FANUC

GE Fanuc Automation

Computer Numerical Control Products

Series 0-TD / 0-GCD

Operator's Manual

GFZ-62544EN/02 February 1997

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.

SAFETY PRECAUTIONS

This section describes the safety precautions related to the use of CNC units. It is essential that these precautions be observed by users to ensure the safe operation of machines equipped with a CNC unit (all descriptions in this section assume this configuration). Note that some precautions are related only to specific functions, and thus may not be applicable to certain CNC units.
Users must also observe the safety precautions related to the machine, as described in the relevant manual supplied by the machine tool builder. Before attempting to operate the machine or create a program to control the operation of the machine, the operator must become fully familiar with the contents of this manual and relevant manual supplied by the machine tool builder.

Contents

1 DEFINITION OF WARNING, CAUTION, AND NOTE

This manual includes safety precautions for protecting the user and preventing damage to the machine. Precautions are classified into Warning and Caution according to their bearing on safety. Also, supplementary information is described as a Note. Read the Warning, Caution, and Note thoroughly before attempting to use the machine.

WARNING

Applied when there is a danger of the user being injured or when there is a damage of both the user being injured and the equipment being damaged if the approved procedure is not observed.

CAUTION

Applied when there is a danger of the equipment being damaged, if the approved procedure is not observed.

NOTE

The Note is used to indicate supplementary information other than Warning and Caution.

❍ **Read this manual carefully, and store it in a safe place.**

2 GENERAL WARNINGS AND CAUTIONS

WARNING

1. Never attempt to machine a workpiece without first checking the operation of the machine. Before starting a production run, ensure that the machine is operating correctly by performing a trial run using, for example, the single block, feedrate override, or machine lock function or by operating the machine with neither a tool nor workpiece mounted. Failure to confirm the correct operation of the machine may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.

2. Before operating the machine, thoroughly check the entered data.
   Operating the machine with incorrectly specified data may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.

3. Ensure that the specified feedrate is appropriate for the intended operation. Generally, for each machine, there is a maximum allowable feedrate. The appropriate feedrate varies with the intended operation. Refer to the manual provided with the machine to determine the maximum allowable feedrate. If a machine is run at other than the correct speed, it may behave unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.

4. When using a tool compensation function, thoroughly check the direction and amount of compensation.
   Operating the machine with incorrectly specified data may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.

5. The parameters for the CNC and PMC are factory–set. Usually, there is not need to change them. When, however, there is not alternative other than to change a parameter, ensure that you fully understand the function of the parameter before making any change.
   Failure to set a parameter correctly may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.

6. Immediately after switching on the power, do not touch any of the keys on the MDI panel until the position display or alarm screen appears on the CNC unit.
   Some of the keys on the MDI panel are dedicated to maintenance or other special operations. Pressing any of these keys may place the CNC unit in other than its normal state. Starting the machine in this state may cause it to behave unexpectedly.

7. This manual supplied with a CNC unit provide an overall description of the machine's functions, including any optional functions. Note that the optional functions will vary from one machine model to another. Therefore, some functions described in the manuals may not actually be available for a particular model. Check the specification of the machine if in doubt.

WARNING

8. Some functions may have been implemented at the request of the machine–tool builder. When using such functions, refer to the manual supplied by the machine–tool builder for details of their use and any related cautions.

NOTE

Programs, parameters, and macro variables are stored in nonvolatile memory in the CNC unit. Usually, they are retained even if the power is turned off. Such data may be deleted inadvertently, however, or it may prove necessary to delete all data from nonvolatile memory as part of error recovery.
To guard against the occurrence of the above, and assure quick restoration of deleted data, backup all vital data, and keep the backup copy in a safe place.

3 WARNINGS AND CAUTIONS RELATED TO PROGRAMMING

This section covers the major safety precautions related to programming. Before attempting to perform programming, read the supplied this manual carefully such that you are fully familiar with their contents.

WARNING

1. Coordinate system setting

If a coordinate system is established incorrectly, the machine may behave unexpectedly as a result of the program issuing an otherwise valid move command.
Such an unexpected operation may damage the tool, the machine itself, the workpiece, or cause injury to the user.

2. Positioning by nonlinear interpolation

When performing positioning by nonlinear interpolation (positioning by nonlinear movement between the start and end points), the tool path must be carefully confirmed before performing programming.
Positioning involves rapid traverse. If the tool collides with the workpiece, it may damage the tool, the machine itself, the workpiece, or cause injury to the user.

3. Function involving a rotation axis

When programming polar coordinate interpolation or normal–direction (perpendicular) control, pay careful attention to the speed of the rotation axis. Incorrect programming may result in the rotation axis speed becoming excessively high, such that centrifugal force causes the chuck to lose its grip on the workpiece if the latter is not mounted securely.
Such mishap is likely to damage the tool, the machine itself, the workpiece, or cause injury to the user.

4. Inch/metric conversion

Switching between inch and metric inputs does not convert the measurement units of data such as the workpiece origin offset, parameter, and current position. Before starting the machine, therefore, determine which measurement units are being used. Attempting to perform an operation with invalid data specified may damage the tool, the machine itself, the workpiece, or cause injury to the user.

5. Constant surface speed control

When an axis subject to constant surface speed control approaches the origin of the workpiece coordinate system, the spindle speed may become excessively high. Therefore, it is necessary to specify a maximum allowable speed. Specifying the maximum allowable speed incorrectly may damage the tool, the machine itself, the workpiece, or cause injury to the user.

WARNING

6. Stroke check

After switching on the power, perform a manual reference position return as required. Stroke check is not possible before manual reference position return is performed. Note that when stroke check is disabled, an alarm is not issued even if a stroke limit is exceeded, possibly damaging the tool, the machine itself, the workpiece, or causing injury to the user.

7. Tool post interference check

A tool post interference check is performed based on the tool data specified during automatic operation. If the tool specification does not match the tool actually being used, the interference check cannot be made correctly, possibly damaging the tool or the machine itself, or causing injury to the user.
After switching on the power, or after selecting a tool post manually, always start automatic operation and specify the tool number of the tool to be used.

8. Absolute/incremental mode

If a program created with absolute values is run in incremental mode, or vice versa, the machine may behave unexpectedly.

9. Plane selection

If an incorrect plane is specified for circular interpolation, helical interpolation, or a canned cycle, the machine may behave unexpectedly. Refer to the descriptions of the respective functions for details.

10. Torque limit skip

Before attempting a torque limit skip, apply the torque limit. If a torque limit skip is specified without the torque limit actually being applied, a move command will be executed without performing a skip.

11. Programmable mirror image

Note that programmed operations vary considerably when a programmable mirror image is enabled.

12. Compensation function

If a command based on the machine coordinate system or a reference position return command is issued in compensation function mode, compensation is temporarily canceled, resulting in the unexpected behavior of the machine.
Before issuing any of the above commands, therefore, always cancel compensation function mode.

4 WARNINGS AND CAUTIONS RELATED TO HANDLING

This section presents safety precautions related to the handling of machine tools. Before attempting to operate your machine, read the supplied this manual carefully, such that you are fully familiar with their contents.

WARNING

1. Manual operation

When operating the machine manually, determine the current position of the tool and workpiece, and ensure that the movement axis, direction, and feedrate have been specified correctly. Incorrect operation of the machine may damage the tool, the machine itself, the workpiece, or cause injury to the operator.

2. Manual reference position return

After switching on the power, perform manual reference position return as required. If the machine is operated without first performing manual reference position return, it may behave unexpectedly. Stroke check is not possible before manual reference position return is performed. An unexpected operation of the machine may damage the tool, the machine itself, the workpiece, or cause injury to the user.

3. Manual numeric command

When issuing a manual numeric command, determine the current position of the tool and workpiece, and ensure that the movement axis, direction, and command have been specified correctly, and that the entered values are valid.
Attempting to operate the machine with an invalid command specified may damage the tool, the machine itself, the workpiece, or cause injury to the operator.

4. Manual handle feed

In manual handle feed, rotating the handle with a large scale factor, such as 100, applied causes the tool and table to move rapidly. Careless handling may damage the tool and/or machine, or cause injury to the user.

5. Disabled override

If override is disabled (according to the specification in a macro variable) during threading, rigid tapping, or other tapping, the speed cannot be predicted, possibly damaging the tool, the machine itself, the workpiece, or causing injury to the operator.

6. Origin/preset operation

Basically, never attempt an origin/preset operation when the machine is operating under the control of a program. Otherwise, the machine may behave unexpectedly, possibly damaging the tool, the machine itself, the tool, or causing injury to the user.

WARNING

7. Workpiece coordinate system shift

Manual intervention, machine lock, or mirror imaging may shift the workpiece coordinate system. Before attempting to operate the machine under the control of a program, confirm the coordinate system carefully.
If the machine is operated under the control of a program without making allowances for any shift in the workpiece coordinate system, the machine may behave unexpectedly, possibly damaging the tool, the machine itself, the workpiece, or causing injury to the operator.

8. Software operator's panel and menu switches

Using the software operator's panel and menu switches, in combination with the MDI panel, it is possible to specify operations not supported by the machine operator's panel, such as mode change, override value change, and jog feed commands.
Note, however, that if the MDI panel keys are operated inadvertently, the machine may behave unexpectedly, possibly damaging the tool, the machine itself, the workpiece, or causing injury to the user.

9. Manual intervention

If manual intervention is performed during programmed operation of the machine, the tool path may vary when the machine is restarted. Before restarting the machine after manual intervention, therefore, confirm the settings of the manual absolute switches, parameters, and absolute/incremental command mode.

10. Feed hold, override, and single block

The feed hold, feedrate override, and single block functions can be disabled using custom macro system variable #3004. Be careful when operating the machine in this case.

11. Dry run

Usually, a dry run is used to confirm the operation of the machine. During a dry run, the machine operates at dry run speed, which differs from the corresponding programmed feedrate. Note that the dry run speed may sometimes be higher than the programmed feed rate.

12. Cutter and tool nose radius compensation in MDI mode

Pay careful attention to a tool path specified by a command in MDI mode, because cutter or tool nose radius compensation is not applied. When a command is entered from the MDI to interrupt in automatic operation in cutter or tool nose radius compensation mode, pay particular attention to the tool path when automatic operation is subsequently resumed. Refer to the descriptions of the corresponding functions for details.

13. Program editing

If the machine is stopped, after which the machining program is edited (modification, insertion, or deletion), the machine may behave unexpectedly if machining is resumed under the control of that program. Basically, do not modify, insert, or delete commands from a machining program while it is in use.

5 WARNINGS RELATED TO DAILY MAINTENANCE

WARNING

1. Memory backup battery replacement

When replacing the memory backup batteries, keep the power to the machine (CNC) turned on, and apply an emergency stop to the machine. Because this work is performed with the power on and the cabinet open, only those personnel who have received approved safety and maintenance training may perform this work.
When replacing the batteries, be careful not to touch the high–voltage circuits (marked ⚠ and fitted with an insulating cover).
Touching the uncovered high–voltage circuits presents an extremely dangerous electric shock hazard.

NOTE

The CNC uses batteries to preserve the contents of its memory, because it must retain data such as programs, offsets, and parameters even while external power is not applied.
If the battery voltage drops, a low battery voltage alarm is displayed on the machine operator's panel or CRT screen.
When a low battery voltage alarm is displayed, replace the batteries within a week. Otherwise, the contents of the CNC's memory will be lost.
Refer to the maintenance section of this manual for details of the battery replacement procedure.

WARNING

2. Absolute pulse coder battery replacement

When replacing the memory backup batteries, keep the power to the machine (CNC) turned on, and apply an emergency stop to the machine. Because this work is performed with the power on and the cabinet open, only those personnel who have received approved safety and maintenance training may perform this work.
When replacing the batteries, be careful not to touch the high–voltage circuits (marked ⚠ and fitted with an insulating cover).
Touching the uncovered high–voltage circuits presents an extremely dangerous electric shock hazard.

NOTE

The absolute pulse coder uses batteries to preserve its absolute position.
If the battery voltage drops, a low battery voltage alarm is displayed on the machine operator's panel or CRT screen.
When a low battery voltage alarm is displayed, replace the batteries within a week. Otherwise, the absolute position data held by the pulse coder will be lost.
Refer to the maintenance section of this manual for details of the battery replacement procedure.

WARNING

3. Fuse replacement

Before replacing a blown fuse, however, it is necessary to locate and remove the cause of the blown fuse.
For this reason, only those personnel who have received approved safety and maintenance training may perform this work.
When replacing a fuse with the cabinet open, be careful not to touch the high–voltage circuits (marked ⚠ and fitted with an insulating cover).
Touching an uncovered high–voltage circuit presents an extremely dangerous electric shock hazard.

Table of contents

I. GENERAL

1 GENERAL

About this manual

This manual consists of the following parts:

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.

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 procedures for batteries.

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

This manual does not describe parameters in detail. For details on parameters mentioned in this manual, refer to the manual for parameters (B–62550EN).

This manual describes all optional functions. Look up the options incorporated into your system in the manual written by the machine tool builder.

Applicable models

The models covered by this manual, and their abbreviations are:

Product name	Abbreviations	
FANUC Series 0–TD	0–TD	Series 0–D
FANUC Series 0–GCD	0–GCD	

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 the FANUC Series 0–D. In the table, this manual is marked with an asterisk (*).

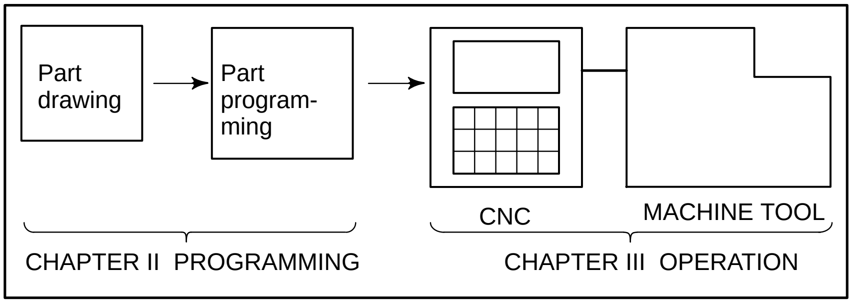
Table 1 Manuals Related to the FANUC Series 0–D

Manual name	Specification number	
FANUC Series 0–TD/MD/GCD/GSD CONNECTION MANUAL (HARDWARE)	B–62543EN	
FANUC Series 0–TD/MD/GCD/GSD CONNECTION MANUAL (FUNCTION)	B–62543EN–1	
FANUC Series 0–TD/GCD OPERATOR'S MANUAL	B–62544EN	*
FANUC Series 0–MD/GSD OPERATOR'S MANUAL	B–62574EN	
FANUC Series 0–TD/MD/GCD/GSD MAINTENANCE MANUAL	B–62545EN	
FANUC Series 0–TD/GCD PARAMETER MANUAL	B–62550EN	
FANUC Series 0–MD/GSD PARAMETER MANUAL	B–62580EN	

1.1 GENERAL FLOW OF OPERATION OF CNC MACHINE TOOL

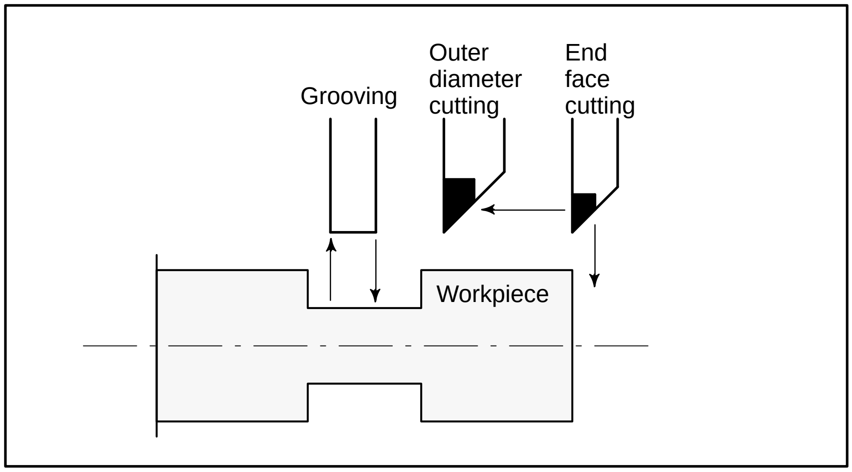
When machining the part using the CNC machine tool, first prepare the program, then operate the CNC machine by using the program.

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 End face cutting	2 Outer diameter cutting	3 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

NOTE

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.

3 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.

4 This manual describes the functions supported by the series and editions of the FANUC Series 0–TD/GCD software indicated below. Note that the functions described in this manual may not be supported by series and editions of the software other than those listed below.

Model	Series	Edition
FANUC Series 0–TD	0671 0672	Edition 04 and later Edition 01 and later
FANUC Series 0–GCD	0881	Edition 01 and later

5 This manual provides a general description of the FANUC Series 0–D. Some functions described in this manual may not, therefore, be available depending on the software series or type.

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**

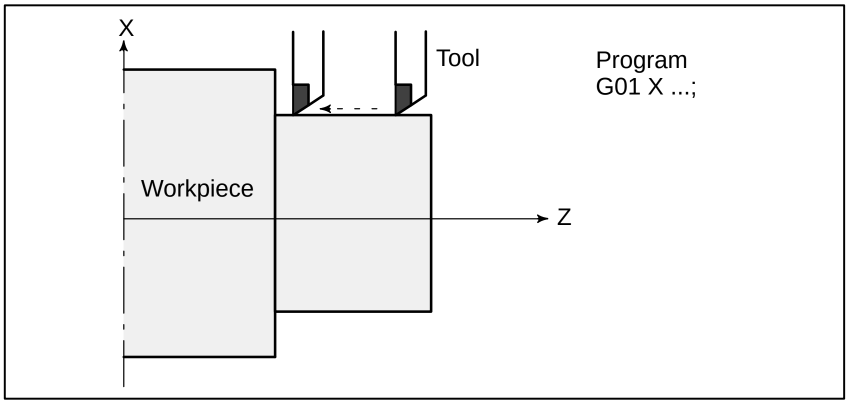


Fig. 1.1 (a) Tool movement along the straight line which is parallel to Z–axis

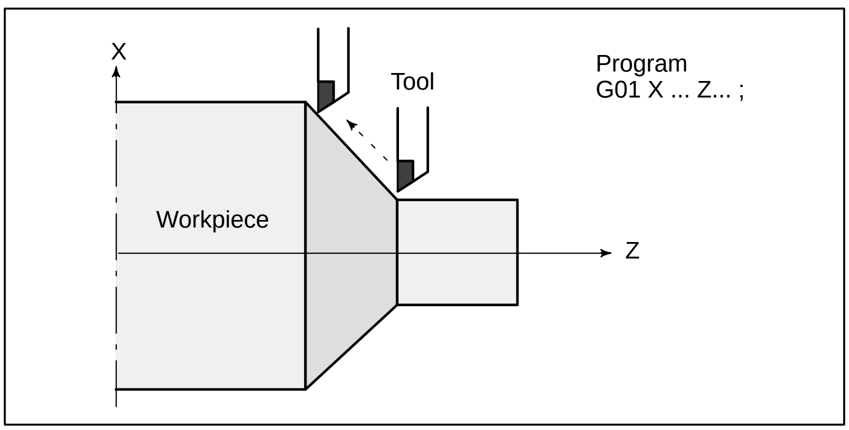


Fig. 1.1 (b) Tool movement along the taper line

- **Tool movement along an arc**

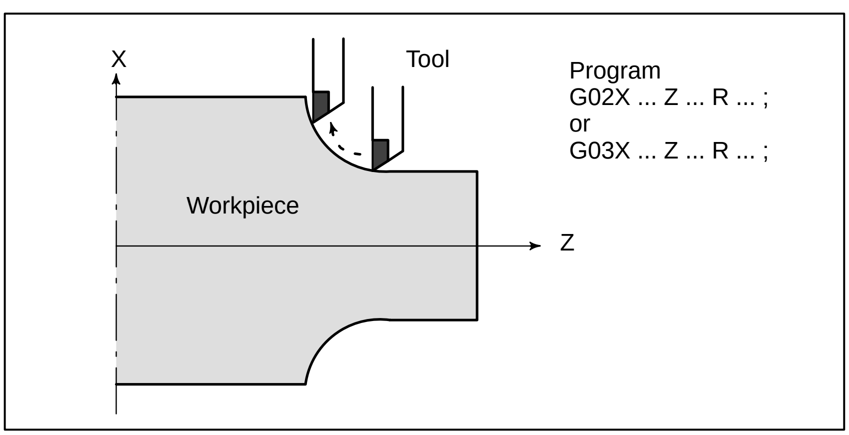


Fig. 1.1 (c) Tool movement along an arc

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.

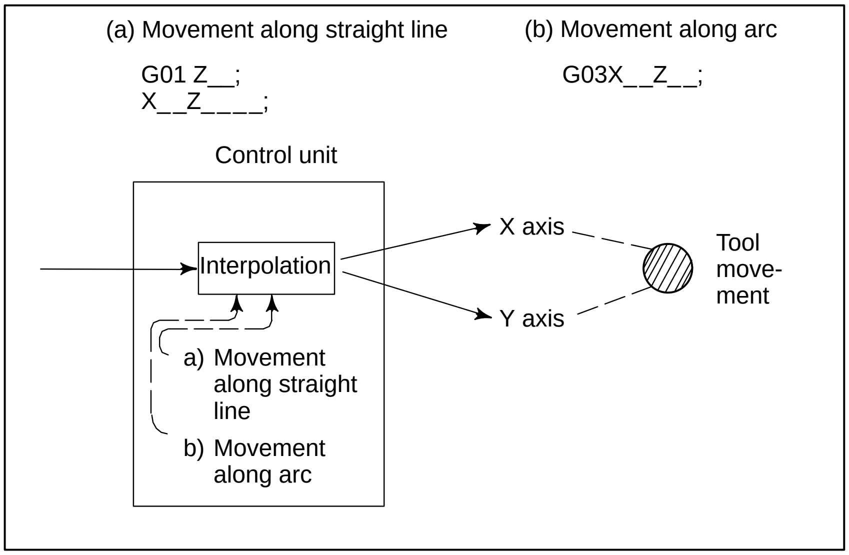


Fig. 1.1 (d) Interpolation function

> **NOTE**
> 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.

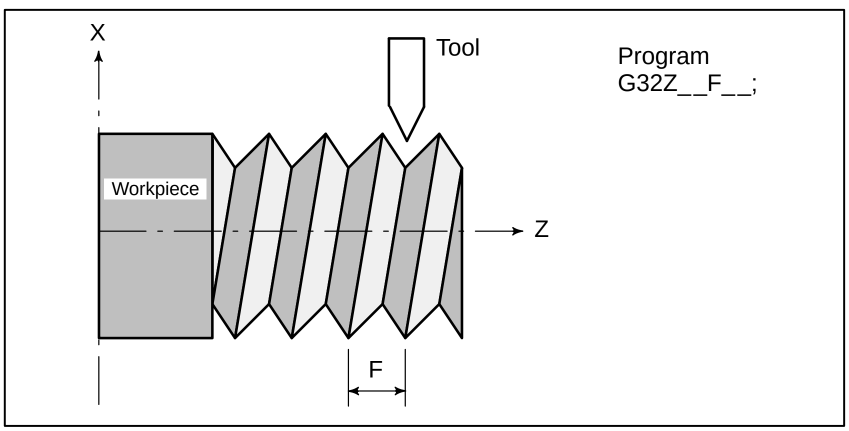


Fig. 1.1 (e) Straight thread cutting

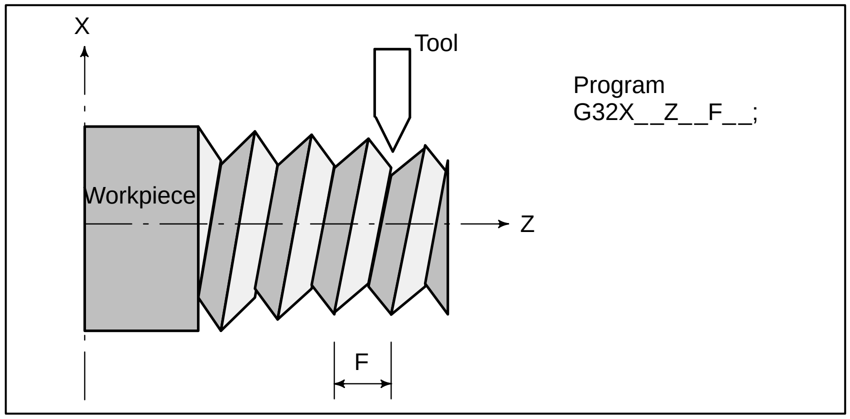


Fig. 1.1 (f) Taper thread cutting

1.2 FEED– FEED FUNCTION

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

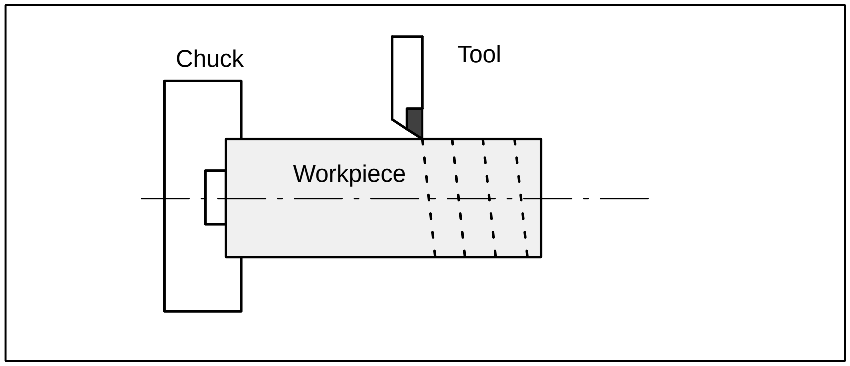


Fig. 1.2 (a) Feed function

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.

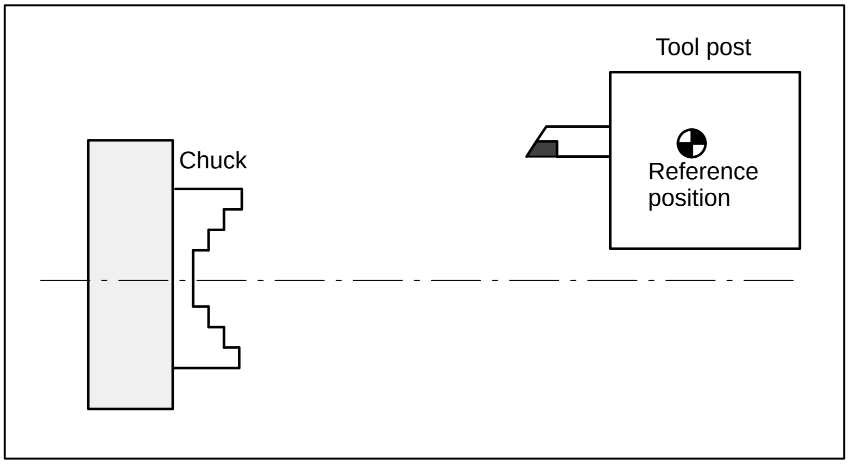


Fig. 1.3.1 (a) Reference position

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

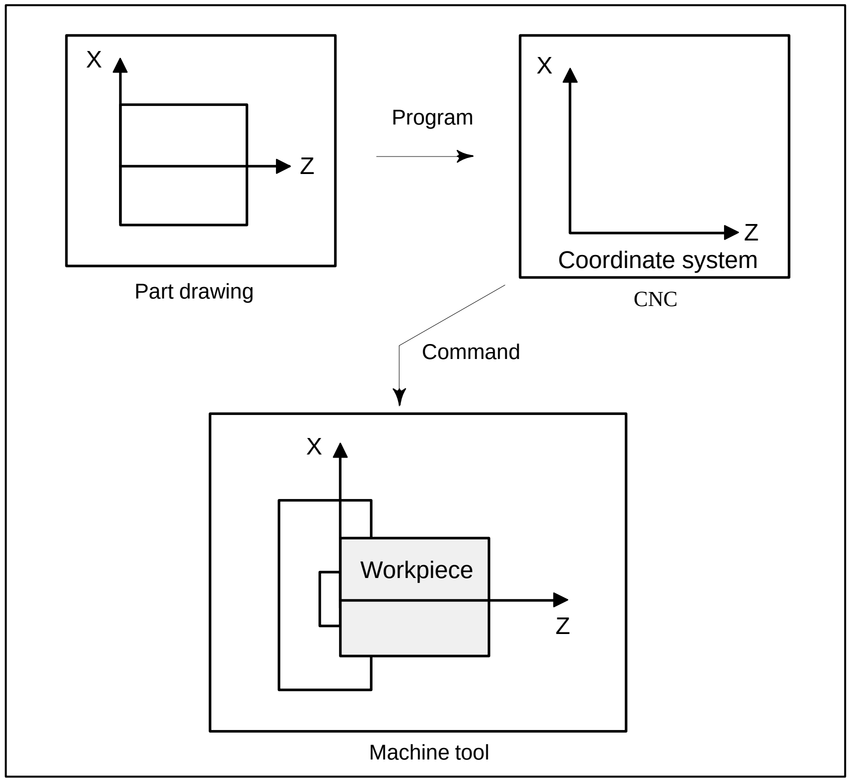


Fig. 1.3.2 (a) Coordinate system

Explanations

- **Coordinate system**

The following two coordinate systems are specified at different locations: (See II–7)

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.

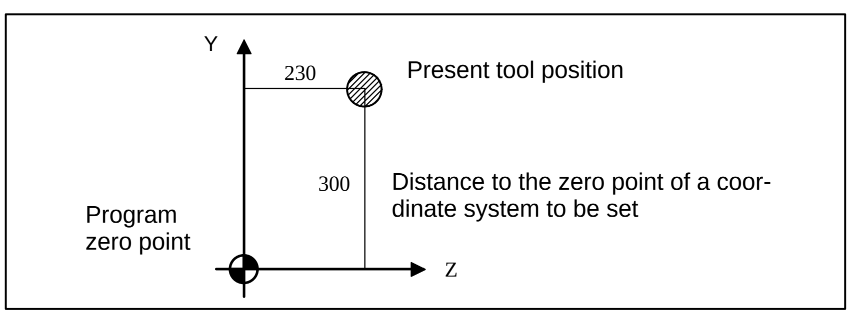


Fig. 1.3.2 (b) Coordinate system specified by the CNC

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

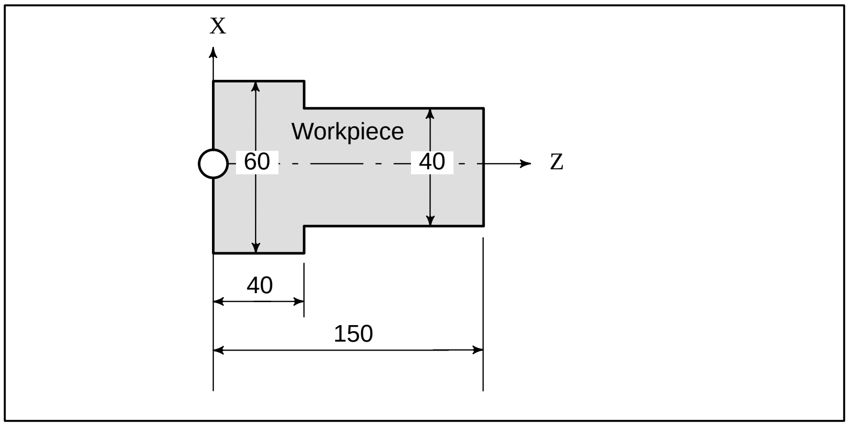


Fig. 1.3.2 (c) Coordinates and dimensions on part drawing

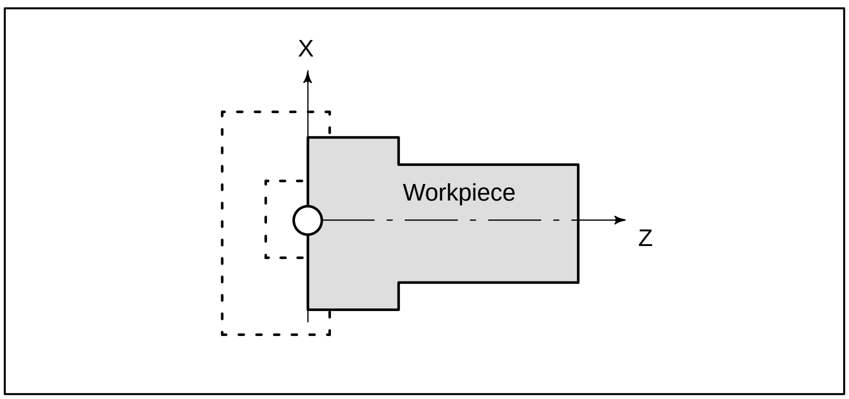


Fig. 1.3.2 (d) Coordinate system on lathe as specified by CNC (made to coincide with the coordinate system on part drawing)

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

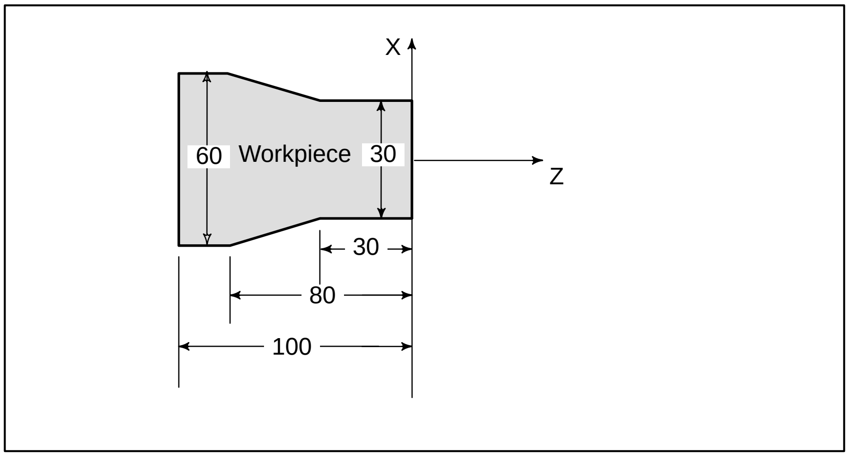


Fig. 1.3.2 (e) Coordinates and dimensions on part drawing

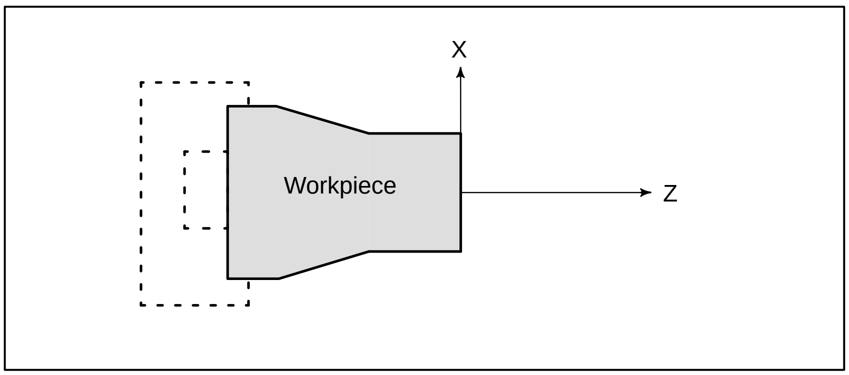


Fig. 1.3.2 (f) Coordinate system on lathe as specified by CNC (made to coincide with the coordinate system on part drawing)

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

Explanations

Coordinate values of command for moving the tool can be indicated by absolute or incremental designation (See II–8.1).

- **Absolute commands**

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.

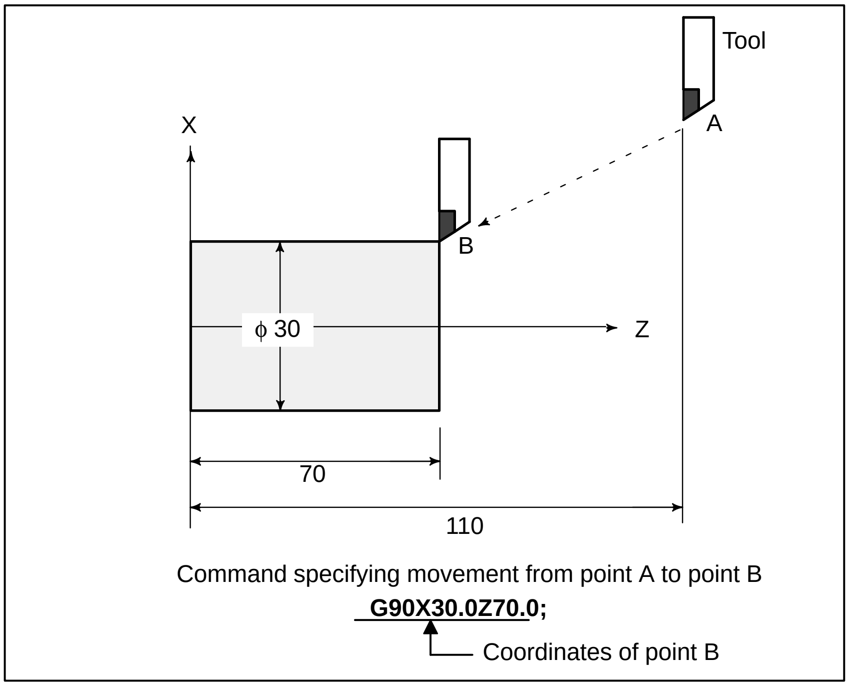


Fig. 1.3.3 (a) Absolute commands

- **Incremental comands**

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

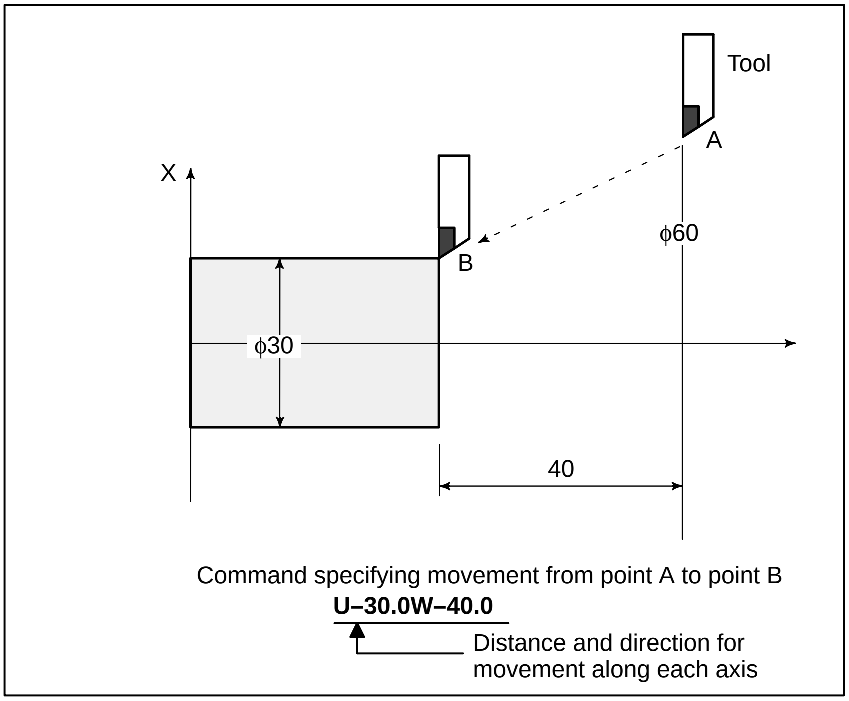


Fig. 1.3.3 (b) Incremental commands

- **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.

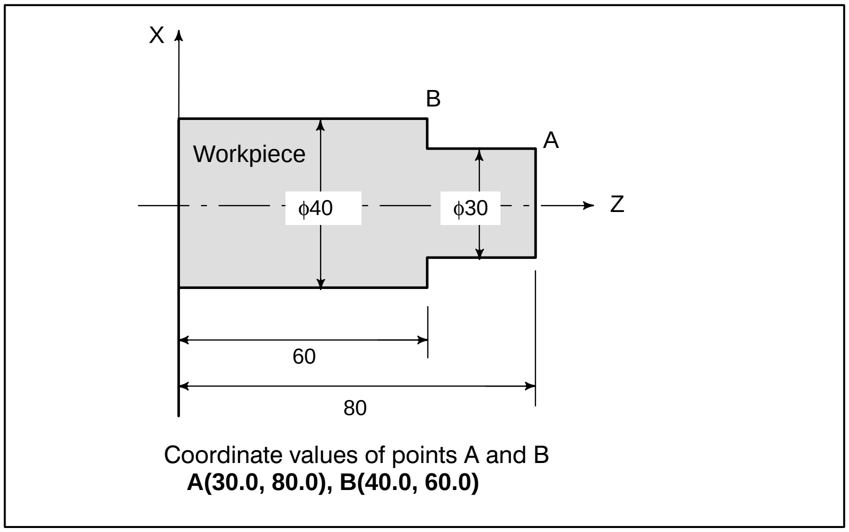


Fig. 1.3.3 (c) Diameter programming

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.

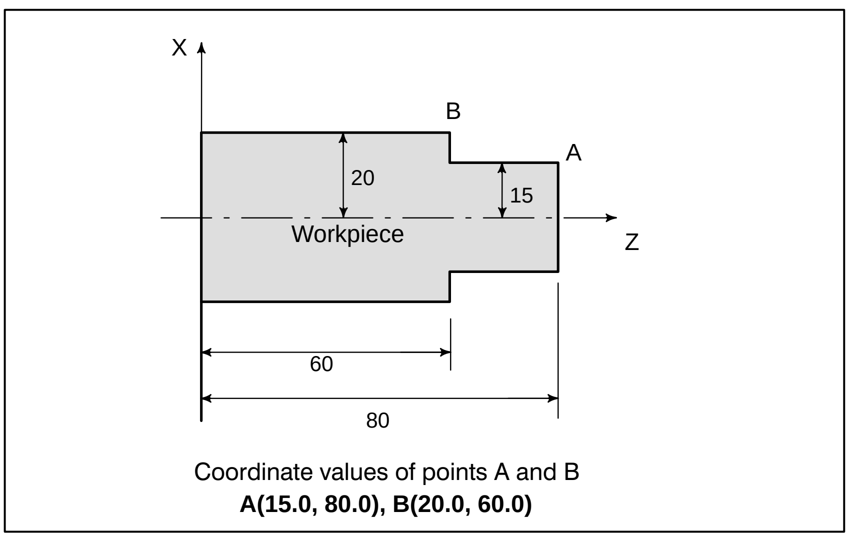


Fig. 1.3.3 (d) Radius programming

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.

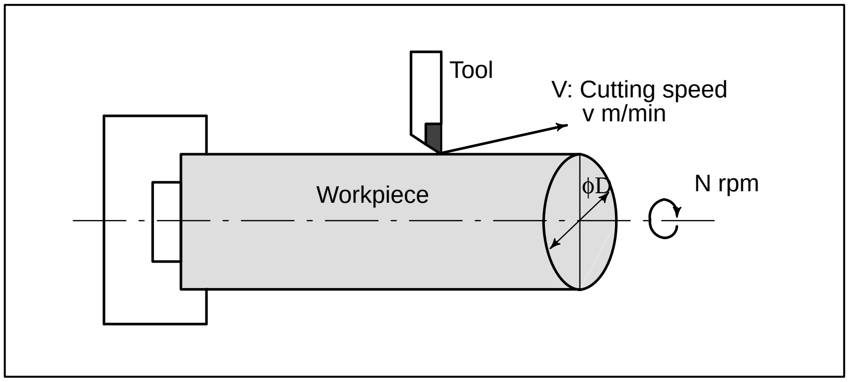


Fig. 1.4 (a) Cutting speed

Examples

<When a workpiece 200 mm in diameter should be machined at a cutting speed of 30 mm/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–9).
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–9.3).

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.

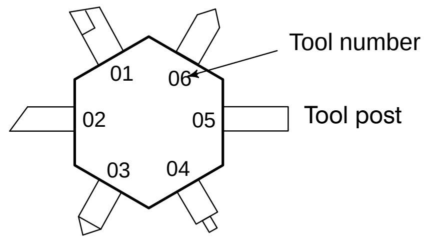


Fig. 1.5 (a) Tool used for various machining

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–10).

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.

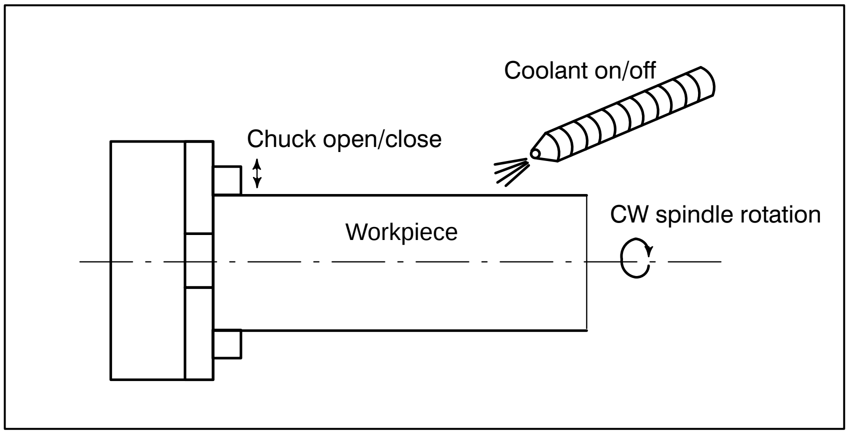


Fig. 1.6 (a) Command for machine operations

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. (See II–11)
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.

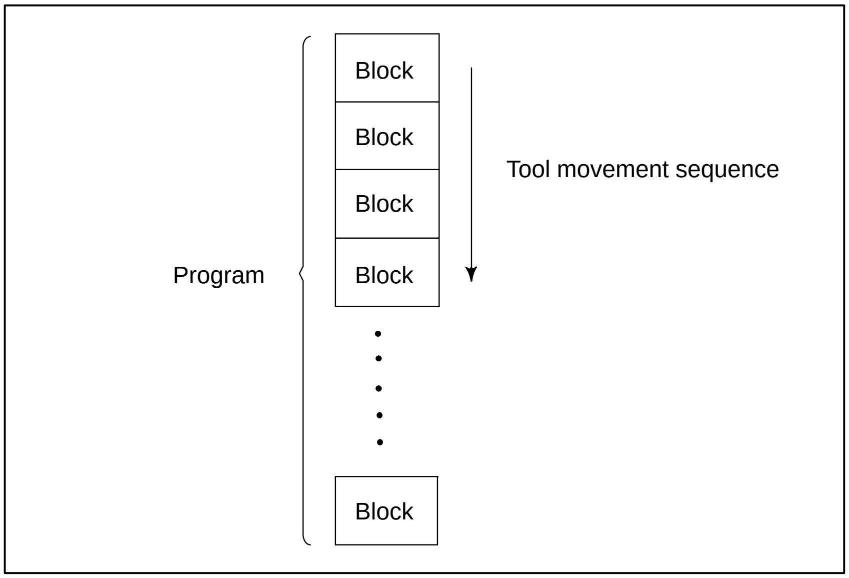


Fig. 1.7 (a) Program configuration

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–12).

Explanations

The block and the program have the following configurations.

- **Block**

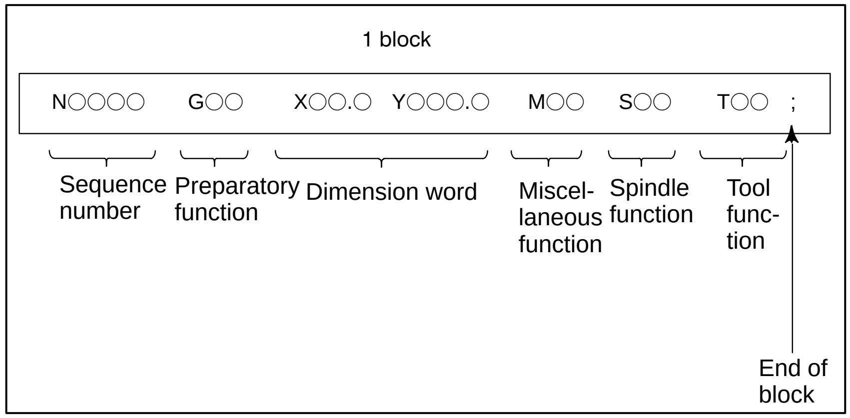


Fig. 1.7 (b) Block configuration

Each block starts with a sequence number which identifies the block, and ends with an end–of–block code which indicates the end of the block. This manual indicates the end–of–block code by ; (LF in the ISO code and CR in the EIA code).

- **Program**

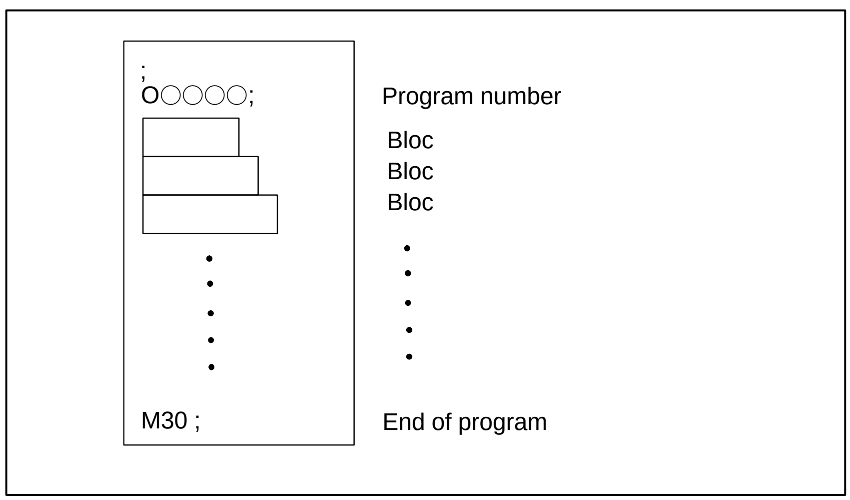
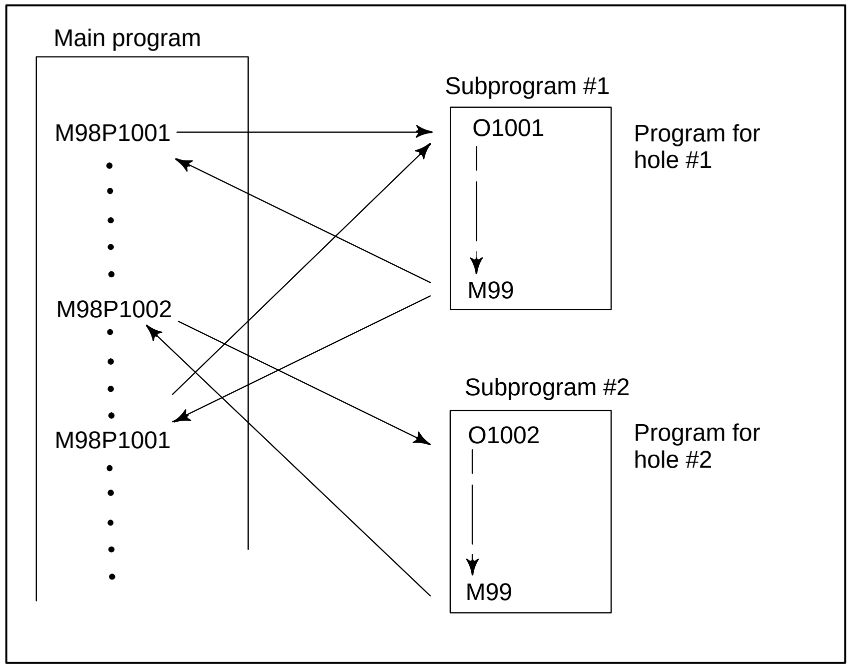


Fig. 1.7 (c) Program configuration

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

- **Tool offset**

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.

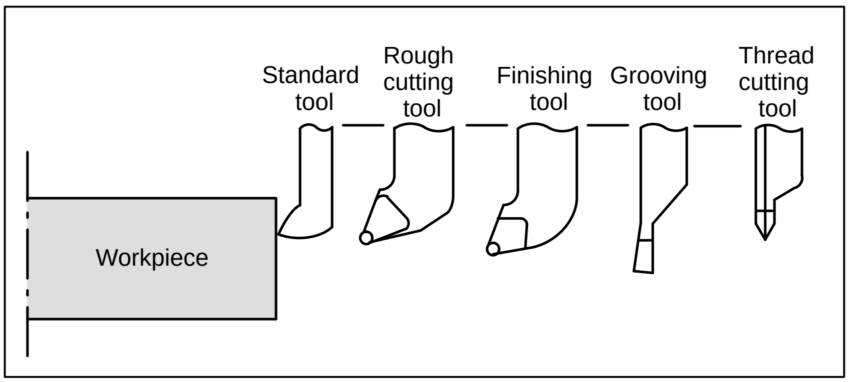
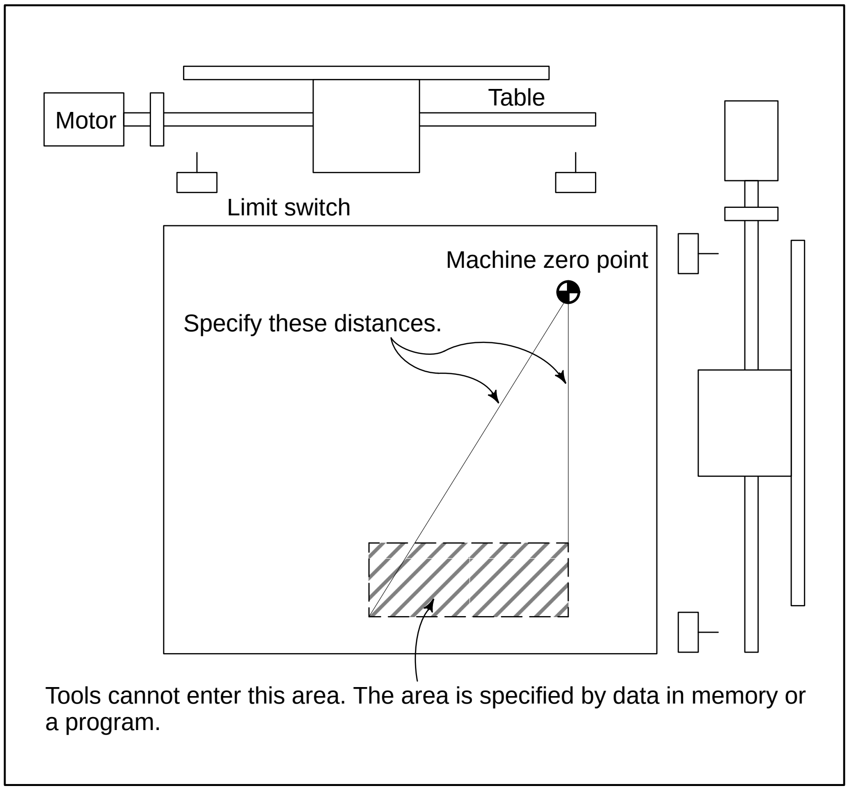


Fig. 1.8 (a) Tool offset

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 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

Item	0–TD	0–GCD
Number of controlled basic axes	2	2
Increase in number of controlled axes (excluding PMC axes)	3 maximum	2 maximum
Number of simultaneously controlled basic axes	2	2
Increase in number of simultaneously controlled axes	2 maximum	2 maximum
PMC–based axis control	None	1 maximum(*1)

*1 Amount by which number of controlled axes can be increased under PMC–based axis control. PMC–based axis control can be applied to a maximum of three axes.

NOTE
In manual operation (such as jog feed, incremental feed, and manual handle feed), the number of simultaneously controlled axes is 1. Note, however, that this number can be increased to 2 by setting bit 4 of parameter No. 049 to 1.

2.2 NAMES OF AXES

The following fixed axis names are used:

	First axis	Second axis	Third axis(*2)
Axis name (absolute programming)	X	Z	C
Incremental programming(*1)	U	W	H

*1 Used for incremental programming when G–code system A is used
*2 Command axis name for third axis. The display axis name varies with the setting of bit 0 of parameter No. 030.

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 2.3.2(a) and 2.3.2 (b)).

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
		0.0001inch(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) Increment system IS–B

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

NOTE
1. The unit inthe 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). G–code system A is standard. Which G–code system is used is specified with bits 1 and 5 of parameter No. 036. Basically, this manual assumes the use of G–code system A. 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 continuous–state 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) Setting bit 0 (G01) of parameter (No.011#6) determines which code, either G00 or G01, is effective.
   (4) Setting bit 3 (G91) of parameter (No.030#7) 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. Alarm 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 gorup are specified in the same block, the G code specified last is valid.
5. G codes are displayed for each group number.

Table 3 G code list (1/2)

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/Helical interpolation CW
G03	G03	G03		Circular interpolation/Helical interpolation CCW
G04	G04	G04	00	Dwell
G10	G10	G10		Data setting
G11	G11	G11		Data setting made cancel
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
G27	G27	G27	00	Reference position return check
G28	G28	G28		Return to reference position
G30	G30	G30		2nd reference position return
G31	G31	G31		Skip function
G32	G33	G33	01	Thread cutting
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, max. spindle speed setting
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 command
G70	G70	G72	00	Finishing cycle (Other than 0–GCD)
G71	G71	G73		Stock removal in turning (Other than 0–GCD)
G72	G72	G74		Stock removal in facing (Other than 0–GCD)
G73	G73	G75		Pattern repeating (Other than 0–GCD)
G74	G74	G76		End face peck drilling (Other than 0–GCD)
G75	G75	G77		Outer diameter/internal diameter drilling (Other than 0–GCD)
G76	G76	G78		Multiple threading cycle (Other than 0–GCD)

Table 3 G code list (2/2)

G code			Group	Function
A	B	C		
G71	G71	G72	01	Traverse grinding cycle (For 0–GCD)
G72	G72	G73		Traverse direct constant–demension grinding cycle (For 0–GCD)
G73	G73	G74		Oscilation grinding cycle (For 0–GCD)
G74	G74	G75		Oscilation direct constant–dimension grinding cycle (For 0–GCD)
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
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
—	G99	G99		Return to R point level

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.

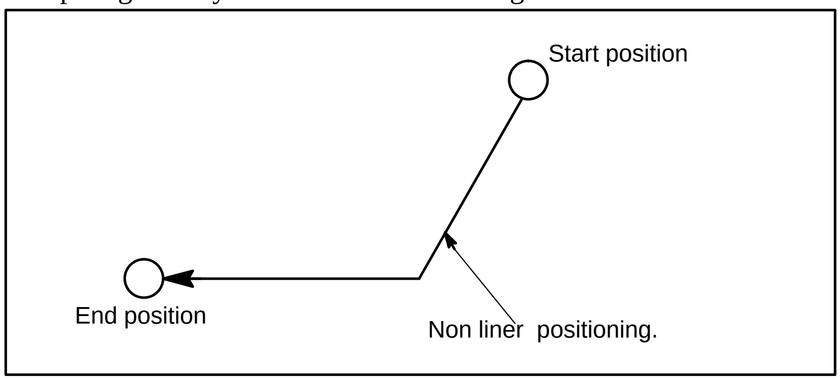
Format

G00 IP_;

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

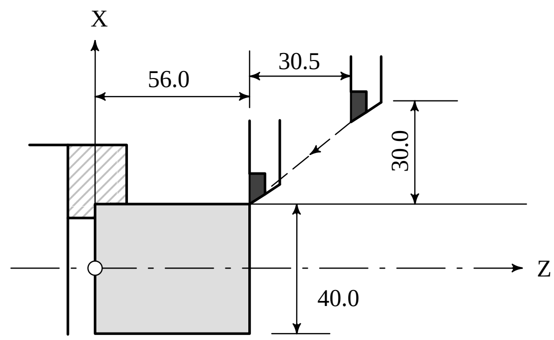
Explanations

Tool path generally does not become a straight line.



The rapid traverse rate in the G00 command is set to the parameter Nos. 518 to 520 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 Nos. 500 to 502.

Examples



< Radius programmimg >
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.

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 commnad, 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 : $F\alpha = \frac{\alpha}{L} \times f$

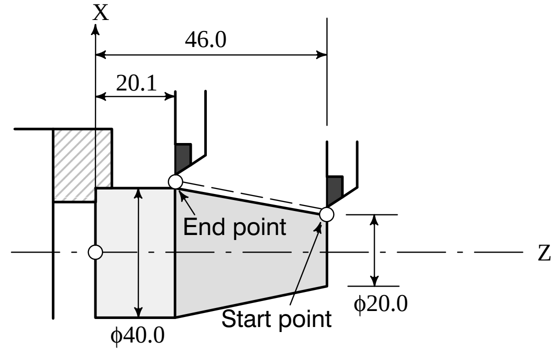
Feed rate of β axis direction : $F_{\beta} = \frac{\beta}{L} \times f$

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

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

$$\mathbf{G17} \left\{\begin{array}{c} \mathbf{G02} \\ \mathbf{G03} \end{array}\right\} \mathbf{Xp\_Yp\_} \left\{\begin{array}{c} \mathbf{I\_J\_} \\ \mathbf{R\_} \end{array}\right\} \mathbf{F\_\ ;}$$

Arc in the ZpXp plane

$$\mathbf{G18} \left\{\begin{array}{c} \mathbf{G02} \\ \mathbf{G03} \end{array}\right\} \mathbf{Xp\_Zp\_} \left\{\begin{array}{c} \mathbf{I\_K\_} \\ \mathbf{R\_} \end{array}\right\} \mathrm{F\_\ ;}$$

Arc in the YpZp plane

$$\mathbf{G19} \left\{\begin{array}{c} \mathbf{G02} \\ \mathbf{G03} \end{array}\right\} \mathbf{Yp\_Zp\_} \left\{\begin{array}{c} \mathbf{J\_K\_} \\ \mathbf{R\_} \end{array}\right\} \mathrm{F\_\ ;}$$

Table 4.3 Description of the Command Format

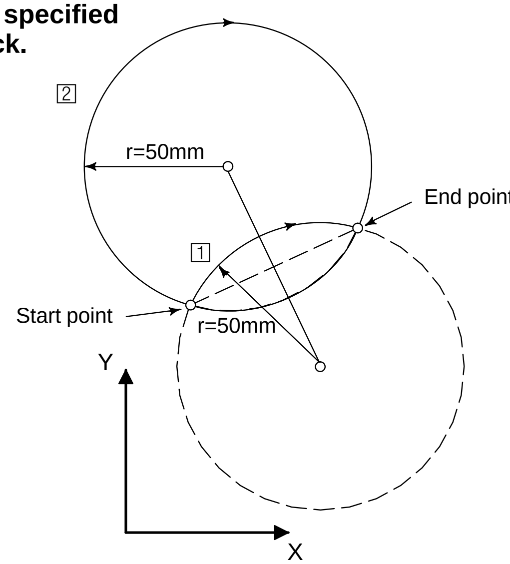
Command	Description
G17	Specification of arc on XpYp plane
G18	Specification of arc on ZpXp plane
G19	Specification of arc on YpZp plane
G02	Circular Interpolation Clockwise direction (CW)
G03	Circular Interpolation Counterclockwise direction (CCW)
$X_{p\_}$	Command values of X axis or its parallel axis (set by parameter Nos. 279 and 280)
$Y_{p\_}$	Command values of Y axis or its parallel axis (set by parameter Nos. 279 and 280)
$Z_{p\_}$	Command values of Z axis or its parallel axis (set by parameter Nos. 279 and 280)
I_	X_p axis distance from the start point to the center of an arc with sign or radius value
J_	Y_p axis distance from the start point to the center of an arc with sign
k_	Z_p axis distance from the start point to the center of an arc with sign
R_	Arc radius with sign fixed to radius value.
F_	Feedrate along the arc

- **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 180k, and the other is more than 180k are considered. An arc with a sector angle of 180kor 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 0kis programmed

G02R ; (The cutter does not move.)

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



- **Feedrate**

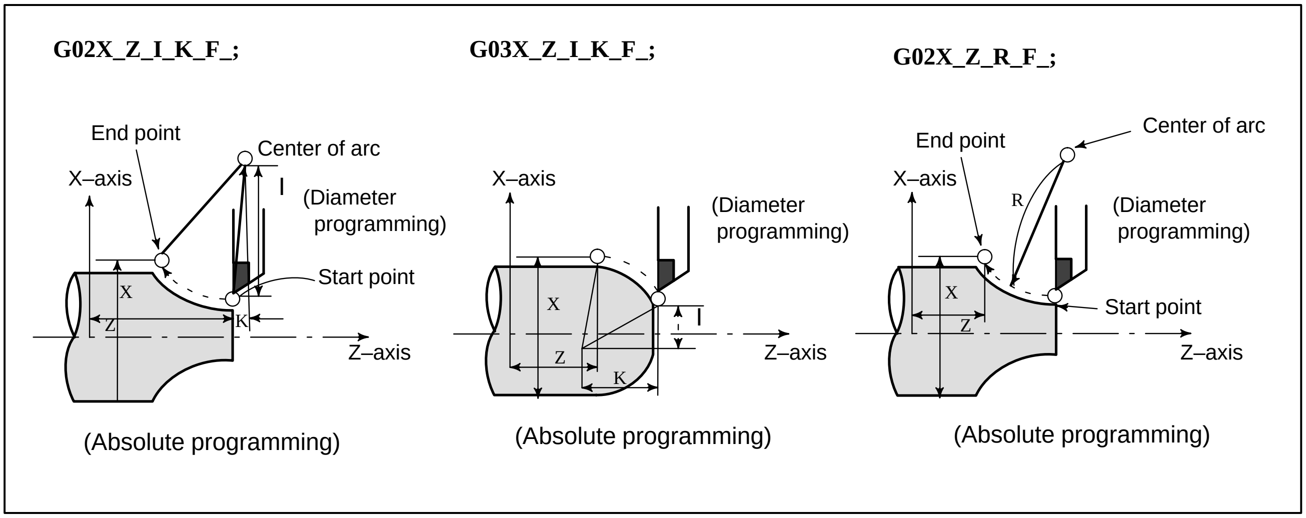
The feedrate in circular interpolation is equal to the feed rate specified by the F code, and the feedrate along the arc (the tangential feedrate of the arc) is controlled to be the specified feedrate.
The error between the specified feedrate and the actual tool feedrate is ±2% or less. However, this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

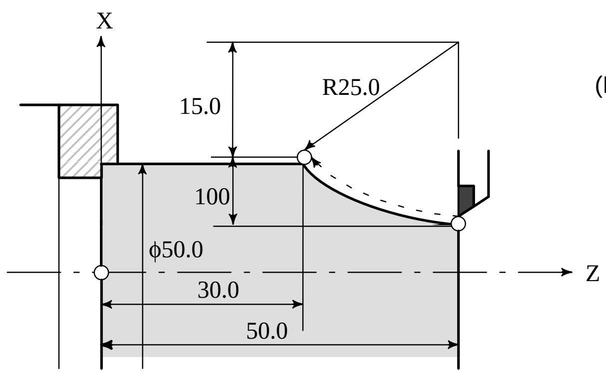
Restrictions

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 (No. 028) is displayed.
When an arc having a center angle approaching 180 degrees is specified in R, a center position calculation error may occur. 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 CONSTANT LEAD THREADING (G32)

Tapered screws and scroll threads in addition to equal lead atraight 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.

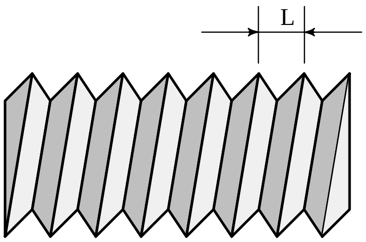


Fig.4.4 (a) Straight Thread

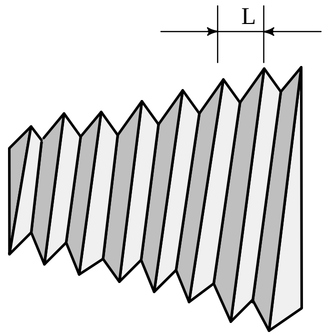


Fig.4.4 (b) Tapered Screw

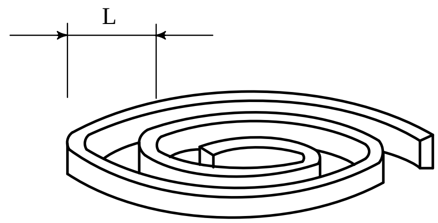


Fig.4.4 (c) Scroll Thread

Format

G32IP_F_;

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

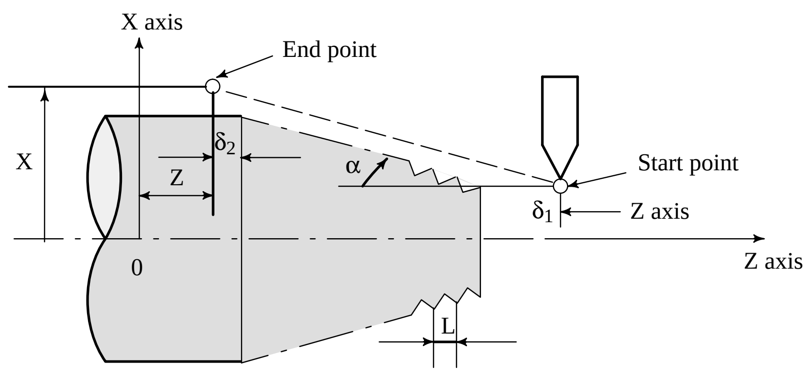


Fig. 4.4 (d) Example of Thread Cutting

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.

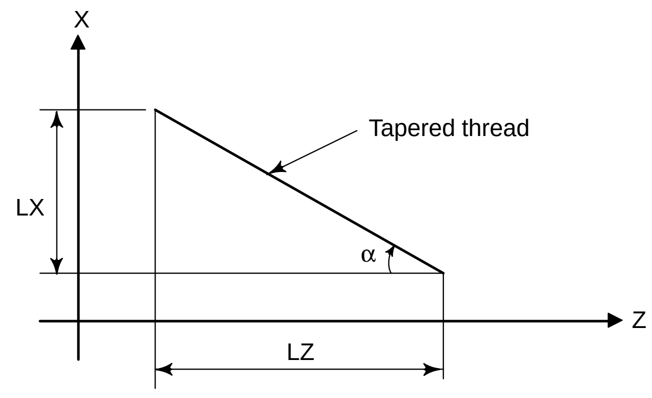


Fig. 4.4 (e) LZ and LX of a Tapered Thread

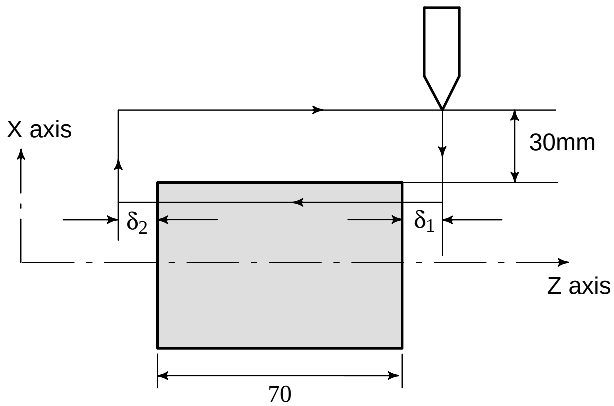
In general, the lag of the servo system, etc. will produce somewhat incorrect leads at the starting and ending points of a thread cut. To compensate for this, a threading length somewhat longer than required should be specified.
Table 4.4 lists the ranges for specifying the thread lead.

Table 4.4 Ranges of lead size that can be specified

	Least command increment
mm input	0.0001 to 500.0000mm
Inch input	0.000001 inch to 9.999999inch

Explanations

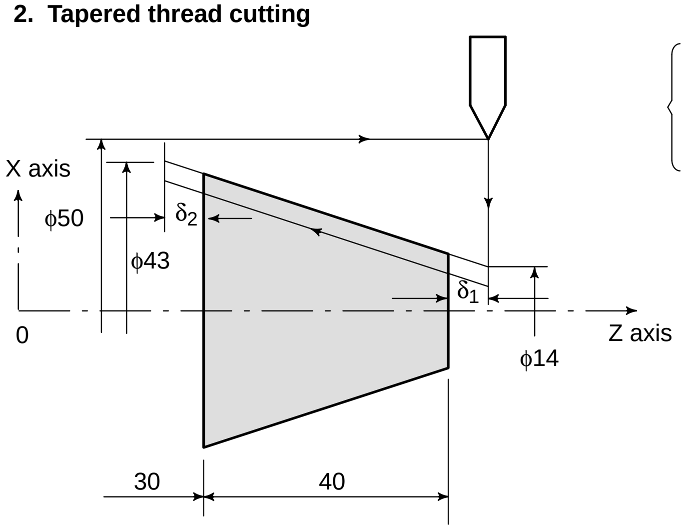
1. Straight thread cutting



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  ;
```

WARNING

1 Feedrate override is effective (fixed at 100%) during thread cutting.

2 it is very dangerous to stop feeding the thread cutter. 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 again pushed during the first block not specifying thread cutting just after thread cutting block or when it has been continuously pushed, the tool stops at the block not specifying 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. However, the lead at the point where the blocks join is incorrect. To obtain the correct lead, the continuous threading option is required.

```
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.

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 is effective in thread cutting mode.

11 Thread cycle retract function is ineffective to G32.

4.5 CONTINUOUS THREAD CUTTING (0–GCD)

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.

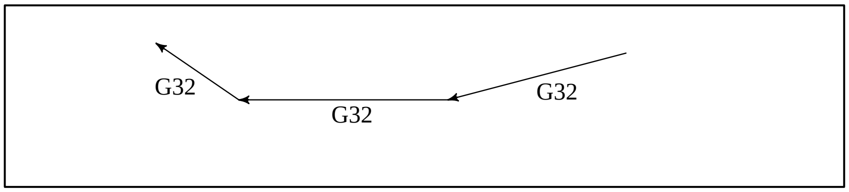


Fig. 4.5 (a) Continuous Thread Cutting

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.

NOTE
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.

4.6 SKIP FUNCTION (G31) (0–GCD)

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.

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
#5064 4th axis coordinate value

WARNING

Disable feedrate override, dry run, and automatic acceleration /deceleration when the feedrate per minute is specified, allowing for an error in the position of the tool when a skip signal is input. By setting bit 3 of parameter No.015, however, feedrate override, dry run, and automatic acceleration/deceleration can be enabled during movement based on the skip function.These functions are enabled when the feedrate per rotation is specified.

NOTE

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 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**

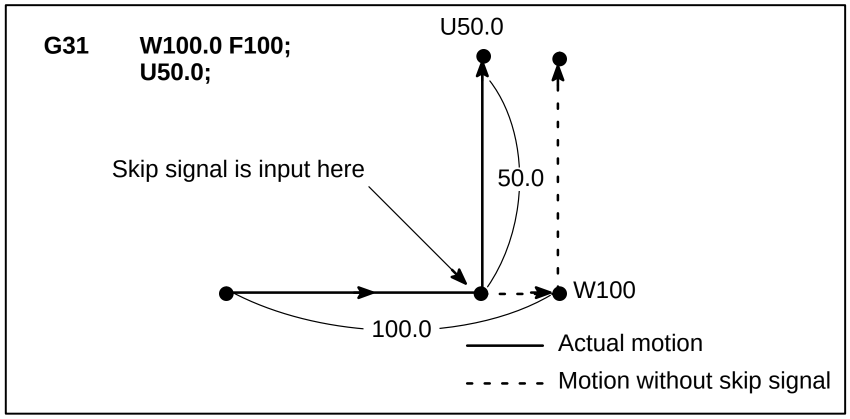


Fig. 4.6 (a) The next block is an incremental command

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

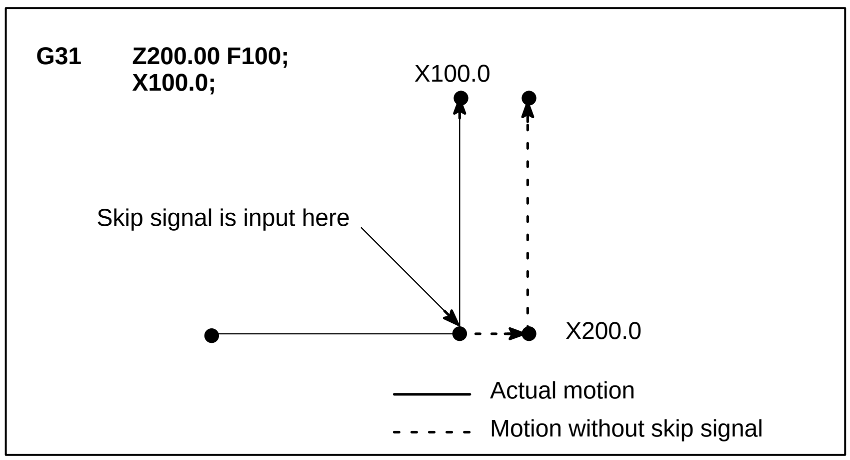


Fig. 4.6 (b) The next block is an absolute command for 1 axis

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

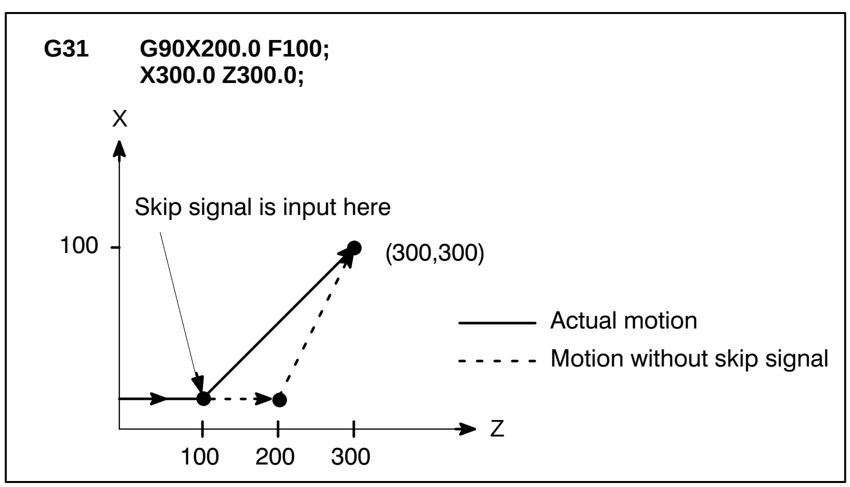


Fig. 4.6 (c) The next block is an absolute command for 2 axes

4.7 MULTI–STEP (0–GCD)

In a block specifying P1 to P4 after G31, the multi–step skip function stores coordinates in a custom macro variable when a skip signal (4–point) is turned on. In a block having Q1 to Q4 specified after G04, a dwell operation can be skipped by turning on the skip signal.
Parameters No. 033 and 034 can be used to select a 4–point 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 No. 035 #0 to #3 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.

