes immediately after the program number.
Up to 31 characters can be used for naming a program within the parentheses. If 31 characters are exceeded, the exceeded characters are not displayed.
Only program number is displayed for the program without any program name.

○ □□□□ (○○○○…○) ;

Program number Program name (up to 31 characters)

- **Software series**

Software series of the system is displayed.
It is used for maintenance ; user is not required this information.

- **Order in which programs are displayed in the program library list**

Programs are displayed in the same order that they are registered in the program library list. However, if bit 4 (SOR) of parameter 3107 is set to 1, programs are displayed in the order of program number starting from the smallest one.

- **Order in which programs are registered**

Immediately after all programs are cleared (by turning on the power while pressing the DELETE key), each program is registered after the last program in the list.
If some programs in the list were deleted, then a new program is registered, the new program is inserted in the empty location in the list created by the deleted programs.

Example) When bit 4 (SOR) of parameter 3107 is 0

1. **After clearing all programs, register programs O0001, O0002, O0003, O0004, and O0005 in this order. The program library list displays the programs in the following order:**
   O0001, O0002, O0003, O0004, O0005

2. **Delete O0002 and O0004. The program library list displays the programs in the following order:**
   O0001, O0003, O0005

3. **Register O0009. The program library list displays the programs in the following order:**
   O0001, O0009, O0003, O0005

11.3.2 Two–path simultaneous editing on the program screen

In two–path control, the programs for both tool posts can be displayed and edited on the same screen when bit 0 (DHD) of parameter No. 3106 is set to 1.
The name of each tool post is displayed above the corresponding program.

Procedure for Two–path Simultaneous Editing on the Program Screen

1 Specify **EDIT** mode for both tool posts.

2 Press function PROG key.

Shared screen (9” CRT)

```
PROGRAM    O1357 N00120          O2468 N00130
(HEAD1)                    (HEAD2)
O1357 (HEAD–1 MAIN PROGRAM);O2468 (HEAD–2 MAIN PROGRAM);
N010 G90 G00 X200.0 Z220.0 N010 G90 G00 X200.0 Z220.0
;                          ;
N020 T0101 ;               N020 T0101 ;
N030 S30000 M03 ;          N030 S30000 M03 ;
N040 G40 G00 X40.0 Z180.0 ;N040 G41 G00 X40.0 Z180.0 ;
N050 G01 Z140.0 F1000.0 ;  N050 G01 Z140.0 F1000.0 ;
N060 X60.0 Z110.0 ;        N060 X60.0 Z110.0 ;
N070 Z90.0 ;               N070 Z90.0 ;
N080 X100.0 Z80.0 ;        N080 X100.0 Z80.0 ;
N090 Z60.0 ;               N090 Z60.0 ;
N100 X140.0 Z40.0 ;        N100 X140.0 Z40.0 ;
N110 X200.0 Z220.0 ;       N110 X200.0 Z220.0 ;
N120 S0 M05                N120 T0100 ;
                           N130 T0102 ;
                           N140 S1000 ;
>N130T0100;M30;            N150 G41 G00 X40.0 Z180.0 ;
  EDIT **** *** *** 16:05:59 HEAD1
[ BG–EDT ][ O SRH ][ SRH + ][ SRH – ][ REWIND ]
```

Shared screen (14” CRT)

```
PROGRAM            O1234 N00010                           O2345 N00100
 (HEAD1)                           (HEAD2)

  O1234 ;                           O2345;
  N10  G00 ;                        N100 G00 ;
  N20  X100.0 ;                     N200 X0 ;
  N30  X200.0 ;                     N300 X50.0 ;
  N40  X300.0 Z300.0 ;              N400 M02 ;
  N50  X400.0 ;                     %
  N60  X500.0 ;
  N70  M02 ;
  %

>_
     EDIT STRT MIN FIN ALM 17:25:01 HEAD1
  [  ][  ][  ][  ][  ][PRGRM][ LIB ][  ][  ][(OPR)][  ]
```

Explanations

- **Shared screen and individual screen**

When the selected tool post is in **EDIT** mode, pressing the **[PRGRM]** soft key displays a shared screen which shows the program for the first tool post on the left and that for the second tool post on the right. However, if the tool post that is not selected fails to satisfy any of the conditions described below, only the individual screen for the selected tool post is displayed.

<Conditions>
- Bit 0 (DHD) of parameter No. 3106 is set to 1.
- The program screen for the selected tool post is the 14” type (when the 14” CRT is being used).
- Both tool posts are set to **EDIT** mode.
- Background editing is not specified for either tool post.

When the mode for the tool post that is not selected is changed from **EDIT** mode on the shared screen, the individual screen (14” type when the 14” CRT is being used) for the selected tool post is displayed.
On the 14” CRT, pressing the **[PRGRM]** soft key toggles between the individual screen (9” type) and the shared screen.

Individual screen (9” CRT)

```
PROGRAM              O1234 N00010                           O2345 N00100
(HEAD1)                                 (HEAD2)
 O1234 ;                                 O2345;
 N10  G00 ;                              N100 G00 ;
 N20  X100.0 ;                           N200 X0 ;
 N30  X200.0 ;                           N300 X50.0 ;
 N40  X300.0 Z300.0 ;                    N400 M02 ;
 N50  X400.0 ;                           %
 N60  X500.0 ;
 N70  M02 ;
 %

>_
    EDIT STRT MIN FIN ALM 17:25:01 HEAD1
  [  ][  ][  ][  ][  ][  ][PRGRM][ LIB ][  ][  ][(OPR)][  ]
```

Individual screen (14” type screen on the 14”CRT)

```
PROGRAM                                     O1234 N00010

 O1234 ;
 N10 G00 ;
 N20 X100.0 ;
 N30 X200.0 ;
 N40 X300.0 Z300.0 ;
 N50 X400.0 ;
 N60 X500.0 ;
 N70 M02 ;
 %

>_
    EDIT STRT MIN FIN ALM 17:25:01 HEAD1
  [  ][  ][  ][  ][  ][  ][PRGRM][ LIB ][  ][  ][(OPR)][  ]
```

Individual screen (9” type screen on the 14”CRT)

```
ACTUAL POSITION                           O1234N00010
                                           F  1000 MM/M
(ABSOLUTE)    (RELATIVE)
X  0.000      X  0.000
Y  0.000      Y  0.000
Z  0.000      Z  0.000
A  0.000      A  0.000               PROGRAM   O1234 N00010
B  0.000      B  0.000               O1234 ;
                                     N10  G00 ;
                                     N20  X100.;
(MACHINE)                            N30  X200.;
X  0.000                             N40  X300. Z300.;
Y  0.000                             N50  X400.;
Z  0.000                             N60  X500.;
A  0.000                             N70  M02 ;
B  0.000                             %

G00  G25
G97  G22                             >_
G67  G80
G99  G67
G21  G54
G40  G18  SCAT

EDIT STRT MIN FIN ALM 17:25:01 HEAD1
[  ][ABS][REL][ALL][  ][  ][PRGRM][ LIB ][  ][  ][(OPR)][  ]
```

- **Editing operation**

Editing is enabled only for the program for the selected tool post. The program for the first or second tool post can be edited on the same screen by selecting either tool post with the tool post selection signal.

- **9" CRT shared screen**

On the 9" CRT, the shared screen consists of 80 digits x 25 lines. If the tool post name specified with parameter No. 3131 contains a character other than alphanumeric and special characters ("# $ % & ' () * + , – . / : ; < = > ? @ [\ ] ^ _ and space), the character will not be displayed correctly. In such a case, operation soft keys **[SRH ↑]** and **[SRH ↓]** are displayed as **[SRH +]** and **[SRH –]**.

Limitations

This function cannot be used for background editing.

11.4 SCREENS DISPLAYED BY FUNCTION KEY OFFSET SETTING

Press function key OFFSET SETTING to display or set tool compensation values and other data.

This section describes how to display or set the following data:

1. Tool offset value
2. Settings
3. Run time and part count
4. Workpiece origin offset value or workpiece coordinate system shift value
5. Custom macro common variables
6. Software operator's panel
7. Tool life management data

This section also describes following functions.

- Direct input of tool offset value
- Direct input of tool offset value measured B
- Counter input of offset value
- Direct input of workpiece coordinate system shift
- Y axis offset
- Sequence number comparison and stop function

The following functions depend on the specifications of the machine tool builder. See the manual issued by the machine tool builder for details.

- Direct input of tool offset value
- Direct input of tool offset value measured B
- Software operator's panel
- Tool life management data

11.4.1 Setting and Displaying the Tool Offset Value

Dedicated screens are provided for displaying and setting tool offset values and tool nose radius compensation values.

Procedure for setting and displaying the tool offset value and the tool nose radius compensation value

1 Press function key [OFFSET SETTING] .

For the two–path control, select the tool post for which tool compensation values are to be displayed with the tool post selection switch.

2 Press chapter selection soft key **[OFFSET]** or press [OFFSET SETTING] several times until the tool compensation screen is displayed.
Different screens are displayed depending on whether tool geometry offset, wear offset, or neither is applied.

```
 OFFSET                                   O0001 N00000
    NO.         X          Z.          R      T
    001       0.000     10.000       0.000    0
    002       0.000      0.000       0.000    0
    003       0.000      0.000       0.000    0
    004      40.000    -40.000       0.000    0
    005       0.000      0.000       0.000    0
    006       0.000      0.000       0.000    0
    007       0.000      0.000       0.000    0
    008       0.000      0.000       0.000    0
 ACTUAL POSITION (RELATIVE)
     U   101.000             W   202.094

> _
MDI **** *** ***       16:05:59
[ OFFSET ][ SETING ][ WORK ][     ][ (OPRT) ]
```

Without tool geometry/wear offset

```
 OFFSET/GEOMETRY                          O0001 N00000
    NO.         X          Z.          R      T
  G 001       0.000      1.000       0.000    0
  G 002       1.486    -49.561       0.000    0
  G 003       1.486    -49.561       0.000    0
  G 004       1.486      0.000       0.000    0
  G 005       1.486    -49.561       0.000    0
  G 006       1.486    -49.561       0.000    0
  G 007       1.486    -49.561       0.000    0
  G 008       1.486    -49.561       0.000    0
 ACTUAL POSITION (RELATIVE)
     U   101.000             W   202.094

> _
MDI **** *** ***       16:05:59
[  WEAR  ][  GEOM  ][ WORK ][     ][ (OPRT) ]
```

With tool geometry offset

```
OFFSET/WEAR                              O0001 N00000
     NO.          X          Z.          R      T
   W 001        0.000      1.000       0.000   0
   W 002        1.486    –49.561       0.000   0
   W 003        1.486    –49.561       0.000   0
   W 004        1.486      0.000       0.000   0
   W 005        1.486    –49.561       0.000   0
   W 006        1.486    –49.561       0.000   0
   W 007        1.486    –49.561       0.000   0
   W 008        1.486    –49.561       0.000   0
 ACTUAL POSITION (RELATIVE)
     U   101.000             W   202.094

> _
MDI **** *** ***       16:05:59
[  WEAR  ][  GEOM  ][ WORK ][     ][ (OPRT) ]
```

With tool wear offset

3 Move the cursor to the compensation value to be set or changed using page keys and cursor keys, or enter the compensation number for the compensation value to be set or changed and press soft key **[NO.SRH]**.

4 To set a compensation value, enter a value and press soft key **[INPUT]**. To change the compensation value, enter a value to add to the current value (a negative value to reduce the current value) and press soft key **[+INPUT]**. Or, enter a new value and press soft key **[INPUT]**.
TIP is the number of the virtual tool tip (see Programming).
TIP may be specified on the geometry compensation screen or on the wear compensation screen.

Explanations

• Decimal point input

A decimal point can be used when entering a compensation value.

• Other method

An external input/output device can be used to input or output a cutter compensation value. See Chapter III–8.
Tool length compensation values can be set using the following functions described in subsequent subsections: direct input of tool offset value, direct–input function B for tool offset measured, and counter input of offset value.

• Tool offset memory

16 groups are provided for tool compensation. The number of groups can be optionally extended to 32, 64, or 99. For the two–path control, the above number of groups can be used for each tool post. Tool geometry compensation or wear compensation can be selected for each group.

- **Disabling entry of compensation values**

In some cases, tool wear compensation or tool geometry compensation values cannot be input because of the settings in bits 0 (WOF) and 1 (GOF) of parameter 3290. The input of tool compensation values from the MDI can be inhibited for a specified range of offset numbers. The first offset number for which the input of a value is inhibited is set in parameter No. 3294. The number of offset numbers, starting from the specified first number, for which the input of a value is inhibited is set in parameter No. 3295.
Consecutive input values are set as follows:

1) When values are input for offset numbers, starting from one for which input is not inhibited to one for which input is inhibited, a warning is issued and values are set only for those offset numbers for which input is not inhibited.
2) When values are input for offset numbers, starting from one for which input is inhibited to one for which input is not inhibited, a warning is issued and no values are set.

- **Displaying radius and TIP**

The radius and TIP are not displayed if the tool tip radius compensation option is not displayed.

- **Changing offset values during automatic operation**

When offset values have been changed during automatic operation, bit 4 (LGT) and bit 6 (LWM) of parameter 5002 can be used for specifying whether new offset values become valid in the next move command or in the next T code command.

LGT	LWM	When geometry compensation values and wear compensation values are separately specified	When geometry compensation values and wear compensation values are not separately specified
0	0	Become valid in the next T code block	Become valid in the next T code block
1	0	Become valid in the next T code block	Become valid in the next T code block
0	1	Become valid in the next T code block	Become valid in the next move command
1	1	Become valid in the next move command	Become valid in the next move command

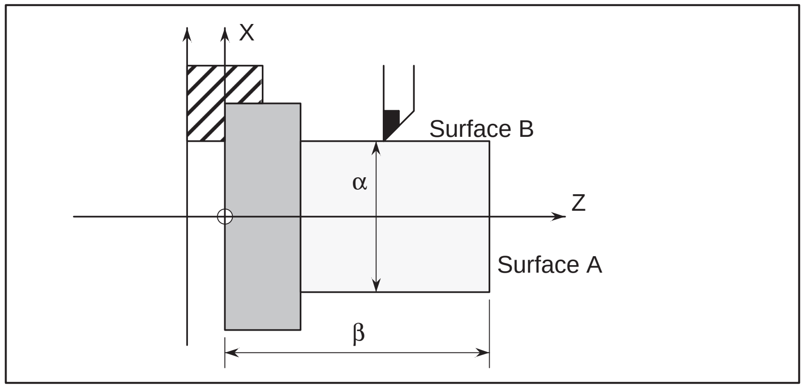
11.4.2 Direct Input of Tool Offset Value

To set the difference between the tool reference position used in programming (the nose of the standard tool, turret center, etc.) and the tool tip position of a tool actually used as an offset value

Procedure for direct input of tool offset value

- **Setting of Z axis offset value**

1 Cut surface A in manual mode with an actual tool.
Suppose that a workpiece coordinate system has been set.



2 Release the tool in X axis direction only, without moving Z axis and stop the spindle.

3 Measure distance β from the zero point in the workpiece coordinate system to surface A.
Set this value as the measured value along the Z–axis for the desired offset number, using the following procedure:

```
OFFSET/GEOMETRY                        O0001 N00000
   NO.          X           Z.           R      T
  G 001       0.000       1.000        0.000   0
  G 002       1.486     -49.561        0.000   0
  G 003       1.486     -49.561        0.000   0
  G 004       1.486       0.000        0.000   0
  G 005       1.486     -49.561        0.000   0
  G 006       1.486     -49.561        0.000   0
  G 007       1.486     -49.561        0.000   0
  G 008       1.486     -49.561        0.000   0
 ACTUAL POSITION (RELATIVE)
     U     0.000            W     0.000
     V     0.000            H     0.000

>MZ120._
MDI **** *** ***      16:05:59
[NO,SRH][ MEASUR ][ INP.C. ][ +INPUT ][ INPUT ]
```

3–1 Press the function key OFFSET SETTING or the soft key **[OFFSET]** to display the tool compensation screen. If geometry compensation values and wear compensation values are separately specified, display the screen for either of them.

3–2 Move the cursor to the set offset number using cursor keys.

3–3 Press the address key Z to be set.

3–4 Key in the measured value (β).

3–5 Press the soft key **[MESURE]**.
The difference between measured value β and the coordinate is set as the offset value.

- **Setting of X axis offset value**

4 Cut surface B in manual mode.

5 Release the tool in the Z–axis direction without moving the X–axis and stop the spindle.

6 Measure the diameter α of surface B.
Set this value as the measured value along the X–axis for the desired offset number in the same way as when setting the value along the Z–axis.

7 Repeat above procedure the same time as the number of the necessary tools. The offset value is automatically calculated and set.
For example, in case α=69.0 when the coordinate value of surface B in the diagram above is 70.0, set 69.0 **[MEASURE]** at offset No.2.
In this case, 1.0 is set as the X–axis offset value to offset No.2.

Explanations

- **Compensation values for a program created in diameter programming**

Enter diameter values for the compensation values for axes for which diameter programming is used.

- **Tool geometry offset value and tool wear offset value**

If measured values are set on the tool geometry compensation screen, all compensation values become geometry compensation values and all wear compensation values are set to 0. If measured values are set on the tool wear compensation screen, the differences between the measured compensation values and the current wear compensation values become the new compensation values.

- **Retracting along two axes**

If a record button is provided on the machine, the tool can retract along two axes when bit 2 (PRC) of parameter 5005 is set and the record signal is used. Refer to the appropriate manual issued by the machine tool builder.

11.4.3 Direct Input of tool offset measured B

The direct input function B for tool offset measured is used to set tool compensation values and workpiece coordinate system shift values.

Procedure for setting the tool offset value

Tool position offset values can be automatically set by manually moving the tool until it touches the sensor.
Refer to the appropriate manual issued by the machine tool builder for actual operation.

1 Execute manual reference position return.
By executing manual reference position return, a machine coordinate system is established.
The tool offset value is computed on the machine coordinate system.

2 Set the offset writing mode signal GOQSM to HIGH.
(Refer to the appropriate manual issued by the machine tool builder for actual operation.)
The CRT display is automatically changed to the tool offset screen (geometry), and the ”OFST” indicator starts blinking in the status indication area in the bottom of the screen, which informs that the offset writing mode is ready.

3 Select a tool to be measured.

4 When the cursor is not coincided with the tool offset number desired to be set, move the cursor to the desired offset number by page key and cursor key.
Besides the cursor can also be coincided with the tool offset number desired to set automatically by the tool offset number input signals (when parameter QNI(No.5005#5)=1).
In this case, the position of the cursor cannot be changed on the tool compensation screen using page keys or cursor keys.

5 Near the tool to the sensor by manual operation.

6 Place the tool edge to a contacting surface of the sensor by manual handle feed.
Bring the tool edge in contact with the sensor. This causes the offset writing signals (+MIT1, –MIT1, +MIT2 or –MIT2) to input to CNC.
The offset writing signal is set to HIGH, and the :

- The axis is interlocked in this direction and its feeding is stopped.
- The tool offset value extracted by the tool offset memory (tool geometry offset value) which corresponds to the offset number shown by the cursor is set up.

7 For both X–axis and Z–axis, their offset value are set by the operations **5** and **6**.

8 Repeat operations **3** to **7** for necessary tools.

9 Set the offset writing signal mode GOQSM to LOW.
The writing mode is canceled and the blinking ”OFST” indicator light goes off.

Procedure for setting the work coordinate system shift amount

Tool position offset values can be automatically set by manually moving the tool until it touches the sensor.
Refer to the appropriate manual issued by the machine tool builder for actual operation.

1 The tool compensation values are then calculated based on the machine coordinates of the tool.

2 Execute manual reference position return.
By executing manual reference position return, the machine coordinate system is established.
The workpiece coordinate system shifting amount is computed based on the machine coordinate system of the tool.

3 Set the workpiece coordinate system shifting amount writing signal mode WOQSM to HIGH.
(Refer to the appropriate manual issued by the machine tool builder for actual operation.)
The CRT display is automatically switches to the workpiece shifting screen, the ”WFST” indicator starts blinking at the status indicator area in the bottom of the screen, which inform that the workpiece coordinate system shifting amount writing mode is ready.

4 Select a tool to be measured.

5 Check tool offset numbers.
The tool offset number corresponding to the tool required for measurement, shall be set in the parameter (No.5020) in advance.
Besides the tool offset number can be set automatically by setting the tool offset number input signal (with parameter QNI(No.5005#5)=1).
Refer to the appropriate manual issued by the machine tool builder for details.

6 Manually approach the tool to an end face of the workpiece.

7 Place the tool edge to the end face (sensor) of the workpiece by manual handle feed.
The workpiece coordinate system shifting amount on the Z–axis is automatically set.

8 Feed the tool.

9 Set the workpiece coordinate system shifting amount writing signal mode WOQSM to LOW.
The writing mode is canceled and the blinking ”WSFT” indicator light goes off.
(Refer to the appropriate manual issued by the machine tool builder for actual operation.)

11.4.4 Counter Input of Offset value

By moving the tool until it reaches the desired reference position, the corresponding tool offset value can be set.

Procedure for counter input of offset value

1 Manually move the reference tool to the reference position.

2 Reset the relative coordinates along the axes to 0 (see subsec. III–11.1.2).

3 Move the tool for which offset values are to be set to the reference position.

4 Select the tool compensation screen. Move the cursor to the offset value to be set using cursor keys.

```
  OFFSET/GEOMETRY                     O0001 N00000
     NO.         X          Z.          R      T
   G 001       0.000      1.000       0.000   0
   G 002       1.486    –49.561       0.000   0
   G 003       1.486    –49.561       0.000   0
   G 004       1.486      0.000       0.000   0
   G 005       1.486    –49.561       0.000   0
   G 006       1.486    –49.561       0.000   0
   G 007       1.486    –49.561       0.000   0
   G 008       1.486    –49.561       0.000   0
  ACTUAL POSITION (RELATIVE)
      U     0.000            W     0.000
      V     0.000            H     0.000
>X_
HND **** *** ***      16:05:59
[NO,SRH][ MEASUR ][ INP.C. ][ +INPUT ][ INPUT ]
```

5 Press address key X (or Z) and the soft key **[INP.C.]**.

Explanations

- **Geometry offset and wear offset**

When the above operations are performed on the tool geometry compensation screen, tool geometry compensation values are input and tool wear compensation values do not change.
When the above operations are performed on the tool wear compensation screen, tool wear compensation values are input and tool geometry compensation values do not change.

11.4.5 Setting the Workpiece Coordinate System Shifting Amount

The set coordinate system can be shifted when the coordinate system which has been set by a G50 command (or G92 command for G code system B or C) or automatic coordinate system setting is different from the workpiece coordinate system assumed at programming.

Procedure for setting the workpiece coordinate system shifting amount

1 Press function key OFFSET SETTING .

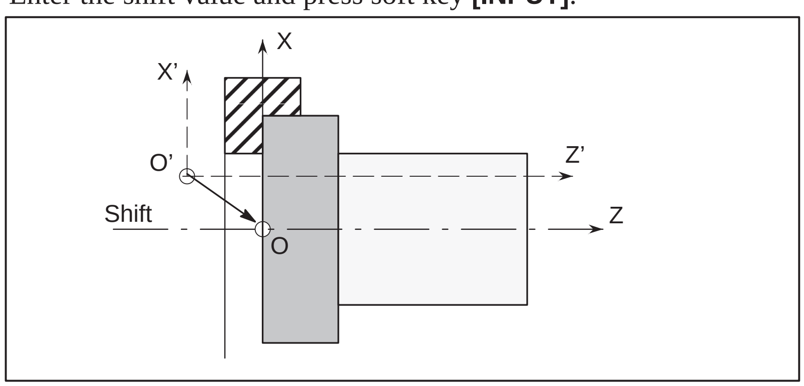
2 Press the continuous menu key ▷ several times until the screen with soft key **[WK.SHFT]** is displayed.

```
 WORK SHIFT                             O0001 N00000

    (SHIFT VALUE) (MEASUREMENT)
     X     0.000   X     0.000
     Z     0.000   Z     0.000




 ACTUAL POSITION (RELATIVE)
     U     0.000            W     0.000


> MZ100._                               S   0  T0000
MDI **** *** ***      16:05:59
[      ][ WK.SHFT ][      ][ +INPUT ][ INPUT ]
```

3 Press soft key **[WK.SHFT]**.

4 Move the cursor using cursor keys to the axis along which the coordinate system is to be shifted.

5 Enter the shift value and press soft key **[INPUT]**.



Explanations

- **When shift values become valid**

Shift values become valid immediately after they are set.

- **Shift values and coordinate system setting command**

Setting a command (G50 or G92) for setting a coordinate system disables the set shift values.

Example When G50 X100.0 Z80.0; is specified, the coordinate system is set so that the current tool reference position is X = 100.0, Z = 80.0 regardless of the shift values.

- **Shift values and coordinate system setting**

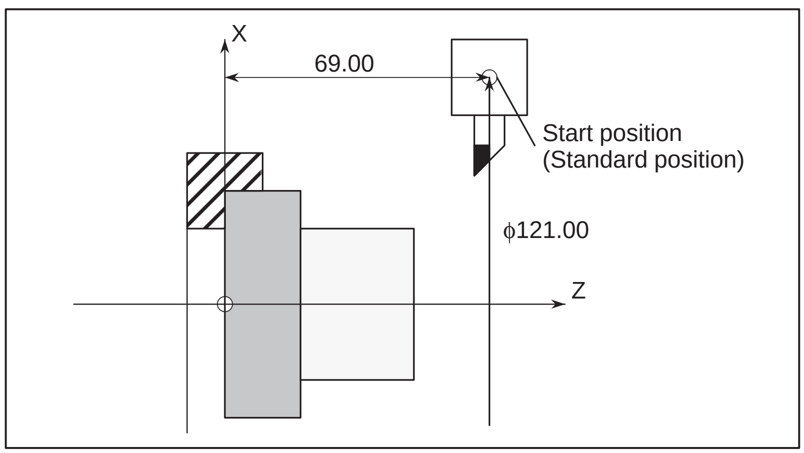
If the automatic coordinate system setting is performed by manual reference position return after shift amount setting, the coordinate system is shifted instantly.

- **Diameter or radius value**

Whether the shift amount on the X axis is diameter or radius value depends on that specified in program.

Examples

When the actual position of the reference point is X = 121.0 (diameter), Z = 69.0 with respect to the workpiece origin but it should be X = 120.0, Z = 70.0, set the following shift values:
X=1.0, Z=–1.0



11.4.6 Y Axis Offset

Tool position offset values along the Y–axis can be set. Counter input of offset values is also possible.
Direct input of tool offset value and direct input function B for tool offset measured are not available for the Y–axis.

