Changes for the Better

MITSUBISHI ELECTRIC

CNC

MELDAS 600L Series

PROGRAMMING MANUAL

BNP-B2232C(ENG)

Introduction

This instruction manual describes the methods of using the high-performance contour control software-fixed type CNC (NC hereafter) MELDAS 600L Series mainly for a lathe.
The programming methods for all of the above models are described, so read this manual thoroughly before starting use.
In respect to the functions related to the multi-axis multi-system, the programming and alarm details for each system are the same as the general-purpose (2-axis, 3-axis) lathe.
Explanations in this manual assume that all functions are provided with all of the above models. However, all options are not necessarily provided with each CNC, so refer to the specifications issued by the machine manufacturer before starting use.
Thoroughly read the "Precautions for Safety" given on the next page to ensure safe use of this numerical control unit.

Details described in this manual

(1) This manual gives general explanations from the standpoint of the NC side.
For explanations concerning individual machine tools, refer to the instruction manual issued by the machine manufacturer.
For items described as "Restrictions, "Usable State", etc., the instruction manual issued by the machine manufacturer takes precedence over this manual.

(2) While every effort has been made to describe special handling in this manual, items not described in this manual should be interpreted as "Not Possible".

(3) The multi-system function is an additional specification. The 3-system model is explained as an example in this manual, but the number of systems that can be used will differ according to the model.
Note that the maximum number of spindle axes will also differ according to the model.
Check the specifications before starting use.

⚠ CAUTION

⚠ For items described in "Restrictions" or "Usable State", the instruction manual issued by the machine manufacturer takes precedence over this manual.
⚠ Items not described in this instruction manual should be interpreted as "Not Possible".
⚠ This manual has been written on the assumption that all option functions are added.
Refer to the specifications issued by the machine manufacturer before starting use.
⚠ Refer to the instruction manual issued by the machine manufacturer for explanations on each machine tool.
⚠ Some screens and functions may differ or may not be usable depending on the NC system version.

Precautions for Safety

Always read the specifications issued by the machine manufacturer, this manual, related manuals and attached documents before installation, operation, programming, maintenance or inspection to ensure correct use.
Understand this numerical control unit, safety items and cautions before using the unit.
This manual ranks the safety precautions into "**DANGER**", "**WARNING**" and "**CAUTION**".

⚠ **DANGER**	When the user may be subject to imminent fatalities or major injuries if handling is mistaken.
⚠ **WARNING**	When the user may be subject to fatalities or major injuries if handling is mistaken.
⚠ **CAUTION**	When the user may be subject to injuries or when physical damage may occur if handling is mistaken.

Note that even items ranked as " ⚠ **CAUTION**", may lead to major results depending on the situation. In any case, important information that must always be observed is described.

⚠ DANGER

Not applicable in this manual.

⚠ WARNING

Not applicable in this manual.

⚠ CAUTION

1. Items related to product and manual

⚠ For items described as "Restrictions" or "Usable State" in this manual, the instruction manual issued by the machine manufacturer takes precedence over this manual.
⚠ Items not described in this instruction manual should be interpreted as "Not Possible".
⚠ This manual is written on the assumption that all option functions are added. Refer to the specifications issued by the machine manufacturer before starting use.
⚠ Some screens and functions may differ or may not be usable depending on the NC system version.

⚠ CAUTION

2. Items related to programming

- ⚠ Because of key chattering etc., during editing, the commands with no value after G become a "G00" operation during running.
- ⚠ ";", "EOB" and "%" "EOR" are expressions used for the explanation. The actual codes are "; (line feed)" and "%" for ISO, and "EOB" (End Of Block) and "EOR" (End Of Record) for EIA.
- ⚠ The commands with no value after G become a "G00" operation during running.
- ⚠ Always carry out dry run operation before actual machining, and confirm the machining program, tool offset amount and workpiece offset amount, etc.
- ⚠ When creating the machining program, select adequate machining conditions, and make sure not to exceed the machine and NC's performance, capacity and limits. Examples given in this manual do not take the machining conditions into consideration.
- ⃠ Do not change fixed cycle programs without the prior approval of the machine manufacturer.
- ⚠ When programming the multi-system, take special care to the movements of the programs for other systems.
- ⚠ During the spindle synchronous control mode, do not turn the rotation command for the slave spindle OFF while the master spindle and slave spindle are chucked on the same workpiece. This will be hazardous as the slave spindle will stop.
- ⚠ Do not issue another axis name change command before axis name change cancel is issued once axis name change is commanded.

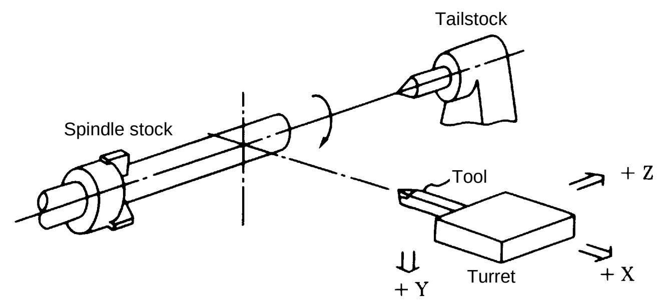
CONTENTS

1. CONTROL AXES

1.1 Coordinate Word and Control Axis

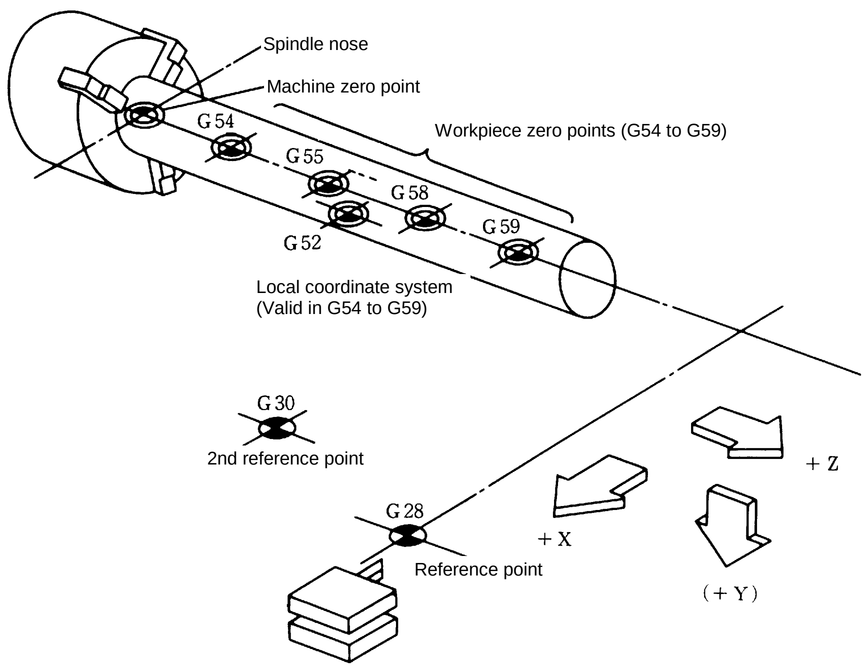
Function and purpose

In the case of a lathe, the axis parallel to the spindle is known as the Z axis and its forward direction is the direction in which the turret moves away from the spindle stock while the axis at right angles to the Z axis is the X axis and its forward direction is the direction in which it moves away from the Z axis, as shown in the figure below.



Coordinate axes and polarities

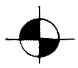
Since coordinates based on the right hand rule are used with a lathe, the forward direction of the Y axis in the above figure which is at right angles to the X-Z plane is downward. It should be borne in mind that an arc on the X-Z plane is expressed as clockwise or counterclockwise as seen from the forward direction of the Y axis. (Refer to the section on circular interpolation.)



Relationship between coordinates

1.2 Coordinate Systems and Coordinate Zero Point Symbols

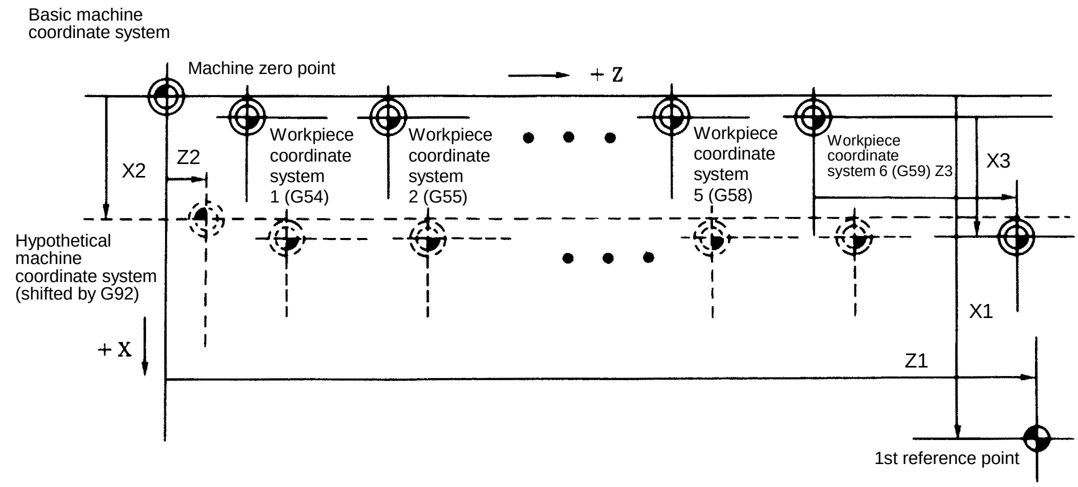
Function and purpose

 : Reference point

 : Machine coordinate zero point

 : Workpiece coordinate zero points (G54 to G59)

Upon completion of the reference point return, the parameters are referred to and automatically set for the basic machine coordinate system and workpiece coordinate systems (G54 to G59).
The basic machine coordinate system is set so that the first reference point is at the position designated by the parameter from the basic machine coordinate zero point (machine zero point).



The local coordinate system (G52) is valid on the coordinate systems designated by the commands for the workpiece coordinate systems 1 to 6.
Using the G92 command, the basic machine coordinate system can be shifted and made the hypothetical machine coordinate system. At the same time, workpiece coordinate systems 1 to 6 are also shifted.

2. INPUT COMMAND UNITS

2.1 Input Command Units

Function and purpose

These are the units used for the movement amounts in the program as commanded by the MDI input. They are expressed in millimeters, inches or degrees (°).

2.2 Input Setting Units

Function and purpose

These are the units of setting data which are used, as with the compensation amounts, in common for all axes.

The input command unit can be selected for each axis and input setting units can be selected in common for the axes by parameters from among the following types. (For further details on settings, refer to the sections about control.)

	Type	Linear axis				Rotation axis (°)
		Millimeter		Inch		
		Diametrical command	Radial command	Diametrical command	Radial command	
Input command unit	#1003 cunit=10	0.001	0.001	0.0001	0.0001	0.001
	=1	0.0001	0.0001	0.00001	0.00001	0.0001
Min. movement unit	IS-B	0.0005	0.001	0.0005	0.0001	0.001
	IS-C	0.00005	0.0001	0.00005	0.00001	0.0001
Input setting unit	IS-B	0.001	0.001	0.0001	0.0001	0.001
	IS-C	0.0001	0.0001	0.00001	0.00001	0.0001

(Note 1) Inch/metric conversion is performed in either of 2 ways: conversion from the Parameter screen ("Initial inch": valid only when the power is turned ON) and conversion using the G command (G20 or G21).
However, when a G command is used for the conversion, the conversion applies only to the input command units and not to the input setting units.
Consequently, the tool offset amounts and other compensation amounts as well as the variable data should be preset to correspond to input setting unit.

(Note 2) The millimeter and inch systems cannot be used together.

3. DATA FORMATS

3.1 Tape Codes

Function and purpose

The tape command codes used for this NC are combinations of alphabet letters (A, B, C...Z), numbers (0, 1, 2...9) and signs (+, –, /...). These alphabet letters, numbers and signs are referred to as characters. Each character is represented by a combination of 8 holes which may, or may not, be present.
These combinations make up what is called codes.
This NC employs the ISO code (R-840).

⚠CAUTION

⚠ ";", "EOB" and "%" "EOR" are expressions used for the explanation. The actual codes are "line feed" and "%" for ISO.

Detailed description

(1) For the sake of convenience, a ";" has been used in the NC display to indicate End Of Block (EOB/LF) which separates one block from another. Do not use the ";" key, however, in actual programming but use the keys in the following table instead.

EOB/EOR keys and displays

Code used / Key used	ISO	NC display
End Of Block	LF or NL	;
End Of Record	%	%

(Note 1) If a code not given in Table of tape codes is assigned during operation, an Illegal address error "P32" will result.

(Note 2) The following codes which exist with ISO can be designated by parameter:
[(left square parenthesis)
] (right square parenthesis)
(sharp sign)
∗ (asterisk)
= (equals sign)
: (colon)
! (exclamation mark) = (queuing code)
$ (dollar sign) = (code designating system number)
Any codes which overlap with existing codes or codes which result in parity H cannot be designated.

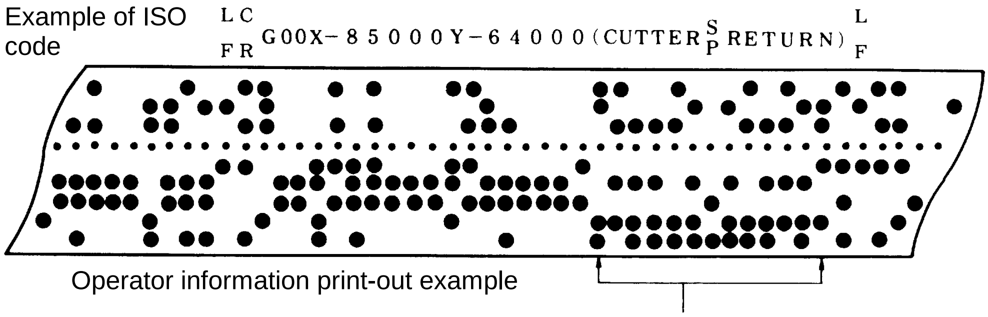
(2) Significant data section (label skip function)

All data up to the first EOB (;), after the power has been turned ON or after operation has been reset, are ignored during automatic operation based on tape, memory loading operation or during a search operation. In other words, the significant data section of a tape extends from the character or number code after the first EOB (;) code after resetting to the point where the reset command is issued.

(3) Control out, control in

When the ISO code is used, all data between control out "(" and control in ")" are ignored by the NC, although these data appear on the setting display unit. Consequently, the command tape name, number and other such data not directly related to control can be inserted in this section.
This information will also be loaded, however, during tape loading.
The system is set to the "control in" mode when the power is turned ON.

Example of ISO code



(4) EOR (%) code

Generally, End Of Record is punched at both ends of the tape. It has the following functions:

(a) Rewind stop when rewinding tape (with tape handler)
(b) Rewind start during tape search (with tape handler)
(c) Completion of loading during tape loading

3.2 Program Formats

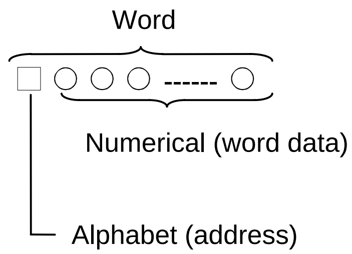
Function and purpose

The prescribed arrangement used when assigning control information to the NC unit is known as the "program format", and the format used with the NC is called the "word address format."

Detailed description

(1) Word and address

A word is a collection of characters arranged in a specific sequence. This entity is used as the unit for processing data and for causing the NC to execute specific operations.
Each word used for the NC consists of an alphabet letter and a number of several digits (sometimes with a "+" or "–" sign placed at the head of the number).



Word configuration

The alphabet letter at the head of the word is the address. It defines the meaning of the numerical information which follows it.
With the NC, "S○=" can be commanded for a multiple number of spindle commands although this applies only to address S.
For details of the types of words and the number of significant digits of numbers used for this NC, refer to Table 1 Format details and abbreviations.

(2) Blocks

A block is a collection of words. It includes the information which is required for the NC to execute one specific operation. One block unit constitutes a complete command. The end of each block is marked with an EOB (End-Of-Block) code.

(3) Programs

A program is a collection of several blocks.

(Note 1) If there is no number after the alphabetic character in the actual program, the value following the alphabetic character will be handled as 0.

(Example) G28XYZ; → G28X0Y0Z0;

Table 1 Format details and abbreviations

Item		Abbreviation
Program number		O8
Sequence number		N5
Preparatory function		G3/G21
Movement command	Input setting unit A 0.01°, mm	X + 62 Z + 62 ✚ + 62
	Input setting unit B 0.001°, mm	X + 53 Z + 53 ✚ + 53
	Input setting unit C 0.0001°, mm	X + 44 Z + 44 ✚ + 44
Movement command, circular, cutter radius	Input setting unit A 0.01°, mm	I + 62 K + 62
	Input setting unit B 0.001°, mm	I + 53 K + 53
	Input setting unit C 0.0001°, mm	I + 44 K + 44
Dwell	Input setting unit A 0.01°, mm	X + 53 P8
	Input setting unit B 0.001°, mm	X + 53 P8
	Input setting unit C 0.0001°, mm	X + 53 P8
Feed function	Input setting unit A 0.01°, mm	F62 (feed per minute) F43 (feed per rotation)
	Input setting unit B 0.001°, mm	F53 (feed per minute) F34 (feed per rotation)
	Input setting unit C 0.0001°, mm	F54 (feed per minute) F25 (feed per rotation)
Tool offset		T1/T2
Miscellaneous function		M2/M8
Spindle function		S2/S5/S8 or S○= n
Tool function		T2/T8
2nd miscellaneous function		A8/B8/C8
Subprogram		P8H5L4
Fixed cycle	Input setting unit A 0.01°, mm	R + 62 Q62 P8 L4
	Input setting unit B 0.001°, mm	R + 53 Q53 P8 L4
	Input setting unit C 0.0001°, mm	R + 44 Q44 P8 L4

(Note 1) " ✚ " denotes the A, B, C, Y, P or R.

(Note 2) The number of digits in the words is checked by the maximum number of digits in the addresses.

3.3 Tape Storage Format

Function and purpose

(1) Storage tape and storage sections

The section which is stored into the memory extends from the character following the head EOB after resetting as far as the EOR code.
The significant codes listed in Table of tape codes in Section 3.1 are the codes in the above storage section which are actually stored into the memory. All other codes are ignored and are not stored.
The data between control out "(" and control in ")" are stored into the memory.

3.4 Optional Block Skip

Function and purpose

This function selectively ignores specific blocks in a machining program which starts with the "/" (slash) code.

Detailed description

(1) Provided that the optional block skip switch is ON, blocks starting with the "/" code are ignored. They are executed if the switch is OFF.
Parity check is valid regardless of whether the optional block skip switch is ON or OFF.
When, for instance, all blocks are to be executed for one workpiece but specific blocks are not to be executed for another workpiece, the same command tape can be used to machine different parts by inserting the "/ " code at the head of those specific blocks.

Precautions for using optional block skip

(1) Put the "/" code at the head of the block. When inserted in a block, this is handled as a division sign.

(Example) N20G1X25./Z25.; ······ NG
(This will be handled as 25. /0, so the error P283 "Divided by zero" will occur.)
/N20G1X25.Z25.; ······ OK

(2) Parity checks (H and V) are conducted regardless of the optional block skip switch state.

(3) The optional block skip is processed immediately before the pre-read buffer.
Consequently, it is not possible to skip up to the block which has been read into the pre-read buffer.

(4) This function is valid even during a sequence number search.

(5) All blocks with the "/" code are also input and output during tape storing and tape output, regardless of the state of the optional block skip switch.

3.5 Program/Sequence/Block Numbers (O, N)

Function and purpose

These numbers are used for monitoring the execution of the machining programs and for calling both machining programs and specific stages in machining programs.

(1) Program numbers are classified by workpiece correspondence or by subprogram units, and they are designated by the address "O" followed by a number with up to 8 digits.

(2) Sequence numbers are attached where appropriate to command blocks which configure machining programs, and they are designated by the address "N" followed by a number with up to 5 digits.

(3) Block numbers are automatically provided inside the NC itself. They are preset to "0" every time a program number or sequence number is read, and they are counted up one at a time unless program numbers or sequence numbers are commanded in blocks which are subsequently read.
Consequently, all the blocks of the machining programs given in the table below can be determined without further consideration by combinations of program numbers, sequence numbers and block numbers.

NC input machining program	NC monitor display		
	Program No.	Sequence No.	Block No.
O12345678 (DEMO.PROG);	12345678	0	0
N100 G00 G90 X120. Z100.;	12345678	100	0
G94 S1000;	12345678	100	1
N102 G71 P210 Q220 I0.2 K0.2 D0.5 F600;	12345678	102	0
N200 G94 S1200 F300;	12345678	200	0
N210 G01 X0 Z95.;	12345678	210	0
G01 X20.;	12345678	210	1
G03 X50. Z80. K–15.;	12345678	210	2
G01 Z55.;	12345678	210	3
G02 X80. Z40. I15.;	12345678	210	4
G01 X100.;	12345678	210	5
G01 Z30.;	12345678	210	6
G02 Z10. K–15.;	12345678	210	7
N220 G01 Z0;	12345678	220	0
N230 G00 X120. Z150.;	12345678	230	0
N240 M02;	12345678	240	0
%	12345678	240	0

3.6 G Code System

Function and purpose

These numbers are used to monitor the execution status of the machining program, or to call a machining program or a specific process in the machining program.
There are 3 G code systems: 1, 2 and 3. Parameters "G code type 1", "G code type 2" and "G code type 3" are used to set the applicable system.
G code system 3 is an additional specification.
The description of the G functions is based on G code system 2 which serves as the standard.

(Note 1) An alarm results when a G code not listed in the table is commanded. ("P34": Illegal G code)

(Note 2) An alarm results when a G code not included in the additional specifications is commanded.

(Example) An alarm ("P50 No spec: Inch/mm") occurs when the inch command G code (G20) is commanded although the inch/mm specifications have not been provided.

Table of G code systems

G code system (standard = 2)			Group	Function name	Reference section in this manual (Section)
1	**2**	**3**			
G00	G00	G00	01	Positioning	6.1
■ G01	■ G01	■ G01	01	Linear interpolation	6.2
G02	G02	G02	01	Circular interpolation (clockwise)	6.3, 6.4
G03	G03	G03	01	Circular interpolation (counterclockwise)	6.3, 6.4
G04	G04	G04	00	Dwell	8.1, 8.2
G09	G09	G09	00	Exact stop	7.8
G10	G10	G10	00	Data setting	12.5
G11	G11	G11	00	Data setting mode cancel	12.1
G12.1	G12.1	G12.1	19	Milling mode ON	6.8
▲ G13.1	▲ G13.1	▲ G13.1	19	Milling mode OFF	
▲ G14	▲ G14	▲ G14	18	Balance cut OFF	17.4
G15	G15	G15	18	Balance cut ON	
G16	G16	G16	02	Y-Z cylindrical plane selection	6.8
■ G17	■ G17	■ G17	02	X-Y plane selection	6.5
■ G18	■ G18	■ G18	02	Z-X plane selection	
■ G19	■ G19	■ G19	02	Y-Z plane selection	
■ G20	■ G20	■ G70	06	Inch command	5.3
■ G21	■ G21	■ G71	06	Metric command	
G22	G22	G22	04	Barrier check ON	15.1
▲ G23	▲ G23	▲ G23	04	Barrier check OFF	

G code system (standard = 2)			Group	Function name	Reference section in this manual (Section)
G27	G27	G27	00	Reference point return check	14.9
G28	G28	G28	00	Reference point return	14.7
G29	G29	G29	00	Return from reference point	
G30	G30	G30	00	2nd reference point return	14.8
G31	G31	G31	00	Skip function	16.1
G32	G33	G33	01	Thread cutting	6.7
G34	G34	G34	01	Variable lead thread cutting	
G35	G35	G35	01	Circular thread cutting (CW)	
G36	G36	G36	01	Circular thread cutting (CCW)	
G37	G37	G37	00	Automatic tool length offset, automatic tool length measurement	16.3
▲ G40	▲ G40	▲ G40	07	Tool nose R compensation cancel	12.4
G41	G41	G41	07	Tool nose R compensation left	
G42	G42	G42	07	Tool nose R compensation right	
■ G43	■ G43	■ G43	08	2nd spindle control OFF	10.6
■ G44	■ G44	■ G44	08	2nd spindle control ON	
G46	G46	G46	07	Tool nose R compensation (automatic selection of direction) ON	12.4
G50	G92	G92	00	Coordinate system setting Spindle clamp speed setting	14.6 10.4
G52	G52	G52	00	Local coordinate system setting	14.11
G53	G53	G53	00	Machine coordinate system selection	14.5
▲ G54	▲ G54	▲ G54	12	Workpiece coordinate system selection 1	14.10
G55	G55	G55	12	Workpiece coordinate system selection 2	
G56	G56	G56	12	Workpiece coordinate system selection 3	
G57	G57	G57	12	Workpiece coordinate system selection 4	
G58	G58	G58	12	Workpiece coordinate system selection 5	
G59	G59	G59	12	Workpiece coordinate system selection 6	
G61	G61	G61	13	Exact stop check mode	7.9
▲ G64	▲ G64	▲ G64	13	Cutting mode	7.10
G65	G65	G65	00	Macro call	13.7.1
G66	G66	G66	14	Macro modal call A	
G66.1	G66.1	G66.1	14	Macro modal call B	
▲ G67	▲ G67	▲ G67	14	Macro modal call cancel	
G68	G68	G68	15	Facing turret mirror image ON	13.8
▲ G69	▲ G69	▲ G69	15	Facing turret mirror image OFF	
G70	G70	G72	09	Finishing cycle	13.2.4
G71	G71	G73	09	Longitudinal rough cutting cycle	13.2.1
G72	G72	G74	09	Face rough cutting cycle	13.2.2
G73	G73	G75	09	Stock removal in rough cutting cycle	13.2.3

G code system (standard = 2)			Group	Function name	Reference section in this manual (Section)
G74	G74	G76	09	Face cut-off cycle	13.2.5
G75	G75	G77	09	Longitudinal cut-off cycle	13.2.6
G76	G76	G78	09	Compound thread cutting cycle	13.2.7
G76.1	G76.1	G76.1	09	2-system simultaneous thread cutting cycle 1	17.13
G76.2	G76.2	G76.2	09	2-system simultaneous thread cutting cycle 2	
▲ G80	▲ G80	▲ G80	09	Hole drilling cycle cancel	13.3.4
G83	G83	G83	09	Deep hole drilling cycle 1 (Z axis)	13.3.1
G79	G83.2	G83.2	09	Deep hole drilling cycle 2	13.4
G84	G84	G84	09	Tap cycle (Z axis)	13.3.2
G85	G85	G85	09	Boring cycle (Z axis)	13.3.3
G87	G87	G87	09	Deep hole drilling cycle (X axis)	13.3.1
G88	G88	G88	09	Tap cycle (X axis)	13.3.2
G89	G89	G89	09	Boring cycle (X axis)	13.3.3
G90	G77	G20	09	Longitudinal cutting fixed cycle	13.1.1
G92	G78	G21	09	Thread cutting fixed cycle	13.1.2
G94	G79	G24	09	Face cutting fixed cycle	13.1.3
■ G96	■ G96	■ G96	17	Constant surface speed control	10.3
■ G97	■ G97	■ G97	17	Constant surface speed control cancel	
■ G98	■ G94	■ G94	05	Asynchronous feed	7.3
■ G99	■ G95	■ G95	05	Synchronous feed	
–	■ G90	■ G90	03	Absolute value command	5.1
–	■ G91	■ G91	03	Incremental value command	
–	▲ G98	▲ G98	10	Hole drilling cycle initial return	13.3
–	G99	G99	10	Hole drilling cycle reference point return	
G110	G110	G110	00	Cross machining command	17.6
G111	G111	G111	00	Axis name change	18.3
G113	G113	G113	00	Spindle synchronization, tool/spindle synchronization cancel	17.8-17.12
G114.1	G114.1	G114.1	00	Spindle synchronization	17.8
G114.2	G114.2	G114.2	00	Tool/spindle synchronization 1 (polygon machining)	17.9
G114.3	G114.3	G114.3	00	Tool/spindle synchronization 2 (hobb machining)	17.10
G115	G115	G115	00	Waiting at designated start point 1	17.2
G116	G116	G116	00	Waiting at designated start point 2	17.3
G117	G117	G117	00	Miscellaneous function output during axis movement	18.1, 18.2
G125	G125	G125	00	Control axis synchronization	17.7
G126	G126	G126	00	Control axis superimposition	17.11
G164	G164	G164	00	Spindle superimposition	17.12
G173	G173	G173	00	Inclined coordinate rotation control	13.16
G200 to	G200 to G999			G macro call	18.3

(Note 1) The " ▲ " mark denotes a G code which is selected within each group when the power is turned ON or when resetting that initializes the modal commands is executed.

(Note 2) The " ■ " mark denotes a G code for which a parameter can be selected as the initial status when the power is turned ON or when resetting that initializes the modal commands is executed. Note that the inch/metric conversion can be made only when the power is turned ON.

⚠CAUTION

⚠ The commands with "no value after G", will be handled as "G00" during operation.

3.7 Precautions Before Machining

Precautions before machining

⚠CAUTION

⚠ Before starting actual machining, always carry out dry operation to confirm the machining program, tool offset amount and workpiece offset amount, etc.

⚠ When creating the machining program, select the appropriate machining conditions, and make sure that the performance, capacity and limits of the machine and NC are not exceeded. The examples do not consider the machining conditions.

4. BUFFER REGISTER

4.1 Pre-read Buffers

Function and purpose

During automatic processing, the contents of 1 block are normally pre-read so that program analysis processing is conducted smoothly. However, during nose R compensation, a maximum of 5 blocks are pre-read for the intersection point calculation including interference check.

The specifications of pre-read buffer are as follows:

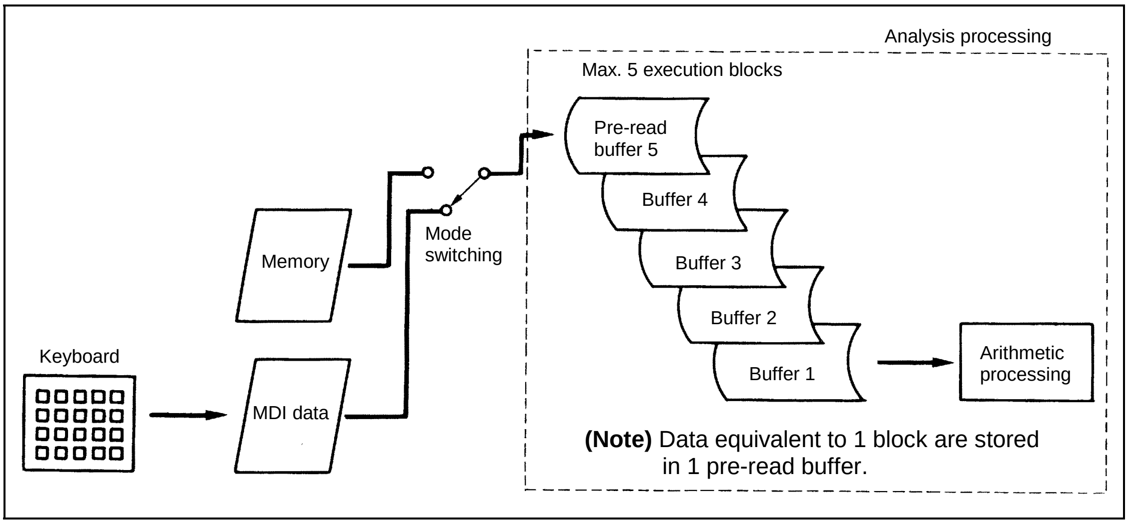
(1) The data of 1 block are stored in this buffer.

(2) Only the significant data in the significant data section are stored into the pre-read buffer.

(3) When codes are sandwiched in the control in or control out mode and the optional block skip function is ON, the data extending from the "/" (slash) code up to the EOB code are not read into the pre-read buffer.

(4) The pre-read buffer contents are cleared with resetting.

(5) When the single block function is ON during continuous operation, the pre-read buffer stores the following block data and then stops operation.

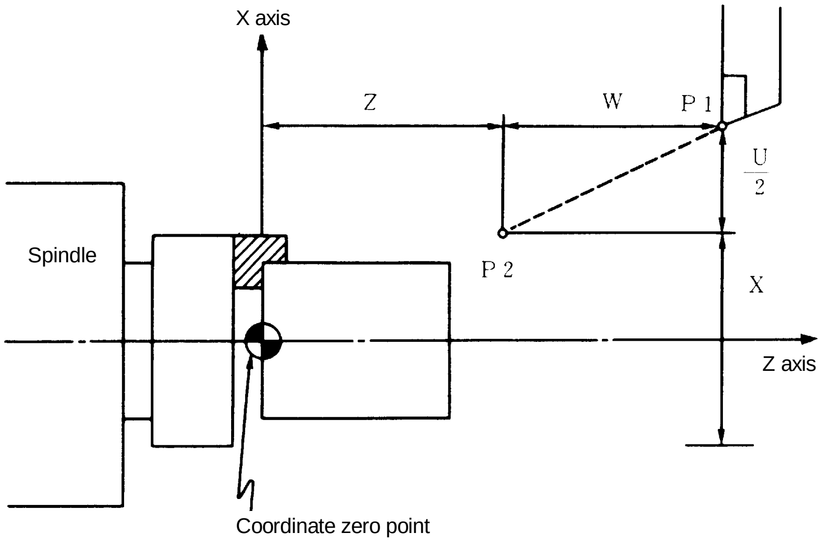


5. POSITION COMMANDS

5.1 Incremental/Absolute Value Commands

Function and purpose

There are 2 methods of issuing tool movement amount commands: the incremental value command and the absolute value command.
The incremental value method applies for coordinates of a point which is to be moved and it issues a command using the distance from the present point, on the other hand, the absolute value method issues a command using the distance from the coordinate zero point. The following figure shows what happens when the tool is moved from point P1 to point P2.



Incremental and absolute value commands

The incremental and absolute value commands for the X and Z axes are identified by addresses when control parameter "#6 ABS/INC Addr." is ON and by G codes (G90/G91) when it is OFF.
Similarly, even with additional axes (C or Y axis), they are differentiated by addresses, or G code.

		Command system	Remarks
Absolute value	X axis	Address X	For setting correspondence between addresses and axes into machine parameters. Absolute and incremental values can be used together in the same block.
	Z axis	Address Z	
	C/Y axis	Address C/Y	
Incremental value	X axis	Address U	
	Z axis	Address W	
	C/Y axis	Address H/V	

(Example)

X_____ W _____ ;

Incremental value command for Z axis

Absolute value command for X axis

Precautions

(1) Coordinate values can be omitted, in which case they are treated as "0".
The absolute and incremental value commands can be differentiated for any axis by the G90 and G91 commands.

(Example) When the C axis has been differentiated by G90/G91.
(when incremental addresses have not been set)

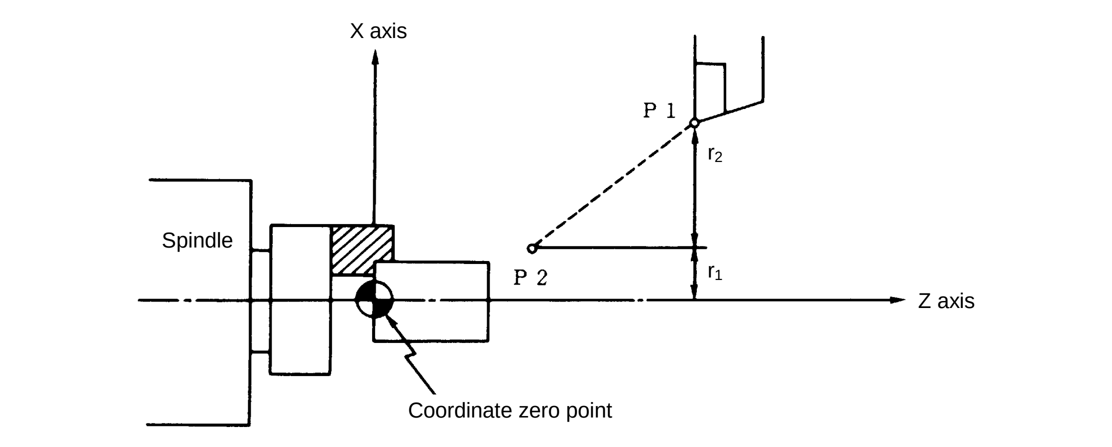
```
G91 X___ W___ C___;
```

- C___: Incremental value command for C axis
- W___: Incremental value command for Z axis
- X___: Absolute value command for X axis

5.2 Radius/Diameter Commands

Function and purpose

The cross sections of workpieces machined on a lathe are circular, and the diameter or radius value of those circles can be used for movement commands in the X-axis direction. A radius command will move the tool by the commanded amount only, but a diameter command will move the tool both in the X-axis direction by an amount equivalent to one-half the command amount only and in the Z-axis direction by the commanded amount only.
This system permits radius or diameter commands to be issued, depending on the parameter setting. The figure below shows the command procedure when the tool is to be moved from point P1 to point P2.



X command		U command		Remarks
Radius	Diameter	Radius	Diameter	Even when a diameter command has been selected, only the U command can be made a radius command by parameter.
$X = r_1$	$X = 2r_1$	$U = r_2$	$U = 2r_2$	

Radius and diameter commands

Precautions

(1) In the above example, the tool moves from P1 to P2 in the minus direction of the X axis and so when an incremental value is issued, the minus sign is given to the numerical value being commanded.

(2) In this manual, diameter commands are used in descriptions of both the X and U axes for the sake of convenience.

5.3 Inch/Metric Conversion (G20, G21)

Function and purpose

These G commands are used to switch between the inch and millimeter (metric) systems.

Command format

G20/G21;	
G20	Inch command
G21	Metric command

Detailed description

The G20 and G21 commands merely select the command units. They do not select the Input units.
G20 and G21 selection is meaningful only for linear axes and it is meaningless for rotation axes.

(Example) Relationship between input command units and G20/G21 commands (with decimal point input type I)

Input command unit "cunit"	Axis type	Command example	"Initial inch" OFF		"Initial inch" ON	
			G21	G20	G21	G20
10	Linear axes	X100;	0.100mm	0.254mm	0.0039 inch	0.0100 inch
	Rotation axes	C100;	0.100°	0.100°	0.100°	0.100°
1	Linear axes	X100;	0.0100mm	0.0254mm	0.00039 inch	0.00100 inch
	Rotation axes	C100;	0.0100°	0.0100°	0.0100°	0.0100°

5.4 Decimal Point Input

Function and purpose

This function enables the decimal point command to be input. It assigns the zero point in millimeter or inch units for the machining program input information that defines the tool paths, distances and speeds.
A parameter selects whether type I (minimum input command unit) or type II (zero point) is to apply for the least significant digit of data without a decimal point.

Command format

○○○○○.○○○	Inch system
○○○○.○○○○	Metric system

Detailed description

(1) The decimal point command is valid for the distances, angles, times and speeds in machining programs.

(2) Refer to the table "Addresses used and valid/invalid decimal point commands" for details on the valid addresses for the decimal point commands.

(3) The number of significant digits in a decimal point command is shown below (for input command unit CS-B).

<table>
<tr><th rowspan="2"></th><th colspan="2">Movement command (linear)</th><th colspan="2">Movement command (rotation)</th><th colspan="2">Feedrate</th><th colspan="2">Dwell (X)</th></tr>
<tr><th>Integer part</th><th>Decimal part</th><th>Integer part</th><th>Decimal part</th><th>Integer part</th><th>Decimal part</th><th>Integer part</th><th>Decimal part</th></tr>
<tr><td rowspan="2">MM (milli-meter)</td><td rowspan="2">0 to 99999.</td><td rowspan="2">.000 to .999</td><td rowspan="2">0 to 99999.</td><td rowspan="2">.000 to .999</td><td>0 to 60000.</td><td>.000 to .999</td><td rowspan="2">0. to 99999.</td><td rowspan="2">.000 to .999</td></tr>
<tr><td>0 to 999.</td><td>.0000 to .9999</td></tr>
<tr><td rowspan="2">INCH (inch)</td><td rowspan="2">0 to 9999.</td><td rowspan="2">.0000 to .9999</td><td rowspan="2">99999. (359.)</td><td rowspan="2">.0 to .999</td><td>0 to 2362.</td><td>.0000 to .9999</td><td rowspan="2">0 to 99.</td><td rowspan="2">.000 to .999</td></tr>
<tr><td>0 to 99.</td><td>.000000 to .999999</td></tr>
</table>

(Note) The top row of Feedrate is for feed per minute and the bottom row is for feed per rotation.

(4) The decimal point command is valid even for commands defining the variable data used in subprograms, etc.

(5) Decimal point commands for decimal point invalid addresses are processed as integer data only and everything below the decimal point is ignored. Addresses which are invalid for the decimal point are D, H, L, M, N, O, P, S and T. All variable commands, however, are treated as data with decimal points.

Precautions

(1) If an arithmetic operator is inserted, the data will be handled as data with a decimal point.

(Example) G00 X123+0; ······ This is the X axis 123mm command. It will not be 123μm.

Example of program

(1) Example of program for decimal point valid address

Specification division / Program example	Decimal point command 1		Decimal point command 2 1 = 1mm
	When 1 = 1μm	When 1 = 10μm	
G0 X123.45 (decimal points are all mm points)	X123.450mm	X123.450mm	X123.450mm
G0 X12345	X12.345mm (last digit is 1μm unit)	X123.450mm	X12345.000mm
#111=123, #112=5.55 X#111 Z#112	X123.000mm, Z5.550mm	X123.000mm, Z5.550mm	X123.000mm, Z5.550mm
#113=#111+#112 (addition)	#113 = 128.550	#113 = 128.550	#113 = 128.550
#114=#111–#112 (subtraction)	#114 = 117.450	#114 = 117.450	#114 = 117.450
#115=#111∗#112 (multiplication)	#115 = 682.650	#115 = 682.650	#115 = 682.650
#116=#111/#112 #117=#112/#111 (division)	#116 = 22.162, #117 = 0.045	#116 = 22.162, #117 = 0.045	#116 = 22.162, #117 = 0.045

Addresses used and valid/invalid decimal point commands

Add-ress	Decimal point com-mand	Application	Re-marks
A	Valid	Coordinate position data	
	Invalid	2nd miscellaneous function code	
	Valid	Angle data	
	Invalid	MRC program number	
	Invalid	Data setting, axis number	
	Valid	Deep hole drilling cycle (2) Safety distance	
B	Valid	Coordinate position data	
	Invalid	2nd miscellaneous function code	
C	Valid	Coordinate position data	
	Invalid	2nd miscellaneous function data	
	Valid	Corner chamfering amount	,C
	Valid	Program tool compensation input Nose R compensation amount (incremental)	
D	Valid	Automatic tool length measurement, deceleration range d	
	Invalid	Data setting, byte type data	
E	Valid	Inch threads Precision thread lead	
F	Valid	Feedrate	
	Valid	Thread lead	
G	Valid	Preparatory function code	
H	Valid	Coordinate position data	
	Invalid	Sequence numbers in subprograms	
	Invalid	Data setting, bit type data	
I	Valid	Circular center coordinates	
	Valid	Nose R compensation/tool radius compensation vector components	
	Valid	Deep hole drilling cycle (2) First cut amount	
J	Valid	Circular center coordinates	
	Valid	Nose R compensation/ nose radius compensation vector components	
	Invalid	Deep hole drilling cycle (2) Dwell at return point	

Add-ress	Decimal point com-mand	Application	Re-marks
K	Valid	Circular center coordinates	
	Valid	Nose R compensation/tool radius compensation vector components	
	Invalid	Hole drilling cycle Number of repetitions	
	Valid	Deep hole drilling cycle (2) Second and subsequent cut amounts	
L	Invalid	Subprogram Number of repetitions	
	Invalid	Program tool compensation input type selection	L2, L10, L11
	Invalid	Data setting selection	L50
	Invalid	Data setting 2-word type data	4 bytes
M	Invalid	Miscellaneous function codes	
N	Invalid	Sequence numbers	
	Invalid	Data setting, data numbers	
O	Invalid	Program numbers	
P	Invalid	Dwell time	
	Invalid	Subprogram call program numbers	
	Invalid	2nd reference point number	
	Invalid	Constant surface speed control, axis number	
	Invalid	MRC finishing shape start sequence number	
	Valid	Cut-off cycle Shift amount/cut amount	
	Invalid	Compound thread cutting cycle, number of cutting passes, chamfering, tool nose angle	
	Valid	Compound thread cutting cycle Thread height	
	Invalid	Program tool compensation input/ Offset number	
	Invalid	Data setting, broad classification number	
	Invalid	Return sequence number from subprogram	
	Valid	Coordinate position data	

(Note 1) Decimal points are all valid in user macro arguments.

Add-ress	Decimal point com-mand	Application	Re-marks
Q	Invalid	Minimum spindle clamp speed	
	Invalid	MRC finishing shape end sequence number	
	Valid	Cut-off cycle Shift amount/cut amount	
	Valid	Compound thread cutting cycle Minimum cut amount	
	Valid	Compound thread cutting cycle First cut amount	
	Valid	Deep hole drilling cycle (1) Cut amount of each pass	
	Invalid	Program tool compensation input Hypothetical tool nose point number	
	Invalid	Deep hole drilling cycle (2) Dwell at cut point	
R	Valid	R-designated arc radius	
	Valid	Corner rounding circular radius	,R
	Valid	Automatic tool length measurement, deceleration range r	
	Valid	MRC longitudinal/face escape amount	
	Invalid	MRC shaping division number	
	Valid	Cut-off cycle, return amount	
	Valid	Cut-off cycle, escape amount	
	Valid	Compound thread cutting cycle, finishing allowance	
	Valid	Compound thread cutting cycle/turning cycle, taper difference	
	Valid	Hole drilling cycle/deep hole drilling cycle (2), distance to reference point	
	Valid	Program tool radius compensation input Nose R compensation amount (absolute)	
	Valid	Coordinate position data	

Add-ress	Decimal point com-mand	Application	Re-marks
S	Invalid	Spindle function codes	
	Invalid	Maximum spindle clamp speed	
	Invalid	Constant surface speed control, surface speed	
	Invalid	Data setting, word type data	2 bytes
T	Invalid	Tool function codes	
U	Valid	Coordinate position data	
	Invalid	2nd miscellaneous function codes	
	Valid	Program tool compensation input	
V	Valid	Coordinate position data	
	Invalid	2nd miscellaneous function codes	
	Valid	Program tool offset input	
W	Valid	Coordinate position data	
	Invalid	2nd miscellaneous function codes	
	Valid	Program tool compensation input	
X	Valid	Coordinate position data	
	Valid	Dwell	
	Invalid	2nd miscellaneous function codes	
	Valid	Program tool compensation input	
Y	Valid	Coordinate position data	
	Invalid	2nd miscellaneous function codes	
	Valid	Program tool compensation input	
Z	Valid	Coordinate position data	
	Invalid	2nd miscellaneous function codes	
	Valid	Program tool compensation input	

6. INTERPOLATION FUNCTIONS

6.1 Positioning (Rapid Traverse); G00

Function and purpose

This command is accompanied by coordinate words. It positions the tool along a linear or non-linear path from the present point as the start point to the end point which is specified by the coordinate words.

Command format

G00 Xx/Uu Zz/Ww;	
x, u, z, w	Coordinate values

The command addresses are valid for all additional axes.

Detailed description

(1) Once this command has been issued, the G00 mode is retained until it is changed by another G function or until the G01, G02, G03 or G33 command in the 01 group is issued. If the next command is G00, all that is required is simply that the coordinate words be specified.

(2) In the G00 mode, the tool is always accelerated at the start point of the block and decelerated at the end point. Execution proceeds to the next block after it has been confirmed that the command pulse of the present block is 0 and that the tracking error of the acceleration/deceleration circuit is 0. The in-position width is set by parameter.

(3) Any G function (G83 to G89) in the 09 group is cancelled (G80) by the G00 command.

(4) Whether the tool moves along a linear or non-linear path is determined by parameter, but the positioning time does not change.

(a) Linear path ········ This is the same as linear interpolation (G01), and the speed is limited by the rapid traverse rate of each axis.

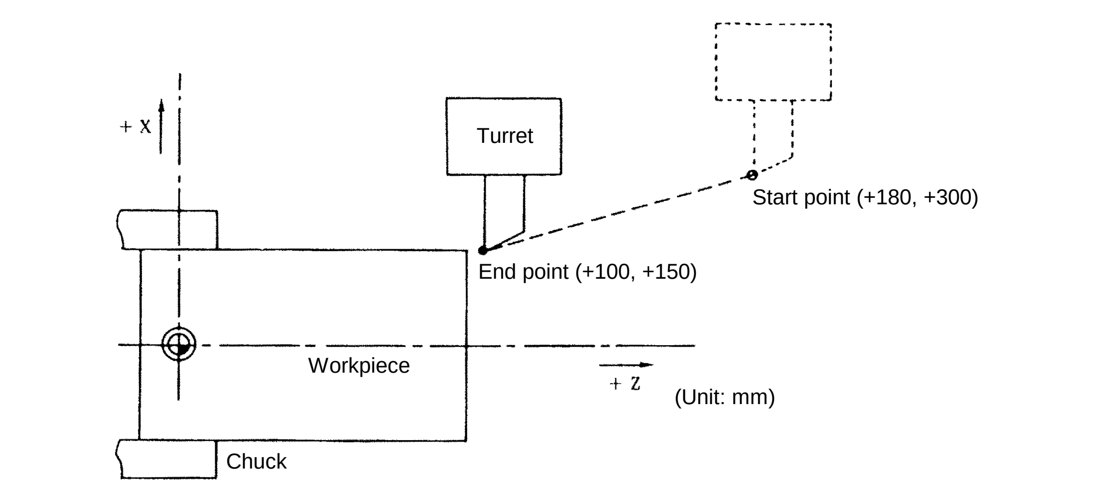
(b) Non-linear path ···· The tool is positioned at the rapid traverse rate independently for each axis.

(5) When no number following the G address, this is treated as G00.

⚠CAUTION

⚠ The commands with "no value after G" will be handled as "G00" during operation.

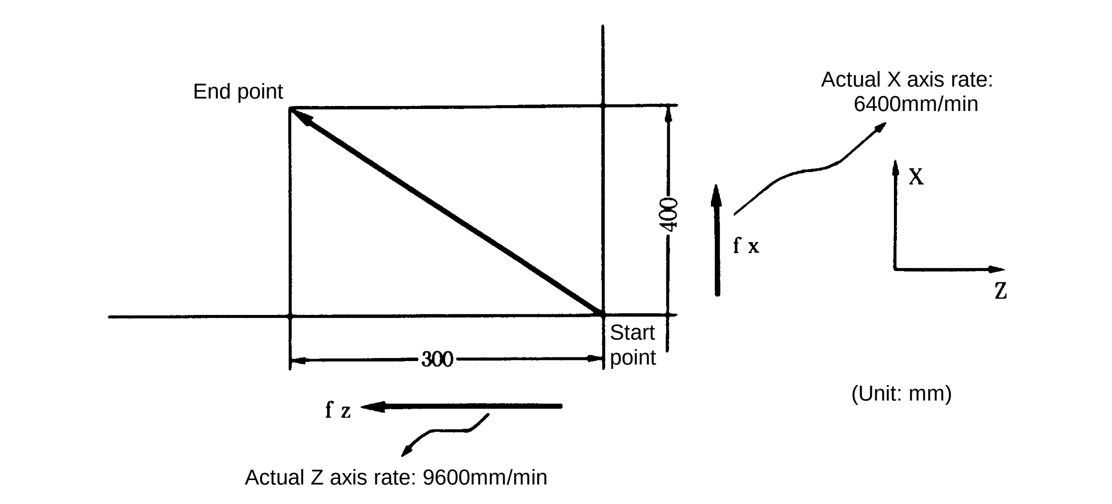
Example of program



G00 X100000 Z150000;	Absolute value command
G00 U-80000 W-150000;	Incremental value command (With an input setting unit of 0.001mm)

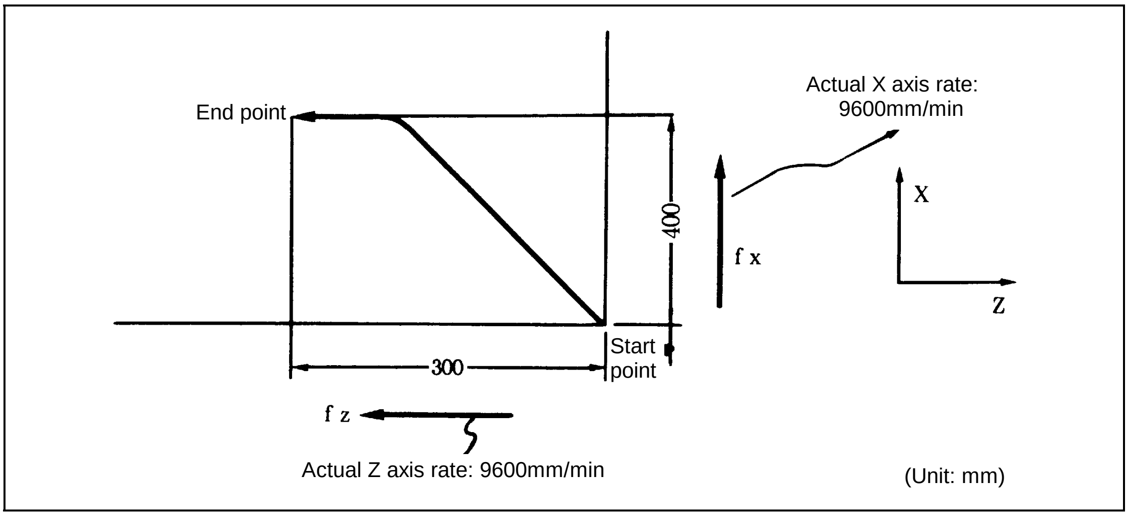
(Note 1) When the "G0 interpolation OFF" user parameter is OFF, the path along which the tool is positioned is the shortest path connecting the start and end points. The positioning speed is automatically calculated so that the shortest distribution time is obtained in order that the commanded speeds for each axis do not exceed the rapid traverse rate.
When, for instance, the X-axis and Z-axis rapid traverse rates are both 9600mm/min, the tool will follow the path in the figure below if the following is programmed:

G00 Z–300000 X400000; (With an input setting unit of 0.001mm)



(Note 2) When the "G0 interpolation OFF" user parameter is ON, the tool will move along the path from the start point to the end point at the rapid traverse rate of each axis.
When, for instance, the X-axis and Z-axis rapid traverse rates are both 9600mm/min, the tool will follow the path in the figure below if the following is programmed:

G00 Z–300000 X400000; (With an input setting unit of 0.001mm)



(Note 3) The rapid traverse rate for each axis with the G00 command differs according to the individual machine and so refer to the instruction manual issued by machine manufacturer.

(Note 4) Rapid traverse (G00) deceleration check
Upon completion of the rapid traverse (G00), execute the next block after the deceleration check time (Td) has elapsed.
The deceleration check time (Td) is as follows, depending on the acceleration/deceleration type.
Linear acceleration/linear deceleration ················ Td = Ts + α
Exponential acceleration/linear deceleration ··········· Td = 2 $\times$ Ts + α
Exponential acceleration/exponential deceleration ······ Td = 2 $\times$ Ts + α
Where Ts is the acceleration/deceleration time constant, α = 0 to 14ms
The time required for the deceleration check during rapid traverse is the longest among the rapid traverse deceleration check times of each axis determined by the rapid traverse acceleration/deceleration time constants and by the rapid traverse acceleration/deceleration mode of the axes commanded simultaneously.

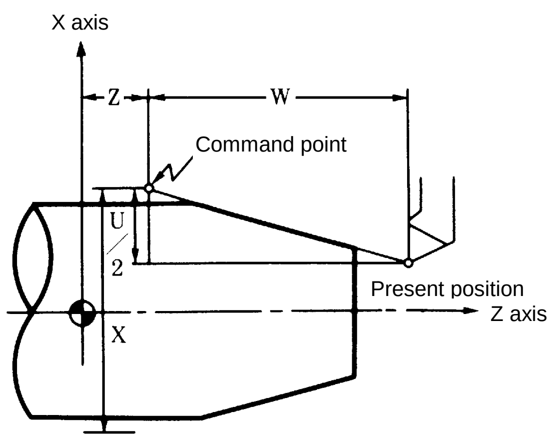
6.2 Linear Interpolation; G01

Function and purpose

This command is accompanied by coordinate words and a feedrate command. It makes the tool move (interpolate) linearly from its present position to the end point specified by the coordinate words at the speed specified by address F. In this case, the feedrate specified by address F always acts as a linear speed in the tool center advance direction.

Command format

G00 Xx/Uu Zz/Ww αα Ff; ("α" is an additional axis)	
x, u, z, w, α	Coordinate values

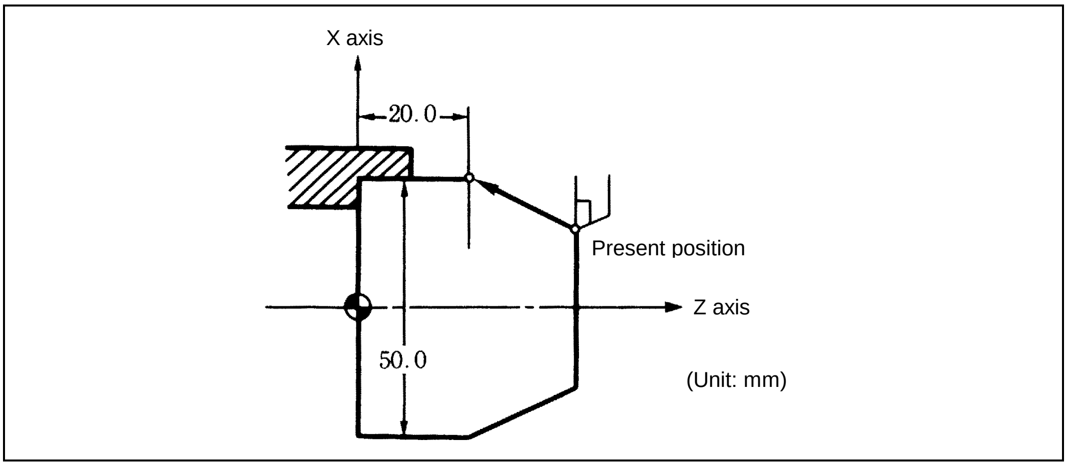


Detailed description

Once this command is issued, the mode is maintained until another G function (G01, G02, G03, G33) in the 01 group which changes the G01 mode is issued. Therefore, if the next command is also G01 and if the feedrate is the same, all that is required to be done is to specify the coordinate words. If no F command is given in the first G01 command block, program error "P62" results.
The feedrate for a rotation axis is commanded by °/min (decimal point position unit). (F300 = 300°/min)
The G functions (G70 to G89) in the 09 group are cancelled (G80) by the G01 command.

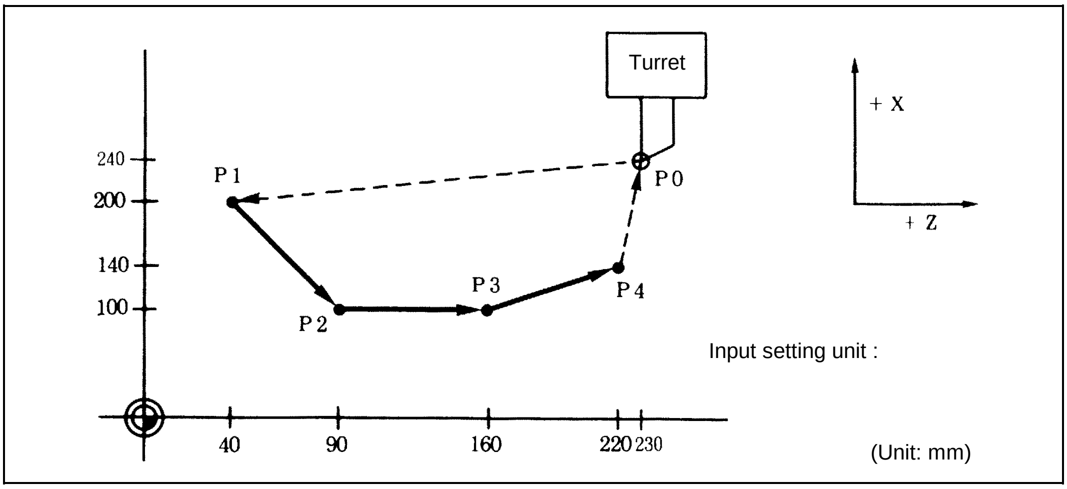
Example of program

(Example 1)



G01 X50.0 Z20.0 F300;	

(Example 2) Cutting in the sequence of $P_1 \rightarrow P_2 \rightarrow P_3 \rightarrow P_4$ at 300mm/min feedrate
$P_0 \rightarrow P_1$, $P_4 \rightarrow P_0$ is for tool positioning



G00 X200000 Z40000;	$P_0 \rightarrow P_1$
G01 X100000 Z90000 F300;	$P_1 \rightarrow P_2$
Z160000;	$P_2 \rightarrow P_3$
X140000 Z220000;	$P_3 \rightarrow P_4$
G00 X240000 Z230000;	$P_4 \rightarrow P_0$

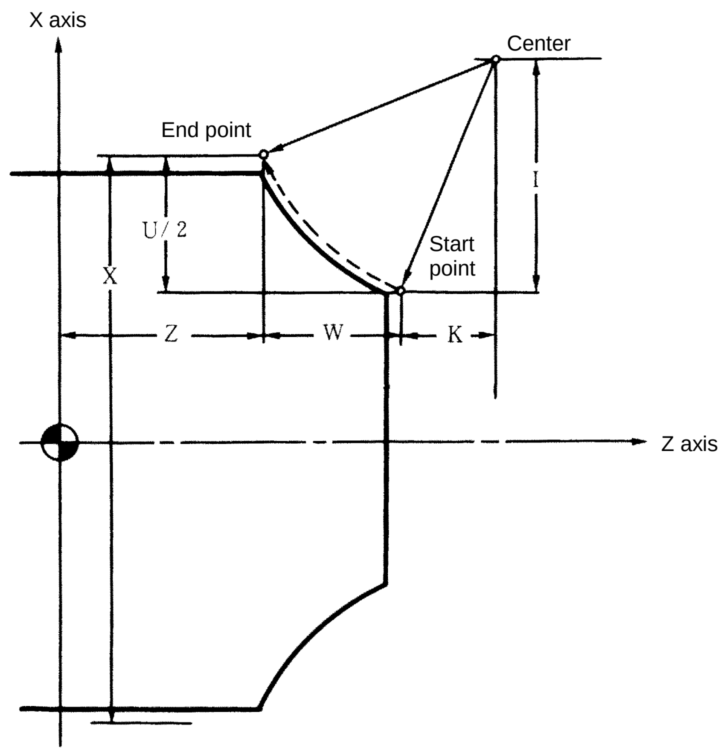
6.3 Circular Interpolation; G02, G03

Function and purpose

These commands serve to move the tool along a circular.

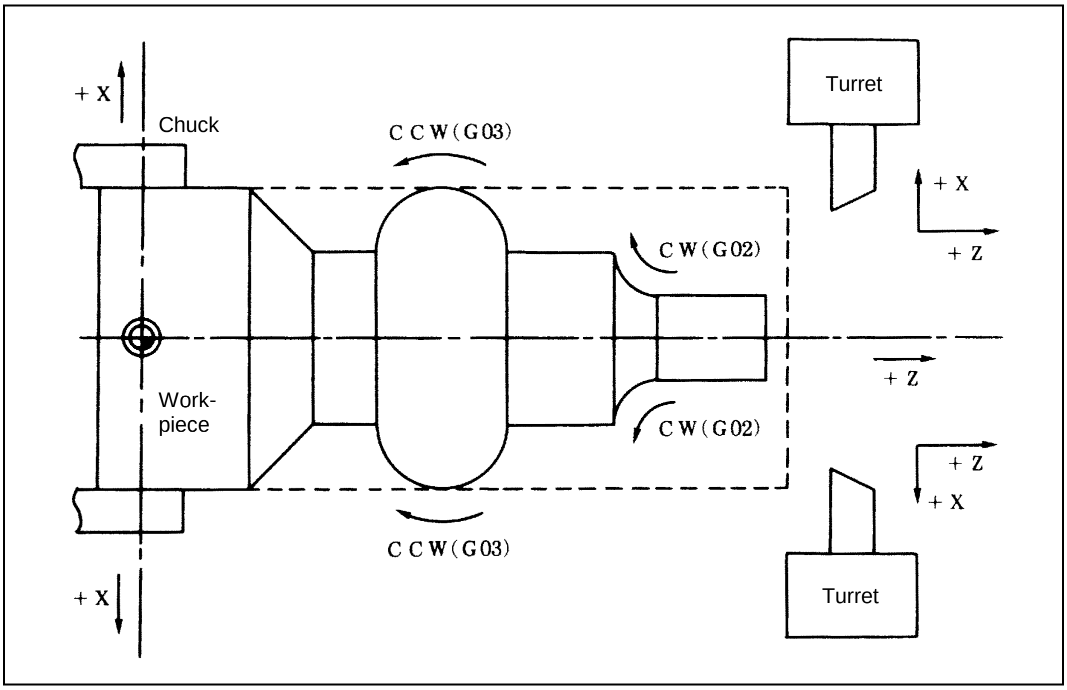
Command format

G02 (G03) Xx/Uu Zz/Ww Ii Kk Ff;	
G02	Clockwise (CW)
G03	Counterclockwise (CCW)
Xx/Uu	Circular end point coordinates, X axis (absolute value of workpiece coordinate system for X, incremental value from present position for U)
Zz/Ww	Circular end point coordinates, Z axis (absolute value of workpiece coordinate system for Z, incremental value from present position for W)
Ii	Circular center, X axis (for I, radius command/incremental value of X coordinate at center as seen from start point)
Kk	Circular center, Z axis (for K, incremental value of Z coordinate at center as seen from start point)
Ff	Feedrate



Detailed description

(1) G02 (or G03) is retained until another G command (G00, G01 or G33) in the 01 group that changes its mode is issued.
The direction of the circular rotation is differentiated by G02 and G03:
G02: CW (Clockwise)
G03: CCW (Counterclockwise)



(2) An arc which extends for more than one quadrant can be executed with a single block command.

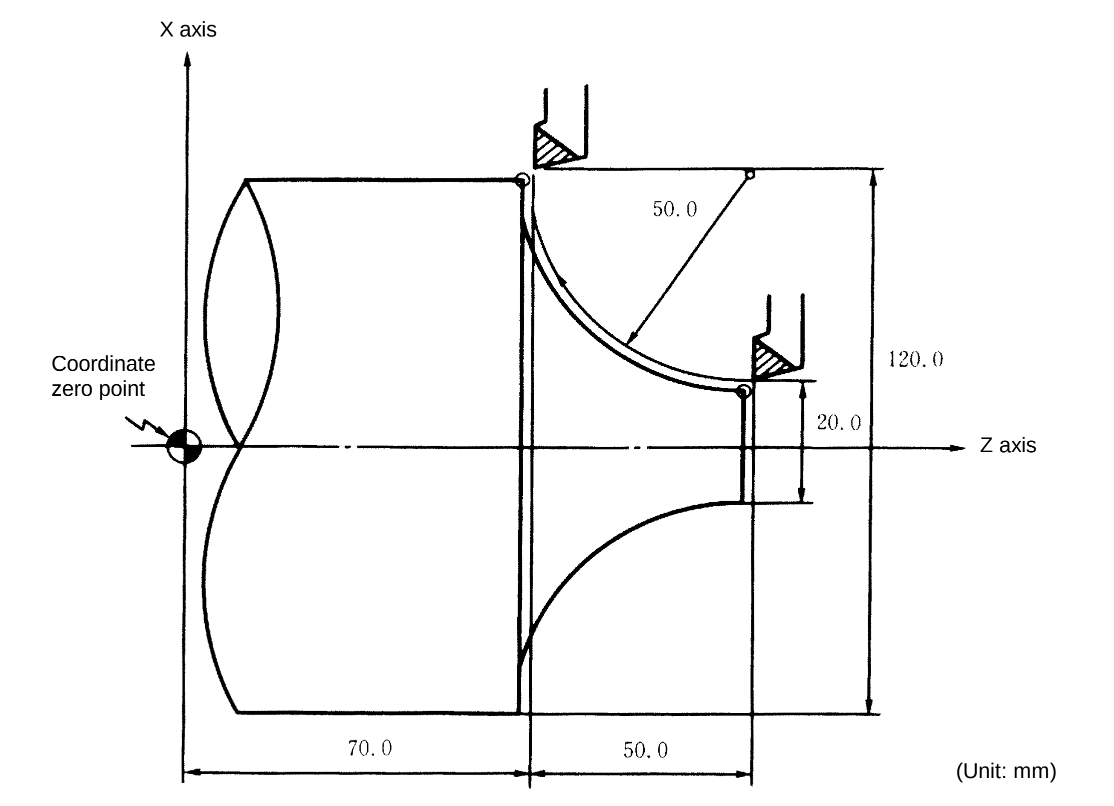
(3) The following information is needed for circular interpolation.

(a)	Rotation direction	: Clockwise (G02) or counterclockwise (G03).
(b)	Circular end point coordinates	: Given by addresses X, Z, U, W.
(c)	Circular center coordinates	: Given by addresses I, K (incremental value commands).
(d)	Feedrate	: Given by address F.

(4) A program error results when I, K or R is not commanded.
Consideration must be given to the sign for I and K since I is the distance in the X-axis direction to the arc center as seen from the start point and K is the distance in the Z-axis direction.

(5) No T commands can be issued in the G2/G3 modal status.
A program error results "P151" if a T command is issued in the G2/G3 modal status.

Example of program



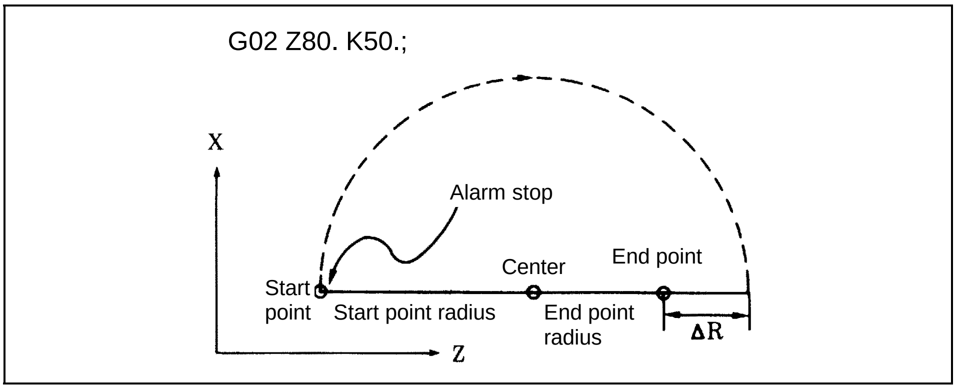
G2 X120.0 Z70.0 I50.0 F200;	Absolute value command
G2 U100.0 W-50.0 I50.0 F200;	Incremental value command

(Note 1) The terms "clockwise" (G02) and "counterclockwise" (G03) used for circular operations are defined as a case where in a right-hand coordinate system, the negative direction is viewed from the positive direction of the coordinate axis which is at right angles to the plane in question.

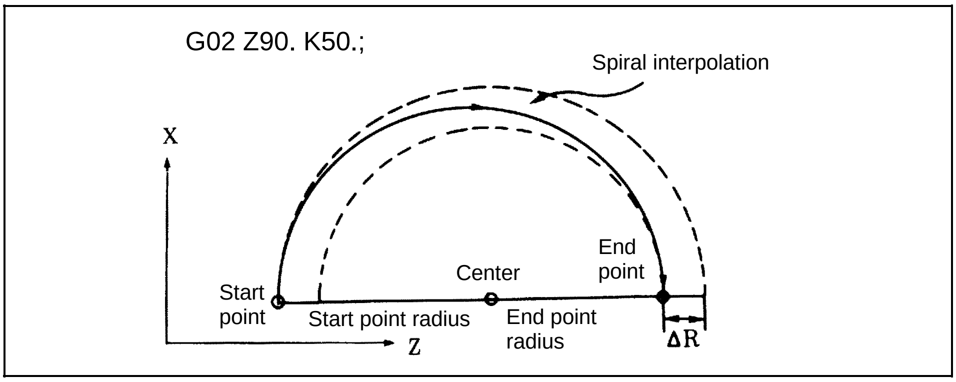
(Note 2) When the end point is the same position as the start point, a 360° circular (full circle) is commanded when the center is commanded using I and K. In this case, always command the end point.

(Note 3) The following occurs when the start and end point radius do not match in a circular command:

(1) Program error "P70" results at the circular start point when error ΔR is greater than the "G02/G03 Error" parameter value.



(2) Spiral interpolation in the direction of the commanded end point results when error ΔR is less than the parameter value.



Although the parameter setting range is from 0 to 100 (input unit), the parameter values in the above examples are assumed to be extremely high in order to facilitate understanding.

6.4 R-designated Circular Interpolation; G02, G03

Function and purpose

Along with the conventional circular interpolation commands based on the circular center coordinate (I, K) designation, these commands can also be issued by directly designating the circular radius R.

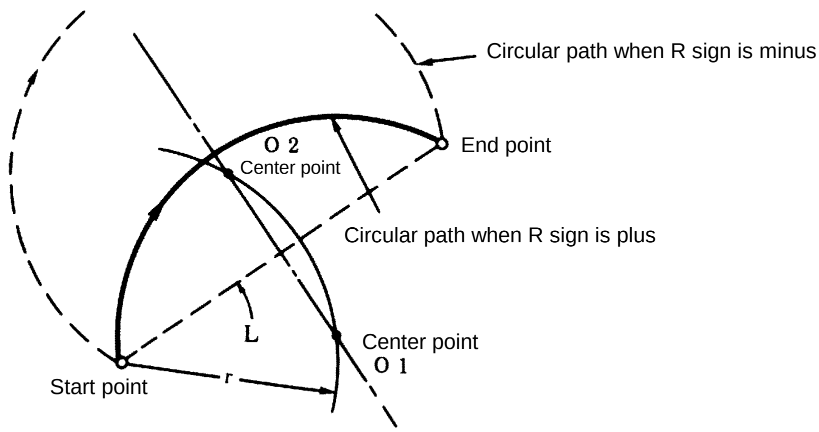
Command format

G02 (G03) Xx/Uu Zz/Ww Rr Ff;

x/u	X-axis end point coordinate
z/w	Z-axis end point coordinate
r	Circular radius
f	Feedrate

Detailed description

The circular center is on the bisector line which is perpendicular to the line connecting the start and end points of the circular. The point, where the circular with the specified radius whose start point is the center intersects the perpendicular bisector line, serves as the center coordinates of the circular command.
If the R sign of the commanded program is plus, the circular command is smaller than a semicircle, if it is minus, the circular command is larger than a semicircle.



The following condition must be met with an R-designated circular interpolation command:

$$\frac{L}{2 \times r} \leq 1$$

Where L is the line from the start point to end point.
If the above conditions are not satisfied and the arc center cannot be calculated, a program error (P71) will occur.
If an R designation and I, K designation are given at the same time in the same block, the circular command with the R designation takes precedence.
In the case of a full-circle command (where the start and end points match), an R designation circular command will be completed immediately if it is issued and no operation will result. An I, K designation circular command should therefore be used in such a case.

Example of program

(Example 1)

G03 Zz1 Xx1 Rr1 Ff1;	R-designated arc on Z-X plane

(Example 2)

G02 Xx1 Zz1 Ii1 Kk1 Rr1 Ff1;	R-designated arc on X-Z plane (When the R designation and I, K designation are contained in the same block, the R designation has priority in processing.)

6.5 Plane Selection; G17, G18, G19

Function and purpose

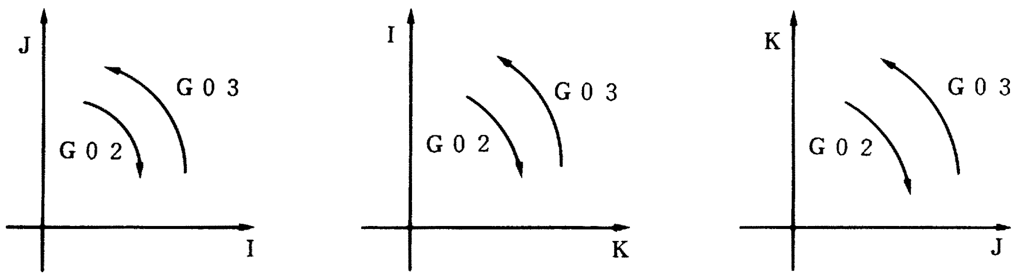
These commands are used to select the control plane and the plane on which the circular exists.
If the 3 basic axes and the parallel axes corresponding to these basic axes are entered as parameters, the commands can select the plane composed of any 2 axes which are not parallel axes. If a rotation axis is entered as a parallel axis, the commands can select the plane containing the rotation axis.
These commands are used to select:

- The plane for circular interpolation
- The plane for nose R compensation
- The plane for milling interpolation

Command format

G17;	**(I-J plane selection)**
G18;	**(K-I plane selection)**
G19;	**(J-K plane selection)**

I, J and K indicate each basic axis or parallel axis.
When the power is turned ON or when the system is reset, the plane set by the control parameters (Initial Z-X plane, Initial Y-Z plane) is selected. If neither of these has been set, the X-Y plane is selected.



G17 (I-J) plane **G18 (K-I) plane** **G19 (J-K) plane**

Parameter entry

	Basic axis	Parallel axis
I	X	Y
J	Y	
K	Z	

Fig. 1 Examples of plane selection parameter entry

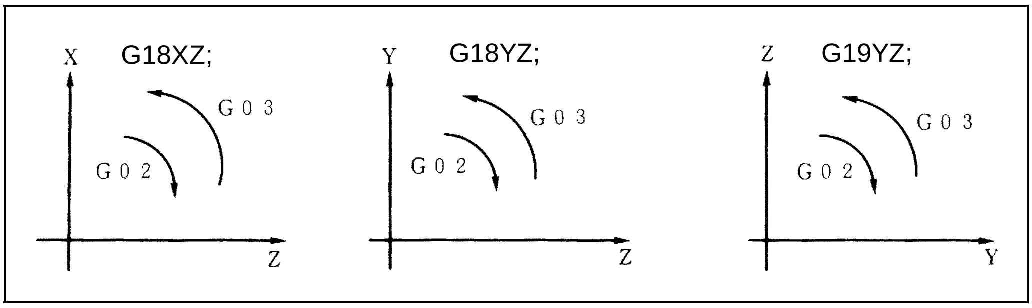
Basic axes and parallel axes can be entered in the parameters. The same axis name can be entered in duplication, but when it is assigned in duplication, the plane is determined by plane selection system (4).
It is not possible to set axes, which have not been entered, as control axes.

Plane selection system

This section describes the plane selection for the parameter entry samples shown in Fig. 1.

(1) Axis addresses assigned in the same block as the plane selection (G17, G18, G19) command determine which of the basic axes or parallel axes are to be in the actual plane selected.

(Example)



(2) Plane selection is not performed with blocks in which the plane selection G code (G17, G18, G19) is not assigned.

G18X_Z_; Z-X plane
Y_Z_; Z-X plane (no plane change)

(3) When the axis addresses are omitted in the block containing the plane selection G codes (G17, G18, G19), it is assumed that the axis addresses of the 3 basic axes have been assigned.

G18; (Z-X plane = G18XZ;)

(4) When the basic axes or their parallel axes are duplicated and assigned in the same block as the plane selection G code (G17, G18, G19), the plane is determined in the order of basic axes, and then parallel axes.

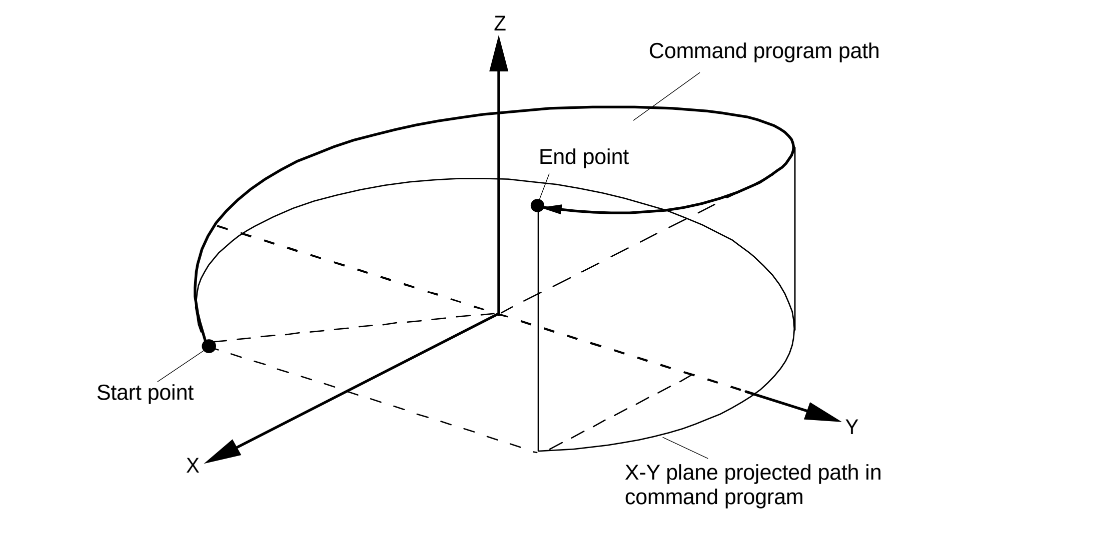
G18XYZ; The Z-X plane is selected.
Therefore, the Y movement is unrelated to the selected plane.

(Note 1) When the "#8121 Initial Z-X plane" in the control parameter is kept ON, the G18 plane is selected when the power is turned ON or when the system is reset.

6.6 Helical interpolation; G17, G18, G19, and G02, G03

Function and purpose

With three orthogonal axes, large diameter screws and solid cams can be machined with simultaneous 3-axis control. With this control, circular interpolation is carried out with two random axes, and at the same time, linear interpolation is carried out with the other axis in synchronization with the circular interpolation.



Command format

G17 G02 (G03) Xx/Uu Yy/Vv Zz/Ww Ii Jj Pp Ff;

G17	Arc plane (G17: X-Y plane, G18: Z-X plane, G19: Y-Z plane) **(Note 2)**
G02 (G03)	Arc rotation direction (G02: clockwise, G03: counterclockwise) **(Note 3)**
Xx/Uu, Yy/Vv	Arc end point coordinates
Zz/Ww	Linear axis end point
Ii, Jj	Arc center coordinates
Pp	Number of pitches
Ff	Feedrate

(Note 1) In this manual, the following setting descriptions are used. I axis: X, J axis: Y, K axis: Z

(Note 2) The linear interpolation axis is the other axis not included in the plane selection.

	Arc plane	Linear interpolation axis
G17	X-Y plane	Z axis
G18	Z-X plane	Y axis
G19	Y-Z plane	X axis

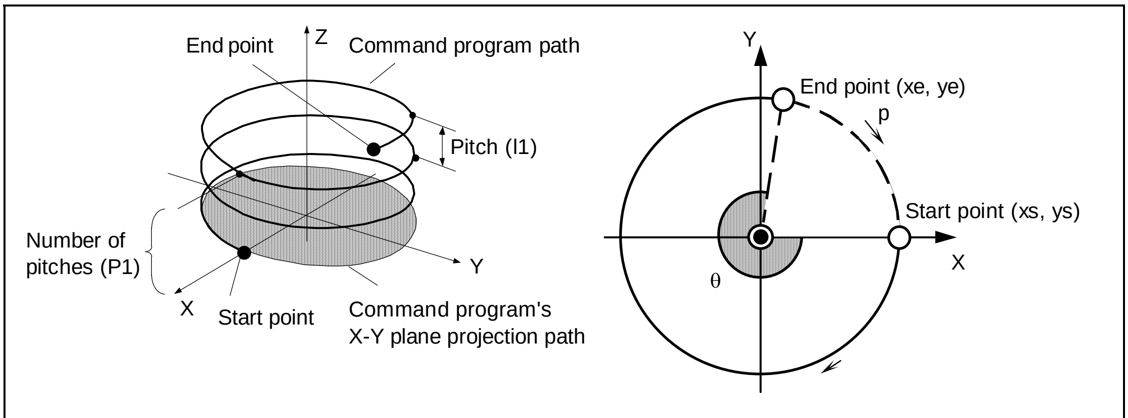
(Note 3) The rotation direction is that looking at the arc plane in the linear interpolation axis' positive to negative direction.

Explanation of addresses

Add-ress	Address meaning	Command range (unit)	Remarks
X/U Y/V	Arc end point coordinates	Coordinate command range (mm/inch) (Decimal point command valid)	1. If a value that exceeds the command range is commanded, a program error will occur. (P35 Setting value range over) 2. If an axis other than one which can be controlled with the command system is commanded, a program error will occur. (P33 Format error).
Z/W	Linear axis end point coordinates	Coordinate command range (mm/inch) (Decimal point command valid)	1. If a value that exceeds the command range is commanded, a program error will occur. (P35 Setting value range over) 2. If an axis other than one which can be controlled with the command system is commanded, a program error will occur. (P33 Format error).
I J	Arc center coordinates	Coordinate command range (mm/inch) (Decimal point command valid)	1. If a value that exceeds the command range is commanded, a program error will occur. (P35 Setting value range over) 2. If an axis other than one which can be controlled with the command system is commanded, a program error will occur. (P33 Format error). 3. Input a radius value. 4. An arc radius value command (R command) cannot be used for the arc center coordinates.
P	Number of pitches (*)	0 to 99	1. If a value that exceeds the command range is commanded, a program error will occur. (P35 Setting value range over) 2. When omitted, this will be handled as P0.
F	Feedrate	Speed command range (mm/inch, inch/min) (Decimal point command valid)	1. Command the speed in the direction of each axis' composite element as the feedrate. 2. If a value that exceeds the command range is commanded, a program error will occur. (P35 Setting value range over)

(**Example:** X-Y plane)

*** Calculating the pitch (l1)**



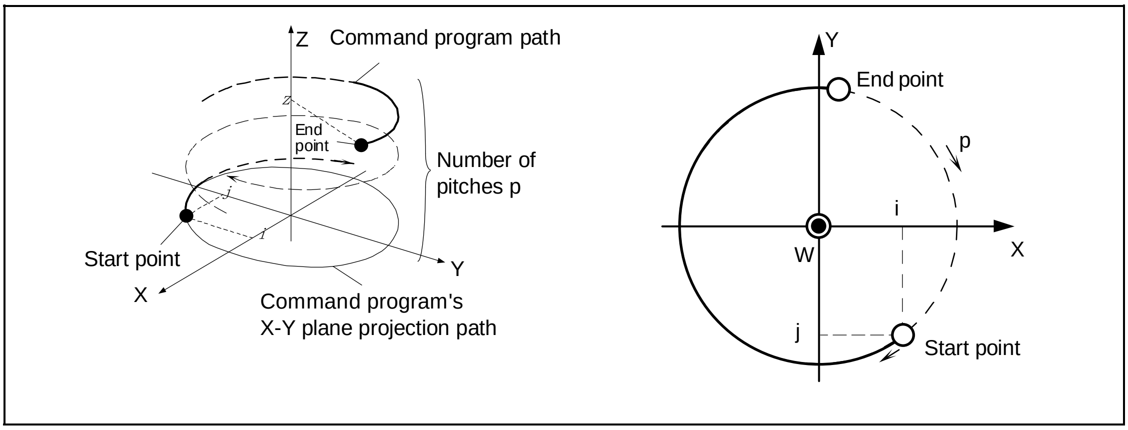
$$l1 = z1/((2\pi \cdot p1 + \theta)/2\pi)$$
$$\theta = \theta e - \theta s = \arctan(ye/xe) - \arctan(ys/xs) \qquad (0 \leq \theta < 2\pi)$$

Whereas, ys and xs indicate the start point coordinates, ye and xe indicate the end point coordinates, z1 indicates the linear axis end point coordinates, and p1 indicates the number of pitches. θe and θs each indicate the phase difference from the arc center coordinates.

Detailed description

The following type of movement will take place when the following type of command is issued.

G17 G02 Xx Yy Zz Ii Jj Pp Ff;



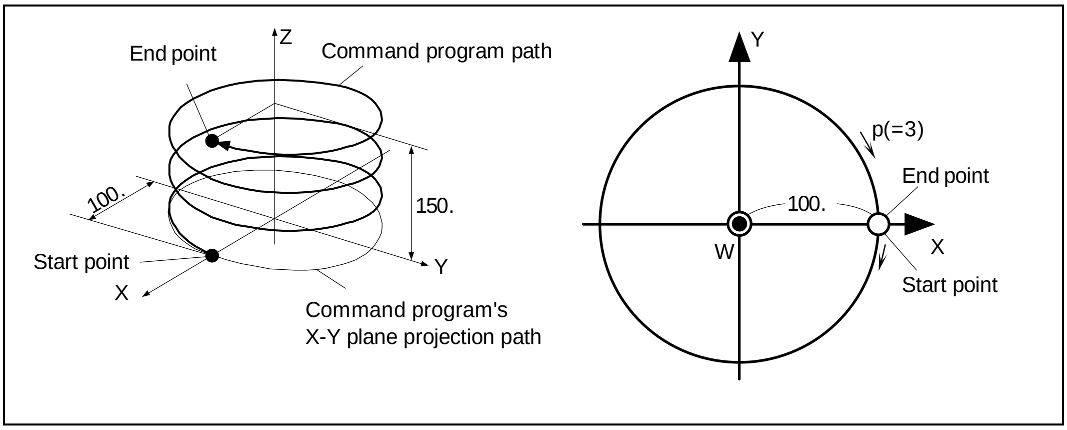
The left drawing shows the process as an exploded view, and the right drawing shows the arc plane from directly above.

Using the point designated as the arc center coordinates as the center, the axis starts to rotate at the feedrate f, and after uniformly cutting from the start point to the end point with the number of pitches designated with p, the axis stops.

Example of program

(Example 1)

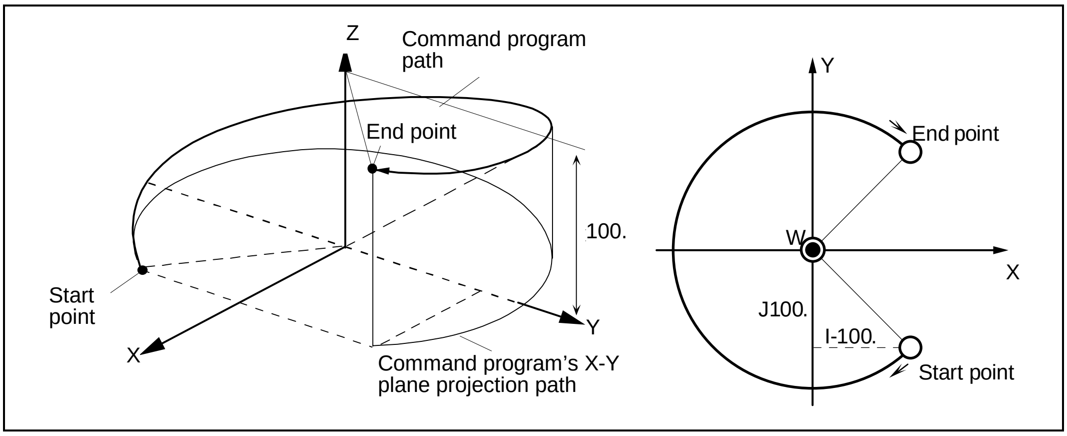
G17 G02 U0. V0. W150. I-100. J0. P3 F300;	Normal command



The left drawing shows the process as an exploded view, and the right drawing shows the arc plane from directly above.
At the start of the block, the axis centers at the point -100mm in the X axis direction and 0mm in the Y axis direction from the workpiece coordinates (start point), and starts cutting at the feedrate 300mm/min while rotating.
While carrying out circular interpolation, the axis uniformly cuts three-step pitches while moving 100mm in the Z axis direction, and then stops.

(Example 2)

G17 G02 U-50. V200. W100. I-100. J100. F1500;	When P command is omitted



The left drawing shows the process as an exploded view, and the right drawing shows the arc plane from directly above.
At the start of the block, the axis centers at the point -100mm in the X axis direction and 100mm in the Y axis direction from the workpiece coordinates (start point), and starts cutting at the feedrate 1500mm/min while rotating.
Since the P command is omitted, the number of pitches is 0. Thus, the axis will cut 100mm in the Z axis direction while carrying out circular interpolation, and then stops.

Precautions and restrictions

(1) When the P designation is omitted, the setting will be the same as P0.

(2) When the arc center coordinates are omitted, "P70 Arc radius error" will occur.

(3) Circular machining is possible by omitting the arc end point coordinates.

(4) If P0 is commanded when the start point and end point coordinates are the same, linear interpolation will take place only in the direction of the linear interpolation axis. If P0 is commanded when the start point and end point coordinates differ, helical interpolation will take place from the start point to end point in one cycle.

(5) A circular interpolation radius command (R command) cannot be issued when carrying out helical interpolation.

(6) If the linear interpolation axis is omitted, the movement will be the same as normal circular interpolation. However, the axis will stop at the end point after carrying out circular interpolation the number of times commanded with P.

(7) If a height axis is set with the same axis designating an arc plane, the latter setting will become valid and normal circular interpolation will take place. However, the circle will be traced only the number of times designated with the P command.

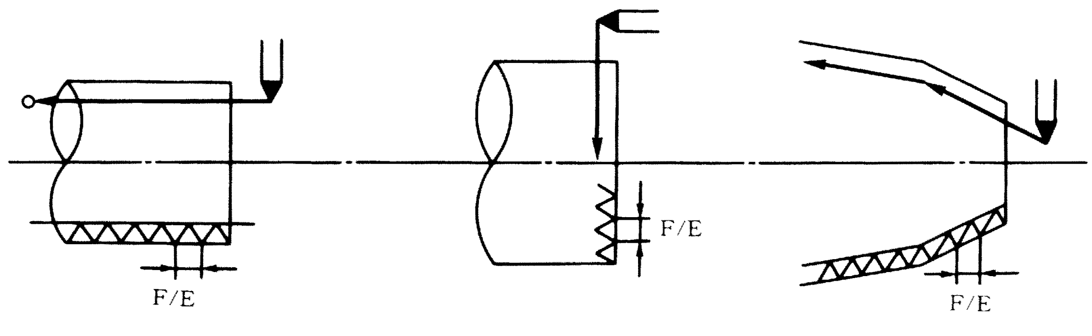
(8) The feedrate is the speed in the direction of each axis' composite element.

6.7 Thread Cutting

6.7.1 Constant lead thread cutting; G33

Function and purpose

The G33 command exercises feed control over the tool which is synchronized with the spindle rotation and so this makes it possible to conduct constant-lead straight thread cutting, tapered thread cutting, and continuous thread cutting.

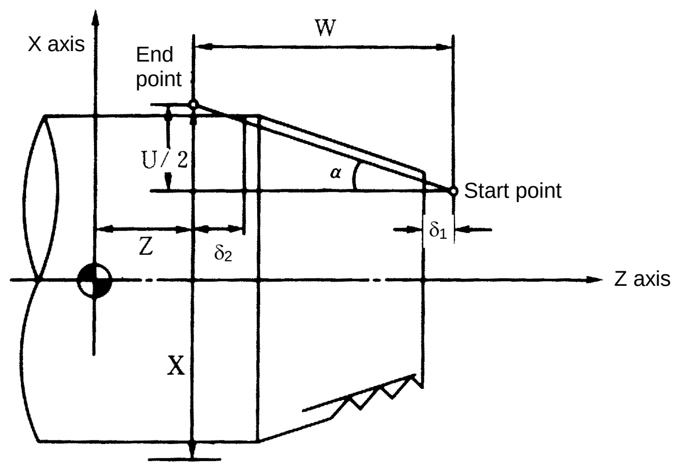


Straight thread **Scrolled thread** **Continuous thread**

Command format

G33 Zz/Ww Xx/Uu Ff Qq ;	(Normal lead thread cutting commands)
Zz, Ww, Xx, Uu	Thread end point addresses and coordinates
Ff	Lead of long axis (axis which moves most) direction
Qq	Thread cutting start shift angle (0.001 to 360.000°)

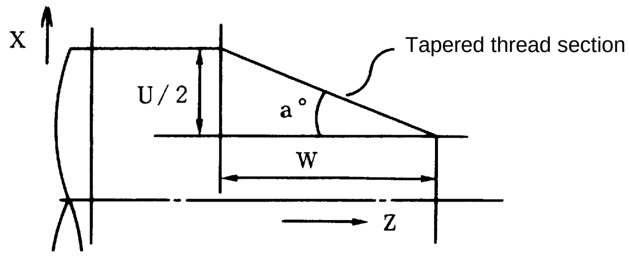
G33 Zz/Ww Xx/Uu Ee Qq ;	(Precision lead thread cutting commands)
Zz, Ww, Xx, Uu	Thread end point addresses and coordinates
Ee	Lead of long axis (axis which moves most) direction
Qq	Thread cutting start shift angle (0.001 to 360.000°)



Detailed description

(1) The E command is also used for the number of threads in inch thread cutting, and whether the threads or precision lead is to be designated can be selected by parameter setting. (The parameter "#8114 Precision thrd cut E" is set to ON for precision lead designation.)

(2) The lead in the long axis direction is commanded for the tapered thread lead.



When a < 45° lead is in Z-axis direction
When a > 45° lead is in X-axis direction
When a = 45° lead can be in either Z or X-axis direction.

Thread cutting metric input

Input unit system	**A (0.01mm)**			**B (0.001mm)**			**C (0.0001mm)**		
Command address	F (mm/rev)	E (mm/rev)	E (threads/ inch)	F (mm/rev)	E (mm/rev)	E (threads/ inch)	F (mm/rev)	E (mm/rev)	E (threads/ inch)
Minimum command unit	1 (= 0.001), (1.=1.000)	1 (= 0.0001), (1.=1.0000)	1 (= 1), (1.=1.0)	1 (= 0.0001), (1.=1.0000)	1 (= 0.00001), (1.=1.00000)	1 (= 1), (1.=1.00)	1 (= 0.00001), (1.=1.00000)	1(=0.000001), (1.=1.000000)	1 (= 1), (1.=1.000)
Command range	0.001 to 9999.999	0.0001 to 9999.9999	0.1 to 9999999.9	0.0001 to 999.9999	0.00001 to 999.99999	0.01 to 999999.99	0.00001 to 99.99999	0.000001 to 99.999999	0.001 to 99999.999

Thread cutting inch input

Input unit system	**A (0.001inch)**			**B (0.0001inch)**			**C (0.00001inch)**		
Command address	F (inch/rev)	E (inch/rev)	E (threads/ inch)	F (inch/rev)	E (inch/rev)	E (threads/ inch)	F (inch/rev)	E (inch/rev)	E (threads/ inch)
Minimum command unit	1(=0.00001), (1.=1.00000)	1(=0.000001) (1.=1.000000)	1 (= 1), (1.=1.000)	1(=0.000001), (1.=1.000000)	1(=0.0000001), (1.=1.0000000)	1 (= 1), (1.=1.0000)	1(=0.0000001), (1.=1.0000000)	1(=0.00000001), (1.=1.00000000)	1(=1), (1.=1.00000)
Command range	0.00001 to 999.99999	0.000001 to 99.999999	0.001 to 99999.999	0.000001 to 99.999999	0.0000001 to 9.9999999	0.0001 to 9999.9999	0.0000001 to 9.9999999	0.00000001 to 0.99999999	0.001 to999.99999

(Note 1) It is not possible to assign a lead where the feedrate as converted into feed per minute exceeds the maximum cutting feedrate.

(3) The constant surface speed control function should not be used for tapered thread cutting commands or scrolled thread cutting commands.

(4) The spindle speed should be kept constant throughout from the rough cutting until the finishing.

(5) If the feed hold function is employed during thread cutting to stop the feed, the thread ridges will lose their shape. For this reason, feed hold does not function during thread cutting.
If the feed hold switch is pressed during thread cutting, block stop will result at the end point of the block following the block in which thread cutting is completed (no longer G33 mode).

(6) The converted cutting feedrate is compared with the cutting feed clamp speed when thread cutting starts, and if it is found to exceed the clamp speed, an operation error will result. **(Note 1)**

(7) In order to protect the lead during thread cutting, a cutting feedrate which has been converted may sometimes exceed the cutting feed clamp speed.

(8) An illegal lead is normally produced at the start of the thread cutting and at the end of the cutting because of servo system delay and other such factors.
Therefore, it is necessary to command a thread length which is determined by adding the illegal lead lengths δ_1 and δ_2 to the required thread length.

(9) The spindle speed is subject to the following restriction:

$$1 \leq R \leq \frac{\text{Maximum feedrate}}{\text{Thread lead}}$$

Where R ≤ Permissible speed of encoder (r/min)
R = Spindle speed (r/min)
Thread lead = mm or inches
Maximum feedrate = mm/min or inch/mm
(this is subject to the restrictions imposed by the machine specifications).

(10) Dry run is valid for thread cutting but the feedrate based on dry run is not synchronized with the spindle rotation.
The dry run signal is checked at the start of thread cutting and any switching during thread cutting is ignored.

(11) Synchronous feed applies for the thread cutting commands even with an asynchronous feed command (G94).

(12) For the spindle override during thread cutting, select either valid or invalid (100% fixed) with the parameters. When the override is valid, the thread will not be cut correctly because of a delay in the servo system if the override is changed during thread cutting. When the override is invalid, the override will change to 100% when operation with an override other than 100% is started. Thus, the thread will not be cut correctly due to a delay in the servo system.

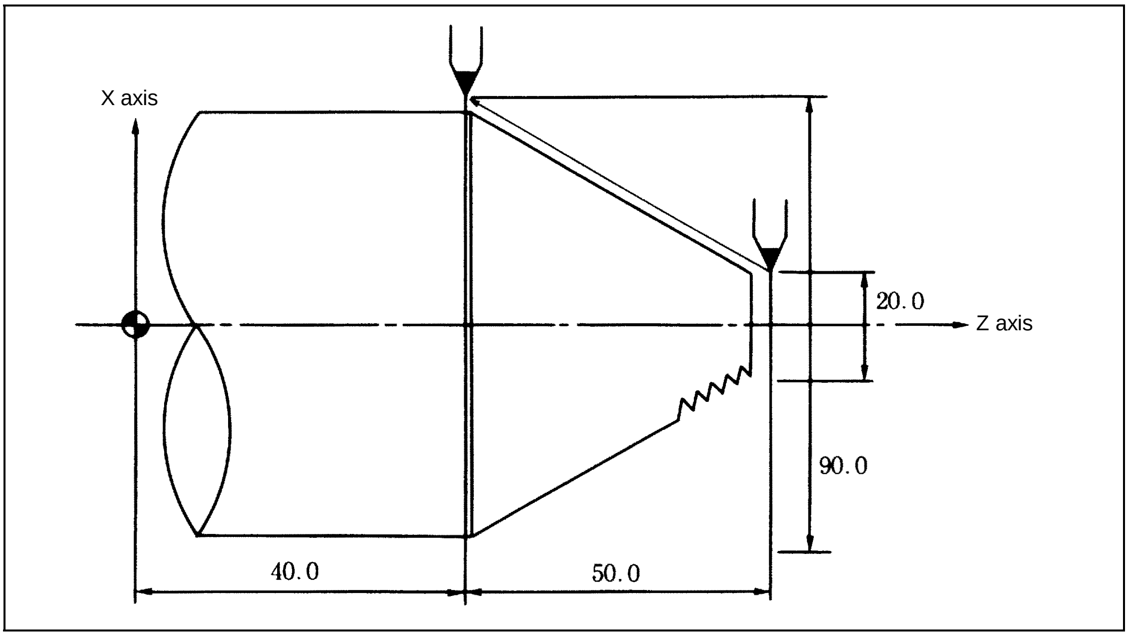
(13) When a thread cutting command is programmed during nose R compensation, the compensation is temporarily canceled and the thread cutting is executed.

(14) When the mode is switched to another automatic mode while G33 is executed, the following block which does not contain a thread cutting command is first executed and then the automatic operation stops.

(15) When the model is switched to the manual mode while G33 is executed, the following block which does not contain a thread cutting command is first executed and then the automatic operation stops. In the case of a single block, the following block which does not contain a thread cutting command is first executed and then the automatic operation stops.

(16) The thread cutting command waits for the single rotation synchronous signal of the rotation encoder and starts movement.
However, movement starts without waiting for this signal when another system issues a thread cutting command during ongoing thread cutting by one particular system. Therefore, thread cutting commands should not be issued by a multiple number of systems.

Example of program



G33 X90.0 Z40.0 E12.34567;	Absolute value command
G33 U70.0 W-50.0 E12.34567;	Incremental value command

6.7.2 Inch thread cutting; G33

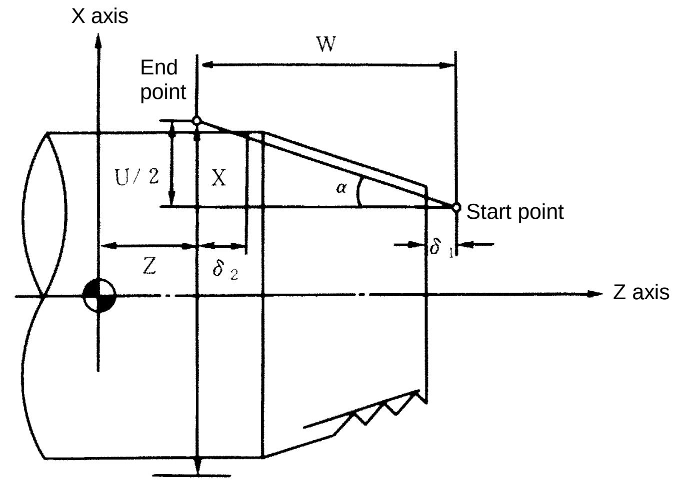
Function and purpose

If the number of threads per inch in the long axis direction is assigned in the G33 command, the feed of the tool synchronized with the spindle speed will be controlled, which means that constant-lead straight thread cutting, tapered thread cutting, and continuous thread cutting can be performed.

Command format

G33 Zz/Ww Xx/Uu Ee Qq;

Zz, Ww, Xx, Uu	Thread end point addresses and coordinates
Ee	Number of threads per inch in direction of long axis (axis which moves most) (decimal point command can also be assigned)
Qq	Thread cutting start shift angle (0.001 to 360.000°)



Detailed description

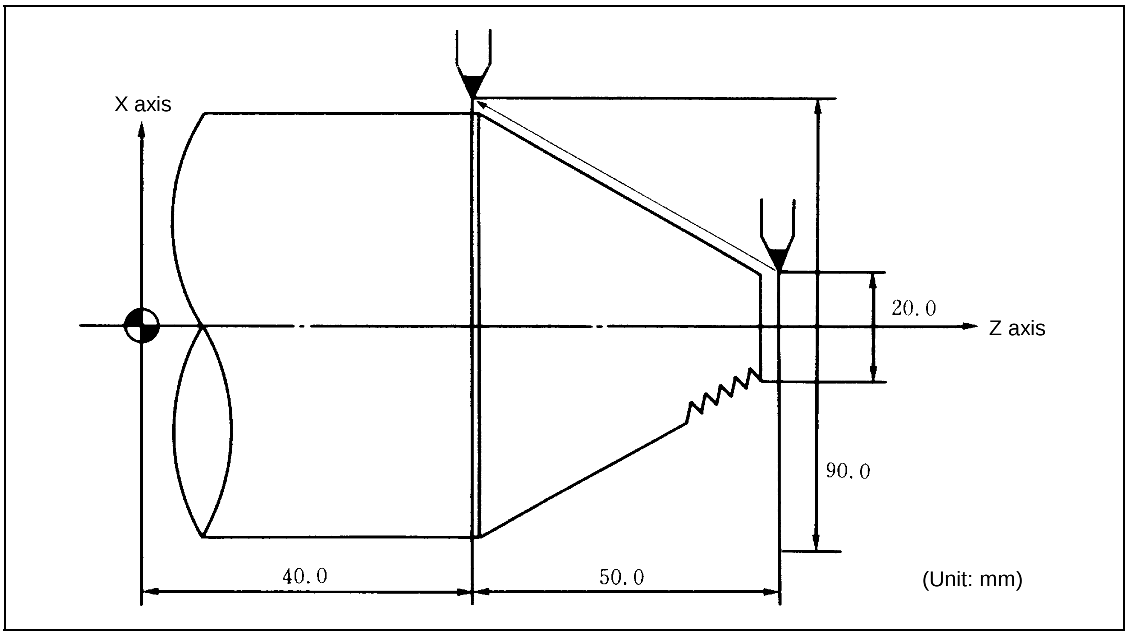
(1) The number of threads in the long axis direction is assigned as the number of threads per inch.

(2) The E code is also used to assign the precision lead length, and whether the thread number or precision lead length is to be designated can be selected by parameter setting. (The parameter "G33 Precision thrd cut E" is set OFF for thread number designation.)

(3) The E command value should be set within the lead value range when the lead is converted.

(4) See Section 6.7.1 on lead thread cutting for other details.

Example of program

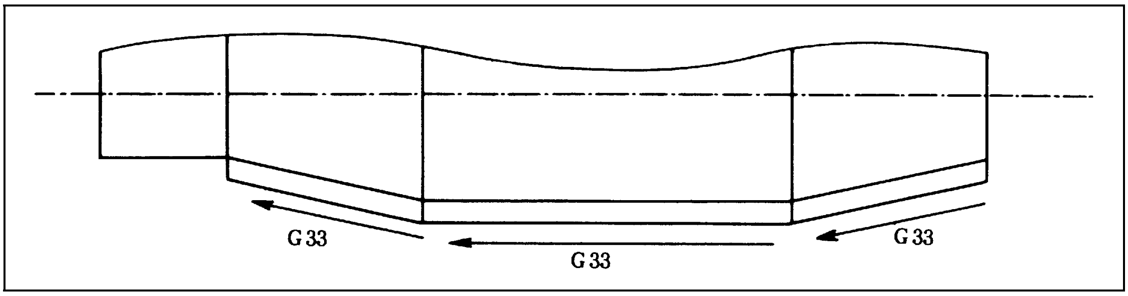


G33 X90.0 Z40.0 E12.0;	Absolute value command
G33 U70.0 W-50.0 E12.0;	Incremental value command

6.7.3 Continuous thread cutting

Function and purpose

Continuous thread cutting is possible by assigning thread cutting commands continuously.
In this way, it is possible to cut special threads whose lead or shape changes.



6.7.4 Variable lead thread cutting

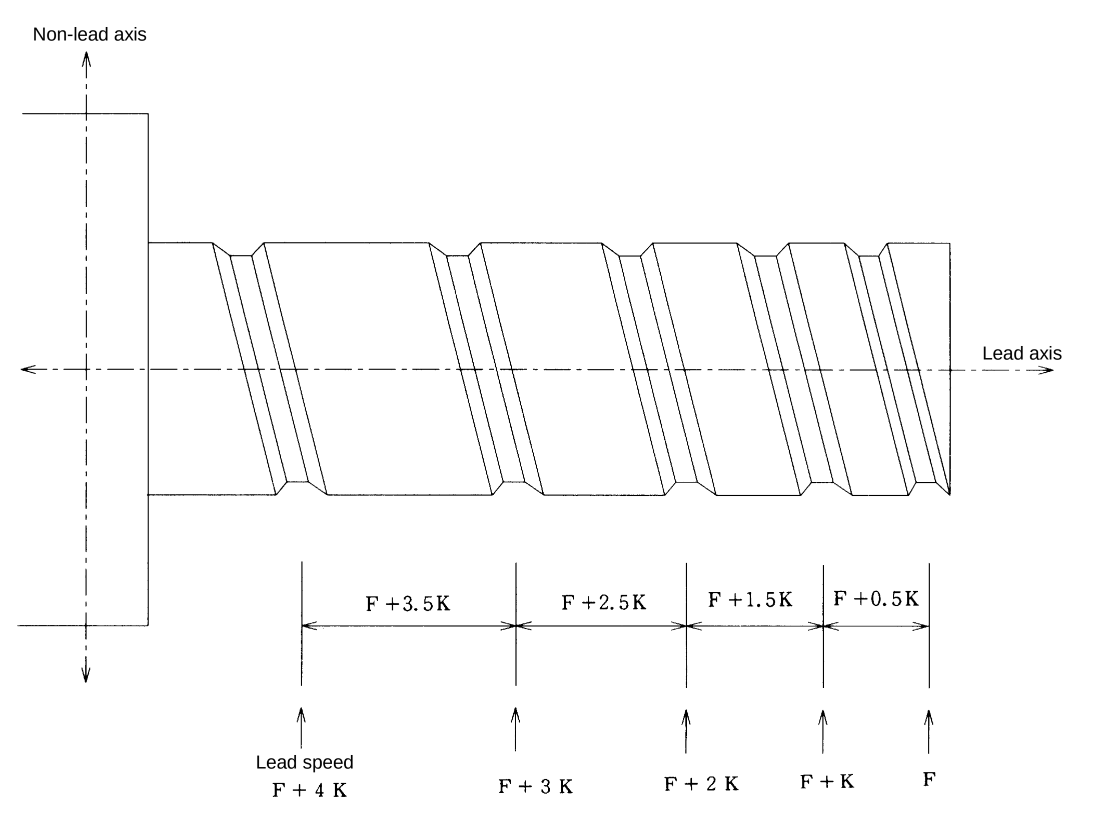
Function and purpose

Variable lead thread cutting is enabled by a command specifying a lead increment or decrement amount per turn of the screw.

Command format

G34 X/U____ Z/W____ F/E____ K____ ;

X/U	X coordinate at the end of thread cutting
Z/W	Z coordinate at the end of thread cutting
F/E	Standard thread lead
K	Lead increment or decrement amount per turn of the screw



Detailed description

(1) The command range is as shown below.

Thread cutting metric input

Input unit system	**A (0.01mm)**		**B (0.001mm)**		**C (0.0001mm)**		**A/B/C**
Command address	F (mm/rev)	E (mm/rev)	F (mm/rev)	E (mm/rev)	F (mm/rev)	E (mm/rev)	K (n ∗ mm/rev) n: Number of pitches Same as F or E (signed)
Minimum command unit	1 (=0.001) (1.=1.000)	1 (=0.0001) (1.=1.0000)	1 (=0.0001) (1.=1.0000)	1 (=0.00001) (1.=1.00000)	1 (=0.00001) (1.=1.00000)	1 (=0.000001) (1.=1.000000)	
Command range	0.001 to 9999.999	0.0001 to 9999.9999	0.0001 to 999.9999	0.00001 to 999.99999	0.00001 to 99.99999	0.000001 to 99.999999	

Thread cutting inch input

Input unit system	**A (0.001inch)**		**B (0.0001inch)**		**C (0.00001inch)**		**A/B/C**
Command address	F (inch/rev)	E (inch/rev)	F (inch/rev)	E (inch/rev)	F (inch/rev)	E (inch/rev)	K (n ∗ inch/rev) n: Number of pitches Same as F or E (signed)
Minimum command unit	1 (=0.00001) (1.=1.00000)	1 (=0.000001) (1.=1.000000)	1 (=0.000001) (1.=1.000000)	1 (=0.0000001) (1.=1.0000000)	1 (=0.0000001) (1.=1.0000000)	1 (=0.00000001) (1.=1.00000000)	
Command range	0.00001 to 999.99999	0.000001 to 99.999999	0.000001 to 99.999999	0.0000001 to 9.9999999	0.0000001 to 9.9999999	0.00000001 to 0.99999999	

(2) A positive value of K indicates incremental pitches.
Movement amount of one block (n pitches) = (F + K) + (F + 2K) + (F + 3K) + · · + (F + nK)

(3) A negative value of K indicates decremental pitches.
Movement amount of one block (n pitches) = (F – K) + (F – 2K) + (F – 3K) + · · + (F – nK)

(4) A program error will occur if the thread lead is not set correctly.

Error number	Meaning	Remedy
P93	**Illegal pitch value** 1) An invalid value is specified for F/E or K in a thread cutting command. 2) The last lead goes outside of the F/E command range.	Specify valid values for F/E and K. **(Reference 1)**

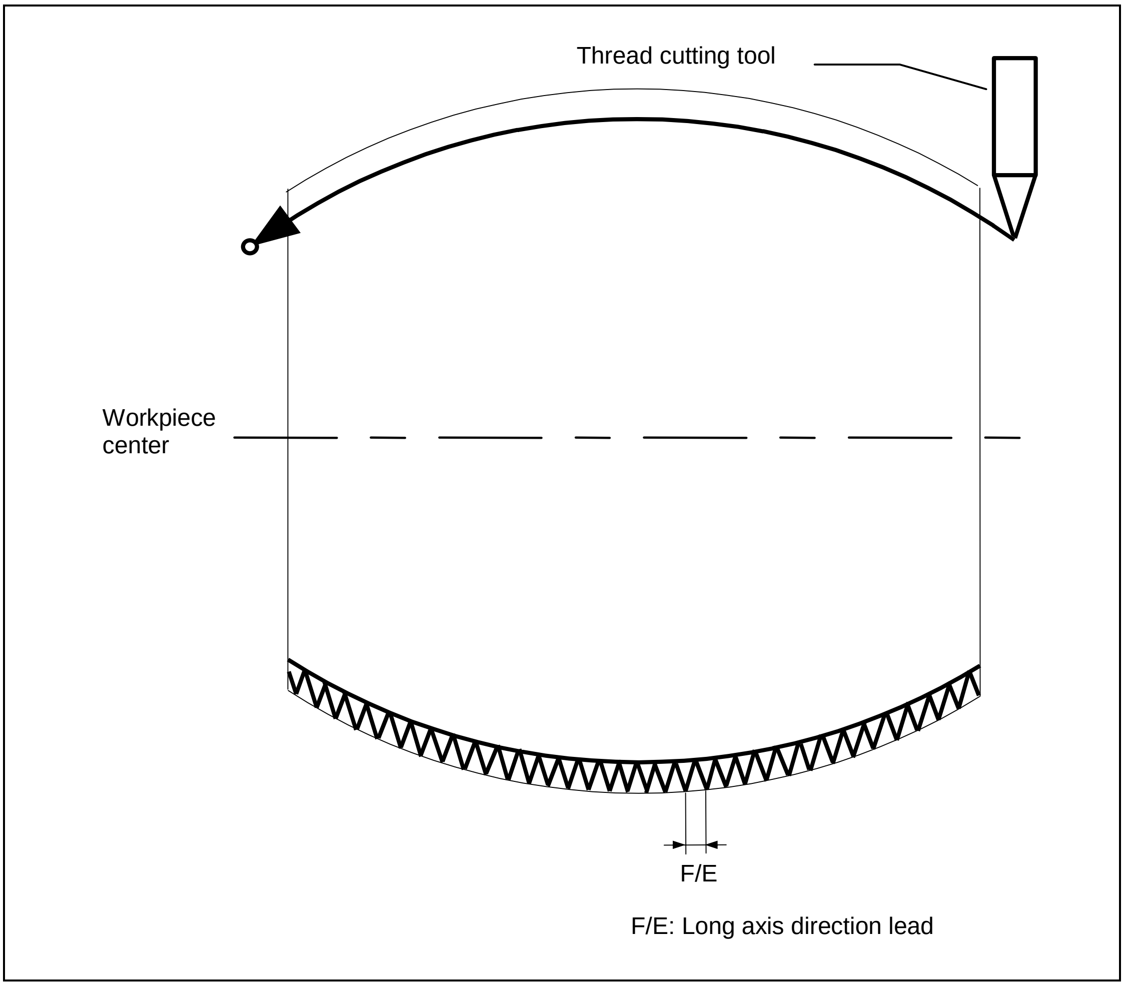
Reference 1) Last lead = $\sqrt{(F^2 + 2KZ)}$
Number of pitches = (–F + last lead)/K
Z : Length of lead axis

(5) The other matters are the same as G33.
Refer to section "6.7.1 Constant lead thread cutting; G33".

6.7.5 Circular thread cutting; G35/G36

Function and purpose

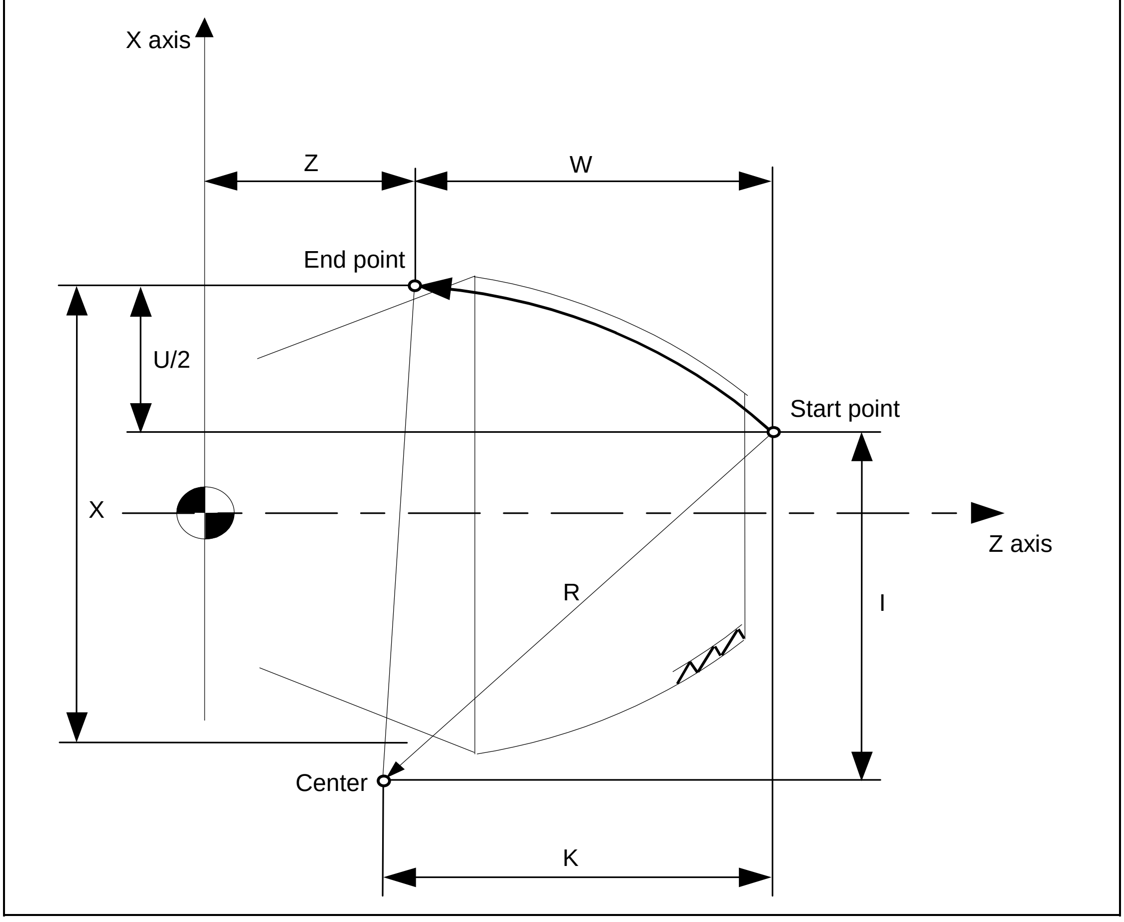
Circular thread cutting is carried out by applying circular interpolation while feeding the tool in synchronization with the spindle rotation.



Command format

G35/G36 Xx/Uu Zz/Ww { Ii Kk / Rr } Ff/Ee Qq ;

G35	Clockwise (CW)
G36	Counterclockwise (CCW)
Xx/Uu	X axis circular end point coordinate (X: absolute value for workpiece coordinate system, U: incremental value from current position)
Zz/Ww	Z axis circular end point coordinate (Z: absolute value for workpiece coordinate system, W: incremental value from current position)
Ii	X axis circular center coordinate (incremental value of circular center looking from start point)
Kk	Z axis circular center coordinate (incremental value of circular center looking from start point)
Rr	Circular radius
Ff/Ee	Lead of long axis (axis with longest movement amount) direction (Ff … Normal lead thread/Ee … Precision lead thread or inch thread)
Qq	Thread cutting start shift angle (0.001 to 360.000°)



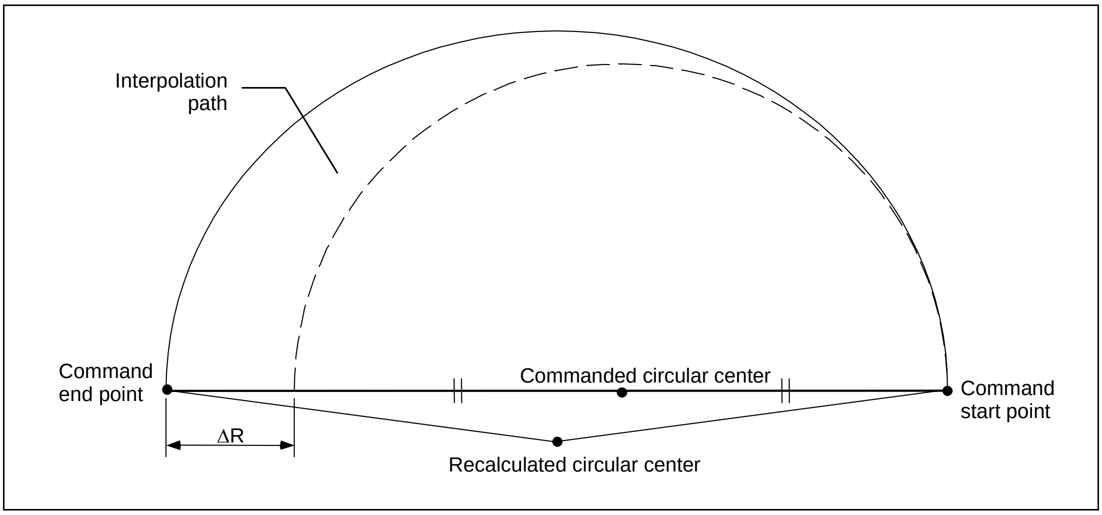
Matters related to command

(1) A program error (P33) will occur if the start point and end point match, or if the circular center angle is more than 180°.