CAUTION
Dwell is not skipped when Qn is not specified and parameters No. 035#0–#3 are not set.

4.8 SKIP FUNCTION BY TORQUE LIMIT ARRIVAL SIGNAL

This function skips the remaining motion of the block by the torque limit arrival signal from the servo motor. It controls such functions as pressing a workpiece to the chuck or transferring a workpiece to another axis.

Format

G31 P99 Motion Command F feedrate ;

Explanations

When commanding the above with limiting the torque of the servo motor, the remaining motion of the block is skipped by asserting the torque limit arrival signal from the motor, and the next block is executed. The skip position is memorized in the ordinary system variables. G31 P99 is valid only on the commanded block.

Generally, the servo lag is not zero after the skip by the torque limit arrival signal. There are two methods to maintain the servo lag. One is leaving the lag as it is.

Another is retrieving the lag to the actual position and making it almost zero. In the latter case, the retrieved position is memorized as the skip position. The parameter (No.389#4) selects either.

• Program Example

G00 Z500.0 ;	Positioning to the start point
M□□ ;	Limiting the torque of servo motor
G31 P99 W22.0 F100.0 ;	Commanding the skip function
G01 Z500.0 ;	Re–positioning to the start point
M○○ ;	Releasing the torque of servo motor

WARNING

1 The torque limit for the servo motor should be commanded before G31 P99. The torque limit can be achieved by using PMC window function. At lease, one torque limit arrival signal of the connected servo motors is turned on, the skip is carried out.

2 The torque limit should be released at the point where the workpiece and the axis is not contacting each other. When resetting the G31 P99, it is interrupted but the torque limit is not released automatically.

NOTE

1 The ordinary skip signal is also active in G31 P99.
2 The in–position is not checked at the end of G31 P99.
3 The mirror image should be cancelled before commanding G31 P99.
4 The nose–R compensation should be cancelled before commanding G31 P99.
5 G31 P99 should not be commanded successively.
6 Dry–run, feedrate and automatic acc./dec. should be ignored during G31 P99. (Parameter No.015#5=0)
7 The servo excess error during stop at the axis is not checked while the torque limit arrival signal turns on.

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 (parameters No. 518 to 521).
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)).

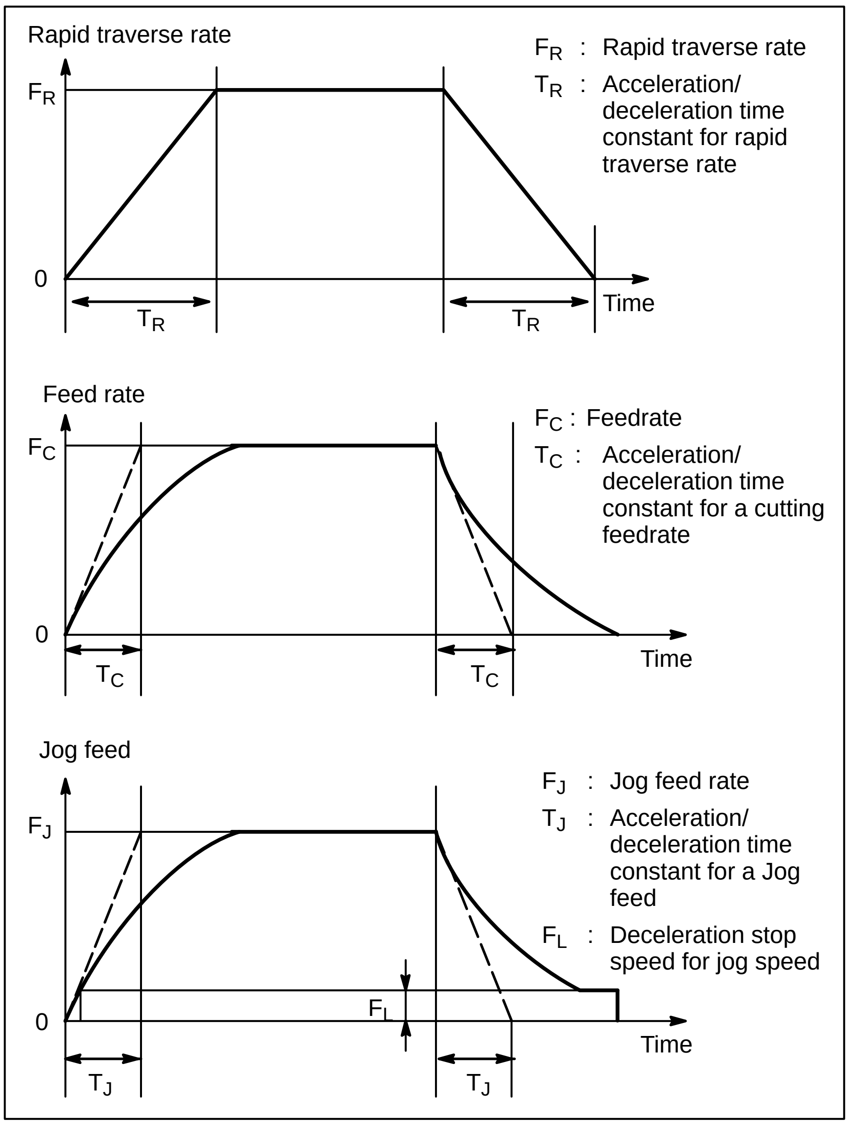


Fig. 5.1 (a) Automatic acceleration/deceleration (example)

- **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)).

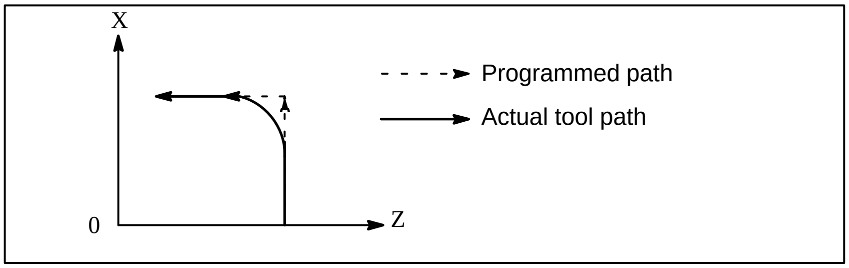


Fig. 5.1 (b) Example of Tool Path between Two Blocks

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

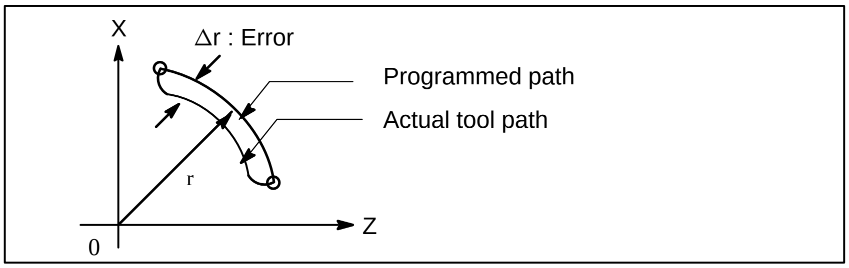


Fig. 5.1 (c) Example of Radial Error in Circular Interpolation

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).
By setting bit 5 of parameter No. 020, however, the in–position check function can be disabled in each positioning block.
A rapid traverse rate is set for each axis by parameters No. 518 to 526, 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 by parameter No. 533.
For detailed information, refer to the appropriate manual of the machine tool builder.

Command value range of the rapid traverse

Metric output	30 to 100,000 mm/min 30 to 100,000 deg/min
Inch output	3.0 to 4,000.0 inch/min 3.0 to 100,000 deg/min

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.

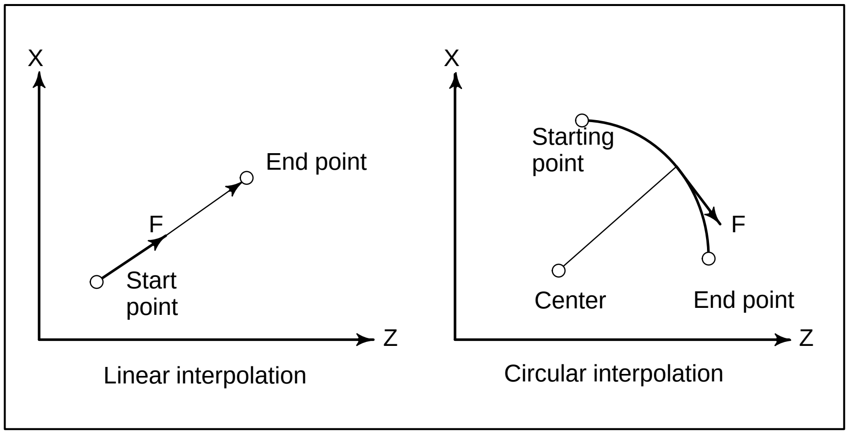


Fig. 5.3 (a) Tangential feedrate (F)

- **Feed per minute (G98)**

After specifying G98, 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 150% (in 10% 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.

- **Feedrate command value range for feed per minute**

	Increment system	
	IS–B	**IS–C**
Metric output	1 to 100,000 mm/min 1 to 100,000 deg/min	1 to 12,000 mm/min 1 to 12,000 deg/min
Inch output	0.01 to 4,000.00 inch/min 0.01 to 6,000.00 deg/min	0.01 to 480.00 inch/min 0.01 to 600.00 deg/min

- **Feedrate decimal point input for feed per minute**

The command value ranges indicated in the above table are also applicable when a decimal point is used in the specification of a feedrate for feed per minute. By setting bit 5 of parameter No. 077, however, the command value ranges for those cases where a decimal point is entered can be modified as indicated below. With 0–GCD, the following command value ranges are applicable even if the parameter is not set. When a feedrate that falls outside the command value ranges shown below is required, specify the feedrate without entering a decimal point.

	Increment system	
	IS–B	**IS–C**
Metric output	0.001 to 99,999.999 mm/min 0.001 to 99,999.999 deg/min	0.001 to 12,000.000 mm/min 0.001 to 12,000.000 deg/min
Inch output	0.00001 to 999.99999 inch/min 0.00001 to 999.99999 deg/min	0.00001 to 480.00000 inch/min 0.00001 to 600.00000 deg/min

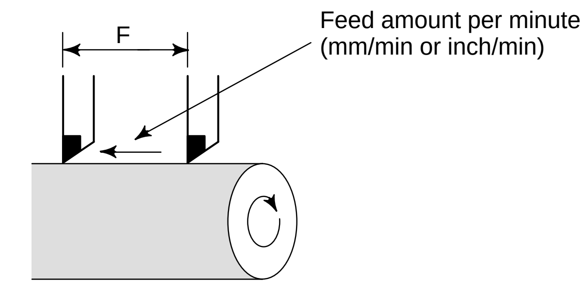


Fig. 5.3 (b) Feed per minute

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

- **Feed per revolution (G99)**

After specifying G99, 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 150% (in 10% 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.

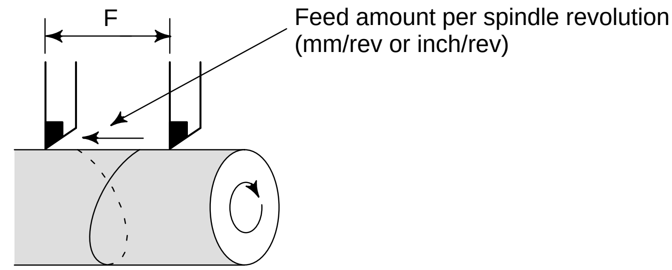


Fig. 5.3 (c) Feed per revolution

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

- **Feedrate command value range for feed per rotation**

	Increment system	
	IS–B	**IS–C**
Metric output	0.0001 to 500.0000 mm/rev	0.0001 to 500.0000 mm/rev
Inch output	0.000001 to 9.999999 inch/rev	0.000001 to 9.999999 inch/rev

The feedrate command value ranges for feed per rotation indicated in the table above indicate the values that can be specified; note that the actual feedrate is clamped to a maximum cutting feedrate.

- **Cutting feedrate clamp**

A common upper limit can be set on the cutting feedrate along each axis with parameter No. 527. 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:

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.

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
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
IS–C	1 to 99999999	0.0001 s

CAUTION
Dwell and Tool offset can not be commanded in the same block.

5.5 DWELL BY TURNING TIMES OF SPINDLE

This function specifies the dwell interval by turning times of the spindle instead of the time interval.
This function is enabled by setting bit 0 of parameter No. 395 accordingly.

Format

```
         ┌ P__________
(G99) G04│ X__________  ;
         └ U__________
```

Explanations

The dwell command by designating turning times of the spindle is possible when commanding G04 during the feed per revolution mode (G99). Turning times of the spindle is designated by P–code, X–code or U–code. The dwell is performed while the spindle rotates the designated times. The setting unit and range of turning times are as follows. The decimal point can be used in X–code and U–code.

	Unit	Range
Standard	0.001 rev	0 to 99999.999 rev
Input unit 1/10	0.0001 rev	0 to 9999.9999 rev

Example

When commanding G99 G04 U2.5;, the dwell is preformed while the spindle rotates 2.5 times.

CAUTION
1 In spite of the parameter setting, the dwell command during the feed per minute is regarded as the ordinary dwell by the time interval.
2 When setting the parameter (No.062#3) to ”1” to expand the maximum spindle speed, This function is not used.

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 two reference positions can be specified by setting coordinates in the machine coordinate system in parameters. However, the first reference position matches the machine zero point.

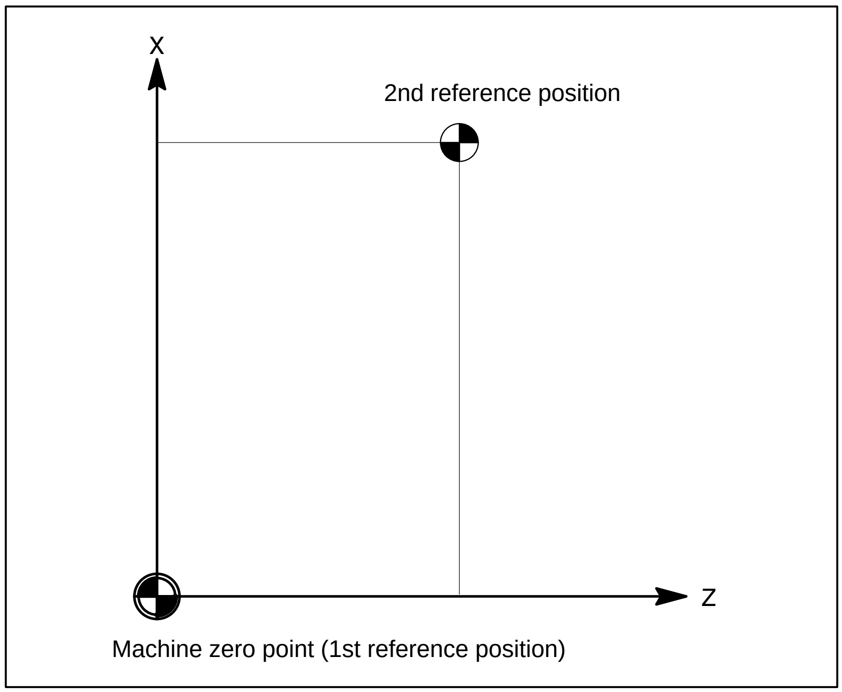


Fig. 6 (a) Machine zero point and reference positions

- **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.

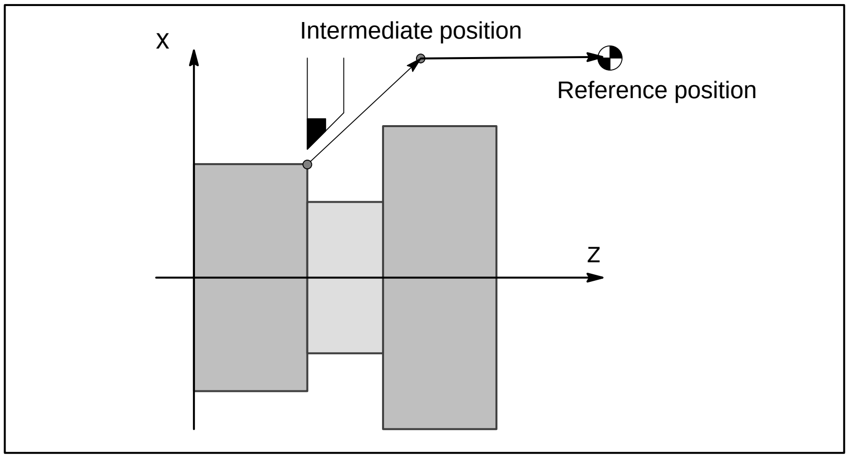


Fig. 6 (b) Reference position return

- **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.)**

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 reference position return (G30)**

In a system without an absolute–position detector, the second 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 0 to 2 of No. 003). 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 input unit. 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 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.

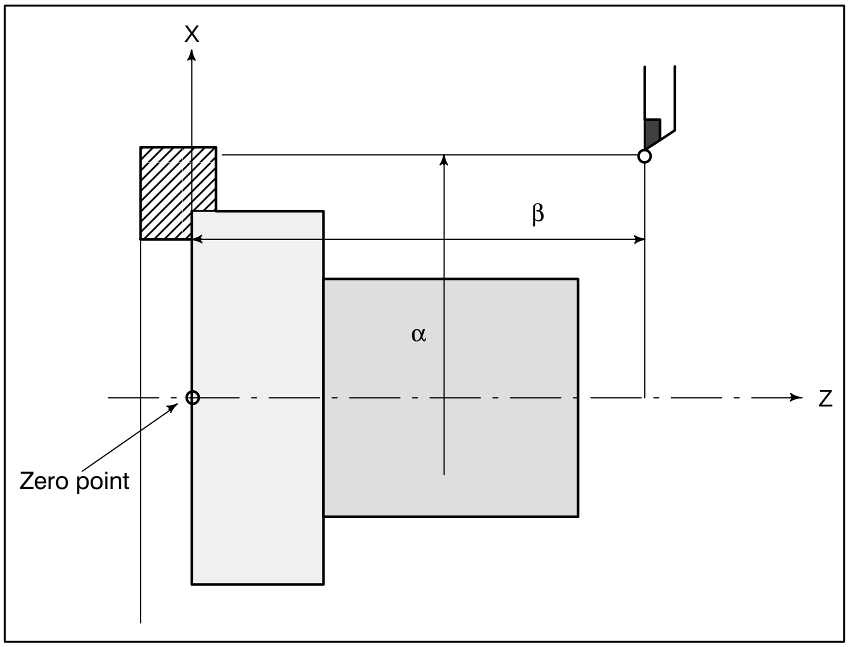


Fig. 7 Tool Position Specified by XαZβ

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_**.

7.1 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. The machine zero point matches the first reference position.
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–4.1). 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 command is specified based on a machine coordinate system, the tool moves by rapid traverse. G53, which is used to select a machine coordinate system, is a one–shot G code; that is, it is valid only in the block in which it is specified. The absolute command is valid, but the incremental 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.
The workpiece coordinate system option (G54 to G59) is required.

7.2 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**).

7.2.1 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 7 of parameter No. 010 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.).
To enable the use of absolute programming, a workpiece coordinate system must be established using one of the methods described above.

Format

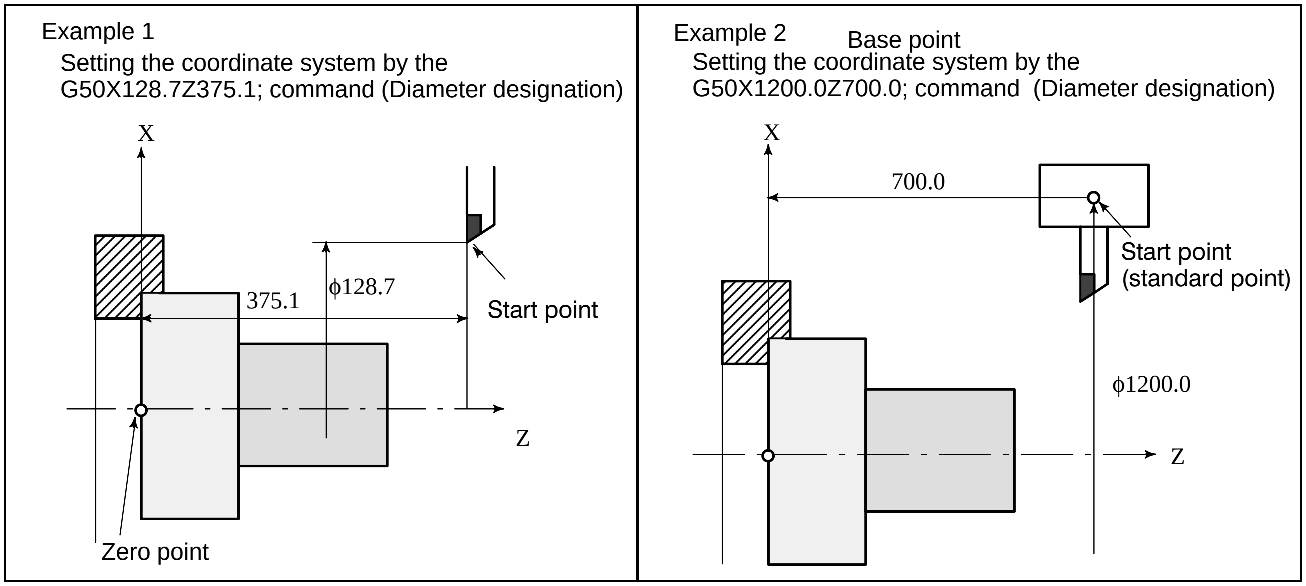
Setting a workpiece coordinate system by G50

```
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


Example 1
Setting the coordinate system by the
G50X128.7Z375.1; command (Diameter designation)
X
375.1
ϕ128.7
Start point
Z
Zero point
Example 2
Base point
Setting the coordinate system by the
G50X1200.0Z700.0; command (Diameter designation)
X
700.0
Start point
(standard point)
ϕ1200.0
Z

7.2.2 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 Section 7.2.1.)

(1) Selecting a workpiece coordinate system set by 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 CRT/MDI panel

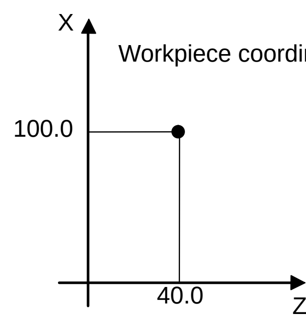
By specifying a G code from G54 to G59, one of the workpiece coordinate systems 1 to 6 can be selected.

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.

Examples

G55 G00 X100.0 Z40.0 ;



In this example, positioning is made to positions (X=100.0, Z=40.0) in workpiece coordinate system 2.

Fig. 7.2.2 (a)

7.2.3 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 CRT/MDI panel (see III–11.4.6)

(2) Programming by G10 or G50

(3) Changing an external workpiece zero point offset value (refer to machine tool builder's manual)

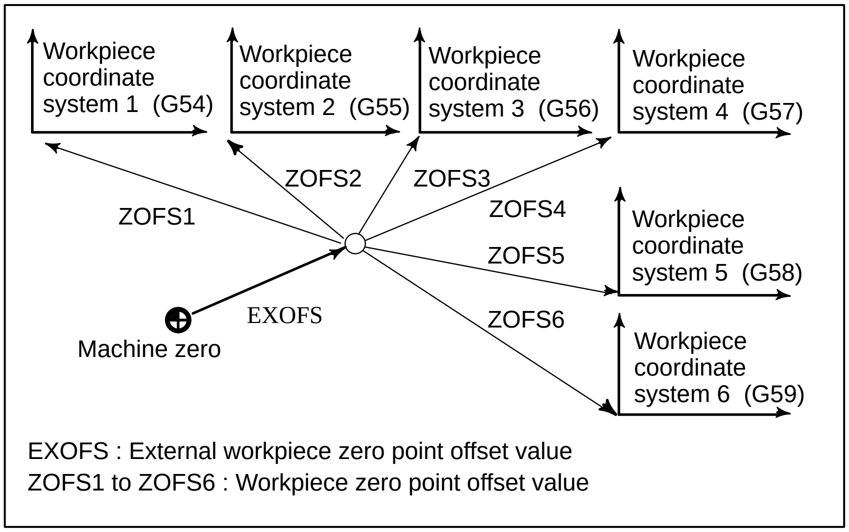


Fig. 7.2.3 (a) Changing an external workpiece zero point offset value or workpiece zero point offset value

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**

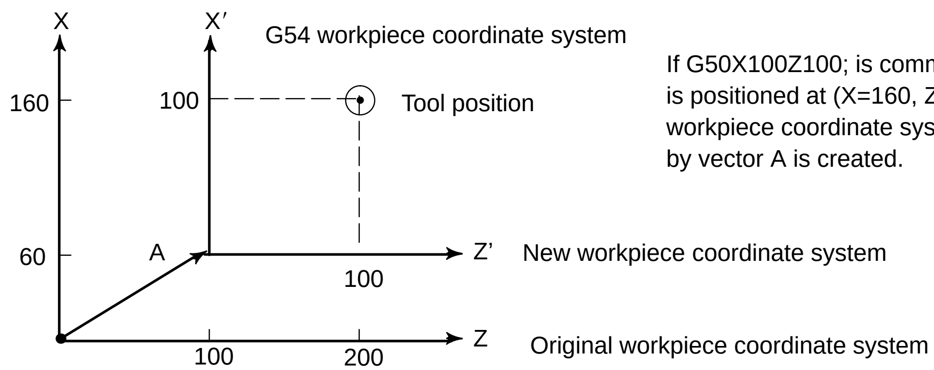
With the G10 command, each workpiece coordinate system can be changed separately.

- **Changing by G50**

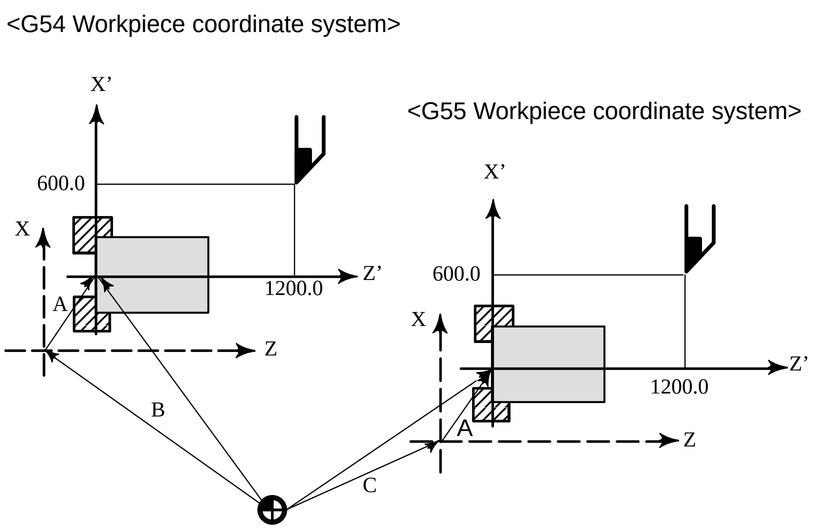
By specifying G50 IP_;, a workpiece coordinate system (selected with a

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 currnt 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 (X=160, Z=200) 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,C : Workpiece zero point offset value in the original workpiece coordinate system

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.

7.2.4 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 beshifted (see III–3.1).
Set the desired shift amount in the work coordinate system shift memory.

Explanations

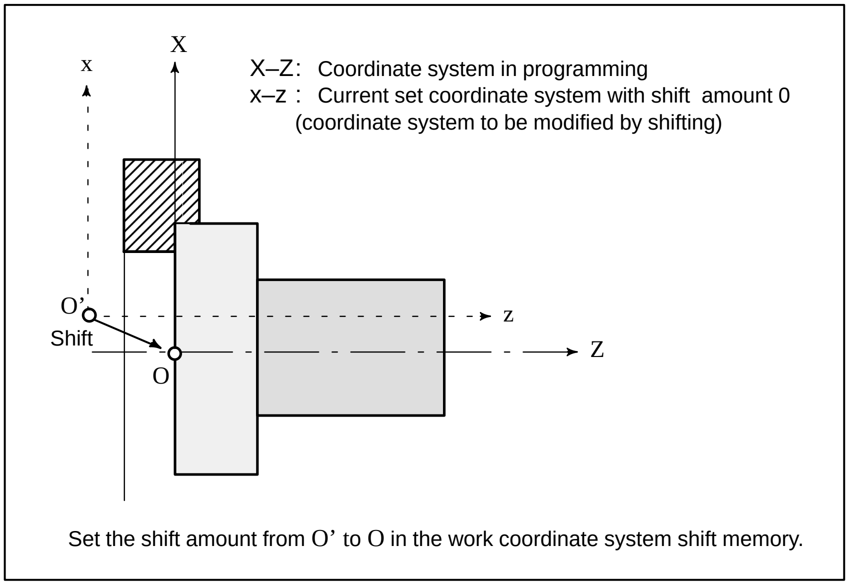


Fig. 7.2.4 (a) Workpiece Coordinate System shift

See Subsection 11.4.4 of Part III for how to specify the distance the work coordinate system is shifted.

7.3 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 G52 **IP_**;, 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.

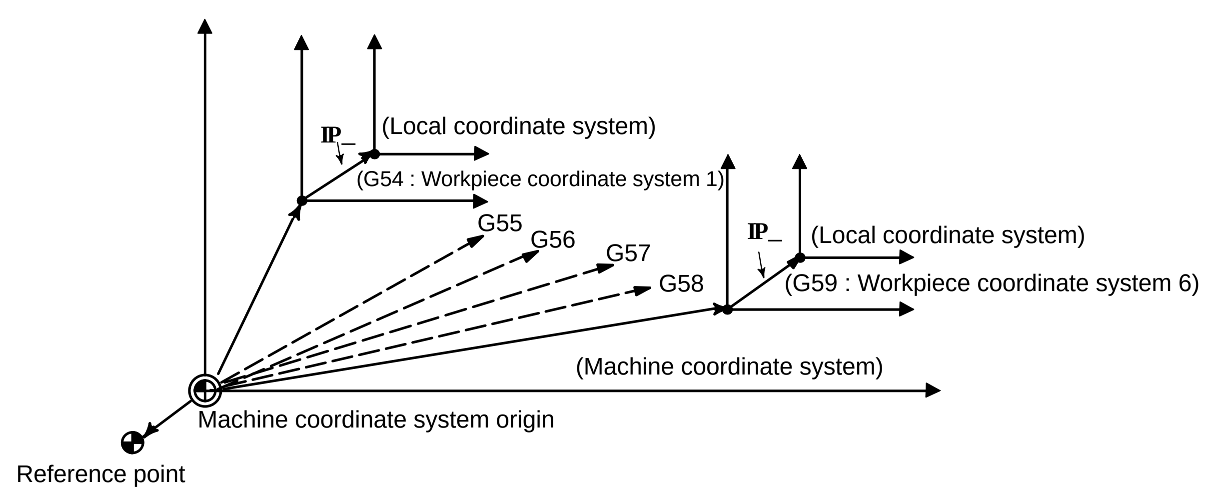


Fig. 7.3 Setting the local coordinate system

WARNING

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 coordinatesystem 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.

7.4 PLANE SELECTION

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

Explanations

Table 7.4 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.
Parameters No. 279 and 280 specifies whether 3rd axis or 4th 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.

NOTE
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 PS alarm No. 212 to be generated.

Examples

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

G17X_Y_;	XY plane,
G17A_Y_;	AY plane
G18X_Z_;	ZX plane
X_Y_;	Plane is unchanged (ZX plane)
G17 ; . .	XY plane
G18 ; . .	ZX plane
G17 A_ ;	AY plane
G18Y_ ;	ZX plane, Y axis moves regardless without any relation to the plane.

8 COORDINATE VALUE AND DIMENSION

This chapter contains the following topics.

8.1 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 Y Z C	U W V H

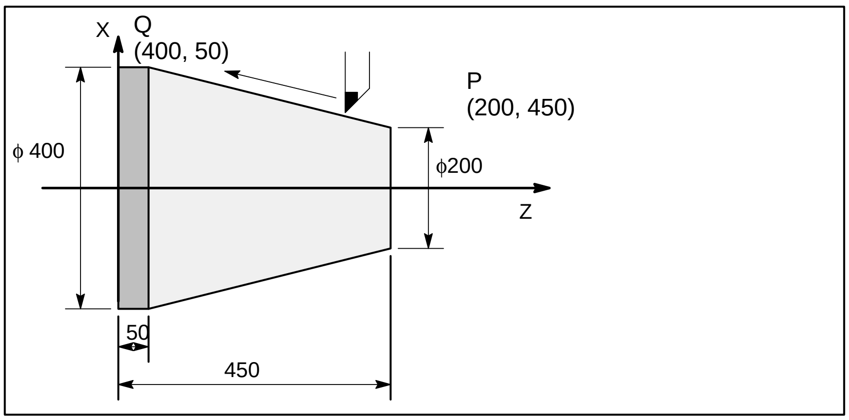
- **G code system B or C**

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 ;



NOTE
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.

8.2 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 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.

WARNING

1 G20 and G21 must not be switched during a program.
2 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.

CAUTION

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.

NOTE

1 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.
2 The inch and metric input can also be switched using setting parameter (inch).

8.3 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, inches or sections. 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 (bit 7 of parameter 015). 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 without 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

WARNING

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.

NOTE

1 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.

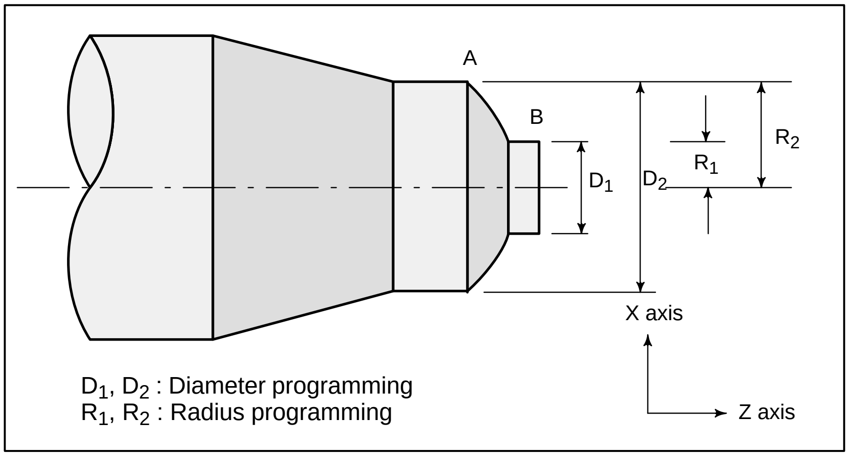
2 When more than eight digits are specified, an alarm occurs. If a value is entered with a decimal point, the number of digits is also checked after the value is converted to an integer according to the least input increment.

Examples:

X1.23456789;	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 alarm occurs.

8.4 DIAMETER AND RADIUS PROGRAMMING

Since the work cross section is usually circular in CNC lathe control programming, dimensions of X axis 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 (No.019#2). When using diameter programming, note the conditions listed in the table8.4(a).

Table 8.4 (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.001#4) 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

9 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.

9.1 SPECIFYING THE SPINDLE SPEED WITH A BINARY CODE

This spindle speed can be specified by address S followed by 2–digit code. A block can contain only one S code. Refer to the appropriate manual provided by the machine tool builder for details such as the execution order when a move command and an S code command are in the same block.

9.2 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.

9.3 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 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.

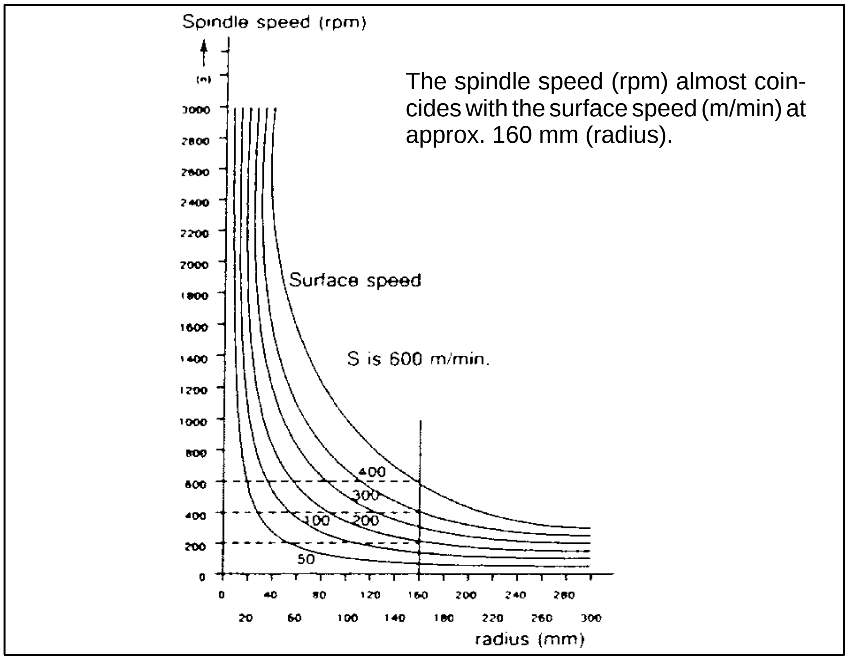


Fig. 9.3 (a) Relation between workpiece radius, spindle speed and surface speed

- **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.

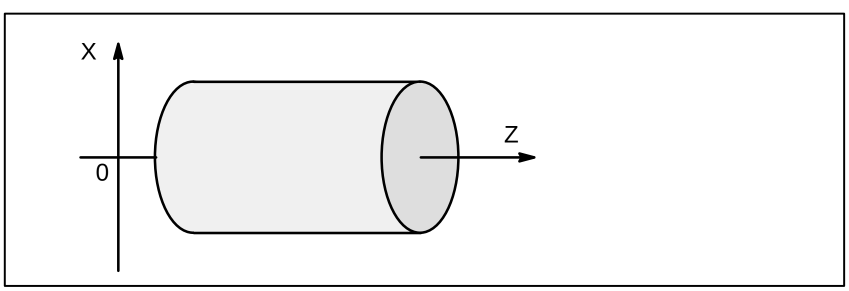
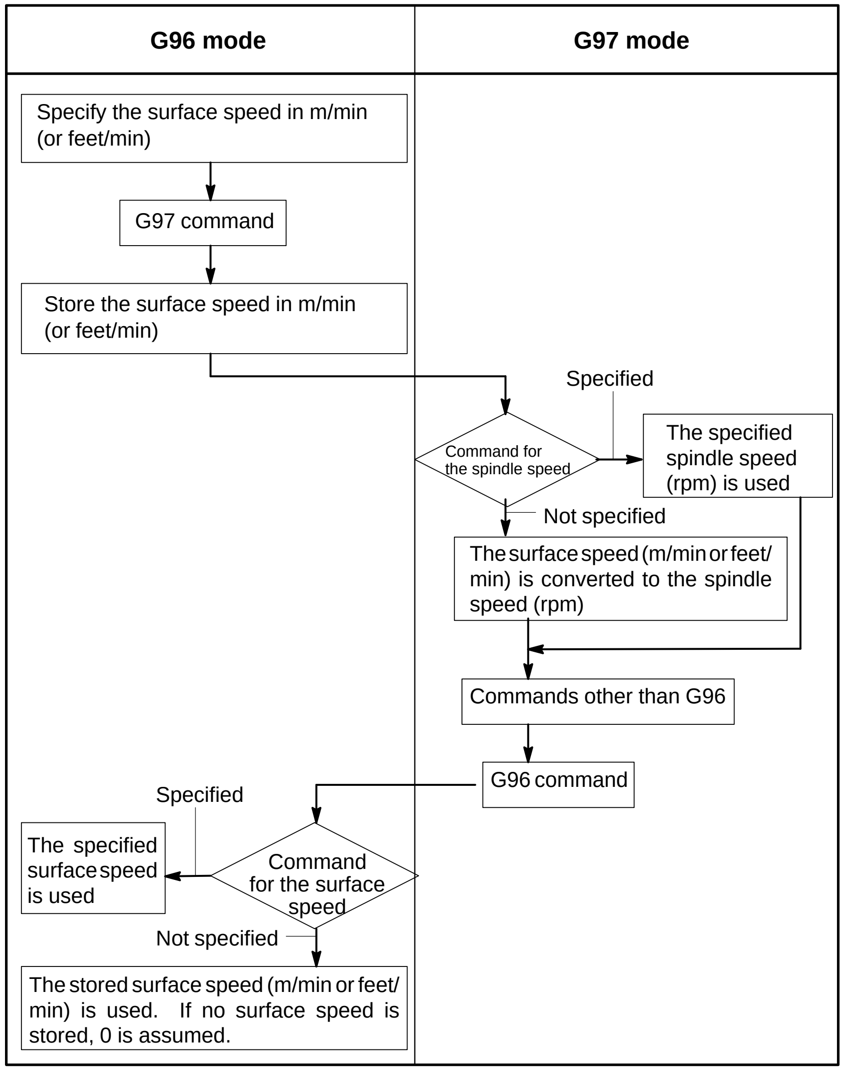


Fig. 9.3 (b) Example of the Workpiece Coordinate System for Constant Surface Speed Control

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



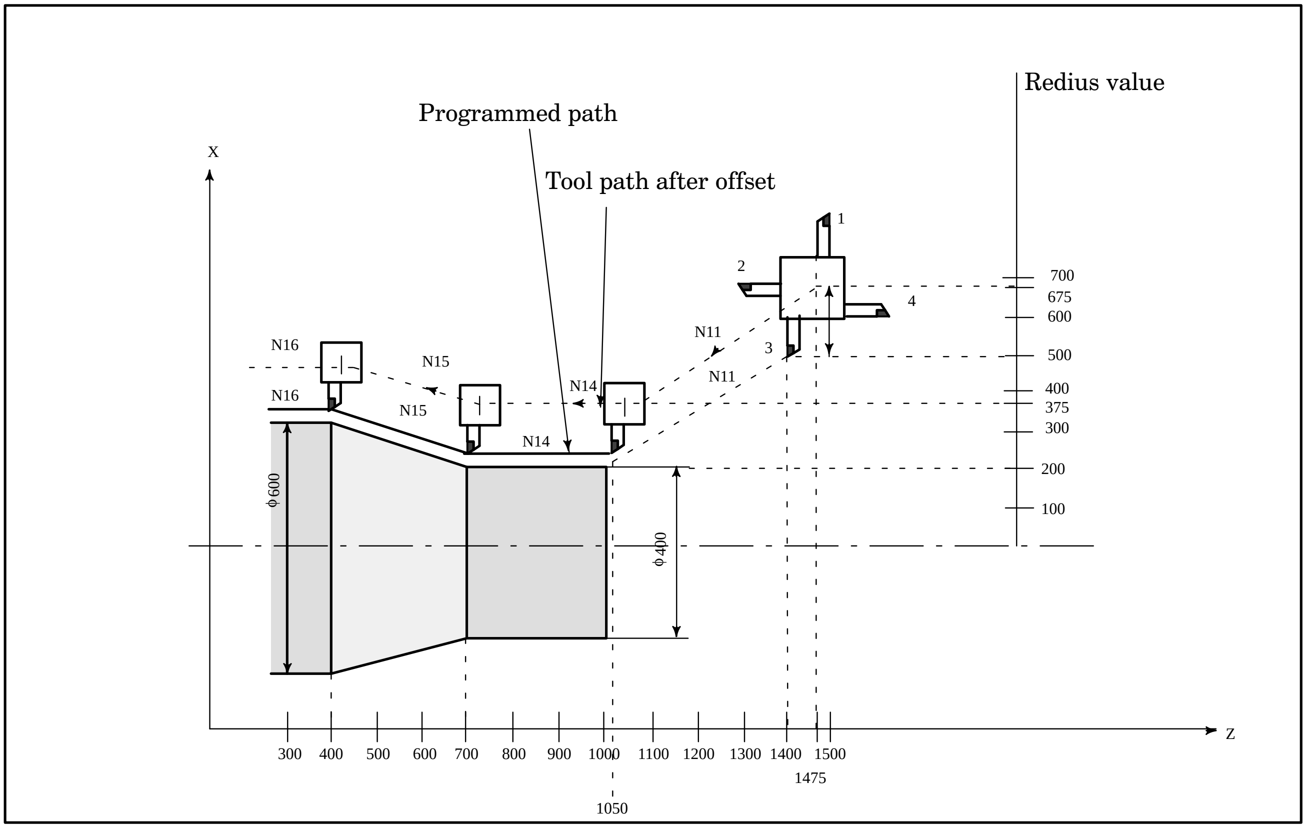
Restrictions

- **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 700.0F1.0 ;
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 TOOL FUNCTION (T FUNCTION)

Tool selection function is available as tool function.

10.1 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 setect 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:

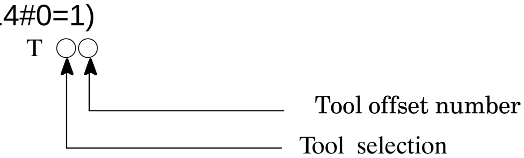
1. Simultaneous execution of the move command and T function commands.
2. Executing T function commands upon completion of move command execution.

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

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.

The above can be used in either of two ways: bit 0 of parameter No. 014 is used to select the method to be used.

1.Last one digit of T–code designates the offset number.
(parameter 014#0=1)



2.Last two digits of T–code designate the offset number.
(parameter 014#0=0)

Explanations

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)
N1G00X100.0Z140.0
N2T0313; .....(Select Tool No. 3 and Offset value No.13)
N3X40.0Z105.0;
Some machines use a 1–digit value to specify tool selection.

11 AUXILIARY FUNCTION

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

11.1 AUXILIARY FUNCTION (M FUNCTION)

When address M followed by 3 digit value 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, and M codes for calling a subprogram (parameters Nos. 240 to 242). 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 019 (M02) 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 12.3** for details .

- **M99 (End of subprogram)**

This code indicates the end of a subprogram.
M99 execution returns control to the main program. See the subprogram **section 12.3** for details.

> **NOTE**
> The block following M00, M01, M02 and M30, is not read into the input buffer register, if present. Similarly, ten M codes which do not buffer can be set by parameters (Nos. 111 and 112). Refer to the machine tool builder’s instruction manual for these M codes.

11.2 MULTIPLE M COMMANDS IN A SINGLE BLOCK

So far, one block has been able to contain only one M code. However, this function allows up to three M codes to be contained in one block.
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. Before this function can be used, bit 7 of parameter No. 065 must be set.

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, or M99 must not be specified together with another M code.
Some M codes other than M00, M01, M02, M30, M98, and M99 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 ; G28G91U0W0 ; : : :	M40M50M60 ; G28G91U0W0 ; : : : : :

11.3 OUTPUTTING SIGNAL NEAR END POINT

This function outputs the signal DEN2 when the tool approaches to the end point of a rapid traverse (G00) or linear interpolation (G01) block where a miscellaneous function (M–code) and a tolerable distance are commanded. As the miscellaneous function can be triggered near the end point by using the DEN2, it shortens the execution time.

Format

G00
G01 **Motion Command , Q____ M____ ;**

Motion Command: Ordinary Motion Command
,Q______: Tolerable Distance
M_______: Ordinary miscellaneous function

Explanations

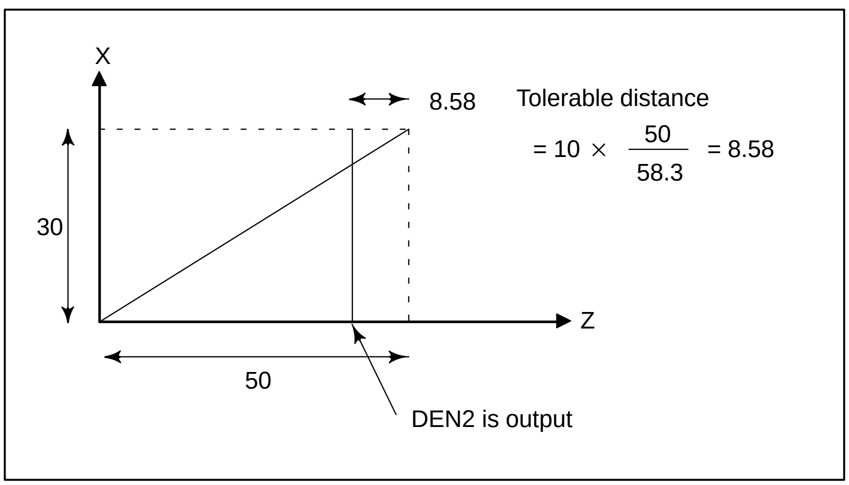
The miscellaneous function (M–code) and the tolerable distance ”,Q” are commanded on the rapid traverse (G00) or linear interpolation (G01) block. The command unit of ”,Q” is the input increment. The decimal point is permitted in ”,Q”. When ”the distance to go” of the longest motion axis decreases to the tolerable distance defined by ”,Q”, the signal DEN2 which means the nearness of the end point is output. The tolerable distance of the longest motion is defined by the following formula.

Tolerable distance of longest motion =

$$\text{Command of ”,Q”} \times \frac{\text{distance of longest motion}}{\text{Block distance}}$$

Program Example

G01 U60.0 W50.0 ,Q10.0 M123 ;



NOTE
1. The special M–codes which are managed internally such as M98 and M99, are not used for M–codes of this function.
2. This function is not used during the multiple turning cycles.
3. This function can not be commanded by MDI operation A.
4. The DEN2 keeps on till the completion signal (FIN) fo the miscellaneous function turns on. When the FIN inputs before reaching the end, the distribution end signal (DEN) is not output at the end.
5. When the function is used on the rapid traverse block of the simultaneous axes motion, there may be the error between the tolerance distance and the actual point where the DEN2 is output.

12 PROGRAM CONFIGURATION

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.

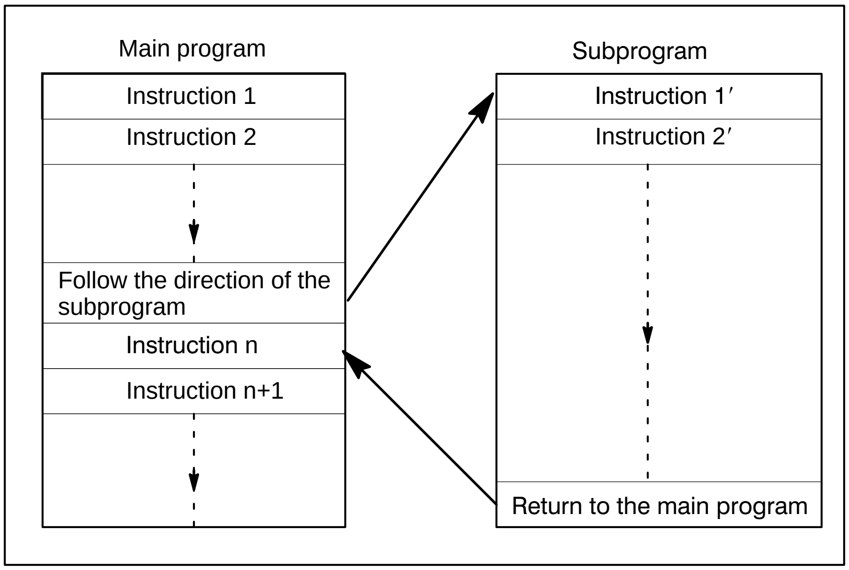


Fig. 12 (a) Main program and Subprogram

The CNC memory can hold up to 63 main programs and subprograms. A main program can be selected from the stored main programs to operate the machine.

Program components

A program consists of the following components:

Table 12 (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

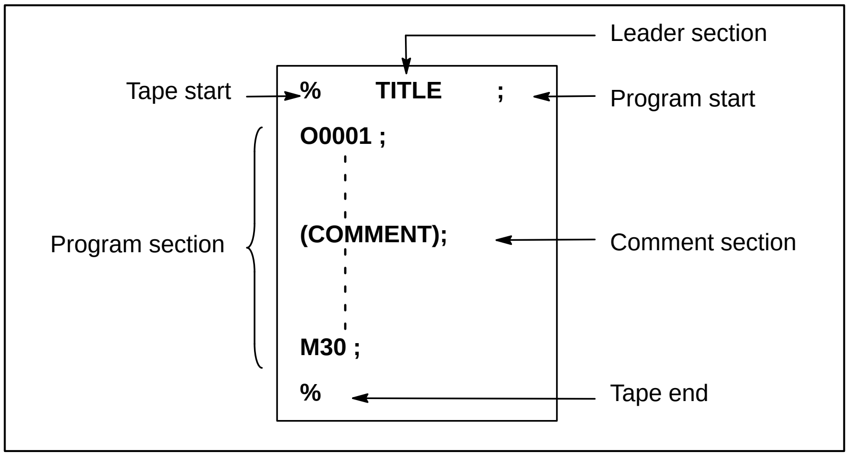


Fig. 12 (b) Program configuration

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
Program number	O0001 ;
Block 1	N1 G91 G00 X120.0 Y80.0 ;
Block 2	N2 G43 Z–32.0 H01 ;
:	:
Block n	Nn M2 ;
Program end	M30 ;

A block contains information necessary for machining, such as a move command or coolant on/off command. Specifying a slash (/) at the start of a block disables the execution of some blocks (see ”optional block skip” in Section 12.2).

12.1 PROGRAM COMPONENTS OTHER THAN PROGRAM SECTIONS

This section describes program components other than program sections. See Section 12.2 for a program section.

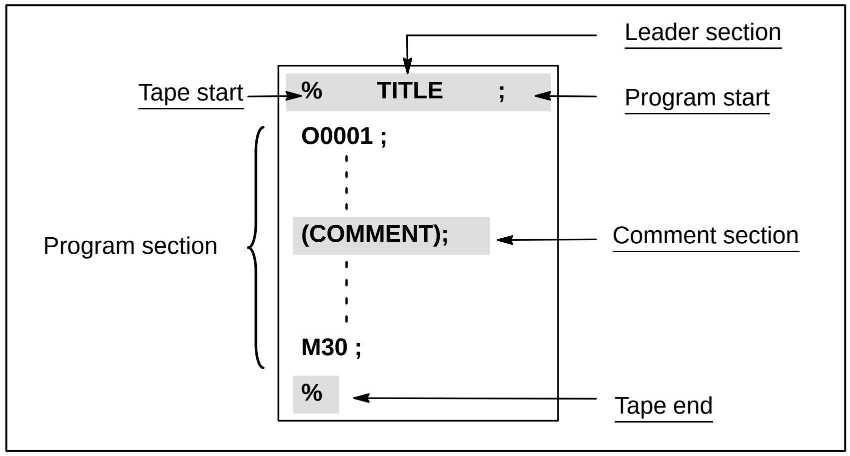


Fig. 12.1 (a) Program configuration

Explanations

- **Tape start**

The tape start indicates the start of a file that contains NC programs. 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 12.1 (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.

Table 12.1 (b) Code of a program start

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

NOTE

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. The user can enter a header, comments, directions to the operator, etc. in a comment section. There is no limit on the length of a comment section.

Table 12.1 (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 command tape 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 6 are ignored, and thus are not read into memory. When data in memory is punched out on paper tape with the punch function, the comment sections are also punched out.

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 punched out or displayed.

During memory operation in memory command mode, all comment sections are ignored.

The TV check function can be used for a comment section by setting parameter (bit 6 of No. 018).

CAUTION

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.

NOTE

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. 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.

Table 12.1 (d) Code of a tape end

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

12.2 PROGRAM SECTION CONFIGURATION

This section describes elements of a program section. See Section 12.1 for program components other than program sections.

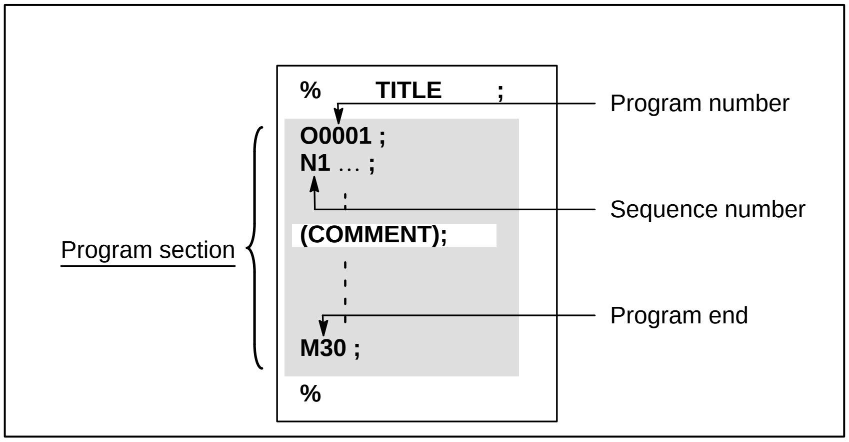


Fig. 12.2 (a) Program configuration

- **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. 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.).

NOTE
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 12.2 (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 9999) 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. 12.2 (b) Sequence number and block (example)

> **NOTE**
> 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 horizontally. 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 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. 11.5.1 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 12.2 (b) indicates the usable addresses and their meanings.
The same address may have different meanings, depending on the preparatory function specification.

Table 12.2 (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

NOTE
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. 12.2 (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 feedrate of up to 100 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 12.2 (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	1–100000mm/min	0.01–4000.00 inch/min
	Increment system IS–C		1–12000mm/min	0.01–480.00 inch/min
Feed per revolution		F	0.0001–500.0000 mm/rev	0.000001–9.999999 inch/rev
Spindle speed function		S	0–20000	0–20000
Tool function		T	0–9999	0–9999
Auxiliary function		M	0–999	0–999
		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

- **Optional block skip**

When a slash is specified at the head of a block, and optional block skip switch on the machine operator panel is set to on, the information contained in the block is ignored in tape operation or memory operation. When optional block skip switch is set to off, the information contained in the block for which / is specified is valid. This means that the operator can determine whether to skip the block containing /.
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.

WARNING

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 ***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.

NOTE

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.

- **Program end**

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

Table 12.2 (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.

WARNING

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.)

12.3 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 Z100.0 M99 ;**

- **Subprogram call**

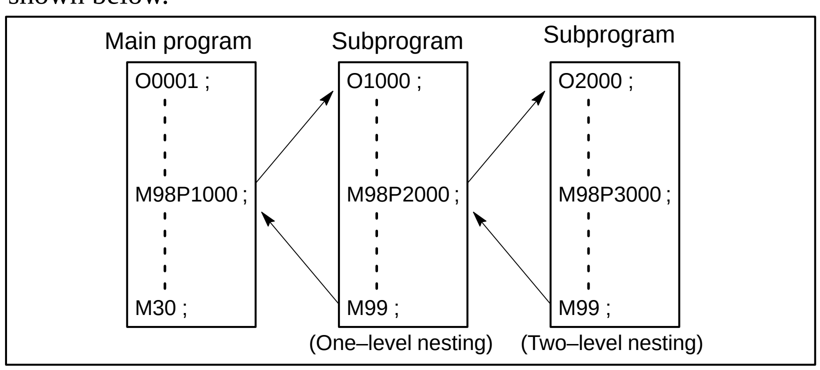
```
M98 P ○○○○ ○○○○ ;
       ↑     ↑
```

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 two levels as shown below.



A single call command can repeatedly call a subprogram up to 999 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.

NOTE

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 ;**

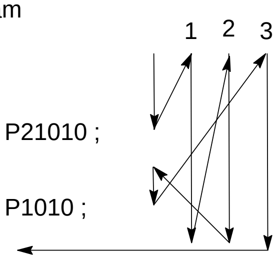
This command specifies ”Call the subprogram (number 1002) five times in succession.” A subprogram call command (M98P_) can be specified in the same block as a move command.

☆ **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	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 … ;                   O1010 … ;
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.

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

- **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 9.3 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 ;            Optional block skip
  N1050 M99 P1020 ;      ON
```

13 FUNCTIONS TO SIMPLIFY PROGRAMMING

This chapter explains the following items:

NOTE

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

13.1 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).

13.1.1 Outer Diameter/Internal Diameter Cutting Cycle (G90)

- **Straight cutting cycle**

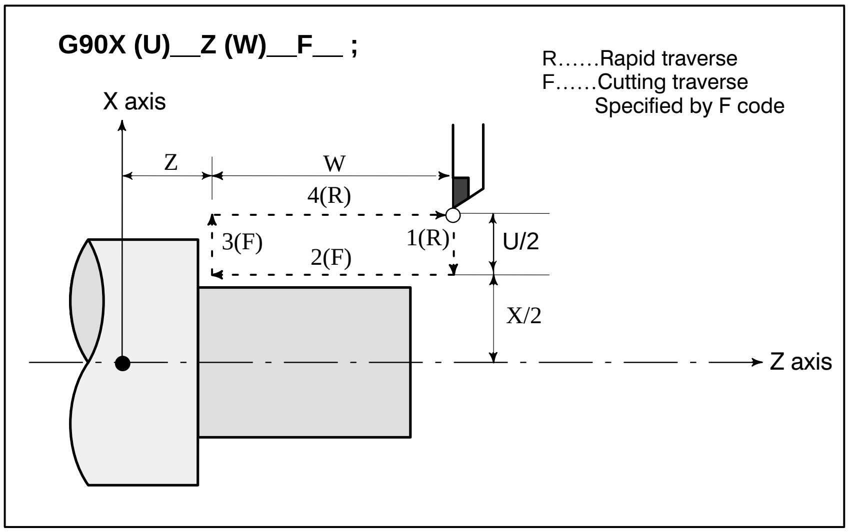


Fig. 13.1.1 (a) Straight Cutting Cycle

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 13.1.1 (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**

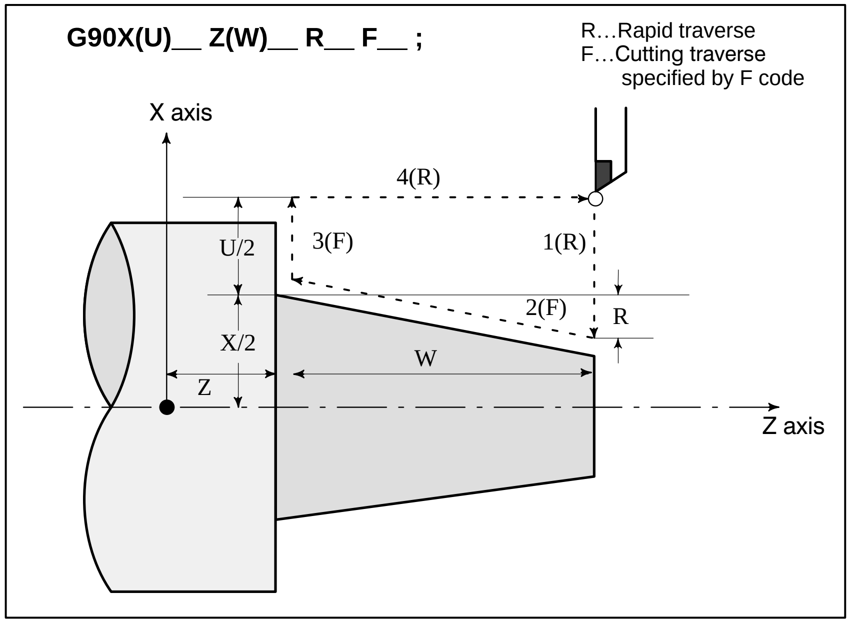
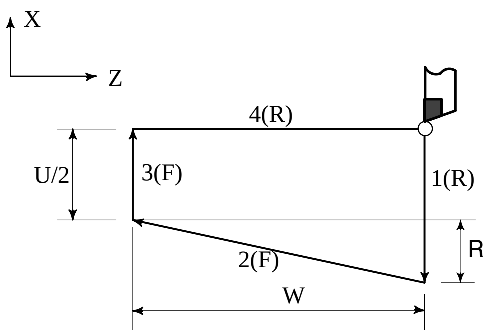


Fig. 13.1.1 (b) Taper Cutting Cycle

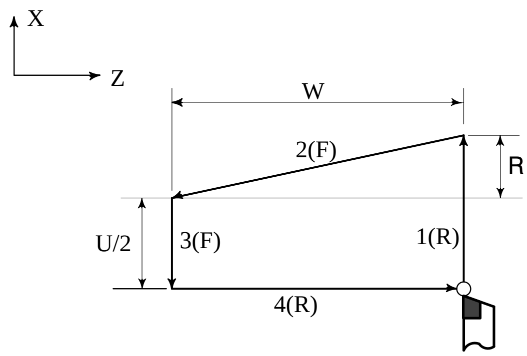
- **Signs of numbers specified in the taper cutting cycle**

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

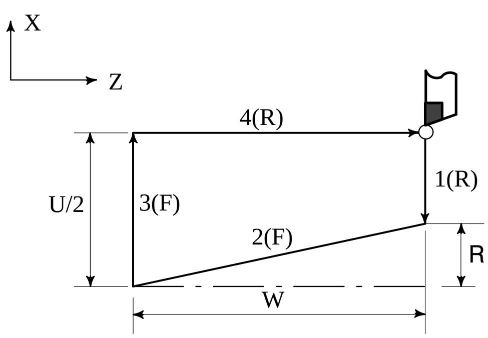
1. $U < 0, W < 0, R < 0$



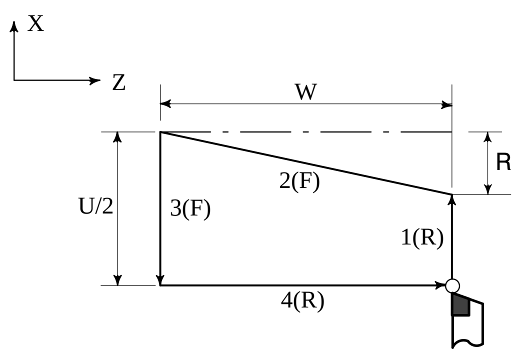
2. $U > 0, W < 0, R > 0$



3. $U < 0, W < 0, R > 0$ at $|R| \leqq \left|\frac{U}{2}\right|$



4. $U > 0, W < 0, R < 0$ at $|R| \leqq \left|\frac{U}{2}\right|$



13.1.2 Thread Cutting Cycle (G92)

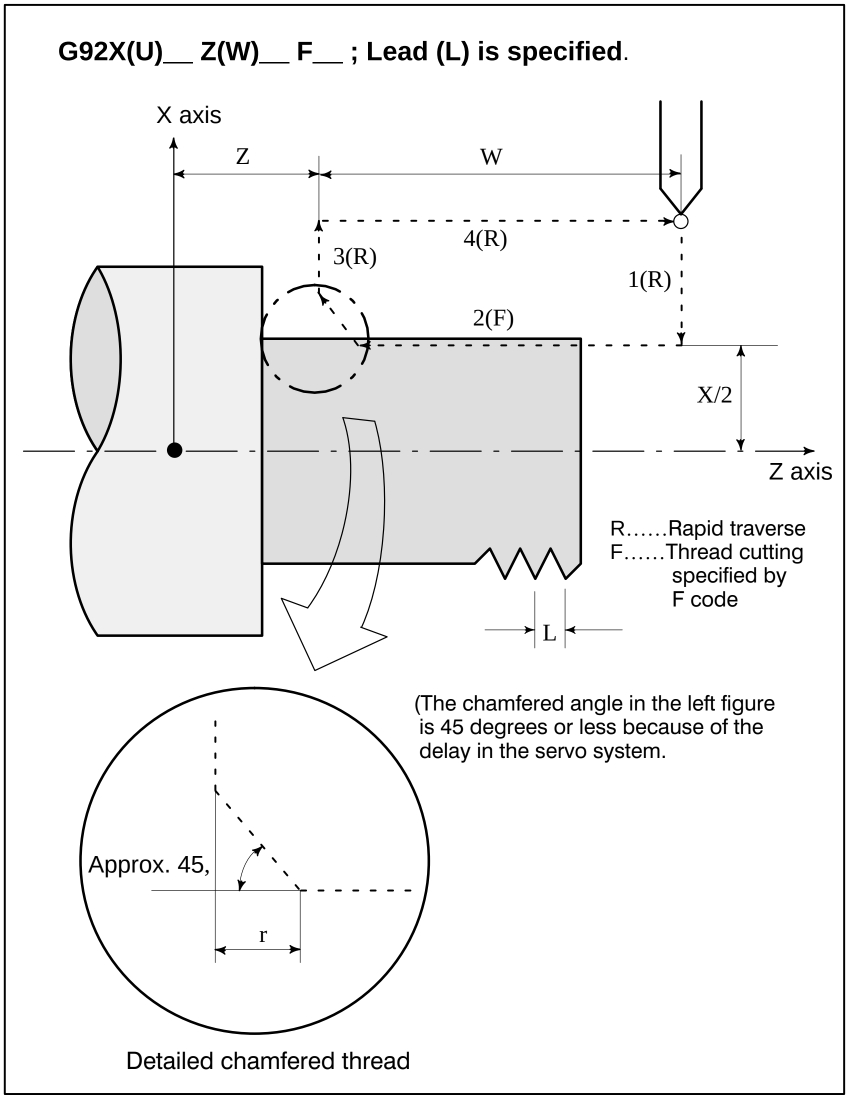


Fig. 13.1.2 (a) Straight Thread Cutting

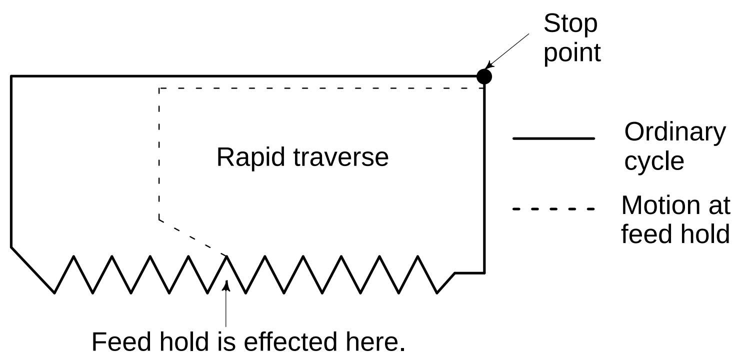
In incremental programming, the sign of numbers following addresses U and W depends on the direction of paths 1 and 2. That is , if the direction of path 1 is the negative along the X axis, the value of U is negative.
The range of thread leads, limitation of spindle speed, etc. are the same as in G32 (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.

WARNING

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.

CAUTION

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**

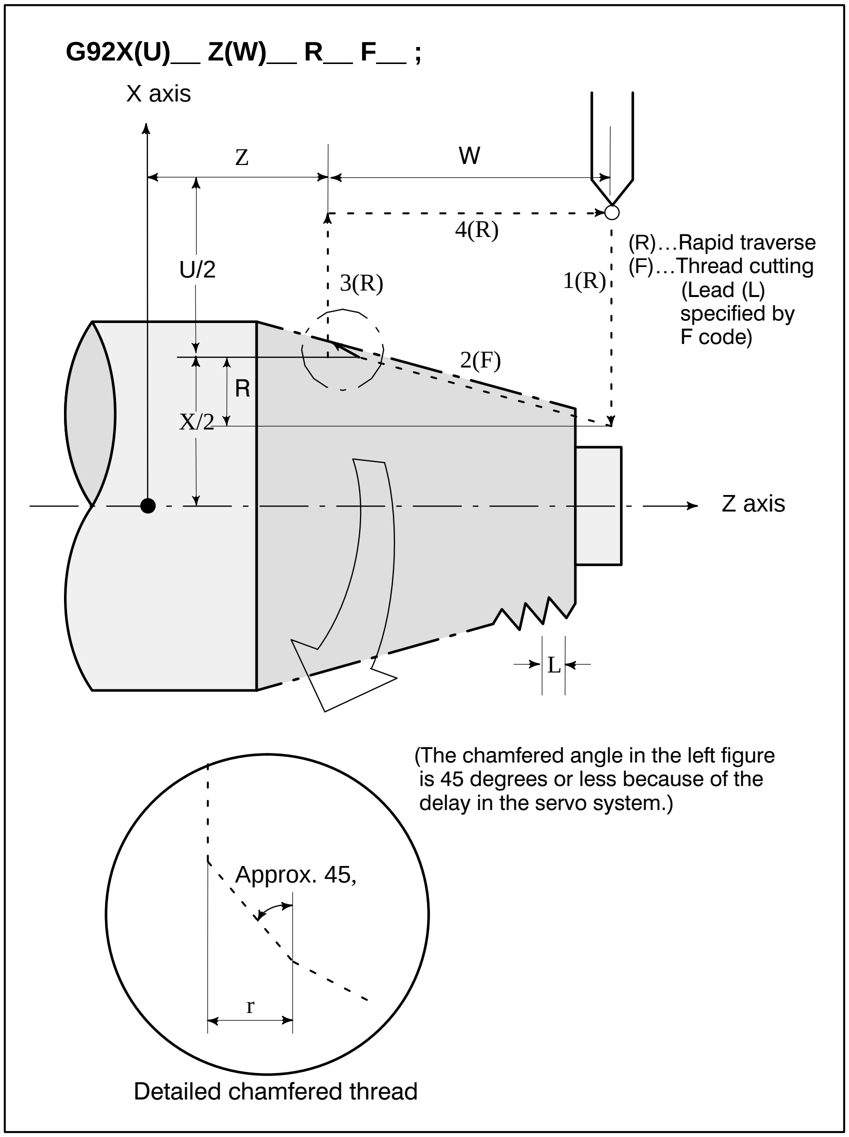


Fig. 13.1.2 (b) Taper thread cutting cycle

13.1.3 End Face Turning Cycle (G94)

- **Face cutting cycle**

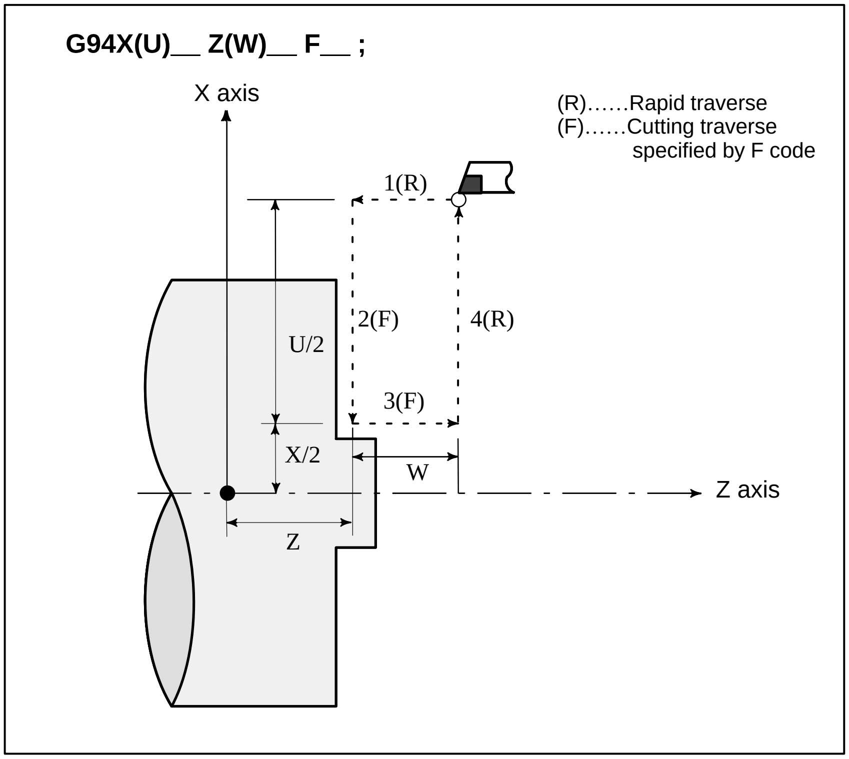


Fig. 13.1.3 (a) Face Cutting Cycle

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**

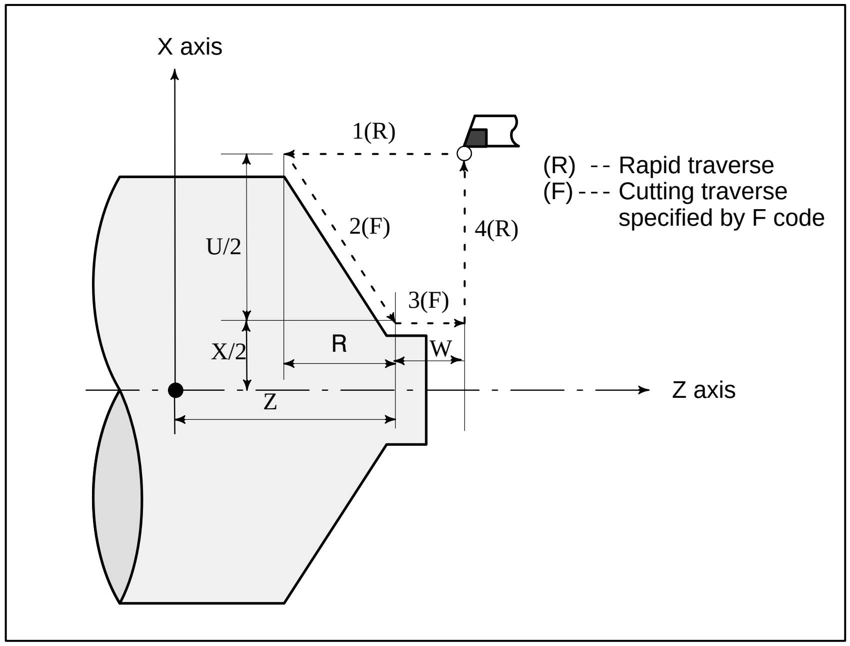
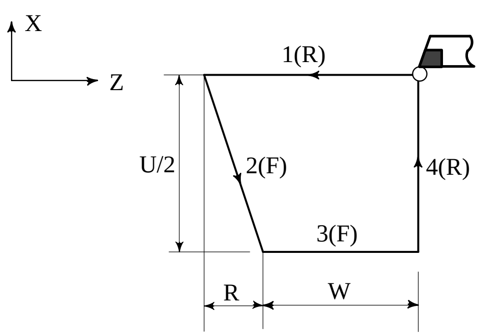
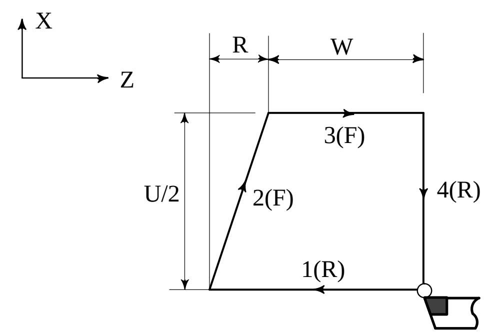
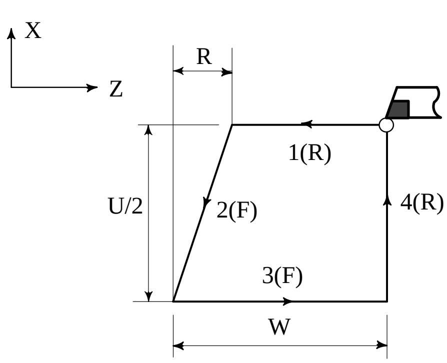
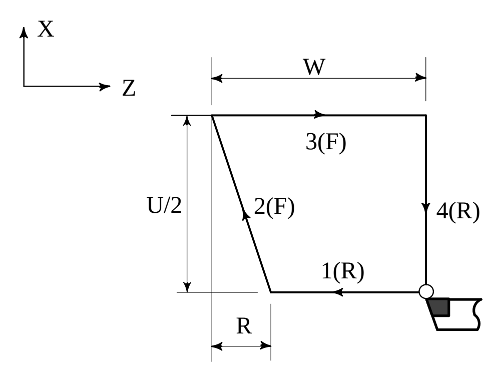


Fig. 13.1.3 (b)

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

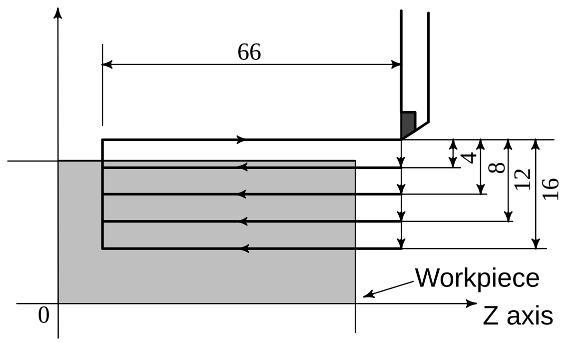
In incremental programming, the relationship between the signs of the numbers following address U, W, and R, and the tool paths are as follows:

<table>
<tr><td>1. U < 0, W < 0, R < 0</td><td>2. U > 0, W < 0, R < 0</td></tr>
<tr><td>
</td><td>
</td></tr>
<tr><td>3. U < 0, W < 0, R > 0
at | R | ≦ | W |</td><td>4. U > 0, W < 0, R<0
at | R | ≦ | W |</td></tr>
<tr><td>
</td><td>
</td></tr>
</table>

NOTE

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.