Procedure for setting the tool offset value of the Y axis

1 Press function key [OFFSET SETTING] .

2 Press the continuous menu key [▷] several times until the screen with soft key **[OFST.2]** is displayed.

3 Press soft key **[OFST.2]**.
The Y axis offset screen is displayed.

```
OFFSET                                   O0001 N00000
   NO.       Y
   01      10.000
   02       0.000
   03       0.000
   04      40.000
   05       0.000
   06       0.000
   07       0.000
   08       0.000
ACTUAL POSITION (RELATIVE)
     U   100.000         W   100.000

>_
 MDI **** *** ***       16:05:59
[ OFST.2 ][ W.SHFT ][      ][       ][ (OPRT) ]
```

3–1 Press soft key **[GEOM]** to display the tool geometry compensation values along the Y–axis.

```
OFFSET/GEOMETRY                          O0001 N00000
   NO.       Y
 G 01      10.000
 G 02       0.000
 G 03       0.000
 G 04      40.000
 G 05       0.000
 G 06       0.000
 G 07       0.000
 G 08       0.000
ACTUAL POSITION (RELATIVE)
     U   100.000         W   100.000

>_
 MDI **** *** ***       16:05:59
[  WEAR  ][  GEOM  ][      ][       ][ (OPRT) ]
```

3–2 Press soft key **[WEAR]** to display the tool wear compensation values along the Y–axis.

```
OFFSET/WEAR                        O0001 N00000
   NO.      Y
  W 01     10.000
  W 02      0.000
  W 03      0.000
  W 04     40.000
  W 05      0.000
  W 06      0.000
  W 07      0.000
  W 08      0.000
ACTUAL POSITION (RELATIVE)
    U   100.000         W   100.000
>_
MDI **** *** ***      16:05:59
[ WEAR ][  GEOM  ][      ][      ][ (OPRT) ]
```

4 Position the cursor at the offset number to be changed by using either of the following methods:

- Move the cursor to the offset number to be changed using page keys and cursor keys.
- Type the offset number and press soft key **[NO.SRH]**.

5 Type the offset value.

6 Press soft key **[WEAR]**. The offset value is set and displayed.

```
OFFSET/WEAR                        O0001 N00000
   NO.      Y
  W 01     10.000
  W 02      0.000
  W 03      0.000
  W 04     40.000
  W 05      0.000
  W 06      0.000
  W 07      0.000
  W 08      0.000
ACTUAL POSITION (RELATIVE)
    U   100.000         W   100.000
>_
MDI **** *** ***      16:05:59
[ NO.SRH ][ MEASUR ][ INP.C. ][ +INPUT ][ INPUT ]
```

Procedure for counter input of the offset value

To set relative coordinates along the Y–axis as offset values:

1 Move the reference tool to the reference point.

2 Reset relative coordinate Y to 0 (see subsec. III–11.1.2).

3 Move the tool for which offset values are to be set to the reference point.

4 Move the cursor to the value for the offset number to be set, press Y , then press soft key **[INP.C.]**.
Relative coordinate Y (or V) is now set as the offset value.

11.4.7 Displaying and Entering Setting Data

Data such as the TV check flag and punch code is set on the setting data screen. On this screen, the operator can also enable/disable parameter writing, enable/disable the automatic insertion of sequence numbers in program editing, and perform settings for the sequence number comparison and stop function.

See Chapter III–10 for automatic insertion of sequence numbers.

See subsection III–11.4.8 for the sequence number comparison and stop function. This subsection describes how to set data.

Procedure for setting the setting data

1 Select the **MDI** mode.

2 Press function key [OFFSET SETTING] .

3 Press soft key **[SETING]** to display the setting data screen.
This screen consists of several pages.
Press page key [PAGE ↑] or [PAGE ↓] until the desired screen is displayed.
An example of the setting data screen is shown below.

```
SETTING (HANDY)                    O0001 N00000

  PARAMETER WRITE = 1(0:DISABLE 1:ENABLE)
  TV CHECK = 0(0:OFF   1:ON)
  PUNCH CODE = 1(0:EIA   1:ISO)
  INPUT UNIT = 0(0:MM    1:INCH)
  I/O CHANNEL =  0(0–3:CHANNEL NO.)
  SEQUENCE NO. = 0(0:OFF   1:ON)
  TAPE FORMAT  = 0(0:NO CNV  1:F15)
  SEQUENCE STOP =     0 (PROGRAM NO.)
  SEQUENCE STOP =     0 (SEQUENCE NO.)


> _
MDI **** *** ***        16:05:59
[ OFFSET ][ SETING ][ WORK ][     ][ (OPRT) ]
```

```
SETTING (HANDY)                    O0001 N00000

  MIRROR IMAGE X= 0 (0:OFF   1:ON)
  MIRROR IMAGE Z= 0 (0:OFF   1:ON)


> _
MDI **** *** ***        16:05:59
[ OFFSET ][ SETING ][ WORK ][     ][ (OPRT) ]
```

4 Move the cursor to the item to be changed by pressing cursor keys [↑] , [↓] , [←] , or [→] .

5 Enter a new value and press soft key **[INPUT]**.

Contents of settings

- **PARAMETER WRITE**

Setting whether parameter writing is enabled or disabled.
0 : Disabled
1 : Enabled

- **TV CHECK**

Setting to perform TV check.
0 : No TV check
1 : Perform TV check

- **PUNCH CODE**

Setting code when data is output through reader puncher interface.
0 : EIA code output
1 : ISO code output

- **INPUT UNIT**

Setting a program input unit, inch or metric system
0 : Metric
1 : Inch

- **I/O CHANNEL**

Using channel of reader/puncher interface.
0 : Channel 0
1 : Channel 1
2 : Channel 2
3 : Channel 3

- **SEQUENCE STOP**

Setting of whether to perform automatic insertion of the sequence number or not at program edit in the EDIT mode.
0 : Does not perform automatic sequence number insertion.
1 : Perform automatic sequence number insertion.

- **TAPE FORMAT**

Setting the F15 tape format conversion.
0 : Tape format is not converted.
1 : Tape format is converted.
See PROGRAMMING for the F15 tape format.

- **SEQUENCE STOP**

Setting the sequence number with which the operation stops for the sequence number comparison and stop function and the number of the program to which the sequence number belongs

- **MIRROR IMAGE**

Setting of mirror image ON/OFF for each axes.
0 : Mirror image off
1 : Mirror image on

- **Others**

Page key [PAGE ↑] or [PAGE ↓] can also be pressed to display the SETTING (TIMER) screen. See subsection III–11.4.9 for this screen.

11.4.8 Sequence Number Comparison and Stop

If a block containing a specified sequence number appears in the program being executed, operation enters single block mode after the block is executed.

Procedure for sequence number comparison and stop

1 Select the **MDI** mode.

2 Press function key OFFSET SETTING .

3 Press chapter selection soft key **[SETING]**.

4 Press page key PAGE ↑ or PAGE ↓ several times until the following screen is displayed.

```
SETTING (HANDY)                      O0001 N00000

  PARAMETER WRITE = 1 (0:DISABLE 1:ENABLE)
  TV CHECK        = 0 (0:OFF   1:ON)
  PUNCH CODE      = 1 (0:EIA   1:ISO)
  INPUT UNIT      = 0 (0:MM    1:INCH)
  I/O CHANNEL     =  0 (0–3:CHANNEL NO.)
  SEQUENCE NO.    = 0 (0:OFF   1:ON)
  TAPE FORMAT     = 0 (0:NO CNV  1:F10/11)
  SEQUENCE STOP   =     0 (PROGRAM NO.)
  SEQUENCE STOP   =    11 (SEQUENCE NO.)


> _
MDI **** *** ***       16:05:59
[ OFFSET ][ SETING ][ WORK ][     ][ (OPRT) ]
```

5 Enter in (PROGRAM NO.) for SEQUENCE STOP the number (1 to 9999) of the program containing the sequence number with which operation stops.

6 Enter in (SEQUENCE NO.) for SEQUENCE STOP (with five or less digits) the sequence number with which operation is stopped.

7 When automatic operation is executed, operation enters single block mode at the block containing the sequence number which has been set.

Explanations

- **Sequence number after the program is executed**

After the specified sequence number is found during the execution of the program, the sequence number set for sequence number compensation and stop is decremented by one. When the power is turned on, the setting of the sequence number is 0.

- **Exceptional blocks**

If the predetermined sequence number is found in a block in which all commands are those to be processed within the CNC control unit, the execution does not stop at that block.

Example

```
N1 #1=1 ;
N2 IF [#1 EQ 1] GOTO 08 ;
N3 GOTO 09 ;
N4 M98 P1000 ;
N5 M99 ;
```

In the example shown above, if the predetermined sequence number is found, the execution of the program does not stop.

- **Stop in the canned cycle**

If the predetermined sequence number is found in a block which has a canned–cycle command, the execution of the program stops after the return operation is completed.

- **When the same sequence number is found several times in the program**

If the predetermined sequence number appears twice or more in a program, the execution of the program stops after the block in which the predetermined sequence number is found for the first time is executed.

- **Block to be repeated a specified number of times**

If the predetermined sequence number is found in a block which is to be executed repeatedly, the execution of the program stops after the block is executed specified times.

11.4.9 Displaying and Setting Run Time,Parts Count, and Time

Various run times, the total number of machined parts, number of parts required, and number of machined parts can be displayed. This data can be set by parameters or on this screen (except for the total number of machined parts and the time during which the power is on, which can be set only by parameters).
This screen can also display the clock time. The time can be set on the screen.

Procedure for Displaying and Setting Run Time, Parts Count and Time

1 Select the MDI mode.

2 Press function key OFFSET SETTING .

3 Press chapter selection soft key **[SETING]**.

4 Press page key PAGE ↑ or PAGE ↓ several times until the following screen is displayed.

```
SETTING (TIMER)                      O0001 N00000

    PARTS TOTAL     =         14
    PARTS REQUIRED  =          0
    PARTS COUNT     =         23

    POWER ON        =      4H 31M
    OPERATING TIME  =      0H  0M  0S
    CUTTING TIME    =      0H 37M  5S
    FREE PURPOSE    =      0H  0M  0S
    CYCLE TIME      =      0H  0M  0S
               DATE =  1993/07/05
               TIME =    11:32:52

> _                                  S   0  T0000
MDI **** *** ***         16:05:59
[ OFFSET ][ SETING ][ WORK ][      ][ (OPRT) ]
```

5 To set the number of parts required, move the cursor to PARTS REQUIRED and enter the number of parts to be machined.

6 To set the clock, move the cursor to DATE or TIME, enter a new date or time, then press soft key **[INPUT]**.

Display items

● PARTS TOTAL

This value is incremented by one when M02, M30, or an M code specified by parameter 6710 is executed. This value cannot be set on this screen. Set the value in parameter 6712.

● PARTS REQUIRED

It is used for setting the number of machined parts required.
When the “0” is set to it, there is no limitation to the number of parts. Also, its setting can be made by the parameter (NO. 6713).

- **PARTS COUNT**

This value is incremented by one when M02, M30, or an M code specified by parameter 6710 is executed. The value can also be set by parameter 6711. In general, this value is reset when it reaches the number of parts required. Refer to the manual issued by the machine tool builder for details.

- **POWER ON**

Displays the total time which the power is on. This value cannot be set on this screen but can be preset in parameter 6750.

- **OPERATING TIME**

Indicates the total run time during automatic operation, excluding the stop and feed hold time.
This value can be preset in parameter 6751 or 6752.

- **CUTTING TIME**

Displays the total time taken by cutting that involves cutting feed such as linear interpolation (G01) and circular interpolation (G02 or G03). This value can be preset in parameter 6753 or 6754.

- **FREE PURPOSE**

This value can be used, for example, as the total time during which coolant flows. Refer to the manual issued by the machine tool builder for details.

- **CYCLE TIME**

Indicates the run time of one automatic operation, excluding the stop and feed hold time. This is automatically preset to 0 when a cycle start is performed at reset state. It is preset to 0 even when power is removed.

- **DATA and TIME**

Displays the current date and time. The date and time can be set on this screen.

Explanations

- **Usage**

When the command of M02 or M30 is executed, the total number of machined parts and the number of machined parts are incremented by one. Therefore, create the program so that M02 or M30 is executed every time the processing of one part is completed. Furthermore, if an M code set to the parameter (NO. 6710) is executed, counting is made in the similar manner. Also, it is possible to disable counting even if M02 or M30 is executed (parameter PCM (No. 6700#0) is set to 1). For details, see the manual issued by machine tool builders.

Restrictions

- **Run time and part count settings**

Negative value cannot be set. Also, the setting of “M” and “S” of run time is valid from 0 to 59.
Negative value may not be set to the total number of machined parts.

- **Time settings**

Neither negative value nor the value exceeding the value in the following table can be set.

Item	Maximum value	Item	Maximum value
Year	2085	Hour	23
Month	12	Minute	59
Day	31	Second	59

11.4.10 Displaying and Setting the Workpiece Origin Offset Value

Displays the workpiece origin offset for each workpiece coordinate system (G54 to G59) and external workpiece origin offset. The workpiece origin offset and external workpiece origin offset can be set on this screen.

Procedure for Displaying and Setting the Workpiece Origin Offset Value

1 Press function key [OFFSET SETTING] .

2 Press chapter selection soft key **[WORK]**.
The workpiece coordinate system setting screen is displayed.

```
 WORK COORDINATES                  O0001 N00000

   NO.      DATA                  NO.      DATA
   00   X   0.000                 02   X 152.580
  (EXT) Z   0.000                (G55) Z 234.000


   01   X  20.000                 03   X 300.000
  (G54) Z  50.000                (G56) Z 200.000


> _                                S    0  T0000
MDI **** *** ***        16:05:59
[ OFFSET ][ SETING ][  WORK ][      ][ (OPRT) ]
```

3 The screen for displaying the workpiece origin offset values consists of two or more pages. Display a desired page in either of the following ways:

Press the page up [PAGE ↑] or page down [PAGE ↓] key.

Enter the workpiece coordinate system number (0: external workpiece origin offset, 1 to 6: workpiece coordinate systems G54 to G59) and press operation selection soft key **[NO.SRH]**.

4 Turn off the data protection key to enable writing.

5 Move the cursor to the workpiece origin offset to be changed.

6 Enter a desired value by pressing numeric keys, then press soft key **[INPUT]**. The entered value is specified in the the workpiece origin offset value. Or, by entering a desired value with numeric keys and pressing soft key **[+INPUT]**, the entered value can be added to the previous offset value.

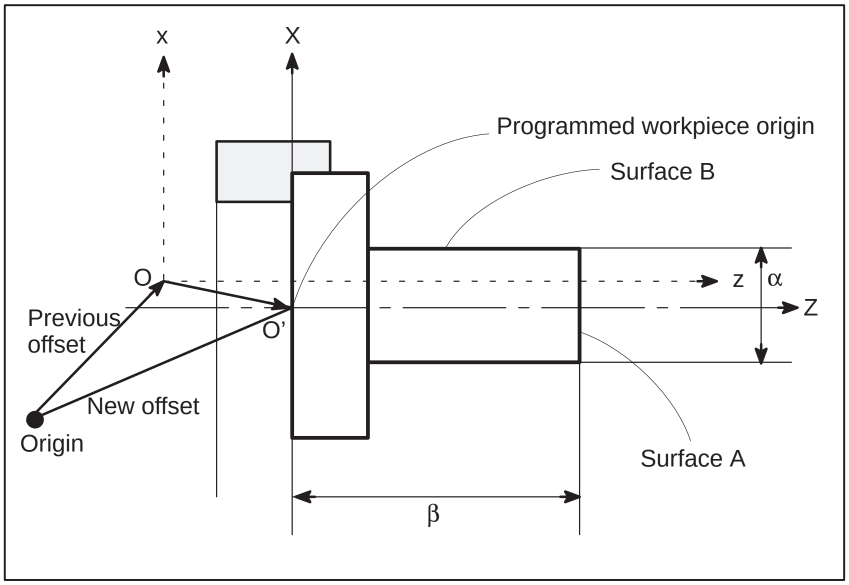
7 Repeat **5** and **6** to change other offset values.

8 Turn on the data protection key to disable writing.

11.4.11 Input of measured workpiece origin offsets

This function is used to compensate for the difference between the programmed workpiece coordinate system and the actual workpiece coordinate system. The measured offset for the origin of the workpiece coordinate system can be input on the screen such that the command values match the actual dimensions.
Selecting the new coordinate system matches the programmed coordinate system with the actual coordinate system.

Procedure for Inputting of Measured Workpiece Origin Offsets



1 When the workpiece is shaped as shown above, cut surface A manually.

2 Move the tool along the X axis without changing the Z coordinate then stop the spindle.

3 Measure distance β between surface A and the programmed origin of the workpiece coordinate system as shown above.

4 Press function key OFFSET SETTING .

5 To display the workpiece origin offset setting screen, press the chapter selection soft key **[WORK]**.

```
WORK COORDINATES                O1234 N56789
(G54)
  NO.      DATA           NO.       DATA
  00   X  0.000           02   X  0.000
 (EXT) Z  0.000           (G55)Z  0.000


  01   X  0.000           03   X  0.000
 (G54) Z  0.000          (G56) Z  0.000



> Z100.                          S   0  T0000
MDI **** *** ***      16:05:59
[ NO.SRH ][ MEASUR ][      ][ +INPUT ][ INPUT ]
```

6 Position the cursor to the workpiece origin offset value to be set.

7 Press the address key for the axis along which the offset is to be set (Z–axis in this example).

8 Enter the measured value (α) then press the **[MEASUR]** soft key.

9 Cut surface B manually.

10 Move the tool along the Z axis without changing the X coordinate then stop the spindle.

11 Measure the diameter of surface A (α) then enter the diameter at X.

Restrictions

- **Consecutive input**

Offsets for two or more axes cannot be input at the same time.

- **During program execution**

This function cannot be used while a program is being executed.

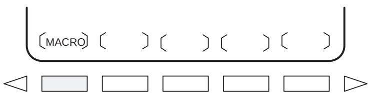
- **Effect from other shift value**

Any shift specified for the workpiece coordinate system or external offset remains effective when this function is used.

11.4.12 Displaying and Setting Custom Macro Common Variables

Displays common variables (#100 to #149 or #100 to #199, and #500 to #531 or #500 to #999) on the CRT. When the absolute value for a common variable exceeds 99999999, ******** is displayed. The values for variables can be set on this screen. Relative coordinates can also be set to variables.

Procedure for displaying and setting custom macro common variables



Continuous menu key

1 Press function key OFFSET SETTING .

2 Press the continuous menu key ▷ , then press chapter selection soft key **[MACRO]**.The following screen is displayed:

```
VARIABLE                          O0001 N00000
  NO.      DATA          NO.       DATA
  100    1000.000        108        0.000
  101       0.000        109    40000.000
  102  -50000.000        110    153020.00
  103       0.000        111     0001.000
  104   1238501.0        112        0.000
  105       0.000        113    20000.000
  106       0.000        114        0.000
  107       0.000        115        0.000
 ACTUAL POSITION (RELATIVE)
    U0.000              W 0.000

> _                               S   0  T0000
MDI **** *** ***       16:05:59
[ NO.SRH ][     ][ INP.C. ][     ][ INPUT ]
```

3 Move the cursor to the variable number to set using either of the following methods:
– Enter the variable number and press soft key **[NO.SRH]**.
– Move the cursor to the variable number to set by pressing page keys ↑ PAGE and/or PAGE ↓ and cursor keys ↑ , ↓ , ← , and/or → .

4 Enter data with numeric keys and press soft key **[INPUT]**.

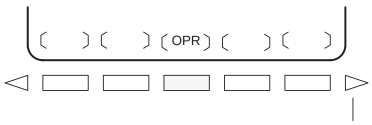
5 To set a relative coordinate in a variable, press address key X or Z , then press soft key **[INP.C.]**.

6 To set a blank in a variable, just press soft key **[INPUT]**. The value field for the variable becomes blank.

11.4.13 Displaying and Setting the Software Operator’s Panel

With this function, functions of the switches on the machine operator’s panel can be controlled from the CRT/MDI panel.
Jog feed can be performed using numeric keys.

Procedure for displaying and setting the software operator’s panel



Continuous menu key

1 Press function key OFFSET SETTING .

2 Press the continuous menu key ▷ , then press chapter selection soft key **[OPR]**.

3 The screen consists of several pages.

Press page key ↑ PAGE or PAGE ↓ until the desired screen is displayed.

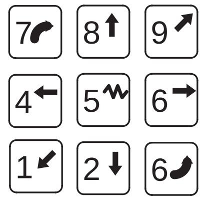
```
OPERATOR’S PANEL                 O0000 N00000

 MODE :  MDI  MEM  EDIT  HND  JOG  REF
 HANDLE AXIS :  HX   HZ   HC   HY
 STEP MULTI. :  *1   *10  *100
 RAPID OVRD. :  100%  50%  25%  F0
 JOG OVRD    :   2.0%
                          **************
 FEED OVRD.  :  100%
                                 ***
ACTUAL POSITION (ABSOLUTE)
    X    0.000        Z    0.000

>_
REF **** *** ***        16:05:59
[  MACRO  ][  MEMu  ][  OPR  ][      ][      ]
```

```
OPERATOR’S PANEL                 O0000 N00000

BLOCK SKIP    :            OFF  ON
SINGLE BLOCK  :            OFF  ON
MACHINE LOCK  :            OFF  ON
PROTECT KEY   :            PROTECT    RELEASE
FEED HOLD     :            OFF

ACTUAL POSITION (ABSOLUTE)
    X    0.000        Z    0.000



MDI **** *** ***        16:05:59
[  MACRO  ][  MENU  ][  OPR  ][      ][      ]
```

4 Move the cursor to the desired switch by pressing cursor key ↑ or ↓ .

5 Push the cursor move key ← or → to match the mark ■ to an arbitrary position and set the desired condition.

6 Press one of the following arrow keys to perform manual continuous feed. Press the 5 key together with an arrow key to perform manual continuous rapid traverse.



Explanations

• Valid operations

The valid operations on the software operator's panel are shown below. Whether to use the CRT or machine operator's panel for each group of operations can be selected by parameter 7200.

Group1 : Mode selection
Group2 : Selection of manual continuous feed axis, manual continuous rapid traverse
Group3 : Selection of manual pulse generator feed axis, selection of manual pulse magnification x1, x10, x100
Group4 : Manual continuous federate, federate override, rapid traverse override
Group5 : Optional block skip, single block, machine lock, dry run
Group6 : Protect key
Group7 : Feed hold

• Display

The groups for which the machine operator's panel is selected by parameter 7200 are not displayed on the software operator's panel.

• Screens on which manual continuous feed is valid

When the CRT indicates other than the software operator's panel screen and diagnostic screen, manual continuous feed is not conducted even if the arrow key is pushed.

• Manual continuous feed and arrow keys

The feed axis and direction corresponding to the arrow keys can be set with parameters (Nos. 7210 to 7217).

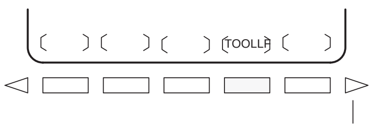
• General purpose switches

Eight optionally definable switches are added as an extended function of the software operator's panel. The name of these switches can be set by parameters as character strings of max. 8 characters. For the meanings of these switches, refer to the manual issued by machine tool builder.

11.4.14 Displaying and Setting Tool Life Management Data

Tool life data can be displayed to inform the operator of the current state of tool life management. Groups which require tool changes are also displayed. The tool life counter for each group can be preset to an arbitrary value. Tool data (execution data) can be reset or cleared. To register or modify tool life management data, a program must be created and executed. See Explanations in this section for details.

Procedure for display and setting the tool life management data



Continuous menu key

1 Press function key [OFFSET SETTING].

2 Press the continuous menu key [▷] to display chapter selection soft key **[TOOLLF]**.

3 Press soft key **[TOOLLF]**.

4 One page displays data on two groups. Pressing page key [PAGE ↓] or [PAGE ↑] successively displays data on the following groups. Up to four group Nos., for which the Tool Change signal is being issued, are displayed at the bottom of each page. An arrow shown in the figure is displayed for five or more groups, if exists.

```
TOOL LIFE DATA  :                O3000 N00060
                       SELECTED GROUP 000
  GROUP  001 : LIFE   0150   COUNT   0000
    0034 0078 0012 0056
    0090 0035 0026 0061
    0000 0000 0000 0000
    0000 0000 0000 0000

  GROUP  002 : LIFE   1400   COUNT   0000
    0062 0024 0044 0074
    0000 0000 0000 0000
    0000 0000 0000 0000
    0000 0000 0000 0000
  TO BE CHANGED  : 003 004 005 006 --->
> _
MEM **** *** ***      16:05:59
[ MACRO ][      ][  OPR  ][ TOOLLF ][ (OPRT) ]
```

5 To display the page containing the data for a group, enter the group number and press soft key **[NO.SRH]**.
The cursor can be moved to an arbitrary group by pressing cursor key [↑] or [↓].

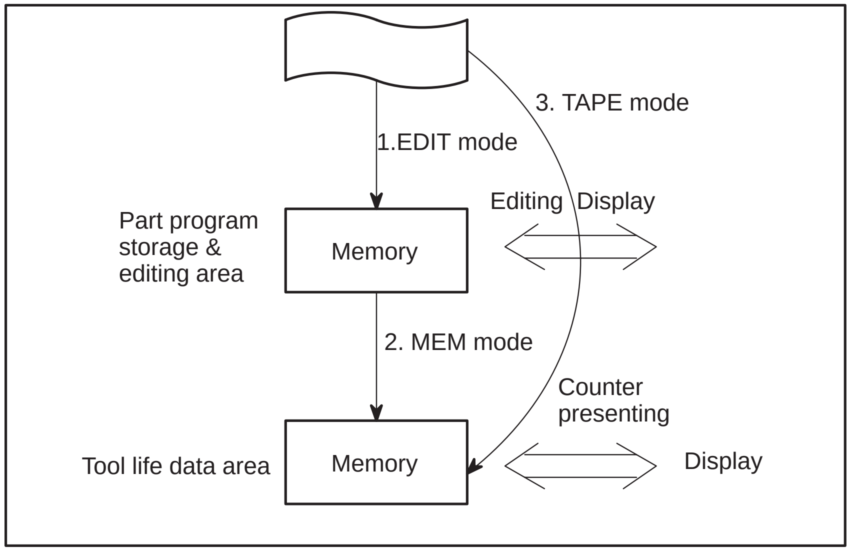
6 To change the value in the life counter for a group, move the cursor to the group, enter a new value (four digits), and press **[INPUT]**. The life counter for the group indicated by the cursor is preset to the entered value. Other data for the group is not changed.

7 To reset the tool data, move the cursor on the group to reset, then press the **[(OPRT)]**, **[CLEAR]**, and **[EXEC]** soft keys in this order.
All execution data for the group indicated by the cursor is cleared together with the marks (@, #, or *).

Explanations

• Registering tool life management data

The tool life management data must be executed to register it in the CNC memory.



1 Load the program for tool life management in the EDIT mode, as with an ordinary CNC tape.
The program will be registered in the part program memory and will be made ready for display and editing.