(2) The following will occur if the start point radius and end point radius do not match.

(a) A program error (P70) will occur if the error ΔR is larger than the "[Setup parameter] #8010 G02/03 Error".

(b) If the error ΔR is smaller than the "[Setup parameter] #8010 G02/03 Error", the start point radius and end point radius will be equally sectioned, and interpolation will start from the new circular center.

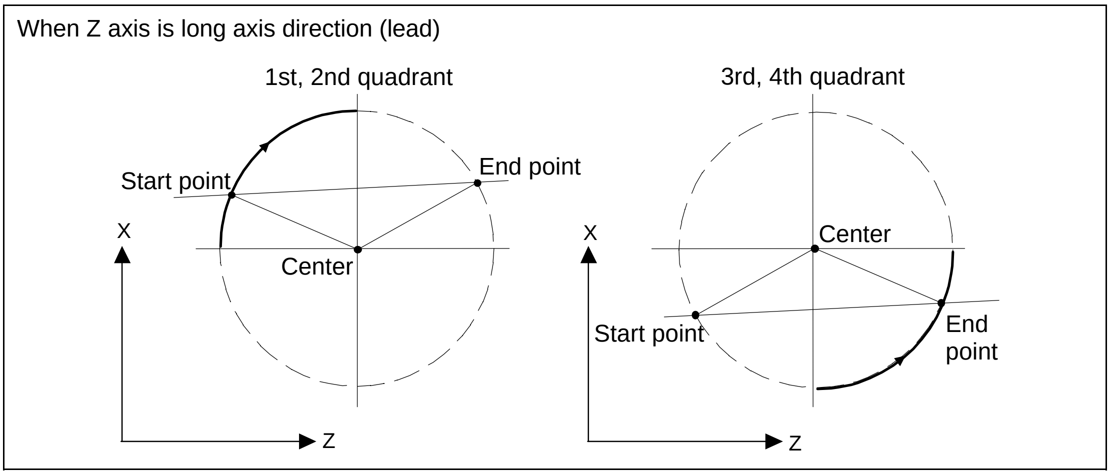


(3) A program error (P33) will occur if the R_ sign is negative.

(4) A program error (P33) will occur if I_K_,R_ are not commanded together.

(5) If I_K_ command and R_ command are issued in the same block, the R_ command will have the priority.

(6) An arc can be issued in two continuous quadrants when the circular center is set as (0,0). A program error (P33) will occur if an arc extending over three or more quadrants is issued.



(7) If the vertical axis and horizontal axis command movement amount is the same, the horizontal axis direction in the selected plane will be the long axis.

Plane selection	**Long axis when movement amount is the same**
G17 : (IJ plane)	I axis
G18 : (KI plane)	K axis
G19 : (JK plane)	J axis

Matters related to speed

(1) If "cutting feedrate > clamp speed (thread cutting clamp speed)" when thread cutting is started, the "M01 operation error 0107" will occur and thread cutting will not start. The lead axis and non-lead axis feedrate is checked in the following manner.

(a) Lead axis .... Checked with the spindle rotation speed and commanded pitch when thread cutting starts.

(b) Non-lead axis............ Feedrate of the section with the largest movement per unit time between the circular start point and end point is checked at the start of thread cutting.

(2) The cutting feedrate may exceed the clamp speed to guarantee the lead during thread cutting. In this case, "M01 operation error 0107" will appear, but thread cutting will continue. Note that only when the circular thread command is successively commanded, if the cutting feedrate exceeds the clamp speed during thread cutting, the automatic operation will stop just before the circular thread command, and "M01 operation error 0107" will appear.

(3) Related parameters

- #2002 clamp (cutting feed clamp speed)
- #2044 thr_clamp (thread cutting clamp speed)

Matters related to continuous thread cutting

(1) Continuous thread cutting is possible by successively commanding the thread cutting command. This makes it possible to cut a lead midway or to cut a special thread having a changing shape.

(2) Continuous thread cutting is possible as arc → arc, arc → constant lead, constant lead → constant lead, constant lead → arc.

Matters related to servo system delay

(1) Illegal leads will result at the start and end of thread cutting due to normal servo system delay, etc.
Thus, command the length including the illegal lead length at the start and end of thread cutting to the required thread length.
Another method is to command the required thread length with the circular thread, and command the illegal lead length before and after (start and end of thread cutting) as a uniform lead thread (G33). (Continuous thread cutting with constant lead → arc → constant lead.)

Relation with other functions

(1) Coordinate system
  (a) A program error (P113) will occur if a circular thread command is used for an orthogonal axis in the selected plane.

(2) Program test
  (a) The thread cutting feedrate is not synchronized with the spindle rotation when dry run is valid. (The thread pitch is not guaranteed.)
  (b) The dry run signal input during thread cutting is invalid.
  (c) The thread cutting feedrate is not synchronized with the spindle rotation during program check. (The thread pitch is not guaranteed.) The circular thread is a reverse run prohibit command, so reverse run is not possible.

(3) Calling, starting and stopping the program
  (a) Feed hold cannot be applied during thread cutting. If the feed hold switch is pressed during thread cutting, block stop will result at the end point of the block following the block in which thread cutting is completed (no longer thread cutting mode).

(4) Functions for supporting machining methods
  (a) Circular thread cutting will function correctly even during mirror image.
  (b) Circular thread cutting will function correctly even during cross.
  (c) The "M01 operation error 1003" will occur if the circular thread command is used during superimposition control.
  (d) A program error (P201) will occur of there is a circular thread command in the finished shape program while using the compound lathe fixed cycle.
  (e) A program error (P385) will occur if a corner R/C is commanded during thread cutting or in the next block's thread cutting.
  (f) The geometric function cannot be used with the circular thread command. A program error (P395 or P70) will occur if the geometric command is issued with the circular thread command.

(5) Inputting the speed
  (a) The thread cutting command will be synchronous feed even during the per minute feed (asynchronous) mode.

(6) Tool diameter
  (a) Thread cutting during nose R compensation will be executed after nose R compensation is temporarily canceled.

(7) Feed
  (a) "Thread cutting time constant (axis specification parameter #2045: thr_t1" can be applied to the acceleration/deceleration of the NC control axis during circular thread cutting.

(8) Interpolation
  (a) A program error (P481) will occur if the thread cutting command is issued during milling.

(9) Spindle
  (a) The spindle motor energy saving mode is invalid during thread cutting even if the ECMD signal is valid.

(10) Machine structure related functions
  (a) A program error (P712) will occur if the circular thread cutting command is issued during the inclined coordinate rotation mode. A program error (P711) will occur if the inclined coordinate rotation command is issued during the circular thread modal.

Precautions and restrictions

(1) Do not issue the circular thread cutting command during constant surface speed control. The threads will not be cut correctly because the spindle rotation speed will change during thread cutting.

(2) If the spindle override is changed during thread cutting, the threads will not be cut correctly due to a delay in the servo system.

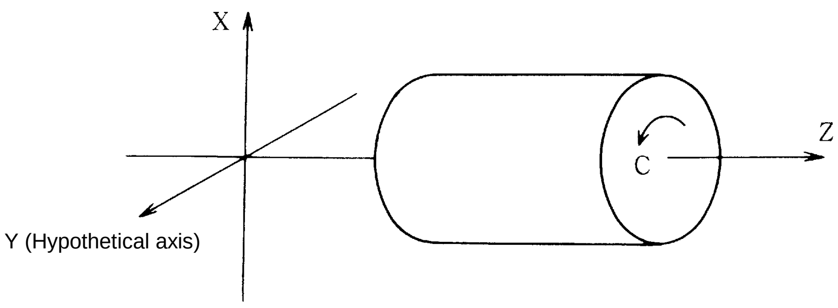
(3) A program error (P39) will occur if the circular thread (G35/G36) command is issued without the additional specifications.

(4) Circular threads cannot be cut with the thread cutting fixed cycle or the compound thread cutting fixed cycle.

6.8 Milling Interpolation; G12.1/G13.1

Function and purpose

Milling interpolation is used to perform contouring control by converting commands programmed in an orthogonal coordinate system into movements of a linear axis and rotation axis (workpiece rotation).



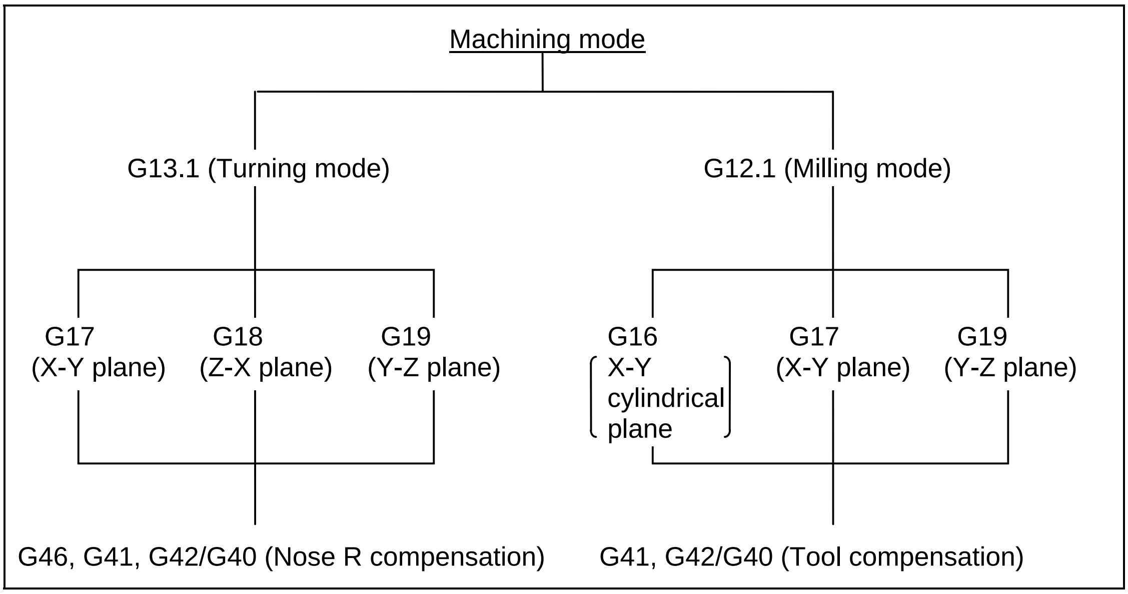
A G12.1 command is issued to perform milling and a G13.1 command is issued to cancel milling and returns to normal turning.

Command format

G12.1;	**Milling mode ON**
G13.1;	**Milling mode OFF (Turning mode)**

The following G codes are used to select milling and set the conditions.

G code	Function	Remarks
G12.1 G13.1	Milling mode ON Milling mode OFF	Default is G13.1.
G16 G17 G19	Selection of Y-Z cylindrical plane Selection of X-Y plane Selection of Y-Z plane	One of G17, G16, and G19 can be defined as the default (when G12.1 is issued) by the parameter.
G41 G42	Tool R compensation left Tool R compensation right	Default is G40.



6.8.1 Selecting milling mode

Detailed description

(1) The G12.1 and G13.1 commands are used to switch between the turning (G13.1) and milling (G12.1) modes.

(2) These commands are modal and the initial mode effective at power ON is the turning mode.

(3) The following requirements must be satisfied before a G12.1 command is issued. Otherwise, a program error results.

  (a) Nose R compensation has been canceled.

  (b) Constant surface speed control has been canceled.

(4) If one of the command axes in the milling mode has not completed reference point return, a program error results.

(5) The G12.1 command automatically cancels an asynchronous mode F command. Therefore, specify an F value in milling mode.

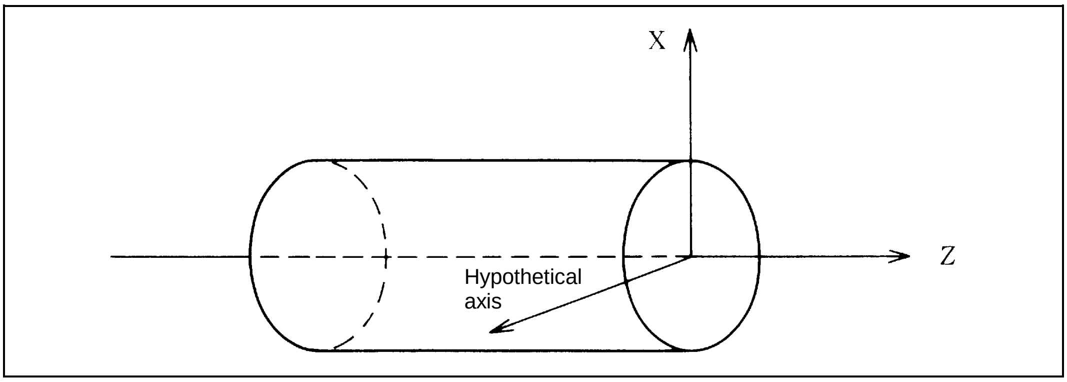
(Note) If G12.1 is executed, while no movement command has been given, after nose R compensation is canceled by an independent G40 command, nose R compensation is canceled in the G12.1 block.

6.8.2 Milling interpolation control and command axes

Detailed description

(1) The two orthogonal linear axes (X axis and Z axis) and a rotation axis are used as control axes for milling interpolation. The rotation axis is defined by a parameter.

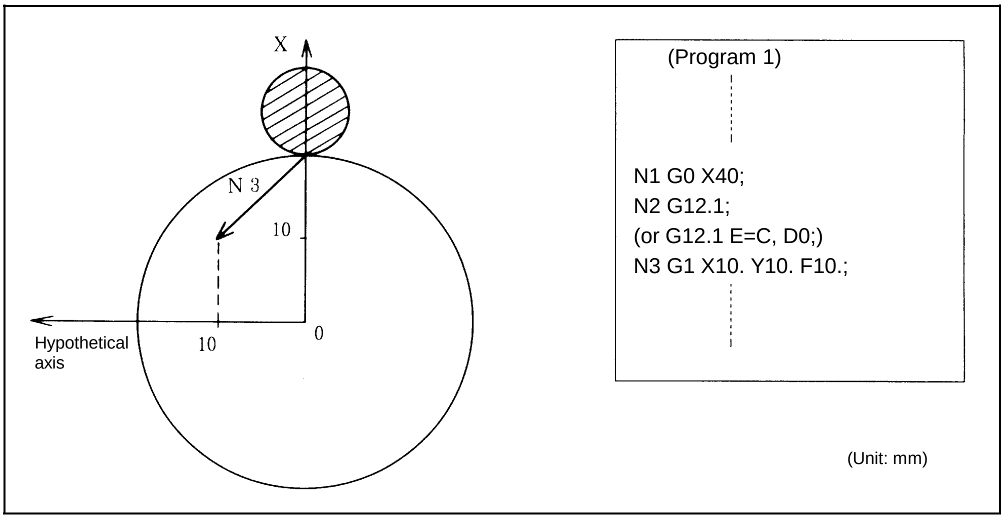
(2) Three orthogonal linear axes are used as the command axes for milling interpolation. They are the X, Z, and a hypothetical axis.
The hypothetical axis is a hypothetical axis for interpolation which intersects the X and Z axes at right angles.
The name of the hypothetical axis is defined by a parameter, either Y or the name of the control rotation axis selected in (1) above.



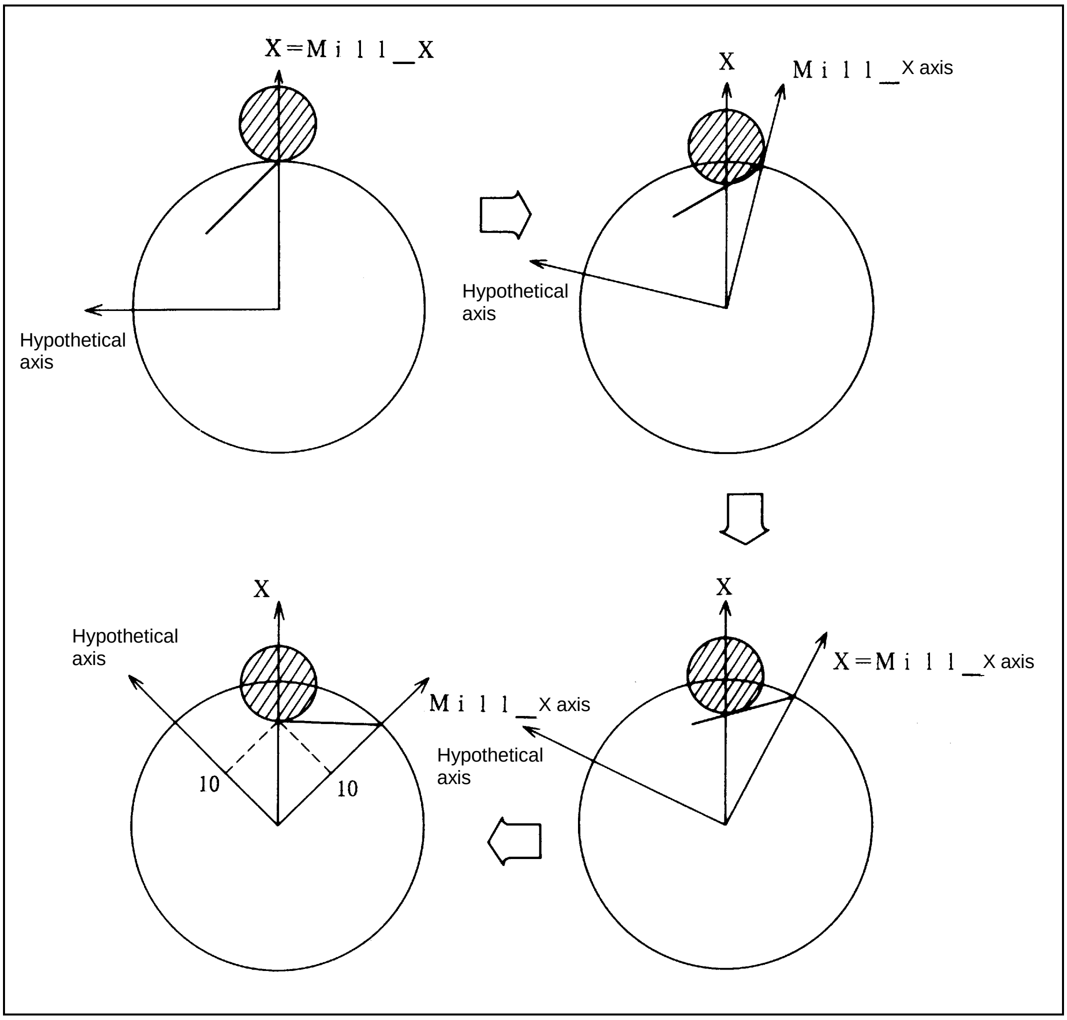
(3) Command axis X for milling is not just the interpolated one of control axis X. It is handled as X in the milling coordinate system when a G12. 1 command is issued.

(4) All values are treated as radius values in the milling coordinate system. (Any parameters commanded with radius/diameter in turning mode are ignored.)

(Example 1)



N3 of program 1 is executed as follows:



Current values
X 28.284 (diameter value display)
C 45.000

(5) Milling interpolation is also available for a two-control-axis system consisting of one linear axis and one rotation axis. The X axis must be used as the linear axis. The rotation and milling hypothetical axes are selected as shown above. In milling mode, the G17 plane must be selected.

(6) The table below lists the incremental axis names of the hypothetical axis used in milling mode. These axis commands handle radius commands only.

mill_c (hypothetical axis parameter)	Absolute axis name	Incremental axis name
0	Y	V
1	Rotation axis name (C)	Rotation axis incremental name (H)

(The following description uses Y for the hypothetical axis name and C for the rotation axis name.)

6.8.3 Selecting a plane during the milling mode

Function and purpose

A plane selection command decides the plane on which the tool moves for circular interpolation or tool radius compensation in milling mode.

Command format

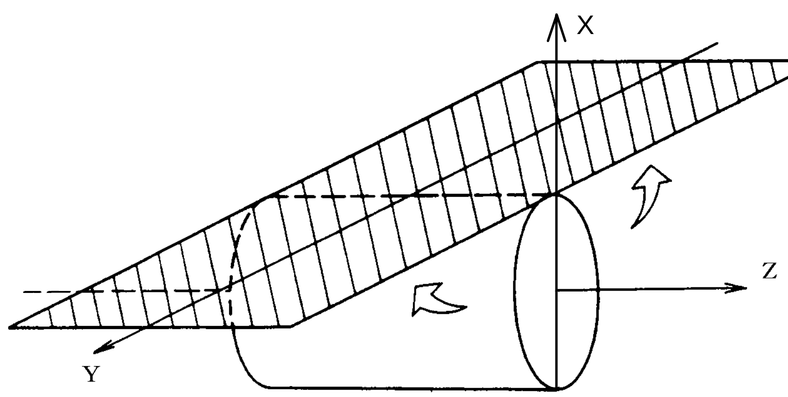
G17/G19;	
G16 C_;	
G16	Y - Z cylindrical plane
C_	Cylindrical radius value
G17	X - Y plane
G19	Y - Z plane

(1) These G commands for plane selection are modal. The G17 plane is automatically selected as the default each time the turning mode is switched to the milling mode by a G12.1 command. When the milling mode is switched back to the turning mode by a G13.1 command, the plane that was selected before the milling mode is entered is restored.

(2) G16 or G19 can also be defined as the default effective when a G12.1 command is issued. A parameter is used for this.

(3) The three planes selected are explained below.

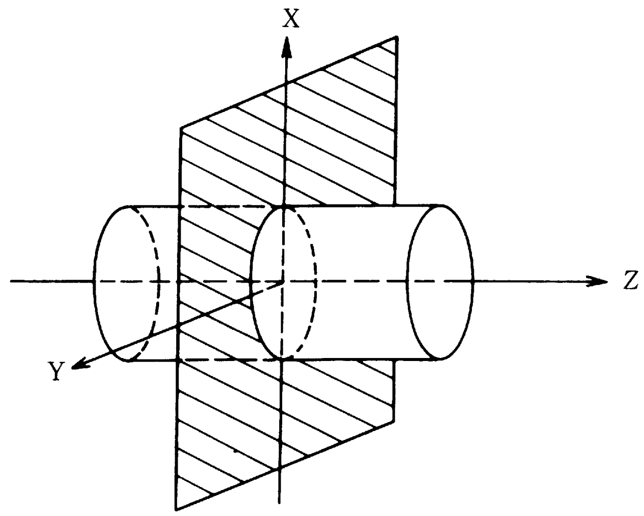
(a) G16
G16 indicates the plane obtained by developing a cylinder with its bottom radius X. This is useful to process the side face of a workpiece.



Y-Z cylindrical plane

(b) G17

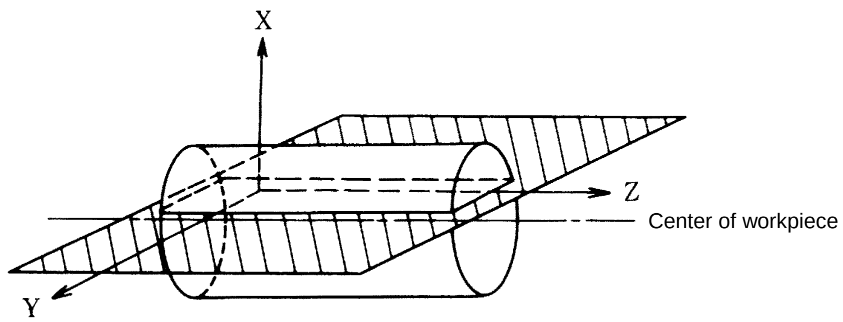
G17 is an X-Y plane in an XYZ orthogonal coordinate system. This is useful to process the end face of a workpiece.



X-Y plane

(c) G19

G19 is a Y-Z plane in an XYZ orthogonal coordinate system.



Y-Z plane

6.8.4 Setting milling coordinate system

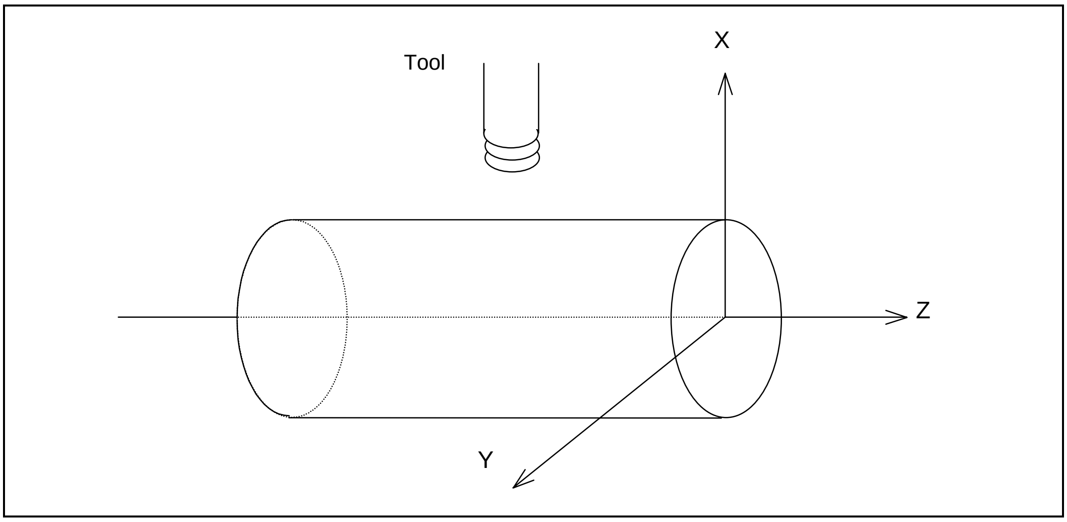
Function and purpose

The coordinate system for the milling mode is set according to the selected plane each time the turning mode (G13.1) is switched to the milling mode by a G12.1 command.

G17 and G19 planes

(1) For the X and Z axes, the current positions are set as radius value on the coordinate value.

(2) The Y axis is decided as the axis which intersects the X and Z axes at right angles. Y=0 is defined in a G12.1 command.



G16 plane

(1) To select a G16 plane, the radius value of a cylinder is specified by G16C_;. If no radius value is specified, the current X axis value is used as the radius value to define a cylinder. If no radius value can be defined, a program error "P485" occurs.

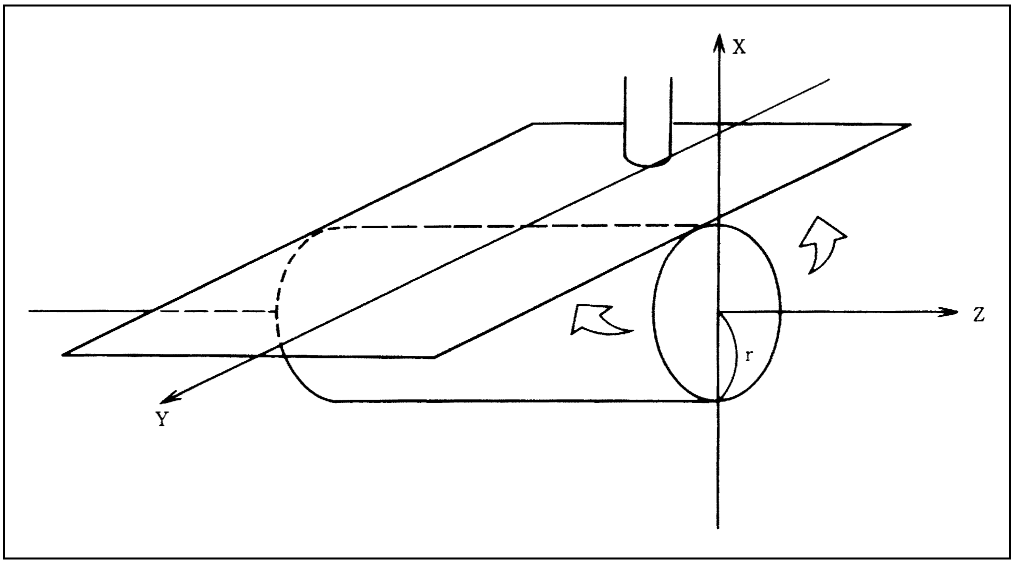
(2) As in normal turning mode, the X axis indicates the distance from the center line of the workpiece.

(3) G16 (Y-Z cylindrical plane) is actually the side of a cylinder.
The X axis indicates the distance from the center line of the workpiece. The Y axis indicates the circumference with the radius value of the bottom of a cylinder defined by a G16 command.

(4) The zero point of the Y axis is the position where a G12.1 command is issued.

(Example)

```
    :                          :
    :                          :
G12.1 G16 G50.;     or      G12.1;
                           G16 C50.;
    :                          :
    :                          :
```



6.8.5 Preparatory functions

Valid G codes in milling mode

Classification	G code	Function	Classification	G code	Function
*	G00	Positioning		G65	Macro call
*	G01	Linear interpolation		G66	Macro modal call A
*	G02	Circular interpolation (CW)		G66.1	Macro modal call B
*	G03	Circular interpolation (CCW)		G67	Macro modal call cancel
	G04	Dwell			
	G09	Exact stop check		G80	Hole drilling cycle cancel
				G83	Deep hole drilling cycle (Z axis)
	G13.1	Turning mode		G84	Tap cycle (Z axis)
				G85	Boring cycle (Z axis)
				G87	Deep hole drilling cycle (X axis)
○	G16	Y-Z cylindrical plane selection		G88	Tap cycle (X axis)
	G17	X-Y plane selection		G89	Boring cycle (X axis)
				G90	Absolute value command
	G19	Y-Z plane selection		G91	Incremental value command
	G22	Barrier check ON		G94	Asynchronous feed
	G23	Barrier check OFF			
				G98	Hole drilling cycle initial point return
				G99	Hole drilling cycle reference point return
				G61	Exact stop mode
	G40	Tool radius compensation cancel			
	G41	Tool R compensation left		G64	Turning mode
	G42	Tool R compensation right			

* : Milling interpolation command
○ : G code effective only in milling mode

(1) If an invalid G code is issued in milling mode, a program error "P481" occurs.

(2) In milling mode, all movement commands are commanded with the coordinate system determined by the selected machining plane. The rotation axis thus cannot be moved by a direct command in milling mode. To perform milling at a specific position of a workpiece, therefore, positioning must have been made in turning mode.

(Example)

```
        :
        :
G0 X100. C180.;          ⇒  Positioning before milling
G12.1;
G0 X50.;
        :
        :
```

(3) If a command for an axis other than X, Z, and Y (rotation axis) is issued in milling mode, a program error results.

(4) In milling mode, the Y axis can be specified by only four G codes: G00, G01, G02, and G03. These are called the milling interpolation commands.

Positioning (G00)

If a G00 command is issued in milling mode, positioning is made to the specified point on the selected plane at a rapid traverse rate.

G00 X/U__ Y/V__ Z/W__;

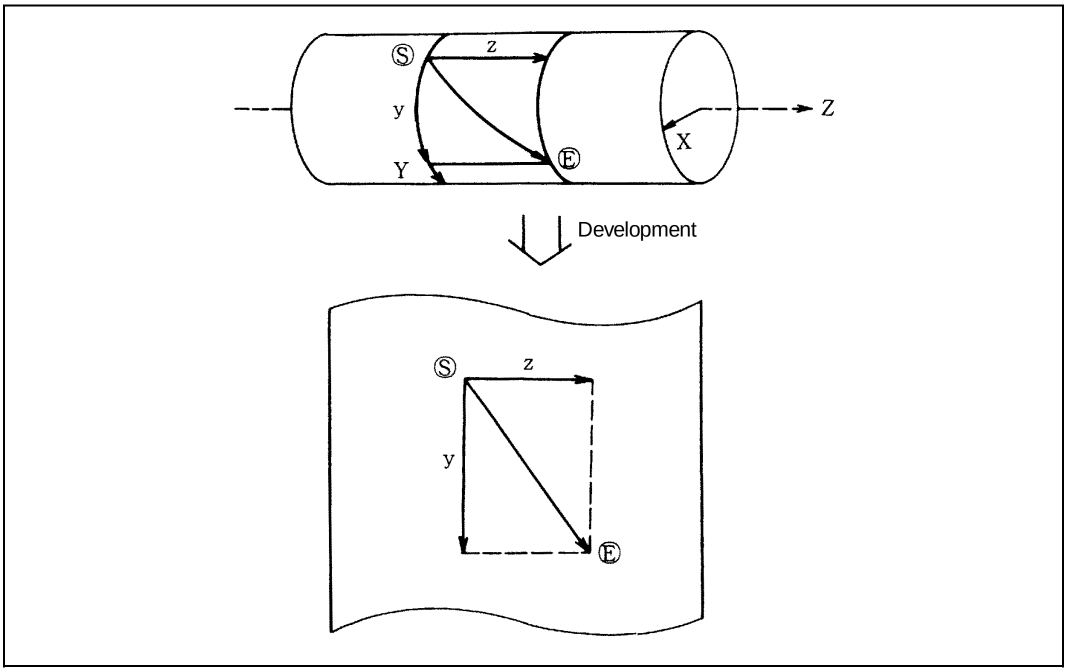
Linear interpolation (G01)

If a G01 command is issued in milling mode, linear interpolation is made to the specified point on the selected plane at the speed specified by an F speed.

(1) G16 mode

Program format

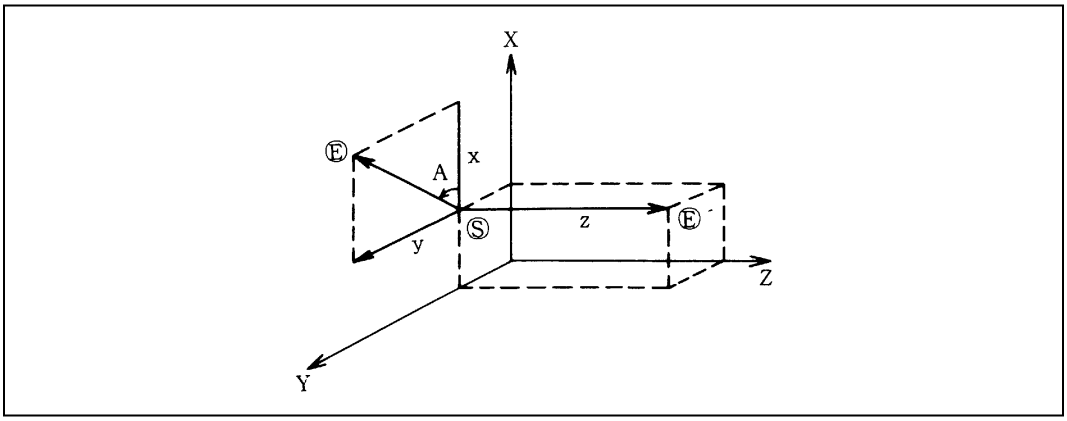
G01 Y/V__ Z/W__ X/U__ F__;



(2) G17 mode

Program format

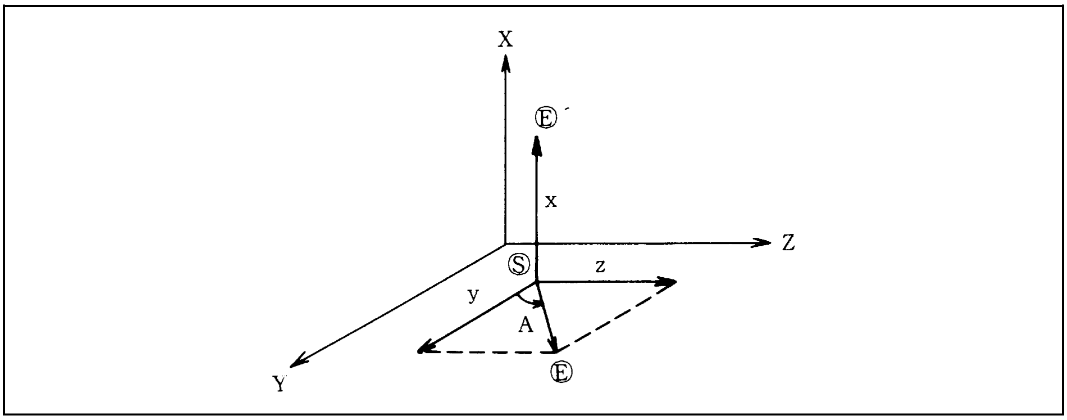
G01 X/U__ Y/V__ Z/W__ F__;



(3) G19 mode

Program format

G01 Y/V__ Z/W__ X/U__ F__;



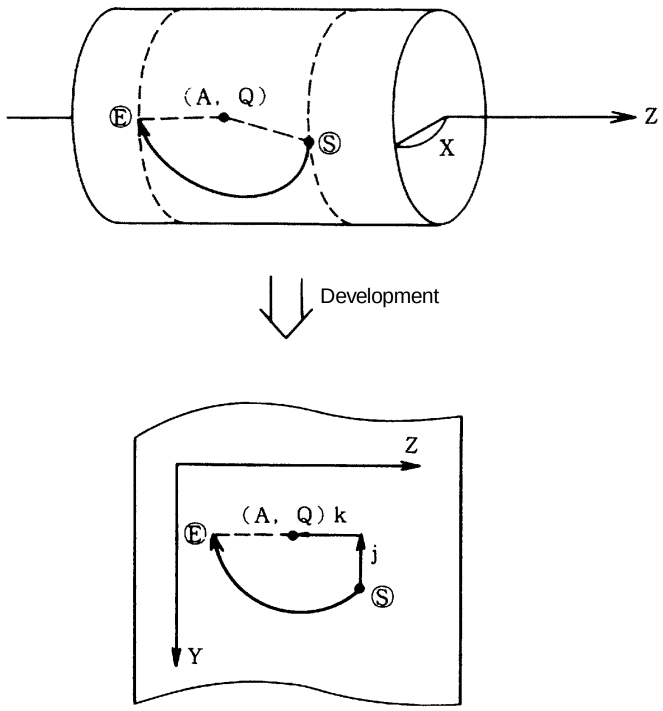
Circular interpolation (G02/G03)

If a G02 or G03 command is issued in milling mode, circular interpolation is performed at the specified speed on the selected plane.

(1) G16 mode

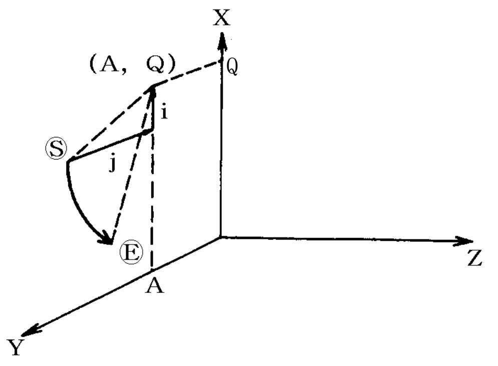
G02/G03 Y/V__ Z/W__ J__ K__ F__;
or
G02/G03 Y/V__ Z/W__ R__ F__;

G02	Circular interpolation (clockwise)
G03	Circular interpolation (counterclockwise)
Y/V	Circular end point coordinate Y axis (Y: absolute value, V: incremental value)
Z/W	Circular end point coordinate Z axis (Z: absolute value, W: incremental value)
J/K	Circular center incremental value (radius command incremental value from the start point to the center)
R	Circular radius
F	Feedrate



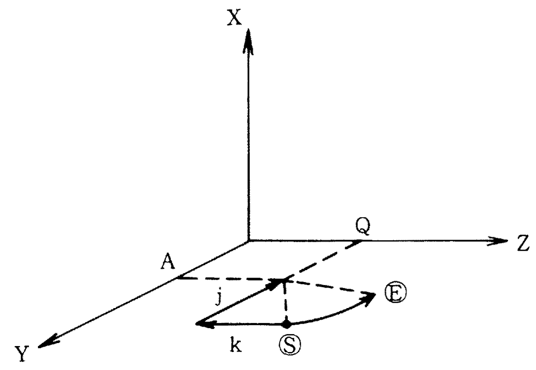
(2) G17 mode

G02/G03 X/U__ Y/V__ I__ J__ F__;	
or	
G02/G03 X/U__ Y/V__ R__ F__;	
X/U	Circular end point coordinate X axis (X: absolute value, U: incremental value)
Y/V	Circular end point coordinate Y axis (Y: absolute value, V: incremental value)
I/J	Circular center incremental value (incremental value from the start point to the center)
R	Circular radius
F	Feedrate



(2) G19 mode

G02/G03 Y/V__ Z/W__ J__ K__ F__ ;	
or	
G02/G03 Y/V__ Z/W__ R__ F__ ;	
Y/V	Circular end point coordinate Y axis (Y: absolute value, V: incremental value)
Z/W	Circular end point coordinate Z axis (Z: absolute value, W: incremental value)
J/K	Circular center incremental value (incremental value from the start point to the center)
R	Circular radius
F	Feedrate



6.8.6 Switching from milling mode to turning mode; G13.1

Detailed description

(1) A G13.1 command is used to cancel the milling mode and return to the turning mode.

(2) The G13.1 command is effective if the following requirement is met. If not, a program error occurs.

  (a) Tool radius compensation has been canceled.

(3) The G13.1 command restores the plane selected before the preceding G12.1 command was issued.

(4) The G13.1 command restores the mode (synchronous or asynchronous) and the F value (if in asynchronous mode) selected before the preceding G12.1 command was issued.

(Note) If G13.1 is executed, while no movement command has been given, after cancellation by an independent G40 command, tool radius compensation is canceled in the G13.1 block.

6.8.7 Feed function

Asynchronous cutting feed

An asynchronous feed mode (G94 command) can use F5.3 digits to specify the feedrate per minute in units of 0.001mm/min. The specifiable range is 0.001 to 60000.000mm/min. If the effective speed exceeds the cutting feed clamp speed, it is clamped by that clamp speed.

(Note 1) Whenever the turning mode is switched to the milling mode by a G12.1 command, the F command modal value is canceled. After mode change, therefore, the feedrate must be set by an F command.
(Note 2) A G12.1 command forces the mode to shift to the asynchronous mode.
(Note 3) When the milling mode is canceled by a G13.1 command, both the feed mode and F command modal value return to the original state before the preceding G12.1 command was issued.

6.8.8 Program support functions

Relation with other functions

The following program support functions are effective in milling mode:
(1) Linear angle command
(2) Variable command
(3) Automatic corner chamfering/corner R
(4) Geometric function
(5) Hole drilling cycle
(6) Subprogram function
(7) User macro

6.8.9 Miscellaneous functions

Relation with other functions

(1) M and B commands can be issued in milling mode.

(2) In milling mode, an S command specifies not the spindle speed but the rotary tool speed.

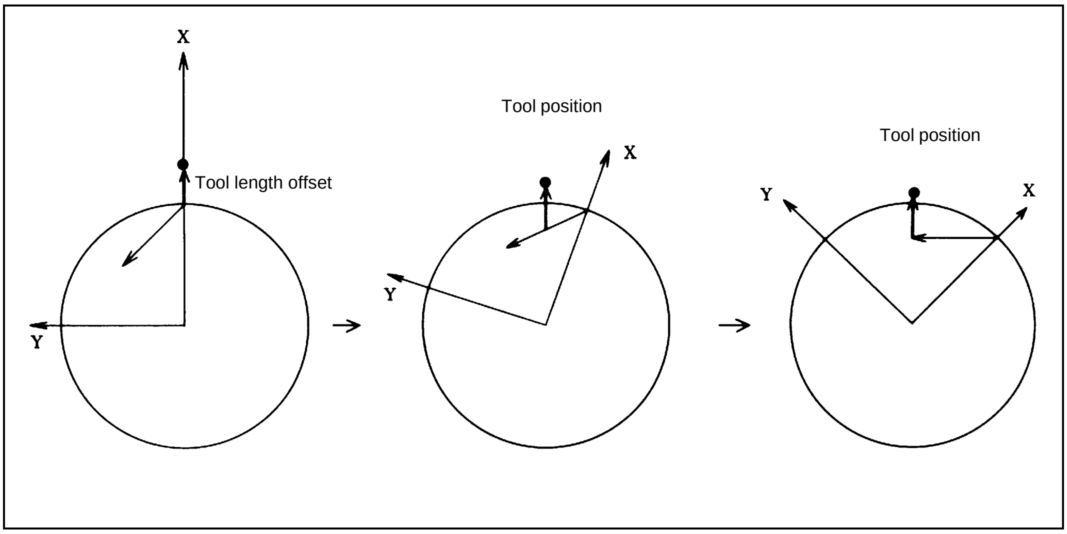
(3) If a T command is issued in milling mode, a program error occurs. Before a G12.1 command is issued, therefore, tool selection must be done.

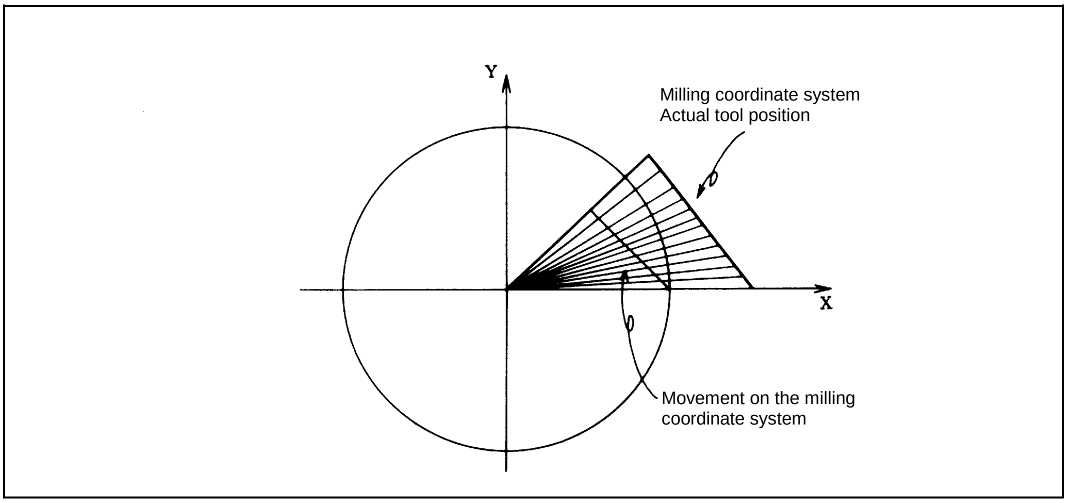
```
    :
T1212;          ⇒ Specify a T command before a G12.1 command.
G0 X100. Z0.;
G12.1;
    :
T1200;          ⇒ In milling mode, a T command causes a program error.
    :
G13.1;
```

6.8.10 Tool offset functions

Tool length offset

(1) In milling mode, tool compensation is performed by adding the tool length offset amount specified on the cutting coordinates converted from the milling coordinate system.

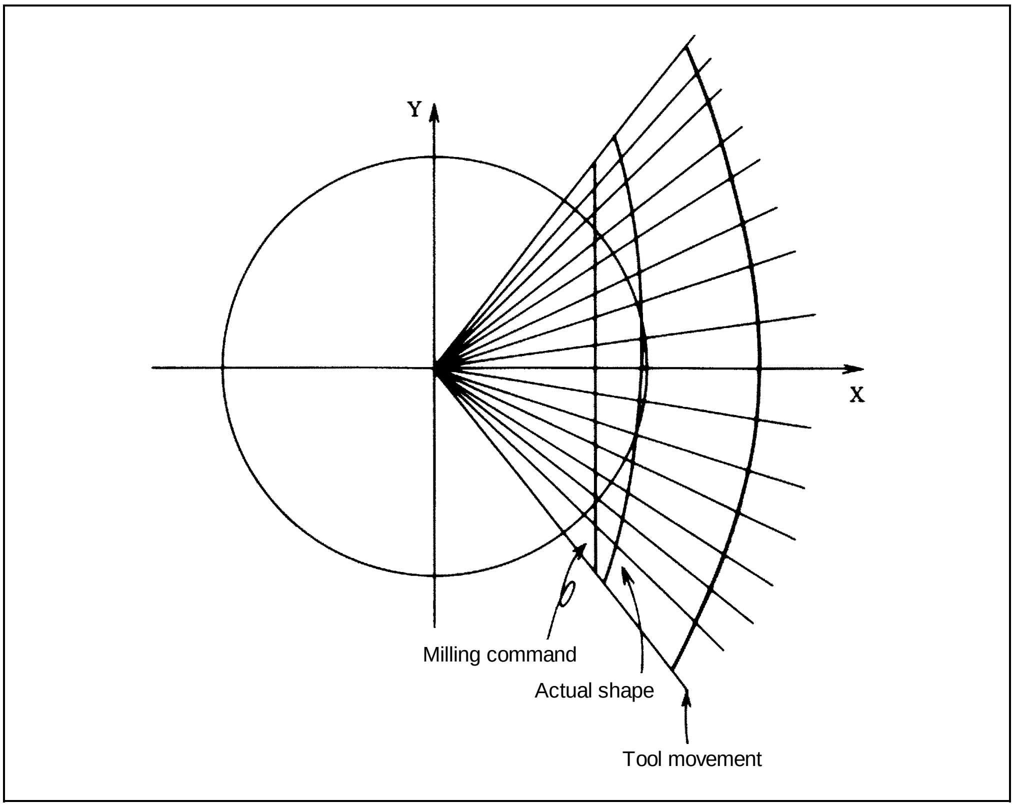




(2) As in (1) above, if the offset amount is different from the actual one, the shape is not corrected normally.

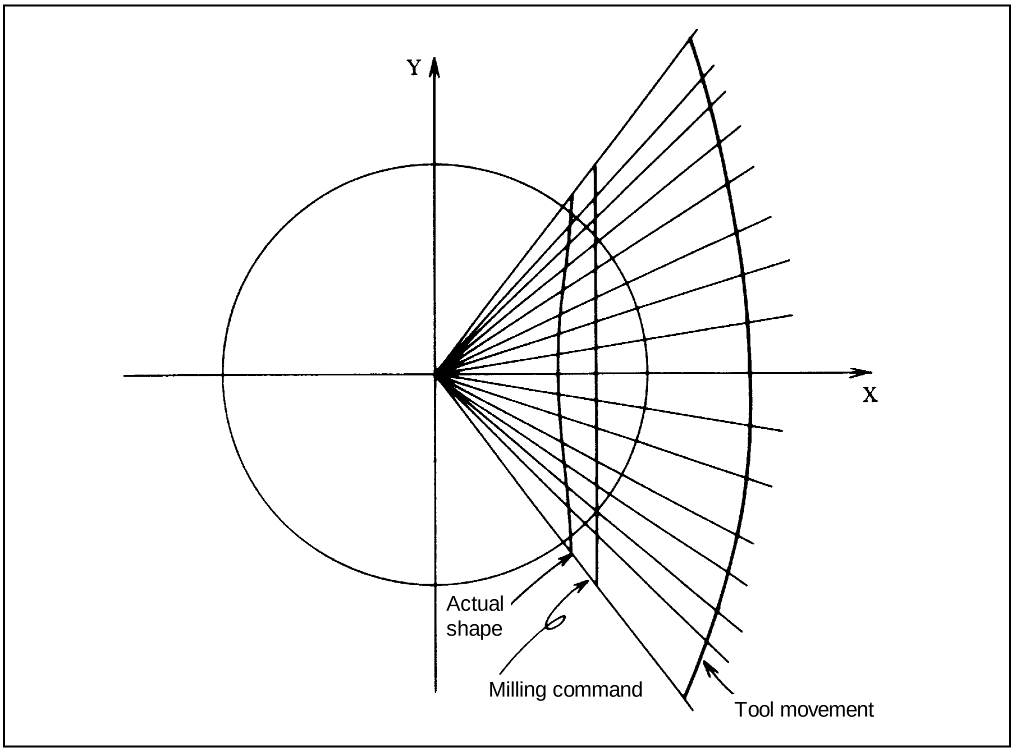
(a) If the offset amount is larger than tool length:

Example: The actual tool length is 15.0 when tool length X = 20.0



(b) If the offset amount is smaller than tool length:

Example: The actual tool length is 25.0 when tool length X = 20.0



Tool radius compensation

The workpiece shape can be compensated in the direction of the vector by the radius amount of the tool specified by a G command (G40 to G42) and selected compensation number.

Command format

G40 Xx Yy;	**Tool radius compensation cancel**
G41 Xx Yy;	**Tool radius compensation (left)**
G42 Xx Yy;	**Tool radius compensation (right)**

(1) A tool radius compensation command must be issued after the milling mode is entered. The tool radius compensation command must be canceled before the turning mode is restored.

(2) A tool compensation number must be specified before the milling mode is entered (before a G12. 1 command is issued). A T command in milling mode causes a program error.

(3) Tool radius compensation is performed on the selected plane.

G17 plane ... XY axes
G19 plane, G16 plane } YZ axes

Tool radius compensation cancel mode

Tool radius compensation is canceled under either of the following conditions:

(1) While a G12.1 command is effective

(2) After a compensation cancel command (G40) is issued

In the compensation cancel mode, the offset vector is 0 and the tool center path matches the programmed path. A program that contains tool radius compensation must end after the compensation is canceled.

Starting tool radius compensation (startup)

Tool radius compensation starts if all the following requirements are met in compensation cancel mode:

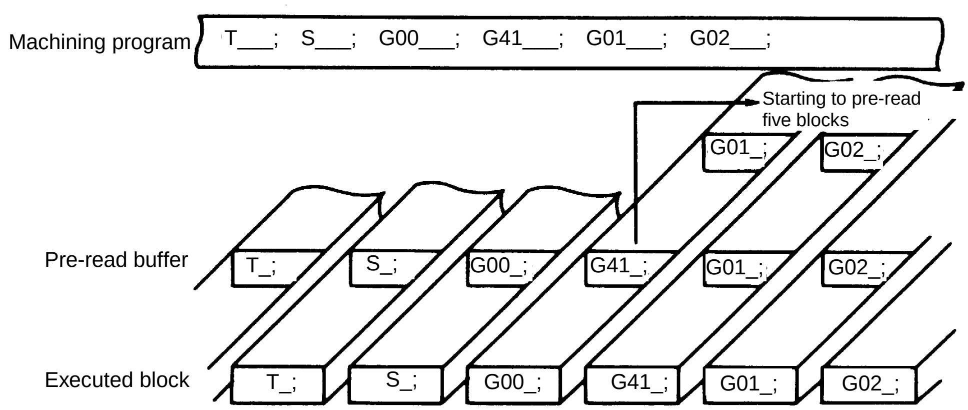
(1) A G41 or G42 command is issued.

(2) The tool radius compensation number is greater than 0 and equal to or less than the maximum compensation number.

(3) The movement command is not a circular command.

Whether in continuous or single block operation, compensation always starts after reading three movement command blocks, or if three movement command blocks are not found, up to five continuous blocks.
Similarly, in compensation mode, up to five blocks are pre-read for compensation operation.

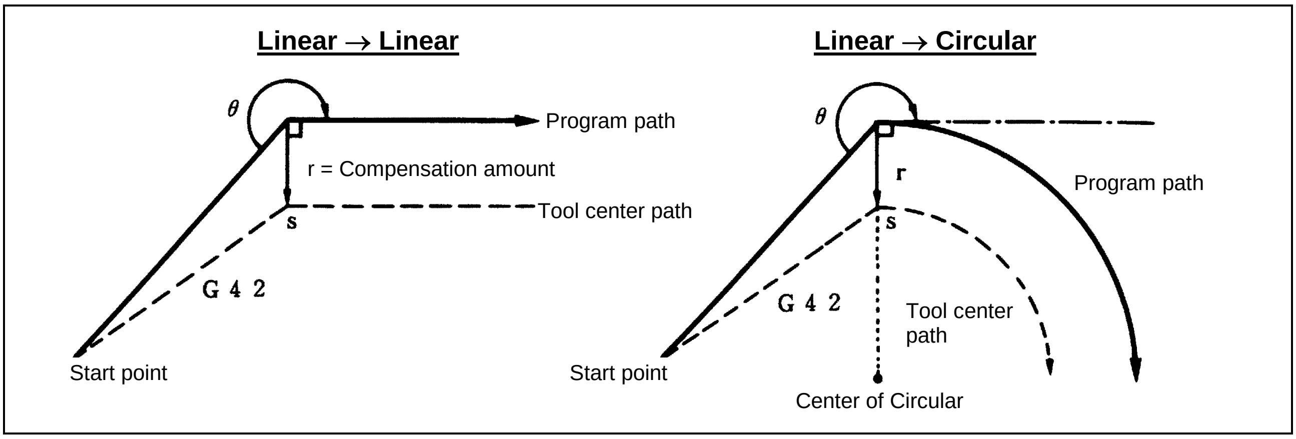
Control state diagram



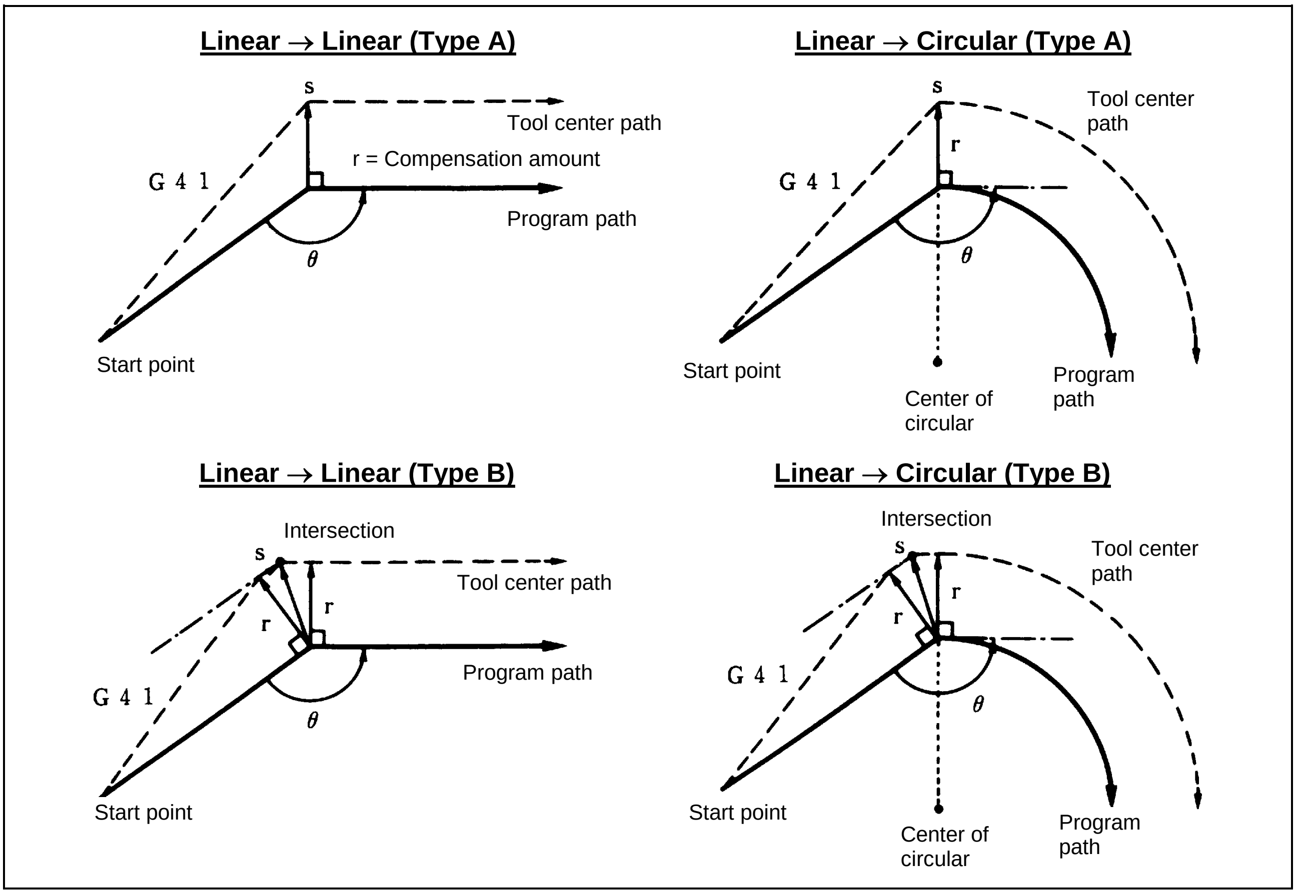
There are two ways of starting tool radius compensation: type A and type B.
The type depends on selection of the control parameter "Radius compen type B".
This type is used in common with the compensation cancel type.
In the following explanatory figure, "S" denotes the single block stop point.

Start operation for tool radius compensation

(1) Machining an inside corner

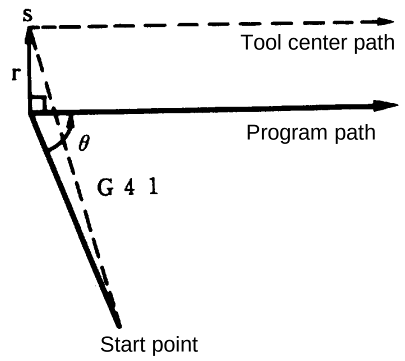


(2) Machining an outside corner (obtuse angle) (Type A or B can be selected by parameter) [90° ≤ θ < 180°]

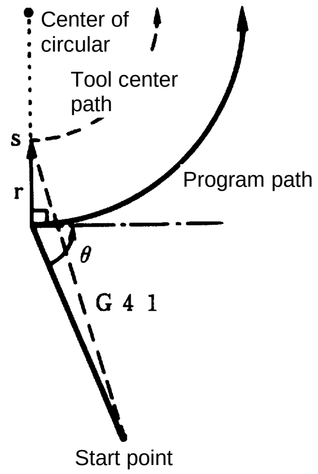


(3) Machining an outside corner (acute angle) (Type A or B can be selected by parameter) [$\theta < 90°$]

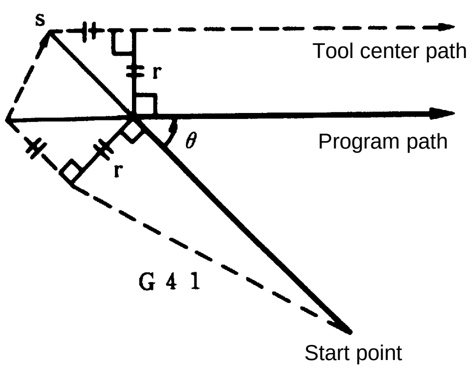
Linear → Linear (Type A)



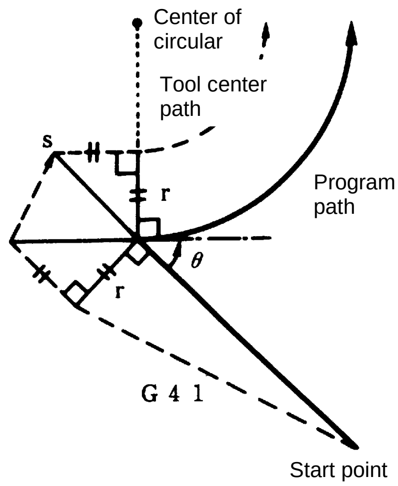
Linear → Circular (Type A)



Linear → Linear (Type B)



Linear → Circular (Type B)

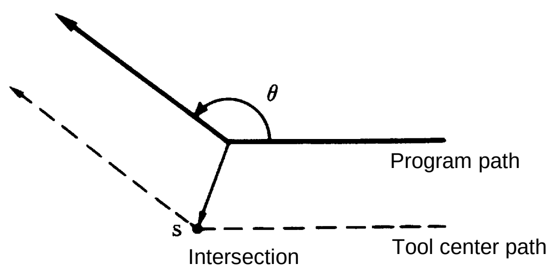


Operations in compensation mode

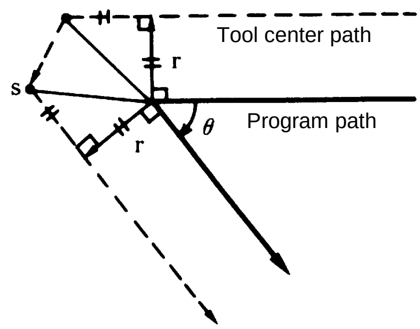
Compensation is valid both for positioning and for interpolation commands such as circular and linear interpolation.
Even if the same compensation command (G41 or G42) is specified in the compensation mode, the command will be ignored.
If four or more blocks not accompanying movement are assigned continuously in the compensation mode, over-cutting or under-cutting will result.

(1) Machining an outside corner

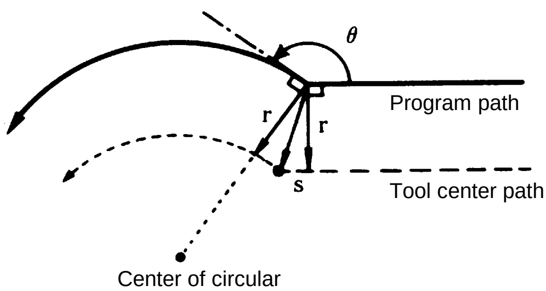
Linear → Linear (90° ≤ θ < 180°)



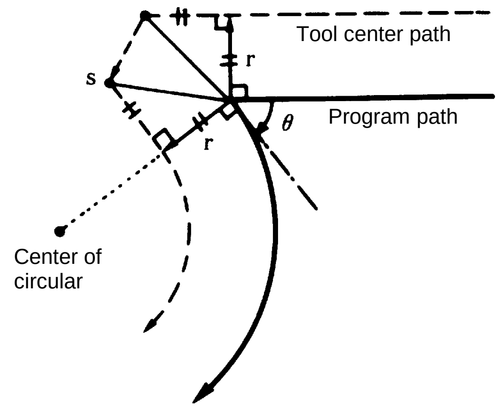
Linear → Linear (0° < θ < 90°)



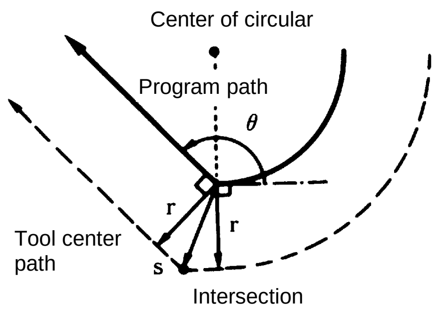
Linear → Circular (90° ≤ θ < 180°)



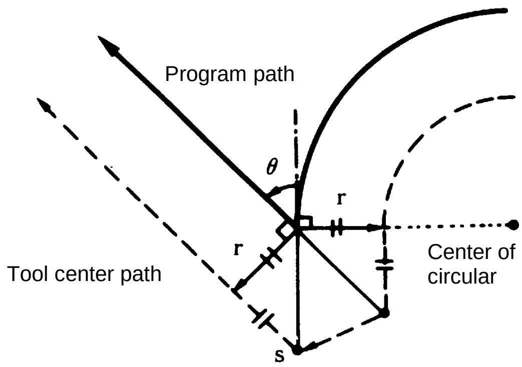
Linear → Circular (0° < θ < 90°)



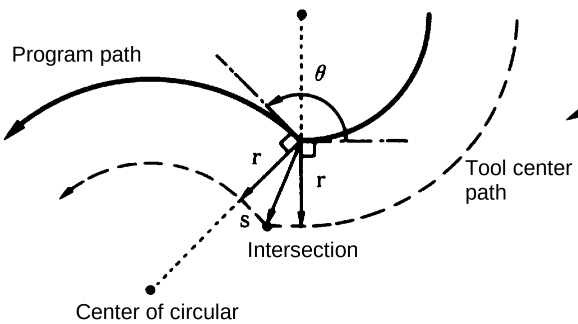
Circular→ Linear (90° ≤ θ < 180°)



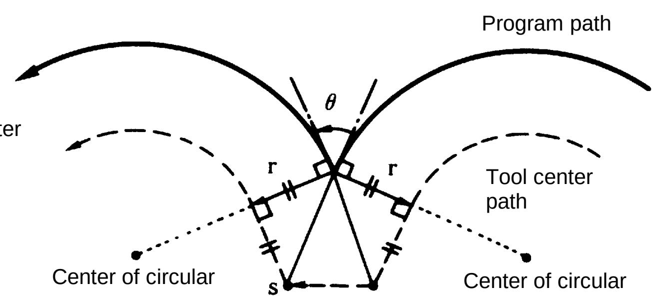
Circular → Linear (0° < θ < 90°)



Circular → Circular (90° ≤ θ < 180°)

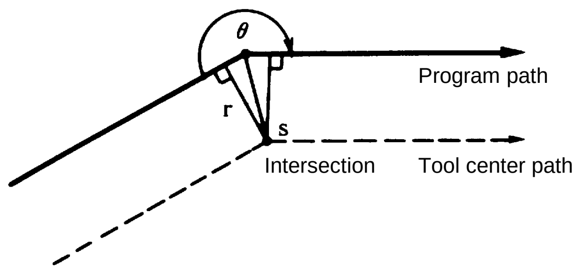


Circular → Circular (0° < θ < 90°)

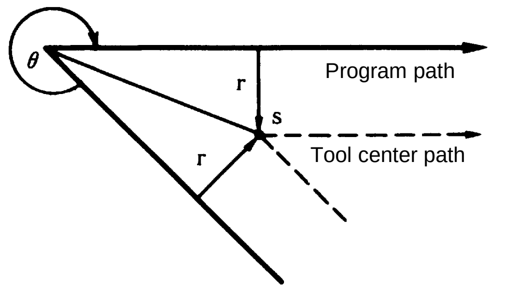


(2) Machining an inside corner

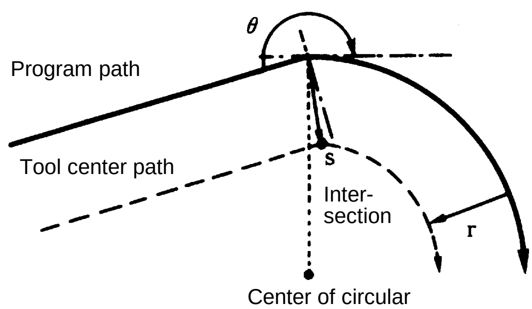
Linear → Linear (Obtuse angle)



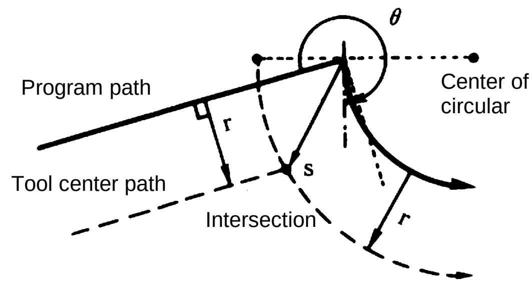
Linear → Linear (Obtuse angle)



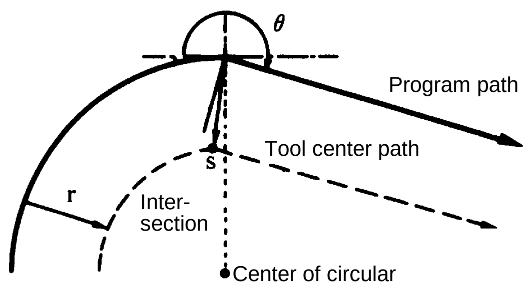
Linear → Circular (Obtuse angle)



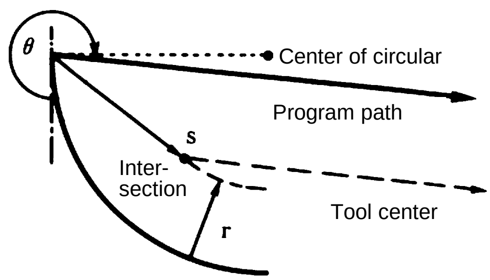
Linear → Circular (Obtuse angle)



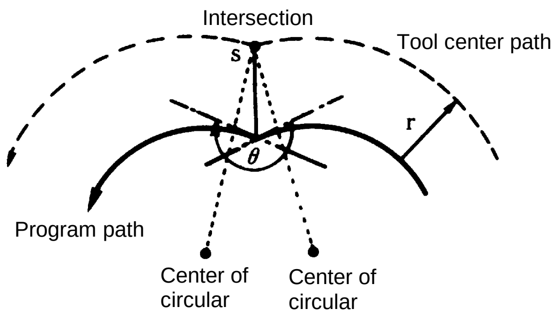
Circular → Linear (Obtuse angle)



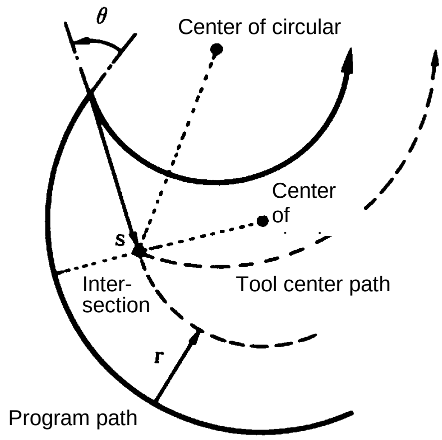
Circular → Linear (Obtuse angle)



Circular → Circular (Obtuse angle)

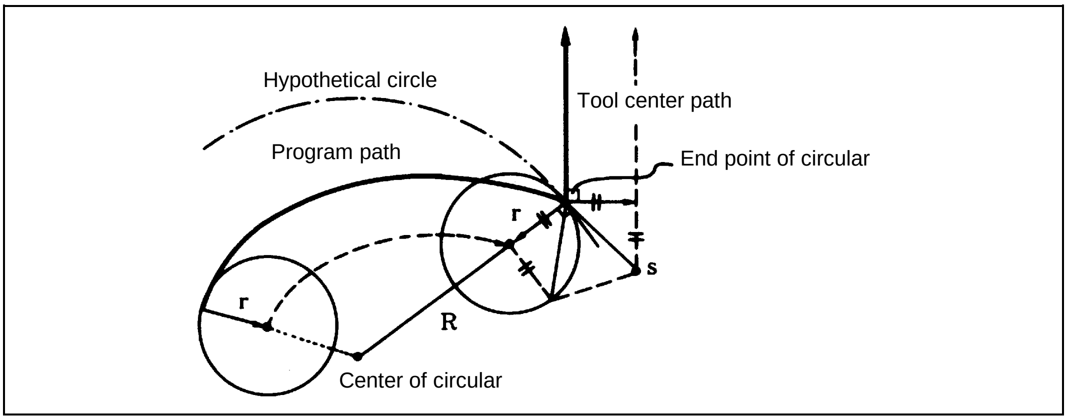


Circular → Circular (Acute angle)



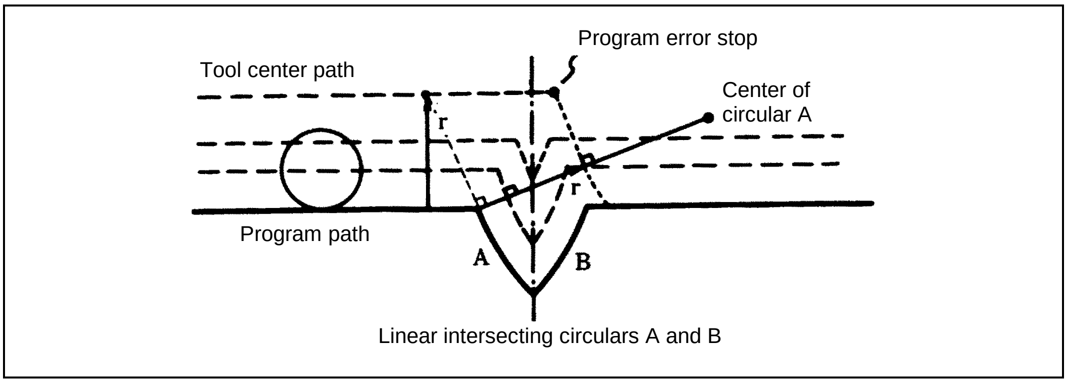
(3) When the arc end point is not on the circular

With a spiral circular command: the area from the arc start point to the end point is interpolated as a spiral arc.
With a normal circular command: if the error after compensation is within the parameter value, it is interpolated as a spiral arc.



(4) When the inside intersection does not exist

In an instance such as that shown in the figure below, the intersection of arcs A and B may cease to exist due to the compensation amount.
In such cases, program error "P152" appears, and the tool stops at the end point of the preceding block.



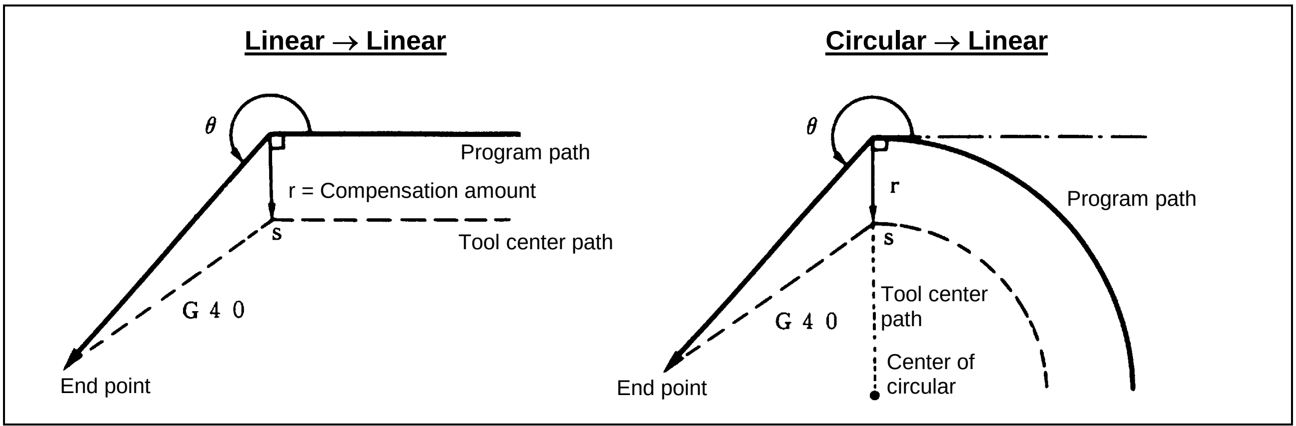
Tool radius compensation cancel

If either of the following conditions is met in the tool radius compensation mode, the compensation will be canceled. However, the movement command must be a command other than a circular command. If an attempt is made to cancel the compensation by a circular command, program error "P151" results.

(1) A G40 command has been executed.
The cancel mode is established once the compensation cancel command has been read, the 5-block pre-read process is suspended, and 1-block pre-read applies instead.

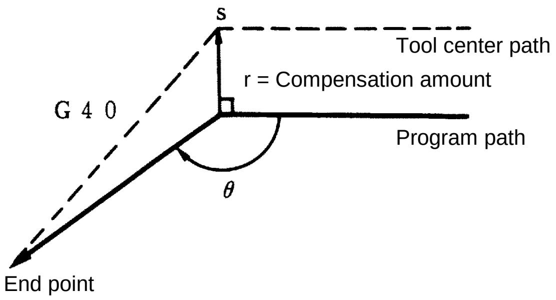
Tool radius compensation cancel operation

(1) Machining an inside corner

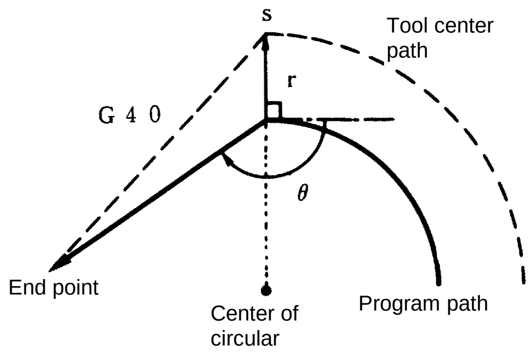


(2) Machining an outside corner (obtuse angle) (Type A or B can be selected by parameter) [90° ≤ θ < 180°]

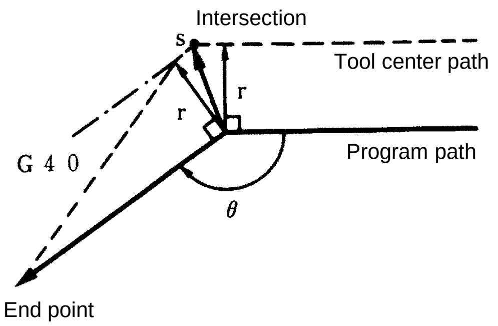
Linear → Linear (Type A)



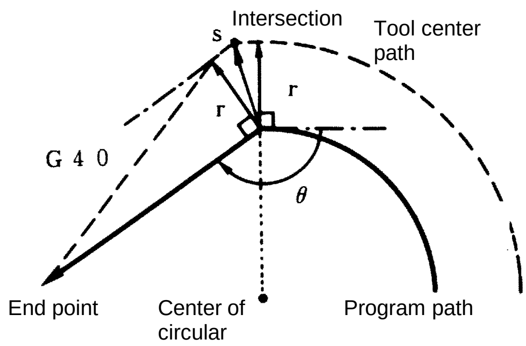
Circular → Linear (Type A)



Linear → Linear (Type B)

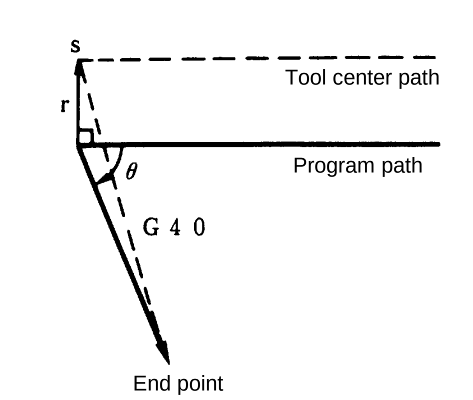


Circular → Linear (Type B)

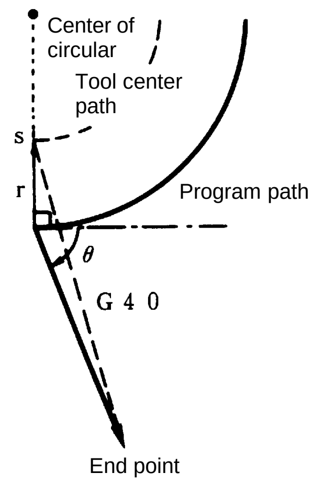


(3) Machining an outside corner (acute angle) (Type A or B elm be selected by parameter) [$\theta < 90°$]

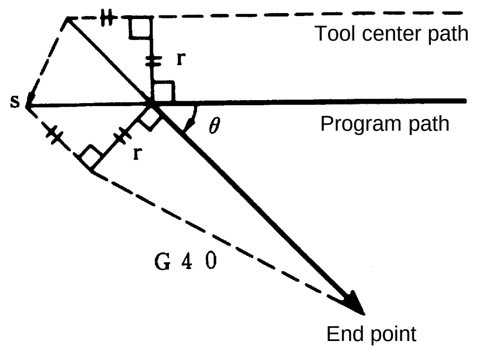
Linear → Linear (Type A)



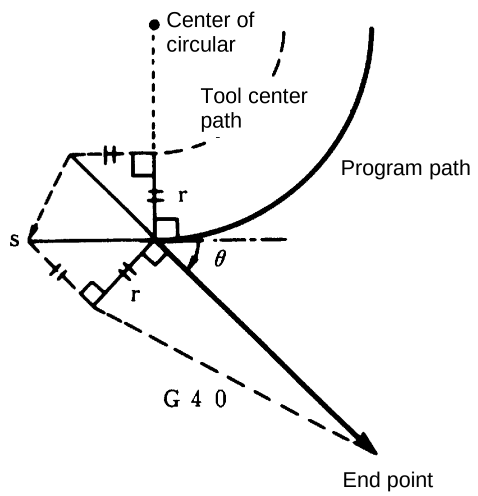
Circular → Linear (Type A)



Linear → Linear (Type B)



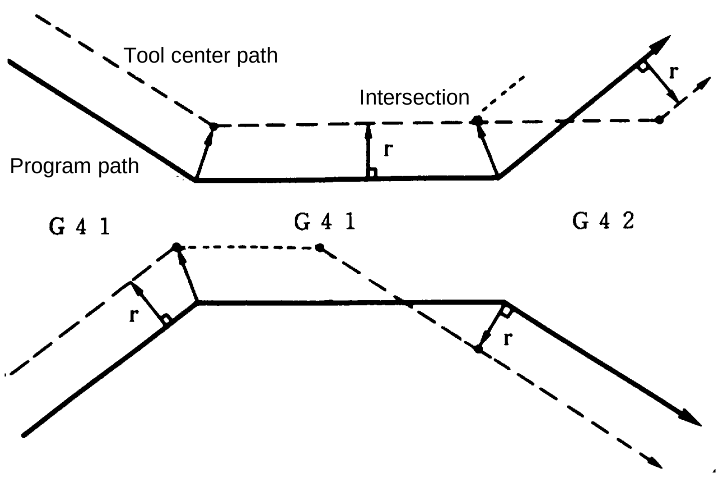
Circular → Linear (Type B)



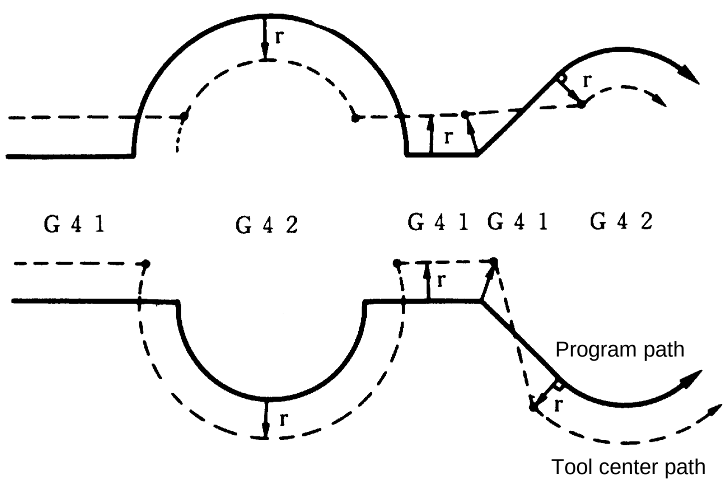
Changing the compensation direction during tool radius compensation

The compensation direction can be changed by changing the compensation command in the compensation mode without the compensation having to be first canceled.
However, no change is possible in the compensation start block and the following block.

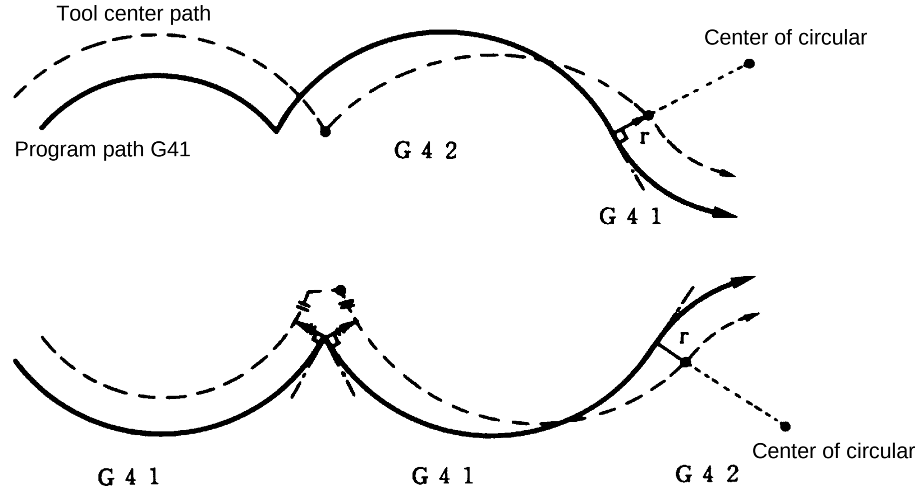
Linear → Linear



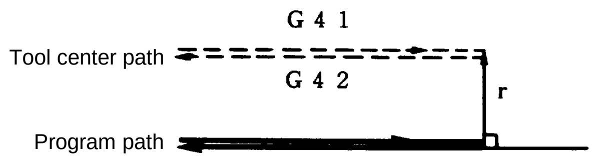
Linear ↔ Circular



Circular → Circular



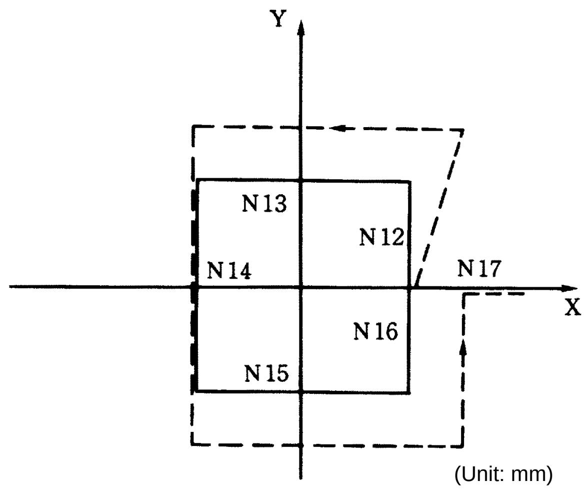
Linear return



G0 block

If there is a block containing a G0 command, the preceding block does not perform intersection operation, the tool comes to the position vertical to the end point, and the G0 block temporarily loses the offset vector. Compensation is not canceled, but instead the tool moves from the intersection vector directly to a point without vector, that is, to the point specified by the program. The offset vector is regenerated by a block containing a G1 command.

```
        :
        :
N10 G12.1;
N11 G42;
N12 G1 Y100..F100;
N13 X–100.;
N14 G0 Y–100.;
N15 G1 X100.;
N16 Y0.;
N17 G40 G0 X200.;
        :
        :
```



(Unit: mm)

Blocks without movement and M commands inhibiting pre-read

The following blocks are known as blocks without movement;

a. M03; ········ M command
b. S12; ········ S command
c. G04 X500; ········ Dwell
d. G10 P01 R50; ········ Compensation amount setting
e. (G17)Z40; ········ Movement but not on compensation plane
f. G90; ········ G code only
(a. – f.: No movement)
g. G91 X0; ········ Movement amount 0 ······ Zero movement amount

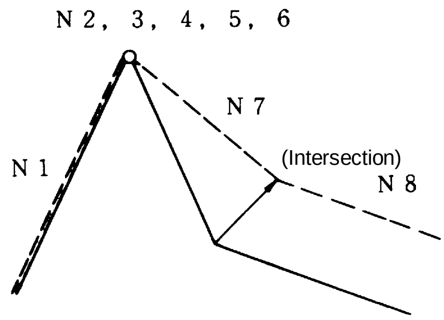
M00, M01, M02, and M30 are treated as M codes inhibiting pre-read.

(1) Blocks without movement commands specified at compensation start

If four or more blocks without movement continue or if M command inhibiting pre-read is issued, offset vectors are not generated.

N1 X30. Y60.;
N2 G41;
N3 G4 X1000;
N4 F100;
N5 S500;
N6 M3;
N7 X20. Y–50.;
N8 X50. Y–20.;

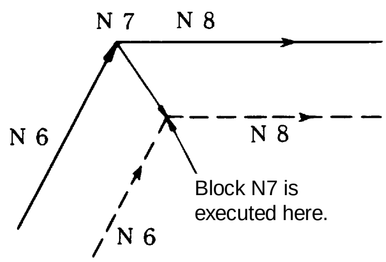
(N2 – N6: Block without movement)



(2) Blocks without movement commands specified in compensation mode

If four or more blocks without movement do not continue in compensation mode and if no M command inhibiting pre-read is issued, intersection vectors are generated as usual.

N6 G91 X100. Y200.;
N7 G04 X P1000; .... Block without movement
N8 X200.;

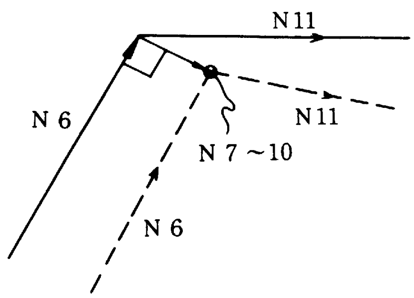


If four or more blocks without movement continue or if M command inhibiting pre-read is issued, offset vectors are generated perpendicularly at the end point of the preceding block.

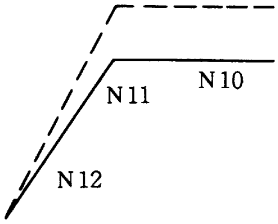
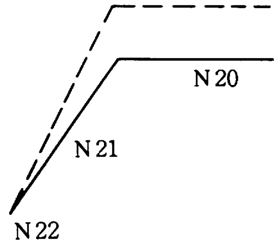
N6 X100. Y200.;
N7 G4 X1000;
N8 F100;
N9 S500;
N10 M4;
N11 X100.;

(N7 – N11: Block without movement)

This may cause cutting.



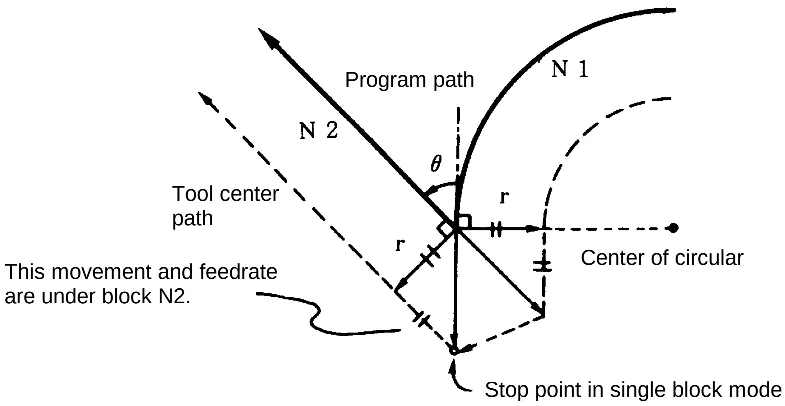
(3) Compensation cancel alone

(a) G1 precedes the block containing G40:	**(b) G0 precedes the block containing G40:**
: N10 G1 X__ Y__; N11 G40; N12 G0 X__ Y__;  	: N20 G1 X__ Y__; N21 G0 X__ Y__; N22 G40; 
Compensation is canceled by a movement command following G40.	Compensation is canceled by a G0 command before G40.

(Note) In program (a), if G13.1 is commanded after G40 without a movement command, cancellation is done at block G13.1.

Corner movement

When a multiple number of offset vectors are created at the joints between movement command blocks, the tool will move in a straight line between those vectors. This action is called corner movement.
When the vectors do not match, the tool moves in order to turn the corner although this movement belongs to the next block. Consequently, operation in the single block mode will execute the previous block + corner movement as a single block and the remaining joining movement + next block will be executed as a single block in the following operation.



6.8.11 Interference check

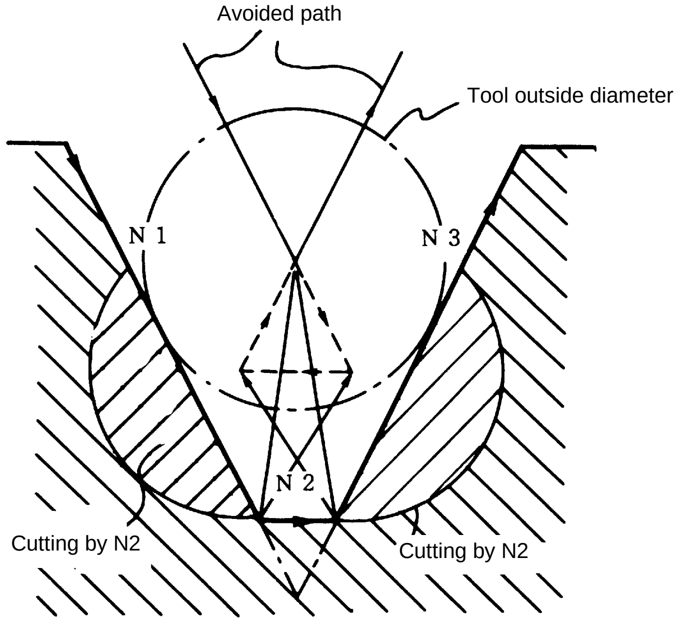
Function and purpose

A tool, whose tool radius has been compensated under the tool radius compensation function by the usual 2-block pre-reading, may sometimes cut into the workpiece. This is known as interference. An interference check is the function which prevents such interference from occurring.
The types of interference check are indicated below, and each can be selected for use by parameter.

Function	Parameter	Operation
Interference check alarm function	Rad compen intrf byp : OFF Interference check invalid signal OFF	Operation stops with a program error before execution of the block causing cutting.
Interference check avoiding function	Rad compen intrf byp : ON Interference check invalid signal OFF	The tool path is changed to prevent cutting from occurring.

(Example)

```
(G41)

N1 G90 G1 X50. Y–100.;
N2 X70. Y–100.;
N3 X120. Y0;
```

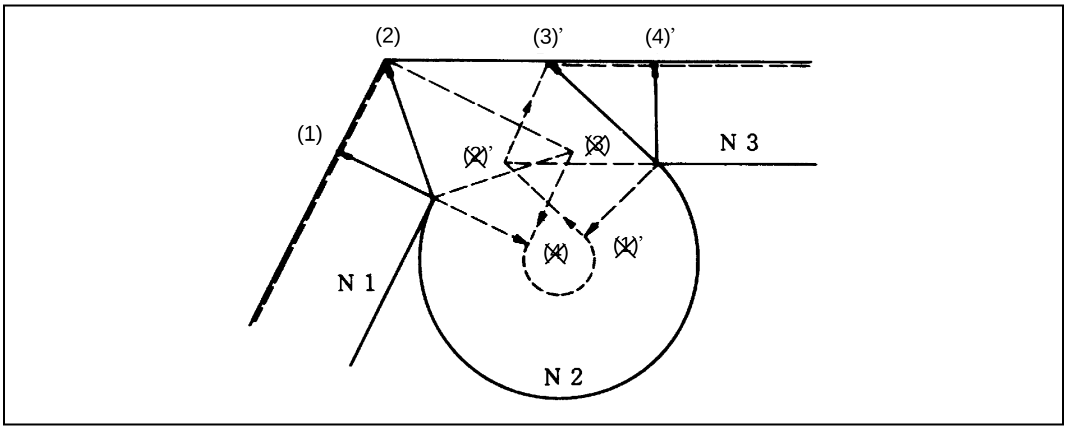


(1) With alarm function

An alarm is given before N1 is executed. The buffer correction function can thus be used to change N1 to the following, enabling machining to continue:
N1 G1 X20. Y-40.;

(2) With avoidance function

The intersection of N1 and N3 is calculated to create interference avoidance vectors.



Examples of interference check:

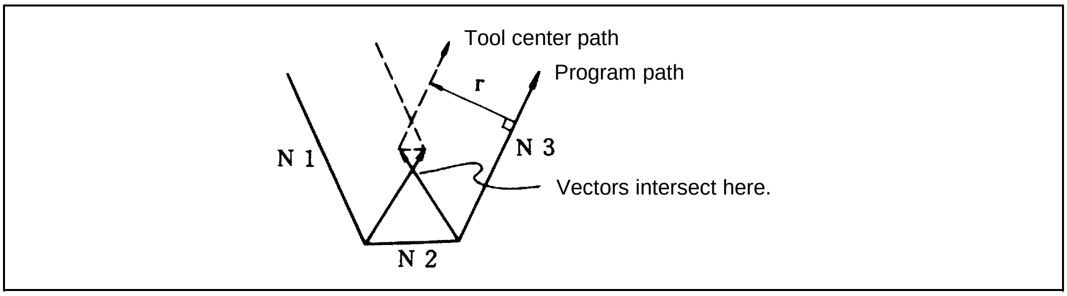
Vector (1) (4)’ check → No interference
↓
Vector (2) (3)’ check → No interference
↓
Vector (3) (2)’ check → Interference → Vectors (3) (2)’ deleted
↓
Vectors (4) (1)’ deleted

As a result of the above processing, vectors (1) (2) (3)’ , and (4)’ remain as valid, and operation is done with the path connecting these vectors as the interference avoidance path.

Detailed description

(1) Conditions regarded as interference

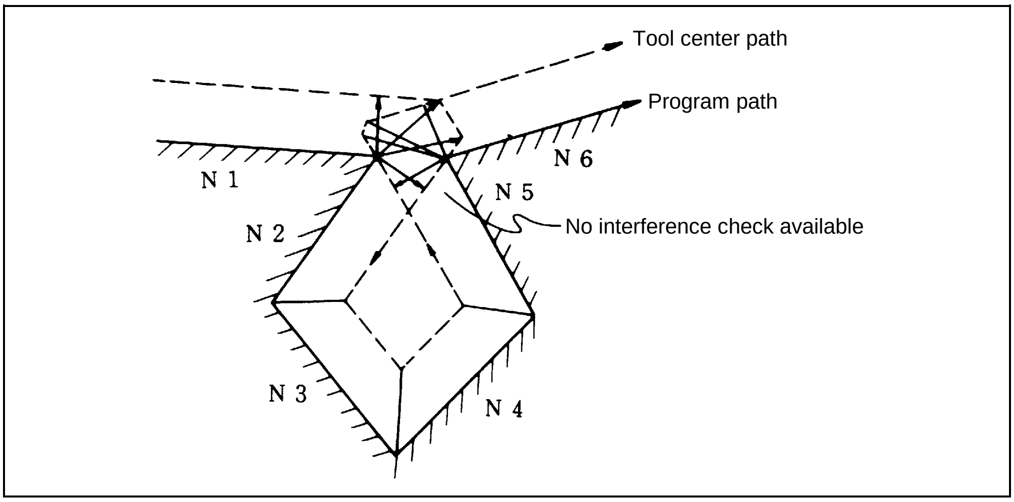
With three blocks containing movement commands of five pre-read blocks, interference is regarded as occurring if the compensation calculation vectors, which have been created at contact of movement commands, intersect each other.



(2) Interference check is not available when:

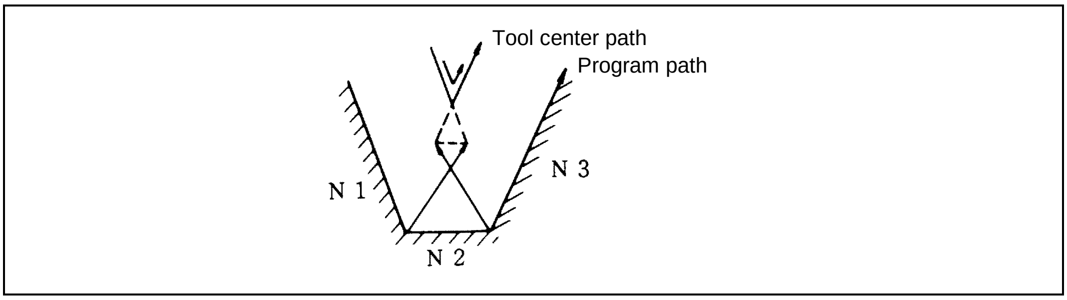
(a) Three blocks containing movement commands cannot be pre-read (three or more blocks of five pre-read blocks do not contain movement commands).

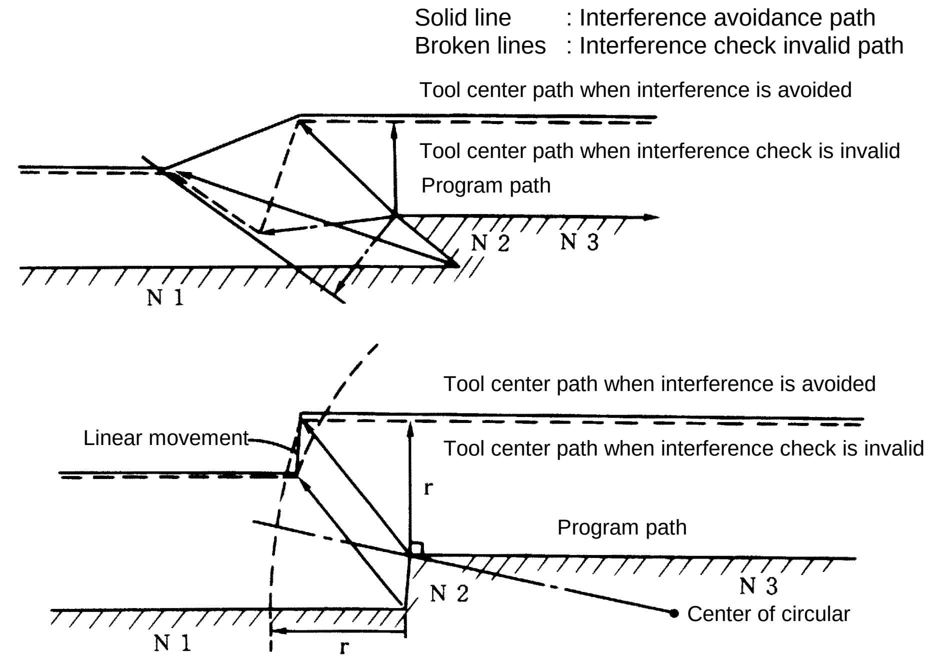
(b) Interference occurs in the fourth or subsequent block containing movement commands.



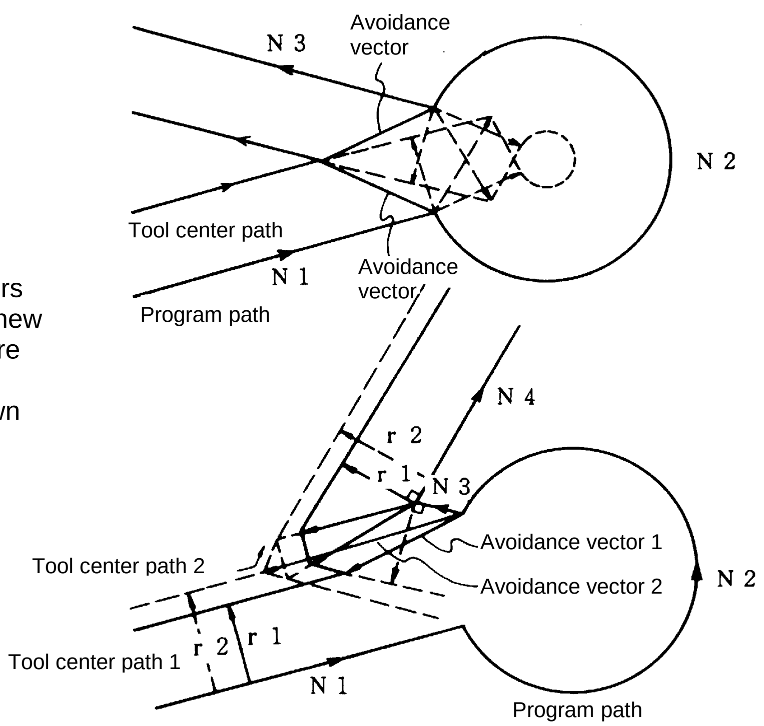
(3) Operation during interference avoidance

If the interference avoidance function is available, the tool moves as follows.

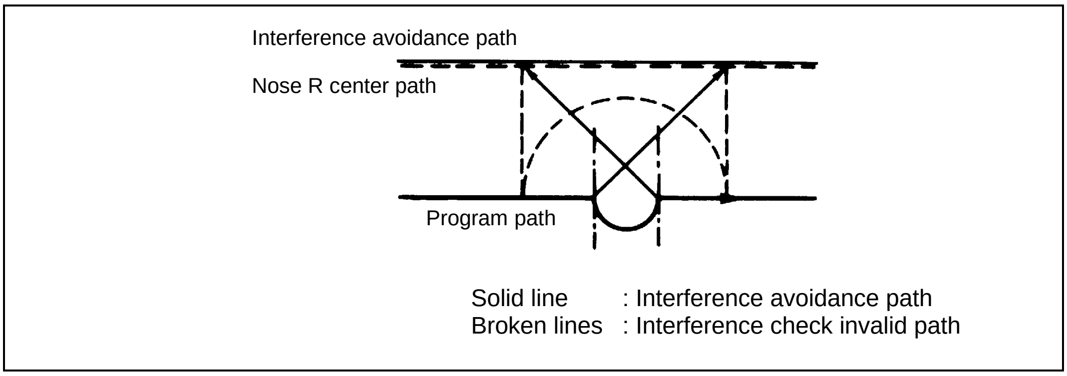




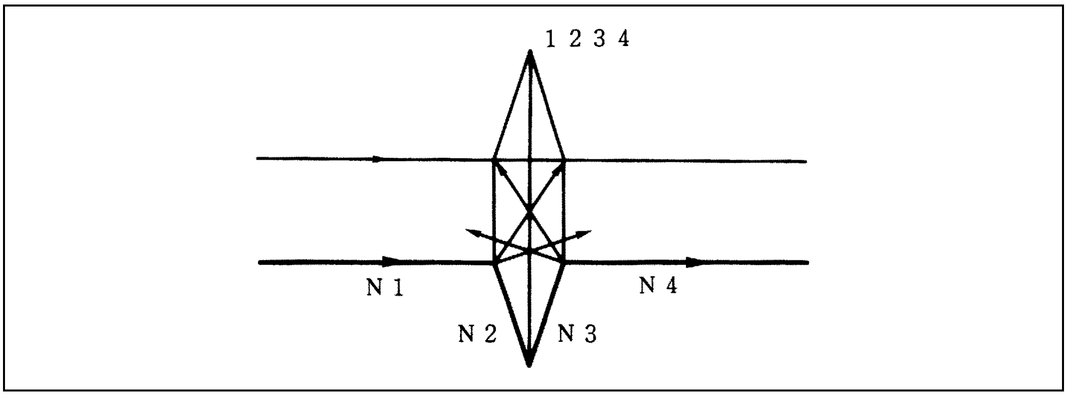
If all interference avoidance line vectors have been deleted, new avoidance vectors are created to avoid interference as shown in the figure at right.



In the figure below, the groove is left uncut.



In the figure below, the tool moves in the opposite direction at N2. After N1 is executed, program error "P153" occurs.

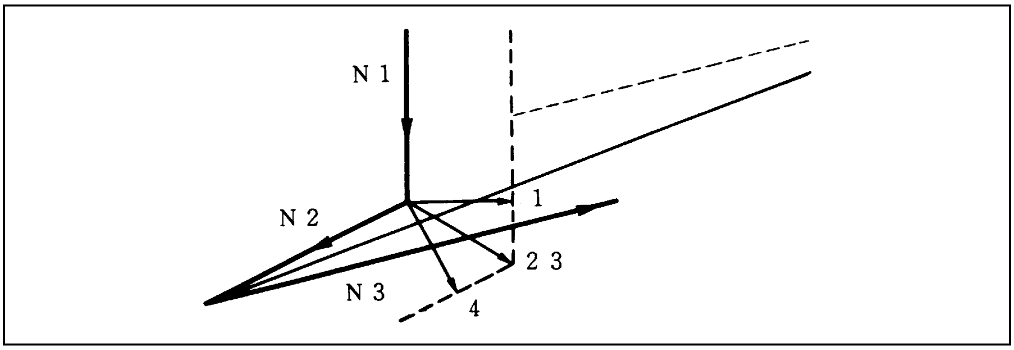


Interference check alarm

An interference check alarm occurs under the following conditions.

(1) With the interference check alarm function selected

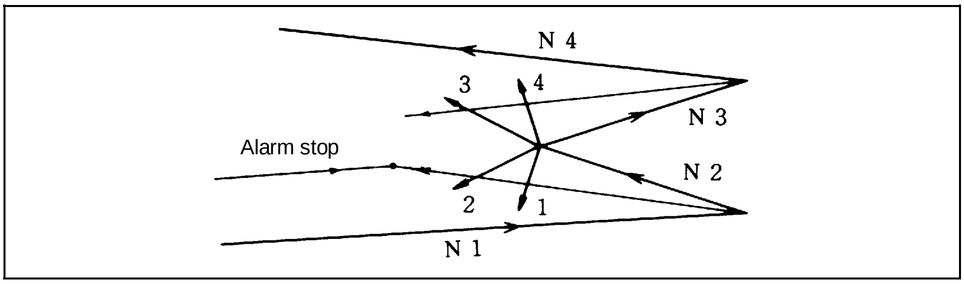
(a) All vectors are deleted at the end point of the current block.
As shown in the figure, if vectors 1 to 4 are all deleted at the end point of the N1 block, program error "P153" results prior to N1 execution.



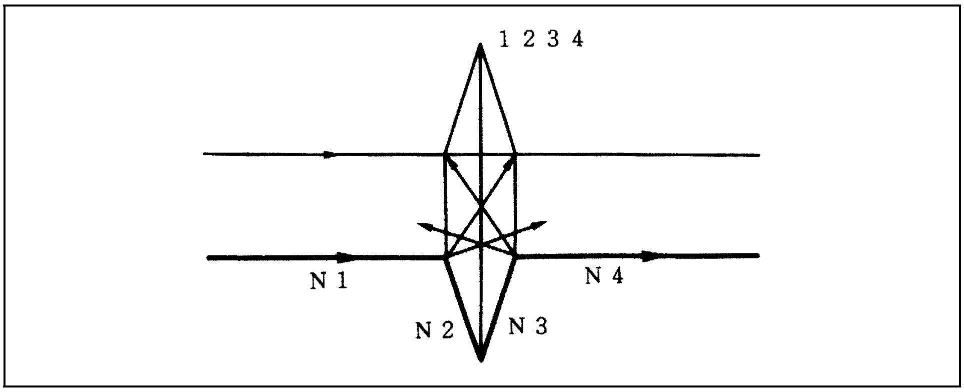
(2) With the interference check avoidance function selected

(b) There are valid vectors at the end point of the following block though all vectors at the end point of the current block were deleted.

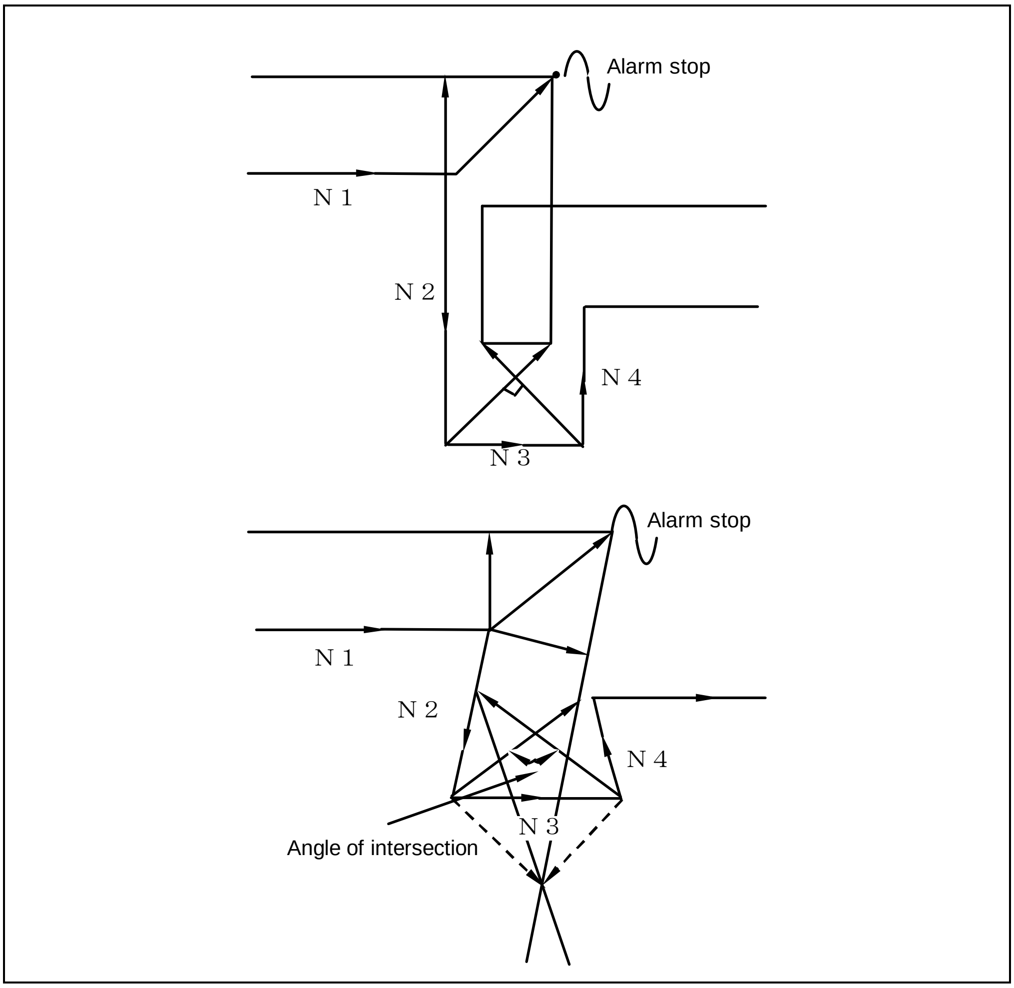
(i) In the figure, if N2 interference check is conducted, the N2 end point vectors are all deleted but the N3 end point vectors are regarded as valid.
This causes program error "P153" at the N1 end point.



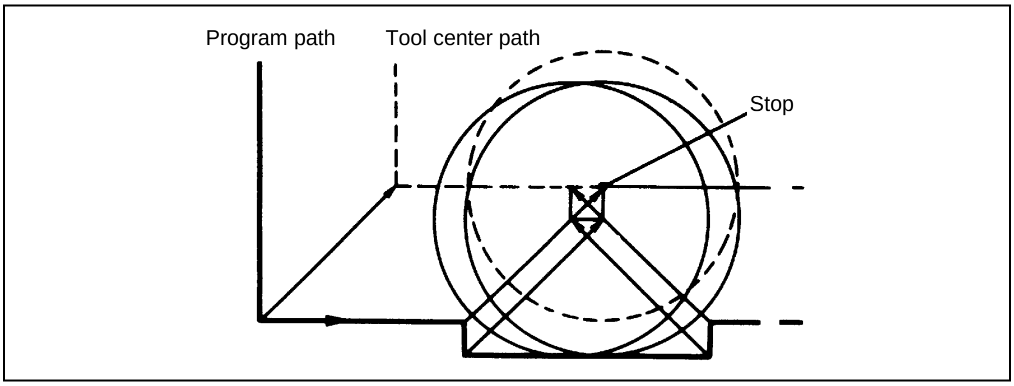
(ii) In the figure, the tool moves in the opposite direction at N2.
This causes program error "P153" after N1 execution.

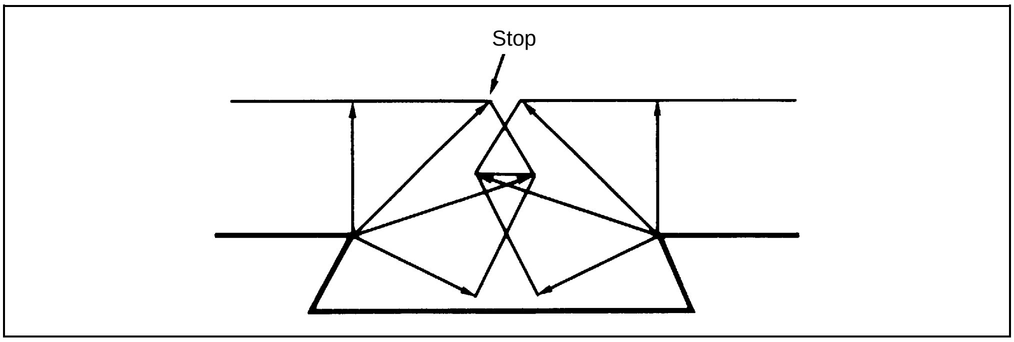


(c) The avoidance vectors cannot be created.
As shown in the figure, even when the conditions for creating avoidance vectors are met, it may still be impossible to create avoidance vectors or the avoidance vectors may interfere with N3. Program error "P153" thus occurs at the N1 end point if the vectors intersect at an angle of 90° or more.



(d) The program advance direction and the advance direction after compensation are reversed.
An interference may be assumed when no interference occurs actually if grooves running in parallel with narrower width between the two than the tool diameter or a bottom-widened groove is programmed.





7. FEED FUNCTIONS

7.1 Rapid Traverse Rate

Function and purpose

The rapid traverse rate can be set independently for each axis. The available speed range depends on the input setting unit. When the input setting unit becomes 1/10, the available speed range also becomes 1/10. For instance, when the input setting unit is 0.1μm, the minimum and maximum speeds become 1/10 of those for the input setting unit of 1μm. Refer to the specifications manual for details of setting ranges. The maximum speed is further limited depending on the machine specifications.
Refer to the specifications manual of the machine for the rapid traverse rate settings.

Two paths are available during positioning: the interpolation type where the area from the start point to the end point is linearly interpolated or the non-interpolation type where movement proceeds at the maximum speed of each axis. The type is selected by parameter. The positioning time is the same in each case.

(Note) Rapid traverse override
Override can be applied by an external input signal for both manual and automatic rapid traverse. There are 2 types which are determined by the PLC specifications.
Type 1: Override in 4 steps: 1%, 25%, 50% and 100%
Type 2: Override in 1% steps from 0% to 100%.

7.2 Cutting Feedrate

Function and purpose

The cutting feedrate is assigned with address F and 8 digits (F8-digit direct designation).
The F8 digits are assigned with a decimal point for a 5-digit integer and a 3-digit fraction. The cutting feedrate is valid for the G01, G02, G03 and G33 commands

Examples (asynchronous feed)

	Feedrate	
G1 X100. Z100. F200;	200.0mm/min	F200. or F200.000 gives the same rate.
G1 X100. Z100. F123.4;	123.4mm/min	
G1 X100. Z100. F56.789;	56.789mm/min	

(Note 1) A program error "P62" results when there is no F command in the first cutting command (G01, G02, G03, G33) after the power has been turned ON.

(Note 2) When a metric system machine is used with inch commands, the maximum speeds for both rapid traverse and cutting feed are indicated in inches after conversion of values in millimeters.

7.3 Synchronous/Asynchronous Feed; G94, G95

Function and purpose

Using the G95 command, it is possible to assign the feed amount per rotation with an F code. When this command is used, the rotary encoder must be attached to the spindle.

Command format

G94;
G95;

G94	Feed per minute (mm/min) (asynchronous feed)
G95	Feed per rotation (mm/rev) (synchronous feed)

The G95 command is a modal command and so it is valid until the G94 command (feed per minute) is next assigned.

(1) The F code command range is as follows.
The movement amount per spindle rotation with synchronous feed (feed per rotation) is assigned by the F code and the command range is as shown in the table below.

Metric input

Input unit system	B (0.001mm)		C (0.0001mm)	
Command mode	Feed per minute	Feed per rotation	Feed per minute	Feed per rotation
Command address	F (mm/min)	E (mm/rev)	F (mm/min)	E (mm/rev)
Minimum command unit	1 (= 1.000), (1. = 1.000)	1 (= 0.0001), (1. = 1.0000)	1 (= 1.0000), (1. = 1.0000)	1 (= 0.00001), (1. = 1.00000)
Command range	0.001 to 480000.000	0.0001 to 999.9999	0.0001 to 108000.0000	0.00001 to 99.99999

Inch input

Input unit system	B (0.0001inch)		C (0.00001inch)	
Command mode	Feed per minute	Feed per rotation	Feed per minute	Feed per rotation
Command address	F (inch/min)	E (inch/rev)	F (inch/min)	E (inch/rev)
Minimum command unit	1 (= 0.01), (1. = 1.0000)	1 (= 0.000001), (1. = 1.000000)	1 (= 0.01), (1. = 1.00000)	1 (= 0.0000001), (1. = 1.0000000)
Command range	0.0001 to 18897.6370	0.000001 to 99.999999	0.00001 to 4251.96850	0.0000001 to 9.9999999

(2) The effective speed (actual movement speed of machine) under feed per rotation conditions is given in the following formula (Formula 1).

FC = F × N × OVR ············ (Formula 1)

FC	Effective speed (mm/min, inch/min)
F	Commanded speed (mm/rev, inch/rev)
N	Spindle speed (r/min)
OVR	Cutting feed override

When a multiple number of axes have been commanded at the same time, the effective speed FC in Formula 1 applies in the vector direction of the command.

(Note 1) The effective speed (mm/min or inch/min), which is produced by converting the commanded speed, the spindle speed and the cutting feed override into the feedrate per minute, appears as the FC on the "Monitor 1" screen of the setting display unit.

(Note 2) When the above effective rate exceeds the cutting feed clamp speed, it is clamped at that clamp speed.

(Note 3) If the spindle speed is zero when synchronous feed is executed, operation alarm "105" results.

(Note 4) During machine lock high-speed processing, the rate will be 60,000 mm/min (or 2,362 inch/min, 60,000°/min) regardless of the commanded speed and spindle speed. When high-speed processing is not undertaken, the speed will be the same as for non-machine lock conditions.

(Note 5) Under dry run conditions, asynchronous speed applies and movement results at the externally set speed (mm/min or inch/min).

(Note 6) Whether asynchronous feed (G94) or synchronous feed (G95) is to be established when the power is turned ON or when M02 or M30 is executed can be selected with the parameter "Initial sychr feed" setting.

7.4 Feedrate Designation and Effects on Control Axes

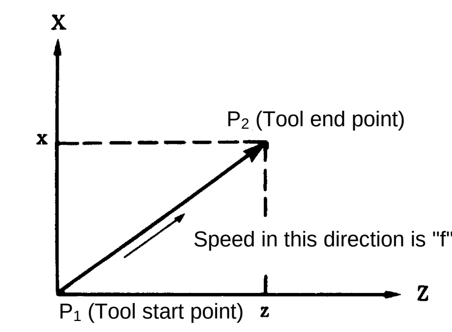
Function and purpose

It has already been mentioned that a machine has various control axes. These control axes can be divided into linear axes which control linear movement and rotation axes which control rotary movement. The feedrate is designed to assign the displacement speed of these axes, and the effect exerted on the tool movement speed which poses problems during cutting differs according to when control is exercised over the linear axes or when it is exercised over the rotation axes.
The displacement amount for each axis is assigned separately for each axis by a value corresponding to the respective axis. The feedrate is not assigned for each axis but assigned as a single value. Therefore, when two or more axes are to be controlled simultaneously, it is necessary to understand how this will work for each of the axes involved.
The assignment of the feedrate is described with the following related items.

When controlling linear axes

Even when only one machine axis is to be controlled or there are two or more axes to be controlled simultaneously, the feedrate which is assigned by the F code functions as a linear speed in the tool advance direction.