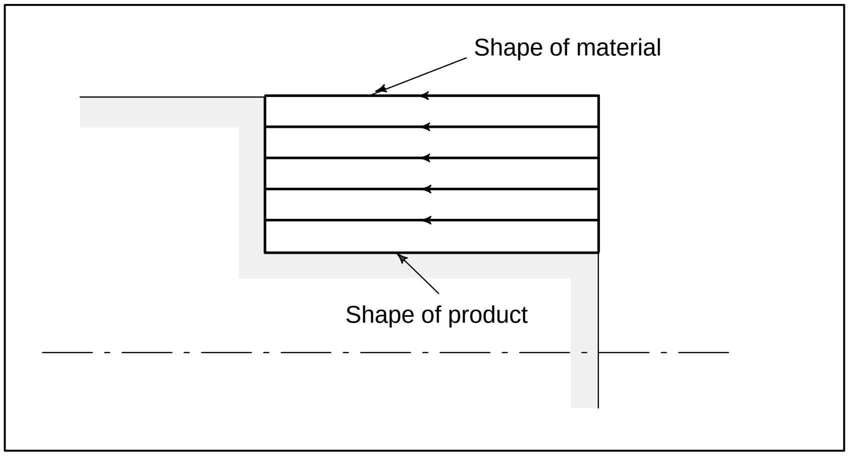
(Example)

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

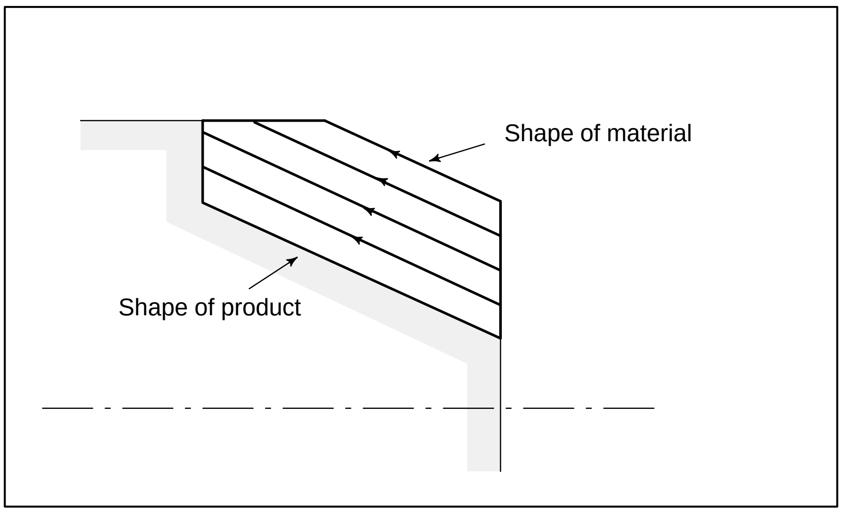
13.1.4 How to Use Canned Cycles (G90, G92, G94)

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

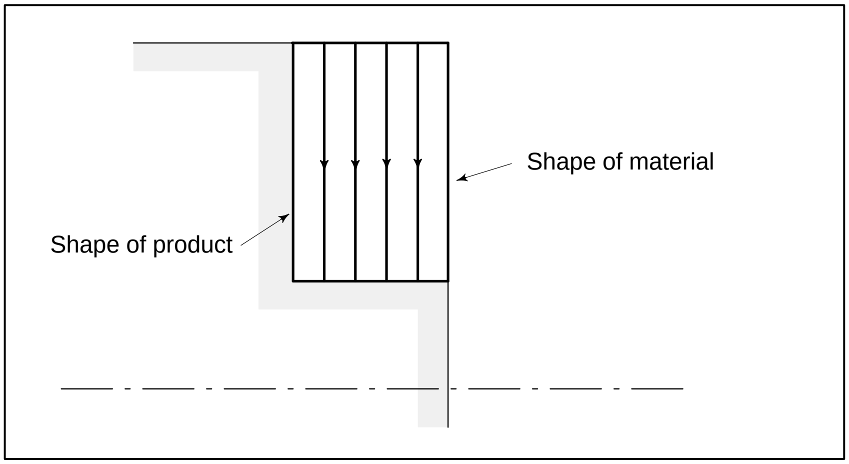
- **Straight cutting cycle (G90)**



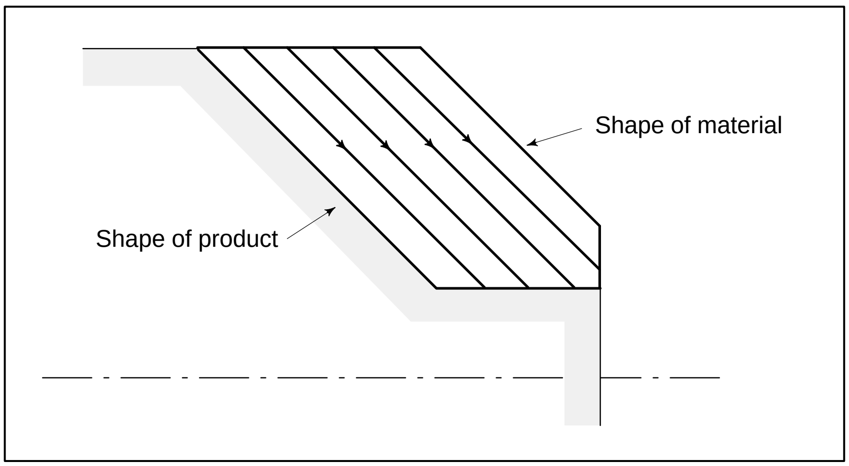
- **Taper cutting cycle (G90)**



- **Face cutting cycle (G94)**



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



13.2 MULTIPLE REPETITIVE CYCLE (G70 TO G76)

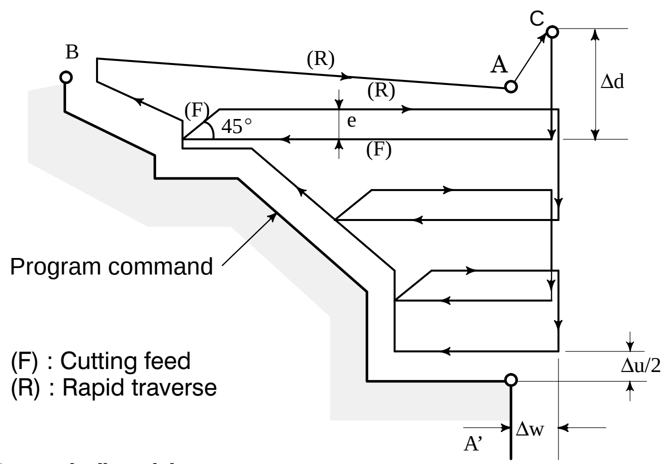
This option canned cycles to make CNC programming easy. For instance, the data of the finish work shape describes the tool path for rough machining. And also, a canned cycles for the thread cutting is available.

13.2.1 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____     The move command between A and B is
            T____     specified in the blocks from sequence
                      number ns to nf.
N (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. 717), 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. 718), 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.

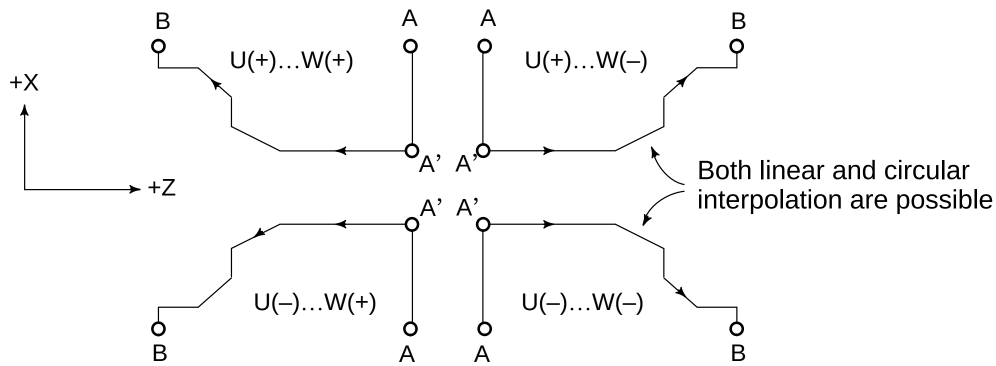
f,s,t : Any F , S, or T function contained in blocks ns to nf in the cycle is ignored, and the F, S, or T function in this G71 block is effective.

Fig. 13.2.1 (a) Cutting Path in Stock Removal in Turning (Type I)

NOTE

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:



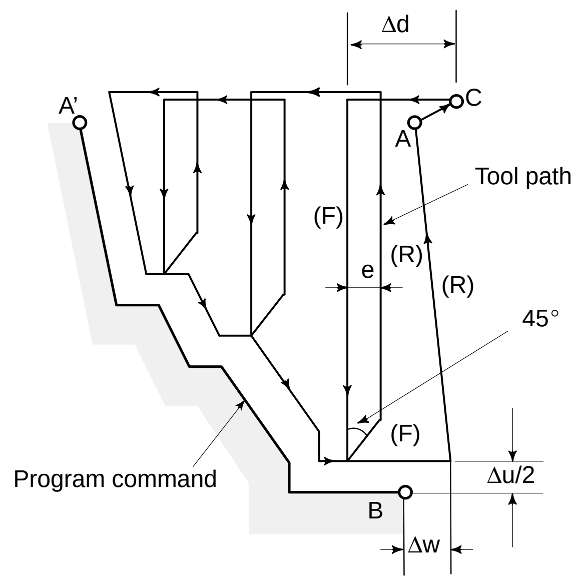
The tool path between A and A’ is specified in the block with sequence number ”ns” including G00 or G01, and in this block, a move command in the Z axis cannot be specified. The tool path between A’ and B must be steadily increasing or decreasing pattern in both X and Z axis. When the tool path between 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”.

4 Nose radius compensation is disabled during cycle operation. When the imaginary tool nose number is 0 or 9, however, a nose radius compensation value is added to U and W.

13.2.2 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(Δd) R(e) ;
G72 P(ns) Q(nf) U(Δu) W(Δw) F(f) S(s) T(t) ;

The meanings of Δd, e, ns, nf, Δu,Δw, f, s, and t are the same as those in G71.

Fig. 13.2.2 (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 :

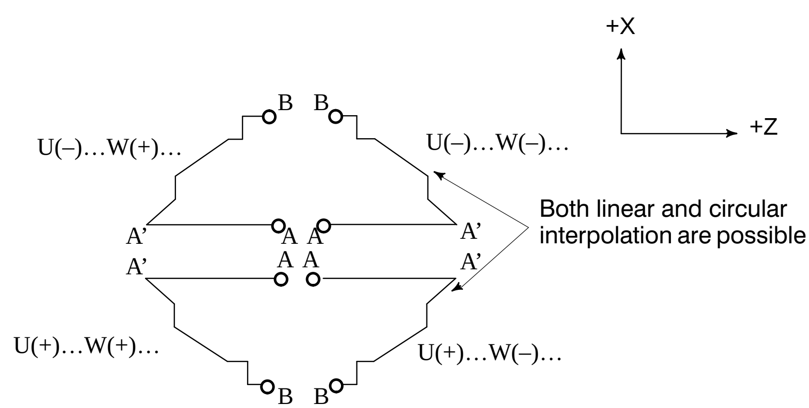
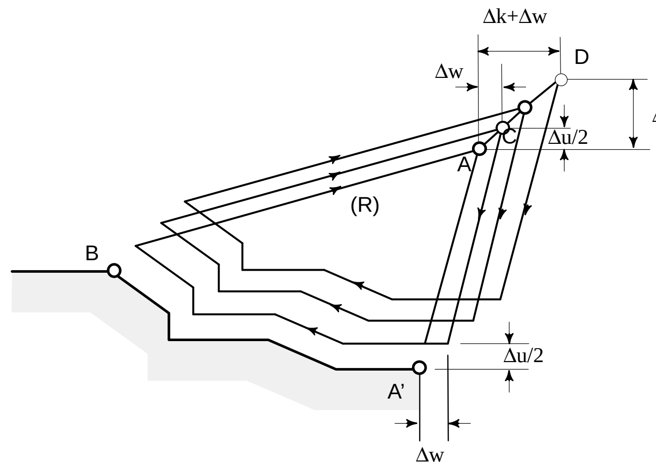


Fig. 13.2.2 (b) Signs of Numbers Specified with U and W in Stock Removal in Facing

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.

13.2.3 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____      The move command between A and B is
            T____      specified in the blocks from sequence
N (nf)..........;      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. 719, 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. 720, 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. 721, 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. 13.2.3 (a) Cutting path in Pattern Repeating

NOTE

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.
3. Nose radius compensation is disabled during cycle operation. When the imaginary tool nose number is 0 or 9, however, a nose radius compensation value is added to U and W.

13.2.4 Finishing Cycle (G70)

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

Format

G70P (ns) Q (nf) ; U (Δu) W (Δw) ;

(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) :Cutting allowance in X direction (diameter/radius specification)

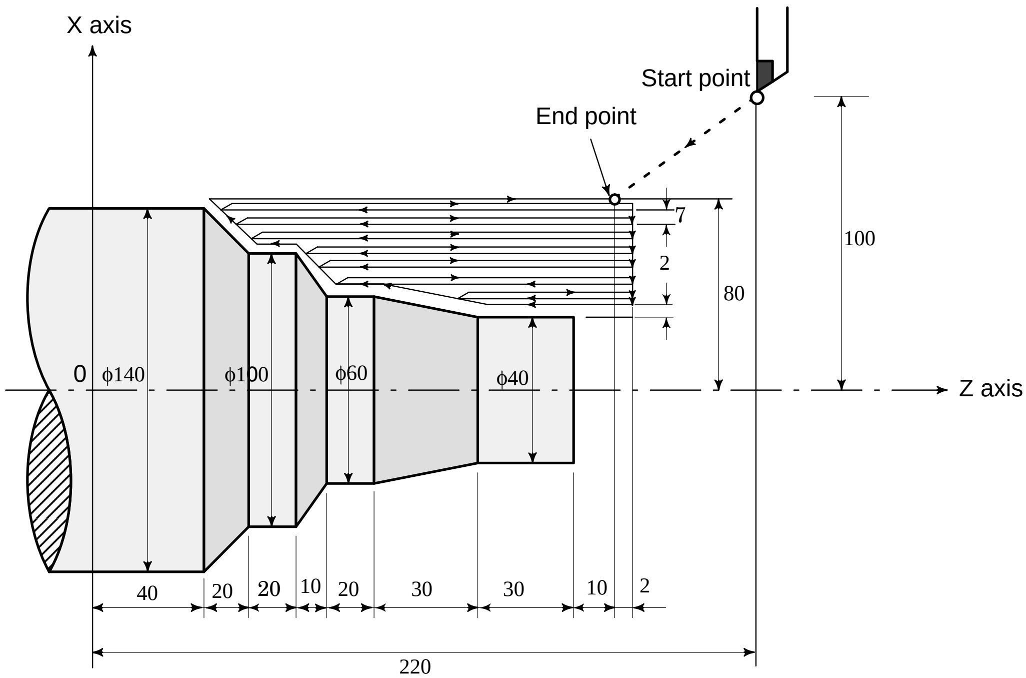
(Δw) :Cutting allowance in Z direction

NOTE

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.
4. Nose radius compensation is enabled during a finishing cycle.
5. Specify cutting allowances U and W when performing finish machining that involves cutting a finishing allowance in several steps.

Examples

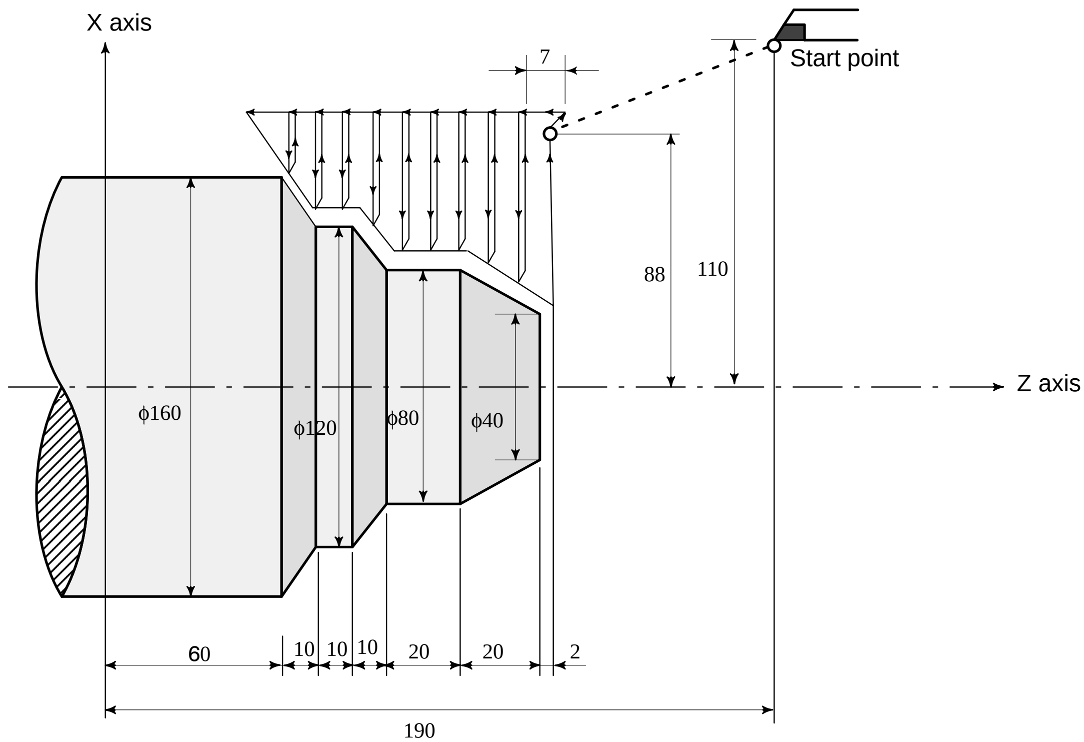
Stock Removal in Turning (G71) (Type I)



(Diameter designation, metric input)

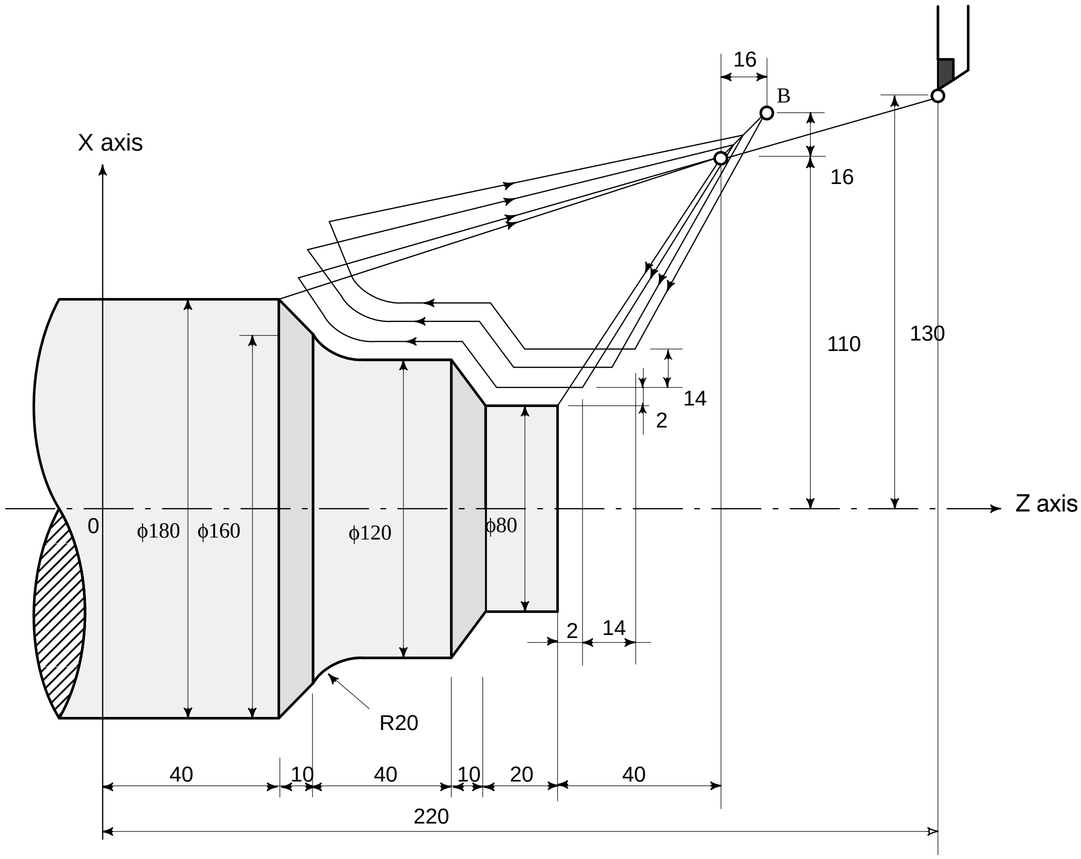
```
N010    G50 X200.0 Z220.0 ;
N011    G00 X160.0 Z180.0 ;
N012    G71 U7.0 R1.0 ;
N013    G71 P014 Q020 U4.0 W2.0 F0.3 S550 ;
N014    G00 X40.0 F0.15 S700 ;
N015    G01 W–40.0;
N016            X60.0 W–30.0 ;
N017            W–20.0 ;
N018            X100.0 W–10.0 ;
N019            W–20.0 ;
N020    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

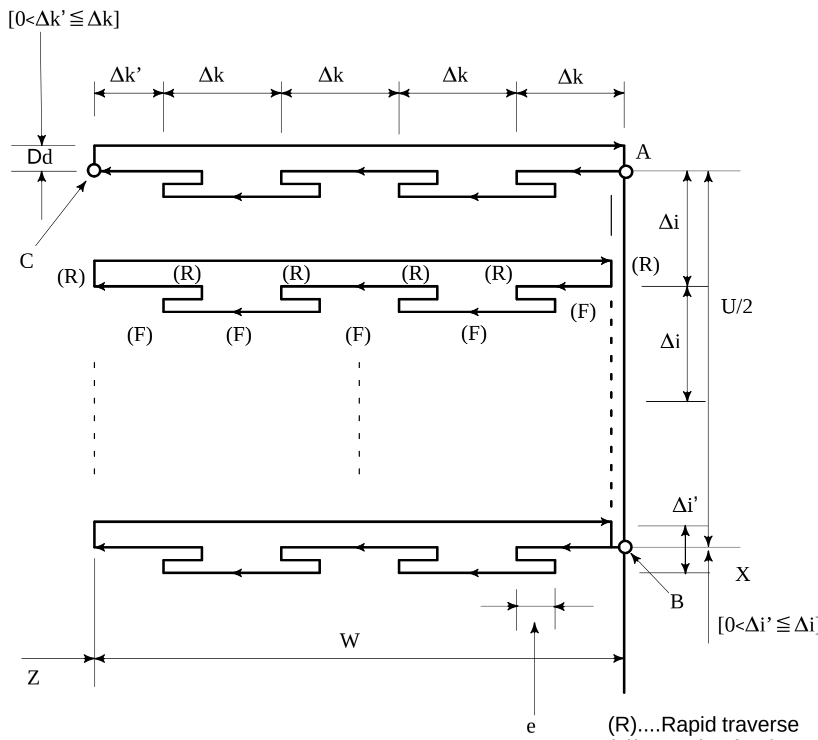


(Diameter designation, metric input)

```
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 ;
N016    X120.0 W–10.0 ;
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 ;
```

13.2.5 End Face Peck Drilling Cycle (G74)

The following program generates the cutting path shown in Fig. 13.2.5 (a). Chip breaking is possible in this cycle as shown below. If X (3) 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. 722, 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 and radius amount (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

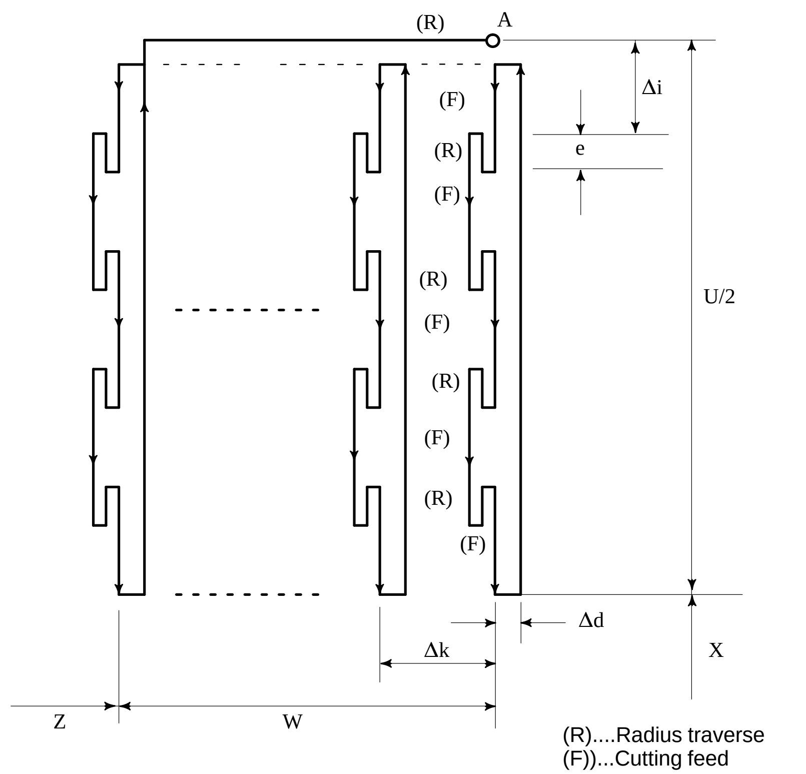
Fig. 13.2.5 (a) Cutting Path in End Face Peek Drilling Cycle

NOTE

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.

13.2.6 Outer Ddiameter / 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. 13.2.6 (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.

13.2.7 Multiple Thread Cutting Cycle (G76)

The thread cutting cycle as shown in Fig.13.2.7 (a) is programmed by the G76 command.

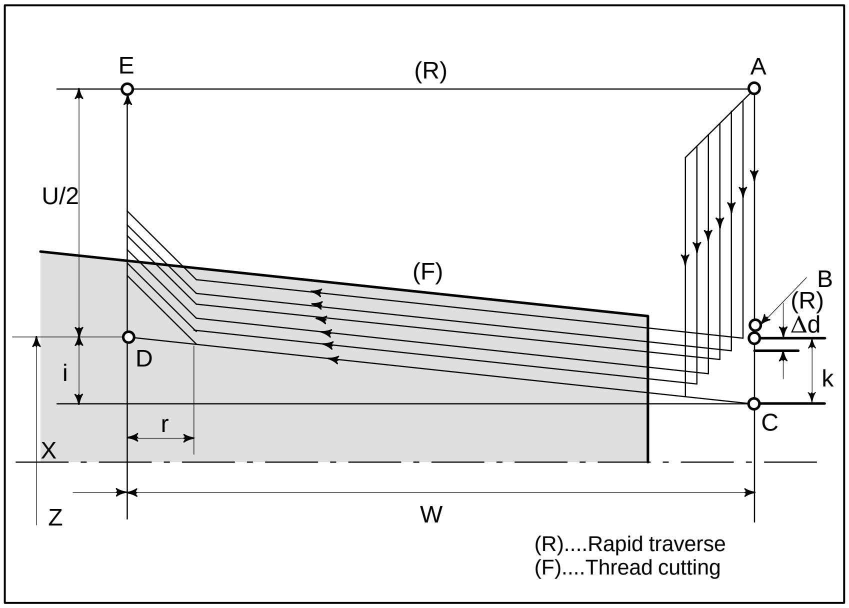
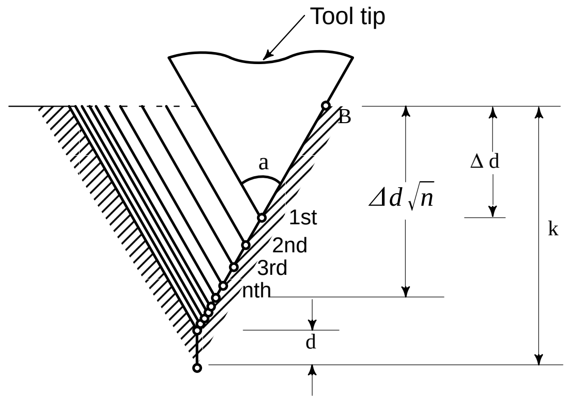


Fig. 13.2.7 (a) Cutting Path in Multiple thread cutting cycle



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. 723, 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. 109, 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. 724, 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. 725, and the parameter is changed by the program command.

d : Finishing allowance (Command with radius amount)
This designation is modal and is not changed until the other value is designated. Also this value can be specified by parameter No. 726, and the parameter is changed by the program command.

i : Taper value command with radius amount
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 (Command with radins amount)

L : Lead of thread (same as G32).

Fig. 13.2.7 (b) Detail of cutting

- **Thread cutting cycle retract**

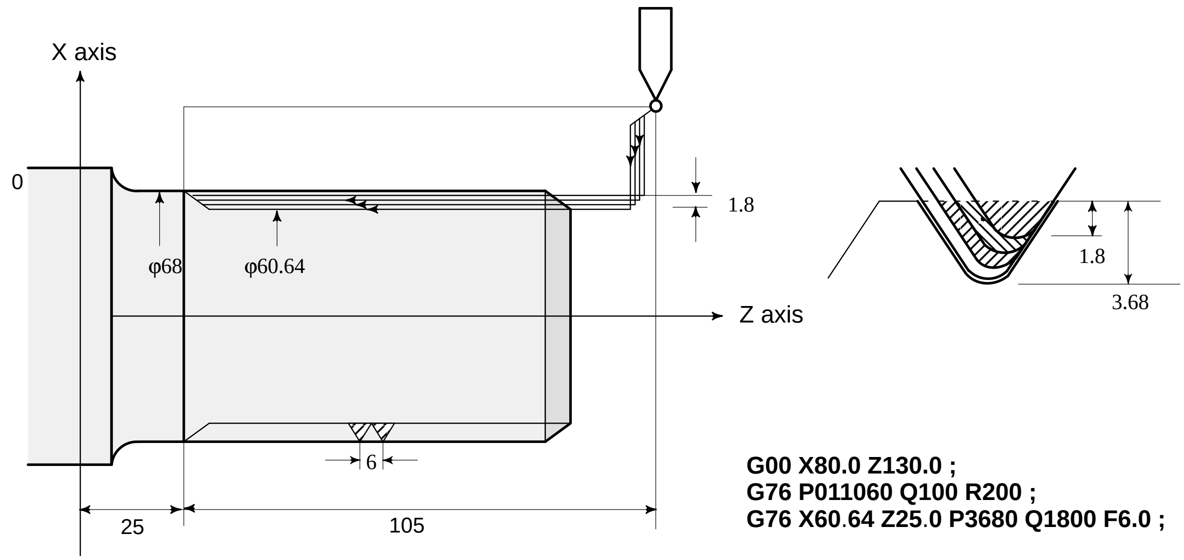
When feed hold is applied during threading in the multiple thread cutting cycle (G76), the tool quickly retracts in the same way as in chamfering performed at the end of the thread cutting cycle. The tool goes back to the start point of the cycle. When cycle start is triggered, the multiple thread cutting 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 13.1.2.

NOTE

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 Δd for the first path, and $\Delta dn=\Delta d\sqrt{n}$ 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.)
   - i : minus (determined by the direction of the tool path AC.)
   - k : plus (always)
   - d : 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 Δdn) 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)



G00 X80.0 Z130.0 ;
G76 P011060 Q100 R200 ;
G76 X60.64 Z25.0 P3680 Q1800 F6.0 ;

13.2.8 Notes on Multiple Repetitive Cycle (G70 to 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 (G70 to G76) 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.

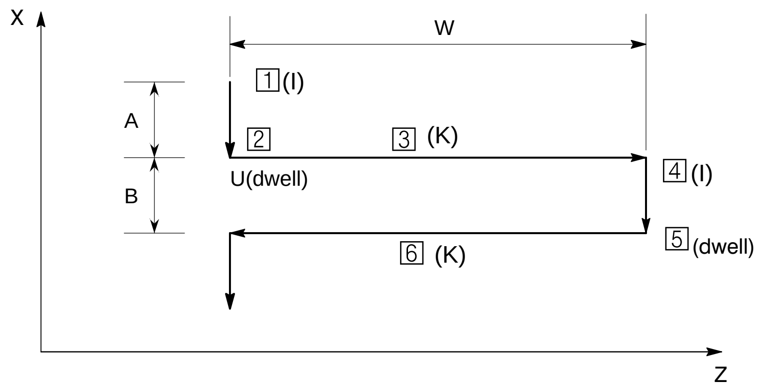
13.3 CANNED GRINDING CYCLE (FOR 0–GCD ONLY)

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.

13.3.1 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

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.

13.3.2 Traverse Direct Fixed–Dimension Grinding Cycle (G72)

Format

G72 P_ A_ B_ W_ U_ I_ K_ H_ ;

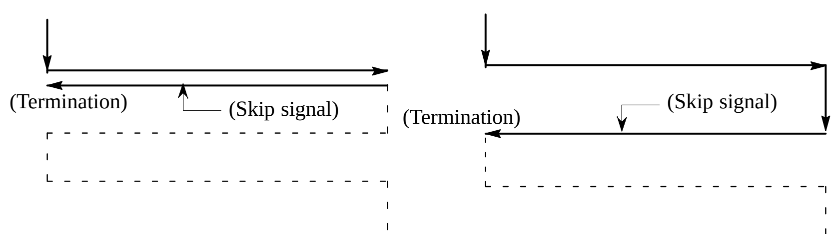
P : Gauge number (1 to 4)
A : First depth of cut
B : Second depth of cut
W: Grinding range
U : Dwell time   Maximum specification time : 99999.999seconds
I : Feedrate of A and B
K : Feedrate of W
H : Number of repetitions   Setting value : 1 to 9999

Explanations

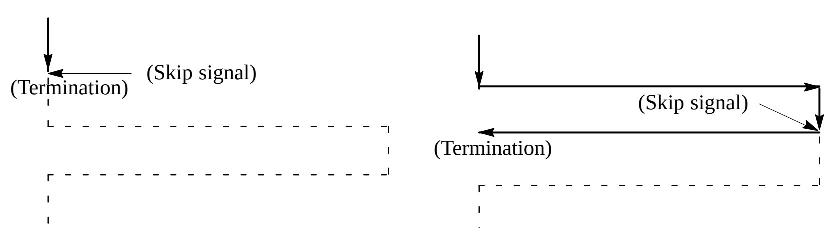
When the multistage skip operation is used, a gauge number can be specified. The method of gauge number specification is the same as the method of multistage skip function. When the multistage skip operation is not used, the conventional skip signal is valid.
The same specifications as G71 apply except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

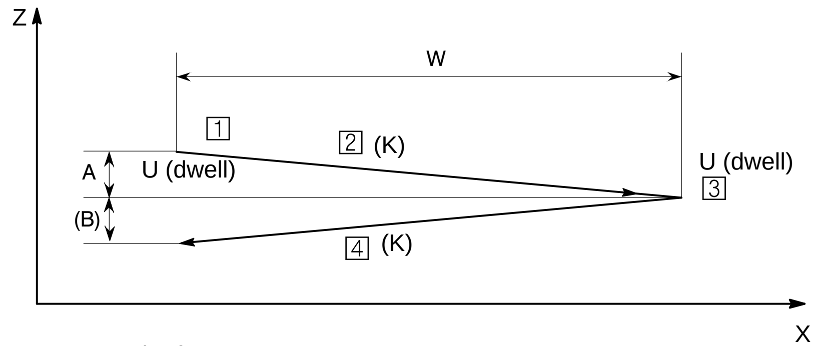


3. The skip signal is valid during dwell, without being affected by parameters DS1 to DS8 (No. 035#0 to #3). Dwell is immediately stopped for return to the Z axis coordinate where the cycle started.

13.3.3 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 : 1 to 9999

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.

13.3.4 Oscillation Direct Fixed–Dimension Grinding Cycle

Format

G74 P_ A_ (B_) W_ U_ K_ H_ ;

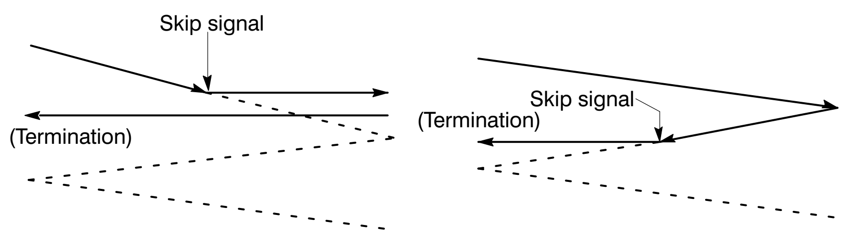
P : Gauge number (1 to 4)
A : Depth of cut
B : Depth of cut
W: Grinding range
U : Dwell time
K : Feedrate of W
H : Number of repetitions Setting value : 1 to 9999

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.

NOTE
1 The data items A, B, W, U, 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.

13.4 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**

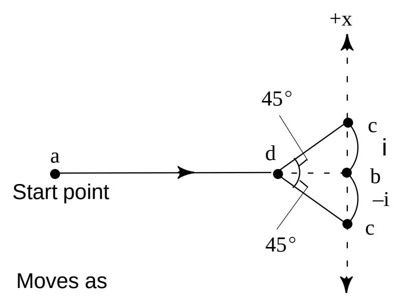
Format	Tool movement
G01Z (W) I (C) ±i ; Specifies movement to point b with an absolute or incremental command in the figure on the right. Whether to use address I or C to specify chamfering is set using bit 4 of parameter No. 029.	 Moves as a→d→c (For –X movement, –i)

Fig. 13.4 (a) Chamfering (Z→X)

- **Chamfering X → Z**

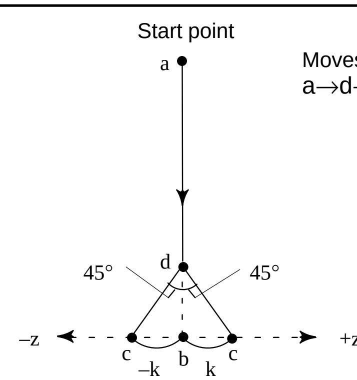
Format	Tool movement
G01X (U) _K (C) ±k ; Specifies movement to point b with an absolute or incremental command in the figure on the right. Whether to use address K or C to specify chamfering is set using bit 4 of parameter No. 029.	 (For –Z movement, –k)

Fig. 13.4 (b) Chamfering (X→Z)

- **Corner R Z → X**

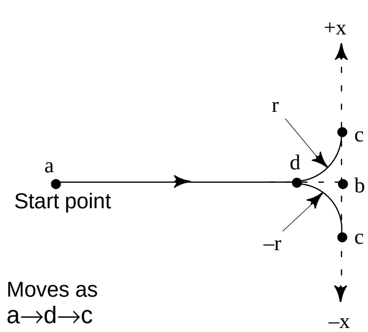
Format	Tool movement
G01Z (W)_R ±r ; Specifies movement to point b with an absolute or incremental command in the figure on the right.	 Moves as a→d→c (For –X movement, –r)

Fig. 13.4 (c) Corner R (Z→X)

- **Corner R**
  X → Z

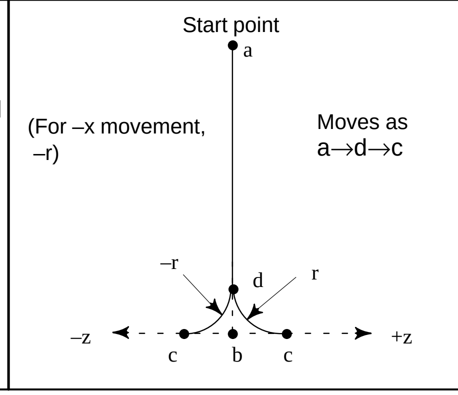
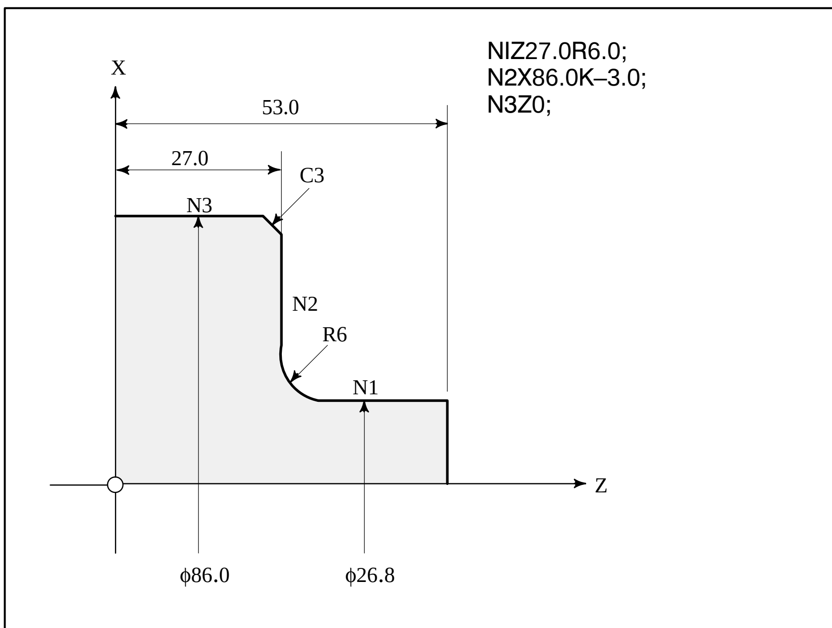
Format	Tool movement
G01X (U)_R ±r ; Specifies movement to point b with an absolute or incremental command in the figure on the right.	

Fig. 13.4 (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.
Specification value of chamfering or corner–R is 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. 13.4 (a) to (d). In incremental programming, specify a distance from point b.

Examples



NOTE

1 The following commands cause an alarm.
   1) Chamfering or corner–R is commanded when X and Z axes are specified by G01. (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. (alarm No. 055)
   3) Next block to the block where chamfering and corner R were specified, has not G01 command. (alarm No. 051, 052)
   4) If more than one of I, K, and R re specified in G01, alarm No. 053 is issued.

2 A single block stops at point c of Fig. 14.5 (a) to 14.5 (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 set I/K for an address for chamfering, fix parameter No.029#4 to 1.

5 If both C and R are specified with G01 in a block, the address specified last is valid.

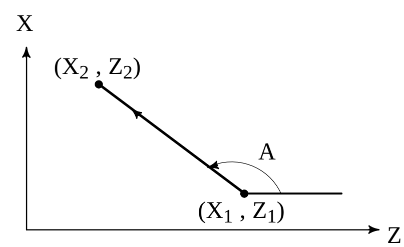
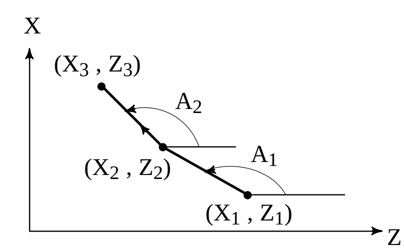
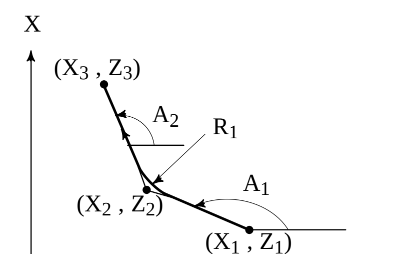
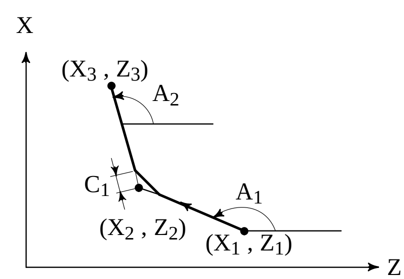
6 Neither chamfering nor corner–R machining can be specified in direct drawing dimension programming.

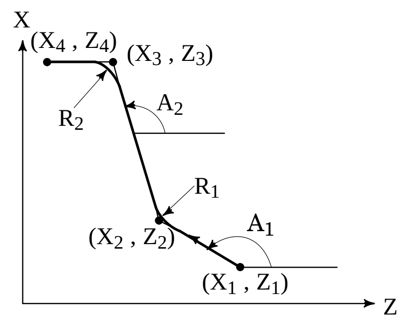
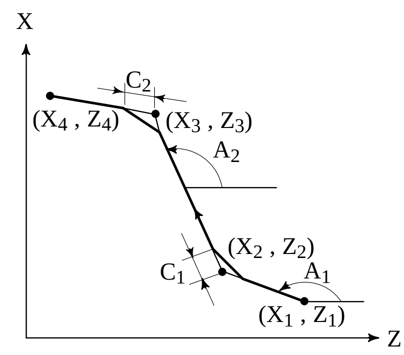
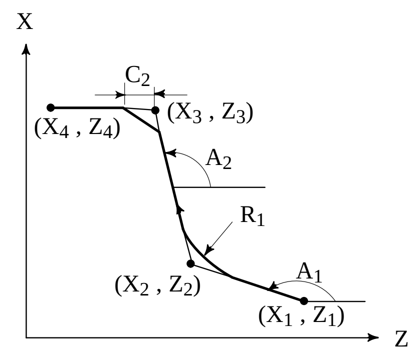
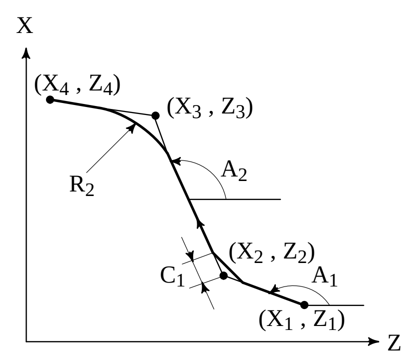
13.5 DIRECT DRAWING DIMENSIONS PROGRAMMING

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 13.5 (a) Commands table

	Commands	Movement of tool
1	X_2_ (Z_2_) A_ ; or 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 :

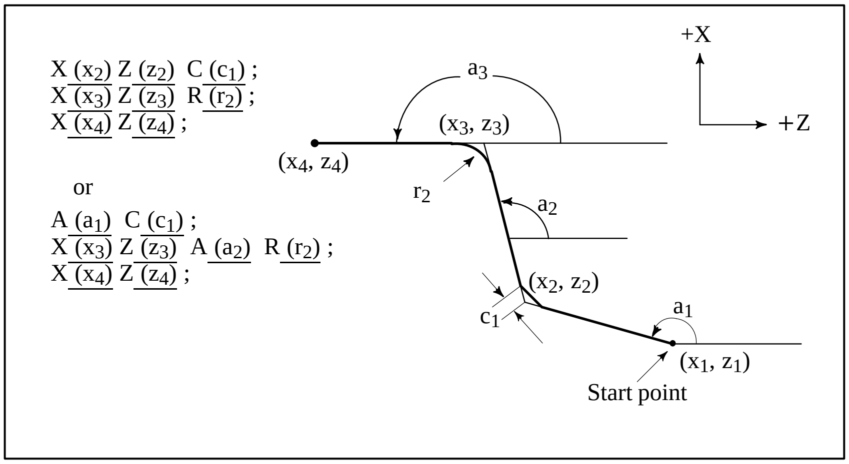


Fig. 13.5 (a) Machining Drawing (example)

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 following address.

- A_.....Straight line
- C_.....Chamfering
- R_.....Corner R

By specifying 1 to parameter No. 029#4 on the system which use C as an axis name, the degree of a straight line or the value of chamfering or corner R can be commanded with a comma before address.

, A_.....Straight line
, C_.....Chamfering
, R_......Corner R

NOTE

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 13.5 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 alarm No.057 occurs.)
   2) Z_ , A_ ; (If a value within 90°±1°or 270°±1°is specified for the angle instruction, the alarm No. 057 occurs.)

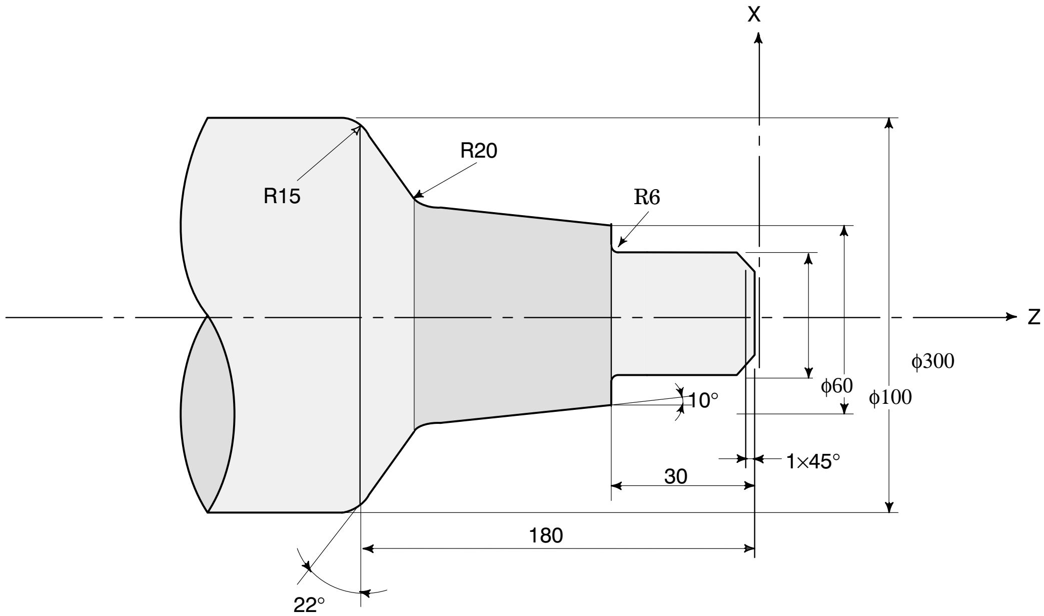
6 An alarm occurs if the angle made by the 2 lines is within ±1°when calculating the point of Intersection.

7 Chamfering or corner R 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 COMPENSATION FUNCTION

This chapter describes the following compensation functions:

14.1 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).

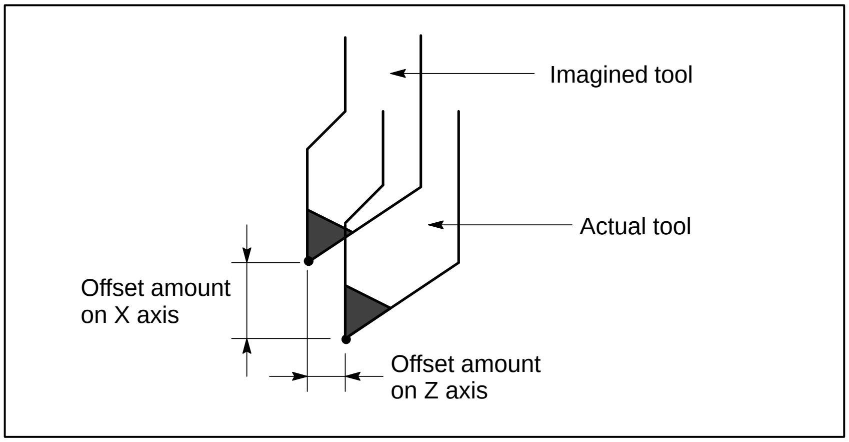


Fig. 14.1 (a) Tool offset

14.1.1 Tool Geometry Offset and Tool Wear Offset

Tool geometry offset is used to compensate for a tool figure or tool attachment position difference. Tool wear offset is used to compensate for tool tip wear. These offset values can be set separately. When no distinction is to be made between these values, set the total of these values as the tool position offset.

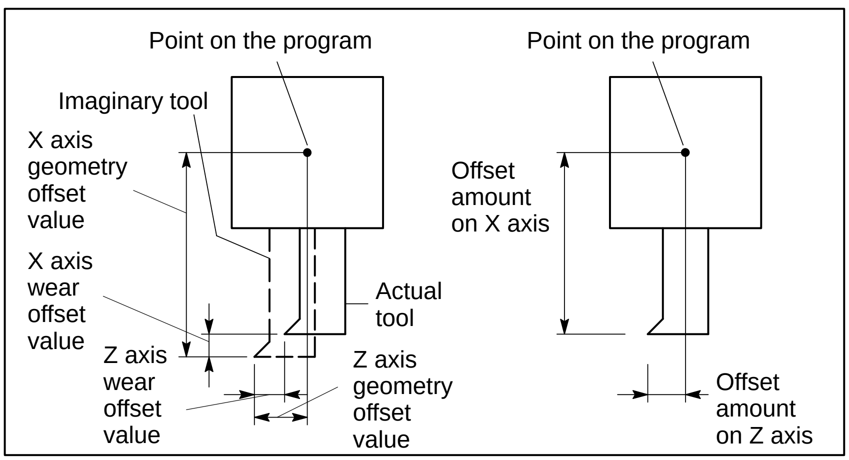


Fig. 14.1.1(a) Difference the tool geometry offset from tool wear offset

Fig. 14.1.1(b) Not difference the tool geometry offset from tool wear offset (The moving is same as tool wear offset)

14.1.2 T code for Tool Offset

There are two methods for specifying a T code as shown in Table 14.1.2(a) and Table 14.1.2(b).

Format

- **Lower digit of T code specifies geometry and wear offset number**

Table 14.1.2 (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 bit 0 of parameter No.014, is set to 1, a tool wear offset number is specified with the last digit of a T code.	When bit 1 of parameter No.013, 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 bit 0 of parameter No.014, 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 14.1.2(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 bit 0 of parameter No.014, is set to 1, a tool wear offset number is specified with the last digit of a T code.	When bit 1 of parameter No.013, is set to 1, 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 bit 0 of parameter No.014, is set to 0, a tool wear offset number is specified with the last two digits of a T code.	

14.1.3 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.

14.1.4 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.

14.1.5 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.

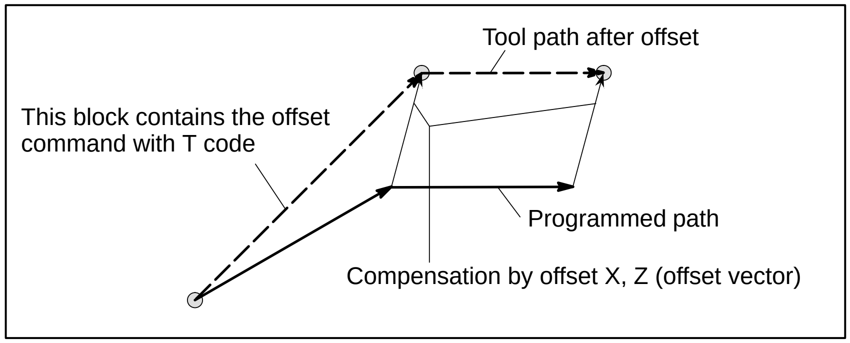


Fig. 14.1.5 (a) Movement of offset (1)

- **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.

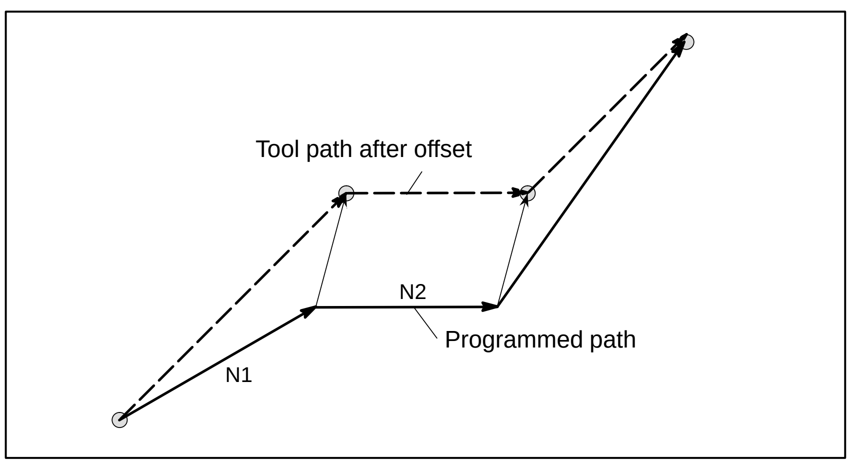


Fig. 14.1.5 (b) Movement of offset (2)

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.

- **Tool wear offset in only T code command**

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.
By setting bit 4 of parameter No. 014, however, an offset movement can be made together with the axis movement specified by the next block.

> **WARNING**
> 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.

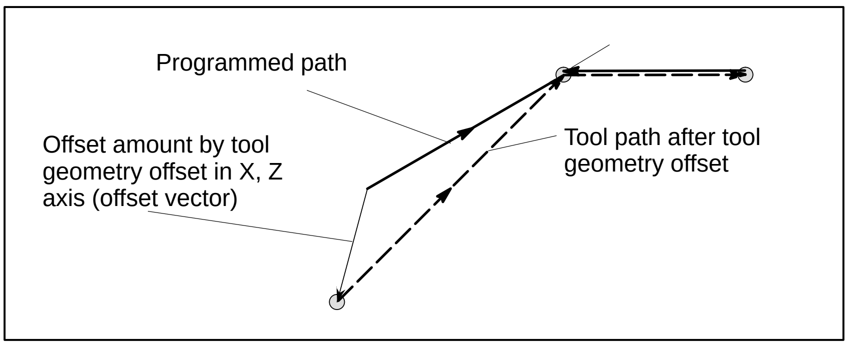


Fig. 14.1.5 (c) Movement of tool geometry offset

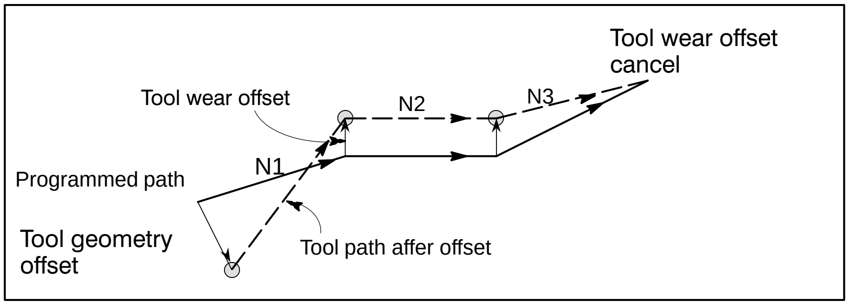
> **NOTE**
> As well as wear offset, the tool can be compensated by parameter setting (No.013#2) to add or subtract the programmed end point of each block.

- **Tool geometry offset cancel**

Tool geometry offset is not canceled, even when offset number 0 is specified. By setting bit 3 of parameter No. 013, however, tool geometry offset can be canceled when offset number 0 is specified.
Furthermore, by setting bit 3 of parameter No. 001/bit 1 of parameter No. 014, tool geometry offset can also be canceled by a reset.

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 bit 1 of parameter No.013, is set 0),
N1 X50.0 Z100.0 T0202 ; Specifies offset number 02
N2 Z200.0 ;
N3 X100.0 Z250.0 T0200 ; Cancels offset



14.2 OVERVIEW OF TOOL NOSE RADIUS COMPENSATION

It is difficult tp produce the compensation necessary to form accurate parts when using only the tool,offset function due to tool nose roundness. The tool nose radius compensation function compensates automatically for the above errors.

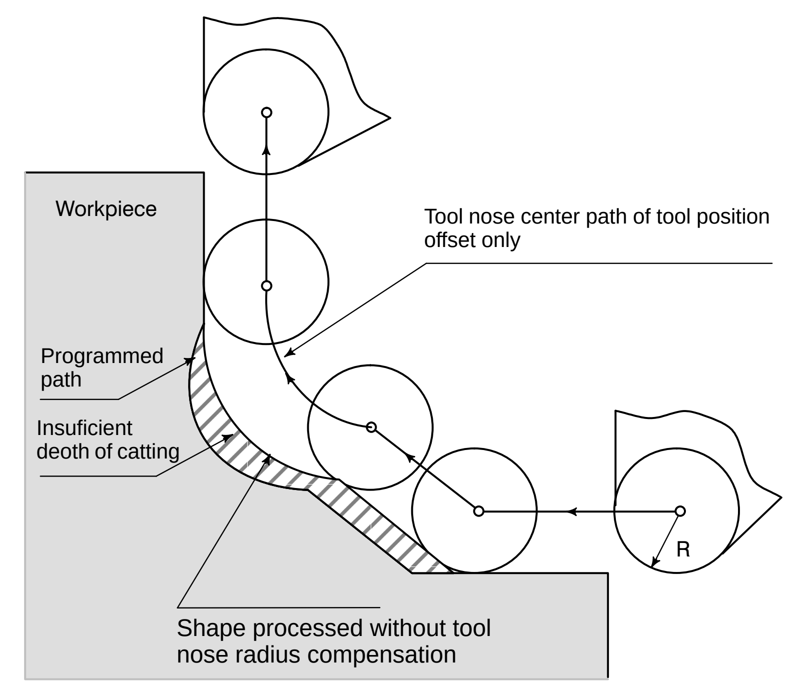


Fig. 14.2 Tool path of tool nose radius compensation

14.2.1 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.

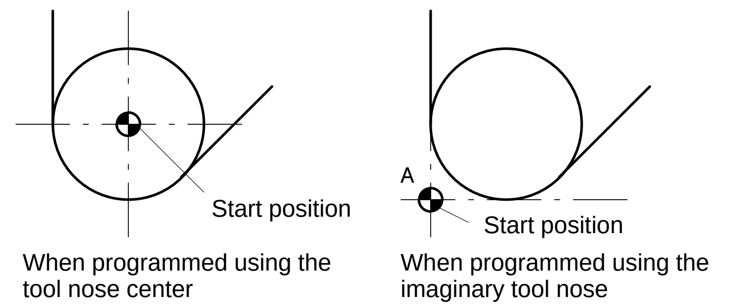


Fig. 14.2.1 (a) Tool nose radius center and imaginary tool nose

NOTE

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.

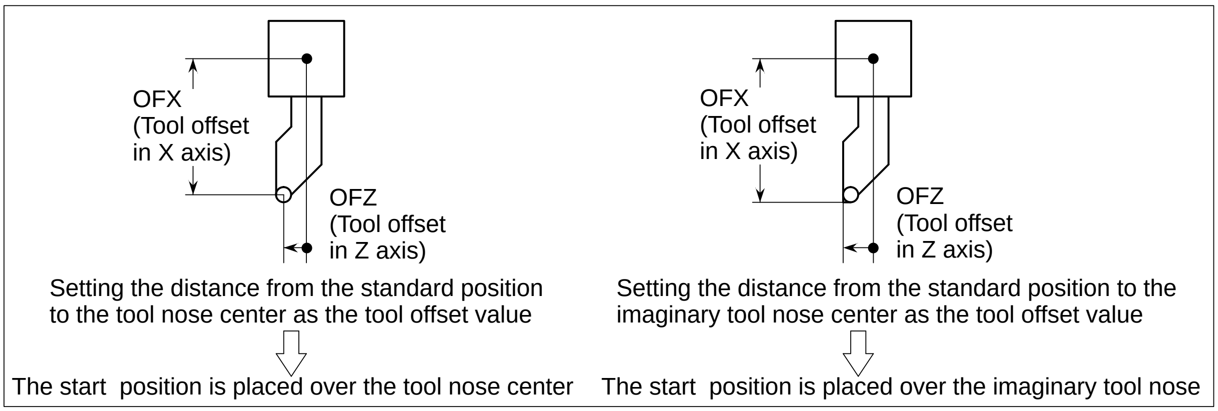


Fig. 14.2.1 (b) Tool offset value when the turret center is placed over the start position

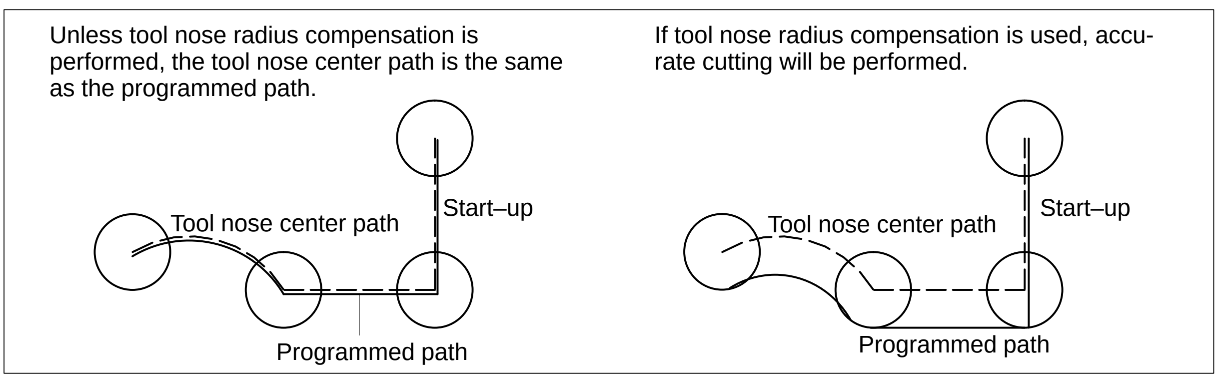


Fig. 14.2.1 (c) Tool path when programming using the tool nose center

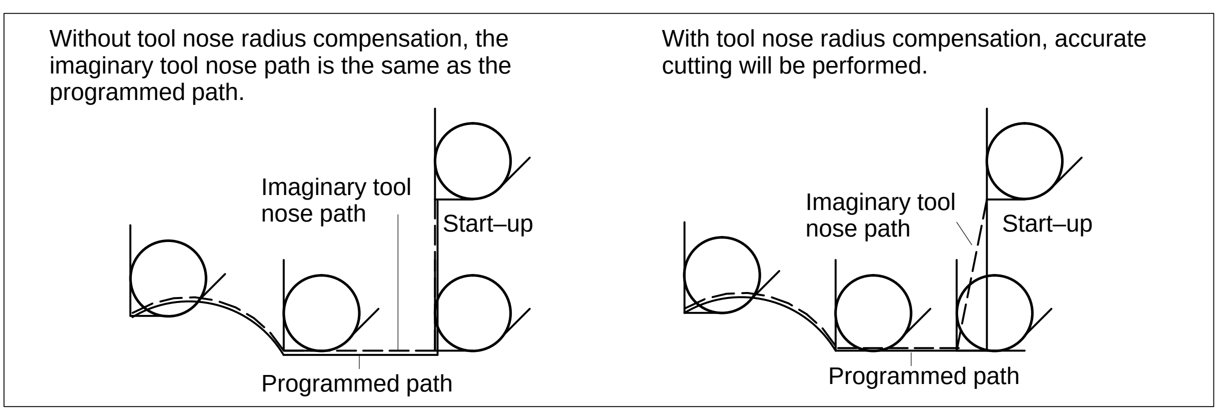


Fig. 14.2.1 (d) Tool path when programming using the imaginary tool nose

14.2.2 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.14.2.2 below together with their corresponding codes.

This Fig14.2.2 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.

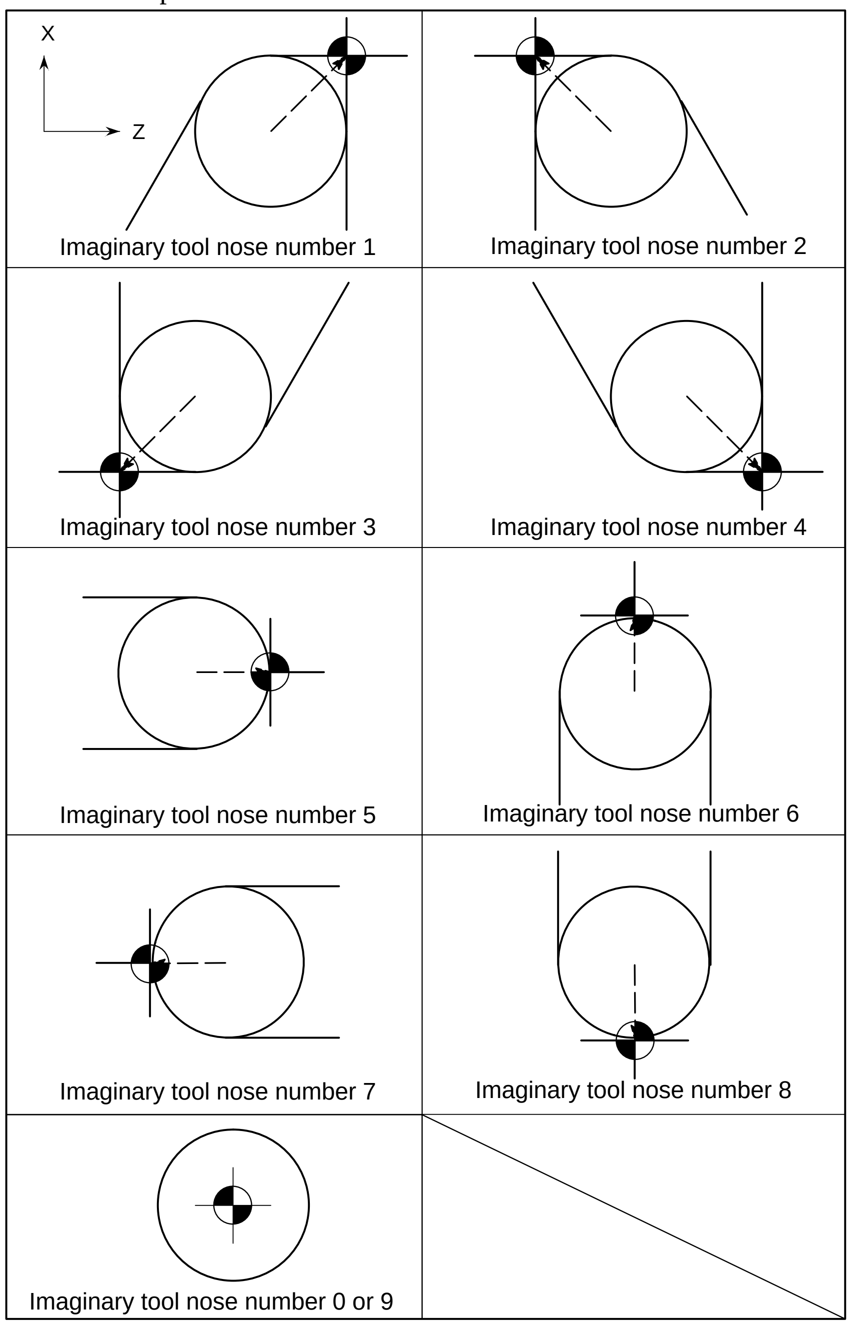


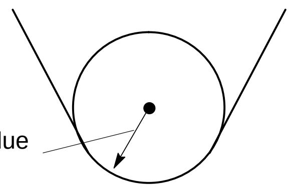
Fig. 14.2.2 Direction of imaginary tool nose

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.

14.2.3 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 14.2.3 (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)
01	0.040	0.020	0.20	1
02	0.060	0.030	0.25	2
:	:	:	:	:
31	0.050	0.015	0.12	6
32	0.030	0.025	0.24	3

When the options of tool geometry compensation and tool wear compensation are selected, the offset values become as follows :

Table 14.2.3 (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)
G01	10.040	50.020	0	1
G02	20.060	30.030	0	2
G03	0	0	0.20	6
G04	:	:	:	:
G05	:	:	:	:
:	:	:	:	:

Table 14.2.3 (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)
W01	0.040	0.020	0	1
W02	0.060	0.030	0	2
W03	0	0	0.20	6
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 14.1.2.

NOTE

When the geometry offset number is made common to the tool selection by the parameter (No.013#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. By setting bit 3 of parameter No. 075, however, the imaginary tool nose direction, specified with a wear offset number, can be enabled.

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

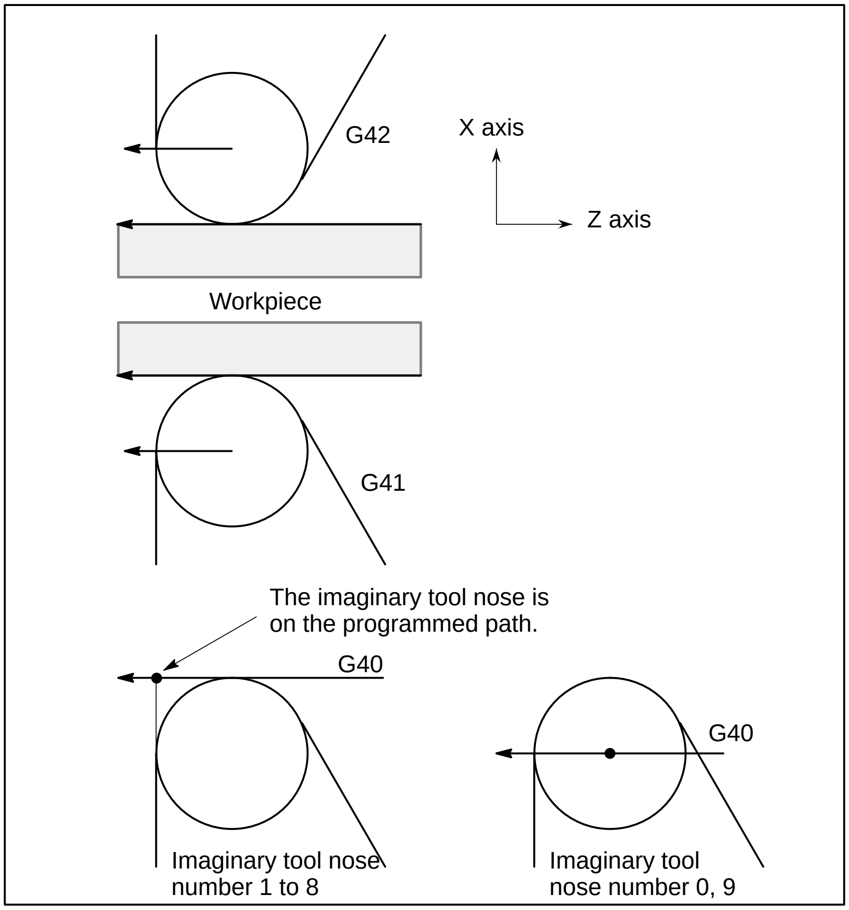
The offset value corresponding to the offset number 0 is always 0.
No offset value can be set to offset number 0.

14.2.4 Work Position and Move Command

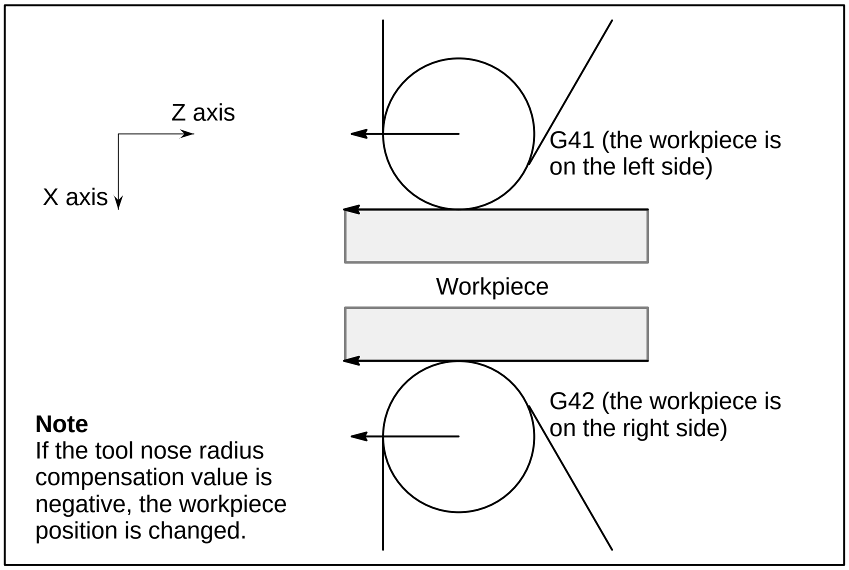
In tool nose radius compensation, the position of the workpiece with respect to the tool must be specified.

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.



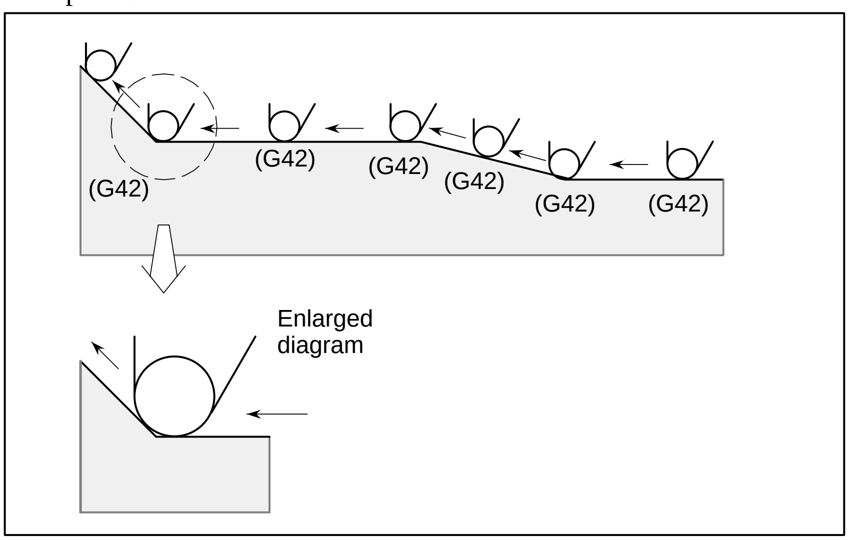
The workpiece position can be changed by setting the coordinate system as shown below.



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.

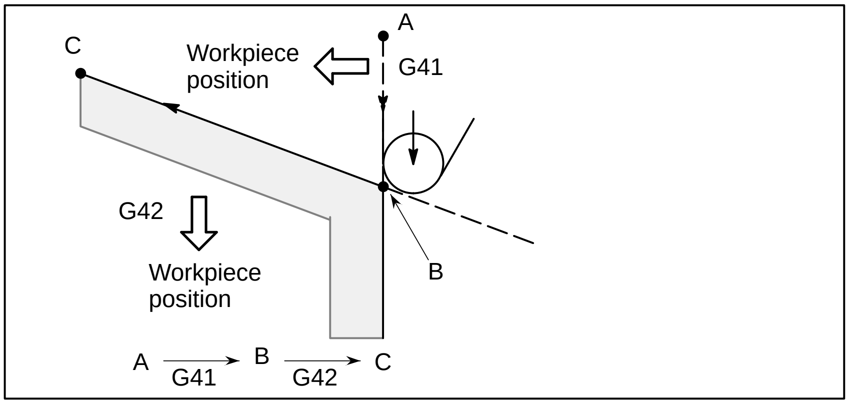
- **Tool movement when the workpiece position does not change**

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.



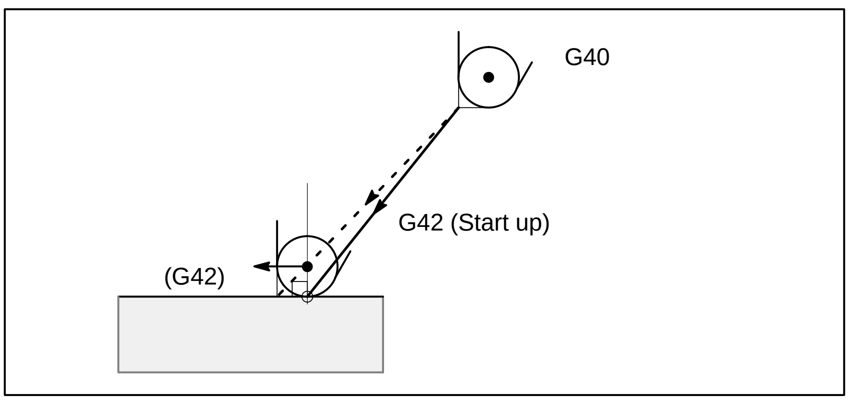
Although the workpiece does not exist on the right side of the programmed path in the above case. the existence of the workpiece is assumed in the movement from A to B. The workpiece direction must not be changed in the block next to the start–up block.

- **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.

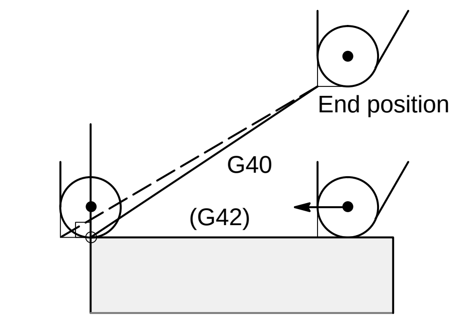


- **Offset cancel**

The block in which the mode changes to G40 from G41 or G42 is called the offset cancel block.

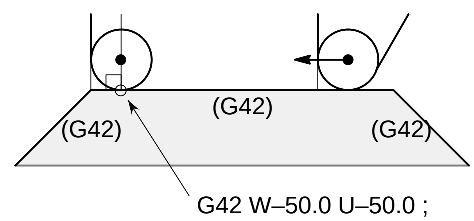
G41 _ ;
G40 _ ; (Offset cancel block)

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.

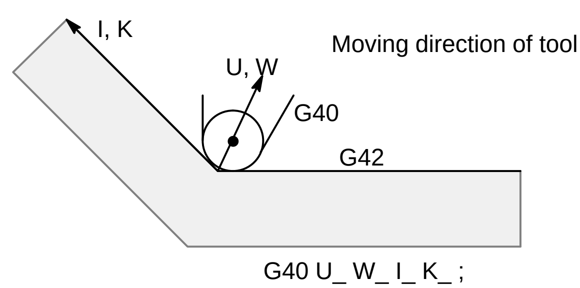


In the block that first specifies G41/G42, the above positioning of the tool nose center is not performed.

- **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 :

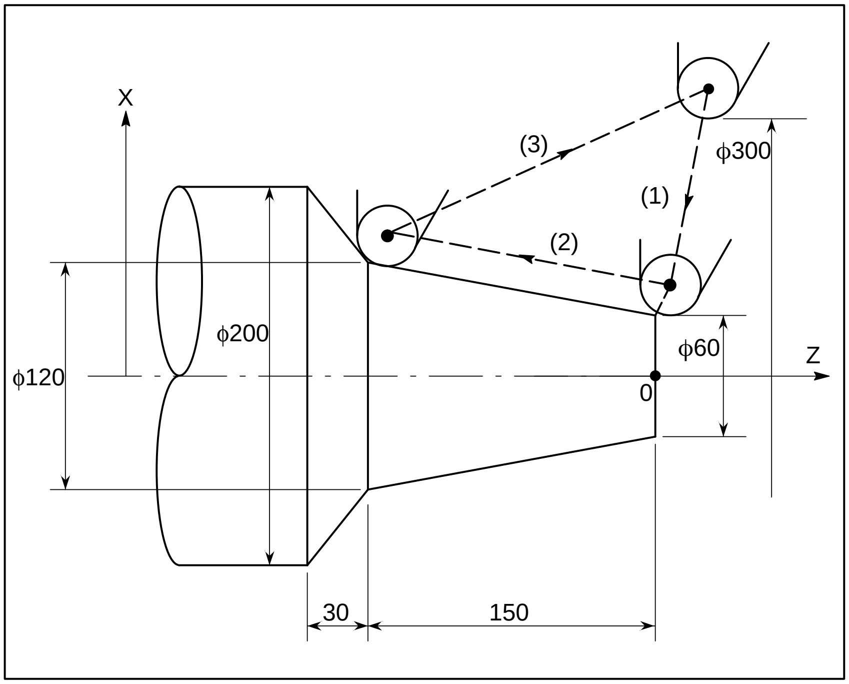
G40 X(U) _ Z(W) _ I _ K _ ;



The workpiece position specified by addresses I and K is the same as that in the preceding block.

G40 X_ Z_ I_ K_ ;	I, J of tool nose radius compensation cancel
G40 G02 X_ Z_ I_ K_ ;	I, J of circular interpolation center

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 ;

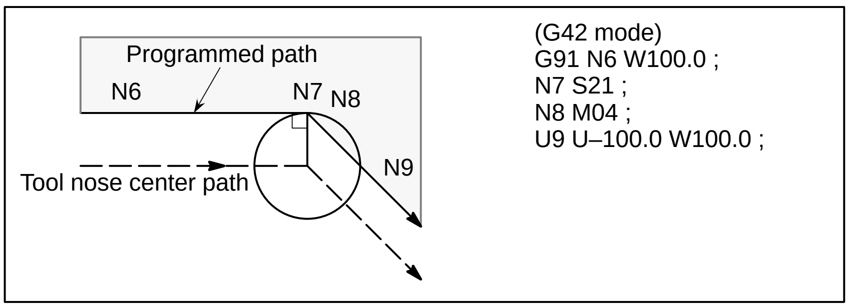
14.2.5 Notes on Tool Nose Radius Compensation

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 X1.0 ;	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.

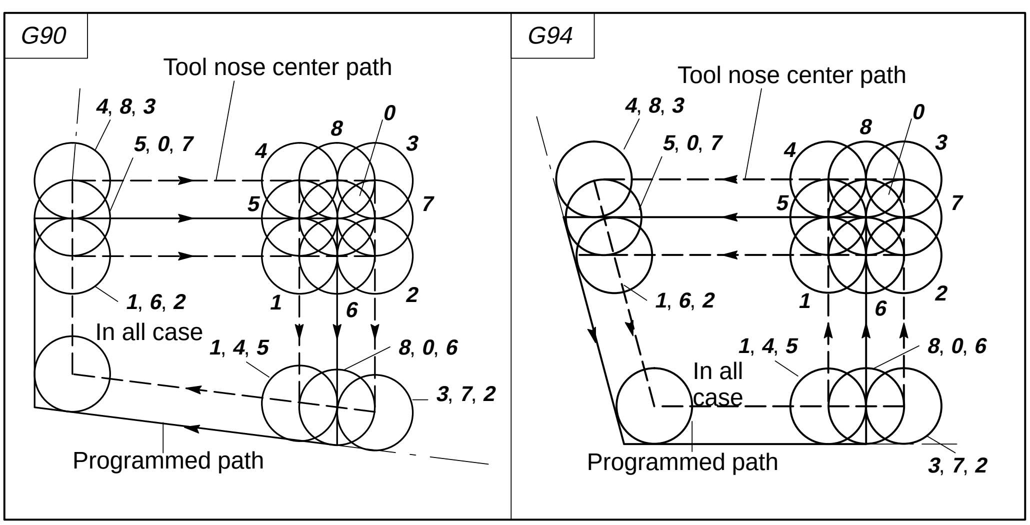


• Tool nose radius compensation with G90 or G94

Tool nose radius compensation with G90 (outer diameter/internal diameter cutting cycle) or G94 (end face turning cycle) is as follows, :

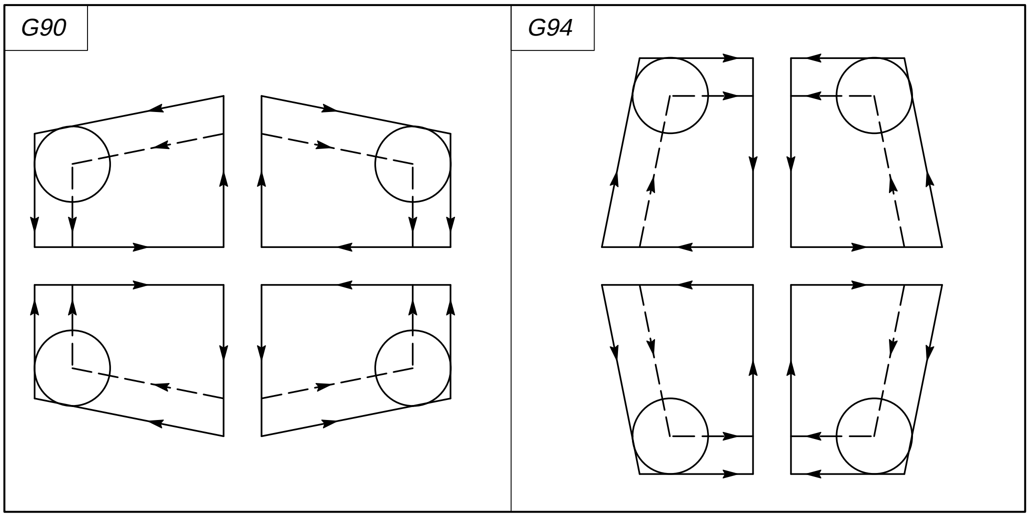
1.Motion for imaginary tool nose numbers

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.

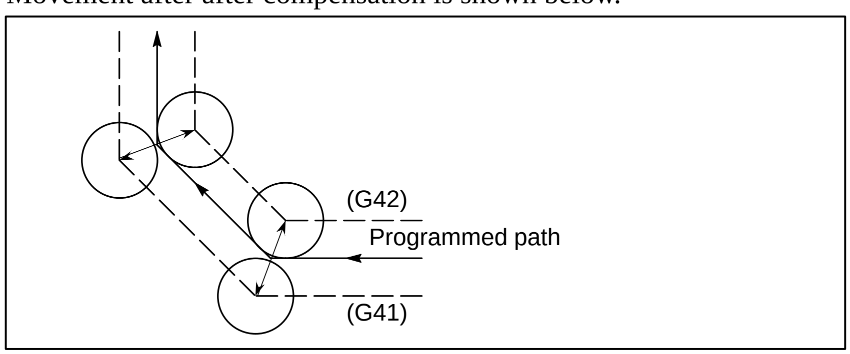
G71 (Stock removal in turning or traverse grinding cycle)
G72 (Stock removal in facing or traverse direct constant–dimension grinding cycle)
G73 (Pattern repeating or Oscillation grinding cycle)

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)

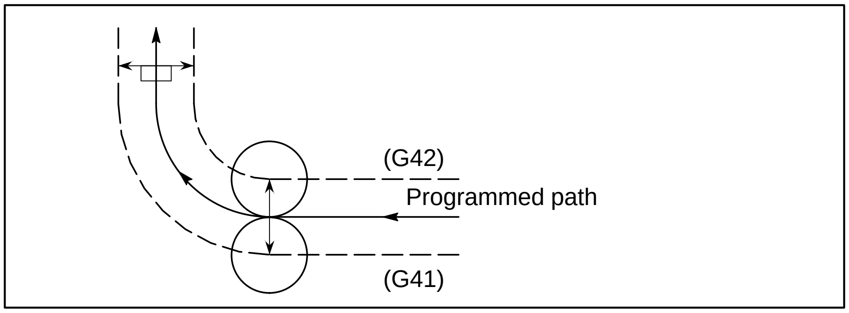
- **Tool nose radius compensation when chamfering is performed**

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.

14.3 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 14.2.
This section consists of the following subsections:

14.3.1 General

- **Tool nose radius center compensation vector**

The tool nose radius compensation 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 compensation) 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, or G32 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 CRT/MDI panel 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 tool path 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**

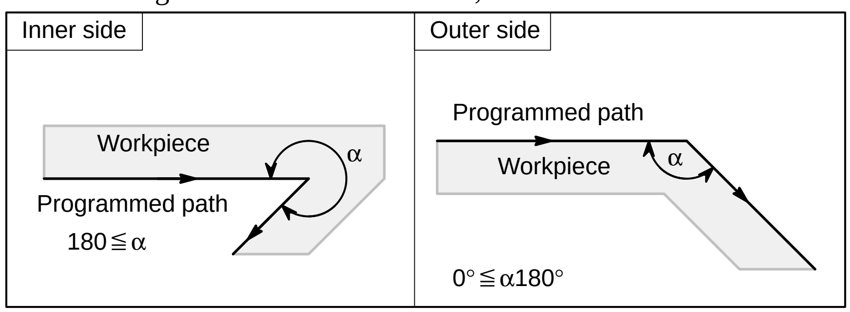
When a block which satisfies all the following conditions is executed in cancel mode, the system enters the offset mode. Control during this operation is called start–up.

- 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, 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 180k, it is referred to as "inner side." When the angle is between 0k and 180k, it is referred to as "outer side."



- **Meaning of symbols**

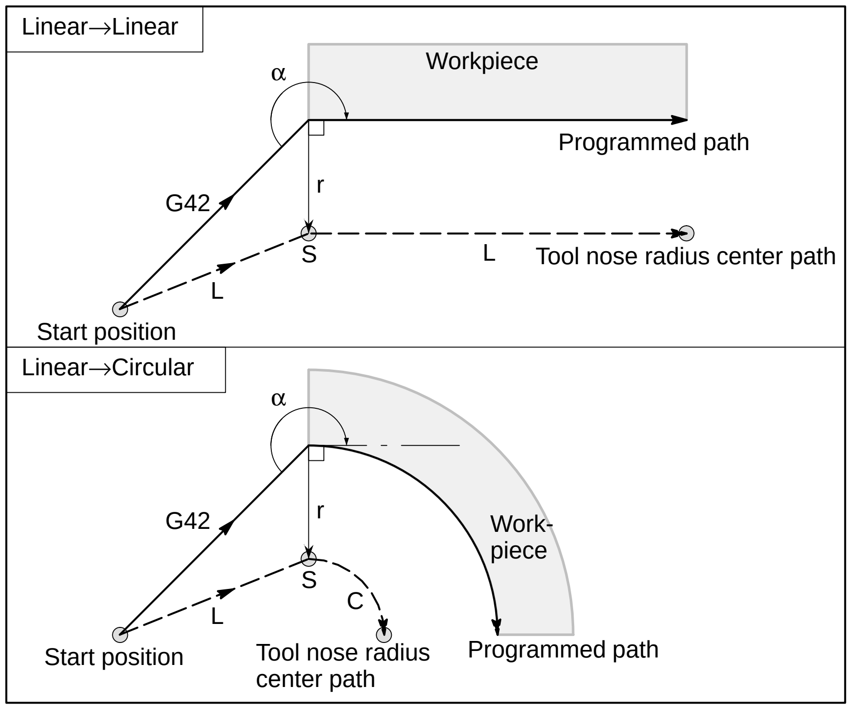
The following symbols are used in subsequent figures:
– ***S*** indicates a position at which a single block is executed once.
– ***SS*** indicates a position at which a single block is executed twice.
– ***SSS*** indicates a position at which a single block is executed three times.
– ***L*** indicates 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.

14.3.2 Tool Movement in Start–Up

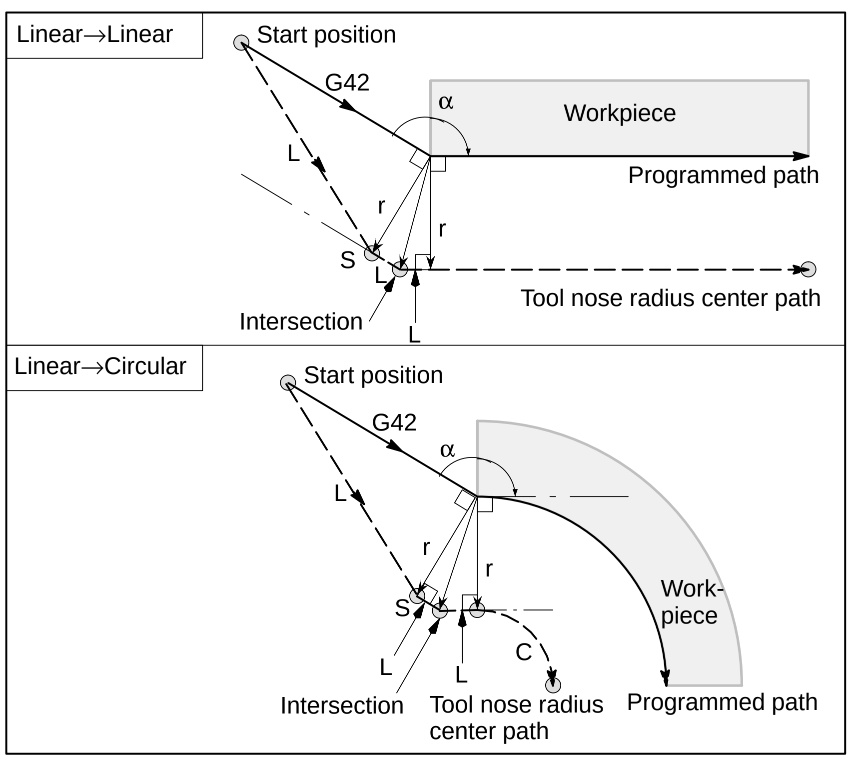
When the offset cancel mode is changed to offset mode, the tool moves as illustrated below (start–up):

Explanations

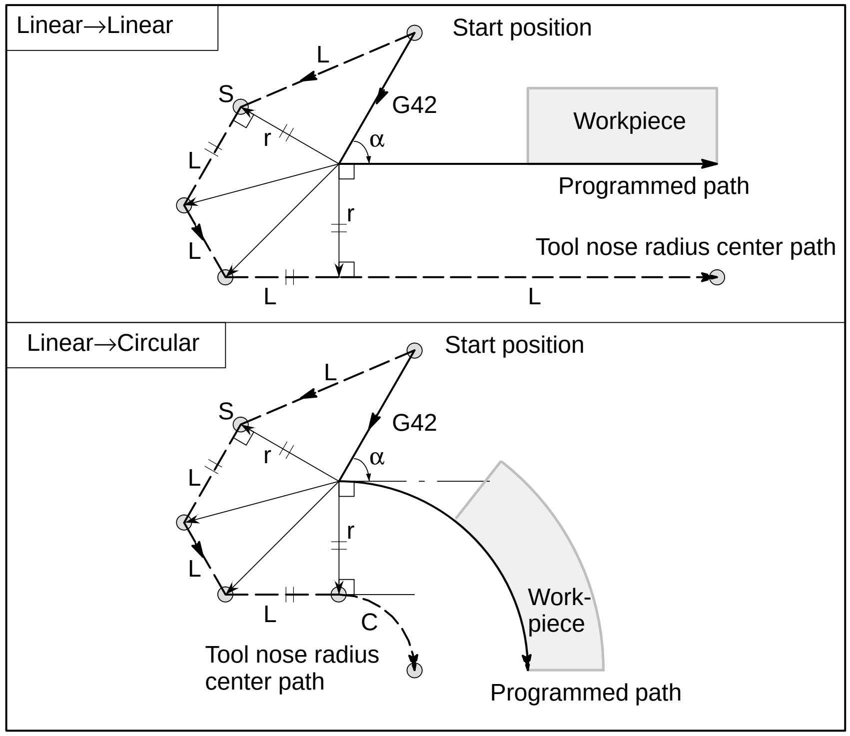
- **Tool movement around an inner side of a corner (180° ≦ α)**



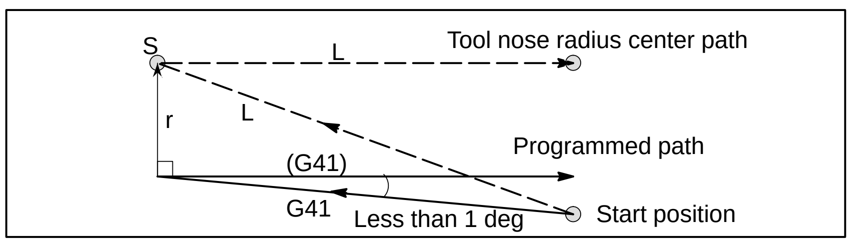
- **Tool movement around the outside of a corner at an obtuse angle (90° ≦ α < 180°)**



- **Tool movement around the outside of an acute angle (α < 90°)**



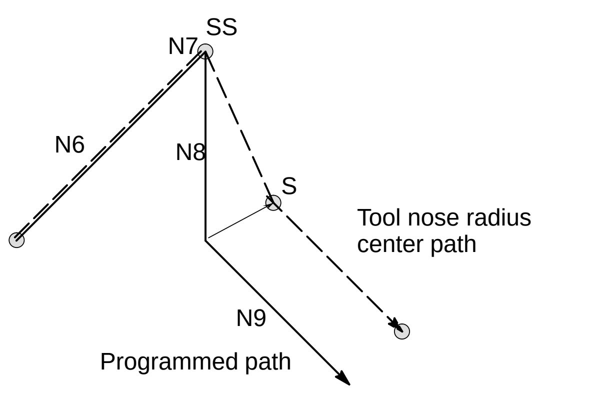
- **Tool movement around the outside linear→linear at an acute angle less than 1 degree (α < 1°)**



- **A block without tool movement specified at start–up**

If the command is specified at start–up, the offset vector is not created.