2 Perform a cycle start operation in the MEM mode to run the program. The data will be stored in the tool life data area of the memory; at the same time, the already existing tool life data of all groups will be cancelled and the life counters will be cleared. Data once stored is not erased by turning the power off.

3 Executing a cycle start operation in the TAPE mode instead of the operation of (1), stores the program contents directly onto the tool life data area. In this case, however, display and editing cannot be done as in (1). TAPE mode is not always prepared according to the machine tool builders.

● Display contents

```
TOOL LIFE DATA  :                O3000 N00060
                     SELECTED GROUP 000
  GROUP  001 : LIFE   0150   COUNT   0007
    0034 0078 0012 0056
    0090 0035 0026 0061
    0000 0000 0000 0000
    0000 0000 0000 0000

  GROUP  002 : LIFE   1400   COUNT   0000
    0062 0024 0044 0074
    0000 0000 0000 0000
    0000 0000 0000 0000
    0000 0000 0000 0000
  TO BE CHANGED  : 003 004 005 006 --->
> _
MEM **** *** ***      16:05:59
[ MACRO ][      ][  OPR  ][ TOOLLF ][ (OPRT) ]
```

- The first line is the title line.
- In the second line the group number of the current command is displayed.
  When there is no group number of the current command, 0 is displayed.
- In lines 3 to 7 the tool life data of the group is displayed.
  The third line displays group number, life and the count used.
  The life count is chosen by parameter LTM (No. 6800#2) as either minutes(or hours) or number of times used.
  In lines 4 to 7, tool numbers are displayed. In this case, the tool is selected in the order, 0034 → 0078 → 0012 → 056 → 0090 ...
  The meaning of each mark before the tool numbers is :
  * : Shows the life has finished.
  # : Shows that the skip command has been accepted.
  @ : Shows that the tool is currently being used.
  The life counter counts for tool with @.
  ”*” is displayed when the next command is issued by the group to which it belongs.
- Lines 8 to 12 are next group life data to the group displayed in lines 3 to 7.
- In the thirteenth line the group number when the tool change signal is being emitted is displayed. The group number display appears in ascending order. When it cannot be completely displayed, ”––>” is displayed.

11.4.15 Setting and Displaying B–axis Tool Compensation

Setting and displaying B–axis tool compensation

1 Press the OFFSET SETTING function key.

2 Press the ▷ continuous menu key several times. Then, press the **[OFST.B]** chapter selection key.

- When the option for tool geometry and wear compensation is not provided,

```
 OFFSET (B–AXIS)                      O0200 N00000
   No.        DATA
    51      –999.999
    52      –999.999
    53      –999.999
    54      –999.999
    55      –999.999
    56      –999.999              .
    57      –999.999
    58      –999.999
    59      –999.999


>_                                    S    0 T0000
 MDI **** *** ***          15:29:51
[ OFST.B ] [      ] [      ] [      ] [      ]
```

- When the option for tool geometry and wear compensation is provided,

```
 OFFSET (B–AXIS)                      O0200 N00000
   NO.        (WEAR)   (GEOMETRY)
    51      –999.999    –999.999
    52      –999.999    –999.999
    53      –999.999    –999.999
    54      –999.999    –999.999
    55      –999.999    –999.999
    56      –999.999    –999.999          .
    57      –999.999    –999.999
    58      –999.999    –999.999
    59      –999.999    –999.999


>_                                    S    0 T0000
 MDI **** *** ***          15:29:51
[ OFST.B ] [      ] [      ] [      ] [      ]
```

4 Position the cursor to the item to be set or modified, using the cursor keys.

5 Enter the value, then press the INPUT key.

Explanations

The offset can be set to a value in the following valid data ranges.

Offset	Metric input	Inch input
IS–B	–999.999 to 999.999	–99.9999 to 99.9999
IS–C	–999.9999 to 999.9999	–99.99999 to 99.9999

Special B–axis offsets are input or output together with usual offsets.
When the option for tool geometry and wear compensation is provided, wear offsets and geometry offsets can be specified separately. A tool offset consists of both the specified wear offset and geometry offset.
In two–path control, tool offsets can be specified for each tool post or both tool posts, depending on the setting of COF, bit 0 of parameter No. 8242.

11.5 SCREENS DISPLAYED BY FUNCTION KEY SYSTEM

When the CNC and machine are connected, parameters must be set to determine the specifications and functions of the machine in order to fully utilize the characteristics of the servo motor or other parts.

This chapter describes how to set parameters on the MDI panel. Parameters can also be set with external input/output devices such as the Handy File (see Chapter III–9).

In addition, pitch error compensation data used for improving the precision in positioning with the ball screw on the machine can be set or displayed by the operations under function key SYSTEM .

See Chapter III–7 for the diagnostic screens displayed by pressing function key SYSTEM .

11.5.1 Displaying and Setting Parameters

When the CNC and machine are connected, parameters are set to determine the specifications and functions of the machine in order to fully utilize the characteristics of the servo motor. The setting of parameters depends on the machine. Refer to the parameter list prepared by the machine tool builder.
Normally, the user need not change parameter setting.

Procedure for displaying and setting parameters

1 Set 1 for **PARAMETER WRITE** to enable writing. See the procedure for enabling/disabling parameter writing described below.

2 Press function key SYSTEM .

3 Press chapter selection soft key **[PARAM]** to display the parameter screen.

```
PARAMETER (SETTING)                O0010 N00002

   0000       SEQ               INI  ISO   TVC
           0    0    0    0    0    0    0
   0001                               FCV
           0    0    0    0    0    0    0    0
   0012                                    MIR
    X      0    0    0    0    0    0    0    0
    Y      0    0    0    0    0    0    0    0
    Z      0    0    0    0    0    0    0    0
   0020 I/O CHANNEL                        0
   0022                                    0

 > _
 MDI **** *** ***      16:05:59
 [ PARAM ][ DGNOS ][  PMC  ][ SYSTEM ][ (OPRT) ]
```

4 Move the cursor to the parameter number to be set or displayed in either of the following ways:

· Enter the parameter number and press soft key **[NO.SRH]** .

· Move the cursor to the parameter number using the page keys, ↑ PAGE and PAGE ↓ , and cursor keys, ↑ , ↓ , ← , and → .

5 To set the parameter, enter a new value with numeric keys and press soft key **[INPUT]** in the MDI mode. The parameter is set to the entered value and the value is displayed.

6 Set 0 for **PARAMETER WRITE** to disable writing.

Procedure for enabling/displaying parameter writing

1 Select the **MDI** mode or enter state emergency stop.

2 Press function key [OFFSET SETTING] .

3 Press soft key **[SETING]** to display the setting screen.

```
SETTING (HANDY)                      O0001 N00000

  PARAMETER WRITE = 1 (0:DISABLE 1:ENABLE)
  TV CHECK        = 0 (0:OFF   1:ON)
  PUNCH CODE      = 1 (0:EIA   1:ISO)
  INPUT UNIT      = 0 (0:MM    1:INCH)
  I/O CHANNEL     =  0 (0–3:CHANNEL NO.)
  SEQUENCE NO.    = 0 (0:OFF   1:ON)
  TAPE FORMAT     = 0 (0:NO CNV  1:F10/11)
  SEQUENCE STOP   =     0 (PROGRAM NO.)
  SEQUENCE STOP   =    11 (SEQUENCE NO.)


> _                                 S    0  T0000
MDI **** *** ***          16:05:59
[ OFFSET ][ SETING ][ WORK ][      ][ (OPRT) ]
```

4 Move the cursor to **PARAMETER WRITE** using cursor keys.

5 Press soft key **[(OPRT)]**, then press **[1: ON]** to enable parameter writing.
At this time, the CNC enters the P/S alarm state (No. 100).

6 After setting parameters, return to the setting screen. Move the cursor to **PARAMETER WRITE** and press soft key **[(OPRT)]** , then press **[0: OFF]**.

7 Depress the [RESET] key to release the alarm condition. If P/S alarm No. 000 has occurred, however, turn off the power supply and then turn it on, otherwise the P/S alarm is not released.

Explanations

● Setting parameters with external input/output devices

See Chapter 8 for setting parameters with external input/output devices such as the Handy File.

● Parameters that require turning off the power

Some parameters are not effective until the power is turned off and on again after they are set. Setting such parameters causes alarm 000. In this case, turn off the power, then turn it on again.

● Parameter list

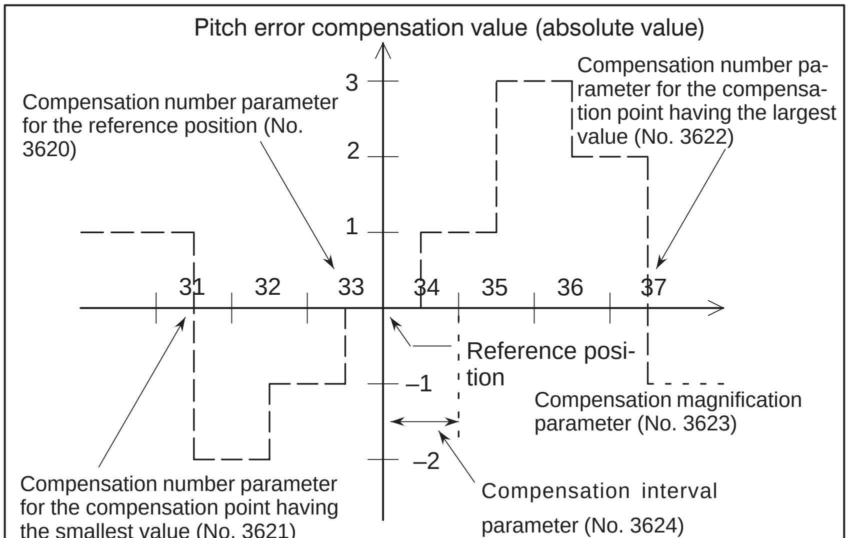
Refer to the FANUC Series 16/18160/180–MODEL B Parameter Manual (B–61810E) for the parameter list.

● Setting data

Some parameters can be set on the setting screen if the parameter list indicates ”Setting entry is acceptable”. Setting 1 for **PARAMETER WRITE** is not necessary when three parameters are set on the setting screen.

11.5.2 Displaying and Setting Pitch Error Compensation Data

If pitch error compensation data is specified, pitch errors of each axis can be compensated in detection unit per axis.
Pitch error compensation data is set for each compensation point at the intervals specified for each axis. The origin of compensation is the reference position to which the tool is returned.
The pitch error compensation data is set according to the characteristics of the machine connected to the NC. The content of this data varies according to the machine model. If it is changed, the machine accuracy is reduced.
In principle, the end user must not alter this data.
Pitch error compensation data can be set with external devices such as the Handy File (see Chapter III–9). Compensation data can also be written directly with the MDI panel.
The following parameters must be set for pitch error compensation. Set the pitch error compensation value for each pitch error compensation point number set by these parameters.
In the following example, 33 is set for the pitch error compensation point at the reference position.



Compensation position number	31	32	33	34	35	36	37
Compensation value to be set	+3	–1	–1	+1	+2	–1	–3

· Number of the pitch error compensation point at the reference position (for each axis): Parameter 3620

· Number of the pitch error compensation point having the smallest value (for each axis): Parameter 3621

· Number of the pitch error compensation point having the largest value (for each axis): Parameter 3622

·Pitch error compensation magnification (for each axis): Parameter 3623

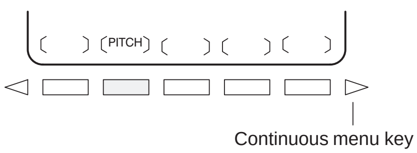
· Interval of the pitch error compensation points (for each axis): Parameter 3624

Procedure for displaying and setting the pitch error compensation data

1 Set the following parameters:

- Number of the pitch error compensation point at the reference position (for each axis): Parameter 3620
- Number of the pitch error compensation point having the smallest value (for each axis): Parameter 3621
- Number of the pitch error compensation point having the largest value (for each axis): Parameter 3622
- Pitch error compensation magnification (for each axis): Parameter 3623
- Interval of the pitch error compensation points (for each axis): Parameter 3624

2 Press function key SYSTEM .



3 Press the continuous menu key ▷, then press chapter selection soft key **[PITCH]**.
The following screen is displayed:

```
PIT–ERROR SETTING                          O0000 N00000

      NO.DATA       NO.DATA       NO.DATA
      0000 0        0010 0        0020 0
      0001 0        0011 0        0021 0
      0002 0        0012 0        0022 0
      0003 0        0013 0        0023 0
  (X) 0004 0        0014 0        0024 0
      0005 0        0015 0        0025 0
      0006 0        0016 0        0026 0
      0007 0        0017 0        0027 0
      0008 0        0018 0        0028 0
      0009 0        0019 0        0029 0
> _
MEM **** *** ***       16:05:59
[ NO.SRH ][ ON:1 ][ OFF:0 ][ +INPUT ][ –INPUT ]
```

4 Move the cursor to the compensation point number to be set in either of the following ways:

- Enter the compensation point number and press the [NO.SRH] soft key.
- Move the cursor to the compensation point number using the page keys, ↑PAGE and PAGE↓, and cursor keys, ↑, ↓, ←, and →.

5 Enter a value with numeric keys and press the **[INPUT]** soft key.

11.6 DISPLAYING THE PROGRAM NUMBER, SEQUENCE NUMBER, AND STATUS, AND WARNING MESSAGES FOR DATA SETTING OR INPUT/OUTPUT OPERATION

The program number, sequence number, and current CNC status are always displayed on the screen except when the power is turned on, a system alarm occurs, or the PMC screen is displayed.
If data setting or the input/output operation is incorrect, the CNC does not accept the operation and displays a warning message.
This section describes the display of the program number, sequence number, and status, and warning messages displayed for incorrect data setting or input/output operation.

11.6.1 Displaying the Program Number and Sequence Number

The program number and sequence number are displayed at the top right on the screen as shown below.

```
PROGRAM                                   O2000 N00130    — Sequence No.
 O1000                                                    — Program No.
 N100 G50 X0 Z0. ;
 N101 G00 X100. Z50. ;;
 N102 G01 X230. Z56. ;
 N103 W–10. ;
 N104 U–120. ;
 N105 M02 ;




> _
EDIT **** *** ***          16:05:59
[ PRGRM ][ CHECK ][ CURRNT ][ NEXT ][ (OPRT) ]
```

The program number and sequence number displayed depend on the screen and are given below:

On the program screen in the EDIT mode on Background edit screen :
The program No. being edited and the sequence number just prior to the cursor are indicated.

Other than above screens :

The program No. and the sequence No. executed last are indicated.

Immediately after program number search or sequence number search :

Immediately after the program No. search and sequence No. search, the program No. and the sequence No. searched are indicated.

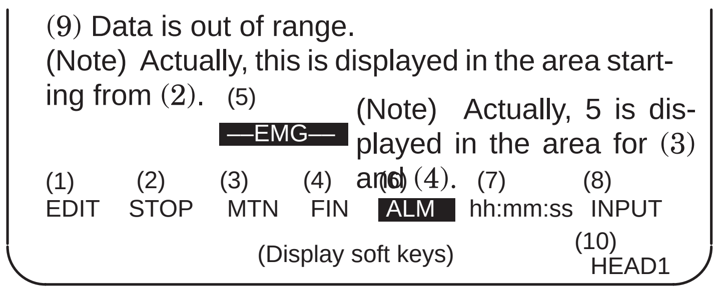
11.6.2 Displaying the Status and Warning for Data Setting or Input/Output Operation

The current mode, automatic operation state, alarm state, and program editing state are displayed on the next to last line on the CRT screen allowing the operator to readily understand the operation condition of the system.

If data setting or the input/output operation is incorrect, the CNC does not accept the operation and a warning message is displayed on the next to last line of the CRT screen. This prevents invalid data setting and input/output errors.

Explanations

- **Description of each display**



Note) Actually, (10) is displayed at the position where (8) is now displayed.

- **(1) Current mode**

MDI : Manual data input, MDI operation
MEM : Automatic operation (Memory operation)
RMT : Automatic operation (DNC operation)
EDIT : Memory editing
HND : Manual handle feed
JOG : Jog feed
TJOG : TEACH IN JOG
THND : TEACH IN HANDLE
INC : Manual incremental feed
REF : Manual reference position return

- **(2) Automatic operation status**

**** : Reset (When the power is turned on or the state in which program execution has terminated and automatic operation has terminated.)
STOP : Automatic operation stop (The state in which one block has been executed and automatic operation is stopped.)
HOLD : Feed hold (The state in which execution of one block has been interrupted and automatic operation is stopped.)
STRT : Automatic operation start–up (The state in which the system operates automatically)

- **(3) Axis moving status/dwell status**

MTN : Indicates that the axis is moving.
DWL : Indicates the dwell state.
*** : Indicates a state other than the above.

- **(4) State in which an auxiliary function is being executed**

FIN : Indicates the state in which an auxiliary function is being executed. (Waiting for the complete signal from the PMC)
*** : Indicates a state other than the above.

- **(5) Emergency stop or reset status**

--EMG-- : Indicates emergency stop.(Blinks in reversed display.)
—RESET— : Indicates that the reset signal is being received.

- **(6) Alarm status**

ALM : Indicates that an alarm is issued. (Blinks in reversed display.)
BAT : Indicates that the battery is low. (Blinks in reversed display.)
Space : Indicates a state other than the above.

- **(7) Current time**

hh:mm:ss – Hours, minutes, and seconds

- **(8) Program editing status**

INPUT : Indicates that data is being input.
OUTPUT : Indicates that data is being output.
SRCH : Indicates that a search is being performed.
EDIT : Indicates that another editing operation is being performed (insertion, modification, etc.)
LSK : Indicates that labels are skipped when data is input.
RSTR : Indicates that the program is being restarted
Space : Indicates that no editing operation is being performed.

- **(9) Warning for data setting or input/output operation**

When invalid data is entered (wrong format, value out of range, etc.), when input is disabled (wrong mode, write disabled, etc.), or when input/output operation is incorrect (wrong mode, etc.), a warning message is displayed. In this case, the CNC does not accept the setting or input/output operation.
The following are examples of warning messages:

Example 1)
When a parameter is entered

```
> 1
EDIT    WRONG MODE

                 (Display soft keys)
```

Example 2)
When a parameter is entered

```
> 999999999
MDI    TOO MANY DIGITS

                 (Display soft keys)
```

Example 3)
When a parameter is output to an external input/output device

```
> _
MEM    WRONG MODE

                 (Display soft keys)
```

- **(10)Tool post name (for the two–path control)**

HEAD1 : Tool post 1 is selected.
HEAD2 : Tool post 2 is selected.
Other names can be used depending on the settings of parameters 3141 to 3147.
The tool post name is displayed at the position where (8) is now displayed. While the program is edited, (8) is displayed.

11.7 SCREENS DISPLAYED BY FUNCTION KEY MESSAGE

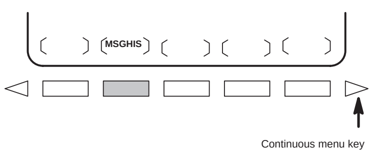
By pressing the MESSAGE function key, data such as alarms, alarm history data, and external messages can be displayed.
For information relating to alarm display, see Section III.7.1. For information relating to alarm history display, see Section III.7.2.
For information relating to external message display, see the relevant manual supplied by the machine tool builder.

11.7.1 External Operator Message History Display

External operator messages can be preserved as history data.
Preserved history data can be displayed on the external operator message history screen.

Procedure for external operator message history display

Procedure



1 Press the MESSAGE function key.

2 Press the continuous menu key ▷, then press the chapter selection soft key **[MSGHIS]**. The screen shown below appears.

Date and Page number
Message nuumber
Display range
(Up to 255 characters)

```
MESSAGE HISTORY                         O0000 N00000
 94/01/01 17:25:00                           PAGE:1
  NO. ****




 MEM STRT MIN FIN ALM 09:36:48
[      ][ MSGHIS ][       ][       ][ (OPRT) ]
```

Up to 255 characters can be specified for an external operator message. By setting MS1 and MS0 (bits 7 and 6 of parameter No. 3113), however, the number of characters that can be preserved as external operator message history data can be restricted, and the number of history data items selected.

Explanations

- **Updating external operator message history data**

When an external operator message number is specified, updating of the external operator message history data is started; this updating is continued until a new external operator message number is specified or deletion of the external operator message history data is specified.

- **Clearing external operator message history data**

To clear external operator message history data, press the [CLEAR] soft key. This clears all external operator message history data. (Set MSGCR (bit 0 of parameter No. 3113) to 1.)
Note that when MS1 and MS0 (bits 7 and 6 of parameter No. 3113), used to specify the number of external operator message history data items to be displayed, are changed, all existing external operator message history data is cleared.

Limitations

- **Two–path control**

When two–path control is exercised, the external operator messages for system 1 are displayed. (The external operator messages for system 2 are not displayed.)

- **Option**

Before this function can be used, the external data input function or optional external message function must be selected.

12 GRAPHICS FUNCTION

The graphic function indicates how the tool moves during automatic operation or manual operation.

12.1 GRAPHICS DISPLAY

It is possible to draw the programmed tool path on the 9–inch or 14–inch CRT screen, which makes it possible to check the progress of machining, while observing the path on the CRT screen.
In addition, it is also possible to enlarge/reduce the screen.
The drawing coordinates (parameter) and graphic parameters must be set before a tool path can be displayed.
With two–path control, the tool paths of two tool posts are displayed on the same screen, one on the right and the other on the left.

Graphics display procedure

Procedure

Set the drawing coordinates with parameter No.6510 before starting drawing. See ”Drawing Coordinate System” for the settings and corresponding coordinates.
For the two–path control, parameter GRL (bit 0 of No. 6500) specifies which tool post is displayed on which side (tool post 1 on the right or tool post 2 on the right).

1 Press function key GRAPH . Press CUSTOM GRAPH for a small MDI unit.

The graphic parameter screen shown below appears. (If this screen does not appear, press soft key **[G.PRM]**.)

```
GRAPHIC PARAMETER                          O0001 N00020

 WORK LENGTH           W=       130000
 WORK DIAMETER         D=       130000
 PROGRAM STOP          N=            0
 AUTO ERASE            A=            1
 LIMIT                 L=            0
 GRAPHIC CENTER        X=        61655
                       Z=        90711
 SCALE                 S=           32
 GRAPHIC MODE          M=            0

                                S    0 T0000
>_
MEM STRT **** FIN    12:12:24       HEAD1
[ G.PRM ][     ][ GRAPH ][ ZOOM ][ (OPRT) ]
```

2 For the two–path control, determine for which tool post the data is specified, using a tool post select signal.
Specify the PROGRAM STOP (N), AUTO ERASE (A), and GRAPHIC CENTER (X,Y) parameters separately for each tool post. The other parameters are common to both tool posts. It does not matter for which tool post they are specified first.

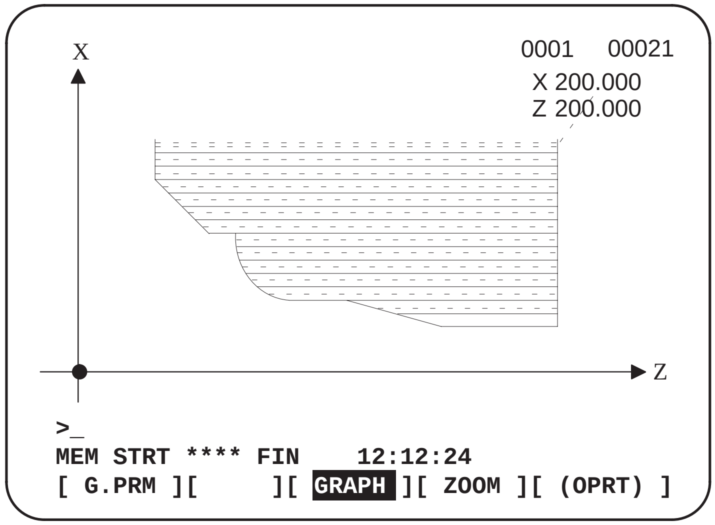
3 Move the cursor with the cursor keys to a parameter to set.

4 Enter data, then press the INPUT key.

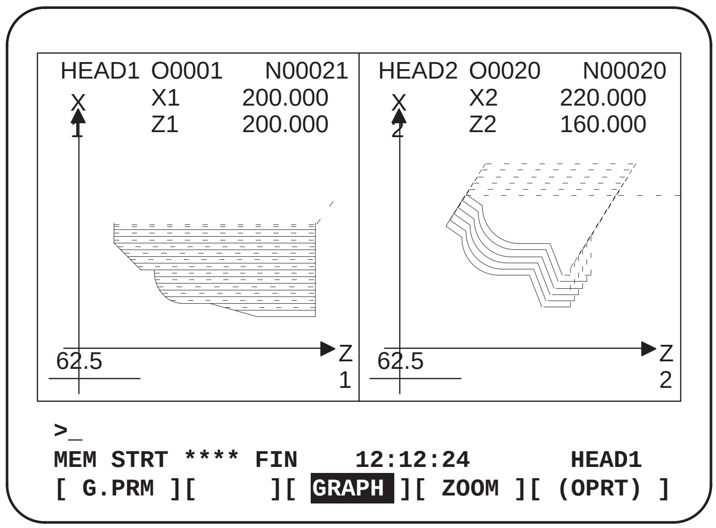
5 Repeat steps 3 and 4 until all required parameters are specified.

6 Press soft key **[GRAPH]**.

7 Automatic or manual operation is started and machine movement is drawn on the screen.



16–TB

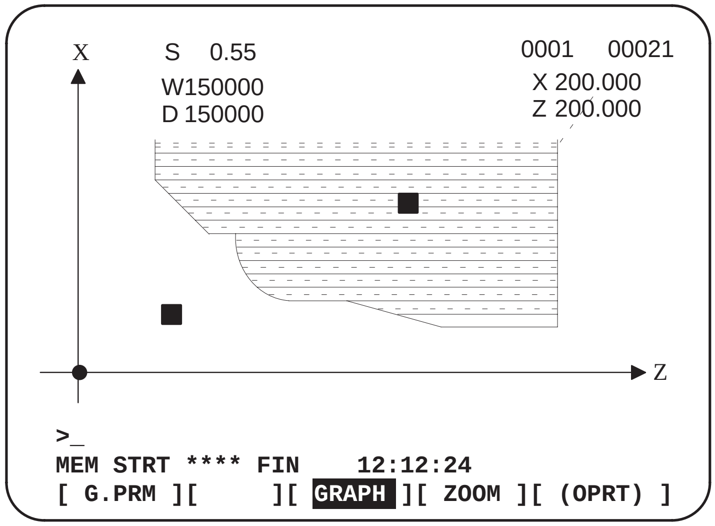


16–TB (two–path lathe control)

- **Magnifying drawings**

Part of a drawing on the screen can be magnified.