(Example) When the feedrate is designated as "f" and linear axes (X and Z) are to be controlled



When only linear axes are to be controlled, it is sufficient to designate the cutting feed in the program. The feedrate for each axis is such that the designated rate is broken down into the components corresponding to the movement amounts.
In the above example:

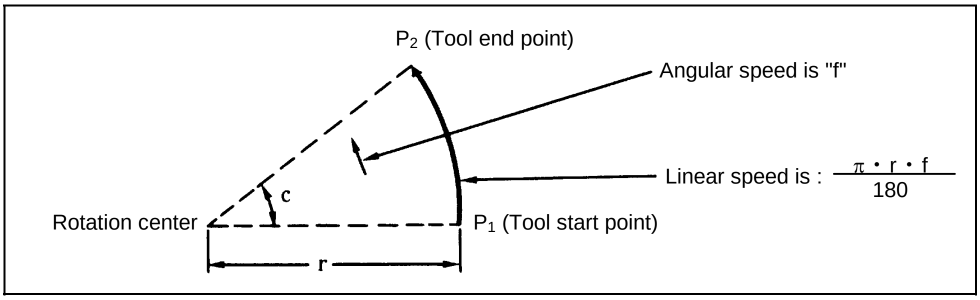
$$\text{Feedrate for X axis} = f \times \frac{x}{\sqrt{x^2 + z^2}}$$

$$\text{Feedrate for Z axis} = f \times \frac{z}{\sqrt{x^2 + z^2}}$$

When controlling rotation axes

When rotation axes are to be controlled, the designated feedrate functions as the rotation speed of the rotation axes or, in other words, as an angular speed.
Consequently, the cutting feed in the tool advance direction, or in other words the linear speed, varies according to the distance between the rotation center and the tool. This distance must be considered when designating the feedrate in the program.

(Example) When the feedrate is designated as "f" and rotation axis (C axis) is to be controlled ("f" units = °/min)



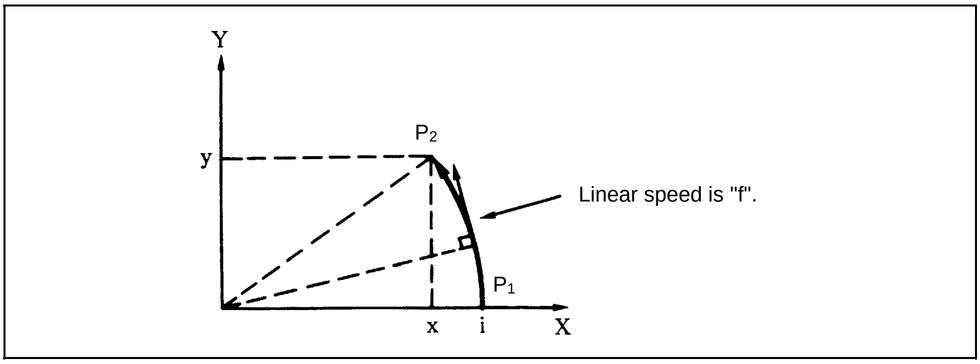
In this case, in order to make the cutting feed (linear speed) in the tool advance direction "fc":

$$fc = f \times \frac{\pi \bullet r}{180}$$

Therefore, the feedrate to be designated in the program must be:

$$f = fc \times \frac{180}{\pi \bullet r}$$

(Note) When the circular interpolation function is used and the tool is moved along the circumference of an arc by the linear control axis, the rate in the tool advance direction, or in other words the tangential direction, will be the feedrate designated in the program.

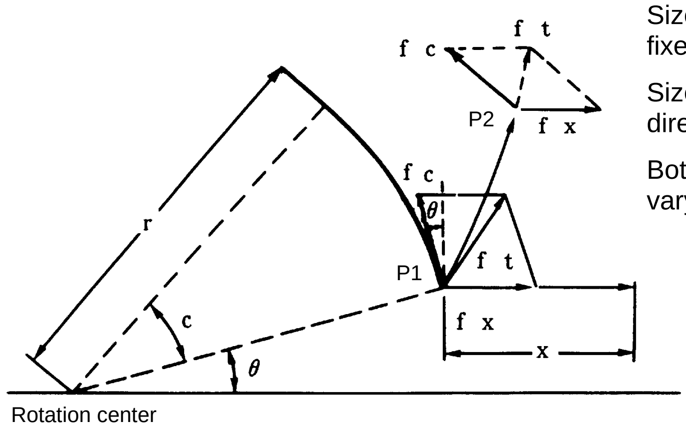


(Example) When the feedrate is designated as "f" and the linear axes (X and Z axes) are to be controlled using the circular interpolation function
In this case, the feedrate of the X and Z axes will change along with the tool movement. However, the combined speed will always be maintained at the constant value "f".

When linear and rotation axes are to be controlled at the same time

The NC unit proceeds in exactly the same way whether linear or rotation axes are to be controlled.
When a rotation axis is to be controlled, the numerical value assigned by the coordinate words (C, H) is the angle and the numerical values assignees by the feedrate (F) are all handled as linear speeds. In other words, 1° of the rotation axis is treated as being equivalent to 1mm of the linear axis.
Consequently, when both linear and rotation axes are to be controlled simultaneously, the components for each axis of the numerical values assigned by F will be the same as for section above (applying "when linear axes are to be controlled"). However, although in this case both the size and direction of the speed components based on linear axis control do not vary, the direction of the speed components based on rotation axis control will change along with the tool movement (their size will not change). This means, as a result, that the combined tool advance direction feedrate with vary along with the tool movement.

(Example) When the feedrate is designated as "f" and linear (X) and rotary (C) axes are to be controlled simultaneously
If the X-axis incremental command value is "x" and the C-axis incremental command value is "c":



Size and direction are fixed for fx.

Size is fixed for fc but direction varies.

Both size and direction vary for ft.

X-axis feedrate (linear speed) "fx" and C-axis feedrate (angular speed) "ω" are expressed as:

$$fx = f \times \frac{x}{\sqrt{x^2 + c^2}} \quad \cdots\cdots \quad (1) \qquad\qquad \omega = f \times \frac{c}{\sqrt{x^2 + c^2}} \quad \cdots\cdots \quad (2)$$

Linear speed "fc" based on C-axis control is expressed as:

$$fc = f \times \frac{\pi \bullet r}{180} \quad \cdots\cdots \quad (3)$$

If the speed in the tool advance direction at start point P_1 is "ft" and the component speeds in the X-axis and Y-axis directions are "ftx" and "fty," respectively, then these can be expressed as:

$$ftx = -r\sin\left(\frac{\pi}{180}\theta\right) \times \frac{\pi}{180}\omega + fx \quad \cdots\cdots \quad (4)$$

$$fty = -r\cos\left(\frac{\pi}{180}\theta\right) \times \frac{\pi}{180}\omega \quad \cdots\cdots\cdots\cdots \quad (5)$$

Where r is the distance between rotation center and tool (in mm units), and
θ is the angle between the P_1 point and the X axis at the rotation center (in ° units).
The combined speed "ft" according to formulas (1), (2), (3), (4) and (5) is:

$$\begin{aligned} ft &= \sqrt{ftx^2 + fty^2} \\ &= f \times \frac{\sqrt{x^2 - x \bullet c \bullet r\sin\left(\frac{\pi}{180}\theta\right)\frac{\pi}{90} + \left(\frac{\pi \bullet r \bullet c}{180}\right)^2}}{\sqrt{x^2 + c^2}} \quad \cdots\cdots \quad (6) \end{aligned}$$

Consequently, feedrate "f" designated by the program must be as follows:

$$f = ft \times \frac{x^2 + c^2}{\sqrt{x^2 - x \bullet c \bullet r\sin\left(\frac{\pi}{180}\theta\right)\frac{\pi}{90} + \left(\frac{\pi \bullet r \bullet c}{180}\right)^2}} \quad \cdots\cdots \quad (7)$$

"ft" in formula (6) is the speed at the P_1 point and the value of θ changes as the C axis rotates, which means that the value of "ft" will also change. Consequently, in order to keep the cutting feed "ft" as constant as possible, the rotation angle which is designated in one block must be reduced to as low as possible and the change width in the θ value must be minimized.

7.5 Thread Cutting Leads

Function and purpose

F7-digit or E8-digit commands for thread leads can be issued for the thread cutting mode (G33, G34, G76, G78 commands).
The thread lead command range is 0.0001 to 999.9999mm/rev (F7 digits) or 0.0001 to 999.99999mm/rev (E8 digits) (with input unit of µm).

Thread cutting metric input

Input unit system	**A (0.01mm)**			**B (0.001mm)**			**C (0.0001mm)**		
Command address	F (mm/rev)	E (mm/rev)	E (threads/ inch)	F (mm/rev)	E (mm/rev)	E (threads/ inch)	F (mm/rev)	E (mm/rev)	E (threads/ inch)
Minimum command unit	1 (= 0.001), (1.=1.000)	1 (= 0.0001), (1.=1.0000)	1 (= 1), (1.=1.0)	1 (= 0.0001), (1.=1.0000)	1 (= 0.00001), (1.=1.00000)	1 (= 1), (1.=1.00)	1 (= 0.00001), (1.=1.00000)	1 (=0.000001), (1.= 1.000000)	1 (= 1), (1.=1.000)
Command range	0.001 to 9999.999	0.0001 to 9999.9999	0.1 to 9999999.9	0.0001 to 999.9999	0.00001 to 999.99999	0.01 to 999999.99	0.00001 to 99.99999	0.000001 to 99.999999	0.001 to99999.999

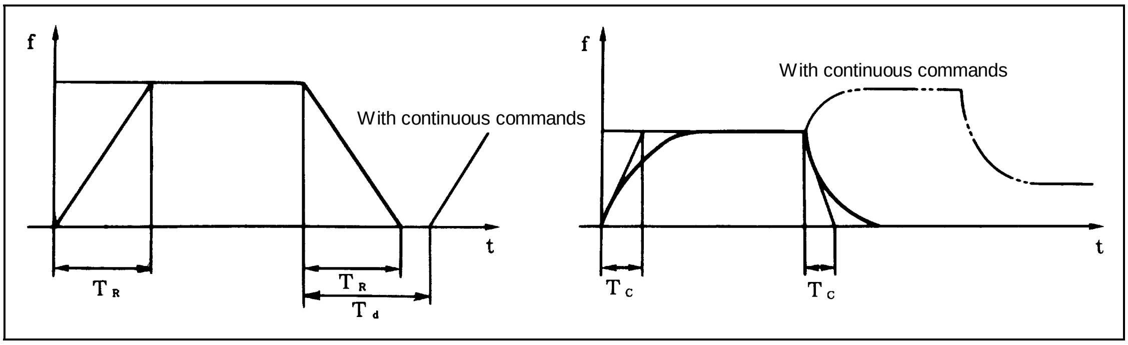
Thread cutting inch input

Input unit system	**A (0.001inch)**			**B (0.0001inch)**			**C (0.00001inch)**		
Command address	F (inch/rev)	E (inch/rev)	E (threads/ inch)	F (inch/rev)	E (inch/rev)	E (threads/ inch)	F (inch/rev)	E (inch/rev)	E (threads/ inch)
Minimum command unit	1 (=0.00001), (1.=1.00000)	1 (=0.000001), (1.=1.000000)	1 (= 1), (1.=1.000)	1(=0.000001), (1.=1.000000)	1 (=0.0000001), (1.= 1.0000000)	1 (= 1), (1.=1.0000)	1 (=0.0000001), (1.= 1.0000000)	1(= 0.00000001), (1.= 1.00000000)	1(=1), (1.=1.00000)
Command range	0.00001 to 999.99999	0.000001 to 99.999999	0.001 to 99999.999	0.000001 to 99.999999	0.0000001 to 9.9999999	0.0001 to 9999.9999	0.0000001 to 9.9999999	0.00000001 to 0.99999999	0.001 to 999.99999

7.6 Automatic Acceleration/Deceleration

Function and purpose

The rapid traverse and manual feed acceleration/deceleration pattern is linear acceleration and linear deceleration. Time constant T_R can be set independently for each axis using parameters in 1ms steps from 1 to 500ms.
The cutting feed (not manual feed) acceleration/deceleration pattern is exponential acceleration/ deceleration. Time constant Tc can be set independently for each axis using parameters in 1ms steps across a range from 1 to 500ms. (Normally, the same time constant is set for all axes.)



Rapid traverse acceleration/ deceleration pattern
(T_R = Rapid traverse time constant)
(T_d = Deceleration check time)

Cutting feed acceleration/ deceleration pattern
(T_c = Cutting feed time constant)

With rapid traverse and manual feed, the following block is executed after the command pulse of the present block has become "0" and the tracking error of the acceleration/deceleration circuit has become "0". However, with cutting feed, the following block is executed as soon as the command pulse of the present block becomes "0" although an external signal (error detect) can detect that the tracking error of the acceleration/deceleration circuit has reached "0" and the following block can be executed.
When the in-position check has been made valid (selected by machine parameter) during the deceleration check, it is first confirmed that the tracking error of the acceleration/deceleration circuit has reached "0", then it is checked that the position deviation is less than the parameter setting value, and finally the following block is executed.
It depends on the machine as to whether the error detect function can be activated by a switch or M function and so reference should be made to the instruction manual issued by the machine manufacturer.

7.7 Rapid Traverse Constant Inclination Acceleration/Deceleration

Function and purpose

This function allows acceleration/deceleration to be carried out at a constant inclination during linear acceleration/acceleration in the rapid traverse mode.
The constant inclination acceleration/deceleration method is effective in improving the cycle time compared to conventional methods.

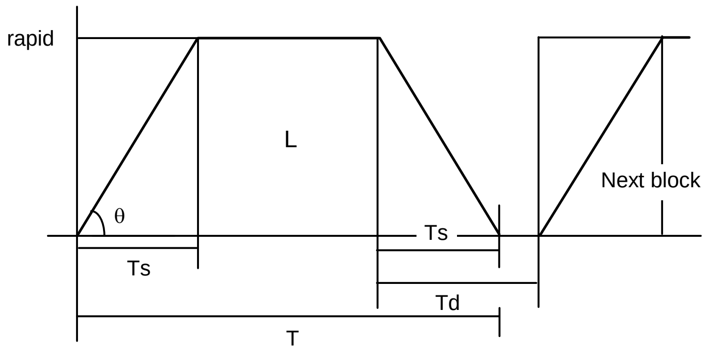
Detailed description

(1) Rapid traverse constant inclination acceleration/deceleration is effective only when rapid traverse is commanded. This is also only effective when the rapid traverse command's acceleration/deceleration mode is linear acceleration and linear deceleration.

Rapid traverse constant inclination acceleration/deceleration selection parameters
(Machine parameter "SP-1")
SP-1/bit1 = 0: Rapid traverse constant time acceleration/deceleration
1: Rapid traverse constant inclination acceleration/deceleration

(2) The acceleration/deceleration pattern for when rapid traverse constant inclination acceleration/ deceleration is carried out is shown below.

<When interpolation distance is longer than acceleration/deceleration distance>



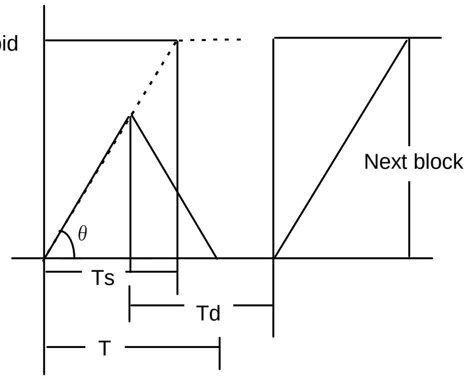
rapid : Rapid traverse rate
Ts : Acceleration/deceleration time constant
Td : Command deceleration check time
θ : Acceleration/deceleration inclination
T : Interpolation time
L : Interpolation distance

$$T = \frac{L}{rapid} + Ts$$

$$Td = Ts + (0 \text{ to } 14ms)$$

$$\theta = \tan^{-1}\left(\frac{rapid}{Ts}\right)$$

<When interpolation distance is shorter than acceleration/deceleration distance>



rapid : Rapid traverse rate
Ts : Acceleration/deceleration time constant
Td : Command deceleration check time
θ : Acceleration/deceleration inclination
T : Interpolation time
L : Interpolation distance

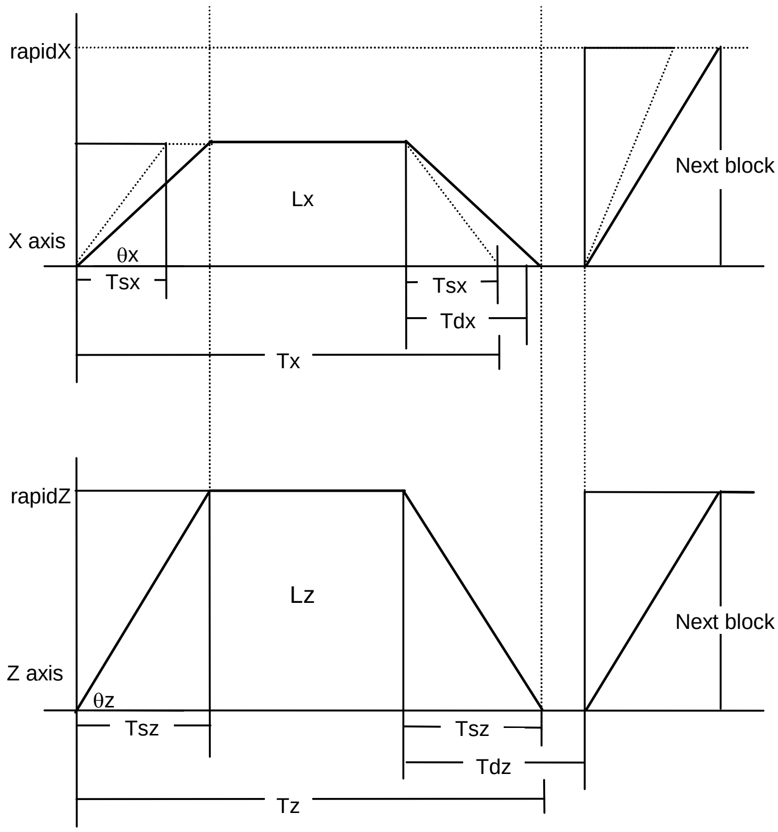
$$T = 2 \times \sqrt{Ts \times L/rapid}$$

$$Td = \frac{T}{2} + (0 \text{ to } 14\text{ms})$$

$$\theta = \tan^{-1}\left(\frac{rapid}{Ts}\right)$$

(3) When 2-axis simultaneous interpolation (linear interpolation) is carried out during rapid traverse constant inclination acceleration/deceleration, the longest time will be applied among the acceleration/deceleration times for all axes commanded at the same time. Note that the time is determined by the rapid traverse rate, rapid traverse acceleration/ deceleration time constant, and interpolation distance.
Thus, linear interpolation will be carried out even when the acceleration/deceleration time constants for each axis differ.

<For 2-axis simultaneous interpolation (for linear interpolation Tsx <Tsz, Lx ≠ Lz)>



When Tsz > Tsx, Tdz > Tdx will be established, and Td = Tdz in this block.

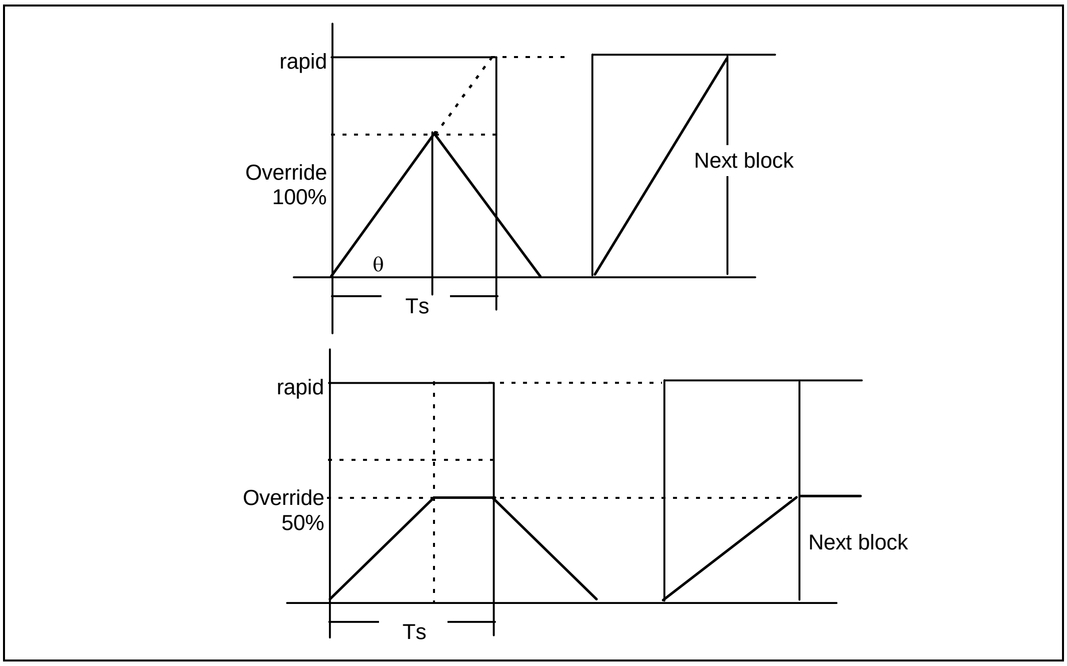
(4) As for the time required for checking the commanded deceleration during rapid traverse constant inclination acceleration/deceleration, the longest time will be applied among the checking times for all axes. Note that the checking time is determined by the rapid traverse acceleration/deceleration time constant and interpolation distance.

(5) The G0 (rapid traverse command) program format for rapid traverse constant inclination acceleration/deceleration is the same as when this function is invalid (constant time acceleration/deceleration).
This function is valid only when G0 (rapid traverse) is commanded.

Relation with other functions

(1) Relation with override and dry run

If the rapid traverse rate changes due to override or dry run, etc., the acceleration/deceleration inclination will change, and constant inclination acceleration/deceleration will not take place.



7.8 Speed Clamp

Function and purpose

This function exercises control over the actual cutting feedrate in which override has been applied to the cutting feedrate command so that the speed clamp value which has been preset independently for each axis is not exceeded.

(Note) Speed clamping is not applied to synchronous feed and thread cutting.

7.9 Exact Stop Check; G09

Function and purpose

In order for roundness to be prevented during corner cutting and for machine shock to be alleviated when the tool feedrate changes suddenly, there are times when it is desirable to start the commands in the following block once the in-position state after the machine has decelerated and stopped or the elapsing of the deceleration check time has been checked. The exact stop check function is designed to accomplish this purpose.
The "inpos" machine parameter among the basic specifications parameters enables control to be exercised either by the deceleration check time or in-position state. The in-position state is valid when "inpos" is set to "1".
The in-position width is set into parameter "ZRZ" on the Servo param screen by the machine manufacturer.

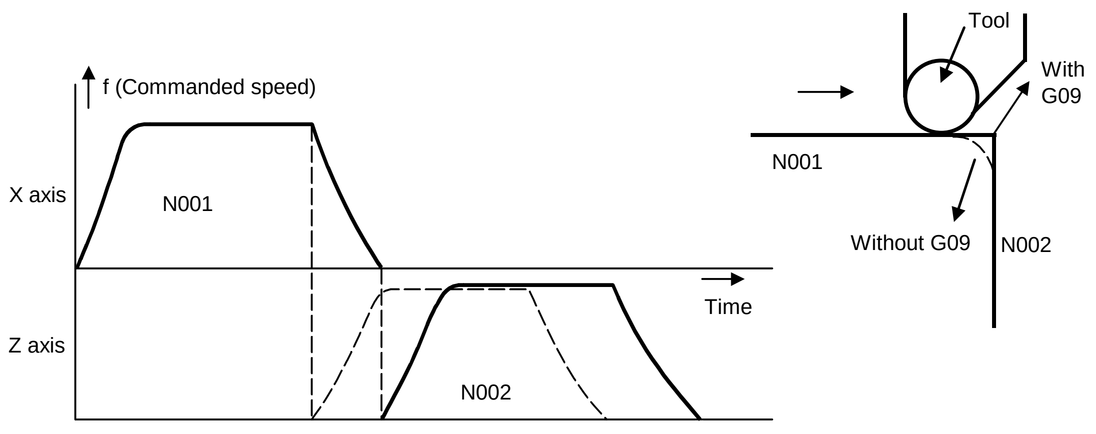
Command format

G09 G01 (G02, G03);

The exact stop check command G09 has an effect only with the cutting command (G01 to G03) in its particular block.

Example of program

N001 G09 G01 X100.000 F150;	The commands in the following block are started once the deceleration check time or in-position state has been checked after the machine has decelerated and stopped.
N002 Z100.000;	



Solid line indicates speed pattern with G09 command.
Broken line indicates speed pattern without G09 command.

Fig. 1 Exact stop check result

Detailed description

(1) With continuous cutting feed

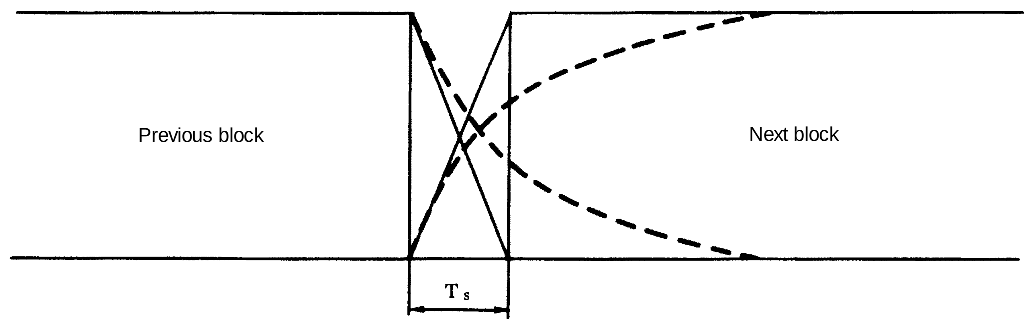


Fig. 2 Continuous cutting feed command

(2) With cutting feed in-position check

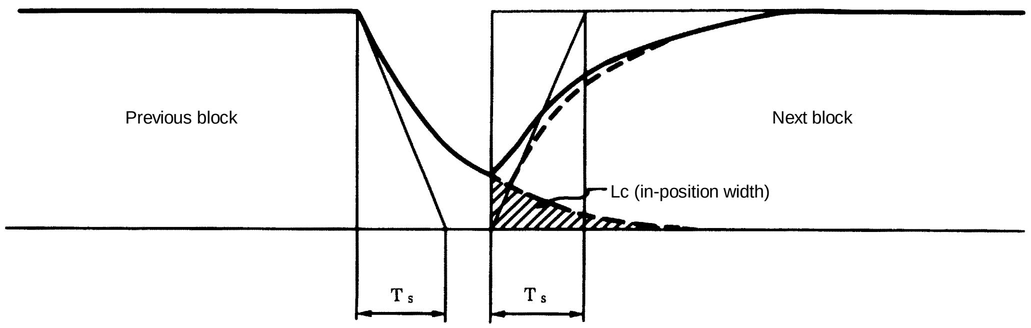
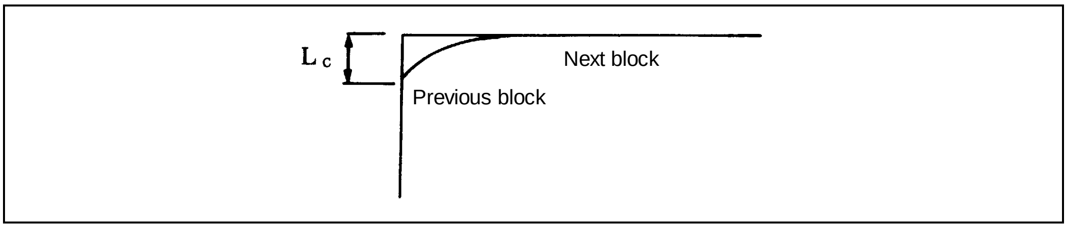


Fig. 3 Block joint with cutting feed in-position check

In Figs. 2 and 3:

Ts = Cutting feed acceleration/deceleration time constant
Lc = In-position width

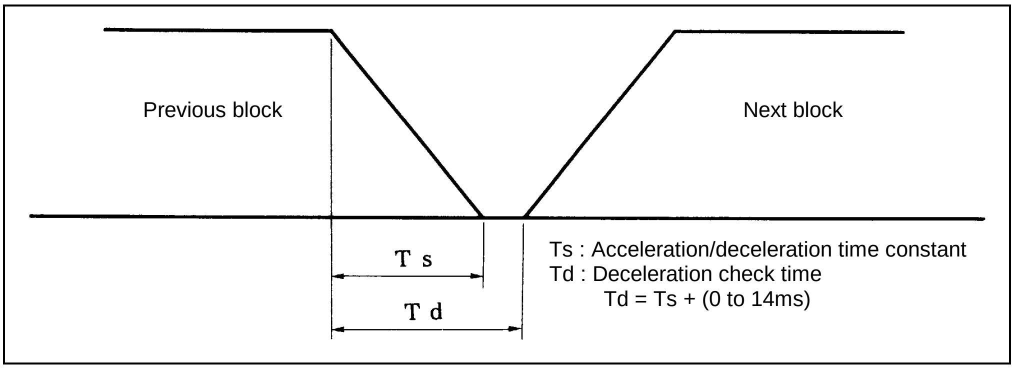
As shown in Fig. 3, the in-position width "Lc" can be set into the "ZRZ" servo parameter as the remaining distance (shaded area in Fig. 3) of the previous block when the next block is started.
The setting unit for the "ZRZ" servo parameter is 0.001mm or 0.0001inch.
The in-position width is designed to reduce the roundness at the workpiece corners to below the constant value.



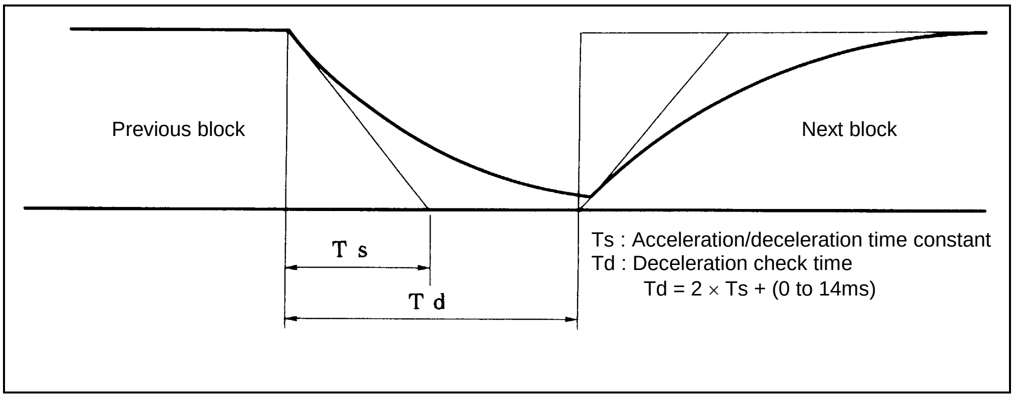
To eliminate corner roundness, set the "ZRZ" servo parameter to zero and perform an in-position check or assign the dwell command (G04) between blocks.

(3) With deceleration check

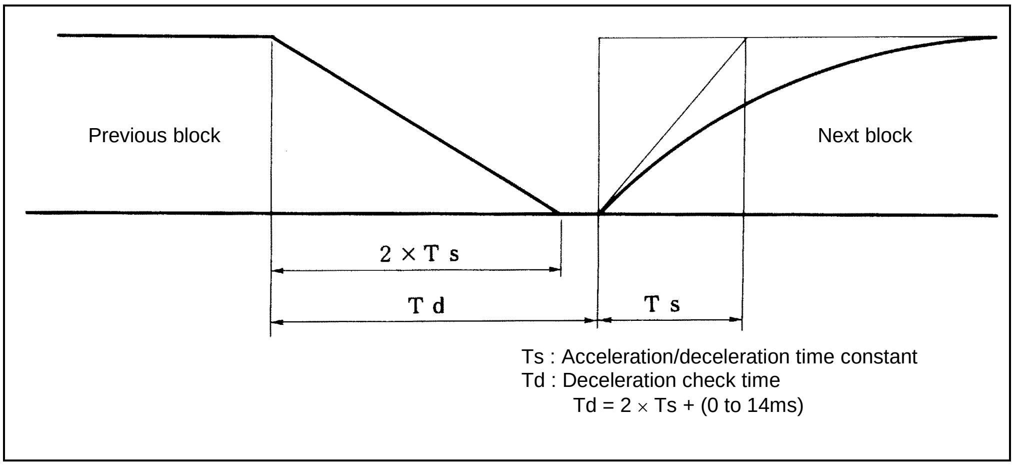
(a) With linear acceleration/deceleration



(b) With exponential acceleration/deceleration



(c) With exponential acceleration/linear deceleration



The time required for the deceleration check during cutting feed is the longest among the cutting feed deceleration check times of each axis determined by the cutting feed acceleration/deceleration time constants and by the cutting feed acceleration/ deceleration mode of the axes commanded simultaneously.

(Note 1) To execute exact stop check in a fixed cycle cutting block, insert command G09 into the fixed cycle subprogram.

7.10 Exact Stop Check Mode; G61

Function and purpose

Exact stop check with G09 confirms the in-position state only for that block, while G61 functions as a modal. Thus, cutting commands (G01 to G03) issued after G61 will all decelerate at the end point of that block, and the in-position state will be checked.
G61 is canceled with the cutting mode command (G64).

Command format

G61;

7.11 Cutting Mode; G64

Function and purpose

In this mode, the next block is successively executed instead of decelerating and stopping between the cutting feed blocks such as with the exact stop check mode (G61). The cutting mode (G64) is entered when the power is turned ON. The cutting mode command is canceled by the exact stop check mode command (G61).

Command format

G64;

7.12 Feed Forward Control

Function and purpose

This function is used to decrease the difference in continuous speed that occurs during servo position loop control.

Detailed description

<Feed forward control>

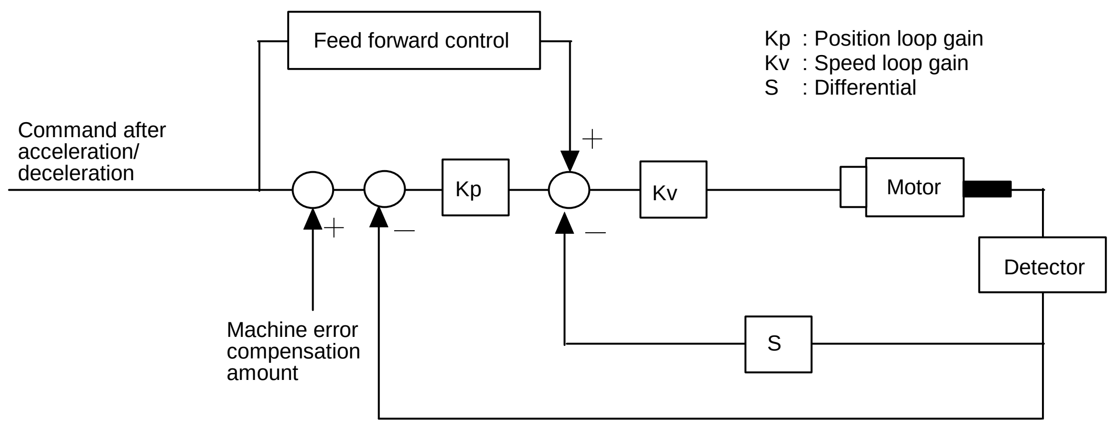
Generally with machine tools, the movement path error at the machine end in respect to the commanded path is caused by the following three accuracies.

(1) Machine static accuracy (assembly accuracy/thermal displacement)
(2) Machine dynamic accuracy (machine displacement caused by movement)
(3) Control system accuracy (error caused by NC control delay)

The (3) control system accuracy is a factor for the following two errors.

- Error caused by smoothing circuit droop amount because of NC control acceleration/ deceleration
- Continuous speed error caused by servo position loop control

The feed forward control function improves the speed error caused by the servo position loop control, which is one of the errors caused by the control system accuracy. (Refer to following diagram.)



Feed forward control

Feed forward parameters (machine parameters "G0fwdg" "fwd-g")
G0fwdg : G0 feed forward gain 0 to 200%
fwd-g : G1 feed forward gain 0 to 200%

Restrictions

(1) Feed forward control is valid for a movement command during automatic operation.
Note that this control will not function on axes for which skip, dog-type zero point return, synchronous tap or hobb machining is being carried out.

(2) If the machine vibrates when the feed forward gain is set to 100%, the feed forward gain must be lowered or the servo system must be adjusted.

8. DWELL

The G04 command can delay the start of the next block.

8.1 Dwell Per Second; (G94) G04

Function and purpose

When the dwell command is given in the feed per minute mode (G94), the execution of the next block is delayed for the designated time.

Command format

G94 G04 X/U__; or G94 G04 P__;

Command unit: 0.001s
Decimal point commands are invalid for address P and if such commands are assigned, everything following the decimal point will be ignored.

Detailed description

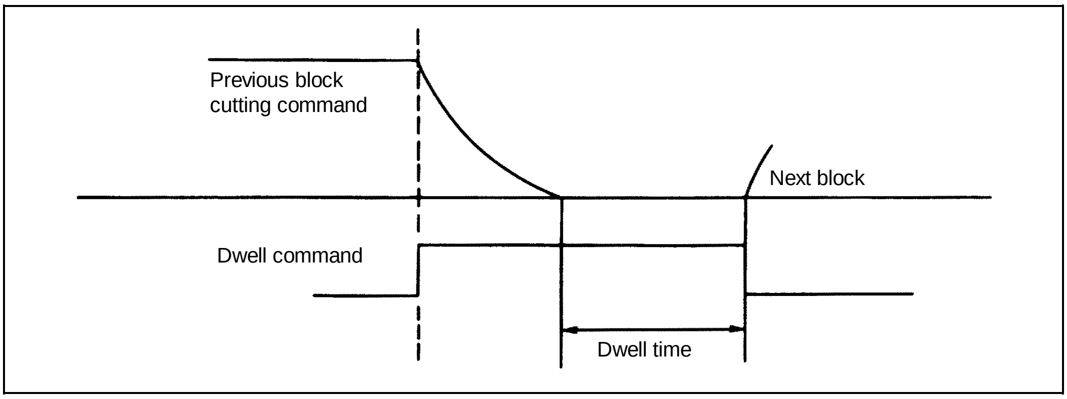
(1) The table below lists the dwell time.

Input setting unit	Command range based on address X, U	Command range based on address P
0.01/0.001mm	0.001 to 99999.999 (s)	1 to 99999999 (×0.001s)
0.0001inch	0.001 to 99999.999 (s)	1 to 99999999 (×0.001s)

(2) When a cutting command is in the previous block, the dwell command starts calculating the dwell time after the machine has decelerated and stopped.
When it is commanded in the same block as an M, S, T or B command, the calculation starts simultaneously.

(3) Dwell is valid even for machine lock but it can be completed immediately by the control parameter "Machine lock rapid".

(4) The per-second dwell can be established regardless of the G94 or G95 mode by means of the control parameter "G04 time fixed".



Example of program

(1) **When input setting unit is 0.01mm, 0.001mm or 0.0001inch**

(Example 1)

G04 X500;	0.5s dwell time

(Example 2)

G04 X5000;	5s dwell time

(Example 3)

G04 X5.;	5s dwell time

(Example 4)

G04 P5000;	5s dwell time

(Example 5)

G04 P12.345;	0.012s dwell time

(2) **When the time designation has been inserted prior to G04 with an input setting unit of 0.0001 inch**

(Example 6)

X5. G04;	50s dwell time (this is equivalent to X50000 G04)

8.2 Dwell Per Rotation; (G95) G04

Function and purpose

When the G04 command is issued in the feed per rotation mode (G95), the spindle first rotates by the designated speed and then the next block is executed.

Command format

G95 G04 X/U__; or G95 G04 P__;

Command unit: 0.001rev
Decimal point commands are invalid for address P and if such commands are assigned, everything following the decimal point will be ignored.

Detailed description

(1) The table below lists the dwell speeds.

Input setting unit	Command range based on address X, U	Command range based on address P
0.01/0.001mm	0.001 to 99999.999 (rev)	1 to 99999999 (× 0.001rev)

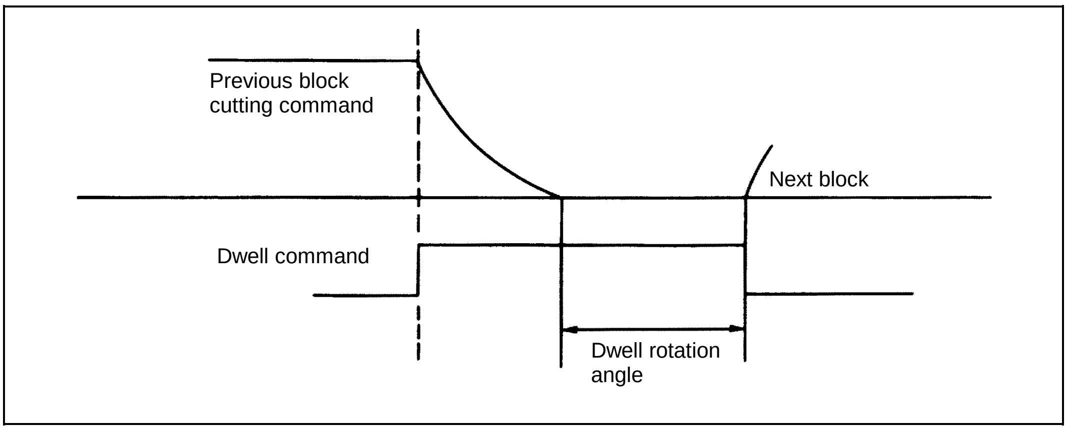
(2) When a cutting command is in the previous block, the dwell command starts controlling the dwell speed after the machine has completed its deceleration. If it is commanded in the same block as an M, S, T or B command, the control is started simultaneously.

(3) Dwell is valid even for machine lock but it can be completed immediately by the control parameter "Machine lock rapid".

(4) Dwell also stops during spindle stop. It is resumed when the spindle starts to rotate again.

(5) The per-second dwell can be fixed by means of the control parameter "G04 time fixed".

(6) This function cannot be used unless a position detection encoder has been mounted on the spindle.



9. MISCELLANEOUS FUNCTIONS

9.1 Miscellaneous Functions (M2-digit BCD)

Function and purpose

The miscellaneous functions are also known simply as M functions, and they include such numerically controlled machine miscellaneous functions as spindle forward run and reverse run, operation stop and coolant ON/OFF.
These functions are designated by a 2-digit number following the address M with this NC, and only one command can be commanded in a single block.

(Example) G00 Xx1 Mm1;

When two or more commands are issued, only the last one will be valid.
The six commands of M00, M01, M02, M30, M98 and M99 are used as miscellaneous commands for specific purposes and so they cannot be used as general miscellaneous commands. This therefore leaves 94 miscellaneous functions which are usable as such commands. Refer to the instruction manual issued by the machine manufacturer for the actual correspondence between the functions and numerical values.
When the M00, M01, M02 and M30 functions are used, the next block is not read into the pre-read buffer due to pre-read inhibiting.
An M function can be specified together with other commands in the same block, and when such a function is specified together with a movement command in the same block, there are two possible sequences in which the commands are executed:

(1) The M function is executed after the movement command.

(2) The M function is executed at the same time as the movement command.

Which of these sequences actually applies depends on the machine specifications.
Processing and completion sequences are required in each case for all M commands except M98 and M99.
The six M functions used for specific purposes will now be described.

Program stop; M00

When the tape reader has read this function, it stops reading the next block. As far as the NC's functions are concerned, only the tape reading is stopped. Whether such machine functions as the spindle rotation and coolant supply are stopped or not differs according to the machine specifications.
Restart is enabled by pressing the automatic start button on the machine operation board.
Whether resetting can be carried out by M00 depends on the machine specifications.

Optional stop; M01

If the tape reader reads the M01 command when the optional stop switch on the machine operation board is ON, it will stop and the same effect as with the M00 command will apply.
If the optional stop switch is OFF, the M01 command is ignored.

(Example)

:	Optional stop switch status and operation
N10 G00 X1000;	Stops at N11 when switch is ON
N11 M01;	Next command (N12) is executed without stopping at N11 when switch is OFF
N12 G01 X2000 Z3000 F600;	
:	

Program end; M02 or M30

This command is normally used in the final block for completing the machining, and so it is primarily used for tape rewinding. Whether the tape is actually rewound or not depends on the machine specifications.
Depending on the machine specifications, the NC is reset by the M02 or M30 command upon completion of tape rewinding and any other commands issued in the same block.
(Although the contents of the command position display counter are not cleared by this reset action, the modal commands and compensation amounts are cancelled.)
The NC stops when the rewinding operation is completed (the in-automatic operation lamp goes off.) To restart the unit, the automatic start button must be pressed or similar steps must be taken.

(Note 1) Independent signals are also output respectively for the M00, M01, M02 and M30 commands and these outputs are each reset by pressing the reset key.

(Note 2) M02 or M30 can be assigned by manual data input MDI.
At this time, commands can be issued simultaneously with other commands just as with the tape.

Subprogram call/completion; M98, M99

These commands are used as the return instructions from branch destination subprograms and branches to subprograms.
M98 and M99 are processed inside the NC and so M code signals and strobe signals are not output.

NC internal processing with M00/M01/M02/M30 commands

NC internal processing suspends pre-reading when the M00, M01, M02 or M30 command has been read. Other tape rewinding operations and the initialization of modals by resetting differ according to the machine specifications.

9.2 Miscellaneous Functions (M8-digit)

Function and purpose

These functions are assigned with an 8-digit (0 to 99999999) number following the address M, and up to 4 commands can be assigned in one block. However, the number of commands which can be assigned in one block depends on the machine specifications.

(Example) G00 Xx Mm_1 Mm_2 Mm_3 Mm_4;

When five or more commands have been assigned in one block, the last four commands will be valid.
Parameters are used to select whether BCD output or binary output is to apply for the M8-digit commands.
The output signals are 8-digit BCD code and start signals or 32-bit binary data with sign and start signals.
As with the M2-digit functions, the M00, M01, M02, M30, M98 and M99 codes are reserved for special applications.
Processing and completion sequences are required for all M commands except for M98 and M99.

9.3 2nd Miscellaneous Functions (A8/B8/C8-digit)

Function and purpose

These serve to assign the indexing table position and other such functions. In this NC, they are assigned by an 8-digit number from 0 to 99999999 following address A, B or C. The machine specifications determine which codes correspond to which positions.
The output signals are command value BCD signals and start signals.
The A, B and C functions can be assigned simultaneously with any other commands but when they are in the same block as movement commands, there are 2 sequences in which the commands are executed, as below. The following two cases may apply depending on the machine specifications.

(1) The A, B or C function is executed after the movement command.

(2) The A, B or C function is executed simultaneously with the movement command.

Processing and completion sequences are required for all 2nd miscellaneous functions.
The table below gives the various address combinations. It is not possible to use an address which is the same for the axis name of an additional axis and 2nd miscellaneous function.

Additional axis name / **2nd miscellaneous function**	**A**	**B**	**C**
A	×	○	○
B	○	×	○
C	○	○	×

(Note) When "A" has been assigned as the 2nd miscellaneous function address, the following commands cannot be used.
(1) Linear angle commands
(2) Geometric I commands

10. SPINDLE FUNCTIONS

10.1 Spindle Functions (S2-digit BCD)

Function and purpose

The spindle functions are also known simply as S functions and they assign the spindle speed. In this NC, they are assigned with a 2-digit number following the address S ranging from 0 to 99, and 100 commands can be specified. In actual fact, however, it depends on the machine specifications as to how many of these 100 functions are used and which numbers correspond to which functions, and thus refer to the instruction manual issued by the machine manufacturer. When a number exceeding 2 digits is assigned, the last 2 digits will be valid.
The S functions can be assigned simultaneously with any other commands but when they are in the same block as movement commands, there are 2 sequences in which the commands are executed, as below. The following two cases may apply depending on the machine specifications.

(1) The S function is executed after the movement command.

(2) The S function is executed simultaneously with the movement command.

Processing and completion sequences are required for all S commands from S00 to S99.

10.2 Spindle Functions (S8-digit)

Function and purpose

These functions are assigned with an 8-digit (0 to 99999999) number following the address S, and only one command can be assigned in one block. The output for S8-digit functions can be either BCD or binary, and this can be selected by parameter.
The output signals are 8-digit BCD signals and start signals or 32-bit binary data with sign and start signals.
Processing and completion sequences are required for all S commands.

10.3 Constant Surface Speed Control; G96, G97

Function and purpose

This function automatically adjusts the spindle rotation speed according to changes in the coordinate values when cutting in the axis direction designated with the program or parameters.
Using this function, cutting can be carried out while maintaining the cutting point speed at a constant level.
When cross machining is commanded, if the axis layout of the constant surface speed control axis changes, the spindle rotation speed will be maintained at the speed applied before the layout of the constant surface speed control axis changed. The spindle rotation speed is maintained until the constant surface speed control axis returns to its original position or until constant surface speed control is canceled.

Command format

G96 Ss Pp; Constant surface speed ON

Ss	Surface speed
Pp	Constant surface speed control axis

G97; Constant surface speed cancel

Address	Address meaning	Command range (unit)	Remarks
S	Surface speed	Initial metric 1 to 99999999 (m/min) Initial inch 1 to 99999999 (feet/min)	• If the command range is exceeded, the "P35 Setting value range over" alarm will occur.
P	Constant surface speed control axis	1 to number of control axes in G96 system	• If P is not commanded, the movement will follow the constant surface speed control axis parameter (base system parameter 1 1117 G96_ax). • When the constant surface speed control axis parameter is set to 0, the 1st axis will be the constant surface speed axis regardless of whether P is commanded or not. • If the command range is exceeded, the "P133 Illegal P No.: G96" alarm will occur. (This will not occur if the constant surface speed control axis parameter is set to 0.)

Detailed description

(1) The constant surface speed control axis is set with parameter (machine parameter "G96 ax").
G96 ax = 0 : Fixed at 1st axis (P command invalid)
1 : 1st axis
2 : 2nd axis
3 : 3rd axis

(2) When the above-mentioned parameter is not 0, the constant surface speed control axis can be assigned with address P.

(Example) When G96 ax = 1

Program	Constant surface speed control axis
G96 S100;	→ 1st axis
G96 S100 P3 ;	→ 3rd axis

(3) Example of selection program and operation

G90 G96 G01 X50. Z100. F300 S200; :	The spindle speed is controlled so that the surface speed is 200m/min.
G97 G01 X50. Z100. F300 S500; :	The spindle speed is controlled to 500 r/min.
M02;	The modal returns to the initial value.

(Note) The initial value of the modal can be selected with parameter.

Explanation of operations

(1) The constant surface speed control axis is set with the parameter (machine parameter "G96 ax").

(2) Designate the spindle surface speed with the S command when the constant surface speed control command is ON. The surface speed can be changed with the S command only during the constant surface speed control mode.

(Example)

Machining program example	State of control axis in constant surface speed command system			Explanation of operation
	1st axis	2nd axis	3rd axis	
: G96 G01 X50. S200 F100 :	X1 ↓ ↓ ↓	Z1 ↓ ↓ ↓	C1 ↓ ↓ ↓	The spindle rotation speed is controlled so that the surface speed for the constant surface speed control axis is 200 (m/min).

(3) By issuing the P command during constant surface speed control, a random axis in the constant surface speed control system can be set as the constant surface speed control axis.
To change the constant surface speed control axis during the constant surface speed control mode, command with the G96 P∗ format.
The surface speed can also be changed by commanding S at the same time.
(When the base system parameter 1 1117 G96_ax is set to 0, the axis will be fixed to the 1st axis.)

(Example 1)

Machining program example	State of control axis in constant surface speed command system			Explanation of operation
	1st axis	2nd axis	3rd axis	
: G96 S200 P1; : :	X1 ↓ ↓ ↓	Z1 ↓ ↓ ↓	C1 ↓ ↓ ↓	The X1 axis will become the constant surface speed control axis. (The spindle rotation will be controlled so that the surface speed for the X1 axis is 200 (m/min).)
G96 P2; :	↓ ↓	↓ ↓	↓ ↓	The Z1 axis will become the constant surface speed control axis.

(Example 2)

Machining program example	State of control axis in constant surface speed command system			Explanation of operation
	1st axis	2nd axis	3rd axis	
: G96 S200 P1; : :	Z1 ↓ ↓ ↓	C1 ↓ ↓ ↓		The Z1 axis will become the constant surface speed control axis. (The spindle rotation will be controlled so that the surface speed for the Z1 axis is 200 (m/min).)

(4) The constant surface speed control and spindle clamp speed command can be issued to the first spindle or the other spindle.
Designate whether the commands are issued to the first spindle or the other spindle with the spindle selection command G code (G43/G44).
Set the parameter (base system parameter 1131 Sselect) to select either the 1st spindle or the other spindle as the default.

Spindle	Spindle targeted for constant surface speed control	
	During G43	During G44
First spindle	○	×
The other spindle	×	○

(Note) The spindle selection command is an option.

(5) Set whether to the calculate the surface speed for the rapid traverse command constantly or at the block's end point with the parameter (base common parameter 2 1312 (G96_G0).

Relation with other functions

(1) Relation with axis exchange command (cross machining control)

(a) If the axis layout of the constant surface speed control axis changes due to an axis exchange command during cross machining control etc., the spindle rotation speed will be maintained at the speed applied before the layout was changed.

(b) If the surface speed is commanded with the S command while the spindle rotation speed is being held, the commanded speed will be validated when the constant surface speed axis returns to its original position.

[When constant surface speed control axis number changes with cross machining control command in constant surface speed command system after constant surface speed is designated]

(Example 1)

Machining program example	State of control axis in constant surface speed command system			Explanation of operation
	1st axis	2nd axis	3rd axis	
G110 X1 Z1 C1; :	X1 ↓	Z1 ↓	C1 ↓	
G96 S100 P1; :	↓ ↓	↓ ↓	↓ ↓	Constant surface speed control is carried out for X1.
G110 Z1; : :	Z1 ↓ ↓	– – –	– – –	Since the 1st axis has changed from the X1 axis to the Z1 axis, the present rotation speed will be held as the spindle speed.
S200; :	↓ ↓	– –	– –	The command for changing the surface speed from 100 (m/min) to 200 (m/min) will not be valid as the constant surface rotation speed is held.
G110 X1; :	X1 ↓	– –	– –	Since the 1st axis has changed to the X1 axis, the constant surface speed will be recalculated. The surface speed will be changed to 200 (m/min).

(Example 2)

Machining program example	State of control axis in constant surface speed command system			Explanation of operation
	1st axis	2nd axis	3rd axis	
G110 X1 Z1 C1; :	X1 ↓	Z1 ↓	C1 ↓	
G96 S100 P2; :	↓ ↓	↓ ↓	↓ ↓	Constant surface speed control is carried out for Z1.
G110 Z1; : :	Z1 ↓ ↓	– – –	– – –	The Z1 axis number in the system changes from the 1st axis to the 2nd axis. Thus, the spindle speed is the constant rotation speed commanded with the constant surface speed final command value for the Z1 axis before G140 Z = Z1 command.
G110 X1 Z1; :	X1 ↓	Z1 ↓	– –	Since the Z1 axis has changed to the 2nd axis, the constant surface speed will be recalculated.

[When constant surface speed control axis number changes with axis exchange command from the system other than the constant surface speed command system after constant surface speed is commanded]

(Example)

Machining program example		State of control axis in constant surface speed command system			Explanation of operation
Constant surface speed command system	Other system	1st axis	2nd axis	3rd axis	
G110 X1 Z1 C1; :	: :	X1 ↓	Z1 ↓	C1 ↓	
G96 S100 P1; :	: :	↓ ↓	↓ ↓	↓ ↓	Constant surface speed control is carried out for X1.
G110 Z1; : :	G110 X1; : :	Z1 ↓ ↓	– – –	– – –	Since the 1st axis has changed from the X1 axis to the Z1 axis, the present rotation speed will be held as the spindle speed.
G110 X1; :	: :	X1 ↓	– –	– –	Since the 1st axis has changed to the X1 axis, the constant surface speed will be recalculated. The surface speed will be changed to 200 (m/min).

(c) When the spindle rotation speed is being maintained at the constant rotation speed after the layout of the constant surface speed axis has changed, if the constant surface speed command is executed again, the spindle rotation speed being maintained will be canceled, and the constant surface speed control command issued again will be executed.

(Example)

Machining program example		State of control axis in constant surface speed command system			Explanation of operation
Constant surface speed command system	Other system	1st axis	2nd axis	3rd axis	
G110 X1 Z1 C1; :	: :	X1 ↓	Z1 ↓	C1 ↓	
G96 S100 P1; :	: :	↓ ↓	↓ ↓	↓ ↓	Constant surface speed control is carried out for X1.
G110 Z1; : :	G110 X1; : :	Z1 ↓ ↓	– – –	– – –	Since the 1st axis has changed from the X1 axis to the Z1 axis, the present rotation speed will be held as the spindle speed.
G96 S100 P1; :	: :	Z1 ↓	– –	– –	Constant surface speed control is carried out for Z1.

Precautions and restrictions

(1) If an axis number that does not exist in the command system is commanded for the constant surface speed command, the program error "P133 Illegal P No.: G96" will occur.

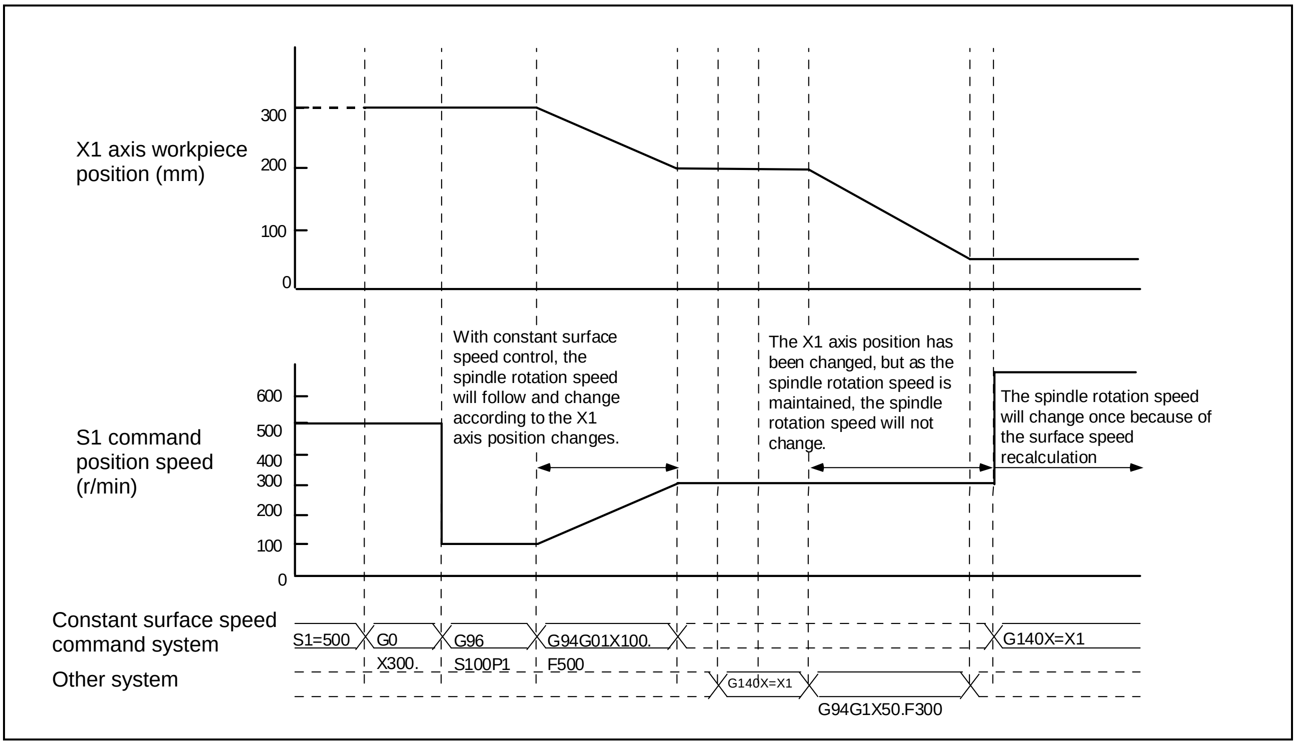
(2) Even if the spindle rotation speed is maintained when the axis is exchanged during constant surface speed control, the speed clamp command will be valid.

(3) If the axis layout changes due to an axis exchange during the constant surface speed control and the position of the axis for constant surface speed changes while the spindle rotation speed is maintained, the rotation speed of the constant surface speed spindle may differ greatly when the control is resumed.

(Example)

<table>
<tr><th colspan="2">Machining program example</th><th colspan="3">State of control axis in constant surface speed command system</th><th rowspan="2">Explanation of operation</th></tr>
<tr><th>constant surface speed command system</th><th>Other system</th><th>1st axis</th><th>2nd axis</th><th>3rd axis</th></tr>
<tr><td>G110 X1 Z1 C1;
S500</td><td>:
:</td><td>X1
↓</td><td>Z1
↓</td><td>C1
↓</td><td></td></tr>
<tr><td rowspan="2">G0 X300.;
:</td><td>:</td><td>↓</td><td>↓</td><td>↓</td><td rowspan="3">Constant surface speed control is carried out for X1.</td></tr>
<tr><td>:</td><td>↓</td><td>↓</td><td>↓</td></tr>
<tr><td>G96 S100 P1;
G94 G1 X200. F500;</td><td>:
:</td><td>↓
↓</td><td>↓
↓</td><td>↓
↓</td></tr>
<tr><td>G110 Z1;
:
:</td><td>G110 X1;
:
G94 G1 X50. F500;
:</td><td>Z1
↓
↓</td><td>–
–
–</td><td>–
–
–</td><td>Since the 1st axis has changed from the X1 axis to the Z1 axis, the present rotation speed will be held as the spindle speed.</td></tr>
<tr><td>G110 X1;
:</td><td>:
:</td><td>X1
↓</td><td>–
–</td><td>–
–</td><td>Constant surface speed control is carried out for Z1.</td></tr>
</table>

(Operation diagram)



(4) If reset is applied during constant surface speed control, the spindle rotation speed will change to "0".

10.4 Spindle Clamp Speed Setting; G92

Function and purpose

The maximum clamp speed of the spindle can be assigned by address S following G92 and the minimum clamp speed by address Q.

Command format

G92 Ss Qq ;	**Spindle clamp speed setting**
Ss	Maximum clamp speed
Qq	Minimum clamp speed

Detailed description

(1) To accommodate gear selection between the spindle and spindle motor, parameters can be used to set the speed range up to 4 stages in 1r/min units.

(2) The lower value among the speed based on the parameter and based on "Ss" of "G92 Ss Qq;" is valid as the upper limit.
Similarly, the higher value among the parameter and "Qq" is as the lower limit.

(3) Set the parameter (sp_6/Bit 3) to select whether to validate the spindle clamp speed setting only during constant surface speed control mode or even for normal spindle rotation commands.

(4) The spindle clamp speed command can be commanded for the first spindle or the other spindles.
Set the spindle selection command G code (G43/G44) to designate whether to issue the commands to the first spindle or the other spindles.
Set the parameter (base system parameter 1131 Sselect) to select either the first spindle or the other spindles as the default.

Spindle	**Spindle targeted for constant surface speed control**	
	During G43	**During G44**
First spindle	○	×
The other spindle	×	○

(Note) The spindle selection command is an option.

(5) Even if the spindle rotation speed is maintained when the axis is exchanged during constant surface speed control, the speed clamp command will be valid.

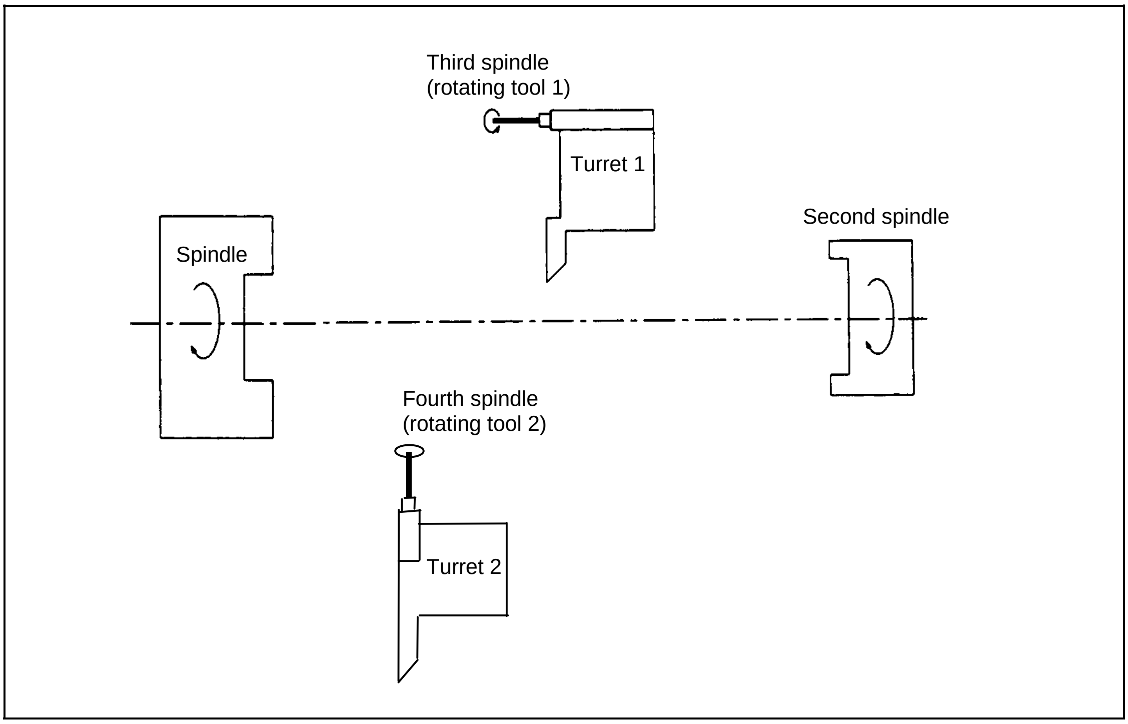
(Example)

Machining program example	State of control axis in constant surface speed command system			Explanation of operation
	1st axis	2nd axis	3rd axis	
G110 X1 Z1 C1; :	X1 ↓	Z1 ↓	C1 ↓	
G96 S100 P2; :	↓ ↓	↓ ↓	↓ ↓	Constant surface speed control is carried out for X1.
G110 Z1; : :	Z1 ↓ ↓	– – –	– – –	Since the 1st axis has changed from the X1 axis to the Z1 axis, the present rotation speed will be held as the spindle speed.
G92 S4500 Q200; :	↓ ↓	– –	– –	The maximum clamp rotation speed for the constant surface speed control spindle will be 4500 (r/min) and the minimum rotation speed will be 200 (r/min).

10.5 Spindle Functions (Multiple Spindles)

Function and purpose

In addition to the spindle (first spindle) control function, the multiple-spindle control function is available.
An S command can be used to specify the spindle speed. To specify spindle speeds for multiple spindles, use an S○ = ××××× command. The S command can be issued from a machining program running in any system.
The second spindle control function is also available to perform cutting in synchronous with the rotation of the second spindle.



(Note) The number of spindles depends on the model. Check the specifications.

10.5.1 Multiple-spindle commands

Function and purpose

The S command can be used to specify individual spindles in the "S○ = ∗∗∗∗∗" format as well as to specify the first spindle in the standard format "S∗∗∗∗∗".

Command format

The command format is as follows:

S○ = ∗∗∗∗∗	**S5-digit analog**
S○ = ∗∗∗∗∗∗∗∗	**S8-digit binary/BCD**
○	A digit from 1 to 9
∗ ··· ∗	Spindle speed or surface speed command value

Detailed description

(1) Each spindle command can be specified by ○ with parameter setting (sname), which is a digit from 1 to 9.

(Example)

Parameter \ Spindle	< 1 >	< 2 >	< 3 >	< 4 >	< 5 >	< 6 >
sname	1	2	3	4	5	6

G97;
S1=3500; Instructs the first spindle to rotate at 3,500 (r/min).
S2=1500; Instructs the second spindle to rotate at 1,500 (r/min).
S3=2000; Instructs the third spindle to rotate at 2,000 (r/min).
S4=2500; Instructs the fourth spindle to rotate at 2,500 (r/min).
S5=2000; Instructs the fifth spindle to rotate at 2,000 (r/min).
S6=3000; Instructs the sixth spindle to rotate at 3,000 (r/min).

(2) Commands for multiple spindles can be issued in one block.

(3) If two or more commands are specified for the same spindle in one block, the last command is valid.

(Example) S1=3500 S1=3600 S1=3700; S1 = 3700 is a valid command.

(4) S commands in the "S∗∗∗∗∗" and "S○ = ∗∗∗∗∗" formats can be specified simultaneously.
The S command in the "S∗∗∗∗∗" format applies to the first spindle. When second spindle control is ON (G44), it applies to the second spindle. The S command in this format cannot be used for the third, fourth, fifth, or sixth spindles. Use the "S○ = ∗∗∗∗∗" format for these spindles.

(5) A command for each spindle can be issued from a program running in any system. Each spindle rotates according to the S command specified last for it. (See the figure below.)
If S commands are simultaneously issued from multiple systems, the command from the largest system number is valid.

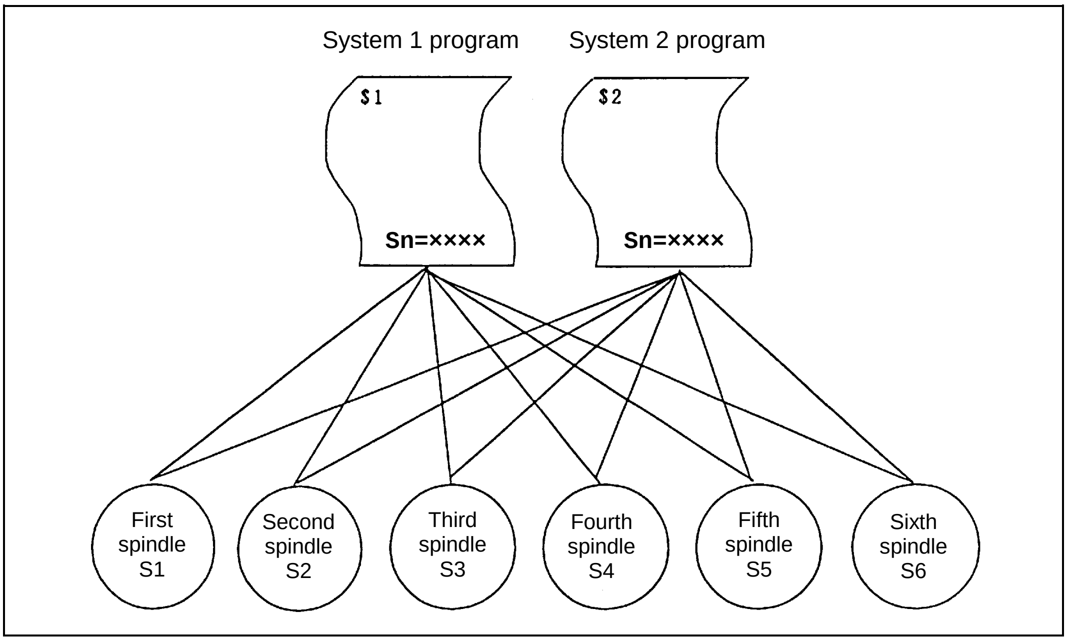
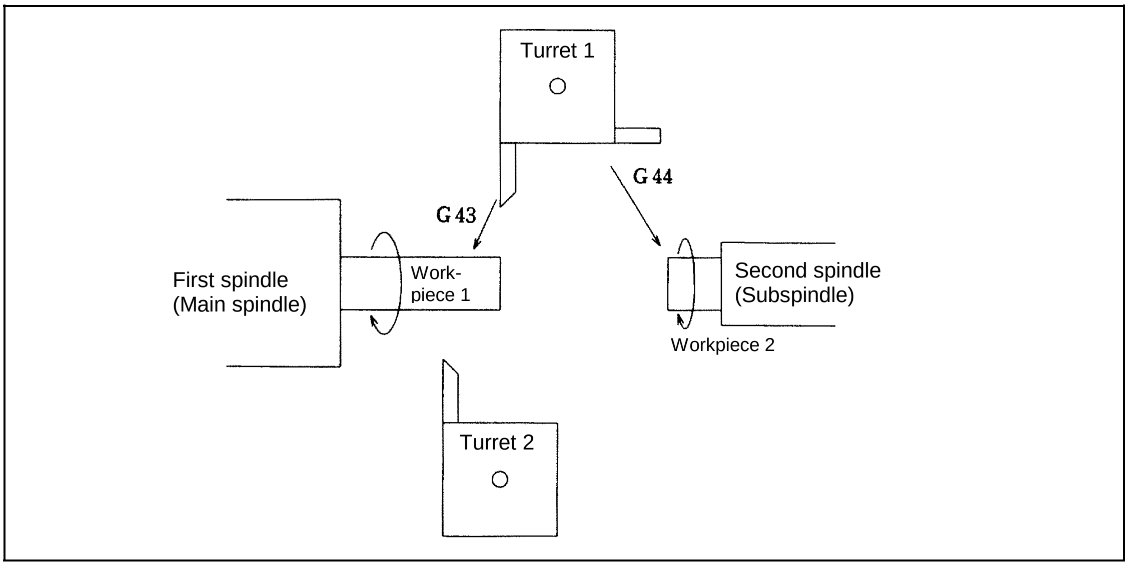


Fig. Spindle commands and system programs

10.6 Second Spindle Control Function

Function and purpose

This function controls whether cutting is carried out synchronized with the first spindle or second spindle.
This function is required to machine the back of a workpiece by passing it from the first spindle to the second spindle. It is also required to cut two workpieces with two spindles and two turrets.



Command format

G43;	**Second spindle control OFF (first spindle control)**
G44;	**Second spindle control ON**

Detailed description

(1) G43 and G44 are modal G codes.

(2) Use of G43 or G44 when the power is turned ON or the system is reset depends on the parameter (Sselect) setting.
The parameter is available for each system.

Sselect = 0: First spindle control
1: Second spindle control

(3) If G43 or G44 is specified with an S command in the same block, the target spindle of the S command varies depending on the order the G43 or G44 and S command are specified.
If the S command is specified first, the previous G43 or G44 mode applies.
If the S command is specified last, the G43 or G44 in the same block applies.

(4) The G43 or G44 command can be issued from any system.

(5) The following control functions are switched after the G43 or G44 command is issued:

(a) Feed per rotation (synchronous feed)
A feedrate command in G95 mode indicates the feedrate per rotation of the first spindle in G43 mode or the second spindle in G44 mode.

(b) S command (S∗∗∗∗∗, S○ = ∗∗∗∗∗), constant surface speed control

Function	G43 mode	G44 mode
S command in G97 or G96 mode Constant surface speed control Command specifying the upper or lower limit of spindle speed in constant surface speed control mode (G92 S__Q__)	Command control for the first spindle	Command control for the second spindle

(Note) Even in G43 mode, the S○ = ∗∗∗∗∗ command can be used to specify the second spindle. Similarly, the S○ = ∗∗∗∗∗ command can be used to specify the first spindle even in G44 mode. In this case, however, the spindle speed needs to be specified even in G96 mode.

Example 1: S1 = First spindle, S2 = Second spindle

<table>
<tr><th rowspan="2"></th><th colspan="2">Spindle speed</th></tr>
<tr><th>First spindle</th><th>Second spindle</th></tr>
<tr><td>G43 ;</td><td></td><td></td></tr>
<tr><td>G97 S1000 (same as for "G97 S1=1000")
:</td><td rowspan="2">S1000 (r/min)</td><td>0 (r/min)</td></tr>
<tr><td>S2=2000;
:</td><td rowspan="2">S2000 (r/min)</td></tr>
<tr><td>G96 S100;
:</td><td rowspan="3">S100 (m/min)

(Note 1)</td></tr>
<tr><td>S2=2500;
:</td><td>S2500 (r/min)</td></tr>
<tr><td>G44 S200;
:</td><td rowspan="2">S200 (m/min)</td></tr>
<tr><td>S1=3000;
:</td><td rowspan="2">S3000 (r/min)</td></tr>
<tr><td>G97 S4000;
:</td><td>S4000 (r/min)</td></tr>
</table>

(Note 1) The G44 command switches constant surface speed control to the second spindle. Therefore, the speed of the first spindle remains unchanged as the speed at "G44 S200;". When "S1=3,000;" is specified, it changes to 3000 (r/min).

10.6.1 Second spindle extension selection

Function and purpose

This function is the second spindle control function extension, and enables the second spindle control command (G44) to be issued to a random spindle between the second and sixth spindles. Set the parameter (machine parameter SP2name) to select the spindle from the second to sixth spindles.

SP2name value	Selected spindle
0	Second spindle
1 to 6	First to sixth spindle
7 to 9	Second spindle

The functions targeted for second spindle control are synchronous feed, thread cutting, spindle rotation command, constant surface speed, dwell per rotation, and spindle clamp speed setting.

Detailed description

(1) Setting the spindle clamp speed

If the spindle targeted for second spindle control is changed, the clamp rotation speed commanded before the changes will not be held. Execute the spindle clamp speed setting command again.

Example of program

G44;	Second spindle control ON
G92 S2000;	Clamp speed setting for spindle set with parameter
S1000;	Rotation command for spindle set with parameter
G95 G1 X100. F1.;	Synchronous feed control for spindle set with parameter
G43;	First spindle control ON (Second spindle control OFF)

Precautions

(1) Serial encoder input of spindle-type servomotor control is not possible.

(2) The spindle rotation speed upper/lower limit over signals are output to the spindle selected with the G43/G44 commands.

(3) The spindle no-signal detection function is carried out on the spindle selected with the G43/G44 commands.

11. TOOL FUNCTIONS

11.1 Tool Functions (T4-digit)

Function and purpose

The tool functions are also known simply as T functions and they assign the tool numbers and tool offset numbers.
They are assigned with a 3- or 4-digit number following the address T, and the first two digits designate the tool number.
One group of T command can be assigned in one block, but refer to the instruction manual issued by the machine manufacturer since the number of T commands which can be used differs according to the individual machine.
Parameter setting is used to determine whether a T command is to be used with 3 or 4 digits.

Command format

```
T  **  *  ;
   |   └──────── Tool offset number
   └──────────── Tool number

T  **  **  ;
   |   └──────── Tool offset number
   └──────────── Tool number
```

Refer to the instruction manual issued by the machine manufacturer for the correspondence between the actual tools and the tool numbers commanded in the program.
BCD codes and start signal are output.
The T functions can be assigned simultaneously with any other commands but when they are in the same block as movement commands, there are 2 sequences in which the commands are executed, as below. The following two cases may apply depending on the machine specifications.

(a) The T function is executed after the movement command.

(b) The T function is executed simultaneously with the movement command.

Processing and completion sequences are required for all T commands.

11.2 Tool Functions (T8-digit)

Function and purpose

These functions are assigned with an 8-digit (0 to 99999999) number following the address T with the first 6 or 7 digits used as the tool number and the last 2 digits or 1 digit used as the tool offset number.
Parameters are set to determine how many digits are used. Refer to the instruction manual issued by the machine manufacturer since the number of T commands which can be used differs according to the individual machine.
One group of T command can be assigned in a block.

Command format

```
T  ******  ** ;
      |     |_______ Tool offset number
      |_____________ Tool number
T  *******  * ;
      |     |_______ Tool offset number
      |_____________ Tool number
```

Refer to the instruction manual issuers by the machine manufacturer for the correspondence between the actual tools and the tool numbers commanded in the program.
6-digit (or 7-digit) BCD signals and start signal or binary data and start signal are output.
The BCD or binary output can be selected with parameter.
Processing and completion sequences are required for all T commands.

11.3 Number of T Command Digits Judgment Function

Function and purpose

The following effects can be achieved by judging the number of digits when the tool compensation command is issued.

(1) When T macro call is valid, whether to call the T macro can be judged with the number of T digits.

(2) If the tool length offset number and wear offset number are commanded individually by the parameters, the wear offset number can be changed without changing the tool length offset number.

Detailed description

When this function is valid, the following actions will take place according to the details of the T command.

Command with 2-digit wear offset number	Command with 1-digit wear offset number	Tool length offset number	Wear offset number	T macro call (Note 2)
T00 (T0)	T0	Hold	Cancel	Not called
T0000 (T000)	T00	Cancel	Cancel	Called
T▲▲00	T▲▲▲0	▲▲(▲▲▲)	Cancel	Called
T■■	T■	Hold	■■(■)	Not called
T00■■	T0■	Cancel	■■(■)	Called
T▲▲■■	T▲▲▲■	▲▲(▲▲▲)	■■(■)	Called

▲ indicates a random tool length offset number, ■ indicates a random wear offset number

(Note 1) In the above example, the tool length offset number and wear offset number are commanded individually.

(Note 2) T macro call indicates the case to call the T macro only when the tool number is commanded.

Example of operation

Upper 2 digits: Tool length offset number, lower 2 digits: Wear offset number

Program	Handling of command	Tool length offset number	Tool wear offset number
T0000	T0000	Cancel	Cancel
T101	T101	01	01
T02	T02	Hold	02
T003	T003	Cancel	03
#501=0001	– –	– –	– –
T#501	T1	Hold	01
T100+10	T110	01	10
T220-200	T20	Hold	20
T[100+10]	T110	01	10
T[0001]	T1	Hold	01

Precautions and restrictions

(1) When using variables (local variables, common variables, etc.) for the T command, if the value has three or more digits, the tool length offset and T macro call will be executed.

(2) When a calculation expression (100 + 10, etc.) is used for the T command, if the value has three or more digits, the tool length offset and T macro call will be executed.

(3) When a value enclosed in brackets [ ] is commanded for the T command, if the value has three or more digits, the tool length offset and T macro call will be executed.

(4) If the T command is issued with a decimal point, the number of digits may be mistaken. Always command T with an integer.

12. TOOL OFFSET FUNCTIONS

12.1 Tool Offset

Function and purpose

Tool offset is performed by the T functions which are commanded with the 3-, 4- or 8-digit number following address T. There are two types of tool offset: tool length offset and tool nose wear offset. There are two ways to issue the commands: the tool length offset and tool nose wear offset are designated by the last 1 digit or 2 digits of the T command or the tool wear offset is assigned by the last 1 digit or 2 digits of the T command and the tool length offset by the tool number. Parameter setting is used to switch between them. A parameter is also used to select the last 1 digit or 2 digits for the offset.
One group of T command can be assigned in a block.

Command format

(1) When designating the tool length and tool nose wear offset number using the last 1 digit or 2 digits of the T command

```
T  **  * ;
   |   └──────────── Tool length offset + tool nose wear offset
   └──────────────── Tool number

T  **  ** ;
   |   └──────────── Tool length offset + tool nose wear offset
   └──────────────── Tool number

T  *******  * ;
      |     └─────── Tool length offset + tool nose wear offset
      └───────────── Tool number

T  ******  ** ;
      |     └─────── Tool length offset + tool nose wear offset
      └───────────── Tool number
```

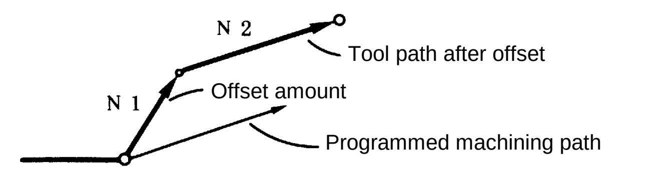
(2) When differentiating between the tool length offset number and tool nose wear offset number

```
T  **  * ;
   |   └──────────── Tool nose wear offset
   └──────────────── Tool number + tool length offset

T  **  ** ;
   |   └──────────── Tool nose wear offset
   └──────────────── Tool number + tool length offset

T  *******  * ;
      |     └─────── Tool nose wear offset
      └───────────── Tool number + tool length offset

T  ******  ** ;
      |     └─────── Tool nose wear offset
      └───────────── Tool number + tool length offset
```

Detailed description

There are two ways to execute tool offset and these can be selected by parameter: executing offset when the T command is executed and executing offset in the block with a movement command without performing offset when the T command is executed.

(1) Offset with T command execution



N1 T0101;
N2 X100. Z200.;

Tool length offset and tool nose wear offset are conducted simultaneously.

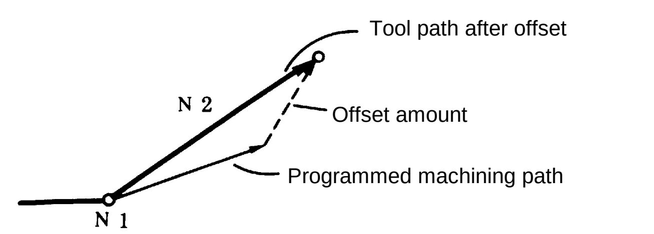
(Note 1) The movement applying to offset with the T command execution is rapid traverse in a G00 modal and cutting feed in the other modals.

(Note 2) When performing offset with T command execution, the path is offset as a linear movement in a circular modal.

(Note 3) When performing offset with T command execution, offset will not function until the arrival of any G command except those listed below when the T command has been assigned in the same block as the G commands listed below.

G04 : Dwell
G10 : Program tool offset input/data setting
G11 : Data setting mode cancel
G65 : User macro simple call
G92 : Coordinate system setting

(2) Offset with movement command



N1 T0101;
N2 X100. Z200.;

Tool length offset and tool nose wear offset are conducted simultaneously.

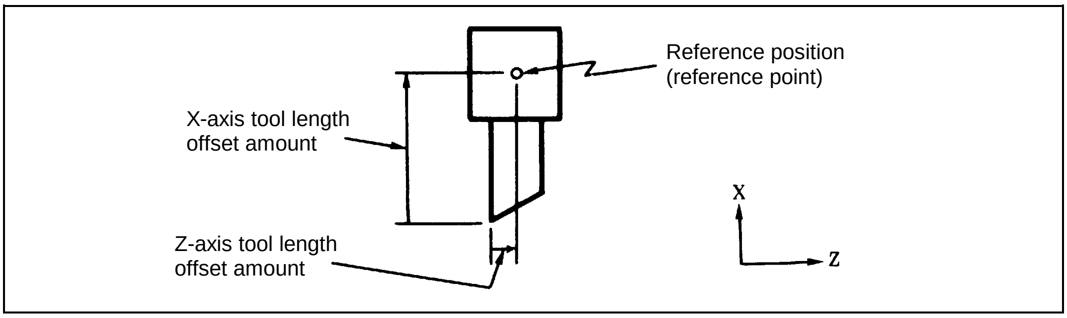
(Note 1) When performing offset with a movement command, offset is applied if the offset amount is lower than the "#8010 G02/03 Error" parameter when offset is performed for the first time with a circular command. If the amount is higher, the program error "P70 Arc end point deviation large" results. (This also applies when the arc command and T command are in the same block for offsetting with T command execution.)

12.2 Tool Length Offset

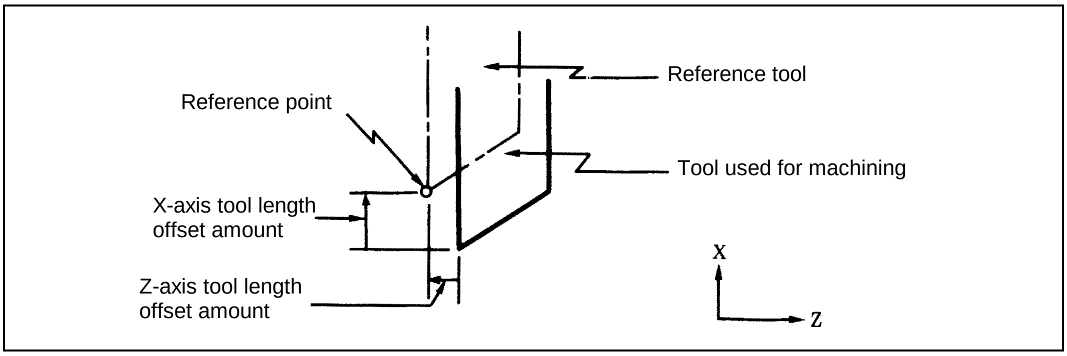
Tool length offset amount setting

This function offsets the tool length with respect to the programmed reference position. This position may generally be set to either the center position of the turret or the tool nose position of the reference tool.

(1) Center position of turret

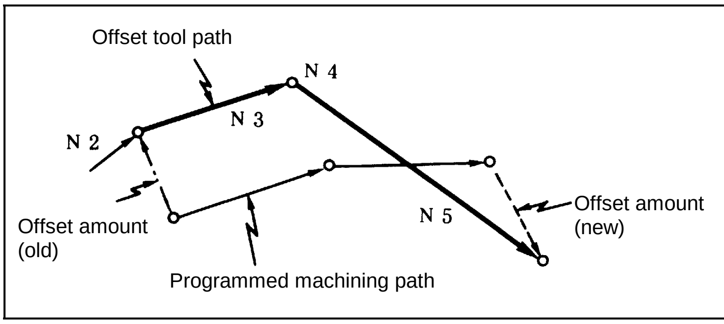


(2) Tool nose position of reference tool



Tool length offset number change

When tool numbers are changed, the tool length offset corresponding to the new tool numbers are added to the movement amounts in the machining program.

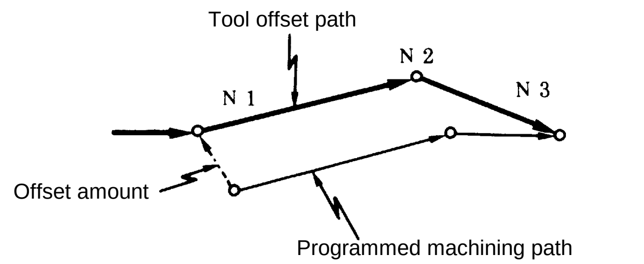



```
N1 T0100;
N2 G1 X10.0 Z10.0 F100;
N3 G1 X13.0 Z15.0 F20.0;
N4 T0200;
N5 G1 X13.0 Z20.0 F25.0;
```

In this example, the tool length offset is applied with the tool number and offset is performed in the block with the movement command.

Tool length offset cancel

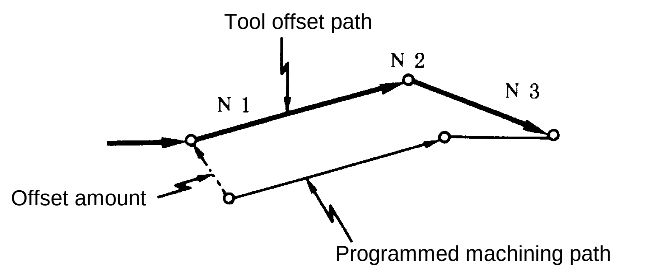
(1) Offset is canceled when "0" has been assigned as the tool length offset number by the T command.



N1 X10.0 Z10.0 F10;
N2 T0000;
N3 G1 X10.0 Z20.0;

In this case, offset is performed by the block with the movement command.

(2) Offset is canceled when the offset amount in the tool length offset number assigned by the T command is "0".



N1 G1 X10.0 Z10.0 F10;
N2 T0100;
N3 G1 X10.0 Z20.0;

In this case, offset is performed by the block with the movement command.

(Note 1) When G28, G29 or G30 is commanded, the machine moves to the position where the offset was canceled and the offset amount is stored in the memory. This means that with the next movement command the machine will move to the offset position.

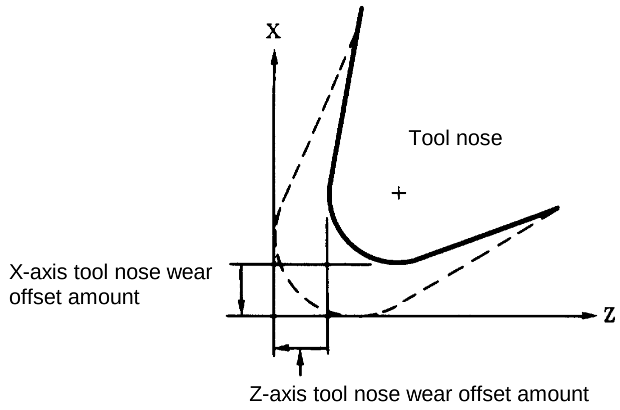
(Note 2) Even if the offset amount of the offset number currently selected by MDI, etc., is changed during automatic operation, the changed offset amount will not be valid unless a T command with the same number is executed again.

(Note 3) The tool length offset and tool nose wear offset amounts are cleared by resetting and by emergency stop. They can be retained by parameters.

12.3 Tool Nose Wear Offset

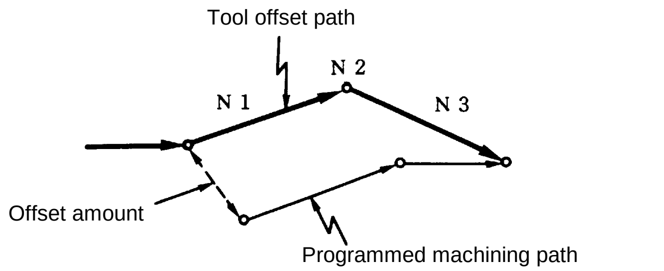
Tool nose wear offset amount setting

The wear sustained by the tool being used can be offset.



Tool nose wear offset cancel

Tool nose wear offset is canceled when "0" has been assigned as the offset number.



N1 G1 X10.0 Z10.0 F10;
N2 T0100;
N3 G1 X10.0 Z20.0;

In this case, offset is performed in the block with the movement command.

(Note 1) When G28, G29 or G30 is commanded, the machine moves to the position where the offset was canceled and the offset amount is stored in the memory. This means that with the next movement command the machine will move to the offset position.

(Note 2) Even if the offset amount of the offset number currently selected by MDI, etc., is changed during automatic operation, the changed offset amount will not be valid unless a T command with the same number is executed again.

(Note 3) The tool length offset and tool nose wear offset amounts are cleared by resetting and by emergency stop. They can be retained by parameters.

12.3.1 Wear offset amount hold

Function and purpose

If the tool length offset number and wear offset number are individually commanded with the parameters, the wear offset number must be designated when the tool length offset number is changed. Thus, to change the tool length offset number without changing the wear offset number, the previously commanded wear offset number must be memorized and commanded. By using this command, the tool length offset number can be commanded while holding the wear offset number.

Command format

T△△97;

△ is a random tool length offset number.
Wear offset amount hold is validated by commanding "97" for the tool wear offset number.

Precautions

(1) When the tool length offset number is the last 2 digits of the T command (same as the wear offset number), this function will be invalid. If "97" is commanded for the wear offset number, a program error (P170 No offset number) will occur.

(2) If the tool wear offset number has only 1 digit, this function will be invalid. If "97" is commanded, the wear offset number will be handled as "7".

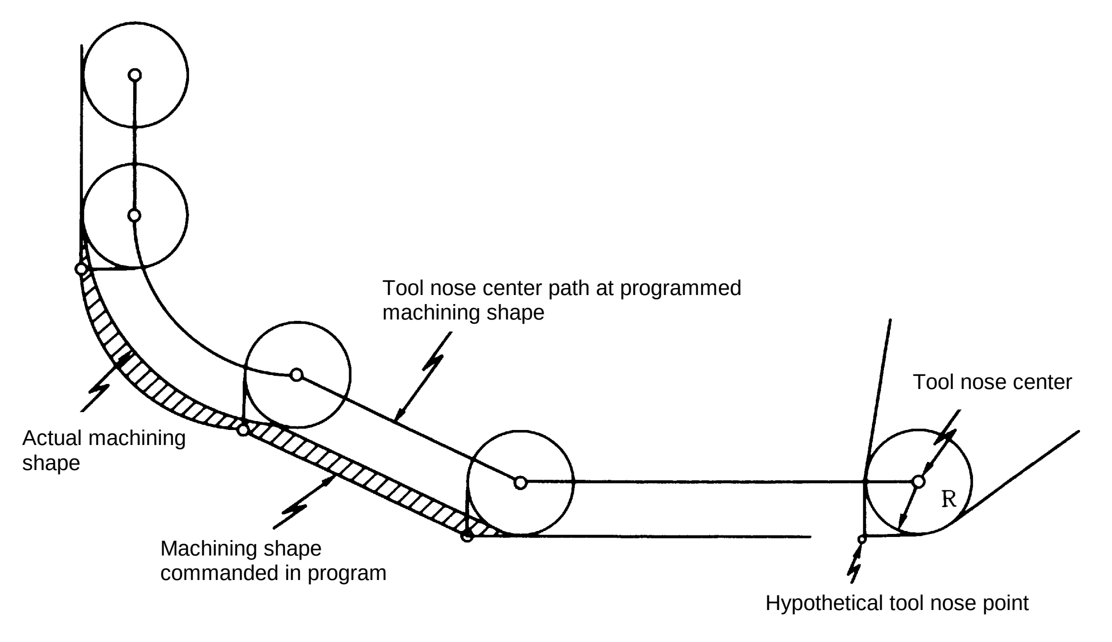
(3) Select the validity of this function with the parameter.
sp_3/bit 9 Wear offset amount hold

0: Invalid
1: Valid

12.4 Nose R Compensation; G40, G41, G42, G46

Function and purpose

The tool nose is generally rounded and so a hypothetical tool nose point is treated as the tool nose for programming purposes. When this is done, an error caused by the tool nose rounding arises during taper cutting or circular cutting between the actually programmed shape and the cutting shape. Nose R compensation is a function for automatically calculating and offsetting this error by setting the tool nose radius value.
The command codes enable the offset direction to be fixed or automatically identified.

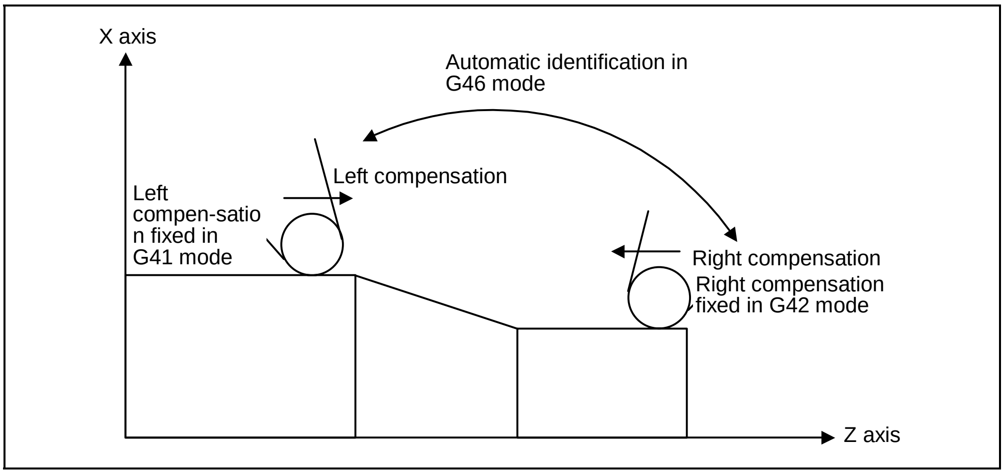


Functions and command format

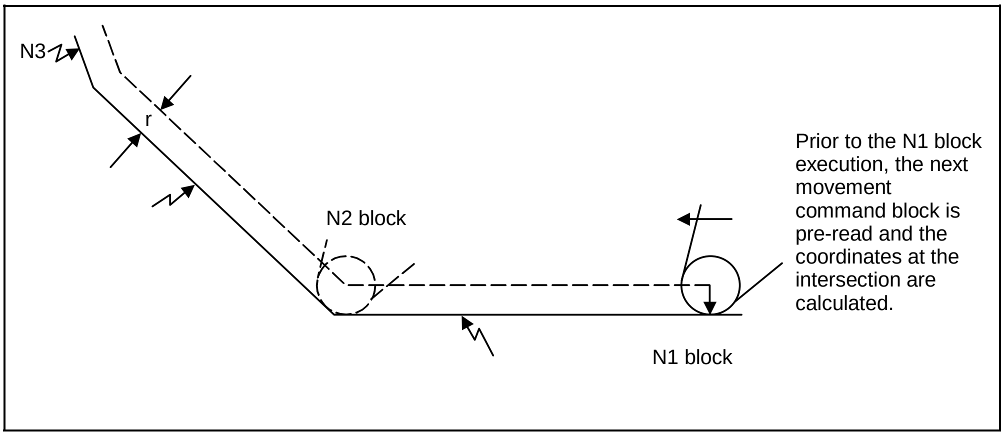
Code	Function	Command format
G40	Nose R compensation mode cancel	G40 (Xx/Uu Zz/Ww Ii Kk);
G41	Nose R compensation left mode ON	G41 (Xx/Uu Zz/Ww);
G42	Nose R compensation right mode ON	G42 (Xx/Uu Zz/Ww);
G46	Nose R compensation automatic direction identification mode ON	G46 (Xx/Uu Zz/Ww);

(Note 1) By means of the preset hypothetical tool nose point and movement commands in the machining program, the G46 nose R compensation function automatically identifies the compensation direction and provides nose R compensation.

(Note 2) G40 serves to cancel the nose R compensation mode.



(Note 3) Nose R compensation pre-reads the data in the following two movement command blocks (up to 5 blocks when there are no movement commands) and controls the nose R center path by the intersection calculation method so that it is offset from the programmed path by an amount equivalent to the nose radius.



(Note 4) In the above figure, "r" is the nose R compensation amount (nose radius).

(Note 5) The nose R compensation amount corresponds to the tool length number and it is preset along with the tool nose point.

(Note 6) If there are 4 or more blocks without movement amounts among 5 continuous blocks, overcutting or undercutting will result. Blocks in which optional block skip is valid are ignored.

(Note 7) Nose R compensation is also valid for fixed cycles (G77 to G79) and for rough cutting cycles (G70, G71, G72, G73). However, in the rough cutting cycles, the finishing shape with the nose R compensation applied will be turned with the compensation canceled and, upon completion of the turning, operation will automatically return to the compensation mode.

(Note 8) With thread cutting commands, compensation is temporarily canceled 1 block before.

(Note 9) A nose R compensation (G41 or G42) command can be assigned during nose R compensation (G46). There is no need to cancel the G40 data compensation.

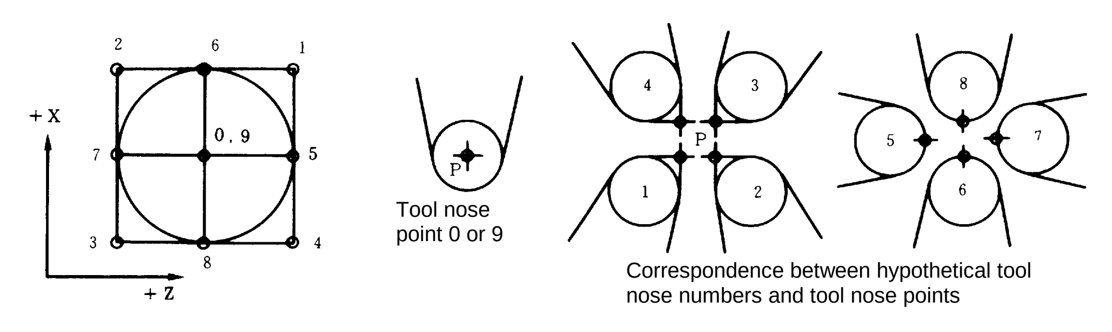
(Note 10) The compensation plane, movement axes and next advance direction vector follow the plane selection command designated by G17, G18 or G19.

G17 ····· X-Y plane X, Y, I, J
G18 ····· Z-X plane Z, X, K, I
G19 ····· Y-Z plane Y, Z, J, K

12.4.1 Tool nose point and compensation directions

Tool nose point

Since the tool nose is generally rounded, the programmed tool nose position is aligned with point P shown in the examples of the figures below.
With nose R compensation, one point among those in the figures below indicating the position relationship is selected for each tool length number and preset. (Selection from points 1 to 8 in the G46 mode and 0 to 9 in the G41/G42 mode.)

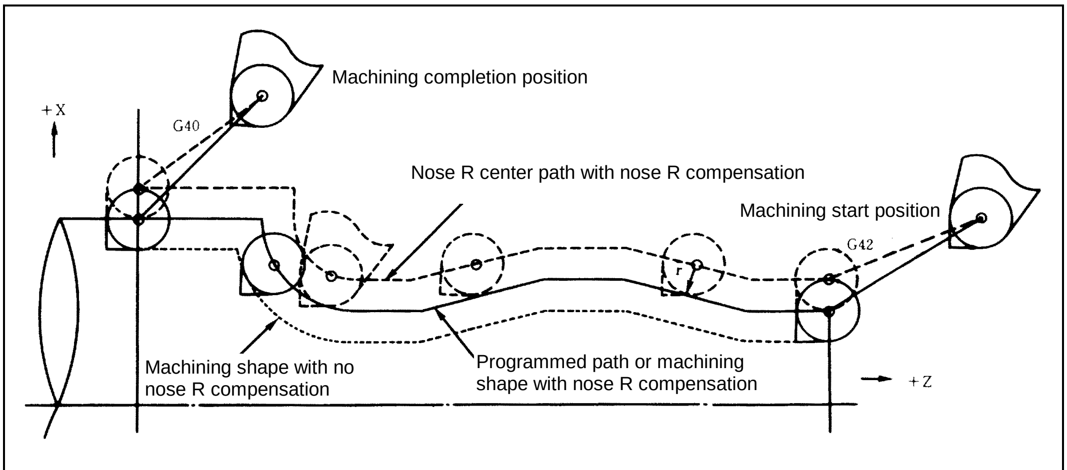


Tool nose point 0 or 9

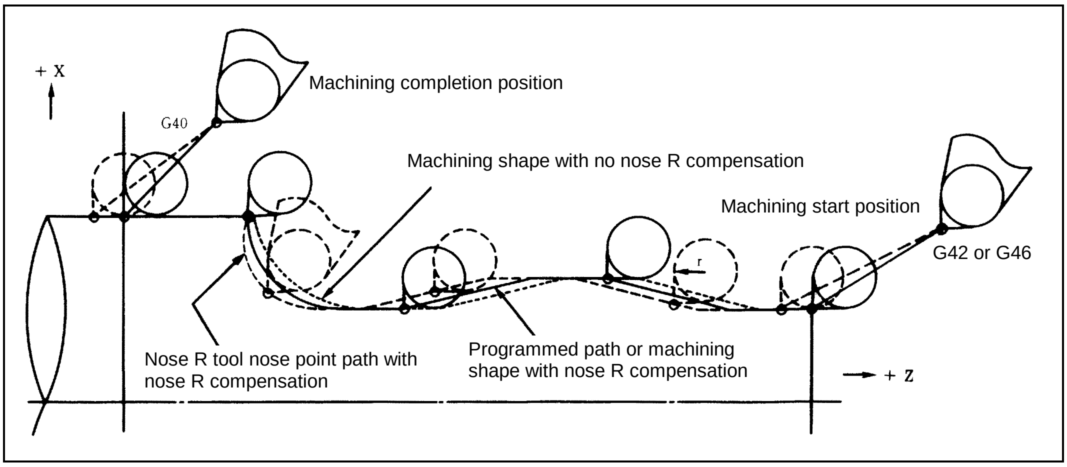
Correspondence between hypothetical tool nose numbers and tool nose points

Tool nose point and compensation operation

(1) When the nose R center has been aligned with the machining start position



(2) When the tool nose point has been aligned with the machining start position



Compensation directions

(1) The compensation direction of the G41/G42 commands is determined by the G41/G42 codes. The direction in a G46 command is automatically determined in accordance with the following table from the relationship between the tool nose points and the commanded movement vectors.

(2) When nose R compensation has been started and the initial movement vector (including G0) corresponds to an "×" mark in the table, the compensation direction cannot be specified and so it is determined by the next movement vector. When the direction cannot be determined even after pre-reading 5 blocks, program error "P156" results.

(3) When an attempt is made to reverse the compensation direction during nose R compensation, program error "P157" results except when the reversal is done in the G00 block. Even if the directions differ before and after the G28, G30 or G53 block, an error will not result since compensation is temporarily canceled. Using a parameter, the tool can also be moved unchanged in the same compensation direction.

(4) When the compensation direction during nose R compensation coincides with an "×" in the following table, the direction complied with the previous compensation direction.

Determining the compensation direction by the movement vectors and tool nose point in command G46

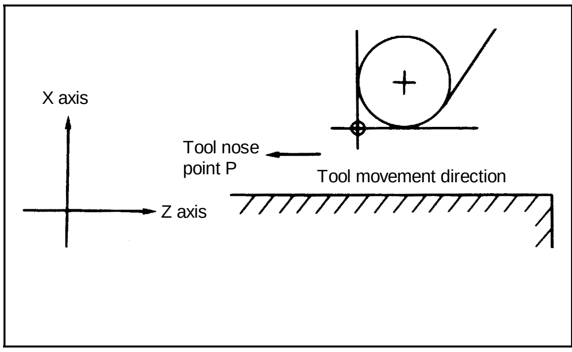
Direction of tool nose advance \ Compensation direction of tool nose		Tool nose points 1	2	3	4	5	6	7	8	Compensation direction of tool nose / Direction of tool nose advance	
Movement vector (tool nose points 1 to 4)		Right	Right	Left	Left	×	Right	×	Left	→	Movement vector (tool nose points 5 to 8)
	↗	×	Right	×	Left	Left	Right	Right	Left		
		Left	Right	Right	Left	Left	×	Right	×	↑	
	↖	Left	×	Right	×	Left	Left	Right	Right		
		Left	Left	Right	Right	×	Left	×	Right	←	
	↙	×	Left	×	Right	Right	Left	Left	Right		
		Right	Left	Left	Right	Right	×	Left	×	↓	
	↘	Right	×	Left	×	Right	Right	Left	Left		

(Note 1) An "×" mark in the tables indicates that the compensation direction is not determined from the movement vector tool nose points.

(Note 2) The"↗" mark denotes a movement vector in the 45° direction. (The other movement vectors are based on this.)

(Note 3) The "" mark denotes a movement vector with a range larger than 45° and smaller than 135°. (The other movement vectors are based on this.)

(Example) With tool nose point 3, movement vector in the Z-axis (–) direction (with ← movement vector)



As shown in the figure on the left, the workpiece is on the X-axis (–) side from the tool nose position and tool movement direction. Consequently, the right side compensation of the workpiece in the direction of the tool's advance serves as the compensation direction.

12.4.2 Nose R compensation operations

Nose R compensation cancel mode

The nose R compensation cancel mode is established by any of the following conditions.

(1) After the power has been turned ON

(2) After the reset button on the setting display unit has been pressed

(3) After the M02 or M03 command with reset function has been executed

(4) After the compensation cancel command (G40) has been executed

(5) After tool number 0 has been selected (T00 has been executed)

The offset vectors are 0 in the compensation cancel mode, and the tool nose point path coincides with the program path.
Programs including nose R compensation must be terminated in the compensation cancel mode.

Nose R compensation start (start-up)

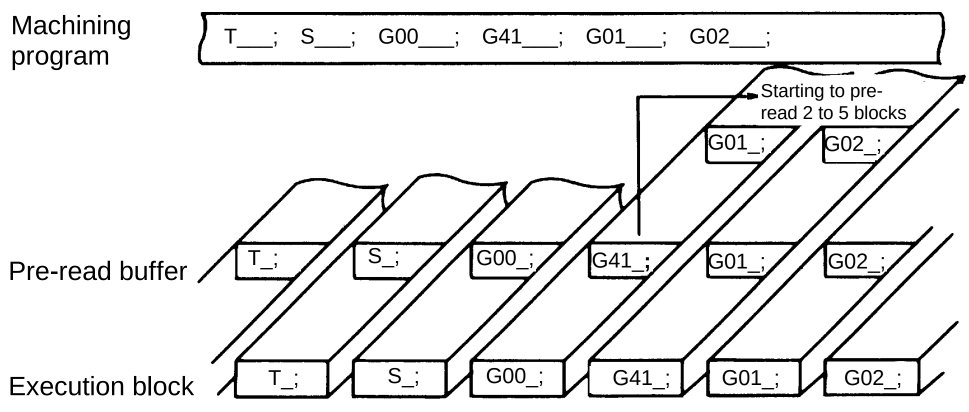
Nose R compensation starts when all the following conditions are met in the compensation cancel mode.

(1) The G41, G42 or G46 command has been issued.

(2) The movement command is any command except a circular command.

At the start of compensation, the operation is executed after at least 2 to 5 blocks have been read continuously for intersection calculation regardless of single block operation (Two flocks are pre-read if movement commands are present; 5 blocks are pre-read if such commands are not present.)
During compensation mode, up to 5 blocks are pre-read and the compensation is calculated.

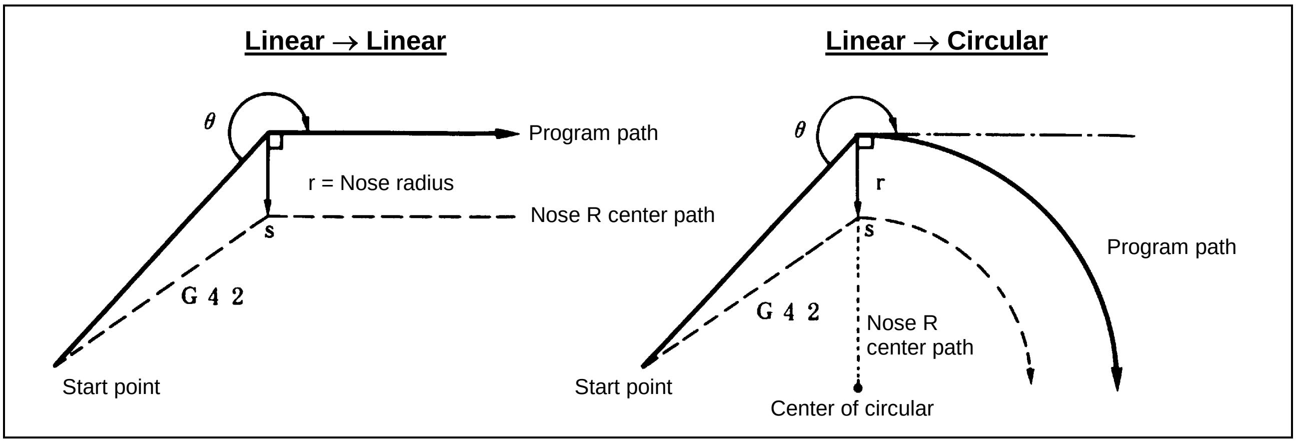
Control mode transition diagram



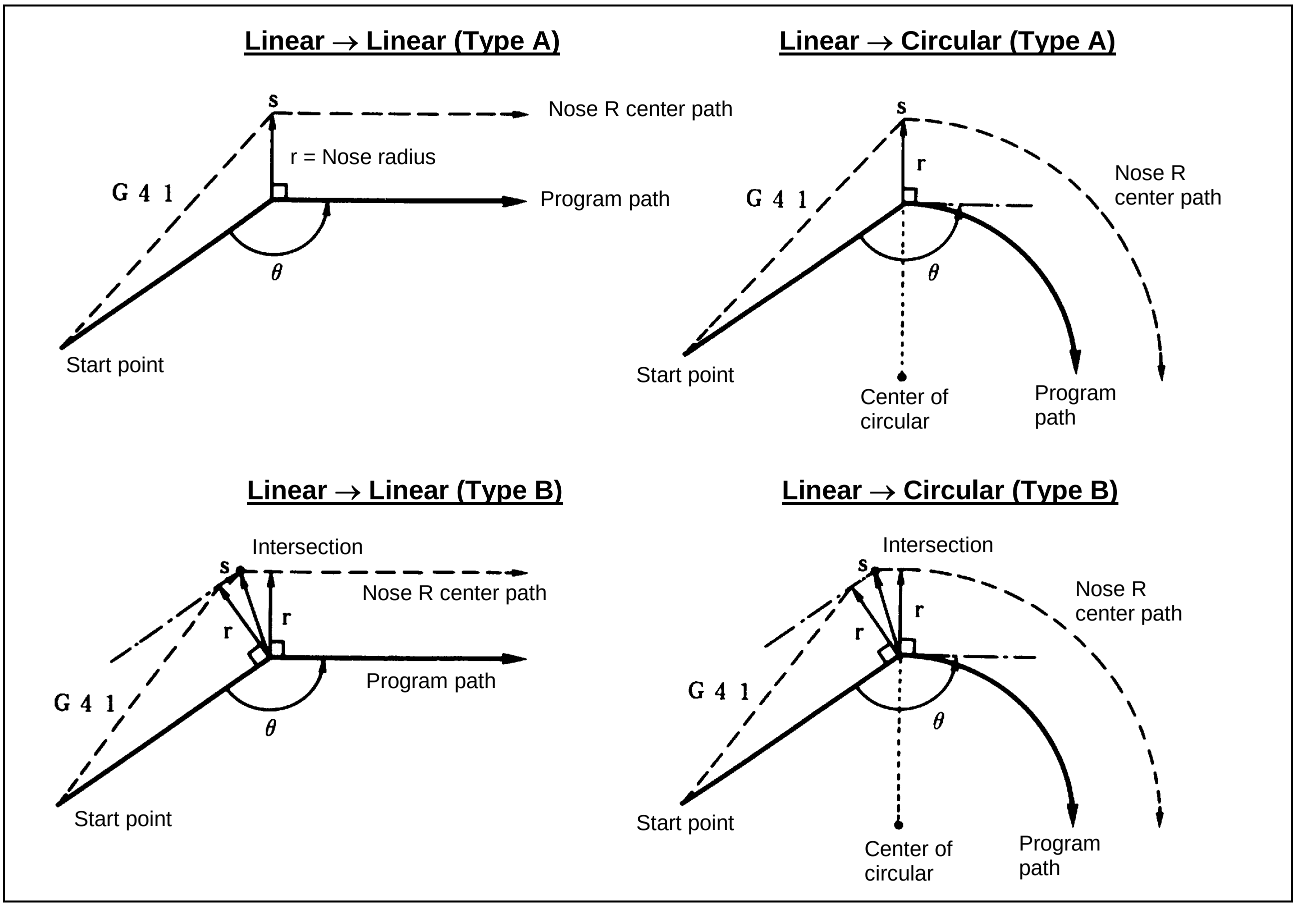
There are two ways of starting the compensation operation: type A and type B.
The type can be selected or de-selected by the control parameter "Radius compen type B".
This type is used in common with the compensation cancel type.
In the following explanatory figure, "S" denotes the single block stop point.

Start operation for tool nose R compensation

(1) Machining an inside corner



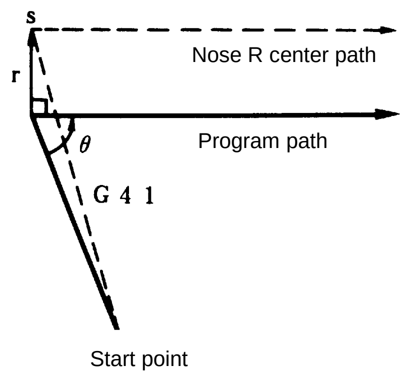
(2) Machining an outside corner (obtuse angle) (elope A or B can be selected it parameter) [90° ≤ θ < 180°]



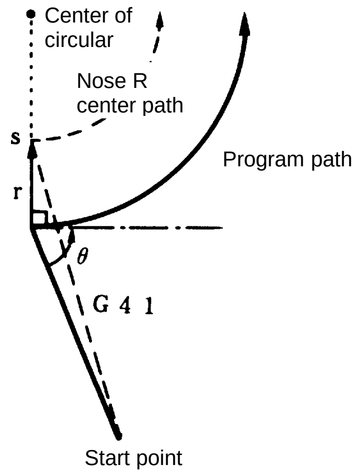
When nose R compensation is started, if G41, G42 or G46 is commanded independently, the tool will not move the amount equal to nose R compensation. The G00 command does not apply nose R compensation. Nose R compensation is applied from the G01, G02 or G03 command. However, even if the axis is commanded, nose R compensation will not be applied if there is no movement.

(3) Machining an outside corner (acute angle) (Type A or B can be selected by parameter) [θ < 90°]

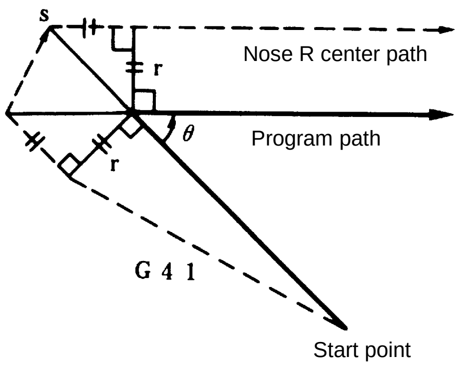
<u>**Linear → Linear (Type A)**</u>



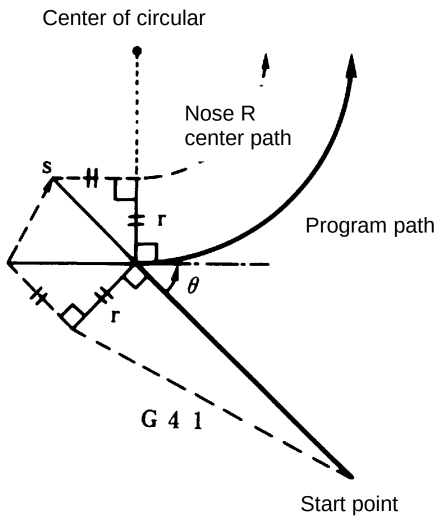
<u>**Linear → Circular (Type A)**</u>



<u>**Linear → Linear (Type B)**</u>



<u>**Linear → Circular (Type B)**</u>



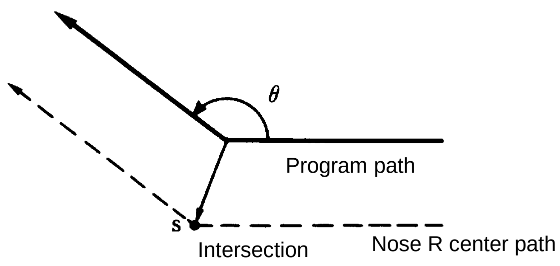
(Note 1) Where is no axis movement command in the same block, compensation is performed perpendicularly to the next block direction.

Operations in compensation mode

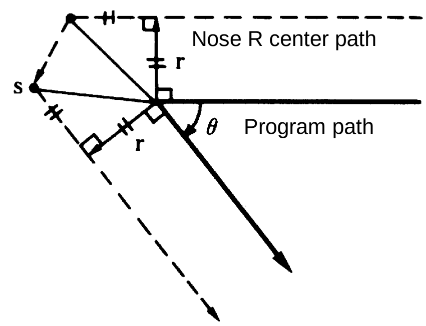
Compensation is valid both for positioning and for interpolation commands such as circular and linear interpolation.
Even if the same compensation command (G41, G42, G46) is issued in a nose R compensation (G41, G42, G46) mode, the command will be ignored.
When 4 or more blocks not accompanying movement are commanded continuously in the compensation mode, overcutting or underwriting will result.
When the M00 command has been issued during nose R compensation, pre-reading is prohibited.