```
G91 G40 ··· ;
      :
N6 U10.0 W10.0 ;
N7 G41 U0 ;
N8 W–10.0 ;
N9 W–10.0 U10.0 ;
```



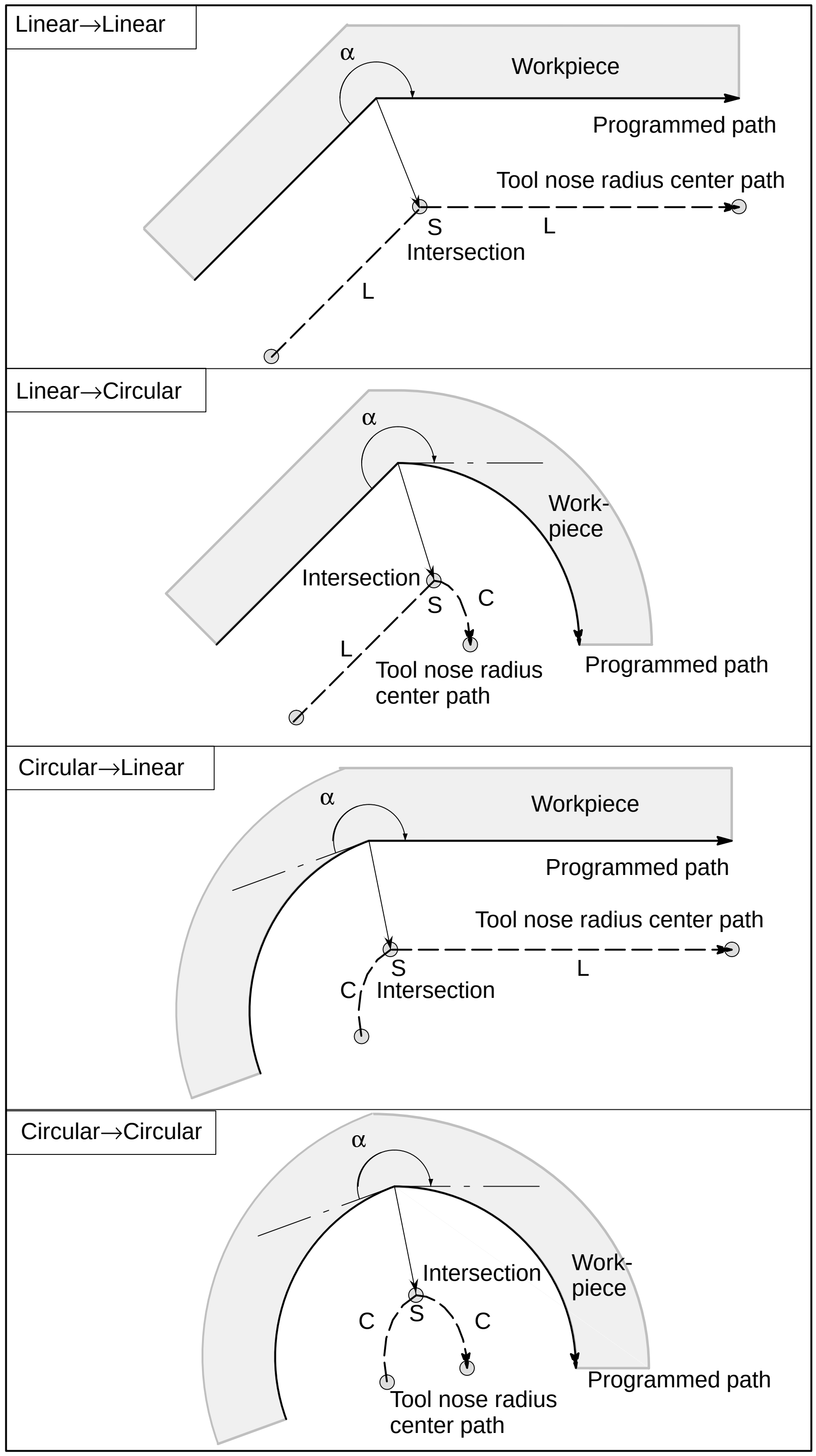
NOTE
For the definition of blocks that do not move the tool, see Subsection 14.3.3.

14.3.3 Tool Movement in Offset Mode

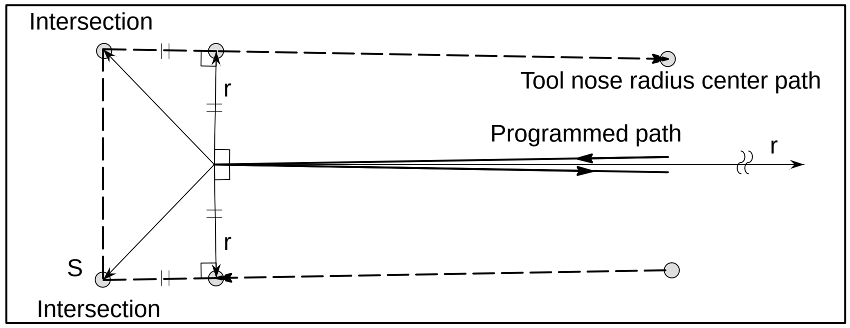
In the offset mode, the tool moves as illustrated below:

Explanations

- **Tool movement around the inside of a corner (180° ≦ α)**

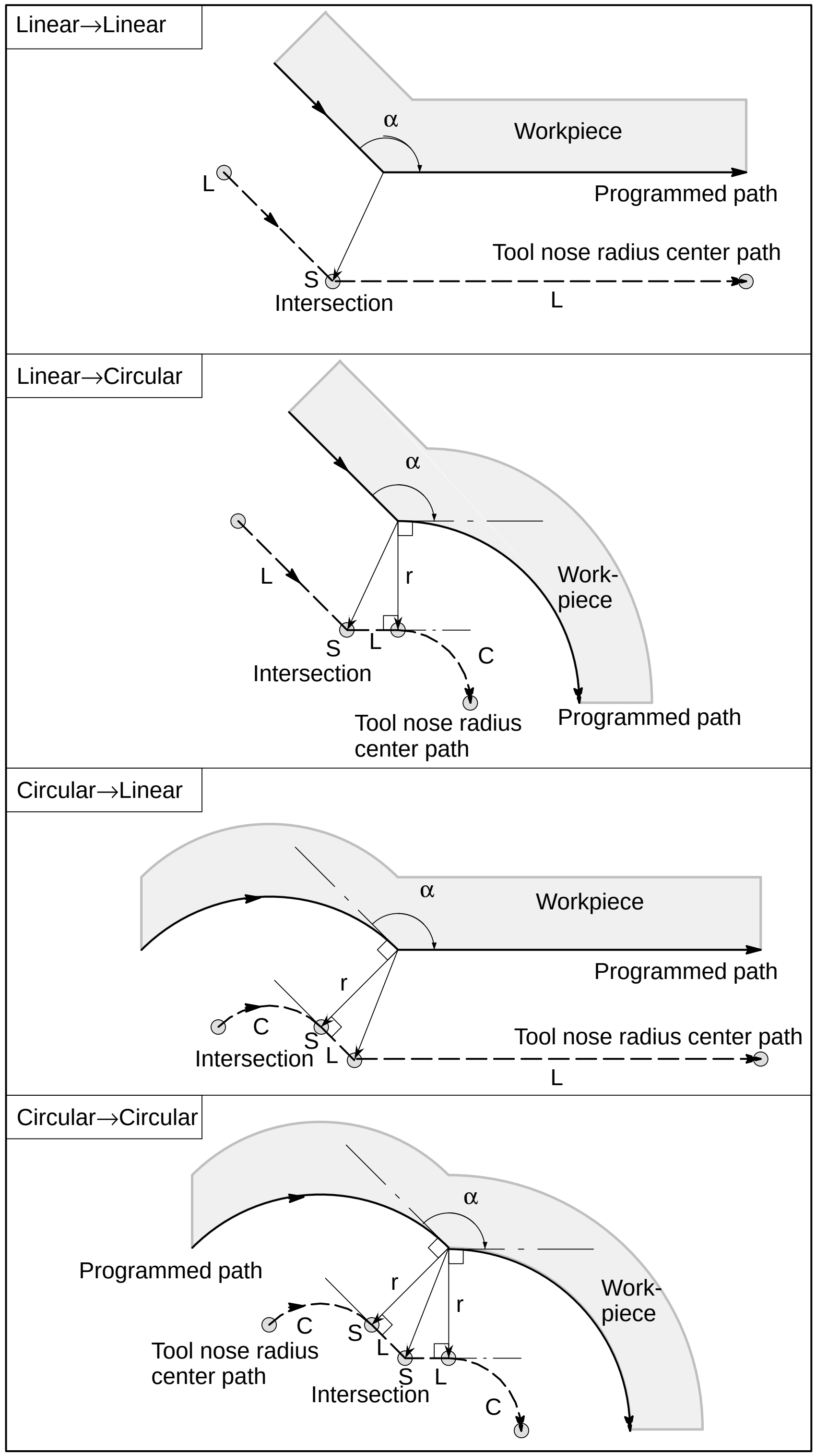


- **Tool movement around the inside ($\alpha < 1°$) with an abnormally long vector, linear → linear**

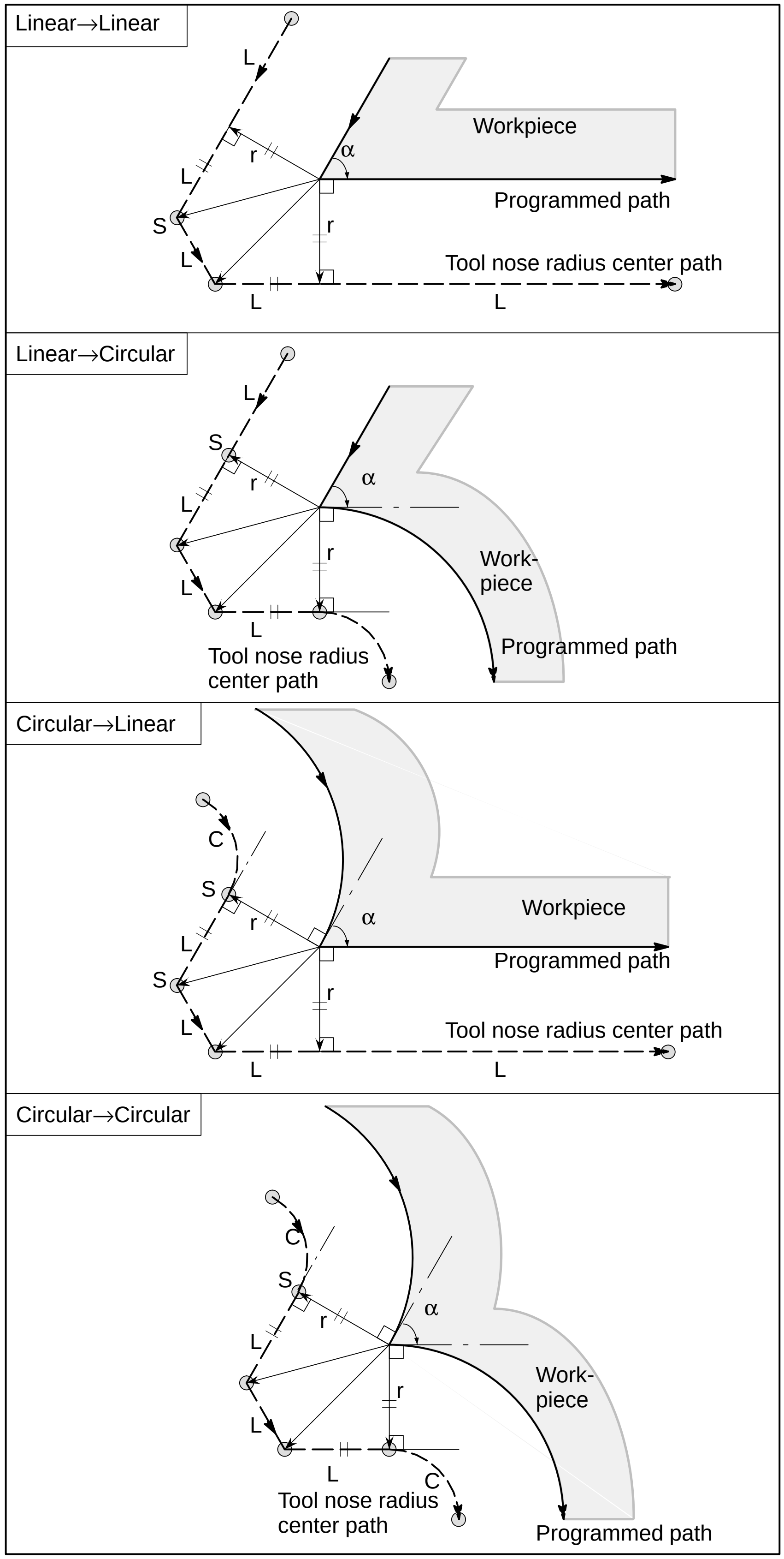


Also in case of arc to straight line, straight line to arc and arc to arc, the reader should infer in the same procedure.

- **Tool movement around the outside corner at an obtuse angle (90°≦α<180°)**



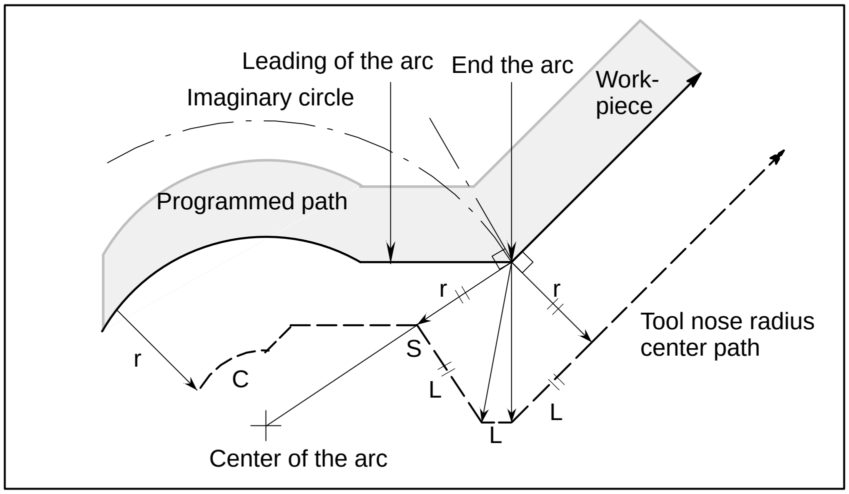
- **Tool movement around the outside corner at an acute angle ($\alpha < 90°$)**



- **When it is exceptional**

End position for the arc is not on the arc

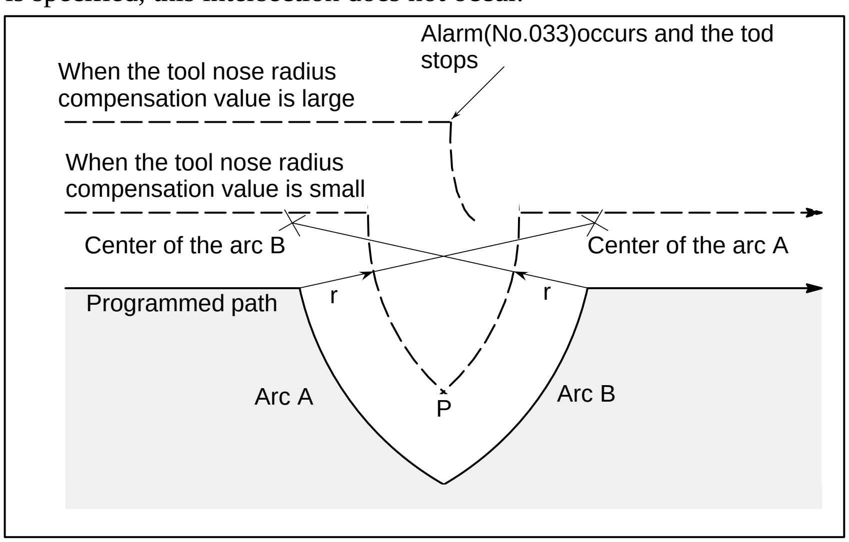
If the end of a line leading to an arc is programmed as the end of the arc by mistake as illustrated below, the system assumes that tool nose radius compensation has been executed with respect to an imaginary circle that has the same center as the arc and passes the specified end position. Based on this 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.

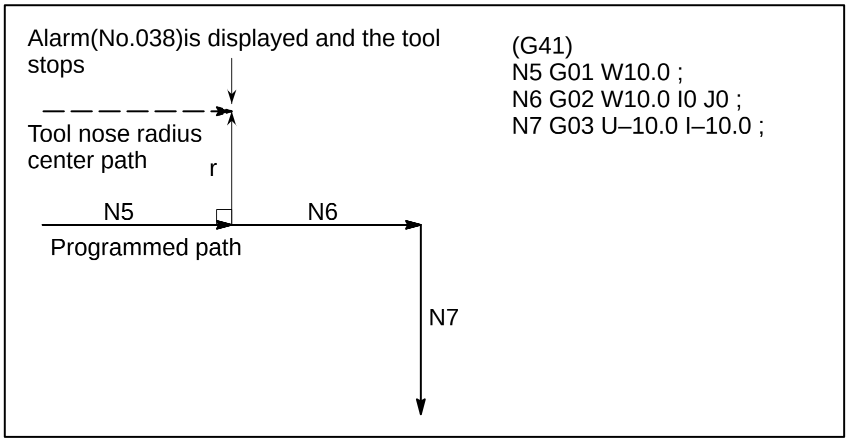
There is no inner intersection

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, alarm 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, alarm (No. 038) is displayed, and the tool will stop at the end position of the preceding block.



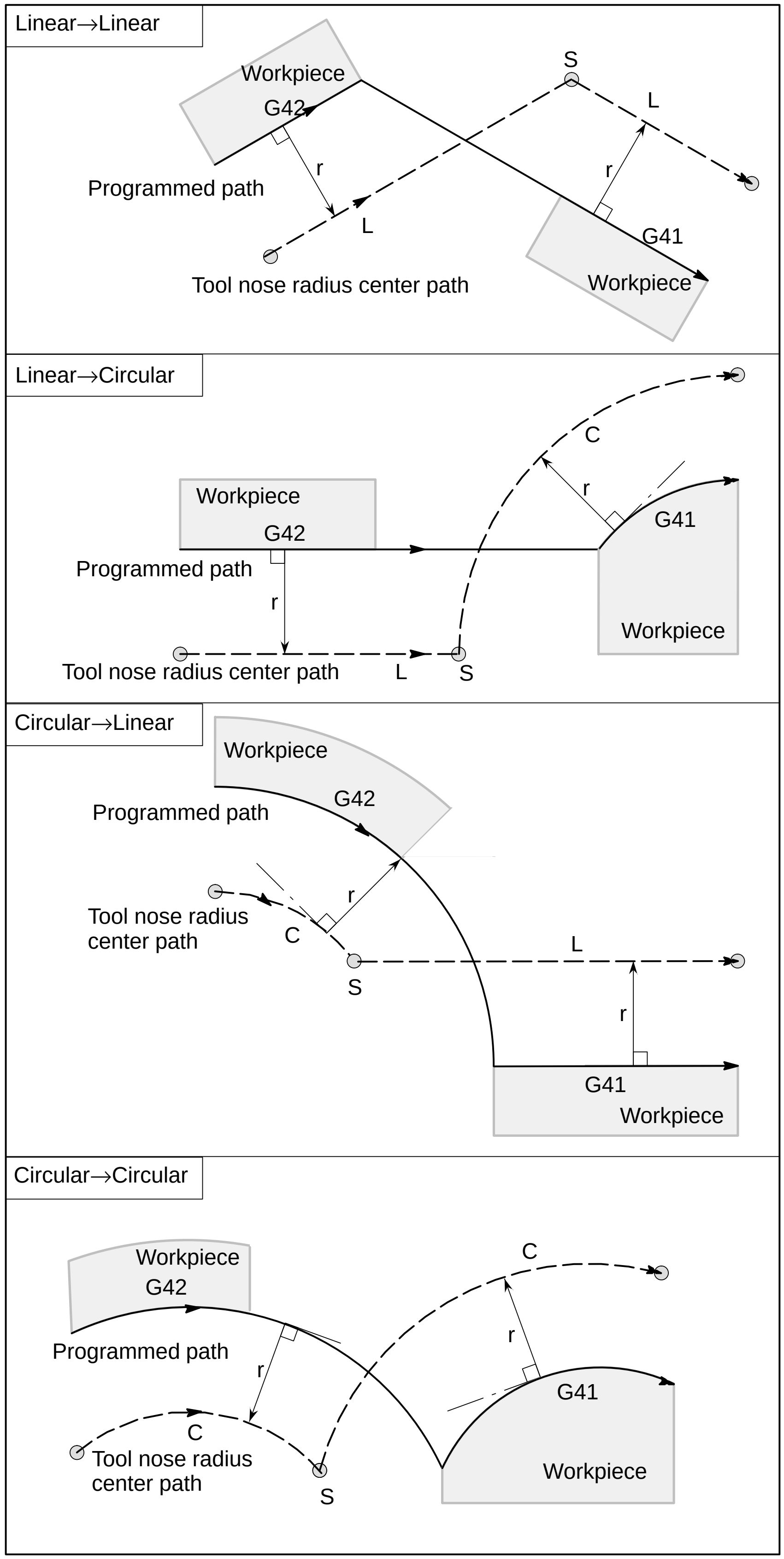
- **Change in the offset direction in the offset mode**

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

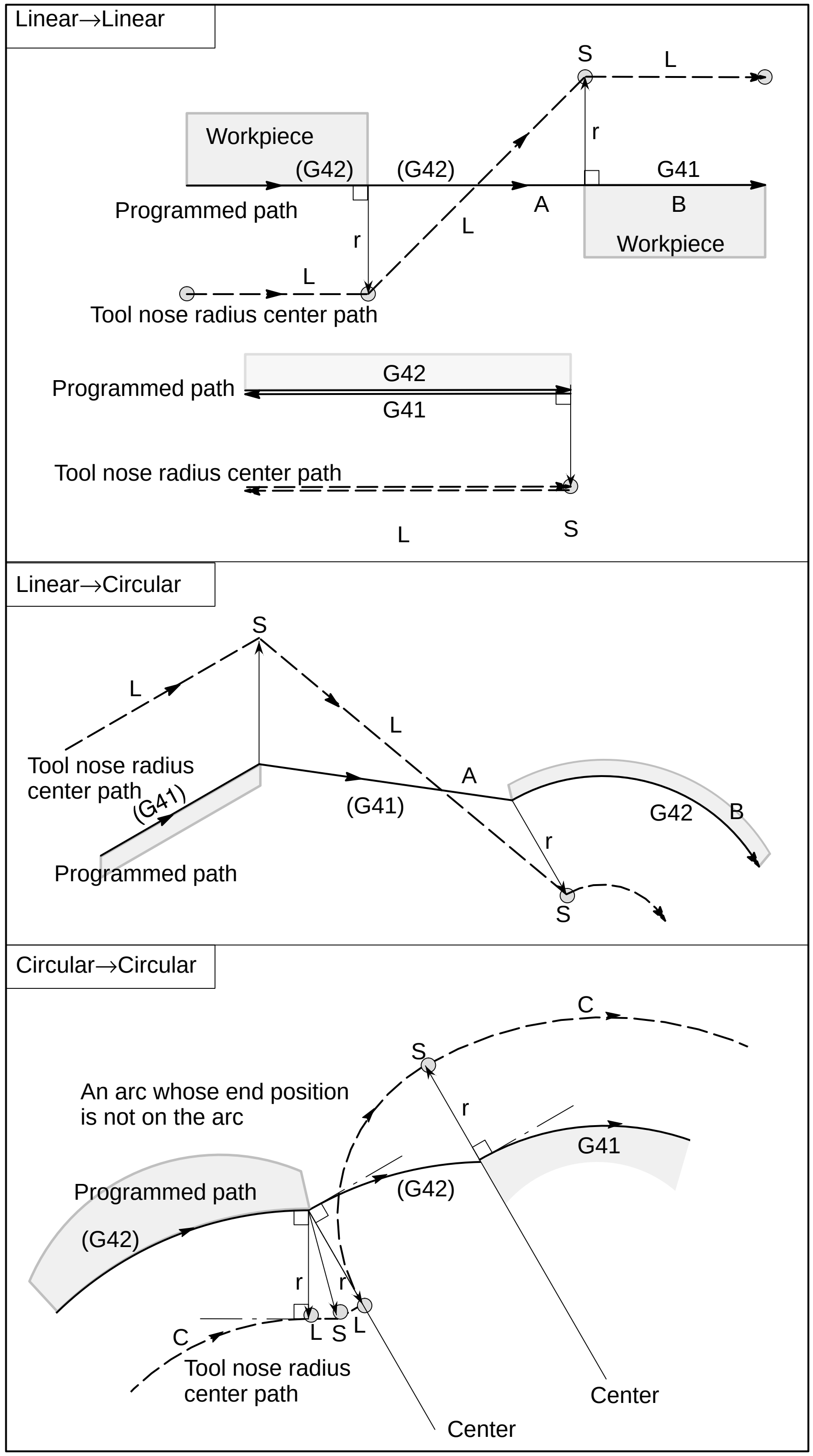
The offset direction can be changed in the offset mode. If the offset direction is changed in a block, a vector is generated at the intersection of the tool nose radius center path of that block and the tool nose radius center path of a preceding block. However, the change is not available in the start–up 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.

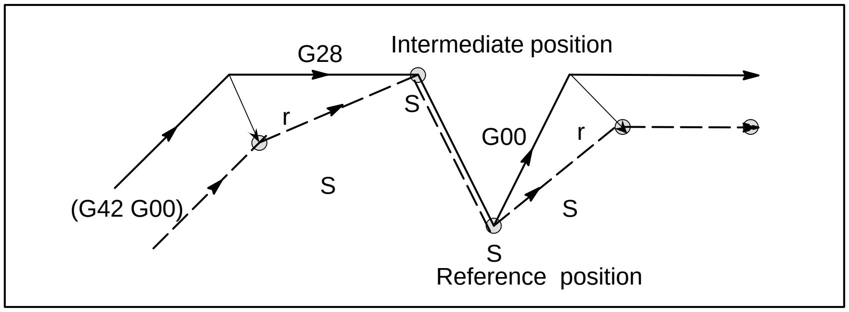


- **Temporary tool nose radius compensation cancel**

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 14.3.2 and 14.3.4.

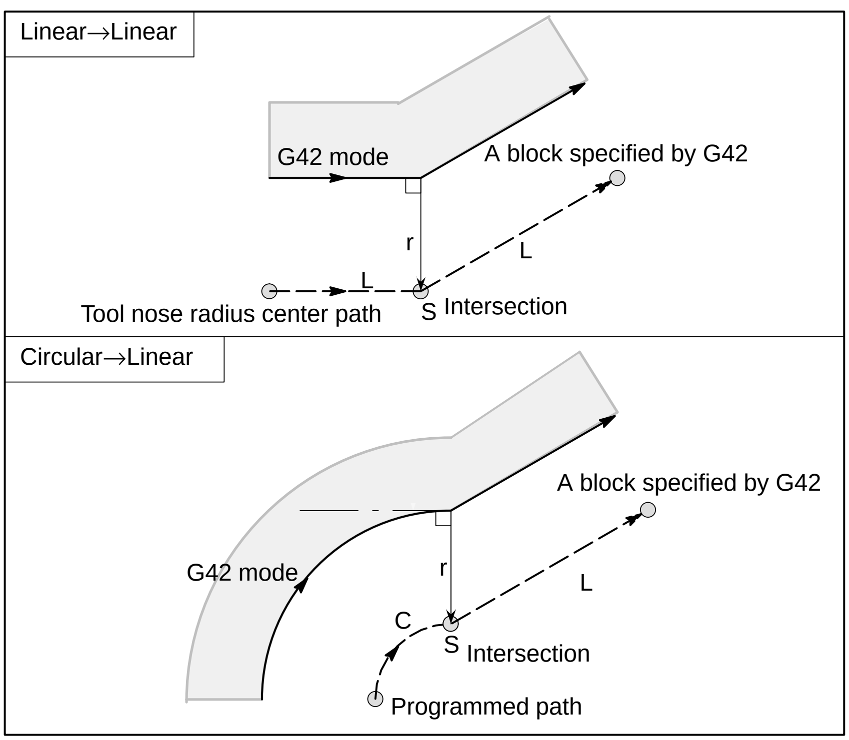
Specifying G28 (automatic return to the reference position) in the offset mode

If G28 is specified in the offset mode, the offset mode is canceled at an intermediate position. Offset mode is restored automatically after the tool is returned to the reference positition.



- **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.

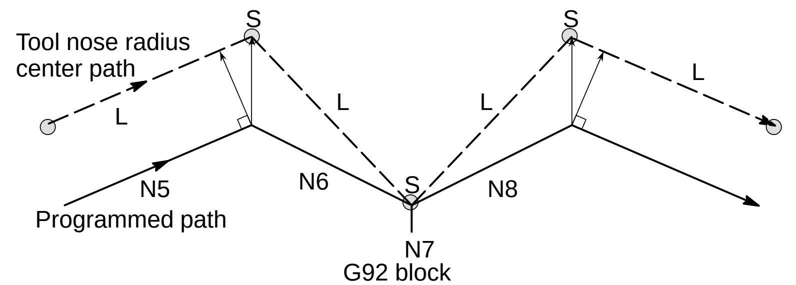


- **Command cancelling the offset vector temporarity**

During offset mode, if G92 (absolute zero point programming) 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.

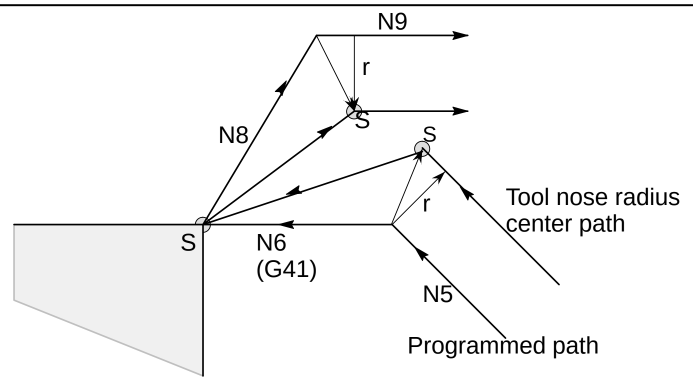
- **Workpiece coordinate system setting (G50)**




```
(G41)
N5 G91 G01 U70.0 W30.0 ;
N6 U–60.0 W30.0 ;
N7 G50 X20.0 Z10.0 ;
N8 G01 X80.0 Z40.0 ;
```

- **Canned cycles (G90, G92, G94) and Multiple repetitive cycles (G71 to G76)**

See Sections 13.1 (G90, G92, G94) and 13.2 (G71 to G76) for the tool nose radius compensation is related canned cycles.