8 Press the GRAPH function key, then the **[ZOOM]** soft key to display a magnified drawing. The magnified–drawing screen contains two zoom cursors (■)

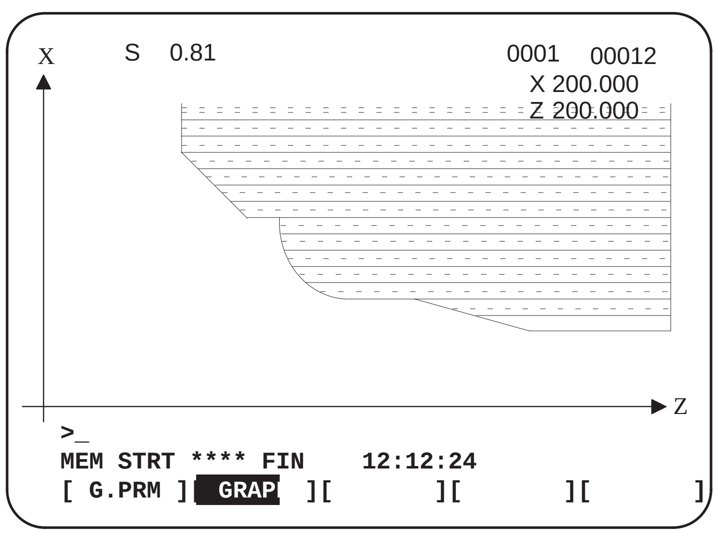


A rectangle that has one of its diagonals defined by the two zoom cursors is magnified to the full size of the screen.
For the two–path control, the zoom cursors are indicated for the selected tool post. Use the tool post select switch to select the tool post corresponding to the drawing to be magnified.

9 Using the cursor keys ↓ ↑ → ←, move the zoom cursors to specify a diagonal for the new screen. Pressing the **[HI / LO]** soft key toggles the zoom cursor to be moved.

10 To make the original drawing disappear, press **[EXEC]**.

11 Resume the previous operation. The part of the drawing specified with the zoom cursors will be magnified.

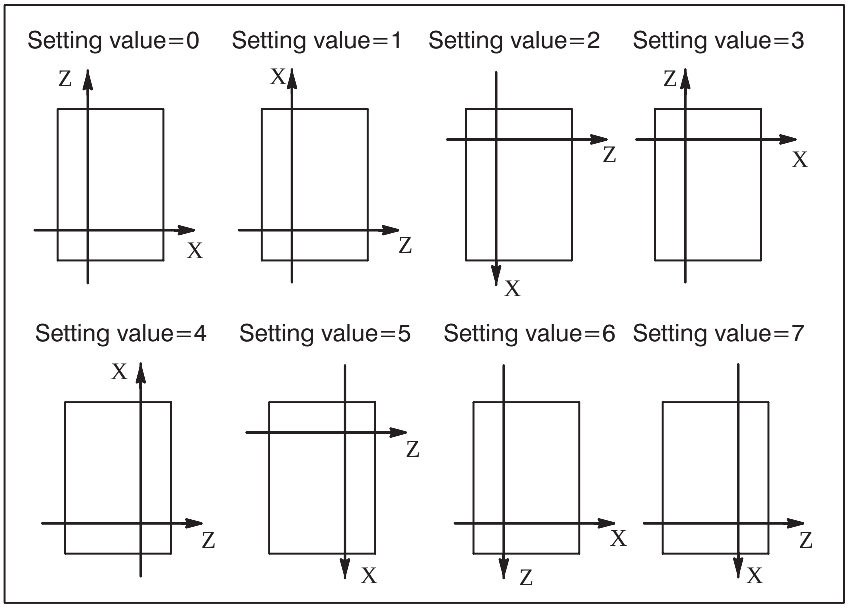


12 To display the original drawing, press the **[NORMAL]** soft key, then start automatic operation.

Explanation

- **Setting drawing coordinate systems**

Parameter No. 6510 is used to set a drawing coordinate system for using the graphic function. The relationships between setting values and drawing coordinate systems are indicated below. With two–path control, a different drawing coordinate system can be selected for each tool post.



- **Graphics parameter**

WORK LENGTH (W), WORK DIAMETER (D)

Specify work length and work diameter. The table below lists the input unit and valid data range.

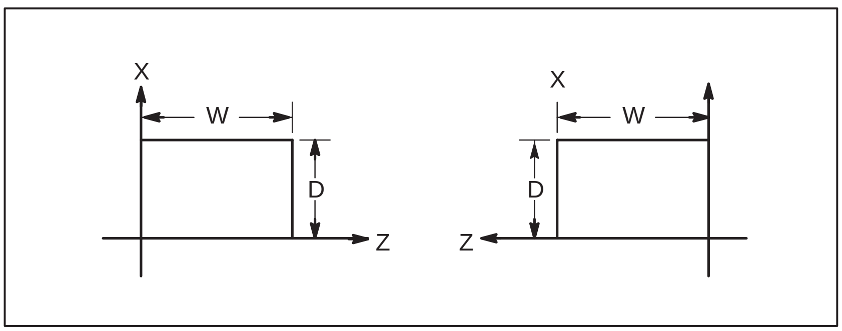


Table 12.1 Unit and Range of Drawing Data

Increment system	Unit		Valid range
	mm input	Inch input	
IS–B	0.001 mm	0.0001 inch	0 to 99999999
IS–C	0.0001 mm	0,00001 inch	

GRAPHIC CENTER (X, Z), SCALE (S)

A screen center coordinate and drawing scale are displayed. A scale screen center coordinate are automatically calculated so that a figure set in WORK LENGTH (a) and WORK DIAMETER (b) can be fully displayed on the screen. So, the user need not set these parameters usually.

A screen center coordinate is defined in the workpiece coordinate system. Table 12. 3. 2 indicates the unit and range. The unit of SCALE is 0.001%.

PROGRAM STOP (N)

Set the sequence number of an end block when part of the program is to be drawn. A value set in this parameter is automatically cancelled (cleared to 0) once a drawing is provided.

AUTO ERASE (A)

If 1 is set, the previous drawing is automatically erased when automatic operation is stated from the reset state. Then, drawing is started.

LIMIT (L)

If 1 is set, the area of stored stroke limit 1 is drawn with double–dot–and–dash lines.

> **Notes**
> The parameter values for drawing are preserved even if power is turned off.

- **Executing drawing only**

Since the graphic drawing is done when coordinate value is renewed during automatic operation, etc., it is necessary to start the program by automatic operation. To execute drawing without moving the machine, therefore, enter the machine lock state.

- **Deleting the previous drawing**

Pressing the **[REVIEW]** soft key on the graphic screen deletes tool paths on it. Setting the graphic parameter as AUTO ERASE (A) = 1 specifies that when automatic operation is started at reset, program execution begins after the previous drawing is erased automatically (AUTO ERASE = 1).

- **Drawing a part of a program**

When necessary to display a part of a program, search the starting block to be drawn by the sequence number search, and set the sequence number of the end block to the PROGRAM STOP N= of the graphic parameter before starting the program under cycle operation mode.

- **Drawing using dashed lines and solid lines**

The tool path is shown with a dashed line (- - - -) for rapid traverse and with a solid line (——) for cutting feed.

- **Displaying coordinates**

The displayed drawing is indicated with coordinates in a work coordinate system.

- **Displaying the machine zero point**

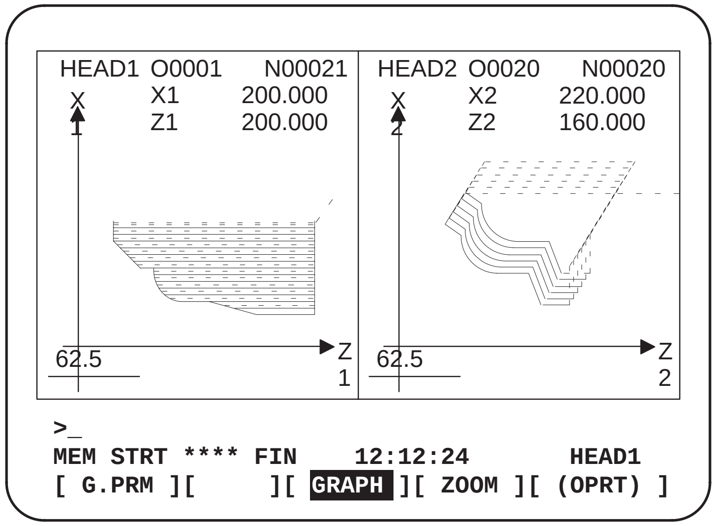
The machine zero point is indicated with a question mark.

- **Switching from a drawing screen to another screen**

Even if the screen is switched to a non–drawing screen, drawing continues. When the drawing screen is displayed again, the entire drawing appears (no parts are missing).

- **Drawing for tool posts 1 and 2 (two–path lathe control)**

For the two–path lathe control, the screen is split vertically, and each half screen displays the tool path for either tool post.



Parameter GRL (bit 0 of NO. 6500) specifies which tool post is to be displayed on which side.

GRL = 0 : Tool post 1 is displayed on the left half, and tool post 2 on the right half.

GRL = 1 ; Tool post 1 is displayed on the right half, and tool post 2 on the left half.

Restrictions

- **Feedrate**

In case the feed rate is considerably high, drawing may not be executed correctly, decrease the speed by dry–run, etc. to execute drawing.

- **Changing the graphic parameters during automatic operation**

After a graphic parameter is changed, the **[REVIEW]** soft key must be pressed to initialize the graphic screen. Otherwise, the change to the graphic parameter is not reflected correctly.

- **Coordinate axis names**

The coordinate axis names are fixed to X or Z. For the two–path control, the first and second axes for tool post 1 are named X1 and Z1, respectively, and the first and second axes for tool post 2 are named X2 and Z2, respectively.

- **Zooming drawings**

If the WORK and DIAMETER graphic parameters are not set correctly, the drawing cannot be magnified. To reduce a drawing, specify a negative value for the SCALE graphic parameter. The machine zero point is indicated with a question mark.

13 HELP FUNCTION

The help function displays on the screen detailed information about alarms issued in the CNC and about CNC operations. The following information is displayed.

- **Detailed information of alarms**

When the CNC is operated incorrectly or an erroneous machining program is executed, the CNC enters the alarm state. The help screen displays detailed information about the alarm that has been issued and how to reset it. The detailed information is displayed only for a limited number of P/S alarms. These alarms are often misunderstood and are rather difficult to understand.

- **Operation method**

If you are not sure about a CNC operation, refer to the help screen for information about each operation.

- **Parameter table**

When setting or referring to a system parameter, if you are not sure of the number of the parameter, the help screen displays a list of parameter Nos. for each function.

Help Function Procedure

Procedure

1 Press the HELP key on the MDI. HELP (INITIAL MENU) screen is displayed.

```
HELP (INITIAL MENU)           O1234   N00001


                ***** HELP *****

                1.  ALARM DETAIL
                2.  OPERATION METHOD
                3.  PARAMETER TABLE






                                    S     0 T0000
 MEM **** *** ***        10:12:25
 [ 1 ALM ][ 2 OPR ][ 3 PARA ][       ][       ]
```

Fig.13(a) HELP (INITIAL MENU) Screen

The user cannot switch the screen display from the PMC screen or CUSTOM screen to the help screen. The user can return to the normal CNC screen by pressing the HELP key or another function key.

ALARM DETAIL screen

2 Press soft key **[1 ALAM]** on the HELP (INITIAL MENU) screen to display detailed information about an alarm currently being raised.

```
>100                           S     0 T0000
MEM **** *** ***        10:12:25
[       ][       ][        ][       ][ SELECT ]
```

Fig.13(b) How to select each ALARM DETAILS

```
HELP (ALARM DETAIL)               O0010 N00001

NUMBER : 027
M'SAGE  :  NO  AXES  COMMANDED  IN  G43/G44
FUNCTION  :  TOOL  LENGTH  COMPENSATION  C
ALARM  :
   IN  TOOL  LENGTH  COMPENSATION  TYPE C,
   NO  AXIS  IS  DESIGNATED  IN  G43  &  G44
   BLOCKS.  IN  TOOL  LENGTH  COMPENSATION
   TYPE  C,  IT  TRIES  TO  LATCH  ON  TO
   ANOTHER  AXIS  WITHOUT  OFFSET  CANCE–
   LING.

>100                           S     0 T0000
MEM **** *** ***        10:12:25
[ 1 ALM ][ 2 OPR ][ 3 PARA ][       ][       ]
```

Alarm No.
Normal explana–tion on alarm
Function classification
Alarm details

Fig.13(c) ALARM DETAIL Screen when P/S Alarm No.27 is issued

Note that only details of the alarm identified at the top of the screen are displayed on the screen.
If the alarms are all reset while the help screen is displayed, the alarm displayed on the ALARM DETAIL screen is deleted, indicating that no alarm is issued.

```
HELP  (ALARM DETAIL)              O1234 N00001


NUMBER    :
M'SAGE    :
FUNCTION  :
ALARM     :

        <<ALARM IS NOT GENERATED>>

  ENTER THE DETAIL–REQUIRED ALARM NUMBER,
  AND PRESS [SELECT] KEY


>100                           S     0 T0000
MEM **** *** ***        10:12:25
[ 1 ALM ][ 2 OPR ][ 3 PARA ][       ][       ]
```

Fig.13(d) ALARM DETAIL Screen when No Alarm is issued

3 To get details on another alarm number, first enter the alarm number, then press soft key **[SELECT]**. This operation is useful for investigating alarms not currently being raised.

```
HELP (ALARM DETAIL)              O1234 N00001


NUMBER    : 100
M'SAGE    : PARAMETER WRITE ENABLE
FUNCTION  :
ALARM     :

        <<ALARM IS NOT GENERATED>>





>100                          S     0 T0000
MEM **** *** ***       10:12:25
[      ][      ][       ][      ][ SELECT ]
```

Fig.13(e) ALARM DETAIL Screen when P/S alarm No.100 is selected

OPERATION METHOD screen

4 To determine an operating procedure for the CNC, press the soft key **[2 OPR]** key on the HELP (INITIAL MENU) screen. The OPERATION METHOD menu screen is then displayed. (See Fig. 13 (e).)

```
HELP (OPERATION METHOD)        O1234   N00001

1. PROGRAM EDIT
2. SEARCH
3. RESET
4. DATA INPUT WITH MDI
5. DATA INPUT WITH TAPE
6. OUTPUT
7. INPUT WITH FANUC CASSETTE
8. OUTPUT WITH FANUC CASSETTE
9. MEMORY CLEAR


                                S     0 T0000
MEM **** *** ***       10:12:25
[ 1 ALM ][ 2 OPR ][ 3 PARA ][       ][       ]
```

Fig.13(f) OPERATION METHOD Menu Screen

To select an operating procedure, enter an item No. from the keyboard then press the **[SELECT]** key.

```
>1                              S     0 T0000
MEM **** *** ***        10:12:25
[       ][       ][        ][       ][ SELECT ]
```

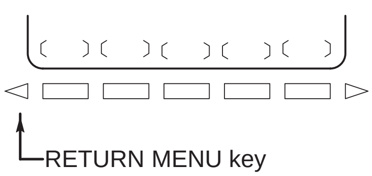
Fig.13(g) How to select each OPERATION METHOD screen

When “1. PROGRAM EDIT” is selected, for example, the screen in Figure 13 (g) is displayed.
On each OPERATION METHOD screen, it is possible to change the displayed page by pressing the PAGE key. The current page No. is shown at the upper right corner on the screen.

```
HELP (OPERATION METHOD)         01234 N00001
<< 1.  PROGRAM EDIT >>                   1/4
*DELETE ALL PROGRAMS
  MODE    : EDIT
  SCREEN  : PROGRAM
  OPR     : (O–9999) – <DELETE>

*DELETE ONE PROGRAM
  MODE    : EDIT
  SCREEN  : PROGRAM
  OPR     : (O+PROGRAM NUMBER) – <DELETE>


>_                              S     0 T0000
MEM **** *** ***        10:12:25
[ 1 ALM ][ 2 OPR ][ 3 PARA ][       ][       ]
```

Each item
Page/maximum page
Operation
Set mode
Operation location
Operating procedure

Fig.13(h) Selected OPERATION METHOD screen



5 To return to the OPERATION METHOD menu screen, press the RETURN MENU key to display “**[2 OPR]**” again, and then press the **[2 OPR]** key again.
To directly select another OPERATION METHOD screen on the screen shown in Figure 13 (h), enter an item No. from the keyboard and press the **[SELECT]** key.

```
>3                              S     0 T0000
MEM **** *** ***        10:12:25
[       ][       ][        ][       ][ SELECT ]
```

Fig.13(i) How to select another OPERATION METHOD screen

PARAMETER TABLE screen

6 If you are not sure of the No. of a system parameter to be set, or to refer to a system parameter, press the **[3 PARA]** key on the HELP (INITIAL MENU) screen. A list of parameter Nos. for each function is displayed. (See Figure 13 (j).)
It is possible to change the displayed page on the parameter screen. The current page No. is shown at the upper right corner on the screen.

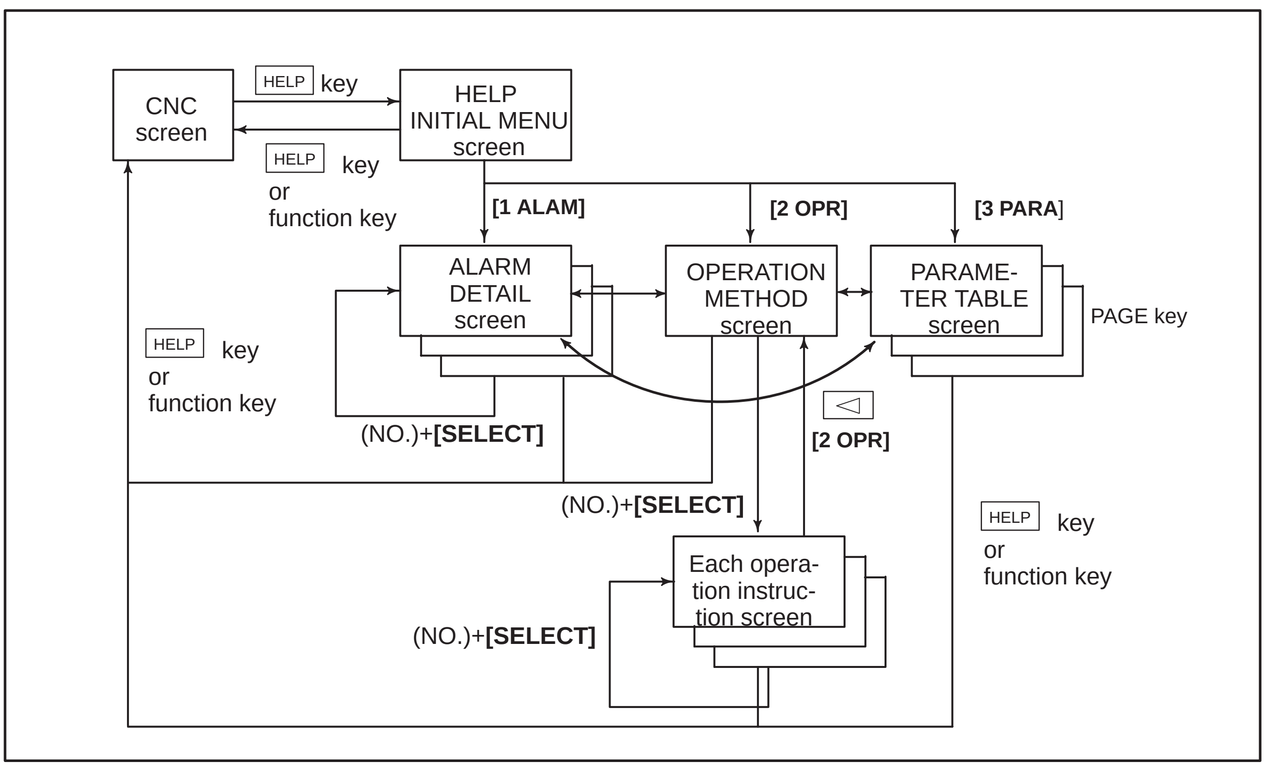
```
HELP (PARAMETER TABLE)          01234  N00001
                                          1/4

* SETTEING   (No. 0000A)
* READER/PUNCHER INTERFACE (No. 0100A)
* AXIS CONTROL
  /SETTING UNIT (No. 1000A)
* COORDINATE SYSTEM(No. 1200A)
* STROKE LIMIT(No. 1300A)
* FEED RATE  (No. 1400A)
* ACCEL/DECELERATION CTRL(No. 1600A)
* SERVORELATED(No. 1800A)
* DI/DO    (No. 3000A)

>_                                S     0 T0000
MEM **** *** ***        10:12:25
[ 1 ALM ][ 2 OPR ][ 3 PARA ][      ][      ]
```

Fig. 13(i) PARAMETER TABLE screen

7 To exit from the help screen, press the HELP key or another function key.

Explanation

● Configuration of the Help Screen



IV. MAINTENANCE

1 METHOD OF REPLACING BATTERY

This chapter describes the method of replacing batteries as follows.

1.1 REPLACING CNC BATTERY FOR MEMORY BACK–UP
1.2 REPLACING BATTERIES FOR ABSOLUTE PULSE CODER
1.3 REPLACING BATTERY FOR ABSOLUTE PULSE CODER (α series Servo Amp Module)

Battery for Memory Backup

The CNC has a battery to backup the memory devices for part programs, offset data, system parameters and so on. When the battery was reaching the low level, the CNC would display a warning sign BAT on the screen before losing the important data in the memory.
When you find the sign BAT, please replace the battery as soon as possible. If you left the CNC power off for more than one week without changing the battery, the data in the CNC memory could get lost.

Battery for Absolute Pulse Coder

When the machine is equipped with absolute encoder. System such as an absolute pulse coder or absolute linear scale, there is a battery for them separately from the battery for memory backup.
When you get an alarm message No. 307 or 308 APC alarm, please replace the battery as soon as possible following the instructions in 1.2 or 1.3, or the absolute position could be lost and it would be required to take a procedure of manual reference point return.

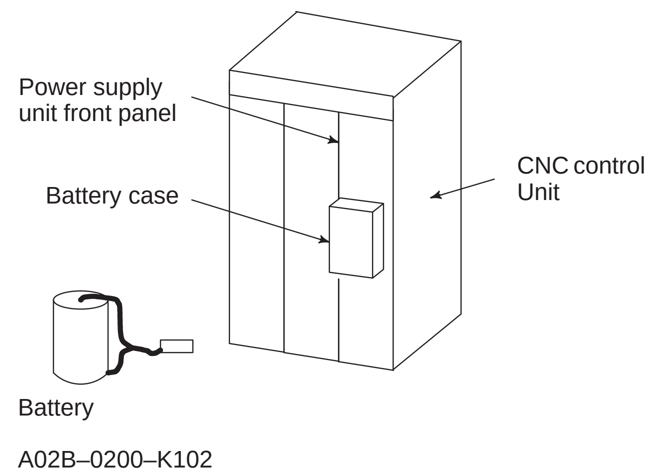
1.1 REPLACING CNC BATTERY FOR MEMORY BACK–UP

Replace CNC battery (lithium battery) for memory back–up by the following procedure.
Prepare lithium battery A02B–0200–K102 in advance.

Procedure for replacing CNC battery for memory back–up

1 Turn machine (NC) power ON. (See Notes below)

2 Remove the battery case on the front panel of power supply unit. The battery case can be removed by holding the top of the case and pulling the case towards you .



3 Remove the connector of the battery.

4 Replace the battery, and connect the connector.

5 Attach the battery case.

6 Turn machine (NC) power OFF.

Notes

The batteries can be replaced whether NC power is ON or OFF. However, the replacement should be done within 30 minutes when the power is OFF. Memory contents might be erased if the NC power is OFF without battery for more than 30 minutes.
When the contents of memory are lost, turning on the power to the CNC may cause a RAM PARITY system alarm to occur. In this case, the CNC cannot be used.

1.2 REPLACING BATTERIES FOR ABSOLUTE PULSE CODER

Replace batteries (alkaline batteries) for absolute pulse coder by the following procedure.
Prepare 4 alkaline batteries (UM–1type) commercially available in advance.

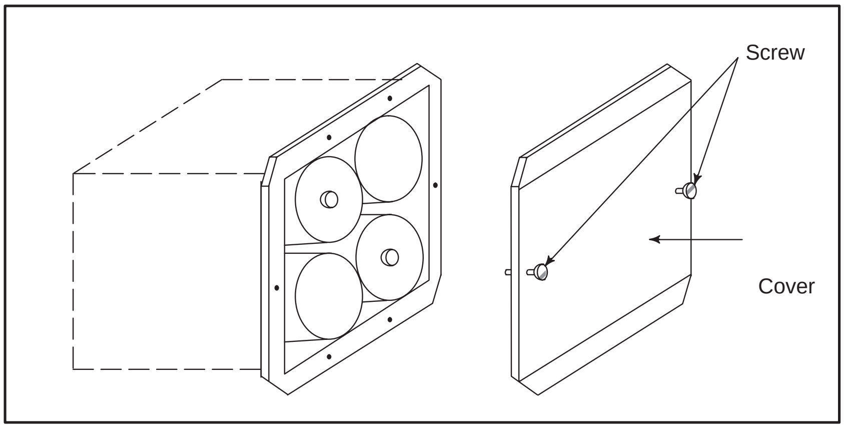
Procedure for replacing batteries for absolute pulse coder

Procedure

1 Turn machine (NC) power ON.

2 Loosen screws on the battery case to remove the cover. For placement of the battery case, refer to the machine tool builder's manual.

3 Replace the batteries in the case. Insert 2 batteries each in the opposite direction as illustrated below.



4 After replacement, install the cover.

5 Turn machine (NC) power OFF

Notes
Replace the batteries for absolute pulse coder when NC power is ON.
Replacing the batteries with power OFF causes the absolute position stored in memory to be lost.

1.3 REPLACING BATTERIES FOR ABSOLUTE PULSE CODER (α Series Servo Amp Module)

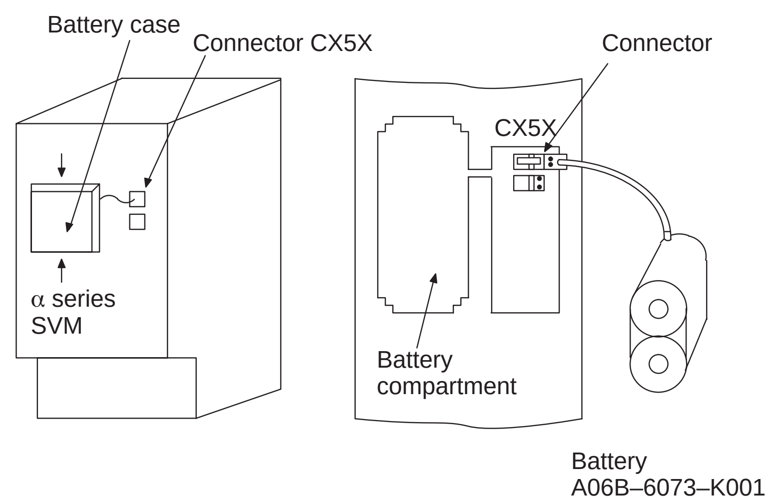
In case that the α series servo drive is used, the battery for absolute pulse coder could be provided on the α series servo amplifier module instead of the battery case as shown in 1.3. In this case the battery is not an Alkaline battery but a lithium battery, A06B–6073–K001. Prepare the battery in advance and replace it by the following procedure.