(1) Machining an outside corner

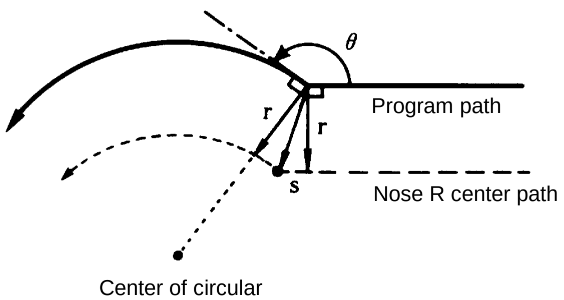
Linear → Linear (90° ≤ θ < 180°)



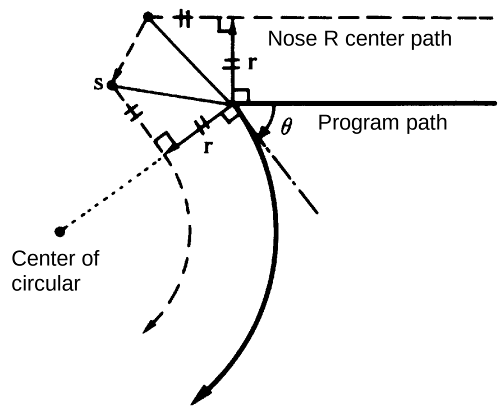
Linear → Linear (0° < θ < 90°)



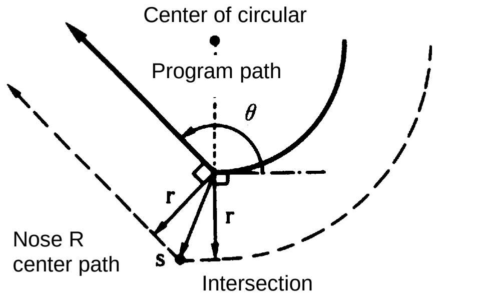
Linear → Circular (90° ≤ θ < 180°)



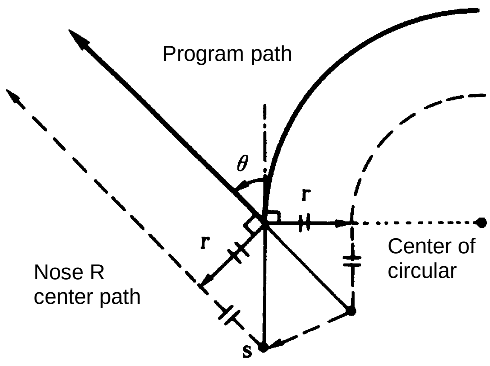
Linear → Circular (0° < θ < 90°)



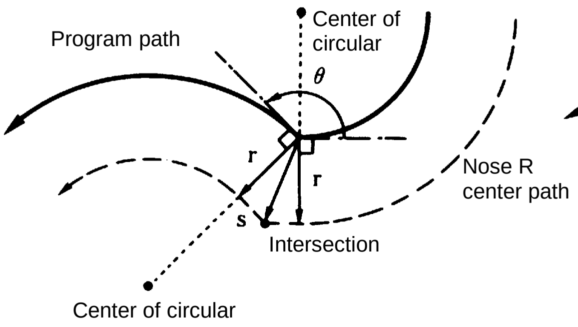
Circular → Linear (90° ≤ θ < 180°)



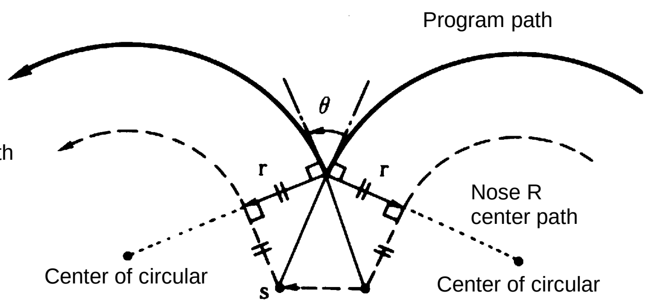
Circular → Linear (0° < θ < 90°)



Circular → Circular (90° ≤ θ < 180°)

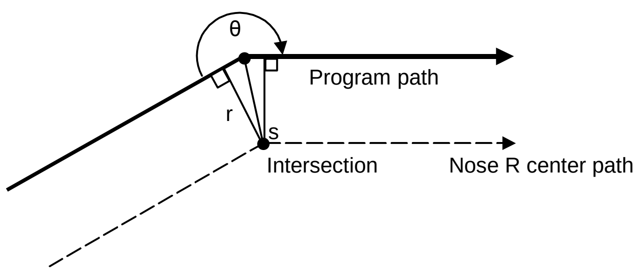


Circular → Circular (0° < θ < 90°)

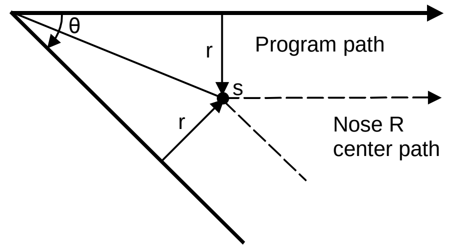


(2) Machining an inner wall

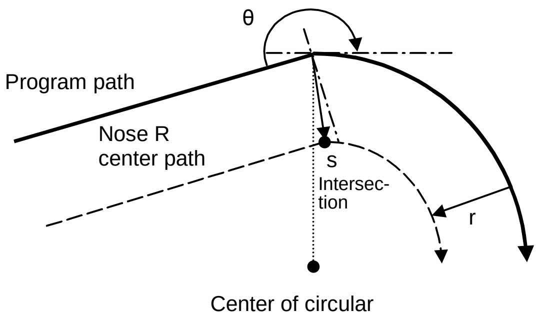
Linear → Linear (Obtuse angle)



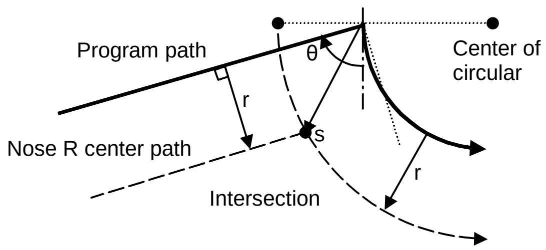
Linear → Linear (Obtuse angle)



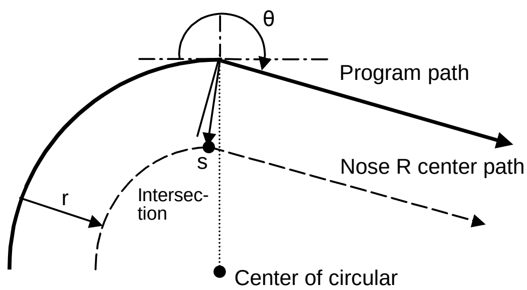
Linear → Circular (Obtuse angle)



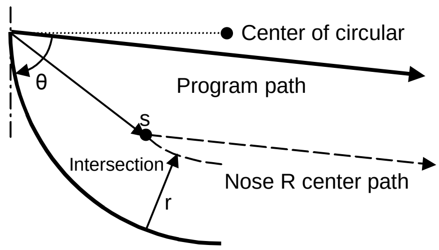
Linear → Circular (Obtuse angle)



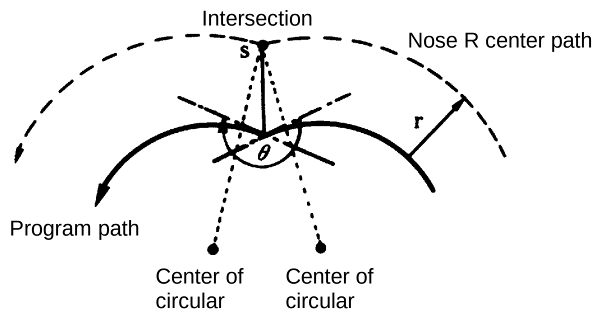
Circular → Linear (Obtuse angle)



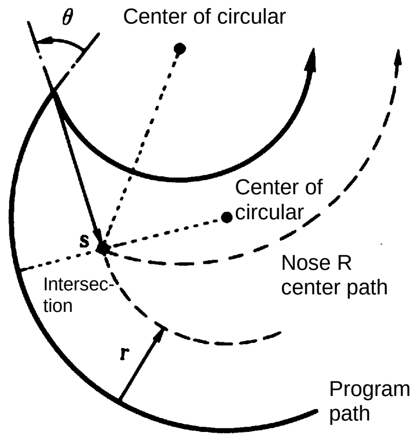
Circular → Linear (Obtuse angle)



Circular → Circular (Obtuse angle)

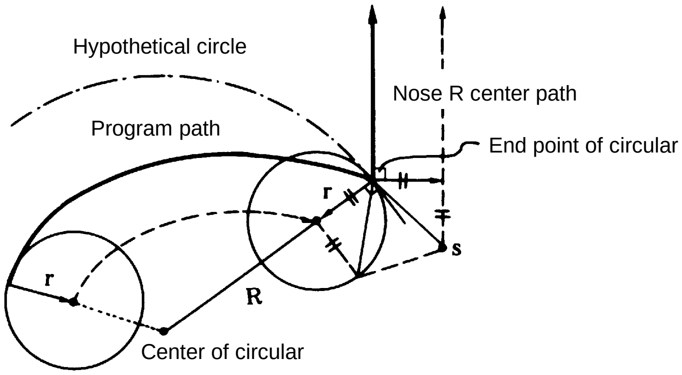


Circular → Circular (Acute angle)



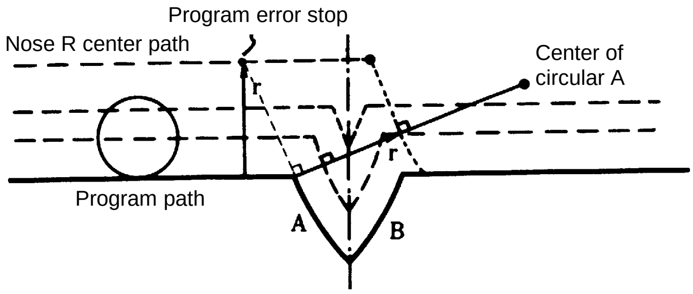
(3) When the arc end point is not on the arc

If the error applying after compensation is within the "G02/03 Error" parameter, the area from the arc start point to the end point is interpolated as a spiral arc.



(4) When the inner intersection does not exist

In an instance such as that shown in the figure below, the intersection of circulars A and B may cease to exist due to the compensation amount. In such cases, program error "P152" appears and the tool stops at the end point of the previous block.



Tool nose radius compensation cancel

If either of the following conditions is met in the nose R compensation mode, the compensation will be canceled. However, the movement command must be a command which is not a circular command.
If the compensation is canceled by a circular command, program error "P151" results.

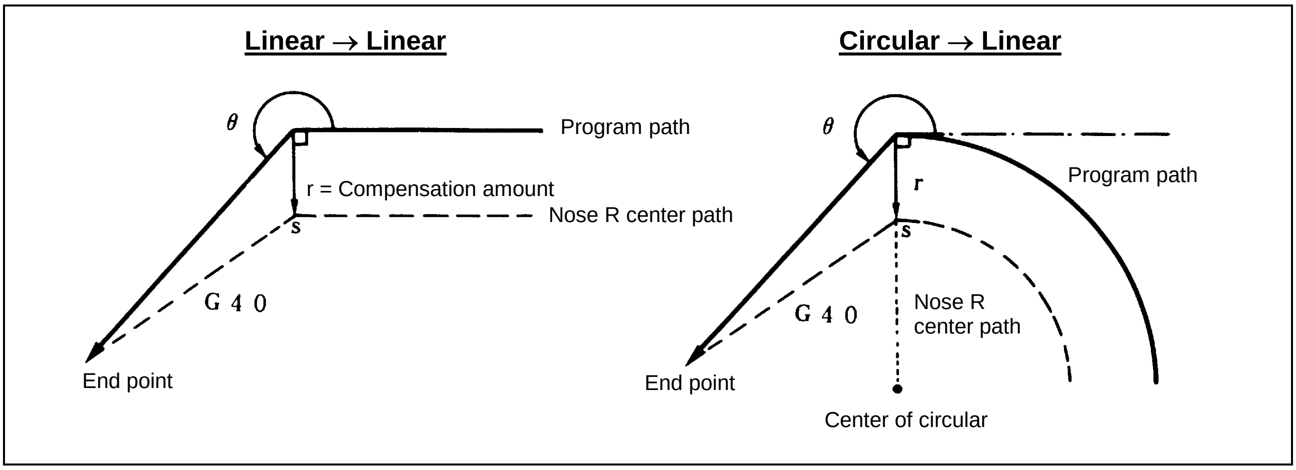
(1) The G40 command has been executed.

(2) The T00 tool number has been executed.

The cancel mode is established once the compensation cancel command has been read, 5-block pre-reading is suspended and 1-block pre-reading is applies instead.

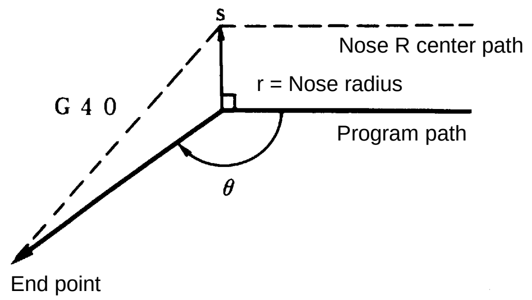
Tool nose radius compensation cancel operation

(1) Machining an inside corner

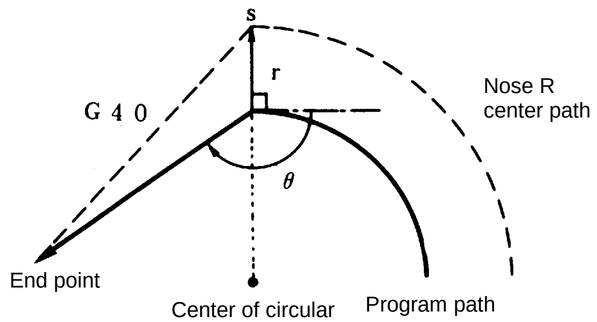


(2) Machining an outside corner (obtuse angle) (type A or B can be selected by parameter)

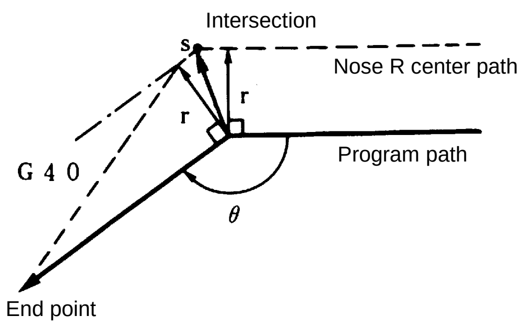
Linear → Linear (Type A)



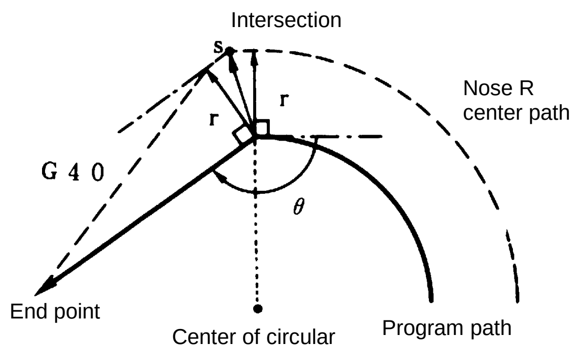
Circular → Linear (Type A)



Linear → Linear (Type B)

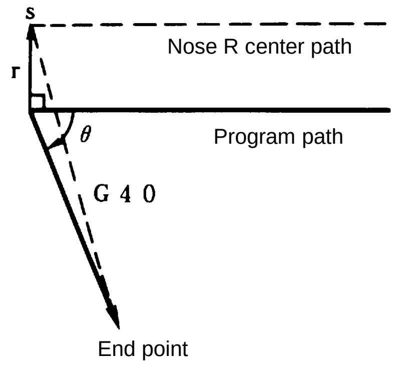


Circular → Linear (Type B)

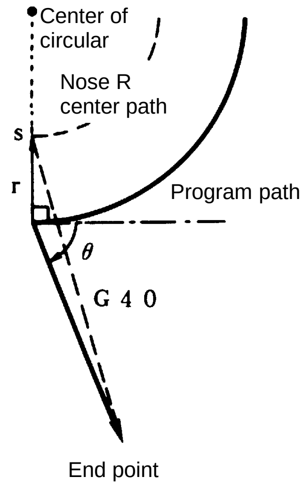


(3) Machining an outside corner (acute angle) (type A or B can be selected by parameter)

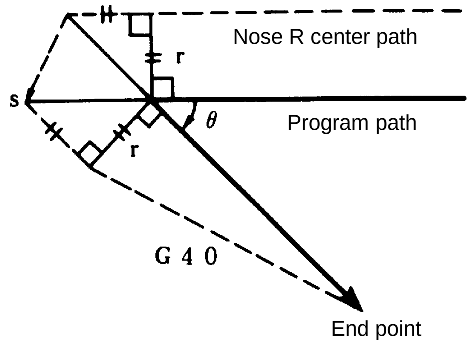
Linear → Linear (Type A)



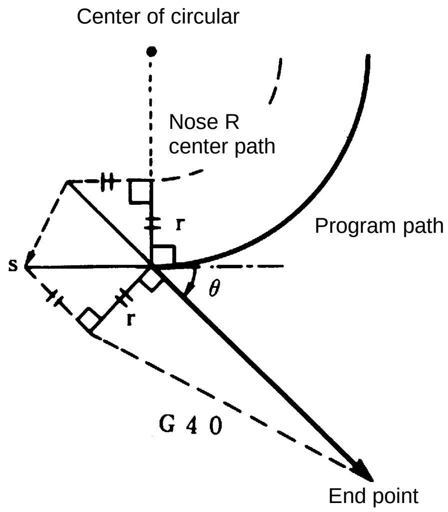
Circular → Linear (Type A)



Linear → Linear (Type B)



Circular → Linear (Type B)



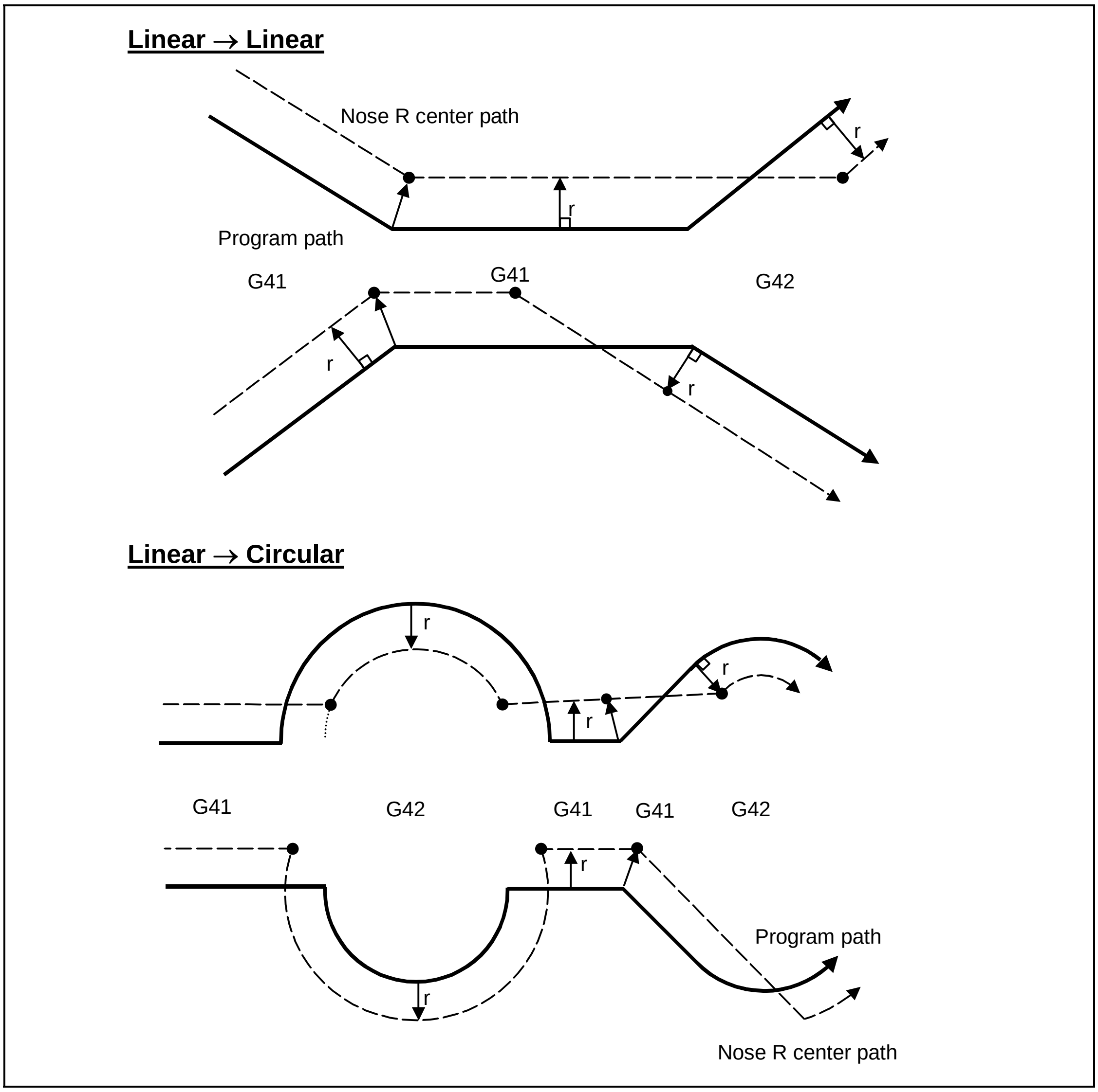
12.4.3 Other operations during nose R compensation

Changing the compensation direction during nose R compensation

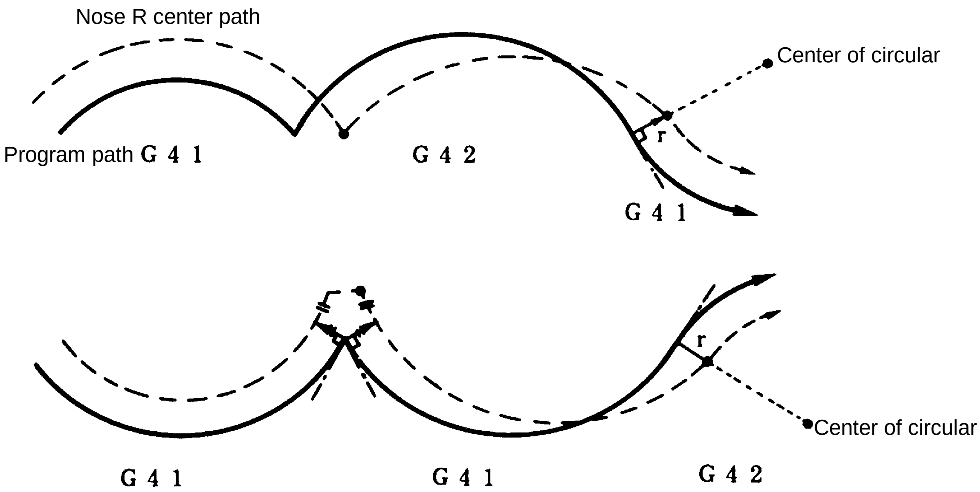
The compensation direction is determined by the nose R compensation commands (G41, G42).

G code	Compensation direction
G41	Left-side compensation
G42	Right-side compensation

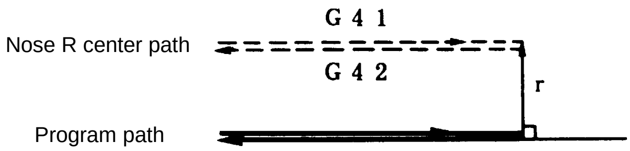
The compensation direction can be changed by changing the compensation command in the compensation mode without the compensation having to be first canceled.
However, no change is possible in the compensation start block and the following block.



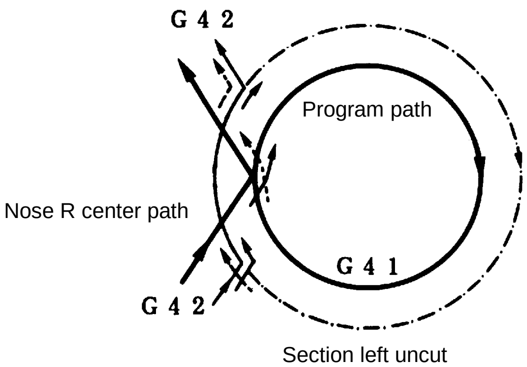
Circular → Circular



Linear return



Arc exceeding 360° due to compensation method

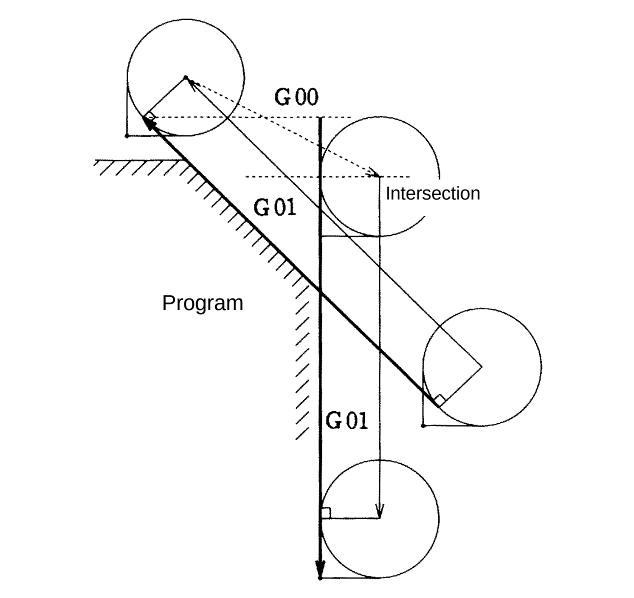
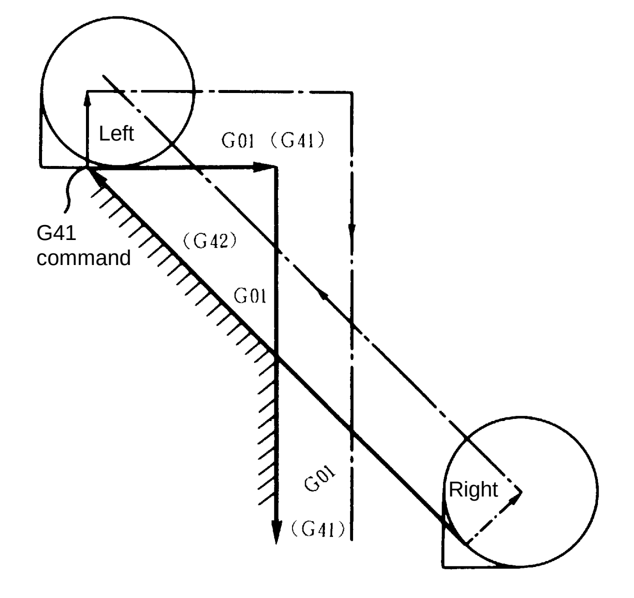


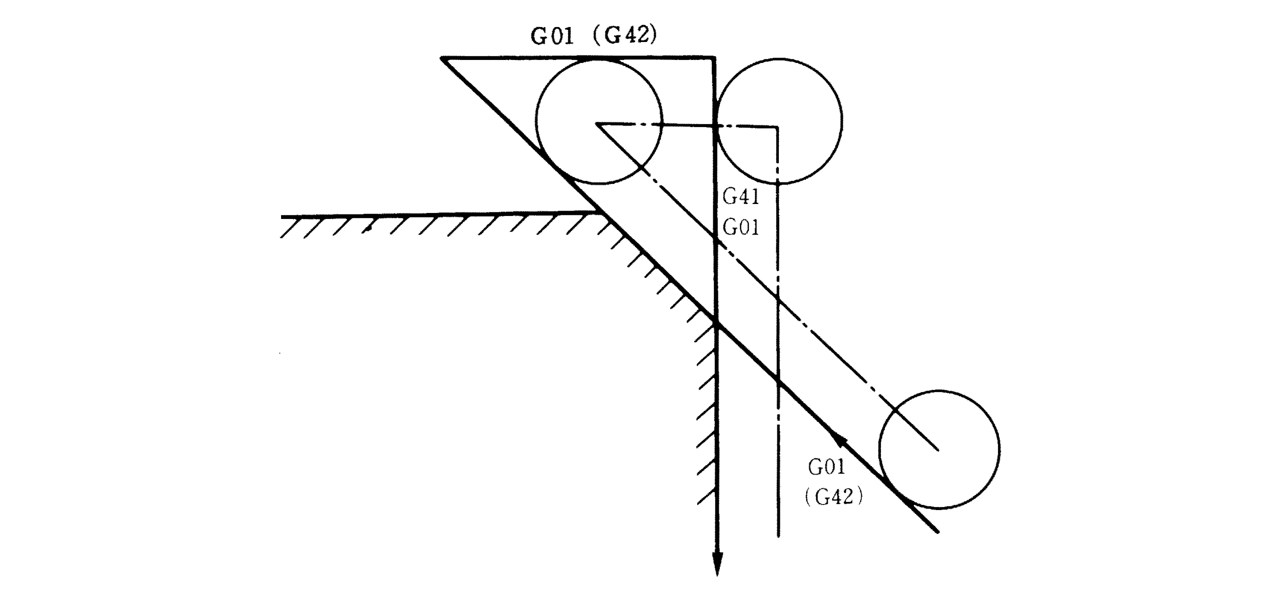
In the cases below, it is possible that the circular may exceed 360°.

a. With compensation direction selection based on G41/G42
b. With I, J, K commanded in G40

In cases like this, compensation is provided shown in the figure on the left and a section will be left uncut.

Nose R compensation of path closed by G46/G41/G42

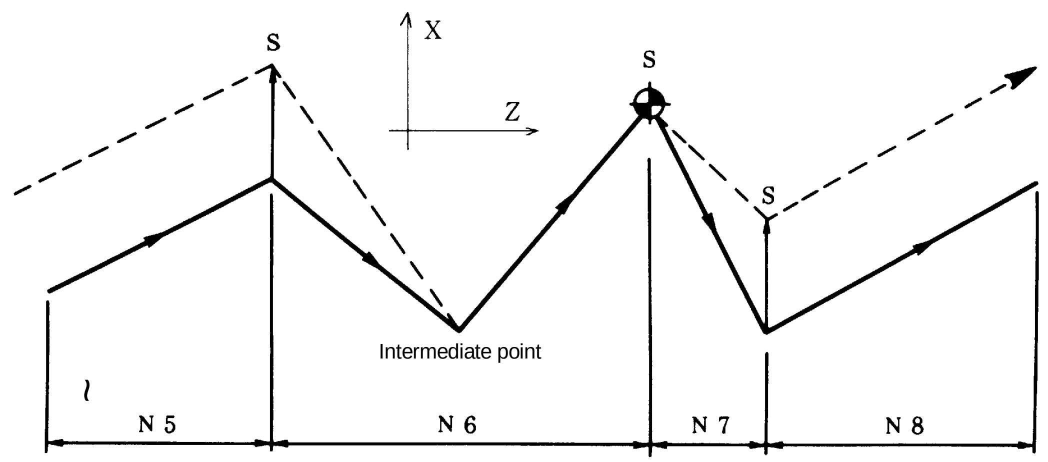
(1) G46 command operation	(2) G42 → G41 command operation
	

(3) G42 → G41 command operation


Command for eliminating offset vectors temporarily

When the following command is issued in the compensation mode, the offset vectors are temporarily eliminated and a return is then made automatically to the compensation mode.
In this case, the compensation is not canceled, and the tool goes directly from the intersection vector to the point without vectors or, in other words, to the programmed command point. When a return is made to the compensation mode, it goes directly to the intersection.

(1) Reference point return command



(G41)

N5 G01 U+60. W+60.;

N6 G28 U–80. W+50.; ← Temporarily no compensation vectors at intermediate point (Reference point when there is no intermediate point)

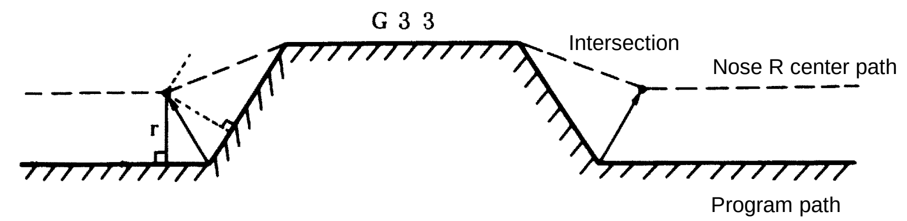
N7 U–120. W+30.;

N8 U+80. W+70.;

(Note 1) The offset vectors do not change with the coordinate system setting (G92) command.

(2) G33 thread cutting command

Nose R compensation does not apply to the G33 block.



(3) Compound fixed cycles

When a compound fixed cycle I command (G70, G71, G72, G73) is assigned, the nose R compensation is temporarily canceled, the finishing shape to which nose R compensation has been applied is turned with the compensation canceled and, upon completion, a return is automatically made to the compensation mode.

Blocks without movement

The following blocks are known as blocks without movement.

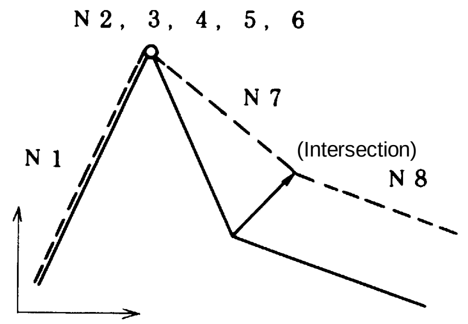
a.	M03; ···········	M command	No movement
b.	S12; ···········	S command	
c.	T0101; ···········	T command	
d.	G04 X500; ···········	Dwell	
e.	G10 P01 R50; ···········	Compensation amount setting	
f.	G92 X600. Z500.; ····	Coordinate system setting	
g.	Y40. ; ···········	Movement but not on compensation plane	
h.	G00; ···········	G code only	
i.	U0 ; ···········	Movement amount 0	··· Movement amount is 0

(1) When commanded at compensation start

Offset vector cannot be generated for a block without movement.

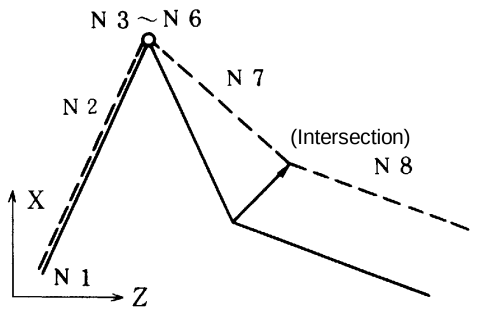
```
N1 U60. W30. T0101;
N2 G41;
N3 G4 X1000;
N4 F100;
N5 S500;
N6 M3;
N7 U–100. W20.;
N8 U–40. W50.;
```

(N2 to N6: Block without movement)




```
N1 T0101;
N2 G41 U120. W30.;
N3 G4 X1000;
N4 F100;
N5 S500;
N6 M3;
N7 U–100. W20.;
N8 U–40. W50.;
```

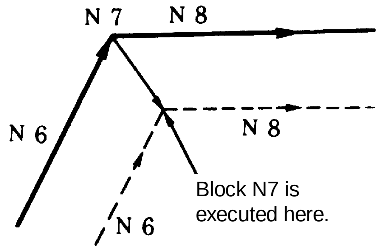
(N3 to N6: Block without movement)



(2) When command is assigned in the compensation mode

When 4 or more blocks without movement does not follow in succession in the compensation mode, the intersection vectors will be created as usual.

N6 U400. W100.;
N7 G04 XP1000; ..... Block without movement
N8 X200.;

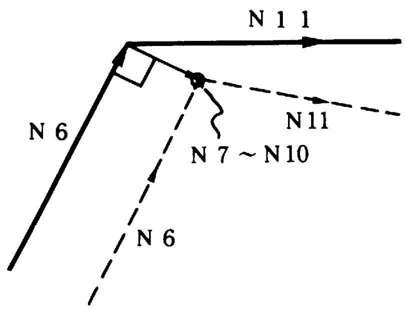


When 4 or more blocks without movement follow in succession, the offset vectors are created perpendicularly at the end point of the previous block.

N6 U200. W200.;
N7 G4 X1000;
N8 F100;
N9 S500;
N10 M4;
N11 X100.;

Block without movement

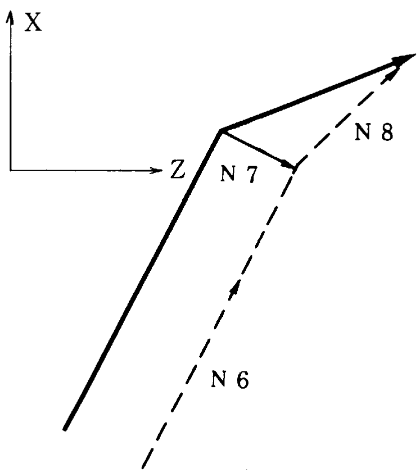
In this case, a cut results.



(3) When commanded together with compensation cancel

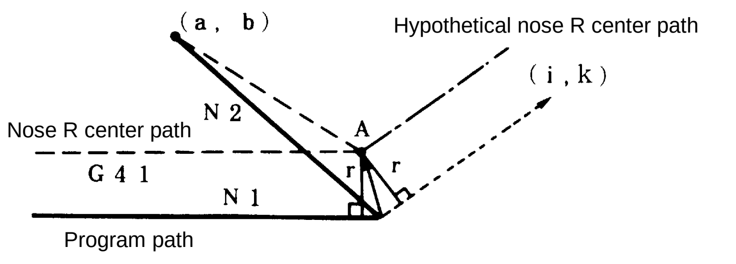
Only the offset vectors are canceled when a block without movement is commanded together with the G40 command.

N6 U400. W100.;
N7 G40 G04 P1000;
N8 U100. W100.;



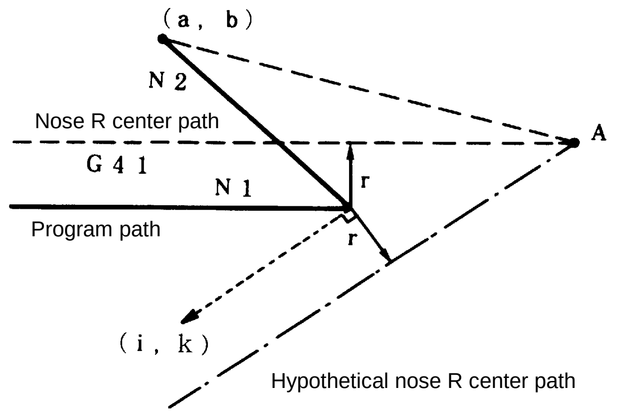
When I, J, K are commanded in G40

(1) When the last movement command block among the 4 blocks before the G40 block is in the G41or G42 mode, it is considered to be commanded in the vector I, J and K direction from the end point of the last movement command, and compensation is canceled after interpolation up to the intersection with the corresponding hypothetical nose R center path. There is no change in the compensation direction.



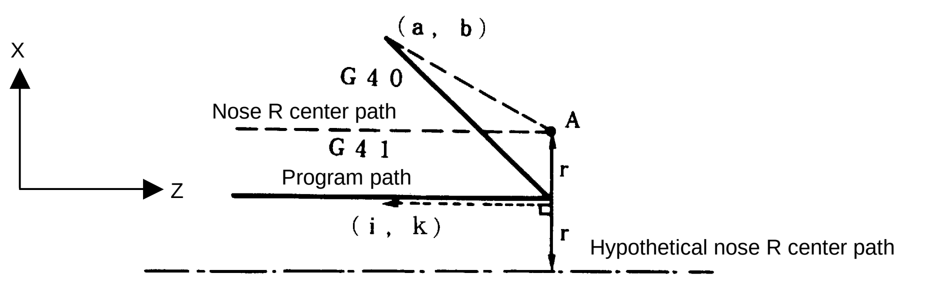
N1 (G41) G1Z__;
N2 G40 XaZbIiKk;

In this case, care is required since the intersection must be sought even when the command vector is incorrect, as in the figure below, regardless of the compensation direction.

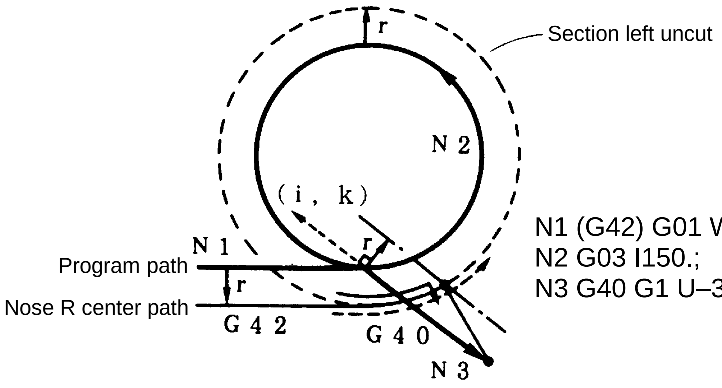


When the I, K signs are incorrect in the above program example

When the intersection calculation and offset vectors cannot be sought, perpendicular vectors are created in the block before G40.



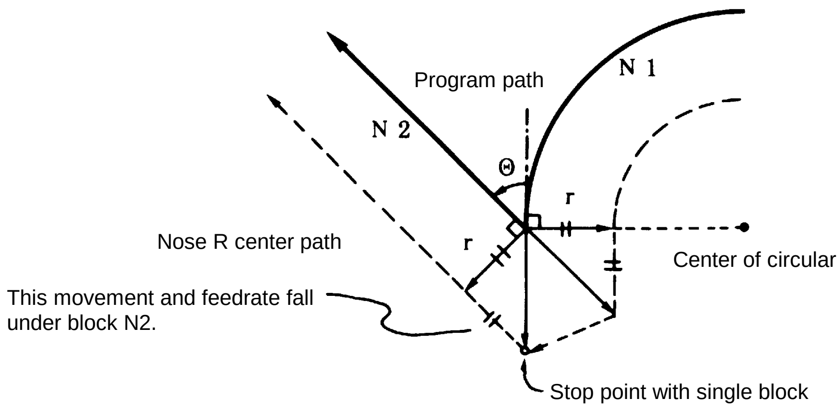
(2) Care should be taken with the G40 command after an circular command since there is a danger of a section being left uncut when the circular exceeds 360° due to the contents of I, J and K.



N1 (G42) G01 W200.;
N2 G03 I150.;
N3 G40 G1 U–300. W150. I100. K–100.;

Corner movement

When a multiple number of offset vectors are created at the joins between movement command blocks, the tool will move in a linear between those vectors. This action is called corner movement.
When the vectors do not coincide, the tool moves in order to machine the corner although this movement is part and parcel of the join block. Consequently, operation in the single block mode will execute the previous block + corner movement as a single block and the remaining joining movement + following block will be executed as a single block in the following operation.



12.4.4 G41/G42 commands and I, J, K designation

Function and purpose

The compensation direction can be intentionally changed by issuing the G41/G42 command and I, J, K in the same block.

Command format

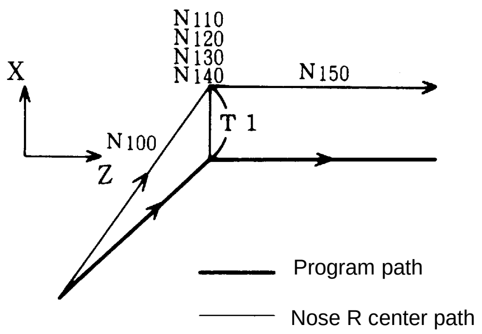
G18 (Z-X plane) G41/G42 X__ Z__ I__ K__ ;

Assign a linear command (G00, G01) in a movement mode.

I, K type vectors (G18 X-Z plane selection)

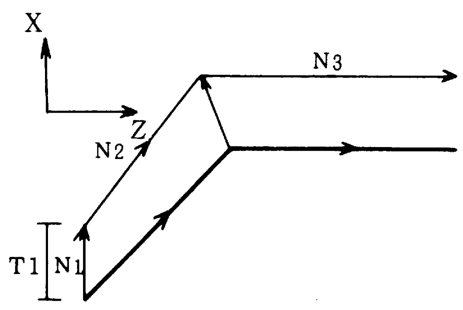
The new I, K type vector (G18 plane) created by this command is now described. (Similar descriptions apply to vector I, J for the G17 plane and to J, K for the G19 plane.)
As shown in the figures below, the vectors with a size equivalent to the compensation amount are made to serve as the I, K type offset vector perpendicularly to the direction designated by I, K without the intersection of the programmed path being calculated. The I, K vector can be commanded even in the mode (G41/G42 mode in the block before) and even at the compensation start (G40 mode in the block before).

(1) When I, K is commanded at compensation start



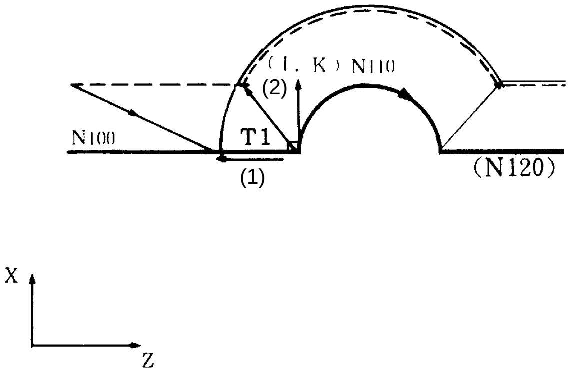
(G40)
≀
N100 G41 U200. W100. K150. T0101;
N110 G04 X1000;
N120 G01 F1000;
N130 S500;
N140 M03;
N150 Z150.;
≀

(2) When there are no movement commands at the compensation start



(G40)
≀
N1 G41 K150. T0101;
N2 U200. W100.;
N3 W150.;
≀

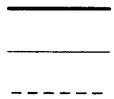
(3) When I, K has been commanded in the mode (G18 plane)



~

(G18 G41 T0010)
N100 G41 G01 W150. I50.;
N110 G02 W100. K50.;
N120 G01 W–150.;

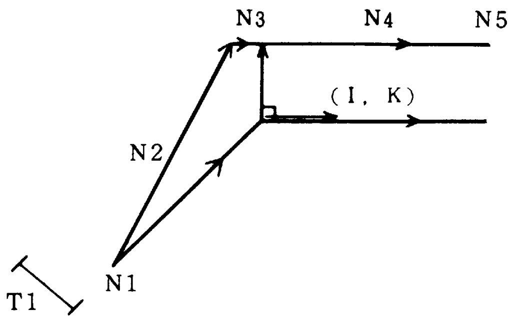
~



Program path
Nose R center path
Path for intersection calculation

(1) I, K type vector
(2) Intersection calculation type vector

(4) When I, K has been commanded in a block without movement



N1 G41 T0101 G01 F1000;
N2 U200. W100.;
N3 G41 K50.;
N4 W150.;
N5 G40;

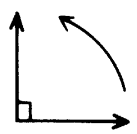
Direction of offset vectors

(1) In G41 mode

Direction produced by rotating the direction commanded by I, K through 90° to the left from the forward direction of the Y axis (3rd axis) as seen from the zero point

(Example 1) With K100.

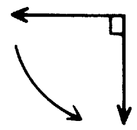
Offset vector direction



(0, 100) IK direction

(Example 2) With K–100.

(0, –100) IK direction



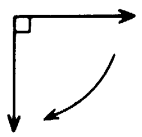
Offset vector direction

(2) In G42 mode

Direction produced by rotating the direction commanded by I, K through 90° to the right from the forward direction of the Y axis (3rd axis) as seen from the zero point

(Example 1) With K100.

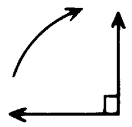
(0, 100) IK direction



Offset vector direction

(Example 2) With K–100.

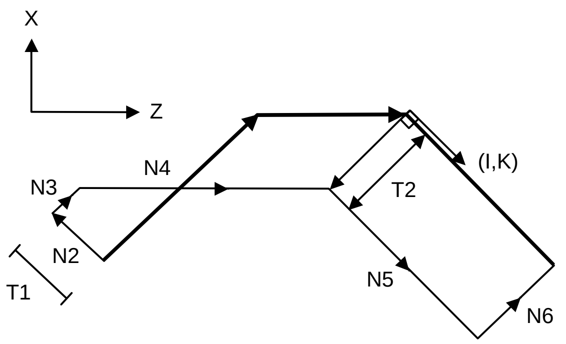
Offset vector direction



(0, –100) IK direction

Selection of offset modal

The G41 or G42 modal can be selected at any time.

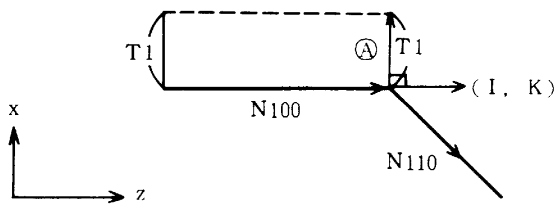



```
N1 G28 X0 Z0;
N2 G41 T0101 F1000;
N3 G01 U200. W100.;
N4 G42 W100. I–100. K100;
N5 X100. Y–100;
N6 G40;
N7 M02;
 %
```

Offset amount for offset vectors

The offset amounts are determined by the offset number (modal) in the block with the I, K designation.

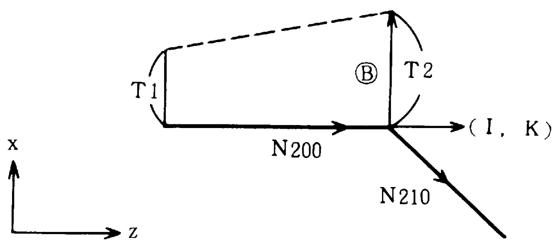
<Example 1>



(G41 T0101)
≀
N100 G41 W150. K50.;
N110 U–200. W100.;
≀

Vector Ⓐ is the offset amount entered in tool offset number modal 1 in the N100 block.

<Example 2>



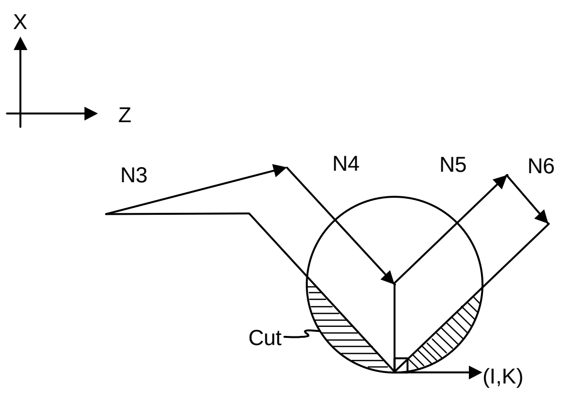
(G41 T0101)
≀
N200 G41 W150. K50. T0102;
N210 U–200. W100.;
≀

Vector Ⓑ is the offset amount entered in tool offset number modal 2 in the N200 block.

Precautions

(1) Issue the I, K type vector in a linear mode (G0, G1). If it is issued in an arc mode at the start of compensation, program error "P151" will result.
An I, K designation in an arc mode functions as an arc center designation in the offset mode.

(2) When the I, K type vector has been designated, it is not deleted (avoidance of interference) even if there is interference. Consequently, overcutting may arise in such a case.




```
N1 G28 X0 Z0;
N2 G41 T0101 F1000;
N3 W100.;
N4 G41 U–200. W100. K10.;
N5 U200. W100.;
N6 G40;
N7 M02;
```

(3) Refer to the following table for the offset methods based on the presence and/or absence of the G41 and G42 commands and I, K (J) command.

G41/G42	I, K (J)	Offset method
No	No	Intersection calculation type vector
No	Yes	Intersection calculation type vector
Yes	No	Intersection calculation type vector
Yes	Yes	I, K type vector No insertion block

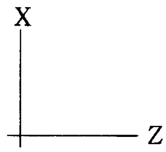
12.4.5 Interrupts during nose R compensation

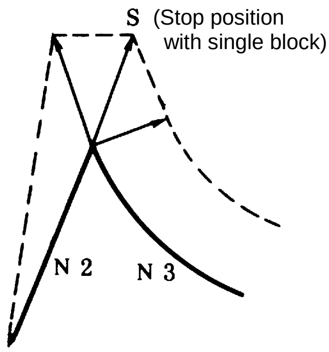
MDI interrupt

Nose R compensation is valid in any automatic operation mode - whether memory or MDI operation.
An interrupt based on MDI will give the result as in the figure blow after block stop during memory operation.

(1) Interrupt without movement (tool path does not change)

```
N1 G41 T0101;
N2 U100. W20.;               MDI interrupt
N3 G3 U–80. W40. R70.;   S1000 M3;
```



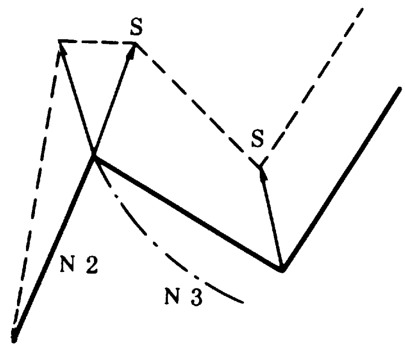


(2) Interrupt with movement

The offset vectors are automatically re-calculated at the movement block after interrupt.

With linear interrupt

```
N1 G41 T0101;
N2 U100. W20.;               MDI interrupt
N3 G3 U–80. W40. R70.;  U–60. W50.;
                        U100. W30.;
```



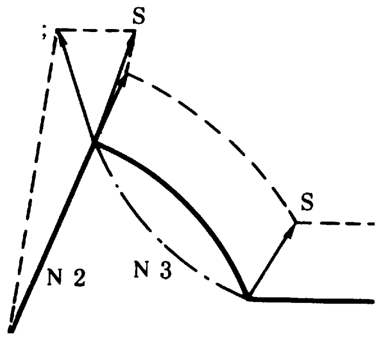
With circular interrupt

N1 G41 T0101;

N2 U100. W20.; MDI interrupt

N3 G3 U–80. W40. R70.; G2 U–80. W40. R70.;

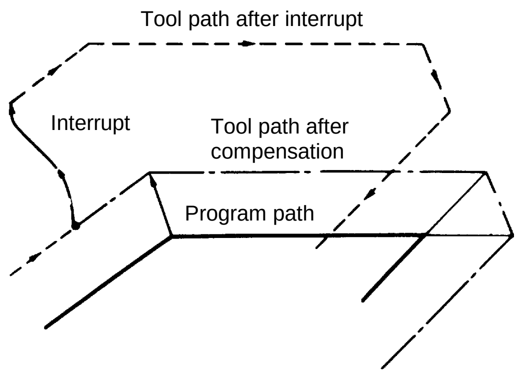
G1 W40.;



Manual interrupt

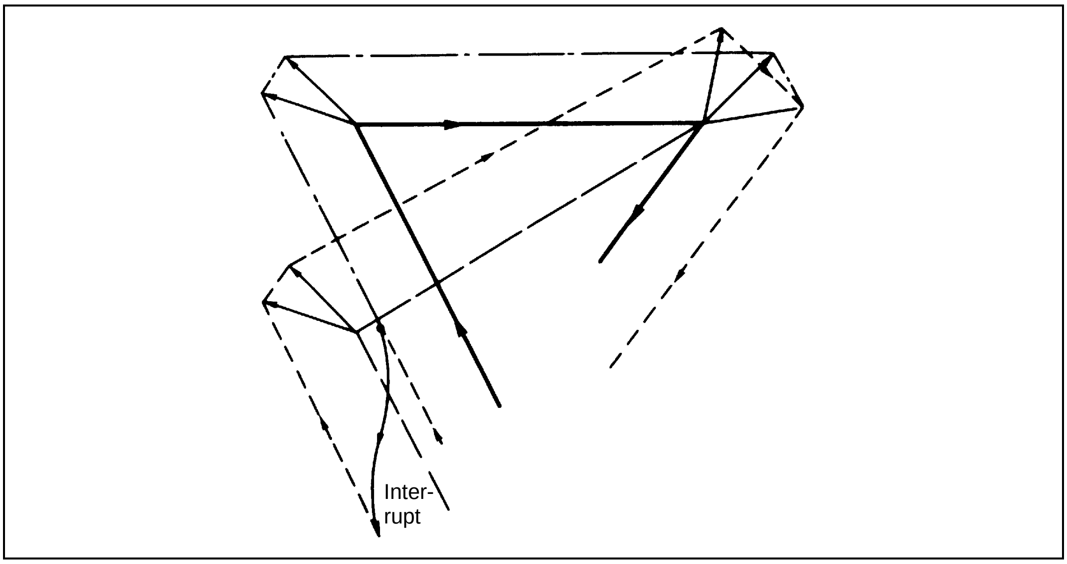
(1) Interrupt with manual absolute OFF

The tool path is shifted by an amount equivalent to the interrupt amount.

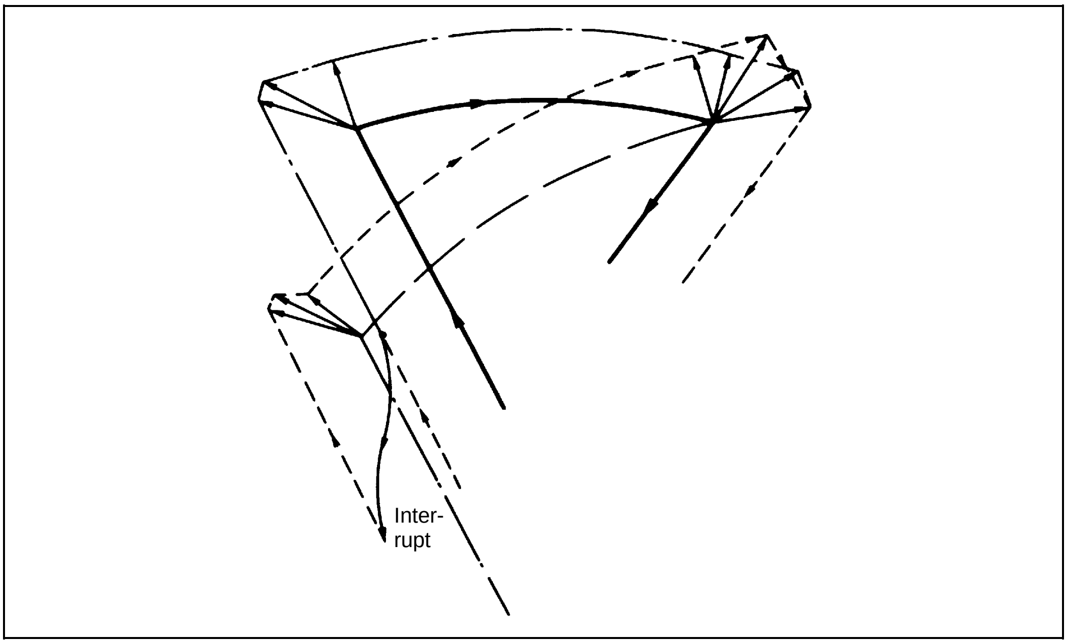


(2) Interrupt with manual absolute ON

In the incremental value mode, the same operation results as with manual absolute OFF.



In the absolute value mode, however, the tool returns to its original path at the end point of the block following the interrupted block, as shown in the figure.



12.4.6 General precautions for nose R compensation

Assigning the compensation amounts

(1) The compensation amount is normally assigned by designating the number of the compensation amount by the last 1 digit or 2 digits of the T code. Depending on the machine specifications, the high-order digits may be used. The T code will remain valid once designated until another T code is subsequently commanded.
Besides being used to designate the compensation amounts for nose R compensation, the T codes are also used to designate the compensation amounts for tool length offset.

(2) The compensation amounts are normally changed when a different tool has been selected in the compensation cancel mode. However, when an amount is changed in the compensation mode, the vectors at the end point of the block are calculated using the compensation amount designated in that block.

Errors during nose R compensation

(1) An error results when any of the following commands are programmed during nose R compensation.
G17, 18, 19 ("P112" when a plane differing from that applying during the compensation has been commanded)
G31 ("P608")
G74, 75, 76 ("P155")
G81 to G89 ("P155")

(2) An error "P158" results when a tool nose point other than 1 to 8 has been designated in the G46 mode.

(3) An error "P156" results when the compensation direction is not determined by the movement vector of the initial cutting command even when the nose R compensation operation has started in the G46 mode and 5 blocks have been pre-read.

(4) An error "P151" results when an circular command is issued in the first or last block of the nose R compensation.

(5) A program error "P157" results when the compensation direction is reversed in the G46 mode.
A parameter can be set to move the tool in the same compensation direction. (Control parameter #8137 G46 no reverse error)

(6) A program error "P152" results during nose R compensation when the intersection is not determined with single block skip in the interference block processing.

(7) A program error results when there is an error in one of the pre-read blocks during nose R compensation.

(8) A program error "P153" results when interference arises under no interference avoidance conditions during nose R compensation.

(9) A program error "P150" results when a nose R compensation command is issued even though the nose R compensation specification has not been provided.

12.4.7 Interference check

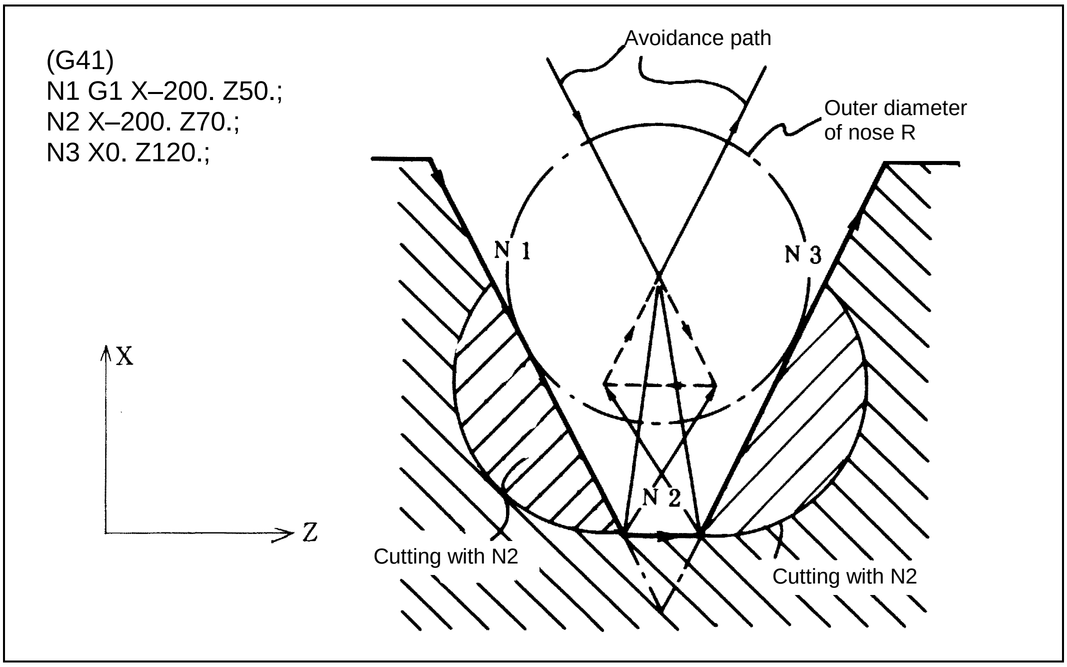
Function and purpose

A tool, whose tool nose has been compensated under the tool nose R compensation function by the usual 2-block pre-read, may sometimes cut into the workpiece. This is known as interference, and interference check is the function which prevents this from occurring.
There are two types of interference check, as indicated below, and each can be selected for use by parameter.

Function	Parameter	Operation
Interference check alarm function	Rad compen intrf byp OFF	A program error results before the execution of the block in which the cut arises, and operation stops.
Interference check avoidance function	Rad compen intrf byp ON	The tool path is changed so that workpiece is not cut into.

Detailed description

(Example)



(1) With alarm function

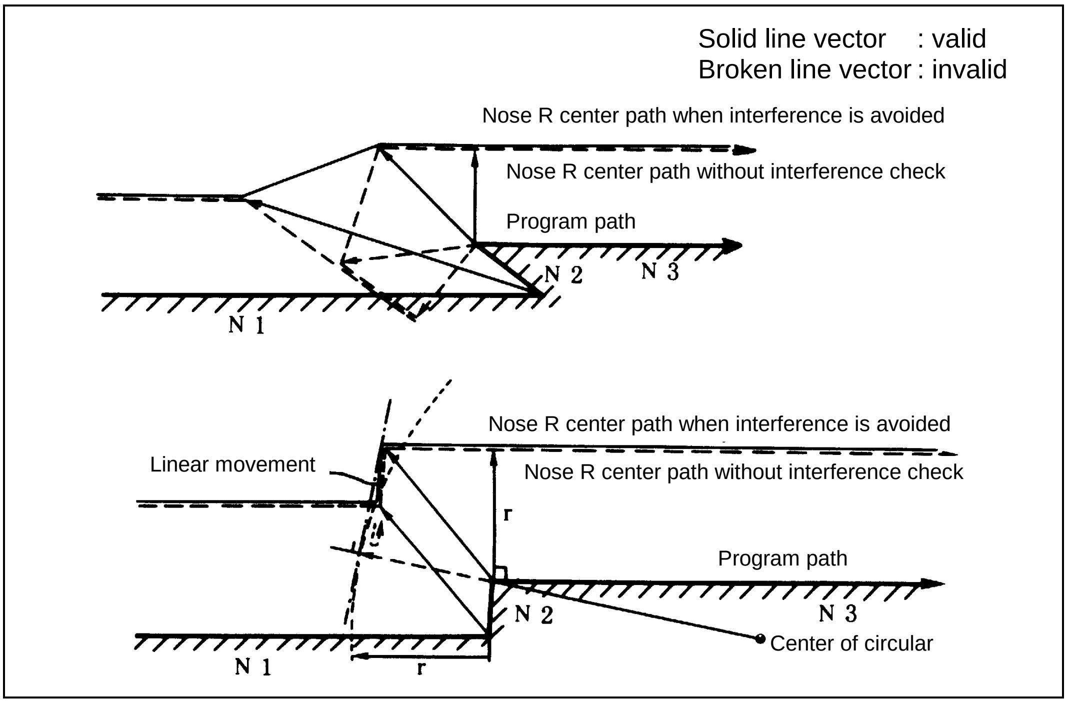
The alarm occurs before N1 is executed and so, using the buffer correction function, N1 can be changed as below and machining can be continued:

N1 G1 X–80. Z20.;

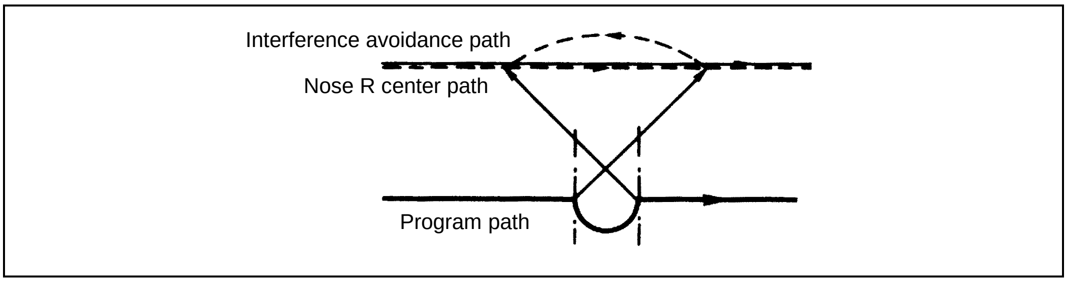
(2) With avoidance function

The intersection of N1 and N3 is calculated and the interference avoidance vectors are created.

Operation during interference avoidance



In the case of the figure below, the groove will be left uncut.

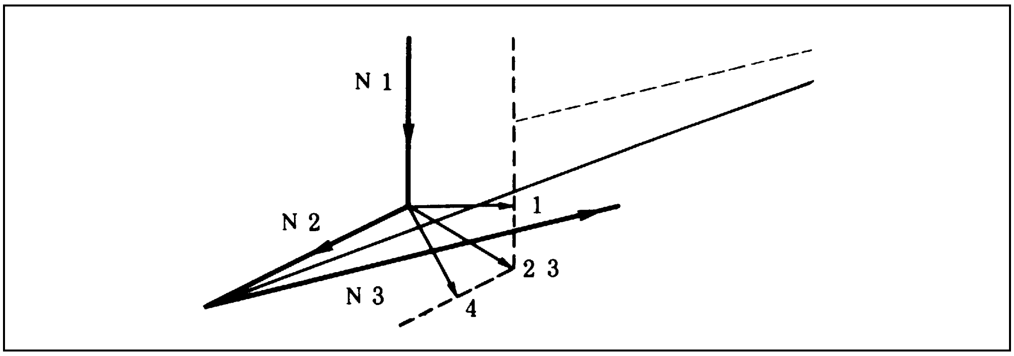


Interference check alarm

The interference check alarm occurs under the following conditions.

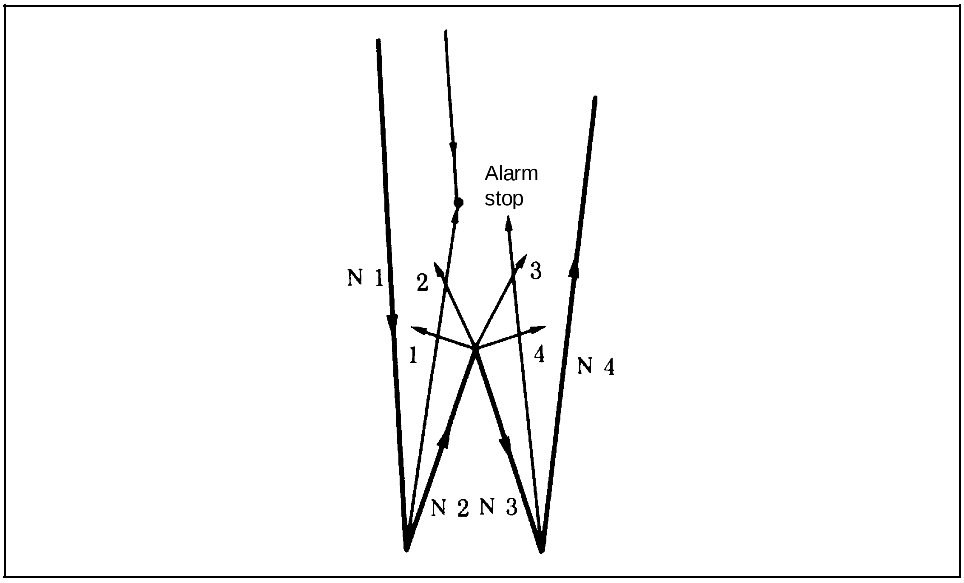
(1) When the interference check alarm function has been selected

(a) When all the vectors at the end block of its own block have been deleted
When, as shown in the figure below, vectors 1 to 4 at the end point of the N1 block have all been deleted, program error "P153" results prior to N1 execution.

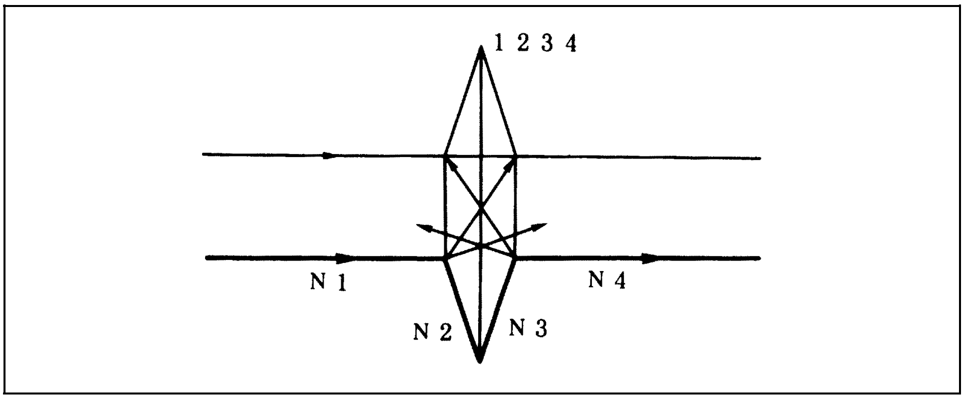


(2) When the interference check avoidance function has been selected

(a) When there are valid vectors at the end point of the following block even when all the vectors at the end point of its own block have been deleted
(i) When, in the figure below, the N2 in interference check is conducted, the N2 end point vectors are all deleted but the N3 end point vectors are regarded as valid. Program error "P153" now occurs at the N1 end point.

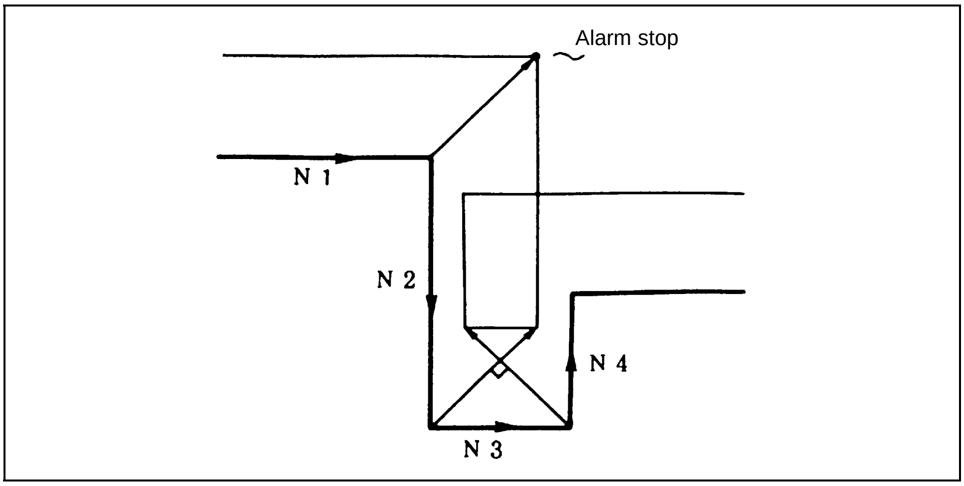


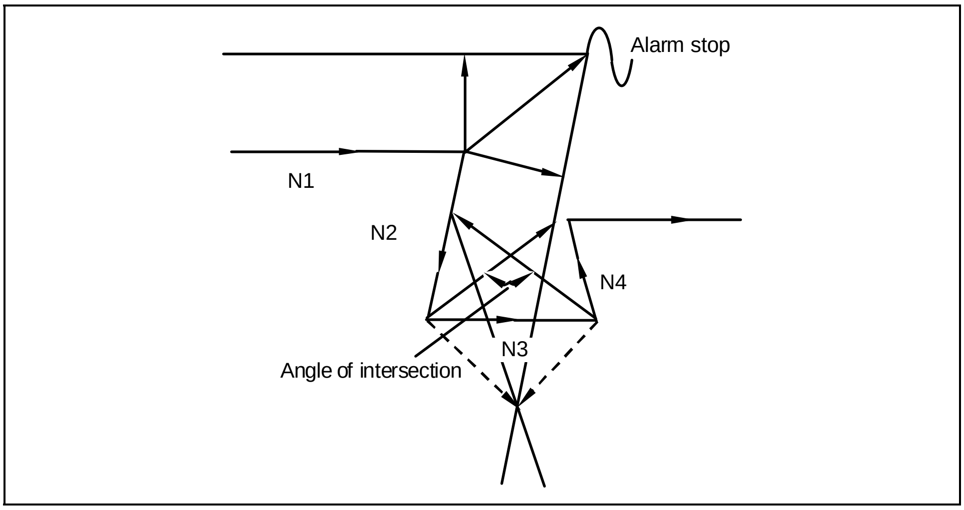
(ii) In a case such as that shown in the figure below, the tool will move in the reverse direction at N2. Program error "P153" now occurs prior to N1 execution.



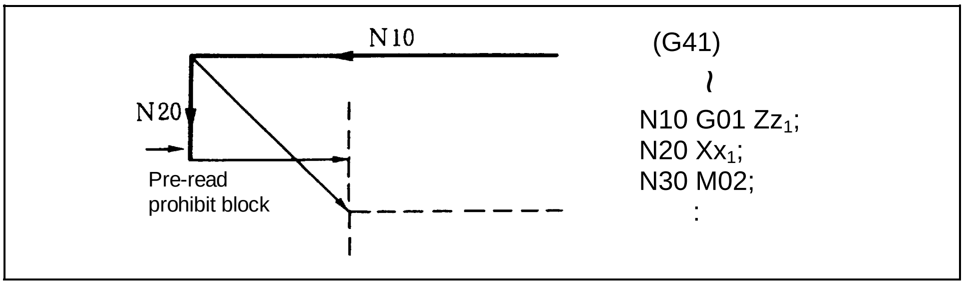
(b) When avoidance vectors cannot be created

(i) Even when, as in the figure below, the conditions for creating the avoidance vectors are met, it may still be impossible to create these vectors or the interference vectors may interfere with N3.
As a result, program error "P153" will occur at the N1 end point when the vector intersecting angle is more than 90°.

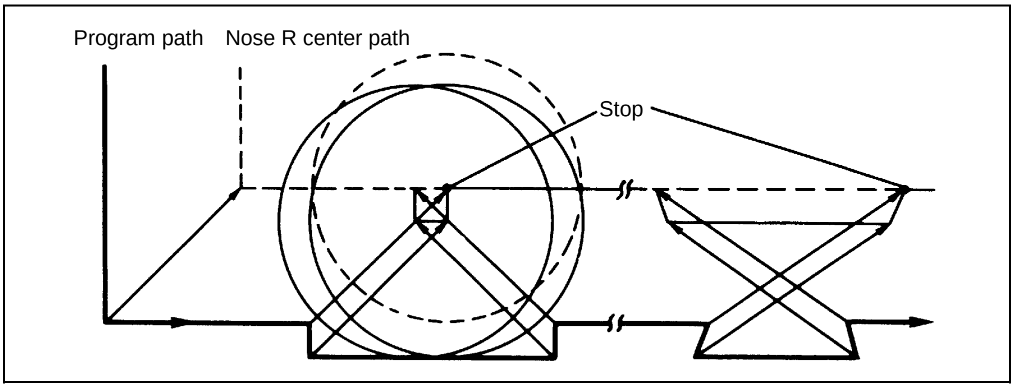




(ii) Avoidance vectors cannot be created when pre-read prohibit blocks are interfered with and so program error "P153" occurs.



(c) When the program advance direction and the advance direction after compensation are reversed.
In the following case, interference is still regarded as occurring even when there is actually no interference.
When grooves which are narrower than the nose R diameter or which have parallel or widening walls are programmed.



12.5 Programmed Tool Offset Input; G10

Function and purpose

The amount of tool offset and workpiece offset can be set or changed by the G10 command. When commanded with absolute values (X, Z, R), the commanded offset amounts serve as the new amounts; when commanded with incremental values (U, W, C), the new offset amounts are equivalent to the commanded amounts plus the current offset amount settings.

Command format

(1) Workpiece offset input (L2)

G10 L2 P__ X__ (U__) Z__ (W__) ;

P_	Offset number
X_	X-axis offset amount (absolute)
U_	X-axis offset amount (incremental)
Z_	Z-axis offset amount (absolute)
W_	Z-axis offset amount (incremental)

(2) Tool length offset input (L10)

G10 L10 P__ X__ (U__) Z__ (W__) R__ (C__) Q__ ;

P_	Offset number
X_	X-axis offset amount (absolute)
U_	X-axis offset amount (incremental)
Z_	Z-axis offset amount (absolute)
W_	Z-axis offset amount (incremental)
R_	Nose R compensation amount (absolute)
C_	Nose R compensation amount (incremental)
Q_	Hypothetical tool nose point

(3) Tool nose wear offset input (L11)

G10 L11 P__ X__ (U__) Z__ (W__) R__ (C__) Q__ ;

P_	Offset number
X_	X-axis offset amount (absolute)
U_	X-axis offset amount (incremental)
Z_	Z-axis offset amount (absolute)
W_	Z-axis offset amount (incremental)
R_	Nose R compensation amount (absolute)
C_	Nose R compensation amount (incremental)
Q_	Hypothetical tool nose point

(4) When there is no L command with tool length offset input (L10) or tool nose wear offset input (L11)

Tool length offset input command : P = 10000 + offset number
(Equivalent to G10 L10 P compensation number)
Tool nose wear offset input : P = offset number
(Equivalent to G10 L11 P compensation number)

Detailed description

(1) The following table shows the offset numbers and the setting ranges of the hypothetical tool nose points.

Add-ress	Signifi-cance of address	Setting range		
		L2	L10	L11
P	Offset number	0: External workpiece offset 1: G54 workpiece offset 2: G55 workpiece offset 3: G56 workpiece offset 4: G57 workpiece offset 5: G58 workpiece offset 6: G59 workpiece offset	When L command is present: 1 to Max. number of offset sets When L command is not present: 10001 to 10000 + Max. number of offset sets	When L command is/is not present: 1 to Max. number of offset sets
Q	Hypothetical tool nose point	—	0 to 9	

(Note 1) The maximum number of offset sets for P (offset number) with tool offset input (L10 or L11) is 20 under the standard specifications and up to a total of 80 with the addition of options.

(2) The setting range for the offset amount is given blow.
Program error "P35" occurs for any value not listed in the table after command unit conversion.
With an incremental value command, the setting range for the offset amount is the sum of the present setting and command value.

Input unit	Tool length offset amount		Tool wear offset amount	
	Metric system	Inch system	Metric system	Inch system
IS - B	±99999.999 (mm)	±9999.9999 (inch)	±99999.999 (mm)	±9999.9999 (inch)
IS - C	±9999.9999 (mm)	±999.99999 (inch)	±9999.9999 (mm)	±999.99999 (inch)

(3) The number of axes that can be commanded is as follows.

- Workpiece offset input : Maximum 5 axes
- Tool length offset input : Maximum 3 axes
- Tool nose wear offset input : Maximum 3 axes

Precautions and restrictions

(1) Offset setting range check
The maximum value of the wear offset amount and the maximum additional value (Setup parameter 8031/8032) for the wear offset input check respectively take precedence for a single-time offset amount in the maximum value and incremental value command of the wear offset amount, and when an amount greater than these values has been commanded, program error "P35" results.

(2) G10 is an unmodal command and is valid only in the commanded block.

(3) The data changed with workpiece offset input is validated at the next workpiece coordinate system selection command. The data changed with tool length offset input or tool nose wear offset input is validated at the next T command.

(4) Address C is treated as an incremental command value of the tool nose radius in the L10 or L11 command. In L10 and L11, address C is handled as the nose R incremental command value, thus, an absolute value cannot be commanded for the axis named "C".

(5) If an illegal L number or P number is commanded, program error "P172" or "P170" will result, respectably. Program error "P170" occurs when the P command is omitted.

(6) Program error "P35" results when the offset amount exceeds the setting range.

(7) X, Z and U, W are input together in a single block but when an address that commands the same offset input (X, U or Z, W) is commanded, the address which is input last is valid.

(8) Offset will be input if even one address following "G10 L (2/10/11) P__" is commanded. The command is ignored when not even a single command has been assigned. (Offset is not input.)

(Example) G10 L10 P3 Z50.;

↓

[Tool length data]

#	Z
3	50.000

Input as per left

(9) When the absolute/incremental address is invalid (when 8106 ABS/INC Addr. is invalid), addition is possible with the incremental amount using the G91 mode + absolute value address command.

(10) Decimal points are valid for offset amounts.

(11) Program error "P171" occurs when this command is input even though the corresponding specifications have not been provided.

(12) G40 to G42 are ignored when they have been commanded in the same block as G10.

(13) Do not command G10 in the same block as fixed cycles and subprogram call commands. This will cause malfunctioning and program errors.

(14) When a T command has been issued in the same block as G10 with Tmove (tool offset operation) as 0 (Base common parameter 1317), the offset will be accomplished in the following block.

12.6 Common System Offset

Function and purpose

In the multi-system NC, when the basic common parameter 1334 Tcom∗ is turned ON, offset data is valid for all common offset systems.
With this convenient function, one tool can be used in multiple systems. Programming is easy; there is no need to distinguish the T command in the machining program.

Command format

```
T □□ □□ ;
       └── Tool offset number
    └───── Tool number
└───────── T command
```

```
T □□ ;
    └── Tool offset number
└────── T command
```

Detailed description

(1) This command is issued with 4 digits or 2 digits that follow the T address. (When two digits are commanded, the tool number will not be output.)

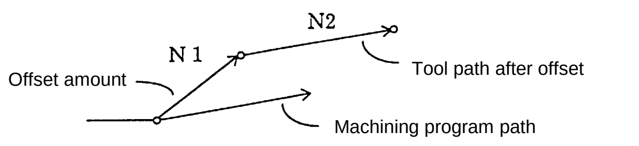
(2) One set of this command can be issued in one block.

(3) The T function can be used simultaneously with any other command. However, when in the same block as the movement command, there are two types of command execution procedures. (Which procedure is suitable depends on the basic common parameter Tmove).

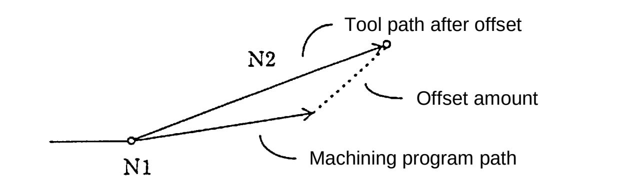
(a) Execute the T function after movement is completed.

(b) Execute T function and movement command at the same time.

(i) Tmove = 0

	N1 T0101; N2 X100. Z200.;

(ii) Tmove = 1

	N1 T0101; N2 X100. Z200.;

13. PROGRAM SUPPORT FUNCTIONS

13.1 Fixed Cycles for Turning

Function and purpose

When performing rough cutting and other such operations during turning, these functions enable shapes normally commanded in several blocks to be commanded in a single block. In other words, they simplify the machining program.

G code	Function
G77	Longitudinal turning cycle
G78	Thread cutting cycle
G79	Face turning cycle

Command format

G77 X/U_ Z/W_ R_ F_ ;

(Same for G78, G79)

Detailed description

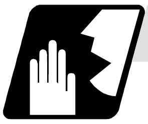
(1) Fixed cycle commands are modal G codes and so they are valid until another command in the same modal group or a cancel command is issued.
The following G code cancel commands are available.

G00, G01, G02, G03
G07,
G09,
G10, G11,
G27, G28, G29, G30,
G31,
G33, G34,
G37,
G92,
G52, G53,
G65,

(2) There are two fixed cycle call types, movement command block call and block-by-block call. These are selected by a control parameter setting.
A movement command block call serves to call the fixed cycle macro subprogram only when there is an axial movement command in the fixed cycle mode. The block-by-block call serves to call the fixed cycle macro subprogram in each block in the fixed cycle mode. Both types are executed until the fixed cycle is canceled.

(3) A manual interrupt can be applied while a turning fixed cycle (G77 to G79) is being executed. Upon completion of the interrupt, however, the tool must be returned to the position where the manual interrupt was applied and then the turning fixed cycle should be resumed. If it is resumed without the tool having been returned, all subsequent operations will deviate by an amount equivalent to the manual interrupt.

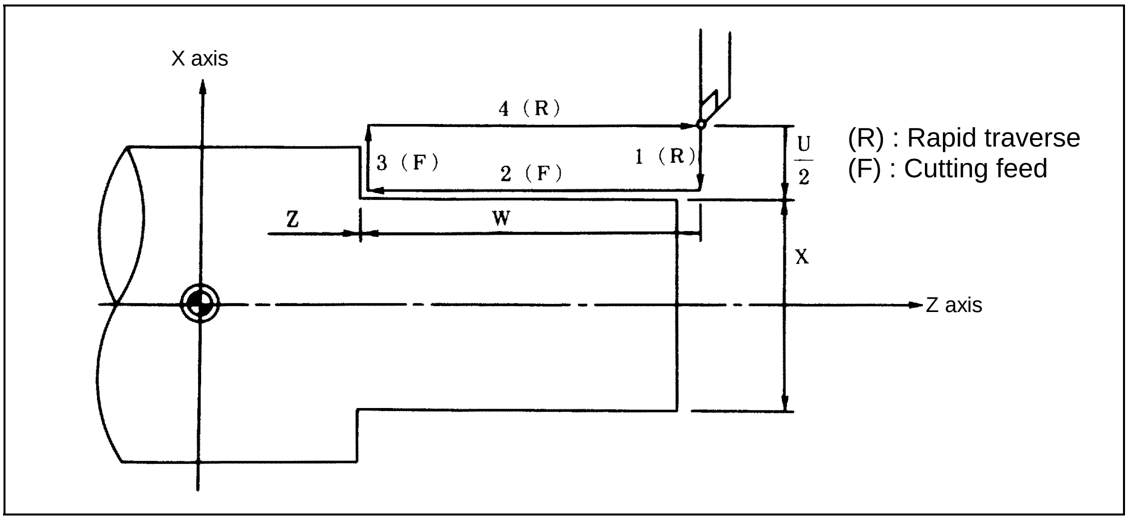
13.1.1 Longitudinal cutting cycle; G77

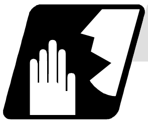


Straight cutting

This function enables continuous straight cutting in the longitudinal direction using the following command.

G77 X/U_ Z/W_ F_;

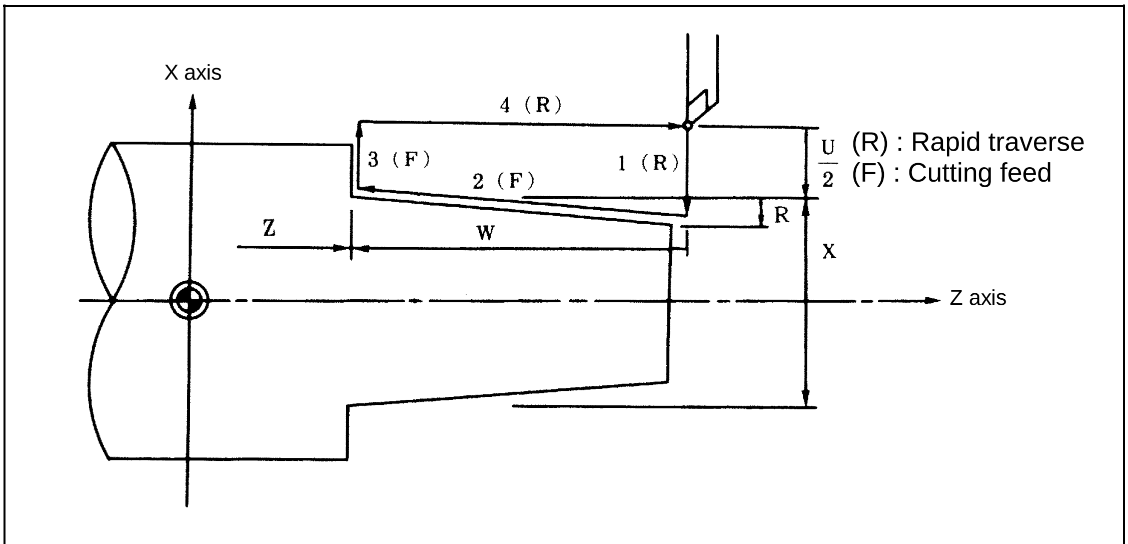




Taper cutting

This function enables continuous taper cutting in the longitudinal direction using the following command.

G77 X/U_ Z/W_ R_ F_;

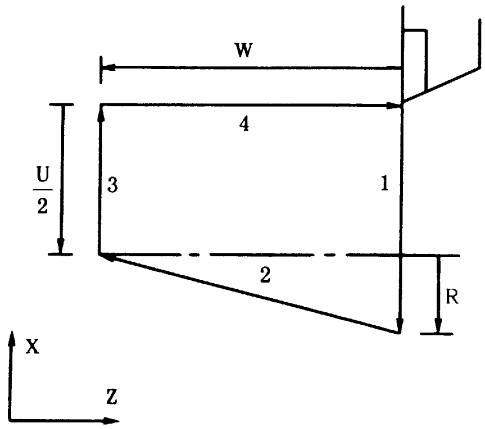


R : Taper depth (radius designation, incremental value, sign required)

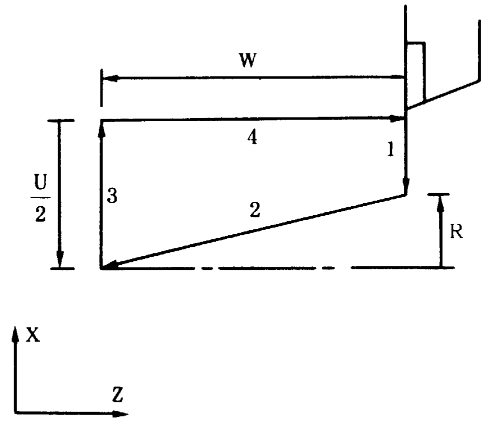
With a single block, the tool stops at the end points of operations 1, 2, 3 and 4.

Depending on the U, W and R signs, the following shapes are created.

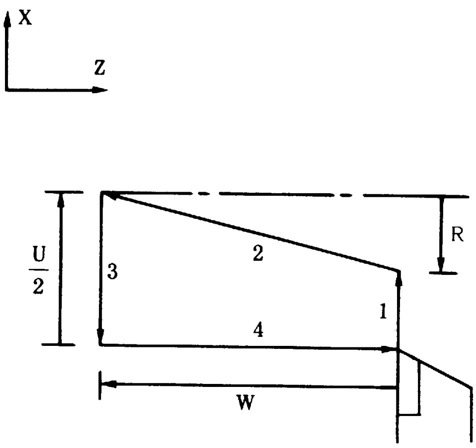
(1) U < 0, W < 0, R < 0



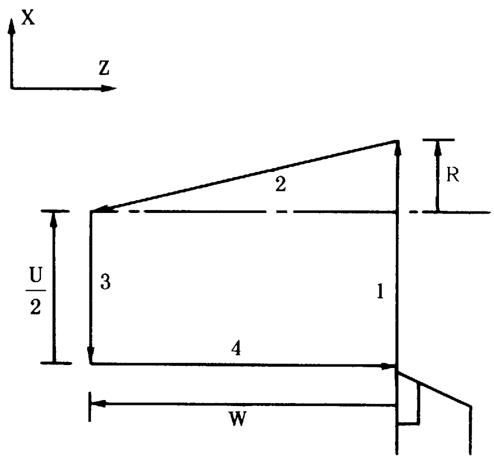
(2) U < 0, W < 0, R > 0



(3) U > 0, W < 0, R < 0



(4) U > 0, W < 0, R > 0



Program error "P191 Taper length error" results with shapes (2) and (3) unless the following condition is satisfied.

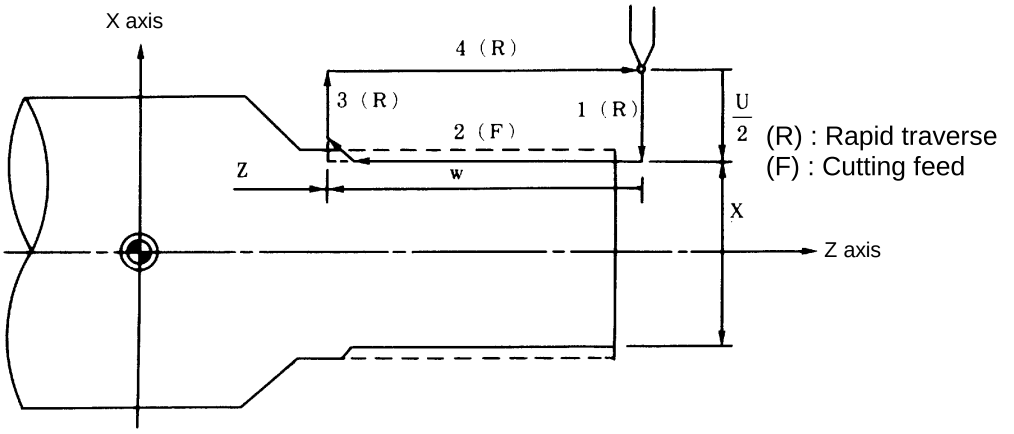
| U/2 | ≥ | r |

13.1.2 Thread cutting cycle; G78

Straight cutting

This function enables straight thread cutting using the following command.

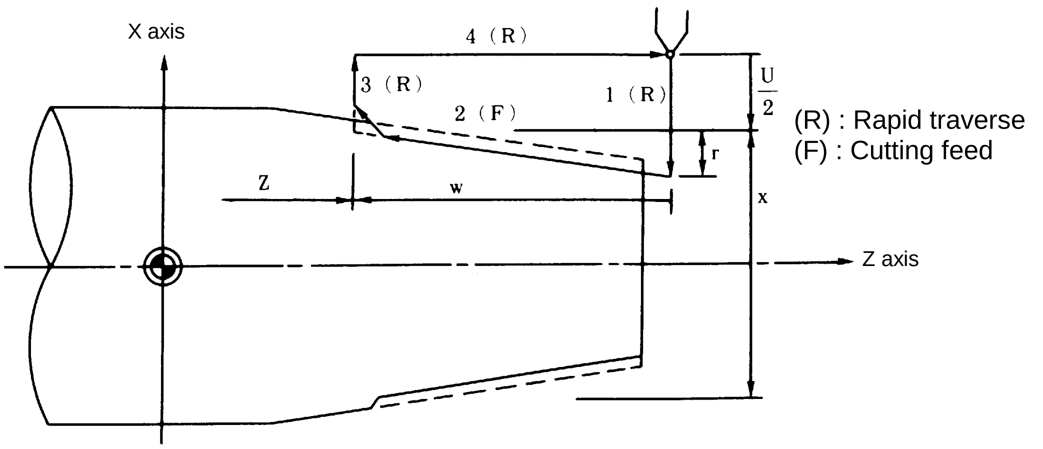
G78 X/U_ Z/W_ F/E_;



Taper cutting

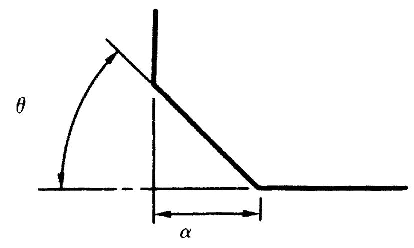
This function enables taper thread cutting using the following command.

G78 X/U_ Z/W_ R_ F/E_;



R: Taper depth (radius designation, incremental value, sign required)

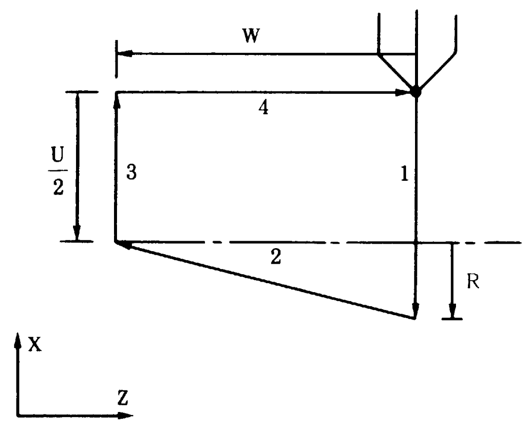
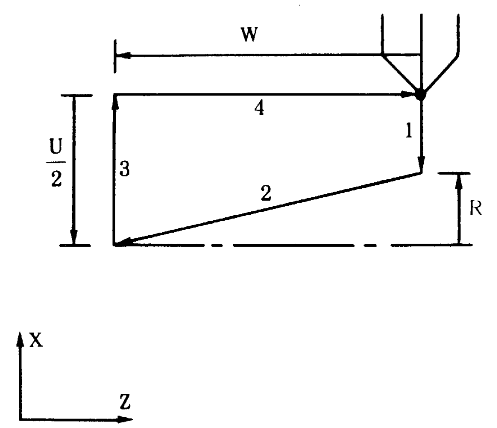
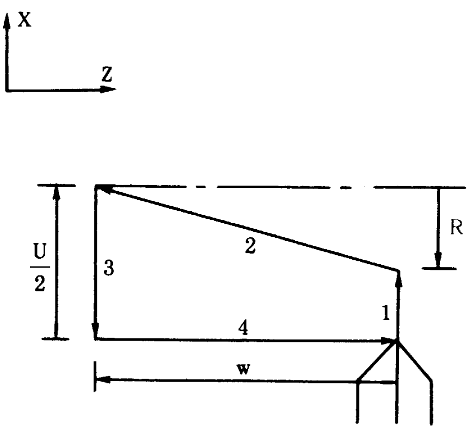
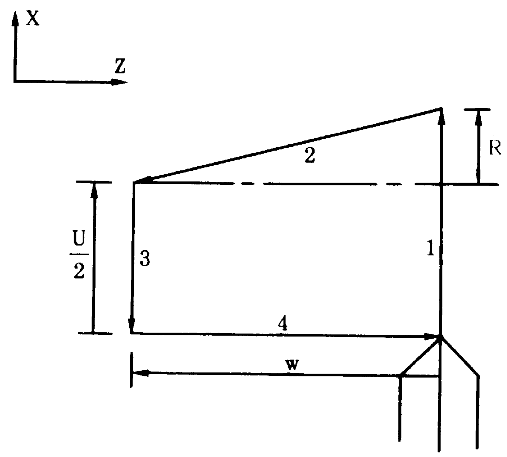
Details for chamfering

	α : Chamfering amount of thread If the thread lead is assumed to be L, then the parameter can be set in 0.1L units from 0 to 12.7L. θ : Chamfering angle of thread The parameter can be set in 1° units from 0 to 89°

With a single block, the tool stops at the end points of operations 1, 3 and 4.

When the feed hold function is applied during a thread cutting cycle, automatic operation will stop if no thread is then being cut. If thread cutting is proceeding, operation stops at the next movement completion position (completion of operation 3) of the thread cutting.
The dry run valid/invalid status does not change during thread cutting.

Depending on the U, W and R signs, the following shapes are created.

(1) U < 0, W < 0, R < 0	**(2) U < 0, W < 0, R > 0**
	
(3) U > 0, W < 0, R < 0	**(4) U > 0, W < 0, R > 0**
	

Program error "P191 Taper length error" results with shape (2) and (3) unless the following condition is satisfied.

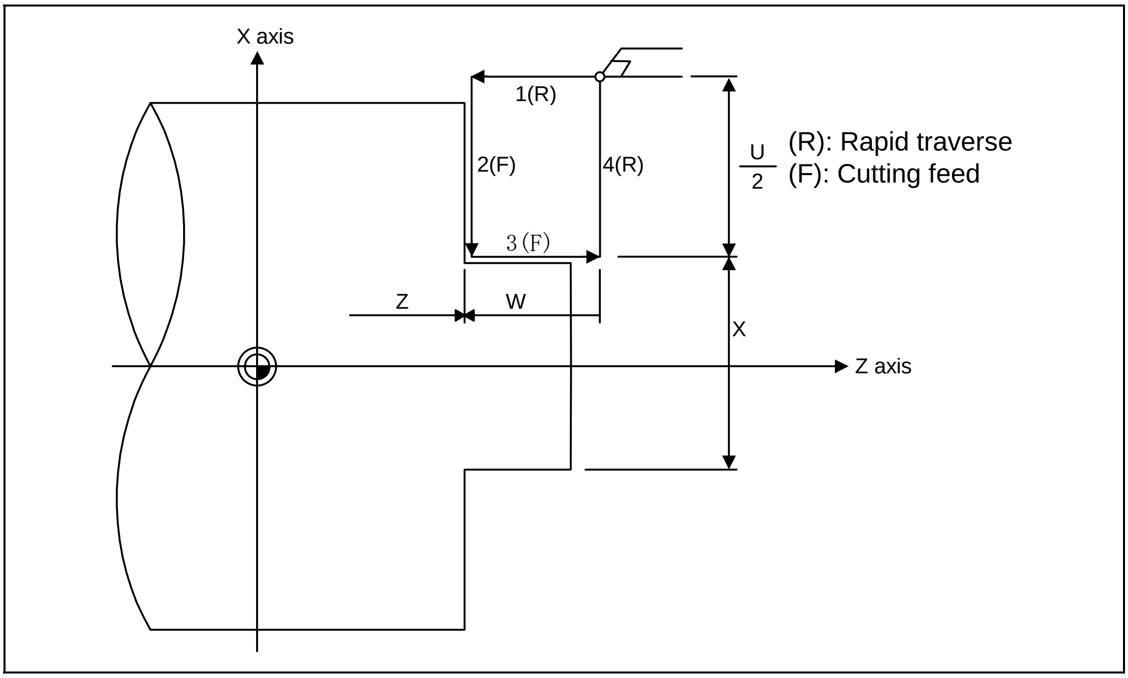
$| U/2 | \geq | r |$

13.1.3 Face cutting cycle; G79

Straight cutting

This function enables continuous straight cutting in the face direction using the following command.

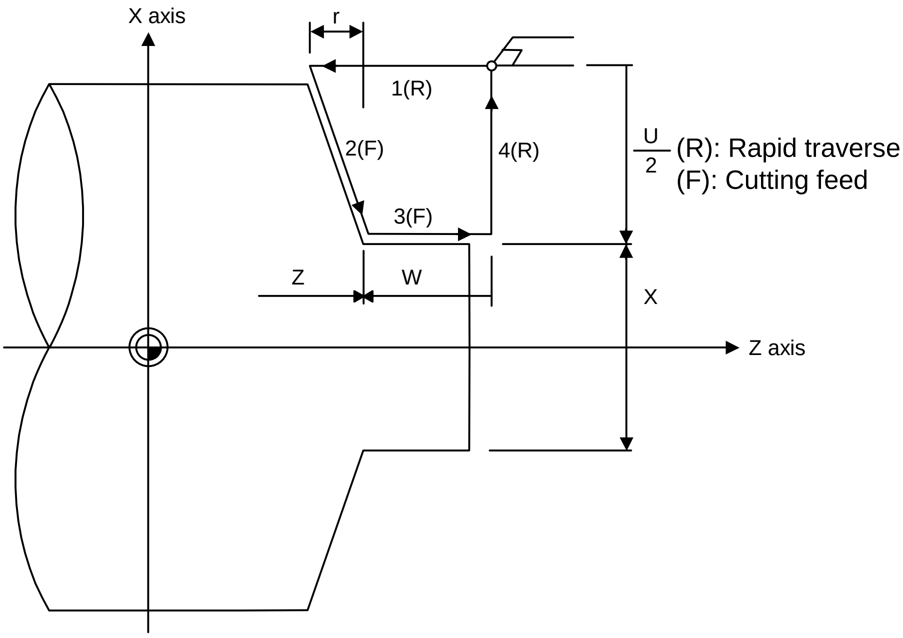
G79 X/U_ Z/W_ F_;



Taper cutting

This function enables continuous taper thread cutting using the following command.

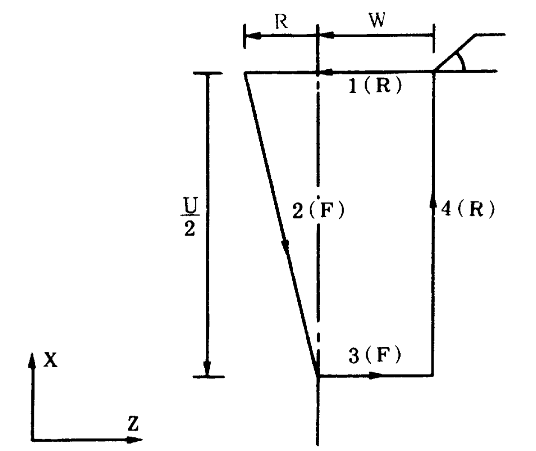
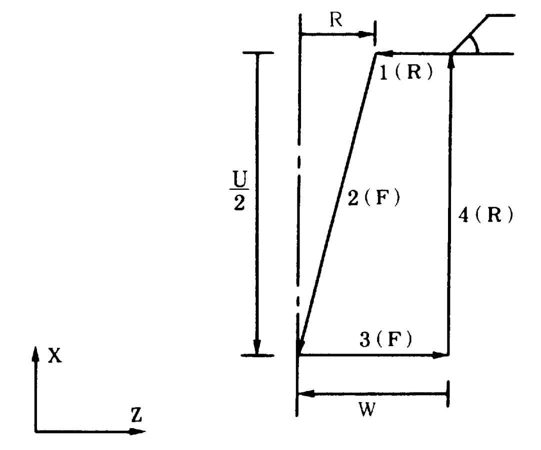
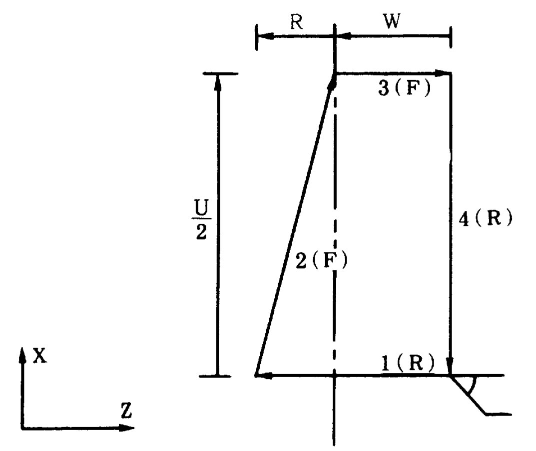
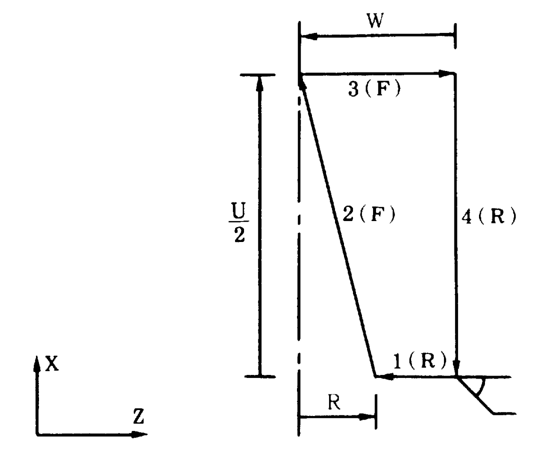
G78 X/U_ Z/W_ R_ F_;



R: Taper depth (radius designation, incremental value, sign required)

With a single block, the tool stops at the end points of operations 1, 2, 3 and 4.

Depending on the U, W and R signs, the following shapes are created.

<table>
<tr><th>(1) U < 0, W < 0, R < 0</th><th>(2) U < 0, W < 0, R > 0</th></tr>
<tr><td>
</td><td>
</td></tr>
<tr><th>(3) U > 0, W < 0, R < 0</th><th>(4) U > 0, W < 0, R > 0</th></tr>
<tr><td>
</td><td>
</td></tr>
</table>

Program error "P191 Taper length error" results with shapes (2) and (3) unless the following condition is satisfied.

| w | ≥ | r |

13.2 Compound Fixed Cycles

Function and purpose

These functions enable prepared fixed cycle to be executed by commanding the corresponding program in a block.
The types of fixed cycles are listed below.

G code	Function	
G70	Finishing cycle	Compound fixed cycles I
G71	Longitudinal rough cutting cycle (finished shape chamfering)	
G72	Face rough cutting cycle (finished shape chamfering)	
G73	Formed material rough cutting cycle	
G74	Face cut-off cycle	Compound fixed cycles II
G75	Longitudinal cut-off cycle	
G76	Compound thread cutting cycle	

(1) If, with any of the above functions for the compound canned cycle I (G70 to G73), the finished shape program has not been entered in the memory, the function cannot be used.

Command format

G70 A_ P_ Q_ ;	
G71 U_ R_ ; **G71 A_ P_ Q_ U_ W_ F_ S_ T_ ;**	
G72 W_ R_ ; **G72 A_ P_ Q_ U_ W_ F_ S_ T_ ;**	
G73 U_ W_ R_ ; **G73 A_ P_ Q_ U_ W_ F_ S_ T_ ;**	
G74 R_ ; **G74 X(U)_ Z(W)_ P_ Q_ R_ F_ ; (Same for G75)**	
G76 P_ R_ ; **G76 X(U)_ Z(W)_ P_ Q_ R_ F_ ;**	

Refer to the explanation of each G code (13.2.1 and following) for details on each address.

Detailed description

(1) The A, P and Q commands of the compound fixed cycles I are described below.

(a) When the A command is not present, P and Q in the program now being executed are called.
When the A command is present and the P command is not present, the head block of the program designated by the A command is treated as the P command.

(b) When the Q command is not present, operation continues until the M99 command is located. When both the Q and M99 commands are not present, operation continues until the final block in the finished shape program.

(2) Two pre-read blocks are required to create a finished shape program.

13.2.1 Longitudinal rough cutting cycle I; G71

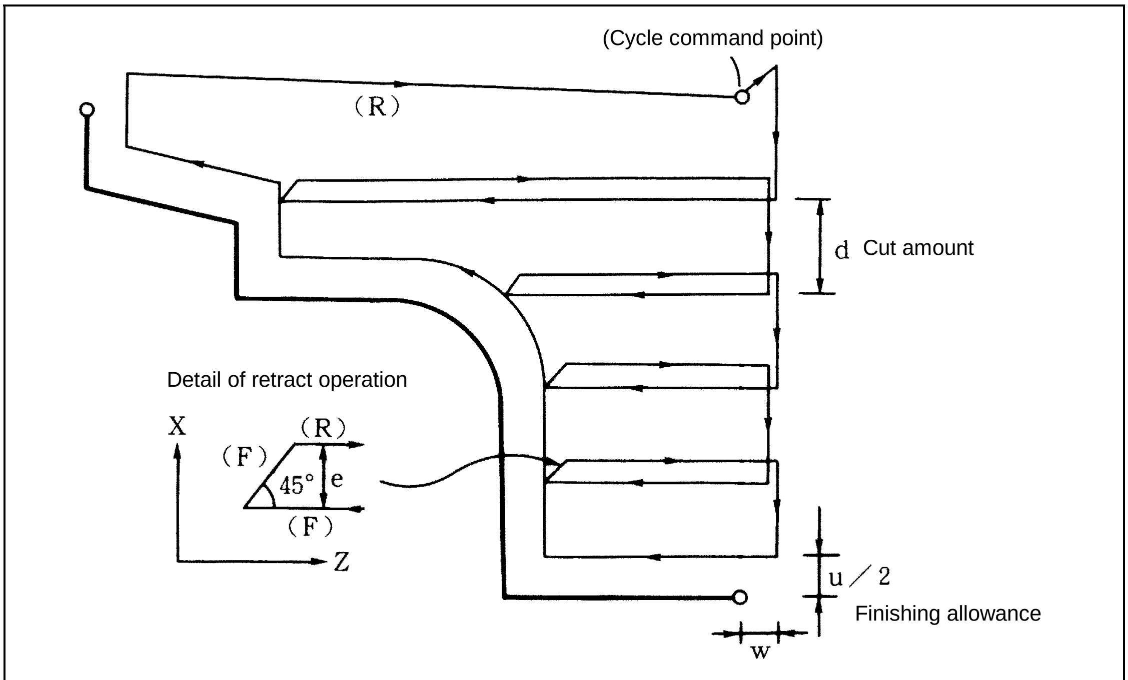
Function and purpose

This function calls the finished shape program and, while automatically calculating the tool path, performs rough cutting in the longitudinal direction.

Command format

G71 Ud Re ;
G71 Aa Pp Qq Uu Ww Ff Ss Tt ;

Ud	Cut amount "d" (cut amount without P, Q commands) (modal) ·········· Variable parameter valid
Re	Retract amount "e" (modal) ········ Variable parameter valid
Aa	Finished shape program number (program being executed when omitted)
Pp	Finished shape start sequence number (program head when omitted)
Qq	Finished shape end sequence number (up to end of program when omitted) Up to M99 command when M99 comes first even if Q command is present
Uu	X-axis direction finishing allowance (diameter or radius designation)
Ww	Z-axis direction finishing allowance
Ff	Cutting feedrate
Ss	Spindle speed
Tt	Tool command

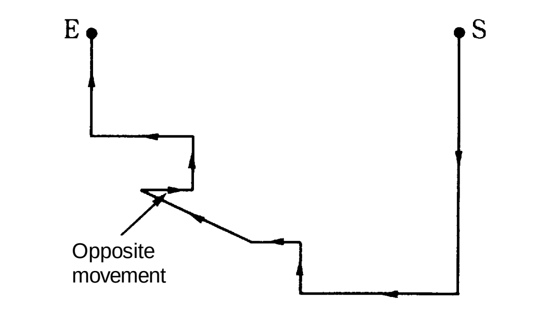
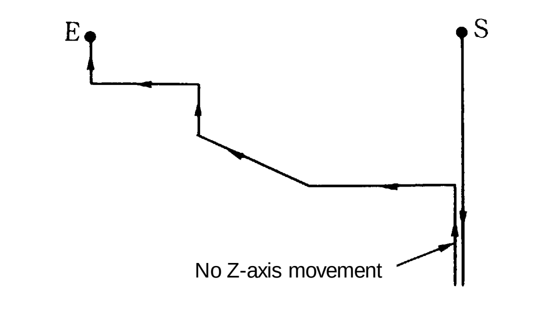


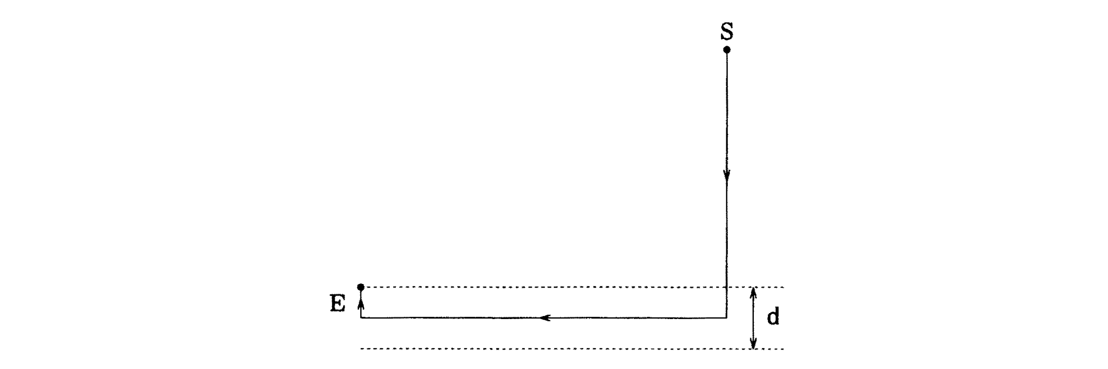
(Note 1) A "variable parameter" is a parameter which can use a parameter setting value without the issue of a program command or a parameter which is that parameter value rewritten by the program command.

(Note 2) The U command applied to the finishing allowance when it is in the same block as the A, P and Q commands.

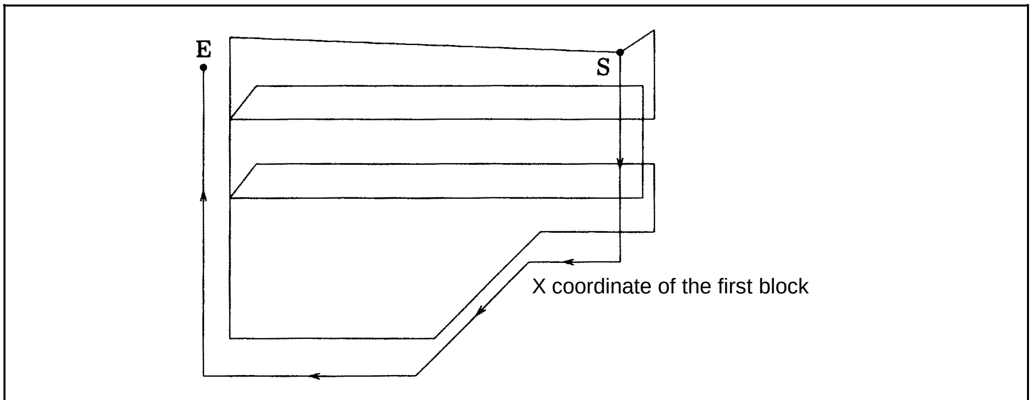
Cutting shape

It must be ensured that the finished shape changes monotonically (increase or decrease only) in both the X- and Z-axis directions.
Program error "P203 D cmnd figure error (MRC)" occurs with the following shapes.

(1) When there is movement away from the Z axis	**(2) When there is no Z axis movement in the second block**
	

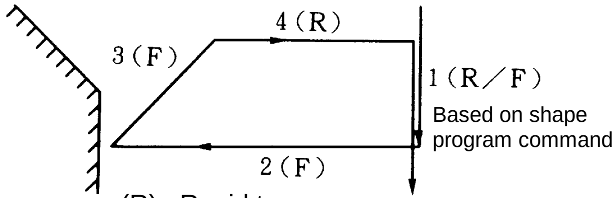
(3) When the depth of the finished shape X axis is smaller than cut amount "d"


(Note) For the following shape, no program error occurs but the program performs rough cutting down to the X coordinate of the first block and traces the specified shape for finishing.



Configuration of a cycle

A cycle is composed as shown below.

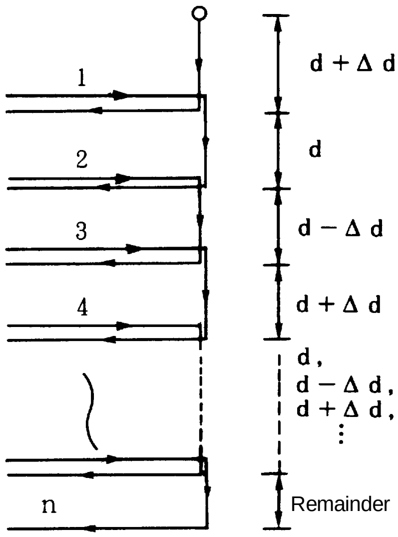


(R) : Rapid traverse
(F) : Cutting feed

Cut amount

The cut amount is designated by "d". However, it is possible to change the cut amount with each cutting pass by setting the cut change amount (Δd) using a parameter.
Program error "P204 E cmnd fixed cycle error (MRC)" results when "d" is less than "Δd".

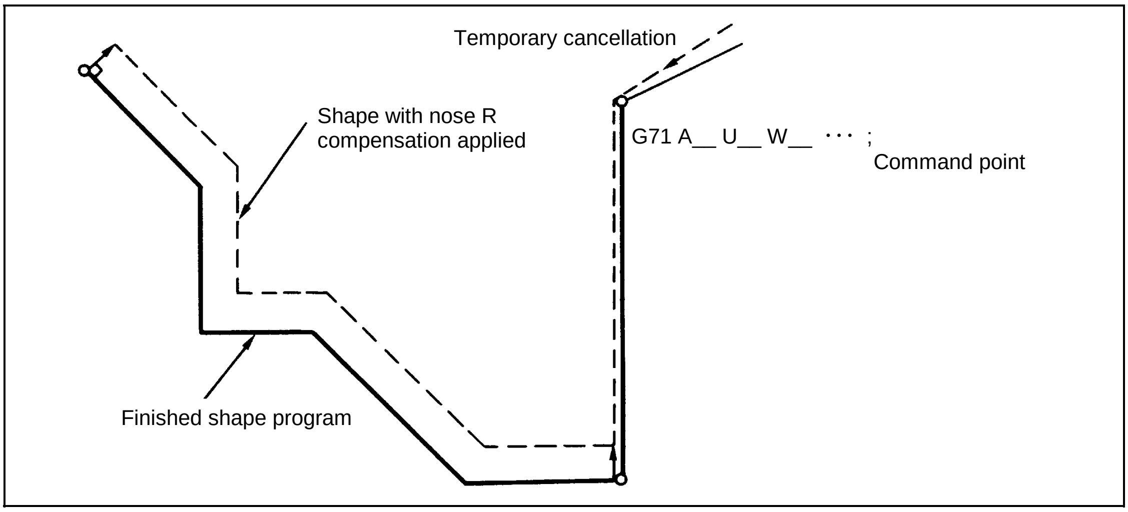
1st cutting pass	d + Δd
2nd cutting pass	d
3rd cutting pass	d – Δd
4th cutting pass	d + Δd
≀	[d, d – Δd, d + Δd are repeated]
Final cutting pass	Remainder



Nose R compensation

When this cycle is commanded with the nose R compensation mode still in force, nose R compensation is applied to the finished shape program covered by this cycle and the cycle is executed for this shape.
However, when this cycle is commanded with the nose R compensation mode still in force, the compensation is applied to the shape as follows.

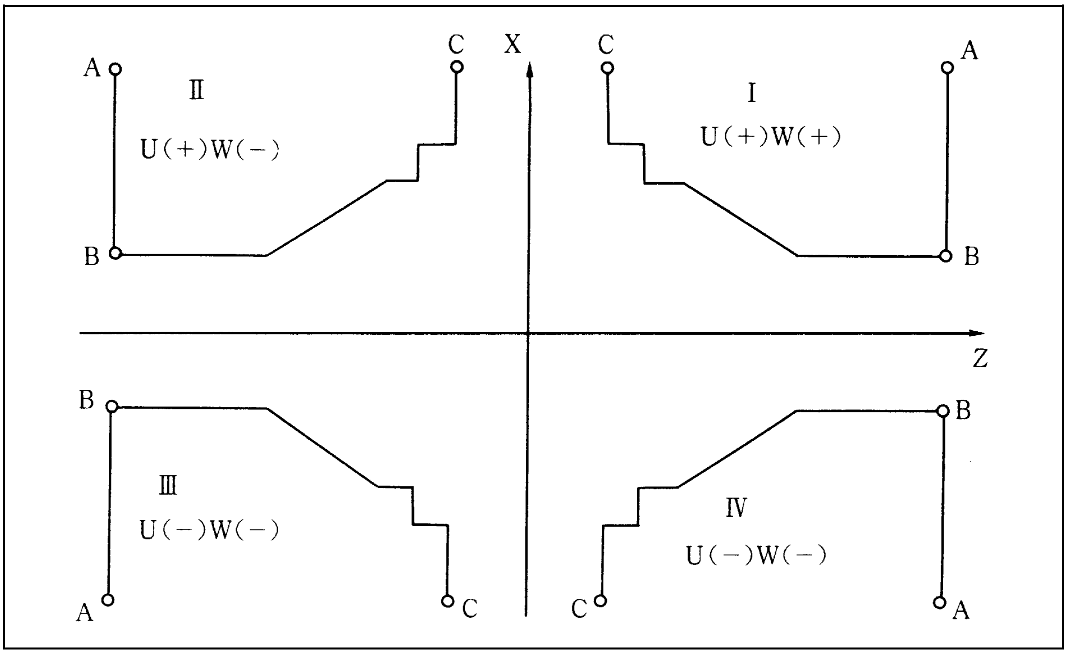
- The mode is temporarily canceled immediately before the cycle.
- It starts up with the finished shape program.
- The end block of this program serves as the pre-read prohibit block.



Others

(1) After the cutting, the remainder is made the cut amount. However, if this amount is less than the value set by parameter, finishing rough cutting is executed and the workpiece is not cut.

(2) Finishing allowance direction
The finishing allowance direction is determined by the shape as follows. A→B→C apply for the finishing program.



13.2.2 Face rough cutting cycle I; G72

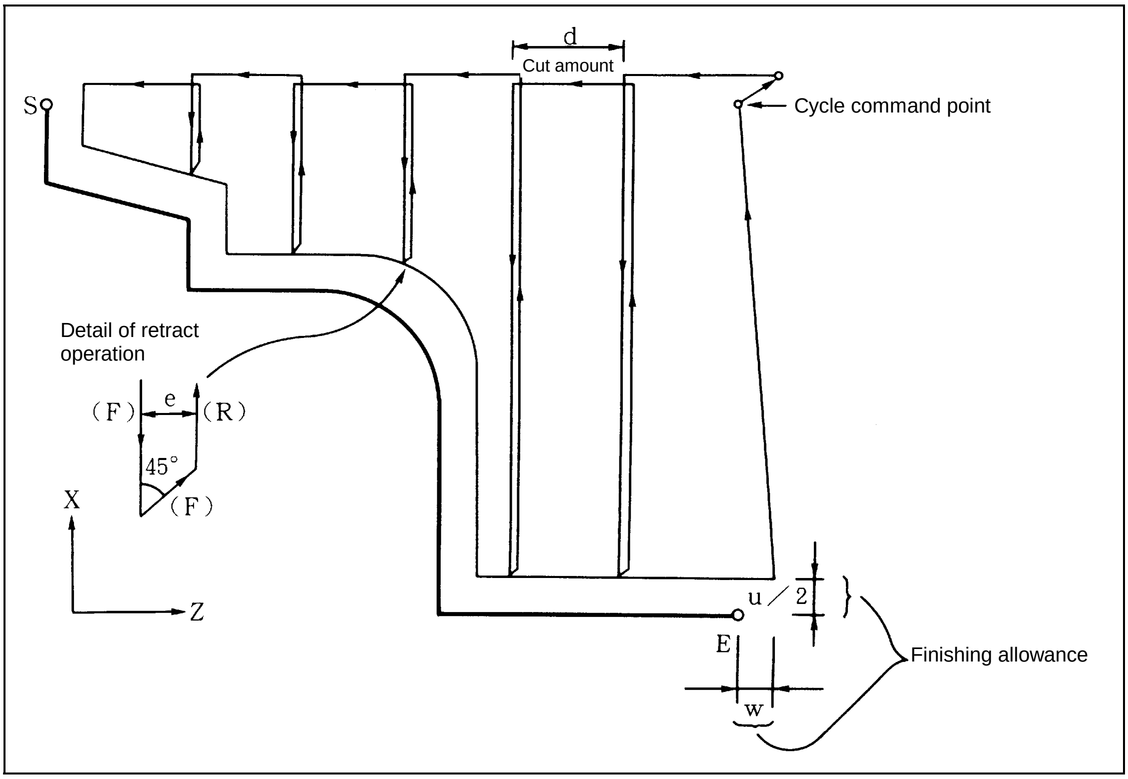
Function and purpose

This function calls the finished shape program and, while automatically calculating the path, performs rough cutting in the face direction.