```
(G42)
N5 G01 U50.0 W–60.0 ;
N6 W–80.0 ;
N7 G90 U–60.0 Z–80.0 I–30.0 ;
N8 G00 U120.0 W50.0 ;
N9 W50.0 ;
```

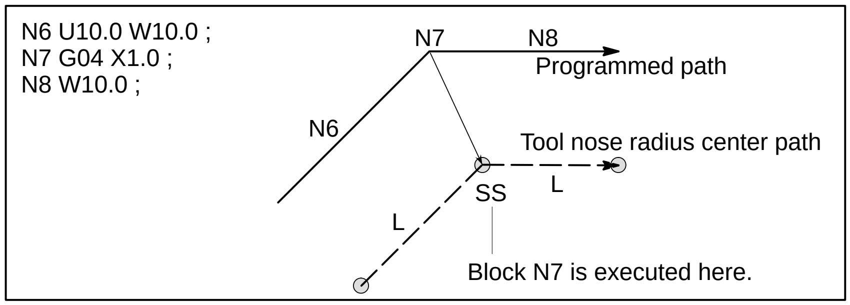
- **A block without tool movement**

The following blocks have no tool movement. In these blocks, the tool will not move even if tool nose radius compensation is effected.

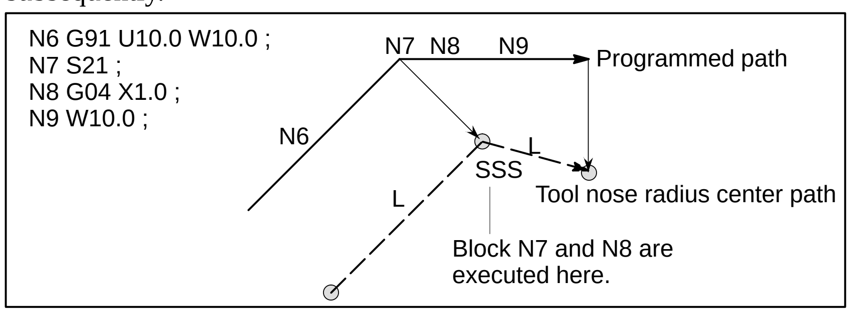
M05 ;	M code output	Commands are no movement.
S21 ;	S code output	
G04 X1.0 ;	Dwell	
G10 P01 X10 Z20 R10.0 ;	tool nose radius compensation value setting/modification	
Y20.0 ;	Move command not included in the offset plane.	
G98 ;	G code only	
U0 ;	Move distance is zero.	

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.



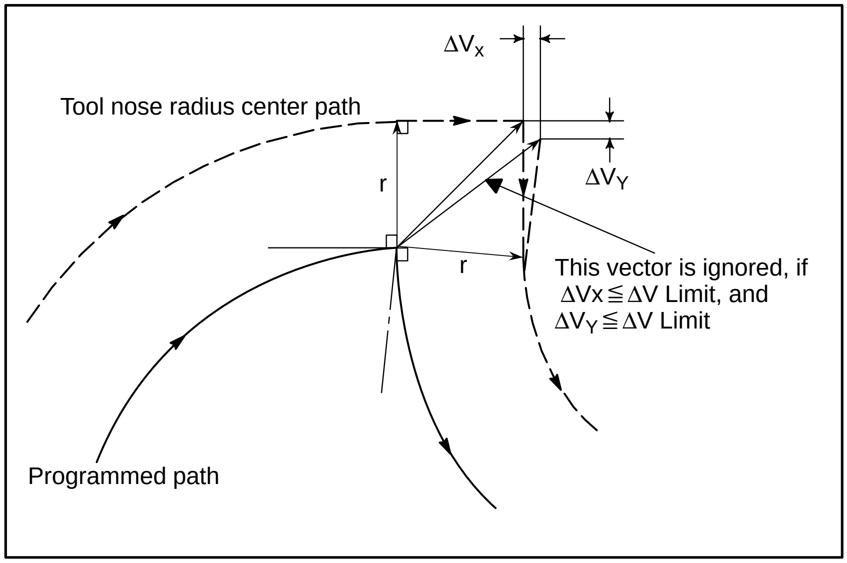
However, when the move distance is zero, even if the block is commanded singly, tool motion becomes the same as that when more than one block of without tool movement are commanded, which will be described subsequently.



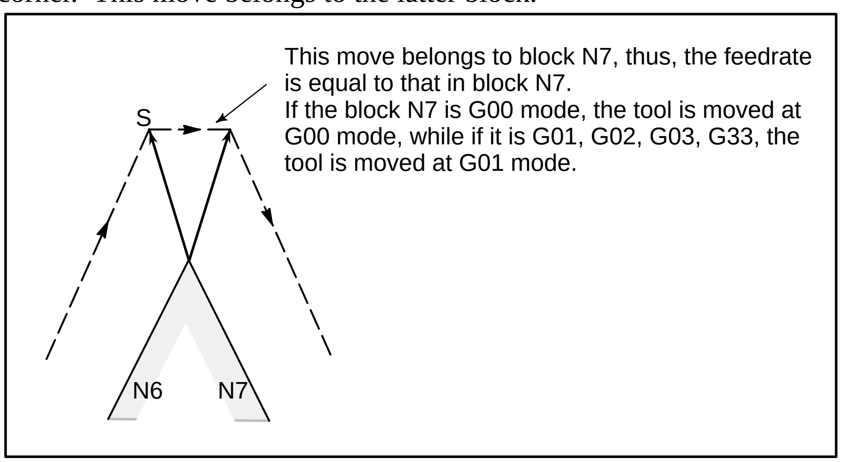
Setting bit 4 of parameter No. 395 enables the successful application of tool nose radius compensation, even when two or more blocks that do not contain tool movement are specified in succession.

- **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 ΔVx≦ΔVlimit and ΔVy≦ΔVlimit, the latter vector is ignored. The ΔVlimit is set in advance by parameter (No. 557).
If these vectors do not coincide, a move is generated to turn around the corner. This move belongs to the latter block.



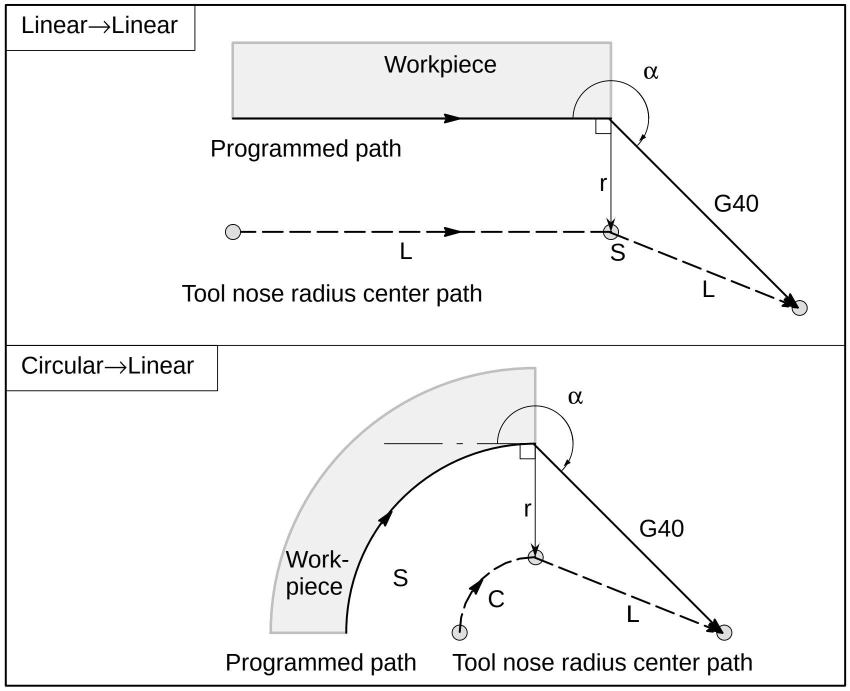
- **Interruption of manual operation**

For manual operation during the tool nose radius compensation, refer to Section III–3.5, " Manual Absolute ON and OFF."

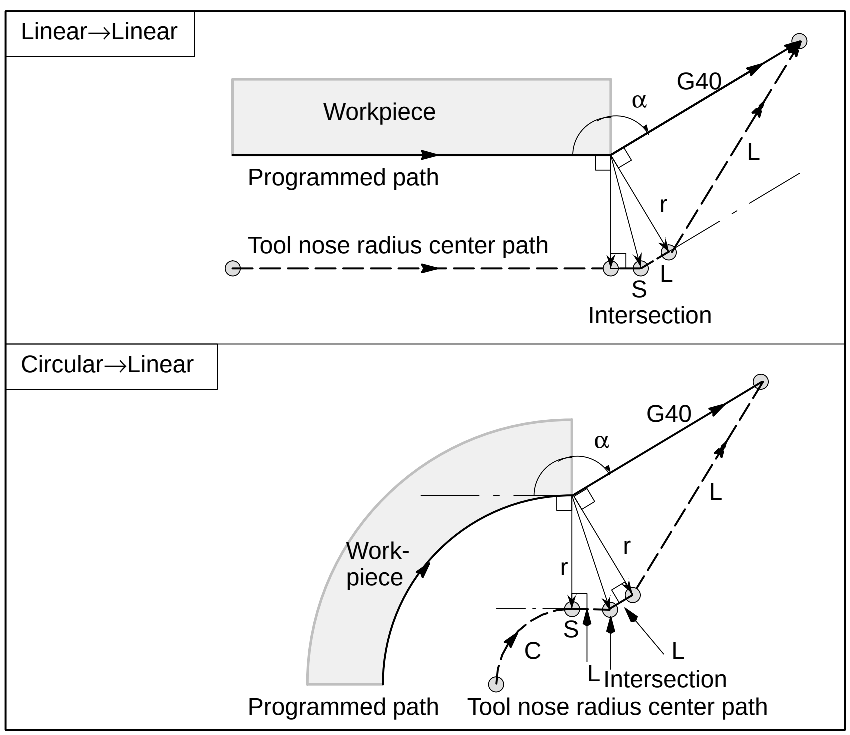
14.3.4 Tool Movement in Offset Mode Cancel

Explanations

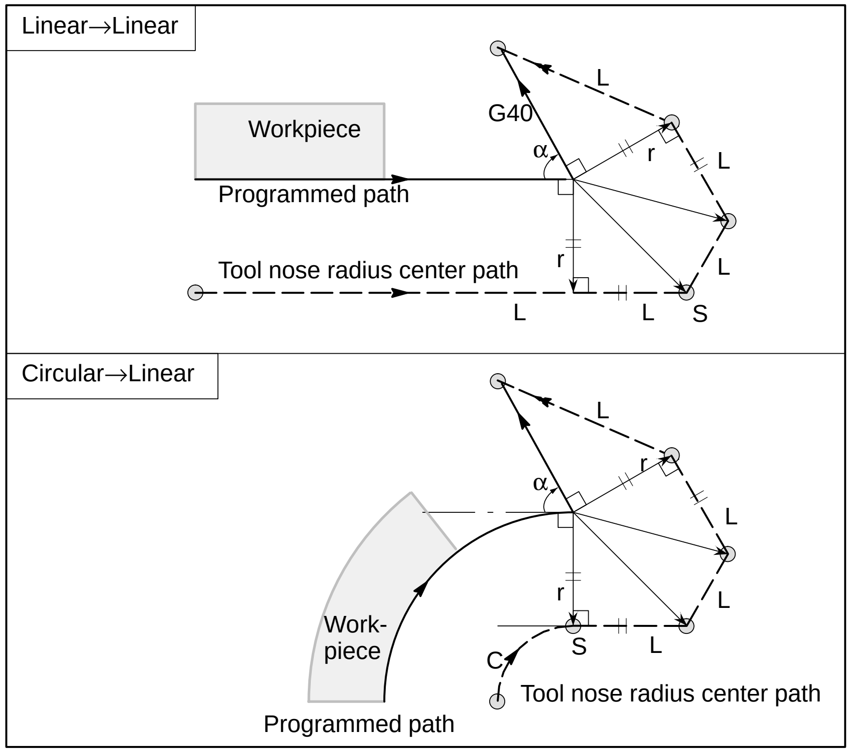
- **Tool movement around an inside corner ($180^\circ \leqq \alpha$)**



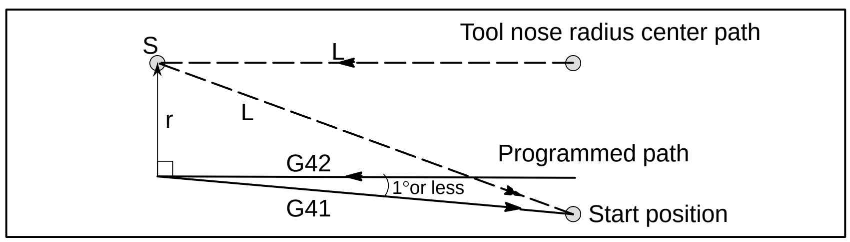
- **Tool movement around an outside corner at an obtuse angle ($90^\circ \leqq \alpha 180^\circ$)**



- **Tool movement around an outside corner at an acute angle ($\alpha < 90°$)**

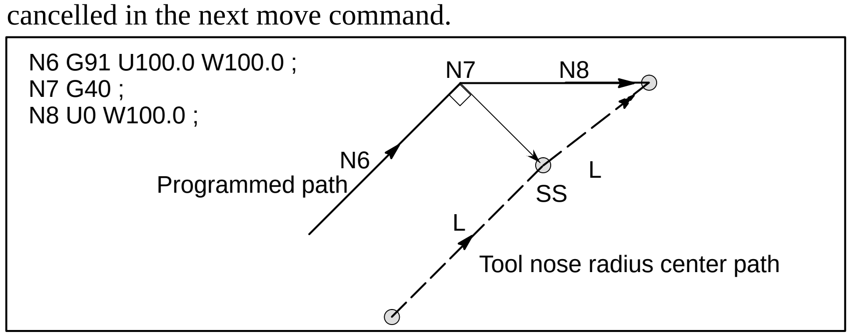


- **Tool movement around the outside linear→linear at an acute angle less than 1 degree ($\alpha < 1°$)**



- **A block without tool movement specified together with offset cancel**

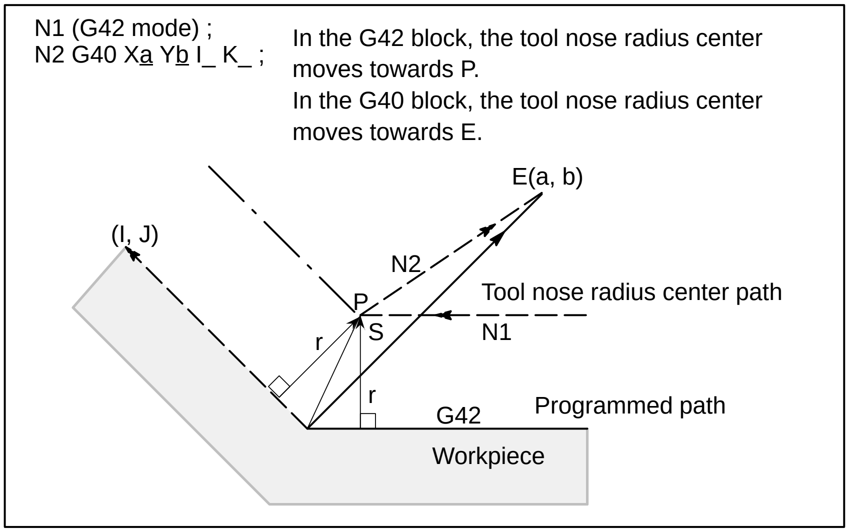
When a block without tool movement is commanded together with an offset cancel, a vector whose length is equal to the offset value is produced in a normal direction to tool motion in the earlier block, the vector is cancelled in the next move command.



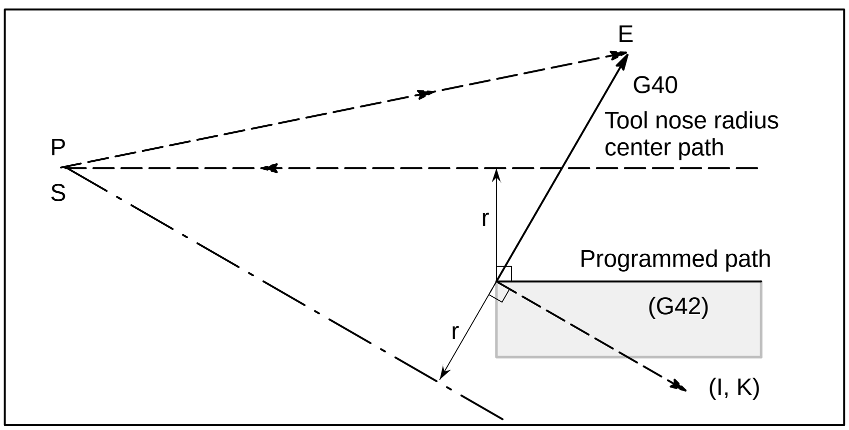
- **Block containing G40 and I_J_K_**

The previous block contains G41 or G42

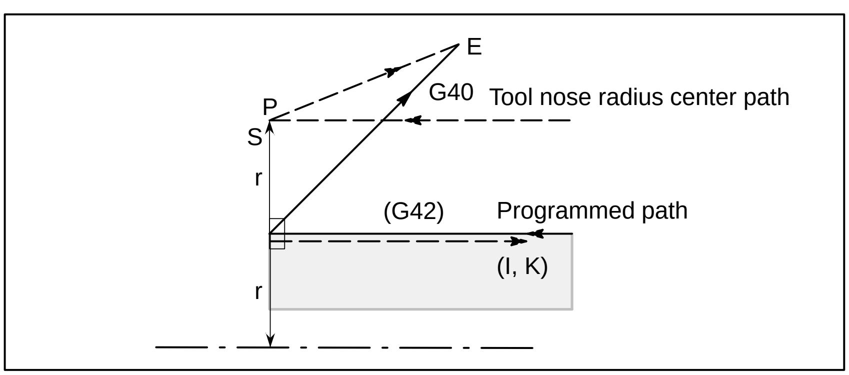
If a G41 or G42 block precedes a block in which G40 and I_, J_, K_ are specified, the system assumes that the path is programmed as a path from the end position determined by the former block to a vector determined by (I,J), (I,K), or (J,K). The direction of compensation in the former block is inherited.



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.



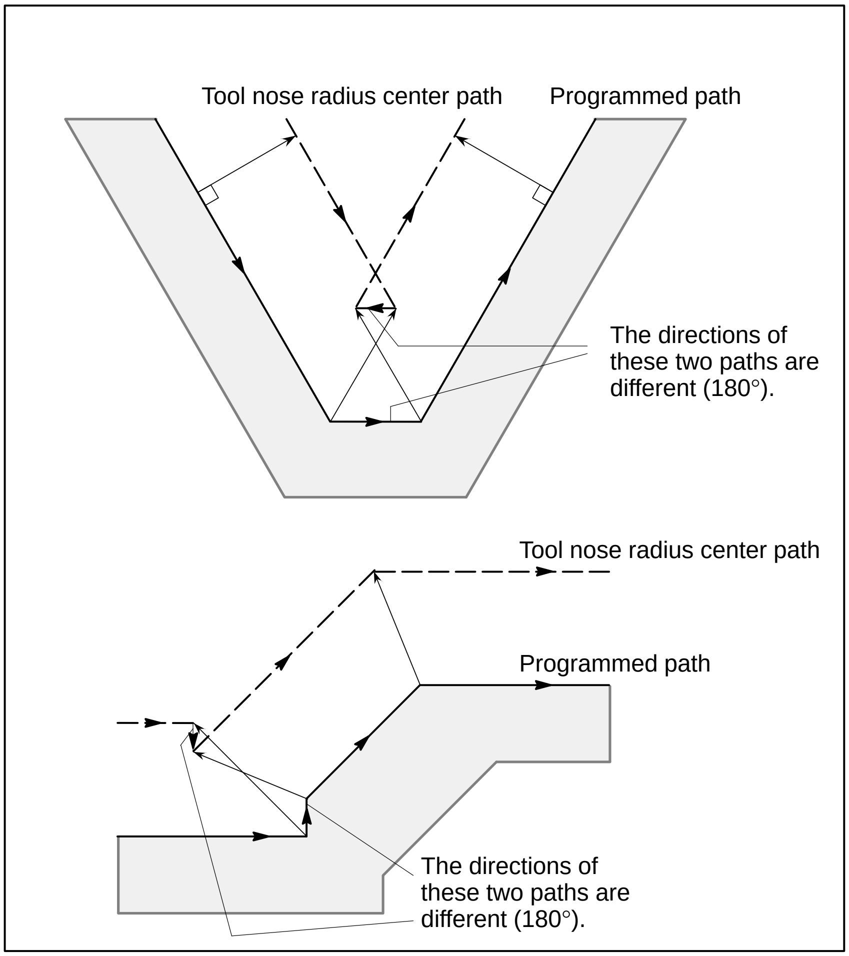
14.3.5 Interference Check

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.

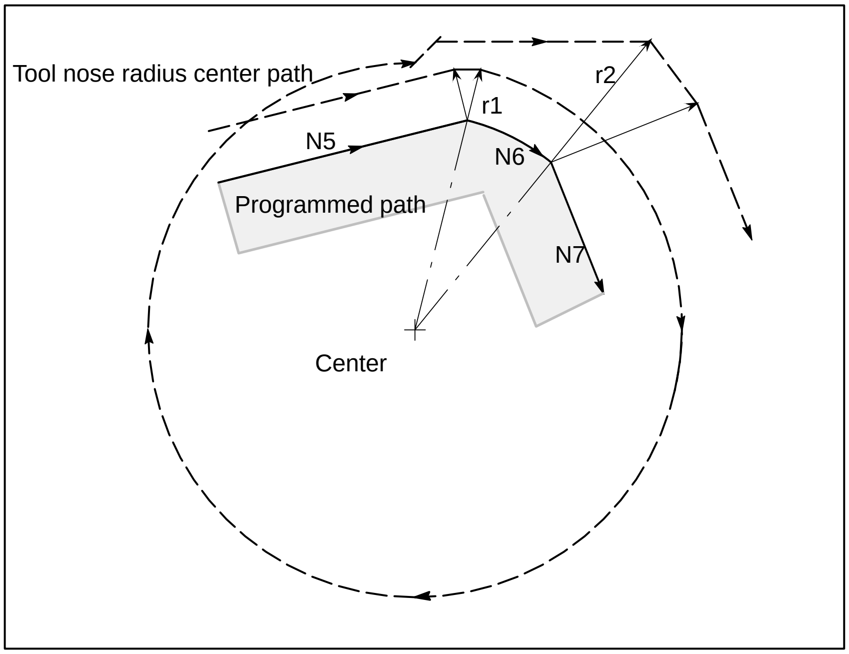
Explanations

- **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).



3 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).



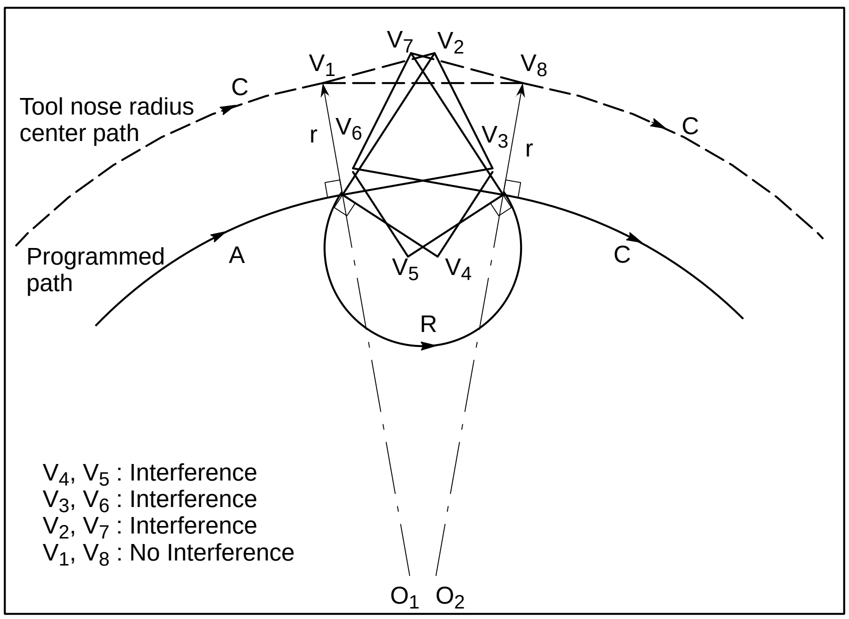
(G41)
N5 G01 U20.0 W80.0 T1 ;
N6 G02 U–16.0 W32.0 I–80.0 K–20.0 T2 ;
N7 G01 U–50.0 W20.0 ;
(Tool compensation value corresponding to T1 : r_1 = 20.0)
(Tool compensation value corresponding to T2 : r_2 = 60.0)

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.

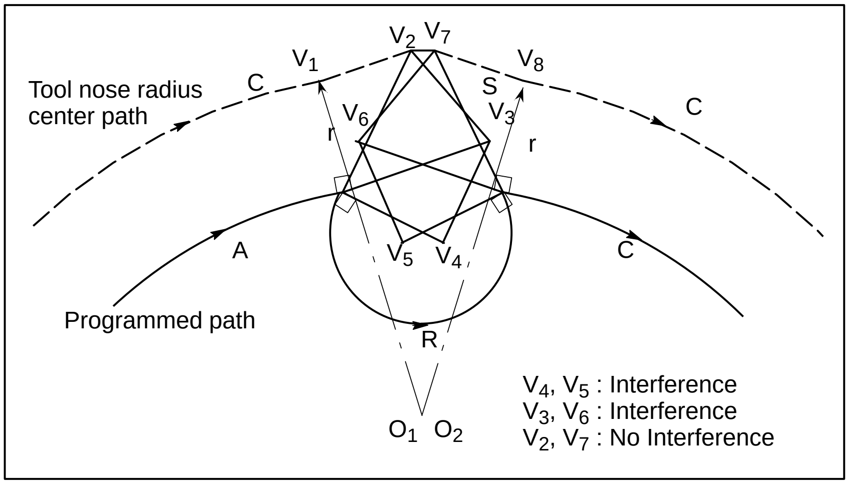
- **Correction of interference in advance**

[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

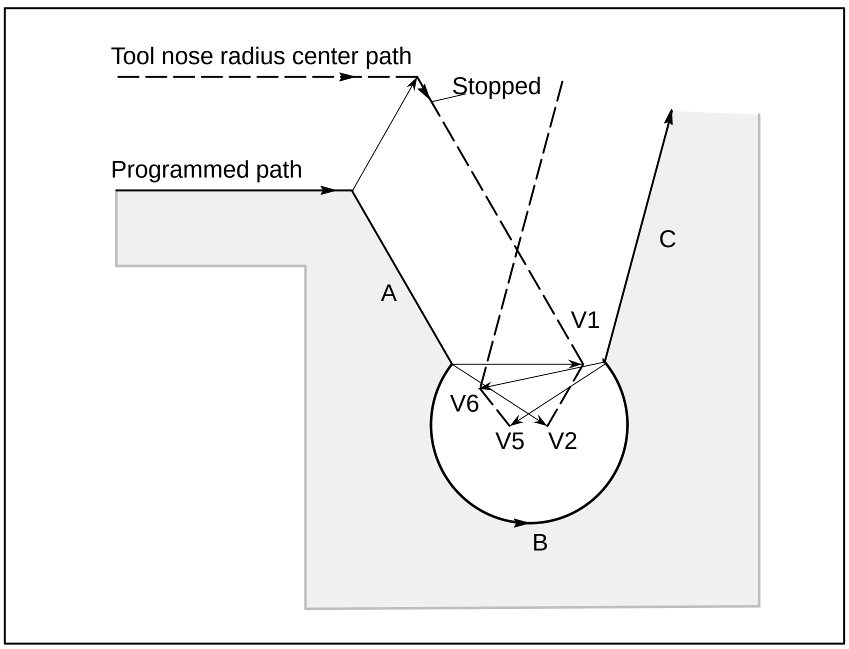


(Example 2) The tool moves linearly from V_1, V_2, V_7, to V_8



2 If the interference occurs after correction 1, the tool is stopped with an alarm.

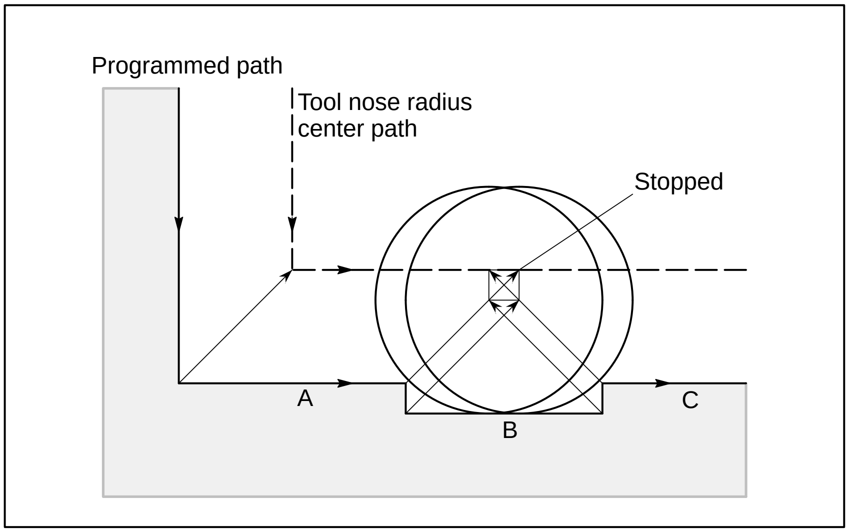
If the interference occurs after correction 1 or if there are only one pair of vectors from the beginning of checking and the vectors interfere, the alarm (No.41) is displayed and the tool is stopped immediately after execution of the preceding block. If the block is executed by the single block operation, 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.

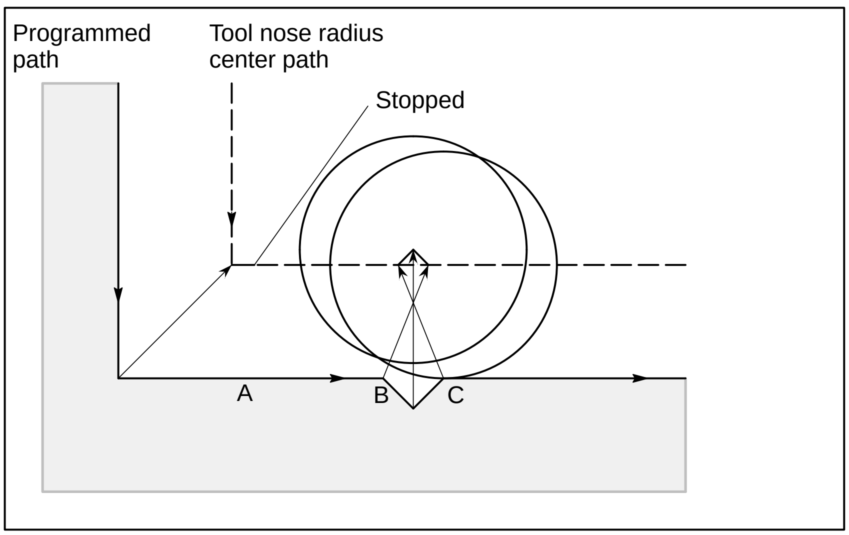
- **When interference is assumed although actual interference does not occur**

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 alarm(No.041) is displayed.

2 Groove which is smaller than the tool nose radius compensation value



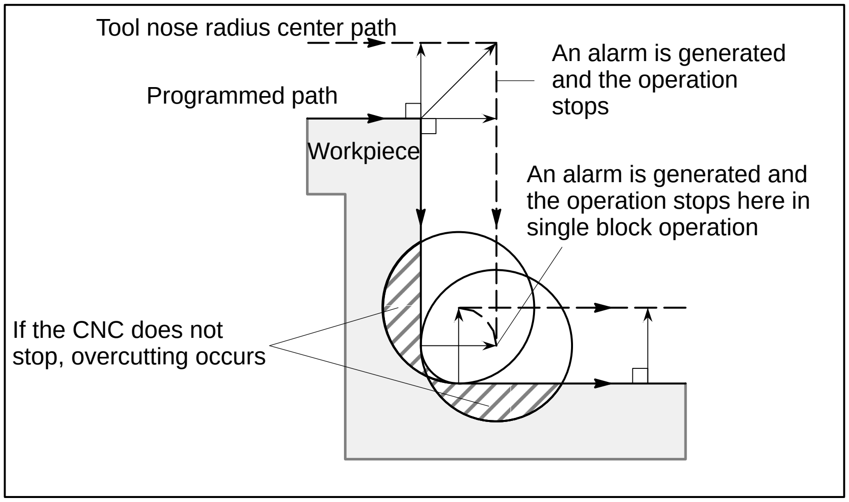
Like 1 , the direction is reverse in block B.

14.3.6 Overcutting by Tool Nose Radius Compensation

Explanations

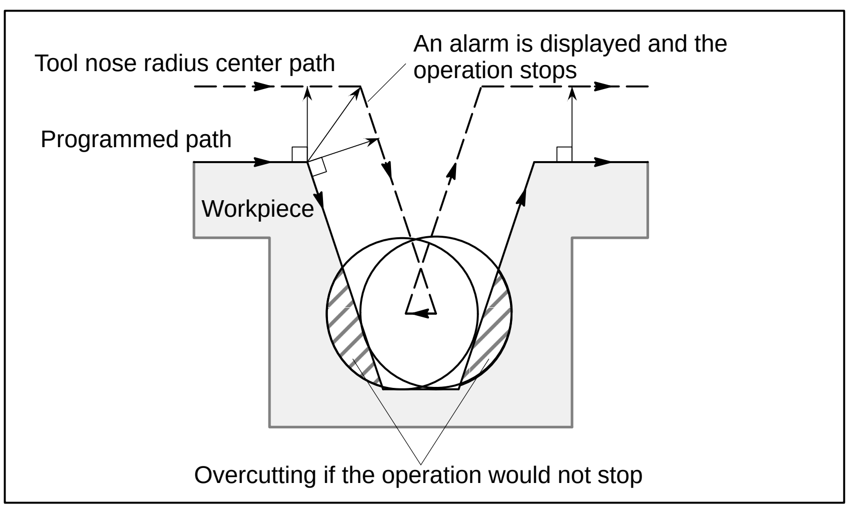
- **Machining an inside corner at a radius smaller than the tool nose radius**

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.



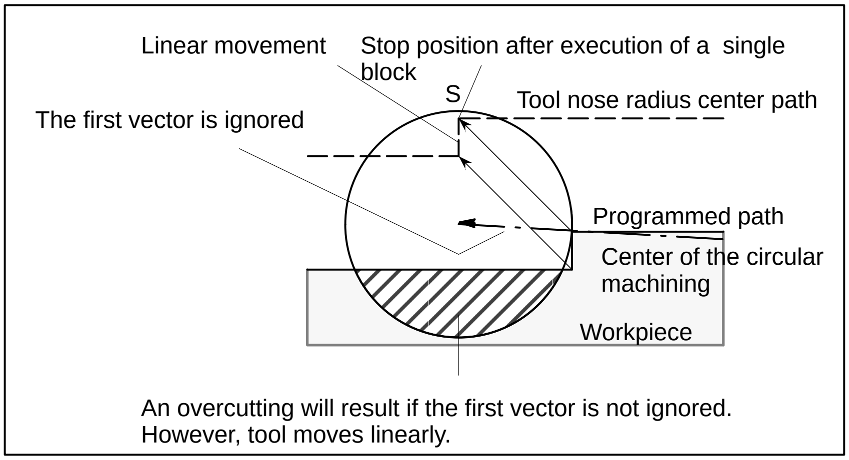
- **Machining a groove smaller than the tool nose radius**

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.



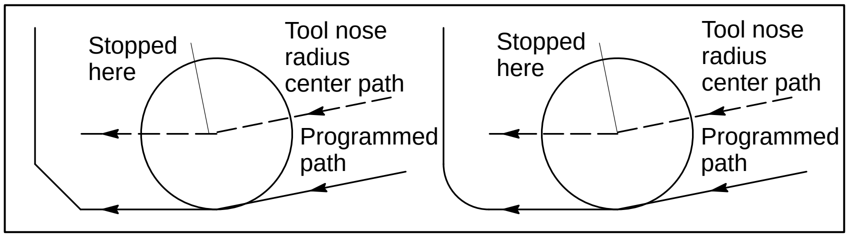
- **Machining a step smaller than the tool nose radius**

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.



14.3.7 Correction in Chamfering and Corner Arcs

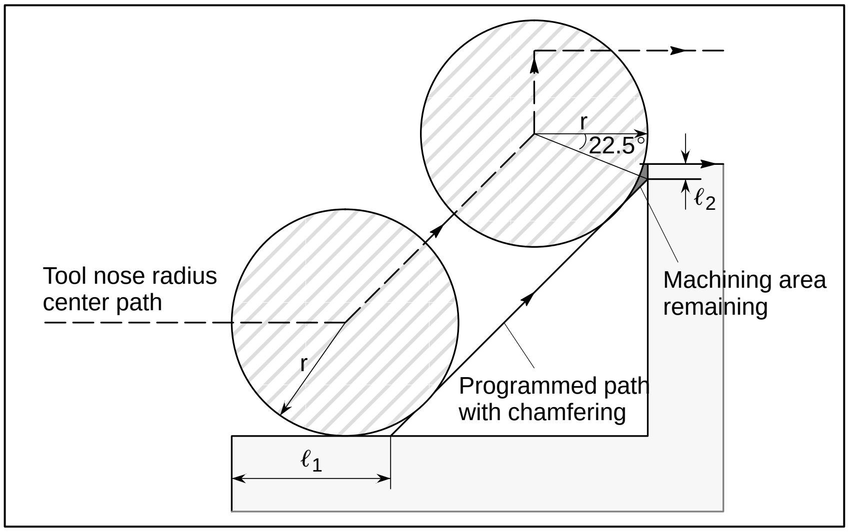
In chamfering or corner arcs, tool nose radius compensation only be performed when an ordinary intersection exists at the corner.
In offset cancel mode, a start–up block or when exchanging the offset direction, compensation cannot be performed, an alarm (No.39) is displayed and the tool is stopped.
In 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 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 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.

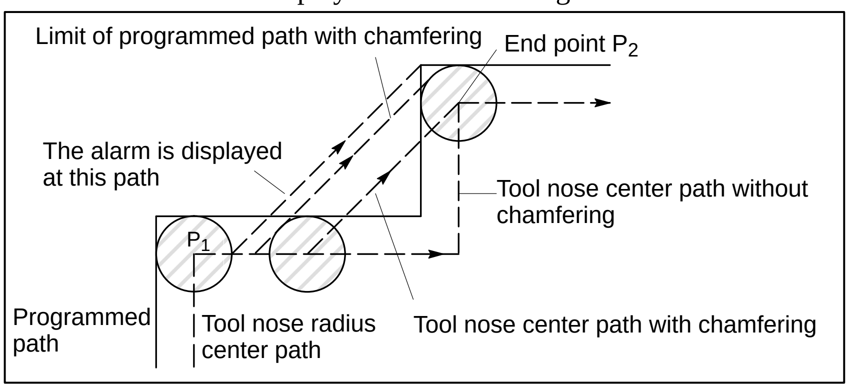


In inner chamfering, if the portion of the programmed path that is not a part of the chamfering (in the above figure ℓ_1 or ℓ_2) is in following range, insufficiently cut are will exist.

$0 \leqq \ell_1$ or $\ell_2 < r \cdot \tan 22.5°$ (r : too nose radius)

Enlarged view on the remaining machining area

Alarm No.52 or 55 is displayed in the following cases :



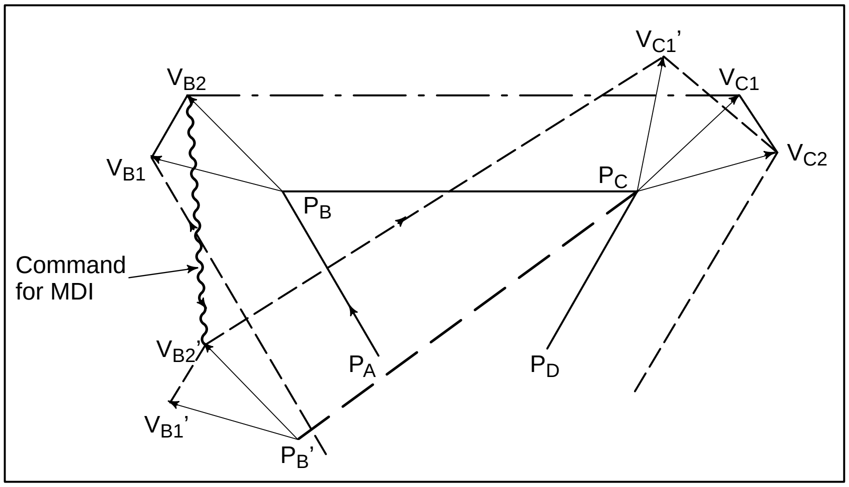
In outer chamfering with an offset, a limit is imposed on the programmed path. The path during chamfering coincides with the intersection points P_1 or P_2 without chamfering, therefore, outer chamfering is limited. In the figure above, the end point of the tool center path with chamfering coincides with the intersection point (P2) of the next block without chamfering. If the chamfering value is more than the limit value specified, alarm No.52 or 55 will be displayed.

14.3.8 Input Command from MDI

Tool nose radius compensation is not performed for commands input from the MDI.

However, when automatic operation using the CNC tape composed of 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.



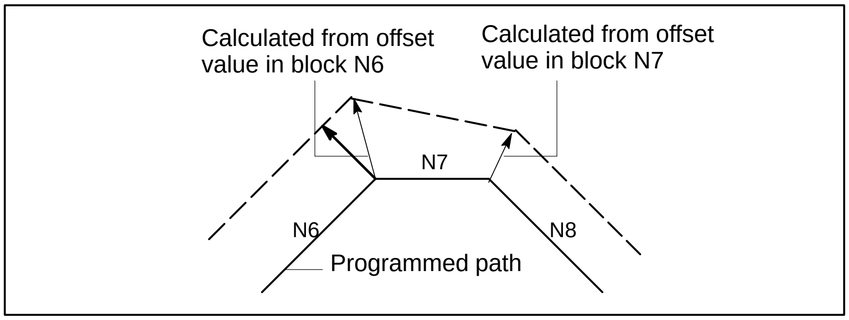
When position P_A, P_B, and P_C are programmed in an absolute command, tool is stopped by the single block function after executing the block from P_A to P_B and the tool is moved by MDI operation. Vectors V_{B1} and V_{B2} are translated to V_{B1}' and V_{B2}' and offset vectors are recalculated for 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.

14.3.9 General Precautions for Offset Operations

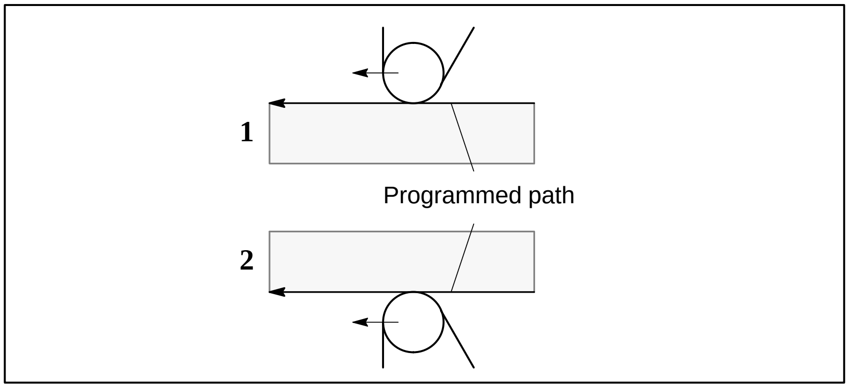
- **Changing the offset value**

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.
Addilionaly, the changing of hypothetical tool nose numer and the changing of tool position offset are same.



- **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.



> **WARNING**
> 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.

14.4 TOOL COMPENSATION 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. 14.4 (a)).
Tool compensation can be specified without differentiating compensation for tool geometry from that for tool wear (Fig.14.4.(b)).
In this case the operation of moving is same as the tool wear offset value

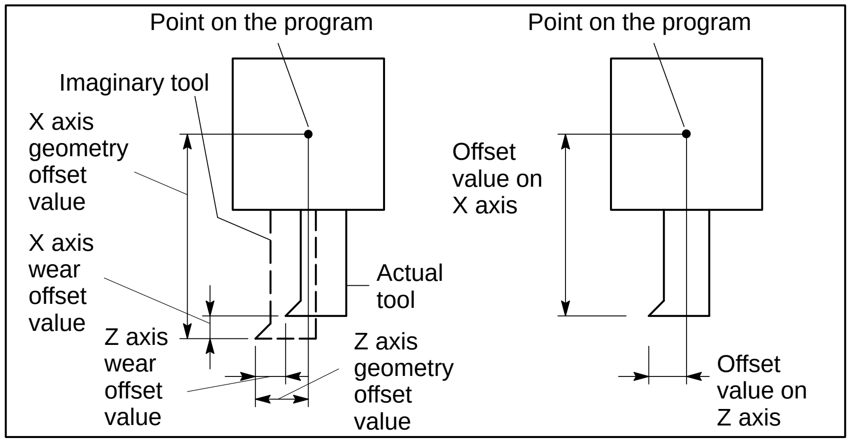


Fig. 14.4(a) Difference the tool geometry offset from tool wear offset

Fig. 14.4(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. 14.1.2 for details.

14.4.1 Tool Compensation and Number of Tool Compensation

- **Valid range of tool compensation values**

Table 14.4.1 shows the valid input range of tool compensation values.

Table 14.4.1 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.0729.

- **Number of tool compensation**

The memory can hold 16 or 32 tool compensation values.

14.4.2 Changing of Tool Offset Value

Offset values can be input by a program using the following command :

Format

G10 P_ X_ Z_ R_ Q_ ;

or

G10 P_ U_ W_ C_ Q_ ;

P : Offset number
- 0 : Command of work coordinate system shift value
- 1–32 : Command of tool wear offset value. Command value is offset number
- 1000+(1–32) : Command of tool geometry offset value. (1–32) : Offset number

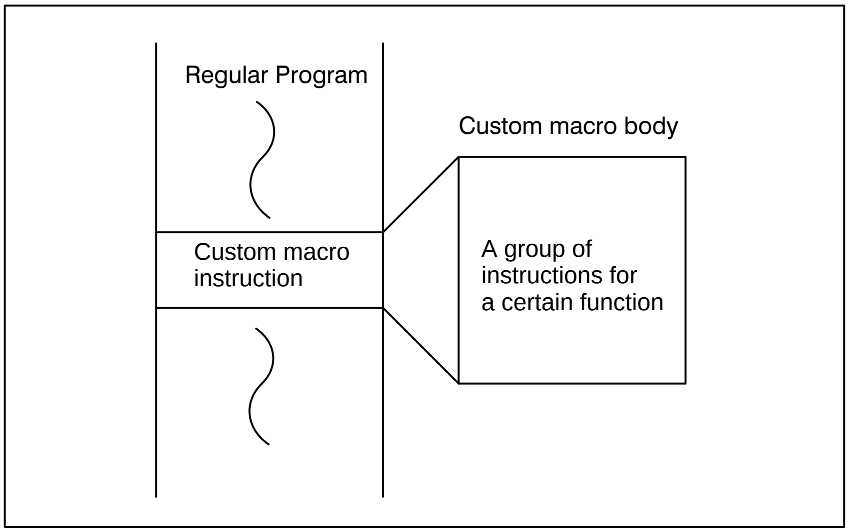
X : Offset value on X axis (absolute)
Z : Offset value on Z axis (absolute)
U : Offset value on X axis (incremental)
W : Offset value on Z axis (incremental)
R : Tool nose radius offset value (absolute)
C : Tool nose radius offset value (incremental)
Q : Imaginary tool nose number

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, W, V, and C is added to the current offset value corresponding to the offset number.

NOTE
1. Addresses X, Z and U, 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 tape by specifying this command successively instead of inputting these values one at a time from the MDI unit.
3. With P0 (workpiece coordinate system shift amount specification), only an X–/Z–axis shift can be entered.

15 CUSTOM MACRO A

A function covering a group of instructions is stored in memory as same as a subprogram. The stored function is presented by one instruction, so that only the representative instruction need be specified to execute the function. This group of registered instructions is called a ”custom macro body” and the representative instruction is called a ”custom macro instruction”. The custom macro body may simply be called a macro. And the custom macro instruction may be called a macro call command.



Programmers need only remember representative macro instructions without having to remember all the instructions in a custom macro body. The three most significant points on custom macros are that variables can be used in the custom macro body, operations can be performed on variables and actual values can be assigned to the variables in custom macro instructions.

15.1 CUSTOM MACRO COMMAND

The custom macro command is the command to call the custom macro body.

15.1.1 M98 (Single call)

Command format is as follows :

Format

M98 P__;

Called macro body program No.

With the above command, the macro body specified by P is called.

15.1.2 Subprogram Call Using M Code

The subprogram can be called using M code set in parameter.

N_ G_ X_ M98P <p> ;

instead of commanding as above, the same operation can be commanded using following command :

N_ G_ X_ M <m> ;

The correspondence of M code <m> which calls subprogram and the program number <p> (O9001 to O9003) of the called subprogram shall be set by parameters (No. 0240 to No. 0242).

For subprogram call, a maximum of 3 among M03 to M255, except M30 and M code which does not buffer (parameter No. 111, 112) can be used.

CAUTION

1. Similarly to M98, signal MF and M code are not output.
2. Delivery of argument is not possible.
3. Subprogram call M code used in the subprogram which is called by M or T code does not executes subprogram call but as an ordinal M code.

15.1.3 Subprogram Call Using T code

When parameter (No. 040 #5) is set beforehand, subprogram (O9000) can be called using T code.

N_ G_ X_ T <t> ;

the above command results in the same operation of command of the following 2 blocks.

#149 = <t> ;

N_ G_ X_ M98 P9000 ;

The T code t__ is stored in a common variable #0149 as an argument.

> **CAUTION**
> 1. It is not possible to command with a same block as that of subprogram call using M code.
> 2. Subprogram call T code used in the subprogram which is called by M or T code does not executes subprogram call but as an original T code.

15.2 CUSTOM MACRO BODY

In the custom macro body, the CNC command, which uses ordinary CNC command variables, calculation, and branch command can be used. The custom macro body starts from the program No. which immediately follows O and ends at M99.

O_____________ ;	PROGRAM NO.
G65 H01 .... ;	CALCULATION COMMAND
G90 G00 X#101 ;	CNC COMMAND USING VARIABLES
G65 H82 .... ;	BRANCH COMMAND
M99;	END OF CUSTOM MACRO

Fig. 15.2 Construction of the custom macro body

15.2.1 Variables

A variable can be specified to make the macro flexible and versatile by applying the calculated variable when calling the macro or when executing the macro itself. Multiple variables are identified from each other by variable numbers.

(1) How to express variables
Variables are expressed by variable numbers following # as shown below.
#i (i = 100, 101 .....)
(Example) #5, #109, #1005

(2) How to quote variables
A numeral following an address can be replaced by a variable. Assume that <Address> #1 or <Address> – #1 is programmed, and it means that the variable value or its complement serves as the command value of the address.
(Example)
F#103 . . . F15 was commanded when #103=15
Z–#110 . . Z–250 was commanded when #110=250
G#130 . . . G3 was commanded when #103=3.
When replacing a variable number with a variable, it is not expressed as ”##100”, for example, but express as ”#9100”. That is, ”9” next to ”#” indicates the substitute of the variable number, while the lower number to be replaced.
(Example)
If #100=105 and #105=–500, ”X#9100” indicates that X–500 was commanded, and ”X–#9100” indicates that X500 was commanded.

NOTE
1 No variable can be quoted at address O and N. Neither O#100 nor N#120 can be programmed.
2 It is not possible to command a value exceeding the maximum command value set in each address. When #30=120, G#30 has exceeded the maximum command value.

(3) Display and setting of variable values
Variable values can be displayed on the CRT screen, and a value can be set in a variable by using the MDI keys.

15.2.2 Kind of Variables

Variables are sorted into common variables and system variables according to variable numbers, and their applications and characters differ from each other.

(1) Common variable #100 to #149 and #500 to #531
Common variables are common to main programs and each macro called from these main programs. That is, #i in a macro is equal to #i in another macro.
Common variables #100 to #149 are cleared when the power is turned off, and reset to ”0” just after power was turned on. Common variables #500 to #531 are not cleared, even if power is turned off, and their values remain unchanged.

(2) System variable
The system variables are defined as variables whose applications remain fixed.

(a) Interface input signals #1000 to #1015, #1032
Interface signals can be known, by reading system variables #1000 to #1015 for reading interface signals.

2^{15}	214	2^{13}	212	2^{11}	210	2^9	28	2^7	26	2^5	24	2^3	22	2^1	20
UI15	UI14	UI13	UI12	UI11	UI10	UI9	UI8	UI7	UI6	UI5	UI4	UI3	UI2	UI1	UI0
#1015	#1014	#1013	#1012	#1011	#1010	#1009	#1008	#1007	#1006	#1005	#1004	#1003	#1002	#1001	#1000

Input signal	Variable value
Contact closed	1
Contact opened	0

By reading system variable #1032, all the input signals can be read at once.

$$\#1032=\sum_{i=0}^{15}\#(1000+i)\times 2^{i}$$

NOTE
1 No value can be substituted into system variables #1000 to #1032.
2 System variables #1000 to #1015 can be displayed by diagnostic function.
No.130 U10 to U17
No.131 U18 to U15
3 System variables #1000 to #1032 can be used only when PMC is combined.

(b) Interface output signals #1100 to #1115, #1132, #1133
A value can be substituted into system variables #1100 to #1115 for sending the interface signals.

2^{15}	214	2^{13}	212	2^{11}	210	2^{9}	28	2^{7}	26	2^{5}	24	2^{3}	22	2^{1}	20
UO15	UO14	UO13	UO12	UO11	UO10	UO9	UO8	UO7	UO6	UO5	UO4	UO3	UO2	UO1	UO0
#1115	#1114	#1113	#1112	#1111	#1110	#1109	#1108	#1107	#1106	#1105	#1104	#1103	#1102	#1101	#1100

Output signal	Variable value
Contact closed	1
Contact open	0

By substituting a value into system variable #1132, all output signals (UO0 to UO15) can be sent out at once.

$$\#1132=\sum_{i=0}^{15} \#(1100+i)\times 2^{i}$$

32 INTERFACE SIGNALS (UO100 TO UO131) CAN BE SENT OUT BY #1133 AT ONCE.

CAUTION
If any other number than '0' or '1' is substituted into system variables #1100 to #1115, it is treated as '1'.

NOTE
1 It is possible to read the values of system variables #1100 to #1133.
2 System variables #1100 to #1115 and #1133 can be displayed by diagnostic function.

DGNOS	No.162	UO0 to UO7
	No.163	UO8 to UO15
	No.196	UO100 to UO107
	No.197	UO108 to UO115
	No.198	UO116 to UO123
	No.199	UO124 to UO131

3 System variables #1100 to #1133 can be used only when PMC is combined.

(c) Tool offset values #2001 to #2932
Offset values can be checked from the values of system variables #2001 to #2932, used to hold tool offset values. By assigning a value to system variable #i, an offset value can be modified.
A tool position offset for the X–axis is handled as a radius when the offset is determined from a specified radius. A tool position offset forthe X–axis is handled as a diameter when the offset is determined from a specified diameter.

	Tool offset number	Tool offset value	Wear offset value	Geometry offset value
X	1 to 32	#2001 to #2032	#2001 to #2032	#2701 to #2732
Z	1 to 32	#2101 to #2132	#2101 to #2132	#2801 to #2832
R	1 to 32	#2201 to #2232	#2201 to #2232	#2901 to #2932
T	1 to 32	#2301 to #2332	#2301 to #2332	#2301 to #2332

(Example) #103=#2005
The X–axis tool position offset value of offset number 5 is assigned to variable #103. When the offset is 1.5 mm, the value of #103 is 1500.

(d) Clock information #3011, 3012 (option)
It is possible to know the year, month, day, hour, minute, and second by reading system variables #3011, #3012.

Kind	System variable
Year, Month, Day	#3011
Hour, Minute, Second	#3102

(Example) When it is May 20, 1993 4:17 5” PM
#3012=19930520,
#3012=161705

(e) Number of necessary parts, number of machined parts
By using system variables, the number of parts required and the number of parts machined can be read and assigned.

Kind	System variable
Number of machined parts	#3901
Number of necessary parts	#3902

(f) Modal information #4001 to #4120
It is possible to know the current values of modal information (modal command given till immediately preceding block) by reading values of system variables #4001 to #4120.

NOTE
Do not substifute a negative value.

Variables	Modal information
#4001	G CODE (GROUP 01)
#4002	G CODE (GROUP 02)
#4003	G CODE (GROUP 03)
:	
#4022	G CODE (GROUP 22)
#4109	F CODE
#4113	M CODE
#4114	SEQUENCE NO.
#4115	PROGRAM NO.
#4119	S CODE
#4120	T CODE

NOTE
The unit will be the one being used when the command is given.

(g) Position information #5001 to #5124

The position information can be known by reading system variables #5001 to #5124. The unit of position information is 0.001 mm in metric input and 0.0001 inch in inch input.

System variables	Position information	Reading while moving	Cutter and tool length compensation
#5001 #5002 #5003	Block end point position of X axis Block end point position of z axis Block end point position of 3rd axis (C axis)	Possible	Not considered. Program command position
#5021 #5022 #5023	X axis coordinate position Z axis coordinate position 3rd axis (Cf axis) coordinate position	Impossible	Considered. Position of tool reference point (Machine coordinate)
#5041 #5042 #5043	Present position of X axis Present position of Z axis Present position of 3rd axis (Cf axis)	Impossible	Considered. Position of tool reference point
#5061 #5062 #5063	Skip signal position of X axis Skip signal position of Z axis Skip signal position of 3rd axis (Cf axis)	Possible	Considered. Position of tool reference point
#5081 #5082	Tool position offset amount or wear offset amount of X axis Tool position offset amount or wear offset amount of Z axis	impossible	
#5121 #5122	Graphic offset amount of X axis Graphic offset amount of Z axis	impossible	

15.2.3 Operation Instruction and Branch Instruction (G65)

General format:

G65HmP#i Q#j R#k ;

m : 01 to 99. An operation instruction or branch instruction function is represented.

#i : Name of variable used to hold the result of an operation

#j : Name of variable on which an operation is to be performed.
(A constant can also be specified.)

#k : Name of variable on which an operation is to be performed.
(A constant can also be specified.)

(Meaning) #i=#j ⊕ #k

Specified using the Hm operator

Example

P#100 Q#101 R#102 . . . #100=#101⊕#102
P#100 Q#101 R15 . . . . . #100=#101⊕15
P#100 Q–100 R#102 . . . #100=–100⊕#102
P#100 Q120 R–50 . . . . . #100=120⊕–50
P#100 Q–#101 R#102 . . #100=–#101⊕#012

CAUTION

1 No decimal point can be put to variable values. Therefore, the meaning of each value is the same as that designated without decimal point when quoted in each address.
(Example) #100 = 10
X#100 0.01 mm (metric input)

2 Those indicating an angle must be expressed by degree, and input increment is 1/1000 degree.
(Example) 100 - - 0.1°

Table 15.2.3

G code	H code	Function	Definition
G65	H01	Definition, substitution	#i = #j
G65	H02	Addition	#i = #j + #k
G65	H03	Subtraction	#i = #j – #k
G65	H04	Product	#i = #j × #k
G65	H05	Division	#i = #j ÷ #k
G65	H11	Logical sum	#i = #j. OR. #k
G65	H12	Logical product	#i = #j. AND. #k
G65	H13	Exclusive OR	#i = #j. XOR. #k
G65	H21	Square root	$\#i = \sqrt{\#j}$
G65	H22	Absolute value	#i = | #j |

Table 15.2.3

G code	H code	Function	Definition
G65	H23	Remainder	#i = #j – trunc (#j / #k) $\times$ #k (trunc : Discard fractions less than 1)
G65	H24	Conversion from BCD to binary	#i = BIN (#j)
G65	H25	Conversion from binary to BCD	#i = BCD (#j)
G65	H26	Combined multiplication/division	#i = (#i $\times$#j) $\div$ #k
G65	H27	Combined square root 1	$\#i = \sqrt{\#J^2 + \#K^2}$
G65	H28	Combined square root 2	$\#i = \sqrt{\#J^2 - \#K^2}$
G65	H31	Sine	#i = #j · SIN (#k)
G65	H32	Cosine	#i = #j · COS (#k)
G65	H33	Tangent	#i = #j · TAN (#k)
G65	H34	Arctangent	#i = ATAN (#j / #k)
G65	H80	Unconditional divergence	GOTOn
G65	H81	Conditional divergence 1	IF#j = #k, GOTOn
G65	H82	Conditional divergence 2	IF#j $\neq$ #k, GOTOn
G65	H83	Conditional divergence 3	IF#j > #k, GOTOn
G65	H84	Conditional divergence 4	IF#j < #k, GOTOn
G65	H85	Conditional divergence 5	IF#j $\geqq$ #k, GOTOn
G65	H86	Conditional divergence 6	IF#j $\leqq$ #k, GOTOn
G65	H99	P/S alarm occurrence	P/S alarm number 500 +n occurrence

- **Variable arithmetic command**

(a) Definition and substitution of variable #i = #j
G65 H01 P#i Q#j ;
[Example] G65 H01 P#101 Q1055 ; (#101=1005)
G65 H01 P#101 Q#110 ; (#101=#110)
G65 H01 P#101 Q–#112 ; (#101=–#112)

(b) Addition #i = #j + #k
G65 H02 P#i Q#j R#k;
[Example] G65 H02 P#101 Q#102 R15 ; (#101=#102+15)

(c) Subtraction #i = #j – #k
G65 H03 P#i Q#j R#k;
[Example] G65 H03 P#101 Q#102 R#103 ; (#101=#102–#103)

(d) Product #i = #j $\times$#k
G65 H04 P#i Q#j R#k;
[Example] G65 H04 P#101 Q#102 R#103 ; (#101=#102$\times$#103)

(e) Division #i = #j ÷#k
G65 H05 P#i Q#j R#k;
[Example] G65 H05 P#101 Q102 R#103 ; (#101=#102÷#103)

(f) Logical sum #i = #j.OR.#k
G65 H11 P#i Q#j R#k;
[Example] G65 H11 P#101 Q102 R#103 ; (#101=#102.OR.#103)

(g) Logical product #i = #j.AND.#k
G65 H12 P#i Q#j R#k ;
[Example] G65 H12 P#101 Q#102 R#103 ; (#101=#102.AND.#103)

(h) Exclusive OR #i = #j.XOR.#k
G65 H13 P#i Q#j R#k ;
[Example] G65 H13 P#101 Q#102 R#103 ; (#101=#102.XOR.#103)

(i) Square root #i = $\sqrt{\#j}$
G65 H21 P#i Q#j ;
[Example] G65 H21 P#101 Q#102 ; (#101= $\sqrt{\#102}$)

(j) Absolute value #i = |#j|
G65 H22 P#i Q#j ;
[Example] G65 H22 P#101 Q#102 ; (#101=|#102|)

(k) Remainder #i = #j – trunc (#j/#k)×#k
trunc : Discard fractions less than 1
G65 H23 P#i Q#j R#k ;
[Example] G65 H23 P#101 Q#102 R#103 ;
(#101=#102–trunc (#102/#103)×#103)

(l) Conversion from BCD to binary #i = BIN (#j)
G65 H24 P#i Q#j ;
[Example] G65 H24 P#101 Q#102 ; (#101=BIN (#102))

(m) Conversion from binary to BCD #i = BCD (#j)
G65 H25 P#i Q#j ;
[Example] G65 H25 P#101 Q#102 ; (#101=BCD (#102))

(n) Combined multiplication/division #i = (#1×#j) ÷ #k
G65 H26 P#i Q#j R#k ;
[Example] G65 H26 P#101 Q#102 R#103 ;
(#101=(#101×#102)(÷#103)

(o) Combined square root 1 #i = $\sqrt{\#j^2 + \#k^2}$
G65 H27 P#i Q#j R#k ;
[Example] G65 H27 P#101 Q#102 R#103 ;
(#101= $\sqrt{\#102^2 + \#103^2}$)

(p) Combined square root 2 #i = $\sqrt{\#j^2 - \#k^2}$
G65 H28 P#i Q#j R#k ;
[Example] G65 H28 P#101 Q#102 R#103 ;
(#101= $\sqrt{\#102^2 - \#103^2}$)

(q) Sine #i = #j × SIN (#k) (degree unit)
G65 H31 P#i Q#j R#k ;
[Example] G65 H31 P#101 Q#102 R#103 ; (#101=#102×SIN(#103)

(r) Cosine #i = #j × COS (#k) (degree unit)
G65 H32 P#i Q#j R#k ;
[Example] G65 H32 P#101 Q#102 R#103 ;
(#101=#102 × COS (#103)

(s) Tangent #i = #j × TAN (#k) (degree unit)
G65 H33 P#i Q#j R#k ;
[Example] G65 H33 P#101 Q#102 R#103 ;
(#101=#102 × TAN (#103)

(t) Arctangent #i = ATAN (#j/#k) (degree unit)
G65 H34 P#i Q#j R#k ; ($0° \leqq$ #i $< 360°$)
[Example] G65 H34 P#101 Q#102 R#103 ;
(#101=ATAN (#102 / #103))

CAUTION

Angle in (q) to (t) must be indicated by degree and the least input increment is 1/1000 degree.

NOTE

1 If either Q or R necessary for each arithmetic operation was not indicated, its value is calculated as ’0’.
2 All figures below decimal point are truncated if each arithmetic result includes decimal point.

(a)Unconditional branch
G65 H80 Pn ; n : Sequence number
[Example] G65 H80 P120 ; (Diverge to N120)

(b) Conditional divergence 1 (#j=#k)
G65 H81 Pn Q#j R#k ; n : Sequence number
[Example] G65 H81 P1000 Q#101 R#102 ;
#101=#102, go to N1000
#101≠#102, go to next

(c) Conditional divergence 2 (#j≠#k)
G65 H82 Pn Q#j R#k ; n : Sequence number
[Example] G65 H82 P1000 Q#101 R#102 ;
#101≠#102, go to N1000
#101=#102, go to

(d) Conditional divergence 3 (#j>#k)
G65 H83 Pn Q#j R#k ; n : Sequence number
[Example] G65 H83 P1000 Q#101 R#102 ;
#101 > #102, go to N1000
#101 ≦ #102, go to next

(e) Conditional divergence 4 (#j<#k)
G65 H84 Pn Q#j R#k ; n : Sequence number
[Example] G65 H84 P1000 Q#101 R#102 ;
#101 < #102, go to N1000
#101 ≧ #102, go to next

(f) Conditional divergence 5 (#j≧#k)
G65 H85 Pn Q#j R#k ; n : Sequence number
[Example] G65 H85 P1000 Q#101 R#102 ;
#101 ≧ #102, go to N1000
#101 < #102, go to next

(g) Conditional divergence 6 (#j≦#k)
G65 H86 Pn Q#j R#k ; n : Sequence number
[Example] G65 H86 P1000 Q#101 R#102 ;
#101 ≦ #102, go to N1000
#101 > #102, go to next

(h) P/S alarm occurrence
G65 H99 Pn ; Alarm No. : 500+n
[Example] G65 H99 P15 ; P/S alarm 515 occurrence

NOTE

1. If positive numbers were designated as sequence numbers at branch designations, they are searched forward first and then, backward. If negative numbers were designated, they are searched backward first and then, forward.
2. Sequence number can also be designated by variables.
   (Example) G65 H81 P#100 Q#101 R#102 ;
   When conditions are satisfied, processing branches to the block having the sequence number designated with #100.

15.2.4 Warning and Notes on Custom Macro

WARNING

Since an integer only is employable as the variable value, in case the operation results with decimal numbers, the figures below decimal point truncated, if an arithmetic result contains a fraction part.

Particularly be careful with the arithmetic sequence, accordingly.

[Example]

When #100=35, #101=10, #102=5, the following results.
#110=#100 ÷ #101 (=3)
#111=#110 × #102 (=15)
#120=#100 × #102 (=175)
#121=#120 ÷ #101 (=17)
#111=15 and #121=17

NOTE

1 How to input ”#”
When ” /# EOB” key is depressed after
address, # code is input

2 It is also possible to give a macro instruction in the MDI mode. However address data other than G65 are not displayed by keying operation.

3 Address H, P, Q and R of The operation and branch instructions must always be written after G65. Address O and N only are writable before G65.
H02 G65 P#100 Q#101 R#102 ; . . . Error
N100 G65 H01 P#100 Q10 ; . . . . . . Correct

4 Single block
Generally, the The operation and branch instructions block does not stop even if single block stop is turned on. However, by setting parameter No. 011#5, it is possible to make single block effective.
This is used for macro testing.

5 Variable values can be taken within a range of -2^{31} to $2^{31}-1$, but they are not displayed correctly, except for –99999999 to 99999999. If they exceed the above range, they are displayed
as *********.

6 It is possible to nest subprograms up to four times.

7 When a custom macro is loaded from a paper tape in the EIA code, ‘&’ code is treated as ’#’, because there is no ’#’ code in the EIA code.

16 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_P_;	For parameters entry
:	
G11;	Parameter entry mode cancel

Meaning of command	
N_:	Parameter No. (4digids)
P_:	Parameter setting value (Leading zeros can be omitted.)

Explanations

- **Parameter setting value (P_)**

Do not use a decimal point in a value set in a parameter (P_).
a decimal point cannot be used in a custom macro variable for P_either.

NOTE

1. Do not fail to perform reference point return manually after changing the pitch error compensation data or backlash compensation data.
2. Other NC statements cannot be specified while in parameter input mode.
3. The canned–cycle mode must be cancelled before entering of parameters. When not cancelled, the drilling motion will be activated.
4. Parameter entry (G10) cannot be specified during axis movement based on PMC axis control.

17 ROTARY AXIS ROLL–OVER

General

The roll–over function prevents coordinates for the rotation axis from overflowing. The roll–over function is enabled by setting bit 1 of parameter 388 to 1.

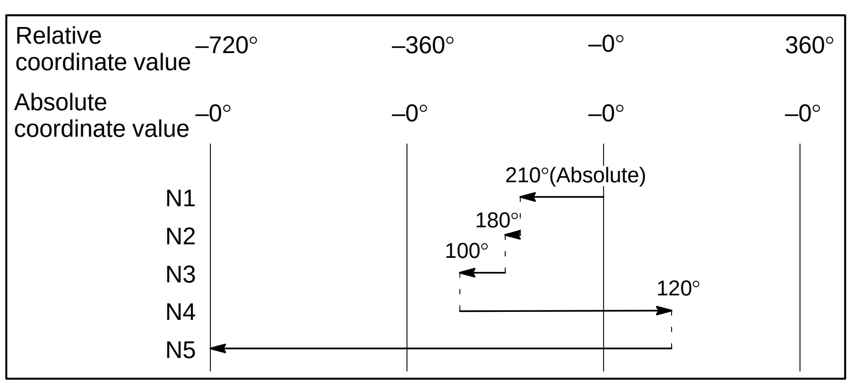
Explanations

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.788, and rounded by the angle corresponding to one rotation. The tool moves in the direction in which the final coordinates are closest when bit 2 of parameter No.388 is set to 0. Displayed values for relative coordinates are also rounded by the angle corresponding to one rotation when bit 3 of parameter No.388 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.788 = 360000). When the following program is executed using the roll–over function of the rotating axis, the axis moves as shown below.

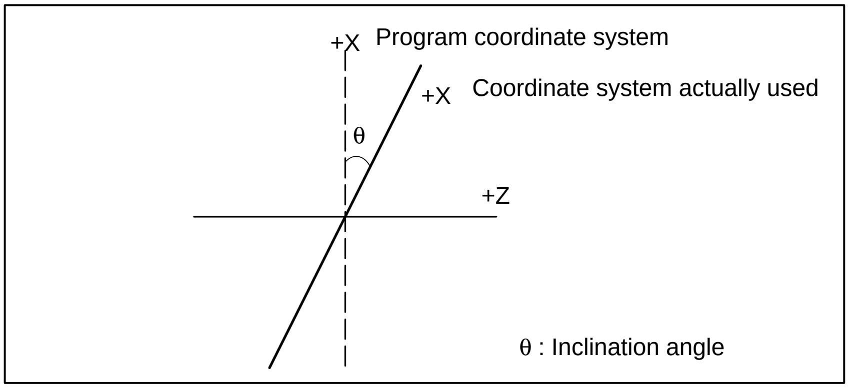
C0 ;	Sequence number	Actual movement value	Absolute coordinate value after movement end
N1 C–150.0 ;	N1	–150	210
N2 C540.0 ;	N2	–30	180
N3 C–620.0 ;	N3	–80	100
N4 H380.0 ;	N4	+380	120
N5 H–840.0 ;	N5	–840	0



18 ANGULAR AXIS CONTROL (0–GCD)

General

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 (No.036#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.755).
Parameter (No.036#2) enables X–axis manual reference point return only with a distance along the X–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.

WARNING

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 05 or +905 is set, an error can occur. A range from +205 to +605 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.

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 which is machine's own.
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.

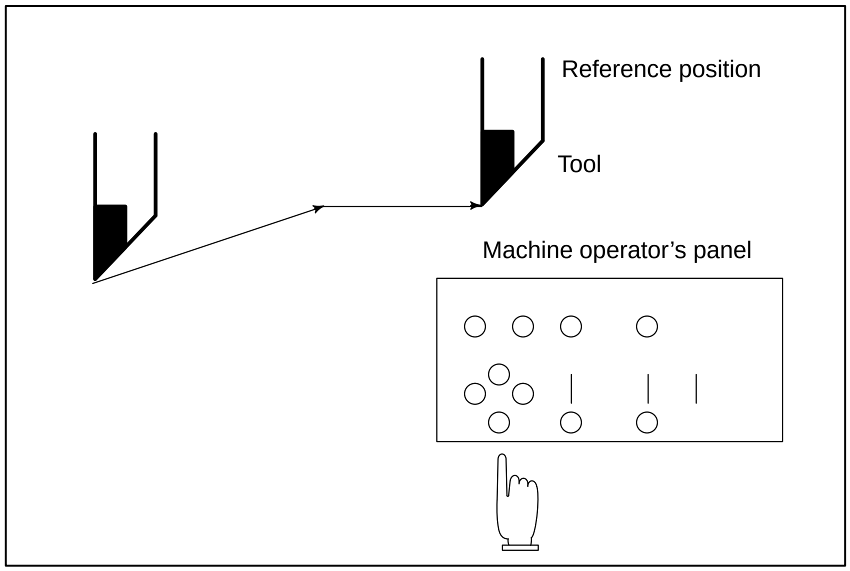


Fig. 1.1 (a) Manual reference position return

The tool can be moved to the reference position also with program commands.
This operation is called automatic reference position return (See Chapter 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.

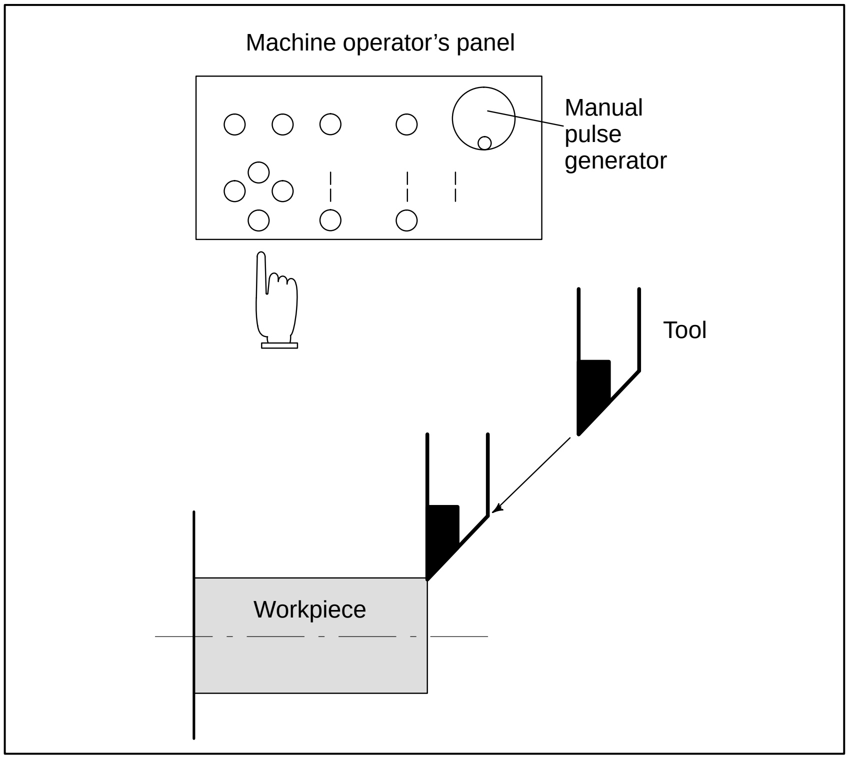


Fig. 1.1 (b) The tool movement by manual operation

The tool can be moved in the following ways:

(i) Jog feed (See Section III–3.2)
The tool moves continuously while a pushbutton 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, DNC and MDI operations. (See Chapter III–4).

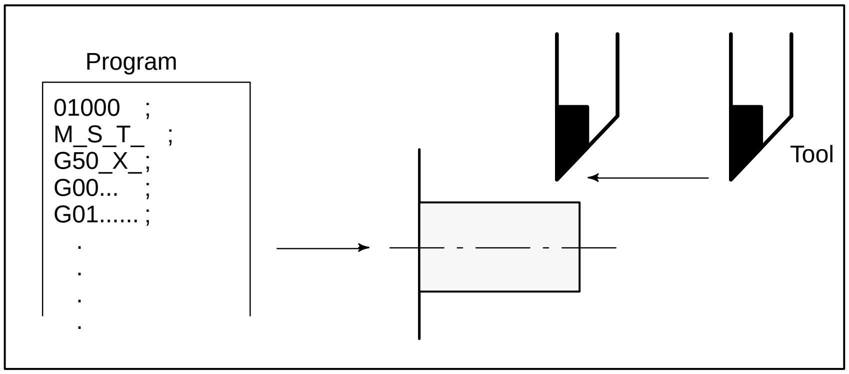


Fig. 1.2 (a) Tool Movement by Programming

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.

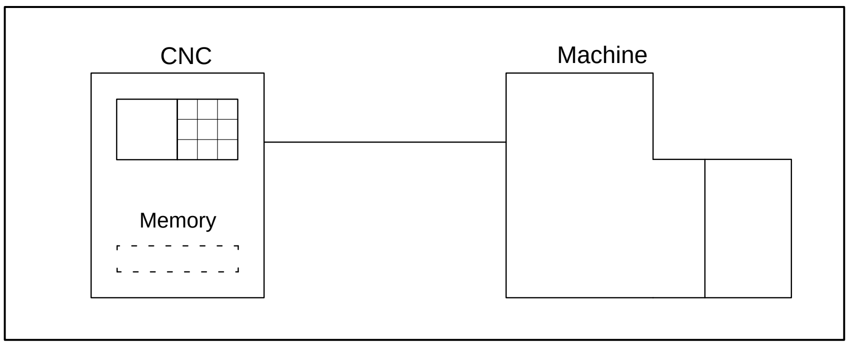


Fig. 1.2 (b) Memory Operation

- **DNC operation**

The machine can operate by reading a program directly from a connected I/O device, without registering the program into CNC memory. This function is useful when a program is too large to be registered into CNC memory.

- **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.

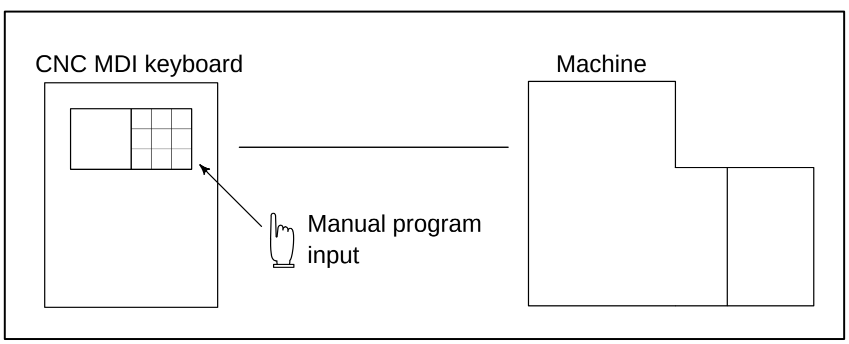


Fig. 1.2 (c) MDI operation

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).

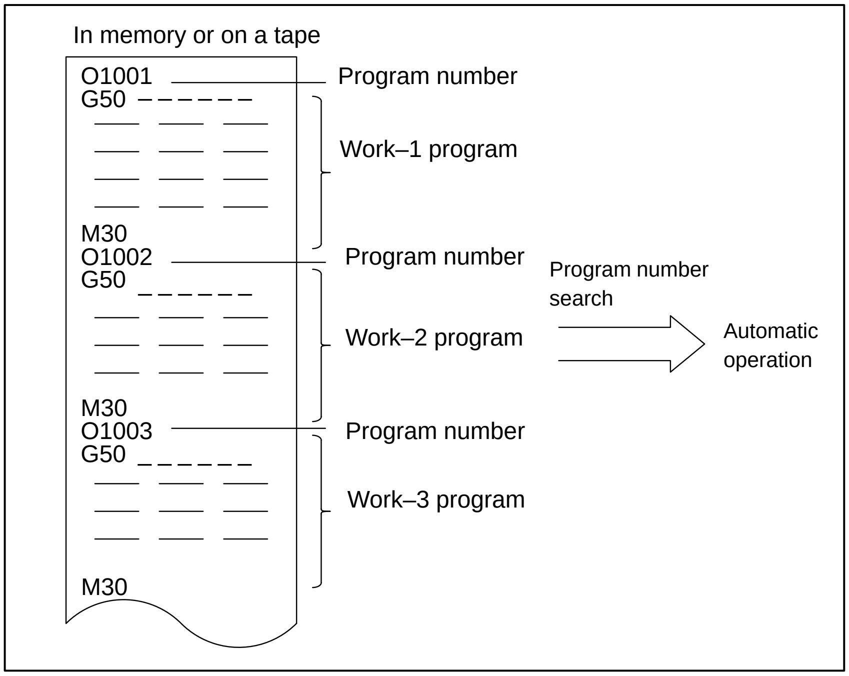


Fig. 1.3 (a) Program Selection for Automatic Operation

• Start and stop (See Chapter III–4)

Pressing the cycle start pushbutton causes automatic operation to start. By pressing the feed hold or reset pushbutton, 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.

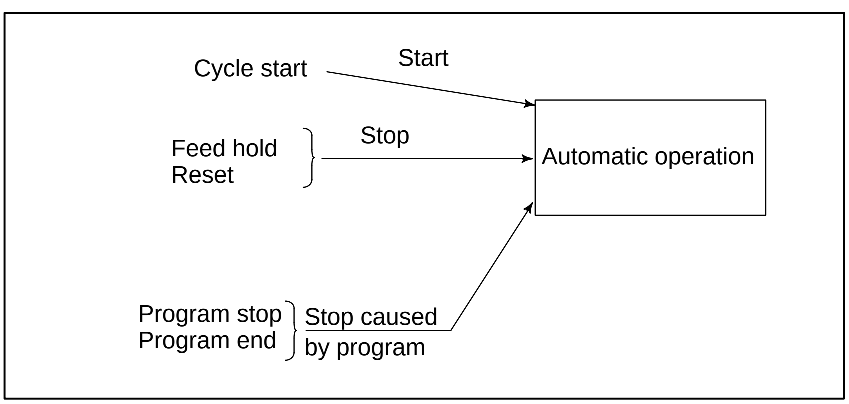


Fig. 1.3 (b) Start and Stop for Automatic Operation

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 Chapter 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.

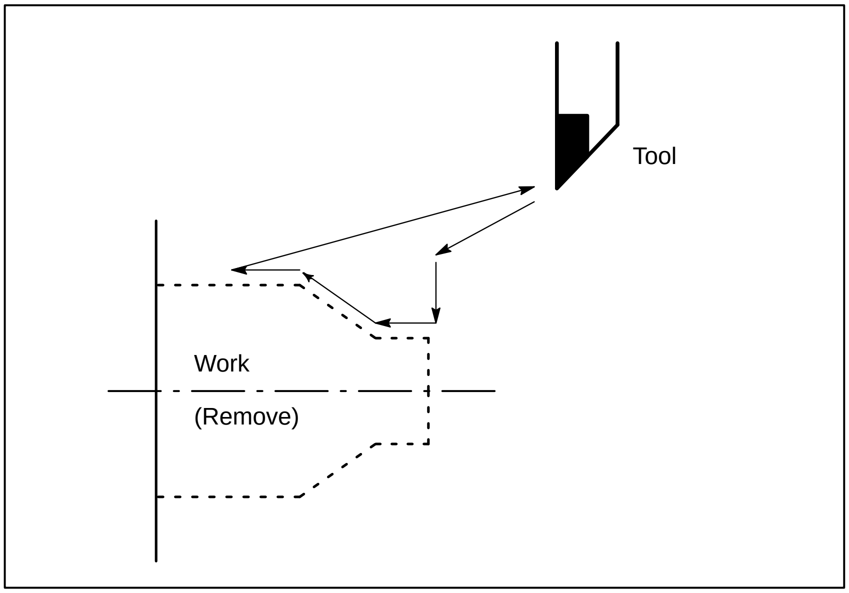


Fig. 1.4.1 (a) Dry run

- **Feedrate override (See Section III–5.2)**

Check the program by changing the program of feed rate command.

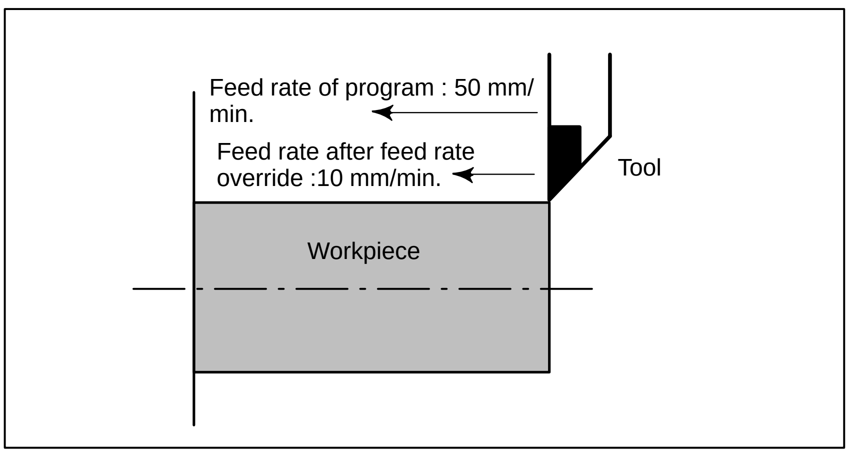


Fig. 1.4.1 (b) Feedrate Override

- **Single block (See Section III–5.5)**

When the cycle start pushbutton 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.

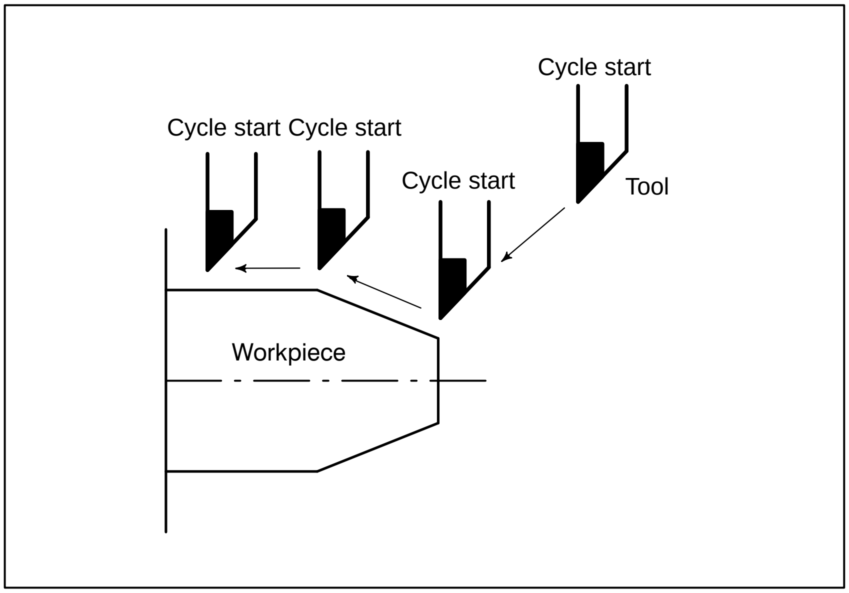


Fig. 1.4.1 (c) Single Block

1.4.2 How to View the Position Display Change without Running the Machine

Explanations

- **Machine lock (See Sections III–5.1)**

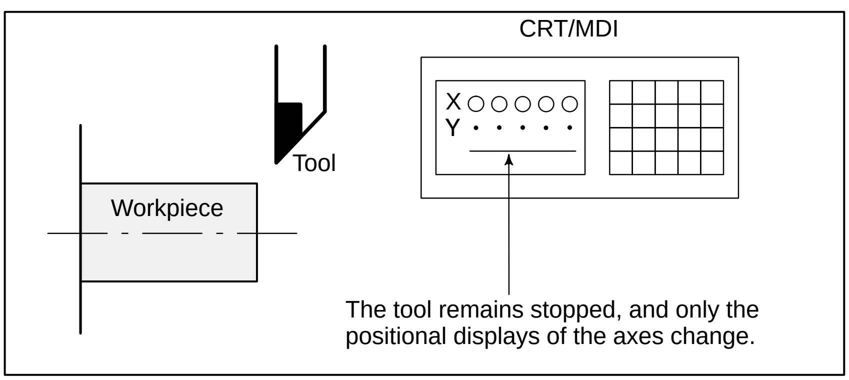


Fig. 1.4.1 (d) Machine Lock

- **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 Chapter III–9).
This operation can be executed using the part program storage/edit function.

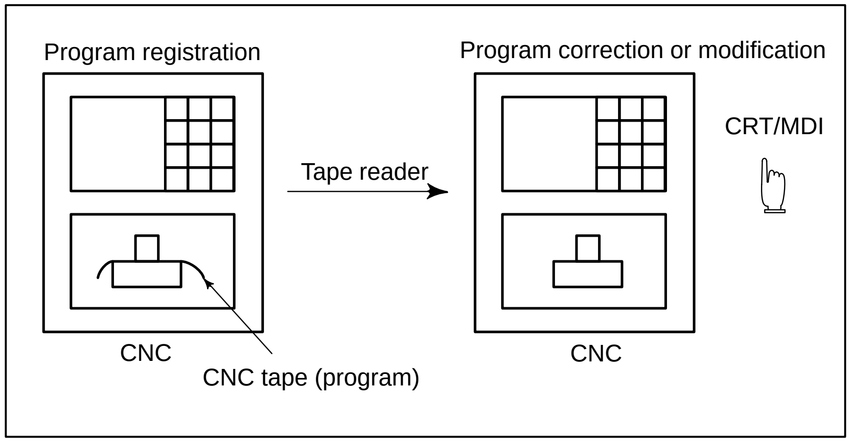


Fig. 1.5 (a) Part Program Editing

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).

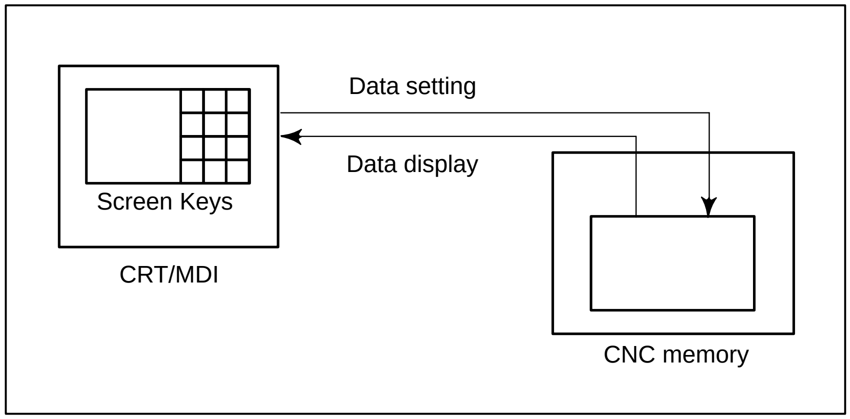


Fig. 1.6 (a) Displaying and Setting Data

Explanations

- **Offset value**

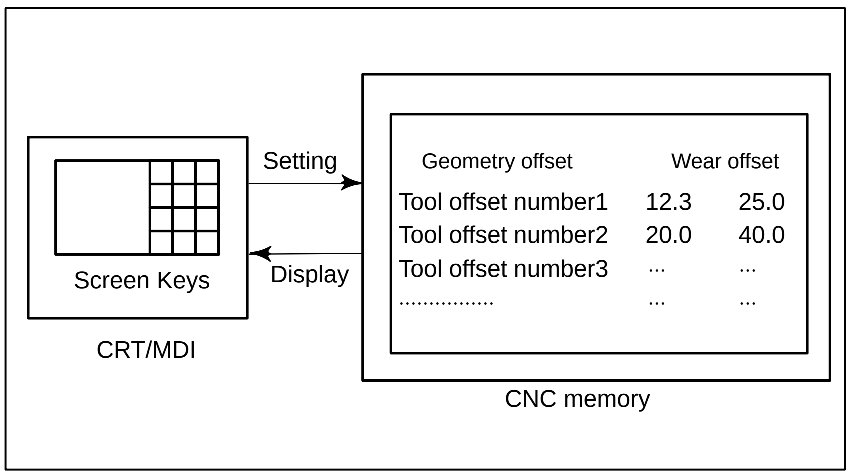


Fig. 1.6 (b) Displaying and Setting Offset Values

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 Subsec. III–11.4.1).

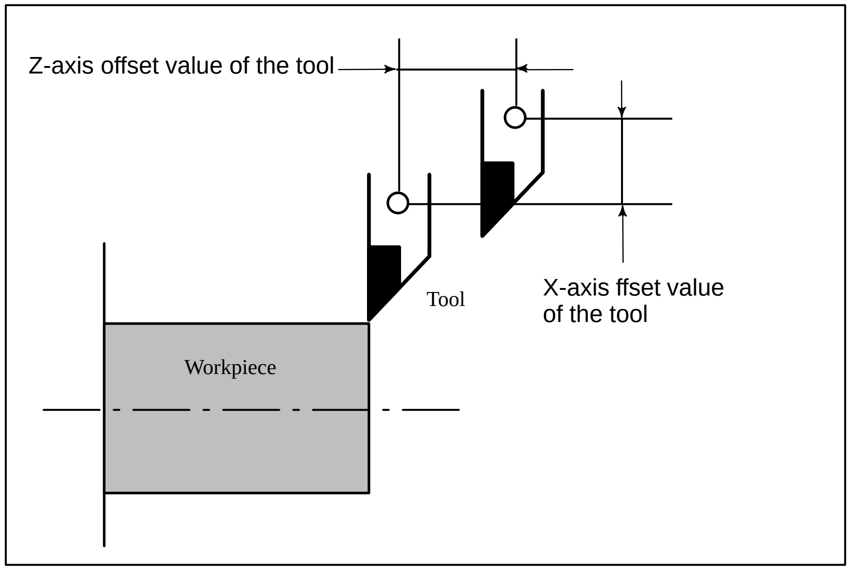


Fig. 1.6 (c) Offset Value

- **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
·Selection of I/O devices
The above data is called setting data (See Subsec. III–11.5.1).

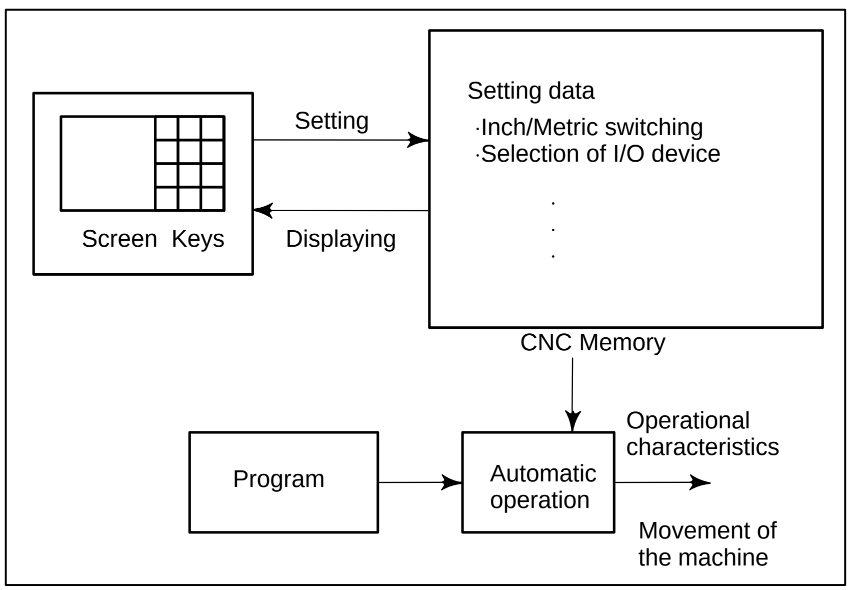


Fig. 1.6 (d) Displaying and Setting Operator's setting data

- **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 Subsec III–11.5.3).

Parameters differ depending on machine tool.

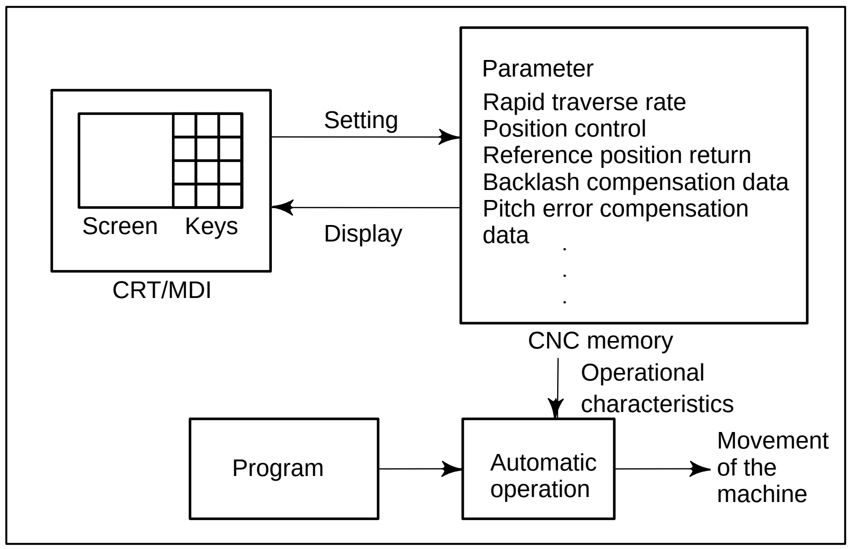


Fig. 1.6 (e) Displaying and setting parameters

- **Data protection key**

A key called the data protection key can be defined. It is used to prevent part programs from being registered, modified, or deleted erroneously (See Chapter III–11).

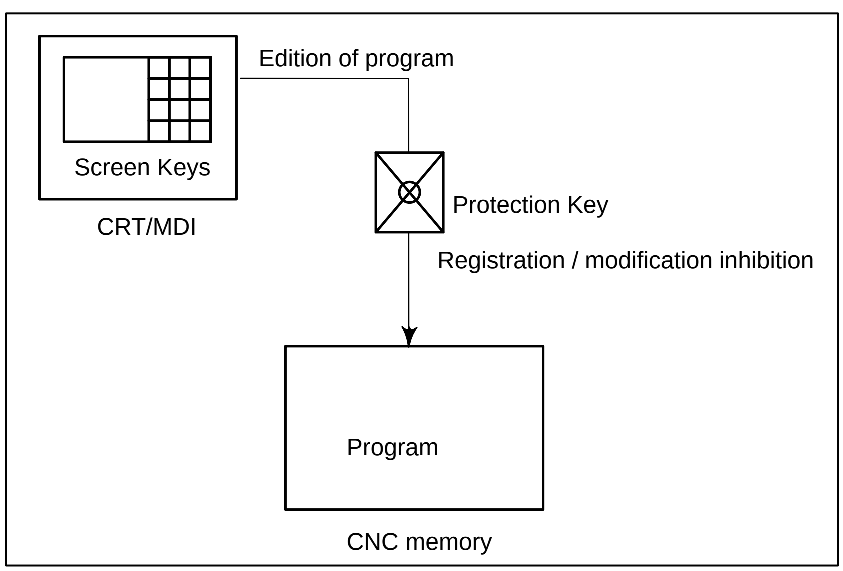


Fig. 1.6 (f) Data Protection Key

1.7 DISPLAY

1.7.1 Program Display (See Subsection III–11.2.1)

The contents of the currently active program are displayed. In addition, the programs scheduled next and the program list are displayed.

Active sequence number

Active program number

```
PROGRAM                                O0001  N0000
O0001 T0101 ;
S550 M08 ;
M45 ;
N010 G50 X200. Z200. ;
N011 G00 X160. Z180. ;
N012 G71 U7. R1. ;
N013 G71 P014 Q020 U4. W2. F0.3 S550 M03
 ;
N014 G00 X40. F0.15 S700 ;



<
                        EDIT
[ PRGRM ][  LIB  ][  I/O  ][        ][        ]
```

Program content

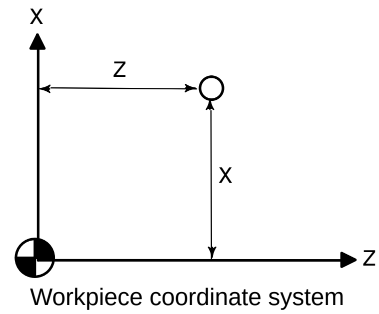
Currently executed program

The cursor indicates the currently executed location

```
PROGRAM                              O0001  N0000
    SYSTEM SDITION      0671 – 04
  PROGRAM NO,  USED :    24 FREE :    39
  MEMORY AREA  USED : 24960 FREE : 97920
PROGRAM LIBRARY LIST
  O0021 O0041 O0615 O0651 O0601 O0645
  O0613 O0021 O1041 O1051 O0010 O2011
  O2505 O0011 O3511 O3148 O3153 O4011
  O4048 O5221 O5111 O5766 O6032 O0001




<
                           EDIT
[ PRGRM ][CONDNS ][        ][        ][        ]
```

1.7.2 Current Position Display (See Subsections III–11.1.1 to 11.1.3)

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.




```
ACTUAL POSITION (ABSOLUTE)      O0001 N0023
 X    200.000
 Z    220.000
 C      0.000



                      PART COUNT  1786
RUN TIME      2H47M   CYCLE TIME  0H 1M47S
ACT.F   3000 MM/M
                        BUF AUTO
[  ABS  ][  REL  ][  ALL  ][  HNDL  ][      ]
```

1.7.3 Alarm Display (See Section III–7.1)

When a trouble occurs during operation, error code and alarm message are displayed on CRT screen. See APPENDIX 7 for the list of error codes and their meanings.

```
ALARM MESSAGE                   O0001  N0011

010  P/S ALARM






          ALARM         BUF MDI
[ ALARM ][       ][  MSG  ][       ][       ]
```

1.7.4 Parts Count Display, Run Time Display (See Subsection III–11.5.2)

When option is selected, two types of run time and number of parts are displayed on the screen.