Procedure for replacing batteries for absolute pulse coder

Procedure

1 Turn machine (NC) power ON.

2 Remove the battery case on the front panel of α series Servo Amp Module (SVM).
The battery case can be removed by holding the top of the case and pulling the case towards you.



3 Remove the connector the battery.

4 Replace the battery , and connect the connector.

5 Attach the battery case.

6 Turn machine (NC)power OFF.

Notes

1 Replace the batteries for absolute pulse coder when NC power is ON.
Replacing the batteries with power OFF causes the absolute position stored in memory to be lost.

2 If your machine is equipped with a separate battery case, follow the instructions in 1.2.

APPENDIX

A TAPE CODE LIST

ISO code										EIA code										Meaning	
Character	8	7	6	5	4		3	2	1	Character	8	7	6	5	4		3	2	1		
0			○	○		○				0			○			○					Number 0
1	○		○	○		○			○	1						○			○		Number 1
2	○		○	○		○		○		2						○		○			Number 2
3			○	○		○		○	○	3				○		○		○	○		Number 3
4	○		○	○		○	○			4						○	○				Number 4
5			○	○		○	○		○	5				○		○	○		○		Number 5
6			○	○		○	○	○		6				○		○	○	○			Number 6
7	○		○	○		○	○	○	○	7						○	○	○	○		Number 7
8	○		○	○	○	○				8					○	○					Number 8
9			○	○	○	○			○	9				○	○	○			○		Number 9
A		○				○			○	a		○	○			○			○		Address A
B		○				○		○		b		○	○			○		○			Address B
C	○	○				○		○	○	c		○	○	○		○		○	○		Address C
D		○				○	○			d		○	○			○	○				Address D
E	○	○				○	○		○	e		○	○	○		○	○		○	?	Address E
F	○	○				○	○	○		f		○	○	○		○	○	○			Address F
G		○				○	○	○	○	g		○	○			○	○	○	○		Address G
H		○			○	○				h		○	○		○	○					Address H
I	○	○			○	○			○	i		○	○	○	○	○			○		Address I
J	○	○			○	○		○		j		○		○		○		○	○		Address J
K		○			○	○		○	○	k		○		○		○		○			Address K
L	○	○			○	○	○			l		○				○		○	○	?	Address L
M		○			○	○	○		○	m		○		○		○	○				Address M
N		○			○	○	○	○		n		○				○	○		○		Address N
O	○	○			○	○	○	○	○	o		○				○	○	○			Address O
P		○		○		○				p		○		○		○	○	○	○		Address P
Q	○	○		○		○			○	q		○		○	○	○					Address Q
R	○	○		○		○		○		r		○			○	○			○		Address R
S		○		○		○		○	○	s			○	○		○		○			Address S
T	○	○		○		○	○			t			○			○		○	○		Address T
U		○		○		○	○		○	u			○	○		○	○			?	Address U
V		○		○		○	○	○		v			○			○	○		○	?	Address V
W	○	○		○		○	○	○	○	w			○			○	○	○		?	Address W
X	○	○		○	○	○				x			○	○		○	○	○	○		Address X
Y		○		○	○	○			○	y			○	○	○	○					Address Y
Z		○		○	○	○		○		z			○		○	○			○		Address Z

ISO code										EIA code										Meaning	
Character	8	7	6	5	4		3	2	1	Character	8	7	6	5	4		3	2	1	Without custom macro B	With custom macro B
DEL	○	○	○	○	○	o	○	○	○	Del		○	○	○	○	o	○	○	○	×	×
NUL						o				Blank						o				×	×
BS	○				○	o				BS			○		○	o		○		×	×
HT					○	o			○	Tab			○	○	○	o	○	○		×	×
LF or NL					○	o		○		CR or EOB	○					o					
CR	○				○	o	○		○	___										×	×
SP	○		○			o				SP				○		o				□	□
%	○		○			o	○		○	ER					○	o		○	○		
(			○		○	o				(2–4–5)				○	○	o		○			
)	○		○		○	o			○	(2–4–7)		○			○	o		○			
+			○		○	o		○	○	+		○	○	○		o				Δ	
–			○		○	o	○		○	–		○				o					
:			○	○	○	o		○		___											
/	○		○		○	o	○	○	○	/			○	○		o			○		
.			○		○	o	○	○		.		○	○		○	o		○	○		
#	○		○			o		○	○	Parameter (No.6012)											
$			○			o	○			___										Δ	
&	○		○			o	○	○		&					○	o	○	○		Δ	
'			○			o	○	○	○	___										Δ	
*	○		○		○	o		○		Parameter (No.6010)										Δ	
,	○		○		○	o	○			,			○	○	○	o		○	○		
;	○		○	○	○	o		○	○	___										Δ	
<			○	○	○	o	○			___										Δ	
=	○		○	○	○	o	○		○	Parameter (No.6011)										Δ	
>	○		○	○	○	o	○	○		___										Δ	
?			○	○	○	o	○	○	○	___										Δ	
@	○	○				o				___										Δ	
”			○					○		___										Δ	
[	○	○		○	○	o		○	○	Parameter (No.6013)										Δ	
]	○	○		○	○	o	○		○	Parameter (No.6014)										Δ	

Notes

1. The symbols used in the remark column have the following meanings.

(Space) :

The character will be registered in memory and has a specific meaning.
If it is used incorrectly in a statement other than a comment, an alarm occurs.

× : The character will not be registered in memory, but will be ignored.

Δ : The character will be registered in memmory, but will be ignored during program execution.

○ : The character will be registered in memory. If it is used in a statement other than a comment, an alarm occurs.

□ : If it is used in a statement other than a comment, the character will not be registered in memory. If it is used in a comment, it will be registered in memory.

2. Codes not in this table are ignored if their parity is correct.

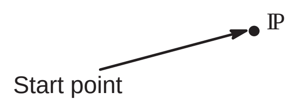
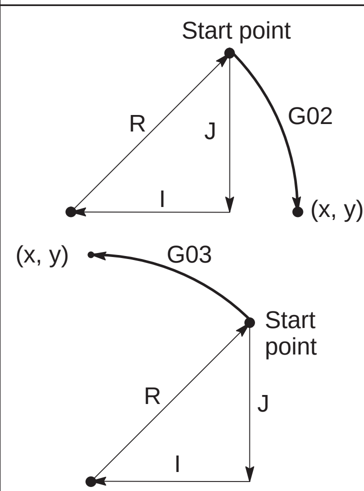
3. Codes with incorrect parity cause the TH alarm. But they are ignored without generating the TH alarm when they are in the comment section.

4. A character with all eight holes punched is ignored and does not generate TH alarm in EIA code.

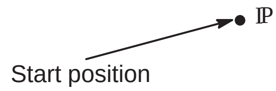
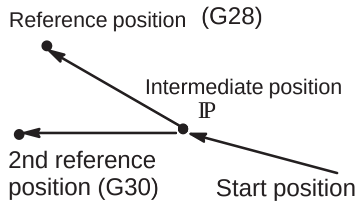
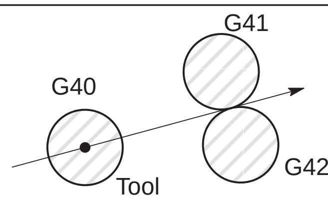
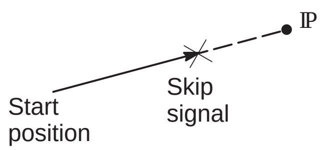
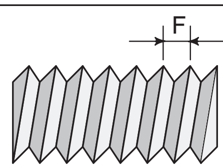
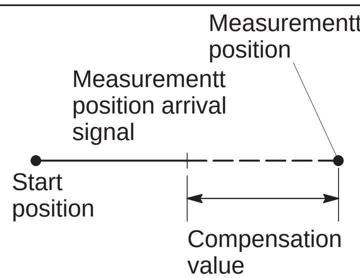
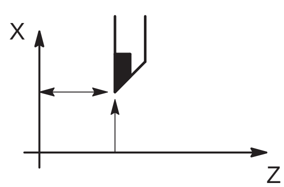
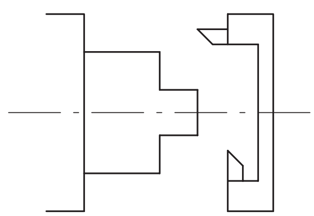
B LIST OF FUNCTIONS AND TAPE FORMAT

Some functions cannot be added as options depending on the model.
In the tables below, IP _:presents a combination of arbitrary axis addresses using X and Z.
x = 1st basic axis (X usually)
z = 2nd basic axis (Z usually)

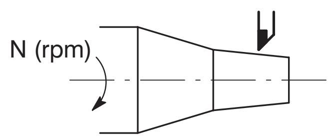
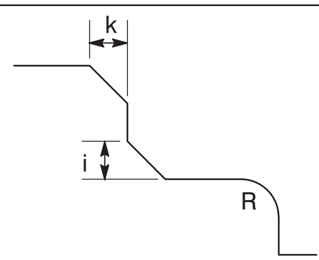
(1/3)

Functions	Illustration	Tape format
Positioning (G00)		G00 IP_ ;
Linear interpolation (G01)		G01 IP_ F_;
Circular interpolation (G02, G03)		{G02 / G03} X_ Z_ {R_ / I_ K_} F_ ;
Dwell (G04)		G04 {X_ / P_} ;
Change of offsetvalue by program(G10)		Tool geometry offset value G10 P_ X_ Z_ R_ Q_ ; P=1000+Geometry offset number Tool wear offset value G10 P_ X_ Z_ R_ Q_ ; P=Wear offset number
Inch/metric conversion (G20, G21)		Inch input : G20 Metric input : G21
Spindle speed fluctuation detection (G25, G26)		G25 ; G26 P_ Q_ R_ ;

(2/3)

Functions	Illustration	Tape format
Reference position return check (G27)		G27 IP_ ;
Reference position return (G28) 2nd, reference position return (G30)		G28 IP_ ; G30 IP_ ;
Cutter compensation (G40, G41, G42)		{G41 / G42} P_ ; P : Tool offset number G40 : Cancel
Skip fubction (G31)		G31 IP_ F_;
Thread cutting (G32)		Equal lead thread cutting G32 IP_ F_;
Automatic tool compensation (G36, G37)		G36 X xa ; G37 Z za ;
Coordinate system setting Spindle speed setting (G50)		G50 X_ Z_ ; Coordinate system setting G50 S_ ; Spindle speed setting
Mirror image for double turret (G68, G69)		G68 ; Mirror image for double turret on G69 ; Mirror image cancel

(3/3)

Functions	Illustration	Tape format
Feed per minute (G98) Feed per revolution (G99)	mm/min inch/min mm/rev inch/rev	G98 … F_ ; G99 … F_ ;
Constant surface speed control (G96/G97)	m/min or feet/min  	G96 S_ ; G97 ; Cancel
Chamfering, Corner R	 	X_ ; $\left\{\begin{array}{l} C \pm k \\ R\_ \end{array}\right\}$ P_ ; Z_ ; $\left\{\begin{array}{l} C \pm i \\ R\_ \end{array}\right\}$ P_ ;
Canned cycle (G71 to G76) (G90, G92, G94)	Refer to II.14. FUNCTIONS TO SIMPLIFY PROGRAMMING	N_ G70 P_ Q_ ; G71 U_ R_ ; G71 P_ Q_ U_ W_ F_ S_ T_ ; G72 W_ R_ ; G72 P_ Q_ U_ W_ F_ S_ T_ ; G73 U_ W_ R_ ; G73 P_ Q_ U_ W_ F_ S_ T_ ; G74 R_ ; G74 X(u)_ Z(w)_ P_ Q_ R_ F_ ; G75 R_ ; G75 X(u)_ Z(w)_ P_ Q_ R_ F_ ; G76 P_ Q_ R_ ; G76 X(u)_ Z(w)_ P_ Q_ R_ F_ ; $\left\{\begin{array}{l} G90 \\ G92 \end{array}\right\}$ X_ Z_ I_ F_ ; G94 X_ Z_ K_ F_ ;

C RANGE OF COMMAND VALUE

Linear axis

- **In case of millimeter input, feed screw is millimeter**

	Increment system	
	IS–B	**IS–C**
Least input increment	0.001 mm	0.0001 mm
Least command increment	X : 0.0005 mm Y : 0.001 mm	X : 0.00005 mm Y : 0.0001 mm
Max. programmable dimension	±99999.999 mm	±9999.9999 mm
Max. rapid traverse ***1**	240000 mm/min	100000 mm/min
Feedrate range ***1**	Feed per minute : 1 to 240000 mm/min Feed per revolution 0.0001 to 500.0000 mm/rev	Feed per minute : 1 to 100000 mm/min Feed per revolution 0.0001 to 500.0000 mm/rev
Incremental feed	0.001, 0.01, 0.1, 1mm/step	0.0001, 0.001, 0.01, 0.1 mm/step
Tool compensation	0 to ±999.999 mm	0 to ±999.9999 mm
Backlash compensation	0 to ±0.255 mm	0 to ±0.255 mm
Dwell time	0 to 99999.999 sec	0 to 99999.999 sec

- **In case of inch input, feed screw is millimeter**

	Increment system	
	IS–B	**IS–C**
Least input increment	0.0001 inch	0.00001 inch
Least command increment	X : 0.00005 inch Y : 0.0001 inch	X : 0.000005 inch Y : 0.00001 inch
Max. programmable dimension	±9999.9999 inch	±393.70078 inch
Max. rapid traverse ***1**	240000 mm/min	100000 mm/min
Feedrate range ***1**	Feed per minute : 0.01 to 9600 inch/min Feed per revolution 0.000001 to 9.999999 inch/rev	Feed per minute : 0.01 to 4000 inch/min Feed per revolution 0.000001 to 9.999999 inch/rev
Incremental feed	0.0001, 0.001, 0.01, 0.1 inch/step	0.00001, 0.0001, 0.001, 0.01 inch/step
Tool compensation	0 to ±99.9999 inch	0 to ±99.9999 inch
Backlash compensation	0 to ±0.255 mm	0 to ±0.255 mm
Dwell time	0 to 99999.999 sec	0 to 9999.9999 sec

- **In case of inch input, feed screw is inch**

	Increment system	
	IS–B	**IS–C**
Least input increment	0.0001 inch	0.00001 inch
Least command increment	X : 0.00005 inch Y : 0.0001 inch	X : 0.000005 inch Y : 0.00001 inch
Max. programmable dimension	±9999.9999 inch	±999.99999 inch
Max. rapid traverse ***1**	9600 inch/min	4000 inch/min
Feedrate range ***1**	Feed per minute : 0.01 to 9600 inch/min Feed per revolution 0.000001 to 9.999999 inch/rev	Feed per minute : 0.01 to 4000 inch/min Feed per revolution 0.000001 to 9.999999 inch/rev
Incremental feed	0.0001, 0.001, 0.01, 0.1 inch/step	0.00001, 0.0001, 0.001, 0.01 inch/step
Tool compensation	0 to ±99.9999 inch	0 to ±99.9999 inch
Backlash compensation	0 to ±0.0255 inch	0 to ±0.0255 inch
Dwell time	0 to 99999.999 sec	0 to 9999.9999 sec

- **In case of millimeter input, feed screw is inch**

	Increment system	
	IS–B	**IS–C**
Least input increment	0.001 mm	0.0001 mm
Least command increment	X : 0.00005 inch Y : 0.0001 inch	X : 0.000005 inch Y : 0.00001 inch
Max. programmable dimension	±99999.999 mm	±9999.9999 mm
Max. rapid traverse ***1**	9600 inch/min	960 inch/min
Feedrate range ***1**	Feed per minute : 1 to 240000 mm/min Feed per revolution 0.0001 to 500.0000 mm/rev	Feed per minute : 1 to 100000 mm/min Feed per revolution 0.0001 to 500.0000 mm/rev
Incremental feed	0.001, 0.01, 0.1, 1mm/step	0.0001, 0.001, 0.01, 0.1 mm/step
Tool compensation	0 to ±999.999 mm	0 to ±999.9999 mm
Backlash compensation	0 to ±0.0255 inch	0 to ±0.0255 inch
Dwell time	0 to 99999.999 sec	0 to 9999.9999 sec

Rotation axis

	Increment system	
	IS–B	**IS–C**
Least input increment	0.001 deg	0.0001 deg
Least command increment	±0.001 deg	±0.0001 deg
Max. programmable dimension	±99999.999 deg	±9999.9999 deg
Max. rapid traverse ***1**	240000 deg/min	100000 deg/min
Feedrate range ***1**	1 to 240000 deg/min	1 to 100000 deg/min
Incremental feed	0.001, 0.01, 0.1, 1deg/step	0.0001, 0.001, 0.01, 0.1 deg/step
Tool compensation	0 to ±999.999 mm	0 to ±999.9999 mm
Backlash compensation	0 to ±0.255 deg	0 to ±0.255 deg

Note

Note

*1 The feedrate range shown above are limitations depending on CNC interpolation capacity.
As a whole system, limitations depending on servo system must also be considered.

D NOMOGRAPHS

D.1 INCORRECT THREADED LENGTH

The leads of a thread are generally incorrect in δ_1 and δ_2, as shown in Fig. D.1 (a), due to automatic acceleration and deceleration.
Thus distance allowances must be made to the extent of δ_1 and δ_2 in the program.

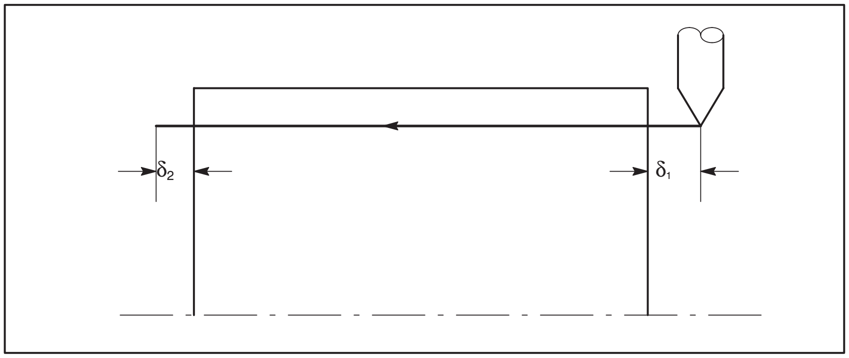


Fig.D.1(a) Incorrect thread position

Explanations

● How to determine δ_2

$$\delta_2 = T_1 V \text{ (mm)} \quad \ldots\ldots\ldots \quad (1)$$

$$V = \frac{1}{60} RL$$

T_1 : Time constant of servo system (sec)
V : Cutting speed (mm/sec)
R : Spindle speed (rpm)
L : Thread feed (mm)

Time constant T_1 (sec) of the servo system: Usually 0.033 s.

● How to determine δ_1

$$\delta_1 = \{t - T_1 + T_1 \exp(-\frac{t}{T_1})\} V \quad \ldots\ldots\ldots \quad (2)$$

$$a = \exp(-\frac{t}{T_1}) \quad \ldots\ldots\ldots \quad (3)$$

T_1 : Time constant of servo system (sec)
V : Cutting speed (mm/sec)

Time constant T_1 (sec) of the servo system: Usually 0.033 s.

The lead at the beginning of thread cutting is shorter than the specified lead L, and the allowable lead error is ΔL. Then as follows.

$$a = \frac{\Delta L}{L}$$

When the value of HαI is determined, the time lapse until the thread accuracy is attained. The time HtI is substituted in (2) to determine δ_1: Constants V and T_1 are determined in the same way as for δ_2. Since the calculation of δ_1 is rather complex, a nomography is provided on the following pages.

- **How to use nomograph**

First specify the class and the lead of a thread. The thread accuracy, α, will be obtained at (1), and depending on the time constant of cutting feed acceleration/ deceleration, the δ_1 value when V = 10mm / s will be obtained at (2). Then, depending on the speed of thread cutting, δ_1 for speed other than 10mm/ s can be obtained at (3).

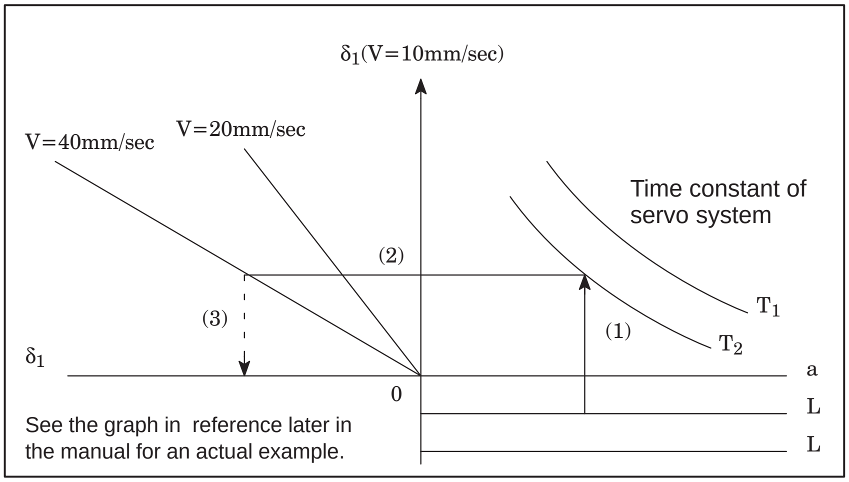


Fig.D.1(b) Nomograph

Notes

The equations for δ_1, and δ_2 are for when the acceleration/deceleration time constant for cutting feed is 0.

D.2 SIMPLE CALCULATION OF INCORRECT THREAD LENGTH

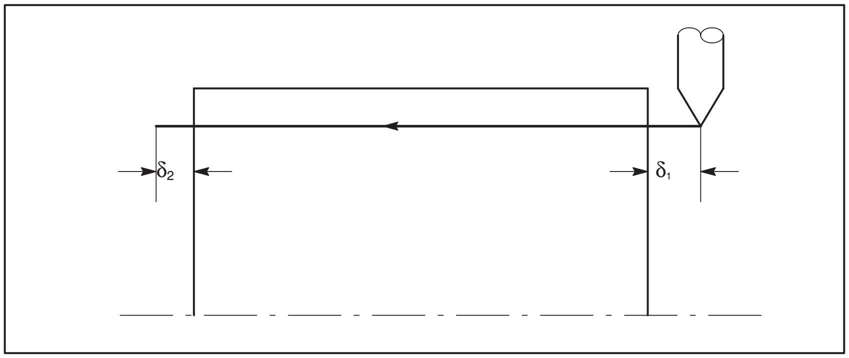


Fig. D.2(a) Incorrect threaded portion

Explanations

● How to determine δ_2

$$\delta_2 = \frac{LR}{1800\,*} \text{ (mm)}$$

R : Spindle speed (rpm)
L : Thread lead (mm)

* When time constant T of the servo system is 0.033 s.

● How to determine δ_1

$$\delta_1 = \frac{LR}{1800\,*}(-1-\ln a) \text{ (mm)}$$

$$= \delta_2(-1-\ln a) \text{ (mm)}$$

R : Spindle speed (rpm)
L : Thread lead (mm)

* When time constant T of the servo system is 0.033 s.

Following a is a permitted value of thread.

a	–1–lna
0.005	4.298
0.01	3.605
0.015	3.200
0.02	2.912

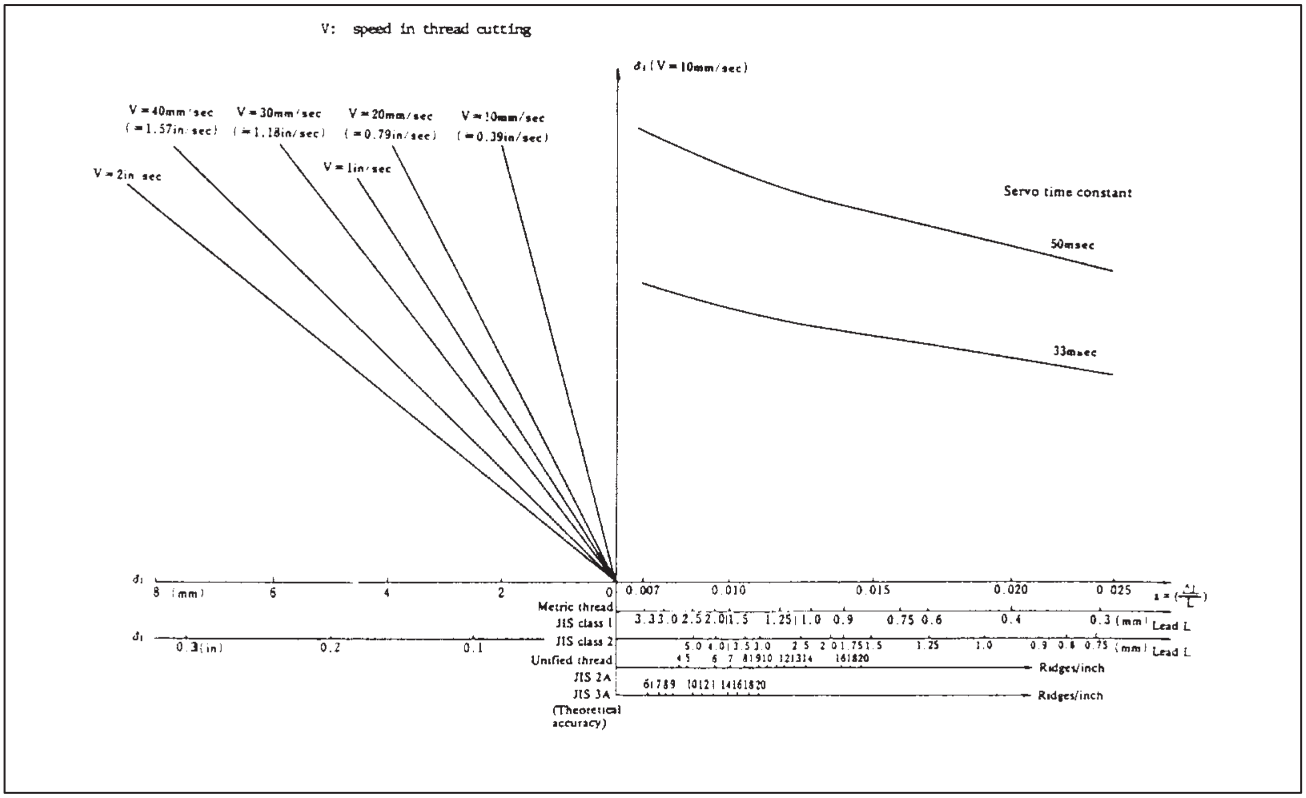
Examples

R=350rpm
L=1mm
a=0.01 then

$$\delta_2 = \frac{350 \times 1}{1800} = 0.194 \text{ (mm)}$$

$$\delta_1 = \delta_2 \times 3.605 = 0.701 \text{ (mm)}$$

- **Reference**



Nomograph for obtaining approach distance δ_1

D.3 TOOL PATH AT CORNER

When servo system delay (by exponential acceleration/deceleration at cutting or caused by the positioning system when a servo motor is used) is accompanied by cornering, a slight deviation is produced between the tool path (tool center path) and the programmed path as shown in Fig. D.3 (a).
Time constant T_1 of the exponential acceleration/deceleration is fixed to 0.