Command format

G72 Wd Re ;
G72 Aa Pp Qq Uu Ww Ff Ss Tt ;

Wd	Cut amount "d" (cut amount without P, Q commands) (modal) ·········· Variable parameter valid
Re	Retract amount "e" (modal) ······· Variable parameter valid
Aa	Finished shape program number (program being executed when omitted)
Pp	Finished shape start sequence number (program head when omitted)
Qq	Finished shape end sequence number (up to end of program when omitted) Up to M99 command when M99 comes first even if Q command is present
Uu	X-axis direction finishing allowance (diameter or radius designation)
Ww	Z-axis direction finishing allowance
Ff	Cutting feedrate
Ss	Spindle speed
Tt	Tool command



The F, S and T commands in the finished shape program are ignored and the value in the rough cutting cycle command or previous value is valid.

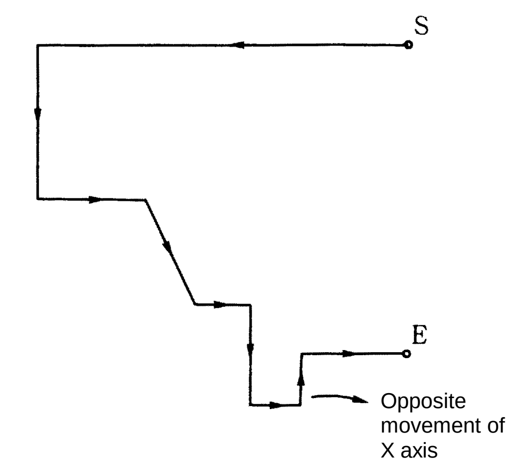
(Note 1) A "variable parameter" is a parameter which can use a parameter setting value without the issue of a program command or a parameter which is that parameter value rewritten by the program command.

(Note 2) The W command applies to the finishing allowance when it is in the same block as the A, P and Q commands.

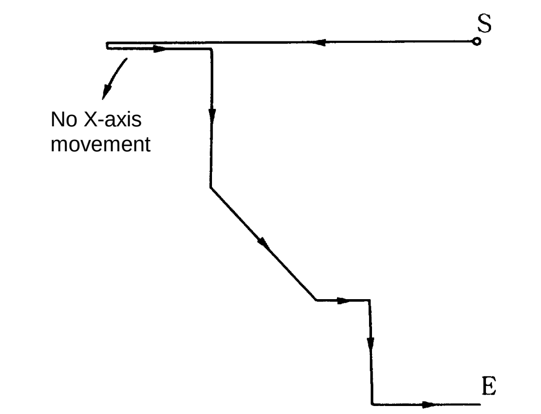
Cutting shape

It must be ensured that the finished shape changes monotonically (increase or decrease only) in both the X- and Z-axis directions.
Program error "P203 D cmnd figure error (MRC)" occurs with the following shapes.

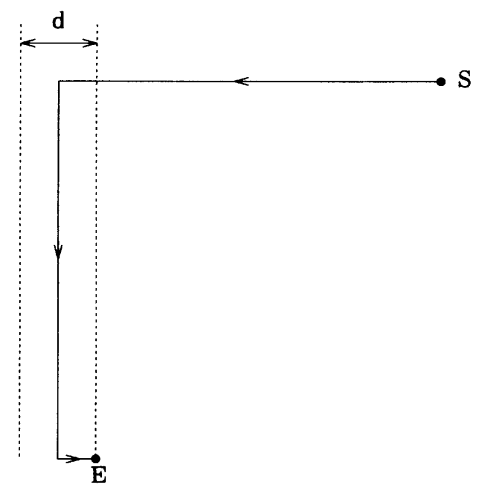
(1) When there is movement away from the X axis



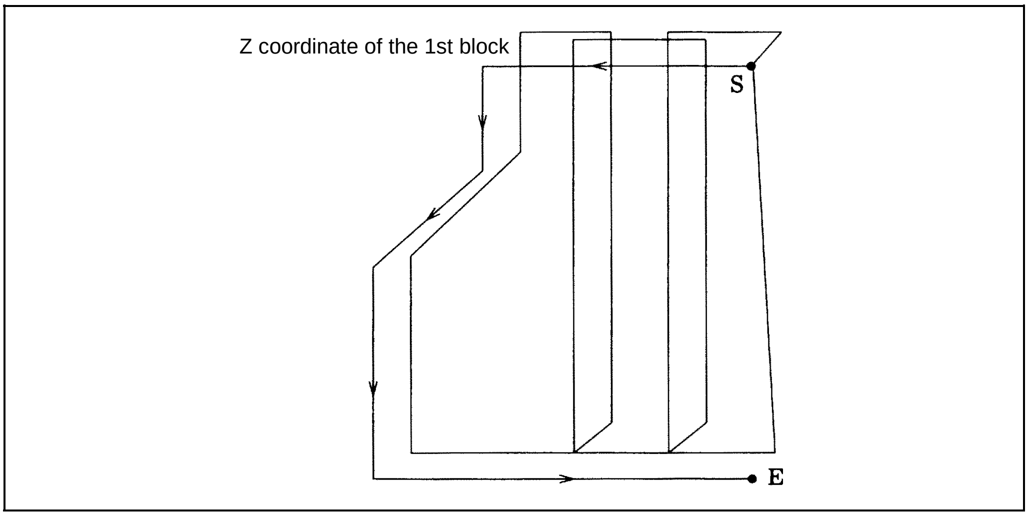
(2) When there is no X axis movement in the second block



(3) When the depth of the finished shape X axis is smaller than cut amount "d"

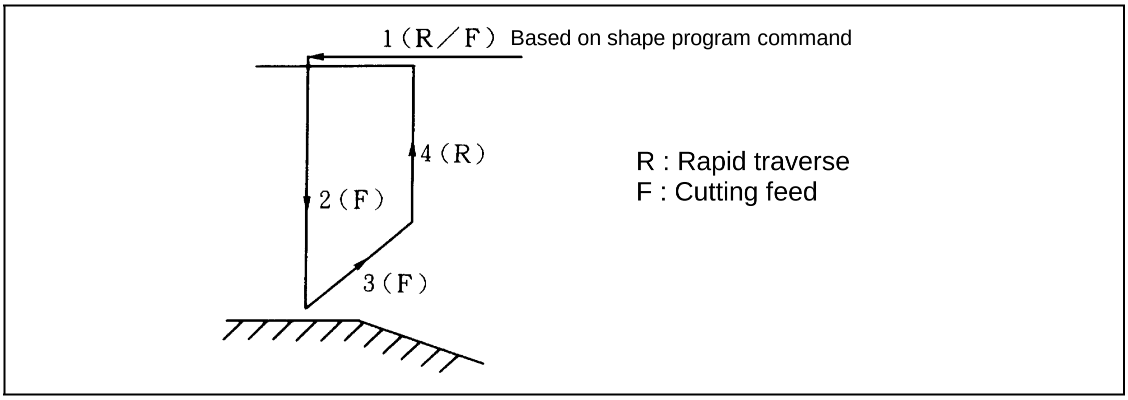


(Note) For the following shape, no program error occurs but the program performs rough cutting down to the Z coordinate of the 1st block and traces the specified shape for finishing.



Configuration of a cycle

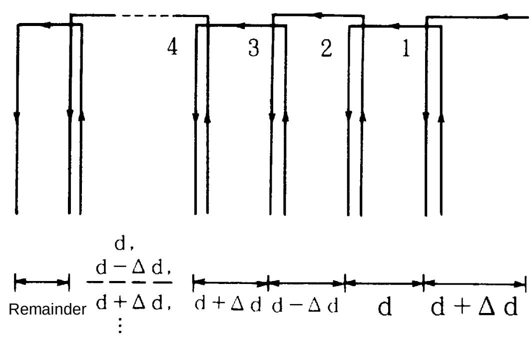
A cycle is composed as below.



Cut amount

The cut amount is designated by "d". However, it is possible to change the cut amount with each cutting pass by setting the cut change amount (Δd) using a parameter.
When d < Δd, a program error "P204 E cmnd fixed cycle error (MRC)" will occur.

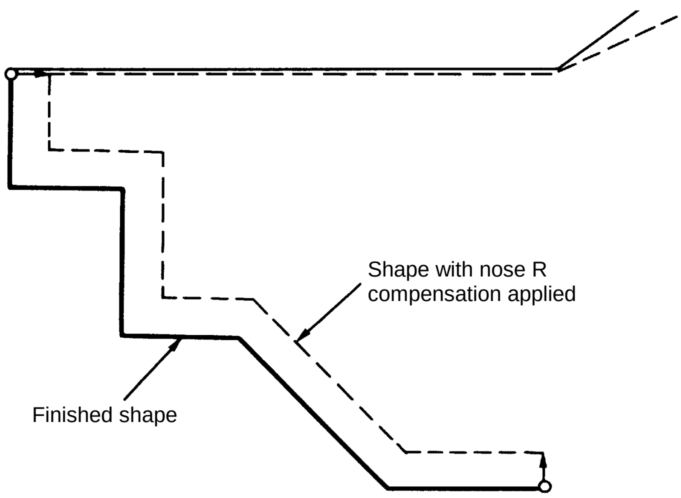
1st cutting pass	d + Δd
2nd cutting pass	d
3rd cutting pass	d – Δd
4th cutting pass	d + Δd
≀	d, d – Δd, d + Δd are repeated
Final cutting pass	Remainder



Nose R compensation

When this cycle is commanded with the nose R compensation mode still in force, nose R compensation is applied to the finished shape program covered by this cycle and the cycle is executed for this shape.
However, when this cycle is commanded with the nose R compensation mode still in force, the compensation is applied to the shape as follows.

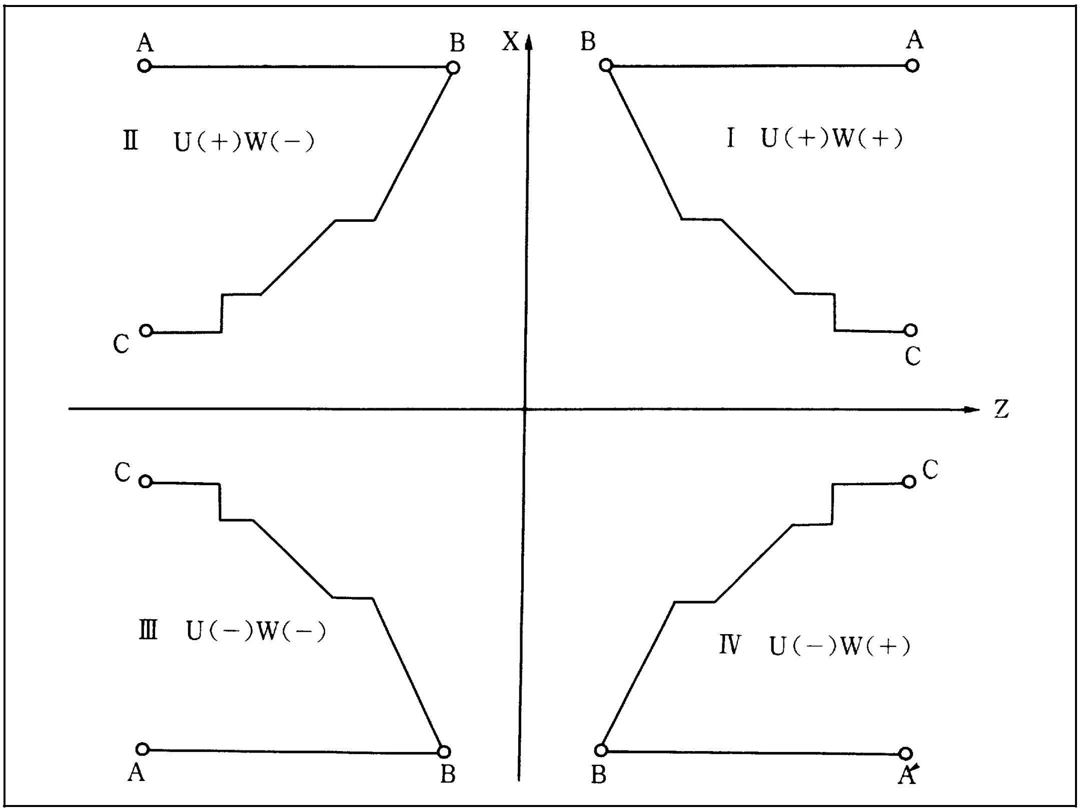
- The mode is temporarily canceled immediately before the cycle.
- It starts up with the finished shape program.
- The end block of this program serves as the pre-read prohibit block.



Others

(1) After the cutting, the remainder is made the cut amount. However, if this amount is less than the value set by parameter, this cycle is not executed.

(2) Finishing allowance direction
The finishing allowance direction is determined by the shape as follows.



13.2.3 Formed material rough cutting cycle; G73

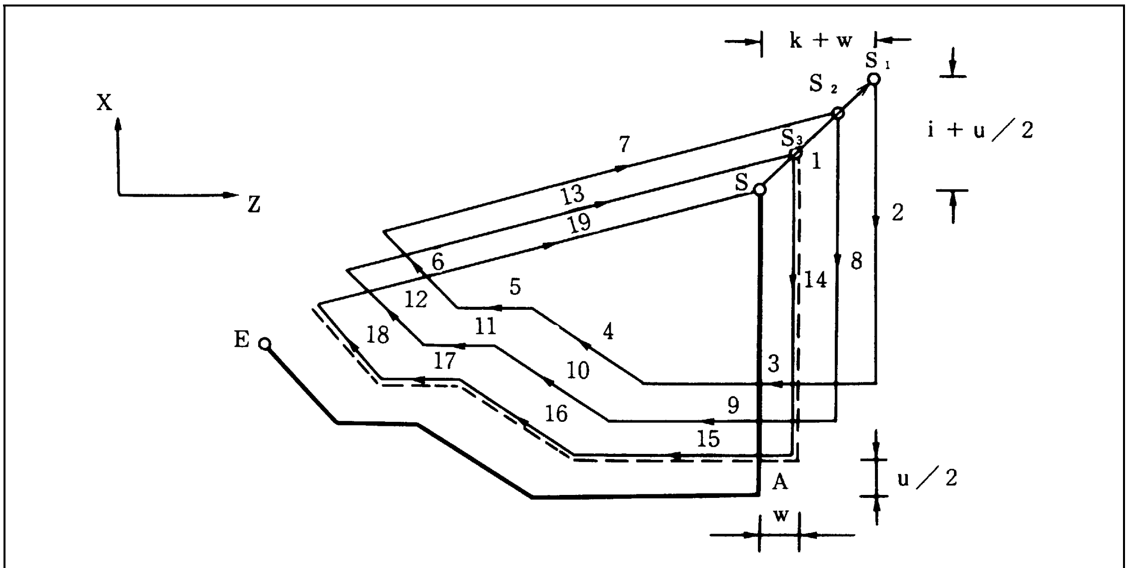
Function and purpose

This calls the finished shape program, automatically calculates the path and performs rough machining while cutting the workpiece into the finished shape.

Command format

G73 Ui Wk Rd ;
G73 Aa Pp Qq Uu Ww Ff Ss Tt ;

Ui	X-axis direction cutting allowance	i	• Cutting allowances when P, Q commands are not present. • Modal data; variable parameters are valid. • Sign is ignored. • Radius designation applies to the cutting allowance.
Wk	Z-axis direction cutting allowance	k	
Rd	Number of divisions	d	
Aa	Finished shape program number (program being executed when omitted)		
Pp	Finished shape start sequence number (program head when omitted)		
Qq	Finished shape end sequence number (up to end of program or M99 when omitted)		
Uu	X-axis direction finishing allowance	u	• Cutting allowance when P, Q commands are present. • Sign is ignored. • Diameter/radius designation changes in accordance with the parameters. • The shift direction is determined by the shape. For details, reference should be made to the "finishing allowance direction" for G71.
Ww	Z-axis direction finishing allowance	w	
Ff	Cutting feedrate (F function)		The F, S and T commands in the finished shape program are ignored and the value in the rough cutting cycle command or previous value is valid.
Ss	Spindle speed (S function)		
Tt	Tool selection (T function)		



(Note 1) A "variable parameter" is a parameter which can use a parameter setting value without the issue of a program command or a parameter which is that parameter value rewritten by the program command.

(Note 2) With a single block, operation stops at the end point of each block.

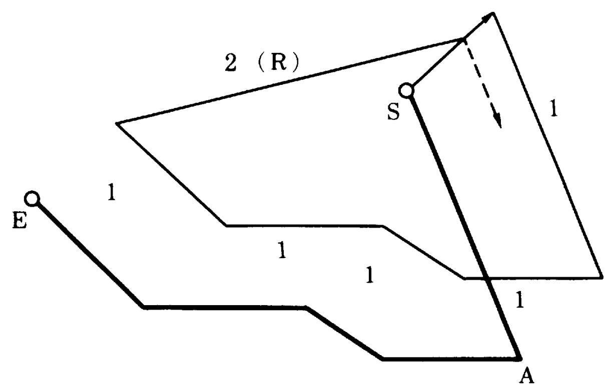
(Note 3) When the finished shape is specified in a separate program or ends with M99, operation proceeds to the block next to G73 after execution of G73.
When the finished shape is specified with P or Q in the same program, operation proceeds to the block next to the one specified by Q after execution of G73.

Finished shape

In the program, S → A → E in the previous program are commanded.
The section between A and E must be a shape with monotonous changes in both the X-axis and Z-axis directions.

1 cycle configuration

A cycle is configured as shown below.



1 : Based on shape program with shape profiling
2 : Rapid traverse on return

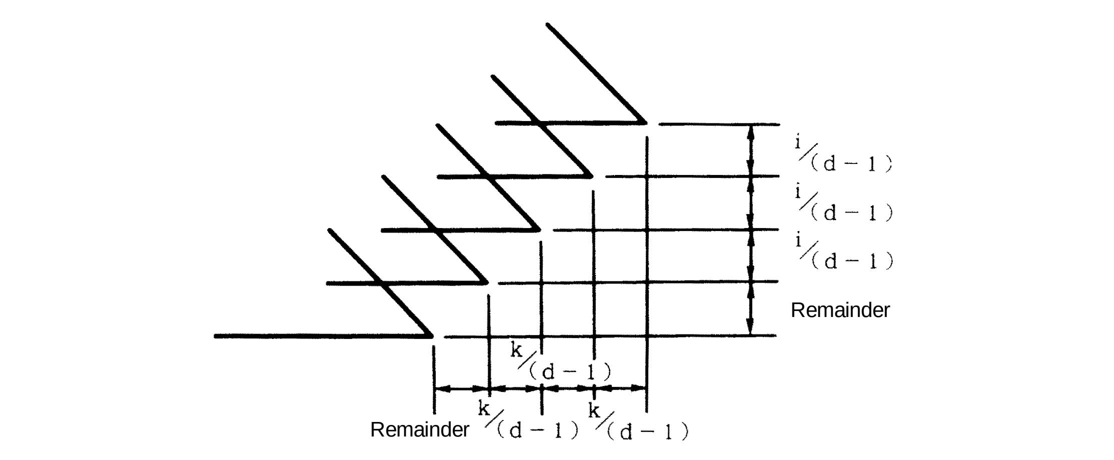
Cut amount

The cut amount is the value produced by dividing the cutting allowances (i, k) by the number of divisions (d–1).

X-axis direction i/(d–1)

Z-axis direction k/(d–1)

When divisions cannot be made, chamfering is performed and adjustment is made at the final pass.



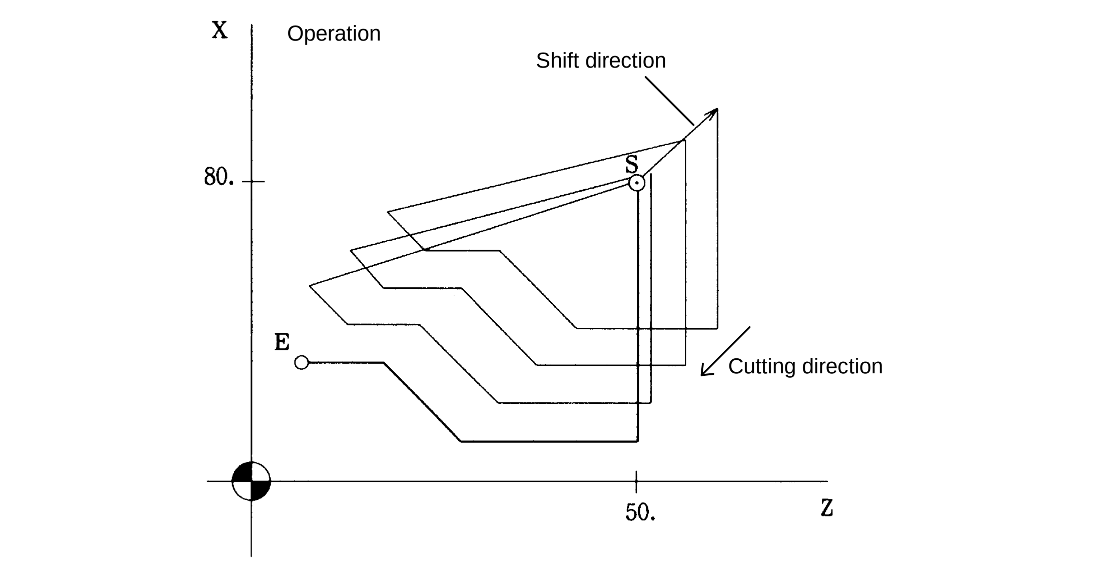
Nose R compensation

When this cycle is commanded with the nose R compensation mode still in force, nose R compensation is applied to the finished shape program covered by this cycle and the cycle is executed for this shape.

However, when this cycle is commanded with the nose R compensation mode still in force, the compensation is temporarily canceled immediately before this cycle and started at the head block of the finished shape program.

Example of program

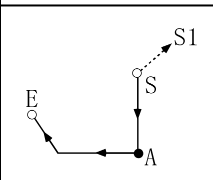
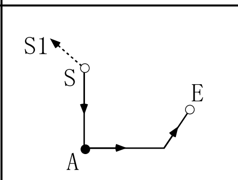
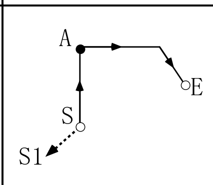
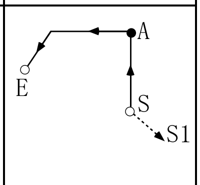
G28 XZ;
G0 X80. Z50.;
G73 U10. W10. R3;
G73 P10. Q20. U2. W2. F100.;
N10 G1 X20. Z50. F150.;
G1 X20. Z30.;
G1 X40. Z20.;
G1 X40. Z10.;
N20 G1 X50. Z5.;
M02;



Others

(1) Cutting direction

The cutting direction is determined according to the following table by the finished program shape.

	1	2	3	4
X axis	**A < E**	**A < E**	**A >= E**	**A >= E**
General Z axis	**–**	**+**	**+**	**–**
Drawing				
X axis	–	–	+	+
Z axis	–	+	+	–

13.2.4 Finishing cycle; G70

Function and purpose

After rough cutting have been carried out by the G71 to G73 commands, finishing cutting can be performed by the following command.

Command format

G70 A_ P_ Q_ ;

A	Finished shape program number (program being executed when omitted)
P	Finished shape start sequence number (program head when omitted)
Q	Finished shape end sequence number (up to end of program when omitted)

(1) The F, S and T commands in the finished shape program are valid during the finishing cycle.

(2) When the G70 cycle is completed, the tool returns to the start point by rapid traverse and the next block is read.

(Example 1) When a sequence number is designated

```
  ≀
N100 G70 P200 Q300;        N200 ······ ;
N110                         ≀
                           N300 ······ ;
N120
  ≀
N200 ┐
  ≀  ├ Finished shape program
N300 ┘
N310
  ≀
```

(Example 2) When a program number is designated

```
  ≀
N100 G70 A100;              O100
N110 ······ ;               G01 X100 Z50 F0.5;
N120 ······ ;                 ≀
  ≀                         M99;
```

If the N100 cycle is executed in either Example 1 or Example 2, the N110 block is executed next.

13.2.5 Face cut-off cycle; G74

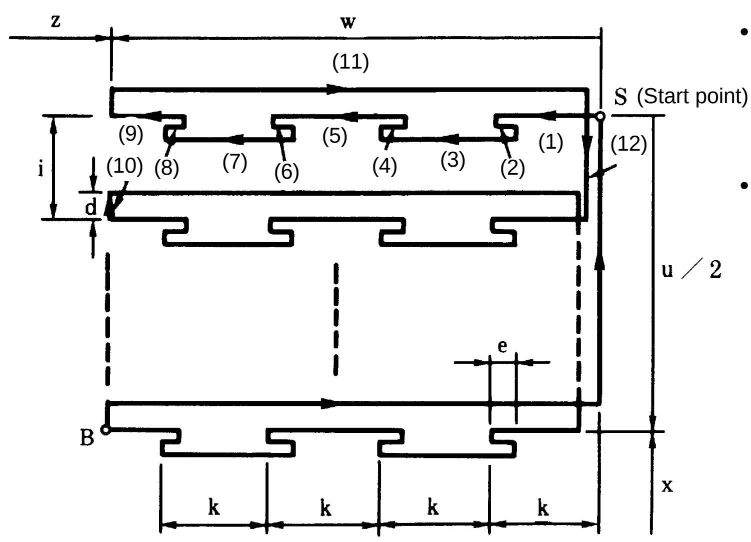
Function and purpose

The G74 fixed cycle automatically enters a groove in the face direction of the workpiece by commanding the coordinates of the groove end point, cut amount, cutter shift amount and cutter escape at the bottom of the cut.

Command format

G74 Re ;
C74 X/(U)x Z/(W)z Pi Qk Rd Ff ;

Re	Return amount (no X/U, P commands) (modal).....Variable parameter valid
X/Ux	B point X coordinate (absolute/incremental value)
Z/Wz	B point Z coordinate (absolute/incremental value)
Pi	Tool shift amount (radius designation, incremental value, sign not required)
Qk	Cut amount (radius designation, incremental value, sign not required)
Rd	Escape at the bottom of the cut
Ff	Feedrate



- Operation (9) as well as operation (12) immediately before the final cycle are executed with the remainder amount.
- Operations (2), (4), (6), (8), (10), (11) and (12) are executed at the rapid traverse rate.

Single block stop

Operation stops at each block from (1) to (12).

Others

(1) "e" can also be set by parameter. (It is rewritten by the program command.)

(2) When X/U and P are omitted or when the values of "x" and "i" are "0", operation will apply to the Z axis only.

(3) A case where the X/U or Z/W command is not present is treated as the assignment of a parameter setting command (G74 Re). If Re as well as the X/U or Z/W command is not present, no processing results.

(4) When the value of cut amount "k" is higher than hole depth "w", the command is executed with "k" equal to "w".

(5) A program error "P204 E cmnd fixed cycle error (MRC)" results in the following cases.

  (a) When "i" is 0 or P has not been commanded even though X/U has been commanded

  (b) When tool shift amount "i" is greater than the "x" movement amount

  (c) When the escape amount "d" is greater than the shift amount "i"

  (d) When the return amount "e" is greater than the cut amount "k"

13.2.6 Longitudinal cut-off cycle; G75

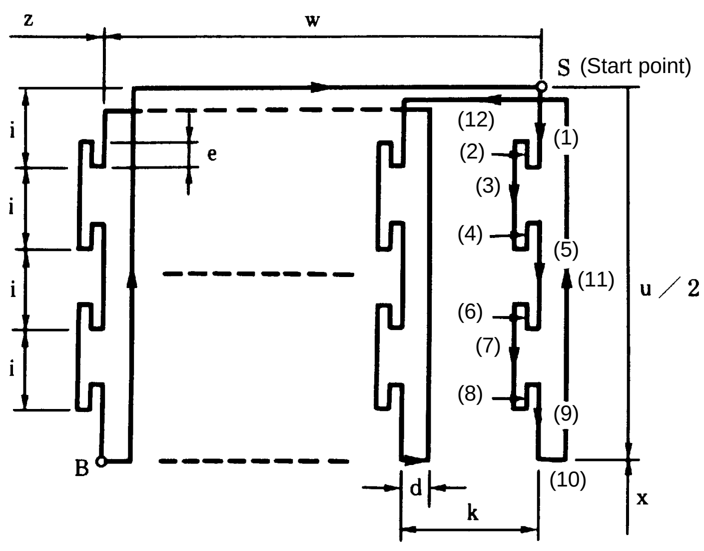
Function and purpose

The G75 fixed cycle automatically enters a groove in the longitudinal direction of the workpiece by commanding the coordinates of the groove end point, cut amount, cutter shift amount and cutter escape at the bottom of the cut.

Command format

G75 Re ;
C75 X/(U)x Z/(W)z Pi Qk Rd Ff ;

Re	Return amount (no X/U, P commands) (modal).....Variable parameter valid
X/Ux	B point X coordinate (absolute/incremental value)
Z/Wz	B point Z coordinate (absolute/incremental value)
Pi	Cut amount (radius designation, incremental value, sign not required)
Qk	Tool shift amount (radius designation, incremental value, sign not required)
Rd	Escape at the bottom of the cut
Ff	Feedrate



- Operation (9) as well as operation (12) immediately before the final cycle are executed with the remainder amount.
- Operations (2), (4), (6), (8), (10), (11) and (12) are executed at the rapid traverse rate.

Single block stop

Operation stops at each block from (1) to (12).

Others

(1) "e" can also be set by parameter. (It is rewritten by the program command.)

(2) When Z/W and Q are omitted or when the values of "z" and "k" are "0", operation will apply to the X axis only.

(3) A case where both the X/U and Z/W commands are not present is treated as the assignment of a parameter setting command (G74 Re). If Re as well as the X/U and Z/W commands is not present, no processing results.

(4) When the value of cut amount "i" is higher than hole depth "u/2", the command is executed with "i" equal to "u/2".

(5) A program error "P204 E cmnd fixed cycle error (MRC)" results in the following cases.

   (a) When "k" is 0 or Q has not been commanded even though Z/W has been commanded

   (b) When tool shift amount "k" is greater than the "z" movement amount

   (c) When the escape amount "d" is greater than the shift amount "k"

   (d) When the return amount "e" is greater than the cut amount "i"

13.2.7 Compound thread cutting cycle; G76

Function and purpose

The G76 fixed cycle enables the workpiece to be cut at the desired angle by designating the thread cutting start point and end point, and it automatically cuts so that the cutting cross section (cutting torque) per cutting pass is constant.
Various longitudinal direction threads can be cut by bearing in mind the command value for the thread end point coordinate and taper height component.

Command format

G76 Pmra QΔd min Rd ;
G76 X/(U) Z/(W) Ri Pk QΔd Ff ;

m	Number of cutting passes for finishing: 00 to 99 (modal)
r	Chamfering amount: 00 to 99 (modal) The chamfering width based on thread lead "λ" is designated by a 2-digit integer without decimal point from 0.0 λ to 9.9 λ.
a	Tool nose angle (thread angle): 00 to 99 (modal) The angle from 0° to 99° is assigned in 1° units. "m", "r" and "a" are commanded in succession in address P. **(Example)** When "m" = 5, "r" = 1.5 and "a" = 0°, P is 051500 and the leading and trailing "0" cannot be omitted.
Δd min	Minimum cut amount (modal) This is the clamping value for guaranteeing the cut amount of a single cutting pass and it is valid only for rough cutting. Finishing is performed by the cutting allowance designated separately. When the calculated value for the cut amount of a single cutting pass is less than Δd min, the clamping is performed at Δd min.
d	Finishing allowance (modal)
X/U	The X coordinate of the end point for the thread is commanded by an absolute or incremental value.
Z/W	The Z coordinate of the end point for the thread is commanded by an absolute or incremental value.
i	Taper height component (radius value) for thread Straight thread when "i" is 0.
k	Thread height is commanded by a positive radial value.
Δd	The cut amount of the first cutting pass is commanded by a positive radius value.
λ	Thread lead

(Note 1) The two above G76 commands cannot be placed in the same block.
The data commanded by P, Q and R are automatically identified according to the presence or absence of the X/U and Z/W axis addresses.
Command the axis address before P, Q or R.

(Note 2) Parameter settings can be used for the above "m", "r", "a", "Δd min" and "d" modal data but these parameter settings are rewritten by the program commands.

(Note 3) The chamfering amount designation is valid even for thread cutting fixed cycles.

(Note 4) Program error "P204 E cmnd fixed cycle error (MRC)" results in the following cases.

(a) When "a" is outside the rating.

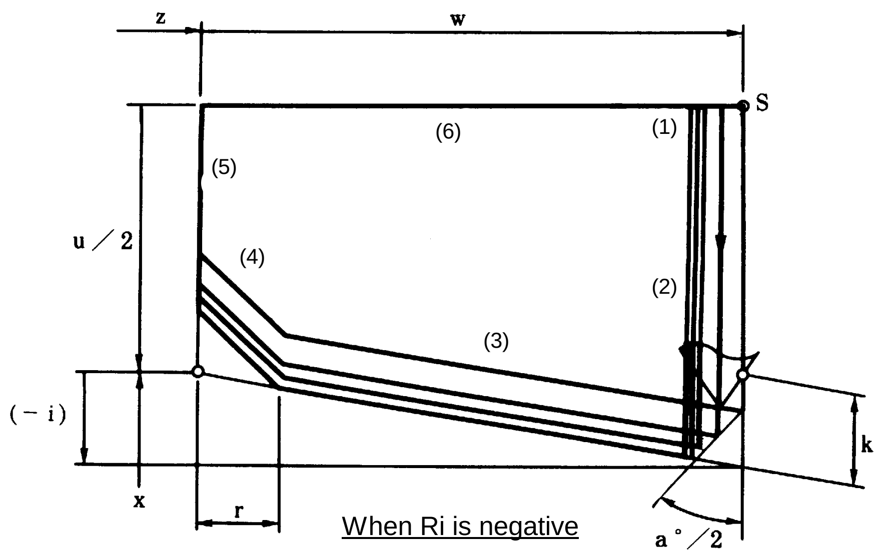
(b) When both the X and Z commands have not been issued or when the start and end point coordinates are the same for either the X or Z command.

(c) When the thread is greater than the movement of the X axis at the thread bottom.

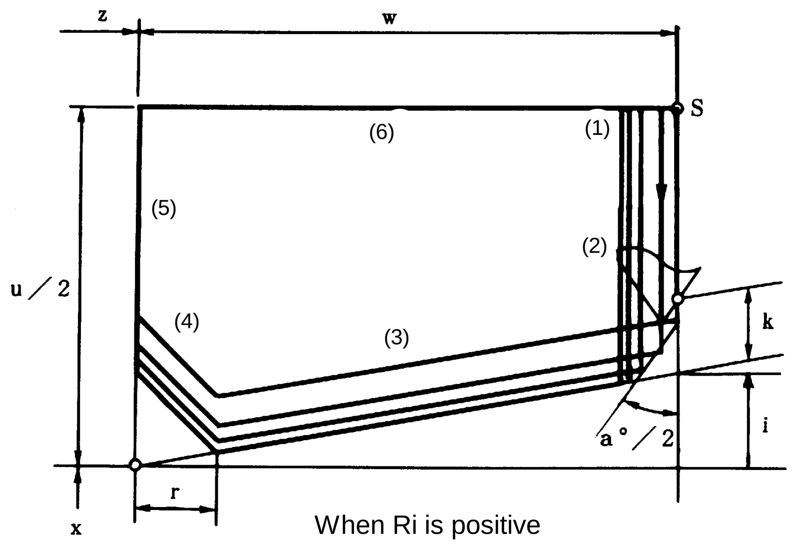
(Note 5) The precautions for the thread cutting command (G33) and thread cutting cycle (G78) should be observed.

1 cycle configuration

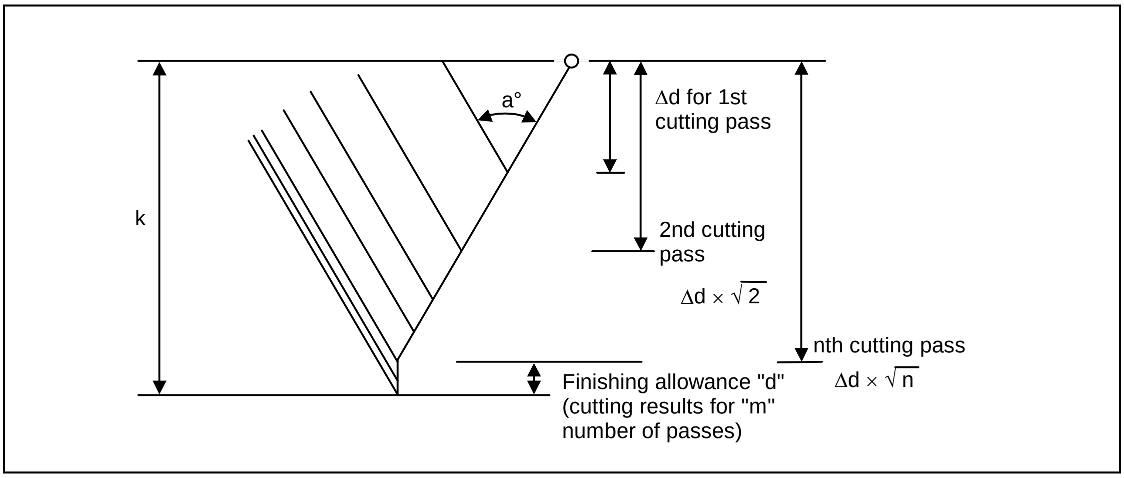
The tool moves at rapid traverse for operations (1), (2), (5) and (6) in the cycle and at the cutting feed based on the F designation for operations (3) and (4).



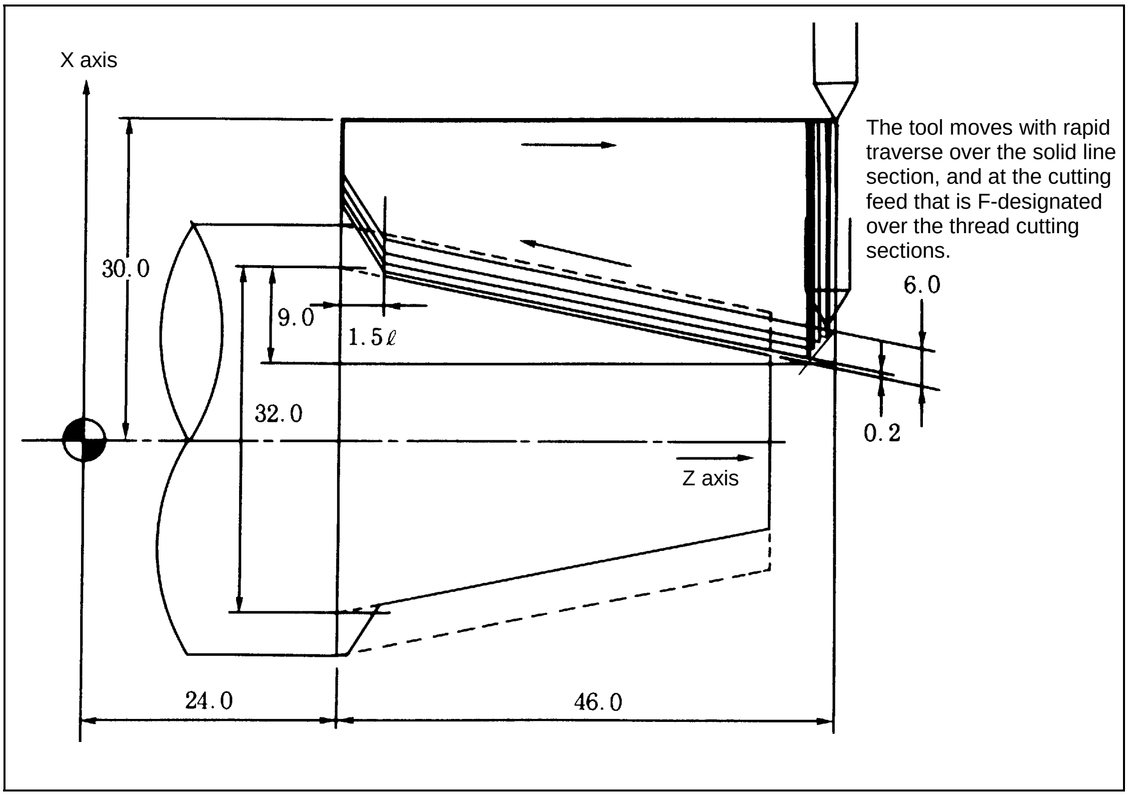
When Ri is negative



When Ri is positive



Example of program



G76 P021560 Q0.1 R0.2;
G76 U–40.0 W–46.0 R9.0 P6.0 Q3.5 F4.0;

Interrupt operation

(1) When the feed hold button is pressed while G76 is being executed, automatic operation will stop upon completion of a block without thread cutting if thread cutting is ongoing. (The feed hold lamp lights immediately in the feed hold mode and it goes off when automatic operation stops.)
If thread cutting is not ongoing, the feed hold lamp lights and the feed hold status is established.

(2) The tool stops upon completion of operations (1), (4) and (5) when the mode is switched to another automatic operation mode during the G76 command execution, when automatic operation is changed to manual operations or when single block operation is conducted.

(3) The dry run valid/invalid status during G76 execution does not change during thread cutting.

13.2.8 Precautions for compound fixed cycles (G70 to G76)

Precautions

(1) Except for the parameters which have been preset on the setting display unit, command all the required parameters in the blocks containing the compound fixed cycle commands.

(2) Provided that the finished shape program has been entered in the memory, compound fixed cycle I commands can be executed in the memory or MDI operation mode.

(3) When executing a G70 to G73 command, ensure that the sequence number of the finished shape program specified with P and Q is not duplicated in that program.

(4) The finished shape program specified by P and Q in the G71 to G73 blocks should be prepared so that the maximum number of blocks is 50 for all the commands for corner chamfering, corner rounding and other commands including the automatic insertion blocks based on nose R compensation.
If this number is exceeded, program error "P202 Block over (MRC)" results.

(5) The finished shape program specified by the G71 to G73 blocks should be a program with monotonous changes (increase or decrease only) for both the X and Z axes.

(6) Blocks without movement in the finished shape program are ignored.

(7) N, F, S, and T commands in the finished shape program are ignored.

(8) When any of the following commands are present in a finished shape program, program error "P201 Program error (MRC)" results.
- Commands related to reference point return (G27, G28, G29, G30)
- Thread cutting (G33)
- Fixed cycles
- Skip functions (G31)

(9) If subprogram call or macro call commands are present in the finished shape program, these commands will also be executed. Note that if the subprogram is called from the last block in the finish shape program, it will not be executed.

(10) Except for thread cutting cycles, operation stops at the end (start) point of each block in the single block mode.

(11) Remember that, depending on whether the sequence or program number is designated, the next block upon completion of the G71, G72 or G73 command will differ.

(a) When the sequence number is designated	**(b) When the program number is designated**
The next block is the next block designated by Q. ≀ N100 G71 P200 Q500 U__ W__ ･･･ ; N200; N300; N400; Finished shape program N500; N600; Operation moves to the N600 block upon completion of the cycle.	The next block is the next block of the cycle command. ≀ N100 G71 A100 U__ W__ ･･･ ; N200 N300 N400 O100 N10 X100. Z50. ; N20 ≀ Operation moves to the N200 block upon completion of the cycle.

(12) The next block applying upon completion of the G70 command is the next block of the command block.

```
≀
N100······ ;
N200;
N300;
N400;
N500;
≀
N1000 G70 P200 Q500;   (or G70   A100 ;)
N1100   ······ ;
≀
```

Operation moves to the N1100 block upon completion of the G70 command.

(13) It is possible to apply a manual interrupt while a compound fixed cycle command (G70 to G76) is being executed. However, upon completion of the interrupt, the tool must first be returned to the position where the interrupt was applied and then the compound fixed cycle must be restarted.
If it is restarted without the tool having been returned, all subsequent movements will deviate by an amount equivalent to the manual interrupt amount.

(14) Compound fixed cycle commands are unmodal commands and so they must be issued every time they are required.

(15) Program error "P203 D cmnd figure error (MRC)" results with the G71 and G72 commands even when, because of nose R compensation, there is no further movement of the Z axis in the second block or the Z axis has moved in the opposite direction.

(16) The common variables set in the G70 to G73 finished shape programs are ignored, and the values set before issuance of the rough cutting cycle command become valid.

(17) If an circular command is present in the first movement block in the finished shape program, program error "P201 Program error (MRC)" results.

13.3 Hole Drilling Fixed Cycles; G80 to G89

Function and purpose

These fixed cycles are used for predetermined sequences of machining operations such as normal positioning, hole drilling, boring and tapping, etc., which are specified in a block. The various sequences available using these functions are listed in the table below.

G code	Hole drilling axis	Hole drilling start	Operation at hole bottom	Return operation	Application
G80	–	–	–	–	Cancel
G83	Z	Cutting feed, intermittent feed	Dwell	Rapid traverse	Deep hole drilling cycle
G84	Z	Cutting feed	Dwell, spindle reverse rotation	Cutting feed	Tapping cycle
G85	Z	Cutting feed	Dwell	Cutting feed	Boring cycle
G87	X	Cutting feed, intermittent feed	Dwell	Rapid traverse	Deep hole drilling cycle
G88	X	Cutting feed	Dwell, spindle reverse rotation	Cutting feed	Tapping cycle
G89	X	Cutting feed	Dwell	Cutting feed	Boring cycle

A fixed cycle mode is canceled when the G80 or any G command in the 01 group is issued. The various data will also be cleared simultaneously to zero.

Command format

(1) Face hole drilling

G8∆ X/U__ C/H__ Z/W__ R__ Q__ P__ F__ D__ M__ S (during synchronous tap) ,R__ K__ ;	
G8∆	Hole machining mode (G83, G84, G85)
X/U__ C/H__	Hole positioning data
Z/W__ R__ Q__ P__ F__ D__ M__ S (during synchronous tap) ,R__	Hole machining data
K__	Number of repetitions

(2) Longitudinal hole machining

G8□ Z/W__ C/H__ X/U__ R__ Q__ P__ F__ D__ M__ S (during synchronous tap) ,R__ K__ ;	
G8□	Hole machining mode (G87, G88, G89)
Z/W__ C/H__	Hole positioning data
X/U__ R__ Q__ P__ F__ D__ M__ S (during synchronous tap) ,R__	Hole machining data
K__	Number of repetitions

(3) Cancel

G80 ;

(4) Data outline and corresponding addresses

(a) Hole machining modes : These are the fixed cycle modes for drilling (G83, G87), tapping (G84, G88) and boring (G85, G89).
These are modal commands and once they have been issued, they will remain valid until another hole machining mode command, the cancel command for the hole drilling fixed cycle or a G command in the 01 group is issued.

(b) Hole positioning data : These are for the positioning of the X (Z) and C axes.
These are unmodal data, and they are commanded block by block when the same hole machining mode is to be executed continuously.

(c) Hole machining data : These are the actual machining mode.
Except for Q, they are modal. Q in the G83 or G87 command is unmodal and is commanded block by block as required.

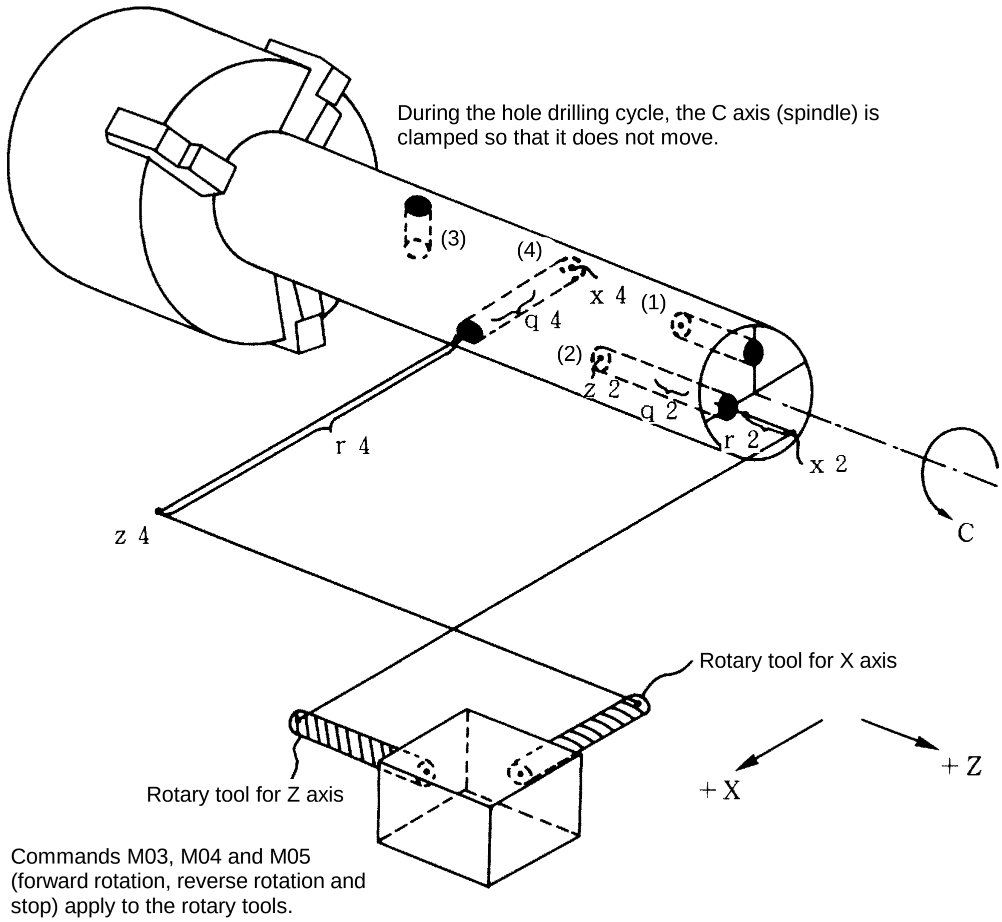
(d) Number of repetitions : This number is designated for hole machining at equal intervals when the same cycle is to be repeated.
The setting range is from 0 to 9999; the decimal point is not valid.
The number is unmodal and is valid only in the block in which it has been assigned. When this number is not designated, it is treated as K1. When K0 is designated, the hole machining data are stored in the memory but no holes will be machined.

Address	Significance
G	Selection of hole machining cycle sequence (G80, G83, G84, G85, G87, G88, G89)
X/U (Z/W*) C/H	Designation of hole position initial point (absolute/incremental value)
Z/W (X/U*)	Designation of hole bottom position (absolute/incremental value from reference point)
R	Designation of R point position (incremental value from initial point) (sign ignored)
Q	Designation of cut amount for each cutting pass with G83 (G87); always incremental value, radius value (sign ignored) Override during return with G84 (G88)
P	Designation of dwell time at hole bottom point; relationship between time and designated value is same as for G04 designation
F/E	Designation of feedrate for cutting feed
D	Designation of tap spindle No. with G84 (G88)
M	Designation of M code for C axis clamp
S	Designation of tap spindle rotation speed during G84 (G88)
,R	Changeover of synchronization/asynchronization with G84 (G88)
K	Designation of number of repetitions, 0 to 9999 (standard value = 1)

* Addresses in parentheses apply for commands G87, G88 and G89.

Outline drawing

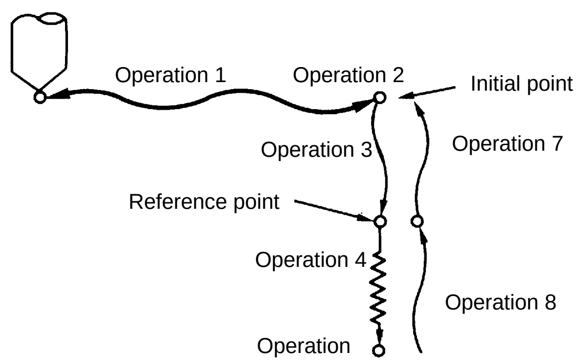
The hole drilling axes for the hole drilling fixed cycle and the positioning, etc., are shown in the outline drawing below.



(1) G83 Xx1 Cc1 Zz1 Rr1 Qq1 Pp1 Ff1 Kk1;
(2) G83 Xx2 Cc2 Zz2 Rr2 Qq2 Pp2 Ff2 Kk2;
(3) G87 Zz3 Cc3 Xx3 Rr3 Qq3 Pp3 Ff3 Kk3;
(4) G87 Zz4 Cc4 Xx4 Rf4 Qq4 Pp4 Ff4 Kk4;

Example of operation

There are 7 actual operations which are each described in turn below.



Operation 1 : This denotes the positioning (by rapid traverse) of the tool to the X (Z) and C axis initial point.
Operation 2 : This is output provided that the M code for C-axis clamping is issued.
Operation 3 : The tool is positioned as far as the R point by rapid traverse.
Operation 4 : Hole machining is conducted by cutting feed.
Operation 5 : This operation takes place at the hole bottom position and it differs according to the fixed cycle mode. Possible actions include rotary tool reverse rotation (M04), rotary tool forward rotation (M03) and dwell.
Operation 6 : The tool is retracted as far as the R point.
Operation 7 : The tool is returned as far as the initial point at the rapid traverse rate.
(Operations 6 and 7 may be a single operation depending on the fixed cycle mode.)

Whether the fixed cycle is to be completed at operation 6 or 7 can be selected by the following G commands.

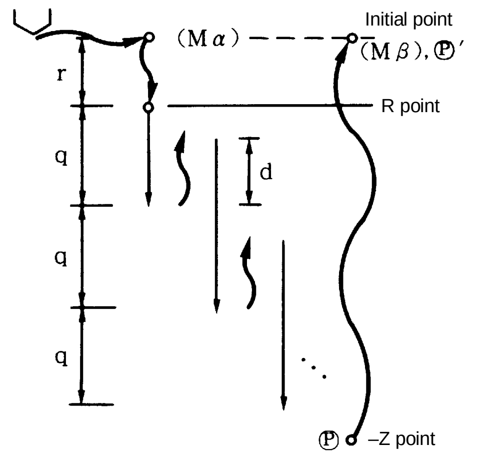
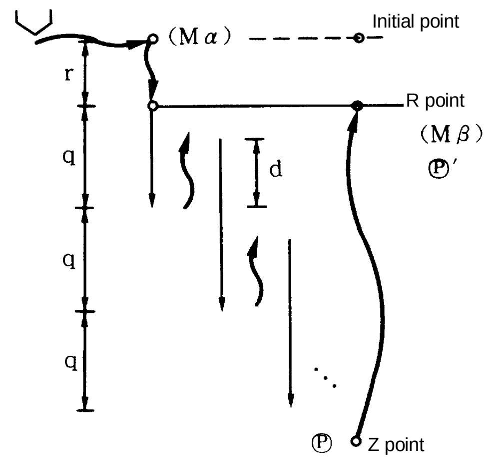
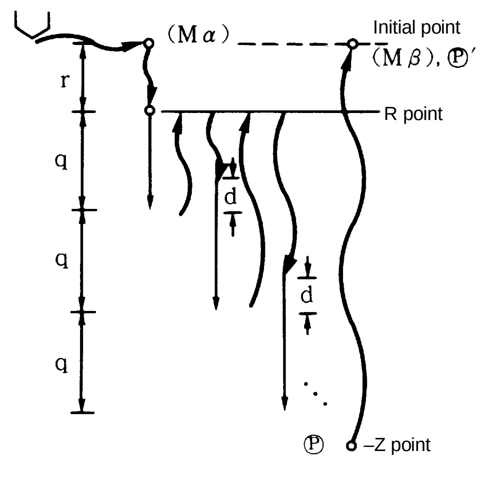
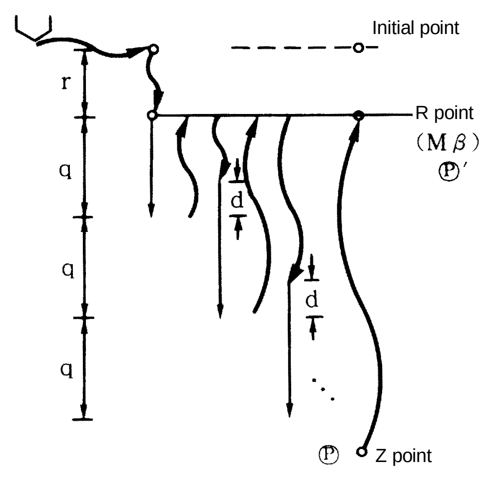
G98 ･･･ Initial level return
G99 ･･･ Reference point level return

These G commands are modal. Once, for instance, G98 is designated, the G98 mode will remain valid until the G99 command is issued. The G98 mode is established in the initialized state when the NC unit is ready to operate.

13.3.1 G83 face deep hole drilling cycle 1 (G87 longitudinal deep hole drilling cycle 1)

When the Q command is present (deep hole drilling)

G83 (G87) X(z)__ C__ Z(x)__ R__ Q__ P__ F__ K__ M__;

<table>
<tr><th>Type</th><th>G98 mode</th><th>G99 mode</th></tr>
<tr><td>A
(high speed)</td><td>
</td><td>
</td></tr>
<tr><td>B
(nor-mal)</td><td>
</td><td>
</td></tr>
</table>

(1) Types A and B can be selected by the control parameter "G83/87 rapid".

(2) Return amount "d" is set by the setup parameter "G83 Retract". The tool returns at rapid traverse.

(3) (Mα) · · · · The M code (Mm) is output when there is a C-axis clamping M code command (Mm).

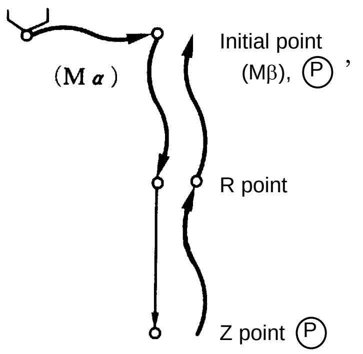
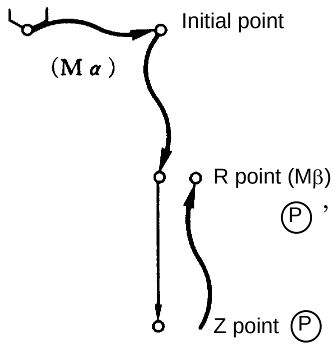
(4) (Mβ) · · · · The C-axis unclamping M code (C-axis clamp M code + 1 = Mm + 1) is output when there is a C-axis clamping M code command (Mm).

(5) Ⓟ · · · · · Dwell is performed for the duration equivalent to the time designated by P.

(6) Ⓟ ' · · · · After the C-axis unclamping M code (Mm + 1) has been output, dwell is performed for the duration equivalent to the time set by the machine parameter "clmp_ D".

When the Q command is not present (drilling)

G83 (G87) X(z)__ C__ Z(x)__ R__ Q__ P__ F__ K__ M__;

G98 mode	G99 mode
	

(1) See Section 13.3.1 "When the Q command is present (deep hole drilling)" for details on Mα, Mβ, P and P '.

13.3.2 G84 face tapping cycle (G88 longitudinal tapping cycle)

Selection of synchronous/asynchronous tapping cycle

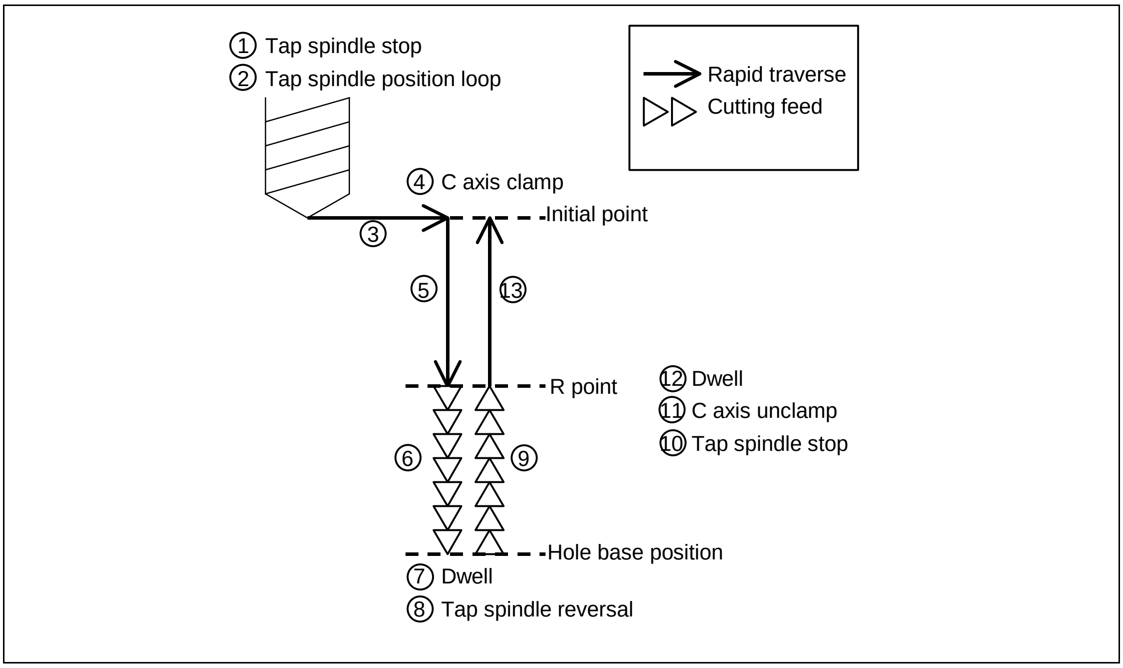
Command ",R1" or ",R2" with the tapping cycle command to select the synchronous tapping cycle, and ",R0" to select the asynchronous tapping cycle.
When ",R1" is commanded, tap spindle zero point return will not be executed before tap cutting starts.
When ",R2" is commanded, tap spindle zero point return is carried out before tap cutting starts. (Tap spindle zero point return type)
Asynchronous tapping takes place when there is no ",R*" command.
During the tapping cycle modal, the previous command will be followed unless there is another command.
During the tapping cycle modal, tapping will not be carried out in a block containing only the ",R*" command. Only the mode will be changed between asynchronous and synchronous. If ",R2" is commanded while operating with ",R0" or ",R1", the tap spindle will be returned to the zero point.

	Positioning command at start of G84 (G88) mode	**Positioning command during G84 (G88) mode**
No ,R command	Asynchronous method	Previous method
,R0 (asynchronous method)	Asynchronous method	Asynchronous method
,R1 (synchronous method: no tap spindle zero point return)	Synchronous method (no tap spindle zero point return)	Synchronous method (no tap spindle zero point return)
,R2 (synchronous method: with tap spindle zero point return)	Synchronous method (with tap spindle zero point return)	When previous method is asynchronous method or synchronous method (without tap spindle zero point return) → Synchronous method (with tap spindle zero point return)
		When previous method is synchronous method (with tap spindle zero point return) → Synchronous method (without tap spindle zero point return)

Operation sequence

The operation sequence for the synchronous tapping cycle is the same as the asynchronous tapping cycle. However, the tap spindle forward run/reverse run M code is not output.

G84(G88) X/U○(Z/W○) C/H○ Z/W○(X/U○) R○ P○ F○/E○ Q○ K○ M○ D○ S○,R○;



① The tap spindle stops.
② The tap spindle start position loop control.
③ The spindle is positioned to the initial point.
(When using the tap spindle zero point return type, the spindle is returned to the zero point.)
④ The C axis clamp M code is output (only when commanded).
⑤ The spindle is positioned to the R point.
⑥ Tap cutting is started at a speed synchronized with the tap spindle.
⑦ The spindle stops for the designated time with dwell (only when commanded).
⑧ The tap spindle is reversed.
⑨ Tap return is executed at the step ⑥ speed x override speed.
⑩ The tap spindle stops.
⑪ The C axis unclamp M code (C axis clamp M+1) is output (only when commanded in step ④).
⑫ The spindle stops for the set time with dwell to unclamp the C axis.
(Only when commanded in step ④.)
⑬ The spindle is positioned to the initial point.
(Only during the hole drilling cycle initial point return mode (G98).)

The override is canceled during steps ⑥ to ⑩ of the tapping cycle, and the override is automatically set to 100%. Dry run and feed hold are also ignored.

If feed hold is input during the tapping cycle, the block will stop after the return operation.
Block stop with single block cannot be applied during steps ⑥ to ⑩ of the tapping cycle. If operation is applied with single block, the block stop will be applied just before step ⑥, immediately after step ⑩ (differs according to return mode (G98/G99) or presence of C axis clamp M code).
The tap spindle's position loop control state is held until the synchronous tapping cycle modal is canceled.

Whether to move the spindle to the initial point in step ⑬ after movement in step ⑫ is completed can be selected with the G command.

G98: Hole drilling cycle initial point return mode

G99: Hole drilling cycle R point return mode

The command is a modal and is held until the mode is changed with a command. The default state is G98.

Relation with other functions

(1) Constant surface speed control
The spindle rotation speed targeted for constant surface speed control does not change during the constant surface speed command in the tapping cycle, or during the tapping cycle command during the constant surface speed control mode. (Constant surface speed control is not applied.) During the tapping cycle, the spindle rotation speed applied at the point the tapping cycle command is executed is held. When the tapping cycle command is finished, the constant surface speed will be obtained from the position of the constant surface speed control axis, and the spindle rotation speed will be changed.

(2) Spindle C axis control
If another spindle position command (synchronous tapping cycle, spindle synchronization control (G114.n)) is commanded during spindle C axis control, or if spindle C axis control is commanded during another spindle position control, the "M01 operation error 1026" will occur.

(3) Spindle synchronization control, tool-spindle synchronization control I or II
If the synchronous tapping cycle command is issued to a related axis in spindle synchronization control or tool-spindle synchronization control I or II, or if the spindle synchronization control or tool-spindle control I or II is commanded to a spindle in the synchronized tapping cycle, the "M01 operation error 1007" will occur.

(4) Spindle orientation
Spindle orientation is invalid even if commanded during the synchronous tapping cycle. If the synchronous tap cycle is commanded during spindle orientation, the synchronous tapping cycle operation will be executed first. In either case, the spindle orientation is valid when the synchronous tapping cycle modal is exited.

(5) Spindle type servo
In the same manner as the normal spindle, inclined constant acceleration/deceleration can be executed during the synchronous tapping cycle by setting the tap spindle rotation speed and tap time constant parameters.
Multi-step acceleration/deceleration is executed when multi-step acceleration/deceleration for synchronous tapping cycle is set.
The position loop gain used for synchronous tap control can be selected with the parameters.

(6) Spindle superimposition control
If the tapping cycle command is issued to a reference spindle during spindle superimposition control, the "M01 operation error 1007" will occur.
The synchronous tapping cycle command (differential tap) can be issued to the superimposition spindle during spindle superimposition control.

(7) Override, spindle override
Override is invalid during tap cutting. The override is canceled, and the override is automatically set to 100%.

(8) Dry run
Dry run is invalid during tap cutting.

(9) Single block
Single block stop is invalid from the start of tap cutting to the end of the tapping cycle. If single block is turned ON, the block will stop after the operation following the tap return operation is completed. (The block stop position will change according to the return mode (G96/G99) or the presence of the C axis clamp M code command.)

(10) Feed hold
Feed hold is invalid during tap cutting. If feed hold is input, the block will stop after tap return is completed.

(11) Program check operation
The thread cutting and tapping cycle in the program check operation include the actual cutting mode and dry operation mode.

(12) Cross machining command (G110)
A program error (P501) will occur if the cross machining command (G110) is executed in the system during fixed cycle mode.

(13) Axis name change
The synchronous tapping cycle can be commanded to an axis name after axis name change is completed.

(14) Control axis synchronization
The synchronous tapping cycle command can be issued to a reference axis.
The synchronous tapping cycle command cannot be issued to a synchronous axis.
Whether to result in the "M01 operation error 1003" or to invalidate the command when a synchronous tapping cycle is issued to a synchronous axis can be set with the parameters.

(15) Control axis superimposition
The synchronous tapping cycle can be issued to a superimposition related axis.

(16) Inclined coordinate rotation
The tapping cycle command will function even during the inclined coordinate rotation mode.
If inclined coordinate rotation is commanded or canceled during the fixed cycle mode, a program error (P34) will occur.

Precautions

(1) If address "F" and "E" are commanded simultaneously in the synchronous tapping cycle command block, the valid address will be determined according to the input unit.
For metric input : Address F will be valid, and the E command will be ignored.
For inch input : Address E will be valid, and the F command will be ignored.

(2) During the synchronous tapping cycle, the tap axis cutting feedrate will be synchronized with the tap spindle rotation speed.
The tap axis cutting feedrate and tap spindle rotation speed are clamped by the tap axis' maximum cutting feedrate or tap spindle maximum rotation speed. The synchronous relation of the tap spindle and tap axis is maintained even when clamped.

(3) The cutting feedrate of the tap axis during the synchronous tapping cycle is synchronized with the tap spindle rotation speed, but the cutting feedrate modal (F modal) does not change.

(4) The reverse run/forward run M code for the tap spindle is not output at the R point or hole base point while the synchronous tapping cycle is being executed. (MF is also not output.) The tap spindle forward run/reverse run does not need to be commanded before the fixed cycle command.

(5) The tap spindle for executing the synchronous tap cycle can be designated with address D. If a spindle that is not serially connected is designated, a program error (P182) will occur.

(6) Whether to carry out forward tapping or reverse tapping when executing the synchronous tap cycle is determined by the sign for address D. (Positive: forward tapping, negative: reverse tapping) If there is no address D command, forward tapping will take place with the 1st spindle. If the 1st spindle is not serially connected, a program error (P182) will occur.

(7) The S, D and F/E commands in the synchronous tapping cycle modal cannot be changed, after once the synchronous tapping cycle is commanded. A program error (P33) will occur if a command is issued. To change the command, cancel the synchronous tapping cycle modal once with G80, and command S, D and F/E with the synchronous tapping cycle command.

(8) If there is no S command for the tap spindle in the first synchronous tapping cycle command after changing to the tap cycle modal, a program error (P181) will occur.

(9) The tap spindle's S modal is updated by the S command for the synchronous tapping cycle command.

(10) S commands and rotation commands issued from another system for the tap spindle in the synchronous tapping cycle are invalid. Note that the modal is updated, so these will be validated when the synchronous tapping cycle is canceled.

(11) A program error (P33) will occur if G84 or G88 is commanded during the synchronous tapping cycle modal. When changing the modal, cancel the synchronous tapping cycle with G80 before commanding G84/G88.

(12) A program error (P33) will occur if the spindle axis' forward run/reverse run M code is not set with spindle parameter "#24sprcmm" in the asynchronous tapping cycle command, or if the following M code is set.

M00/M01/M02/M30	M command
M98/M99	Sub-spindle control command

If the spindle is not designated with address D, M3/M4 will be applied unconditionally.

(13) During synchronous tap cycle cutting, an error of several pulses (detection unit) will occur between the tap axis and tap spindle even in the constant state (state with no speed changes). Generally, the synchronization error is larger than the constant state in the transient state (state with speed fluctuations during acceleration/deceleration). This is caused by a difference in the tap axis and tap spindle's responsiveness. The differences in responsiveness occur due to the various machine system, including drive section, conditions (drive section capacity, inertia, etc.). Ideally, the various conditions of two synchronized axes should be set to the same when possible to reduce the synchronization error.

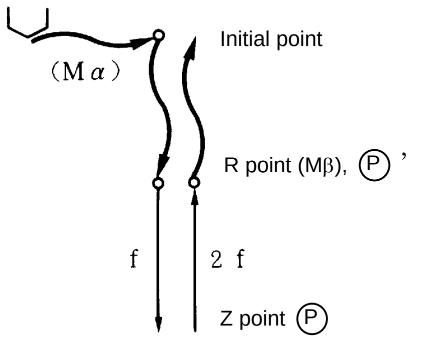
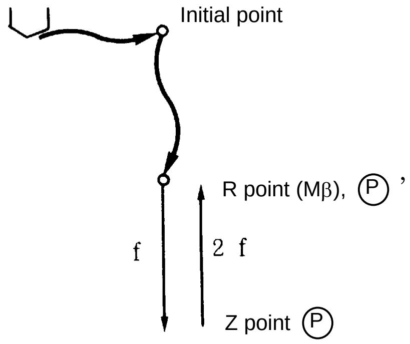
(14) The override during return (Q command) is an option. A program error (P39) will occur if issued when the option is not available.

(15) The tap spindle zero point return (during ,R2 command) is an option. A program error (P39) will occur if issued when the option is not available.

13.3.3 G85 face boring cycle (G89 longitudinal boring cycle)

Detailed description

G85 (G89) X(z)_ C_ Z(x)_ R_ P_ F_ K_ M_;

G98 mode	G99 mode
	

(1) See Section 13.3.1 "When the Q command is present (deep hole drilling)" for details on Mα, Mβ, P and P '.

(2) The tool returns to the R point at a cutting feedrate which is double the designated feedrate command. However, it does not exceed the maximum cutting feedrate.

13.3.4 G80 hole drilling fixed cycle cancel

Detailed description

This command cancels the hole drilling fixed cycles (G83, G84, G85, G87, G88, G89). The hole machining mode and hole machining data are also canceled.

13.3.5 Precautions for using hole drilling fixed cycles

Precautions

(1) When the G84 and G88 fixed cycle commands are issued, the rotary tool must be rotated in the prescribed direction beforehand using a miscellaneous function (M3, M4).

(2) If the basic axis, additional axis and R data are present in the block, hole drilling is performed in a fixed cycle mode; it will not be performed if the data are not present.
Even if the X-axis data are present, hole drilling will not result if a dwell (G04) time command is present in the block.

(3) The hole machining data (Q, P) data should be commanded in the block (block including the basic axis, additional axis and R data) in which the holes are machined.
The modal data will not be updated even if these data are commanded in a non-hole drilling block.

(4) When resetting is applied during the execution of the G85 (G89) command, the F medals may change.

(5) The hole drilling fixed cycles are also canceled by any G code in the 01 group except G80.
If it is commanded in the same block as the fixed cycle, the fixed cycle will be ignored.
m = 01 group code, n = hole drilling fixed cycle code

(a) $\underbrace{\text{Gm}}_{\text{Executed}}\ \underbrace{\text{Gn}}_{\text{Ignored}}\ \underbrace{\text{X(z)\_\_ C\_\_ Z(x)\_\_}}_{\text{Executed}}\ \underbrace{\text{R\_\_ Q\_\_ P\_\_ K\_\_}}_{\text{Ignored}}\ \underbrace{\text{F\_\_}}_{\text{Memorized}}\ ;$

(b) $\underbrace{\text{Gn}}_{\text{Executed}}\ \underbrace{\text{Gm X(z)\_\_ C\_\_ Z(x)\_\_}}_{\text{Executed}}\ \underbrace{\text{R\_\_ Q\_\_ P\_\_ K\_\_}}_{\text{Ignored}}\ \underbrace{\text{F\_\_}}_{\text{Memorized}}\ ;$

(Examples) G01 G83 X100. C30. Z50. R–10. Q10. P1 F100.;
G83 G01 X100. C30. Z50. R–10. Q10. P1 F100.;
In both cases, "G01 X100. C30. Z50. F100." is executed.

(6) When a miscellaneous command is issued in the same block as the fixed cycle command, it is output at the same time as the initial positioning.
However, if the C-axis clamping M code which has been set by parameter (Base specification parameter 2-16 clmp_M) is commanded in the same block, the M code will be output after positioning (operation 2).
After the holes have been machined, the tool returns to the return point (initial point in G98 mode; R point in G99 mode). Then the C-axis unclamping M code (clamp M + 1) is output and dwell is performed for the duration of time set by the parameter (Base specification parameter 2-17 clmp_D).
When the number of cutting passes has been designated, the above control is exercised for the initial pass only except for the C-axis clamping M code. In the case of the C-axis clamping/unclamping M commands, operation is modal and the code is output with each cutting pass until the operation is canceled by the fixed cycle cancel command.

(7) When a tool length offset command (T function) is issued in a hole drilling fixed cycle mode, execution will follow the tool length offset function.

(8) When a hole drilling fixed cycle command is issued during nose R compensation, program error "P155 Fixed cyc exec during compen" results.

(9) Operation is fixed to initial point level return with the G code system 1. It is not possible to change the return level using G98/G99. It should be borne in mind that if G98 or G99 is commanded, a different function will be executed.

13.4 Deep Hole Drilling Cycle 2; G83.2

Function and purpose

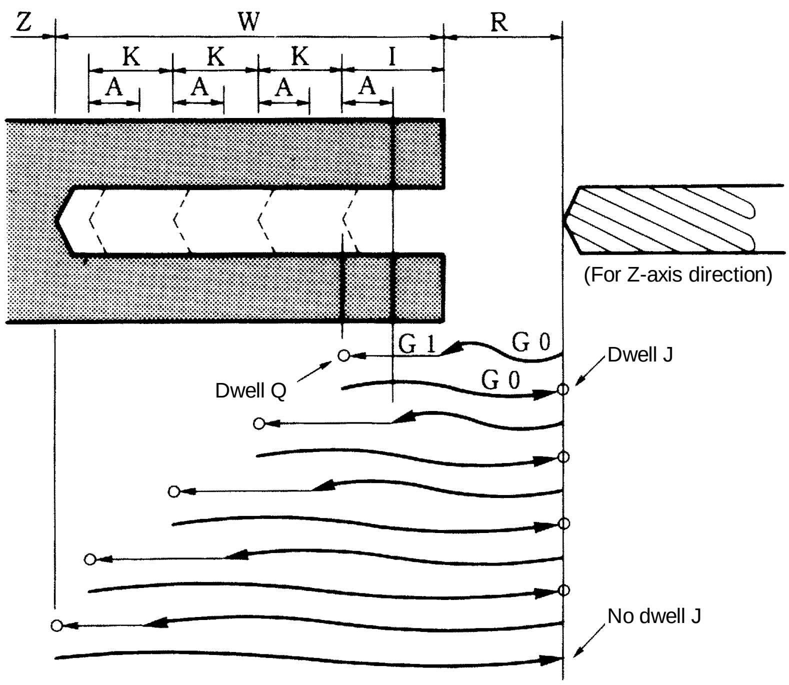
The deep hole drilling cycle 2 drills deep holes in the X-axis or Z-axis direction by commanding the X or Z coordinate of the end point and the cut amount with cutting feed.

Command format

G83.2 W/Z/U/X_ R_ I_ K_ A_ Q_ J_ F_ ;

W/Z/U/X	Incremental value from hole drilling and cutting start point/coordinates of hole bottom (with sign)
R	Incremental value (no sign) from present position up to hole drilling start point (always radius value with incremental value)
I	Cut amount of first cutting pass (no sign) (always radius value with incremental value)
K	Cut amount of second and subsequent cutting passes (no sign) (always radial value with incremental value)
A	Drill stop safety distance for second and subsequent cutting passes (no sign) (always radial value with incremental value)
Q	Dwell time at cut point (no sign, decimal point invalid)
J	Dwell time at return point (no sign, decimal point invalid)
F	Cutting feedrate

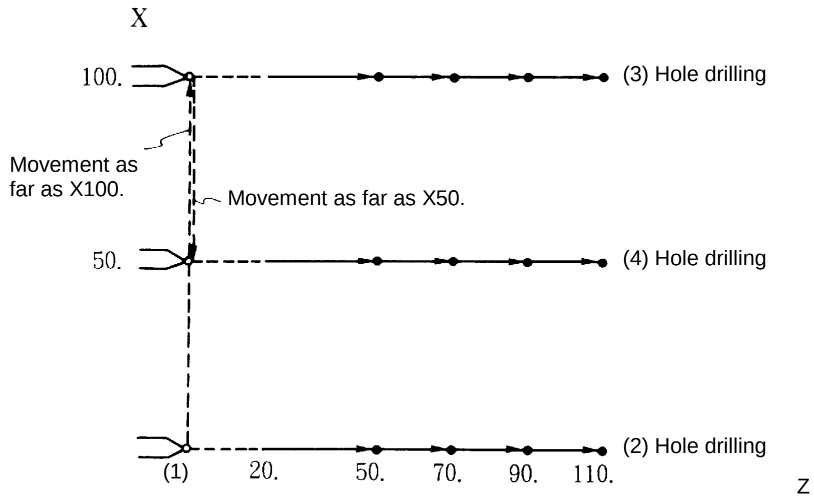
Operation



Example of program (when deep hole drilling cycle 2 is used as a modal command)

G28 X Z;
G0 X0. Z0.; ······································· (1)
G83.2 Z110. R20. I30. K20. A5. Q1000
J500. F300. ; ······························ (2)
X100.; ·· (3)
X50.; ··· (4)
M02;

Operation



Detailed description

A, P and Q commands for compound fixed cycles are as follows.

(1) When the drill stop safety distance (address "A") command is not given, the setup parameter "G83 Retract" parameter setting value is used.

(2) The deep hole drilling cycle 2 is a modal command and so it will remain valid until a command in the same modal group or the cancel command (G80) is issued.

(3) If the command either for the cut amount (address "I") of the first cutting pass or the cut amount (address "K") of the second and subsequent passes is not present (including a command value of 0), the command value which is present will be used and the operation will be executed with both I and K equal to the command value.
If both commands are not present, hole drilling is conducted once as far as the hole bottom.

(4) When the axis address of the hole drilling axes has been commanded a multiple number of times in a block, the address commanded last is valid.

(5) The deep hole drilling cycle 2 is also canceled by any G code in the 01 group except G80. If it is commanded in the same block as the fixed cycle, the fixed cycle will be ignored.
m = 01 group code, n = hole drilling fixed cycle code

(a) $\underbrace{\text{Gm}}_{\text{Executed}}\ \underbrace{\text{Gn}}_{\text{Ignored}}\ \underbrace{\text{X(z)\_\_ C\_\_ Z(x)\_\_}}_{\text{Executed}}\ \underbrace{\text{R\_\_ I\_\_ K\_\_ A\_\_ Q\_\_ J\_\_}}_{\text{Ignored}}\ \underbrace{\text{F\_\_}}_{\text{Memorized}}\ ;$

(b) $\underbrace{\text{Gn}}_{\text{Executed}}\ \underbrace{\text{Gm X(z)\_\_ C\_\_ Z(x)\_\_}}_{\text{Executed}}\ \underbrace{\text{R\_\_ I\_\_ K\_\_ A\_\_ Q\_\_ J\_\_}}_{\text{Ignored}}\ \underbrace{\text{F\_\_}}_{\text{Memorized}}\ ;$

(Examples) G01 G83.2 Z50. R–10. I8. K10. A3. Q1000 J500 F100.;
G83.2 G01 Z50. R–10. I8. K10. A3. Q1000 J500 F100.;
In both cases, "G01 Z50. F100." is executed.

(6) When a miscellaneous function is commanded in the same block as the deep hole drilling cycle 2 command, it is output at the same time as the initial positioning.

(7) When a tool length offset command (T function) is issued in the deep hole drilling cycle 2 mode, execution will follow the tool length offset function.

(8) If the basic axis, additional axis or R data are in the block during the deep hole drilling cycle 2 mode, hole drilling is performed; if the data are not present, no holes are machined.
Even if the X-axis data are present, no holes will be machined if the dwell (G04) time command is present in the block.

(9) Command the hole machining data (A, I, K, Q, J) in the block (including the basic axis, additional axis or R data) in which the hole drilling operation is conducted.
Even if they are commanded in a block with no hole drilling operation, the modal data will not be updated.

(10) A program error "P33 Format error" results with the following commands.

(a) When both the X hole drilling axis (command address X or U) and the Z hole drilling axis (command address Z or W) have been commanded

(b) When any axis except X or Z axes (any command address except X, U, Z and W) has been commanded

(11) When the feed hold button is pressed while the deep hole drilling cycle 2 is being executed, feed hold results at that point, and when automatic operation is restarted, the remainder is executed.

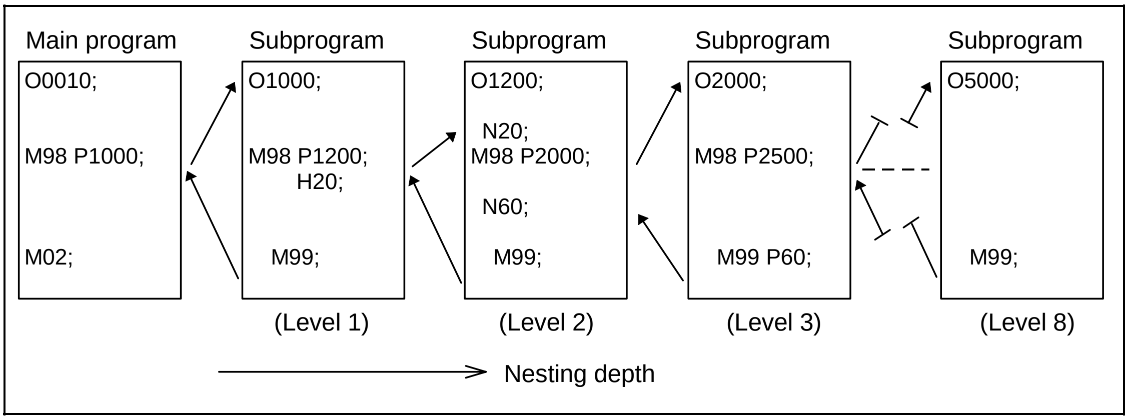
(12) When an interrupt based on manual operation is performed during the feed hold (manual ABS switch ON), the operation in the deep hole drilling cycle 2 modal is shifted from the automatic operation restart by an amount equivalent to the movement caused by the interrupt.

(13) With single block operation, block stop results upon completion of the deep hole drilling cycle 2 command.

13.5 Subprogram Control; M98, M99

Function and purpose

Fixed sequences or repeatedly used parameters can be stored in the memory as subprograms which can then be called from the main program when required. M98 serves to call subprograms and M99 serves to return operation from the subprogram to the main program. Furthermore, it is possible to call other subprograms from particular subprograms and the nesting depth can include as many as 8 levels.



The table below shows the functions which can be executed by adding and combining, subprogram control functions and fixed cycle functions.

	Case 1	Case 2	Case 3	Case 4
1. Subprogram control 2. Fixed cycles	No No	Yes No	Yes Yes	No Yes
Function				
1. Memory operation	○	○	○	○
2. Subprogram call	×	○	○	×
3. Subprogram variable designation **(Note 2)**	×	○	○	×
4. Subprogram nesting level call **(Note 3)**	×	○	○	×
5. Fixed cycles	×	×	○	○
6. Subprogram editing for fixed cycle	×	×	○	○

(Note 1) "○" denotes a function which can be used and "×" a function which cannot be used.

(Note 2) Variables cannot be transferred with the M98 command but variable commands in subprograms can be used provided that the variable command option is available.

(Note 3) A maximum of 8 nesting levels can be called form the nesting depth.

Command format

Subprogram call

M98 P__ H__ L__ ;	
P__	Program number of subprogram to be called (own program if omitted) P can be omitted only during memory operation and MDI operation.
H__	Sequence number in subprogram to be called (head block if omitted)
L__	Number of subprogram repetitions (L1 if omitted)

Return to main program from subprogram

M99 P__ L__ ;	
P__	Sequence number of return destination (returned to block following block to be called if omitted)
L__	Number of times after repetition number has been changed

Creating and entering subprograms

Subprograms have the same format as machining programs for normal memory operation except that the subprogram completion instruction M99(P_ L_); is entered as an independent block at the last block.

○△△△△△△△△△△△△△;	Program number as subprogram number
··························· ;	
··························· ;	
:	Program number as subprogram number
:	
··························· ;	
M99;	Subprogram return command
% (EOR)	Entry completion code (% with ISO code and EOR with EIA code)

(1) The above program is entered by "editing" operations at the setting display unit. For further details, refer to the section on "program editing" in the instruction manual.

(2) Only those subprograms numbers ranging from 1 to 99999999 designated by the optional specifications can be used. When there are no program numbers on the tape, they are entered as the setting number for "program input."

(3) Up to 8 nesting levels can be used for calling programs from subprograms, and program error "P230" results if this number is exceeded.

(4) No distinction between main programs and subprograms is made since they are entered in the sequence in which they were read. This means that main programs and subprograms should not be given the same numbers. (If they are, error "E11" appears during entry.)

; O ○○○○ ; · · · · · · · · · · · · · · · · · · · ; : : M99 ; %	Subprogram A

O △△△△ ; · · · · · · · · · · · · · · · · · · · ; : : M99 ; % ;	Subprogram B

O □□□□ ; · · · · · · · · · · · · · · · · · · · ; : : M99 ; % ;	Subprogram C

(Note 1) Main programs can be entered during memory or MDI operation but subprograms must be entered in the memory.

(Note 2) Besides the M98 command, subprogram nesting is subject to the following commands:

- G65 : Macro call
- G66 : Modal call
- G66.1 : Modal call
- G code call
- Miscellaneous function call (M, S, T, etc.)
- MDI interrupt

(Note 3) Subprogram nesting is not subject to the following commands which can be called even beyond the 8th nesting level.

- Fixed cycles
- Pattern cycles

Subprogram execution

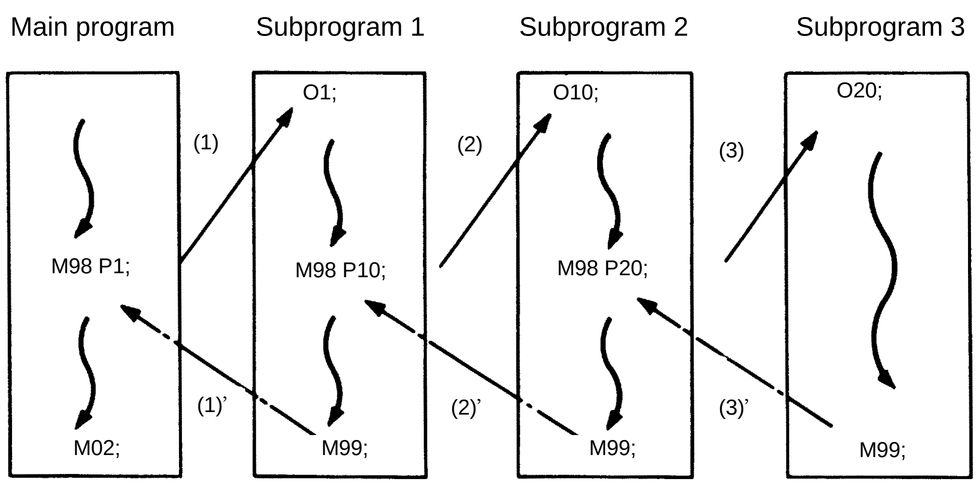
Subprogram call

M98 Pp1 Hh1 Ll1 ;

M98	Subprogram call command
Pp1	Subprogram number to be called by the numerical value of p1 with up to 8 digits
Hh1	Any sequence number within the subprogram to be called by the numerical value of h1 with up to 5 digits
Ll1	Number of repetitions from 1 to 9999 with numerical value of l1 up to 4 digits; if L is omitted, the function is executed once; with l0, there is no execution. For instance, "M98 P1 L3;" is equivalent to the following: M98 P1; M98 P1; M98 P1;

Example of program 1

When there are 3 subprogram calls (known as 3 nesting levels)



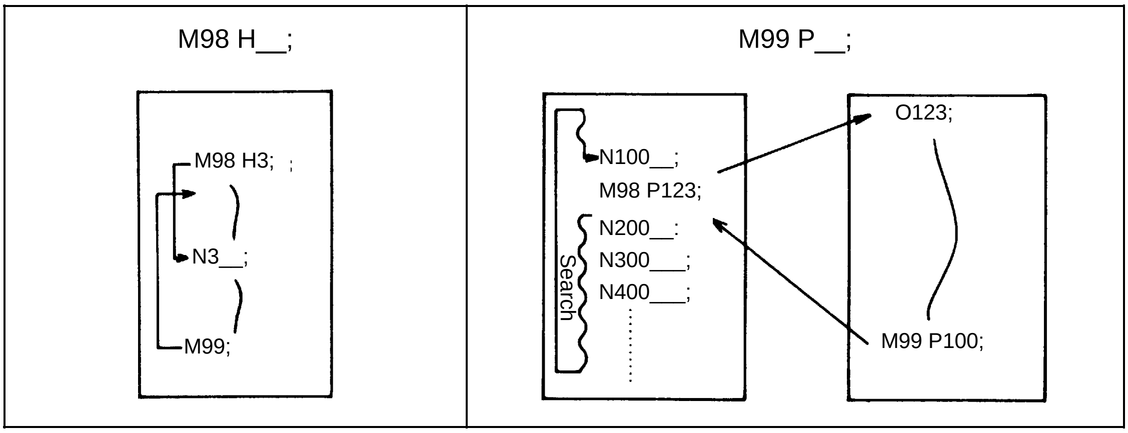
Sequence of execution : (1) → (2) → (3) → (3)’ → (2)’ → (1)’

(1) For nesting, the M98 and M99 commands should always be paired off on a 1:1 basis (1) for (1)’, (2) for (2)’.

(2) Modal information can be rewritten according to the execution sequence without distinction between main programs and subprograms. This means that after calling a subprogram, attention must be paid to the modal data status when programming.

Example of program 2

The "M98 H_;, M99 P_;" commands designate the sequence numbers in a program with a call instruction.



Example of program 3

Main program M98 P2;

O1 ; : : M99 ; %	Subprogram 1

O2 ; : N200 : M99 ; %	Subprogram 2

O3 ; : N200 : M99 ; %	Subprogram 3

(Note 1) When the O2 N200 block is searched with the memory search function, the modal data from O2 to N200 are updated.

(Note 2) The same sequence number can be used with different subprograms.

(Note 3) When the subprogram is to be repeatedly used, it will be repeatedly executed for "l1" times provided that "M98 Pp1 Lλ1;" is programmed.

Other precautions

(1) Program error "P23" results when the designated program number (P) is not located.

(2) Single block stop does not occur with the "M98 P_;, M99;" block. If any address except O, N, P, L or H is used, single block stop can be executed. (With "X100. M98 P100;", operation branches to O100 after "X100." is executed.)

(3) When M99 is commanded by the main program, operation returns to the head. (This is same for MDI.)

(4) Bear in mind that the search operation will take time when the sequence number is designated by "M99 P_".

13.6 Variable Commands

Function and purpose

Programming can be endowed with flexibility and general-purpose capabilities by designating variables, instead of giving direct numerical values to particular addresses in a program, and by assigning the values or those variables as required when executing a program.

Command format

#△△△ = ○○○○○○○○ or #△△△ = [formula]

Detailed description

(1) Variable expressions — **Example**

		Example
(a) #m ··········	m = value consisting of 0 to 9	#100
(b) #[f] ··········	f = one of the following in the formula	# [– #120]
	Numerical value m	123
	Variable	#543
	Formula Operator Formula	#110 + #119
	– (minus) formula	– #120
	[Formula]	[#119]
	Function [formula]	SIN [#110]

(Note 1) The 4 standard operators are +, –, ∗ and / .
(Note 2) Functions cannot be used unless the user macro specifications are available.
(Note 3) Program error "P241" results when a variable number is negative.
(Note 4) Examples of incorrect variable expressions are given below.

Incorrect		Correct
#6/2	→	# [6/2]
# – – 5	→	# [– [–5]]
# – [#1]	→	# [– #1]

(2) Types of variables

The following table gives the types of variables.

Type of variable	Number	Function	Remarks
Common variables	100 to 149:Per system 500 to 549:Common for systems	Can be used in common throughout main, sub and macro programs.	50 + 50*n sets
	100 to 199:Per system 500 to 599:Common for systems		100 + 100*n sets
	100 to 149:Per system 100100 to 800149:Common for systems 500 to 549:Common for systems		450 sets
	100 to 199:Per system 100100 to 800199:Common for systems 500 to 599:Common for systems		900 sets
Local variables	1 to 32	Can be used for local variables in macro programs.	User macro specifications required
System variables	1000 to	Application is fixed by system.	User macro specifications required
Fixed cycle variables	1 to 32	Local variables in fixed cycle programs	User macro specifications required

(Note 1) All common variables are retained even when the power is turned OFF.

(Note 2) The common variables are divided into the following two types.

Common variables 1 : Variables that can be used commonly through all systems.

500 to 549 (599)

100100 to 100149 (100199) : Common with system 1 common variables 2 "100 to 149 (199)"

200100 to 200149 (200199) : Common with system 2 common variables 2 "100 to 149 (199)"

:

800100 to 800149 (800199) : Common with system 8 common variables 2 "100 to 149 (199)"

Common variables 2 : Variables that can be used commonly within the system program.

100 to 149 (99)

#100 to #149 and #500 to #549 can be used for the variable command 50+50*n sets (type A), and #100 to #199 and #500 to #599 can be used for the 100+100*n sets (type B).

(3) Variable quotations

Variables can be used for all addresses except O, N and / (slash).

(a) When the variable value is used directly:
X#1 ·················· Value of #1 is used as the X value.

(b) When the complement of the variable value is used:
X–#2 ················· Value with the #2 sign changed is used as the X value.

(c) When defining variables:
#3 = #5 ·············· Variable #3 uses the equivalent value of variable #5.
#1 = 1000 Variable #1 uses the equivalent value 1000 (which is treated as 1000.).

(d) When defining the variable arithmetic formula:
#1 = #3 + #2 –100 The value of the operation result of #3 + #2 – 100. is used as the #1 value.
X [#1 + #3 + 1000] The value of the operation result of #1 + #3 + 1000 is used as the X value.

(Note 1) A variable cannot be defined in the same block as an address. It must be defined in a separate block.

Incorrect		Correct
X #1= #3 + 100 ;	→	#1= #3 + 100 ;
		X#1 ;

(Note 2) Up to five sets of square parentheses [ ] may be used.

#543 = – [[[[[#120]/2 + 15.]∗3 – #100]/#520 + #125 + #128]∗ #130 + #132] ;

(Note 3) There are no restrictions on the number of characters and number of variables for variable definition.

(Note 4) The variable values should be within a range from 0 to ±99999999.
If this range is exceeded, the operations may not be conducted properly.

(Note 5) The variable definitions are valid from the moment that the variables are actually defined.

#1 = 100 ; ······················ #1 = 100
#1 = 200 #2 = #1 + 200 ; ·········· #1 = 200, #2 = 400
#3 = #1 + 300 ; ·················· #3 = 500

(Note 6) Variable quotations are always regarded as having a decimal point at the end.

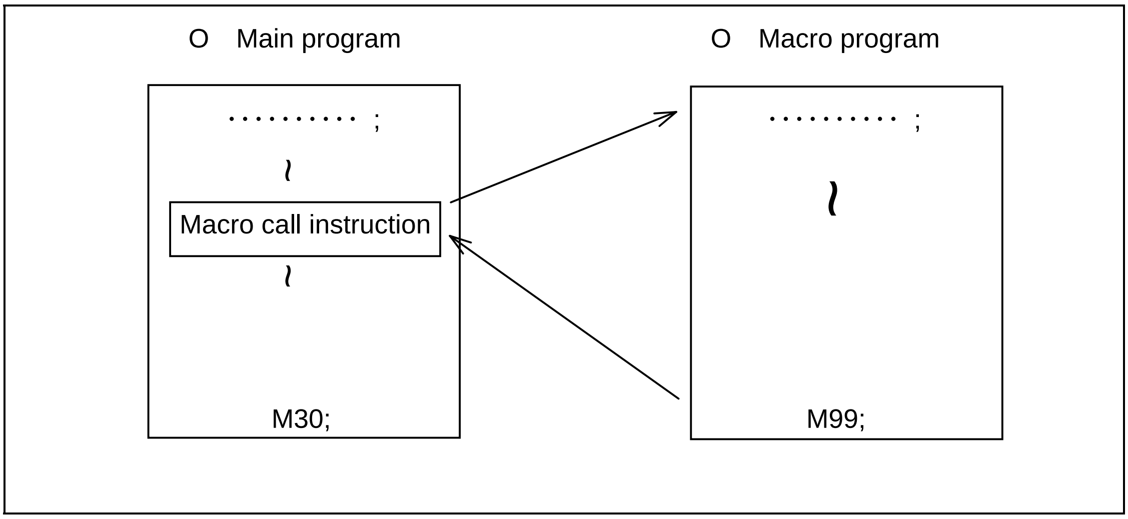
When #100 = 10,
then "X#100;" is treated as X10.

13.7 User Macro

13.7.1 User macro commands; G65, G66, G66.1, G67

Function and purpose

By combining the user macros with variable commands, it is possible to use macro program call, operation, data input/output with PLC, control, decision, branch and many other instructions for measurement and other such applications.



Macro programs use variables, operation instructions and control instructions to create subprograms which function to provide dedicated control.
These dedicated control functions (macro programs) are called by the macro call instructions exactly when required from the main program.

Detailed description

(1) When the G66 or G66.1 command is entered, the specified user macro program will be called after each block has been executed or after the movement command in the block with the movement commands has been executed until the G67 (cancel) command is entered.

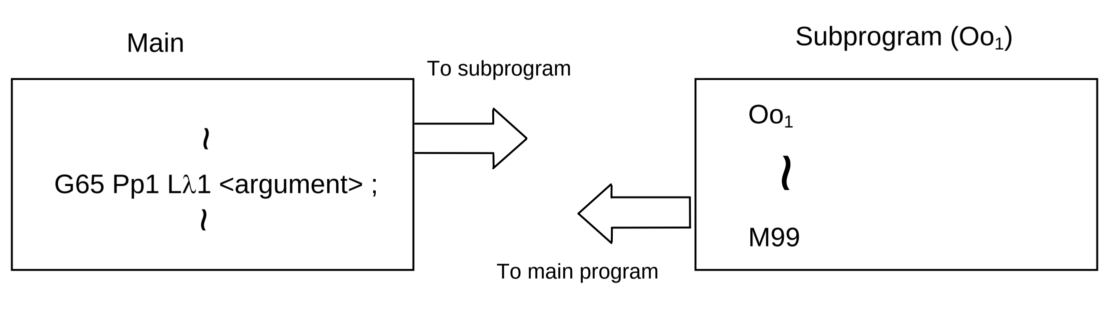
(2) The G66 (G66.1) and G67 commands must be paired in the same program.

13.7.2 Macro call instruction

Function and purpose

Included among the macro call commands are the simple calls which apply only to the instructed block and also modal calls (types A and B) which apply to each block in the call modal.

Simple macro calls



M99 is used to conclude the user macro subprogram.

Format

G65 P__ L__ <argument> ;	
P_	Program No.
L_	No. of repetitions

When the <argument> must be transferred as a local variable to a user macro subprogram, the actual value should be designated after the address.
Regardless of the address, a sign and decimal point can be used in the argument. There are 2 ways in which arguments are designated.

(1) Argument designation Ⅰ

Format : A_ B_ C_ ... X_ Y_ Z_

Detailed description

(a) Arguments can be designated using any address except G, L, N, O and P.

(b) Except for I, J and K, there is no need for designation in alphabetical order.

(c) I, J and K must be designated in alphabetical order.

I_ J_ K_ ...... Correct
J_ I_ K_ ...... Incorrect

(d) Addresses which do not need to be designated can be omitted.

(e) The following table shows the correspondence between the addresses which can be designated by argument designation Ⅰ and the variable numbers in the user macro main body.

Address and variable number correspondence		Call instructions and usable address	
Argument designation I address	Variable in macro	G65, G66	G66.1
A	#1	○	○
B	#2	○	○
C	#3	○	○
D	#7	○	○
E	#8	○	○
F	#9	○	○
G	#10	×	×*
H	#11	○	○
I	#4	○	○
J	#5	○	○
K	#6	○	○
L	#12	×	×*
M	#13	○	○
N	#14	×	×*
O	#15	×	×
P	#16	×	×*
Q	#17	○	○
R	#18	○	○
S	#19	○	○
T	#20	○	○
U	#21	○	○
V	#22	○	○
W	#23	○	○
X	#24	○	○
Y	#25	○	○
Z	#26	○	○

○ : Can be used.
× : Cannot be used.
* : Can be used while G66.1 command is modal.

(2) Argument designation II

Format : A_ B_ C_ I_ J_ K_ I_ J_ K_

Detailed description

(a) In addition to addresses A, B and C, up to 10 groups of arguments with I, J, K serving as 1 group can be designated. K10 (#33) of the tenth group, however, is ignored.

(b) When the same address is duplicated, designate the addresses in the specified order.

(c) Addresses which do not need to be designated can be omitted.

(d) The following table shows the correspondence between the addresses which can be designated by argument designation II and the variable numbers in the user macro main body.

Argument designation II address	Variable within macro	Argument designation II address	Variable within macro
A	#1	J5	#17
B	#2	K5	#18
C	#3	I6	#19
I1	#4	J6	#20
J1	#5	K6	#21
K1	#6	I7	#22
I2	#7	J7	#23
J2	#8	K7	#24
K2	#9	I8	#25
I3	#10	J8	#26
J3	#11	K8	#27
K3	#12	I9	#28
I4	#13	J9	#29
J4	#14	K9	#30
K4	#15	I10	#31
I5	#16	J10	#32

(Note 1) The numbers 1 to 10 accompanying I, J and K denote the sequence of the commanded groups and they are not required for the actual instructions.

(3) Using arguments designations I and II together

If addresses corresponding to the same variable are commanded when both types I and II are used to designate arguments, the latter address is valid.

(Example 1)

Call instruction G65 A1.1 B–2.2 D3.3 I4.4 I7.7 ;

Variable

#1 : 1.1

#2 : –2.2

#3 :

#4 : 4.4

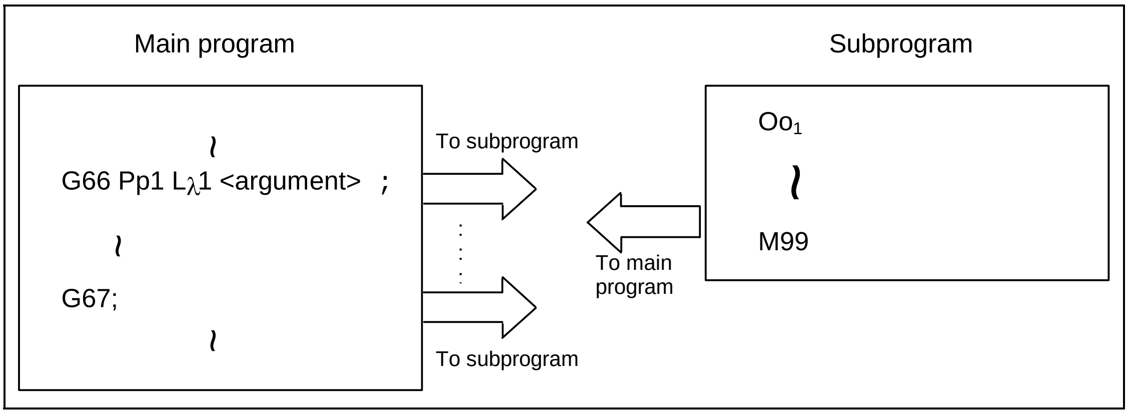
#5 :

#6 :

#7 : ~~3.3~~ 7.7

In the above example the last I7.7 argument is valid when both arguments D3.3 and I7.7 are commanded for the #7 variable.

Modal call A (movement command call)



When the block with a movement command is commanded between G66 and G67, the movement command is first executed and then the designated user macro subprogram is executed. The number of times the subprogram is executed is "λ1" times with each call.
The <argument> is the same as for a simple call.

Format

G66 P__ L__ <argument> ;	
L_	No. of repetitions
P_	Program No.

Detailed description

(1) When the G66 command is entered, the specified user macro subprogram will be called after the movement command in the block with the movement commands has been executed until the G67 (cancel) command is entered.

(2) The G66 and G67 commands must be paired in the same program.
A program error will result when G67 is issued without the G66 command.

Modal call B (for each block)

The specified user macro subprogram is called unconditionally for each command block which is assigned between G66.1 and G67 and the subprogram is executed λ1 times.
<argument> is the same as a simple call command.

Format

G66.1 P__ L__ <argument> ;	
L_	No. of repetitions
P_	Program No.

Detailed description

(1) In the G66.1 mode, everything except the O, N and G codes in the various command blocks which are read are handled as the argument without being executed.

(2) The same applies as when G65 P_ is assigned at the head of a block for all significant blocks in the G66.1 mode.

(Example 1)

N100 G01 G90 X100. Z200. F400 R1000;
in the G66.1 P1000; mode is the same as:
N100 G65 P1000 G01 G90 X100. Z200. F400 R1000;

(Note 1) The call is performed even in the G66.1 command block in the G66.1 mode and the correspondence between the argument address and the variable number is the same as for G65 (simple call).

(3) The range of the G and N command values which can be used a new as variables in the G66.1 mode is subject to the restrictions applying to value as normal NC command values.

(4) O, sequence numbers N and modal G code are updated as modal information.

G code macro call

User macro subprograms with prescribed program numbers can be called merely by issuing the G code command.

Format

G×× <argument> ;	
G××	G code for macro call

Detailed description

(1) The above instruction functions in the same way as the instructions below, and parameters are set for each G code to determine the correspondence with the instructions.

a: M98 P△△△△ ;
b: G65 P△△△△△△ <argument>;
c: G66 P△△△△△△ <argument>;
d: G66.1 P△△△△△△ <argument>;

When the parameters corresponding to c and d above are set, issue the cancel command (G67) either in the user macro or after the call code has been commanded so as to cancel the modal call.

(2) The correspondence between the "××" which conducts the macro call and the program number P△△△△ of the macro to be called is set by parameter.

(3) Up to 10 G codes from G00 to G255 can be used with this instruction. (Those G codes, such as G00, G01 and G02, which have already been clearly defined for specific applications under EIA Standards cannot be used.)

(4) The commands cannot be issued during a user macro subprogram which has been called by a G code.

Miscellaneous command macro call (for M, S, T, B code macro call)

The user macro subprograms of the specified program number can be called merely by issuing an M (or S, T, B) code. (Only entered codes apply for M but all S, T and B codes apply.)

Format

<u>Mm</u> ; (or Ss ;, Tt ;, Bb ;)	
Mm	M code for macro call (or S, T, B code)

Detailed description

(1) The above instruction functions in the same way as the instructions below, and parameters are set for each M code to determine the correspondence with the instructions. (Same for S, T and B codes)

```
a: M98     P△△△△ ;          } M98, Mm are not output
b: G65     P△△△△△  Mm ;
c: G66     P△△△△△  Mm ;
d: G66.1   P△△△△△  Mm ;
```

When the parameters corresponding to c and d above are set, issue the cancel command (G67) either in the user macro or after the call code has been commanded so as to cancel the modal call.

(2) The correspondence between the "Mm" which conducts the macro call and the program number P△△△△ of the macro to be called is set by parameter. Up to 10 M codes from M00 to M95 can be entered.
Any code except the codes basically required by the machine and the M0, M1, M2, M3 and M96 to M99 codes can be entered.

(3) As with M98, the display appears on the CRT screen of the setting display unit but the M codes and MF are not output.

(4) Even if the miscellaneous command entered above is issued during a user macro subprogram called by the M code, macro call will not result and it will be handled as an normal miscellaneous command.

(5) All S, T and B codes call the subprograms in the prescribed program numbers of the corresponding S, T and B functions.

Differences between M98 and G65 commands

(1) The argument can be designated for G65 but not for M98.

(2) The sequence number can be designated for M98 but not for G65, G66 and G66.1.

(3) M98 executes a subprogram after all the commands except M, P, H and L in the M98 block have been executed, but G65 branches to the subprogram without any further operation.

(4) When any address except O, N, P, H or L is included in the M98 block, single block stop results. This is not the case with G65.

(5) The level of the M98 local variables is fixed but it can be varied in accordance with the nesting depth for G65. (#1, for instance, has the same significance either before or after M98 but a different significance in each case with G65.)

(6) The M98 nesting depth extends up to 8 levels in combination with G65, G66 and G66.1. The G65 nesting depth extends up to only 4 levels in combination with G66 and G66.1.

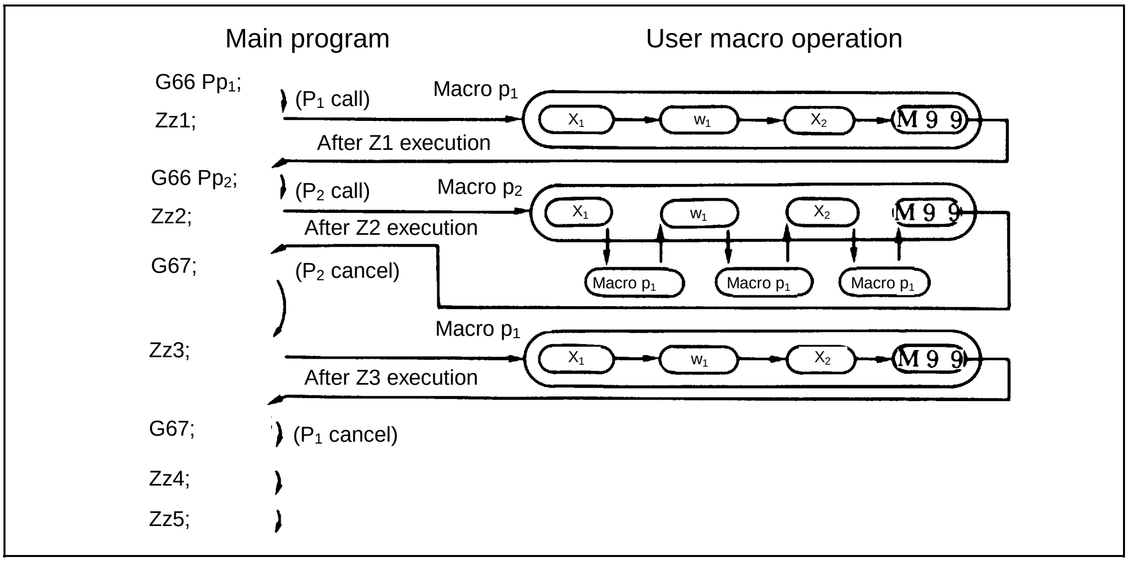
Macro call command nesting depth

Up to 4 nesting levels are available for macro subprogram calls based on simple call or modal call.
The argument with a macro call instruction is valid only on the called macro level. Since the nesting depth for macro calls extends up to 4 levels, the argument can be used as a local variable for the program with each respective macro call.

(Note 1) When a G65, G66, G66.1, G code macro call or miscellaneous command macro call is conducted, this is regarded as nesting level 1 and the level of the local variables is also incremented by one.

(Note 2) The designated user macro subprogram is called every time the movement command is executed with modal call A. However, when the G66 command has been duplicated, the next user macro subprogram is called every time an axis is moved even with movement commands in the macro.
User macro subprograms are called in sequence from the subprogram commanded last.

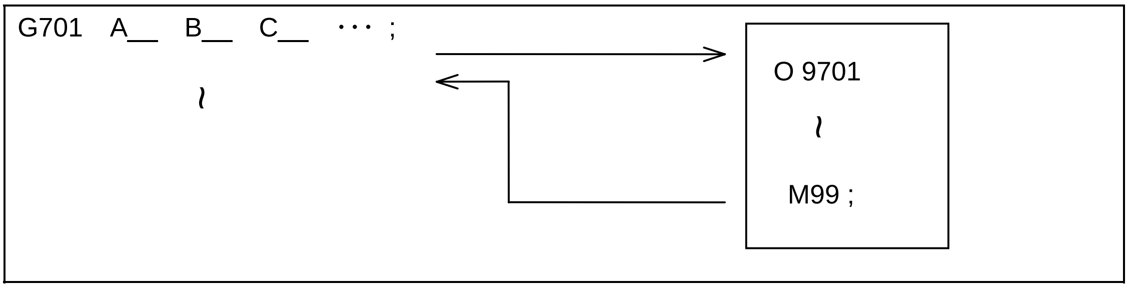
(Example 1)



13.7.3 G code for macro

Function and purpose

G200 to G999 can be used as the user macro instruction.
With this instruction, the execution of NC moves to the designated program.



Command format

G □×× ;	
□	2 to 9 value

Detailed description

(1) The macro program number corresponding to the macro instruction is determined by the last two digits after the G code and the value of the third digit of the set parameter.
It is interpreted that there are no macros registered when the parameter is set to 0, and a program error (P34) will occur.

(2) The G code macro call returns to the main program with the subprogram's M99 command.

(3) Macro restrictions such as the argument address and local variables follow the macro call type corresponding commands (M99, G65, G66, G66.1) set by the respective parameters.

13.7.4 Variables

Function and purpose

Both the variable specifications and user macro specifications are required for the variables which are used with the user macros.
The offset amounts of the local, common and system variables among the variables for this NC unit are retained even when the unit's power is turned OFF.

Multiple use of variables

When the user macro specifications apply, variable numbers can be turned into variables (multiple use of variables) or replaced by <formula>. Only one of the arithmetical (+, –, ×, ÷) operations can be conducted with <formula>.

(Example 1) Multiple use of variables

#1=10 #10=20 #20=30;	# [# [#1]]=# [#10] from #1=10.
#5=# [# [#1]];	# [#10]=#20 from #10=20.
	Therefore, #5=#20 or #5=30.

#1=10 #10=20 #20=30	# [#[#1]]=# [#10] from #1=10.
#5=1000;	# [#10]=#20 from #10=20.
# [# [#1]]=#5;	Therefore, #20=#5 or #20=1000.

(Example 2) Replacing variable numbers with <formula>

#10=5;	
# [#10+1]=1000;	In which case, #6=1000.
# [#10–1]=–1000;	In which case, #4=–1000.
# [#10∗3]=100;	In which case, #15=100.
# [#10/2]=–100;	In which case, #3=–100.

Undefined variables

Variables applying with the user macro specifications such as variables which have not been used even once after the power was turned ON or local variables not quoted by the G65, G66 or G66.1 commands can be used as <vacant>. Also, variables can forcibly be set to <vacant>. Variable #0 is always used as the <vacant> variable and nothing can be defined in the left-side member.

(1) Arithmetic expressions

#1 = #0 ; ··············· #1 = <vacant>
#2 = #0 + 1 ; ············ #2 = 1
#3 = 1 + #0 ; ············ #3 = 1
#4 = #0∗10 ; ············ #4 = 0
#5 = #0 + #0 ; ··········· #5 = 0

> It should be borne in mind that <vacant> in an arithmetic expression is handled in the same way as "0".
> <Vacant> + <Vacant> = 0
> <Vacant> + <Constant> = Constant
> <Constant> + <Vacant> = Constant

(2) Variable quotations

When undefined variable only are quoted, they are ignored up to the address.
When #1 = <Vacant>

G0 X#1 Z1000; ··············· Equivalent to G0 Z1000;
G0 X#1 + 10 Z1000; ·········· Equivalent to G0 X10 Z1000;

(3) Conditional expressions (#0 is <vacant>.)

When #101 = <Vacant>	When #101 = 0
#101 EQ #0 <Vacant> = <Vacant> established	#101 EQ #0 0 = <Vacant> not established
#101 NE 0 <Vacant> ≠ 0 established	#101 NE 0 0 ≠ 0 not established
#101 GE #0 <Vacant> ≥ 0 established	#101 GE #0 0 ≥ <Vacant> established
#101 GT 0 <Vacant> > 0 not established	#101 GT 0 0 > 0 not established
#101 LE #0 <Vacant> ≤ <Vacant> established	#101 LE #0 0 ≤ <Vacant> established
#101 LT0 <Vacant> < 0 not established	#101 LT 0 0 < 0 not established

13.7.5 Types of variables

Common variables (#100 to #149 and #500 to #549) (Example of type A)

These variables are used in common from any position.
For details, refer to the section on the variable commands.

Local variables (#1 to #32)

These can be defined as an <argument> when a macro subprogram is called or used locally within main programs and subprograms. They can be duplicated regardless of the relationship existing between macros (up to 4 levels).

G65 Pp1 Lλ1 <argument> ;	
p1	Program number
λ1	Number of repetitions

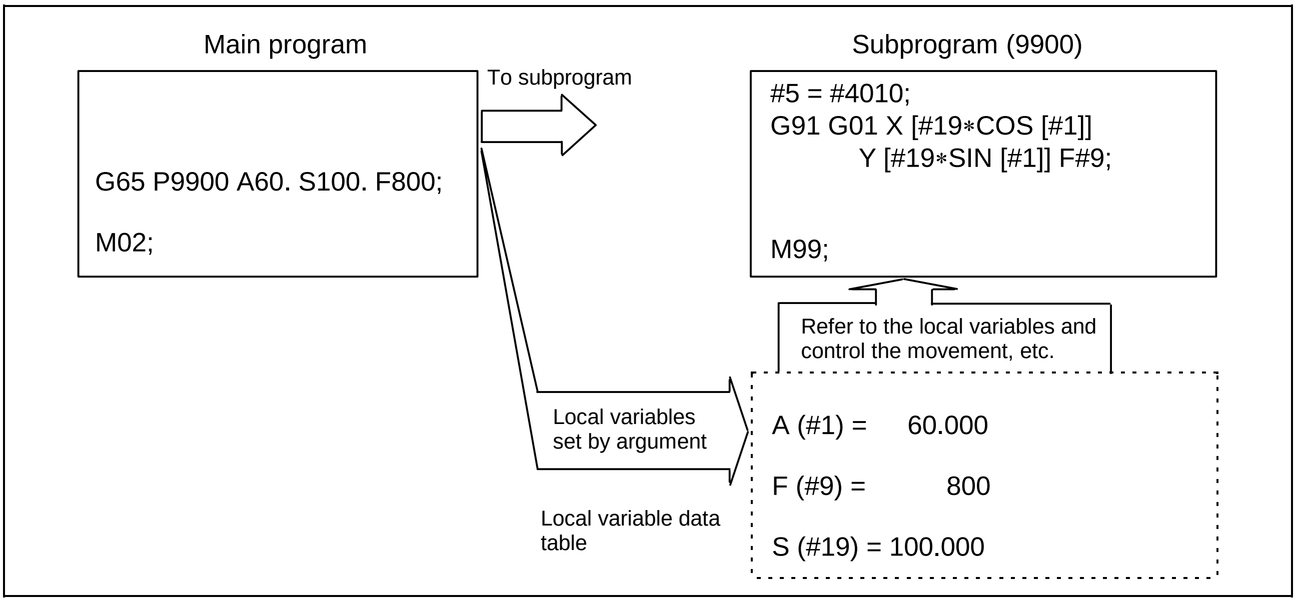
The <argument> is assumed to be Aa1 Bb1 Cc1...... Zz1.
The following table shows the correspondences between the addresses designated by <argument> and the local variable numbers used in the user macro main body.

Call command		Argument address	Local variable number
G65, G66	G66.1		
○	○	A	#1
○	○	B	#2
○	○	C	#3
○	○	D	#7
○	○	E	#8
○	○	F	#9
×	×∗	G	#10
○	○	H	#11
○	○	I	#4
○	○	J	#5
○	○	K	#6
×	×∗	L	#12
○	○	M	#13
×	×∗	N	#14
×	×	O	#15
×	×∗	P	#16

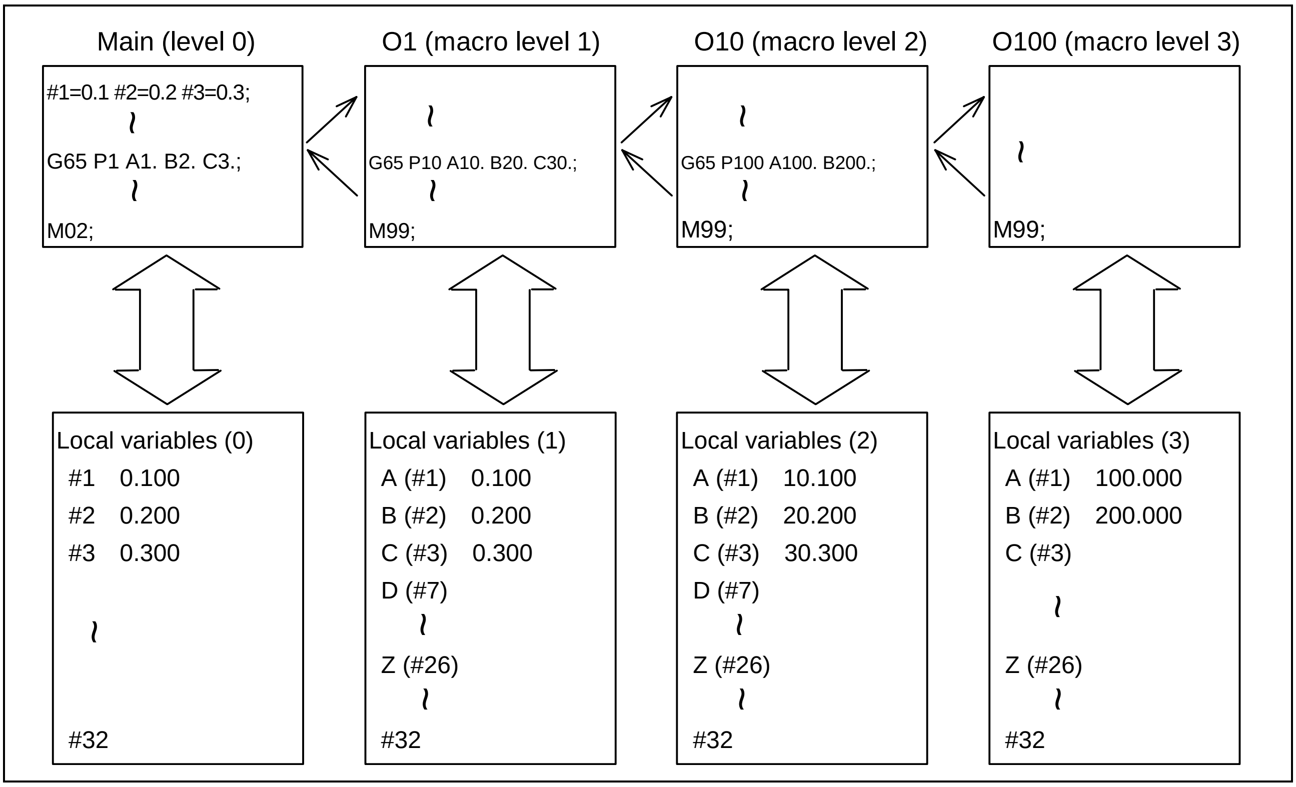
Call command		Argument address	Local variable number
G65, G66	G66.1		
○	○	Q	#17
○	○	R	#18
○	○	S	#19
○	○	T	#20
○	○	U	#21
○	○	V	#22
○	○	W	#23
○	○	X	#24
○	○	Y	#25
○	○	Z	#26
			#27
			#28
			#29
			#30
			#31
			#32

"×" in the above table denotes an argument address which cannot be used. However, provided that the G66.1 mode has been established, an argument address denoted by the asterisk can be added for use.
Note that there are no corresponding addresses for those marked with –.