```
ACTUAL POSITION (ABSOLUTE)      O0001 N0023
 X      200.000
 Z      220.000
 C        0.000

                           PART COUNT  1786
RUN TIME       2H47M   CYCLE TIME  0H 1M47S
ACT.F   3000 MM/M
                             BUF AUTO
[  ABS  ][  REL  ][  ALL  ][  HNDL  ][      ]
```

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.

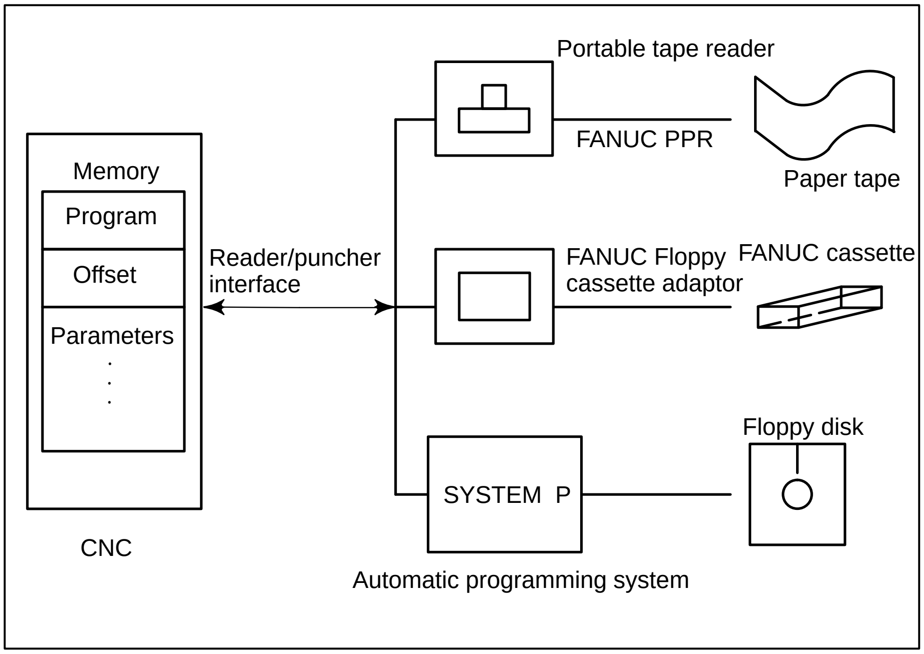


Fig. 1.8 (a) Data Output

2 OPERATIONAL DEVICES

The peripheral devices available include the CRT/MDI panel 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 CRT/MDI PANELS

Figs. 2.1 (a) to 2.1 (f) show the CRT/MDI.

9” CRT/MDI small monochrome (with soft key) . . . . Fig.2.1(a)

External view

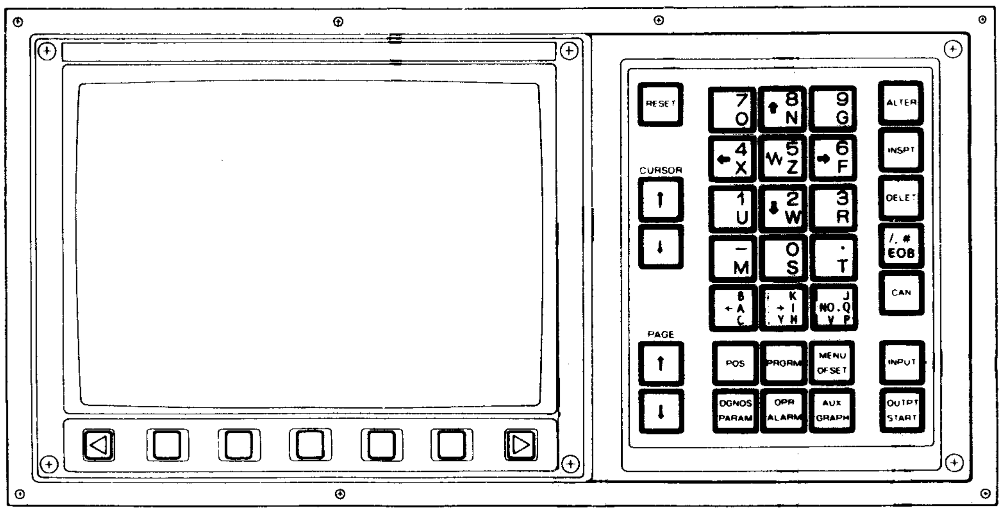


Fig. 2.1 (a) 9” CRT/MDI small monochrome (with soft key)

Explanation of the keyboard

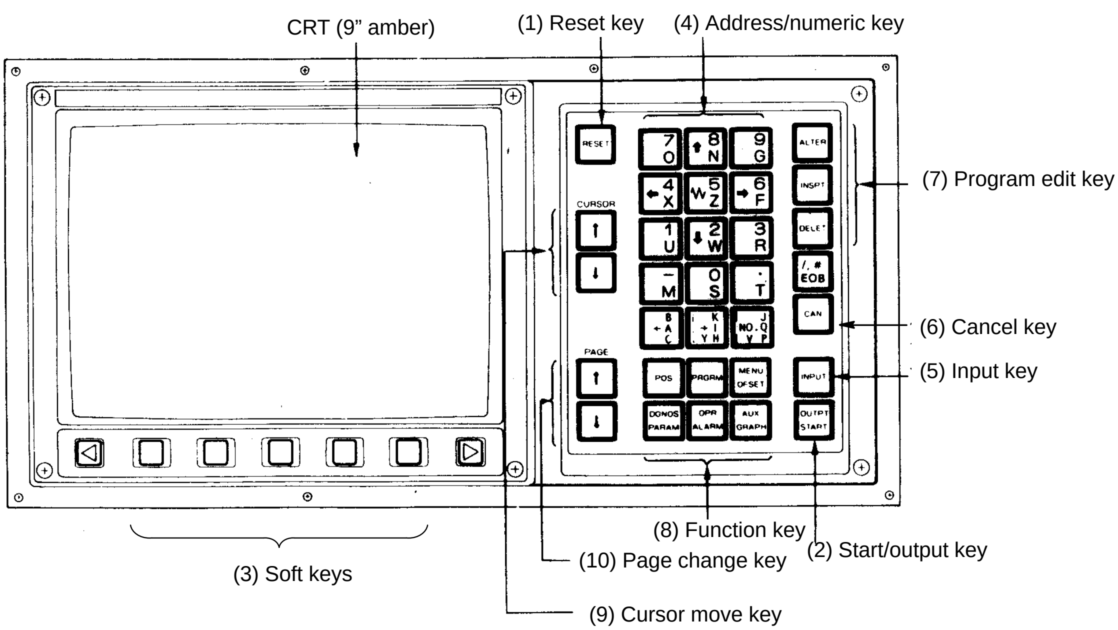


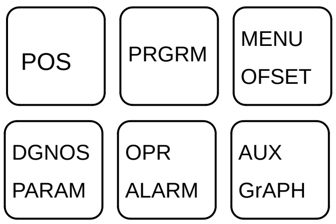
Fig. 2.1 (b) 9” monochrome or color CRT/MDI panel (horizontal type)

Table 2.1 Explanation of the MDI keyboard

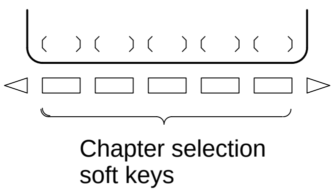
Number	Name	Explanation
1	RESET key RESET	Press this key to reset the CNC, to cancel an alarm, etc.
2	OUTPUT key, START key OUTPT START	Used to start MDI or automatic operation. The use of this key depends on the machine tool builder. Refer to the manual of the machine tool builder. This key is used also to output data to an I/O unit.
3	Soft keys (option)	The soft keys have various functions, according to the Applications. The soft key functions are displayed at the bottom of the CRT screen.
4	Address and numeric keys N) ← 4 …	Press these keys to input alphabetic, numeric, and other characters.
5	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 [IN-PUT] key of the soft keys. This operation is same as using I/O devices.
6	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
7	Program edit keys ALTER INSRT DELET	Press these keys when editing the program. ALTER : Alteration INSRT : Insertion DELET : Deletion
8	Function keys POS PRGRM …	Press theses keys to switch display screens for each function. See sec. 2.2 for detailas of the function keys.
9	Cursor move keys CURSOR ↑ ↓	There are two different cursor move keys. ↓ :This key is used to move the cursor in a downward or forward direction. ↑ : This key is used to move the cursor in an upward or reverse direction.
10	Page change keys PAGE ↑ ↓	Two kinds of page change keys are described below. ↓ : This key is used to changeover the page on the CRT screen in the forward direction. ↑ : This key is used to changeover the page on the CRT screen in the reverse direction.

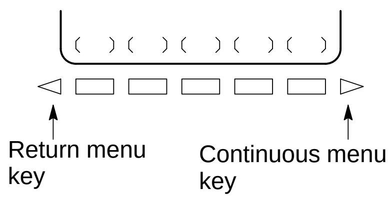
2.2 FUNCTION KEYS AND SOFT KEYS

2.2.1 General Screen Operations



Function keys





1 Press a function key on the CRT/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 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.

If soft keys are not provided, press the key with the identical function to select a desired chapter. The description of the soft keys is automatically displayed according to the optional configuration. The description of the soft keys can be displayed regardless of the optional configuration if bit 7 of parameter 048 is set accordingly.

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 panels:

Key	Description
POS	Press this key to display the **position screen**.
PRGRM	Press this key to display the **program screen**.
MENU OFSET	Press this key to display the **offset/screen**.
DGNOS PARAM	Press this key to display the **setting screen / parameter screen / diagnosis screen.**
OPR ALARM	Press this key to display the **alarm screen. / operator message screen.**
AUX GRAPH	This key is not used.

2.2.3 Key Input and Input Buffer

Explanations

● For standard key

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.

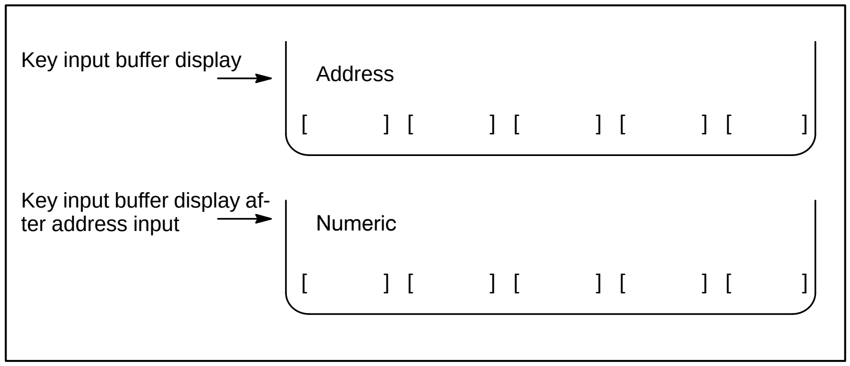
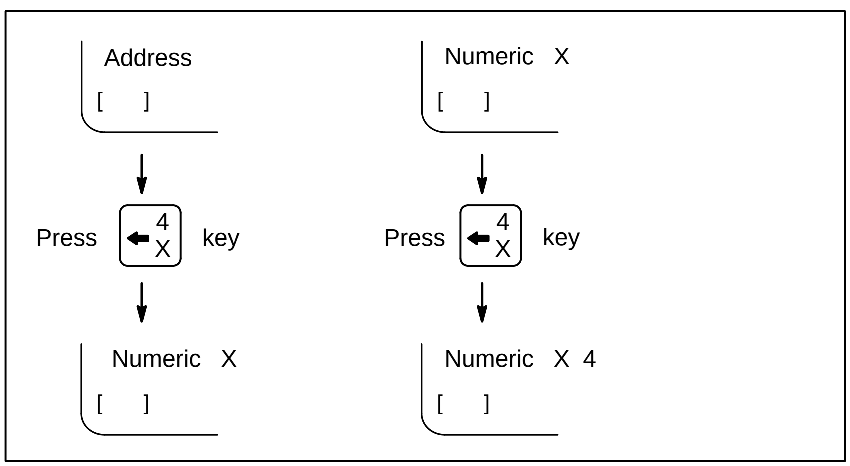
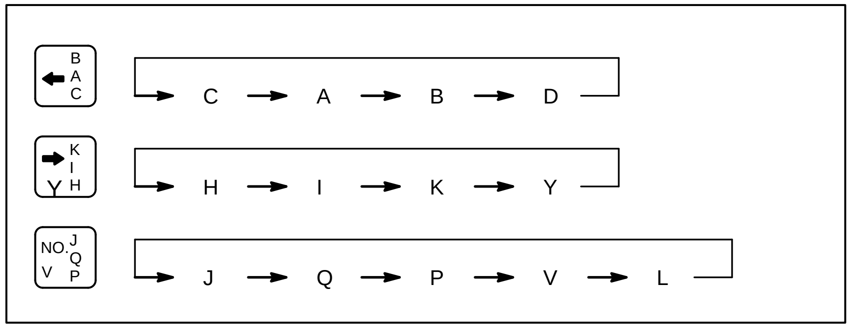


Fig. 2.2.3 Key input buffer display

On the standard key panel, the same key is used to input both an address and a numeric value.
When the key input buffer begins with ”ADDRESS”, pressing that key inputs the address. When the key input buffer begins ”NUMERIC”, pressing that key inputs the numeric value.



Data of one word (address + numeric value) can be entered into the key input buffer at one time. The following data input keys are used to input addresses. Each time the key is pressed, the input address changes as shown below:



Pressing the CAN key deletes all the data input to the key input buffer.

When bit 7 of parameter 0394 is set to 1, each press of the CAN key deletes only the most recently entered character during data input using the parameter, diagnostic, or offset screen.

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 8

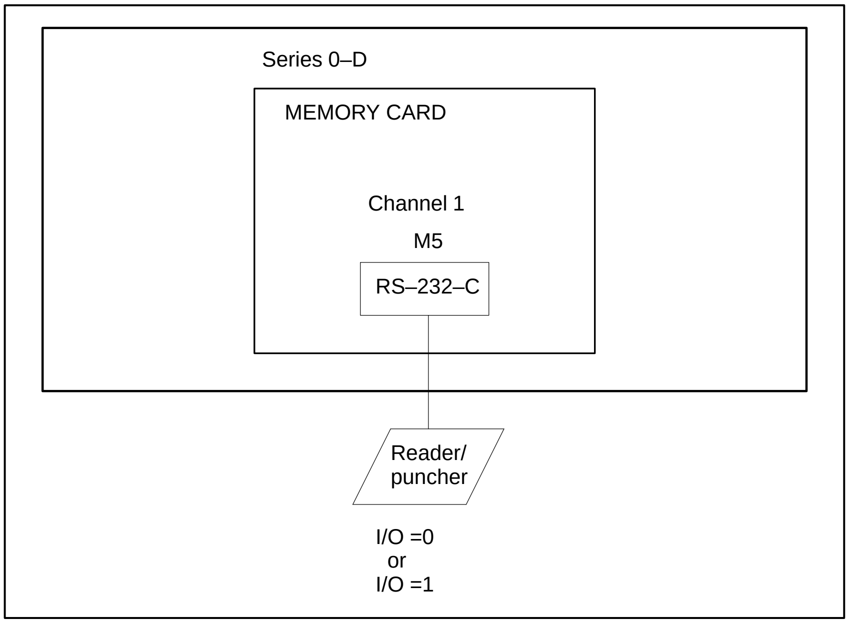
The following data can be input/output to or from external input/output devices:

1. Programs
2. Offset data
3. Parameters

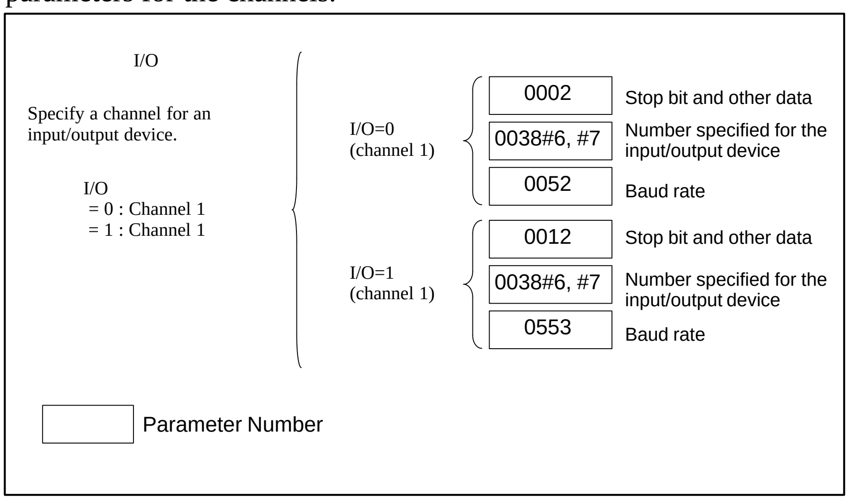
For how data is input and output, see Chapter 8.

Parameter

Before an external input/output device can be used, parameters must be set as follows.



Series 0–D has one channel 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.
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.

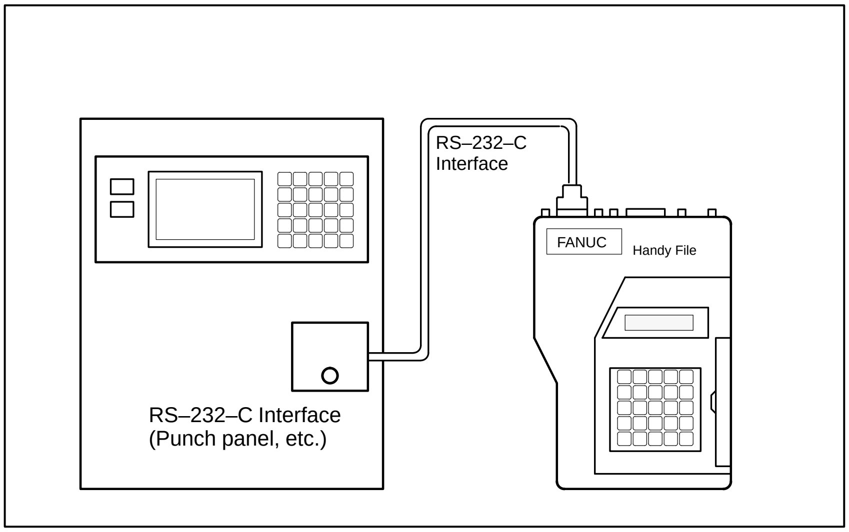


2.3.1 FANUC Handy File

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.

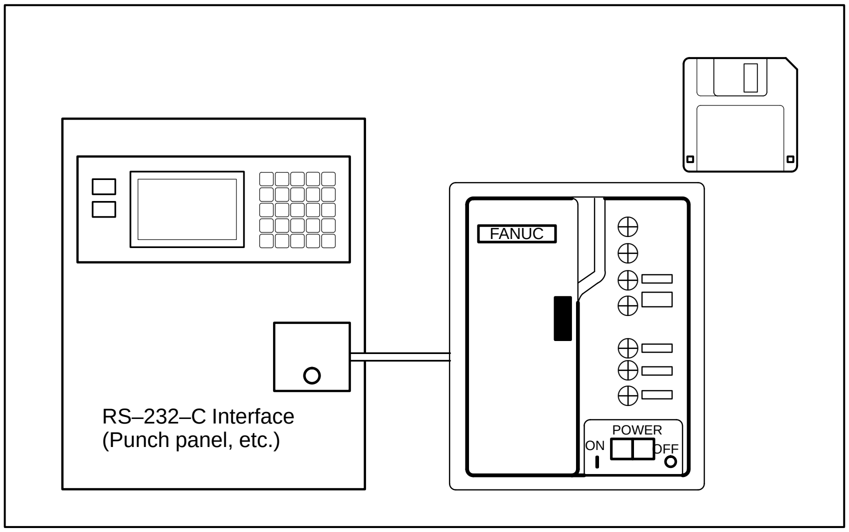
The Handy File uses 3.5–inch floppy disks, which do not have the problems of paper tape (i.e., noisy during input/output, easily broken, and bulky).

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

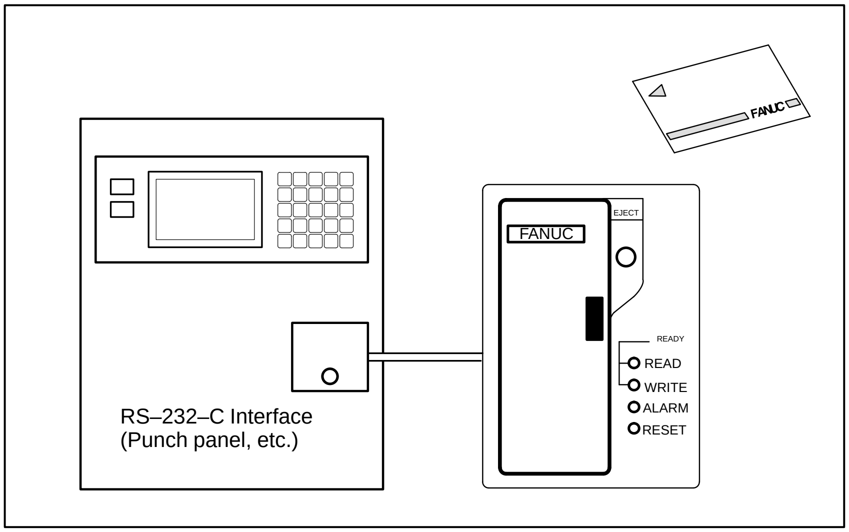
When the Floppy Cassette is connected to the NC, machining programs stored in the NC can be saved on a Floppy Cassette, and machining programs saved in the Floppy Cassette can be transferred to the NC.



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.

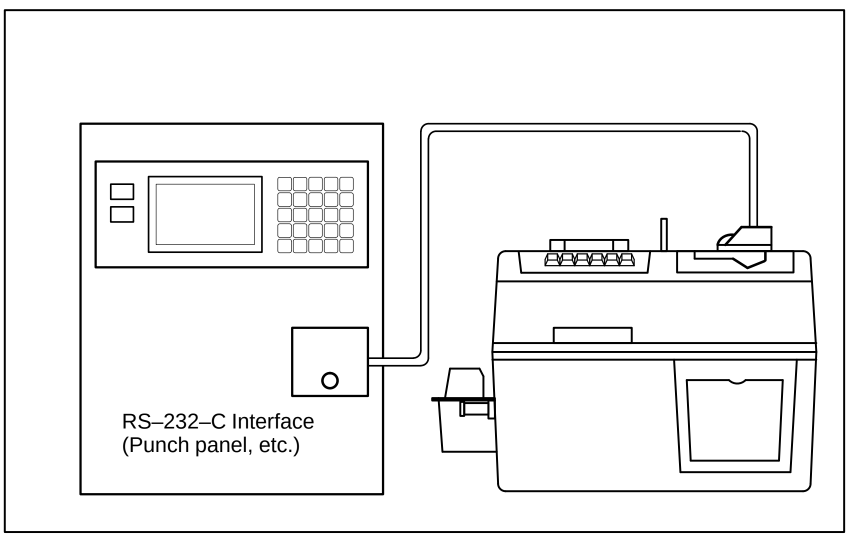
When an FA Card is connected to the CNC via the card adapter, NC machining programs stored in the CNC can be transferred to and saved in an FA Card. Machining programs stored on an FA Card can also be transferred to the CNC.



2.3.4 FANUC PPR

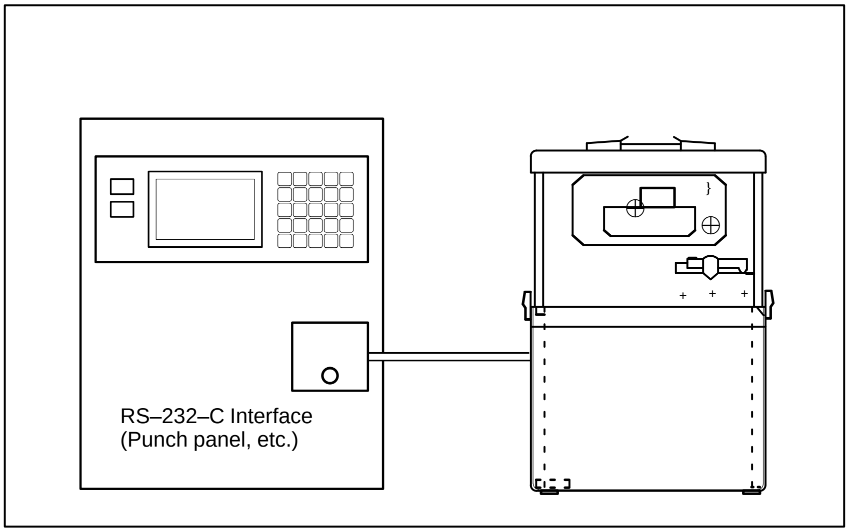
The FANUC PPR consists of three units: A printer, paper tape punch, and paper tape reader.

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. If the screen shown in Subsection 2.4.1(b) is displayed, a system failure may have occurred.
If the machine tool is in the emergency stop state, the software configuration screen, shown in 2.4.1(b), appears.

Press the POS function key on the CRT/MDI panel, 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)       O0001 N0023
 X      200.000
 Z      220.000
 C        0.000



                             PART COUNT  1786
RUN TIME       2H47M         CYCLE TIME  0H 1M47S
ACT.F   3000 MM/M
                               BUF AUTO
[  ABS  ][  REL  ][  ALL  ][  HNDL  ][       ]
```

Fig. 2.4.1(a) Screen of position display

4 Check that the fan motor is rotating.

> **WARNING**
> When pressing the <POWER ON> key. Until the positional or alarm screen is displayed, do not touch keys of CRT/MDI panel. Some keys are used for the maintenance or special operation purpose. When they are pressed, unexpected operation may be caused.

Display of software configuration

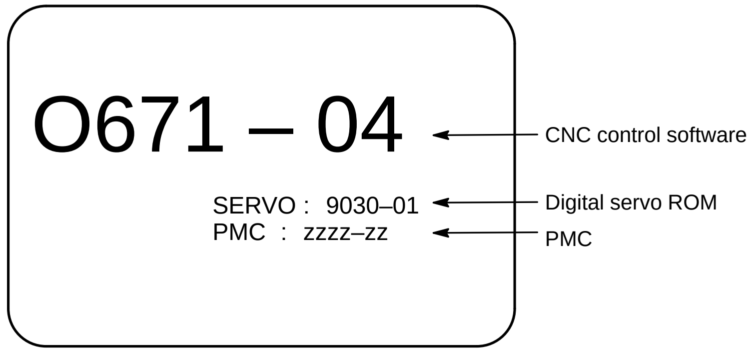


Fig. 2.4.1(b) Display of software configuration

2.4.2 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.

> **NOTE**
> Refer to the machine tool builder's manual for turning off the power to the machine.

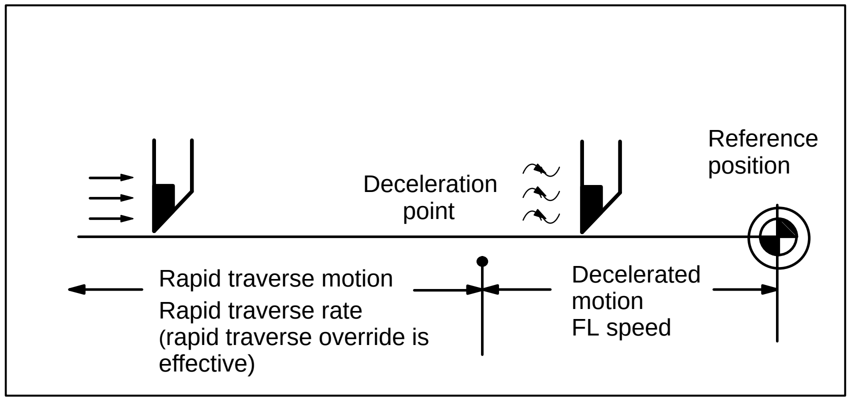
3 MANUAL OPERATION

MANUAL OPERATION are four kinds as follows :

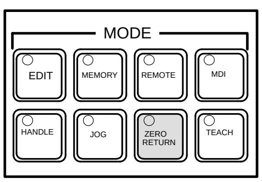
1. **Manual reference position return**
2. **Jog feed**
3. **Incremental feed**
4. **Manual handle feed**
5. **Manual absolute on/off**

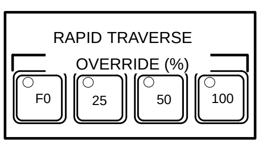
3.1 MANUAL REFERENCE POSITION RETURN

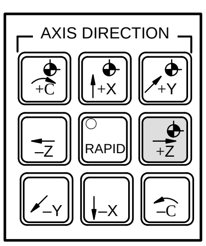
The tool is returned to the reference position as follows :
The tool is moved in the direction specified in parameter (bit0 to 2 of No.0003) for each axis with the reference position return switch on the machine operator's panel. The tool moves to the deceleration point at the rapid traverse rate, then moves to the reference position at the FL speed. The rapid traverse rate and FL speed are specified in parameters (No. 0518 to 0520).
Fourstep 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 two axes simultaneously when specified so in parameter (bit4 of No.0049).



Procedure for Manual Reference Position Return Operation





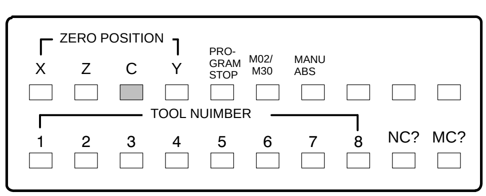


1 Press the reference position return switch, one of the mode selection swithces.

2 To decerease 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**

Bit7 of parameter No.0010 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 0708 to 0710, 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 $X=\alpha, Z=\gamma$ when reference position return is performed.

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.

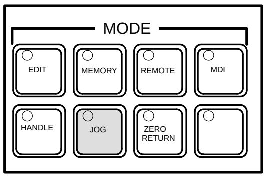
- **Reference position return completion LED**

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 JOG FEED



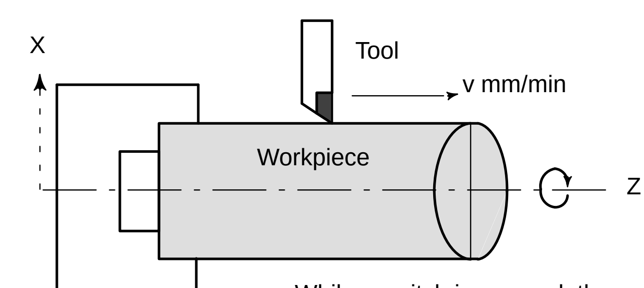
In the jog mode, pressing a feed axis and direction selection switch on the machine operator's panel continuously moves the tool along the selected axis in the selected direction.
The jog feedrate is described following table 3.2.

Table 3.2 Jog Feedrate

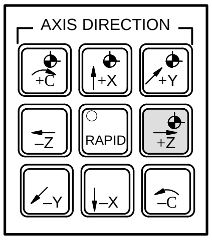
Rotary switch position	Feedrate		Rotary switch position	Feedrate	
	Metric input (mm/min)	Inch input (inch/min)		Metric input (mm/min)	Inch input (inch/min)
0	0	0	8	50	2.0
1	2.0	0.08	9	79	3.0
2	3.2	0.12	10	126	5.0
3	5.0	0.2	11	200	8.0
4	7.9	0.3	12	320	12
5	12.6	0.5	13	500	20
6	20	0.8	14	790	30
7	32	1.2	15	1260	50

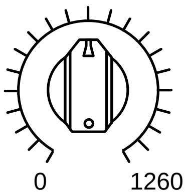
NOTE
The feedrate error (about ”3%) affects on the feedrate in the table above.

The jog feedrate can be adjusted with the jog feedrate override dial.
Pressing the rapid traverse switch moves the tool at the rapid traverse feedrate regardless of the postiotion of the jog 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.0049#4).

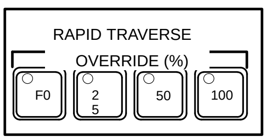


Procedure for Jog Feed Operation





JOG FEED RATE OVERRIDE



1 Press the jog 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 described table 3.2. The tool stops when the switch is released.

3 The jog feedrate can be adjusted with the jog feedrate override switch.

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.

Restrictions

- **Acceleration/decelera– tion for rapid traverse**

Accerleration/deceleration method and time constant for rapid traverse are the same as G00 in programmed command.

- **Change of modes**

Changing the mode to the jog mode while pressing a feed axis and direction selection switch does not enable jog feed. To enable jog feed, enter the jog mode first, then press a feed axis and direction selection switch.

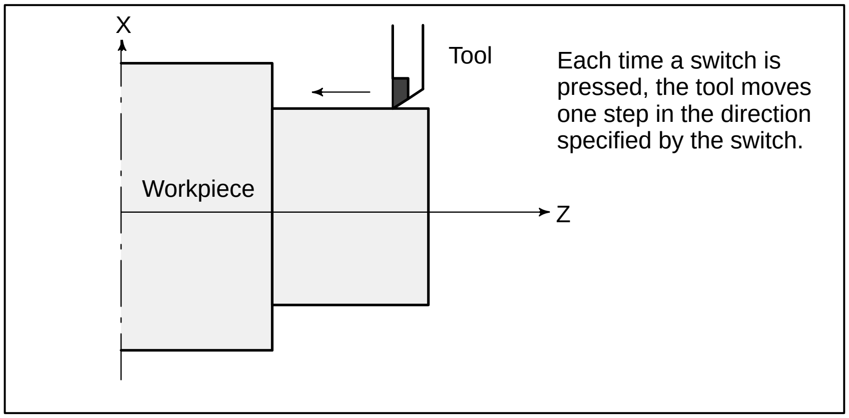
- **Rapid traverse prior to reference position return**

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 JOG feedrate. This function can be disabled by setting parameter (No.0010#0).

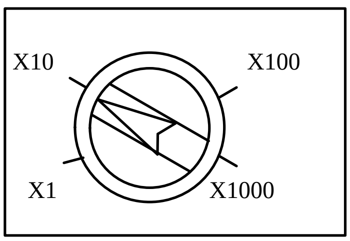
3.3 INCREMENTAL FEED

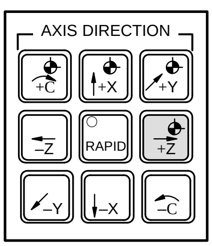
In the incremental (INC) mode, pressing a feed axis and direction selection switch on the machine operator's panel moves the tool one step along the selected axis in the selected direction. The minimum distance the tool is moved is the least input increment. Each step can be 10, 100, or 1000 times the least input increment.

This mode is effective when a manual pulse generator is not connected.



Procedure for Incremental Feed Operation





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

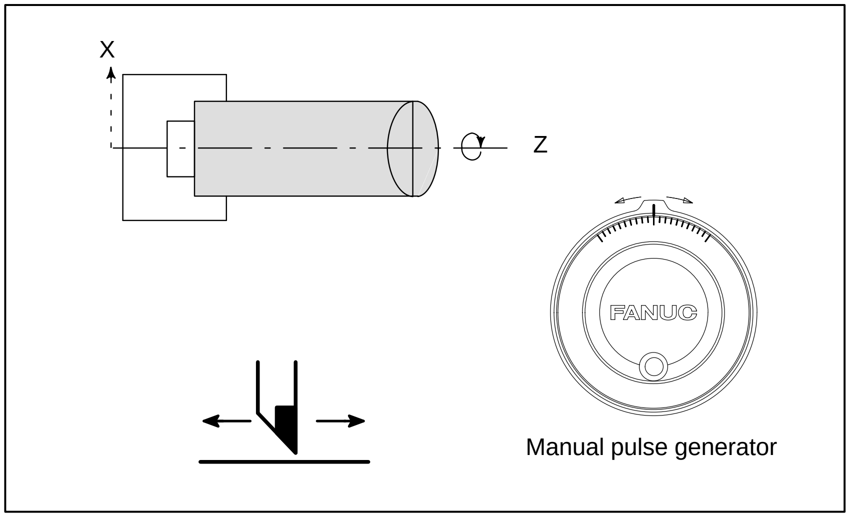
- **Travel distance specified with a diameter**

The distance the tool travels along the X–axis can be specified with a diameter.

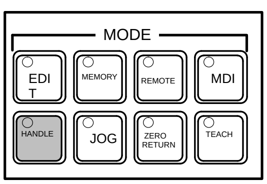
3.4 MANUAL HANDLE FEED

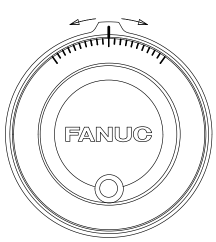
In the handle mode, the tool can be minutely moved by rotating the manual pulse generator on the machine operator's panel. Select the axis along which the tool is to be moved with the handle feed axis selection switches.

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. 0121 and 0699).



Procedur for Manual Handle Feed Operation





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 handle feed in Jog mode**

Parameter (bit 0 of No. 0013) enables or disables the manual handle feed in the JOG mode.
When the parameter (bit 0 of No. 0013) is set 1,both manual handle feed and incremental feed are enabled.

- **Availability of manual handle feed in TEACH IN JOG mode**

Parameter (bit 6 of No. 0002) enables or disables the manual handle feed in the TEACH IN JOG mode.

- **A command to the MPG exceeding rapid traverse rate**

Parameter (bit 4 of No. 0060) 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**

Parameter (No.0386#0 to 2) 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.

WARNING

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.

NOTE

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.

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.

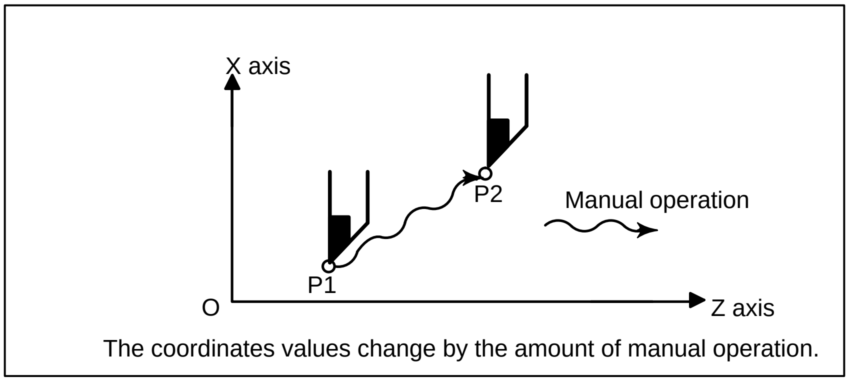


Fig. 3.5 (a) Coordinates with the switch ON

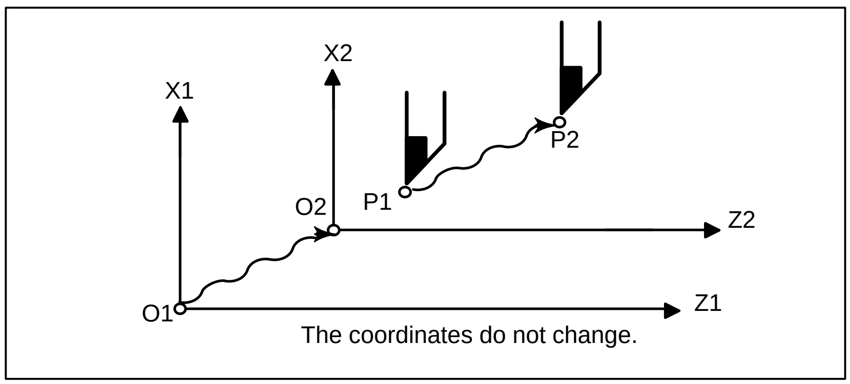


Fig. 3.5 (b) Coordinates with the switch OFF

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.

```
G01   X100.0Z100.0F010 ;    1
      X200.0Z150.0     ;    2
      X300.0Z200.0     ;    3
```

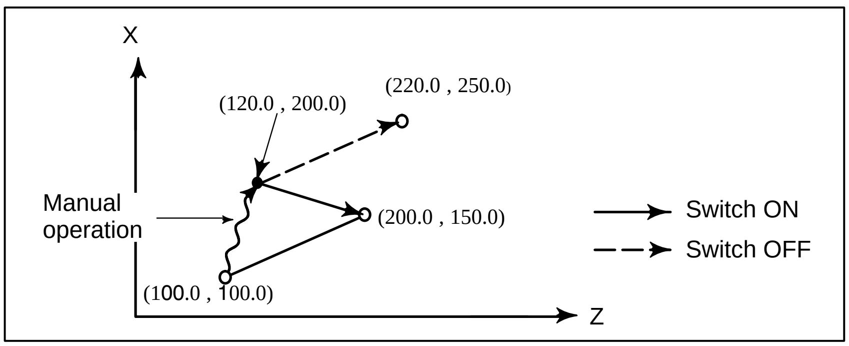
The subsequent figures use the following notation:

→ 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.

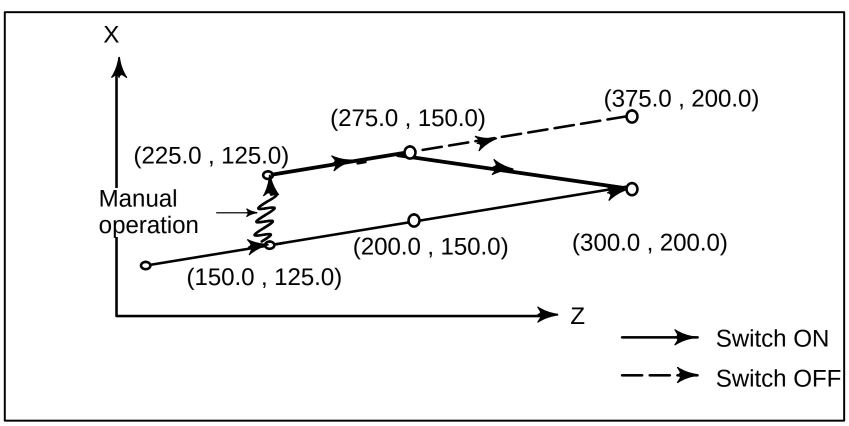
- **Manual operation after the end of block**

Coordinates when block $\boxed{2}$ has been executed after manual operation (X–axis +20.0, Z–axis +100.0) at the end of movement of block $\boxed{1}$.



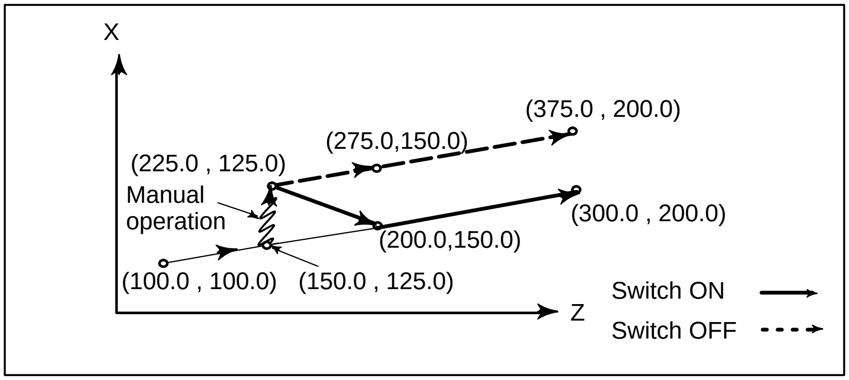
- **Manual operation after a feed hold**

Coordinates when the feed hold button is pressed while block $\boxed{2}$ is being executed, manual operation (X–axis + 75.0) is performed, and the cycle start button is pressed and released



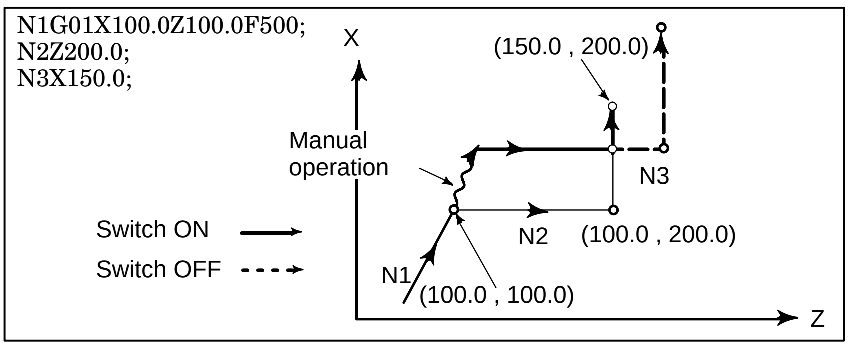
- **When reset after a manual operation following a feed hold**

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.



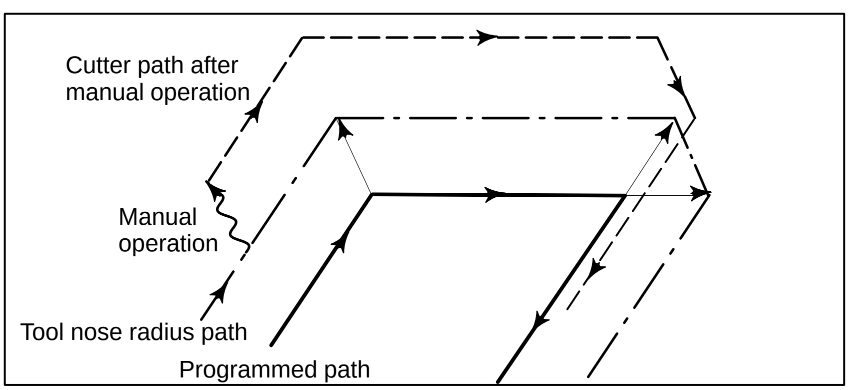
- **When the next move block is an incremental**

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.

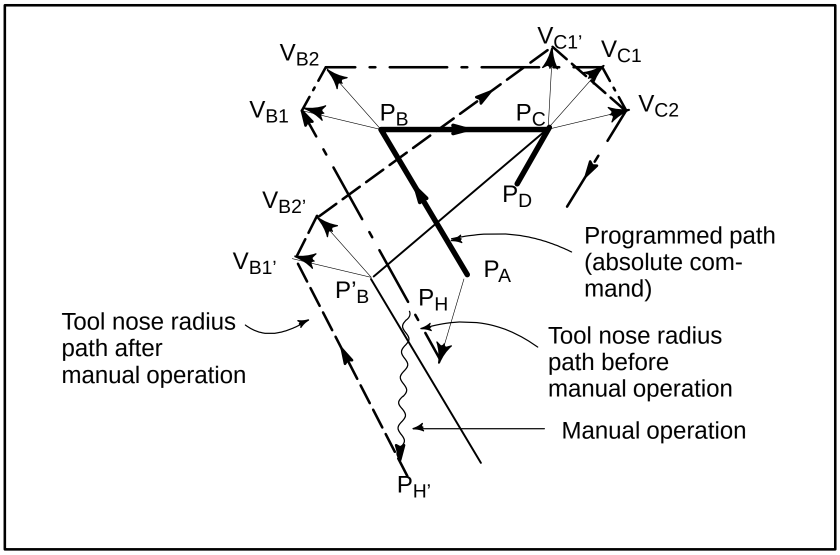


When the switch is ON during tool nose radius compensation

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.

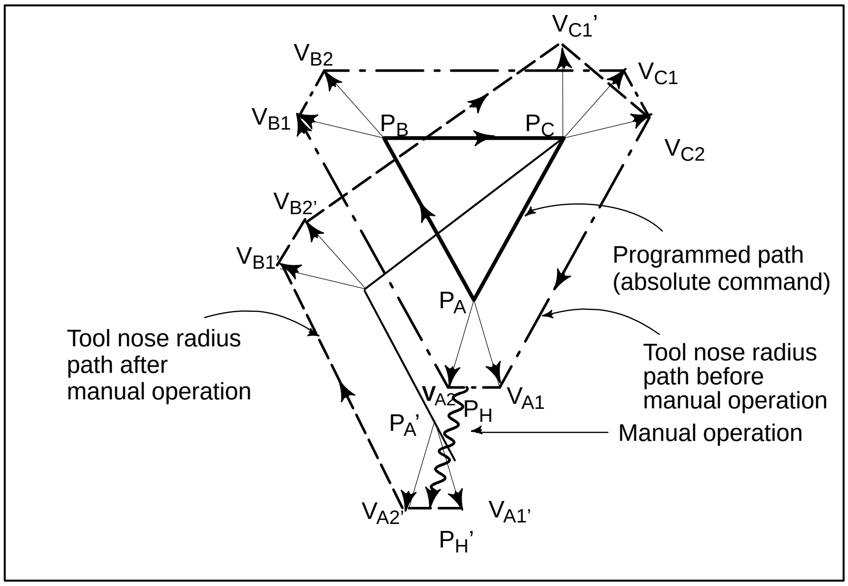
Manual operation performed in other than cornering

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

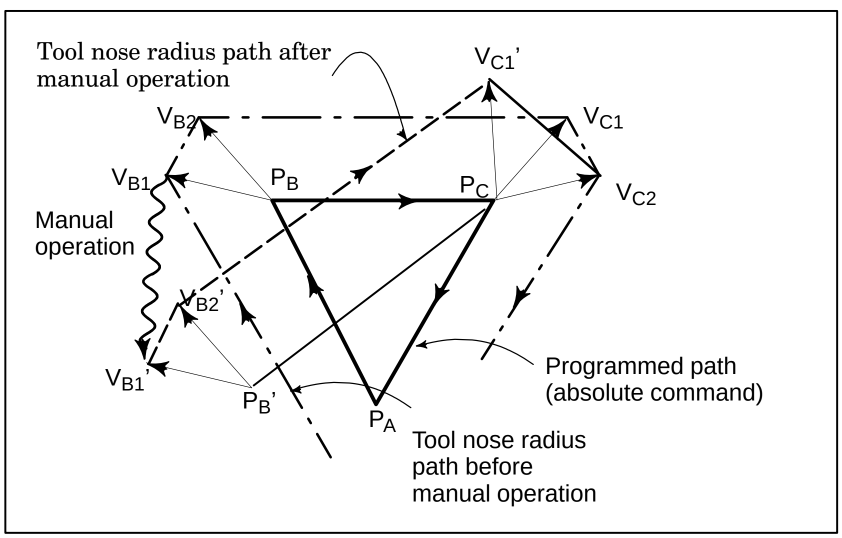
This is an example when manual operation is performed during cornering. $V_{A2'}$, $V_{B1'}$, and $V_{B2'}$ are vectors moved in parallel with V_{A2}, V_{B1} and V_{B2} by the amount of manual movement. The new vectors are calculated from V_{C1} and V_{C2}. Then correct tool nose radius compensation 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**
Function for operating the machine by reading a program from the I/O unit

·**Search for a sequence number**
Function for starting operation from a sequence number in the middle of a program

·**MIRROR IMAGE**
Function for enabling mirror–image movement along an axis 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 panel is pressed, automatic operation terminates and the reset state is entered.
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 **AUTO** mode selection switch.

2 Select a program from the registered programs. To do this, follow the steps below.

2–1 Press $\boxed{\text{PRGRM}}$ to display the program screen.

2–2 Press address $\boxed{\text{O}}$.

2–3 Enter a program number using the numeric keys.

2–4 Press the $\boxed{\downarrow}$ soft key.

3 Press the cycle start switch on the machine operator's panel. Automatic operation starts, and the cycle start lamp goes on. When automatic operation terminates, the cycle start lamp goes off.

4 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 lamp goes on and the cycle start lamp 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 lamp is on, machine operation restarts.

b. Terminating memory operation

Press the $\boxed{\text{RESET}}$ key on the CRT/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 panel 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.

4.2 MDI OPERATION

In the **MDI** mode, a program can be inputted 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 – A

Example of X10.5 Z200.5;

One command block can be entered from the CRT/MDI for execution.

1 Press MDI key on the mode select switch.

2 Press the PRGRM button.

3 Press soft key **[MDI]** to display a screen with MDI at the top left.

```
PROGRAM                              O0001 N0020
     (MDI)                        (MODAL)
                                 G00 F   0.3000
                                 G97 M   003
                                 G69 S   00550
                                 G99 T   0101
                                 G21
                                 G40 WX  0.000
                                 G25 WZ  0.000
                                 G22 SRPM550
                                 G54 SSPM0
                                      SMAX 32767
                                      SACT 0
ADRS.
                                   MDI
[ PRGRM ][CURRNT ][ NEXT  ][  MDI  ][ RSTR  ]
```

4 Input ”X 10.5” by address/numeric key.

5 Press INPUT key.

The data, Z 10.5, is input and displayed. If you are aware of an error in the keyed–in number before pressing the INPUT key , press the CAN key and key in X and the correct number again.

6 Input ”Z 200.5” by address/numeric key.

7 Press INPUT key.

The data, Z200.5 is input and displayed. If you pressed wrong number keys, correct the operation following the instruction described above.

```
PROGRAM                               O0001 N0020
     (MDI)                          (MODAL)
     .X     10.500               G00 F   0.3000
     .Y    200.500               G97 M   003
                                 G69 S   00550
                                 G99 T   0101
                                 G21
                                 G40 WX  0.000
                                 G25 WZ  0.000
                                 G22 SRPM550
                                 G54 SSPM0
                                      SMAX 32767
                                      SACT 0
ADRS.
                                   MDI
[ PRGRM ][CURRNT ][ NEXT  ][  MDI  ][ RSTR  ]
```

8 Press the OUTPT START key.

Press the cycle start button on the machine operator's panel (depending on the machine tool).

Cancel before pressing the START button

To modify X10.5 Z200.5 to X10.5, cancel Z200.5, by following the steps described below:

1 Press Z CAN INPUT keys.

2 Press the OUTPT START or the cycle start button on the machine operator's panel.

> **WARNING**
> Modal G codes cannot be cancelled. Enter the correct data again.

Limitations

- A single MDI operation executes a single input block. Two or more blocks cannot be executed at one time.
- The end–of–block symbol (;) need not be entered.
- A subprogram call or macro call cannot be specified.
- The input block is cleared when the MDI operation is completed or when a reset is specified.

4.3 DNC OPERATION

DNC operation enables machine operation by reading a program directly from the connected I/O unit. The program is not registered in CNC memory. This method is useful when a program is too large to be registered in CNC memory.

Operating procedure for DNC operation

1 Select the MDI mode and set the channel of the connected I/O unit in the I/O field on the setting screen.

2 Select the **AUTO** mode.

3 Call the beginning of the program in the I/O unit.

4 Input the DNCI signal. (For the actual operation, refer to the manual of the machine tool builder.)

5 Press the **CYCLE START** key.

DNC operation is started. Operation can be started and resumed in the same manner described for memory operation.

Explanations

· The program running under DNC operation can call a subprogram from memory.

· The program running under DNC operation can specify a custom macro. However, repeat and branch instructions cannot be programmed.

· The program running under DNC operation can call a macro program from memory.

· In DNC operation, no sequence number can be specified in the command for returning control from the called subprogram or macro program to the calling program (M99P****).

· The DC3 code is output and reading is stopped at the end of each block (each time EOB is read). To read blocks continuously, specify bit 7 of parameter 0390 accordingly.

· During DNC operation, no program can be displayed; only the current block and next block can be displayed.

· During DNC operation, F is displayed as the address of the program number at the top right corner of the CRT screen.

4.4 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 ;
N1234 X100.0 Z100.0 ;
S12 ;
:
Selected program → ***O0002 ;***
N2345 X20.0 Z20.0 ;
Target sequence number is found. → ***N2346 X10.0 Z10.0 ;***
:
O0003 ;
:

This section is searched starting at the beginning. (Search operation is performed only within a program.)

Procedure for sequence number search

1 Select **AUTO** mode.

2 Press PRGRM.

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 cursor ⬇ key.

7 Upon completion of search operation, the sequence number searched for is displayed in the upper–right corner of the CRT screen.

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 NC 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)

Limitations

To ignore a P/S alarm (Nos. 003 to 010) during a search for a sequence number, set bit 1 of parameter 0051 accordingly.

- **Searching in sub–program**

During sequence number search operation, M98Pxxxx (subprogram call) is not executed. So an 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.

```
Main program          Subprogram
O1234                 O5678
   :                     :
   :                  N8888
M98 P5678 ;              :
   :                  M99 ;
   :                     :
```

If an attempt is made to search for N8888 in the example above, an alarm is raised.

Alarm

Number	Contents
60	Command sequence number was not found in the sequence number search.

4.5 MIRROR IMAGE (FOR 0–TD ONLY)

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.

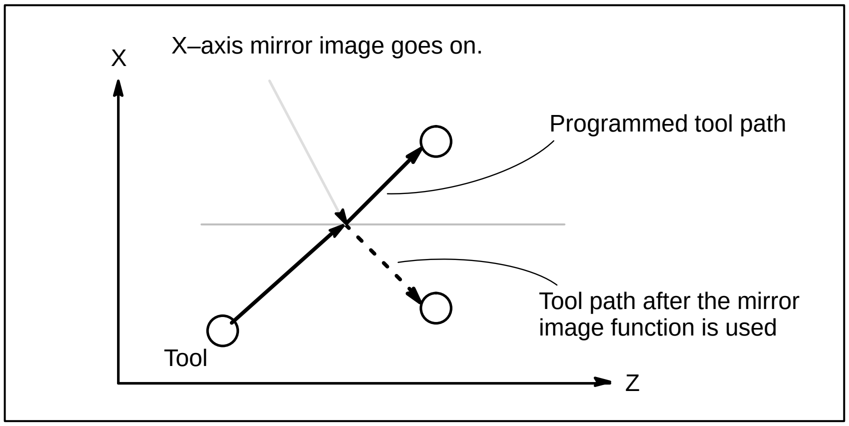


Fig. 4.5 (a) Mirror Image

Procedure

For operation procedure, refer to the manual supplied by the machine tool builder.

Restrictions

The mirror image function is not offective in the of movement during manual operation, the movement from an intermediate point to the reference position during automatic reference position return.

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 Block

5.1 MACHINE LOCK AND AUXILIARY FUNCTION LOCK

To display the change in the position without moving the tool, use machine lock.
In addition, auxiliary function lock, which disables M, S, and T commands, is available for checking a program together with machine lock.

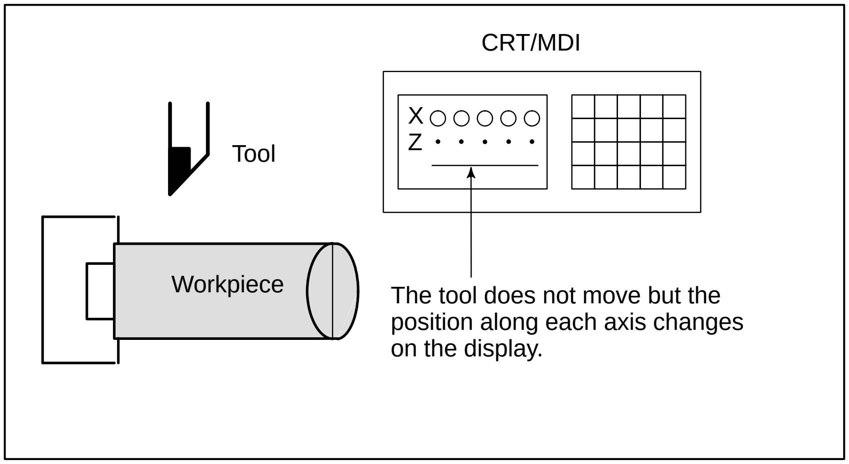


Fig. 5.1 Machine lock

Procedure for Machine Lock and Auxiliary Function Lock Operation

- **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.
After an automatic operation is executed with the machine lock function, the relationship in position between workpiece coordinates and machine coordinates prior to the automatic operation may be changed. If the relationship is changed, reset the workpiece coordinate system by specifying the command for setting the coordinate system or by performing a manual reference position return. Refer to the appropriate manual provided by the machine tool builder for machine lock.

- **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.

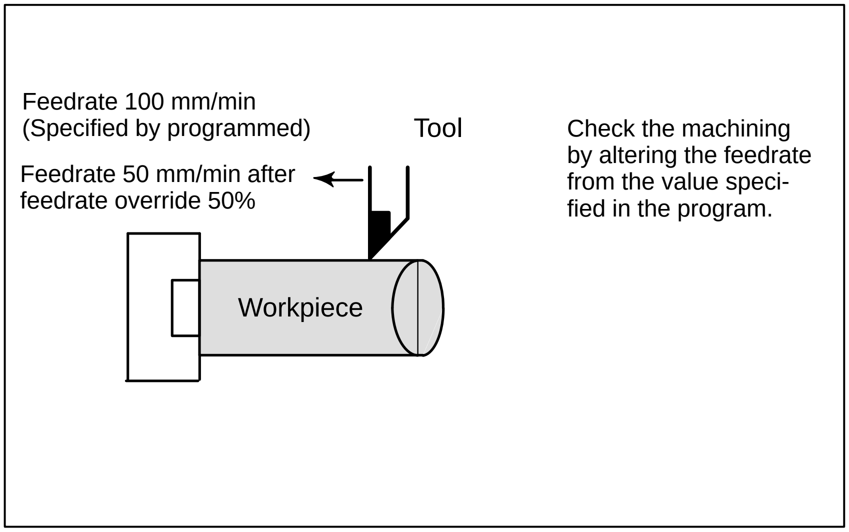
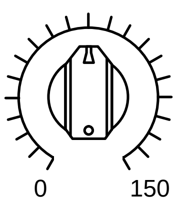


Fig. 5.2 Feedrate override

Procedure for Feedrate Override Operation



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 jog 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 150% (10% step). 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. 0533).

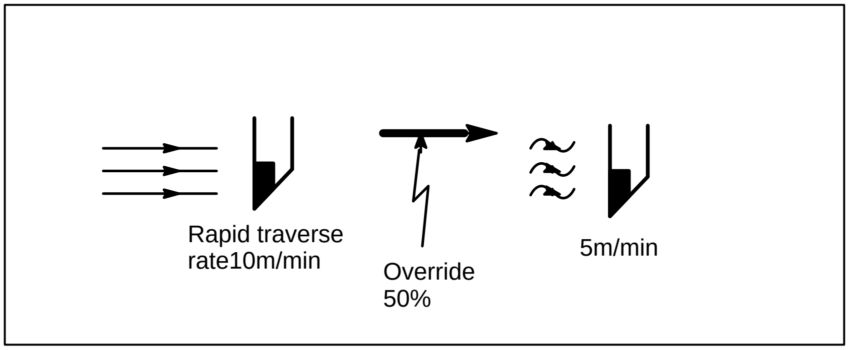
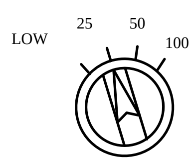


Fig. 5.3 Rapid traverse override

Procedure for Rapid Traverse Override Operation



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 taht the workpiece is removed from the table.

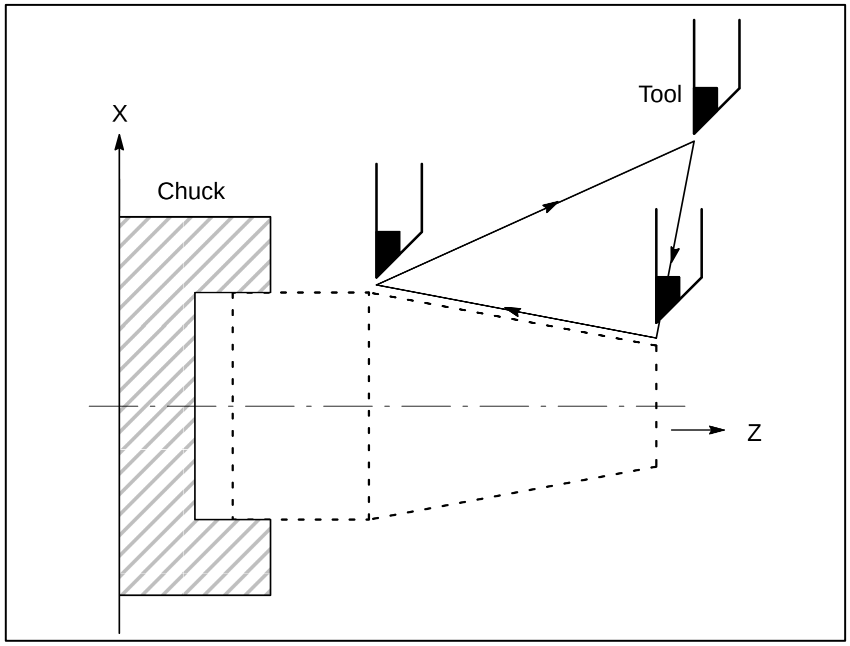


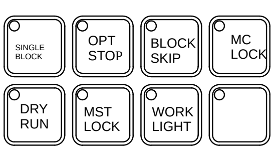
Fig. 5.4 Dry run

Procedure for Dry Run Operation

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	Max. jog feedrate
OFF	Jog feed rate or rapid traverse rate ***1)**	Jog feed rate

*1: Job feed rate when parameter (bit 6 of No. 0001) is 1.
Rapid traverse rate when parameter is 0.

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.

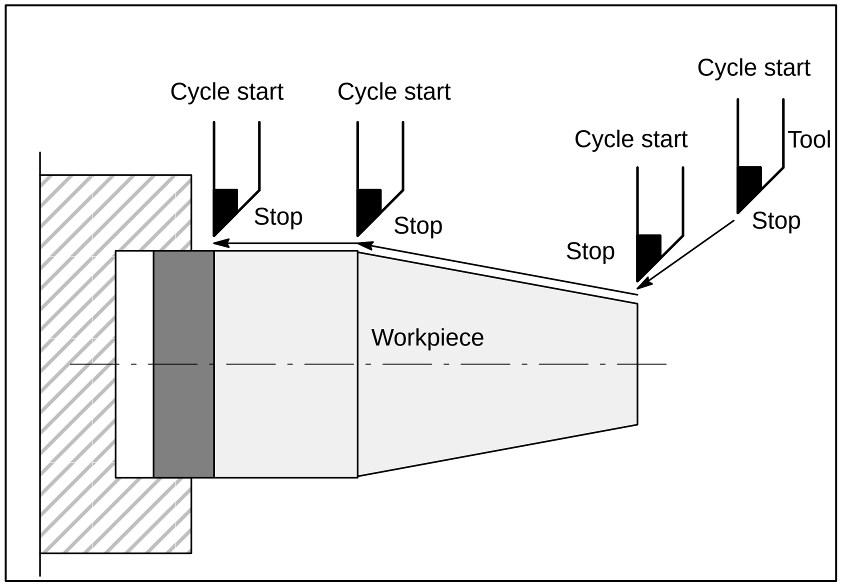


Fig. 5.5 Single block

Procedure for Single Block Operation

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.

S : Single block — — ➤ Rapid traverse ——➤ Cutting feed

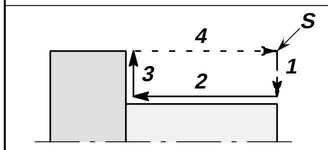
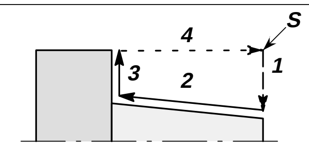
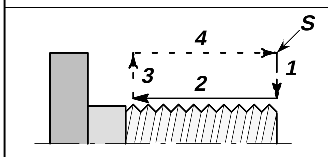
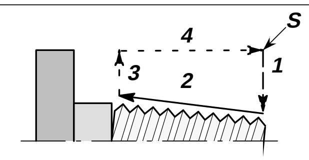
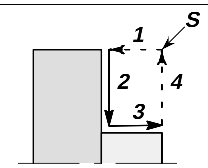
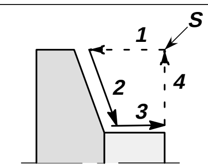
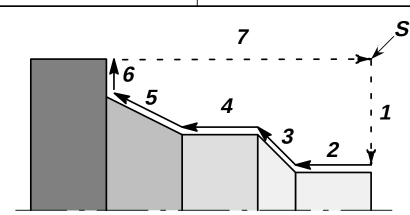
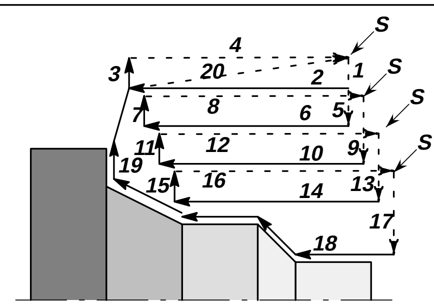
	Tool path		Explanation
☆G90 (Outer/inner turning cycle)	Straight cutting cycle 	Taper cutting cycle 	Tool path 1 to 4 is assumed as one cycle. After 4 is finished, a stop is made.
☆G92 (Threading cycle)	Straight threading cycle 	Taper threading cycle 	Tool path 1 to 4 is assumed as one cycle. After 4 is finished, a stop is made.
☆G94 (End surface turning cycle)	Straight end surface cutting cycle 	Taper end surface cutting 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

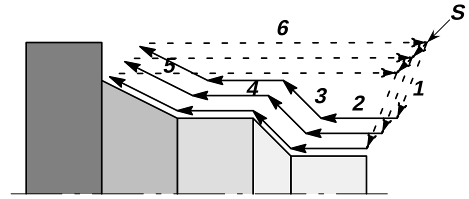
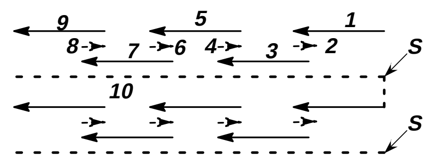
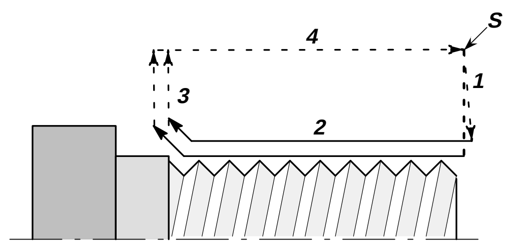
	Tool path	Explanation
☆G73 (Closed–loop cutting cycle)		Tool path 1 to 6 is assumed as one cycle. After 10 is finished, a stop is made.
☆G74 (End surface cutting–off cycle) G75 (Outer/inner surface cutting–off cycle)	 This figure shows the case for G74. G75 is the same.	Tool path 1 to 10 is assumed as 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_;. or G65. However, single block stop is even performed in a block with M98P_ command, if the block contains an address other than O, N or P.

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.

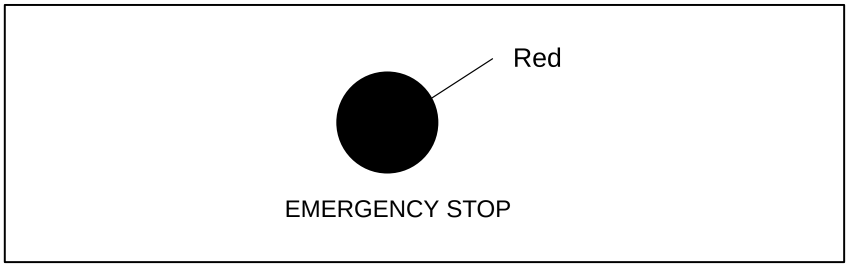


Fig. 6.1 Emergency stop

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 Z axis direction tool limit switch, the tool decelerates and stops because of working the limit switch and an OVER TRAVEL is displayed.

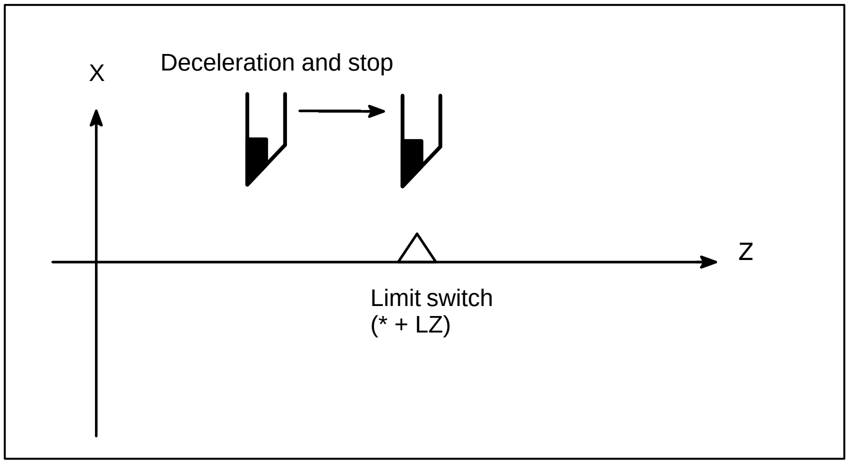


Fig. 6.2 Overtravel

Explanation

- **Releasing overtravel**

Press the reset button to reset the alarm after moving the tool to the Z axis minus direction by manual operation. For details on operation, refer to the operator's manual of the machine tool builder.

- **Alarm**

No.	Message	Description
520	Overtravel: +Z	The tool has exceeded the hardware–specified overtravel limit along the positive zth axis.

The overtravel limit signal (*+LZ) can be enabled or disabled, depending on bit 2 of parameter 0015. For details, refer to the manual of the machine tool builder.

6.3 STROKE CHECK

There areas which the tool cannot enter can be specified with stored stroke check 1/2.

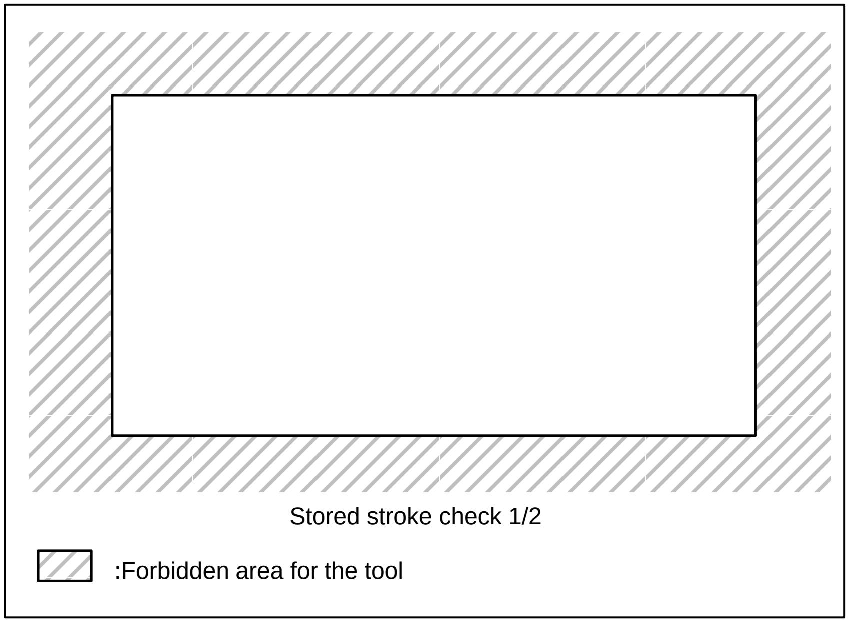


Fig. 6.3 (a) Stroke check

When the tool exceeds a stored stroke check, 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 check 1/2**

Parameters (Nos.0700 to 0705 or Nos.0770 to 0775) 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. The parameter used depends on the signal from the machine.

- **Checkpoint for the forbidden area**

The parameter setting value 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.

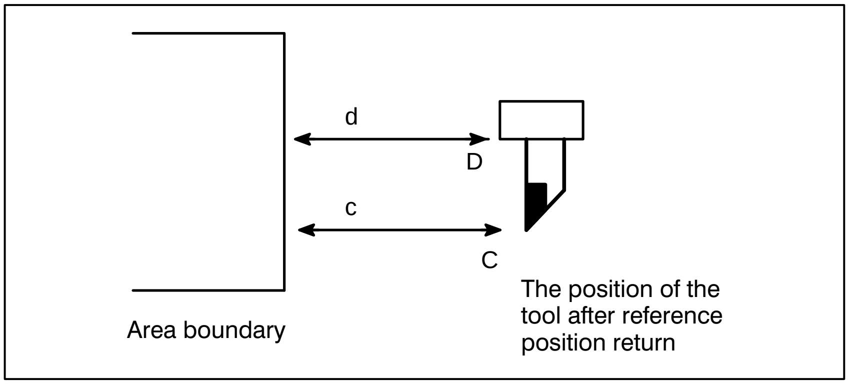


Fig. 6.3 (d) Setting the forbidden area

- **Amount of overrun relative to stored stroke check**

The maximum amount of overrun L (mm), relative to the stored stroke check, is calculated using the following formula, where F (mm/min) is the maximum rapid traverse rate:

L = F/7500

When L is set, the tool can enter a previously set inhibited area by up to L mm. You can also set bit 7 of parameter No. 0076 to stop the tool up to L mm outside the set inhibited area. In the latter case, the tool does not enter the inhibited area.

- **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.

- **Releasing the alarms**

When a stroke check alarm is issued, manually move the tool out of the inhibited area, in the direction opposite to that indicated in the alarm message. Then, press the reset key to release the alarm. If the tool has entered two inhibited areas at the same time, such that it cannot be manually moved out of either area, apply emergency stop, change the stroke check parameters to release the alarm, then manually move the tool out of the inhibited areas.

> **NOTE**
> In setting a forbidden area, if the two points to be set are the same, the area is as follows:
> When the forbidden area is check 1/2, all areas are forbidden areas.

Alarms

Number	Message	Contents
5n0	OVER TRAVEL: +n	Exceeded the n–th axis + side stored stroke check 1/2.
5n1	OVER TRAVEL: –n	Exceeded the n–th axis – side stored stroke check 1/2.

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.

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.

```
ALARM MESSAGE                          O0001  N0011

  511  OVER TRAVEL : –X










           ALARM                 MDI
[ ALARM ][  OPR  ][  MSG  ][        ][        ]
```

- **Another method for alarm displays**

In some cases, the alarm screen does not appear, but an ALM blinks at the bottom of the screen.

```
PARAMETER                              O0001  N0013

   (SETTING 1)
    TVON= 0
    ISO = 1   (0:EIA 1:ISO )
    INCH= 0   (0:MM  1:INCH)
    I/O = 0
    SEQ = 1





NO. TVON
           ALARM                    MDI
[ PARAM ][ DGNOS ][        ][ SV–PRM ][       ]
```

In this case, display the alarm screen as follows:

1. Press the function key OPR ALARM .

2. Press the 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 250: Program errors *1
No. 3n0 to 3n8: Absolute pulse coder (APC) alarms *2
No. 3n9 : Serial pulse coder (SPC) alarms *2
No. 400 to 495: Servo alarms
No. 510 to 581: Overtravel alarms
No. 600 to 607: PMC alarms
No. 700 to 704: Overheat alarms
No. 910 to 998: System alarms

*1) 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.

*2) n is number of control axis.

7.2 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 DGNOS PARAM.

2 Press the soft 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) – Press NO. key.
–Key input the number of the diagnostic data to be displayed.
–Press INPUT key.

```
DIAGNOSTIC                              O0001  N0011
   NO.     DATA            N0.     DATA
  0000   00000001         0010    00000000
  0001   00000000         0011    00000000
  0002   00000000         0012    00000000
  0003   00000000         0013    00000000
  0004   10000000         0014    00000000
  0005   00000000         0015    00000000
  0006   00000000         0016    00100010
  0007   00000000         0017    00100000
  0008   00000000         0018    10100000
  0009   00000000         0019    00000000

NO. 0000
                               AUTO
[ PARAM ][ DGNOS ][       ][SV-PRM ][       ]
```

Data of self–Diagnosis

	#7	#6	#5	#4	#3	#2	#1	#0
0700		CSCT	CITL	COVZ	CINP	CDWL	CMTN	CFIN

When a digit is "1", the corresponding status is effective.