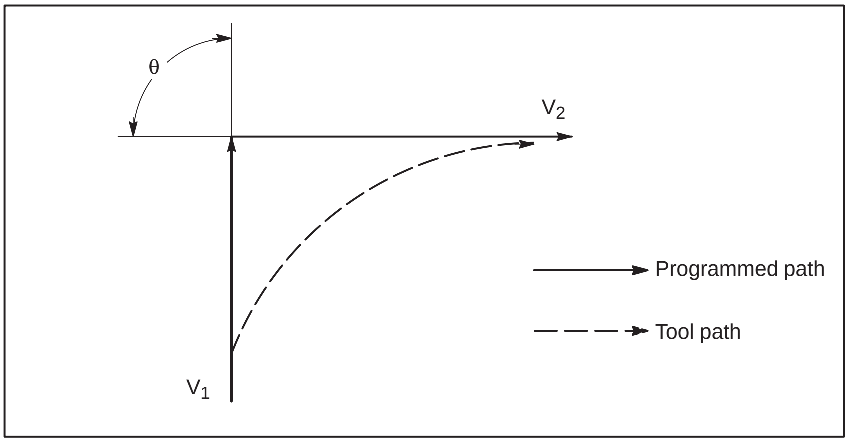


Fig. D.3 (a) Slight deviation between the tool path and the programmed path

This tool path is determined by the following parameters:
· Feedrate (V_1, V_2)
· Corner angle (θ)
· Exponential acceleration / deceleration time constant (T_1) at cutting ($T_1 = 0$)
· Presence or absence of buffer register.

The above parameters are used to theoretically analyze the tool path and above tool path is drawn with the parameter which is set as an example. When actually programming, the above items must be considered and programming must be performed carefully so that the shape of the workpiece is within the desired precision.
In other words, when the shape of the workpiece is not within the theoretical precision, the commands of the next block must not be read until the specified feedrate becomes zero. The dwell function is then used to stop the machine for the appropriate period.

Analysis

The tool path shown in Fig. D.3 (b) is analyzed based on the following conditions:
Feedrate is constant at both blocks before and after cornering.
The controller has a buffer register. (The error differs with the reading speed of the tape reader, number of characters of the next block, etc.)

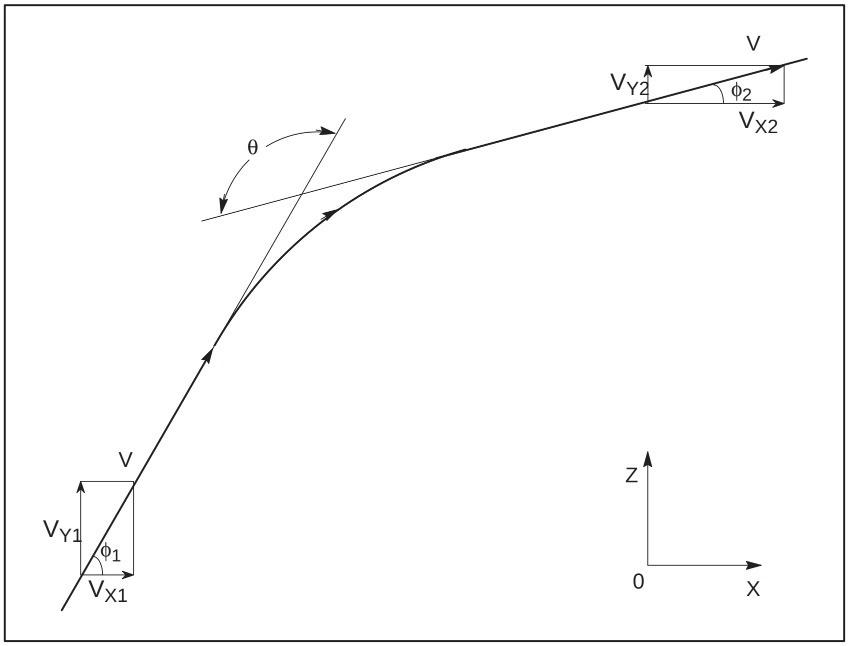


Fig. D.3(b) Example of tool path

- **Description of conditions and symbols**

$$V_{X1} = V\cos\phi_1$$

$$V_{Y1} = V\sin\phi_1$$

$$V_{X2} = V\cos\phi_2$$

$$V_{Y2} = V\sin\phi_2$$

V : Feedrate at both blocks before and after cornering
V_{X1} : X–axis component of feedrate of preceding block
V_{Y1} : Y–axis component of feedrate of preceding block
V_{X2} : X–axis component of feedrate of following block
V_{Y2} : Y–axis component of feedrate of following block
θ : Corner angle
ϕ_1 : Angle formed by specified path direction of preceding block and X–axis
ϕ_2 : Angle formed by specified path direction of following block and X–axis

- **Initial value calculation**

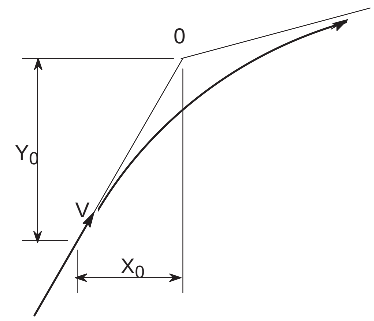


Fig. D.3(c) Initial value

The initial value when cornering begins, that is, the X and Y coordinates at the end of command distribution by the controller, is determined by the feedrate and the positioning system time constant of the servo motor.

$$X_0 = V_{X1}(T_1 + T_2)$$

$$Y_0 = V_{Y1}(T_1 + T_2)$$

T_1:Exponential acceleration / deceleration time constant. (T=0)
T_2:Time constant of positioning system (Inverse of position loop gain)

- **Analysis of corner tool path**

The equations below represent the feedrate for the corner section in X–axis direction and Y–axis direction.

$$V_X(t) = (V_{X2}-V_{X1})[1-\frac{V_{X1}}{T_1-T_2}\{T_1\exp(-\frac{t}{T_1})-T_2\exp(-\frac{t}{T_2})\} + V_{X1}]$$

$$= V_{X2}[1-\frac{V_{X1}}{T_1-T_2}\{T_1\exp(-\frac{t}{T_1})-T_2\exp(-\frac{t}{T_2})\}]$$

$$V_Y(t) = \frac{V_{Y1}-V_{Y2}}{T_1-T_2}\{T_1\exp(-\frac{t}{T_1})-T_2\exp(-\frac{t}{T_2})\} + V_{Y2}$$

Therefore, the coordinates of the tool path at time t are calculated from the following equations:

$$X(t) = \int_0^t V_X(t)dt-X_0$$

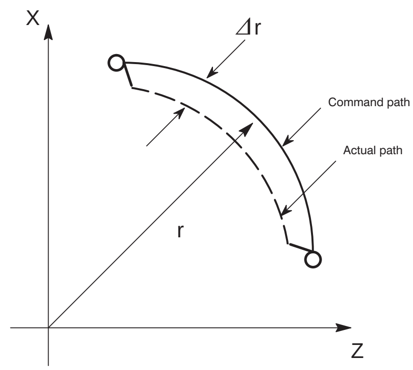
$$= \frac{V_{X2}-V_{X1}}{T_1-T_2}\{T_1^{\,2}\exp(-\frac{t}{T_1})-T_2^{\,2}\exp(-\frac{t}{T_2})\}-V_{X2}(T_1 + T_2-t)$$

$$Y(t) = \int_0^t V_Y(t)dt-Y_0$$

$$= \frac{V_{Y2}-V_{Y1}}{T_1-T_2}\{T_1^{\,2}\exp(-\frac{t}{T_1})-T_2^{\,2}\exp(-\frac{t}{T_2})\}-V_{Y2}(T_1 + T_2-t)$$

D.4 RADIUS DIRECTION ERROR AT CIRCLE CUTTING

When a servo motor is used, the positioning system causes an error between input commands and output results. Since the tool advances along the specified segment, an error is not produced in linear interpolation. In circular interpolation, however, radial errors may be produced, specially for circular cutting at high speeds.
This error can be obtained as follows:



$$\Delta r = \frac{1}{2}(T_1^{\,2} + T_2^{\,2})\frac{V^2}{r} \quad \ldots\ldots\ (1)$$

- Δr : Maximum radius error (mm)
- v : Feedrate (mm/s)
- r : Circle radius (mm)
- T_1 : Exponential acceleration/deceleration time constant (sec) at cutting (T=0)
- T_2 : Time constant of positoning system (sec). (Inverse of positon loop gain)

In the case of bell–shaped acceleration/deceleration and linear acceleration/deceleration after cutting feed interpolation, an approximation of this radius error can be obtained with the following expression:

$$\Delta r = (\frac{1}{24}T_1^{\,2} + \frac{1}{2}T_2^{\,2})\frac{V^2}{r}$$

Thus, the radius error in the case of bell–shaped acceleration/deceleration and linear acceleration/deceleration after interpolation is smaller than in case of exponential acceleration/deceleration by a factor of 12, excluding any error caused by a servo loop time constant.

Fig. D.4(a) Radius direction error of circular cutting

Since the machining radius r (mm) and allowable errorΔr (mm) of the workpiece is given in actual machining, the allowable limit feedrate v (mm /sec) is determined by equation (1).
Since the acceleration/deceleration time constant at cutting which is set by this equipment varies with the machine tool, refer to the manual issued by the machine tool builder.

E STATUS WHEN TURNING POWER ON, WHEN CLEAR AND WHEN RESET

Parameter 3402 (CLR) is used to select whether resetting the CNC places it in the cleared state or in the reset state (0: reset state/1: cleared state).
The symbols in the tables below mean the following :
○:The status is not changed or the movement is continued.
×:The status is cancelled or the movement is interrupted.

Item		When turning power on	Cleared	Reset
Setting data	Offset value	○	○	○
	Data set by the MDI setting operation	○	○	○
	Parameter	○	○	○
Various data	Programs in memory	○	○	○
	Contents in the buffer storage	×	×	○ : MDI mode × : Other mode
	Display of sequence number	○	○ (Note 1)	○ (Note 1)
	One shot G code	×	×	×
	Modal G code	Initial G codes. (The G20 and G21 codes return to the same state they were in when the power was last turned off.)	Initial G codes. (G20/G21 are not changed.)	○
	F	Zero	Zero	○
	S, T, M	×	○	○
	K (Number of repeats)	×	×	×
Work coordinate value		Zero	○	○

Item		When turning power on	Cleared	Reset
Action in operation	Movement	×	×	×
	Dwell	×	×	×
	Issuance of M, S and T codes	×	×	×
	Tool offset	×	Depending on parameter LVK(No.5003#6)	○ : MDI mode Other modes depend on parameter LVK(No.5003#6).
	Tool nose radius compensation	×	×	○ : MDI mode × : Other modes
	Storing called subprogram number	×	× (Note 2)	○ : MDI mode × : Other modes (Note 2)
Output signals	CNC alarm signal AL	Extinguish if there is no cause for the alarm	Extinguish if there is no cause for the alarm	Extinguish if there is no cause for the alarm
	Reference position return completion LED	×	○ (× : Emergency stop)	○ (× : Emergency stop)
	S, T and B codes	×	○	○
	M code	×	×	×
	M, S and T strobe signals	×	×	×
	Spindle revolution signal (S analog signal)	×	○	○
	CNC ready signal MA	ON	○	○
	Servo ready signal SA	ON (When other than servo alarm)	ON (When other than servo alarm)	ON (When other than servo alarm)
	Cycle start LED (STL)	×	×	×
	Feed hold LED (SPL)	×	×	×

Notes

1. When heading is performed, the main program number is displayed.
2. When a reset is performed during execution of a subprogram, control returns the main program.
   Execution cannot be started from the middle of the subprogram.

F CHARACTER–TO–CODES CORRESPONDENCE TABLE

Character	Code	Comment	Character	Code	Comment
A	065		6	054	
B	066		7	055	
C	067		8	056	
D	068		9	057	
E	069			032	Space
F	070		!	033	Exclamation mark
G	071		”	034	Quotation mark
H	072		#	035	Hash sign
I	073		$	036	Dollar sign
J	074		%	037	Percent
K	075		&	038	Ampersand
L	076		’	039	Apostrophe
M	077		(	040	Left parenthesis
N	078		)	041	Right parenthesis
O	079		*	042	Asterisk
P	080		+	043	Plus sign
Q	081		,	044	Comma
R	082		–	045	Minus sign
S	083		.	046	Period
T	084		/	047	Slash
U	085		:	058	Colon
V	086		;	059	Semicolon
W	087		<	060	Left angle bracket
X	088		=	061	Equal sign
Y	089		>	062	Right angle bracket
Z	090		?	063	Question mark
0	048		@	064	HAtI mark
1	049		[	091	Left square bracket
2	050		^	092	
3	051		o	093	Yen sign
4	052		]	094	Right square bracket
5	053		_	095	Underscore

Note
Each Japanese dakuten or handakuten is counted as one character.

G ALARM LIST

1) Program errors (P/S alarm)

Number	Message	Contents
000	PLEASE TURN OFF POWER	A parameter which requires the power off was input, turn off power.
001	TH PARITY ALARM	TH alarm (A character with incorrect parity was input). Correct the tape.
002	TV PARITY ALARM	TV alarm (The number of characters in a block is odd). This alarm will be generated only when the TV check is effective.
003	TOO MANY DIGITS	Data exceeding the maximum allowable number of digits was input. (Refer to the item of max. programmable dimensions.)
004	ADDRESS NOT FOUND	A numeral or the sign “ – ” was input without an address at the beginning of a block. Modify the program .
005	NO DATA AFTER ADDRESS	The address was not followed by the appropriate data but was followed by another address or EOB code. Modify the program.
006	ILLEGAL USE OF NEGATIVE SIGN	Sign “ – ” input error (Sign “ – ” was input after an address with which it cannot be used. Or two or more “ – ” signs were input.) Modify the program.
007	ILLEGAL USE OF DECIMAL POINT	Decimal point “ . ” input error (A decimal point was input after an address with which it can not be used. Or two decimal points were input.) Modify the program.
009	ILLEGAL ADDRESS INPUT	Unusable character was input in significant area. Modify the program.
010	IMPROPER G–CODE	An unusable G code or G code corresponding to the function not provided is specified. Modify the program.
011	NO FEEDRATE COMMANDED	Feedrate was not commanded to a cutting feed or the feedrate was inadequate. Modify the program.
014	ILLEGAL LEAD COMMAND	In variable lead threading, the lead incremental and decremental outputted by address K exceed the maximum command value or a command such that the lead becomes a negative value is given. Modify the program.
020	OVER TOLERANCE OF RADIUS	In circular interpolation (G02 or G03), difference of the distance between the start point and the center of an arc and that between the end point and the center of the arc exceeded the value specified in parameter No. 3410.
021	ILLEGAL PLANE AXIS COMMANDED	An axis not included in the selected plane (by using G17, G18, G19) was commanded in circular interpolation. Modify the program.
022	CIRCULAR INTERPOLATION	In circular interpolation, radius R, or the distance between the start point and the center of the arc, I, J, or K, has not been specified.
020	G NO CIRCLE RADIUS	When circular interpolation is specified, neither R (specifying an arc radius), nor I, J, and K (specifying the distance from a start point to the center) is specified.
023	ILLEGAL RADIUS COMMAND	In circular interpolation by radius designation, negative value was commanded for address R. Modify the program.
028	ILLEGAL PLANE SELECT	In the plane selection command, two or more axes in the same direction are commanded. Modify the program.

Number	Message	Contents
029	ILLEGAL OFFSET VALUE	The offset values specified by T code is too large. Modify the program.
030	ILLEGAL OFFSET NUMBER	The offset number in T function specified for tool offset is too large. Modify the program.
031	ILLEGAL P COMMAND IN G10	In setting an offset amount by G10, the offset number following address P was excessive or it was not specified. Modify the program.
032	ILLEGAL OFFSET VALUE IN G10	In setting an offset amount by G10 or in writing an offset amount by system variables, the offset amount was excessive.
033	NO SOLUTION AT NRC	A point of intersection cannot be determined for tool nose radius compensation. Modify the program.
034	NO CIRC ALLOWED IN ST–UP /EXT BLK	The start up or cancel was going to be performed in the G02 or G03 mode in tool nose radius compensation. Modify the program.
035	CAN NOT COMMANDED G31	Skip cutting (G31) was specified in tool nose radius compensation mode. Modify the program.
037	CAN NOT CHANGE PLANE IN NRC	The offset plane is switched in tool nose radius compensation. Modify the program.
038	INTERFERENCE IN CIRCULAR BLOCK	Overcutting will occur in tool nose radius compensation because the arc start point or end point coincides with the arc center. Modify the program.
039	CHF/CNR NOT ALLOWED IN NRC	Chamfering or corner R was specified with a start–up, a cancel, or switching between G41 and G42 in tool nose radius compensation. The program may cause overcutting to occur in chamfering or corner R. Modify the program.
040	INTERFERENCE IN G90/G94 BLOCK	Overcutting will occur in tool nose radius compensation in canned cycle G90 or G94. Modify the program.
041	INTERFERENCE IN NRC	Overcutting will occur in tool nose radius compensation. Modify the program.
046	ILLEGAL REFERENCE RETURN COMMAND	Other than P2, P3 and P4 are commanded for 2nd, 3rd and 4th reference position return command.
050	CHF/CNR NOT ALLOWED IN THRD BLK	Chamfering or corner R is commanded in the thread cutting block. Modify the program.
051	MISSING MOVE AFTER CHF/CNR	Improper movement or the move distance was specified in the block next to the chamfering or corner R block. Modify the program.
052	CODE IS NOT G01 AFTER CHF/CNR	The block next to the chamfering or corner R block is not G01. Modify the program.
053	TOO MANY ADDRESS COMMANDS	In the chamfering and corner R commands, two or more of I, K and R are specified. Otherwise, the character after a comma(”,”) is not C or R in direct drawing dimensions programming. Modify the program.
054	NO TAPER ALLOWED AFTER CHF/CNR	A block in which chamfering in the specified angle or the corner R was specified includes a taper command. Modify the program.
055	MISSING MOVE VALUE IN CHF/CNR	In chamfering or corner R block, the move distance is less than chamfer or corner R amount.
056	NO END POINT & ANGLE IN CHF/CNR	Neither the end point nor angle is specified in the command for the block next to that for which only the angle is specified (A). In the chamfering common, I(K) is commanded for the X(Z) axis.
057	NO SOLUTION OF BLOCK END	Block end point is not calculated correctly in direct dimension drawing programming.
058	END POINT NOT FOUND	Block end point is not found in direct dimension drawing programming.

Number	Message	Contents
059	PROGRAM NUMBER NOT FOUND	In an external program number search or external workpiece number search, a specified program number was not found. Otherwise, a program specified for searching is being edited in background processing. Check the program number and external signal. Or discontinue the background editing.
060	SEQUENCE NUMBER NOT FOUND	Commanded sequence number was not found in the sequence number search. Check the sequence number.
061	ADDRESS P/Q NOT FOUND IN G70–G73	Address P or Q is not specified in G70, G71, G72, or G73 command. Modify the program.
062	ILLEGAL COMMAND IN G71–G76	1 The depth of cut in G71 or G72 is zero or negative value. 2 The repetitive count in G73 is zero or negative value. 3 The negative value is specified to Δi or Δk is zero in G74 or G75. 4 A value other than zero is specified to address U or W, though Δi or Δk is zero in G74 or G75. 5 A negative value is specified to Δd, though the relief direction in G74 or G75 is determined. 6. Zero or a negative value is specified to the height of thread or depth of cut of first time in G76. 7 The specified minimum depth of cut in G76 is greater than the height of thread. 8 An unusable angle of tool tip is specified in G76. Modify the program.
063	SEQUENCE NUMBER NOT FOUND	The sequence number specified by address P in G70, G71, G72, or G73 command cannot be searched. Modify the program.
064	SHAPE PROGRAM NOT MONOTONOUSLY	A target shape which is not monotonous increase or decrease was specified in a repetitive canned cycle (G71 or G72).
065	ILLEGAL COMMAND IN G71–G73	1 G00 or G01 is not commanded at the block with the sequence number which is specified by address P in G71, G72, or G73 command. 2. Address Z(W) or X(U) was commanded in the block with a sequence number which is specified by address P in G71 or G72, respectively. Modify the program.
066	IMPROPER G–CODE IN G71–G73	An unallowable G code was commanded between two blocks specified by address P in G71, G72, or G73. Modify the program.
067	CAN NOT OPERATE IN MDI MODE	G70, G71, G72, or G73 command with address P and Q was specified. Modify the program.
069	FORMAT ERROR IN G70–G73	The final move command in the blocks specified by P and Q of G70, G71, G72, or G73 ended with chamfering or corner R.
070	NO PROGRAM SPACE IN MEMORY	The memory area is insufficient. Delete any unnecessary programs, then retry.
071	DATA NOT FOUND	The address to be searched was not found. Or the program with specified program number was not found in program number search. Check the data.
072	TOO MANY PROGRAMS	The number of programs to be stored exceeded 63 (basic), 125 (option), 200 (option), 400 (option), or 1000 (option). Delete unnecessary programs and execute program registration again.
073	PROGRAM NUMBER ALREADY IN USE	The commanded program number has already been used. Change the program number or delete unnecessary programs and execute program registration again.

Number	Message	Contents
074	ILLEGAL PROGRAM NUMBER	The program number is other than 1 to 9999. Modify the program number.
075	PROTECT	An attempt was made to register a program whose number was protected.
076	ADDRESS P NOT DEFINED	Address P (program number) was not commanded in the block which includes an M98, G65, or G66 command. Modify the program.
077	SUB PROGRAM NESTING ERROR	The subprogram was called in five folds. Modify the program.
078	NUMBER NOT FOUND	A program number or a sequence number which was specified by address P in the block which includes an M98, M99, M65 or G66 was not found. The sequence number specified by a GOTO statement was not found. Otherwise, a called program is being edited in background processing. Correct the program, or discontinue the background editing.
079	PROGRAM VERIFY ERROR	In memory or program collation,a program in memory does not agree with that read from an external I/O device. Check both the programs in memory and those from the external device.
080	G37 ARRIVAL SIGNAL NOT ASSERTED	In the automatic tool compensation function (G36, G37), the measurement position reach signal (XAE or ZAE) is not turned on within an area specified in parameter 6254 (value ε). This is due to a setting or operator error.
081	OFFSET NUMBER NOT FOUND IN G37	Automatic tool compensation (G36, G37) was specified without a T code. (Automatic tool compensation function) Modify the program.
082	T–CODE NOT ALLOWED IN G37	T code and automatic tool compensation (G36, G37) were specified in the same block. (Automatic tool compensation function) Modify the program.
083	ILLEGAL AXIS COMMAND IN G37	In automatic tool compensation (G36, G37), an invalid axis was specified or the command is incremental. Modify the program.
085	COMMUNICATION ERROR	When entering data in the memory by using Reader / Puncher interface, an overrun, parity or framing error was generated. The number of bits of input data or setting of baud rate or specification No. of I/O unit is incorrect.
086	DR SIGNAL OFF	When entering data in the memory by using Reader / Puncher interface, the ready signal (DR) of reader / puncher was turned off. Power supply of I/O unit is off or cable is not connected or a P.C.B. is defective.
087	BUFFER OVERFLOW	When entering data in the memory by using Reader / Puncher interface, though the read terminate command is specified, input is not interrupted after 10 characters read. I/O unit or P.C.B. is defective.
088	G LAN FILE TRANS ERROR (CHANNEL-1)	File data transfer over the OSI–Ethernet was terminated as a result of a transfer error.
089	G LAN FILE TRANS ERROR (CHANNEL-2)	File data transfer over the OSI–Ethernet was terminated as a result of a transfer error.
090	REFERENCE RETURN INCOMPLETE	The reference position return cannot be performed normally because the reference position return start point is too close to the reference position or the speed is too slow. Separate the start point far enough from the reference position, or specify a sufficiently fast speed for reference position return. Check the program contents.
091	REFERENCE RETURN INCOMPLETE	In the automatic operation halt state, manual reference position return cannot be performed.
092	AXES NOT ON THE REFERENCE POINT	The commanded axis by G27 (Reference position return check) did not return to the reference position.

Number	Message	Contents
094	P TYPE NOT ALLOWED (COORD CHG)	P type cannot be specified when the program is restarted. (After the automatic operation was interrupted, the coordinate system setting operation was performed.) Perform the correct operation according to th operator's manual.
095	P TYPE NOT ALLOWED (EXT OFS CHG)	P type cannot be specified when the program is restarted. (After the automatic operation was interrupted, the external workpiece offset amount changed.) Perform the correct operation according to th operator's manual.
096	P TYPE NOT ALLOWED (WRK OFS CHG)	P type cannot be specified when the program is restarted. (After the automatic operation was interrupted, the workpiece offset amount changed.) Perform the correct operation according to th operator's manual.
097	P TYPE NOT ALLOWED (AUTO EXEC)	P type cannot be directed when the program is restarted. (After power ON, after emergency stop or P/S alarm 94 to 97 were reset, no automatic operation was performed.) Perform automatic operation.
098	G28 FOUND IN SEQUENCE RETURN	A command of the program restart was specified without the reference position return operation after power ON or emergency stop, and G28 was found during search. Perform the reference position return.
099	MDI EXEC NOT ALLOWED AFT. SEARCH	After completion of search in program restart, a move command is given with MDI.
100	PARAMETER WRITE ENABLE	On the PARAMETER(SETTING) screen, PWE(parameter writing enabled) is set to 1. Set it to 0, then reset the system.
101	PLEASE CLEAR MEMORY	The power turned off while rewriting the memory by program edit operation. If this alarm has occurred, press <RESET> while pressing <PROG>, and only the program being edited will be deleted. Register the deleted program.
109	P/S ALARM	A value other than 0 or 1 was specified after P in the G08 code, or no value was specified.
111	CALCULATED DATA OVERFLOW	The result of calculation is out of the allowable range (-10^{47} to -10^{-29}, 0, and 10^{-29} to 10^{47}).
112	DIVIDED BY ZERO	Division by zero was specified. (including tan 90°) Modify the program.
113	IMPROPER COMMAND	A function which cannot be used in custom macro is commanded. Modify the program.
114	FORMAT ERROR IN MACRO	There is an error in other formats than <Formula>. Modify the program.