(1) Local variables in subprograms can be defined by means of the <argument> designation during macro call. (Local variables can be used freely in those subprograms.)



(2) Local variables can be used independently on each of the macro calls levels (4 levels). Local variables are also provided independently for the main program (macro level 0). Arguments cannot be used for the level 0 local variables.



The statuses of the local variables appear on the setting display unit.
For details, refer to the Instruction Manual as well as to the sections covering the operation of the setting display unit and the local variables.

Macro interface inputs (#1000 to #1035)

The status of the interface input signals can be ascertained by reading out the values of variable numbers 1000 to 1035. Available value which has been read out can be only one of 2 values: 1 or 0 (1: contact closed, 0: contact open). All the input signals from #1000 to #1031 can be read at once by reading out the value of variable number 1032. Variable numbers #1000 to #1035 are for readout only, and nothing can be placed in the left side member of their operation formula.

System variable	No. of points	Interface input signal	System variable	No. of points	Interface input signal
#1000	1	Register R72 bit 0	#1016	1	Register R73 bit 0
#1001	1	Register R72 bit 1	#1017	1	Register R73 bit 1
#1002	1	Register R72 bit 2	#1018	1	Register R73 bit 2
#1003	1	Register R72 bit 3	#1019	1	Register R73 bit 3
#1004	1	Register R72 bit 4	#1020	1	Register R73 bit 4
#1005	1	Register R72 bit 5	#1021	1	Register R73 bit 5
#1006	1	Register R72 bit 6	#1022	1	Register R73 bit 6
#1007	1	Register R72 bit 7	#1023	1	Register R73 bit 7
#1008	1	Register R72 bit 8	#1024	1	Register R73 bit 8
#1009	1	Register R72 bit 9	#1025	1	Register R73 bit 9
#1010	1	Register R72 bit 10	#1026	1	Register R73 bit 10
#1011	1	Register R72 bit 11	#1027	1	Register R73 bit 11
#1012	1	Register R72 bit 12	#1028	1	Register R73 bit 12
#1013	1	Register R72 bit 13	#1029	1	Register R73 bit 13
#1014	1	Register R72 bit 14	#1030	1	Register R73 bit 14
#1015	1	Register R72 bit 15	#1031	1	Register R73 bit 15

System variable	No. of points	Interface input signal
#1032	32	Register R72, R73
#1033	32	Register R74, R75
#1034	32	Register R76, R77
#1035	32	Register R78, R79

Macro interface outputs (#1100 to #1135)

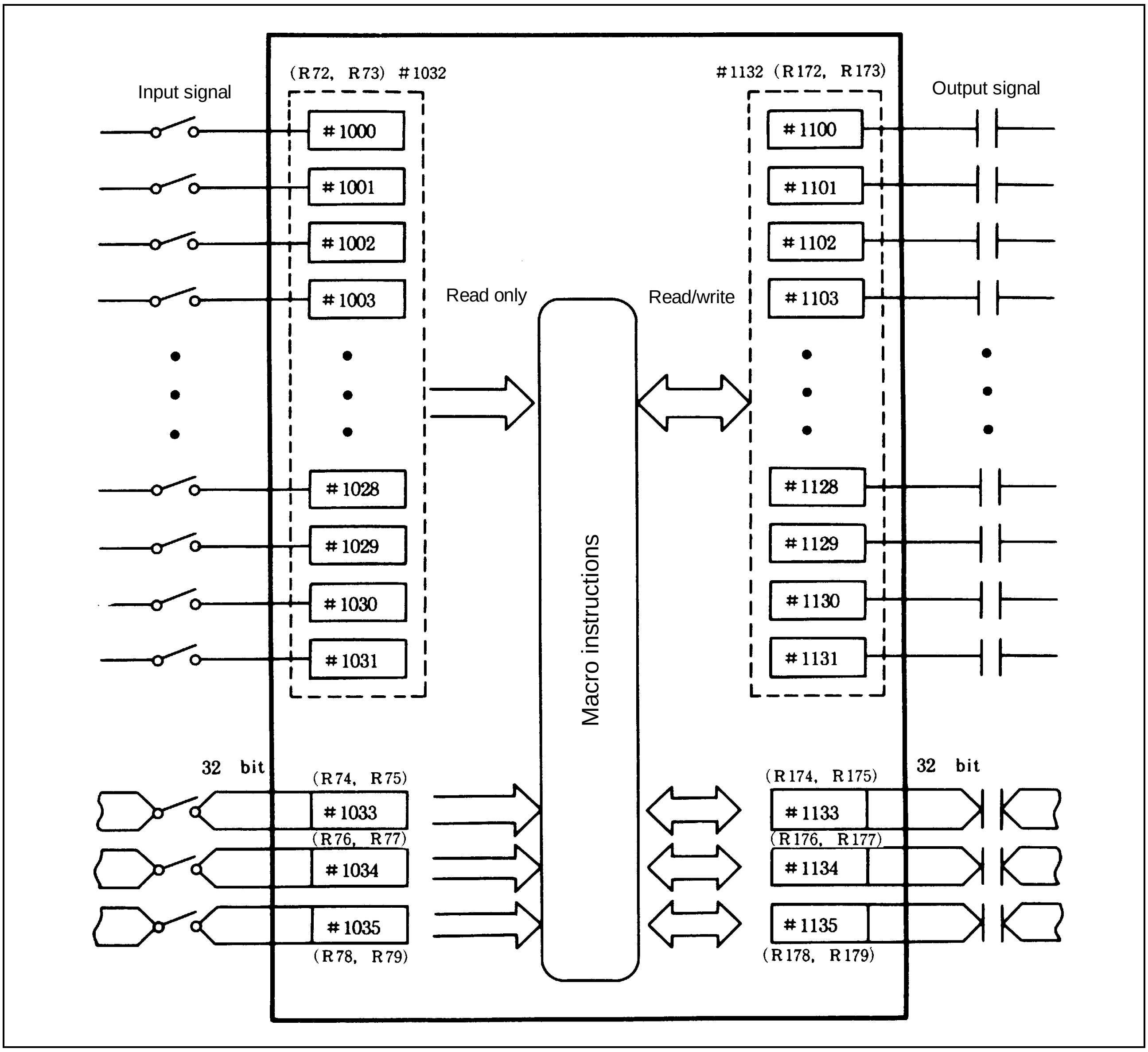
The interface output signals can be sent by substituting values in variable numbers 1100 to 1135. An output signal can be only 0 or 1.
All the output signals from #1100 to #1131 can be sent at once by substituting a value in variable number 1132. (2^0 to 2^{31})
The status of the writing and output signals can be read in order to offset the #1100 to #1135 output signals.

System variable	No. of points	Interface output signal	System variable	No. of points	Interface output signal
#1100	1	Register R172 bit 0	#1116	1	Register R173 bit 0
#1101	1	Register R172 bit 1	#1117	1	Register R173 bit 1
#1102	1	Register R172 bit 2	#1118	1	Register R173 bit 2
#1103	1	Register R172 bit 3	#1119	1	Register R173 bit 3
#1104	1	Register R172 bit 4	#1120	1	Register R173 bit 4
#1105	1	Register R172 bit 5	#1121	1	Register R173 bit 5
#1106	1	Register R172 bit 6	#1122	1	Register R173 bit 6
#1107	1	Register R172 bit 7	#1123	1	Register R173 bit 7
#1108	1	Register R172 bit 8	#1124	1	Register R173 bit 8
#1109	1	Register R172 bit 9	#1125	1	Register R173 bit 9
#1110	1	Register R172 bit 10	#1126	1	Register R173 bit 10
#1111	1	Register R172 bit 11	#1127	1	Register R173 bit 11
#1112	1	Register R172 bit 12	#1128	1	Register R173 bit 12
#1113	1	Register R172 bit 13	#1129	1	Register R173 bit 13
#1114	1	Register R172 bit 14	#1130	1	Register R173 bit 14
#1115	1	Register R172 bit 15	#1131	1	Register R173 bit 15

System variable	No. of points	Interface output signal
#1132	32	Register R172, R173
#1133	32	Register R174, R175
#1134	32	Register R176, R177
#1135	32	Register R178, R179

(Note 1) The last values of the system variables. #1100 to #1135 sent are retained as "1" or "0". (They are not cleared even with resetting.)

(Note 2) The following applies when any number except "1" or "0" is substituted into #1100 to #1131.
<Vacant> is treated as "0".
Any number except "0" and <vacant> is treated as "1".
Any value less than "0.00000001" is indefinite.


Input signal
(R72, R73) #1032
#1000
#1001
#1002
#1003
#1028
#1029
#1030
#1031
Read only
Read/write
Macro instructions
#1132 (R172, R173)
Output signal
#1100
#1101
#1102
#1103
#1128
#1129
#1130
#1131
32 bit
(R74, R75)
#1033
(R76, R77)
#1034
#1035
(R78, R79)
(R174, R175)
#1133
(R176, R177)
#1134
#1135
(R178, R179)
32 bit

Tool offset (#10001 to or #2001 to)

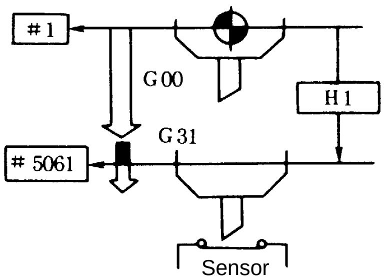
Variable number range		Contents
#10001 to #10000 + n	#2001 to #2000 + n	1st axis tool length offset amount
#11001 to #11000 + n	#2701 to #2700 + n	1st axis wear offset amount
#12001 to #12000 + n	———	3rd axis tool length offset amount
#13001 to #13000 + n	———	3rd axis wear offset amount
#14001 to #14000 + n	#2101 to #2100 + n	2nd axis tool length offset amount
#15001 to #15000 + n	#2801 to #2800 + n	2nd axis wear offset amount
#16001 to #16000 + n	#2201 to #2200 + n	Nose R compensation amount
#17001 to #17000 + n	#2901 to #2900 + n	Nose R wear compensation amount
#18001 to #18000 + n	#2301 to #2300 + n	Tool nose point

Tool data can be read and values substituted using the variable numbers.
Either the numbers in the #10000 order or #2000 order can be used.
The last 3 digits of the variable numbers correspond to the tool offset number.
The tool offset data are configured as data with a decimal point in the same way as for other variables. Consequently, this decimal point must be commanded when data below the decimal point is to be entered.

Programming example		Common variables	Tool offset data
#101 = 1000; #10001 = #101; #102 = #10001;	After execution →	#101 = 1000.0 #102 = 1000.0	H = 1000.000

(Example 1) Calculation and tool offset data setting

G28 X0 T0101;	Zero point return
M06;	Tool change (T0101)
#1 = #5001;	Start point storage
G00 X–200.;	Rapid traverse to safe position
G31 X–50. F100;	Skip measuring
#10001 = #5061 – #1;	Measurement distance calculation and tool offset data setting
≀	



(Note) In this example no consideration is given to the delay in the skip sensor signal. #5001 is the X-axis start point position and #5061 is the X-axis skip coordinates, and indicated is the position at which the skip signal is input while G31 is being executed.

Workpiece coordinate system offset (#5201 to #5323)

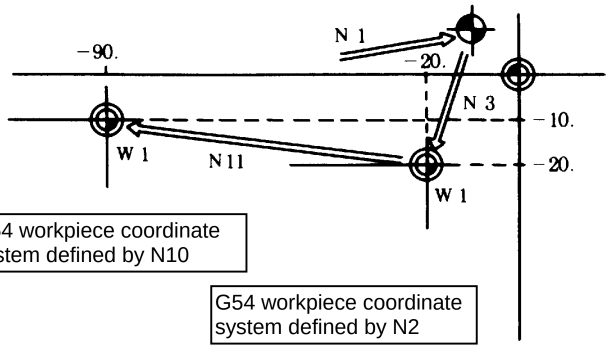
By using variable numbers 5201 to 5323, it is possible to read out the workpiece coordinate system offset data or to substitute values.

(Note) The number of axes which can be controlled differs according to the NC specifications. The last digit in the variable number corresponds to the control axis number.

Axis No. / Coordinate name	1st axis	2nd axis	3rd axis	Remarks
External workpiece offset	#5201	#5202	#5203	External workpiece offset specifications are required.
G54	#5221	#5222	#5223	Workpiece coordinate system offset specifications are required.
G55	#5241	#5242	#5243	
G56	#5261	#5262	#5263	
G57	#5281	#5282	#5283	
G58	#5301	#5302	#5303	
G59	#5321	#5322	#5323	

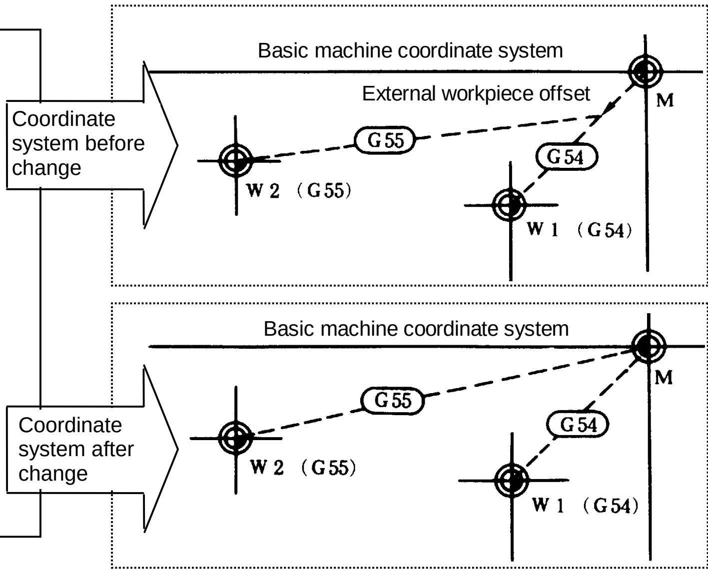
(Example 1)

```
N1 G28 X0Z0;
N2 #5221 = –20. #5223 = –20.;
N3 G00 G54 X0Z0;
    ~
N10 #5221 = –10. #5223 = –90.;
N11 G00 G54 X0Z0;
    ~
M02;
```



(Example 2)

```
        ~

N100  #5221 = #5221 + #5201;
      #5222 = #5222 + #5202;
      #5241 = #5241 + #5201;
      #5242 = #5242 + #5202;
      #5201 = 0    #5202 = 0;

        ~
```



This is an example where the external workpiece offset values are added to the workpiece coordinate (G54, G55) system offset values without changing the position of the workpiece coordinate systems.

NC alarm (#3000)

The NC unit can be forcibly set to the alarm state by using variable number 3000.

Format

#3000 = 70 (CALL #PROGRAMMER #TEL #530);	
70	Alarm number
CALL #PROGRAMMER #TEL #530	Alarm message

Any alarm number from 1 to 9999 can be specified.
The alarm message must be less than 31 characters long.
The "P277 : Macro alarm message" appears in the <NC alarm> column on diagnosis page 1 while the alarm number and alarm message 70: (CALL #PROGRAMMER #TEL #530) is indicated in the <operator massage>.

Example of program (alarm when #1 = 0)

```
~
1F[#1NE0]GOTO100;
#3000=70 (CALL #PROGRAMMER #TEL #530);

N100
~
```

Stops with NC alarm

<NC ALARM> DIAG 1.

P277 : Macro alarm message

<Operator message>

CALL #PROGERAMMER #TEL #530 70

(Note 1) Alarm number "0" is not displayed and any number exceeding "9999" cannot be indicated.

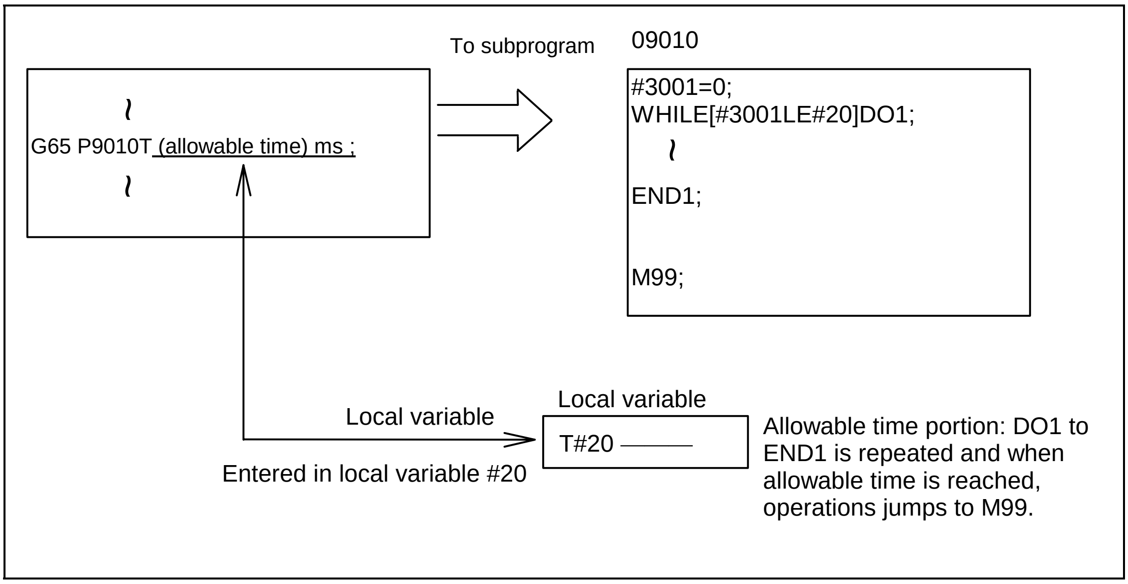
(Note 2) The characters "@" and "~" cannot be used in the alarm message. If the "@" or "~" character is used in the alarm message, a program error "P34" will occur.

Integrating time (#3001, #3002)

The integrating time can be read when the power is turned ON or during automatic operation or automatic start or values can be substituted by using variable numbers 3001 and 3002.

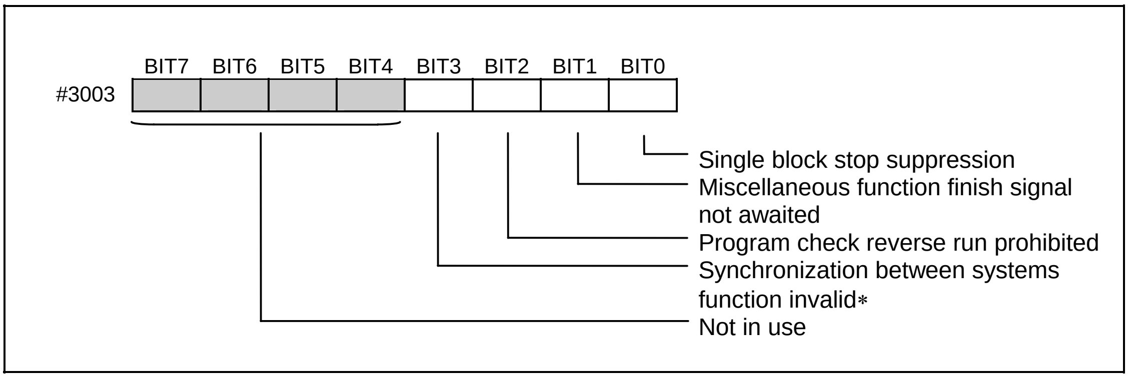
Type	Variable number	Unit	Contents when power is turned ON	Initialization of contents	Count conditions
Power ON	3001	1ms	Same as when power is turned OFF	Value substituted for variable	At all times while power is ON
Automatic start	3002				In-automatic start

The integrating time returns to "0" in about 2.44 ×10ms (approximately 7.7 years).



Suppression of single block stop and miscellaneous function finish signal waiting (#3003)

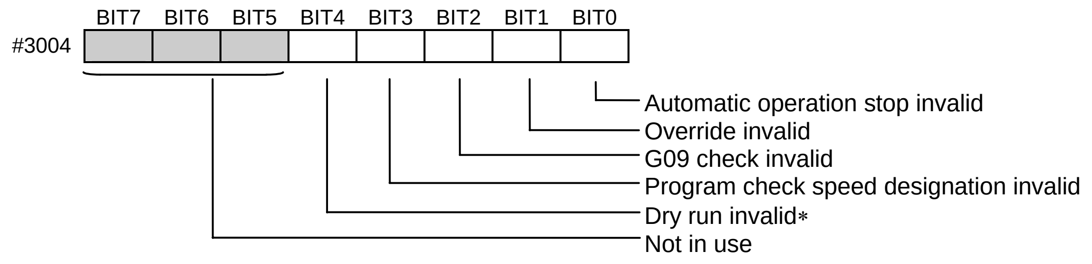
By substituting the values below in variable number 3003, it is possible to suppress single block stop in the subsequent blocks or to advance to the next block without waiting for the miscellaneous function (M, S, T, B) finish signal (FIN).



(Note 1) #3003 is cleared to zero by resetting.

Feed hold, feedrate override, G09 valid/invalid (#3004)

By substituting the values below in variable number 3004, it is possible to make the feed hold, feedrate override and G09 functions either valid or invalid in the subsequent blocks.



(Note 1) #3004 is cleared to zero by resetting.

(Note 2) The functions are valid when the above bits are "0" and invalid when they are "1".

Message display & stop (#3006)

By using variable number 3006, the message display is stopped after the previous block has been executed and, if the message display data have been commanded, then the corresponding message will be indicated.

Format

#3006=1 (TAKE FIVE);	
TAKE FIVE	Message

(Note 1) The message should not be longer than 31 characters and it should be enclosed within round parentheses.

(Note 2) #3006=1 is fixed to "1".

(Note 3) The characters "@" and "~" cannot be used in the message. If the "@" or "~" character is used in the message, a program error "P34" will occur.

Mirror image (#3007)

By reading variable number 3007, it is possible to ascertain the status of mirror image at a particular point in time for each axis.
The axes correspond to the bits of #3007.
Contents of each bit is indicated as follows.

{ 0: Mirror image invalid
1: Mirror image valid }

Bit	15	14	13	12	11	10	9	8	7	6	5	4	3	2	1	0
nth axis														3	2	1

G command medals (#4001 to #4021 and #4201 to #4221)

Using variable numbers 4001 to 4021, it is possible to read the modal commands which have been issued up to the block immediately before.
Similarly, it is possible to read the medals in the block being executed with variable numbers #4201 to #4221.

Variable number		Function
Pre-read block	**Execution block**	
#4001	#4201	Interpolation mode: G00:0, G01:1, G02:2, G03:3, G33:33
#4002	#4202	Plane selection: G17:17, G18:18, G19:19
#4003	#4203	Absolute/incremental: G90:90, G91:91
#4004	#4204	Barrier check: G22:22, G23:23
#4005	#4205	Feed designation: G94:94, G95:95
#4006	#4206	Inch/metric: G20:20, G21:21
#4007	#4207	Tool nose radius compensation: G40:40, G41:41, G42:42, G46:46
#4008	#4208	Second spindle control: G43:43, G44:44
#4009	#4209	Fixed cycle: G80:80, G73 to 79:73 to 79, G81 to G89:81 to 89
#4010	#4210	Return level: G98:98, G99:99
#4011	#4211	
#4012	#4212	Workpiece coordinate system: G54 to G59:54 to 59
#4013	#4213	Acceleration/deceleration:G61 to G62:61 to 62, G64 to 64
#4014	#4214	Macro modal call: G66:66, G66.1:66.1, G67:67
#4015	#4215	Facing turret mirror image: G68:68, G69:69
#4016	#4216	
#4017	#4217	Constant surface speed control: G96:96, G97:97
#4018	#4218	Balance cut: G14:14, G15:15
#4019	#4219	Milling: G12.1:12.1, G13.1:13.1
#4020	#4220	
#4021	#4221	

(Example)

```
G28 X0 Z0;
G00 X150. Z200;
G65 P300 G02 W–30. K–15. F1000;
M02;

O300
#1=#4001; → Group 01 G modal (pre-read) #1 = 2.0
#2=#4201; → Group 01 G modal (now being executed) #2 = 0.0
G#1W#24;
M99;
%
```

Other modals (#4101 to #4120 and #4301 to #4320)

Using variable numbers 4101 to 4120, it is possible to read the modal commands assigned up to the block immediately before.
Similarly, it is possible to read the medals in the block being executed with variable numbers #4301 to #4320.

Variable number		Modal information
Pre-read	Execution	
#4101	#4301	
#4102	#4302	
#4103	#4303	
#4104	#4304	
#4105	#4305	
#4106	#4306	
#4107	#4307	Tool wear offset No.
#4108	#4308	
#4109	#4309	Feedrate F (asynchronous)
#4110	#4310	

Variable number		Modal information
Pre-read	Execution	
#4111	#4311	
#4112	#4312	
#4113	#4313	Miscellaneous function M
#4114	#4314	Sequence number N
#4115	#4315	Program number O
#4116	#4316	
#4117	#4317	
#4118	#4318	
#4119	#4319	Spindle function S
#4120	#4320	Tool function T

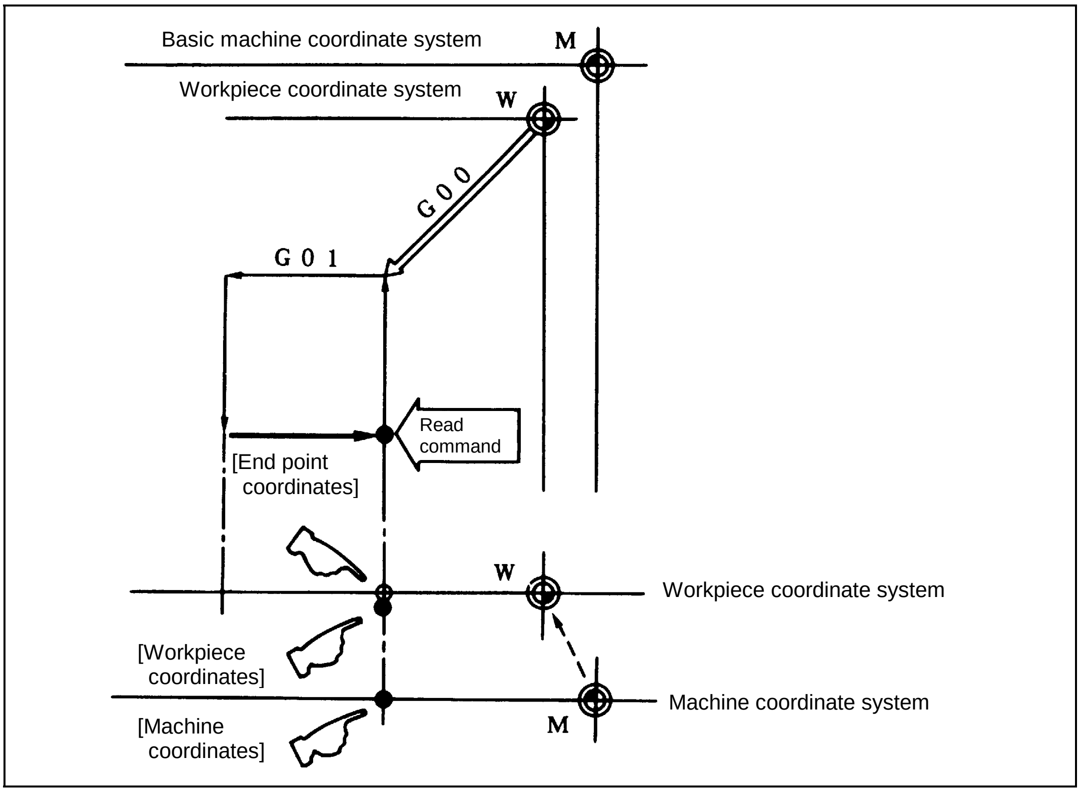
Position information (#5001 to #5103)

The end point coordinate system, machine coordinate value, workpiece coordinate value, skip coordinate value, tool position offset amount and servo deviation amount in the previous block can be read using the variable numbers 5001 to 5103.

Position information / Axis No.	End point coordinate of block immediately before (program)	Machine coordinate	Workpiece coordinate	Skip coordinate (workpiece coordinate)	Tool position offset amount	Servo deviation amount
1	#5001	#5021	#5041	#5061	#5081	#5101
2	#5002	#5022	#5042	#5062	#5082	#5102
3	#5003	#5023	#5043	#5063	#5083	#5103
Remarks (reading during movement)	Yes	No	No	Yes	No	Yes

(Note) The number of axes which can be controlled differs according to the NC specifications.
The last digit of the variable number corresponds to the control axis number.

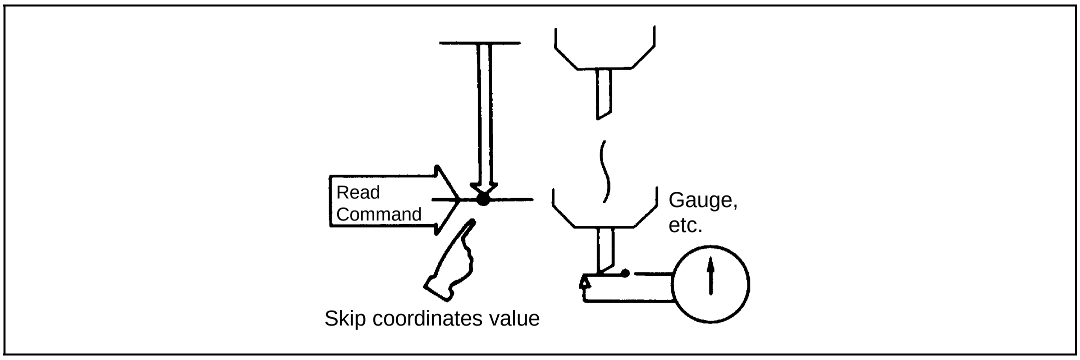
Variable number	Position information
#5121	Approach axis collision position
#5122	Sampling designated axis collision position



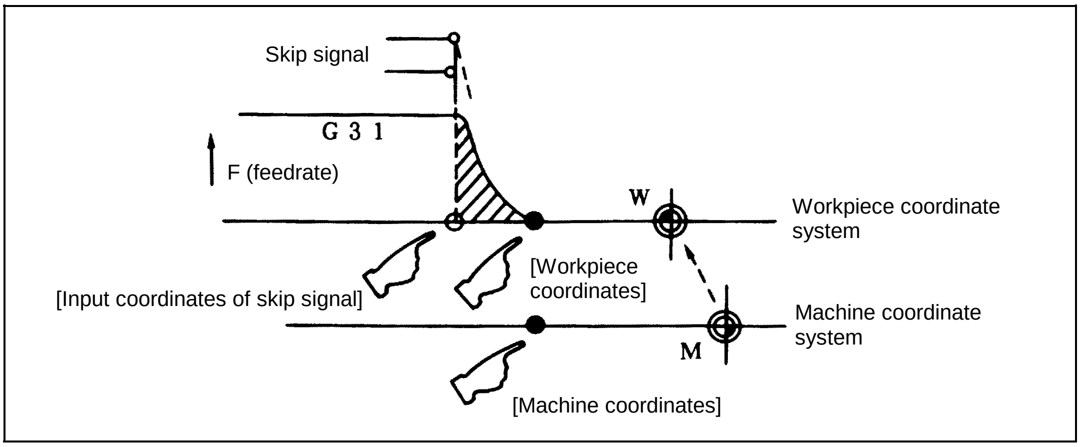
(1) The positions of the end point coordinates and skip coordinates are positions in the workpiece coordinate system.

(2) The end point coordinates, skip coordinates and servo deviation amounts can be read even during movement. However, it must first be checked that movement has stopped before reading the machine coordinates and workpiece coordinates.

(3) The position where the skip signal is turned ON in the G31 block is indicated for the skip coordinates. The end point position is indicated when the skip signal has not been turned ON. (For further details, refer to the section on tool length measurement.)



(4) The tool nose position where the tool offset and other such factors are not considered is indicated as the end point position. The tool reference point position with consideration given to tool offset is indicated for the machine coordinates, workpiece coordinates and skip coordinates.



For "●", check stop and then proceed to read.
For "○", reading is possible during movement.

The position of the skip signal input coordinate value is the position in the workpiece coordinate system. The coordinate value in variable numbers #5061 to #5063 memorize the moments when the skip input signal during movement was input and so they can be read at any subsequent time.
For further details, reference should be made to the section on the "skip" function.

Variable name setting and quotation

Any name (variable name) can be given to common variables #500 to #519. It must be composed of not more than 7 alphanumerics and it must begin with an alphabetic character.

Format

SETVNn[NAME1,NAME2,·····];	
n	Head number of variable which assigns name
NAME1	#n name (variable name)
NAME2	#n + 1 name (variable name)

Variable names are separated by a comma ",".

Detailed description

(1) Once variable names have been set, they will not be cleared even when the power is turned OFF.

(2) Variables in programs can be quoted by their variable names. In cases like this, the variables should be enclosed in square parentheses.

(Example 1) G01X[#POINT1];

(3) The variable numbers, data and variable names appear on the CRT screen of the setting display unit.

(Example 2)

Program... SETVN500[A234567, DIST, TOOL25];

```
[Common variables]

        #500      –12345.678      A234567
        #501        5670.000      DIST
        #502        –156.500      TOOL25
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~
        #518          10.000      NUMBER
    Common variable #(502)DATA(–156.5) NAME (TOOL25)
```

(Note) Do not include character strings (SIN, COS, etc.) used for operation commands, etc. in the variable name.

13.7.6 Operation commands

A variety of operations can be performed between variables.

Command format

#i=<formula>;

<Formula> is a combination of constants, variables, functions and operators.
Constants can be used instead of #j and #k below.

(1) Definition and substitution of variables	#i = #j	Definition, substitution
(2) Addition operation	#i = #j + #k	Addition
	#i = #j – #k	Subtraction
	#i = #j OR #k	Logical sum (at every bit of 32 bits)
	#i = #j XOR #k	Exclusive OR (at every bit of 32 bits)
(3) Multiplication operation	#i = #j ∗ #k	Multiplication
	#i = #j / #k	Division
	#i = #j MOD #k	Remainder
	#i = #j AND #k	Logical product (at every bit of 32 bits)
(4) Functions	#i = SIN [#k]	Sine
	#i = COS [#k]	Cosine
	#i = TAN [#k]	Tangent (sinθ/cosθ used for tanθ)
	#i = ATAN [#k]	Arctangent (ATAN or ATN may be used)
	#i = ACOS [#k]	Arccosine
	#i = SQRT [#k]	Square root (SQRT or SQR may be used)
	#i = ABS [#k]	Absolute value
	#i = BIN [#k]	Conversion from BCD to BINARY
	#i = BCD [#k]	Conversion from BINARY to BCD
	#i = ROUND [#k]	Rounding off (ROUND or RND may be used)
	#i = FIX [#k]	Discard fractions less than 1
	#i = FUP [#k]	Add for fractions less than 1
	#i = LN [#k]	Natural logarithm
	#i = EXP [#k]	Exponent with e (=2.718...) as bottom

(Note 1) A value without a decimal point is basically treats as a value with a decimal point at the end (1 = 1.000).

(Note 2) Offset amounts from #10001 and workpiece coordinate system offset values from #5201 are handled as data with a decimal] point. Consequently, data with a decimal point will be produced even when data without a decimal point have been defined in the variable numbers.

(Example)

Program	Common variables after execution
#101 = 1000 ; #10001 = #101 ; #102 = #10001;	→ #101 1000.000 #102 1000.000

(Note 3) The <formula> after a function must be enclosed in the square parentheses.

Sequence of operations

(1) The sequence of the operations (1) to (3) is, respectively, the functions followed by the multiplication operation followed in turn by the addition operation.

#101=#111+#112∗SIN[#113]

(1) Function
(2) Multiplication operation
(3) Addition operation

(2) The part to be given priority in the operation sequence should be enclosed in brackets. Up to 5 pairs of such brackets including those for the functions may be used.

#101=SQRT [[[#111 – #112] ∗ SIN [#113] + [#114] ∗ #115] ;

First pair of brackets
Second pair of brackets
Third pair of brackets

Examples of operation commands

(1) Main program and argument designation	G65P100A10B20.; #101 = 100.000 #102 = 200.000;	#1 10.000 #2 20.000 #101 100.000 #102 200.000
(2) Definition and substitution	#1 = 1000 #2 = 1000. #3 = #101 #4 = #102 #5 = #5041	#1 1000.000 #2 1000.000 #3 100.000 } From common variables #4 200.000 #5 –10.000 } From offset amount
(3) Addition and subtraction (+, –)	#11 = #1+1000 #12 = #2–50. #13 = #101+#1 #14 = #5041–3. #15 = #5041+#102	#11 2000.000 #12 950.000 #13 1100.000 #14 –13.000 #15 190.000
(4) Logical sum (OR)	#3 = 100 #4 = #3OR14	#3 = 01100100 14 = 00001110 #4 = 01101110 = 110
(5) Exclusive OR (XOR)	#3 = 100 #4 = #3XOR14	#3 = 01100100 14 = 00001110 #4 = 01101010 = 106
(6) Multiplication and division (∗, /)	#21 = 100∗100 #22 = 100.∗100 #23 = 100∗100 #24 = 100.∗100. #25 = 100/100 #26 = 100./100. #27 = 100/100. #28 = 100./100. #29 = 5041∗#101 #30 = 5041/#102	#21 10000.000 #22 10000.000 #23 10000.000 #24 10000.000 #25 1.000 #26 1.000 #27 1.000 #28 1.000 #29 –1000.000 #30 –0.050

	Remainder (MOD)	#31 = #19MOD#20	#19/#20 = 48/9 = 5 with 3 over
(7)	Logical product (AND)	#9 = 100 #10 = #9AND15	#9 = 01100100 15 = 00001111 #10 = 00000100 = 4
(8)	Sin (SIN)	#501 = SIN [60] #502 = SIN [60.] #503 = 1000∗SIN [60] #504 = 1000∗SIN [60.] #505 = 1000.∗SIN [60] #506 = 1000.∗SIN [60.] **(Note)** SIN [60] is equivalent to SIN [60.]	#501 0.860 #502 0.860 #503 866.025 #504 866.025 #505 866.025 #506 866.025
(9)	Cosine (COS)	#541 = COS [45] #542 = COS [45.] #543 = 1000∗COS [45] #544 = 1000∗COS [45.] #545 = 1000.∗COS [45] #546 = 1000.∗COS [45.] **(Note)** COS [45] is equivalent to COS [45.]	#541 0.707 #542 0.707 #543 707.107 #544 707.107 #545 707.107 #546 707.107
(10)	Tangent (TAN)	#551 = TAN [60] #552 = TAN [60.] #553 = 1000∗TAN [60] #554 = 1000∗TAN [60.] #555 = 1000.∗TAN [60] #556 = 1000.∗TAN [60.] **(Note)** TAN [60] is equivalent to TAN [60.]	#551 1.732 #552 1.732 #553 1732.051 #554 1732.051 #555 1732.051 #556 1732.051
(11)	Arctangent (ATAN or ATN)	#561 = ATAN [173205/100000] #562 = ATAN [173205/100.] #563 = ATAN [173.205/100000] #564 = ATAN [173.205/100.] #565 = ATAN [1.732]	#561 60.000 #562 60.000 #563 60.000 #564 60.000 #565 59.999
(12)	Arccosine (ACOS)	#521 = ACOS [100./141.421] #522 = ACOS [100./141.421] #523 = ACOS [1000./1414.213] #524 = ACOS [10./14.142] #525 = ACOS [0.707]	#521 45.000 #522 45.000 #523 45.000 #524 44.999 #525 45.009
(13)	Square root (SQR or SQRT)	#571 = SQRT [1000] #572 = SQRT [1000.] #573 = SQRT [10.∗10.+20.∗20.] #574 = SQRT [#14∗#14 + #15∗#15] **(Note)** In order to increase the accuracy, proceed with the operation inside parentheses.	#571 31.623 #572 31.623 #573 22.361 #574 190.444
(14)	Absolute value (ABS)	#576 = –1000 #577 = ABS [#576] #3 = 70. #4 = –50. #580 = ABS [#4 – #3]	#576 –1000.000 #577 1000.000 #580 120.000
(15)	BIN, BCD	#1 = 100 #11 = BIN [#1] #12 = BCD [#1]	 #11 64 #12 256

(16)	Rounding off (ROUND or RND)	#21 = ROUND [14/3] #22 = ROUND [14./3] #23 = ROUND [14/3.] #24 = ROUND [14./3.] #25 = ROUND [–14/3] #26 = ROUND [–14./3] #27 = ROUND [–14/3.] #28 = ROUND [–14./3.]	#21 5 #22 5 #23 5 #24 5 #25 –5 #26 –5 #27 –5 #28 –5
(17)	Discarding fractions below decimal point (FIX)	#21 = FIX [14/3] #22 = FIX [14./3] #23 = FIX [14/3.] #24 = FIX [14./3.] #25 = FIX [–14/3] #26 = FIX [–14./3] #27 = FIX [–14/3.] #28 = FIX [–14./3.]	#21 4.000 #22 4.000 #23 4.000 #24 4.000 #25 –4.000 #26 –4.000 #27 –4.000 #28 –4.000
(18)	Adding fractions less than 1 (FUP)	#21 = FUP [14/3] #22 = FUP [14./3] #23 = FUP [14/3.] #24 = FUP [14./3.] #25 = FUP [–14/3] #26 = FUP [–14./3] #27 = FUP [–14/3.] #28 = FUP [–14./3.]	#21 5.000 #22 5.000 #23 5.000 #24 5.000 #25 –5.000 #26 –5.000 #27 –5.000 #28 –5.000
(19)	Natural logarithms (LN)	#101 = LN [5] #102 = LN [0.5] #103 = LN [–5]	#101 1.609 #102 –0.693 Error "P282"
(20)	Exponents (EXP)	#104 = EXP [2] #105 = EXP [1] #106 = EXP [–2]	#104 7.389 #105 2.718 #106 0.135

Operation accuracy

As shown in the following table errors will be generated when performing operations once and these errors will be accumulated by repeating the operations.

Operation format	Average error	Maximum error	Type of error
a = b + c a = b – c	2.33×10^{-10}	5.32×10^{-10}	Min. $\left\lvert \frac{\varepsilon}{b} \right\rvert , \left\lvert \frac{\varepsilon}{c} \right\rvert$
a = b∗c	1.55×10^{-10}	4.66×10^{-10}	Relative error $\left\lvert \frac{\varepsilon}{a} \right\rvert$
a = b/c	4.66×10^{-10}	1.86×10^{-9}	
a = b	1.24×10^{-9}	3.73×10^{-9}	
a = SIN [b] a = COS [b]	5.0×10^{-9}	1.0×10^{-8}	Absolute error $\lvert \varepsilon \rvert ^{\circ}$
a = ATAN [b/c]	1.8×10^{-6}	3.6×10^{-6}	

(Note 1) SIN/COS is calculated for the function TAN.

Notes on reduced accuracy

(1) Addition and subtraction

It should be noted that when absolute values are used subtractively in addition or subtraction, the relative error cannot be kept below 10^{-8}.
For instance, it is assumed that the real value produced as the operation calculation result of #10 and #20 are as follows (these value cannot be substituted directly):
#10 = 2345678988888.888
#20 = 2345678901234.567
Performing #10 – #20 will not produce #10 – #20 = 87654.321.
There are 8 decimal digits in the variables and so the values of #10 and #20 will be as follows (strictly speaking, the internal values will differ somewhat from the value below because they are binary numbers):
#10 = 2345679000000.000
#20 = 2345678900000.000
Consequently, #10 – #20 = 100000.000 will generate a large error.

(2) Logical relation

EQ, NE, GT, LT, GE and LE are basically the same as addition and subtraction and so care should be taken with errors. For instance, to determine whether or not #10 and #20 are equal in the above example:
IF[#10EQ#20]
It is not always possible to provide proper evaluation because of the above-mentioned error.
Therefore when the error is evaluated as in the following expression:
IF [ABS [#10–#20]LT200000]
and the difference between #10 and #20 falls within the designated range error, both values should be considered equal.

(3) Trigonometric functions

Absolute errors are guaranteed with trigonometric functions but since the relative error is not under 10^{-8}, care should be taken when dividing or multiplying after having used a trigonometric function.

13.7.7 Control commands

The flow of programs can be controlled by IF ~ GOTO ~ and WHILE ~ DO ~.

Branching

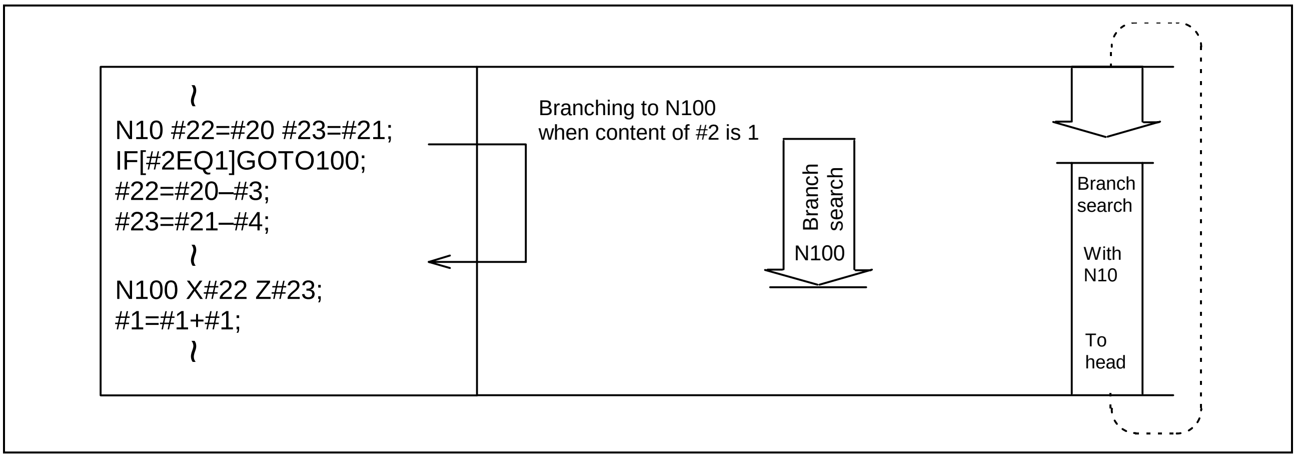
Format

IF [conditional expression] GOTO n ; (where n = sequence number in the program)

When the condition is satisfied, control branches to "n" and when it is not satisfied, the next block is executed.
IF [conditional expression] can be omitted and, when it is, control passes to "n" unconditionally.
The following types of [conditional expressions] are available.

#i EQ #j	= When #i and #j are equal
#i NE #j	≠ When #i and #j are not equal
#i GT #j	> When #i is greater than #j
#i LT #j	< When #i is less than #j
#i GE #j	≥ When #i is #j or more
#i LE #j	≤ When #i is #j or less

"n" of "GOTO n" must always be in the same program. Alarm "P231" will result if it is not. A formula or variable can be used instead of #i, #j and "n".
In the block with sequence number "n" which will be executed after a "GOTO n" command, the sequence number Nn must always be at the head of the block. Otherwise, alarm "P231" will result.
If "/" is at the head of the block and Nn follows, control can be branched to the sequence number.



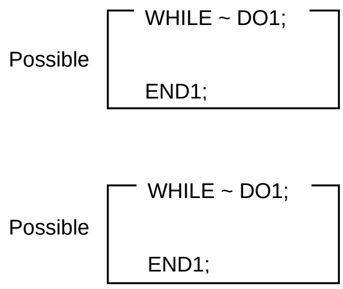
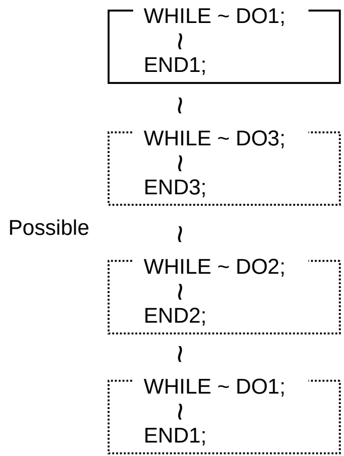
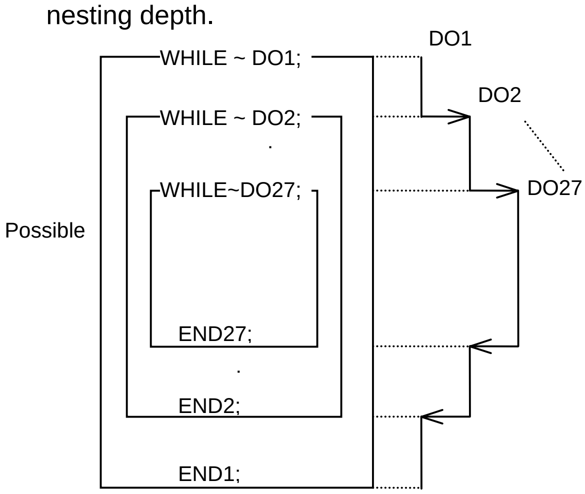
(Note) When the sequence number of the branch destination is searched, the search is conducted up to the end of the program (% code) from the block following "IF..... ;" and if it is not found, it is conducted up to the block before "IF ..... ;". Therefore, branch searches in the opposite direction to the program flow will take longer to execute compared with branch searches in the forward direction.

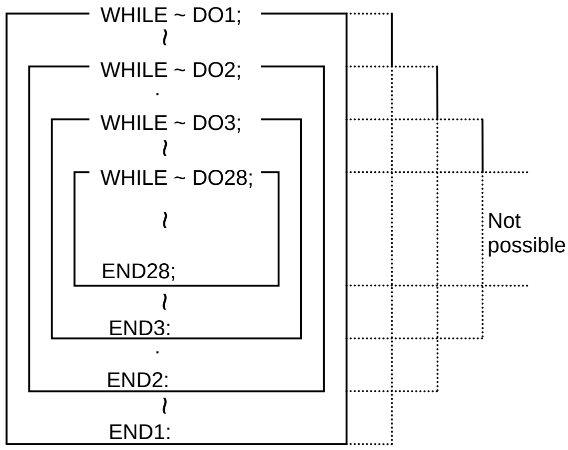
Repetition

Format

```
WHILE [conditional expression] DOm ; (m =1, 2, 3 ..... 127)
    ~
END m ;
```

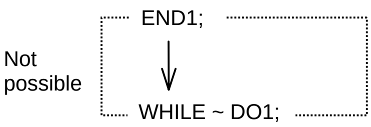
While the conditional expression is established, the blocks from the following block to ENDm are repeatedly executed; when it is not established, execution moves to the block after ENDm. DOm may come before WHILE.
"WHILE [conditional expression] DOm" and "ENDm" must be used as a pair. If "WHILE [conditional expression]" is omitted, these blocks will be repeatedly in infinitum. The repeating identification numbers range from 1 to 127 (DO1, DO2, DO3, ... DO127). Up to 27 nesting levels can be used.

<table>
<tr>
<td>(1) Same identification number can be used any number of times.

</td>
<td>(2) Any number may be used for the WHILE~ DOm identification number.

</td>
</tr>
<tr>
<td>(3) Up to 27 nesting levels for WHILE~Dom. "m" is any number form 1 to 127 for the nesting depth.


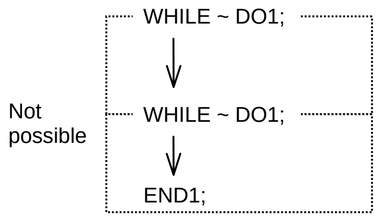
(Note) With nesting, "m" which has been used once cannot be used.</td>
<td>(4) The number of WHILE~DOm nesting levels cannot exceed 27.

</td>
</tr>
</table>

(Note) When the fixed cycle is used during the WHILE~DOm interval, the WHILE~DOm nesting levels will be less than 27.

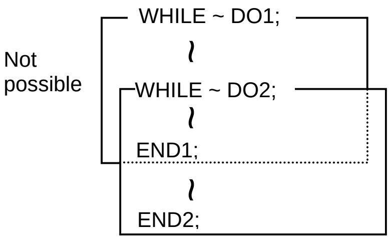
(5) WHILE ~ DOm must be designated first and ENDm last.



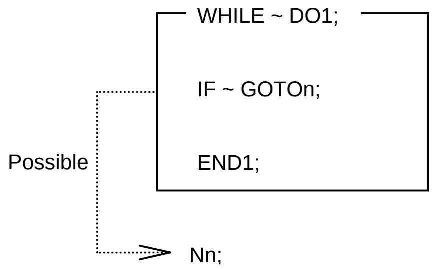
(6) WHILE ~ DOm and ENDm must correspond on a 1:1 (pairing) basis in the same program.



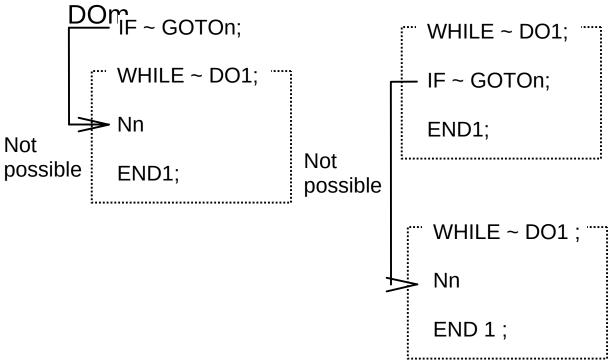
(7) Two WHILE ~ DOm's must not overlap.



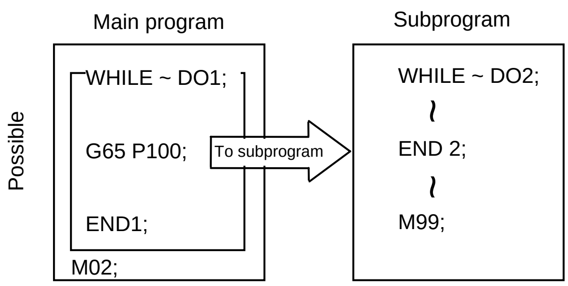
(8) Branching externally is possible from the WHILE ~ DOm range.



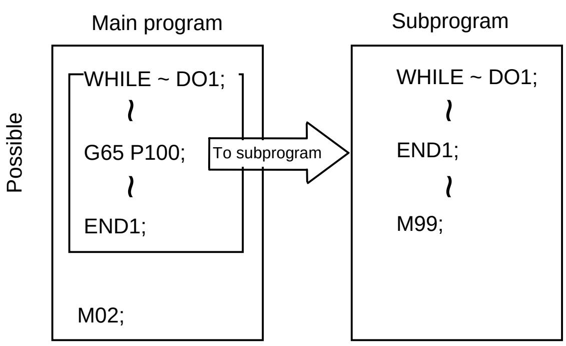
(9) No branching is possible inside WHILE ~ DOm.



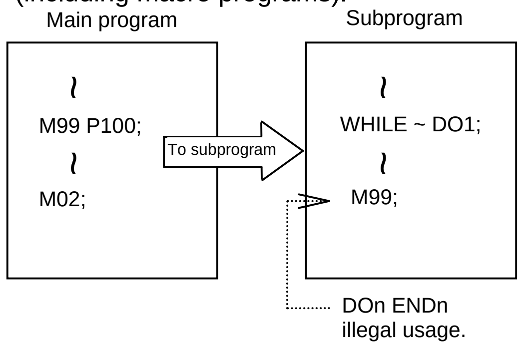
(10) Subprograms can be called by M98, G65 or G66 between WHILE ~ DOm's.



(11) Calls can be initiated by G65 or G66 between WHILE ~ DOm's and commands can be issued again from 1. Up to 27 nesting levels are possible for the main program and subprograms.



(12) A program error results with M99 unless WHILE and END are paired within the subprograms (including macro programs).



13.7.8 Precautions

Precautions

(1) When the user macro commands are employed, it is possible to use the M, S, T and other NC control commands together with the operation, decision, branching and other macro commands for creating the machining programs. When the former commands are made into NC executable statements and the latter commands into macro statements, the macro statement processing should be accomplished as quickly as possible in order to minimize the machining time, because such processing is not directly related to machine control.
As a result, the parameter (control parameter "Macro single") can be decided upon and the macro statements can be processed in parallel with the execution of the NC executable statement.
(The parameter can be set OFF during normal machining to process all the macro statements together or set ON during a program check to execute the macro statements block by block. This enables the setting to be made in accordance with the intended objective in mind.)

Example of program

```
G91 G28 X0 Z0;

G92 X0 Z0;
G00 X–100. Z–100.;
#101=100.∗COS [210,];      ← Macro statement
#103=100.∗SIN [210.];
G01 X#101 Z#103 F800;
```

Macro statements are:

(a) Operation commands (blocks including "=")

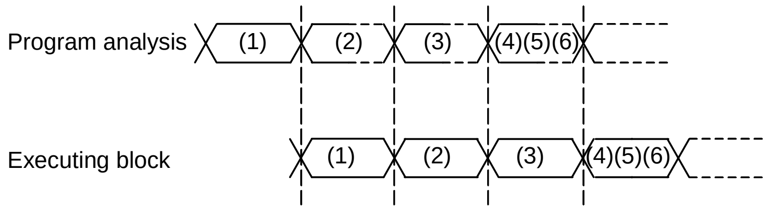
(b) Control commands (blocks including GOTO, DO~END, etc.)

(c) Macro call commands (including macro calls based on G codes and cancel command (G65, G66, G66.1, G67))

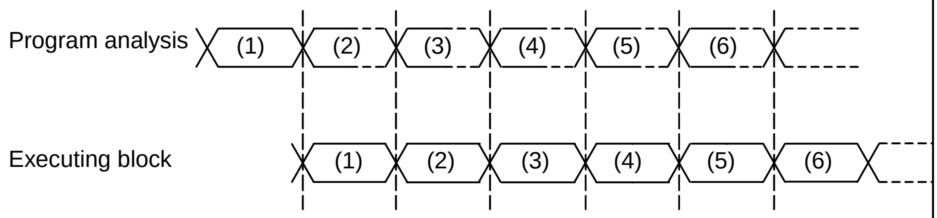
NC statements are all those statements which are not macro statements.

Flow of processing

Macro single OFF



Macro single ON



Machining program display

Macro single OFF	[Executing] N3 G00 X–100. Z–100.; [Next command] N6 G01 X#101 Z#103 F800;	N4, N5 and N6 are processed in parallel with the control of the NC executable statement of N3. N6 is an NC executable statement and so it is displayed as the next command. If the N4, N5 and N6 analysis is in time during N3 control, the machine movement will be continuously controlled.
Macro single ON	[Executing] N3 G00 X–100. Z–100.; [Next command] N4 #101=100.∗COS[210.];	N4 is processed in parallel with the control of the NC executable statement of N3, and it is displayed as the next command. N5 and N6 are analyzed and N6 is executed after N3 has finished, and so the machine control is held on standby during the N5 and N6 analysis time.

13.8 Double-Turret Mirror Image; G68, G69

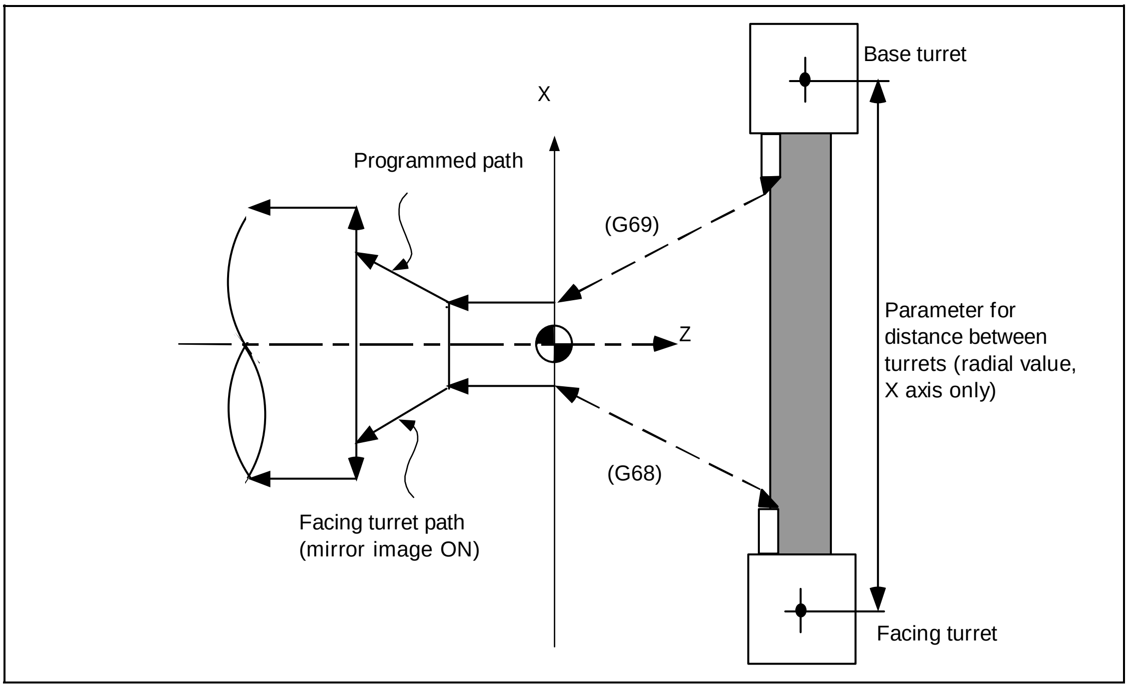
Function and purpose

With machines in which the base turret and facing turret are integrated, this function enables the workpiece to be cut by the tools on the facing turret using the programs prepared at the base turret side. (See figure below).
The distance between the two turrets is set beforehand with the parameter.

Command format

G68;	
G69;	
G68	Double-turret mirror image ON
G69	Double-turret mirror image Cancel

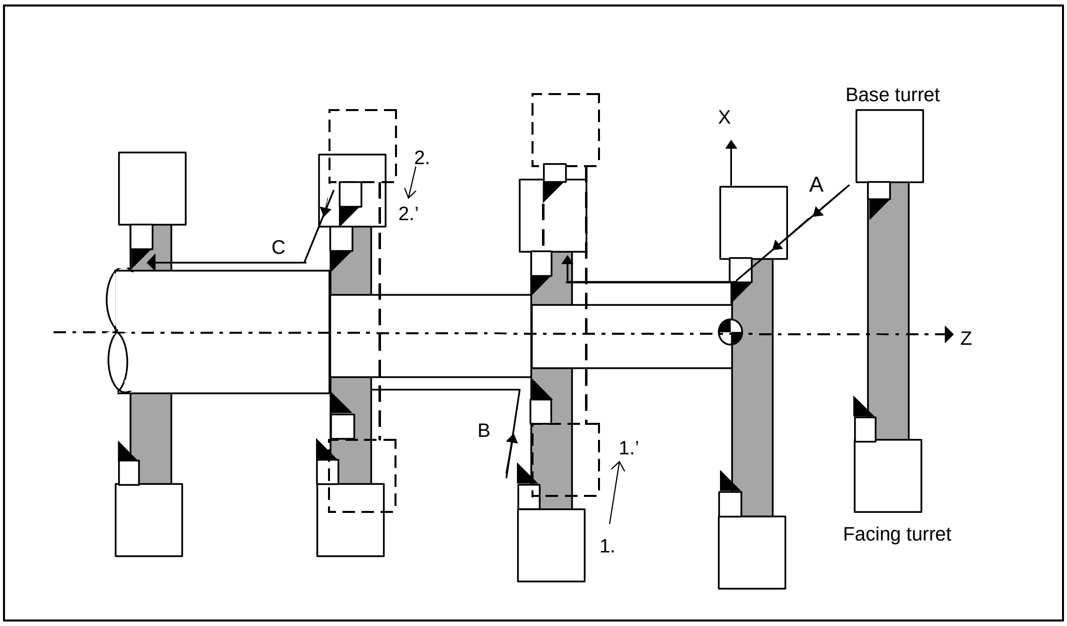
When the G68 command is issued, the subsequent programmed coordinate systems are shifted to the facing turret side and the movement direction of the X axis is made the opposite of that commanded by the program. When the G69 command is issued, the subsequent programmed coordinate systems are returned to the base turret side.



Program and example of operation

(1) Example of operation by absolute value command

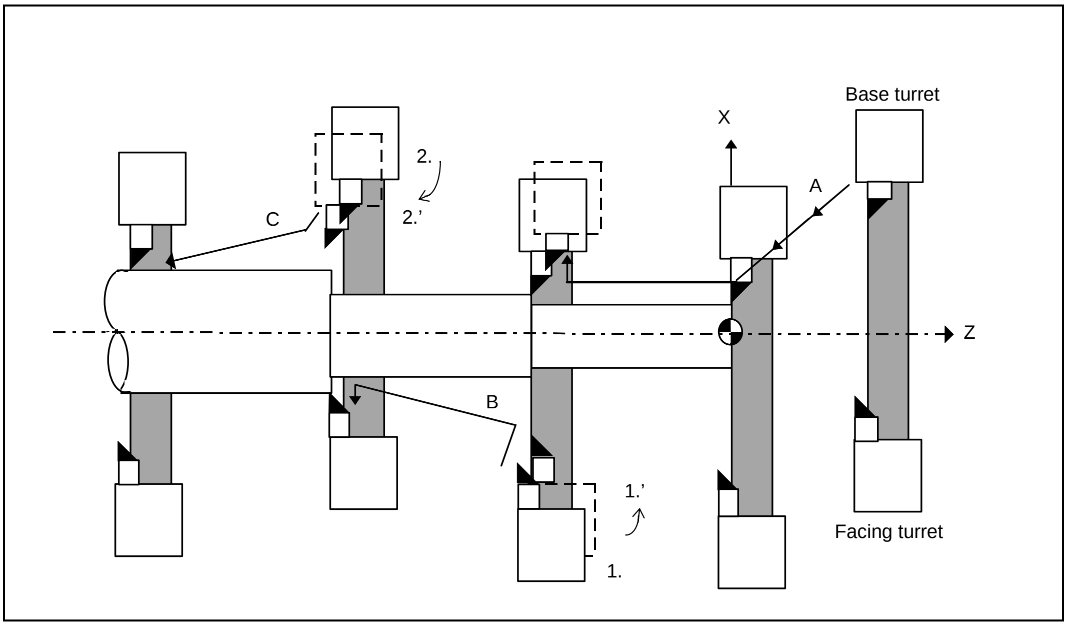
T0101; G00 X10. Z0.; G01 Z-40. F400; X20.;	Selection of base turret	Machining by base turret ··········· A
G68; T0202; G00 X20. Z-40.; G01 X20. Z-80. F200; X30.;	Double-turret mirror image ON Selection of facing turret ······· [1]	Machining by facing turret ··········· B
G69; T0101; G00 X30. Z-80.; G01 X30. Z-120. F400;	Double-turret mirror image cancel Selection of base turret ······· [2]	Machining by base turret ··········· C



A value determined from the turret interval parameter is added for movement by the first X-axis command issued after the double-turret mirror image is turned ON. In the above operation example, program [1] block causes movement 1 to 1'.
Similarly, a value determined from the turret interval parameter is subtracted for movement by the first X-axis command issued after the double-turret mirror image is canceled. In the above operation example, program [2] block causes movement from 2 to 2'.

(2) Example of operation by incremental value command

T0101; G00 X0. Z0.; G01 Z-40. F400; X20.;	Selection of base turret	Machining by base turret ············ A
G68; T0202; G00 U-10. W0.; G01 X20. Z-80. F200; X30.;	Double-turret mirror image ON Selection of facing turret ······· [1]’	Machining by facing turret ············ B
G69; T0101; G00 U-10. W0.; G01 X30. Z-120. F400;	Double-turret mirror image cancel Selection of base turret ······· [2]’	Machining by base turret ············ C

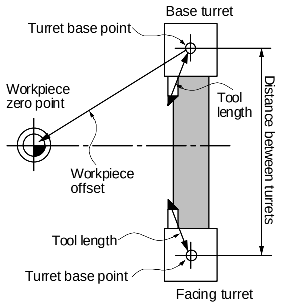
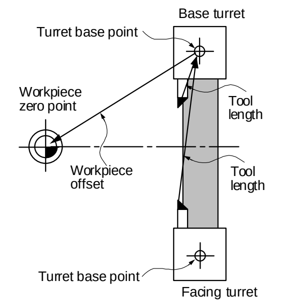
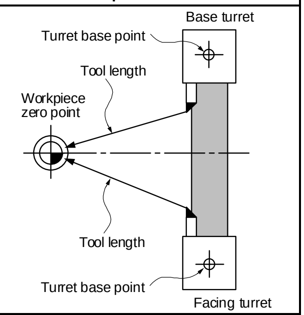


A incremental value command issued after the double-turret mirror image is turned ON causes movement by the amount of the X-axis movement in the opposite direction specified by the program command. Block [1]’ made by changing program [1] in "(1) Example of operation by absolute value command" to an incremental command causes the tool to move +10. by reversing –10. In the above operation example, the tool moves from 1 to 1’.
The same applies to an incremental value command issued after the double-turret mirror image is canceled. Block [2]’ made by changing program [2] in "(1) Example of operation by absolute value command" to an incremental command causes the tool to move –10. In the above operation example, the tool moves from 2 to 2’.

Turret offset amount

(1) Tool length offset amount

The tool length is the distance from the tool nose to the tool length base point. This definition is the same for the facing turret. However, the offset amount setting differs according to the position of the tool length base point, as shown below. (The settings are approached from the types in the table below.)

	Type A	Type B	Type C
Workpiece zero point	Workpiece face center	Workpiece face center	Workpiece face center
Tool length base point	Turret base points	Base turret base point	Workpiece face center
Distance between turrets	Distance between base points of both turrets (radius value)	0	0
Workpiece offset	Workpiece zero point – tool length base point of base turret	Workpiece zero point – tool length base point of base turret	0
Tool length	Tool length base point – tool nose position	Tool length base point – tool nose position	Tool length base point – tool nose position
Outline drawings			

The above outline drawings apply when the machine parameter (base specifications parameter) "mirrA" is 0. If it is 1, the value is set with the tool of the facing turret in the same direction as the tool of the base turret. At this time, the tool length base point is the tool length base point of the base turret.
When "mirrA" is 1, the sign for the X-axis tool length wear amount for the facing tool is reversed and the tool nose point is set to the opposite side (for instance, 2 → 3). When tool length measurement is executed with "mirrA" as 0, the data will be accepted with "mirrA" as 0.

(Setting examples)

	mirrA=0		mirrA=1	
	X	Z	X	Z
Workpiece offset	–120.	–110.	–120.	–110.
Tool length of base turret	80.	35.	80.	35.
Tool wear amount of base turret	–20.	–5.	–20.	–5.
Tool nose point of base turret	3		3	
Tool length of facing turret	150.	40.	120.	40.
Tool wear amount of facing turret	–20.	–5.	–20.	–5.
Tool nose point of facing turret	2		3	
Distance between turret	0		0	

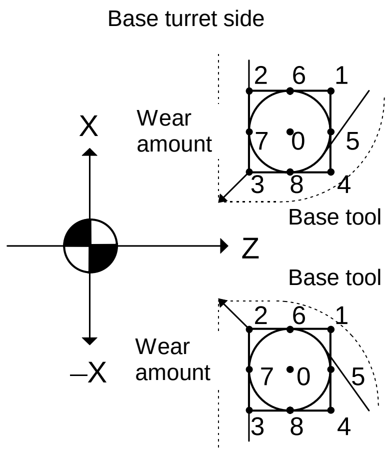
(2) Tool length wear offset amount

The tool length wear offset amount is the distance from the current tool nose to the original tool nose. The original tool nose is that value applying when the tool length offset amount was set.

(3) Tool nose point with nose R compensation

The tool nose point with tool nose R compensation is as follows. It is common for both the base and facing turrets.

Tool wear offset amount and nose R tool nose point



(4) Distance between turrets

The distance between the turrets is the distance from the tool length base point of the facing turret to the tool length base point of the base turret. It is set by parameter only for the X axis.
"0" is set when the tool length base point is common.
Machine parameter (Base specifications parameter) mirofs
Setting range: 0 to 99999.999 (mm) (radial value)

13.9 Corner Chamfering, Corner Rounding Function Ⅰ

Chamfering at any angle or corner rounding is performed automatically by adding ",C_" or ",R_" to the end of the block to be commanded first among those command blocks which shape the corner with lines only.

13.9.1 Corner chamfering (,C_)

Function and purpose

The corner is chamfered in such a way that the positions produced by subtracting the lengths commanded by ",C_" from the hypothetical starting and final corners which would apply if no chamfering were to be performed are connected.

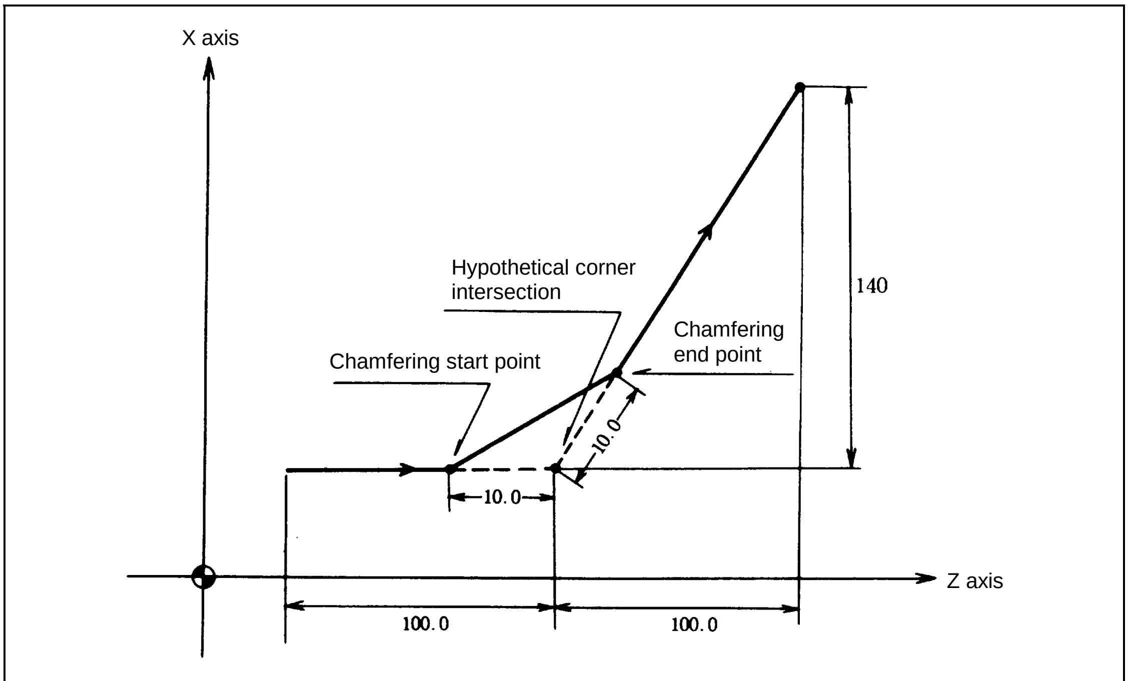
Command format

N100 G01 X__ Z__ ,C__ ; **N200 G01 X__ Z__ ;**	
C__	Length up to chamfering starting point or end point from hypothetical corner

Chamfering is performed at the point where N100 and N200 intersect.

Example of program

G01 W100. ,C10. F100;
U280. W100.;



Detailed description

(1) The start point of the block following the corner chamfering serves as the hypothetical corner intersection.

(2) When the comma in ",C" is not present, it is handled as a C command.

(3) When both "C_" and "R_" are commanded in the same block, the latter command is valid.

(4) Tool offset is calculated for the shape which has already been subjected to corner chamfering.

(5) Program error "P382" results when the block following the corner chamfering block does not contain a linear command.

(6) Program error "P383" results when the movement amount in the corner chamfering block is less than the chamfering amount.

(7) Program error "P384" results when the movement amount in the block following the corner chamfering block is less than the chamfering amount.

(8) When corner chamfering I is commanded, 2 blocks are pre-read for calculating the intersection.

13.9.2 Corner rounding (,R_)

Function and purpose

The imaginary corner, which would exist if the corner were not to be rounded, is rounded with the arc having the radius which is commanded by ",R_" only when configured of linear lines.

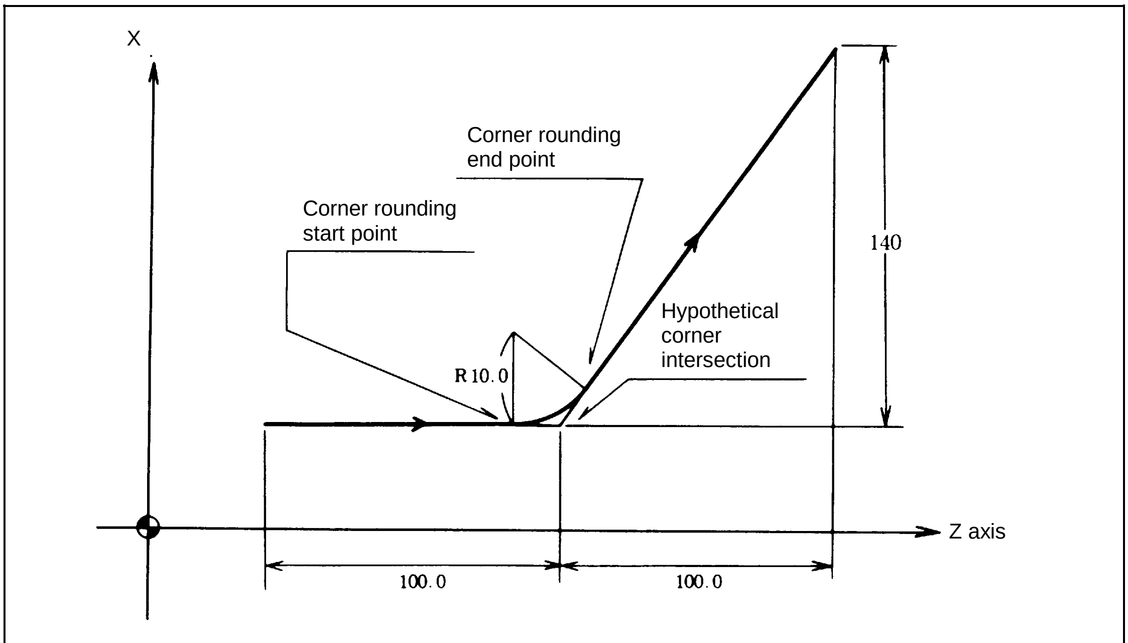
Command format

N100 G01 X__ Z__ ,R__ ; **N200 G01 X__ Z__ ;**	
R__	Circular radius of corner rounding

Rounding is performed at the point where N100 and N200 intersect.

Example of program

G01 W100. ,R10. F100;
U280. W100.;



Detailed description

(1) The start point of the block following the corner rounding serves as the hypothetical corner intersection.

(2) When the comma in ",R" is not present, it is handled as an R command.

(3) When both "C_" and "R_" are commanded in the same block, the latter command is valid.

(4) Tool offset is calculated for the shape which has already been subjected to corner rounding.

(5) Program error "P382" results when the block following the corner rounding block does not contain a linear command.

(6) Program error "P383" results when the movement amount in the corner rounding block is less than the R value.

(7) Program error "P384" results when the movement amount in the block following the corner rounding block is less than the R value.

(8) When corner rounding I is commanded, 2 blocks are pre-read for calculating the contact.

13.10 Corner Chamfering, Corner Rounding Function II

This function enables chamfering and corner rounding by adding ",C_" or ",R_" after the first commanded block among the command blocks which configure the corner by an circular or line with any continuous angle. Chamfering and corner rounding can be conducted with both absolute and incremental commands.

13.10.1 Corner chamfering (,C_)

Function and purpose

Corner chamfering is conducted by issuing the ",C_" command in the first block among 2 blocks including a continuous arc. In the case of a circular, the length of the chord applies.

Command format

N100 G03 X__ Z__ I__ K__ ,C_ ;	
N200 G01 X__ Z__ ;	
C__	Length from hypothetical corner to start or end point of chamfering

Chamfering is performed at the intersection between N100 and N200,

Example of program

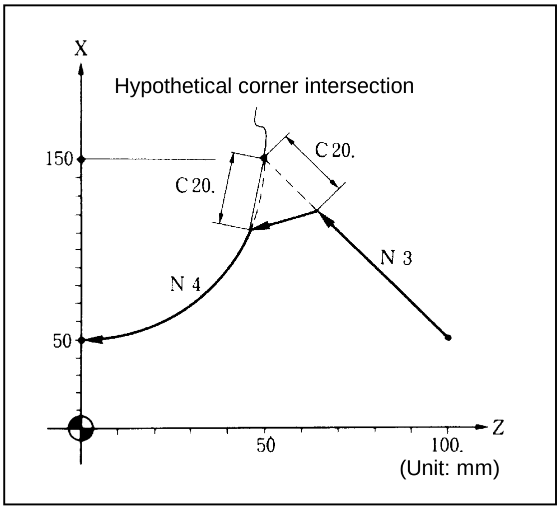
(1) Linear – circular

Absolute value commands

N1 G28 XZ;
N2 G00 X50. Z100.;
N3 G01 X150. Z50. ,C20. F100;
N4 G02 X50. Z0 I0 K–50.;
:

Incremental value commands

N1 G28 XZ;
N2 G00 U25. W100.;
N3 G01 U50. W–50. ,C20. F100;
N4 G02 U–50. W–50. I0 K–50.;
:



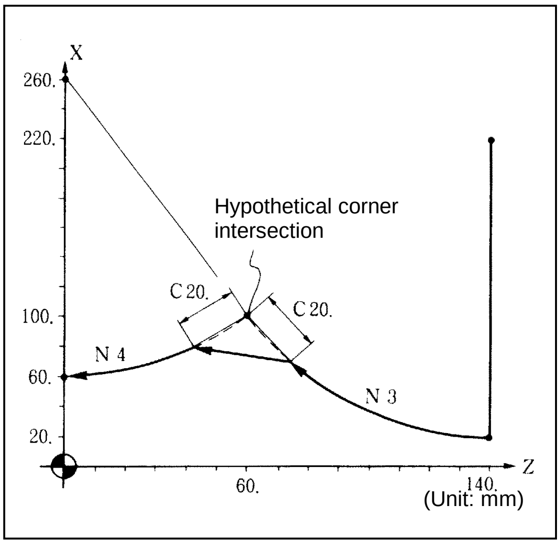
(2) Circular – circular

Absolute value commands

N1 G28 X Z;
N2 G00 X20. Z140.;
N3 G02 X100. Z60. I100. K0. ,C20. F100;
N4 X60. Z0 I80. K–60.;
:

Incremental value commands

N1 G28 X Z;
N2 G00 U10. W140.;
N3 G02 U40. W–80. R100. ,C20. F100;
N4 U–20. W–60. I80. K–60.;
:



Detailed description

(1) Both the corner chamfering and corner rounding I and II options are required in order to use this function.
Program error "P381" results when the function is assigned without an option.

(2) The start point of the block following the corner chamfering serves as the hypothetical corner intersection.

(3) When the comma in ",C" is not present, it is handled as a C command.

(4) When both "C_" and "R_" are commanded in the same block, the latter command is valid.

(5) Tool offset is calculated for the shape which has already been subjected to corner chamfering.

(6) Program error "P385" results when the command in corner chamfering or the following block contains a positioning command or thread cutting command.

(7) Program error "P382" results when the block following the corner chamfering block contains any command except a group 01 G command or any other command.

(8) Program error "P383" results when the movement amount in the corner chamfering block is less than the chamfering amount.

(9) Program error "P384" results when the movement amount in the block following the corner chamfering block is less than the chamfering amount.

(10) Radius command values apply for corner chamfering even with diameter commands.

(11) When corner chamfering II is commanded, 2 blocks are pre-read for calculating the intersection.

13.10.2 Corner rounding (,R_)

Function and purpose

Corner rounding is accomplished by commanding "R_" in the first block for 2 blocks including a continuous circular.

Command format

N100 G03 X__ Z__ I__ K__ ,R_ ;
N200 G01 X__ Z__ ;

R__ Circular radius of corner rounding

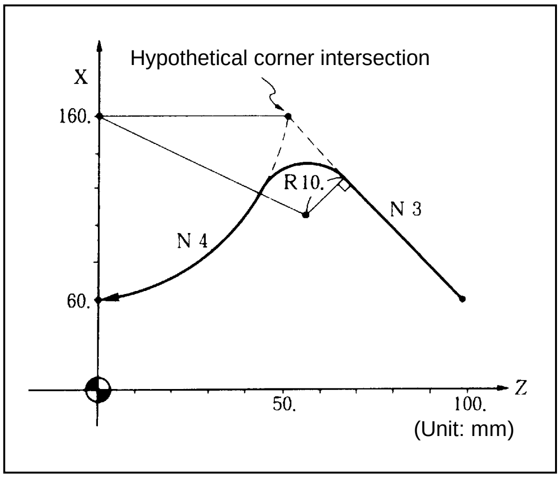
Rounding is performed at the intersection where N100 and N200 intersect.

Example of program

(1) Linear – circular

Absolute value commands

N1 G28 XZ;
N2 G00 X60. Z100.;
N3 G01 X160. Z50. ,R10. F100;
N4 G02 X60. Z0 I0 K–50.;
:



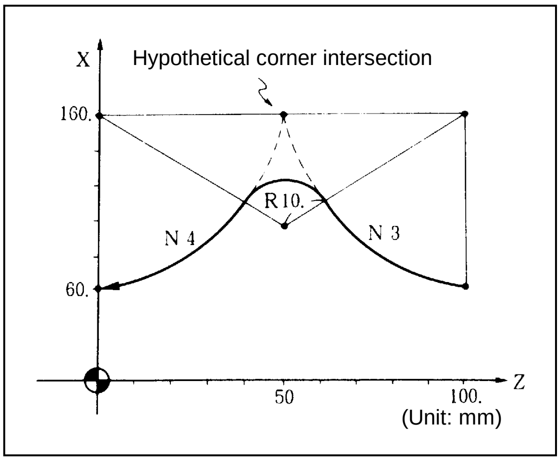
Incremental value commands

N1 G28 XZ;
N2 G00 U30. W100.;
N3 G01 U50. W–50. ,R10. F100;
N4 G02 U–50. W–50. I0 K–50.;
:

(2) Circular – circular

Absolute value commands

N1 G28 XZ;
N2 G00 X60. Z100.;
N3 G02 X160. Z50. ,R10. F100;
N4 X60. Z0 R50.;
:



Incremental value commands

N1 G28 XZ;
N2 G00 U30. W100.;
N3 G02 U50. W–50. I50. K0 ,R10. F100;
N4 U–50. W–50. I0 K–50.;
:

Detailed description

(1) Both the corner chamfering and corner rounding I and II options are required in order to use this function.
Program error "P381" results when the function is assigned without an option.

(2) The start point of the block following the corner rounding serves as the hypothetical corner intersection.

(3) When the comma in ",R" is not present, it is handled as an R command.

(4) When both "C_" and "R_" are commanded in the same block, the latter command is valid.

(5) Tool offset is calculated for the shape which has already been subjected to corner rounding.

(6) Program error "P385" results when the command in corner rounding or the following block contains a positioning command or thread cutting command.

(7) Program error "P382" results when the block following the corner rounding block contains any command except a group 01 G command or any other command.

(8) Program error "P383" results when the movement amount in the corner rounding block is less than the R value.

(9) Program error "P384" results when the movement amount in the block following the corner rounding block is less than the R value.

(10) Radius command values apply for corner rounding even with diameter commands.

(11) When corner rounding II is commanded, 2 blocks are pre-read for calculating the intersection.

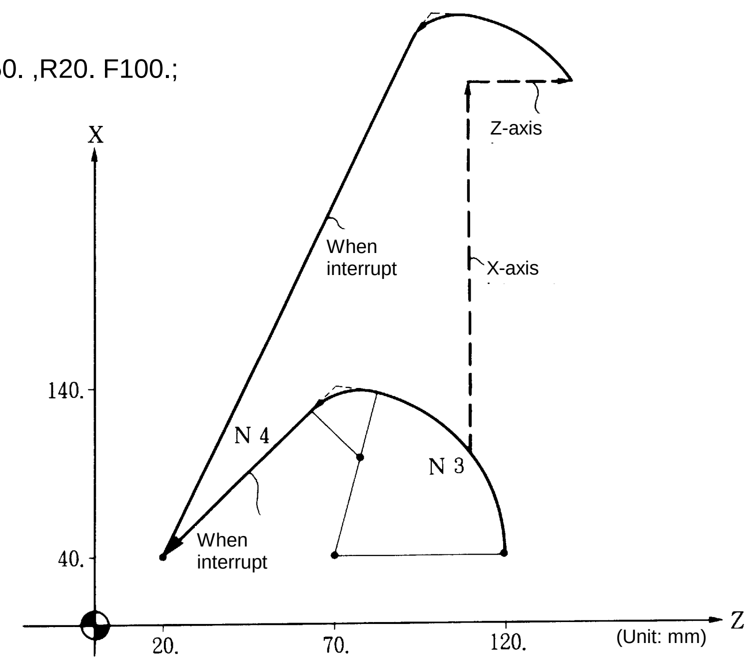
13.10.3 Interrupt during corner chamfering/rounding

Detailed description

(1) The operations are shown below for manual interrupt during corner chamfering or corner rounding.

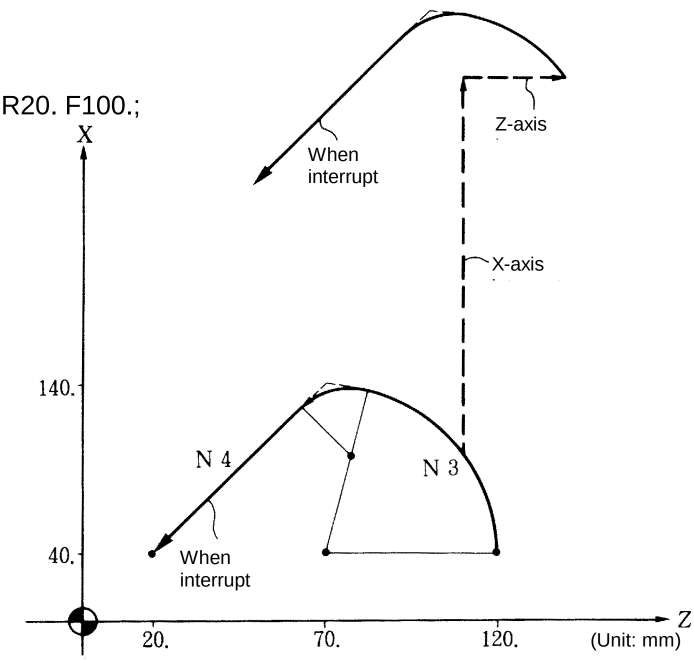
With an absolute value command and manual absolute switch ON.

```
N1 G28 XZ;
N2 G00 X40. Z120.;
N3 G03 X140. Z70. K–50. ,R20. F100.;
N4 G01 X40. Z20.;
        :
```



With an incremental value command and manual absolute switch OFF

```
N1 G28 XZ;
N2 G00 U20. W120.;
N3 G03 U50. W–50. K–50. ,R20. F100.;
N4 G01 U–50. W–50.;
        :
```



(2) With a single block during corner chamfering or rounding, the tool stops after corner chamfering or rounding is executed.

13.11 Linear Angle Command

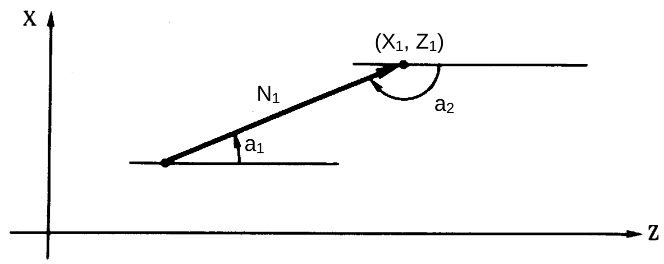
Function and purpose

The end point coordinates are calculated automatically by commanding the linear angle and one of the end point coordinate axes.

Command format

N1 G01 A<u>a1</u> Z<u>z1</u> (Xx1) ;
N1 G01 X__ Z__ ;

This designates the angle and the X or Z axis coordinates.



Detailed description

(1) The angle is from the + direction of the horizontal axis on the selected plane. The counter-clockwise (CCW) direction is considered to be + and the clockwise direction (CW) –.

(2) Either of the axes on the selected plane is commanded for the end point.

(3) The angle is ignored when the angle and the coordinates of both axes are commanded.

(4) When only the angle has been commanded, this is treated as a geometric command.

(5) The angle of either the start point (a1) or end point (a2) may be used.

(6) The function cannot be used when address A is used for the axis name or as the 2nd miscellaneous function.

(7) This function is valid only for the G01 command; it is not valid for other interpolation or positioning commands.

13.12 Geometric Command

13.12.1 Geometric command ⅠA

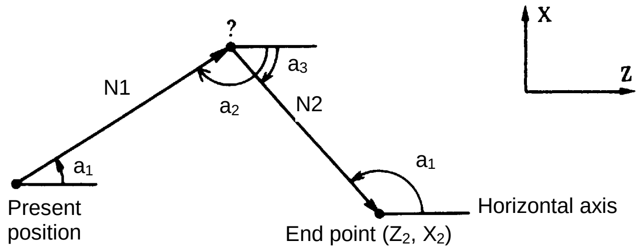
Function and purpose

When it is difficult to determine the point at which the two straight lines intersect with a continuous linear interpolation command, the end point of the first straight line will be automatically calculated inside the NC and the movement command will be controlled provided that the inclination of the first straight line as well as the end point coordinates and inclination of the second straight line are commanded.

Command format

N1 G01 Aa1 (A–a2) Ff1 ;
N2 Xx2 Zz2 A–a2 (Aa3) Ff2 ;

N1 G01 Aa1 (A–a2) Ff1 ;	This designates the angle and feedrate.
N2 Xx2 Zz2 A–a2 (Aa3) Ff2 ;	This designates the next block end point coordinates, angle and feedrate.



Detailed description

(1) Program error "P396" results when the geometric command is not on the selected plane.

(2) The inclination is expressed as the angle that is formed with the horizontal axis + direction on the selected plane. The counterclockwise (CCW) direction is considered to be + and the clockwise direction (CW) –.

(3) Inclination "a" ranges from $-360.000 \le a \le 360.000$.

(4) The inclination of the straight line can be commanded on the start or end point side. The start or end point side of the commanded inclination is identified automatically inside the NC unit.

(5) The end point coordinates of the second block should be commanded with absolute values. When incremental values are used, program error "P393" will result.

(6) The feedrate can be commanded for each block.

(7) When the angle where the two straight lines intersect is less than 1°, program error "P392" will result.

(8) Program error "P396" results when the plane is selected in the 1st block and 2nd block.

(9) This function is ignored when address A is used for the axis name or as the 2nd miscellaneous function.

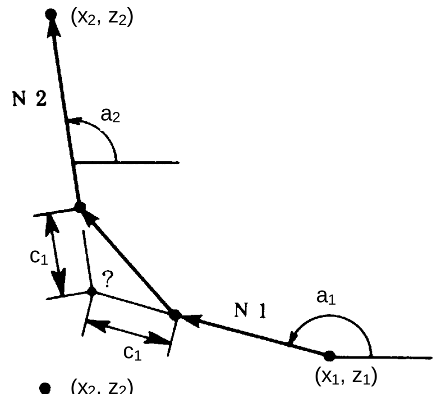
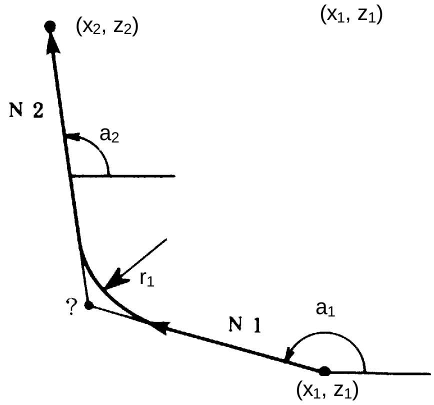
(10) Single block stop is possible at the end point of the 1st block.

(11) Program error "P394" results when the 1st and 2nd blocks do not contain the G01 or G33 command.

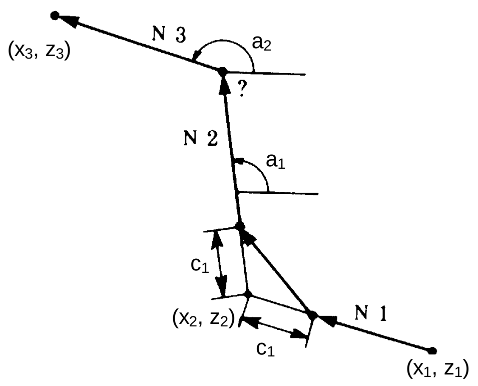
(12) When geometric IA is commanded, 2 blocks are pre-read for calculating the intersection.

Relation with other functions

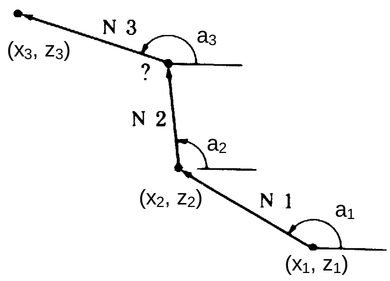
(1) Corner chamfering or corner rounding can be commanded after the angle command in the first block.

(Example 1) N1 Aa1 ,Cc1; N2 Xx2 Zz2 Aa2;	
(Example 2) N1 Aa1 ,Rr1; N2 Xx2 Zz2 Aa2;	

(2) The geometric command IA can be issued after the corner chamfering or corner rounding command.

<table>
<tr><td>(Example 3)
N1 Xx2 Zz2 ,Cc1;
N2 Aa1;
N3 Xx3 Zz3 Aa2;</td><td>
</td></tr>
</table>

(3) The geometric command IA can be issued after the linear angle command.

<table>
<tr><td>(Example 4)
N1 Xx2 Aa1;
N2 Aa2;
N3 Xx3 Zz3 Aa2;</td><td>
</td></tr>
</table>

13.13 Program Parameter Input; G10/G11

Function and purpose

The parameters set from the setting display unit can be changed with the machining programs. Only the user parameters, machine parameters and PLC parameters can be changed.

Command format

The parameters are set with the following format in the data setting mode.

G10 L60; Parameter inputting command		
P major classification number	**N data number**	**H bit type data ;**
P major classification number	**A axis number** **D byte type data**	**N data number** **;**
P major classification number	**A axis number** **S word type data**	**N data number** **;**
P major classification number	**A axis number** **L 2-word type data ;**	**N data number**
G11; Parameter inputting mode cancel (parameter inputting completed)		

There are 8 types of data formats according to the type of parameter (axis-common and axis-independent) and data type, as listed below.

With axis-common data

Axis-common bit type parameter ···········	P______N______H______;
Axis-common byte type parameter ·········	P______N______D______;
Axis-common word type parameter ·········	P______N______S______;
Axis-common 2-word type parameter ······	P______N______L______;

With axis-independent data

Axis-independent bit type parameter ········	P______A______N______H______;
Axis-independent byte type parameter ·····	P______A______N______D______;
Axis-independent word type parameter ····	P______A______N______S______;
Axis-independent 2-word type parameter ···	P______A______N______L______;

(Note 1) The sequence of addresses in a block must be as shown above.

(Note 2) Designate the axis number of the system commanded with the G10 for the axis number designated with address A.

[High-speed type parameter inputting command]

G10 L61; Parameter inputting command			
P major classification number	**A axis number**	**N data number**	Set the parameters in the data setting mode
	D byte type data ;		
P major classification number	**A axis number**	**N data number**	
	S word type data ;		
P major classification number	**A axis number**	**N data number**	
	L 2-word type data ;		
G11; Parameter inputting mode cancel (parameter inputting completed)			

When the high-speed type parameter inputting command is used, the parameter input process can be completed faster than the normal parameter inputting command.
The parameters that can be set with the high-speed parameter input command are limited to the axis specification parameters and servo parameters.
There are three types of data section formats used according to the data type.

Axis-independent byte type parameter	P2 A_____N_____D_____;
Axis-independent word type parameter	P2 A_____N_____S_____;
Axis-independent 2-word type parameter	P2 A_____N_____L_____;

(Note 1) The sequence of addresses in a block must be as shown above.

(Note 2) The axis number designated with address A is the axis number calculated with the following formula.

[Designated axis number] = ([System number] -1) × 5 + [Axis number in system]

(The basic definition number is used for the system number and axis number in system.)

(Note 3) The address N command value is valid only for value corresponding to the axis specification parameter and servo parameter.
(Refer to "Appendix 6. Correspondence of program parameter input N numbers".)

Detailed description

(1) Command address

The table below lists the addresses used in the data formats and their meanings.

Address	Meaning	Details of commands
P	Major classification number command address	This assigns the major classification number of the parameter data using a 2-digit number (positive integer) following P.
A	Axis number command address	This enables independent data to be set for each control axis using an axis number following A when the input data are axis-independent data.
N	Data number command address	This assigns the data numbers (inherent number for each data) for the parameter data using a 5-digit number (positive integer) following N. Refer to "Appendix 6" for the correspondences between the P numbers, N numbers and data contents.
H	Bit type data command address	This is used for the data command in which both the byte size parameter and bit command can be assigned. The first digit in the 2-digit number (positive integer) following H assigns the command bit while the last digit assigns the parameter value (0 or 1). Example of application: Hd0: Sets bit "d" OFF. (d: 0 to 7) Hd1: Sets bit "d" ON. (d: 0 to 7)
D	Byte type data command address	This assigns the byte type parameter value (decimal) with a number (integer) following D.
S	Word type data command address	This assigns the word type parameter value (decimal) with a number (integer) following S.
L	2-word type data command address	This assigns the 2-word type parameter value (decimal) with a number (integer) following L.

(Note) Refer to the "Data types" in Appendix 6 for information on which addresses are used for the data command addresses.

(2) Major classification number (P number)

The correspondence of the major classification number and parameter data type is shown below.

Major classification number (P number)	Type of parameter	Axis data	Remarks
1	System-common/ axis-common parameter		This user or machine parameter is common to each system or axis. The same parameter is changed from a machining program running in any system.
2	System-independent/ axis-independent parameter	○	This user or machine parameter is independent of each system or axis. The parameter designating each system from each system's machining program is changed.
3	Machine error compensation information	○	
4	Machine error compensation amount		
5	PLC constant		
6	PLC timer		
7	PLC counter		
8	Bit selection parameter		
13	System-independent/ axis-common parameter		This user or machine parameter is independent of each system and common to each axis. The parameter designating each system from each system's machining program is changed.
15	Communication parameter		
16	Position switch parameter		This machine parameter is independent of each system and common to each axis. The parameter designating each system from each system's machining program is changed.
17	Spindle-type servo parameter		The axis number is designated with an N number. (P number for M500L compatibility)
20	Superimposition error compensation parameter		
21	Spindle parameter	○	

(3) Data range

The general data range determined by each data type is shown below.

<table>
<tr><th>Data command address</th><th>Data type</th><th>Data range</th></tr>
<tr><td>H</td><td>Bit</td><td><table><tr><th></th><th>Bit0</th><th>Bit1</th><th>Bit2</th><th>Bit3</th><th>Bit4</th><th>Bit5</th><th>Bit6</th><th>Bit7</th></tr><tr><td>OFF</td><td>00</td><td>10</td><td>20</td><td>30</td><td>40</td><td>50</td><td>60</td><td>70</td></tr><tr><td>ON</td><td>01</td><td>11</td><td>21</td><td>31</td><td>41</td><td>51</td><td>61</td><td>71</td></tr></table></td></tr>
<tr><td>D</td><td>Byte</td><td>–128 to +255</td></tr>
<tr><td>S</td><td>Word</td><td>–32768 to +65535</td></tr>
<tr><td>L</td><td>2-word</td><td>–199999998 to 199999998</td></tr>
</table>

(Note) The data range in which each parameter value can be set differs according to each parameter. When creating the program, confirm the data range in Appendix 6 "Setting range (unit)", and input the correct value.

(4) Parameter command unit

The "unit" given in Appendix 6 refers to the minimum setting unit of the parameter data. Refer to the following table and command the correct data values for the "command unit", "interpolation unit" and "speed unit".
For the metric/inch classification in the following table, follow the initial inch parameter state when setting the user parameters, and follow the constant inch input parameter state when setting the machine parameters.

(a) Command unit

Input unit type	Linear axis		Rotary axis
	mm	inch	
B	0.001mm	0.0001 inch	0.001°
C	0.0001mm	0.00001 inch	0.0001°

(b) Interpolation unit

Input unit type	Linear axis		Rotary axis
	mm	inch	
B	0.0005mm	0.00005 inch	0.0005°
C	0.00005mm	0.000005 inch	0.00005°

(c) Speed unit

Input unit type	Linear axis		Rotary axis
	mm	inch	
B	1.0mm/min	0.1 inch/min	1.0°
C	0.1mm/min	0.01 inch/min	0.1°

(5) Examples of data formats

(a) Axis-common parameters

Format used	Data type	N number setting
P1 N____H____ ;	Bit	(Refer to "Appendix 6" for details)
P1 N____D________ ;	Byte	
P1 N____S________ ;	Word	
P1 N____L________ ;	2-word	

(Example) Setting the 2nd miscellaneous function code (#1109 M2name) to "B".
P13 N40 D66; (The "B" ASCII code is 0x42 [hexadecimal] → 66 [decimal])

(b) Axis-independent parameters

Format used	Data type	A number setting	N number setting
P2 A____N____H____ ;	Bit	1 to No. of control axes	(Refer to "Appendix 6" for details)
P2 A____N____D________ ;	Byte		
P2 A____N____S________ ;	Word		
P2 A____N____L________ ;	2-word		

(Example) Setting the X-axis rapid traverse rate (#2001 rapid) to 15000.
P2 A1 N8 L15000;

(c) **Machine error compensation information (Machine error compensation parameter)**

Format used	Data type	A number setting	N number setting
P3 A___N___S_______ ; P3 A___N___L_______ ;	Word 2-word	1 to 12	(Refer to "Appendix 6" for details)

(Example 1) The second set of the machine error compensation sets the 2nd axis of system 1 in the basic axis (#4001 cmpax).
P3 A2 N1 S256; (System 1 - 2nd axis is 0x0100 [hexadecimal] → 256 [decimal])

(Example 2) The first set of the machine error compensation sets 10000 in the division pitch (#4007 spcdv).
P3 A1 N7 L20000; (Setting interpolation unit)

(d) **Machine error compensation amount (Machine error compensation parameter)**

Format used	Data type	N number setting
P4 N____D________ ;	Byte	1 to 128 × No. of control axes

(Example) Set –4 for the compensation data #4305.
P4 N5 D–4;

(e) **PLC constant**

Format used	Data type	N number setting
P5 N____L________ ;	2-word	1 to 48

(Example) Set 100 for the PLC constant #6308.
P5 N8 L100;

(f) **PLC timer**

Format used	Data type	N number setting
P6 N____S________ ;	Word	0 to 103

(Example) Set 100 for the PLC timer #6017.
P6 N17 S100;

(g) **PLC counter**

Format used	Data type	N number setting
P7 N____S________ ;	Word	0 to 23

(Example) Set 20 for the PLC counter #6203.
P7 N3 S20;

(h) **Bit selection parameter**

Format used	Data type	N number setting
P8 N____H________ ; P8 N____D________ ;	Bit Byte	1 to 96 (contents fixed for 45 to 95) (Refer to "Appendix 6" for details.)

(Example 1) Set the #6401 bit pattern to 01010101.
P8 N1 D85; (01010101 [binary] → 85 [decimal])

(Example 2) Turn #6406 bit 3 ON (set to 1).
P8 N6 H31;

Example of program

(Example 1) Setting the G71 minimum cut to 1mm

:	
G10 L60;	
P13 N320 L2000;	G71 minimum cut (1mm/0.5µm = 2000) [interpolation unit]
G11;	
:	

(Example 2) Decimal point command type 1/2 setting

:	
G10 L60;	
P1 N912 H51;	Setting to decimal point type 2
G11;	
:	
G00 X-100 Z-200;	Equivalent to X-100. Z-200.
:	
G10 L60;	
P1 N912 H50;	Setting to decimal point type 1
G11;	
:	
G00 X-100 Z-200;	Equivalent to X-0.1. Z-0.2
:	

(Example 3) Making the soft limit invalid

:	
G10 L60;	
P2 A1 N641 H21;	X-axis soft limit invalid
P2 A2 N641 H21;	Z-axis soft limit invalid
G11;	
:	

(Example 4) Setting the X-axis rapid traverse rate to 24000mm/min

:	
G10 L60;	
P2 A1 N8 L24000;	X-axis rapid traverse rate
G11;	
:	

Precautions and restrictions

(1) The parameter inputting function option is required in order to use this function. Program error "P420" results when it is commanded without the option.

(2) Program error "P421" results in an illegal parameter number (major classification number, axis number or data number). The same error occurs when the setting data (bit type, byte type, word type or 2-word type data) have exceeded the data range.

(3) Program error "P421" results if in the parameter inputting format the major classification number, data number and setting data for axis-common data or the major classification number, axis number, data number and setting data for axis-independent data are not commanded.

(4) Program error "P421" results when any G command except G11 and any address except P, A, N, H, D, S and L are commanded in the parameter inputting mode.

(5) Program error "P421" results when a parameter inputting command is issued during a fixed cycle modal or nose R compensation modal.

(6) Command the axis number for the axis-independent parameter input in the basic definition axis row. Note that even if the axis layout changes with the cross machining command or random axis exchange command, the parameter will be input in the basic definition axis row.

(7) If two or more setting data items with different types are commanded in the same block, the program error "P421" will occur.

(8) When a program error has occurred in the parameter inputting mode, first remedy the error and then proceed again with the parameter inputting.

(9) If the same address is commanded two or more addresses in the same block, the latter command will be valid.

(10) It is not possible to use the binary type data command address "B" with the parameter inputting. Use the byte type data command address "D" instead.

(11) Always command G11; (parameter input mode cancel) at the end of the data setting.

(12) Note that the parameter input data format differs from the parameter tape.

(13) Only the user parameters, machine parameters and PLC parameters can be set with program parameter input.

(14) Some parameters are not validated immediately when input, and the power must be turned OFF/ON to validate them. (Refer to the Parameter Manual.)

(15) If a value other than the axis specification parameter or servo parameter value is issued for the major classification or classification number (P, N address command value) when the high-speed type parameter input command (G10L61) is commanded, that command will be invalid.

(16) Up to 20 parameters can be commanded at once with the high-speed type parameter input command (G10L61).

```
G10 L61 P2  ・・・・・・・  ;  ┐
    :                       ├  Up to 20 parameters can be commanded.
    :                       │
G10 L61 P2 ・・・・・・・・  ;  ┘
G11 ;
```

If more than 20 parameters are commanded, a program error (P421) will occur.

(17) Compatibility with older model (M500L)

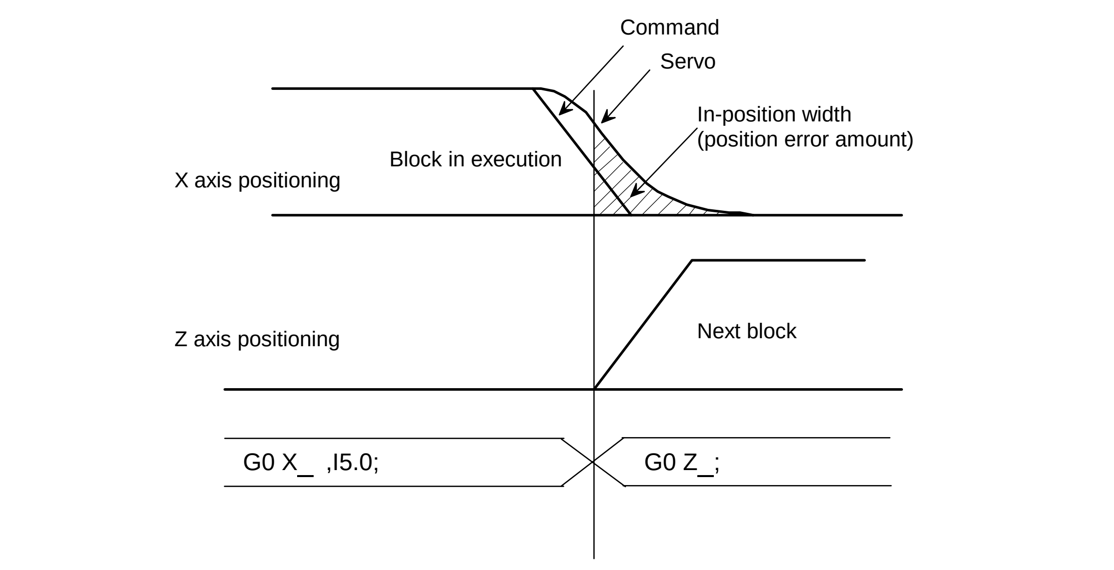
(a) The parameters changed with the parameter input command will not return to the original setting when the power is turned OFF/ON.
(The parameters that returned to the original setting when the M500L power was turned OFF/ON will not return to the original setting with the M600L.)

(b) If a designation differing from "Appendix 6" is made, the set data may differ.

13.14 Programmable In-position Check

Function and purpose

This function allows the in-position width (position error amount), a condition for shifting from the execution block to the next block's process, to be commanded from the machining program with the positioning (rapid traverse: G0) command block and with the linear interpolation (G01) and circular interpolation (G02/G03) commands.
In blocks that do not interfere with the workpiece, the machining time can be shortened by commanding the in-position width.



Command format

The in-position width is designated with the " ,I" command in the block to validate this function.

(1) Positioning command (G00)

G00 X__ Z__ ,I<u>inpos</u>;

(2) Linear interpolation (G01)/circular interpolation (G02/G03) command

G01 X__ Z__ F__ ,I<u>inpos</u>; **G02(G03) X__ Z__ R__ F__ ,I<u>inpos</u>;**

The ",I" command is valid only in the following cases in which deceleration check is carried out.

(a) When the error detect switch is ON.

(b) When G09 (exact stop check) is commanded in the same block.

(c) When G61 (exact stop check mode) is selected.

(d) When the next block is G00.

(3) Cycle command

G83 X__ Z__ R__ Q__ P__ F__ ,Iinpos;

This function is valid between the G0 and G0 block in the cycle when commanded in a cycle such as the hole drilling fixed cycle, turning fixed cycle or compound type turning cycle.

(4) Command address

,Iinpos inpos	In-position width (inpos) designation Command unit : mm/inch, decimal point command valid Command range : For minimum command unit 0.001mm/0.0001inch 0 to 999.999mm or 0 to 99.9999inch

If a command exceeding the command range is issued, a program alarm (Setting value range over) will occur.
In addition to numeric values, variables and macro operation expressions can be set for the inpos value.

(Example) ,I [#100]
,I [#101∗1.2]
,I [2.3/SIN [60]]

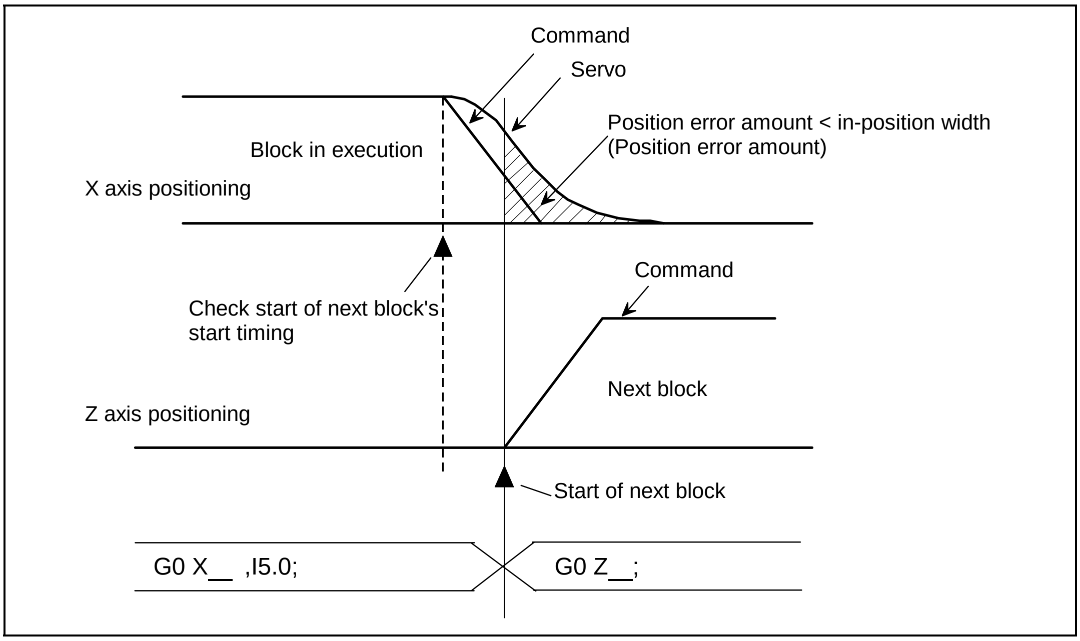
(Note) This function is invalid if the ",I" command is commanded in an independent block or if it is commanded in a block not containing the above valid commands.

Detailed description

(1) When the position error amount of the block being executed is less than the in-position width designated with this command, execution of the next block will start.

(2) The in-position width for this command is valid only in the command block, so blocks that do not have the in-position width command will use the deceleration check method.

(3) If there are multiple moving axes, the system will check whether the combined position error amount of each of the moving axes is less than the in-position width for this command, and then will start execution of the next block.



Position error amount = (end point machine position – feedback machine position)

Example of program

Metric command mode, error detect OFF

:	
G00 X100. Z100. ,I5.0;	Execute in-position width at 5mm
G00 X30.;	Execute deceleration check method with parameters
G01 Z-30. F10. ,I1.0;	In-position width command value is invalid as the G01-G01 commands continue (an error does not occur)
G01 X50. Z-50. ,I2.0;	As deceleration check is valid with G01-G00 command, execute in-position width at 2mm
G00 X80. ,I [#100];	Execute in-position width with variable #100 value
:	
:	
G09 X20. Z-40. ,I3.0;	G09 is commanded and deceleration check is valid, so execute in-position width at 3mm
:	
:	
G83 Z-50. R5. Q4. F20. ,I0.5;	Execute G0-G0 block in cycle with in-position width 0.5mm

Precautions

(1) The in-position width command value and the position error amount are compared at a set time. Thus, the timing for starting the next block may be slightly delayed from the set parameter value.

(2) In a fine segment block with a fast feedrate, the timing for starting the next block may be delayed.

(3) When the calculation request (Y233) is input with the PLC signal, the timing for starting the next block may be delayed.

(4) This function is invalid for axes in the automatic machine lock state.

(5) When this function is used for axes having an acceleration/deceleration pattern combination (example: rapid traverse linear acceleration/deceleration + rapid traverse primary delay acceleration/deceleration), an error will occur in the in-position width.

(6) At sections where the axis movement direction is reversed, this function will be invalidated to prevent a load from being applied on the motor.

Relation with other functions

(1) At the instant the superimposition synchronization command (G125, G126) or synchronization command during axis movement (G128) is executed with another system, in movement blocks that have systems with axes related to these functions, the timing for starting the next block may be delayed.

(2) If the next block reads in the position information with the coordinate system setting command (G92), machine coordinate system selection command (G53) or variable, the timing for starting the next block may be delayed.

(3) The I command is invalid during the milling machining mode. (An error will not occur.)

13.15 Positioning (G00)/Machine Coordinate System Selection (G53) Feedrate Designation

Function and purpose

The axis feedrate can be designated for when G00 (positioning command) or G53 (machine coordinate system selection) is designated.

Command format

The feedrate for moving is commanded with a ",F" command. This rate is used to move in the G00 block, G00 mode, G53 movement block, to the initial point of the hole position in the hole drilling fixed cycle block, and to move to the initial point of the hole position in the block during the hole drilling fixed cycle.

<Designation of feedrate for G00 block>

```
G00 X_ Z_ ,F1000 ;
```

<Designation of feedrate for movement command in G00 mode>

```
G00 ;
X_ Z_ ,F1000 ;
```

<Designation of feedrate for G53 block>

```
G53 X_ Z_ ,F1000 ;
```

<Designation of feedrate for command to move to initial point of hole position in hole drilling cycle>

```
G8x X_ .......... ,F1000 ;
X_ .......... ,F500 ;
   :
G80 ;
   :
```

G83: Deep hole drilling cycle (face)
G84: Tapping cycle (face)
G85: Boring cycle (face)

```
G8x Z_ .......... ,F1000 ;
Z_ .......... ,F500 ;
   :
G80 ;
   :
```

G87: Deep hole drilling cycle (longitudinal)
G88: Tapping cycle (longitudinal)
G89: Boring cycle (longitudinal)

Detailed description

(1) The ",F" command is valid only in the commanded block.

(2) If the ",F" command is issued in a block other than the G00, G00 mode, G53 or the movement block to the initial point of the hole position in the hole drilling cycle, ",F" will be ignored.

(3) Command range

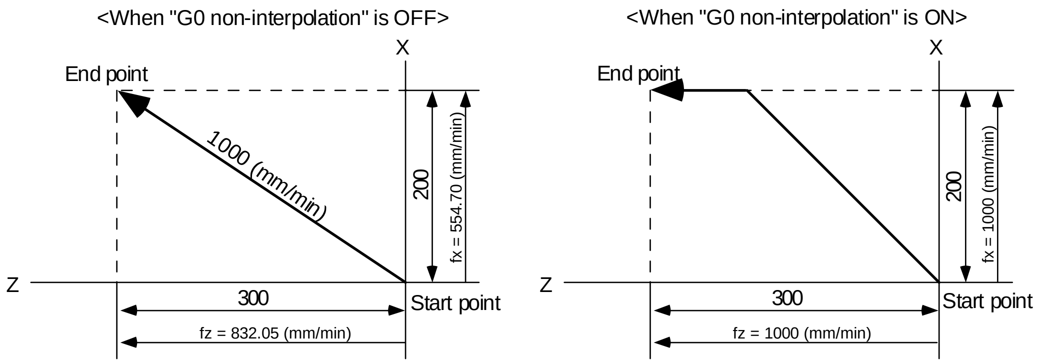
Input command unit (CS)	mm/inch	Command range	Handling of decimal point	
B	mm system	0.001 to 480000.000 (mm/min)	1 (=1.000)	1. (=1.000)
	inch system	0.0001 to 18897.6378 (inch/min)	1 (=1.0000)	1. (=1.0000)
C	mm system	0.0001 to 108000.0000 (mm/min)	1 (=1.0000)	1. (=1.0000)
	inch system	0.00001 to 4251.96850 (inch/min)	1 (=1.00000)	1. (=1.00000)

If the command range is exceeded, a program error (P35) will occur. (This also applies when a negative value is applied.)
Even if the ",F" command is commanded in a block other than the G00, G00 mode, G53 or the movement block to the initial point of the hole position in the hole drilling cycle, if the command range is exceeded a program error (P35) will occur.

(4) The handling of the ",F" command differs according to the state of the control parameter "#8113 G0 non-interpolation".

Control parameter "#8113 G0 non-interpolation"	Handling of ",F" command
OFF	Handled as the interpolation speed
ON	Handled as the commanded speed for each axis.

Feedrate for G00 X200. Z300. ,F1000 command



fx: X axis effective speed fz: Z axis effective speed fx: X axis effective speed fz: Z axis effective speed

(5) If ",F" is not commanded, the rapid traverse rate (Note 1) set with the axis specifications parameter will be valid.

(6) The ",F" command speed is clamped by the rapid traverse rate (Note 1) set with the axis specifications parameter.
The speed clamp method differs according to the state of control parameter "#8113 G0 non-interpolation".

#8113 G0 non-interpolation	Speed clamp
ON	When the commanded ",F" (interpolation speed) is converted into a per axis speed, if there is any axis that exceeds the rapid traverse rate (Note 1) parameter, the interpolation speed will be calculated so that the rapid traverse rate is not exceeded.
OFF	Any axis with a commanded ",F" (per axis speed) exceeding the rapid traverse rate (Note 1) parameter will be clamped by the parameter speed. (An axis that does not exceed the rapid traverse rate parameter will move at the commanded speed.)

(Note 1) Normally, the axis specification parameter's rapid traverse rate (#2001 rapid) is selected for the rapid traverse rate parameter. However, when using the random superimposition control/NC axis superimposition control related axis, one of the following is selected according to the superimposition related axis movement direction and movement mode: rapid traverse rate (#2001 rapid), 2-axis superimposition rapid traverse rate 0 (#2021 plrap0), 2-axis superimposition rapid traverse rate 1 (#2022 plrap1), 3-axis superimposition rapid traverse rate 0 (#2037 pl3rap0), 3-axis superimposition rapid traverse rate 1 (#2038 pl3rap1) or 3-axis superimposition rapid traverse rate 2 (#2039 pl3rap2).