CFIN : The M, S, O, or T function is being executed.

CMTN : A move command in the cycle operation is being executed.

CDWL : Dwell is being executed.

CINP : An in–posiiton check is being executed.

COVZ : Override is at 0%.

CITL : Interlock signal (STLK) is turned on.

CSCT : Speed arrival signal of spindle is turned on.

	#7	#6	#5	#4	#3	#2	#1	#0
0701			CRST					

CRST : One of the following : The reset button on the MDI panel, emergency stop, or remote reset is on.

	#7	#6	#5	#4	#3	#2	#1	#0
0712	STP	REST	EMS		RSTB			CSU

Indicates automatic operation stop or feed hold status. These are used for troubleshooting.

STP : The flag which stops the automatic operation. This is set at the following condition.
·External reset signal is turned on.
·Emergency stop signal is turned on.
·Feed hold signal is turned on.
·Reset button on the CRT/MDI panel is turned on.
·The mode is changed to the manual mode, such as JOG, HANDLE/STEP, TEACH INJOG, TEACH IN HANDLE.
·Other alarm is generated.

REST : This is set when one of the external reset, emergency stop, or reset button is set on.

EMS : This is set when the emergency stop is set on.

RSTB : This is set when the reset button is on.

CSU : This is set when the emergency stop is turned on, or when the servo alarm has been generated.

8 DATA INPUT/OUTPUT

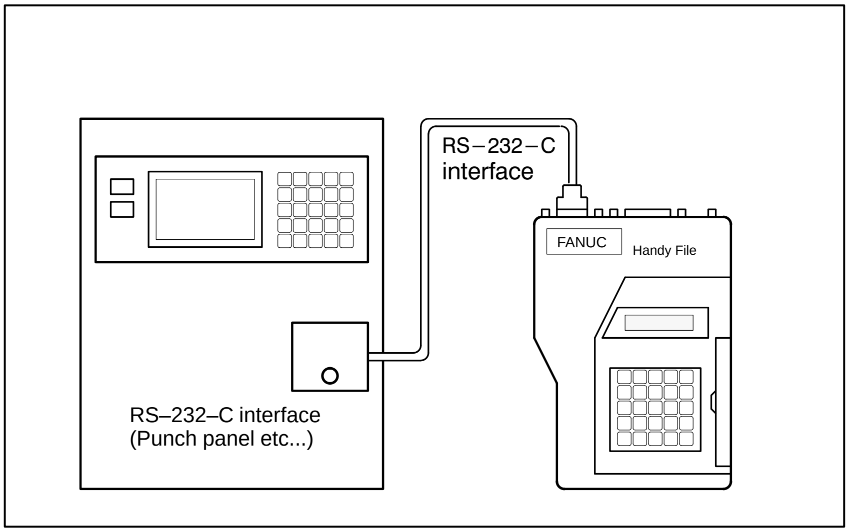
NC data is transferred between the NC and external input/output devices such as the Handy File.

The following types of data can be entered and output :

1.Program
2.Offset data
3.Parameter
4.Pitch error compensation data

Before an input/output device can be used, the input/output related parameters must be set.

For how to set parameters, see Chapter 2 **OPERATIONAL DEVICES**.



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[READ]or[PUNCH] key), is called a [file]. 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

Continuation 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.

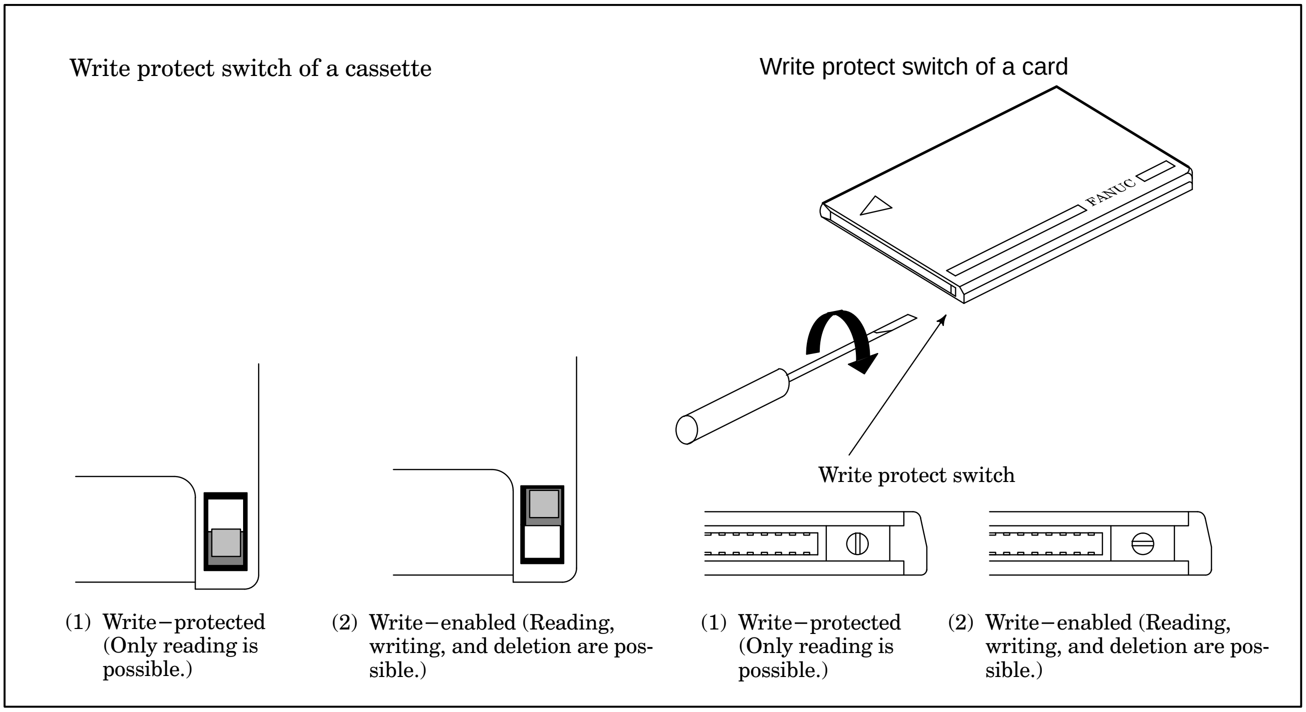


Fig. 8.1 Protect switch

• Writing memo

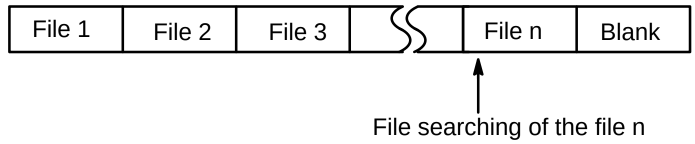
Once written in the cassette or card, data can subsequently be read out by correspondence between the data contents and file numbers. This correspondence cannot be verified, unless the data contents and file numbers are output to the CNC and displayed.
To display the contents,write the file numbers and 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 Operation

1 Press the EDIT or **AUTO** switch on the machine operator's panel.

2 Press function key PRGRM .

3 Enter address N .

4 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.

5 Press soft key INPUT .

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 PRGRM. Display program screen.

4 Enter address N .

5 Enter the number (from 1 to 9999) of the file to delete.

6 Press soft key OUTPT START

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
1 to (k>1) . . . 1 to (k>1)
k . . . . Deleted
(k+1) to n . . . k to (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.

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 **8.2**.

4 Press function key [PRGRM]

5 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.

6 Press soft key [INPUT]

The program is input and the program number specified is assigned to the program.

7 To stop input operation halfway, press the [RESET] key.

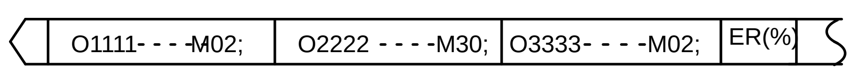
Explanations

- **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 alarm (P/S 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 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.

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.

- **Input with the soft keys**

The soft keys can be used to input a program.

Procedure for progarm input with the soft keys

1 Display the program screen in EDIT mode or background edit mode.

2 Press the **[I/O]** soft key.

3 Input address O, then the program number. If this step is skipped, the program number on the NC tape is automatically selected.

4 Press the **[READ]** soft key. To abandon input at any point, press the **[STOP]** soft key.

5 After input has been completed, press the **[CAN]** soft key to display the program screen again.

```
PROGRAM                          O0615 N0000
    SYSTEM EDITION 0671 – 04
  PROGRAM NO.  USED :    24 FREE : 39
  MEMORY AREA  USED : 24960 FREE : 97920
PROGRAM LIBRARY LIST
  O0001 O0010 O0011 O0021 O0041 O0601
  O0613 O0615 O0645 O0651 O1021 O1041
  O1051 O2011 O2505 O3148 O3153 O3511
  O4011 O4048 O5111 O5221 O5766 O6032




<
                             EDIT
[ PUNCH ][  READ  ][  CAN  ][       ][  STOP  ]
```

- **Program input in background edit mode**

Program input is identical to that in foreground edit mode. If the RESET key is pressed to abandon input in the background while a program is being executed, however, the program execution is also halted. To input a program in background edit mode, use the soft keys.

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.

2 To output to an NC tape, specify the punch code system (ISO or EIA) using a parameter. To output to floppy, specify ISO.

3 Press the EDIT switch on the machine operator's panel.

4 Press function key PRGRM.

5 Enter address O .

6 Enter a program number. If –9999 is entered, all programs stored in memory are output.

7 Press soft key OUTPT START

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 86 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 (No.0002#7 or 0012#7) to 1. This parameter makes the feed is not output, utilizing the memory efficiently.

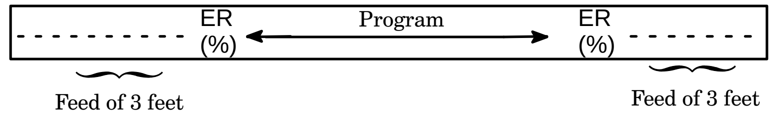
• On the memo record

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.

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

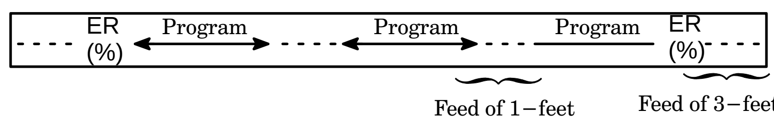
To set LF code only, set parameter bit 7 of No.0070.

- **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.

- **Output with the soft keys**

The soft keys can be used to input a progarm.

Procedure for program output with the soft keys

1 Display the program screen in EDIT mode.

2 Press the **[I/O]** soft key.

3 Input address O , then the program number. If–9999 is entered as the program number, all programs in memory are output.

4 Press the **[READ]** soft key. To abandon input at any point, press the **[STOP]** soft key.

5 After input has been completed, press the **[CAN]** soft key to display the program screen again.

```
PROGRAM                            O0615 N0000
    SYSTEM EDITION 0671 – 04
  PROGRAM NO.  USED :    24 FREE : 39
  MEMORY AREA  USED : 24960 FREE : 97920
PROGRAM LIBRARY LIST
  O0001 O0010 O0011 O0021 O0041 O0601
  O0613 O0615 O0645 O0651 O1021 O1041
  O1051 O2011 O2505 O3148 O3153 O3511
  O4011 O4048 O5111 O5221 O5766 O6032




<
                             EDIT
[ PUNCH ][  READ  ][  CAN  ][       ][  STOP  ]
```

- **Program output in background edit mode**

Program input is identical to that in foreground edit mode. If the RESET key is pressed to abandon input in the background while a program is being executed, however, the program execution is also halted. A program that is currently selected in the foreground can also be output.

> **NOTE**
> Some machines also use the OUTPT START key, used to start punching of the CNC tape in the background, as the cycle start key to start automatic operation. If such a machine is being used, punch the CNC tape in a mode other than automatic operation mode. (To check whether the key is used for both purposes, refer to the manual provided by the machine tool builder.)

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 **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.

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 8.2.

4 Press function key MENU OFSET , and display offset screen.

5 Press soft key INPUT .

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.

2 Specify the punch code system (ISO or EIA) using a parameter.
To output to floppy, specify ISO.

3 Press the EDIT switch on the machine operator's panel.

4 Press function key OFSET, and display offset screen.

5 Press soft key OUTPT START .
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
:P=0 . . . . Work sheet
:P=Wear offset number . . . . For wear offset amount
:p=10000+geometry offset number . . For geometry offset amount
X:Offset value on X axis
Z:Offset value on Z axis
Q:Imaginary tool nose number
R:Tool nose radius offset value

8.6 INPUTTING AND OUTPUTTING PARAMETERS AND PITCH ERROR COMPENSATION DATA

Pitch error compensation data is part of the parameter data. The same input/output operation as for other parameters can be used for pitch error compensation data. This section describes the method of parameter input/output operation.

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 Section **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.

2 When using a floppy, search for the required file according to the procedure in Section **8.2**.

3 Press the EMERGENCY STOP button on the machine operator's panel.

4 Press function key [DGNOS PARAM], and display parameter screen.

5 Enter 1 in response to the prompt for PWE (writing parameters). Alarm P/S100 appears.

6 Press soft key [INPUT].

Parameters are read into memory. Upon completion of input, the "INPUT" indicator at the lower–right corner of the screen disappears.

7 Enter 0 in response to the prompt for PWE (writing parameters).

8 Turn the power to the NC back on.

9 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.

2 Specify the punch code system (ISO or EIA) using a parameter. To output a floppy, specify ISO.

3 Press the EDIT switch on the machine operator's panel.

4 Press function key [DGNOS PARAM], and display parameter screen.

5 Press soft key [OUTPT START] .

All parameters (pitch error compensation is inclded) are output in the defined format.

Explanations

- **Output format**

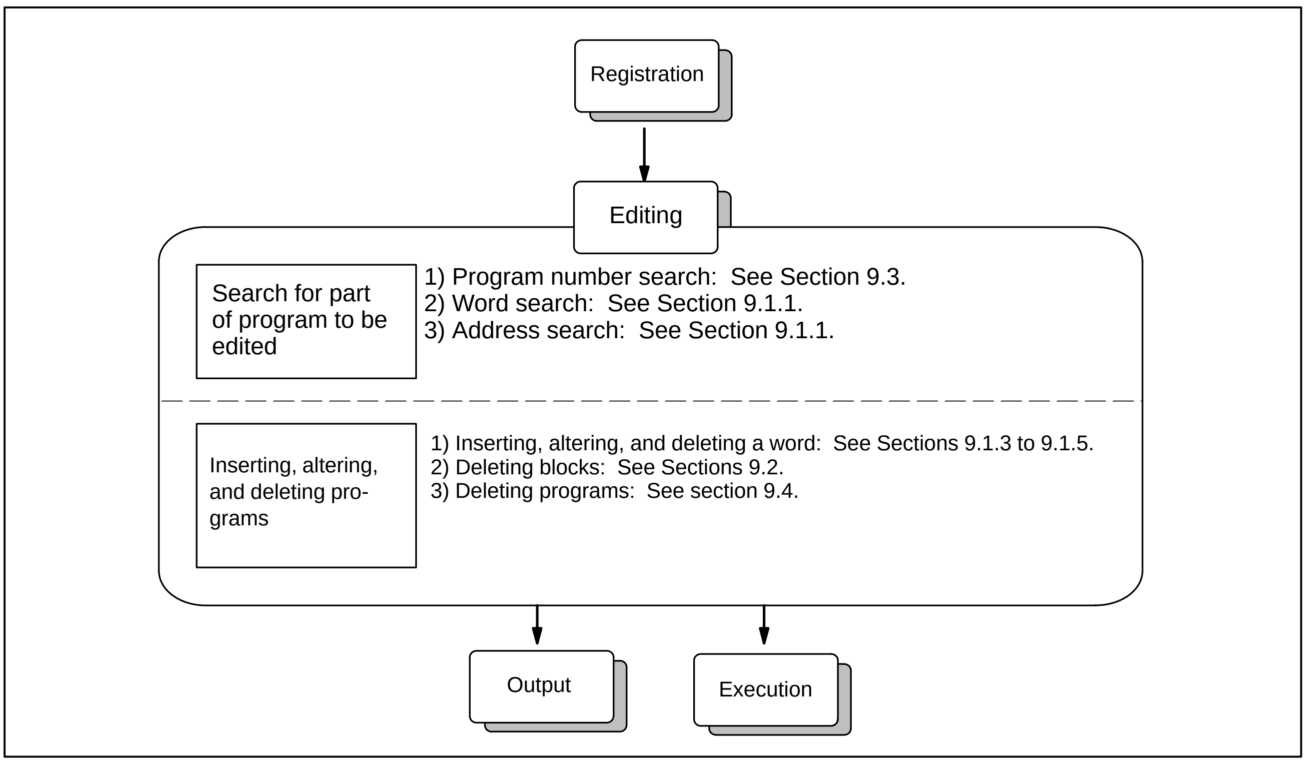
Output format is as follows:
N_P_;

N_ Parameter No.
P_ Parameter setting value .

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. This chapter also describes program 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 function PRGRM key and display the program screen.

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

- **Data input during editing**

To insert or modify a word during editing, the following data is entered.

- **When the standard key panel is being used**
One word (a single alphabetic character followed by a numeric value or symbol) is entered.

- **Editing B with the standard key panel**
Even if the standard key panel is being used, editing B can be enabled by specifying bit 7 of parameter 018 accordingly. Two or more addresses can be input at one time. If the INPUT key is pressed after a single word (a single alphabetic character followed by a numeric value or symbol) is input, another word can be input. After all data has been entered, press the edit key to start editing.
To enable editing B, note the following:

· The INPUT key is used to identify a breakpoint between words.
A program cannot be input or output while a program is displayed. Input or output a program on the program directory screen.

· Input a program number as one word containing address O.

· Up to 32 characters can be entered at one time.

· Each time the CAN key is pressed, only the most–recently entered character is deleted.

WARNING

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. The cursor is positioned to the address of the 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  N1234
 O0050  ;
 N1234  X100.0  Z1250.0 ;
 S12  ;
 N5678  M03  ;
 M02  ;
 %
```

3 Holding down the cursor key [↓] or [↑] scans words continuously.

4 Pressing the page key [↓] displays the next page and searches for the first word of the page.

5 Pressing the page key [↑] displays the previous page and searches for the first word of the page.

6 Holding down the page key [↓] or [↑] displays one page after another.

Procedure for searching a word

Example) of Searching for S12

```
PROGRAM                              O0050  N1234
 O0050  ;
 N1234  X100.0  Z1250.0  ;   <----  N1234 is being searched for/ scanned currently.
 S12  ;                      <----  S12 is searched for.
 N5678  M03  ;
 M02  ;
 %
```

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 cursor key [↓] starts search operation.

Upon completion of search operation, the cursor is displayed at ”S” of S12. Pressing the cursor key [↑] rather than the cursor [↓] key performs search operation in the reverse direction.

Procedure for searching an address

Example) of Searching for M03

```
PROGRAM                              O0050  N1234
 O0050  ;
 N1234  X100.0  Z1250.0  ;   <----  N1234 is being searched for/ scanned currently.
 S12  ;
 N5678  M03  ;               <----  M03 is searched for.
 M02  ;
 %
```

1 Key in address [M] .

2 Press the cursor key [↓].

Upon completion of search operation, the cursor is displayed at ”M” of M03. Pressing the [↑] key rather than the [↓] 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 two 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

1 Select **AUTO** or **EDIT** mode.

2 Press function PROG key and display the program.

3 Press the address key O .

4 Press the cursor 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 [INSRT] key.

Example of Inserting T15

Procedure

1 Search for or scan Z1250.0.

```
Program                                O0050  N1234
 O0050  ;
 N1234  X100.0  Z1250.0  ;
 S12  ;
 N5678  M03  ;
 M02  ;
 %
```

Z1250.0 is searched for/ scanned.

2 Key in [T] [1] [5] .

3 Press the [INSRT] key.

```
Program                                O0050  N1234
 O0050  ;
 N1234  X100.0  Z1250.0  T15  ;
 S12  ;
 N5678  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  N1234
 O0050  ;
 N1234  X100.0  Z1250.0  T15  ;
 S12  ;
 N5678  M03  ;
 M02  ;
 %
```

T15 is searched for/scanned.

2 Key in M 1 5 .

3 Press the ALTER key.

```
Program                                O0050  N1234
 O0050  ;
 N1234  X100.0  Z1250.0  M15  ;
 S12  ;
 N5678  M03  ;
 M02  ;
 %
```

T15 is changed to M15.

9.1.5 Deleting a Word

Procedure for deleting a word

1 Search for or scan a word to be deleted.

2 Press the DELET key.

Example of deleting X100.0

Procedure

1 Search for or scan X100.0.

```
Program                                   O0050  N1234
 O0050 ;
 N1234 X100.0 Z1250.0 M15 ;
 S12 ;
 N5678 M03 ;
 M02 ;
 %
```

X100.0 is searched for/ scanned.

2 Press the DELET key.

```
Program                                   O0050  N1234
 O0050 ;
 N1234 Z1250.0 M15 ;
 S12 ;
 N5678 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 DELET key.

Example of deleting a block of No.1234

Procedure

1 Search for or scan N1234.

```
Program                                  O0050  N1234
 O0050  ;
 N1234  Z1250.0  M15  ;
 S12  ;
 N5678  M03  ;
 M02  ;
 %
```

N1234 is searched for/ scanned.

2 Key in EOB .

3 Press the DELET key.

```
Program                                  O0050  N1234
 O0050  ;
 S12  ;
 N56789  M03  ;
 M02  ;
 %
```

Block containing N1234 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 [DELET] key.

Example of deleting blocks from a block containing N01234 to a block containing N56789

Procedure

1. Search for or scan N1234.

```
Program                              O0050  N1234
 O0050 ;
 N1234  Z1250.0  M15 ;    <------  N1234 is searched for/ scanned.
 S12 ;
 N56789  M03 ;
 M02 ;
 %
```

2. Key in [N] [5] [6] [7] [8] .

```
Program                              O0050  N1234
 O0050 ;
 N1234  Z1250.0  M15 ;  }
 S12 ;                  } <------  Underlined part is deleted.
 N5678  M03 ;           }
 M02 ;
 %
```

3. Press the [DELET] key.

```
Program                              O0050  N1234
 O0050 ;
 M02 ;   <------  Blocks from block containing N1234 to block containing N5678 have been deleted.
 %
```

9.3 PROGRAM NUMBER SEARCH

When memory holds multiple programs, a program can be searched for. There are two methods as follows.

Procedure for program number search

Method 1

1. Select **EDIT** or **AUTO** mode.
2. Press [PRGRM] key to display the program screen.
3. Key in address [O] .
4. Key in a program number to be searched for.
5. Press the cursor 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 **AUTO** mode.
2. Press [PRGRM] key to display the program screen.
3. Key in address [O] .
4. Press the cursor key [↓].
   In this case, the next program in the directory is searched for .

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 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.

9.4.1 Deleting One Program

A program registered in memory can be deleted.

Procedure for deleting one program

1 Select the **EDIT** mode.

2 Press PRGRM to display the program screen.

3 Key in address O .

4 Key in a desired program number.

5 Press the DELET key.
The program with the entered program number is deleted.

9.4.2 Deleting All Programs

All programs registered in memory can be deleted.

Procedure for deleting all programs

1 Select the **EDIT** mode.

2 Press PRGRM to display the program screen.

3 Key in address O .

4 Key in –9999.

5 Press edit key DELET to delete all programs.

9.5 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).
During background editing, all programs cannot be deleted at once.

Procedure for background editing

1 Press function key PRGRM , display program screen.

2 Press soft key ▷ at the right side, and press soft key **[BG–EDT]**.
The background editing screen is displayed (PROGRAM (BG–EDIT) is displayed at the top left of the screen).

3 Edit a program on the background editing screen in the same way as for ordinary program editing.

4 After editing is completed, press soft key **[BG–EDT].**

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.

CAUTION

1 Free space in memory is used for background editing. A program to be subjected to background editing is copied into the free area in memory, then the original program is deleted. Subsequently, editing starts.
Background editing can be executed if the part program storage has sufficient free area to which the target program can be copied and if program registration is allowed in terms of number. If background editing is repeated, the number of deleted areas will increase.
To use these deleted areas efficiently, memory must be reorganized.

2 If the reset key is pressed to abandon program input or output in background editing, the machining in the foreground will also be halted. To input or output a program in the background, therefore, use the soft keys. To halt the input or output, press the **[STOP]** soft key.

3 If a reset by M02/M30 of the machining program in the foreground is executed during program input or output in background editing, program input or output is halted. Program input or output can be prevented from being halted by the reset in the foreground if bit 2 of parameter 076 is specified accordingly.

4 In background editing, program input or output by the external activation signal (MINP) or input/output unit external control is inhibited.

9.6 REORGANIGING MEMORY

If the available part program storage is 120 m or more, or if the background editing function is supported, repeated program editing will create many small, unused areas in memory. Reorganizing memory arranges these unused areas into a single, contiguous area that can be used by programs.

Procedure for Reorganiging Memory

Procedure 1 (RESET key)

Press the emergency stop, external reset, or reset key. The procedure for reorganizing memory is automatically started.

Procedure 2 (soft key)

1. Select **EDIT** mode.
2. Press the PRGRM key to display the program.
3. Press the **[LIB]** soft key.
4. Press the **[REORGANIZE]** soft key.

NOTE

1. One memory is reorganized, the system searches for the beginning of the selected program and the cursor is returned to that point.
2. If the power is turned off during the memory reorganization, alarm 101 occurs when the power is subsequently turned on. Before turning the power off after resetting an alarm, first check whether memory reorganization has been completed. While memory reorganization is being performed, EDIT blinks at the bottom right corner of the screen.
3. As described in procedure 1, above, the memory reorganization procedure is automatically started when a reset is performed. Memory reorganization can be prevented from being started by a reset if bit 0 of parameter 056 is specified accordingly.

10 CREATING PROGRAMS

Programs can be created using any of the following methods:

- MDI keyboard

This chapter describes creating programs using the MDI panel. 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 9.

Procedure for Creating Programs Using the MDI Panel

Procedure

1. Enter the **EDIT** mode.
2. Press the PRGRM key, and display program screen.
3. Press address key O and enter the program number.
4. Press the INSRT key.
5. Create a program using the program editing functions described in Chapter 9.

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 0550.

Procedure for automatic insertion of sequence numbers

Procedure

1 Set 1 for SEQUENCE (see subsec. 11.5.1).

2 Enter the **EDIT** mode.

3 Press PRGRM 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 INSRT 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 INSRT.

7 Enter each word of a block.

8 Press EOB.

9 Press INSRT. 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 10, N20 inserted and

displayed below the line where a new block is specified.

```
PROGRAM                              O0040  N0020
O0040 ;
N10 G50 X0 Z0 ;
N20
%









<
                                 EDIT
[ PRGRM ][  LIB  ][  I/O  ][        ][        ]
```

10 ·In the example above, if N20 is not necessary in the next block, pressing the DELET key after N20 is displayed deletes N20.
·To insert N100 in the next block instead of N20, enter N100 and press ALTER after N20 is displayed. N100 is registered and initial value is changed to 100.

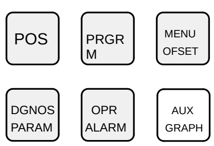
11 SETTING AND DISPLAYING DATA

General

To operate a CNC machine tool, various data must be set on the CRT/MDI panel. 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.
This chapter describes the procedure, assuming that the soft keys are used to select a desired chapter. If the soft keys are not supported, press the key having the equivalent function two or more times to select the desired chapter.

Explanations

·Screen transition chart



MDI function keys
(Shaded keys ( ) are described in this chapter.)

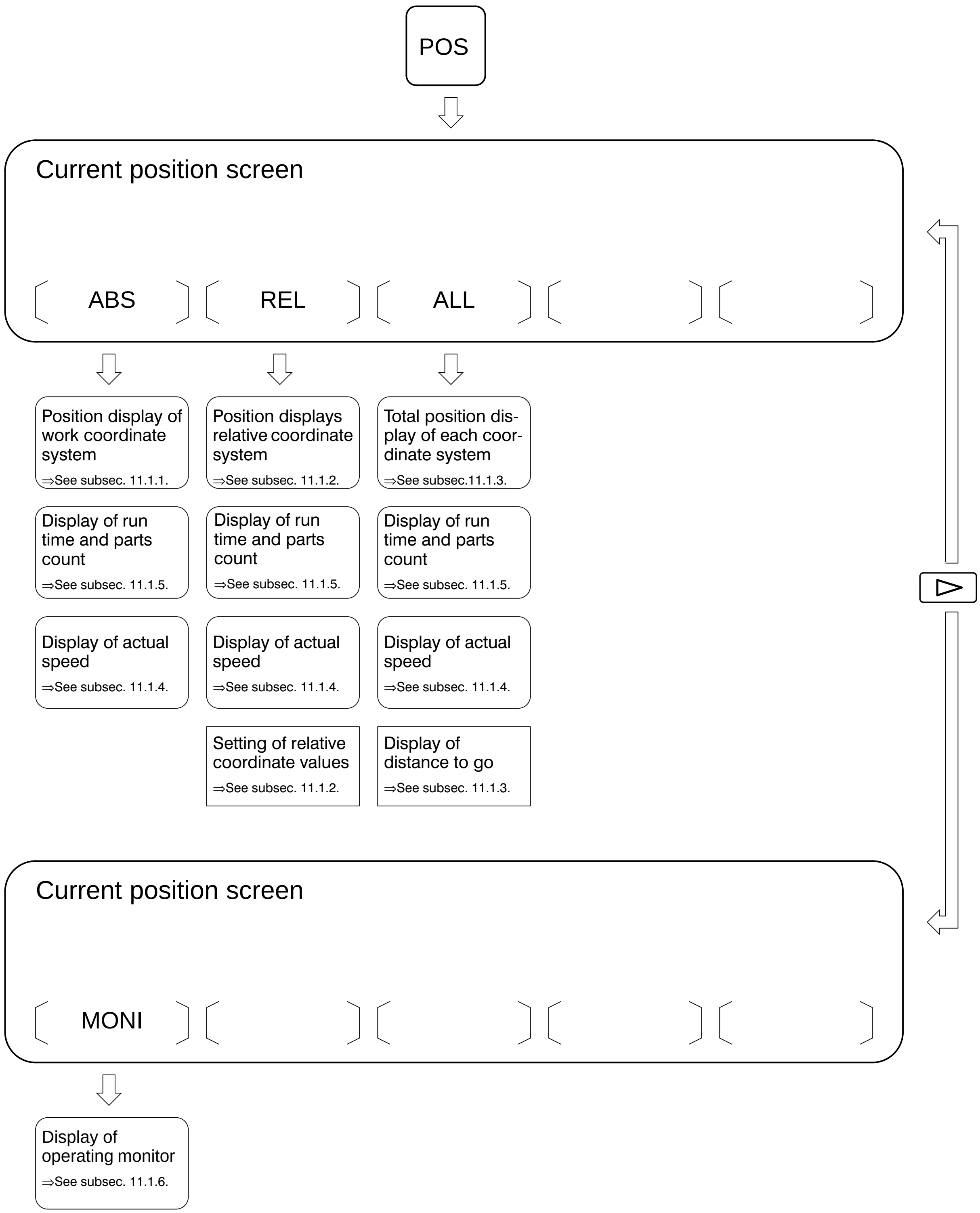
The screen transition for when each function key on the MDI panel is pressed is shown below. The subsections referenced for each screen are also shown. See the appropriate subsection for details of each screen and the setting procedure on the screen. See other chapters for screens not described in this chapter.
Function key AUX GRAPH is not used.

● Data protection key

The machine may have a data protection key to protect part programs. Refer to the manual issued by the machine tool builder for where the data protection key is located and how to use it.

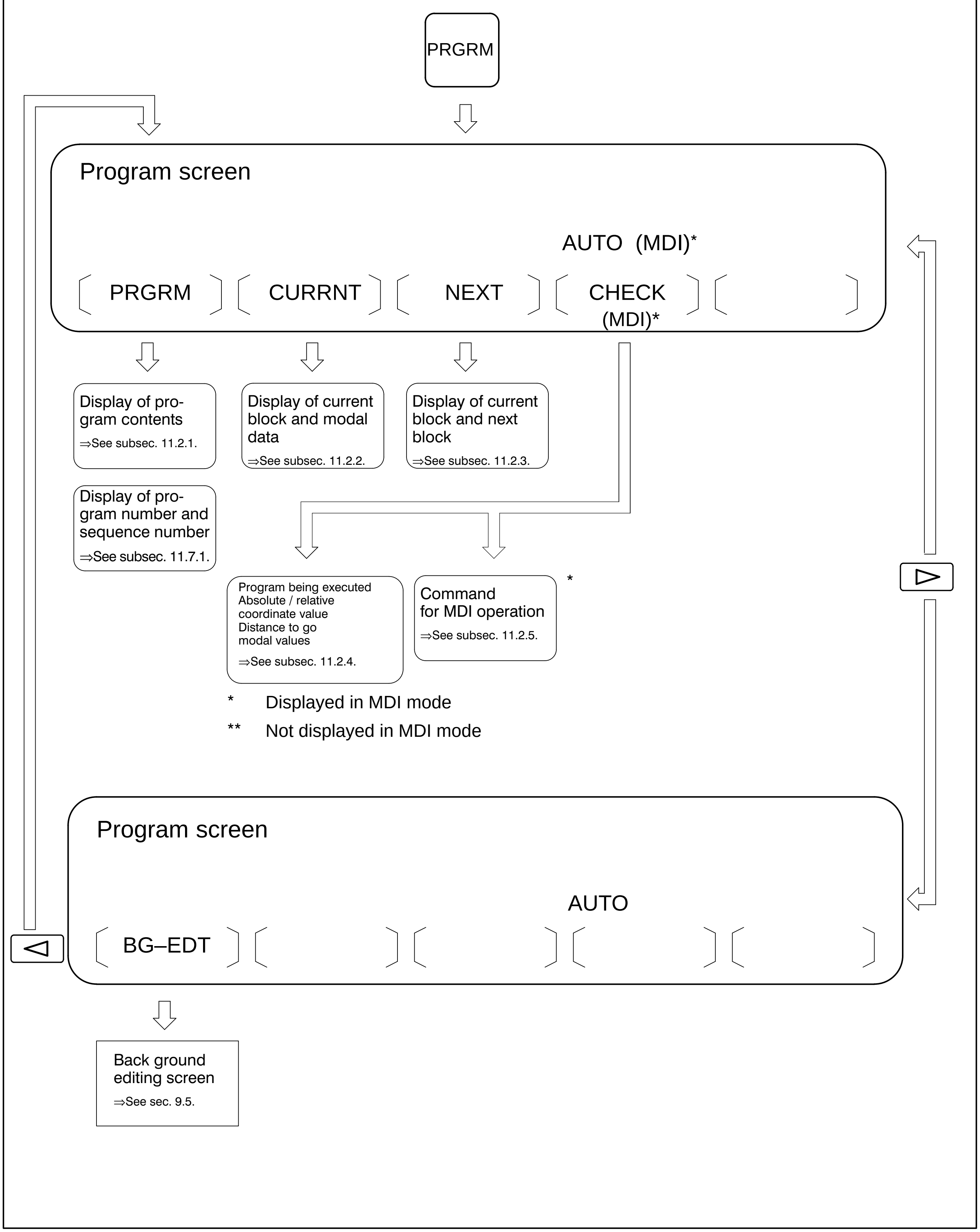
POSITION DISPLAY SCREEN

Screen transition triggered by the function key POS



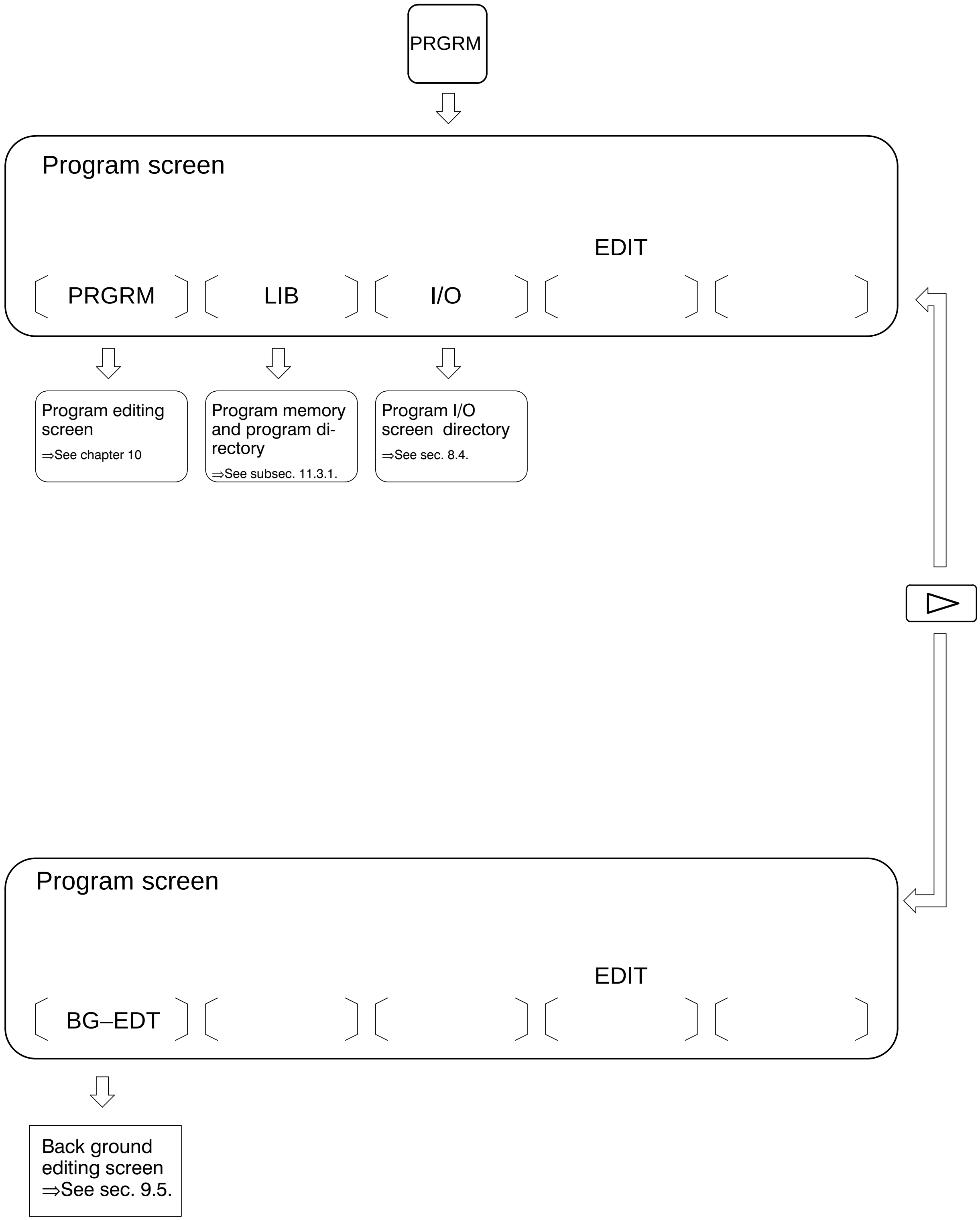
PROGRAM SCREEN

Screen transition triggered by the function key PRGRM in the AUTO or MDI mode



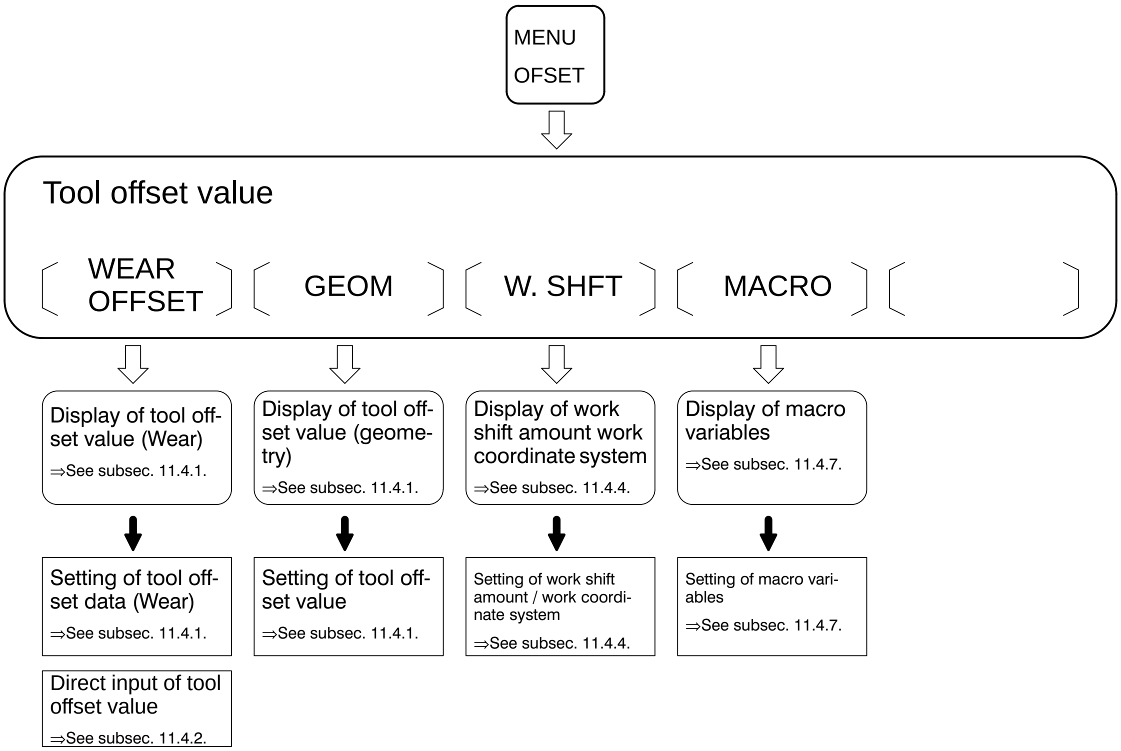
PROGRAM SCREEN

Screen transition triggered by the function key PRGRM in the EDIT mode



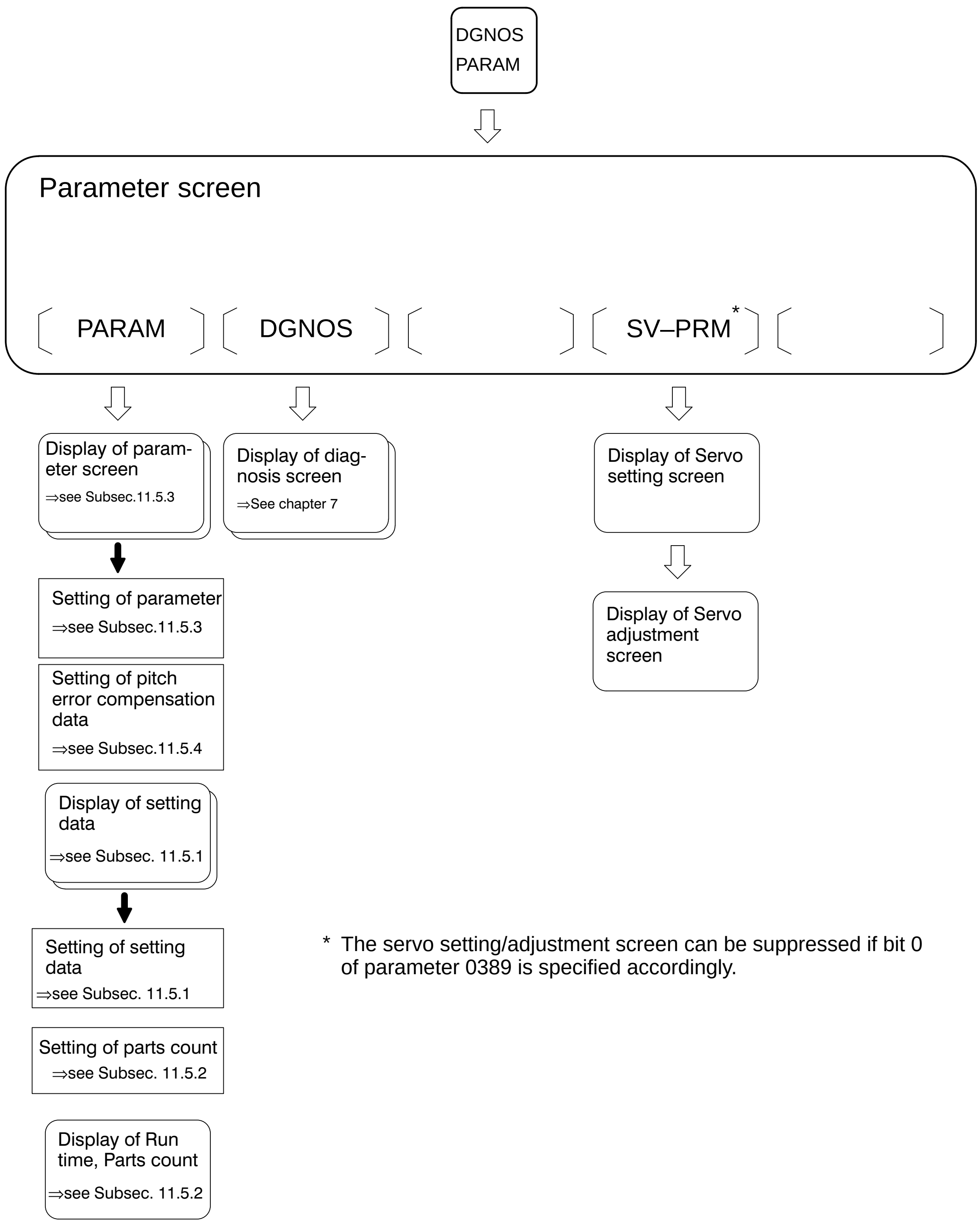
OFFSET SCREEN

Screen transition triggered by the function key MENU OFSET



PARAMETER/DIAGNOSTIC SCREEN

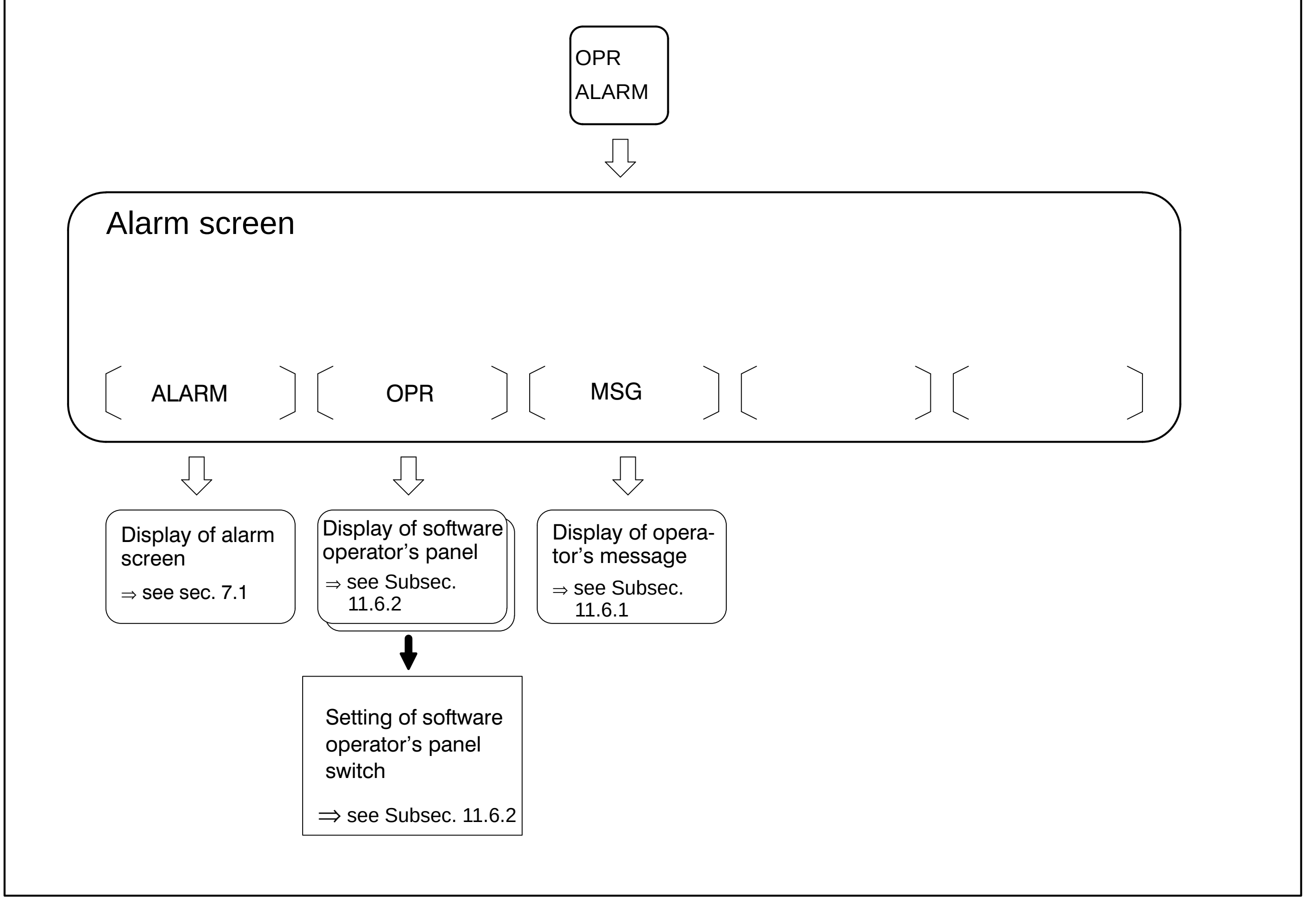
Screen transition triggered by the function key DGNOS PARAM



* The servo setting/adjustment screen can be suppressed if bit 0 of parameter 0389 is specified accordingly.

ALARM SCREEN

Screen transition triggered by the function key OPR ALARM



- **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
		Counter input of offset value	Subsec. 11.4.3
2	Workpiece coordinate system setting	Workpiece coordinate system shift value	Subsec. 11.4.4
		Workpiece origin offset value	Subsec. 11.4.6
3	Setting data (handy)	Parameter write TV check Punch code (EIA/ISO) Input unit (mm/inch) I/O channel Automatic insert of Sequence No.	Subsec. 11.5.1
5	Setting data (timer)	Parts required	Subsec. 11.5.2
6	Macro variables	Custom macro common variables (#100 to #149) (#500 to #531)	Subsec. 11.4.7
7	Parameter	Parameter	Subsec. 11.5.3
		Pitch error compensation data	Subsec. 11.5.4
8	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. 11.6.2
9	Current position display screen	Reset of relative coordinate value.	Subsec. 11.1.2

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.

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]**.

```
ACTUAL POSITION (ABSOLUTE)      O0001 N0023
 X      200.000
 Z      220.000
 C        0.000



                           PART COUNT  1786
RUN TIME       2H47M   CYCLE TIME  0H 1M47S
ACT.F   3000 MM/M
                             BUF AUTO
[  ABS  ][  REL  ][  ALL  ][        ][       ]
```

Explanations

- **Display including compensation values**

Tool compensation values and other values are displayed on the screen displaying a position in the workpiece coordinate system. The programmed point (position of the tool tip) is also displayed.

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]**.

```
ACTUAL POSITION (RELATIVE)      O0001 N0023
 U    200.000
 W    220.000
 H      0.000

                           PART COUNT  23
RUNTIME     3H30M          CYCLE TIME  0H 2M14S
ACT.F  3000 MM/M
                                    AUTO
[  ABS  ][  REL  ][  ALL  ][       ][       ]
```

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 reset the axis coordinate to a specified value

1 Key in the address of the axis name (X, Z, etc.) on the relative coordinate screen. The entered axis address blinks. Two or more axis names can be input.

2 Press the CAN key. The relative coordinates of the axis having the blinking address are reset to 0.

Procedure topreset a value for a specified axis

1 Key in the desired axis name and value on the relative coordinate screen. The entered axis address blinks.

2 Press the INPUT key. The relative coordinate of the axis with the blinking address is preset to the specified value.

To enable this operation, specify bit 0 of parameter 0064 accordingly. In this mode, a reset cannot be performed for the specified axis. To reset the coordinate, key in 0 as the preset value.

- **Display including compensation values**

Bit 1 of parameter 0001 can be used to select whether the displayed values include tool length offset and cutter compensation.

- **Presetting by setting a coordinate system**

Bit 1 of parameter 0002 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 command or when the manual reference position return is made.

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.

Procedure for displaying overall position display screen

1 Press function key POS .

2 Press soft key **[ALL]**.

```
ACTUAL POSITION                  O0100 N0000
     (RELATIVE)           (ABSOLUTE)
    U     0.000          X   200.179
    W     0.000          Z   220.000
    H     0.000          C     0.000

     (MACHINE)         (DISTANCE TO GO)
    X  –118.170          X     0.000
    Z   –21.470          Z     0.000
    C     0.676          C     0.000

                       PART COUNT   23
RUN TIME     3H30M     CYCLE TIME  0H2M14S
ACT.F       0 MM/M
                           AUTO
[  ABS  ][  REL  ][  ALL  ][       ][       ]
```

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 AUTO 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 also be used by setting bit 0 of parameter 0063.

11.1.4 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 2 of parameter 0028.

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)      O0001 N0023
 X      200.000
 Z      220.000
 C        0.000

                           PART COUNT  1786
RUN TIME      2H47M   CYCLE TIME  0H 1M47S
ACT.F   3000 MM/M
                             BUF AUTO
[  ABS  ][  REL  ][  ALL  ][        ][       ]
```

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).

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.

- **Actual feedrate display on the other screen**

The program check screen also displays the actual feedrate.

11.1.5 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 (ABSOLUTE)      O0001 N0023
 X     200.000
 Z     220.000
 C       0.000

                           PART COUNT  1786
RUN TIME       2H47M   CYCLE TIME   0H 1M47S
ACT.F   3000 MM/M
                             BUF AUTO
[  ABS  ][  REL  ][  ALL  ][        ][       ]
```

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 0219 is executed.
The number is reset to 0 when CAN key is pressed after address P key is pressed.

- **RUN TIME**

Indicates the total run time during automatic operation, excluding the stop and feed hold time.
The number is reset to 0 when CAN key is pressed after address R key is pressed.

- **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 11.5.1.

- **Parameter setting**

The number of machined parts and run time cannot be set on current position display screens. They can be set on the setting screen.

- **Incrementing the number of machined parts**

Bit 3 (PCM) of parameter 0040 is used to specify whether the number of machined parts is incremented each time M02, M03, or an M code specified by parameter 0219 is executed, or only each time an M code specified by parameter 0219 is executed.

11.1.6 Operating Monitor Display

This function displays the load of basic feed axes and 1st spindle with serial interface. And also, it is possible to display the speed of 1st spindle with serial interface.
This function is basic.

Explanations

- **Operation**

1 The position screen is selected by pressing the function key POS .

2 The rightmost key **[>]** on the soft–keys is pressed.

3 The soft–key **[MONI]** is pressed.

By the above operation, the operating monitor screen is displayed. Pressing the page keys ↑ ↓ can display the screen instead of the above operation (2) and (3).

- **Operating Monitor Screen**

```
OPERATING MONITOR                    O0001 N0000
   (LOAD METER)


  X:**********     0%   S1:**********    0%
  Z:**********     0%
                         (SPEED METER  RPM)
                        S1:**********    0%






[ MONI  ][       ][       ][       ][       ]
```

(1) LOAD METER (X to Z)
The load of the axis is displayed by percentages to the rated torque of the axis.
One ”*” marks on the bar graph denotes 10%.

(2) LOAD METER (S1)
The load of 1st spindle is displayed by percentages to the rated power of the spindle.
One ”*” marks on the bar graph denotes 20%.

(3) SPEED METER (S1)
The speed of 1st spindle is displayed by RPM. One ”*” marks on the bar graph denotes 10% of the maximum speed of the spindle.

NOTE
1 The loads of only the basic axes are displayed.
2 The load and speed of only 1st spindle with serial interface are displayed.Those for the 2nd serial spindle and analog interface spindle are not displayed.
3 This function is enabled by setting bit 5 of parameter No. 060 to 1.

11.2 SCREENS DISPLAYED BY FUNCTION KEY PRGRM (IN AUTO MODE OR MDI MODE)

This section describes the screens displayed by pressing function key PRGRM in AUTO or MDI mode.The first four of the following screens display the execution state for the program currently being executed in AUTO or MDI mode and the last screen displays the command values for MDI operation in the MDI mode:

1. Program contents display screen
2. Current block display screen
3. Next block display screen
4. Program check screen
5. Program screen for MDI operation

11.2.1 Program Contents Display

Displays the program currently being executed in AUTO mode.

Procedure for displaying the program contents

1 Press function key PRGRM to display a program screen.

2 Press chapter selection soft key **[PRGRM]**.
The cursor is positioned at the block currently being executed.

```
PROGRAM                         O0100 N0013
S550 M08 ;
M45 ;
N010 G50 X200. Z200. ;
N011 G00 X160. Z180. ;
N012 G71 U7. R1. ;
N013 G71 P014 Q020 U4. W2. F0.3 S550 M03
 ;
N014 G00 X40. F0.15 S700 ;




                         AUTO
[ PRGRM ][CURRNT ][ NEXT  ][ CHECK ][       ]
```

11.2.2 Current Block Display Screen

Displays the block currently being executed and modal data in the AUTO or MDI mode.

Procedure for displaying the current block display screen

1 Press function key PRGRM .

2 Press soft key **[CURRNT]**.
The block currently being executed and modal data are displayed.

```
PROGRAM                               O0001 N0013
    (CURRENT)                    (MODAL)
  G71 U     4.000              G00 F    0.3000
      W     2.000              G97 M       003
      F    0.3000              G69 S     00550
      M       003              G99 T      0101
      S     00550              G21
      P        14              G40 WX    0.000
      Q        20              G25 WZ    0.000
                               G22 SRPM    550
                               G54 SSPM      0
                                    SMAX 32767
                                  SACT 0

                             AUTO
[ PRGRM ][CURRNT ][ NEXT  ][ CHECK ][       ]
```

11.2.3 Next Block Display Screen

Displays the block currently being executed and the block to be executed next in the AUTO or MDI mode.

Procedure for displaying the next block display screen

1 Press function key PRGRM.

2 Press chapter selection soft key **[NEXT]**.
The block currently being executed and the block to be executed next are displayed.

```
PROGRAM                              O0100 N0000
    (CURRENT)            (NEXT)
  G71 U     4.000       G71.X   140.000
      W     2.000       G97 M   -20.000
      F    0.3000
      M       003
      S     00550
      P        14
      Q        20



                           AUTO
[ PRGRM ][CURRNT ][ NEXT  ][ CHECK ][       ]
```

11.2.4 Program Check Screen

Displays the program currently being executed, current position of the tool, and modal data in the AUTO mode.

Procedure for displaying the program check screen

1 Press function key PRGRM.

2 Press soft key **[CHECK]**.
The program currently being executed, current position of the tool, and modal data are displayed.

```
PROGRAM                          O0100 N0000
N013 G71 P014 Q020 U4. W2.   F0.3 S550 M03
 ;
N014 G00 X40. F0.15 S700 ;
N015 G01 W–40. ;
 (RELATIVE)   (DIST TO GO)    (G)
 U   –72.000  X     0.000  G00 G99 G25
 W   –76.000  Z     0.000  G97 G21 G22
 H     0.000  C     0.000  G69 G40 G54
                           SPRM    550
                           SSPM      0
 F    0.3000  S00550       SMAX  32767
 M003         T0101        SACT      0
ACT.F        0 MM/M
                      BUF AUTO
[ PRGRM ][CURRNT ][ NEXT  ][ CHECK ][      ]
```

Explanations

- **Program display**

For the program currently being executed, the block currently being executed is displayed first.

- **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 parameter (No.0028#0).

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

Procedure

1 Press function key PRGRM .

2 Press soft key **[MDI]**.
The program input from the MDI and modal data are displayed.

(1) MDI operation – A

```
PROGRAM                        O0001 N0020
        (MDI)            (MODAL)
  G00.X   100.000      G00 F    0.3000
  G80.Z   200.000      G97 M       003
                       G69 S     00550
                       G99 T      0101
                        G21
                       G40 WX    0.000
                       G25 WZ    0.000
                       G22 SRPM    550
                       G54 SSPM      0
                           SMAX  32767
                           SACT      0
ADRS.
                           MDI
[ PRGRM ][CURRNT ][ NEXT  ][  MDI  ][       ]
```

Explanations

- **MDI operation**

See Section 4.2 for MDI operation.

11.3 SCREENS DISPLAYED BY FUNCTION KEY PRGRM (IN THE EDIT MODE)

This section describes the screens displayed by pressing function key PRGRM in the EDIT mode. Function key PRGRM in the EDIT mode can display the program editing screen and the library screen (displays memory used and a list of programs).

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.

2 Press function key PRGRM.

3 Press soft key **[LIB]**.

```
PROGRAM                                O0615 N0000
    SYSTEM EDITION 0671 – 04
  PROGRAM NO.  USED :    24 FREE : 39
  MEMORY AREA  USED : 24960 FREE : 97920
PROGRAM LIBRARY LIST
  O0001 O0010 O0011 O0021 O0041 O0601
  O0613 O0615 O0645 O0651 O1021 O1041
  O1051 O2011 O2505 O3148 O3153 O3511
  O4011 O4048 O5111 O5221 O5766 O6032





<
                           EDIT
[ PRGRM ][CONDNS ][       ][       ][       ]
```

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 (No. 0040#0) to 1.

```
PROGRAM                          O0615  N0000
    STSTEM EDITION       0671 – 04
  PROGRAM NO.  USED :     24 FREE :39
  MEMORY AREA  USED : 24960 FREE : 97920
PROGRAM LIBRARY LIST
 O0001 (TEST–PRO)
 O0010 (MILLING)
 O0011 (THREADING)
 O0021 (TURNING–ROUGH–1)
 O0041 (TURNING–ROUGH–2)
 O0601 (GROOVING)
 O0613 (TURNING–FINE–1)
 O0615 (TURNING–FINE–2)
<
                              EDIT
[ PRGRM ][CONDNS ][       ][       ][       ]
```

- **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.

○ □□□□ (ΔΔΔ…Δ) ;

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 of parameter 0040 is set to 1, programs are displayed in the order of program number starting from the smallest one.

Explanations

(Example) Assume SOR (bit 4 of parameter No. 0040) is set to 0.

(1) After clearing the program, register O0001, O0002, O0003, O0004, and O0005 in this order. Then, displaying a program listing produces the following output:
O0001 O0002 O0003 O0004 O0005

(2) Delete O0002 and O0004. Then, displaying a program listing produces the following output:
O0001 O0003 O0005

(3) Register O0009. Then, displaying a program listing produces the following output:
O0001 O0009 O0003 O0005

- **Free area**

Unused areas resulting from background editing are included when the number of programs or memory areas used is counted. After memory is reorganized, the unused area becomes a free area.

11.4 SCREENS DISPLAYED BY FUNCTION KEY [MENU OFSET]

Press function key [MENU OFSET] 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. Workpiece origin offset value or workpiece coordinate system shift value
3. Custom macro common variables
4. Software operator's panel

This section also describes following functions.
- Direct input of tool offset value
- Counter input of offset value
- Direct input of workpiece coordinate system shift

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
- Software operator's panel

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 [MENU OFSET] .

2 Press soft key **[OFFSET], [WEAR], [GEOM]**.

```
OFFSET                                   O0001  N0013
   NO.       X          Z           R       T
   01      1.350      5.230      0.0000    0
   02     -2.580     13.540      0.0000    0
   03      5.843    -10.256      0.0000    0
   04     50.245    100.235      0.0000    3
   05     -3.124     36.520      0.3003    0
  _06    -16.369     53.248      0.2504    0
   07    -10.587      0.000      0.0000    0
   08     -0.258    -12.354      0.0000    0
ACTUAL POSITION (RELATIVE)
    U      0.000        W      0.000
    H      0.000
ADRS.
                                 MDI
[OFFSET ][       ][W.SHFT ][ MACRO ][       ]
```

Without tool geometry/wear offset

```
OFFSET / GEOMETRY                  O0001  N0001
  NO.       X          Z           R       T
 G_01     30.500     50.300     0.0000    0
 G 02    -23.580   -100.300     0.0000    0
 G 03    123.850     10.200     0.0000    0
 G 04     55.300   -150.600     0.0000    3
 G 05    -56.800     25.700     0.3003    0
 G 06   -148.300     35.700     0.2504    0
 G 07     45.800    200.500     0.0000    0
 G 08   -159.600      0.400     0.0000    0
ACTUAL POSITION (RELATIVE)
    U      0.000        W      0.000
    H      0.000
ADRS.
                           MDI
[ WEAR  ][ GEOM  ][W.SHFT ][ MACRO ][       ]
```

With tool geometry offset

```
OFFSET / WEAR                      O0001  N0001
   NO.       X          Z           R       T
 W 01      1.350      5.230     0.0000    0
 W 02     -2.580      3.542     0.0000    0
 W 03      0.843     -0.542     0.0000    0
 W 04      0.245      0.235     0.0000    3
 W 05     -3.124      1.520     0.0353    0
 W 06     -6.369      3.248     0.0854    0
 W 07     -0.587      0.000     0.0000    0
 W 08     -0.258     -0.354     0.0000    0
ACTUAL POSITION (RELATIVE)
    U      0.000        W      0.000
    H      0.000
ADRS.
                           MDI
[ WEAR  ][ GEOM  ][W.SHFT ][ MACRO ][       ]
```

With tool wear offset

3 Move the cursor to the compensation value to be set or changed using page keys and cursor keys, or press NO. key and enter the compensation number for the compensation value to be set or changed and press INPUT key.

4 To set a compensation value, enter a value and press soft INPUT key.
TIP is the number of the virtual tool tip (see Programming).
Number of the virtural tool 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 8.
Tool length compensation values can be set using the following functions described in subsequent subsections: direct input of tool offset value, and counter input of offset value.

- **Tool offset memory**

32 groups are provided for tool compensation. 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 can be prohibited cannot be input because of the settings in bits 0 and 1 of parameter 0078.

- **Displaying radius and number of the virtural tool tip**

Specify the radius using address R and the number of the virtual tool tip using address T.

- **Changing offset values during automatic operation**

When offset values have been changed during automatic operation, bit 2 and of parameter 0013 and bit 4 of parameter 0014 can be used for specifying whether new offset values become valid in the next move command or in the next T code command.

0013 #2	0014 #4	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

- **Incremental input**

To enable the setting of compensation values, press an incremental command address key (such as U) to set incremental input mode. In this mode, the previous compensation value and the entered value are summed to give the new compensation value.

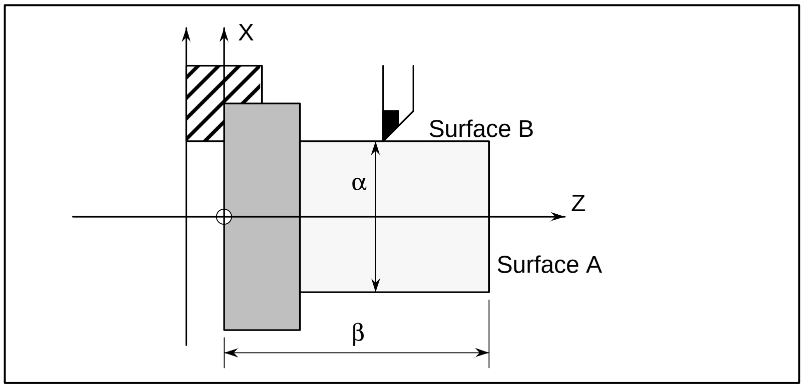
11.4.2 Direct Input of Tool Offset Value

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                                O0001 N0013
   NO.       X          Z          R  T
   01      1.350       5.230      0.0000
   02     -2.580      13.540      0.0000
   03      5.843     -10.256      0.0000
   04     50.245     100.235      0.0000
   05     -3.124      36.520      0.3003
  _06    -16.369      53.248      0.2504
   07    -10.587       0.000      0.0000
   08     -0.258     -12.354      0.0000
ACTUAL POSITION (RELATIVE)
    U      0.000         W      0.000
    H      0.000
ADRS.
                           MDI
[OFFSET ][       ][W.SHFT ][ MACRO ][       ]
```

3–1 Press the function key MENU OFFSET or the soft key **[OFFSET], [WEAR],** or **[GEOM]** to display the tool compensation screen.

3–2 Move the cursor to the set offset number using cursor keys.

3–3 Press address M key and press the address key Z to be set.

3–4 Key in the measured value (β).

3–5 Press the INPUT key.

The difference between measured value β and the absolute 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 M X 69.0 INPUT 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 geometry compensation values become the new compensation values.

- **Retracting along two axes**

If a record button is provided on the machine, after the tool retracts along two axes offset setting can be used when bit 4 of parameter 0015 is set and the record signal is used. Refer to the appropriate manual issued by the machine tool builder.

- **Enabling or disabling the direct input of a tool offset value**

To enable the direct input of a tool offset value, set bit 5 of parameter 010 accordingly.

11.4.3 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. 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                              O0001 N0013
   NO.       X          Z          R  T
   01      1.350       5.230     0.0000
   02     -2.580      13.540     0.0000
   03      5.843     -10.256     0.0000
   04     50.245     100.235     0.0000
   05     -3.124      36.520     0.3003
  _06    -16.369      53.248     0.2504
   07    -10.587       0.000     0.0000
   08     -0.258     -12.354     0.0000
ACTUAL POSITION (RELATIVE)
    U      0.000         W      0.000
    H      0.000
ADRS.
                           MDI
[OFFSET ][       ][W.SHFT ][ MACRO ][       ]
```

5 Press address key X (or Z) and the INPUT key.

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.

- **Enabling/disabling the function**

Bit 6 of parameter No. 008 is used to enable/disable this function.

11.4.4 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

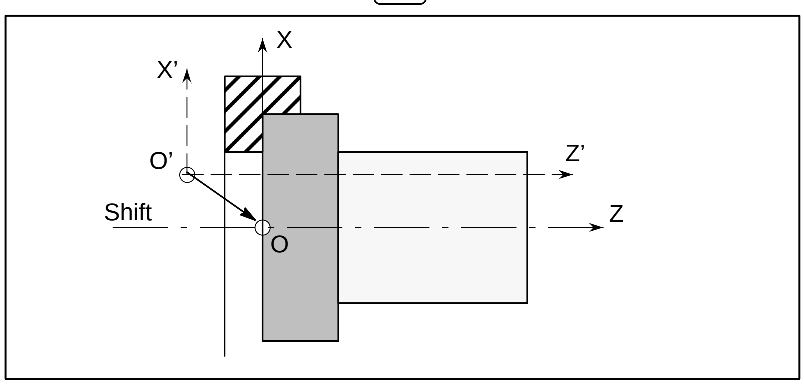
1 Press function key MENU OFSET .

```
WORK SHIFT                          O0001 N00013

   (SHIFT VALUE)        (MEASUREMENT)
    X     -2.605          X     63.587
    Z    -79.582          Z    120.946
    C      0.000          C      0.000




ACTUAL POSITION (RELATIVE)
    U     66.192        W    119.448
    H      0.000
ADRS.
                           JOG
[ WEAR  ][ GEOM  ][ WORK  ][ MACRO ][       ]
```

2 Press soft key **[WK.SHFT]**.

3 Move the cursor using cursor keys to the axis along which the coordinate system is to be shifted.

4 Enter the shift value and press INPUT key.



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**

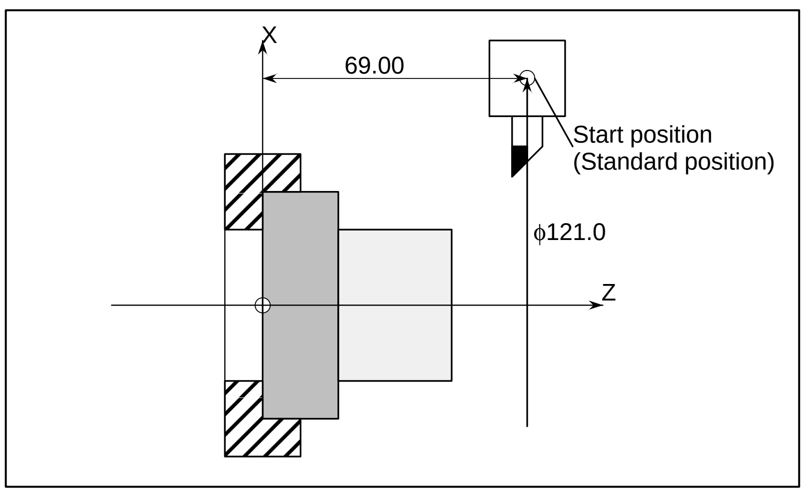
If the automatic coordinate system setting is performed by manual reference position return after shift amount setting, the coordinate system is shifted instantly.

- **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

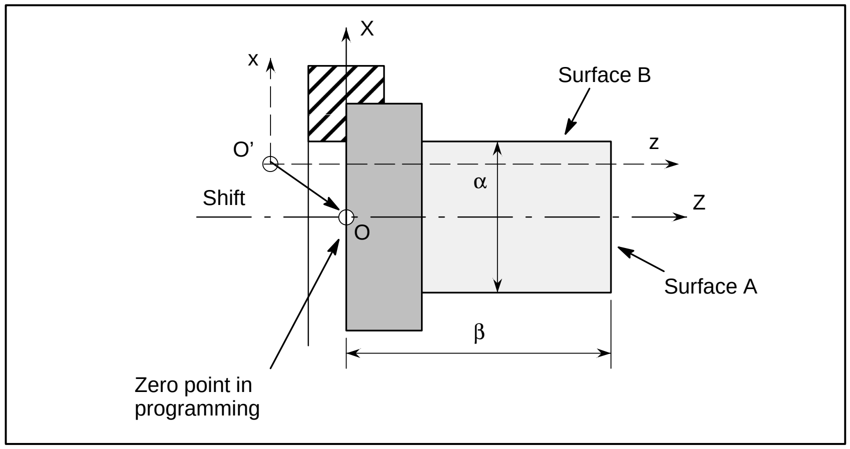


- **Incremental input**

Press an incremental command address key, such as the address key for the axis for which the shift is to be set, to set incremental input mode. In this mode, the previous shift and the entered value are summed to give the new shift.

11.4.5 Direct Measured Value Input for Work Coordinate System Shift

When the work coordinate system set with a G50 command or the automatic coordinate system setting function is different from the coordinate system used in programming, the coordinate system can be shifted by storing the measured distance directly as follows.



1 Cut the workpiece along surface A using the standard tool in manual operation.

2 Release the tool only in X axis direction without moving Z axis and stop the spindle.

3 Measure distance"β", distance between zero point to surface A, in the figure above. And store this value in the work coordinate system shift memory.

3–1 Push function key [MENU OFSET], and soft key **[W. SHT]** then display the WORK SHIFT screen.

3–2 Push address key [M] to input measured distance.

3–3 Push address key [Z] .

3–4 Input measured distance (β)

```
WORK SHIFT                          O0001 N00013

   (SHIFT VALUE)       (MEASUREMENT)
    X     –2.605         X     63.587
    Z    –79.582         Z    120.946
    C      0.000         C      0.000




ACTUAL POSITION (RELATIVE)
    U     66.192        W    119.448
    H      0.000
ADRS.
                           JOG
[ WEAR  ][ GEOM  ][ WORK  ][ MACRO ][       ]
```

3–5 Push INPUT button.

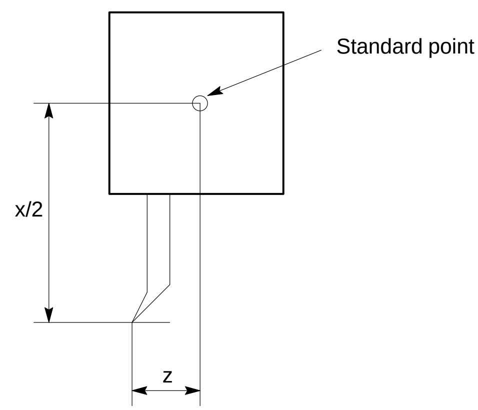
4 Cut surface B in manual mode.

5 Release the tool in Z axis direction without moving X axis and stop the spindle.

6 Measure the diameter a at surface B. And input this distance as X value in the work coordinate system memory.
The shift amount, 0' to 0, is automatically set to the work coordinate system memory. The work coordinate system is shifted immediately and coincides with the coordinate system expected at programming.
If the offset amount of the standard tool is zero, it is said that a work coordinate system is set, where the coordinate values of the tool tip is X=0.0 and Z=0.0 when the tip of the standard tool is positioned at the zero point of the coordinate system.

NOTE

when the tool offset amount (x, z) has already been set on the standard tool as in the figure above and the tool offset function is effective and the direct measured value input for the work coordinate system shift is performed, a work coordinate system is set in which the coordinate values of the standard point are X=0.0 and Z=0.0 when the standard point is positioned at the zero point in the work coordinate system.



11.4.6 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 MENU OFSET.

2 Press chapter selection soft key **[W.SHIFT]** and **[WORK]**.
The workpiece coordinate system setting screen is displayed.

```
WORK COORDINATES                O0001 N0013

   NO.      (SHIFT)     NO.      (G55)
   00   X     -2.605    02   X   -153.480
        Z    -79.582         Z   -304.600
        C      0.000         C      0.000

   NO.      (G54)       NO.      (G56)
   01   X   -125.300  __03   X   -211.350
        Z   -251.200         Z   -295.470
        C      0.000         C      0.000


ADRS.
                                  MDI
[ WEAR  ][ GEOM  ][W.SHFT ][ MACRO ][       ]
```

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 ↑ or page down ↓ key.

Press NO. key and enter the workpiece coordinate system number (0: external workpiece origin offset, 1 to 6: workpiece coordinate systems G54 to G59) and press INPUT key.

4 Turn off the data protection key to enable writing.

5 Move the cursor to the workpiece origin offset to be changed.

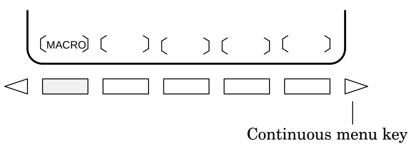
6 Press address key to set and enter a desired value by pressing numeric keys, then press INPUT key. The entered value is specified in the workpiece origin offset value.