Number	Message	Contents
115	ILLEGAL VARIABLE NUMBER	A value not defined as a variable number is designated in the custom macro or in high–speed cycle cutting. The header contents are improper in a high speed cycle cutting. This alarm is given in the following cases: 1. The header corresponding to the specified machining cycle number called is not found. 2. The cycle connection data value is out of the allowable range (0 – 999). 3. The number of data in the header is out of the allowable range (0 – 32767). 4. The start data variable number of executable format data is out of the allowable range (#20000 – #85535). 5. The storing data variable number of executable format data is out of the allowable range (#85535). 6. The storing start data variable number of executable format data is overlapped with the variable number used in the header. Modify the program.
116	WRITE PROTECTED VARIABLE	The left side of substitution statement is a variable whose substitution is inhibited. Modify the program.
118	PARENTHESIS NESTING ERROR	The nesting of bracket exceeds the upper limit (quintuple). Modify the program.
119	ILLEGAL ARGUMENT	The SQRT argument is negative, BCD argument is negative, or other values than 0 to 9 are present on each line of BIN argument. Modify the program.
122	QUADRUPLE MACRO MODAL–CALL	The macro modal call is specified in double. Modify the program.
123	CAN NOT USE MACRO COMMAND IN DNC	Macro control command is used during DNC operation. Modify the program.
124	MISSING END STATEMENT	DO – END does not correspond to 1 : 1. Modify the program.
125	FORMAT ERROR IN MACRO	<Formula> format is erroneous. Modify the program.
126	ILLEGAL LOOP NUMBER	In DOn, $1 \leqq n \leqq 3$ is not established. Modify the program.
127	NC, MACRO STATEMENT IN SAME BLOCK	NC and custom macro commands coexist. Modify the program.
128	ILLEGAL MACRO SEQUENCE NUMBER	The sequence number specified in the branch command was not 0 to 9999. Or, it cannot be searched. Modify the program.
129	ILLEGAL ARGUMENT ADDRESS	An address which is not allowed in <Argument Designation > is used. Modify the program.
130	ILLEGAL AXIS OPERATION	An axis control command was given by PMC to an axis controlled by CNC. Or an axis control command was given by CNC to an axis controlled by PMC. Modify the program.
131	TOO MANY EXTERNAL ALARM MESSAGES	Five or more alarms have generated in external alarm message. Consult the PMC ladder diagram to find the cause.
132	ALARM NUMBER NOT FOUND	No alarm No. concerned exists in external alarm message clear. Check the PMC ladder diagram.
133	ILLEGAL DATA IN EXT. ALARM MSG	Small section data is erroneous in external alarm message or external operator message. Check the PMC ladder diagram.
135	SPINDLE ORIENTATION PLEASE	Without any spindle orientation , an attempt was made for spindle indexing. Perform spindle orientation.
136	C/H–CODE & MOVE CMD IN SAME BLK.	A move command of other axes was specified to the same block as spindle indexing addresses C, H. Modify the program.

Number	Message	Contents
137	M–CODE & MOVE CMD IN SAME BLK.	A move command of other axes was specified to the same block as M–code related to spindle indexing. Modify the program.
138	G SUPERIMPOSED DATA OVERFLOW	In PMC axis control, the increment for pulse distribution on the CNC and PMC side are too large when the superimposed control extended function is used.
139	CAN NOT CHANGE PMC CONTROL AXIS	An axis is selected in commanding by PMC axis control. Modify the program.
145	ILLEGAL COMMAND G112/G113	The conditions are incorrect when the polar coordinate interpolation starts or it is canceled. 1) In modes other than G40, G12.1/G13.1 was specified. 2) An error is found in the plane selection. Parameters No. 5460 and No. 5461 are incorrectly specified. Modify the value of program or parameter.
146	IMPROPER G CODE	G codes which cannot be specified in the polar coordinate interpolation mode was specified. See section II–4.4 and modify the program.
150	ILLEGAL TOOL GROUP NUMBER	Tool Group No. exceeds the maximum allowable value. Modify the program.
151	TOOL GROUP NUMBER NOT FOUND	The tool group commanded in the machining program is not set. Modify the value of program or parameter.
152	NO SPACE FOR TOOL ENTRY	The number of tools within one group exceeds the maximum value registrable. Modify the number of tools.
153	T–CODE NOT FOUND	In tool life data registration, a T code was not specified where one should be. Correct the program.
155	ILLEGAL T–CODE IN M06	In the machining program, M06 and T code in the same block do not correspond to the group in use. Correct the program.
156	P/L COMMAND NOT FOUND	P and L commands are missing at the head of program in which the tool group is set. Correct the program.
157	TOO MANY TOOL GROUPS	The number of tool groups to be set exceeds the maximum allowable value. (See parameter No. 6800 bit 0 and 1) Modify the program.
158	ILLEGAL TOOL LIFE DATA	The tool life to be set is too excessive. Modify the setting value.
159	TOOL DATA SETTING INCOMPLETE	During executing a life data setting program, power was turned off. Set again.
160	MISMATCH WATING M–CODE (only with two path control)	Different M code is commanded in heads 1 and 2 as waiting M code. Modify the program.
161	COMMAND G68/G69 INDEPENDENTLY (only with two path control)	G68 and G69 are not independently commanded in balance cut. Modify the program.
169	ILLEGAL TOOL GEOMETRY DATA (only with two path control)	Incorrect tool figure data in interference check.
175	ILLEGAL G107 COMMAND	Conditions when performing circular interpolation start or cancel not correct. To change the mode to the cylindrical interpolation mode, specify the command in a format of “G07.1 rotation–axis name radius of cylinder.”
176	IMPROPER G–CODE IN G107	Any of the following G codes which cannot be specified in the cylindrical interpolation mode was specified. 1) G codes for positioning, such as G28, G76, G81 – G89, including the codes specifying the rapid traverse cycle 2) G codes for setting a coordinate system: G50, G52 3) G code for selecting coordinate system: G53 G54–G59 Modify the program.
177	CHECK SUM ERROR (G05 MODE)	Check sum error Modify the program.

Number	Message	Contents
178	G05 NOT ALLOWED IN G41/G42 MODE	G05 was commanded in the G41/G42 mode. Correct the program.
179	PARAM. (NO. 7510) SETTING ERROR	The number of controlled axes set by the parameter 7510 exceeds the maximum number. Modify the parameter setting value.
180	COMMUNICATION ERROR (REMOTE BUF)	Remote buffer connection alarm has generated. Confirm the number of cables, parameters and I/O device.
194	SPINDLE COMMAND IN SYNCHRO–MODE	A contour control mode, spindle positioning (Cs–axis control) mode, or rigid tapping mode was specified during the serial spindle synchronous control mode. Correct the program so that the serial spindle synchronous control mode is released in advance.
197	C–AXIS COMMANDED IN SPINDLE MODE	The program specified a movement along the Cf–axis when the signal CON(DGN=G027#7) was off. Correct the program, or consult the PMC ladder diagram to find the reason the signal is not turned on.
199	MACRO WORD UNDEFINED	Undefined macro word was used. Modify the custom macro.
200	ILLEGAL S CODE COMMAND	In the rigid tapping, an S value is out of the range or is not specified. The maximum values for S which can be specified in rigid tapping is set in parameters 5241 to 5243. Change the setting in the parameter or modify the program.
201	FEEDRATE NOT FOUND IN RIGID TAP	In the rigid tapping, no F value is specified. Correct the program.
202	POSITION LSI OVERFLOW	In the rigid tapping, spindle distribution value is too large.
203	PROGRAM MISS AT RIGID TAPPING	In the rigid tapping, position for a rigid M code (M29) or an S command is incorrect. Modify the program.
204	ILLEGAL AXIS OPERATION	In the rigid tapping, an axis movement is specified between the rigid M code (M29) block and G84 (G88) block. Modify the program.
205	RIGID MODE DI SIGNAL OFF	Rigid tapping signal (DGNG061 #1) is not 1 when G84 (G88) is executed though the rigid M code (M29) is specified. Consult the PMC ladder diagram to find the reason the signal is not turned on.
210	CAN NOT COMAND M198/M099	1 M198 and M199 are executed in the schedule operation. Or M198 is executed in the DNC operation. Modify the program. 2 In a multiple repetitive pocketing canned cycle, an interrupt macro was specified, and M99 was executed.
211	G31 (HIGH) NOT ALLOWED IN G99	G31 is commanded in the per revolution command when the high–speed skip option is provided. Modify the program.
212	ILLEGAL PLANE SELECT	The direct drawing dimensions programming is commanded for the plane other than the Z–X plane. Correct the program.
213	ILLEGAL COMMAND IN SYNCHRO–MODE	Movement is commanded for the axis to be synchronously controlled.
214	ILLEGAL COMMAND IN SYNCHRO–MODE	Coordinate system is set or tool compensation of the shift type is executed in the synchronous control. Correct the program.
217	DUPLICATE G251 (COMMANDS)	G51.2 or G251 is further commanded in the polygon machining mode. Modify the program.
218	NOT FOUND P/Q COMMAND IN G251	P or Q is not commanded in the G251 block, or the command value is out of the range. Modify the program.
219	COMMAND G250/G251 INDEPENDENTLY	G251 and G250 are not independent blocks.
220	ILLEGAL COMMAND IN SYNCHR–MODE	In the synchronous operation, movement is commanded by the NC program or PMC axis control interface for the synchronous axis.
221	ILLEGAL COMMAND IN SYNCHR–MODE	Polygon machining synchronous operation and axis control or balance cutting are executed at a time. Modify the program.
224	RETURN TO REFERENCE POINT	Not returned to reference point before cycle start.

Number	Message	Contents
225	SYNCHRONOUS/MIXED CONTROL ERROR (only with two path control only)	This alarm is generated in the following circumstances. (Searched for during synchronous and mixed control command. 1 When there is a mistake in axis number parameter setting. 2 When there is a mistake in control commanded. Modify the program or the parameter.
226	ILLEGAL COMMAND IN SYNCHRO–MODE (only with two path control only)	A travel command has been sent to the axis being synchronized in synchronous mode. Modify the program or the parameter.
229	CAN NOT KEEP SYNCHRO–STATE (only with two path control only)	This alarm is generated in the following circumstances. 1 When the synchro/mixed state could not be kept due to system overload. 2 The above condition occurred in CNC devices (hardware) and synchro–state could not be kept. (This alarm is not generated in normal use conditions.)
231	FORMAT ERROR IN G10 OR L50	Any of the following errors occurred in the specified format at the programmable–parameter input. 1 Address N or R was not entered. 2 A number not specified for a parameter was entered. 3 The axis number was too large. 4 An axis number was not specified in the axis–type parameter. 5 An axis number was specified in the parameter which is not an axis type. 6 An attempt was made to reset bit 4 of parameter 3202 (NE9) or change parameter 3210 (PSSWD) when they are protected by a password. Correct the program.
232	ILLEGAL AXIS COMMAND IN HELICAL	Three or more axes were specified as helical axes in the helical interpolation mode.
233	DEVICE BUSY	When an attempt was made to use a unit such as that connected via the RS–232–C interface, other users were using it.
239	BP/S ALARM	While punching was being performed with the function for controlling external I/O units ,background editing was performed.
240	BP/S ALARM	Background editing was performed during MDI operation.
244	P/S ALARM	In the skip function activated by the torque limit signal, the number of accumulated erroneous pulses exceed 32767 before the signal was input. Therefore, the pulses cannot be corrected with one distribution. Change the conditions, such as federates along axes and torque limit, and try again.
245	T–CODE NOT ALOWED IN THIS BLOCK	One of the G codes, G50, G10, and G04, which cannot be specified in the same block as a T code, was specified with a T code.
5010	END OF RECORD	The end of record (%) was specified.
5016	ILLEGAL COMBINATION OF M CODE	M codes which belonged to the same group were specified in a block. Alternatively,an M code which must be specified without other M codes in the block was specified in a block with other M codes.
5018	POLYGON AXIS SPPED ERROR	The rotating speed ratio of the command value cannot be maintained in the G51.2 mode , because the spped of the spindle or the polygon turning synchronous axis exceeds the clamp value or itis too slow.
5020	PARAMETER OF RESTART ERROR	An erroneous parameter was specified for restarting a program.
5030	ILLEGAL COMMAND (G100)	The end command (G110) was specified before the registration start command (G101, G102, or G103) was specified for the B–axis.

Number	Message	Contents
5031	ILLEGAL COMMAND (G100, G102, G103)	While a registration start command (G101, G102, or G103) was being executed, another registration start command was specified for the B–axis.
5032	NEW PRG REGISTERED IN B–AXS MOVE	While the machine was moving about the B–axis, at attempt was made to register another move command.
5033	NO PROG SPACE IN MEMORY B–AXIS	Commands for movement about the B–axis were not registered because of insufficient program memory.
5034	PLURAL COMMAND IN G110	Multiple movements were specified with the G110 code for the B–axis.
5035	NO FEEDRATE COMMANDED B–AXIS	A feedrate was not specified for cutting feed about the B–axis.
5036	ADDRESS R NOT DEFINED IN G81–G86	Point R was not specified for the canned cycle for the B–axis.
5037	ADDRESS Q NOT DEFINED IN G83	Depth of cut Q was not specified for the G83 code (peck drilling cycle). Alternatively, 0 was specified in Q for teh B–axis.
5038	TOO MANY START M–CODE COMMAND	More than six M codes for starting movement about the B–axis were specified.
5039	START UNREGISTERED B–AXIS PROG	An attempt was made to execute a program for the B–axis which had not been registered.
5040	CAN NOT COMMANDED B–AXIS MOVE	The machine could not move about the B–axis because parameter No.8250 was incorrectly specified, or because the PMC axis system could not be used.
5041	CAN NOT COMMANDED G110 BLOCK	Blocks containing the G110 codes were successively specified in tool–tip radius compensation for the B–axis.
5046	ILLEGAL PARAMETER (ST.COMP)	Parameters related to straightness compensation have been erroneously specified. Possible causes are as follows : 1 Invalid axis numbers have been assigned to move or compensation axes. 2 The number of pitcherror compensation points between the maximum positive and maximum negative points exceeds 128. 3 Straightness compensation point numbers have been assigned in other than ascending order. 4 Straightness compensation points could not belocated between the maximum positive and maximum negative pitch error compensation points. 5 The amount of compensation per compensation point is too large or too small.
5051	M–NET CODE ERROR	Abnormal character reception (Characters except code used to transmit)
5052	M–NET ETX ERROR	”ETX” code is abnormal.
5053	M–NET CONNECT ERROR	Connection time supervision error (parameter No.175)
5054	M–NET RECEIVE ERROR	Boring time supervision error (parameter No.176)
5055	M–NET PRT/FRM ERROR	Vertical parity or framing error detection
5056	M–NET BOARD SYSTEM DOWN	Transmit time–out error (parameter No. 177) ROM parity error CPU interruption detection of not listed above
5058	G35/G36 FORMAT ERROR	A command for changing the major axis was specified during circular threading. Alternatively, the leangth of the major axis was specified to be 0.
5059	RADIUS IS OUT OF RANGE	During circular interpolation, the center of the arc specified with I, J, and K caused the radius to exceed nine digits.

Number	Message	Contents
5073	NO DECIMAL POINT	A decimal point is not specified for a command for which a decimal point must be specified.
5074	ADDRESS DUPLICATION ERROR	The same address appears more than once in a block. Alternatively, a block contains two or more G codes belonging to the same group.
5082	DATA SERVER ERROR	Details are displayed on the data server message screen.

2) Background edit alarm

Number	Message	Contents
070 to 074 085 to 087	BP/S alarm	BP/S alarm occurs in the same number as the P/S alarm that occurs in ordinary program edit.
140	BP/S alarm	It was attempted to select or delete in the background a program being selected in the foreground. **(Note)** Use background editing correctly.

Note

Alarm in background edit is displayed in the key input line of the background edit screen instead of the ordinary alarm screen and is resettable by any of the MDI key operation.

3) Absolute pulse coder (APC) alarm

Number	Message	Contents
300	n AXIS NEED ZRN	Manual reference position return is required for the nth–axis (n=1 – 8).
301	APC ALARM:n AXIS COMMUNICATION	nth–axis (n=1 – 8) APC communication error. Failure in data transmission Possible causes include a faulty APC, cable, or servo interface module.
302	APC ALARM:n AXIS OVER TIME	nth–axis (n=1 – 8) APC overtime error. Failure in data transmission. Possible causes include a faulty APC, cable, or servo interface module.
303	APC ALARM:n AXIS FRAMING	nth–axis (n=1 – 8) APC framing error. Failure in data transmission. Possible causes include a faulty APC, cable, or servo interface module.
304	APC ALARM:n AXIS PARITY	nth–axis (n=1 – 8) APC parity error. Failure in data transmission. Possible causes include a faulty APC, cable, or servo interface module.
305	APC ALARM:n AXIS PULSE MISS	nth–axis (n=1 – 8) APC pulse error alarm. APC alarm. APC or cable may be faulty.
306	APC ALARM:n AXIS BATTERY ZERO	nth–axis (n=1 – 8) APC battery voltage has decreased to a low level so that the data cannot be held. APC alarm. Battery or cable may be faulty.
307	APC ALARM:n AXIS BATTERY DOWN 1	nth–axis (n=1 – 8) axis APC battery voltage reaches a level where the battery must be renewed. APC alarm. Replace the battery.
308	APC ALARM:n AXIS BATTERY DOWN 2	nth–axis (n=1 – 8) APC battery voltage has reached a level where the battery must be renewed (including when power is OFF). APC alarm .Replace battery.
309	APC ALARM:n AXIS ZRN IMPOSSIBLE	An attempt was made to perform reference position return without rotating the motor through one or more turns. Rotate the motor through one or more turns, turn off the power then on again, then perform reference position return.

4) Serial pulse coder (SPC) alarms

When either of the following alarms is issued, a possible cause is a faulty serial pulse coder or cable.

Number	Message	Contents
350	SPC ALARM: n AXIS PULSE CODER	The n axis (axis 1–8) pulse coder has a fault. Refer to diagnosis display No. 202 for details.
351	SPC ALARM: n AXIS COMMUNICATION	n axis (axis 1–8) serial pulse coder communication error (data transmission fault) Refer to diagnosis display No. 203 for details.

- **The details of serial pulse coder alarm No.350**

The details of serial pulse coder alarm No. 350 (pulse coder alarm) are displayed in the diagnosis display (No. 202) as shown below.

	#7	#6	#5	#4	#3	#2	#1	#0
202		CSA	BLA	PHA	RCA	BZA	CKA	SPH

CSA : The serial pulse coder is defective. Replace it.
BLA : The battery voltage is low. Replace the batteries. This alarm has nothing to do with alarm 350 (serial pulse coder alarm).
PHA : The serial pulse coder or feedback cable is defective. Replace the serial pulse coder or cable.
RCA : The serial pulse coder is defective. Replace it.
BZA : The pulse coder was supplied with power for the first time. Make sure that the batteries are connected.
Turn the power off, then turn it on again and perform a reference position return. This alarm has nothing to do with alarm 350 (serial pulse coder alarm).
CKA: The serial pulse coder is defective. Replace it.
SPH : The serial pulse coder or feedback cable is defective. Replace the serial pulse coder or cable.

- **The details of serial pulse coder alarm No.351**

The details of serial pulse coder alarm No. 351 (communication alarm) are displayed in the diagnosis display (No. 203) as shown below.

	#7	#6	#5	#4	#3	#2	#1	#0
203	DTE	CRC	STB	PRM				

DTE : The serial pulse coder encountered a communication error. The pulse coder, feedback cable, or feedback receiver circuit is defective. Replace the pulse coder, feedback cable, or NC–axis board
CRC : The serial pulse coder encountered a communication error. The pulse coder, feedback cable, or feedback receiver circuit is defective. Replace the pulse coder, feedback cable, or NC–axis board.
STB : the serial pulse coder encountered a communication error. The pulse coder, feedback cable, or feedback receiver circuit is defective.
Replace the pulse coder, feedback cable, or NC–axis board.
PRM: An invalid parameter was found. Alarm 417 (invalid servo parameter) is also issued.

5) Servo alarms

Number	Message	Contents
400	SERVO ALARM: n AXIS OVERLOAD	The n–th axis (axis 1–8) overload signal is on. Refer to diagnosis display No. 201 for details.
401	SERVO ALARM: n AXIS VRDY OFF	The n–th axis (axis 1–8) servo amplifier READY signal (DRDY) went off.
404	SERVO ALARM: n AXIS VRDY ON	Even though the n–th axis (axis 1–8) READY signal (MCON) went off, the servo amplifier READY signal (DRDY) is still on. Or, when the power was turned on, DRDY went on even though MCON was off. Check that the servo interface module and servo amp are connected.
405	SERVO ALARM: (WRONG ZRN)	Position control system fault. Due to an NC or servo system fault in the reference position return, there is the possibility that reference position return could not be executed correctly. Try again from the manual reference position return.
407	SERVO ALARM: EXCESS ERR	The difference in synchronous axis position deviation exceeded the set value.
409	SERVO ALARM:EXCESS ERROR	An abnormal load on the servo motor was detected. Alternatively, an abnormal load on the spindle motor was detected in Cs mode.
410	SERVO ALARM: n AXIS EXCESS ERR	The position deviation value when the n–th axis (axis 1–8) stops is larger than the set value. Note) Limit value must be set to parameter No.1829 for each axis.
411	SERVO ALARM: n AXIS EXCESS ERR	The position deviation value when the n–th axis (axis 1–8) moves is larger than the set value. Note) Limit value must be set to parameter No.1828 for each axis.
413	SERVO ALARM: n AXIS LSI OVER	The contents of the error register for the n–th axis (axis 1–8) are beyond of the range of -2^{31} to 2^{31}. This error usually occurs as the result of an improperly set parameters.
414	SERVO ALARM: n AXIS DETECT ERR	N–th axis (axis 1–8) digital servo system fault. Refer to diagnosis display No. 200 and No.204 for details.
415	SERVO ALARM: n AXIS MOTION OVER	A speed higher than 511875 units/s was attempted to be set in the n–th axis (axis 1–8). This error occurs as the result of improperly set CMR.
416	SERVO ALARM: n AXIS DISCONNECT	Position detection system fault in the n–th axis (axis 1–8) pulse coder (disconnection alarm). Refer to diagnosis display No. 201 for details.
417	SERVO ALARM: n AXIS DGTL PARAM	This alarm occurs when the n–th axis (axis 1–8) is in one of the conditions listed below. (Digital servo system alarm) 1) The value set in Parameter No. 2020 (motor form) is out of the specified limit. 2) A proper value (111 or –111) is not set in parameter No.2022 (motor revolution direction). 3) Illegal data (a value below 0, etc.) was set in parameter No. 2023 (number of speed feedback pulses per motor revolution). 4) Illegal data (a value below 0, etc.) was set in parameter No. 2024 (number of position feedback pulses per motor revolution). 5) Parameters No. 2084 and No. 2085 (flexible field gear rate) have not been set. 6) A value outside the limit of {1 to the number of control axes} or a non–continuous value (Parameter 1023 (servo axis number) contains a value out of the range from 1 to the number of axes,or an isolated value (for example, 4 not prceded by 3).was set in parameter No. 1023 (servo axisnumber).
421	SERVO ALARN : n AXIS EXCESS ER (D)	While the dual position feedback function is being applied, an excessive difference was detected between a semi–closed loop error and closed loop error. Check the dual position conversion factor set in parameter Nos. 2078 and 2079.

- **Details of servo alarm No.414**

The details of servo alarm No. 414 are displayed in the diagnosis display (No. 200 and No.204) as shown below.

	#7	#6	#5	#4	#3	#2	#1	#0
200	OVL	LV	OVC	HCA	HVA	DCA	FBA	OFA

OVL : An overload alarm is being generated.
(This bit causes servo alarm No. 400. The details are indicated in diagnostic data No.201).

LV : A low voltage alarm is being generated in servo amp.
Check LED.

OVC : A overcurrent alarm is being generated inside of digital servo.

HCA : An abnormal current alarm is being generated in servo amp.
Check LED.

HVA : An overvoltage alarm is being generated in servo amp.
Check LED.

DCA : A regenerative discharge circuit alarm is being generated in servo amp.
Check LED.

FBA : A disconnection alarm is being generated.
(This bit causes servo alarm No.416.The details are indicated in diagnostic data No. 201)

OFA : An overflow alarm is being generated inside of digital servo.

	#7	#6	#5	#4	#3	#2	#1	#0
204		OFS	MCC	LDA	PMS			

OFS : A current conversion error has occurred in the digital servo.

MCC : A magnetic contactor contact in the servo amplifier has welded.

LDA : The LED indicates that serial pulse coder C is defective

PMS : A feedback pulse error has occurred because the feedback cable is defective.

- **Details of servo alarms No. 400 and No.416**

The details of servo alarms No. 400 and No. 416 are displayed in the diagnosis display (No. 201) as shown below.

	#7	#6	#5	#4	#3	#2	#1	#0
201	ALD			EXP				

When OVL equal 1 in diagnostic data No.200 (servo alarm No. 400 is being generated):

ALD 0 : Motor overheating
1 : Amplifier overheating

When FBAL equal 1 in diagnostic data No.200 (servo alarm No. 416 is being generated):

ALD	EXP	Alarm details
1	0	Built–in pulse coder disconnection (hardware)
1	1	Separately installed pulse coder disconnection (hardware)
0	0	Pulse coder is not connected due to software.

6) Over travel alarms

Number	Message	Contents
500	OVER TRAVEL : +n	Exceeded the n–th axis (axis 1–8) + side stored stroke limit I. (Parameter No.1320 or 1326 **Notes**)
501	OVER TRAVEL : –n	Exceeded the n–th axis (axis 1–8) – side stored stroke limit I. (Parameter No.1321 or 1327 **Notes**)
502	OVER TRAVEL : +n	Exceeded the n–th axis (axis 1–8) + side stored stroke limit II. (Parameter No.1322)
503	OVER TRAVEL : –n	Exceeded the n–th axis (axis 1–8) – side stored stroke limit II. (Parameter No.1323)
504	OVER TRAVEL : +n	Exceeded the n–th axis (axis 1–8) + side stored stroke limit III. (Parameter No.1324)
505	OVER TRAVEL : –n	Exceeded the n–th axis (axis 1–8) – side stored stroke limit III. (Parameter No.1325)
506	OVER TRAVEL : +n	Exceeded the n–th axis (axis 1–8) + side hardware OT.
507	OVER TRAVEL : –n	Exceeded the n–th axis (axis 1–8) – side hardware OT.
508	INTERFERENCE : +n	When n–axis is moving in the positive direction, it interferes with the order tool post (only with two–path control)
509	INTERFERENCE : –n	When n–axis is moving in the negative direction, it interferes with the order tool post (only with two–path control)
510	OVER TRAVEL : +n	A stroke limit check, made before starting movement, found that the end point of a block falls within the plus (+) side inhibited area along the n–axis defined by a stroke limit. Correct the program.
511	OVER TRAVEL : –n	A stroke limit check, made before starting movement, found that the end point of a block falls within the minus (–) side inhibited area along the N–axis defined by a stroke limit. Correct the program.