Example of program

(1) Feedrate command during G00 block, or G00 mode (For G00 interpolation)

:	
G00 X100. Z100. ,F1000;	Moves at the XZ composite speed 1000 (mm/min).
X200. Z200.;	Interpolates at the maximum feedrate that does not exceed the rapid traverse rate parameter set for each X and Z axis.
X300. Z300. ,F2500;	Moves at the XZ composite speed 2500 (mm/min).
:	

(2) Feedrate command during G53 mode

:	
G53 X100. Z100. ,F3000;	Moves at the XZ composite speed 3000 (mm/min).
:	

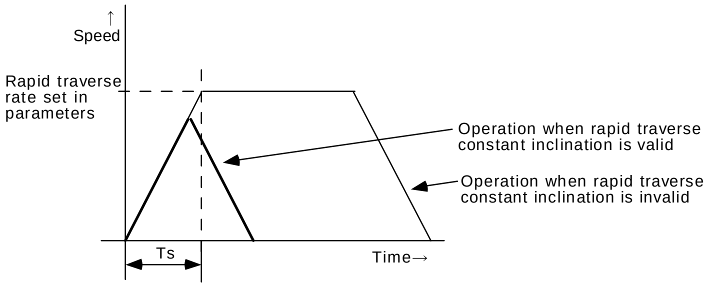
(3) Movement speed command to initial point of hole position in hole drilling cycle (For longitudinal tapping cycle)

:	
G88 X-20. Z30 R5. F1.D3 S500 ,R1 ,F2000;	Moves to the hole position's initial point (Z30.) at 2000(mm/min). (Moves at the X axis rapid traverse rate (parameter setting value) from the hole position's initial point to the R point, and from the R point to the hole position's initial point after tap cutting is completed.)
X-20. Z35. R5.;	Moves with the Z axis rapid traverser rate (parameter setting value) to the hole position's initial point (Z35.).
X-20. Z40. R5. ,F3000;	Moves to the hole position's initial point (Z40.) at 3000(mm/min). (Moves at the X axis rapid traverse rate (parameter setting value) from the hole position's initial point to the R point, and from the R point to the hole position's initial point after tap cutting is completed.)
G80;	
:	

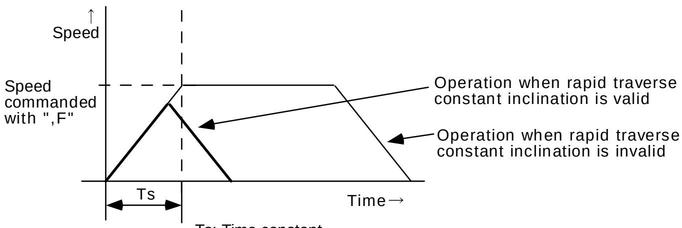
Relation with other functions

(1) Rapid traverse constant inclination control
When ",F" is commanded, constant inclination control is carried out in respect to the speed commanded with ",F".

・When ",F" is not commanded



・When ",F" is commanded



Ts: Time constant

(2) Rapid traverse override
This is the override for the ",F" command.

(3) Override cancel
Override cancel is invalid in respect to the rapid traverse override even when ",F" is commanded. If manual override is validated when dry run is selected, override cancel will be valid for the cutting override.

(4) Dry run
Dry run is valid when control parameter (#8101 G0 dry run) is ON, and rapid traverse is OFF.
The axis will move at the set manual feedrate.
Cutting feed override is also valid when manual override valid is turned ON.

(5) External deceleration
This is valid even when ",F" is commanded.

(6) Milling
The ",F" command is valid also during the milling mode.

(7) Program check
The movement speed during the program check of the block with valid ",F" command is calculated from the ",F" command and program check speed.

(8) High-speed simple program check
The time is calculated based on the ",F" command.

(9) Control axis superimposition (G126)
The ",F" command is clamped according to the rapid traverse rate parameter selected by the superimposition related axis movement direction and movement mode. These parameters include: 2-axis superimposition rapid traverse rate 0 (#2021 plrap0), 2-axis superimposition rapid traverse rate 1 (#2022 plrap1), 3-axis superimposition rapid traverse rate 0 (#2037 pl3rap0), 3-axis superimposition rapid traverse rate 1 (#2038 pl3rap1) and 3-axis superimposition rapid traverse rate 2 (#2039 pl3rap2).

(10) Programmable in-position check
This is valid even when ",F" is commanded.

Precautions

(1) If ",F" is commanded when the G00/G53 feedrate command option is not available, a program error (P39) will occur.

(2) The ",F" command is valid only in the commanded block.

(3) The ",F" command and "F" command can be commanded in the same block. The "F" command will be the feedrate for cutting feed in this case.

(4) ",F" will be ignored if commanded in a block other than the G00, G53, G00 mode, or movement block to initial point of hole position in hole drilling cycle, or in a block that does not have a movement command (axis address command).

(5) When the tool compensation operation parameter (basic common parameter #1317 Tmove) is set to 0 (compensate even blocks without a movement command), if ",F" is commanded for a tool compensation command (T command) block that has no movement command, the compensation movement will take place at the commanded speed only in the G00 mode.

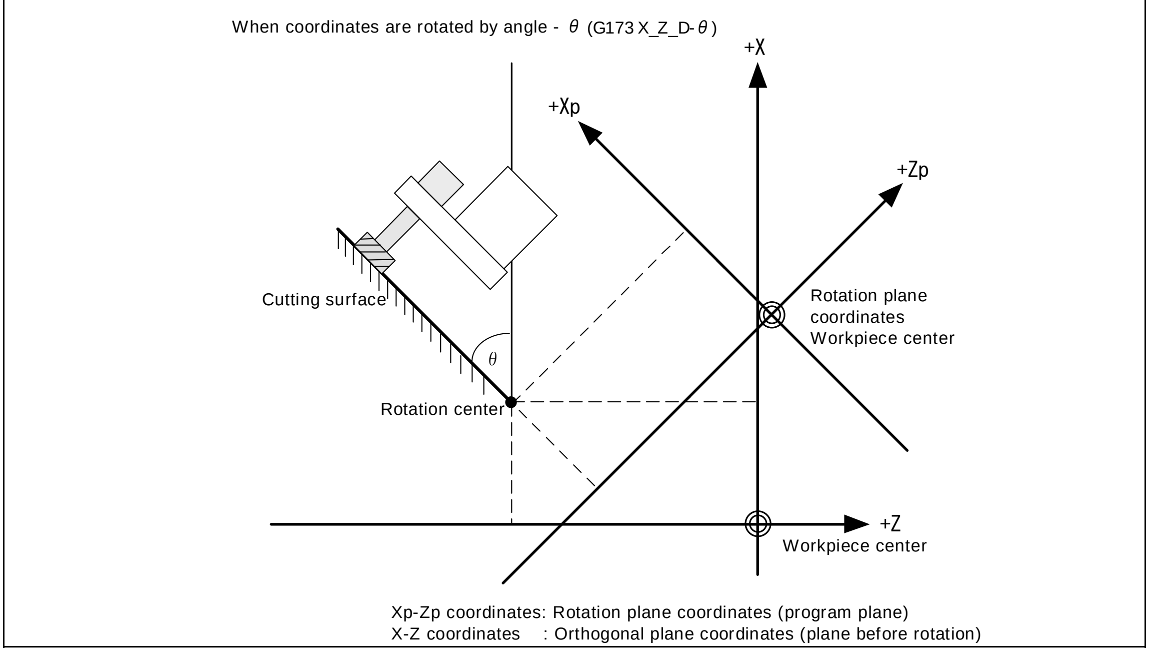
(6) If ",F" is commanded in a nose R cancel command (G40) block that has not movement, the nose R cancel operation will take place at the commanded speed only in the G00 mode.

(7) If the ",F" command exceeds the command range, a program error (P35) will occur.
   (a) A program error (P35) will also occur if a negative value is commanded.
   (b) Even if the ",F" command is issued in a block other than G00, G53, G00 mode, or movement block to initial point of hole position in hole drilling cycle, a program error (P35) will occur if the command range is exceeded.

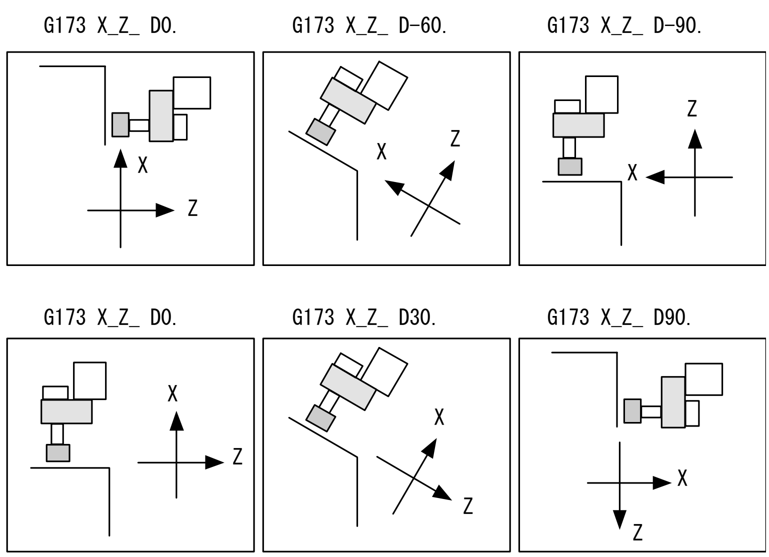
13.16 Inclined Coordinate Rotation; G173

Function and purpose

The inclined coordinate rotation command allows the coordinate system for the IK plane, which configures the coordinate system, to be rotated by the designated angle using the J axis as the rotation center axis (or it allows the coordinate system for the JK plane to be rotated by the designated angle using the I axis as the rotation center axis).
When machining a complicated shape having a position rotated from the program coordinate system, the original machining shape program is created with a non-rotating coordinate system. Then, using the inclined coordinate rotation command to designate the rotation angle, a random machining shape can be programmed easily.
The coordinate system that rotates each coordinate system (workpiece coordinate system, local coordinate system) at the point where the inclined coordinate rotation command is issued is configured.



Examples of inclined coordinate rotation command and movements



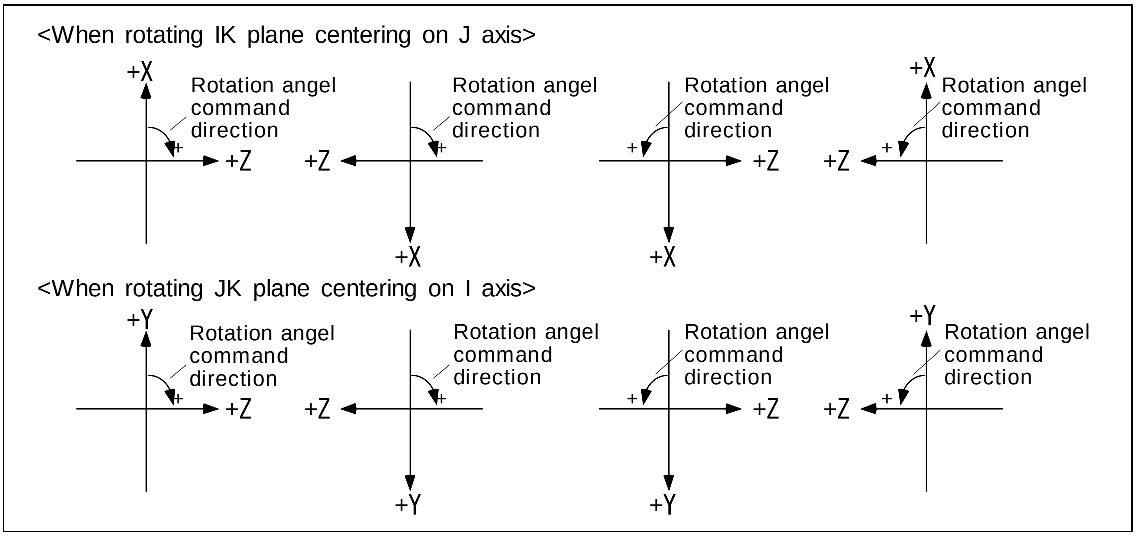
Command format

G173 Xx Zz Dd ;	**Inclined coordinate rotation ON (Rotate IK plane centering on J axis)**
G173 Yy Zz Dd ;	**Inclined coordinate rotation ON (Rotate JK plane centering on I axis)**
Xx/Yy/Zz : Rotation center coordinates	
Dd : Rotation angle	

G173; or G173 X0 Z0 D0;	**Inclined coordinate rotation cancel**
G173; or G173 Y0 Z0 D0;	**Inclined coordinate rotation cancel**

Address	**Meaning of address**	**Command range (unit)**				**Remarks**
X/Y/Z	Rotation center coordinates	Command mode	Input unit system	Internal input unit	Command range	• The position from the local coordinate zero point on the orthogonal plane is commanded with an absolute value. • The rotation center coordinate cannot be omitted.
		Metric	0.001 (mm)	0.00001 (inch)	-99999.999 to 99999.999	
				0.0001 (mm)	-9999.999 to 9999.999	
			0.0001 (mm)	0.001 (mm)	-9999.9999 to 9999.9999	
				0.0001 (mm)	-9999.9999 to 9999.9999	
		Inch	0001 (inch)	0.0001 (inch)	-9999.9999 to 9999.9999	
				0.00001 (inch)	-999.9999 to 999.9999	
			00001 (inch)	0.0001 (inch)	-999.99999 to 999.99999	
				0.00001 (inch)	-999.99999 to 999.99999	
D	Rotation angle	Input unit system		Internal input unit	Command range	• Looking from the rotation center, the direction to rotate in the Z axis (+) direction is set as the rotation angle (+) direction by using the X axis (+) direction (or Y axis (+) direction) as 0°. • Rotation angle (D) is a modal, and will not change until a new angle is commanded. • The rotation angle (D) can be omitted. If omitted when the initial G173 is commanded, D will be set to 0. If G173 is commanded during the inclined coordinate rotation mode, this will be a modal D command. • Rotation angle (D) is an absolute value command regardless of the G90/G91 command.
		0.001 (°)		0.001 (°)	-359.999 to 359.999	
				0.0001(°)	-359.999 to 359.999	
		0.0001 (°)		0.001(°)	-359.9999 to 359.9999	
				0.0001(°)	-359.9999 to 359.9999	

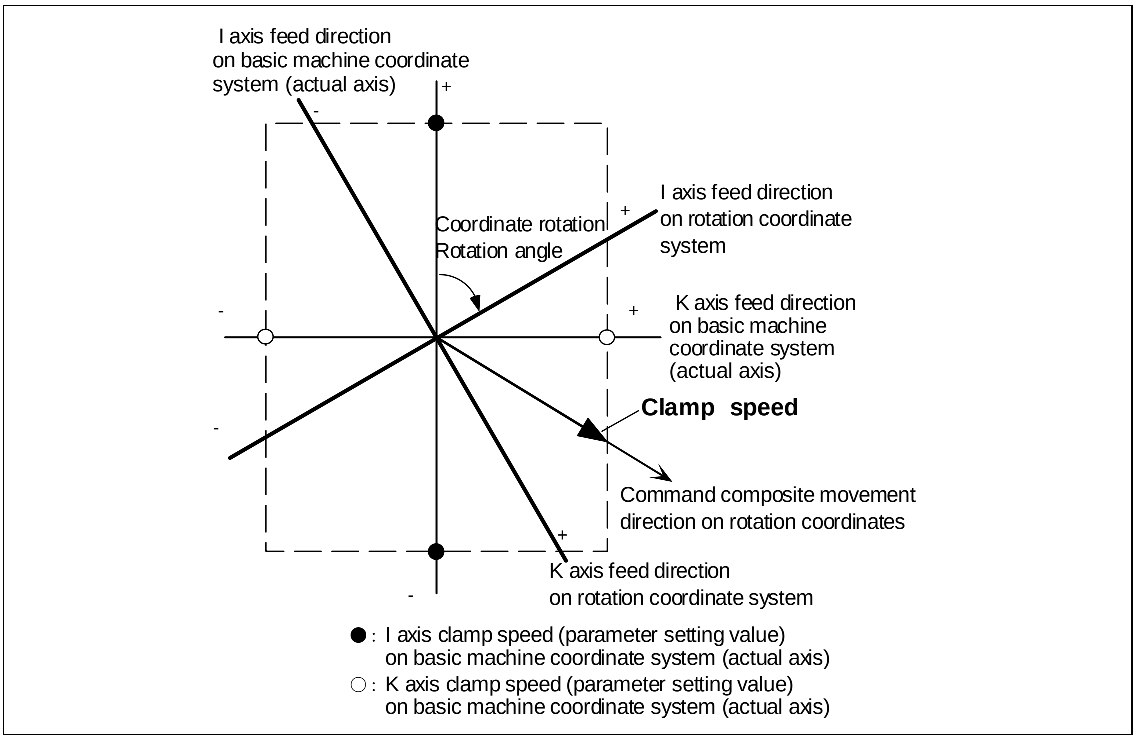
* The relation of the polarity and rotation angle command direction in the basic machine coordinate system for the axis targeted for inclined coordinate rotation is shown below.



Feedrate

The parameter setting value for the actual axis is used as the feedrate for the axis targeted for inclined coordinate rotation.
During the inclined coordinate rotation mode, the rapid traverse rate and cutting feed clamp speed are obtained considering the rotation angle from the basic machine coordinate system.

(Example) Clamp speed for cutting feed (G01) in inclined coordinate rotation mode (IK plane)



Input setting unit

When using the inclined coordinate rotation control, set the parameters with an internal input unit.
The relation of the input unit system and the internal input unit system parameter settings is shown below.

No.	Parameter				Machining program command	Internal input unit	Command output minimum unit
	#1005 iout	#8117 Initial inch	Input unit	#1362 sunit			
1	OFF (Metric output)	OFF (Metric command)	0.001 (mm)	0	0.001 (mm)	0.001 (mm)	0.001 (mm)
2				1	0.001 (mm)	0.0001 (mm)	0.0001 (mm)
3			0.0001 (mm)	0	0.0001 (mm)	0.001 (mm)	0.0001 (mm)
4				1	0.0001 (mm)	0.0001 (mm)	0.0001 (mm)
5		ON (Inch command)	0.0001 (inch)	0	0.0001 (inch)	0.0001 (inch)	0.001 (mm)
6				1	0.0001 (inch)	0.00001 (inch)	0.0001 (mm)
7			0.00001 (inch)	0	0.00001 (inch)	0.0001 (inch)	0.0001 (mm)
8				1	0.00001 (inch)	0.00001 (inch)	0.0001 (mm)
9	ON (Inch output)	OFF (Metric command)	0.001 (mm)	0	0.001 (mm)	0.001 (mm)	0.0001 (inch)
10				1	0.001 (mm)	0.0001 (mm)	0.00001 (inch)
11			0.0001 (mm)	0	0.0001 (mm)	0.001 (mm)	0.00001 (inch)
12				1	0.0001 (mm)	0.0001 (mm)	0.00001 (inch)
13		ON (Inch command)	0.0001 (inch)	0	0.0001 (inch)	0.0001 (inch)	0.0001 (inch)
14				1	0.0001 (inch)	0.00001 (inch)	0.00001 (inch)
15			0.00001 (inch)	0	0.00001 (inch)	0.0001 (inch)	0.00001 (inch)
16				1	0.00001 (inch)	0.00001 (inch)	0.00001 (inch)

Relation with other functions

Function name	Operation	Remarks
Positioning (G00)	This will function even during the inclined coordinate rotation mode. Note that even when G00 non-interpolation is selected, positioning including the axis targeted for inclined coordinate rotation in the inclined coordinate rotation mode will be positioned with interpolation.	The feedrate will be controlled with the rotation coordinates so that the rapid traverse rate (parameter setting) set for the actual axis is not exceeded.
Linear interpolation (G01) Circular interpolation (G02/G03) Helical interpolation	These will function even during the inclined coordinate rotation mode.	The feedrate will be controlled with the rotation coordinates so that the cutting clamp speed (parameter setting) set for the actual axis is not exceeded.
Milling interpolation (G12.1/G13.1)	The program error (P711) will occur if inclined coordinate rotation is commanded during the G12.1 mode. The program error (P712) will occur if G12.1 is commanded during the inclined coordinate rotation mode.	
Override	The override will be applied in respect to the speed on the rotation coordinate system during the inclined coordinate rotation mode.	
Automatic acceleration/deceleration after interpolation	This will function even during the inclined coordinate rotation mode.	Set the time constants (parameter settings) for the axes targeted for the set rotation coordinates to the same value.
Constant inclination acceleration/deceleration	This will function even during the inclined coordinate rotation mode.	
Thread cutting	The program error (P711) will occur if inclined coordinate rotation is commanded during the thread cutting modal. The program error (P712) will occur if thread cutting is commanded during the inclined coordinate rotation mode.	
Manual mode (Jog, incremental, handle)	This will function even during the inclined coordinate rotation mode. The axis selection will be valid for the axes on the rotation coordinate system. Note that if the following occurs during the inclined coordinate rotation mode, the "M01 Operation error 1121" will occur: If a feed axis is selected for the inclined coordinate rotation two target axes, the "M01 Operation error 1121" will occur. (If the above operation is carried out while axis 1 is moving, both axes will stop.)	The error can be reset with the reset signal input after one or both axes selection signals for the axis targeted for inclined coordinate rotation (axis selection valid signal for handle) is turned OFF.
Manual mode (manual random feed)	During the inclined coordinate rotation mode, if a command is issued to the axis targeted for inclined coordinate rotation, the "M01 Operation error 1121" will occur.	The error can be reset with the reset signal input.
Constant surface speed control	This will function even during the inclined coordinate rotation mode.	The surface speed is calculated with the coordinate system on the rotation coordinates.
Tool length, wear compensation	These will function even during the inclined coordinate rotation mode. Whether to hold the compensation amount at reset is determined by the parameter settings.	The tool compensation amount with the inclined coordinate rotation command is not canceled. Command the tool compensation to match the tool being used on the rotation coordinates.

Function name	Operation	Remarks
Nose R compensation	This will function even during the inclined coordinate rotation mode. The compensation amount will be canceled at reset. If the inclined coordinate rotation command is issued during the nose R compensation mode, the program error (P711) will occur.	
Machine coordinate system setting (G53)	This will function even during the inclined coordinate rotation mode. The axis will move to the machine coordinate position on the rotation coordinate system.	
Coordinate system setting (G92)	This will function even during the inclined coordinate rotation mode.	
Workpiece coordinate system selection (G54 to G59)	This will function even during the inclined coordinate rotation mode.	
Local coordinate system setting (G52)	This will function even during the inclined coordinate rotation mode.	
External workpiece coordinate system offset	This will function even during the inclined coordinate rotation mode.	
Plane selection	The plane will change to the G17 plane when inclined coordinate rotation is commanded. This will function even during the inclined coordinate rotation mode. (Other planes can be changed to.)	The state before inclined coordinate rotation will be held even after the rotation coordinate system is changed.
Manual reference point return	During the inclined coordinate rotation mode, if this is commanded to an axis targeted for inclined coordinate rotation, the "M01 Operation error 1121" will occur.	The error can be reset with the reset signal input after the axis selection signal for the axis targeted for inclined coordinate rotation is turned OFF.
Automatic reference point return (G28/G29) 2nd, 3rd, 4th reference point return (G30) Reference point check (G27)	During the inclined coordinate rotation mode, if these are commanded to an axis targeted for inclined coordinate rotation, the program error (P712) will occur.	
Absolute position detection (dog-type)	During the inclined coordinate rotation mode, if this is commanded to an axis targeted for inclined coordinate rotation, the "M01 Operation error 1121" will occur.	The error can be reset with the reset signal input after the axis selection signal for the axis targeted for inclined coordinate rotation is turned OFF.
Absolute position detection (dogless-type)	During the inclined coordinate rotation mode, an error will occur if the absolute position setting operation is attempted to an axis targeted for inclined coordinate rotation. <Error> When data is set in "Abs pos set #1201" from setting screen: Setting not possible When data is set in "Abs pos set #1201" from DDB: DDB error will turn ON	
Absolute position detection (automatic dogless)	During the inclined coordinate rotation mode, if this is commanded to an axis targeted for inclined coordinate rotation, the "M01 Operation error 1121" will occur.	The error can be reset with the reset signal input after the axis selection signal for the axis targeted for inclined coordinate rotation is turned OFF.
Tool position return	During the inclined coordinate rotation mode, if this is commanded to an axis targeted for inclined coordinate rotation, the program error (P712) will occur.	

Function name	Operation	Remarks
Automatic machine lock	During the inclined coordinate rotation mode, the automatic machine lock signal for two axes targeted for inclined coordinate rotation is valid when the signal is selected for two axes comprising a set. During the inclined coordinate rotation mode, if the automatic machine lock signal is selected for only one of two axes targeted for inclined coordinate rotation, the error "M01 Operation error 1121" will occur when the machining program block is started. If the automatic machine lock signal is selected for only one of the two axes targeted for inclined coordinate rotation during a block in the inclined coordinate rotation mode, an error will occur when the block ends.	The error can be reset by turning the automatic machine lock signal ON or OFF for both axes targeted for the inclined coordinate rotation set.
	In the inclined coordinate rotation canceled state, if a hypothetical coordinate setting command is issued when automatic machine lock signal is selected for only one of the two axes targeted for inclined coordinate rotation, the error "M01 Operation error 1120" will occur.	The error can be reset with the reset signal input.
Manual machine lock	During the inclined coordinate rotation mode, the manual machine lock signal for two axes targeted for inclined coordinate rotation is valid when the signal is selected for two axes comprising a set. If the manual machine lock signal is selected for one of the two axes targeted for inclined coordinate rotation during the inclined coordinate rotation mode and manual operation mode, the error "M01 Operation error 1121" will occur. During the inclined coordinate rotation mode, if any of the axes in the inclined coordinate rotation setting system is moving with the manual mode, and the manual machine lock signal is selected for only one of the two axes targeted for inclined coordinate rotation, all axes in the system will stop and an error will occur.	The error can be reset by turning the manual machine lock signal ON or OFF for both axes targeted for inclined coordinate rotation.
	If the hypothetical coordinate setting command is issued with the DDBS while inclined coordinate rotation is canceled and the manual machine lock signal is selected for only one of the two axes targeted for inclined coordinate rotation, the error "M01 Operation error 1120" will occur.	The error can be reset with the reset signal input.
Program check operation	Reverse run extending over the inclined coordinate rotation command/cancel command is not possible. Reverse run of the inclined coordinate rotation command/cancel command is prohibited.	
NC reset	The inclined coordinate rotation mode is canceled by resetting.	
Automatic handle interrupt Manual/automatic simultaneous	During the inclined coordinate rotation mode, if a command is issued to an axis targeted for inclined coordinate rotation, the "M01 Operation error 1121" will occur.	The error can be reset with the reset signal input.
Manual ABS	This will function even during the inclined coordinate rotation mode. Note that if the inclined coordinate rotation command/cancel command is issued when manual ABS is OFF, the manually moved amount will be added to the position after coordinate conversion. (Same as manual ABS ON state.)	
Hole drilling fixed cycle Synchronous tapping cycle (G84/G88) Turning fixed cycle	This will function even during the inclined coordinate rotation mode. If inclined coordinate rotation command/cancel command is issued during the fixed cycle mode, the program error (P34) will occur.	

Function name	Operation	Remarks
Compound lathe cutting fixed cycle	If the compound lathe cutting fixed cycle is commanded during the inclined coordinate rotation mode, the program error (P712) will occur. If inclined coordinate rotation command is issued during the fixed cycle mode, the program error (P34) will occur.	
Mirror image	This will function even during the inclined coordinate rotation mode. The mirror image will be applied on the axis on the rotation coordinates. If inclined coordinate rotation is commanded during mirror image, the program error (P711) will occur.	
Double-turret mirror image T command double-turret mirror image	If these are commanded during the inclined coordinate rotation mode, the program error (P712) will occur. If inclined coordinate rotation is commanded during mirror image, the program error (P711) will occur.	
Position information retrieving variables		
Previous block end point coordinates (#5001 to)	The status of the rotation coordinate system is read during the inclined coordinate rotation mode.	
Machine coordinates (#5021 to)		
Workpiece coordinates (#5041 to)		
Skip coordinates (#5061 to)		
Tool offset amount (#5081 to)		
Servo deviation amount (#5101 to)	The status of the basic coordinate system (actual axis) is read.	
Collision detection machine position (#5121 to)	The status of the basic coordinate system (actual axis) is read.	
Start point wait (G115/G116) Miscellaneous function during axis movement output (G117)	During the inclined coordinate rotation mode, these will function on the rotation coordinate system. If the inclined coordinate rotation command is issued while waiting, the "M01 Operation error 1120" will occur.	
Tool-spindle synchronization II (hobbing)	If this is commanded during the inclined coordinate rotation mode, the program error (P712) will occur If inclined coordinate rotation is commanded during the tool-spindle synchronization II mode, the program error (P711) will occur.	
Cross machining control (G110)	During the inclined coordinate rotation mode, if the random axis exchange command is issued to an axis targeted for inclined coordinate rotation, the "M01 Operation error 1121" will occur.	
Control axis synchronization control (G125) Control axis superimposition control (G126)	During the inclined coordinate rotation mode, if these are commanded to the axis targeted for inclined coordinate rotation, the error "M01 Operation error 1121" will occur. If inclined coordinate rotation is commanded during the superimposition/synchronization mode, the error "M01 Operation error 1120" will occur.	The error can be reset with the reset signal input.

Function name	Operation	Remarks
Peripheral axis synchronization	During the inclined coordinate rotation mode, if the axis targeted for inclined coordinate rotation is commanded as the synchronous axis, the error "M01 Operation error 1121" will occur. If inclined coordinate rotation is commanded during the synchronous mode with the synchronous axis as the axis targeted for inclined coordinate rotation, the error "M01 Operation error 1120" will occur.	The error can be reset with the reset signal input.
Programmable in-position check Automatic error detection	These will function even during the inclined coordinate rotation mode.	
Backlash compensation Memory-type pitch error compensation Relative position error compensation External machine coordinate system compensation	These will function on the basic coordinate system (actual axis).	
Skip (G31) Automatic tool length measurement	If these are commanded during the inclined coordinate rotation mode, the program error (P712) will occur.	
Manual skip Manual tool length measurement I, II	During the inclined coordinate rotation mode, if these are commanded to the axis targeted for inclined coordinate rotation, the error "M01 Operation error 1121" will occur.	
Emergency stop	The inclined coordinate rotation mode will be canceled by emergency stop.	
Stored stroke limit Chuck, tailstock barrier	The check will be carried out with the basic machine coordinates (actual axis) even during the inclined coordinate rotation mode.	
Interlock	During the inclined coordinate rotation mode, this will function in respect to the axis on the rotation coordinate system.	
Interference check I Interference check II	The check will be carried out with the basic machine coordinates (actual axis) even during the inclined coordinate rotation mode.	
Servo OFF Axis removal	During the inclined coordinate rotation mode, if the servo OFF signal is selected for one of the axes targeted for inclined coordinate rotation, the error "M01 Operation error 1121" will occur. This also applies to the axis removal signal.	The error can be reset by turning the servo OFF signal ON or OFF for both axes targeted for inclined coordinate rotation.
	If the servo is OFF for the axis targeted for inclined coordinate rotation and the inclined coordinate rotation command is issued, the error "M01 Operation error 1120" will occur. This also applies to the axis removal signal.	The error can be reset with the reset signal input.
Position switch	The check will be carried out on the rotation coordinate system during the inclined coordinate rotation mode.	
Torque constant control Proportional torque pressing control	During the inclined coordinate rotation mode, if these are commanded to the axis targeted for inclined coordinate rotation, the error "M01 Operation error 1121" will occur.	The error can be reset by inputting the reset signal after the torque constant control request signal is turned OFF.
	If the inclined coordinate rotation command is issued while inclined coordinate rotation is canceled and torque constant control is selected for the axis targeted for inclined coordinate rotation, the error "M01 Operation error 1120" will occur. This also applies to the proportional torque pressing control.	The error can be reset with the reset signal input.

Function name	Operation	Remarks
Inclined axis control	The inclined coordinate rotation control in the inclined axis control system will follow the limits for the inclined axis control. If G170/G171 is commanded during the inclined coordinate rotation mode, the program error (P712) will occur.	
Machine position display	During the inclined coordinate rotation mode, the machine position for the basic machine coordinates (actual axis) will be displayed.	
Display of positions other than machine position	The position in the rotation coordinate system will be displayed. The position display will change when inclined coordinate rotation is commanded.	

Precautions

(1) When using the inclined coordinate rotation command, each coordinate system (workpiece coordinates, local coordinates, external workpiece coordinates, coordinate system settings) at the commanded point will configure the rotated coordinate system. In the same manner, when the cancel command is issued, the coordinate system that rotates (rotates to 0° position) each coordinate system at that point will be configured.

(2) The plane will change to the G17 plane when the inclined coordinate rotation command is issued. To change to another plane during the inclined coordinate rotation mode, execute the plane selection command again. When the cancel command is issued, the selection will return to the plane on which inclined coordinate rotation was commanded.

(3) During the inclined coordinate rotation mode, the inclined coordinate rotation command can be issued again to change the rotation center coordinates and rotation angle.

(4) The workpiece coordinates, local coordinates, external workpiece coordinates and coordinate system setting can be commanded during the inclined coordinate rotation mode.

(5) The axis address command in the inclined coordinate rotation mode is issued as a radius value command.
The coordinate value display will be a radius value display, except for the machine value.

(6) When changing to the inclined coordinate rotation mode with different rotation center axis, a program error (P33) will occur if the inclined coordinate rotation mode is not canceled before changing to the next mode.

(7) If another G code is commanded in the same block as the inclined coordinate rotation command, or if another G code is commanded in the same block as the inclined coordinate rotation cancel command, the program error (P34) will occur.

(8) After the coordinate rotation setting is completed, this command will proceed to the next block.

(9) A program error will occur in the following cases.

<table>
<tr><th>Details</th><th>Error No.</th></tr>
<tr><td>When inclined coordinate rotation control option is not provided</td><td>P39</td></tr>
<tr><td>When an address (excluding sequence No.) other than the rotation coordinate center command or rotation angle command is issued in the G173 command block</td><td>P32</td></tr>
<tr><td>When the rotation center command is not commanded with the two axes, I axis (J axis) and K axis in the G173 command block</td><td rowspan="2">P33</td></tr>
<tr><td>When a command to change to an inclined coordinate rotation mode with difference rotation center axis is issued before canceling the inclined coordinate rotation mode</td></tr>
<tr><td>When another G code is commanded in the G173 command block</td><td rowspan="2">P34</td></tr>
<tr><td>When inclined coordinate rotation is commanded or canceled during the fixed cycle mode</td></tr>
<tr><td>When inclined coordinate rotation is commanded or canceled while mirror image is being applied for the axis targeted for inclined coordinate rotation</td><td rowspan="3">P711</td></tr>
<tr><td>When inclined coordinate rotation is commanded or canceled during the milling mode</td></tr>
<tr><td>When inclined coordinate rotation is commanded or canceled during the nose R compensation mode</td></tr>
</table>

The above program errors will also occur if a command setting a same value as the currently set coordinate rotation angle is issued. (When the rotation angle does not change before and after the command.)

(10) "M01 Operation error 1120" will occur in the following cases.

Conditions for "M01 Operation error 1120" occurrence
When the axis targeted for inclined coordinate rotation to be rotated at the inclined coordinate rotation commanded by G173 is in the following state:
When axis targeted for inclined coordinate rotation is not found in the command system
When the axis is related to control axis synchronization control, control axis superimposition control, random axis superimposition control, and synchronization during axis movement, or is a synchronous axis for peripheral axis synchronization control.
When in the start point designation wait state (G115/G116)
When in the miscellaneous function output during axis movement (G117) wait state
When one of the axes targeted for inclined coordinate rotation is in the machine lock state
When the axis targeted for inclined coordinate rotation is in the servo OFF state
When the axis targeted for inclined coordinate rotation is removed
When zero point return is incomplete; except for the inclined coordinate rotation cancel command
When in the automatic handle interrupt mode
When manual/automatic simultaneous is selected

The above operation errors will also occur if a command setting a same value as the currently set coordinate rotation angle is issued. (When the rotation angle does not change before and after the command.)

(11) The inclined coordinate rotation is canceled when the power is turned ON.

(12) Set the acceleration/deceleration mode and acceleration/deceleration time constants to the same values for the two axes targeted for inclined coordinate rotation.

(13) The spindle rotation speed will change if the rotation coordinate is changed during the constant surface speed mode.

(14) If all axes in the system where the inclined coordinate rotation was commanded are not stopped, the command will be executed after all axes have stopped. When the inclined coordinate rotation command is issued when all axes in the system where the inclined coordinate rotation was commanded are not stopped, if the inclined coordinate rotation setting is disabled during the interval to the completion of inclined coordinate rotation control (until all axes stop in the system where inclined coordinate rotation is commanded), the inclined coordinate rotation control will be stopped.

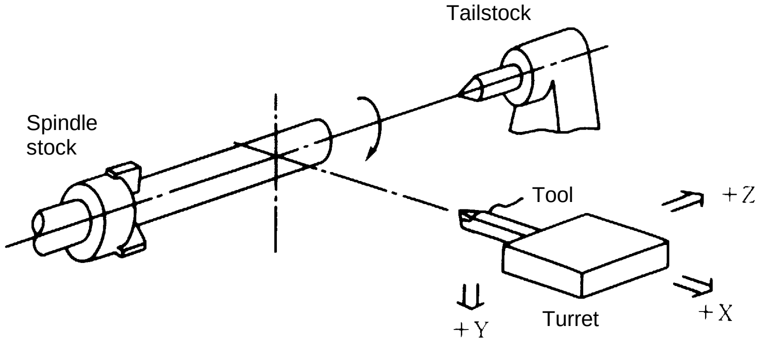
(15) The name of the axis (actual axis) targeted for inclined coordinate rotation is displayed for the name of the axis when an alarm or error occurs during the inclined coordinate rotation mode.

14. COORDINATE SYSTEM SETTING FUNCTIONS

14.1 Coordinate Words and Control Axes

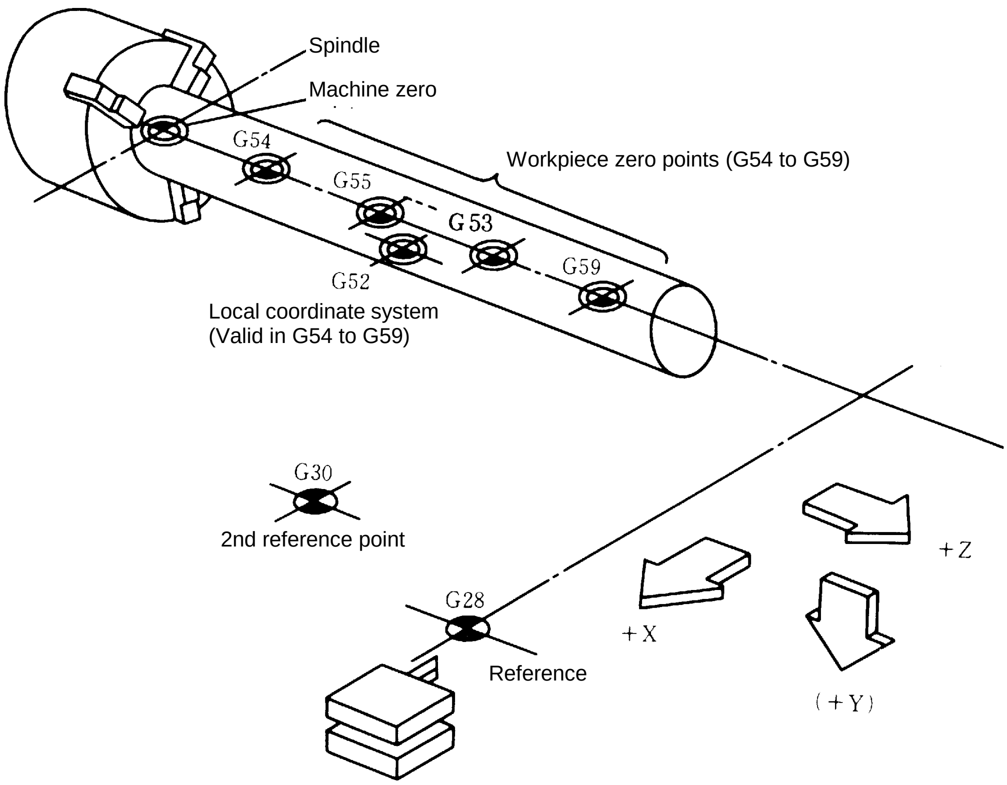
Outline

In the case of a lathe, the axis parallel to the spindle is known as the Z axis and its forward direction is the direction in which the turret moves away from the spindle stock while the axis at right angle to the Z axis is the X axis and its forward direction is the direction in which it moves away from the Z axis, as shown in the figure below.



Coordinate axes and polarities

Since coordinates based on the right hand rule are used with a lathe, the forward direction of the Y axis in the above figure which is at right angles to the X-Z plane is downward. It should be borne in mind that an arc on the X-Z plane is expressed as clockwise or counterclockwise as seen from the forward direction of the Y axis. (Refer to the section on circular interpolation.)

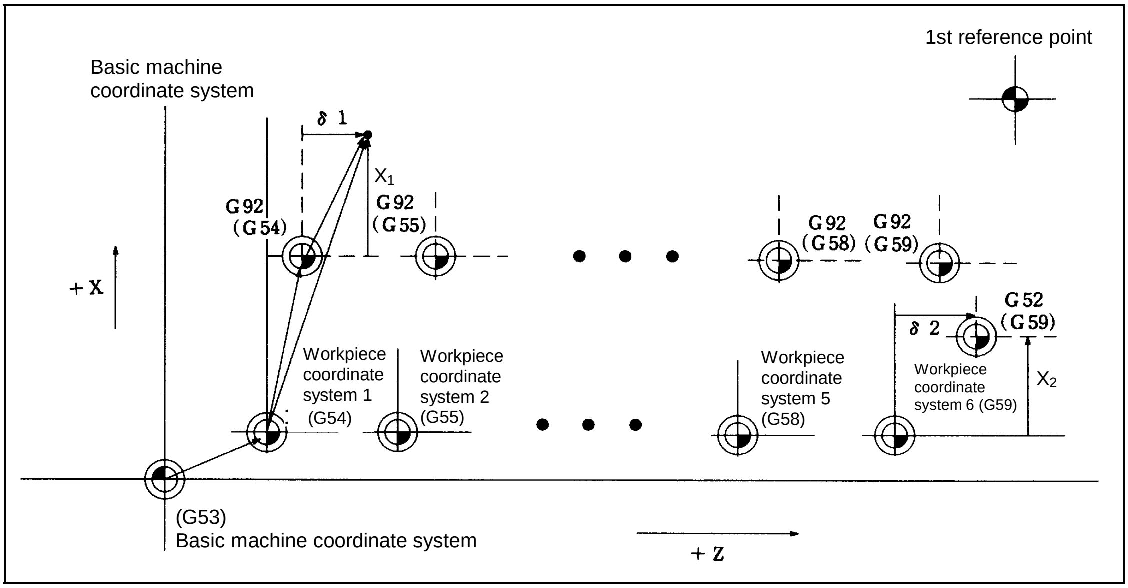


Relationship between coordinates

14.2 Basic Machine, Workpiece and Local Coordinate Systems

Outline

The basic machine coordinate system is fixed in the machine and it denotes that position which is determined inherently by the machine.
The workpiece coordinate systems are used for programming and in these systems the reference point on the workpiece is set as the coordinate zero point.
The local coordinate systems are created on the workpiece coordinate systems and they are designed to facilitate the programs for parts machining.
Upon completion of the reference point return, the basic machine coordinate system and workpiece coordinate systems (G54 to G59) are automatically set with reference to the parameters.
The basic machine coordinate system is set so that the first reference point is brought to the position specified by the parameter from the basic machine coordinate zero point (machine zero point).



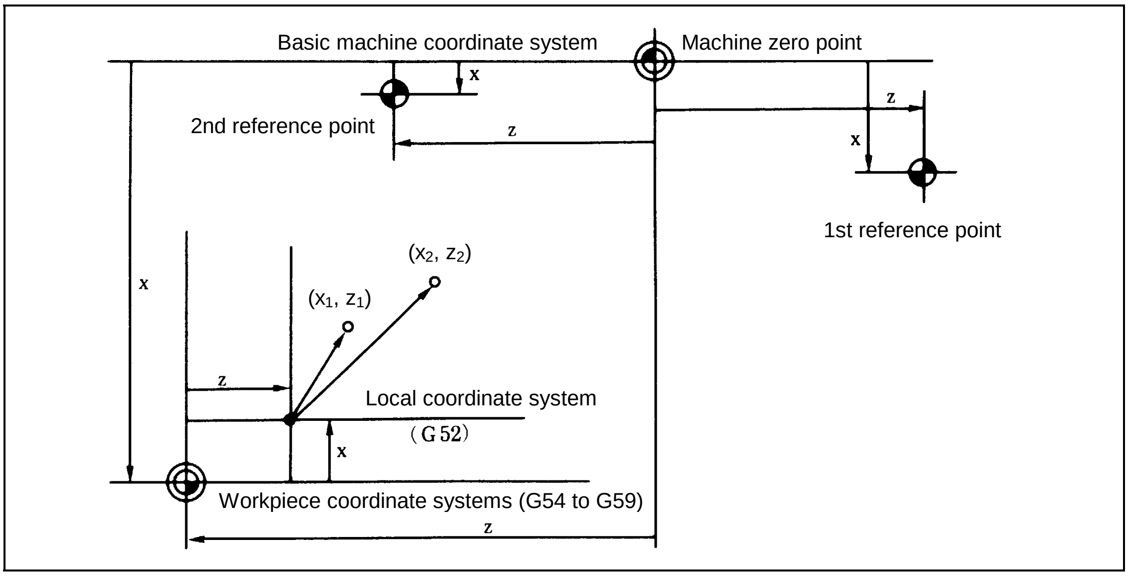
The local coordinate systems (G52) are valid on the coordinate systems designated by workpiece coordinate systems 1 to 6.
When the G52 Xx2 Zz2; command is issued while the G59 workpiece coordinate system is selected, the G59 workpiece coordinate system shifts to the G52 (G59) point in the figure.
The other workpiece coordinate systems (G54 to G58) are not affected.

The automatic coordinate system (G92) is valid in all workpiece coordinate systems 1 to 6.
When the G92 Xx1 Zz1; command is issued at point A while the G54 workpiece coordinate system is selected, the G54 workpiece coordinate system shifts to the G92 (G54) point in the figure.
The other workpiece coordinate systems (G55 to G59) are also shifted.

14.3 Machine Zero Point and 2nd Reference Point (Zero Point)

Outline

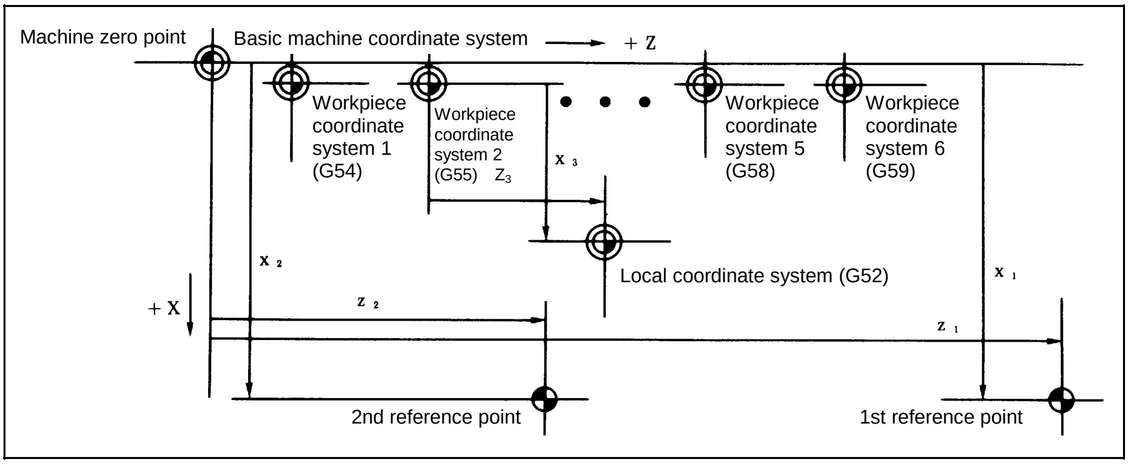
The machine zero point serves as the reference for the basic machine coordinate system. It is inherent to the machine and is determined by the reference (zero) point return.
The 2nd reference (zero) point relates to the position of the coordinates which have been set beforehand by parameter from the basic machine coordinate system zero point.



14.4 Automatic Coordinate System Setting

Function and purpose

When the tool has arrived at the reference point by dog-type reference point return after the NC's power has been turned ON, this function create the various coordinate systems in accordance with the parameter values input beforehand from the setting display unit.
The actual machining program is programmed over the coordinate systems which have been set above.



Detailed description

(1) The coordinate systems created by this function are as follows:

(a) Basic machine coordinate system

(b) Workpiece coordinate systems (G54 to G59)
The local coordinate system (G52) is canceled.

(2) The parameters set in the NC unit all provide the distance from the basic machine coordinate system zero point. Therefore it is decided at which position in the basic machine coordinate system the 1st reference point should be set and then the zero point positions of the workpiece coordinate systems are set.

(3) When the automatic coordinate system setting function is executed, the following functions are canceled: workpiece coordinate system shift based on G92, local coordinate system setting based on G52, workpiece coordinate system shift based on origin setting and workpiece coordinate system shift based on manual interrupt.

(4) When a parameter has been used to select the dog-type of first manual reference point return or automatic reference point return after the power has been turned ON, the dog-type reference point return will be executed for the 2nd and subsequent manual reference point returns or automatic reference point returns.

⚠CAUTION

⚠ **If the workpiece coordinate offset amount is changed during automatic operation (including single block operation), the changes will be valid from the next block of the command several blocks later.**

14.5 Machine Coordinate System Selection; G53

Function and purpose

The tool is moved to the assigned position on the basic machine coordinate system by the G53 command and the coordinate commands that follow.

Command format

G53 G00 Xx Zz αα;
G53 G00 Uu Ww ββ;

αα	Additional axis
ββ	Incremental command axis of the additional axis

Detailed description

(1) When the power is turned ON, the basic machine coordinate system is automatically set as referenced to the reference (zero) point return position, which is determined by the automatic or manual reference (zero) point return.

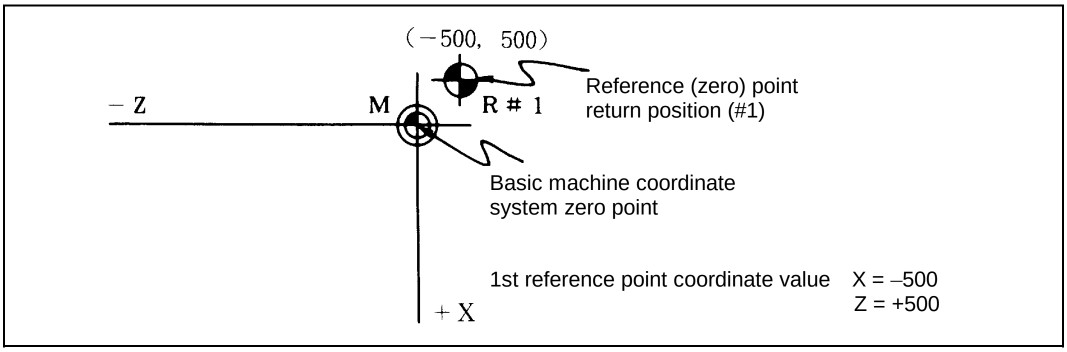
(2) The basic machine coordinate system is not changed by the G92 command.

(3) The G53 command is valid only in the block in which it has been designated.

(4) In the incremental value command mode (U,W,β), the G53 command provides movement with the incremental value in the coordinate system being selected.

(5) The 1st reference point coordinates denote the distance from the basic machine coordinate system zero point to the reference (zero) point return position.

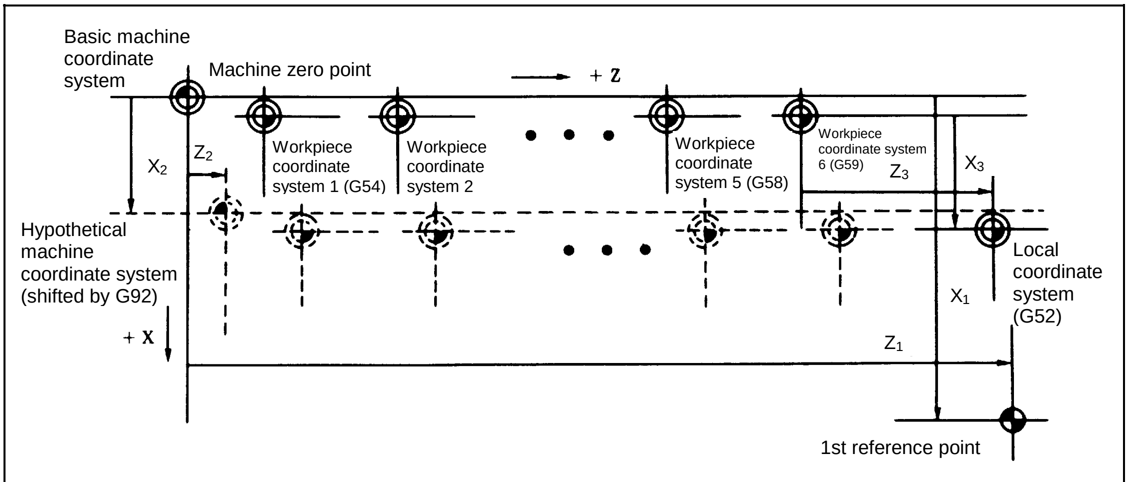
(6) The G53 command always moves at rapid traverse.



14.6 Coordinate System Setting; G92

Function and purpose

This function places the tool at the desired position, and the coordinate system is set by assigning the coordinate system setting command G22 at that position.
This system can be set as desired though normally the X and Y axes are set so that the workpiece center serves as the zero point and the Z axis is set so that the workpiece end serves as the zero point.



Command format

G92 Xx2 Zz2 $\alpha\alpha$2;	
$\alpha\alpha$2	Additional axis

Detailed description

(1) The basic machine coordinate system is shifted by the G92 command, the hypothetical machine coordinate system is created, and at the same time all workpiece coordinate systems 1 to 6 are also shifted.

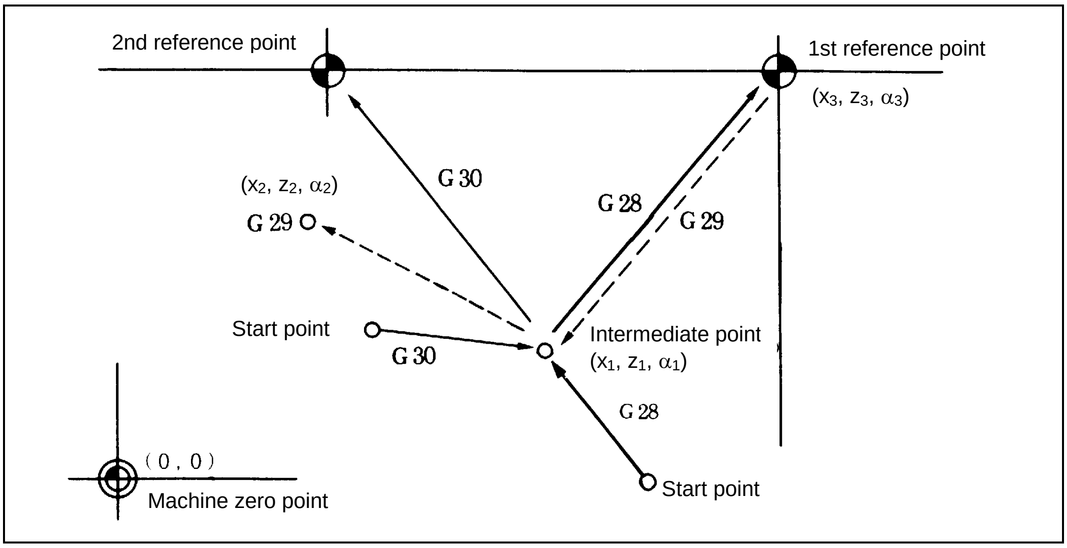
(2) When G92 and S or Q are assigned, the spindle clamp speed is set. (Refer to the section on Spindle Clamp Speed Setting)

14.7 Reference Point Return; G28, G29

Function and purpose

(1) After the commanded axes have been positioned by G0, they are returned respectively at rapid traverse to the first reference (zero) point when G28 is commanded.

(2) By commanding G29, the axes are first positioned independently at high speed to the G28 or G30 intermediate point and then positioned by G0 at the commanded position.



Command format

G28 Xx1 Zz1 $\alpha\alpha$1;	**Additional axis**	**Automatic reference point return**
G29 Xx2 Zz2 $\alpha\alpha$2;	**Additional axis**	**Start position return**
$\alpha\alpha$1/$\alpha\alpha$2	Additional axis	

Detailed description

(1) The G28 command is equivalent to the following:

G00 Xx1 Zz1 $\alpha\alpha1$;
G00 Xx3 Zz3 $\alpha\alpha3$;

In this case, x3, z3 and $\alpha3$ are the reference point coordinates and they are set by parameters as the distance from the basic machine coordinate system zero point.

(2) After the power has been turned ON, the axes which have not been subject to manual reference (zero) point are returned by the dog-type of return just as with the manual type. In this case, the return direction is regarded as the command sign direction. For the second and subsequent returns, the return is made at high speed to the reference (zero) point which was stored at the first time. (The dog-type return can aisle be set by parameter for the second and subsequent times.)

(3) When reference (zero) point return is completed, the zero point arrival output signal is output and also #1 appears at the axis name line on the setting display unit screen.

(4) The G29 command is equivalent to the following:

G00 Xx1 Zz1 $\alpha\alpha1$;
G00 Xx2 Zz2 $\alpha\alpha2$;

} Positioning from the reference point to intermediate point is carried out with independent rapid traverse (non-interpolation type) for each axis.

In this case, x2, z2 and $\alpha2$ are the coordinate value of the G28 or G30 intermediate point.

(5) Program error "P430" results when G29 is executed when automatic reference (zero) point return (G28) is not performed after the power has been turned ON.

(6) The intermediate point coordinate value (x1, z1, $\alpha1$) of the positioning point are assigned by absolute/incremental value commands.

(7) G29 is valid for either G28 or G30 but the commanded axes are positioned after a return has been made to the latest intermediate point.

(8) The tool offset will be temporarily canceled during reference point return unless it is already canceled, and the intermediate point will be the offset position.

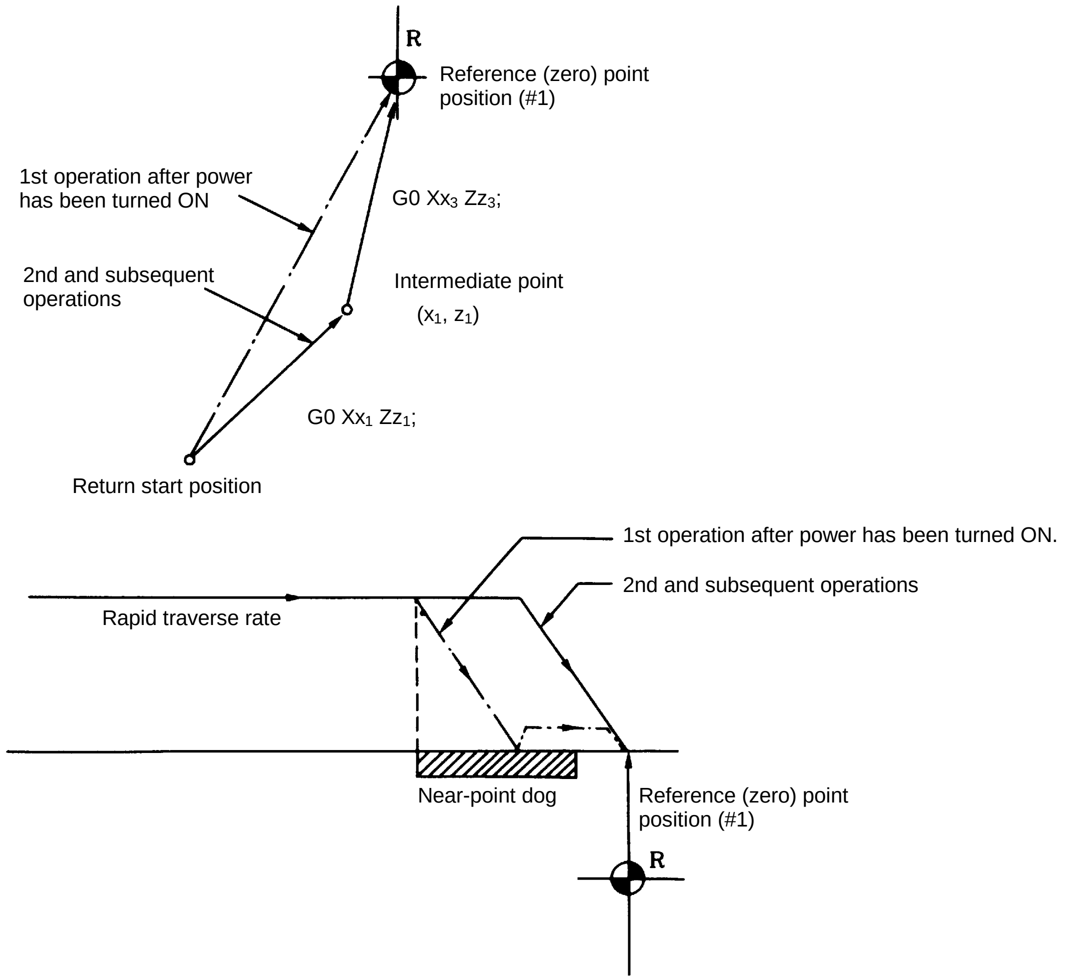
(9) The intermediate point can also be ignored by parameter setting.

(10) Control from the intermediate point to the reference (zero) point is ignored for reference (zero) point return in the machine lock status. The next block is executed when the commanded axis survives as far as the intermediate point.

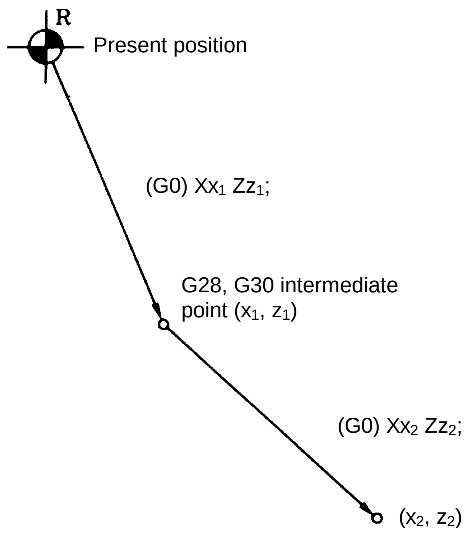
(11) Mirror image is valid from the start point to the intermediate point during reference (zero) point return in the mirror image mode and the tool will move in the opposite direction to that of the command. However, mirror image is ignored from the intermediate point to the reference (zero) point and the tool will move to the reference (zero) point.

Example of program

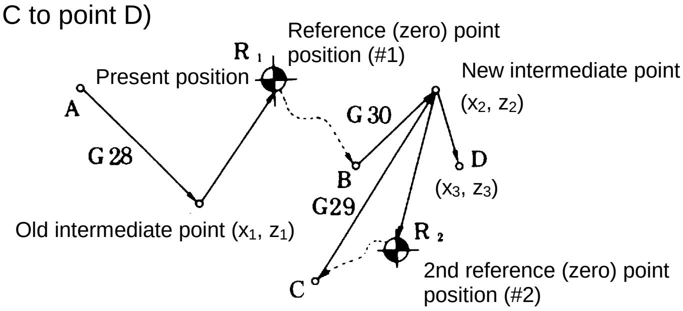
(Example 1) G28 Xx_1 Zz_1;



(Example 2) G29 Xx_2 Zz_2;



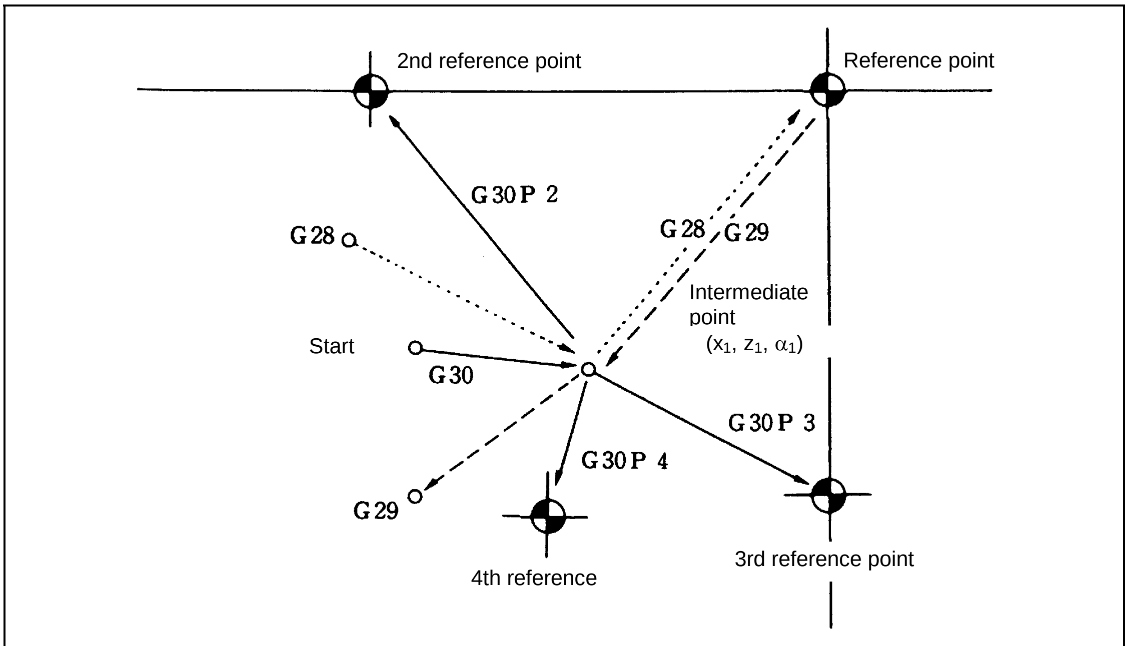
(Example 3) G28 Xx_1 Zz_1;
: (From point A to reference (zero) point)
G30 Xx_2 Zz_2;
: (From point B to 2nd reference (zero) point)
G29 Xx_3 Zz_3;
(From point C to point D)



14.8 2nd, 3rd, and 4th Reference (Zero) Point Return; G30

Function and purpose

The tool can return to the 2nd, 3rd, or 4th reference (zero) point by specifying G30 P2 (P3 or P4).



Command format

G30 P2 (P3, P4) Xx1 Zz1 $\alpha\alpha$1;	
$\alpha\alpha$1	Additional axis

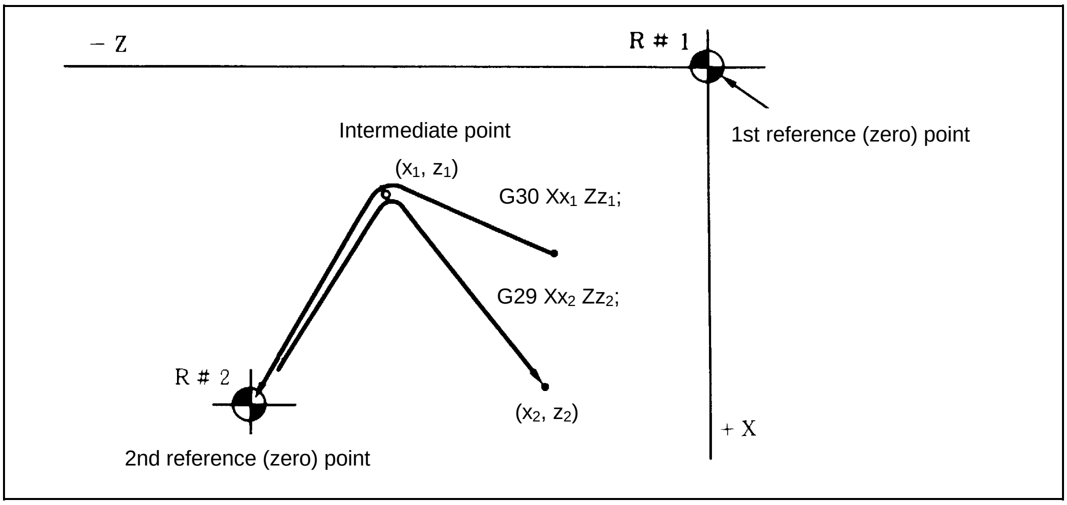
Detailed description

(1) The 2nd, 3rd, or 4th reference (zero) point return is specified by P2, P3, or P4. A command without P or with P0, P1, P5 or a greater P number is ignored, returning the tool to the 2nd reference (zero) point.

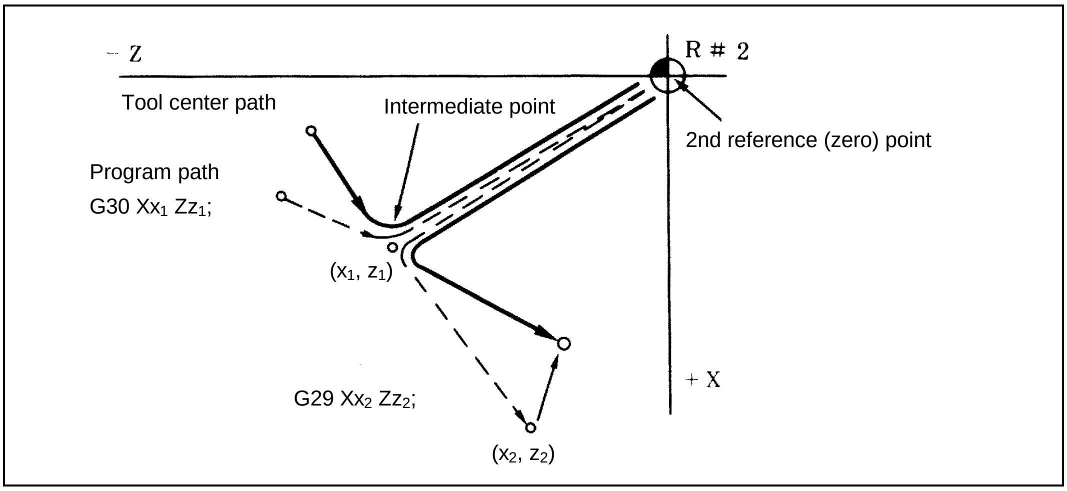
(2) In the 2nd, 3rd, or 4th reference (zero) point return mode, as in the 1st reference (zero) point return mode, the tool returns to the 2nd, 3rd, or 4th reference (zero) point via the intermediate point specified by G30.

(3) The 2nd, 3rd, and 4th reference (zero) point coordinates refer to the positions specific to the machine, and these can be checked with the setting display unit.

(4) If G29 is specified after completion of returning to the 2nd, 3rd, and 4th reference (zero) points, the intermediate point used last is used as the intermediate point for returning by G29.



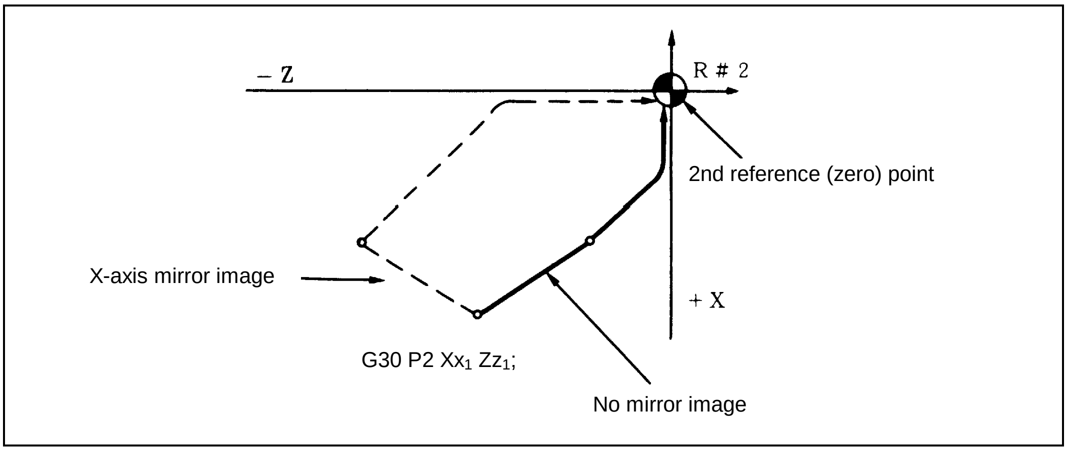
(5) With reference (zero) point return on a plane during compensation, the tool moves without nose R compensation (zero compensation) from the intermediate point as far as the reference (zero) point. With a subsequent G29 command, the tool move without nose R compensation from the reference (zero) point to the intermediate point and it move with such compensation until the G29 command from the intermediate point.



(6) The tool offset amount for the axis involved is canceled temporarily after the 2nd reference (zero) point return.

(7) With 2nd reference (zero) point return in the machine lock status, control from the intermediate point to the reference (zero) point will be ignored. When the commanded axis reaches as far as the intermediate point, the next block will be executed.

(8) With 2nd reference (zero) point return in the mirror image mode, mirror image will be valid from the start point to the intermediate point and the tool will move in the opposite direction to that of the command. However, mirror image is ignored from the intermediate point to the reference (zero) point and the tool moves to the reference (zero) point.



14.9 Reference Point Check; G27

Function and purpose

This command first positions the tool at the position assigned by the program and then, if that positioning point is the 1st reference point, it outputs the reference point arrival signal to the machine in the same way as with the G28 command. Therefore, when a machining program is prepared so that the tool will depart from the 1st reference point and return to the 1st reference point, it is possible to check whether the tool has returned to the reference point after the program has been run.

Command format

G27 Xx1 Zz1 $\alpha\alpha$1 Pp1;

G27	Check command
Xx1 Zz1 $\alpha\alpha$1	Return control axis
Pp1	Check number P1 : 1st reference point check P2 : 2nd reference point check

Detailed description

(1) If the P command has been omitted, the 1st reference point will be checked.

(2) The number of axes whose reference points can be checked simultaneously depends on the number of axes which can be controlled simultaneously.

(3) An alarm results if the final command point is not the reference point.

14.10 Workpiece Coordinate System Setting and Offset; G54 to G59

Function and purpose

(1) The workpiece coordinate systems are for facilitating the programming of workpiece machining in which the reference point of the workpiece to be machined is to serve as the zero point.

(2) These commands enable the tool to move to the positions in the workpiece coordinate system. There are 6 workpiece coordinate systems which are used for programming. (G54 to G59)

(3) Among the workpiece coordinate systems currently selected by these commands, any workpiece coordinate system with coordinates which have been commanded by the present position of the tool is reset. (The "present position of the tool" includes the offset amounts for tool radius, tool length and tool position offset.)

(4) An hypothetical machine coordinate system with coordinates which have been commanded by the present position of the tool is set by this command. (The "present position of the tool" includes the offset amounts for nose R, tool length and tool offset.) (G54, G92)

Command format

(1) Workpiece coordinate system selection (G54 to G59)

G54 Xx1 Zz1 αα1 ;	
αα1	Additional axis

(2) Workpiece coordinate system setting (G54 to G59)

(G54) G92 Xx1 Zz1 αα1 ;	
αα1	Additional axis

Detailed description

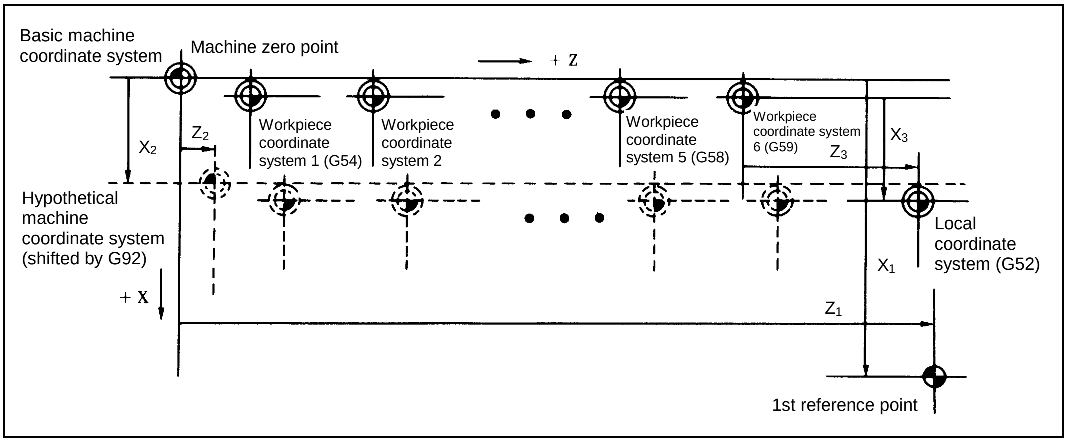
(1) With any of the G54 to G59 commands, the nose R compensation amounts for the commanded axes will not be canceled even if workpiece coordinate system selection is commanded.

(2) The G54 workpiece coordinate system is selected when the power is turned ON.

(3) Commands G54 to G59 are modal commands (group 12).

(4) The coordinate system will move with G92 in a workpiece coordinate system.

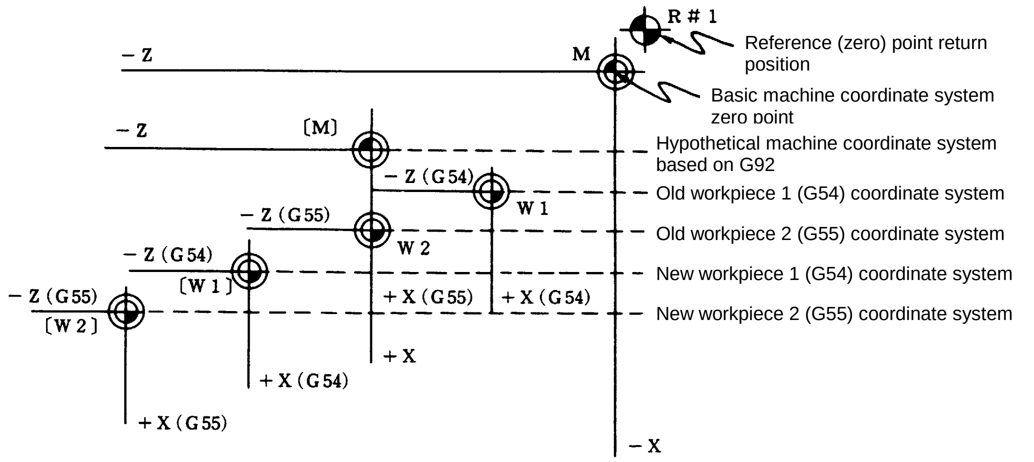
(5) The offset setting in a workpiece coordinate system denotes the distance from the basic machine coordinate system zero point.



(6) The offset settings of workpiece coordinate systems can be changed any number of times. (They can also be changed by G10 L2 Pp1 Xx1 Zz1.)

(7) A new workpiece coordinate system 1 is set by issuing the G92 command in the G54 (workpiece coordinate system 1) mode. At the same time, the other workpiece coordinate systems 2 to 6 (G55 to G59) will move in parallel and new workpiece coordinate systems 2 to 6 will be set.

(8) An hypothetical machine coordinate system is formed at the position which deviates from the new workpiece reference (zero) point by an amount equivalent to the workpiece coordinate system offset amount.



After the power has been turned ON, the hypothetical machine coordinate system is matched with the basic machine coordinate system by the first automatic (G28) or manual reference (zero) point return.

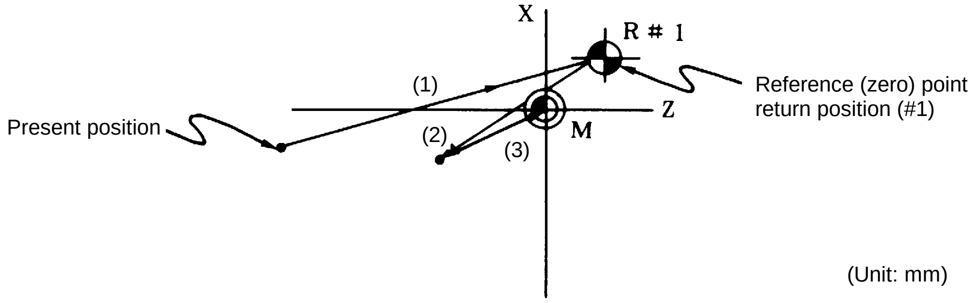
(9) By setting the hypothetical basic machine coordinate system, the new workpiece coordinate system will be set at a position which deviates from that hypothetical basic machine coordinate system zero point by an amount equivalent to the workpiece coordinate system offset amount.

(10) When the first automatic (G28) or manual reference (zero) point return is completed after the power has been turned ON, the basic machine coordinate system and workpiece coordinate systems are set automatically in accordance with the parameter settings.

Example of program

(Example 1)

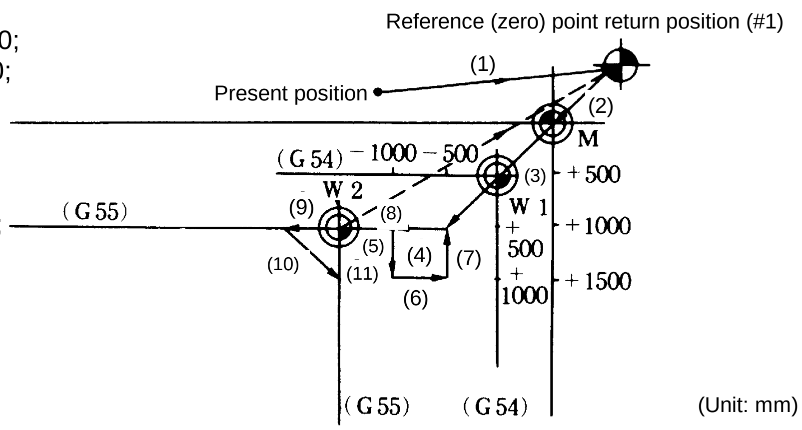
(1) G28 X0 Z0;
(2) G53 X+500 Z–1000;
(3) G53 X0 Z0;



When the 1st reference point coordinate is zero, the basic machine coordinate system zero point and reference (zero) point return position (#1) will coincide.

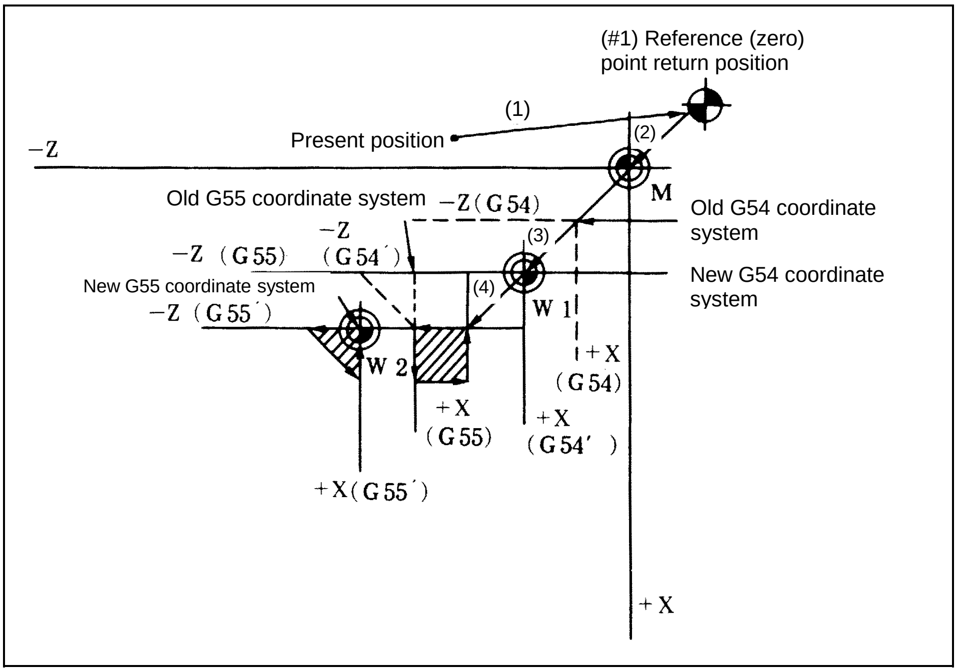
(Example 2)

(1) G28 X0 Z0;
(2) G00 G53 X0 Z0;
(3) G54 X+500 Z–500;
(4) G01 W–500 F100;
(5) U+500;
(6) W+500;
(7) U–500;
(8) G00 G55 X0 Z0;
(9) G01 Z–500 F200;
(10) X+500 Z0;
(11) G28 X0 Z0;



(Example 3) When workpiece coordinate system G54 has deviated (+500, –500) in example 2 (It is assumed that (3) to (10) in example 2 have been entered in subprogram O1111.)

(1) G28 X0 Z0;	
(2) G00 G53 X0 Z0;	(Not required when there is no basic machine coordinate system offset.)
(3) G54 X+500 Z–500;	Amount by which workpiece coordinate system deviates
(4) G92 X0 Z0;	New workpiece coordinate system is set.
(8) M98 P1111;	



(Note) The workpiece coordinate system will deviate each time steps (3) to (5) are repeated. The reference point return (G28) command should therefore be issued upon completion of the program.

⚠CAUTION

⚠ **If the workpiece coordinate system offset amount is changed during single block stop, the changes will be valid from the next block.**

14.11 Local Coordinate System Setting; G52

Function and purpose

The local coordinate systems can be set independently on the G54 to G59 workpiece coordinate systems using the G52 command so that the commanded position serves as the programmed zero point.
The G52 command can also be used instead of the G92 command to change the deviation between the zero point in the machining program and the machining workpiece zero point.

Command format

G54 (G54 to G59) G52 Xx1 Zz1 ;

Detailed description

(1) The G52 command is valid until a new G52 command is issued and the tool does not move. This command comes in handy for employing another coordinate system without changing the zero point positions of the workpiece coordinate systems (G54 to G59).

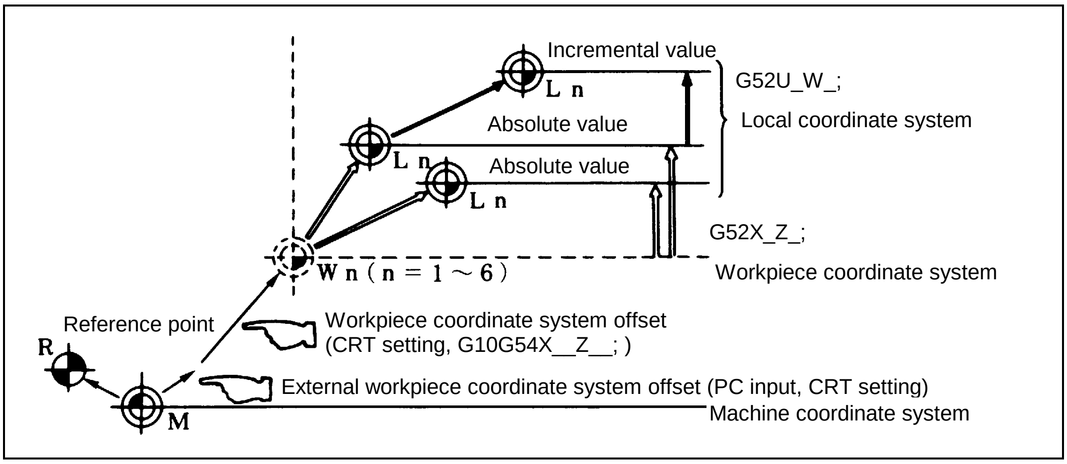
(2) The local coordinate system offset is cleared after power-ON.

(3) The local coordinate system is canceled by (G54 to G59) G52 X0 Z0;.

(4) Coordinate commands in the absolute value mode cause the tool to move to the local coordinate system position.

(5) RESET1 does not clear the local coordinate system offset.

(6) RESET2 or RESET&REWIND clears the local coordinate system offset.

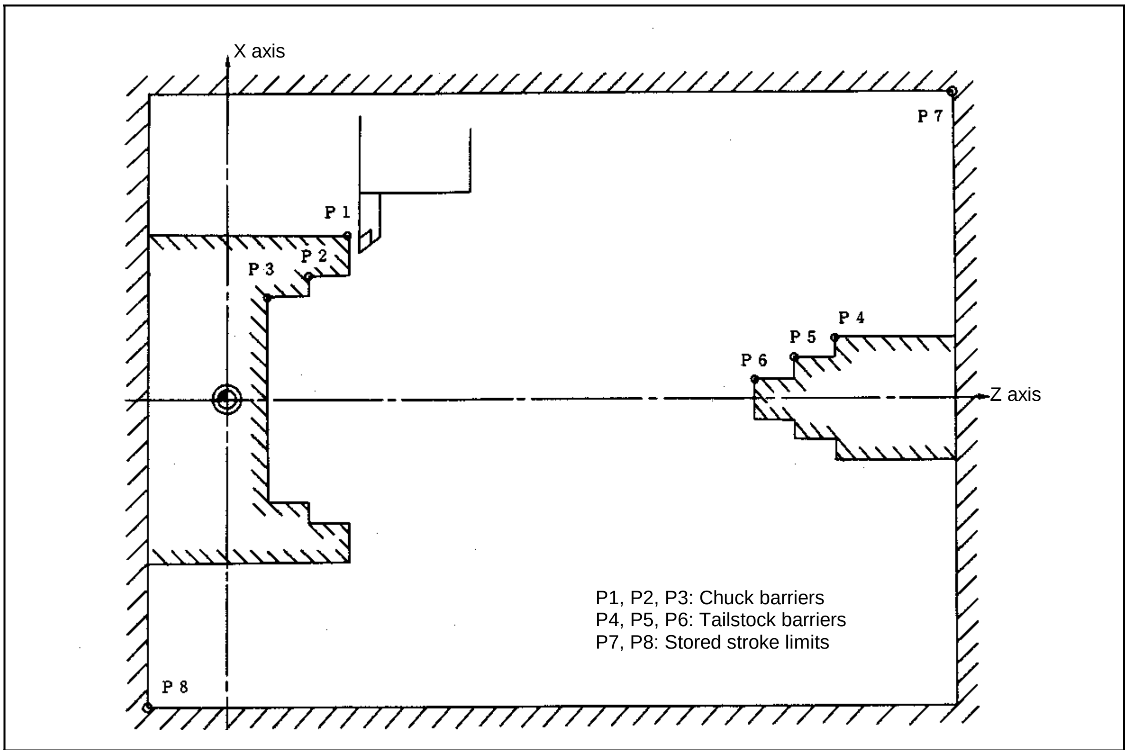


15. PROTECTION FUNCTIONS

15.1 Chuck Barriers/Tailstock Barriers

Function and purpose

The chuck barriers and tailstock barriers serve to limit the movement range of the tool nose points and they are provided in order to prevent collisions with the chuck or tailstock which may arise due to programming errors. The tool is automatically stopped at the barrier limit with movement commands which exceed the range set by parameter.



Command format

G22;	**Barriers valid**
G23;	**Barriers invalid**

Detailed description

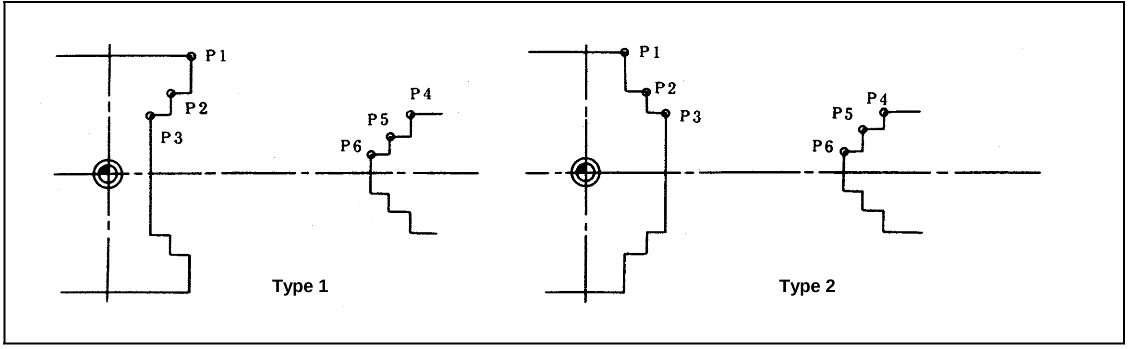
(1) When the machine is about to go beyond the movement range, it stops and at the same time an alarm is displayed. This alarm is released by NC resetting.

(2) This function is invalid during the machine lock mode, but is valid during system synchronized machine lock mode.

(3) This function is valid when all the axes, for which the chuck and tailstock barriers have been set, have completed their return to the reference points.

(4) When the stored stroke check function is available and the stored stroke limit area has been set, the chuck/tailstock barrier function is simultaneously valid with the stored stroke check function.

Chuck barrier/tailstock barrier setting



(1) For both the chuck and tailstock barriers, 3 points can be input as parameters and these are set with the machine coordinates. Points P1, P2 and P3 represent the chuck barriers; points P4, P5 and P6 represent the tailstock barriers.

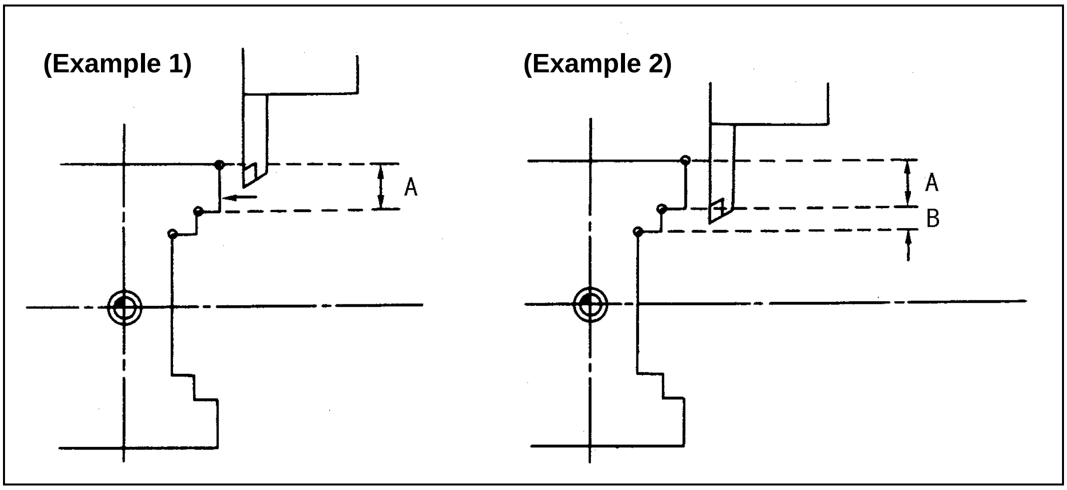
(2) The barrier area is symmetrical to the Z axis and when the barrier point P_ X-axis coordinates have a minus value, the sign is turned into a plus and the coordinates are checked.
The absolute values of the X-axis coordinates for each barrier point must be set as follows:

P1 ≥ P2 ≥ P3, P4 ≥ P5 ≥ P6

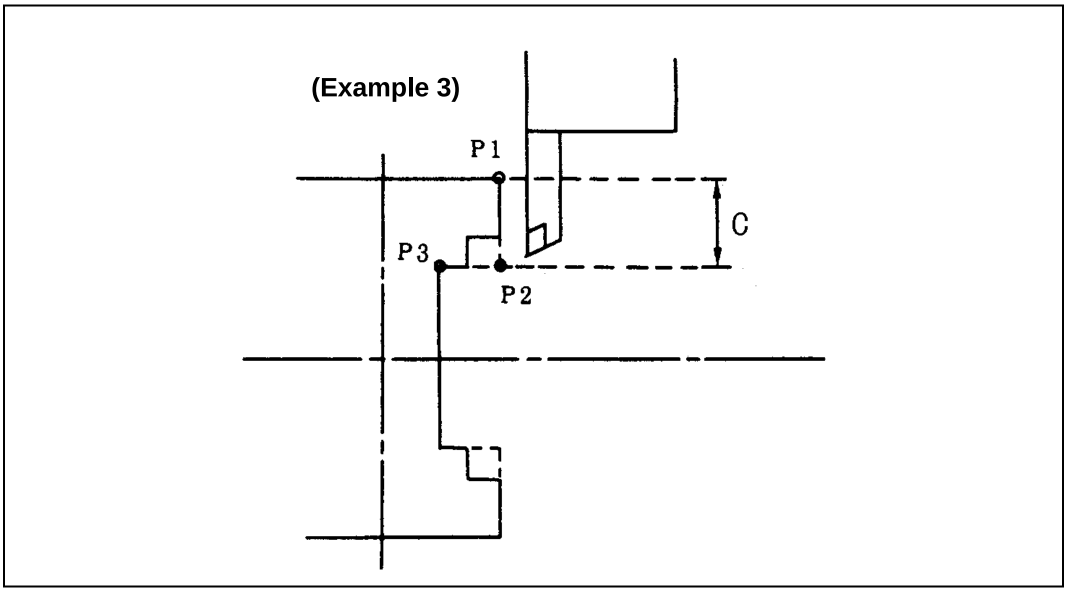
(Z-axis coordinates do not need to conform to this sequence.)

Restrictions for chuck barriers/tailstock barriers

(1) The following points must be borne in mind since there is only one check point from the tool for the chuck barriers/tailstock barriers.
When, in the examples given below, the tool nose width offset amount is set and the tool is moved in the direction of the arrow in the figure in order to conduct the check at the hypothetical tool nose point, the tool will automatically stop at the barrier limit because the check point is in range A of (Example 1). However, in (Example 2), there is a danger that the tool and chuck will collide in range A since the check point is in range B.



In order to avoid collision, the tool can be stopped at the barrier limit when barrier points P1, P2 and P3 are set so that the check point is in range C, as shown in (Example 3).



(2) When the tool enters inside the barrier area and the alarm occurs, the NC system is first reset to release the alarm and then the tool can be moved in the opposite direction to that in which it was moving before.

(3) There is no barrier area for an axis without the reference point return function. Consequently, there is no barrier alarm for such an axis.

(4) When the barriers have been made valid after the tool has entered the barrier area during cancellation, an alarm occurs as soon as any attempt is made by the tool to move. In a case like this, first release the alarm by resetting the NC system and issue the G23 command so that the tool will avoid the area or change the barrier point settings.

16. MEASUREMENT SUPPORT FUNCTIONS

16.1 Skip Function; G31

Function and purpose

When the skip signal is input externally during linear interpolation based on the G31 command, the machine feed is stopped immediately, the remaining distance is discarded and the command in the following block is executed.

Command format

G31 Xx/Uu Zz/Ww Yy/Vv Ff ;	
x, z, y, u, w, v	Axis coordinate value; they are commanded as absolute or incremental values.
f	Feedrate (mm/min)

Linear interpolation can be executed using this function. If the skip signal is input externally while this command is being executed, the machine will stop, the remaining commands will be canceled and operation will be executed from the next block.

Detailed description

(1) If Ff is commanded as the feedrate, command speed f will apply; if it not commanded, the "G31 skip" set in the parameter will serve as the feedrate. In either case, the F modal will not be updated.

(2) The machine will not automatically accelerate and decelerate with the G31 block.

(3) Override is invalid with the G31 command and when machine lock is OFF, and it is fixed at 100%. Dry run is also invalid. The stop conditions (feed hold, interlock, override zero and stroke end) are valid. External deceleration is also valid.

(4) The G31 command is unmodal and so it needs to be commanded each time.

(5) If the skip signal is input during G31 command start, the G31 command will be completed immediately.

(6) When a skip signal has not been input until the G31 block completion, the G31 command will also be completed upon completion of the movement commands.

(7) When the G31 command is issued during nose R compensation, program error "P608" will result.

(8) When there is no F command in the G31 command and the parameter speed is also zero, program error "P603" will result.

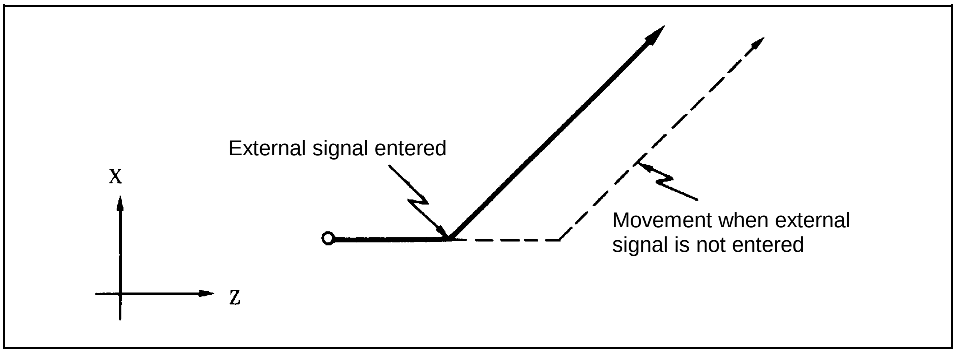
(9) With machine lock is ON, the skip signal will be ignored and execution will continue as far as the end of the block.

(10) The "G31 skip conditions" are set on the parameter screen.
The "G31 skip speed" is set on the machine manufacturer parameter screen.

Execution of G31

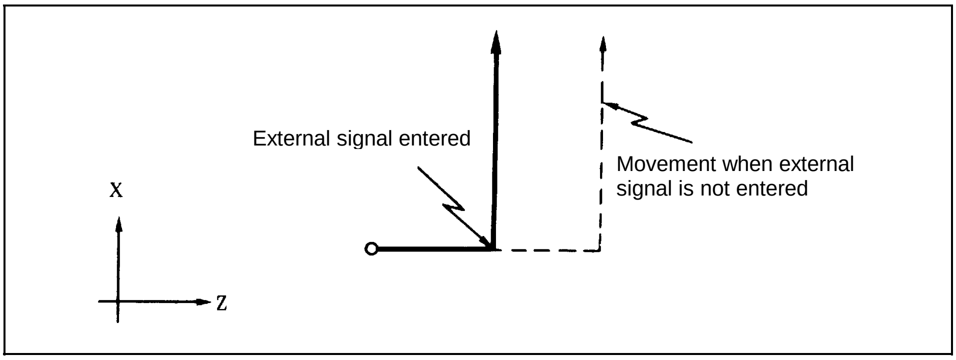
(Example 1) When the next block is an incremental value command

G31 Z1000 F100;
G01 U2000 W1000;



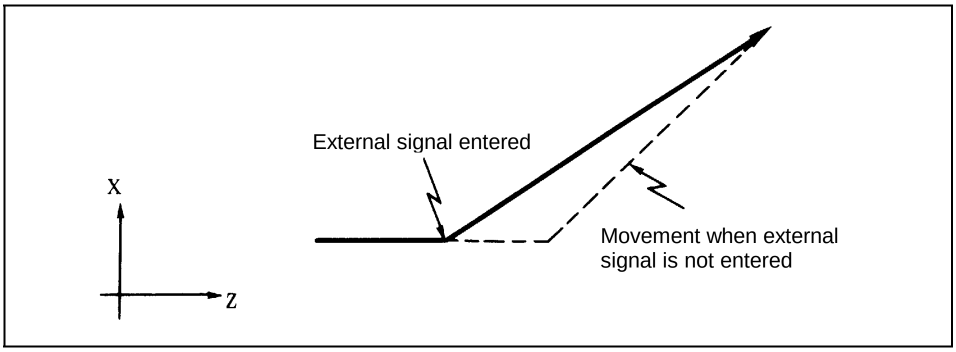
(Example 2) When the next block is a one axis movement command with absolute value

G31 Z1000 F100;
G01 X1000;



(Example 3) When the next block is a two axes movement command with absolute value

G31 Z1000 F100;
G01 X1000 Z2000;



Detailed description (Readout of skip coordinates)

As the skip signal inputting coordinate positions have been stored in the system variables #5061 (1st axis) to #5063 (3rd axis), it is possible to use them with user macro.

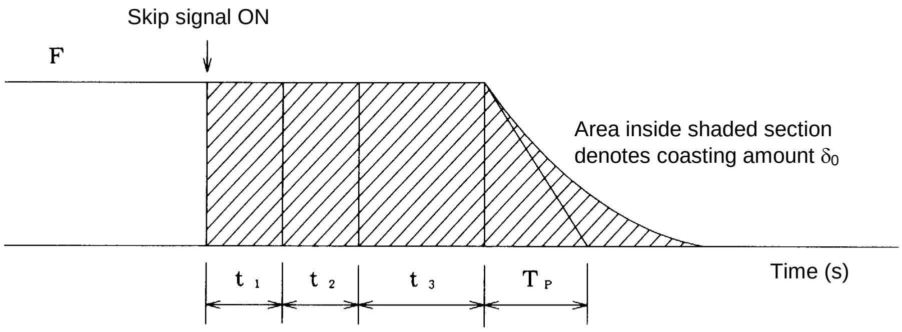
```
    ~
G00 X–100.;
G31 X–200. F60;     ―― Skip command
#101 = #5061        ―― Skip signal input coordinate values (workpiece
    ~                  coordinate system) are readout to #101.
```

Detailed description (G31 Coasting)

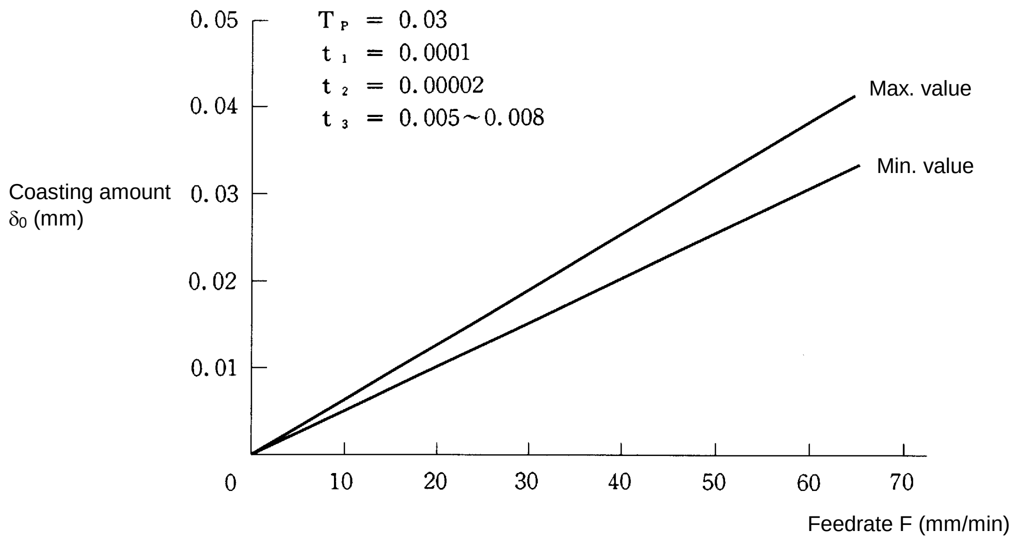
The coasting amount from when the skip signal is input during the G31 command until the machine stops differs according to the "G31 skip speed" and the F command in G31.
The coasting amount can be calculated from the following formula.

$$\delta_0 = \underbrace{\frac{F}{60} * t_1}_{\delta_1} + \underbrace{\frac{F}{60} * (t_2 + t_3)}_{\delta_2} + \underbrace{\frac{F}{60} * Tp}_{\delta_3}$$

δ_0	:	Coasting amount (mm)
F	:	G31 skip speed (mm/min)
Tp	:	Position loop time constant (s) = (position loop gain)
t_1	:	Sensor response delay time in the machine side (s)
t_2	:	Response delay time for NC internal skip signal input (0.00002s)
t_3	:	Response delay time for NC internal skip signal output (0.005 to 0.008s)
δ_1	:	Coasting amount due to sensor response delay time (mm)
δ_2	:	Coasting amount due to NC internal response delay time (mm)
δ_3	:	Coasting amount due to position loop time constant (mm)



Stop pattern with skip signal input



Relationship between coasting amount and feedrate (example)

Detailed description (Skip coordinate readout error)

Macro variables (#5061 to) can be used to readout the skip coordinate value determined by the following expression:
Skip coordinate value = Position where skip signal is ON + ε_1 + ε_2

$$\varepsilon_1 = \frac{F}{60} * (t_1 + t_2) \text{ (mm)}$$

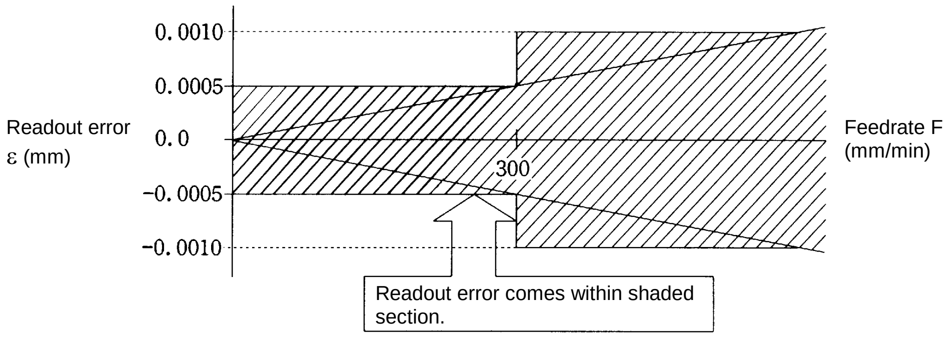
$$\varepsilon_2 = \pm \frac{F}{60} * 0.000111 \text{ (mm)}$$

ε_1	:	Response delay error (mm)
ε_2	:	Readout error (mm)
F	:	G31 skip speed (mm/min)
t_1	:	Sensor response delay time in the machine side (s)
t_2	:	Response delay time for NC internal skip signal input (0.00002s)

(Note 1) Response delay error ε_1 can be compensated. However, it cannot be compensated if the sensor response delay time t_1 in the machine side fluctuates.

(Note 2) Readout error ε_2 is an operation error in the NC and cannot be compensated.

(Note 3) The minimum value of readout error ε_2 is ± 0.0005mm.



Relationship between readout error and feedrate

Examples of coasting amount compensation

(1) Compensating for skip signal input coordinate value

```
#110 = Skip feedrate ;
#111 = Response delay time t1;
```

```
        ≀
G31 X100. F100;        Skip command
G04;                   Machine stop check
#101 = #5061;          Skip signal input coordinate readout
#102 = #110*#111/60;   Coasting amount based on response delay time
#105 = #101–#102;      Skip signal input coordinates
        ≀
```

(2) Compensating for workpiece coordinate value

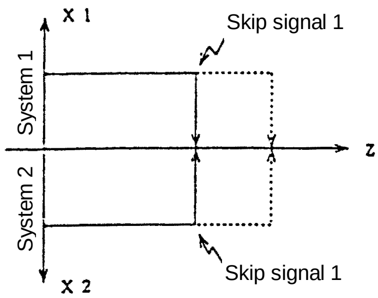
```
#110 = Skip feedrate ;
#111 = Response delay time t1;
#112 = Position loop time constant Tp;
```

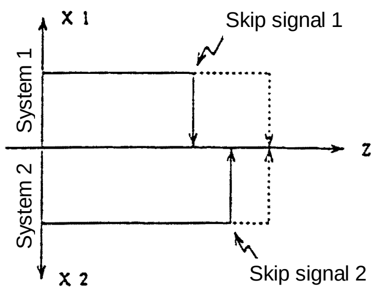
```
        ≀
G31 X100. F100;        Skip command
G04;                   Machine stop check
#101 = #5061;          Skip signal input coordinate readout
#102 = #110*#111/60;   Coasting amount based on response delay
time
#103 = #110*#112/60;   Coasting amount based on position loop time
                       constant
#105 = #101–#102–#103; Skip signal input coordinates
        ≀
```

16.2 Multi-step Skip Function; G31

Function and purpose

During linear interpolation by the skip command (G31), skipping can be executed according to the conditions of the skip signal command P1.
With separate systems, when multi-step skip functions are commanded simultaneously, if the input skip signals are the same, the simultaneous skip operation is executed. If the input skip signals are different, the skip operation is executed according to their respective skip signals. The skip operation is the same as the normal skip command (without G31 P command.)

<table>
<tr>
<td>

When both system 1 and system 2 have the same skip signal</td>
<td>

When system 1 and system 2 have separate skip signals</td>
</tr>
</table>

During the dwell command (G04) , the remaining dwell time is canceled with the skip conditions set by the parameters, and the next block is executed. In the same manner, in the speed dwell the remaining speed is also canceled, and the next block is executed.

Command format

G31 Xx Zz αα Pp Ff ;	
Xx Zz αα	Command format axis coordinate word and target coordinate value
Pp	Skip signal command (see table below)
Ff	Feedrate (mm/min)

(1) The skip speed is issued with command speed F. Note that the F modal is not updated.

(2) The skip signal command is issued with the skip signal command P. P is commanded in the range of 1 to 255. When the command range is exceeded, a program error will occur.

Skip signal command	Valid skip signals							
P	8	7	6	5	4	3	2	1
1								○
2							○	
3							○	○
4						○		
5						○		○
6						○	○	
7						○	○	○
8					○			
~	~	~	~	~	~	~	~	~
253	○	○	○	○	○	○		○
254	○	○	○	○	○	○	○	
255	○	○	○	○	○	○	○	○

(Note) The number of skip signal points differ according to the machine type.

16.3 Automatic Tool Length Measurement; G37

Function and purpose

These functions issue the command values from the measuring start position as far as the measurement time, move the tool in the direction of the measurement position, stop the machine once the tool has arrived at the sensor, cause the NC system to calculate automatically the difference between the coordinate values at that time and the coordinate values of the commanded measurement position and provide this difference as the tool offset amount.
When offset is already being applied to a tool, it moves the tool toward the measurement position with the offset still applied, and if a further offset amount is generated as a result of the measurement and calculation, it provides further offset of the present offset amount.
If there is only one type of offset amount, this is automatically compensated for; if there is distinction between a tool length offset amount and wear offset amount, the wear amount is automatically compensated for.

Command format

G37 α_ R_ D_ F_;

α	Measuring axis address and coordinates of measurement position ···· X, Y, Z
R	This commands the distance between the measurement position and point where the movement is to start at the measuring speed. (Radial value fixed/incremental value)
D	This commands the range within which the tool is to stop. (Radial value fixed/incremental value)
F	This commands the measuring feedrate.

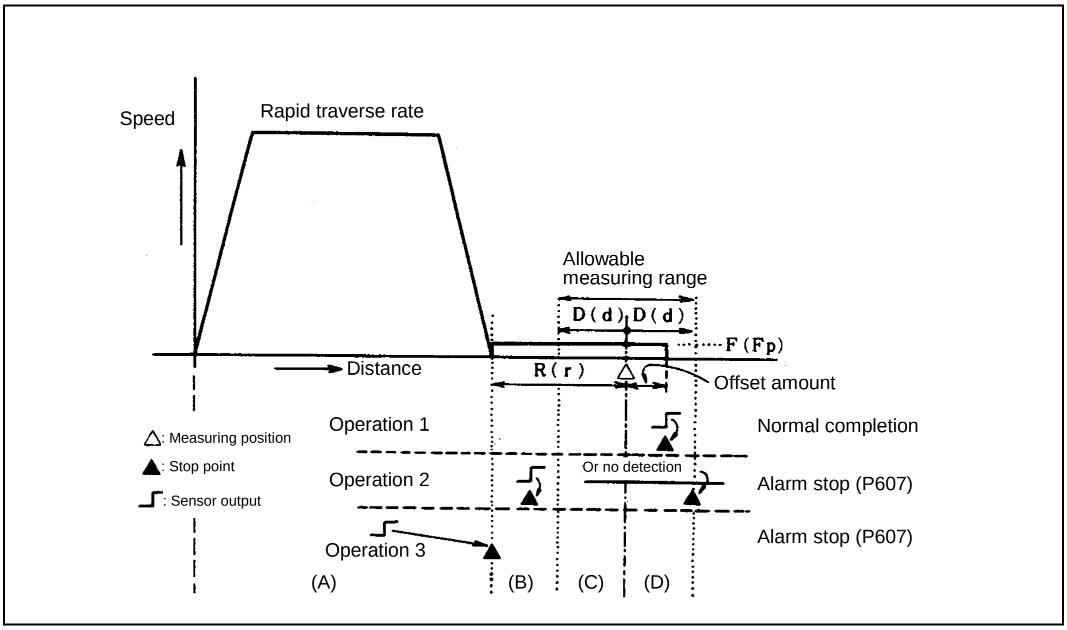
When R_, D_ or F_ is omitted, the value set in the parameter is used instead.

<Parameter> ("T-leng measure" on Setup param screen)

• Auto TLM	speed (Measuring feedrate (Fp))	0 to 60000 [mm/min]
•	zone r (Deceleration range r)	0 to 99999.999 [mm]
•	zone d (Measurement range d)	0 to 99999.999 [mm]

Detailed description

(1) Operation with G37 command



(2) The sensor signal (measuring position arrival signal) is used in common with the skip signal.

(3) The feedrate will be 1mm/min if the F command and parameter measurement speed are 0.

(4) During the synchronous feed mode, the tool moves by synchronous feed [mm/rev].

(5) An updated offset amount is valid unless it is assigned from the following T command of the G37 command.

(6) Excluding the corresponding values at the PLC side, the delay and fluctuations in the sensor signal processing range from 0 to 0.2ms at the NC side only.
As a result, the measuring error shown below is caused.
Maximum measuring error [mm] = Measuring speed [mm/min] × 1/60 × 0.2 [ms]/1000

(7) The machine position coordinates at that point in time are read by sensor signal detection, and the machine will overtravel and stop at a position equivalent to the servo droop.
Maximum overtravel [mm] = Measuring speed [mm/min] × 1/60 × 1/Position loop gain [1/s]
The standard position loop gain is 33 [1/s].

Precautions

(1) Program error "P600" results when G37 is commanded by an NC system which is not provided with the tool length measurement option.

(2) Program error "P604" results when no axis has been commanded in the G37 block or when two or more axes have been commanded.

(3) If the T code is not commanded in the G37 block, the program error "P605" will occur. Note that if the last one digit or last two digits of the T code is 0, the program error "P606" will occur.

(4) Program error "P606" results when the T code is not commanded prior to the G37 block. Even when the T code has been commanded, program error "P606" occurs when the last one or two digits of the T code are zero.

(5) Program error "P607" results when the sensor signal was input outside the allowable measuring range or when the sensor signal was not detected even upon arrival at the end point. However, with operation 3 in the example of the above figure, measurement is considered to be normal when area (B) is not present.

(6) When a manual interrupt is applied while the tool is moving at the measuring speed, a return must be made to the position prior to the interrupt and then operation must be resumed.

(7) The data commanded in G37 or the parameter setting data must meet the following conditions:

$$|\text{Measurement point – start point}| > \text{R command or parameter r} > \text{D command or parameter d}$$

(8) When the D command and parameter d in (7) above are zero, operation will be completed normally only when the commanded measurement point and sensor signal detection point coincide. Otherwise, program error "P607" will result.

(9) When the R and D commands as well as parameters r and d in (7) above are all zero, program error "P607" will result regardless of whether the sensor signal is present or not after the tool has been positioned at the commanded measurement point.

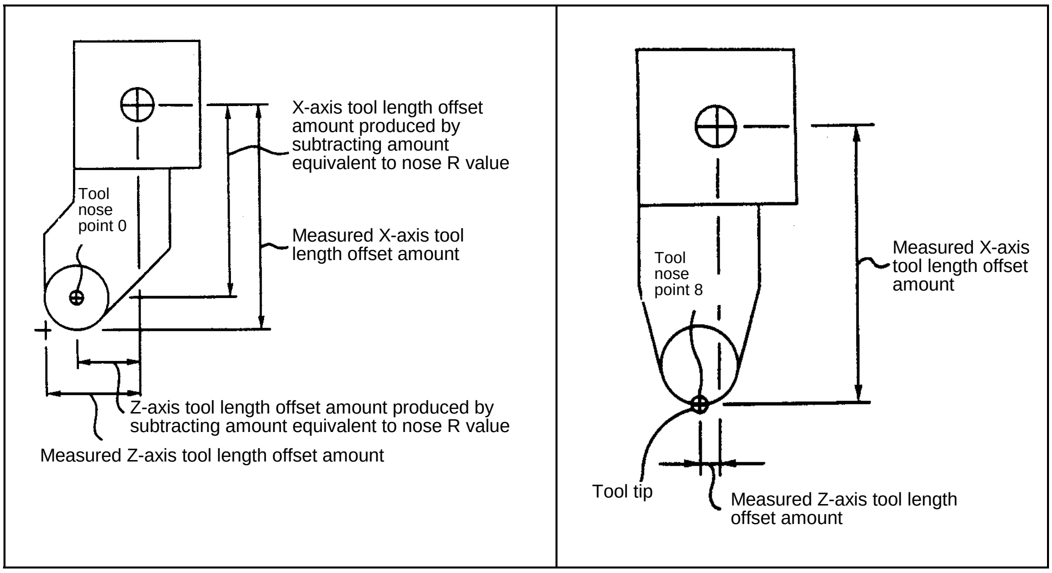
(10) When the measuring command distance is less than the allowable measuring range, the tool will be confined to the allowable measuring range in each case.

(11) When the measuring command distance is less than the measuring speed movement distance, the tool will move at the measuring speed in each case.

(12) When the allowable measuring range is greater than the measuring speed movement distance, the tool will move across the allowable measuring range at the measuring speed.

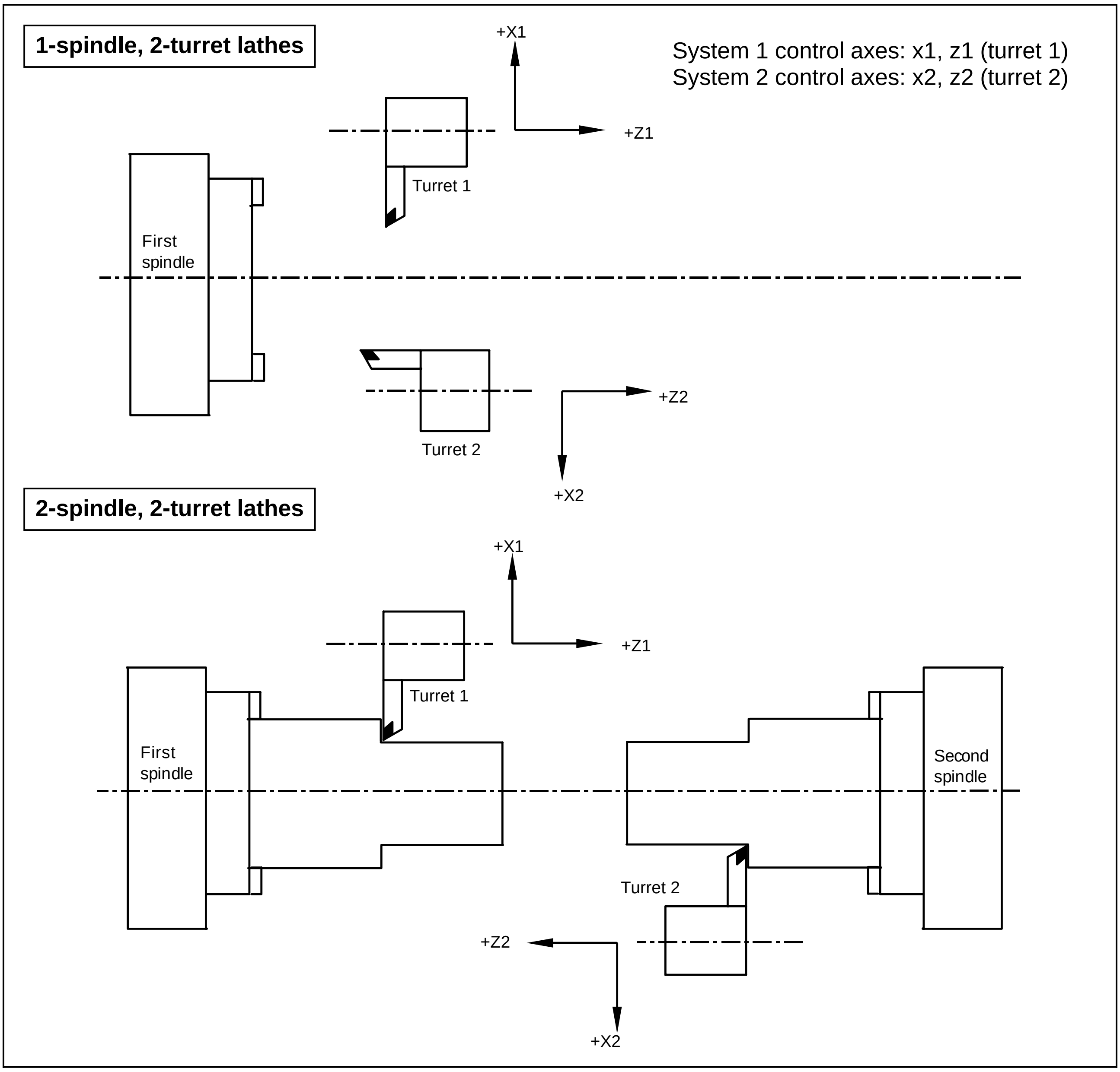
(13) The nose R compensation must be canceled before the G37 command is issued.

(14) Even when the nose R compensation option has been provided, the nose R value and tool nose point numbers will not be considered and the tool length offset amount will be calculated.
When the tool nose point numbers are to be set to 0, an amount equivalent to the nose R value should be deducted from the measured tool length offset amount.
When the tool nose point numbers (tool nose shape) are 5, 6, 7 and 8, the tool length should be measured at the tip of the tool.



17. MULTI-AXIS, MULTI-SYSTEM COMPOUND CONTROL FUNCTIONS

Examples of target lathes



The system controls two or more moving components (turrets, additional axes, etc.) simultaneously and independently as shown in the figures above.

17. MULTI-AXIS, MULTI-SYSTEM COMPOUND CONTROL FUNCTIONS

Basics of multi-system programming

As multiple systems (turrets) move independently, the machining programs for the systems can be configured simply by designating the system numbers at the head of the programs and then by proceeding in the same way as for a 2-axis (or 3-axis) lathe. The synchronizing functions are used if the movements of the multiple systems need to be synchronized.

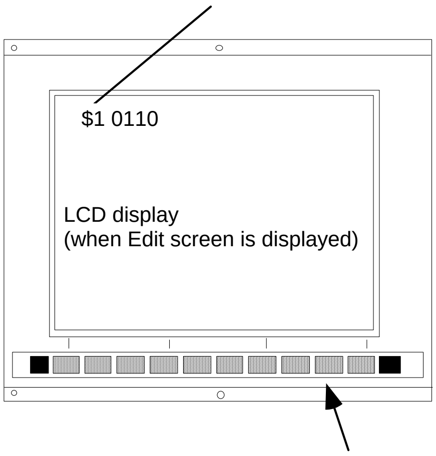
(1) Preparing multi-system programs

A multi-system program uses the same program number for control purposes so only one tape is used. Searches during operations can be executed simply by designating the program number once.

```
O110;   ← Program number
$1;     ← System 1 program designation code
  :   ┐
  :   ├ System 1 program
  :   ┘
$2;     ← System 2 program designation code
  :   ┐
  :   ├ System 2 program
  :   ┘
%
```

System 1 designation	$1;
System 2 designation	$2;
System 3 designation	$3;
:	:

Multi-system programs are prepared after the system designation codes that indicate that the systems have been inserted at the head of the programs. Refer to the manuals issued by the machine manufacturer to confirm which turret is allocated to which system.
When the machining programs are input on the editing screen of the NC setting display unit, the system select menu keys are used to designate the systems. This also applies to MDI operation programs.