7 Repeat **5** and **6** to change other offset values.

11.4.7 Displaying and Setting Custom Macro Common Variables

Displays common variables 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



1 Press function key MENU OFSET.

2 Soft key **[MACRO]**.The following screen is displayed:

```
VARIABLE                           O0001 N0013
  NO.     DATA          NO.       DATA
  100  01000000         108       0000
  101      0000         109   40000000
  102 -50000000         110   00150846
  103      0000         111   00001000
  104  12385010         112   02000000
  105      0000         113   00071232
  106      0000        _114   00024864
  107      0000         115       0000
ACTUAL POSITION (RELATIVE)
    U    71.232           W     24.864
    H     0.000
NO.  144=
                            MDI
[ WEAR  ][ GEOM  ][W.SHFT ][ MACRO ][       ]
```

3 Move the cursor to the variable number to set using either of the following methods:

– Press NO. key and enter the variable number and press INPUT key.

– Move the cursor to the variable number to set by pressing page keys ↑ and/or ↓ and cursor keys ↑ and/or ↓.

4 Enter data with numeric keys and press INPUT key.

5 To set a relative coordinate in a variable, press address key X or Z with pressing EOB key, then press INPUT key.

11.5 SCREENS DISPLAYED BY FUNCTION KEY DGNOS PARAM

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 8).

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. The setting data for operating the machine can also be displayed and set.

See Chapter 7 for the diagnostic screens displayed by pressing function key DGNOS PARAM .

11.5.1 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, and enable/disable the automatic insertion of sequence numbers in program editing.
See Chapter 9 for automatic insertion of sequence numbers.
This subsection describes how to set data.

Procedure for setting the setting data

1 Select the **MDI** mode.

2 Press function key DGNOS PARAM .

3 Press soft key **[W.PRM]**.
This screen consists of several pages.

4 Press NO. key, and press 0 key, INPUT ley.

To switch each screen of setting data, press page key ↑ or ↓.

```
PARAMETER                          O0001  N0001

   (SETTING  1)
    TVON  = 1
    ISO   = 1   (0:EIA 1:ISO )
    INCH  = 0   (0:MM  1:INCH)
    I/O   = 0
    SEQ   = 1





NO. TVON
                           MDI
[ PARAM ][ DGNOS ][       ][SV–PRM ][       ]
```

```
PARAMETER                          O0001  N0001

   (SETTING  2)
   _PWE   = 0  (0:DISABLE 1:ENABLE)
    TAPEF = 0
   (SEQUENCE STOP)
    PRGNO =    0
    SEQNO =    0

PART TOTAL    =      23
PART REQUIRED =       0
PART COUNT    =      23
 RUN TIME      3H36M  CYCLE TOME  0H 0M 0S
NO. PWE
                           MDI
[ PARAM ][ DGNOS ][       ][SV–PRM ][       ]
```

5 Move the cursor to the item to be changed by pressing cursor keys ↑ , ↓ .

6 Enter a new value and press INPUT key.

Contents of settings

- **TV ON**

Setting to perform TV check.
0 : No TV check
1 : Perform TV check

- **ISO**

Setting code when data is output through reader puncher interface.
0 : EIA code output
1 : ISO code output (To output to floppy cassette, be sure to set to 1.)

- **INCH**

Setting a program input unit, inch or metric system
0 : Metric
1 : Inch

- **I/O**

Using channel of reader/puncher interface.
0 : Channel 1
1 : Channel 1

- **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.

11.5.2 Displaying and Setting Run Time, Parts Count

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).

Procedure for Displaying and Setting Run Time, Parts Count

1 Select the MDI mode.

2 Press function key DGNOS PARAM .

3 Press chapter selection soft key **[PARAM]**.

4 Press page key ⬇ to display the following screen.

```
PARAMETER                          O0001  N0001

   (SETTING  2)
   _PWE   = 0  (0:DISABLE 1:ENABLE)
    TAPEF = 0
   (SEQUENCE STOP)
    PRGNO =    0
    SEQNO =    0

PART TOTAL    =      23
PART REQUIRED =       0
PART COUNT    =      23
 RUN TIME      3H36M  CYCLE TOME  0H 0M 0S
NO. PWE
                          MDI
[ PARAM ][ DGNOS ][       ][SV–PRM ][       ]
```

5 To set the number of parts required and parts count, move the cursor to PARTS REQUIRED or PARTS COUNT and enter the number of parts, and press INPUT key.

6 To set the operationg time or cutting time, move the cursor to DATE or TIME, enter a new hour/minute/second, then press INPUT key.

Display items

- **PARTS TOTAL**

This value is incremented by one when M02, M30, or an M code specified by parameter 219 is executed. This value cannot be set on this screen. Set the value in parameter 0779.

- **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. 0600).

- **PARTS COUNT**

This value is incremented by one when M02, M30, or an M code specified by parameter 219 is executed. 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.

- **OPERATING 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.

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. 219) is executed, counting is made in the similar manner. Also, it is possible to disable counting even if M02 or M30 is executed (parameter (No. 0040#3) is set to 1). For details, see the manual issued by machine tool builders.

11.5.3 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 Press function key DGNOS PARAM .

2 Press chapter selection soft key **[PARAM]** to display the parameter screen.

```
PARAMETER                              O0001  N0013
   NO.       DATA          NO.       DATA
  0001     00000000      0011     00000000
  0002     00000001      0012     00000000
  0003     00000000      0013     00000000
  0004     01110111      0014     00000100
  0005     01110111      0015     00000000
  0006     01110111      0016     00000000
  0007     01110111      0017     11111111
  0008     00000000      0018     00000000
  0009     00000000      0019     10000000
  0010     11100000      0020     00000000

NO. 0001 =
                                   MDI
[ PARAM ][ DGNOS ][       ][SV–PRM ][       ]
```

3 Move the cursor to the parameter number to be set or displayed in either of the following ways:

· Press NO. key and enter the parameter number and press INPUT key.

· Move the cursor to the parameter number using the page keys, ⬆ and ⬇ , and cursor keys, ⬆ , ⬇ .

4 To set the parameter set 1 for **PWE (PARAMETER WRITE)** to enable writing. See the procedure for setting data writing described below.

5 Enter a new value with numeric keys and press INPUT key in the MDI mode. The parameter is set to the entered value and the value is displayed.

6 Set 0 for **PWE (PARAMETER WRITE)** to disable writing.

7 To release the alarm, press the RESET key.

Explanations

- **Setting parameters with external input/output devices**

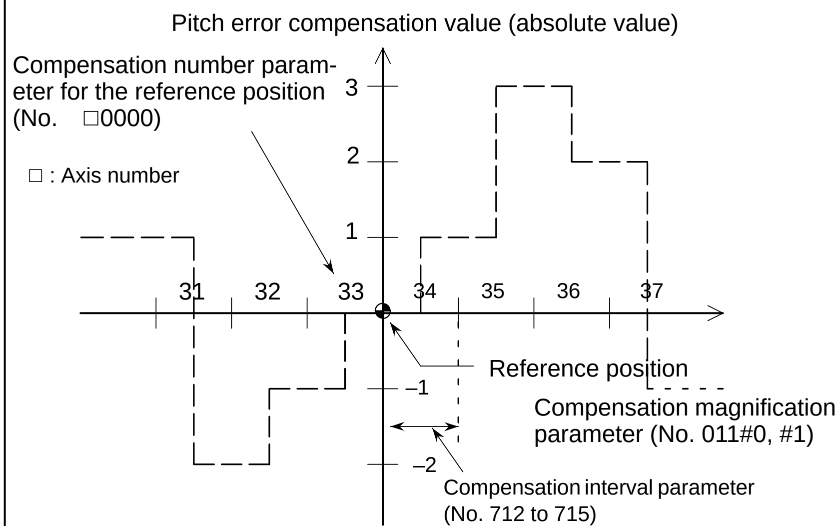
See Chapter 8 for setting parameters with external input/output devices such as the Handy File.

- **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.

11.5.4 Displaying and Setting Pitch Error Compensation Data

If pitch error compensation data is specified, pitch errors of each axis can be compensated 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. 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 8). 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 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
Parameter	□032	□033	□034	□035	□036	□037	□038
Compensation value to be set	+3	–1	–1	+1	+2	–1	–3

· Number of the pitch error compensation point at the reference position (□ : axis number) : Parameter No. □000
· Pitch error compensation magnification : Parameter No. 011#0, #1
· Interval of the pitch error compensation points : Parameter No. 0712 to 0713
· Setting compensation value : Parameter No. □001 + compensation points

Explanations

- **Compensation point number**

128 compensation points from No. 0 to 127 are available for each axis. Specify the compensation number for the reference position of each axis in the corresponding parameter (Parameter n000, n: axis number).

- **Compensation value**

Specify the compensation value in the corresponding parameter (Parameter n001 + compensation point number, n: axis number).

Restrictions

Compensation value range

Compensation values can be set within the range from –7 x compensation magnification (output unit) to +7 x compensation magnification (detection unit). The compensation magnification can be set 1, 2, 4, 8 in parameter No. 011#0, #1. The units of the compensation value can be changed to the detection units if bit 7 of parameter 035 is specified accordingly.

Intervals of compensation points

The pitch error compensation points are arranged with equally spaced. Set the space between two adjacent positions for each axis to the parameter (No. 712 to 713).
Valid data range is 0 to 99999999.
The minimum interval between pitch error compensation points is limited and obtained from the following equation:
Minimum interval of pitch error compensation points = maximum feedrate (rapid traverse rate) / 7500
Unit: mm, inches, deg, and mm/min, inches/min, deg/min

Pitch error compensation of the rotary axis

For the rotating axis, the interval between the pitch error compensation points shall be set to one per integer of the amount of movement (normally 360°) per rotation. The sum of all pitch error compensation amounts per rotation must be made to 0.

Conditions where pitch error compensation is not performed

Note that the pitch error is not compensated in the following cases:

- When the machine is not returned to the reference position after turning on the power. If an absolute–position detector is provided and if the reference position has been determined, pitch error compensation is carried out.
- If the interval between the pitch error compensation points is 0.

Examples

- **For linear axis (X axis)**

· Machine stroke: –400 mm to +800 mm
· Interval between the pitch error compensation points: 50 mm
· No. of the compensation point of the reference position: 40

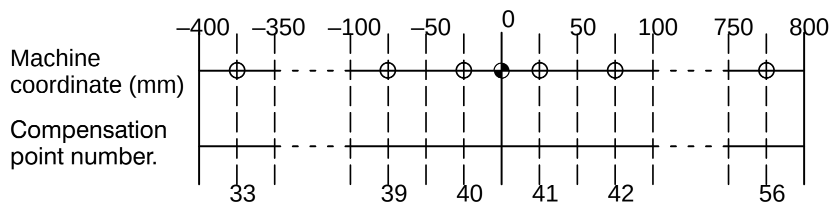
If the above is specified, the No. of the farthest compensation point in the negative direction is as follows:

No. of the compensation point of the reference position – (Machine stroke on the negative side/Interval between the compensation points) + 1
= 40 – 400/50 + 1= 33

No. of the farthest compensation point in the positive direction is as follows:

No. of the compensation point of the reference position + (Machine stroke on the positive side/Interval between the compensation points)
= 40 + 800/50= 56

The correspondence between the machine coordinate and the compensation point No. is as follows:



Compensation values are output at the positions indicated by ○ .

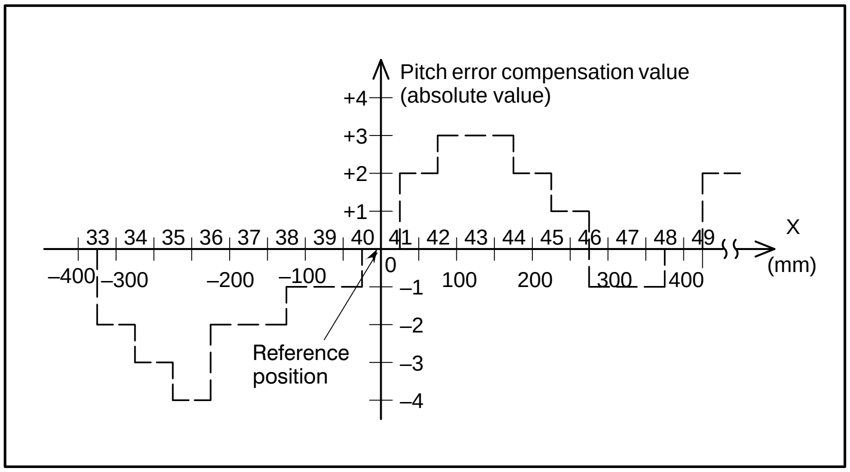
Therefore, set the parameters as follows:

Parameter	Setting value
1000 : Compensation number for the reference position	40
011#0, #1 : Compensation magnification	#0=0, #1=0
0712 : Interval between pitch error compensation points	50000

The compensation amount is output at the compensation point No. corresponding to each section between the coordinates.
The following is an example of the compensation amounts.

Compensation position number	33	34	35	36	37	38	39	40	41
Parameter	1034	1035	1036	1037	1038	1039	1040	1041	1042
Compensation value	+2	+1	+1	–2	0	–1	0	–1	+2

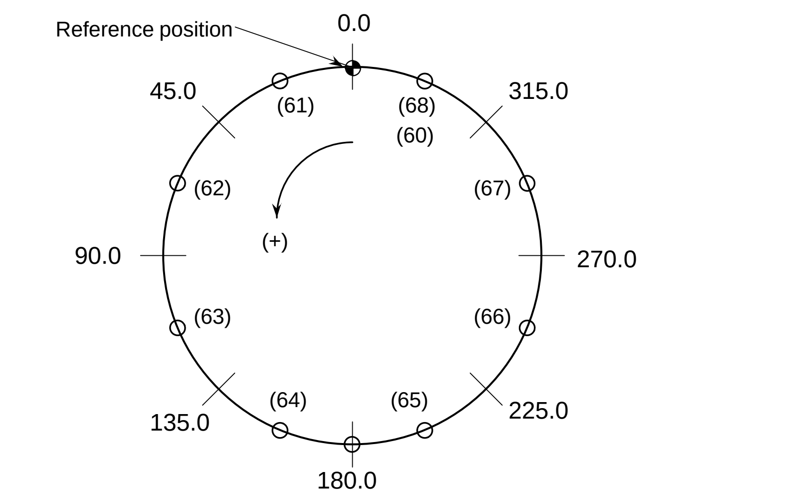
Compensation position number	42	43	44	45	46	47	48	49	56
Parameter	1043	1044	1045	1046	1047	1048	1049	1050	1057
Compensation value	+1	0	–1	–1	–2	0	+1	+2	1



- **For rotary axis (C axis)**

·Amount of movement per rotation: 360°
· Interval between pitch error compensation points: 45°
· No. of the compensation point of the reference position: 60
If the above is specified, the No. of the farthest compensation point in the negative direction for the rotating axis is always equal to the compensation point No. of the reference position.
The No. of the farthest compensation point in the positive direction is as follows:
No. of the compensation point of the reference position + (Move amount per rotation/Interval between the compensation points)
= 60 + 360/45= 68

The correspondence between the machine coordinate and the compensation point No. is as follows:



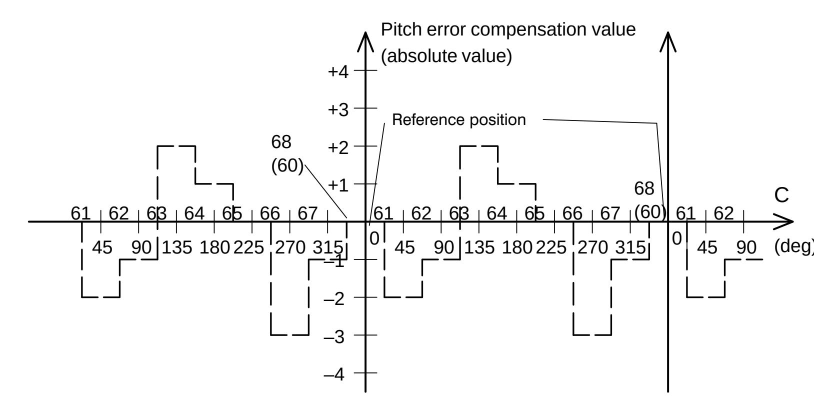
Compensation values are output at the positions indicated by ○ .

If the sum of the compensation values for positions 61 to 68 is not 0, pitch error compensation values are accumulated for each rotation, causing positional deviation.
The same value must be set for compensation points 60 and 68.
Therefore, set the parameters as follows:

Parameter	Setting value
4000 : Compensation number for the reference position	60
011#0, #1 : Compensation magnification	#0=0, #1=0
715 : Interval between pitch error compensation points	45000

The following is an example of compensation amounts.

Compensation position number	60	61	62	63	64	65	66	67	68
Parameter	4061	4062	4063	4064	4065	4066	4067	4068	4069
Compensation value	+1	–2	+1	+3	–1	–1	–3	+2	+1



11.6 SCREENS DISPLAYED BY FUNCTION KEY OPR ALARM

The alarm message and operator message can be displayed by pressing the OPR ALARM key. The software operator's panel can also be displayed and specified. For details of how to display the alarm message, see Chapter 7.

11.6.1 Displaying Operator Message

The operator message function displays a message on the PMC screen.

Procedure for Displaying Operator Message

1 Press function key OPR ALARM

2 press soft key **[MSG]**.

```
OPERATOR MESSAG                   O0000 N0000

   NO. 2001
COOLANT ON










                                S  0  T
                              MDI
[ ALARM ][  OPR  ][  MSG  ][      ][      ]
```

Explanations

- If the operator message display function is enabled, the screen is automatically switched to the operator message screen.
- A PMC command can also be used to clear the operator message.
- For details of the contents of the operator message, and how to clear the message, refer to the manual provided by the machine tool builder.

11.6.2 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

1 Press function key OFFSET SETTING .

2 Press chapter selection soft key **[OPR]**.

3 The screen consists of several pages.
Press page key ↑ or ↓ until the desired screen is displayed.

```
OPERATOR'S PANEL                 O0001 N0001

_MODE :  MDI   ■AUTO  EDIT  HNDL  JOG  ZRN
 HANDLE AXIS   :  ■HX  HZ  H3
 HANDLE MULT.  :  ■*1   *10  *100
 RAPID OVRD.   :  ■100%  50%   25%   F0
 JOG FEED      :   500 MM/MIN
          ████████████████████████████ * *
 FEED OVERD :  90%
          ████████████████████* * * * * *
ACTUAL POSITION (ABSOLUTE)
    X   164.000         Z    182.000
    C     0.000

                             MDI
[ ALARM ][ OPR   ][  MSG  ][ MACRO ][       ]
```

```
OPERATOR'S PANEL                 O0001 N0001

_BLOCK SKIP    :    ■OFF       ON
 SINGLE BLOCK  :     OFF      ■ON
 MACHINE LOCK  :     OFF      ■ON
 DRY RUN       :     OFF      ■ON
 PROTECT KEY   :     PROTECT    RELEASE
 FEED HOLD     :    ■OFF       ON


ACTUAL POSITION (ABSOLUTE)
    X   164.000         Z    182.000
    C     0.000

                             MDI
[ ALARM ][ OPR   ][  MSG  ][       ][       ]
```

4 Move the cursor to the desired switch by pressing cursor key ↑ or ↓ .

5 Push the numerical 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 jog feed. Press the [5] key together with an arrow key to perform jog rapid traverse.

[7↖]	[8↑]	[9↗]
[4←]	[5]	[6→]
[1↙]	[2↓]	[6↘]

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 0017.

Group1 : Mode selection
Group2 : Selection of jog feed axis, jog rapid traverse
Group3 : Selection of manual pulse generator feed axis, selection of manual pulse magnification x1, x10, x100
Group4 : Jog 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 0017 are not displayed on the software operator's panel.

- **Screens on which jog feed is valid**

When the CRT indicates other than the software operator's panel screen and diagnostic screen, jog feed is not conducted even if the arrow key is pushed.

- **Jog feed and arrow keys**

The feed axis and direction corresponding to the arrow keys can be set with parameters (Nos. 0130 to 0137).

- **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.

11.7 DISPLAYING THE PROGRAM NUMBER, SEQUENCE NUMBER, AND STATUS, AND WARNING MESSAGES FOR DATA SETTING

The program number, sequence number, and current CNC status are always displayed on the screen except when the power is turned on or a system alarm occurs.
This section describes the display of the program number, sequence number, and status.

11.7.1 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                       O0001 N0000
O0001 T0101 ;
S550 M08 ;
M45 ;
N010 G50 X200. Z200. ;
N011 G00 X160. Z180. ;
N012 G71 U7. R1. ;
N013 G71 P014 Q020 U4. W2. F0.3 S550 M03
 ;
N014 G00 X40. F0.15 S700 ;



<
                        EDIT
[ PRGRM ][  LIB  ][  I/O  ][       ][       ]
```

The program number and sequence number displayed depend on the screen and are given below:

On the program screen in the EDIT mode or 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.

11.7.2 Displaying the Status and Warning for Data Setting

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.

Explanations

- **Description of each display**

```
NOT READY
            ALARM BAT BUF EDIT          INPUT
            (Display soft keys)
```

1. Current mode

MDI : Manual data input
AUTO : Automatic operation
RMT : Automatic operation (Tape operation, or such like)
EDIT : Memory editing
HNDL : Manual handle feed
JOG : Jog feed
TJOG : (TEACH IN JOG)
THND : (TEACH IN HANDLE)
STEP : Manual incremental feed
ZRN : Manual reference position return

2. Alarm status

ALARM : Indicates that an alarm is issued.
BAT : Indicates that the battery is low.

3. Other status display

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.)
COMPARE : Indicates that the program is being collated.
LSK : Indicates that labels are skipped when data is input.
BUF : Indicates that the block to be executed next is being read.
NOT READY : Indicates that the system is in the emergency stop state.

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

When the message ”BAT” appears at the bottom of the screen, replace the backup batteries for the CNC memory according to the procedure described below.

Procedure for replacing CNC battery for memory back–up

1 Have three commercially available fresh alkaline R20 (D) batteries handy.

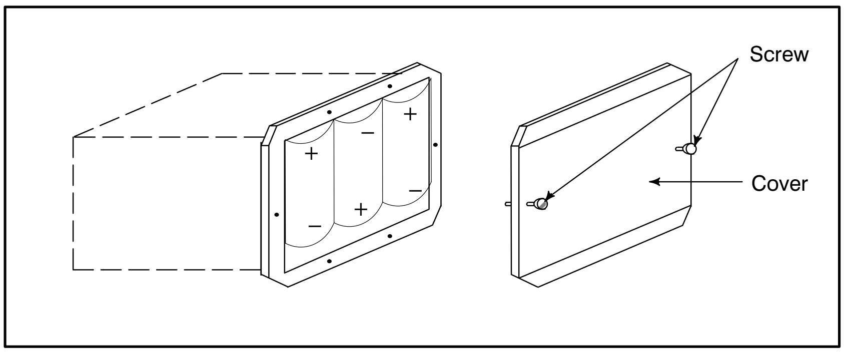
2 Switch on the machine (CNC). (When replacing the batteries, keep the CNC switched on. If you replace the batteries with the CNC switched off, the memory contents will be lost.)

3 Loosen the screws for the battery case cover and remove it. For the location of the battery case, refer to the relevant operating manual issued by the machine builder.

4 Replace the batteries in the battery case with the fresh ones you have handy. Be careful for the orientation of the batteries. Of the three batteries, the middle one is in the orientation opposite to the other two.

5 After replacing the batteries, put the battery case cover back in place.

6 Press the RESET key and ensure the message ”BAT” disappears.



1.2 REPLACING BATTERIES FOR ABSOLUTE PULSE CODER

If absolute pulse coder alarm 3n7 (where n is an axis number) occurs, replace the batteries (alkaline) for the absolute pulse coder according to the procedure described below.

Procedure for replacing batteries for absolute pulse coder

Procedure

1 Have three commercially available fresh alkaline R20 (D) batteries handy.

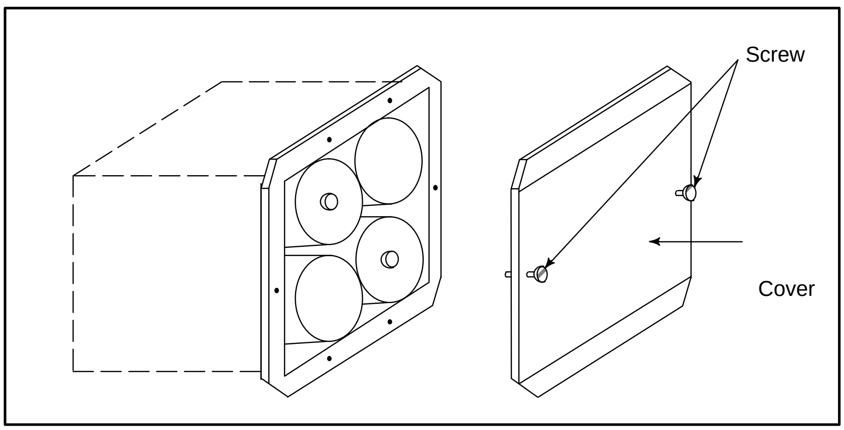
2 Switch on the machine (CNC). (When replacing the batteries, keep the CNC switched on. If you replace the batteries with the CNC switched off, the memory contents will be lost.)

3 Loosen the screws for the battery case cover and remove it. For the location of the battery case, refer to the relevant operating manual issued by the machine builder.

4 Replace the batteries in the battery case with the fresh ones you have handy. Be careful for the orientation of the batteries. (Place the two batteries in the orientation opposite to the other two as shown below.)

5 After replacing the batteries, put the battery case cover back in place.

6 Switch the machine (CNC) off and on again, and ensure that alarm 3n7 does not occur.



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	
DEL	○	○	○	○	○	∘	○	○	○	Del		○	○	○	○	∘	○	○	○	×
NUL						∘				Blank						∘				×
BS	○				○	∘				BS			○		○	∘		○		×
HT					○	∘			○	Tab			○	○	○	∘	○	○		×
LF or NL					○	∘		○		CR or EOB	○					∘				
CR	○				○	∘	○		○	___										×
SP	○		○			∘				SP				○		∘				□
%	○		○			∘	○		○	ER					○	∘		○	○	
(			○		○	∘				(2–4–5)				○	○	∘		○		
)	○		○		○	∘			○	(2–4–7)		○			○	∘		○		
+			○		○	∘		○	○	+		○	○	○		∘				×
–			○		○	∘	○		○	–		○				∘				
:			○	○	○	∘		○		___										
/	○		○		○	∘	○	○	○	/			○	○		∘			○	
.			○		○	∘	○	○		.		○	○		○	∘		○	○	
#	○		○			∘		○	○	Parameter (No.0044)										
$			○			∘	○			___										Δ
&	○		○			∘	○	○		&					○	∘	○	○		Δ
▾			○			∘	○	○	○	___										Δ
*	○		○		○	∘		○		Parameter (No.0042)										Δ
,	○		○		○	∘	○			,			○	○	○	∘		○	○	
;	○		○	○	○	∘		○	○	___										Δ
<			○	○	○	∘	○			___										Δ
=	○		○	○	○	∘	○		○	Parameter (No.0043)										Δ
>	○		○	○	○	∘	○	○		___										Δ
?			○	○	○	∘	○	○	○	___										Δ
@	○	○				∘				___										Δ
”			○					○		___										Δ
[	○	○		○	○	∘		○	○	Parameter (No.0053)										Δ
]	○	○		○	○	∘	○		○	Parameter (No.0054)										Δ

NOTE

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 and will be ignored.

∆ : The character will be registered in memory, 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.

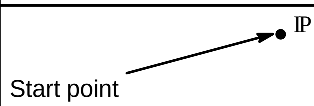
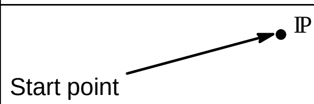
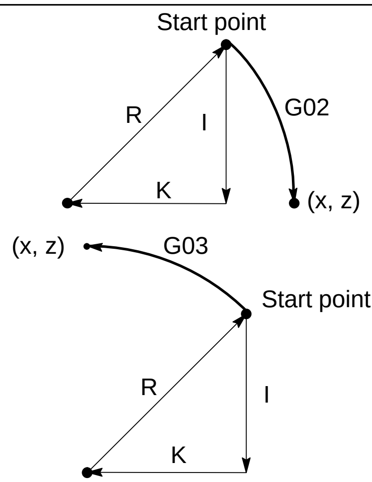
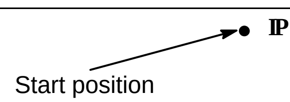
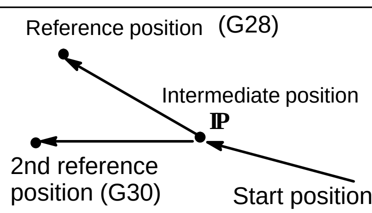
2 Codes not in this table are ignored if their parity is correct.

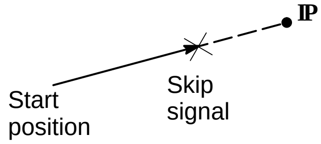
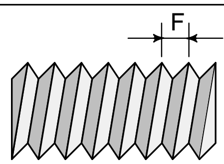
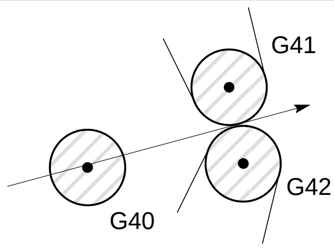
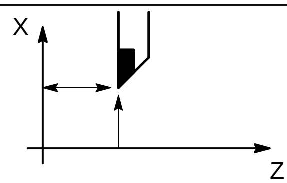
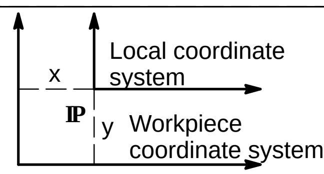
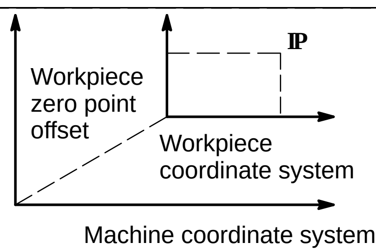
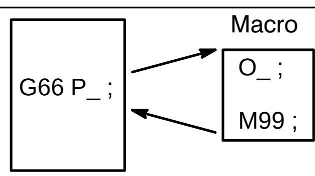
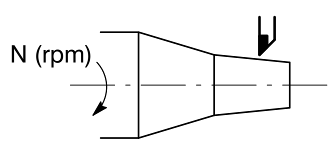
3 Codes with incorrect parity cause the TH alarm. But they are ignored without generating the TH alarm when they are in the comment section.

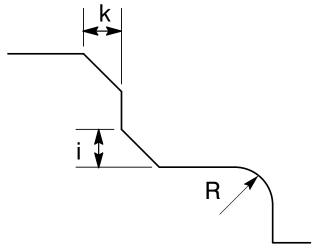
4 A character with all eight holes punched is ignored and does not generate TH alarm in EIA code.

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)

Functions	Illustration	Tape format
Positioning (G00)		G00 IP_ ;
Linear interpolation (G01)		G01 IP_ F_;
Circular interpolation (G02, G03)		$\left\{\begin{matrix}\text{G02}\\\text{G03}\end{matrix}\right\}$ X_ Z_ $\left\{\begin{matrix}\text{R\_}\\\text{I\_ K\_}\end{matrix}\right\}$ F_ ;
Dwell (G04)		G04 $\left\{\begin{matrix}\text{X\_}\\\text{P\_}\end{matrix}\right\}$;
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
Reference position return check (G27)		G27IP _ ;
Reference position return (G28) 2nd, reference position return (G30)		G28 IP _ ; G30 IP_ ;

Functions	Illustration	Tape format
Skip fubction (G31)		G31 IP _ F_;
Thread cutting (G32)		Equal lead thread cutting G32 IP_ F_;
Cutter compensation (G40, G41, G42)		G41 G42 IP _ ; G40 : Cancel
Coordinate system setting Spindle speed setting (G50)		G50 IP _ ; Coordinate system setting G50 S_ ; Spindle speed setting
Local coordinate system setting (G52)		G52 IP_ ;
Machine coordinate system selection (G53)		G53 IP_ ;
Workpiece coordinate system selection (G54 to G59)		G54 : G59 IP_ ;
Custom macro (G65, G66, G67)		Macro command G65 H_ P_ Q_ R_ ; Modal call G66 P_<argument> G67 ; cancel
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

Functions	Illustration	Tape format
Chamfering, Corner R		$X\_\ ;\left\{\begin{array}{l}C(K)\pm k\\ R\_\end{array}\right\}P\_\ ;$ $Z\_\ ;\left\{\begin{array}{l}C(I)\pm i\\ R\_\end{array}\right\}P\_\ ;$
Canned cycle (G71 to G76) (G90, G92, G94)	Refer to II.13. 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_ ;
tool offset		P_ T_

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		0.001 mm	0.0001 mm
Max. programmable dimension		±99999.999 mm	±9999.9999 mm
Max. rapid traverse **Notes**		100000 mm/min	24000 mm/min
Feedrate range **Notes**	Per min	1 to 100000 mm/min	1 to 12000 mm/min
	Per rev	0.0001 to 500.0000 mm/rev	0.0001 to 500.0000 mm/rev
Incremental feed		0.001, 0.01, 0.1, 1 mm/step	0.0001, 0.001, 0.01, 0.1 mm/step
Tool compensation		0 to ±999.999 mm	0 to ±999.9999 mm
Dwell time		0 to 99999.999 sec	0 to 9999.9999 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		0.001 mm	0.0001 mm
Max. programmable dimension		±9999.9999 inch	±393.70078 inch
Max. rapid traverse **Notes**		100000 mm/min	24000 mm/min
Feedrate range **Notes**	Per min	0.01 to 4000 inch/min	0.01 to 12000 inch/min
	Per rev	0.000001 to 9.999999 inch/rev	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 ±99.99999 inch
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		0.0001 inch	0.00001 inch
Max. programmable dimension		±9999.9999 inch	±999.99999 inch
Max. rapid traverse **Notes**		4000 inch/min	960 inch/min
Feedrate range **Notes**	Per min	0.01 to 4000 inch/min	0.01 to 480 inch/min
	Per rev	0.000001 to 9.999999 inch/rev	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 ±99.99999 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		0.0001 inch	0.00001 inch
Max. programmable dimension		±99999.999 mm	±9999.9999 mm
Max. rapid traverse **Notes**		4000 inch/min	960 inch/min
Feedrate range **Notes**	Per min	1 to 100000 mm/min	1 to 12000 mm/min
	Per rev	0.0001 to 500.0000 mm/rev	0.0001 to 500.0000 mm/rev
Incremental feed		0.001, 0.01, 0.1, 1 mm/step	0.0001, 0.001, 0.01, 0.1 mm/step
Tool compensation		0 to ±999.999 mm	0 to ±999.9999 mm
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 **Note**	240000 deg/min	100000 deg/min
Feedrate range (metric input) **Note**	1 to 100000 deg/min	1 to 24000 deg/min
Feedrate range (inch input) **Note**	1 to 6000 deg/min	1 to 600 deg/min
Incremental feed	0.001, 0.01, 0.1, 1 deg/step	0.0001, 0.001, 0.01, 0.1 deg/step

NOTE

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.

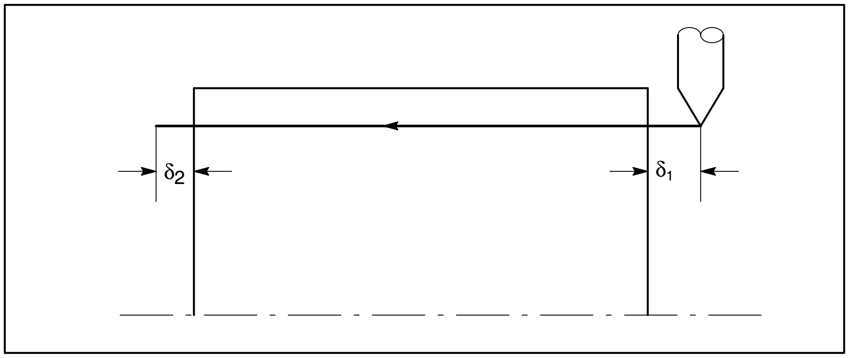


Fig. D.1 (a) Incorrect thread position

Explanations

- **How to determine δ_2**

$$\delta_2 = T_1 V \text{ (mm)} \quad \ldots\ldots\ (1)$$

$$V = \frac{1}{60} RL$$

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**

$$\delta_1 = \{t - T_1 + T_1 \exp(-\frac{t}{T_1})\} V \quad \ldots\ldots\ (2)$$

$$a = \exp(-\frac{t}{T_1}) \quad \ldots\ldots\ (3)$$

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 ”a” 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, *a*, will be obtained at $\boxed{1}$, and depending on the time constant of cutting feed acceleration/ deceleration, the δ_1 value when V = 10mm / s will be obtained at $\boxed{2}$. Then, depending on the speed of thread cutting, δ_1 for speed other than 10mm/ s can be obtained at $\boxed{3}$.

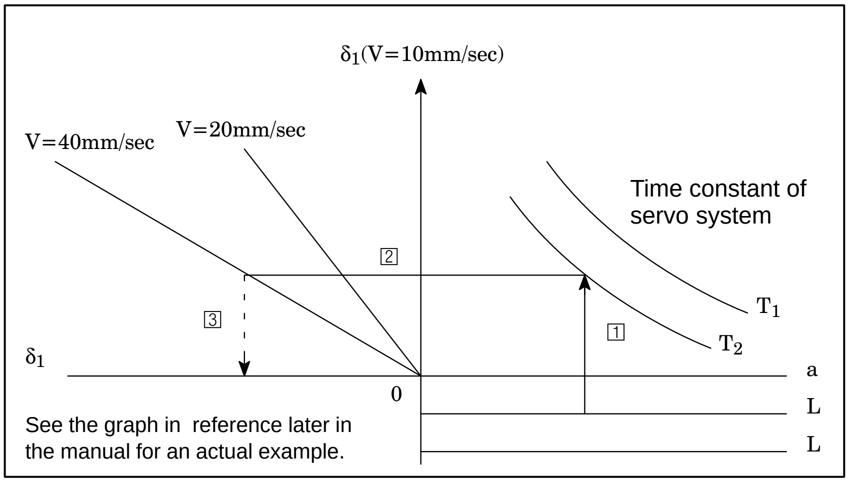


Fig. D.1 (b) Nomograph

> **NOTE**
> The equations for d1, and d2 are for when the acceleration/deceleration time constant for cutting feed is 0.

D.2 SIMPLE CALCULATION OF INCORRECT THREAD LENGTH

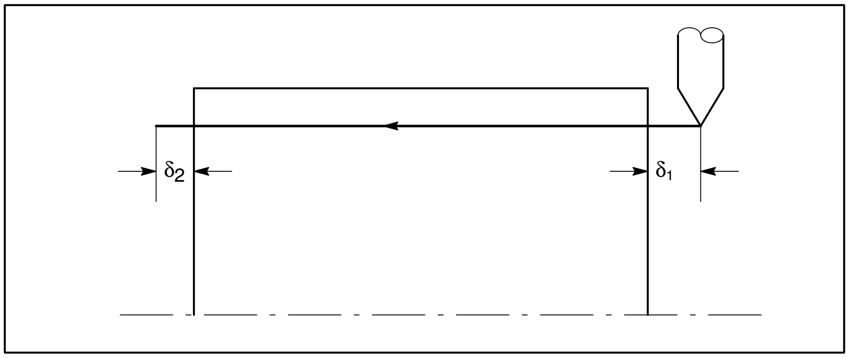


Fig. D.2 (a) Incorrect threaded portion

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**

$$\delta_1 = \frac{LR}{1800*}(-1-lna) \quad \text{(mm)}$$

$$= \delta_2(-1-lna) \quad \text{(mm)}$$

R : Spindle speed (rpm)
L : Thread lead (mm)

* When time constant T of the servo system is 0.033 s.

Following a is a permited 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

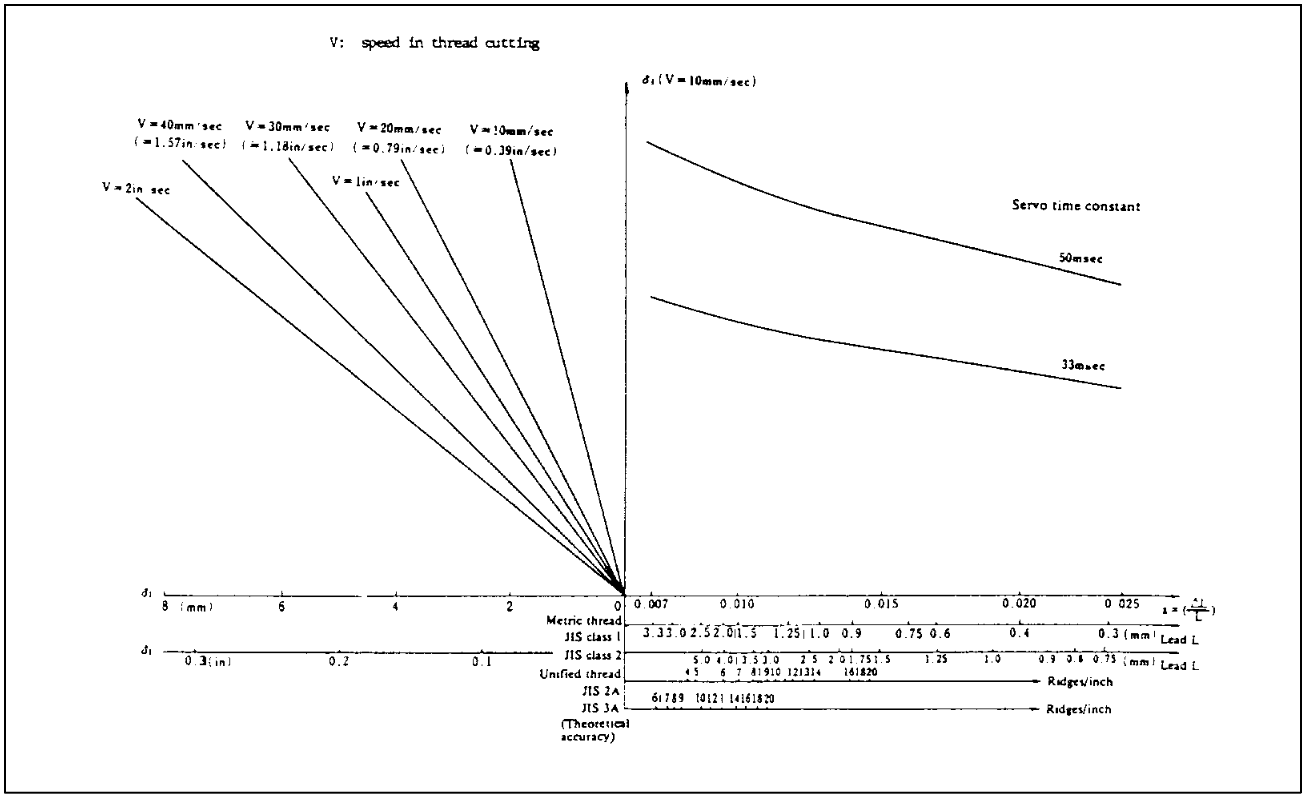
L=1mm

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.

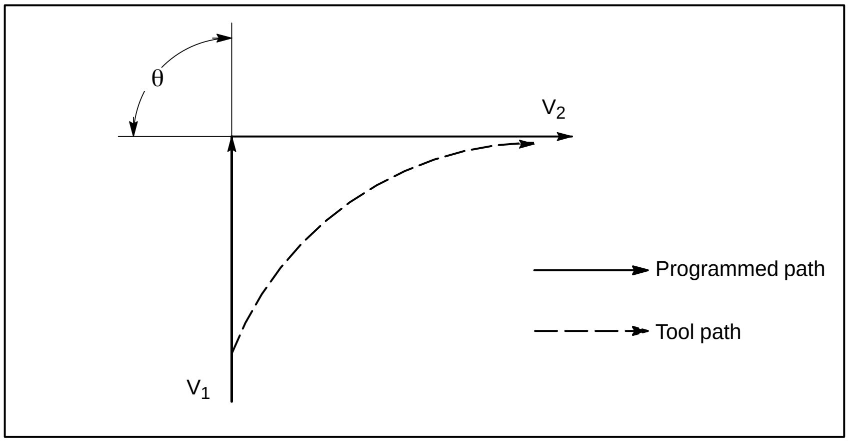


Fig. D.3 (a) Slight deviation between the tool path and the programmed path

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 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.)

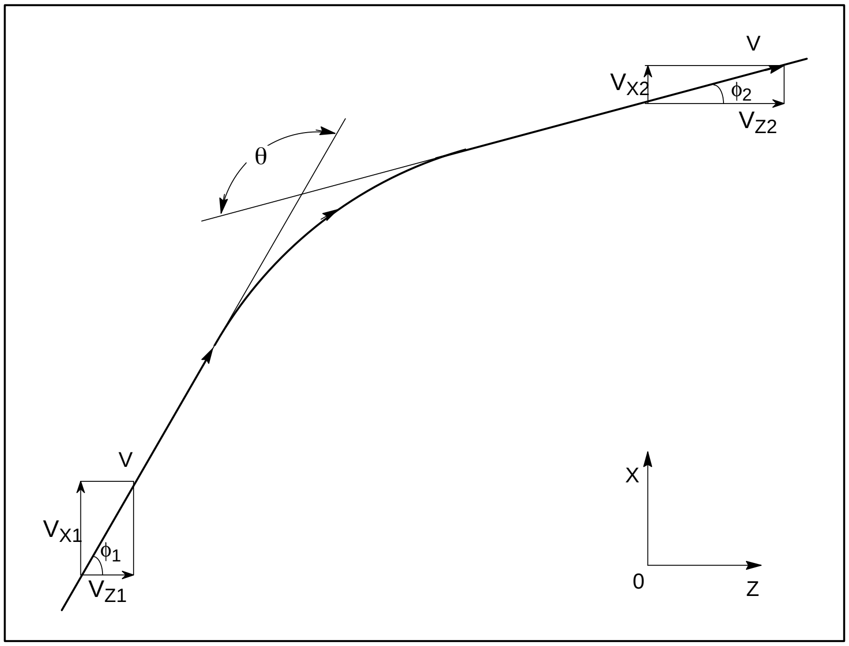


Fig. D.3 (b) Example of tool path

- **Description of conditions and symbols**

$$V_{X1} = V \sin \phi_1$$

$$V_{Z1} = V \cos \phi_1$$

$$V_{X2} = V \sin \phi_2$$

$$V_{Z2} = V \cos \phi_2$$

V : Feedrate at both blocks before and after cornering
V_{X1} : X–axis component of feedrate of preceding block
V_{Z1} : Z–axis component of feedrate of preceding block
V_{X2} : X–axis component of feedrate of following block
V_{Z2} : Z–axis component of feedrate of following block
θ : Corner angle
ϕ_1 : Angle formed by specified path direction of preceding block and Z–axis
ϕ_2 : Angle formed by specified path direction of following block and Z–axis

- **Initial value calculation**

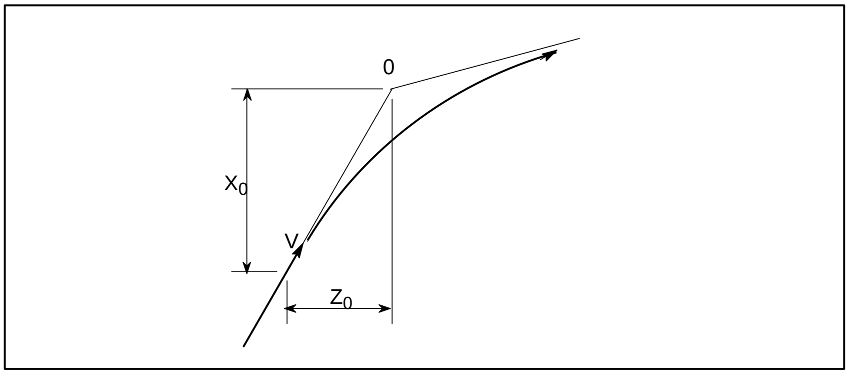


Fig. D.3 (c) Initial value

The initial value when cornering begins, that is, the Z 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.

$$X_0 = V_{X1}(T_1 + T_2)$$

$$Z_0 = V_{Z1}(T_1 + T_2)$$

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 Z–axis direction.

$$V_X(t) = \frac{V_{X1}-V_{X2}}{T_1-T_2}\{T_1\exp(-\frac{t}{T_1})-T_2\exp(-\frac{t}{T_2})\} + V_{X2}$$

$$V_Z(t) = \frac{V_{Z1}-V_{Z2}}{T_1-T_2}\{T_1\exp(-\frac{t}{T_1})-T_2\exp(-\frac{t}{T_2})\} + V_{Z2}$$

Therefore, the coordinates of the tool path at time t are calculated from the following equations:

$$X(t) = \int_0^t V_X(t)dt - X_0$$

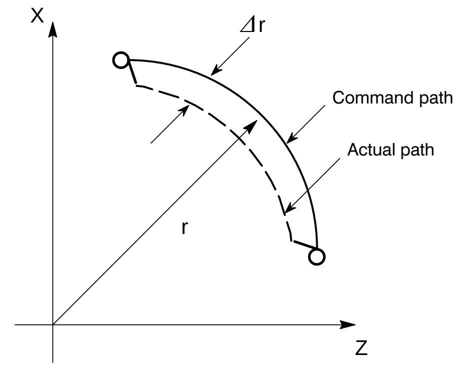
$$= \frac{V_{X2}-V_{X1}}{T_1-T_2}\{T_1^2\exp(-\frac{t}{T_1})-T_2^2\exp(-\frac{t}{T_2})\} - V_{X2}(T_1 + T_2 - t)$$

$$Z(t) = \int_0^t V_Z(t)dt - Z_0$$

$$= \frac{V_{Z2}-V_{Z1}}{T_1-T_2}\{T_1^2\exp(-\frac{t}{T_1})-T_2^2\exp(-\frac{t}{T_2})\} - V_{Z2}(T_1 + T_2 - t)$$

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, sepecially for circular cutting at high speeds.
This error can be obtained as follows:



$$\Delta r = \frac{1}{2}(T_1^{\,2} + T_2^{\,2})\frac{V^2}{r} \qquad \text{.......} \quad (1)$$

- ⊿r : 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)

Since the machining radius r (mm) and allowable error ⊿r (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 045#6 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 compensation	×	Depending on parameter (No.0001#3)	○ : MDI mode Other modes depend on parameter (No.0001#3).
	Tool nose rudius 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)	×	×	×

NOTE

1. When heading is performed, the main program number is displayed as sequence number.
2. When a reset is performed during execution of a subprogram, control returns 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		¥	093	Yen sign
4	052		]	094	Right square bracket
5	053		_	095	Underscore

G ALARM LIST

1) Program errors (P/S alarm)

Number	Meaning	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.
008	PROGRAM HAS AN ERROR AT THE END	The program does not end with M02/M30/M99 and the execution of EOR (%) was attempted instead. Correct 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.
015	TOO MANY AXES COMMANDED	An attempt was made to move the machine along the axes, but the number of the axes exceeded the specified number of axes controlled simultaneously. Alternatively, in a block where where the skip function activated by the torque–limit reached signal (G31 P99/P98) was specified, either moving the machine along an axis was not specified, or moving the machine along multiple axes was specified. Specify movement only along one axis.
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. 0876.
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.
023	ILLEGAL RADIUS COMMAND	In circular interpolation by radius designation, negative value was commanded for address R. Modify the program.

Number	Meaning	Contents
028	ILLEGAL PLANE SELECT	In the plane selection command, two or more axes in the same direction are commanded. Modify the program.
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 tool 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 CRC	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 vertical line. 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 comman, 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.

Number	Meaning	Contents
058	END POINT NOT FOUND	Block end point is not found in direct dimension drawing programming.
059	PROGRAM NUMBER NOT FOUND	In an external program 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 eiting.
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, thoughthe 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 cannot be made by monotonic machining 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 beween two blocks specified by address P in G71, G72, or G73. Modify the program.
067	CAN NOT ERROR IN MDI MODE	G70, G71, G72, or G73 command with address P and Q. Modify the program.
068	TEN OR MORE POCKETS	The number of pockets is greater than or equal to ten for G71 or G72 of type II.
069	FORMAT ERROR IN G70–G73	the final move command in the blocks specified by P and Q of G70, G71, G72, and G73 ended with chamfering or corner R. Modify the program.
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). Delete unnecessary programs and execute program registeration 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 registeration again.

Number	Meaning	Contents
074	ILLEGAL PROGRAM NUMBER	The program number is other than 1 to 9999. Modify the program number.
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 number of subprograms called exceeded the limit.
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 (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.
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	MANUAL RETURN TO THE REFERENCE POSITION IS IMPOSSIBLE BECAUSE OF A TEMPORARY STOP.	A manual return to the reference position cannot be made because the system is in the temporary stop state. After pressing the RESET key, execute manual return to the reference position.
092	AXES NOT ON THE REFERENCE POINT	Automatic reference position return (G28) or the commanded axis by G27 (Reference position return check) did not return to the reference position.
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.

Number	Meaning	Contents
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 94 to 97 reset, no automatic operation is 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 was turned off while memory was being rewritten by program edit operation. When this alarm is issued, clear the program by setting the setting parameter (PWE) to 1, then turning on the power again while holding down the <DELETE> key.
109	P/S ALARM	A value other than 0 or 1 was specified after P in the G08 code, or no value was specified.
110	DATA OVERFLOW	The absolute value of fixed decimal point display data exceeds the allowable range. Modify the program.
111	CALCULATED DATA OVERFLOW	The result of calculation turns out to be invalid, an alarm No.111 is issued. -10^{47} to -10^{-29}, 0, 10^{-29} to 10^{47} Modify the program.
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	Custom macro A contains an undefined H code in a G65 block. Custom macro B contains an error in a format other than <expression>. Correct the program.

Number	Meaning	Contents
115	ILLEGAL VARIABLE NUMBER	A value not defined as a variable number is designated in the custom macro or in high–speed cycle machining. The header contents are improper. This alarm is given in the following cases: High speed cycle machining 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 last 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 datais 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. Or BCD argument is negative, and other values than 0 to 9 are present on each line of BIN argument. Modify the program.
122	DUPLICATE 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	Custom macro A contains an address that cannot be specified in a G65 block. Custom macro B contains a format error in <expression>. Correct 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 attept 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	Meaning	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.
139	CAN NOT CHANGE PMC CONTROL AXIS	An axis is selected in commanding by PMC axis control. Modify the program.
145	ILLEGAL CONDITIONS IN POLAR COORDINATE INTERPOLATION	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. 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.
147	MOVE COMMAND TOO LARGE	The tool passes the coordinate origin, causing the amount of travel about the rotation axis to be too great. Modify the program or set bit 2 of parameter No. 399 accordingly.
150	ILLEGAL TOOL GROUP NUMBER	Tool Group No. of tool life management exceeds the maximum allowable value. Modify the program. Alternatively, modify the tool life data.
151	TOOL GROUP NUMBER NOT FOUND	The tool group of tool life management 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 of tool life management exceeds the maximum value registerable. 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 of tool life management 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 of tool life management is set. Correct the program.
157	TOO MANY TOOL GROUPS	The number of tool groups of tool life management to be set exceeds the maximum allowable value. 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 of tool life management, power was turned off. Set again.
175	ILLEGAL G107 COMMAND	Conditions when performing circular interpolation start or cancel not correct. Modify the program.
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: 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.
178	G05 COMMANDED IN G41/G42 MODE	G05 was commanded in the G41/G42 mode. Correct the program.
179	PARAM. SETTING ERROR	The number of controlled axes set by the parameter 597 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.

Number	Meaning	Contents
194	SPINDLE COMMAND IN SYN-CHRO–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.
195	MODE CHANGE ERROR	The control mode of the serial spindle cannot be changed. Check the Ladder diagram of the PMC.
197	C–AXIS COMMANDED IN SPINDLE MODE	The program specified a movement along the Cf–axis when the signal CON 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 tap, an S value is out of the range or is not specified. The range for S values which can be specified in rigid tapping is set in parameter 5243. Change the setting in the parameter or modify the program.
201	FEEDRATE NOT FOUND IN RIGID TAP	In the rigid tap, no F value is specified. Correct the program.
202	POSITION LSI OVERFLOW	In the rigid tap, spindle distribution value is too large.
203	PROGRAM MISS AT RIGID TAP-PING	In the rigid tap, position for a rigid M code (M29) or an S command is incorrect. Modify the program.
204	ILLEGAL AXIS OPERATION	In the rigid tap, an axis movement is specified between the rigid M code (M29) block and G84 (G74) block. Modify the program.
205	RIGID MODE DI SIGNAL OFF	Rigid mode DI signal is not ON when G84 (G74) is executed though the rigid M code (M29) is specified.Consult the PMC ladder diagram to find the reason the DI signal is not turned on.
210	CAN NOT COMAND M198/M199	M198 and M199 are executed in the schedule operation. M198 is executed in the DNC operation. Modify the program.
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 SYN-CHRO–MODE	Movement is commanded for the axis to be synchronously controlled.
214	ILLEGAL COMMAND IN SYN-CHRO–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)	G251 is further commanded in the G251 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 INDEPEN-DENTLY	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 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.
233	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.

2) Background edit alarm

Number	Meaning	Contents
???	BP/S alarm	BP/S alarm occurs in the same number as the P/S alarm that occurs in ordinary program edit. (070, 071, 072, 073, 074 085,086,087 etc.)
140	BP/S alarm	It was attempted to select or delete in the background a program being selected in the foreground. Use background editing correctly.

3) Absolute pulse coder (APC) alarm

Number	Meaning	Contents
3n0	nth–axis origin return	Manual reference position return is required for the nth–axis (n=1 – 8).
3n1	APC alarm: nth–axis communication	nth–axis APC communication error. Failure in data transmission Possible causes include a faulty APC, cable, or servo interface module.
3n2	APC alarm: nth–axis over time	nth–axis APC overtime error. Failure in data transmission. Possible causes include a faulty APC, cable, or servo interface module.
3n3	APC alarm: nth–axis framing	nth–axis APC framing error. Failure in data transmission. Possible causes include a faulty APC, cable, or servo interface module.
3n4	APC alarm: nth–axis parity	nth–axis APC parity error. Failure in data transmission. Possible causes include a faulty APC, cable, or servo interface module.
3n5	APC alarm: nth–axis pulse error	nth–axis APC pulse error alarm. APC alarm.APC or cable may be faulty.
3n6	APC alarm: nth–axis battery voltage 0	nth–axis APC battery voltage has decreased to a low level so that the data cannot be held. APC alarm. Battery or cable may be faulty.
3n7	APC alarm: nth–axis battery low 1	nth–axis axis APC battery voltage reaches a level where the battery must be renewed. APC alarm. Replace the battery.
3n8	APC alarm: nth–axis battery low 2	nth–axis APC battery voltage has reached a level where the battery must be renewed (including when power is OFF). APC alarm.

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	Meaning	Contents
3n9	SPC ALARM: n AXIS PULSE CODER	The n axis pulse coder has a fault.

- **The details of serial pulse coder alarm No.3n9**

The details of serial pulse coder alarm No. 3n9 are displayed in the diagnosis display (No.760 to 767, 770 to 777) as shown below.

	#7	#6	#5	#4	#3	#2	#1	#0
760 to 767		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 (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 (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.

	#7	#6	#5	#4	#3	#2	#1	#0
770 to 777	DTE	CRC	STB					

DTE : The serial pulse coder encountered a communication error. The pulse coder, feedbak 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.

5) Servo alarms

Number	Meaning	Contents and actions
400	SERVO ALARM: 1, 2TH AXIS OVERLOAD	1–axis, 2–axis overload signal is on. Refer to diagnosis display No. 720 or 721 for details.
401	SERVO ALARM: 1, 2TH AXIS VRDY OFF	1–axis, 2–axis servo amplifier READY signal (DRDY) went off.
402	SERVO ALARM: 3, 4TH AXIS OVERLOAD	3–axis, 4–axis overload signal is on. Refer to diagnosis display No. 722 or 723 for details.
403	SERVO ALARM: 3, 4TH AXIS VRDY OFF	3–axis, 4–axis servo amplifier READY signal (DRDY) went off.
404	SERVO ALARM: n–TH 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 axis card and servo amplifierr are connected.
405	SERVO ALARM: ZERO POINT RETURN FAULT	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.
406	SERVO ALARM: 7, 8TH AXIS OVER LOAD 7, 8TH AXIS VRDY OFF	7–axis, 8–axis overload signal is on. Refer to diagnosis display No. 726 or 727 for details. 7–axis, 8–axis servo amplifier READY signal (DRDY) went off.
4n0	SERVO ALARM: n–TH AXIS – EXCESS ERROR	The position deviation value when the n–th axis stops is larger than the set value. Note) Limit value must be set to parameter for each axis.
4n1	SERVO ALARM: n–TH AXIS – EXCESS ERROR	The position deviation value when the n–th axis moves is larger than the set value. Note) Limit value must be set to parameter for each axis.
4n3	SERVO ALARM: n–th AXIS – LSI OVERFLOW	The contents of the error register for the n–th axis exceeded ^2^{31} power. This error usually occurs as the result of an improperly set parameters.
4n4	SERVO ALARM: n–TH AXIS – DETECTION RELATED ERROR	N–th axis digital servo system fault. Refer to diagnosis display No. 720 and No.727 for details.
4n5	SERVO ALARM: n–TH AXIS – EXCESS SHIFT	A speed higher than 4000000 units/s was attempted to be set in the n–th axis. This error occurs as the result of improperly set CMR.
4n6	SERVO ALARM: n–TH AXIS – DISCONNECTION	Position detection system fault in the n–th axis pulse coder (disconnection alarm).
4n7	SERVO ALARM: n–TH AXIS – PARAMETER INCORRECT	This alarm occurs when the n–th axis is in one of the conditions listed below. (Digital servo system alarm) 1) The value set in Parameter No. 8n20 (motor form) is out of the specified limit. 2) A proper value (111 or –111) is not set in parameter No. 8n22 (motor revolution direction). 3) Illegal data (a value below 0, etc.) was set in parameter No. 8n23 (number of speed feedback pulses per motor revolution). 4) Illegal data (a value below 0, etc.) was set in parameter No. 8n24 (number of position feedback pulses per motor revolution). 5) Parameters No. 8n84 and No. 8n85 (flexible field gear rate) have not been set. 6) An axis selection parameter (from No. 269 to 274) is incorrect. 7) An overflow occurred during parameter computation.
490	SERVO ALARM: 5TH AXIS OVER LOAD	5–axis, 6–axis overload signal is on. Refer to diagnosis display No. 724 or 725 for details.
491	SERVO ALARM: 5, 6TH VRDY OFF	5–axis, 6–axis servo amplifier READY signal (DRDY) went off.

Number	Meaning	Contents and actions
494	SERVO ALARM: 5, 6TH AXIS VRDY ON	The axis card ready signal (MCON) for axes 5 and 6 is off, but the servo amplifier ready signal (DRDY) is not. Alternatively, when the power is applied, the DRDY is on, but the MCON is not. Ensure that the axis card and servo amplifier are connected.
495	SERVO ALARM: 5, 6TH AXIS ZERO POINT RETURN	This is a position control circuit error. It is likely that a return to the reference position failed because of an error in the NC or the servo system. Retry a return to the reference position.

NOTE
If an excessive spindle error alarm occurs during rigid tapping, the relevant alarm number for the tapping feed axis is displayed.

- **Details of servo alarm No.4n4**

The detailed descriptions of servo alarm number 4n4 are displayed with diagnosis numbers 720 to 727 in the sequence of axis numbers.

	#7	#6	#5	#4	#3	#2	#1	#0
720 to 727	OVL	LV	OVC	HCAL	HVA	DCAL	FBAL	OFAL

OVL : An overload alarm is being generated.
(This bit causes servo alarm No. 400, 402, 406, 490).

LV : A low voltage alarm is being generated in servo amp.
Check LED.

OVC : A overcurrent alarm is being generated inside of digital servo.

HCAL : An abnormal current alarm is being generated in servo amp.
Check LED.

HVAL : An overvoltage alarm is being generated in servo amp.
Check LED.

DCAL : A regenerative discharge circuit alarm is being generated in servo amp. Check LED.

FBAL : A disconnection alarm is being generated.
(This bit causes servo alarm No.4n6.)

OFAL : An overflow alarm is being generated inside of digital servo.

6) Spindle alarms

Number	Meaning	Contents and remedy
408	SPINDLE SERIAL LINK START FAULT	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.
409	SPINDLE ALARM DETECTION	A spindle amplifier alarm occurred in a system with a serial spindle. The alarm is indicated as "AL–XX" (where XX is a number) on the display of the spindle amplifier. For details, see Section 14. Setting bit 7 of parameter No. 0397 causes the spindle amplifier alarm number to appear on the screen.

7) Over travel alarms

Number	Meaning	Contents and remedy
5n0	OVER TRAVEL : +n	Exceeded the n–th axis + side stored stroke check 1, 2.
5n1	OVER TRAVEL : –n	Exceeded the n–th axis – side stored stroke check 1, 2.
5n2	OVER TRAVEL : +n	Exceeded the n–th axis + side stored stroke check 3.
5n3	OVER TRAVEL : –n	Exceeded the n–th axis – side stored stroke check 3.
5n4	OVER TRAVEL : +n	Exceeded the n–th axis + side stored stroke check 4.
5n5	OVER TRAVEL : –n	Exceeded the n–th axis – side stored stroke check 4.
520	OVER TRAVEL : +Z	A hardware overtravel occurred in the positive direction of the Z–axis.

8) Macro alarms

Number	Meaning	Contents and remedy
500 to 599	MACRO ALARM	This alarm is related to the custom macro, macro executor, or order–made macro (including conversational program inputs). Refer to the relevant manual for details. (The macro alarm number may coincide with an overtravel alarm number. However, they can be distinguished from each other because the overtravel alarm number is accompanied with the description of the alarm.

9) PMC alarms

Number	Meaning	Contents and remedy
600	PMC ALARM : INVALID INSTRUC-TION	An invalid–instruction interrupt occurred in the PMC.
601	PMC ALARM : RAM PARITY	A PMC RAM parity error occurred.
602	PMC ALARM : SERIAL TRANSFER	A PMC serial transfer error occurred.
603	PMC ALARM : WATCHDOG	A PMC watchdog timer alarm occurred.
604	PMC ALARM : ROM PARITY	A PMC ROM parity error occurred.
605	PMC ALARM : OVER STEP	The maximum allowable number of PMC ladder program steps was exceeded.
606	PMC ALARM : I/O MODULE AS-SIGNMENT	The assignment of I/O module signals is incorrect.
607	PMC ALARM : I/O LINK	An I/O link error occurred. The details are listed below.

Number	Details of PMC alarm (No. 607)
010	* Communication error (SLC (master) internal register error)
020	* An SLC RAM bit error occurred (verification error).
030	* An SLC RAM bit error occurred (verification error).
040	No I/O unit has been connected.
050	32 or more I/O units are connected.
060	* Data transmission error (no response from the slave)
070	* Communication error (no response from the slave)
080	* Communication error (no response from the slave)
090	An NMI (for other than alarm codes 110 to 160) occurred.
130	* An SLC (master) RAM parity error occurred (detected by hardware).
140	* An SLC (slave) RAM parity error occurred (detected by hardware).
160	* SLC (slave) communication error * AL0 : Watchdog timer DO clear signal received * IR1 : CRC or framing error Watchdog timer alarm Parity error

Hardware errors are indicated with an asterisk (*).

10) Overheat alarms

Number	Meaning	Contents and remedy
700	OVERHEAT: CONTROL UNIT	Control unit overheat Check that the fan motor operates normally, and clean the air filter.
704	Overheat: Spindle	The spindle overheated during spindle variation detection. Check the cutting conditions.

11) M–NET alarm

Number	Meaning	Contents and remedy
899	M–NET INTERFACE ALARM	This alarm is related to a serial interface for an external PLC. The details are listed below.

Number	Details of M–NET alarm (No. 899)
0001	Abnormal character (character other than transmission codes) received
0002	”EXT” code error
0003	Connection time monitor error (parameter No. 0464)
0004	Polling time monitor error (parameter No. 0465)
0005	Vertical parity or framing error detected
0257	Transmission time–out error (parameter No. 0466)
0258	ROM parity error
0259	Overrun error detected
Others	CPU interrupt detected

12) System alarms

(These alarms cannot be reset with reset key.)

Number	Meaning	Contents and remedy
910	MAIN RAM PARITY	This RAM parity error is related to low–order bytes. Replace the memory PC board.
911	MAIN RAM PARITY	This RAM parity error is related to high–order bytes. Replace the memory PC board.
912	SHARED RAM PARITY	This parity error is related to low–order bytes of RAM shared with the digital servo circuit. Replace the axis control PC board.
913	SHARED RAM PARITY	This parity error is related to high–order bytes of RAM shared with the digital servo circuit. Replace the axis control PC board.
914	SERVO RAM PARITY	This is a local RAM parity error in the digital servo circuit. Replace the axis control PC board.
915	LADDER EDITING CASSETTE RAM PARITY	This RAM parity error is related to low–order bytes of the ladder editing cassette. Replace the ladder editing cassette.
916	LADDER EDITING CASSETTE RAM PARITY	This RAM parity error is related to high–order bytes of the ladder editing cassette. Replace the ladder editing cassette.
920	WATCHDOG ALARM	This is a watchdog timer alarm or a servo system alarm for axis 1 to 4. Replace the master or axis control PC board.
921	SUB CPU WATCHDOG ALARM	This is a watchdog timer alarm related to the sub–CPU board or a servo system alarm for axis 5 or 6. Replace the sub–CPU board or the axis–5/6 control PC board.
922	7/8 AXIS SERVO SYSTEM ALARM	This is a servo system alarm related to axis 7 or 8. Replace the axis–7/8 control PC board.
930	CPU ERROR	This is a CPU error. Replace the master PC board.
940	PC BOARD INSTALLATION ERROR	PC board installation is incorrect. Check the specification of the PC board.
941	MEMORY PC BOARD CONNECTION ERROR	The memory PC board is not connected securely. Ensure that the PC board is connected securely.

Number	Meaning	Contents and remedy
945	SERIAL SPINDLE COMMUNICATION ERROR	The hardware configuration is incorrect for the serial spindle, or a communication alarm occurred. Check the hardware configuration of the spindle. Also ensure that the hardware for the serial spindle is connected securely.
946	SECOND SERIAL SPINDLE COMMUNICATION ERROR	Communication is impossible with the second serial spindle. Ensure that the second serial spindle is connected securely.
950	FUSE BLOWN ALARM	A fuse has blown. Replace the fuse (+24E; F14).
960	SUB CPU ERROR	This is a sub–CPU error. Replace the sub–CPU PC board.
998	ROM PARITY	This is a ROM parity error. Replace the ROM board in which the error occurred.

13) External alarm

Number	Meaning	Contents and remedy
1000	EXTERNAL ALARM	This alarm was detected by the PMC ladder program. Refer to the relevant manual from the machine builder for details.

14) Alarms Displayed on spindle Servo Unit

Alarm No.	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.
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 cirtcuit	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.

Alarm No.	Meaning	Description	Remedy
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 ciucuit 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 internalto LSI for serial data transfer. This check is made only when power is turned on.	Remove cause, then reset alarm.
AL–33	Insufficient DC link section charging	Detects insuffieient charging of direct current power supply voltage in power circuit section when magnetic contactor in amplifier is turned on (such as open phase and defectifve 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.

Alarm No.	Meaning	Description	Remedy
AL–39	Alarm for indicating failure in detecting 1–rotation signal for Cs contouring control	Detects 1–rotaion signal detection failure in Cs contouring contorl.	Make 1–rotaion 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–rotaion signal adjustment.
AL–41	Alarm for indicating failure in detecting position coder 1–rotaion 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 signasl 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 abnormailty	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 ciucuit 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.
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**

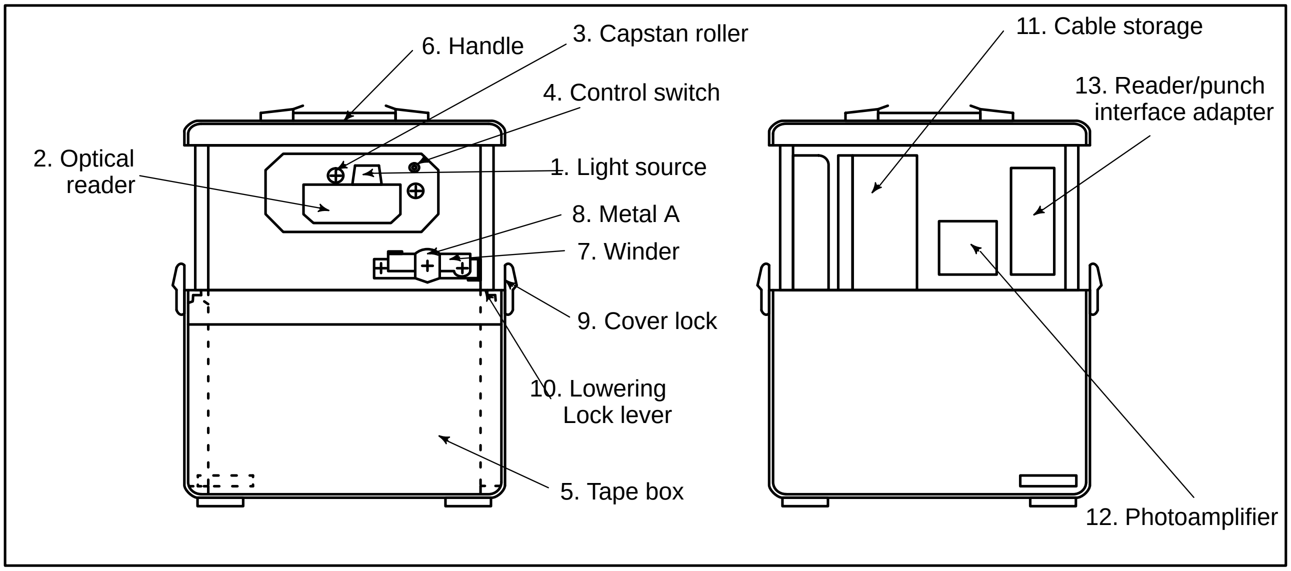


Table H Description of Each Section

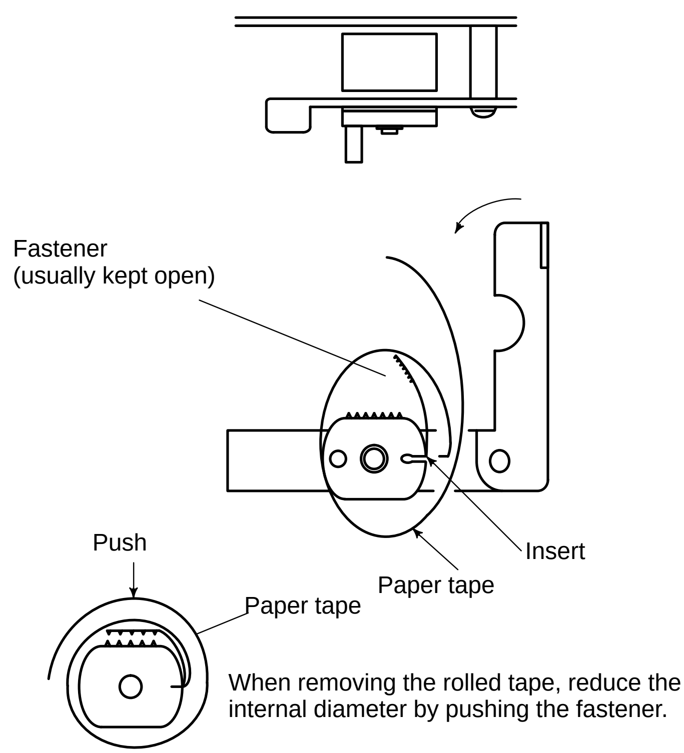
No.	Name	Descriptions
1	Light Sources	An LED (Light emitting diode) is mounted for each channel and for the feed hole (9 diodes in total). A built–in Stop Shoe functions to decelerate the tape. The light source is attracted to the optical reader by a magnet so that the tape will be held in the correct 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 lightsource. 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.

Table H Description of Each Section

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

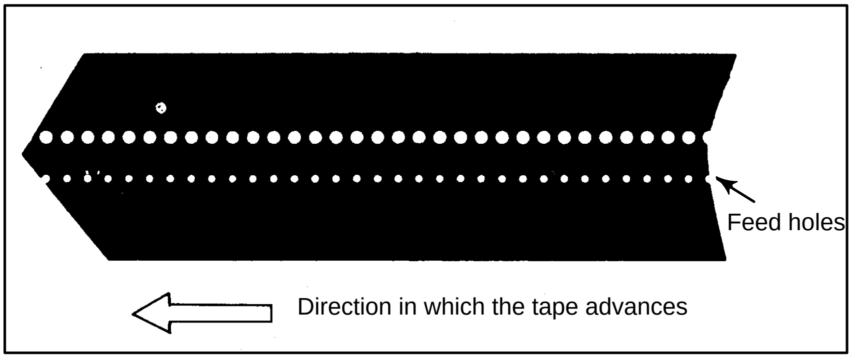
1 Unlock the cover locks 9. Raise the tape reader with the handle 6 until it clicks, then lower the tape reader. The tape reader then appears and is secured. Check that the lowering lock levers 10 are horizontal.

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.

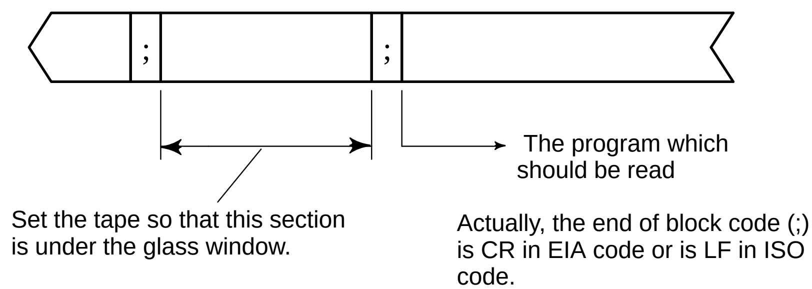
15 Raise the tape reader with the handle **6** to unlock the latch, then gently lower it.

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.



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.

Index

[A]

[B]

[C]

[D]

[E]

[F]

[G]

[H]

[I]

[J]

[K]

[L]

[M]

[N]

[O]

[P]

[R]

[S]

[T]

[V]

[W]

Revision Record

FANUC Series 0–TD/0–GCD OPERATOR’S MANUAL (B–62544EN)

Revision	Date	Contents	Revision	Date	Contents
02	Feb.,’97	Total revision			
01	Dec., ’94				