Notes

Over travel alarms No. 504 and No. 505 are provided only with the T series.
Parameters 1326 and 1327 are effective when EXLM(stroke limit switch signal) is on.

7) Overheat alarms

Number	Message	Contents
700	OVERHEAT: CONTROL UNIT	Control unit overheat Check that the fan motor operates normally, and clean the air filter.
701	OVERHEAT: FAN MOTOR	The fan motor on the top of the cabinet for the control unit is overheated. Check the operation of the fan motor and replace the motor if necessary.
704	OVERHEAT: SPINDLE	Spindle overheat in the spindle fluctuation detection (1)If the cutting load is heavy, relieve the cutting condition. (2)Check whether the cutting tool is share. (3)Another possible cause is a faulty spindle amp.

8) Rigid tapping alarm

Number	Message	Contents
740	RIGID TAP ALARM : EXCESS ERROR	During rigid tapping, the position deviation of the spindle in the stop state exceeded the setting.
741	RIGID TAP ALARM : EXCESS ERROR	During rigid tapping, the position deviation of the spindle in the stop state exceeded the setting.
742	RIGID TAP ALARM : LSI OVER FLOW	During rigid tapping, an LSI overflow occurred on the spindle side.

9) Serial spindle alarms

Number	Message	Contents
749	S–SPINDLE LSI ERROR	A communication error occurred for the serial spindle. The cause may be the disconnection of an optical cable or the interruption of the power to the spindle amplifier. (Note) Unlike alarm No. 750, this alarm occurs when a serial communication alarm is detected after the spindle amplifier is normally activated.
750	SPINDLE SERIAL LINK ERROR	This alarm is generated when the spindle control unit is not ready for starting correctly when the power is turned on in the system with the serial spindle. The four reasons can be considered as follows: 1) An improperly connected optic cable, or the spindle control unit's power is OFF. 2) When the NC power was turned on under alarm conditions other than SU–01 or AL–24 which are shown on the LED display of the spindle control unit. In this case, turn the spindle amplifier power off once and perform startup again. 3) Other reasons (improper combination of hardware) This alarm does not occur after the system including the spindle control unit is activated. 4) The second spindle (when SP2, bit 4 of parameter No. 3701, is 1) is in one of the above conditions 1) to 3). See diagnostic display No. 409 for details.
751	SPINDLE–1 ALARM DETECT (AL–XX)	This alarm indicates in the NC that an alarm is generated in the spindle unit of the system with the serial spindle. The alarm is displayed in form AL–XX (XX is a number). Refer to **(11) Alarms displayed on spindle servo unit** .The alarm number XX is the number indicated on the spindle amplifier. The CNC holds this number and displays on the screen.
752	SPINDLE–1 MODE CHANGE ERROR	This alarm is generated if the system does not properly terminate a mode change. The modes include the Cs contouring, spindle positioning, rigid tapping, and spindle control modes. The alarm is activated if the spindle control unit does not respond correctly to the mode change command issued by the NC.
754	SPINDLE–1 ABNORMAL TORQUE ALM	An abnormal load on the first spindle motor was detected.
761	SPINDLE–2 ALARM DETECT (AL–XX)	Refer to spindle alarm No. 751. (For 2nd axis)
762	SPINDLE–2 MODE CHANGE ERROR	Refer to spindle alarm No. 752.(For 2nd axis)
764	SPINDLE–2 ABNORMAL TORQUE ALM	Same as for alarm No. 754 (for the second spindle)
771	SPINDLE–3 ALARM DETECT (AL–XX)	Same as for alarm No. 751 (for the third spindle)

Number	Message	Contents
772	SPINDLE–3 MODE CHANGE ERROR	Same as for alarm No. 752 (for the third spindle)
774	SPINDLE–3 ABNORMAL TORQUE ALM	Same as for alarm No. 754 (for the third spindle)

- **The details of spindle alarm No.750**

The details of spindle alarm No. 750 are displayed in the diagnosis display (No. 409) as shown below.

	#7	#6	#5	#4	#3	#2	#1	#0
409					SPE	S2E	S1E	SHE

SPE **0 :** In the spindle serial control, the serial spindle parameters fulfill the spindle unit startup conditions.
1 : In the spindle serial control, the serial spindle parameters do not fulfill the spindle unit startup conditions.

S2E **0 :** The second spindle is normal during the spindle serial control startup.
1 : The second spindle was detected to have a fault during the spindle serial control startup.

S1E **0 :** The first spindle is normal during the spindle serial control startup.
1 : The first spindle was detected to have a fault during the spindle axis serial control startup.

SHE **0 :** The serial communications module in the CNC is normal.
1 : The serial communications module in the CNC was detected to have a fault.

10) System alarms

(These alarms cannot be reset with reset key.)

Number	Message	Contents
900	ROM PARITY	ROM parity error (CNC/OMM/Servo) Replace the number of ROM.
910	RAM PARITY : (4N)	RAM parity error in the tape memory RAM module. Clear the memory or replace the module. After this operation, reset all data including the parameters.
911	RAM PARITY: (4N+1)	RAM parity error in the tape memory RAM module. Clear the memory or replace the module. After this operation, reset all data including the parameters.
912	RAM PARITY: (4N+2)	RAM parity error in the tape memory RAM module. Clear the memory or replace the module. After this operation, reset all data including the parameters.
913	RAM PARITY : (4N+3)	RAM parity error in the tape memory RAM module. Clear the memory or replace the module. After this operation, reset all data including the parameters.
914	SRAM PARITY (2N)	RAM parity error for part program storage RAM or additional SRAM. Clear memory or replace the main CPU board or additional SRAM. Then, re–specify all data including parameters.
915	SRAM PARITY (2N+1)	
916	DRAM PARITY	RAM parity error in the DRAM module. Replace the DRAM module.
920	SERVO ALARM (1/2/3/4 AXIS)	Servo alarm (1st to 4th axis). A watchdog alarm or a RAM parity error in the servo module occurred. Replace the servo control module on the main CPU board.
922	SERVO ALARM (5/6/7/8 AXIS)	Servo alarm (5th to 8th axis). A watchdog alarm or a RAM parity error in the servo module occurred. Replace the servo control module on the option 2 board.

Number	Message	Contents
924	SERVO MODULE SETTING ERROR	The digital servo module is not installed. Check that the servo control module or servo interface module on the main CPU or option 2 board is mounted securely.
926	SERVO ALARM (1/2/3/4/5/6 AXIS)	Servo alarm (first to sixth axis). A RAM parity error in the servo module or a watchdog alarm has occurred. Replace the servo control module on the main CPU board.
930	CPU INTERRUPUT	CPU error (abnormal interrupt) The main CPU board is faulty.
950	PMC SYSTEM ALARM	Fault occurred in the PMC. The PMC control module on the main CPU board or option 3 board may be faulty.
951	PMC–RC WATCH DOG ALARM	Fault occurred in the PMC–RC (watchdog alarm).Option 3 board may be faulty.
970	NMI OCCURRED IN BOC	RAM parity error or NMI occurred in the PMC–RB or PMC–RA2 module.
971	NMI OCCURRED IN SLC	An alarm condition occurred in the interface with an I/O unit. For PMC–RA and PMC–RB, check that the PMC control module on the main CPU board is connected to the I/O unit securely. For PMC–RC, check that the PMC control module on the option 3 board is connected to the I/O unit is supplied with power and that the interface module is intact.
972	NMI OCCURRED IN OTHER MODULE	NMI occurred in a board other than the main CPU board.
973	NON MASK INTERRUPT	NMI occurred for an unknown reason.
974	F–BUS ERROR	BUS error of FANUC BUS. The main CPU board or option 1 to 3 boards may be faulty.
975	BUS ERROR (MAIN)	Main CPU board bus error. The main CPU board may be faulty.

(11) Alarms Displayed on spindle Servo Unit

Number	Meaning	Description	Remedy
”A” display	Program ROM abnormality (not installed)	Detects that control program is not started (due to program ROM not installed, etc.)	Install normal program ROM
AL01	Motor overheat	Detects motor speed exceeding specified speed excessively.	Check load status. Cool motor then reset alarm.
AL02	Excessive speed deviation	Detects motor speed exceeding specified speed excessively.	Check load status. Reset alarm.
AL03	DC link section fuse blown	Detects that fuse F4 in DC link section is blown (models 30S and 40S).	Check power transistors, and so forth. Replace fuse.
AL04	Input fuse blown. Input power open phase.	Detects blown fuse (F1 to F3), open phase or momentary failure of power (models 30S and 40S).	Replace fuse. Check open phase and power supply refenerative circuit operation.
AL05	Control power supply fuse blown	Detects that control power supply fuse AF2 or AF3 is blown (models 30S and 40S).	Check for control power supply short circuit . Replace fuse.
AL–07	Excessive speed	Detects that motor rotation has exceeded 115% of its rated speed.	Reset alarm.
AL–08	High input voltage	Detects that switch is flipped to 200 VAC when input voltage is 230 VAC or higher (models 30S and 40S).	Flip switch to 230 VAC.

Number	Meaning	Description	Remedy
AL–09	Excessive load on main circuit section	Detects abnormal temperature rise of power transistor radiator.	Cool radiator then reset alarm.
AL–10	Low input voltage	Detects drop in input power supply voltage.	Remove cause, then reset alarm.
AL–11	Overvoltage in DC link section	Detects abnormally high direct current power supply voltage in power circuit section.	Remove cause, then reset alarm.
AL–12	Overcurrent in DC link section	Detects flow of abnormally large current in direct current section of power circuit	Remove cause, then reset alarm.
AL–13	CPU internal data memory abnormality	Detects abnormality in CPU internal data memory. This check is made only when power is turned on.	Remove cause, then reset alarm.
AL–15	Spindle switch/output switch alarm	Detects incorrect switch sequence in spindle switch/output switch operation.	Check sequence.
AL–16	RAM abnormality	Detects abnormality in RAM for external data. This check is made only when power is turned on.	Remove cause, then reset alarm.
AL–18	Program ROM sum check error	Detects program ROM data error. This check is made only when power is turned on.	Remove cause, then reset alarm.
AL–19	Excessive U phase current detection circuit offset	Detects excessive U phase current detection circuit offset. This check is made only when power is turned on.	Remove cause, then reset alarm.
AL–20	Excessive V phase current detection circuit offset	Detects excessive V phase current detection circuit offset. This check is made only when power is turned on.	Remove cause, then reset alarm.
AL–24	Serial transfer data error	Detects serial transfer data error (such as NC power supply turned off, etc.)	Remove cause, then reset alarm.
AL–25	Serial data transfer stopped	Detects that serial data transfer has stopped.	Remove cause, then reset alarm.
AL–26	Disconnection of speed detection signal for Cs contouring control	Detects abnormality in position coder signal(such as unconnected cable and parameter setting error).	Remove cause, then reset alarm.
AL–27	Position coder signal disconnection	Detects abnormality in position coder signal (such as unconnected cable and adjustment error).	Remove cause, then reset alarm.
AL–28	Disconnection of position detection signal for Cs contouring control	Detects abnormality in position detection signal for Cs contouring control (such as unconnected cable and adjustment error).	Remove cause, then reset alarm.
AL–29	Short–time overload	Detects that overload has been continuously applied for some period of time (such as restraining motor shaft in positioning).	Remove cause, then reset alarm.
AL–30	Input circuit overcurrent	Detects overcurrent flowing in input circuit.	Remove cause, then reset alarm.
AL–31	Speed detection signal disconnection motor restraint alarm or motor is clamped.	Detects that motor cannot rotate at specified speed or it is detected that the motor is clamped. (but rotates at very slow speed or has stopped). (This includes checking of speed detection signal cable.)	Remove cause, then reset alarm.
AL–32	Abnormality in RAM internal to LSI for serial data transfer. This check is made only when power is turned on.	Detects abnormality in RAM interior LSI for serial data transfer. This check is made only when power is turned on.	Remove cause, then reset alarm.

Num- ber	Meaning	Description	Remedy
AL–33	Insufficient DC link section charging	Detects insufficient charging of direct current power supply voltage in power circuit section when magnetic contactor in amplifier is turned on (such as open phase and defective charging resistor).	Remove cause, then reset alarm.
AL–34	Parameter data setting beyond allowable range of values	Detects parameter data set beyond allowable range of values.	Set correct data.
AL–35	Excesive gear ratio data setting	Detects gear ratio data set beyond allowable range of values.	Set correct data.
AL–36	Error counter over flow	Detects error counter overflow.	Correct cause, then reset alarm.
AL–37	Speed detector parameter setting error	Detects incorrect setting of parameter for number of speed detection pulses.	Set correct data.
AL–39	Alarm for indicating failure in detecting 1–rotation signal for Cs contouring control	Detects 1–rotation signal detection failure in Cs contouring contorl.	Make 1–rotation signal adjustment. Check cable shield status.
AL–40	Alarm for indicating 1–rotation signal for Cs contouring control not detected	Detects that 1–rotation signal has not occurred in Cs contouring control.	Make 1–rotation signal adjustment.
AL–41	Alarm for indicating failure in detecting position coder 1–rotation signal.	Detects failure in detecting position coder 1–rotation signal.	Make signal adjustment for signal conversion circuit. Check cable shield status.
AL–42	Alarm for indicating position coder 1–rotation signal not detected	Detects that position coder 1–rotation signal has not issued.	Make 1–rotation signal adjustment for signal conversion circuit.
AL–43	Alarm for indicating disconnection of position coder signal for differential speed mode	Detects that main spindle position coder signal used for differential speed mode is not connected yet (or is disconnected).	Check that main spindle position coder signal is connected to connector CN12.
AL–46	Alarm for indicating failure in detecting position coder 1–rotation signal in thread cutting operation.	Detects failure in detecting position coder 1–rotation signals in thread cutting operation.	Make 1–rotation signal adjustment for signal conversion circuit Check cable shield status.
AL–47	Position coder signal abnormality	Detects incorrect position coder signal count operation.	Make signal adjustment for signal conversion circuit. Check cable shield status.
AL–48	Position coder 1–rotation signal abnormality	Detects that occurrence of position coder 1–rotation signal has stopped.	Make 1–rotation signal adjustment for signal conversion circuit.
AL–49	The converted differential speed is too high.	Detects that speed of other spindle converted to speed of local spindle has exceeded allowable limit in differential mode.	Check the position coder state of the other side.
AL–50	Excessive speed command calculation value in spindle synchronization control	Detects that speed command calculation value exceeded allowable range in spindle synchronization control.	Check parameters such as a position gain.
AL–51	Undervoltage at DC link section	Detects that DC power supply voltage of power circuit has dropped (due to momentary power failure or loose contact of magnetic contactor).	Remove cause, then reset alarm.
AL–52	ITP signal abnormality I	Detects abnormality in synchronization signal (ITP signal)with CNC (such as loss of ITP signal).	Remove cause, then reset alarm.

Number	Meaning	Description	Remedy
AL–53	ITP signal abnormality II	Detects abnormality in synchronization signal (ITP signal) with CNC (such as loss of ITP signal).	Remove cause, then reset alarm.
AL–54	Overload current alarm	Detects that excessive current flowed in motor for long time.	Check if overload operation or frequent acceleration/deceleration is performed.
AL–55	Power line abnormality in spindle switching/output switching	Detects that switch request signal does not match power line status check signal.	Check operation of magnetic contractor for power line switching. Check if power line status check signal is processed normally.

H OPERATION OF PORTABLE TAPE READER

Portable tape reader is the device which inputs the NC program and the data on the paper tape to CNC.

- **Names and descriptions of each section**

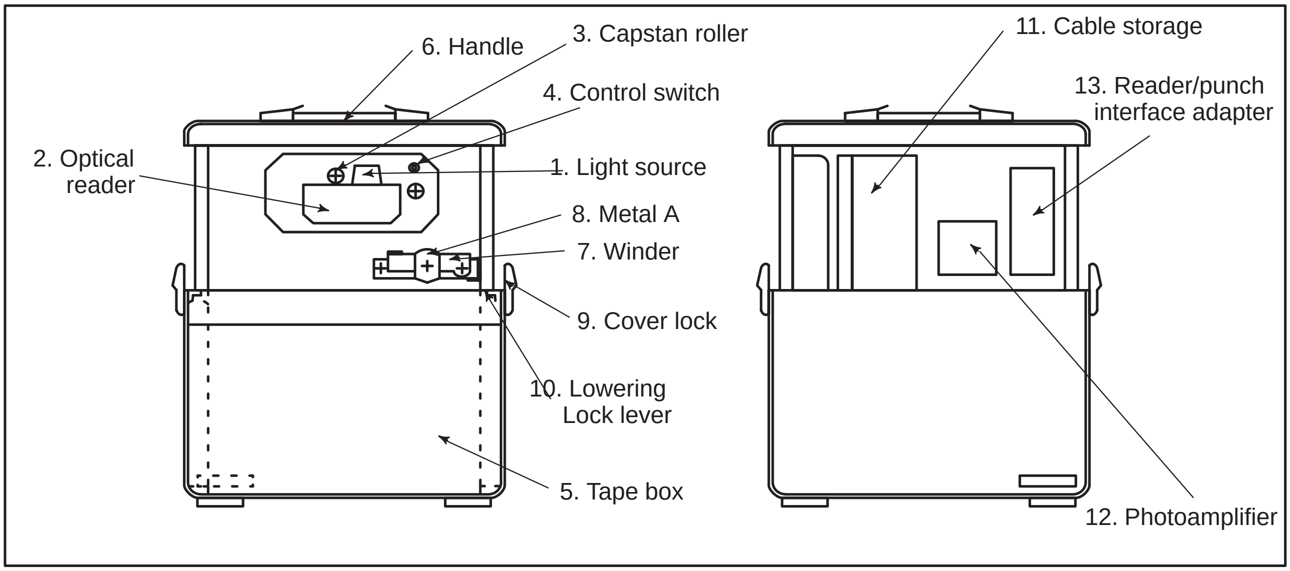
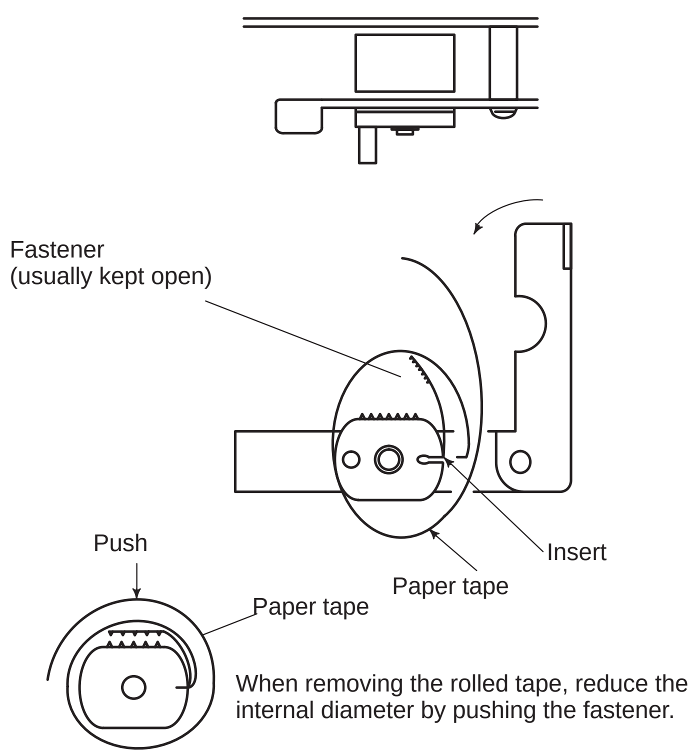


Table 1 Description of Each Section

No.	Name	Descriptions
1	Light Sources	An LED (Light emitting diode) is mounted for each channel and for the feed hole (9 diodes in total). A built–in Stop Shoe functions to decelerate the tape. The light source is attracted to the optical reader by a magnet so that the tape will be held in the correct position. This unit can be opened upward, by turning the tape reader control switch to the RELEASE position (this turns off the magnet).
2	Optical Reader	Reads data punched on the tape, through a glass window. Dust or scratches on the glass window can result in reading errors. Keep this window clean.
3	Capstan Roller	Controls the feeding of tape as specified by the control unit.
4	Tape Reader Control Switch	A 3–position switch used to control the Tape Reader. **RELEASE** ----- The tape is allowed to be free, or used to open the light-source. When loading or unloading the tape, select this position. **AUTO** -------- The tape is set to fixed position by the Stop Shoe. The feed and stop of the tape is controlled by the CNC. To input data from tape, the Light Source must be closed and this position must be selected. **MANUAL** ----- The tape can be fed in the forward reading direction. if another position is selected, the tape feed is stopped.
5	Tape Box	A Tape Box is located below the Tape Reader. A belt used to draw out a paper tape is located inside the box. The paper tape can easily be pulled out using this belt. The tape box accomodates 15 meters of tape.
6	Handle	Used to carry the tape reader.
7	Winder	Used to advance or rewind the tape.
8	Metal A	
9	Cover lock	Be sure to use the lock for fastening the cover before carrying the tape reader.

No.	Name	Descriptions
10	Lowering lock lever	When the tape reader is raised, the latch mechanism is activated to fix the tape reader. Thus, the tape reader is not lowered. The latch is locked with the lowering lock lever. The latch is therefore not unlocked even when the tape reader is raised with the handle. When the latch is locked, the lever is horizontal. To store the tape reader in the box, push the lever to release the lock, then raise the tape reader with the handle to unlock the latch. When the latch is unlocked, the tape reader can be stored in the box. When storing the tape reader, secure it with the cover lock.
11	Cable storage	Used to store rolled power and signal cables. The cable length is 1.5 m.
12	Photoamplifier	For the tape reader
13	Reader/punch interface adapter	200 VAC input and 5 VDC output power and reader/punch interface adapter PCB

Procedure for Operating the Portable Tape Reader

Preparations

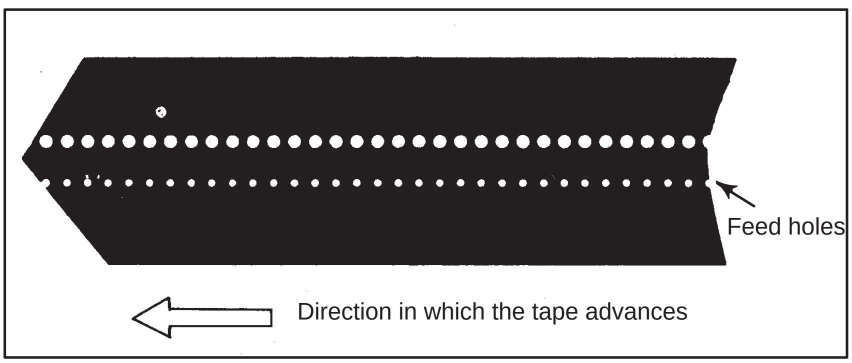
1 Unlock the cover locks **9**. Raise the tape reader with the handle **6** until it clicks, then lower the tape reader. The tape reader then appears and is secured. Check that the lowering lock levers **10** are horizontal.

2 Take out the signal and power cables from the cable storage **11** and connect the signal cable with the CNC reader/punch interface port and the power cable with the power supply.

Setting the tape

3 Turn the control switch to the RELEASE position.

4 Lift the Light Source Unit, and insert an NC tape between the gap. The tape must be positioned as shown in the figure, when viewed looking downward.



5 Pull the tape until the top of the tape goes past the Capstan roller.

6 Check that the NC tape is correctly positioned by the Tape Guide.

7 Lower the Light Source.

8 Turn the switch to the AUTO position.

9 Suspend the top and rear–end of the tape in the Tape Box.

Removing the tape

10 Turn the switch to the RELEASE position.

11 Lift the Light Source and remove the tape.

12 Lower the Light Source

Storage

13 Store the cables in the cable storage **11**.

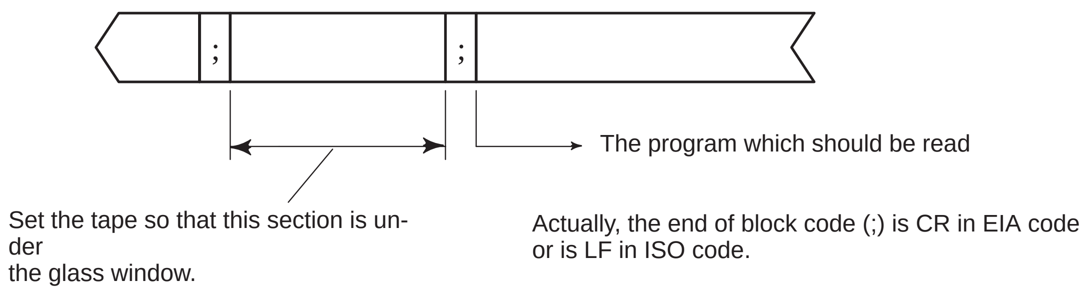
14 Push the lowering lock lever **10** at both sides down.

15 Raise the tape reader with the handle **6** to unlock the latch, then gently lower it.

16 Lock the cover lock **9** and carry the tape reader with the handle **6**.

NOTE 1 SETTING OF A TAPE

When the NC tape is loaded, the Label Skip function activates to read but skip data until first End of Block code (CR in EIA code or LF in ISO code) is read. When loading an NC tape, the location within the tape, from which data reading should be started must properly be selected and the NC tape should be set as shown in the figure below.



NOTE 2 DISCONNECTION AND CONNECTION OF A PORTABLE TAPE READER CONNECTION CABLE

Don’t disconnect or connect CNC tape reader connection cable (signal cable) without turning off the CNC power supply, otherwise the PCB of the tape reader and master PCB of CNC controller may be broken.Turn off the CNC power supply before disconnecting or connecting the connection cable, accordingly.

– Numbers –

– A –

– B –

– C –

– D –

– E –

– F –

– G –

– H –

– I –

– K –

– L –

– M –

– N –

– O –

– P –

– R –

– S –

– T –

– U –

– V –

– W –

– Y –

Revision Record

FANUC Series 16/18/160/180–TB OPERATOR'S MANUAL (B–62444E/03)

Edition	Date	Contents	Edition	Date	Contents
03	Feb., '95	● Following functions were added. ○ Rigid tapping ○ G53, G28, G30 and G30.1 commands when tool position offset is applied ○ G53, G28, G30, and G30.1 commands in tool–tip radius compensation mode ○ Distribution processing termination monitoring function for the high–speed machining command (G05) ○ Tool withdrawal and return (G10.6) ○ Pattern data display ○ DNC operation ○ Special single–block control ○ Stroke limit check prior to performing movement ○ External operator message history display ● Correction of erros			
02	Nov., '94	● Series 18/160/180–TB were added. ● "Fuse replacement" in IV MAINTENANCE part was deleted. ● "Replacement of battery for absolute pulse coder for α series Amp." wa added.			
01	Mar., '94				