(2) System designation codes

System 1 machining program designation: $1;
System 2 machining program designation: $2;
System 3 machining program designation: $3;
: :

(a) Make the system designation code an independent command. If it is designated with other commands, that data will not be valid.

(b) The system designation code commands can be arranged in any order.

(c) When the system designation codes have been omitted, the program will be saved in the memory as the system 1 program.

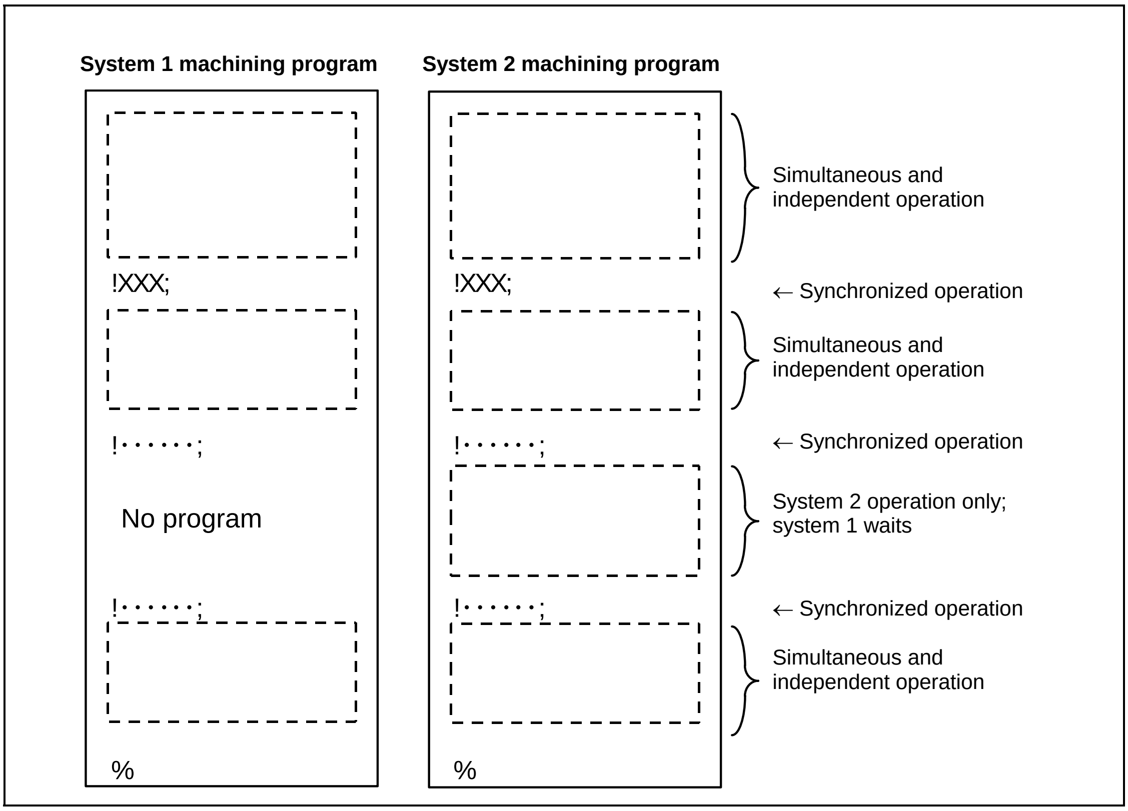
⚠CAUTION

⚠ When programming a multi-system, carefully observe the movements caused by other systems' programs.

17.1 Synchronizing Operation between Systems

Function and purpose

The multi-axis, multi-system compound control NC system can simultaneously run multiple machining programs independently. The synchronizing-between-systems function is used in cases when, at some particular point during operation, the operations of systems 1 and 2 are to be synchronized or in cases when the operation of only one system is required.

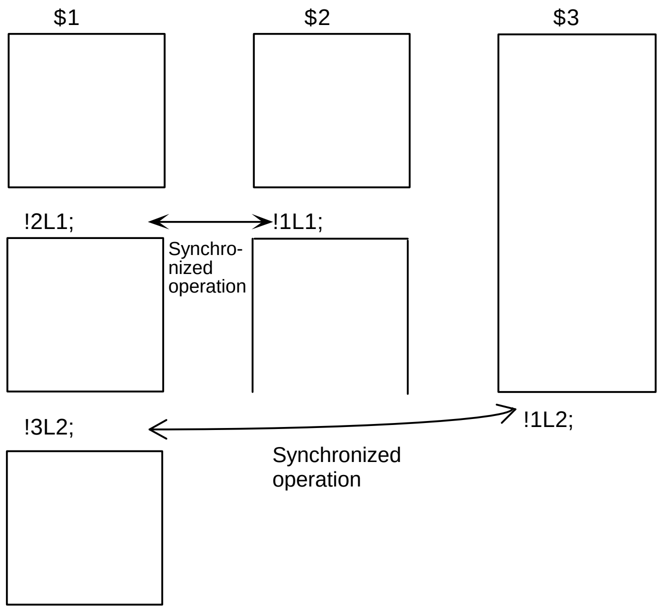


Command format

(1) Command for synchronizing with system n

!nLl;

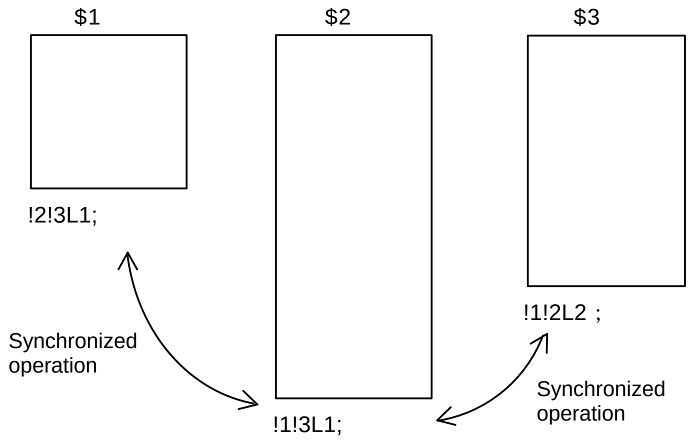
n : System number
l : Synchronizing number 01 to 9999



(2) Command for synchronizing among three systems

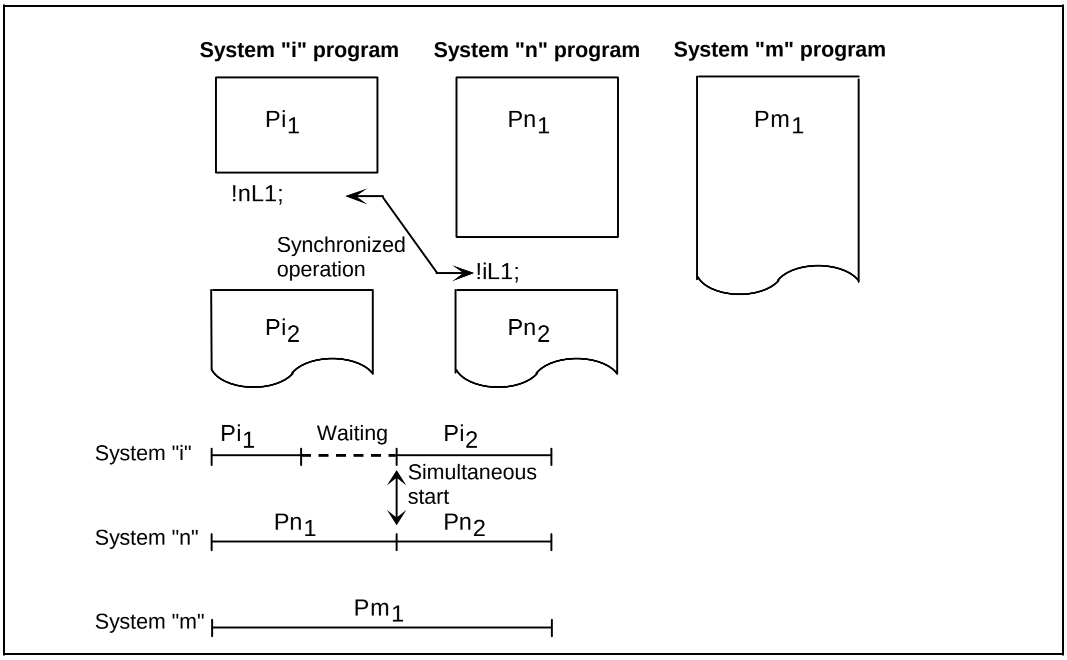
!n!m･･･Ll;

n, m : System number $n \neq m$
l : Synchronizing number 01 to 9999

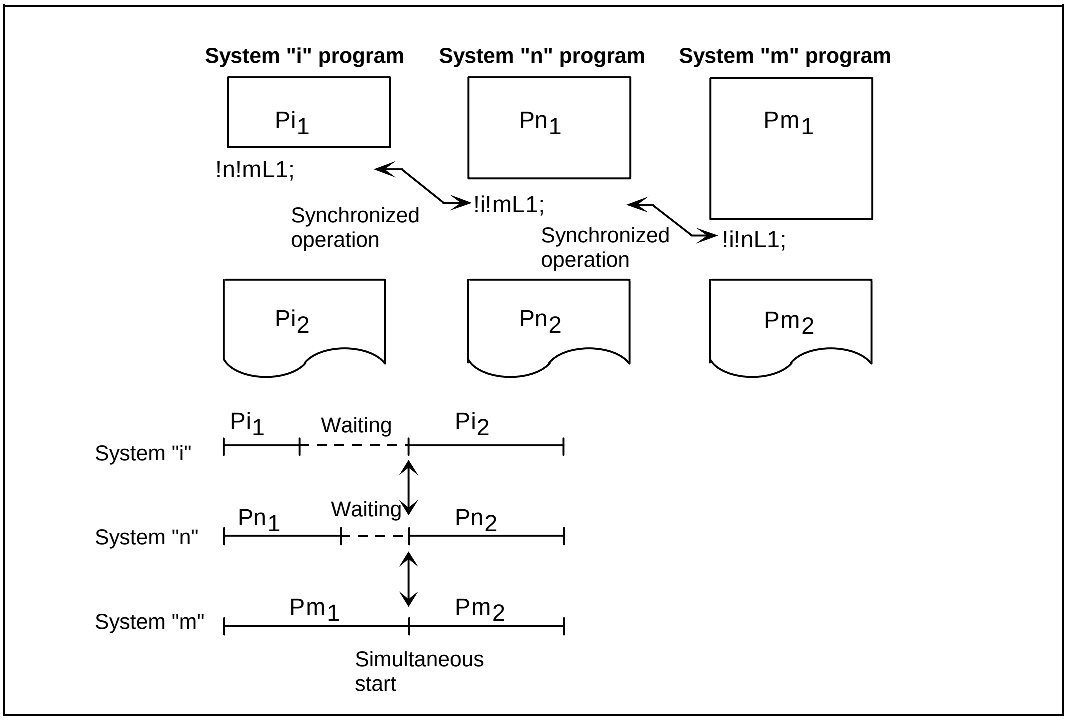


Detailed description

(1) When the !nLl code is issued from the system "i" program, the operation of that program will wait until the !iLl code is issued from the system "n" program.
When the !iLl code is issued, the programs of both systems "i" and "n" will start running simultaneously.



(2) Synchronizing among three systems is as follows. When the !n!mLl command is issued from the system "i" program, the system "i" program operation will wait until the !i!mLl command is issued from the system "n" program and the !i!nLl command is issued from the system "m" program.
When the synchronizing commands are issued, "i", "n" and "m" programs will start operating simultaneously.



(3) A program error (P35) will occur if an illegal system number is commanded for the synchronization command, or if a decimal point is added to the system number issued in the synchronization command.

(Example) !2 → Program error (P35) will occur.

(4) When an M, S or T command or movement command has been issued with a synchronizing command, it will be executed after synchronizing.

(Example) !2L5 G00 X100.;
G00 X100. is executed after synchronization with the second system.

(5) Synchronizing is done only while the system to be synchronized is operating automatically. If this is not possible, the synchronizing command will be ignored and operation will advance to the next block.

(6) The L command is the synchronizing identification number. The same numbers are synchronized but when they are omitted, the numbers are handled as L0.

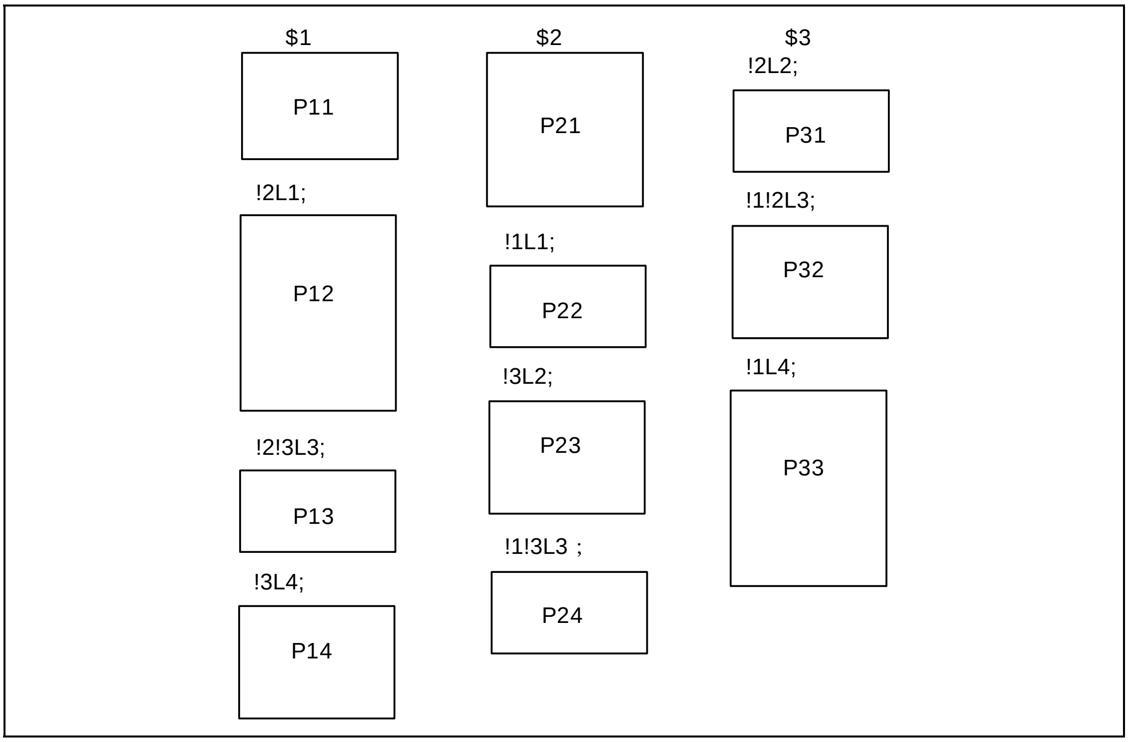
(7) The synchronizing command designates the number of the other system number to be synchronized, and can also be issued along with its own system number.

(Example) System "i" command: !i!n!mLl;

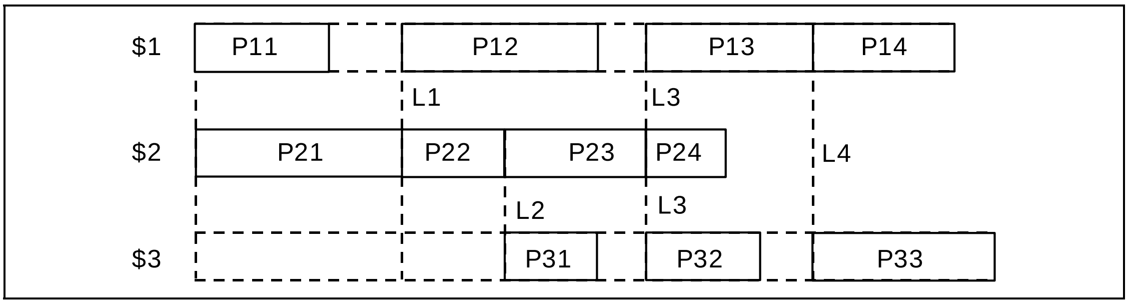
(8) When the system number is omitted (only ! is input), system 1 will be handled as !2 and system 2 as !1.
A command containing only ! cannot be used for synchronizing system 3 and the following systems. If ! is used in all systems, system 3 and the following will enter the waiting state at !, and the program will not proceed to the next block.

(9) "SYN" appears on the operating status display during synchronizing.

Example of synchronizing between systems



The above programs are executed as follows:



17.2 Start Point Designation Synchronizing (Type 1); G115

Function and purpose

The synchronizing point can be placed in the middle of the block by designating the start point.

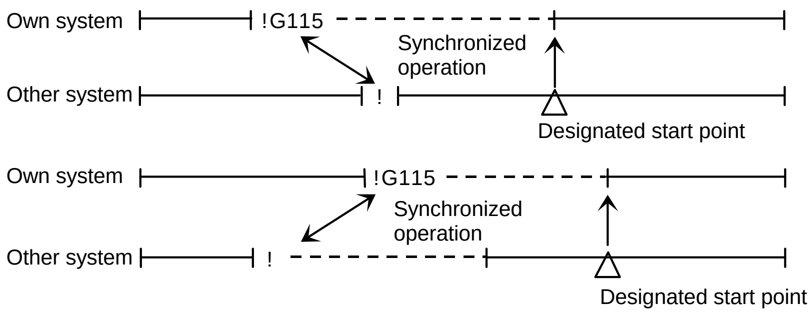
Command format

!nL1	**G115**	**X_ Z_ C_ ;**
!nL1		Synchronizing command
G115		G command
X_ Z_ C_		Start point (designated other system's coordinate value)

Detailed description

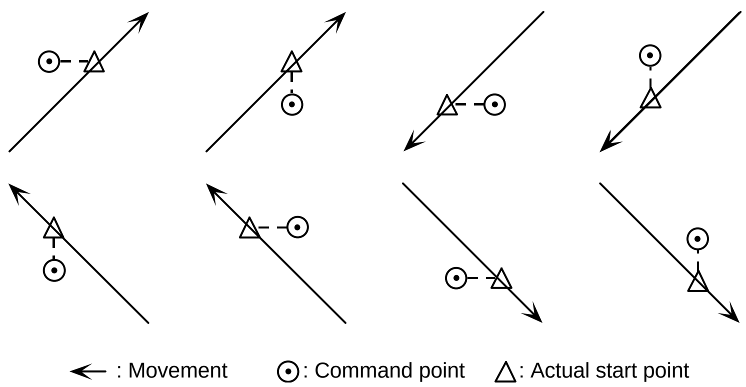
(1) The other system starts first when synchronizing is executed.

(2) The own system waits for the other system to move and reach the designated start point, and then starts.



(3) When the start point designated by G115 is not on the next block movement path of the other system, the own system starts once the other system has reached all of the start point axis coordinates.

Example:



(4) The start point check is executed only for the axis designated by G115.

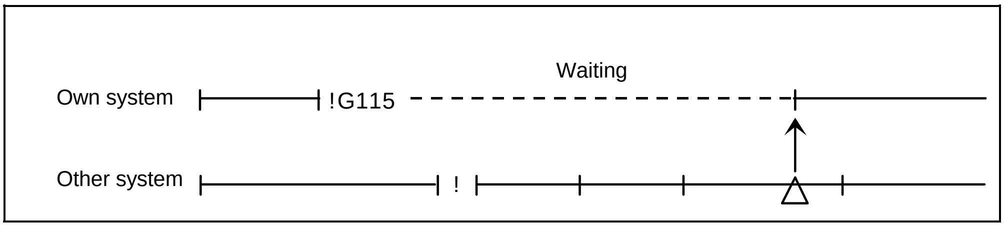
(Example) !L2 G115 X100.;

Once the other system reaches X100., the own system will start. The other axes are not checked.

(5) The following operation is executed by parameters (control parameter #8142 Start point alarm) when the start point cannot be determined by the next block movement of the other system.

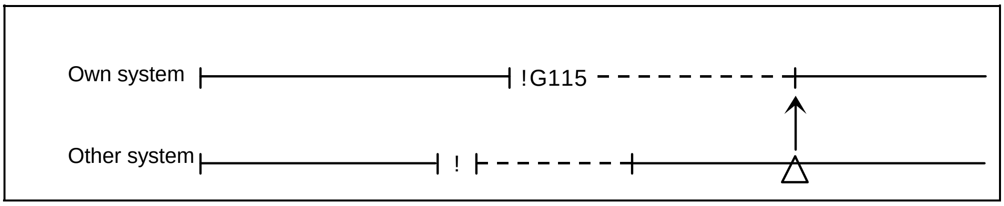
(a) When the parameter is ON

Operation waits until the start point is reached by the movement in the next and subsequent blocks.



(b) When the parameter is OFF

The own system starts upon completion of the next block movement.



(6) The waiting status continues when the G115 command has been duplicated between systems.

(7) Designate the start point using the workpiece coordinates of the other system.

(8) Program error "P33" occurs when the G115 command is issued for 3 systems.

(9) The single block stop function does not apply for the G115 block.

(10) When the G115 command is issued continuously in 2 or more blocks, the block in which it was issued last will be valid.

17.3 Start Point Designation Synchronizing (Type 2); G116

Function and purpose

The synchronizing point can be placed in the middle of a block by designating the start point.

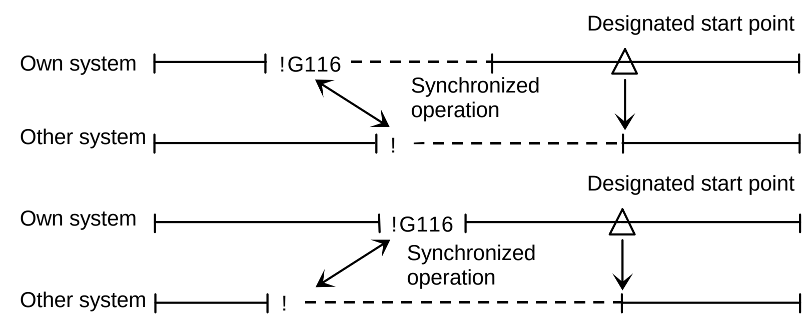
Command format

!nL1 G116 X_ Z_ C_ ;	
!nL1	Synchronizing command
G116	G command
X_ Z_ C_	Start point (designated other system's coordinate value)

Detailed description

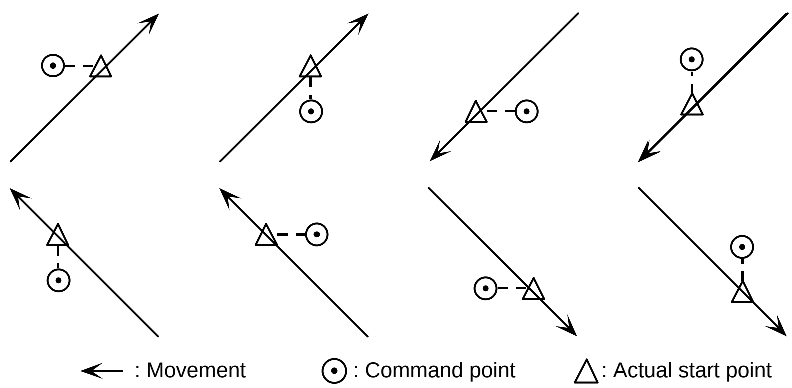
(1) The own system starts first when synchronizing is performed.

(2) The other system waits for the own system to move and reach the designated start point, and then starts.



(3) When the start point designated by G116 is not on the next block movement path of the own system, the other system starts once the own system has reached all of the start point axis coordinates.

Example:



(4) The start point check is executed only for the axis designated by G116.

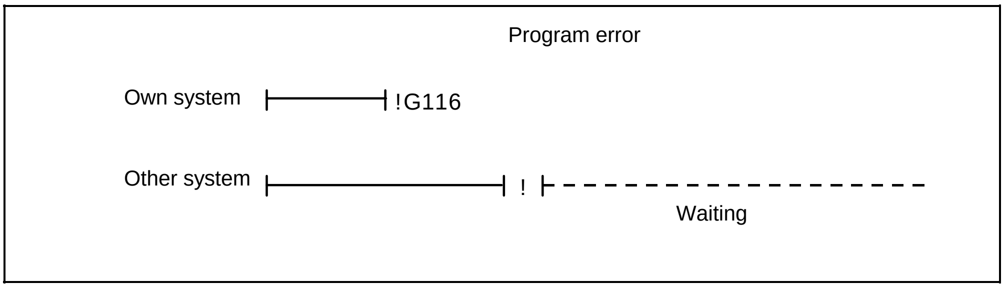
(Example) !L1 G116 X100.;

Once the own system reaches X100., the other system will start. The other axes are not checked.

(5) The next operation is executed by parameters (control parameter #8142 Start point alarm) when the start point cannot be determined by the next block movement of the own system.

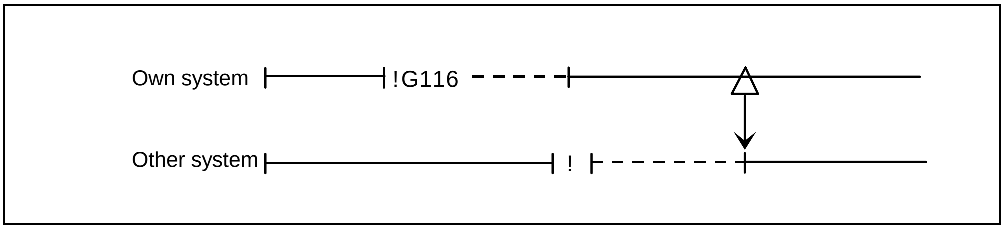
(a) When the parameter is ON

Program error "P33" occurs before the own system moves.

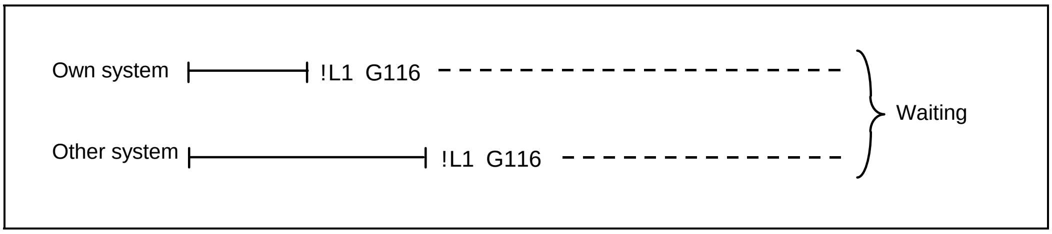


(b) When the parameter is OFF

The other system starts upon completion of the next block movement.

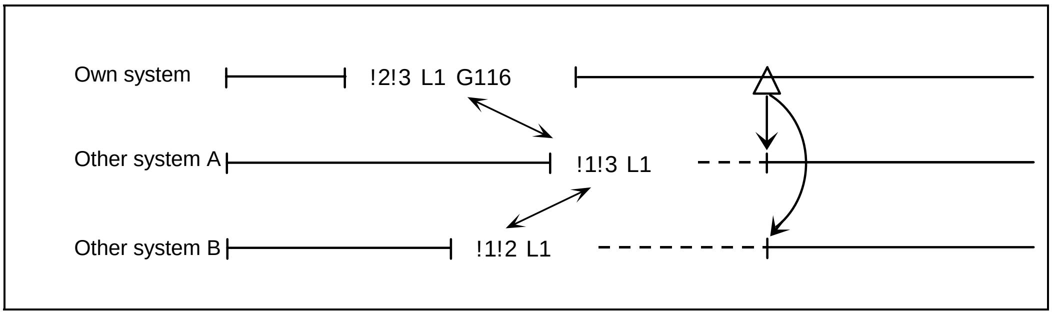


(6) Operation remains stopped when the G116 command has been duplicated between systems.



(7) Designate the start point using the workpiece coordinates of each system.

(8) The two other systems start when the G116 command is issued for 3 systems.



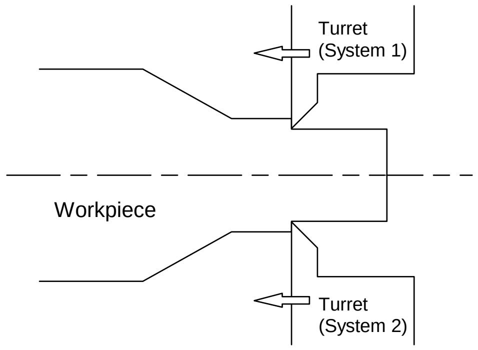
(9) The single block stop function does not apply for the G116 block.

(10) When the G116 command is issued continuously in 2 or more blocks, the block in which it was issued last will be valid.

17.4 Balance Cut Command; G15, G14

Function and purpose

This function can completely synchronize the movement start timing of the system 1 and system 2 turrets.



Command format

G15;	
G14;	
G15	Balance cut command ON (modal)
G14	Balance cut command OFF (modal)

(1) G15 and G14 are modal commands. In the NC initialized state, the G14 balance cut command OFF mode is entered.

(2) When the G15 command is issued, the synchronization between systems will be maintained for all blocks until the G14 command is issued.

Detailed description

The sections of blocks enclosed by G15 and G14 are created in the machining programs for system 1 and system 2.
The sections enclosed by G15 and G14 are synchronized one block at a time in each system.

<System 1>	**<System 2>**	
:	:	
G15;	G15;	
N10 G01 X10. F100;	N10 G01 X10. F50;	← The start timing of the $1 and $2 blocks is synchronized.
N20 G01 X20.;	N20 G01 X20.;	←
:	:	
:	:	
G14	G14	
:	:	
:	:	

(1) If G15 is commanded with one system, the operation will stop and wait until G15 is commanded in the other system.

(2) If G15 is commanded in both system 1 and system 2, synchronization will be carried out one block at a time until G14 is commanded.

(3) After G14 is commanded in both systems, system 1 and system 2 will operate independently.

Example of operation

```
<System 1>          <System 2>
    ⋮                   ⋮
G15                 G15                    ··· (1)
G00 X40. Z0.        G00 X-40. Z250.        ··· (2)
G01 W-30. F1000     G01 W-130. F500        ··· (3)
G01 U40. W-70.      G01 X-80. Z50. F1000   ··· (4)
G14                 G14                    ··· (5)
G01 X100. Z50.      S200                   ··· (6)
G01 Z30.            G00 X-100.
    ⋮                   ⋮
```

(1) Balance cut is turned ON with the G15 command.

(2) (3) Processing of this block will end faster for system 1 than system 2. The system will wait for the system 2 process to end, and will shift the next block for both system 1 and system 2.

(4) The movement amount and movement speed are the same for each system, so synchronized movement will take place.

(5) Balance cut is turned OFF with the G14 command.

(6) Each system will operate independently after this.

Precautions and restrictions

(1) If G15 is commanded in one system, the system will wait for G15 to be commanded in the other system. Synchronization will start when G15 has been commanded in both systems. If G14 is commanded first in one system, the other system will enter the waiting state. Shifting to the next block will not be possible. Thus, always match the number of blocks between the ON mode and OFF mode for both system 1 and system 2 when commanding balance cut.

```
System 1 program    System 2 program
 ≀                   ≀
G15-----------------G15
G01 Z__ F__         G01 Z__ F__
 ≀    ---------------≀
G14                 G14
 ≀                   ≀
```

System 1 can shift to the next block, but system 2 will continue waiting and cannot shift to the next block.

```
System 1 program    System 2 program
 ≀                   ≀
G15-----------------G15
G01 Z__ F__         G01 Z__ F__
 ≀    ---------------≀
G14                 G14
 ≀                   ≀
```

By matching the number of blocks between G15 and G14, both systems can shift to the blocks following G14.

(2) Take special care when using the synchronizing command with the G15 command.
If one system is waiting for synchronization with the ! (synchronization) code and the other system enters the synchronization state with the G15 command, both systems will be in the synchronization state, and will not shift to the next block. Command so that waiting for synchronization with G15 and waiting for synchronization with the ! code do not occur simultaneously.

System 1 program **System 2 program**

```
 ≀                     ≀
G15                    !
G01 Z__ F__           G15
 ≀                    G01 Z__ F__
G14                    ≀
 ≀                    G14
                       ≀
```

Stops in the waiting state at the sections shown with arrows.

(3) If the ! (synchronization) code is commanded in the G15 mode, it will be handled as a one block command with no movement, and synchronization will not be carried out. Thus, do not command this.

(4) This function is valid only for system 1 and system 2. If commanded for the system 3 and following, the system will enter the waiting state at the G15 block and will not shift to the next block. Thus, do not command this.

(5) A block that does not contain movement such as G4, M, S, T, B command or a variable command is handled as 1 block, and synchronization will be carried out.

(6) If G14 is designated when G15 is not declared (when balance cut is OFF), the G14 command will be handled as a block with no process.

(7) If subprogram call, macro call or PLC interrupt are carried out while balance cut is ON, the block configuring the subprogram will be handled as one block, and synchronization will be carried out.

(8) Even when the multi-system single block operation is valid, the synchronization per block will have the priority during G15.

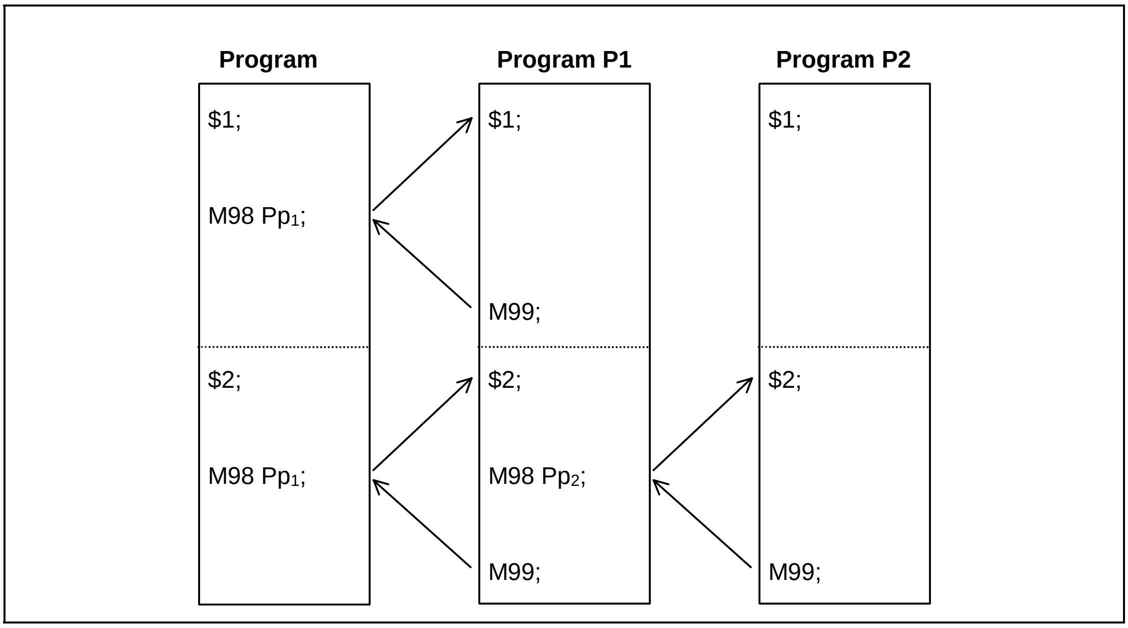
17.5 Program Call Control

Function and purpose

The system program called when a subprogram or other program is called from the main program is explained below.

Detailed description

Basic rule: When the call command is issued from a system "n" program, the system "n" program will be called.



(Note 1) If the program of system "n" to be called has not been saved in the memory, a check will be executed to confirm whether it has been saved in system 1. If it has been saved, the system 1 program will be called. If not, a program error occur.

(Note 2) Common program should be saved in system 1 regardless of the system.

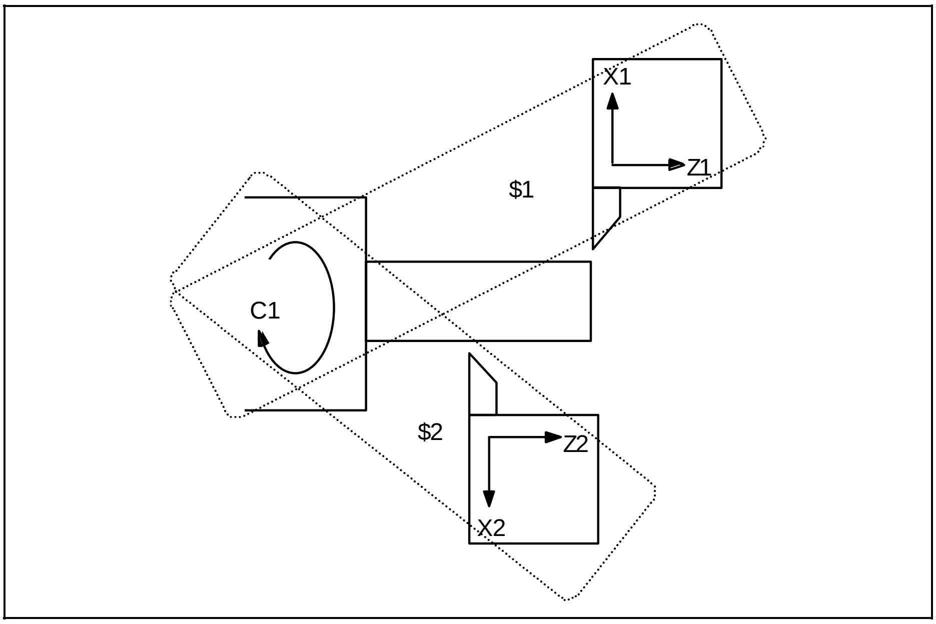
17.6 Cross Axis Control; G110

Function and purpose

The cross machining command serves to change the system definitions of the NC control axes.
For those control axes (cross axes) of which the system definitions can be changed, this command enables control from multiple systems among those systems that are separate from the basic system.

Actual example: When controlling C axis which is the basic axis of system 1 by system 2

	System 1	System 2
Basic definition	X1 Z1 C1	X2 Z2
Cross axis	C1 axis	–
Cross command program	G110 X1 Z1;	G110 X2 Z2 C1;
Control axes before cross command	X1, Z1, C1	X2, Z2
After cross command	X1 Z1	X2 Z2 C1



Command format

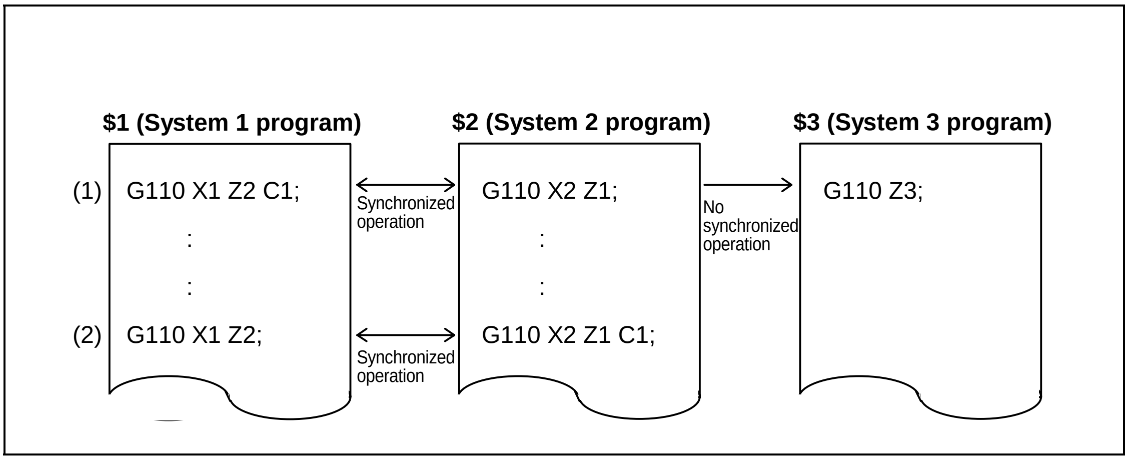
G110 Xn Zm Cl .... ;

G110	Cross machining command code
X, Z, C ...	Axis name
n, m, l ...	System number 1 to system max.

The axis names and system numbers are the basic definition names and numbers given by parameters, and the definitions are changed so that the corresponding axes can be controlled by the relevant system program.
The details of the definition changes are valid from the following block.

Example of program

Basic definition $1 X Z C
$2 X Z
$3 Z



(1) Z1 and Z2 system interchange

(2) C1 axis system change

Detailed description

(1) Program error "P503" occurs when the axes corresponding to the axis names and system numbers do not exist or when an axis (set by machining parameter) has been designated which cannot be controlled by the relevant system.

(2) All of the axes controlled by the relevant system are designated. Only the designated axes can be controlled by the subsequent program.

(3) The cross machining command is required with all systems to be changed as the definitions of the axes and systems change. The NC system synchronizes operation with all the changed systems until the command is issued. After synchronized operation, it processes the changes in the definitions.

(4) Up to 5 axes can be controlled by the systems. Program error "P503" occurs when a number exceeding 5 is designated.

(5) When the same axis name has been designated in the same system, it serves as the system number for a subsequent axis.

(Example) G110 X1 Z2 Z1 ;

This is ignored.

(6) The following will occur when another G code or address is issued in the same block.

(a) When a modal G command has been designated in the same block, the modal will be changed, but the axis address and other commands are ignored.

(b) When an unmodal G command has been designated in the same block, G110 is ignored if that command comes after G110 and the unmodal G command will be valid. On the other hand, if that command comes before G110, the unmodal G command will be ignored and G110 will be valid.

Relation with other functions

(1) Program error "P501" occurs when the cross machining command is issued in any of the following states, so do not issue the cross machining command.

(a) Nose R mode

(b) Milling mode

(c) Balance cut mode

(d) Fixed cycle (turning, hole drilling) mode

(e) Facing turret mirror image mode

(2) G modal commands

G modal commands are not changed by the cross machining command. The modals are saved by each system.

(3) Plane selection axes

The three basic axes for plane selection and the corresponding parallel axis names are saved beforehand as parameters for each system.
Axes are replaced and axes are moved to other systems by the cross machining commands, but the saved plane selection axis will remain unchanged.
Consider the cross machining command when saving the parameters.

(Example 1)

Basic definition: System 1 X1 Z1 System 2 X2 Z2
Cross machining command: $1 G110 X1 Z2; $2 G110 X2 Z1; Z axis replacement
Basic axis parameter save

<System 1>

	Basic axis
I	X
J	
K	Z

<System 2>

	Basic axis
I	X
J	
K	Z

Since the saved parameters are not changed by the cross machining command, the G18 plane axes are X1Z2 axis (system 1) and X2Z1 axis (system 2).

(Example 2)

Basic definition: System 1 X1 Z1 System 2 X2 Z2 Y2
Cross machining command: $1 G110 X1 Z1 Y2; $2 G110 X2 Z2; Y axis shifts to $1
Basic axis parameter save · · · · · Y-axis plane is enabled even for system 1.

<System 1>

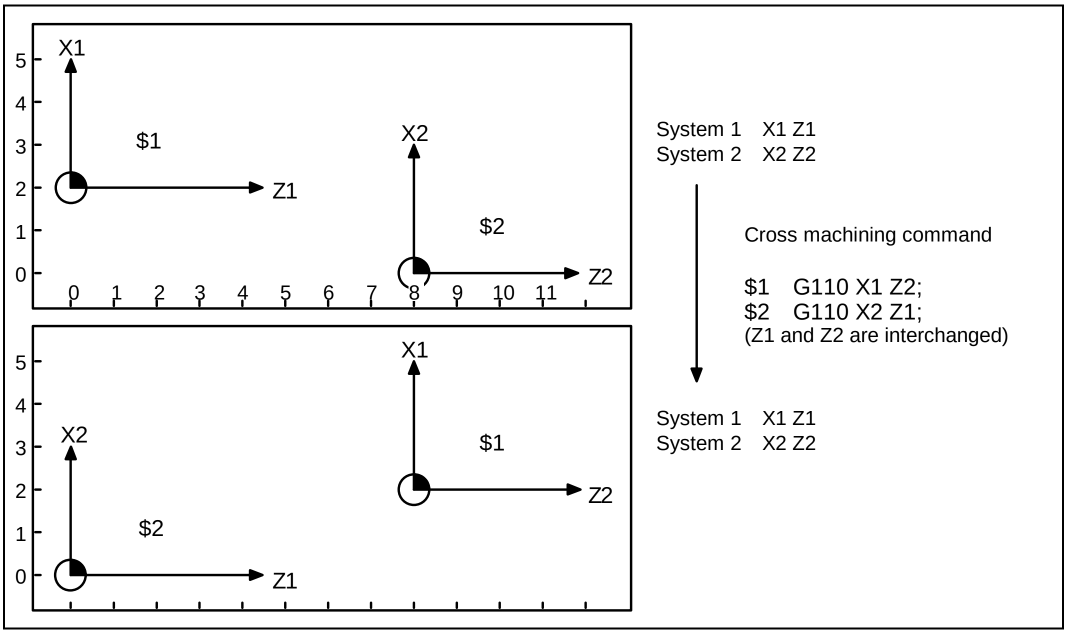
	Basic axis
I	X
J	Y
K	Z

<System 2>

	Basic axis
I	X
J	Y
K	Z

(4) Coordinate system function

The reference point and machine zero point characteristic to the machine at each axis are not changed by the cross machining command.
Since the workpiece coordinate and local coordinate systems are created on the basis of the machine zero point, the coordinate system zero points for each axis also remain unchanged. However, if this is considered on the XZ plane as in the figure below, the coordinate zero points of the systems are changed by cross machining.



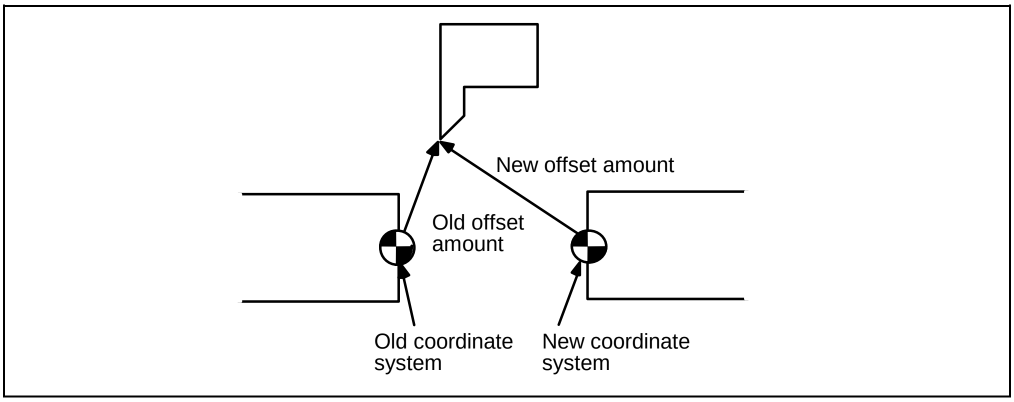
This means that the workpiece coordinate and local coordinate systems for programming will differ from the coordinate systems needed by the programmer because of the cross machining command. In this case, issue the coordinate system commands after the cross machining command and create new coordinate systems.

(5) Tool length and wear offset

The tool length and wear offset amounts are stored for each axis even when the cross machining command is issued. Even after this command has been issued, the program can be run without change. However, the following points should be observed.

(a) The 1st and 2nd axes in each system are the axes for which the tool length and wear offset functions are valid. These functions are not valid for the 3rd and subsequent axes. Therefore, if offset is required for particular axes, issue the cross machining command so that the 1st and/or 2nd axis will apply.

(b) When the tool length offset amount is entered as the amount from the programmed coordinate zero point up to the tool nose, it will not be applied properly if the coordinate zero point differs from the cross machining command. First, set an offset amount that meets the new coordinate zero point, and then issue the T command.



(6) Setting and display functions

There are two setting and display functions, aligned setting and display after the cross machining command, and setting and display with the basic definitions unchanged. These are known as cross setting and display and basic definition setting and display.
Refer to the following table to confirm which functions apply to which items.

Setting and display item	Cross setting and display	Basic definition setting and display
Present position, workpiece coordinate position, machine position, program position, remaining command, manual interrupt amount, error compensation amount	○	
Tool wear offset		○
Tool length offset		○
Workpiece coordinate offset		○
User parameter: axis parameter		○
User parameter: setup 2 (barrier data)		○
Graphic display		○
Servo monitor display		○
Machine parameter: basic specification 1		○
Machine parameter: axis specification		○
Machine parameter: zero point return		○
Machine parameter: servo		○
Alarm system/axis display	○	
Nose R, wear		○

(7) Manual operation

The manual operation axes can be operated manually only when they are part of a basic definition system.
An operation error will occur when an axis selected as a feed axis is used by the cross machining command of another system. Select the feed axis again after that axis has returned to the basic definition axis.

(Example)

	\$1	\$2
Basic definition	X1 Z1 C1	X2 Z2
After cross machining command	X2 Z1 C1	X1 Z2

When manual operation is to be set for system 2 while only system 1 is operating automatically, it is possible to manually operate the 1st axis of the system 2 when the basic definition applies. However, this will not be possible after the cross machining command.

(a) Manual interrupt during automatic operation

Whether the operation of an axis of another system that is cross machining using the cross machining command is to be enabled by a manual interrupt during automatic operation depends on the selection of the parameter (setting made by machine parameter).

- When the parameter is invalid
  The cross machining axis cannot be operated by the manual interrupt.
- When the parameter is valid
  Even the cross machining axis can be operated by the manual interrupt.
  Select the axis in the basic definition system for operating the cross machining axis.

(Example) The X1 axis in system 2 moves when the 1st axis in system 1 is selected under the following conditions:

Basic definition axes	System 1 X1, Z1	System 2 X2, Z2
With cross machining command	System 1 X2, Z1	System 2 X1 Z2

(Note) Jog feed and step feed are valid with manual interrupt; handle feed and optional feed are not valid.

(8) Automatic operation

(a) During automatic operation start

During automatic operation start, the first block is executed after a return is made to the basic definition axes. This means that if the system to be operated automatically does not have the basic definition axes, it waits until it does have them. In other words, the first automatic operation block is the same as that for the cross machining command that is set to the basic definition axes of the relevant system.

(Example) System 1 operation with basic definition axes of X1, X2, C1 for system 1 and X2, Z2 for system 2

System 1 machining program

```
G28 X Z C;
G00 X100;
    :
```

System 2 machining program

```
G28 X Z;              (i)
    :
G110 X1 Z2;           (ii)
    :
G110 X2 Z2;           (iii)
    :
```

When the block before (iii) from (ii) in the system 2 machining program is executed, the synchronized operation occurs until (iii) in the system 2 machining program is reached even if the system 1 machining program is operated automatically. (This is the same as issuing G110 X1 Z1 C1; at the head of the system 1 machining program.)

(b) During reset

Reset system axes return to the basic definitions. However, if a reset system basic definition axis is being used by another system, it will not return but will be set to the reset mode. The axis will return to the basic definition depending on whether the system in which it is used issues a cross machining command or conducts resetting. (All systems will have basic definition axes when they end automatic operation.)

(Example) Resetting system 1 when X1, Z1 and C1 of system 1, X2 and Z2 of system 2, and X3 of system 3 are the basic definition axes

System 1 machining program

```
    :
G110 X3 Z2 C1 ;
    :
G01 X100 ;            (i)
```

System 2 machining program

```
    :
G110 X2 Z1 ;
    :
G01 X100 ;            (ii)
    :
```

When systems 1 and 2 are operating automatically and system 3 is in the reset mode, the axes in system 1 in the reset mode will be X1 Z2 C1 when system 1 is reset while system 1 is executing block (i) and system 2 is executing block (ii) (Z1 is used by system 2.)

(c) When issuing the cross machining command to an axis in a system operating in the manual mode

When using an axis in a system operating manually with a cross machining command from another system, that system will execute the cross machining command. The operation of the manually operating system will be synchronized until it is no longer in the manual operation mode, and then the single block stop mode will be entered. Perform the start-up procedure again.

(Example) When the cross machining command is issued by system 2 with system 1 operating in the manual mode

System 2 machining program

```
G110 X1 Z2;   ←  System 1 waits until it is no longer in the
    :            manual operation mode and then the single
                 block stop mode is established.
```

17.7 Control Axis Synchronization ; G125

Function and purpose

This command can be used to synchronize the operation of an axis in any selected system with an axis in another system.

Command format

(1) Synchronization control ON

G125 D_ = H_	
D (slave axis name)	Command the axis to be moved as the slave axes during synchronization control.
H (master axis name)	Command the axis to be the reference for the slave axis.

(2) Synchronization control OFF

G125 Slave axis name

Detailed description

(1) Issue the slave axis and master axis commands on the basis of the basic axis definitions.

	System 1	**System 2**	**System 3**
[Axis name]	X Z	X Z	X Z
[Basic axis definition]	X1 Z1	X2 Z2	X3 Z3

(2) When issuing the synchronous and synchronous cancel commands, the synchronous command for establishing the timing between the slave axis and master axis must be issued in the block before the synchronous and synchronous cancel command.

(3) The synchronous and synchronous cancel commands are valid when both the slave and master axes are stopped. When issuing the synchronous and synchronous cancel commands from a system that does not contain the master axis, a dwell command should be issued in the block after the block with the synchronous command for the system that contains the master axis. If the dwell command is not issued, the synchronization will be executed after the master axis has stopped. During synchronization, do not issue movement command to the slave axis.

(4) By adding the "–" sign to the master axis in the control axis synchronous command (G125), the slave axis can be synchronized in the reverse direction with respect to the master axis.

(5) During synchronization, manual and automatic movement commands cannot be issued for the slave axis.
Whether an alarm should occur or the command should be invalidated if an axis movement command is issued is set in Base common parameter "syncch".

(6) The soft limit, chuck barrier and hardware OT of the synchronous axis are checked by the machine coordinate system.
All systems stop when soft limit, chuck barrier or hardware OT applies even to one axis.
During simulation and machine lock, however, the soft limit, chuck barrier and hardware OT are not checked.

(7) An alarm occurs when dog-type zero point return is done during synchronization.

(8) The coordinate system of the slave axis during control axis synchronization does not move with the master axis but follows the coordinate system before the synchronization.

(9) An alarm occurs when a command accompanying tool compensation or coordinate system shift of the slave axis is issued during synchronization.

(10) An alarm occurs if the commanded axis does not exist as the basic definition axis or if the slave axis and master axis are not parallel axes (with the same axis name).

(11) An alarm occurs when executing a synchronous command that includes two or three axes that are being superimposed.

(12) An alarm occurs when the cross machining command is issued to the slave axis or master axis during synchronization.

(13) Two or more synchronous commands cannot be issued in the G125 block.
An alarm will occur if two or more commands are issued.

(14) When one axis has been synchronized with another axis without the synchronous cancel command being issued, the previous synchronization will be canceled.

[System 1]	[System 2]		[System 3]
Machining program	Machining program		Machining program
↓	↓		↓
N10 !L1;	N5 !L1;		↓
N11 G125 X2=X1;	↓	No movement command for synchronous axis	↓
↓	↓		↓
N30 !2!3L2;	N20 !1!3L2;		N15 !1!2L2;
↓	↓		N16 G125 X2=X3;
↓	↓		↓

- The X2 axis is synchronized with the X1 axis in the N11 block of system 1, the X2 axis synchronizing with X1 axis in the N16 block of system 3 is canceled, and the X2 axis is synchronized with X3 axis.

Example of program

(1) When issuing the command from a system containing the master axis

(a) Forward direction synchronization

[System 1]	[System 2]	
Machining program	Machining program	
↓	↓	
N9 !L1;	N5 !L1;	
N10 G125 X2=X1;	↓	No movement command for slave axis
↓	↓	
N19 !L2;	N10 !L2;	
N20 G125 X2 ;	↓	
↓		

- The X2 axis is synchronized with the X1 axis in the N10 block of system 1, and the X2 axis synchronization is canceled in the N20 block of system 1.

(b) Reverse direction synchronization

```
 [System 1]            [System 2]
Machining program     Machining program
       ↓                     ↓
N14 !L1;              N10 !L1;
N15 G125 X2=X1;              ↓             } No movement command for
       ↓                     ↓             } slave axis
N24 !L2;              N15 !L2;
N25 G125 X2;                 ↓
       ↓                     ↓
```

- The X2 axis is synchronized in the reverse direction with the X1 axis in the N15 block of system 1, and the X2 axis synchronization is canceled in the N25 block of system 1.

(c) When issuing the command from a system that does not contain the master axis

```
 [System 1]            [System 2]                              [System 3]
Machining program     Machining program                       Machining program
       ↓                     ↓                                        ↓
N9 G4 X0;             N15 !1!3L1;        }                    N5 !1!2L1;
N10 !2!3L1;                  ↓           } No movement        N16 G125 X2=X1;
       ↓                     ↓           } command for                ↓
N39 G4 X0;                   ↓           } slave axis                 ↓
N30 !2!3L2;           N40 !1!3L2;        }                    N10 !1!2L2;
       ↓                     ↓                                N11 G125 X2;
       ↓                     ↓                                        ↓
```

- The X2 axis is synchronized with the X1 axis in the N6 block of system 3, and the X2 axis synchronization is canceled in the N11 block of system 3.

17.8 Spindle Synchronization; G114.1, G113

Function and purpose

In a machine having two spindles, serially connected, the rotation speed and phase of one spindle are controlled in synchronization with the rotation of the other spindle.
This is used when the rotation speed of two spindles need to be matched, such as when using the second spindle to grasp a workpiece grasped by the first spindle, or to change the spindle rotation speed when one workpiece is graphed by both the first and second spindles.

Command format

(1) Spindle synchronization control command

The master spindle and slave spindle are designated, and both spindles are synchronized.
By commanding the slave spindle phase shift amount, the phases of the master spindle and slave spindle can be aligned.

G114.1 H_ D_ R_ ;	**Spindle synchronous control ON**
H	Master spindle
D	Slave spindle
R	Slave spindle phase spindle amount

(2) Spindle synchronization control cancel command

The synchronous state of the two spindles rotating in synchronization with the spindle synchronous command is canceled.

G113; Spindle synchronization control OFF

Address	Meaning of address	Command range (unit)	Remarks
H	Master spindle Set the number of the spindle to be used as the master spindle.	1 to 6 Command machine parameter Sname value	• If there is no command, a program error will occur. (P33 Format error) • If the same value as the D command is commanded, a program error will occur. (P33 Format error) • If a spindle that is not serially connected (HDLC connection), a program error will occur. (P33 Format error)
D	Slave spindle Set the number of the spindle to be synchronized with the master spindle.	1 to 6 or –1 to –6 Command the machine parameter Sname value or the Sname value with – sign value	• If there is no command, a program error will occur. (P33 Format error) • If the same value as the H command is commanded, a program error will occur. (P33 Format error) • The rotation direction of the slave spindle in respect to the master spindle is commanded with the D sign. • If a spindle that is not serially connected (HDLC connection), a program error will occur. (P33 Format error)
R	Slave spindle phase shift amount Set the shift amount from the slave spindle's reference point (one rotation signal).	0 to 359.999 (°) or 0 to 359999 ($° \times 10^{-3}$)	• The commanded shift amount is applied in the clockwise direction of the spindle. • The minimum resolution of the commanded shift amount is as follows. For semi-closed : 360/4096 (°) For full closed : (360/4096) ∗ K (°) K: Spindle and encoder gear ratio. • If there is no R command, phase alignment will not be carried out.

(Note 1) If the G164 command is issued to a spindle during the spindle synchronization (G114.n) mode, or if the G114.n command is issued to a spindle during the spindle superimposition mode, an operation error will occur. (M01 operation error 1005)

(Note 2) If a value exceeding the command range is commanded, a program error will occur. (P35 Setting value range over)

(Note 3) A program error will occur if H or D is not commanded. (P33 Format error)

Rotation speed and rotation direction

(1) The rotation speed and rotation direction of the master spindle and slave spindle during spindle synchronization control are the rotation speed and rotation direction commanded to the master spindle.
Note that the slave spindle's rotation direction can be set to the opposite of the master spindle's direction from the program.

(2) The master spindle's rotation speed and rotation direction can be changed during spindle synchronization control. Constant surface speed control of the master spindle is also possible.

(3) The slave spindle's rotation command is valid during spindle synchronization. When the spindle synchronization control is commanded, if neither a forward run nor a reverse run command is issued to the slave spindle, the slave spindle will not start rotating, and instead will wait for synchronization.
The slave spindle will start rotating when a forward run or reverse run command is input in this state. Note that the slave spindle's rotation direction will be that commanded from the program.
If spindle stop is commanded (if both forward run and reverse run are turned OFF) to the slave spindle during spindle synchronization control, the slave spindle rotation will stop.

(4) The rotation speed command (S command) and the constant surface speed control are invalid for the slave spindle during the spindle synchronization control mode. Note that the modal will be updated, so these will be validated when spindle synchronization is canceled.

Relative position spindle synchronization control (Rotation synchronization)

(1) If the relative position spindle synchronization control command (no R address command) is issued with the G114.1 command, the slave spindle rotating at a random rotation speed will accelerate or decelerate to the rotation speed pre-commanded for the master spindle, and will enter the relative position spindle synchronization control state.

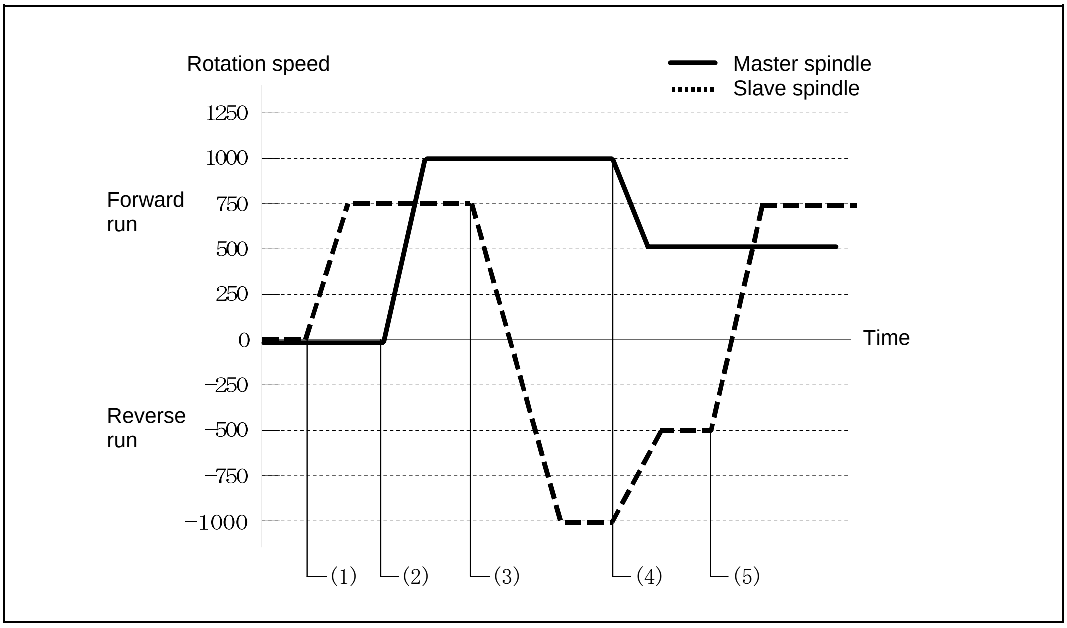
(2) If the master spindle's command rotation speed is changed during relative position spindle synchronization control, the synchronization state will be maintained following the spindle acceleration/deceleration time constants set in the parameters, and the commanded rotation speed will be attained.

(3) Constant surface speed control can be applied to the master spindle even when two spindles are grasping one workpiece in the relative position spindle synchronization control state.

(4) The rotation speed command (S command) and the constant surface speed control are invalid for the slave spindle during the spindle synchronization control mode. Note that the modal will be updated, so these will be validated when spindle synchronization is canceled.

(5) The following type of operation will take place.

M23 S2=750; :	････Forward run second spindle (slave spindle) at 750r/min. (Speed command)	(1)
M03 S1=1000; :	････Forward run first spindle (master spindle) at 1000r/min. (Speed command)	(2)
G114.1 H1 D-2; :	････Synchronize second spindle (slave spindle) to first spindle (master spindle) with reverse run.	(3)
S1=500; :	････Change the first spindle (master spindle) rotation speed to 500r/min.	(4)
G113;	････Cancel spindle synchronization.	(5)



Absolute position spindle synchronization control (Phase synchronization)

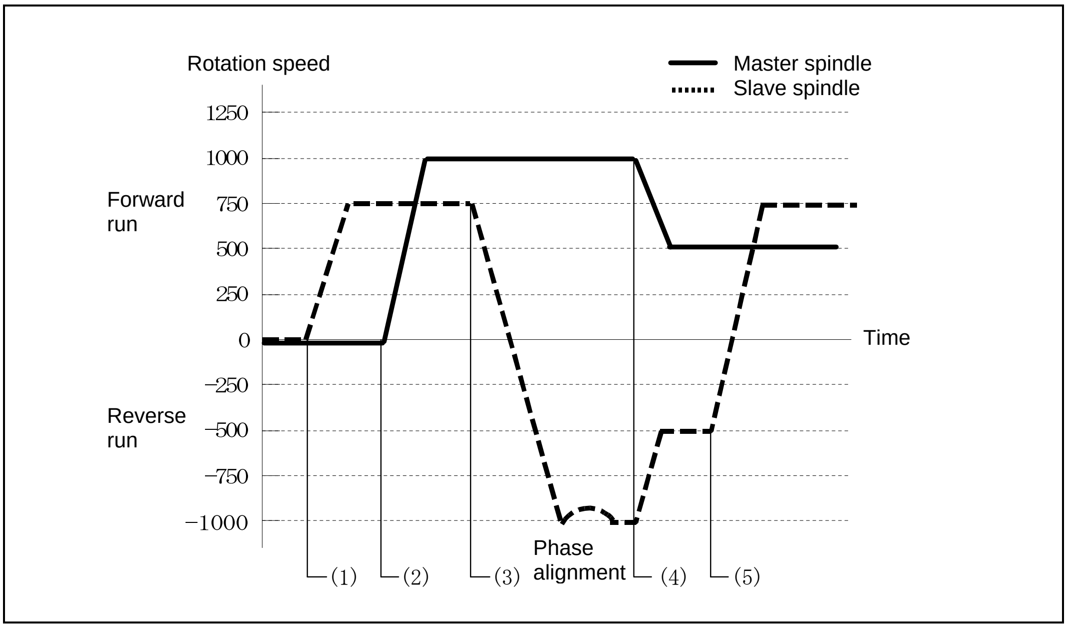
(1) If the absolute position spindle synchronization control command (with R address command) is issued with the G114.1 command, the slave spindle rotating at a random rotation speed will accelerate or decelerate to the rotation speed pre-commanded for the master spindle, and will enter the relative position spindle synchronization control state.
The phase is aligned so that the rotation phase commanded with the R address is reached, and the absolute position spindle synchronization state is entered.

(2) If the master spindle's command rotation speed is changed during absolute position spindle synchronization control, the synchronization state will be maintained following the spindle acceleration/deceleration time constants set in the parameters, and the commanded rotation speed will be attained.

(3) Constant surface speed control can be applied to the master spindle even when two spindles are grasping one workpiece in the absolute position spindle synchronization control state.

(4) The following type of operation will take place.

M23 S2=750; :	･･･	Forward run second spindle (slave spindle) at 750r/min. (Speed command)	(1)
M03 S1=1000; :	･･･	Forward run first spindle (master spindle) at 1000r/min. (Speed command)	(2)
G114.1 H1 D-2 Rxx; :	･･･	Synchronize second spindle (slave spindle) to first spindle (master spindle) with reverse run. Shift slave spindle phase by R command value.	(3)
S1=500; :	･･･	Change the first spindle (master spindle) rotation speed to 500r/min.	(4)
G113;	･･･	Cancel spindle synchronization.	(5)



Precautions for programming

(1) To enter the relative position spindle control mode while the master spindle and slave spindle are chucking the same workpiece, turn the master spindle/slave spindle rotation commands ON before turning the spindle synchronization control mode ON.

(Program example)

```
$1                                               $3
 :                                                :
M6;         ......First spindle chuck             :
                  close                          M25 S2=0;         ......Second spindle stops at
 :                                                                       S=0
!3;                                               :
M5 S1=0;    ......First spindle stops at         !1;               ......Synchronized operation
                  S=0                            M15;              ......Second spindle chuck
 :                                                                       close
 :                                               M24;              ......Second spindle rotation
M3;         ......First spindle rotation                                 command ON
                  command ON                      :
!3;                                               :
 :                                               !1;               ......Synchronized operation
 :                                               G114.1 H1 D-2;    ......Relative position spindle
 :                                                                       synchronization control
S1=1500;    ......Synchronous rotation at                                mode ON
                  S = 1500
```

(2) To chuck the same workpiece with the master spindle/slave spindle during the absolute position spindle synchronization control mode, chuck after aligning the phases. If the phases are aligned after chucking the same workpiece, an excessive load will be applied on the workpiece, and correct operation will not be possible.

(Program example)

```
$1                                                    $3
 :                                                     :
M6;              ......First spindle chuck             :
                       close
 :                                                     :
M3 S1=1500;      ......First spindle rotation         G114.1 H1 D-2     ......Absolute position spindle
                       command ON                                              synchronization control
 :                     (S=1500)                       R0;                     mode ON
 :                                                     :
 :                                                    M24;              ......Second spindle rotation
                                                                              command ON
 :                                                     :
 :                                                    M15;              ......Second spindle chuck
                                                                              close
 :                                                     :                      (Note 1)
```

(Note) Close the chuck after confirming that the synchronization complete signal (X269) has turned ON (phase alignment complete).

⚠CAUTION

⚠ **Do not turn the slave spindle rotation command OFF when the master spindle/slave spindle are both chucking the same workpiece in the spindle synchronization control mode. A hazardous state will be created as the slave spindle will stop.**

Precautions

(1) The spindle rotating with spindle synchronization control will stop when emergency stop is applied. The spindle synchronization control mode will be canceled.

(2) The rotation speed clamp during spindle synchronization control will follow the smaller clamp value set for the master spindle or slave spindle.

(3) Orientation of the master spindle and slave spindle is not possible during the spindle synchronization control mode. To carry out orientation, cancel the spindle synchronization control first.

(4) The rotation speed command (S command) is invalid for the slave spindle during the spindle synchronization control mode. Note that the modal will be updated, so this will be validated when spindle synchronization control is canceled.

(5) The constant surface speed control is invalid for the slave spindle during the spindle synchronization control mode. Note that the modal will be updated, so this will be validated when spindle synchronization is canceled.

(6) The rotation speed (S command) and constant surface speed control for the slave spindle will be validated when spindle synchronization control is canceled. Thus, the slave spindle may carry out different operations when this control is canceled.

17.9 Tool/Spindle Synchronization 1 (Polygon); G114.2, G113

Function and purpose

In a machine having a rotary tool controlled with a serial connection and having a spindle controlled with a serial connection as the workpiece axis, polygon machining can be carried out by controlling the workpiece axis rotation in synchronized with the rotation of the rotary tool axis.
The serial connection control axis spindle and rotary tool axis can be carried out with MDS-∗-SP or MDS-∗-SPJ2.

Command format

(1) Polygon machining mode command

This command sets the polygon machining mode that rotates the two axes in synchronization with differing speeds by designating the rotary tool axis and workpiece axes and the rotation ratio (No. of rotary tool teeth and No. of workpiece angles) of the two designated axes (spindle and spindle).

G114.2 H_ D_ E_ L_ R_;	**Tool/spindle synchronization control 1 (polygon machining mode) ON**
H	Rotary tool axis
D	Workpiece axis
E	Rotary tool axis rotation ratio
L	Workpiece axis rotation ratio
R	Workpiece axis phase shift amount

(2) Polygon machining mode cancel command

This command cancels the synchronization state of rotating two spindles by the spindle synchronization command.

G113;	**Tool/spindle synchronization control 1 (polygon machining mode) OFF**

Add-ress	Meaning of address	Command range (unit)	Remarks
H	Rotary tool axis Select the spindle number of the rotation axis.	1 to 6 Command machine parameter Sname value	1. If there is no command, a program error will occur. (P33 Format error) 2. If the same value as the D command is commanded, a program error will occur. (P33 Format error) 3. If a spindle that is not serially connected (HDLC connection), a program error will occur. (P33 Format error)
D	Workpiece axis Select the spindle number of the workpiece axis.	1 to 6 or −1 to −6 Command the machine parameter Sname value or the Sname value with – sign value	1. If there is no command, a program error will occur. (P33 Format error) 2. If the same value as the H command is commanded, a program error will occur. (P33 Format error) 3. The rotation direction of the workpiece axis in respect to the rotary tool axis is commanded with the D sign. 4. If a spindle that is not serially connected (HDLC connection), a program error will occur. (P33 Format error)
E	Rotary tool axis rotation ratio Set the rotation ratio (No. of rotary tool gear teeth) of the rotary tool axis.	1 to 999	1. If there is no command, the rotation ratio will be interpreted as 1.
L	Workpiece axis rotation ratio Set the rotation ratio (No. of workpiece corners) of the workpiece axis.	1 to 999	1. If there is no command, the rotation ratio will be interpreted as 1.
R	Workpiece axis phase shift amount Set the shift amount from the workpiece axis' reference point (one rotation signal).	0 to 359.999 (°)	1. The commanded shift amount is applied in the clockwise direction of the spindle. 2. The minimum resolution of the commanded shift amount is as follows. For semi-closed : 360/4096 (°) For full closed : (360/4096) ∗ K (°) K: Spindle and encoder gear ratio. 3. If there is no R command, phase alignment will not be carried out.

(Note 1) If a value exceeding the command range is commanded, a program error will occur. (P35 Setting value range over)

Precautions for programming

(1) The axis address (X, Z, C, U, W) command in the same block as G114.2 will be ignored.
(Example) G114.2 ··· X_;
Ignored.

(2) If a modal is commanded in the same block as G114.2, the modal will be updated.
(Example) G114.2 ··· G01_;
The group 01 modal is set to G01.

(3) If a miscellaneous command (M, S, T) is commanded in the same block as G114.2, the miscellaneous command will be executed simultaneously with the change to the rotary tool machining mode.
(Example) G114.2 ··· M03;
M03 is executed simultaneously with G114.2.

(4) If there is a group 00 G code command in the same block as G114.2, the G code commanded last in the block will have the priority.
(Example) G114.2 ··· G04 P30;
G04 P30 is executed.

(5) The tool/spindle synchronization control 1 (polygon machining) mode cannot be commanded during the spindle synchronization control (spindle rotation synchronization) mode or the tool/ spindle synchronization control 2 (hobb machining) mode.
Only one of the three types of spindle synchronization control can be commanded at once.

Example of program

```
 :
 :
M03 S1=0;                 ......First spindle forward run
Txx 00;                   ......Rotary tool selection
M83 S4=500;               ......Fourth spindle forward run
G00 X40. Z-5.;
 :
 :
G114.2 H4 D1 E1 L10 R0;   ......Tool/spindle synchronization 1 (polygon machining mode) ON
                                Rotary tool axis: Fourth spindle
                                Workpiece axis: First spindle
                                No. of rotary tool gear teeth: 1
                                Rotation ratio: No. of workpiece corners as 10
                                Workpiece axis phase shift amount: 0°.

                                As S1 forward run, rotation in synchronized with S4 starts.
                                The phase is aligned with shift amount 0°.
                                The S1 rotation speed is 50 r/min (S4 : S1 = 10 : 1).

G99;                      ......Synchronous feed mode selection
M__                       ......Spindle synchronization complete confirmation
G00 X18.;
G01 Z20. F0.1;            ......First cut in
G00 X40.;                       Z axis feedrate is 0.1mm per workpiece axis rotation
    Z-5.;
 :
 :
G00 X14.;
G01 Z20. F0.1;            ......Final cut in
G00 X40.;                       Z axis feedrate is 0.1mm per workpiece axis rotation
    Z-5.;

G113;                     ......Spindle synchronization cancel

M85;                      ......Fourth spindle stop
M05;                      ......First spindle stop
 :
```

Rotation speed and rotation direction

The rotary tool axis and workpiece axis rotation speed and rotation direction during tool/spindle synchronization 1 (polygon machining) are as follows.

(1) The rotation speed and rotation direction of rotary tool axis are the rotation speed commanded with the S command and the rotation direction commanded with the M command, etc., for the spindle selected as the rotary tool axis.

(2) The workpiece axis rotation speed is determined by the No. of rotary tool gear teeth and No. of workpiece corners commanded with G114.2.

$$Sw = Sh * \frac{E}{L}$$

Sw : Workpiece axis rotation speed (r/min)
Sh : Rotary tool axis rotation speed (r/min)
L : Rotary tool axis rotation ratio (No. of rotary tool gear teeth)
E : Workpiece axis rotation ratio (No. of workpiece corners)

(3) The workpiece axis rotation direction is determined by the sign of the address D commanded with G114.2. In other words, when the D sign is "+", the workpiece axis rotates in the same direction as the rotary tool axis, and when "–", the workpiece axis rotates in the reverse direction of the rotary tool axis.

(4) After tool/spindle synchronization 1 (polygon machining) is commanded, the relation of the rotary tool axis and workpiece axis rotation is held in all operation modes of automatic and manual modes until spindle synchronization cancel (G113) is commanded or until the spindle synchronization cancel signal is input.
Even during feed hold, the rotary tool axis and workpiece axis synchronization state is held.

Polygon machining when workpiece axis is spindle

When using the first to sixth spindles as the workpiece axis, the workpiece axis will be controlled in the following manner.

(1) When the tool/spindle synchronization 1 (polygon machining) mode is commanded, if neither a forward run command nor a reverse run command is input for the workpiece axis, the workpiece axis will not start rotating even if the polygon axis is rotating, and instead will wait for synchronization.
If a forward run command or reverse run command is input for the workpiece axis in this state, the workpiece axis will start rotation.

(2) If spindle stop is commanded (both forward run command and reverse run command are turned OFF) in respect to the workpiece axis during the tool/spindle synchronization 1 (polygon machining) mode, the workpiece axis rotation will stop even when the rotary tool axis is rotating.

(3) The rotation speed command (S command) and constant surface speed command are invalid in respect to the workpiece axis during the tool/spindle synchronization 1 (polygon machining) mode. Note that the modal will be updated, so these will be validated after the tool/spindle synchronization is canceled.

(4) If a rotary tool axis rotation speed that exceeds the workpiece axis maximum rotation speed is commanded, the rotary tool axis rotation speed will be clamped so that the workpiece axis rotation speed does not exceed the workpiece axis maximum rotation speed.

Acceleration/deceleration control

(1) The rotary tool axis will accelerate or decelerate following the spindle synchronization acceleration/deceleration time constant (spt) of the spindle selected as the rotary tool axis.

Phase alignment control

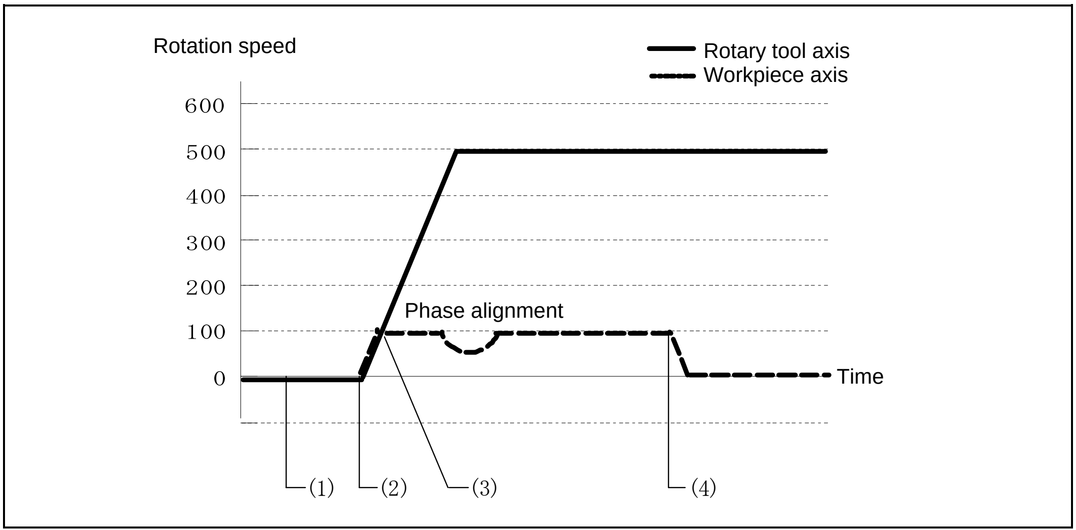
(1) If the tool/spindle synchronization control 1 (with R command) is commanded with the G114.2 command, the workpiece axis rotating at a random speed will accelerate or decelerate to the speed following the rotary tool axis and workpiece axis rotation ratio command, and the tool/spindle synchronization control state will be entered.
After that, the phase will be aligned to match the rotation phase commanded with the R address.

(2) The workpiece axis phase shift amount is commanded as the shift amount from the workpiece axis' reference point (one rotation signal). There is no shift amount in respect to the rotary tool axis.

(3) If the rotary tool axis' command rotation speed is changed during the tool/spindle synchronization control state, acceleration/deceleration following the acceleration/ deceleration time constant that was set in the parameter while maintaining the synchronous state, and then the commanded rotation speed will be attained.

(4) The following type of operation will take place.

M03 S1=0;	· · · ·	Forward run (speed command) first spindle (workpiece axis)	(1)
:			
Txx 00;	· · · ·	Rotary tool selection	
M83 S4=500;	· · · ·	Forward run (speed command) fourth spindle (rotary tool axis)	(2)
:			
G114.2 H4 D1 E1 L5 Rxx;	· · · ·	Forward run first spindle (workpiece axis) and synchronize with fourth spindle (rotary tool axis). Shift workpiece axis phase by amount of R command value.	(3)
:			
:			
G113;	· · · ·	Cancel tool/spindle synchronization 1.	(4)



Precautions and restrictions

(1) The tool/spindle synchronous control 1 (polygon machining) mode cannot be commanded during the spindle synchronization control (spindle rotation synchronization) mode or the tool/spindle synchronization control 2 (hobb machining) mode.
Only one of the three types of spindle synchronization control can be commanded at once.

(2) Restrictions regarding phase alignment control are as follows.

(a) Make sure that the rotation ratio of spindle (and rotary tool spindle) actual rotation speed and encoder speed has the following relation.

Spindle rotation speed/encoder rotation speed = n ("n" is an integer of 1 or more)

If this relation is not established, the encoder reference point will not stay at a constant position of the spindle, and thus the phase (position) will deviate with each phase alignment command.
Note that even in this case, if the No. of rotary tool gear teeth (No. of workpiece corners) is equivalent to the rotation ratio, the blade and workpiece phase (position) will not deviate. (Following relation)

(Rotary tool spindle rotation speed ∗ No. of rotary tool gear teeth)/
encoder rotation speed = n ("n" is an integer of 1 or more)

(b) During phase alignment control, phase alignment is carried out following each spindle encoder's reference point, so if the positional relation of the workpiece and reference point (rotary tool and reference point) deviates when the power is turned OFF/ON or when the tool is changed, etc., the phase will deviate.

17.10 Tool/Spindle Synchronization 2 (Hobb Machining); G114.3, G113

Function and purpose

In a machine having a rotary tool axis controlled with serial connection (HDLC connection) as the hobb axis, and having a C axis as the workpiece axis, spur gears can be machined by controlling the workpiece axis rotation in synchronization with the hobb axis rotation.
Helical gears can be machined by compensating the workpiece axis in synchronized with the Z axis movement during synchronization control of the hobb axis and workpiece axis.
Serial connection (HDLC connection) control of the spindle and rotary tool axis is possible from the MDS-A-SP Series.

Command format

The hobb machining mode for spur gears in which two axes are synchronously rotated at different speeds is entered by designating the hobb axis and workpiece axis, and by designating the rotation ratio (hobb threads and number of gear teeth) for the two designated axes (spindle and spindle, or spindle and C axis).
The hobb machining mode for helical gears is entered by additionally designating the gear torsion angle and module or diametrical pitch.

(1) Hobb machining mode commands (for spur gears)

G114.3 H_ D_ E_ L_ R_ ;	**Tool/spindle synchronization control 2 (hobb machining mode) ON**
H	Hobb axis
D	Workpiece axis
E	Hobb axis rotation ratio
L	Workpiece axis rotation ratio
R	Workpiece axis phase shift amount

(2) Hobb machining mode commands (for helical gears)

G114.3 H_ D_ E_ L_ P_ Q_ R_ ;	**Tool/spindle synchronization control 2 (hobb machining mode) ON**
H	Hobb axis
D	Workpiece axis
E	Hobb axis rotation ratio
L	Workpiece axis rotation ratio
P	Gear torsion angle
Q	Module or diametrical pitch
R	Workpiece axis phase shift amount

(3) Hobb machining mode cancel command

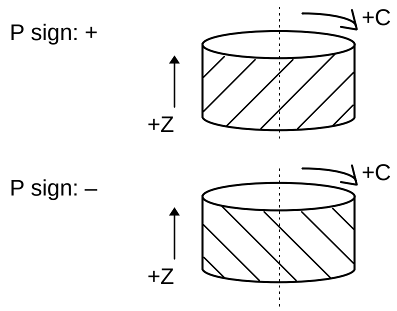
The synchronous state of the two spindles rotating in synchronization with the tool/spindle synchronization command is canceled.

G113 ;	**Tool/spindle synchronization control 2 (hobb machining mode) OFF**

Add-ress	Meaning of address	Command range (unit)	Remarks
H	Hobb axis Set the spindle number of the hobb axis.	1 to 6 Set the machine parameter Sname value.	• If there is no command, a program error will occur. (P33 Format error) • If the same value as the D command is commanded, a program error will occur. (P33 Format error) • If a spindle that is not serially connected (HDLC connected), a program error will occur. (P33 Format error)
D	Workpiece axis Set the spindle number of the workpiece axis.	1 to 9 or –1 to –9 ±1: Not used ±2: Not used ±3: Not used ±4: Not used ±5: Not used ±6: Not used ±7: Not used ±8: Not used ±9: C axis	• If there is no command, a program error will occur. (P33 Format error) • If the same value as the H command is commanded, a program error will occur. (P33 Format error) • The rotation direction of the workpiece axis in respect to the hobb axis is commanded with the D sign. • If the D sign is "+", when the hobb axis rotates in the forward direction, the workpiece axis will also rotate in the forward direction. If the D sign is "–", when the hobb axis rotates in the forward direction, the workpiece axis will rotate in the reverse direction. • If a spindle that is not serially connected (HDLC connected) is selected, or if the C axis is selected when there is no C axis, a program error will occur. (P33 Format error)
E	Hobb axis rotation ratio Set the hobb axis rotation ratio (hobb threads).	0 to 999	• If there is no command, the rotation ratio will be interpreted as 1. • If E0 is commanded, the workpiece axis will stop (synchronized with the Z axis for a helical gear). **(Note 1)**
L	Workpiece axis rotation ratio Set the workpiece axis rotation ratio (number of gear teeth).	1 to 999	• If there is no command, the rotation ratio will be interpreted as 1.

(Note 1) When address E = 0 is commanded, the workpiece axis will not rotate. Do not use this except for special cutting (cutting of only part of the gears, etc.).

(Note 2) If a value exceeding the command range is commanded, a program error will occur. (P35 Setting value range over)

<table>
<tr><th>Add-
ress</th><th>Meaning of address</th><th>Command range (unit)</th><th>Remarks</th></tr>
<tr><td>P
(use with helical gears)</td><td>Gear torsion angle
Command the torsion angle for the helical gears.</td><td>–89000 to +89000
(–89 to +89°)
0.001°
Minimum command unit: °
Decimal point input possible.</td><td>• If there is no P command, or if P0 is commanded, spur gears will be machined.
• To move the Z axis in the pulse direction after entering the hobb machining mode, command the direction that the workpiece axis is twisted.
<table>
<tr><th>P sign</th><th>Direction that workpiece axis twists</th></tr>
<tr><td>+</td><td>+ direction</td></tr>
<tr><td>–</td><td>– direction</td></tr>
</table>

</td></tr>
<tr><td>Q
(use with helical gears)</td><td>Module
Command the normal module for helical gears.</td><td>For metric input:
Designate module
100 to 25000
(0.1 to 25mm)
Minimum command unit: 0.001mm

For inch input:
Designate diametrical pitch
1000 to 250000
(0.1 to 25inch^{-1})
Minimum command unit: 0.0001inch^{-1}</td><td>• If there is no Q command for helical gears (when P is designated), a program error will occur. (P33 Format error)
• For spur gears (when P is not designated, or P0 is commanded), the Q command will be ignored.</td></tr>
<tr><td>R</td><td>Workpiece axis phase shift amount
Command the amount to shift from the workpiece axis reference point to synchronize with the hobb axis reference point.</td><td>0 to 359999
(0 to 359.999°)
Minimum command unit: 0.001°
Decimal point input possible.</td><td>• The commanded shift amount will be applied in the workpiece axis counter's positive direction.
• Phase alignment will not take place if there is no R command.</td></tr>
</table>

(Note 1) If a value exceeding the command range is commanded, a program error will occur. (P35 Setting value range over)

Rotation speed and rotation direction

The hobb axis and workpiece axis rotation speed and rotation direction during tool/spindle synchronization control 2 (hobb machining) are as follows.

(1) The rotation speed and rotation direction of hobb axis are the rotation speed commanded with the S command and the rotation direction commanded with the M command, etc., for the spindle selected as the hobb axis.

(2) The workpiece axis rotation speed is determined by the hobb threads and number of gear teeth commanded with G114.3.

$$Sw = Sh * \frac{E}{L}$$

Sw : Workpiece axis rotation speed (r/min)
Sh : Hobb axis rotation speed (r/min)
E : Hobb axis rotation ratio (hobb threads)
L : Workpiece axis rotation ratio (number of gear teeth)

(3) The workpiece axis rotation direction is determined by the sign of the address D commanded with G114.3. In other words, when the D sign is "+", the workpiece axis will rotate in the same direction as the hobb axis, and when "–", the workpiece axis will rotate in the direction opposite the hobb axis.

(4) After tool/spindle synchronization control 2 (hobb machining) is commanded the relation of the hobb axis and workpiece axis rotation is held in all operation modes of automatic and manual modes until spindle synchronization cancel (G113) is commanded or until the spindle synchronization cancel signal is input.
Even during feed hold, the hobb axis and workpiece axis synchronization state is held.

(5) If the speed is changed in the cutting feed block during the tool/spindle synchronization control 2 (hobb machining) mode, cutting of the gears will not be guaranteed. Thus, feed hold and override will not be applied on the cutting feed block during the tool/spindle synchronization control 2 (hobb machining) mode.

Workpiece axis (C axis) operation

(1) The C axis will start rotating in synchronization with the hobb axis simultaneously with the commanding of G114.3.

(2) An automatic movement command cannot be issued to the C axis during the tool/spindle synchronization control 2 (hobb machining) mode. If commanded, an operation error (M01 1013) will occur.
If a manual movement command is issued to the C axis during the tool/spindle synchronization control 2 (hobb machining) mode, the manual movement will be superimposed on the C axis movement with tool/spindle synchronization. In this case, the C axis selection signal and axis moving signal will be output.

(3) The C axis coordinate system is not established (zero point return incomplete state) during the tool/spindle synchronization control 2 (hobb machining) mode. Thus, the automatic movement command cannot be issued to the C axis. If commanded, a program error will occur.

(4) The C axis rotation speed is determined according to the hobb axis rotation speed, so designate the hobb axis rotation speed so that the C axis cutting clamp speed is not exceeded.

(5) The C axis counter on each screen will be updated as shown below during the tool/spindle synchronization control 2 (hobb machining) mode.

(a) When C axis is a rotary-type rotation axis.
The axis will rotate in the 0.000 to 359.999 range in the normal manner.

(b) When C axis is a linear-type rotation axis
The axis will rotate in the 360° range including the machine position when hobb machining starts.

(Example)

Machine position at start of hobb machining	Rotation range
125.000 (°)	0.000 to 359.999 (°)
750.500 (°)	720.000 to1080.000 (°)
–252.200 (°)	–360.000 to 0.000 (°)

(6) If the hobb machining command is issued before the C axis completes zero point return, a program error (P430 Zero return not completed) will occur.

(7) The C axis can be used for tool/spindle synchronization control 2 (hobb machining) even when it has not completed zero point return.

(8) If the C axis is used for tool/spindle synchronization control 2 (hobb machining) mode, the C axis coordinates after too/spindle synchronization is canceled will not be guaranteed. Thus, to continue using the C axis, the dog-type zero point return must be carried out again, and the coordinate system must be set again.
(The C axis will enter the zero point return incomplete state by the tool/spindle synchronization control 2 (hobb machining) mode command.)

(9) The relation of the hobb axis and workpiece axis rotation is held in all the operation modes of automatic and manual modes after tool/spindle synchronization control 2 (hobb machining) is commanded until spindle synchronization cancel (G113) is commanded, or until the spindle synchronization cancel signal is input.

(10) The C axis selection signal and axis moving signal are not output during the tool/spindle synchronization control 2 (hobb machining) mode.

(11) The external deceleration, interlock and machine lock input signals are invalid for the C axis movement synchronized with the tool/spindle during the tool/spindle synchronization control 2 (hobb machining) mode. These are valid for manual movement commands.

(12) If a servo OFF signal is input for the C axis during the tool/spindle synchronization control 2 (hobb machining) mode, the C axis rotation will stop, and an operation error will occur.

Acceleration/deceleration control

(1) The hobb axis will carry out multi-step acceleration/deceleration with the spindle synchronization acceleration/deceleration time constant (spt) set for the spindle selected as the hobb axis.

Phase alignment control (Machine configuration capable of phase alignment)

To carry out phase alignment during hobb machining, the spindle commanded as the hobb axis must satisfy one of the following conditions. The method used for phase alignment is automatically judged by the NC according to the spindle parameter (SFNC5, GRA1 to 4, GRAB1 to 4) values set for the hobb axis.

(1) Phase alignment is possible if full-closed or semi-closed control is being used for the hobb axis, the deceleration rate is 1/1, and the hobb axis encoder has a Z phase point.

(2) If semi-closed control is being used and the decelerate rate is 1/N (N is an integer other than 1), a limit switch must be installed near the hobb axis zero point so that the hobb axis zero point can be established.

Phase alignment control (Phase alignment operation)

(1) When tool/spindle synchronization control 2 (with R command) is commanded with G114.3, the C axis commanded as the workpiece axis will enter the tool/spindle synchronization (hobb machining) control state.

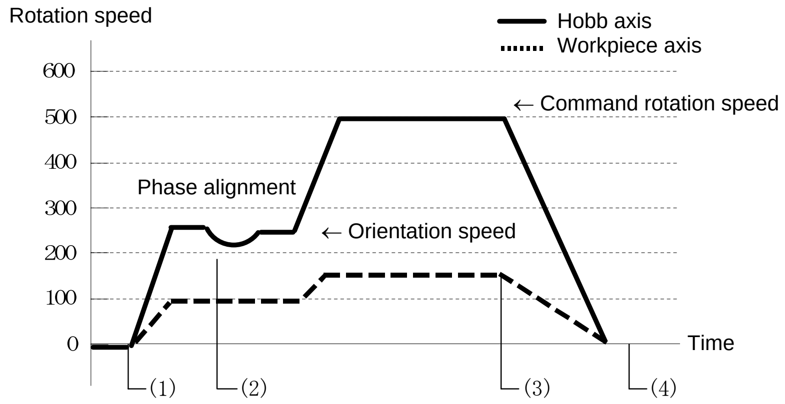
(2) The hobb axis will start rotation at the orientation rotation speed set in the parameters with the first S command issued for the hobb axis after the hobb machining control state is entered.
At this time, the C axis, which is the workpiece axis, will reach the rotation speed following the rotation ratio command for the hobb axis and workpiece axis.
If this command rotation speed is 0r/min, the hobb axis will not start rotating, and instead will wait for the next S command.

(3) The hobb axis and workpiece axis phases will be aligned in this state.

(4) After the phases are aligned, the hobb axis will accelerate/decelerate to the rotation speed commanded with the S command. The workpiece axis will accelerate/decelerate to the rotation speed obtained based on the hobb axis rotation speed allowing for the hobb axis and workpiece axis rotation ratio, and will enter the synchronized state.

(5) The following type of operation will take place.

```
Txx00;                ···· Select the rotary tool.                                             (1)
M83 S4=0;             ···· Forward run the fourth spindle (hobb axis). (Rotation speed is 0)
   :
G114.3 H4 D9 E1       ···· Hobb machining mode (phase alignment at phase difference 0) ON     (2)
L5 R0;
S4=500;               ···· Rotate fourth spindle (hobb axis) at 500r/min.
   :
M85;                  ···· Stop fourth spindle.                                                (3)
G113;                 ···· Cancel tool/spindle synchronization 2.                              (4)
```



Compensation control with workpiece axis

(1) Automatic compensation

The workpiece axis is controlled while constantly allowing for hobb axis delay (advance) caused by disturbance, etc. This is especially effective in increasing the workpiece accuracy during heavy cutting.
Automatic compensation is validated with parameters.
[Spindle NC parameter] (Machine parameter)
sps_1/BIT0 - Tool/spindle synchronization control 2 (hobb machining) compensation selection
OFF : No compensation
ON : Hobb axis delay (advance) is compensated with workpiece axis

(2) Command compensation

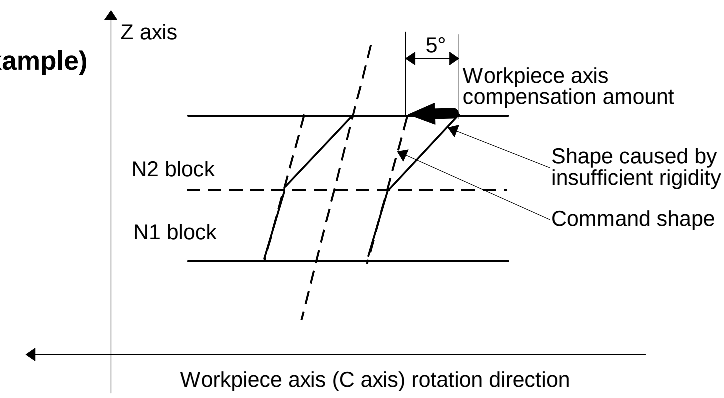
Errors in the cutting workpiece shape caused by insufficient machine rigidity, etc., is compensated with the C axis (workpiece axis) command in the machining program.

(a) Command the workpiece axis compensation amount as an incremental value.

(b) Command the workpiece axis compensation amount direction in the workpiece axis rotation direction using a "+" command, and in the direction opposite the workpiece axis rotation using a "–" command.

(c) The workpiece axis compensation amount is superimposed by accelerating/ decelerating the C axis by the commanded movement amount during rotation synchronized with the spindle.
The acceleration/deceleration mode and time constants will follow the C axis (workpiece axis) parameters.

Example)



Program

```
G114.3 H1 D9 E1 L10 P30. Q100.;
S1=100;
G98;
N1 G01 Z20. F10;
N2 G91 G01 Z20. C5.;
   :
   :
   :
```

Workpiece axis compensation amount

Example of program

(1) To not carry out phase alignment

```
 :
 :
Txx00;                ......Rotary tool selection
M83 S4=0;             ......Fourth spindle stop
G00 X40. Z-5.;
 :
 :
G114.3 H4 D5 E1 L10;  ......Tool/spindle synchronization 2 (hobb machining mode) ON
                            Hobb axis: Fourth spindle, workpiece axis: C axis
                            Hobb threads: 1, rotation ratio: 10 teeth
S4=500;               ......C axis starts forward run synchronized with S4.
                            The C axis rotation speed is 50r/min (S4 : C = 10 : 1).

M77;                  ......Spindle speed synchronization completion confirmation

G98;                  ......Asynchronous feed mode selection

G00 X18.;
G01 Z20. F10;         ......First cut
G00 X40.;                   Z axis feedrate is 10mm/min
    Z-5.;
 :
 :
G00 X14.;
G01 Z20. F10;         ......Final cut
G00 X40.;                   Z axis feedrate is 10mm/min
    Z-5.;

G113;                 ......Spindle synchronization cancel

M85;                  ......Fourth spindle stop
 :
```

(2) To carry out phase alignment

```
  :
  :
Txx00;                  ......Rotary tool selection
M83 S4=0;               ......Fourth spindle stop
G00 X40. Z-5.;
  :
  :
G114.3 H4 D5 E1 L10 R0; ......Tool/spindle synchronization 2 (hobb machining mode) ON
                              Hobb axis: Fourth spindle, workpiece axis: C axis
                              Hobb threads: 1, rotation ratio: 10 teeth
S4=500;                 ......C axis starts forward run synchronized with S4.
                              The phases are aligned at the commanded shift
                              amount 0 while rotating the hobb axis with the spindle
                              orientation speed (parameter). After the phases are
                              aligned, the hobb axis rotates at 500r/min.
                              The C axis starts forward run synchronized with S4.
                              The C axis rotation speed is 50r/min (S4 : C = 10 : 1).

M77;                    ......Spindle speed synchronization completion confirmation

G98;                    ......Asynchronous feed mode selection

G00 X18.;
G01 Z20. F10;           ......First cut
G00 X40.;                     Z axis feedrate is 10mm/min
     Z-5.;
  :
  :
G00 X14.;
G01 Z20. F10;           ......Final cut
G00 X40.;                     Z axis feedrate is 10mm/min
     Z-5.;

G113;                   ......Spindle synchronization cancel

M85;                    ......Fourth spindle stop
  :
```

Precautions for programming

(1) The axis address (X, Z, C, U, W) command in the same block as G114.3 will be ignored.
(Example) G114.3 ･･･ X_;
└──── Ignored.

(2) If a modal is commanded in the same block as G114.3, the modal will be updated.
(Example) G114.3 ･･･ G01_;
└──── The group 01 modal is set to G01.

(3) If an miscellaneous command (M, S, T) is commanded in the same block as G114.3, the miscellaneous command will be executed simultaneously with the change to the rotary tool machining mode.
(Example) G114.3 ･･･ M03;
└──── M03 is executed simultaneously with G114.3.

(4) If there is a group 00 G code command in the same block as G114.3, the G code commanded last in the block will have the priority.
(Example) G114.3 ･･･ G04 P30;
└──── G04 P30 is executed.

(5) The tool/spindle synchronization control 2 (hobb machining) mode cannot be commanded during the spindle synchronization control (spindle rotation synchronization) mode or the tool/ spindle synchronization control 1 (polygon machining) mode.
Only one of the three types of spindle synchronization control can be commanded at once.

Precautions and restrictions

(1) The tool/spindle synchronization control 2 (hobb machining) mode cannot be commanded during the spindle synchronization control (spindle rotation synchronization) mode or tool/ spindle synchronization control 1 (polygon machining) mode.
Only one of the three types of spindle synchronization control can be commanded at once.

(2) When cutting helical gears, correct cutting feed will not be possible in the synchronous feed mode, so always cut in the asynchronous feed mode.

(3) The spindle orientation rotation speed (parameter: sori) for executing phase alignment must be set less than the following value when using a limit switch.

$$\text{sori} < (\theta/360 \times 60000) / (t1 + t2)$$

θ : Angle (°) between hobb axis zero point and position where limit switch turns ON
t1 : NC interpolation cycle (ms)
t2 : Time for signal from limit switch to be input in NC (ms)

(Example) When θ = 30 (°), interpolation cycle = 7.1 (ms), signal conveyance time = 10 (ms)

$$\text{sori} < (30/360 \times 60000) / (7.1 + 10) = 293 \text{ (r/min)}$$

(4) To carry out phase alignment when machining spur gears, correct phase alignment will not be possible if the Z axis is moving, so always carry out phase alignment control when the Z axis is stopped.

(5) The linear-type rotation axis for the absolute position system cannot be used as the hobb machining workpiece axis. If used, an error (Absolute position lost 0002) will occur after the power is turned OFF/ON.

(6) If hobb machining control is carried out using the linear-type rotation axis as the hobb axis, the current value will be illegal when the hobb machining is canceled. In this case, preset the counter after canceling hobb machining.

(7) Execute the G114.3 command when the hobb axis rotation speed is 0, and the spindle rotation command (forward or reverse run) is ON.
If G114.3 is commanded while the hobb axis is rotating or when the rotation command is OFF, the following types of trouble will occur.

(a) Phase alignment will not be carried out correctly.

(b) If the hobb axis rotation speed is high, the workpiece axis will suddenly accelerate and the servo alarm will occur.

(8) If address E is set to 0, phase alignment will not be carried out. Address R will be ignored even if commanded.

(9) Command E as 0 to command hobb machining with a sub-micrometric system.
The submicron system cannot be used if address E is set to a value other than 0.

(10) Issue the commands for the C axis (workpiece axis) from the machining program as a G0 incremental value or G1 incremental value. If issued with a G0 absolute value, the C axis (workpiece axis) will not move as commanded because of the rotation axis shortcut function.

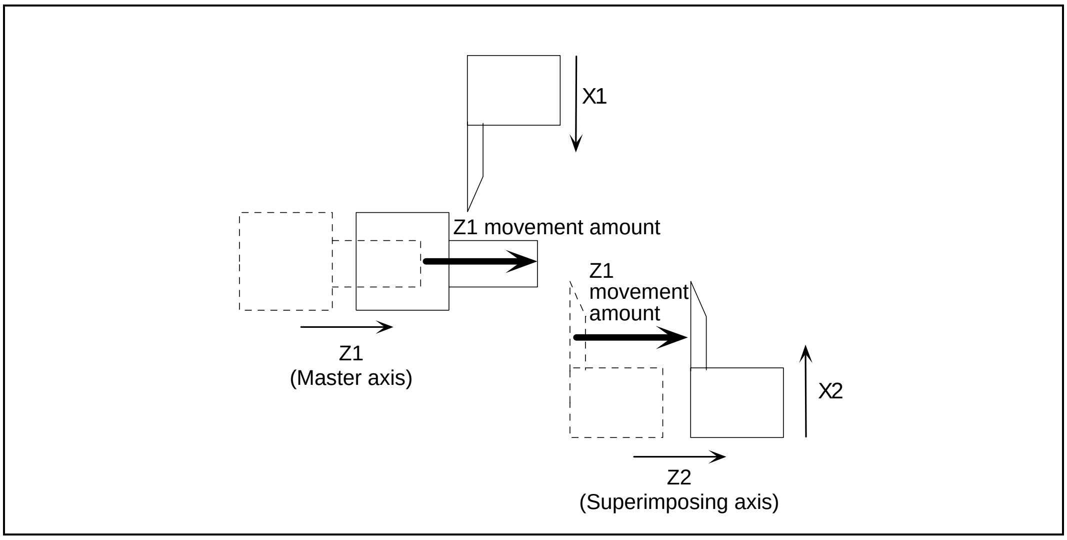
17.11 Control Axis Superimposition; G126

Function and purpose

An axis in a randomly selected system can be superimposed with another system's axis and controlled.
The superimposition of up to three axes can be controlled.

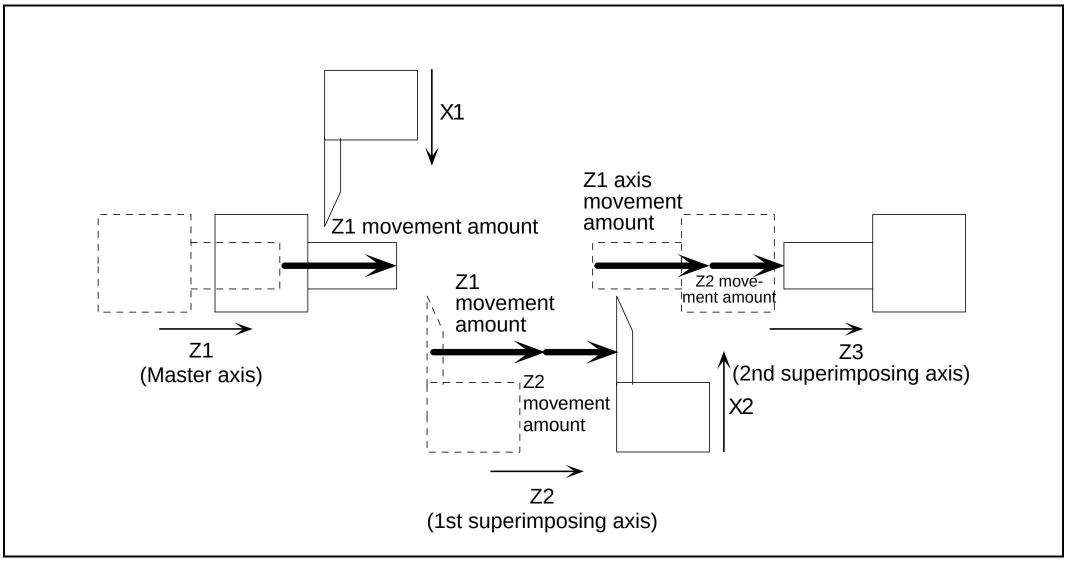
<For 2-axis superimposition>

The superimposing axis (Z2) will move by the amount that the master axis (Z1) moves.



<For 3-axis superimposition>

The 1st superimposing axis (Z2) and 2nd superimposing axis (Z3) will move by the amount that the master axis (Z1) moves.
The 2nd superimposing axis (Z3) will also move by the amount that the 1st superimposing axis (Z2) moves.



Command format

(1) Superimposition start command

G126 Superimposing axis name = Master axis name;	
	Spindle superimposition control start
Superimposing axis name	Command the axis to be handled as the superimposing axis during superimposition control.
Master axis name	Command the axis to be handled as the master axis during superimposition control.

The superimposition start command can be issued even from a system that does not contain a master axis or superimposing axis.

(Note) Command the name set in the parameters (axname2) for each name.
If an axis that is not set in the parameters is commanded, a program error (P32 illegal error) will occur.

(2) Superimposition end command

G126 Superimposing axis name;	
	Spindle superimposition control end
Superimposing axis name	Command the axis to be handled as the superimposing axis during superimposition control.

The superimposition end command can be issued even from a system that does not contain a superimposing axis.

(Note) Command the name set in the parameters (axname2) for each axis.
If an axis that is not set in the parameters is commanded, a program error (P32 Illegal address) will occur.
If an axis that is not the superimposing axis is commanded during superimposition control, this command will be ignored.

Explanation of operation

(1) Superimposition start

G126 Z2=Z1;
:
:
:
:

1) Wait for all axes in system containing Z2/Z1 axes to complete deceleration.
2) Change time constants for all axes in system containing Z2/Z1 axes to 2-axis superimposition time constants set in parameters.
3) Start 2-axis superimposition control of Z2 and Z1 axes.

G126 Z3=Z2;
:
:
:
:

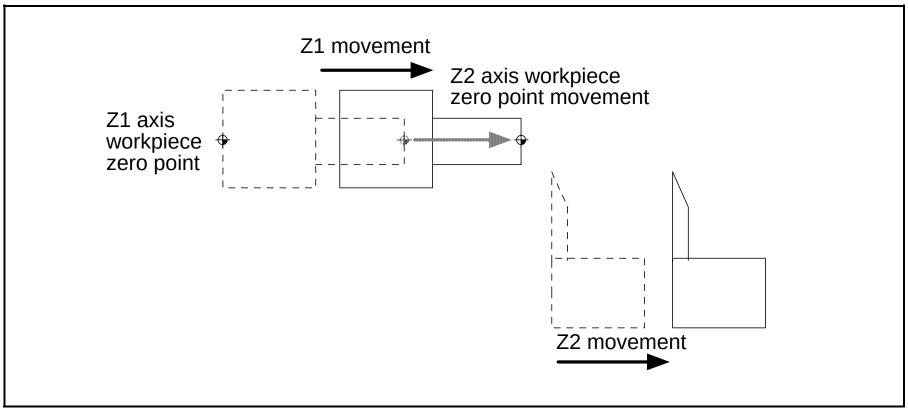
4) Wait for all axes in system containing Z3/Z2/Z1 axes to complete deceleration.
5) Change time constants for all axes in system containing Z3/Z2/Z1 axes to 3-axis superimposition time constants set in parameters.
6) Start 3-axis superimposition control of Z3, Z2 and Z1 axes.

(Note) Establish the workpiece coordinates for the superimposition axes (Z2, Z3) after starting superimposition using the workpiece coordinate system shift command (G92) in the machining program.

(2) Superimposing axis workpiece coordinate system

When the master axis moves, the superimposing axis workpiece coordinate zero point will move according to that movement. The superimposing axis will move by the amount that the master axis moves to maintain the workpiece position.

(Example) Master axis: Z1, superimposing axis: Z2



(3) Master axis and superimposing axis feedrate

(a) For 2-axis superimposition

When movement commands are issued to both the master axis and superimposing axis, if the direction that the superimposing axis moves in synchronization with the master axis and the direction that the superimposing axis moves with its own command are the same, the superimposing axis movement speed will be faster than compared to when moving with its own command.
In this case, the load applied on the superimposing axis will be too large when using the normal speed clamp process, so the master axis, superimposing axis rapid traverse rate and clamp speed are calculated using the following table.

Master axis / Superimposition	Stop	Rapid traverse	Cutting feed	Thread cutting
Stop		Master axis: Follows rapid Superimposing axis: Stop	Master axis: Follows clamp Superimposing axis: Stop	Master axis: **Note 1** Superimposing axis: Stop
Rapid traverse	Master axis: Stop Superimposing axis: Follows rapid	Master axis: Follows plrap0 Superimposing axis: Follows plrap0	Master axis: Follows clamp Superimposing axis: Follows plrap1	Master axis: **Note 1** Superimposing axis: Follows plrap1
Cutting feed	Master axis: Stop Superimposing axis: Follows clamp	Master axis: Follows plrap1 Superimposing axis: Follows clamp	Master axis: Follows plclmp Superimposing axis: Follows plclmp	Master axis: **Note 2** Superimposing axis: Follows plclmp
Thread cutting	Master axis: Stop Superimposing axis: **Note 1**	Master axis: Follows plrap1 Superimposing axis: **Note 1**	Master axis: Follows plclmp Superimposing axis: **Note 2**	Master axis: **Note 2** Superimposing axis: **Note 2**

(Note 1) If the spindle rotation speed exceeds "clamp" when starting thread cutting, the cutting will not start.
(M01 operation error 0107 Spindle rotation speed over will occur.)

(Note 2) If the spindle rotation speed exceeds "plclmp" when starting thread cutting, the cutting will not start.
(M01 operation error 0107 Spindle rotation speed over will occur.)

(b) For 3-axis superimposition

In the same manner as 2-axis superimposition, the movement of the 1st superimposing axis and 2nd superimposing axis will be faster than when using the 1st superimposing axis and 2nd superimposing axis movement commands, so the 1st superimposing axis, 2nd superimposing axis rapid traverse rates and clamp speeds are calculated using the following table.

			Master				
			Stop	Rapid traverse		Cutting feed (thread cutting)	
Super-imposition 1	**Superimposition 2**			**+**	**–**	**+**	**–**
Stop	Stop		Master – – –	Master+rapid	Master–rapid	Master+clamp	Master–clamp
			Superimposing 1 – – –	Superimposing 1 – – –	Superimposing 1 – – –	Superimposing 1 – – –	Superimposing 1 – – –
			Superimposing 2 – – –	Superimposing 2 – – –	Superimposing 2 – – –	Superimposing 2 – – –	Superimposing 2 – – –
	Rapid traverse	+	Master – – –	Master+plrap0	Master–rapid	Master+clamp	Master–clamp
			Superimposing 1 – – –	Superimposing 1 – – –	Superimposing 1 – – –	Superimposing 1 – – –	Superimposing 1 – – –
			Superimposing 2+ rapid	Superimposing 2+ plrap0	Superimposing 2+ rapid	Superimposing 2+ plrap1	Superimposing 2+ rapid
		–	Master – – –	Master+rapid	Master–plrap0	Master+clamp	Master–clamp
			Superimposing 1 – – –	Superimposing 1 – – –	Superimposing 1 – – –	Superimposing 1 – – –	Superimposing 1 – – –
			Superimposing 2– rapid	Superimposing 2– rapid	Superimposing 2– plrap0	Superimposing 2– rapid	Superimposing 2– plrap1
	Cutting feed (thread cutting)	+	Master – – –	Master+plrap1	Master–rapid	Master+plclmp	Master–clamp
			Superimposing 1 – – –	Superimposing 1 – – –	Superimposing 1 – – –	Superimposing 1 – – –	Superimposing 1 – – –
			Superimposing 2+ clamp	Superimposing 2+ clamp	Superimposing 2+ clamp	Superimposing 2+ plclmp	Superimposing 2+ clamp
		–	Master – – –	Master+rapid	Master–plrap1	Master+clamp	Master–plclmp
			Superimposing 1 – – –	Superimposing 1 – – –	Superimposing 1 – – –	Superimposing 1 – – –	Superimposing 1 – – –
			Superimposing 2– clamp	Superimposing 2– clamp	Superimposing 2– clamp	Superimposing 2– clamp	Superimposing 2– plclmp

Superimposition 1		Superimposition 2		Master: Stop	Master: Rapid traverse +	Master: Rapid traverse –	Master: Cutting feed (thread cutting) +	Master: Cutting feed (thread cutting) –
Rapid traverse	+	Stop		Master –––	Master+plrap0	Master–rapid	Master+clamp	Master–clamp
				Superimposing 1+ rapid	Superimposing 1+ plrap0	Superimposing 1+ rapid	Superimposing 1+ plrap1	Superimposing 1+ rapid
				Superimposing 2 –––	Superimposing 2 –––	Superimposing 2 –––	Superimposing 2 –––	Superimposing 2 –––
		Rapid traverse	+	Master –––	Master+pl3rap0	Master–rapid	Master+clamp	Master–clamp
				Superimposing 1+ plrap0	Superimposing 1+ pl3rap0	Superimposing 1+ rapid	Superimposing 1+ pl3rap1	Superimposing 1+ plrap0
				Superimposing 2+ plrap0	Superimposing 2+ pl3rap0	Superimposing 2+ rapid	Superimposing 2+ pl3rap1	Superimposing 2+ plrap0
			–	Master –––	Master+plrap0	Master–rapid	Master+clamp	Master–clamp
				Superimposing 1+ rapid	Superimposing 1+ plrap0	Superimposing 1+ rapid	Superimposing 1+ plrap1	Superimposing 1+ rapid
				Superimposing 2– rapid	Superimposing 2– rapid	Superimposing 2– rapid	Superimposing 2– rapid	Superimposing 2– rapid
		Cutting feed (thread cutting)	+	Master –––	Master+pl3rap1	Master–rapid	Master+pl3clmp1	Master–clamp
				Superimposing 1+ plrap1	Superimposing 1+ pl3rap1	Superimposing 1+ rapid	Superimposing 1+ pl3rap2	Superimposing 1+ plrap1
				Superimposing 2+ clamp	Superimposing 2+ clamp	Superimposing 2+ clamp	Superimposing 2+ pl3clmp1	Superimposing 2+ clamp
			–	Master –––	Master+plrap0	Master–rapid	Master+clamp	Master–clamp
				Superimposing 1+ rapid	Superimposing 1+ plrap0	Superimposing 1+ rapid	Superimposing 1+ plrap1	Superimposing 1+ rapid
				Superimposing 2– clamp	Superimposing 2– clamp	Superimposing 2– clamp	Superimposing 2– clamp	Superimposing 2– clamp
	–	Stop		Master –––	Master+rapid	Master–plrap0	Master+clamp	Master–clamp
				Superimposing 1– rapid	Superimposing 1– rapid	Superimposing 1– plrap0	Superimposing 1– rapid	Superimposing 1– plrap1
				Superimposing 2 –––	Superimposing 2 –––	Superimposing 2 –––	Superimposing 2 –––	Superimposing 2 –––
		Rapid traverse	+	Master –––	Master+rapid	Master–plrap0	Master+clamp	Master–clamp
				Superimposing 1– rapid	Superimposing 1– rapid	Superimposing 1– plrap0	Superimposing 1– rapid	Superimposing 1– plrap1
				Superimposing 2+ rapid	Superimposing 2+ rapid	Superimposing 2+ rapid	Superimposing 2+ rapid	Superimposing 2+ rapid
			–	Master –––	Master+rapid	Master–pl3rap0	Master+clamp	Master–clamp
				Superimposing 1– plrap0	Superimposing 1– rapid	Superimposing 1– pl3rap0	Superimposing 1– plrap0	Superimposing 1– pl3rap1
				Superimposing 2– plrap0	Superimposing 2– rapid	Superimposing 2– pl3rap0	Superimposing 2– plrap0	Superimposing 2– pl3rap1
		Cutting feed (thread cutting)	+	Master –––	Master+rapid	Master–plrap0	Master+clamp	Master–clamp
				Superimposing 1– rapid	Superimposing 1– rapid	Superimposing 1– plrap0	Superimposing 1– rapid	Superimposing 1– plrap1
				Superimposing 2+ clamp	Superimposing 2+ clamp	Superimposing 2+ clamp	Superimposing 2+ clamp	Superimposing 2+ clamp
			–	Master –––	Master+rapid	Master–pl3rap1	Master+clamp	Master–pl3clmp1
				Superimposing 1– plrap1	Superimposing 1– rapid	Superimposing 1– pl3rap1	Superimposing 1– plrap1	Superimposing 1– pl3rap2
				Superimposing 2– clamp	Superimposing 2– clamp	Superimposing 2– clamp	Superimposing 2– clamp	Superimposing 2– clamp

				Master				
				Stop	Rapid traverse		Cutting feed (thread cutting)	
Super-imposition 1		Superimposition 2			+	–	+	–
Cutting feed (thread cutting)	+	Stop		Master –––	Master+plrap1	Master–rapid	Master+plclmp	Master–clamp
				Superimposing 1+ clamp	Superimposing 1+ clamp	Superimposing 1+ clamp	Superimposing 1+ plclmp	Superimposing 1+ clamp
				Superimposing 2 –––	Superimposing 2 –––	Superimposing 2 –––	Superimposing 2 –––	Superimposing 2 –––
		Rapid traverse	+	Master –––	Master+pl3rap1	Master–rapid	Master+pl3clmp1	Master–clamp
				Superimposing 1+ clamp	Superimposing 1+ clamp	Superimposing 1+ clamp	Superimposing 1+ pl3clmp1	Superimposing 1+ clamp
				Superimposing 2+ plrap1	Superimposing 2+ pl3rap1	Superimposing 2+ rapid	Superimposing 2+ pl3rap2	Superimposing 2+ plrap1
			–	Master –––	Master+plrap1	Master–plrap0	Master+plclmp	Master–clamp
				Superimposing 1+ clamp	Superimposing 1+ clamp	Superimposing 1+ clamp	Superimposing 1+ plclmp	Superimposing 1+ clamp
				Superimposing 2– rapid	Superimposing 2– rapid	Superimposing 2– plrap0	Superimposing 2– rapid	Superimposing 2– plrap1
		Cutting feed (thread cutting)	+	Master –––	Master+pl3rap2	Master–rapid	Master+pl3clmp0	Master–clamp
				Superimposing 1+ plclmp	Superimposing 1+ pl3clmp1	Superimposing 1+ clamp	Superimposing 1+ pl3clmp0	Superimposing 1+ plclmp
				Superimposing 2+ plclmp	Superimposing 2+ pl3clmp1	Superimposing 2+ clamp	Superimposing 2+ pl3clmp0	Superimposing 2+ plclmp
			–	Master –––	Master+plrap1	Master–plrap1	Master+plclmp	Master–plclmp
				Superimposing 1+ clamp	Superimposing 1+ clamp	Superimposing 1+ clamp	Superimposing 1+ plclmp	Superimposing 1+ clamp
				Superimposing 2– clamp	Superimposing 2– clamp	Superimposing 2– clamp	Superimposing 2– clamp	Superimposing 2– plclmp
	–	Stop		Master –––	Master+rapid	Master–plrap1	Master+clamp	Master–plclmp
				Superimposing 1– clamp	Superimposing 1– clamp	Superimposing 1– clamp	Superimposing 1– clamp	Superimposing 1– plclmp
				Superimposing 2 –––	Superimposing 2 –––	Superimposing 2 –––	Superimposing 2 –––	Superimposing 2 –––
		Rapid traverse	+	Master –––	Master+plrap0	Master–plrap1	Master+clamp	Master–plclmp
				Superimposing 1– clamp	Superimposing 1– clamp	Superimposing 1– clamp	Superimposing 1– clamp	Superimposing 1– plclmp
				Superimposing 2+ rapid	Superimposing 2+ plrap0	Superimposing 2+ rapid	Superimposing 2+ plrap1	Superimposing 2+ rapid
			–	Master –––	Master+rapid	Master–pl3rap1	Master+clamp	Master–pl3clmp1
				Superimposing 1– clamp	Superimposing 1– clamp	Superimposing 1– clamp	Superimposing 1– clamp	Superimposing 1– pl3clmp1
				Superimposing 2– plrap1	Superimposing 2– rapid	Superimposing 2– pl3rap1	Superimposing 2– plrap1	Superimposing 2– pl3rap2
		Cutting feed (thread cutting)	+	Master –––	Master+plrap1	Master–plrap1	Master+plclmp	Master–plclmp
				Superimposing 1– clamp	Superimposing 1– clamp	Superimposing 1– clamp	Superimposing 1– clamp	Superimposing 1– plclmp
				Superimposing 2+ clamp	Superimposing 2+ clamp	Superimposing 2+ clamp	Superimposing 2+ plclmp	Superimposing 2+ clamp
			–	Master –––	Master+rapid	Master–pl3rap2	Master+clamp	Master–pl3clmp0
				Superimposing 1– plclmp	Superimposing 1– clamp	Superimposing 1– pl3clmp1	Superimposing 1– plclmp	Superimposing 1– pl3clmp0
				Superimposing 2– plclmp	Superimposing 2– clamp	Superimposing 2– pl3clmp1	Superimposing 2– plclmp	Superimposing 2– pl3clmp0

(4) Superimposition start

The case for executing the end operation from the Z3-Z2-Z1 3-axis superimposition state is shown below.

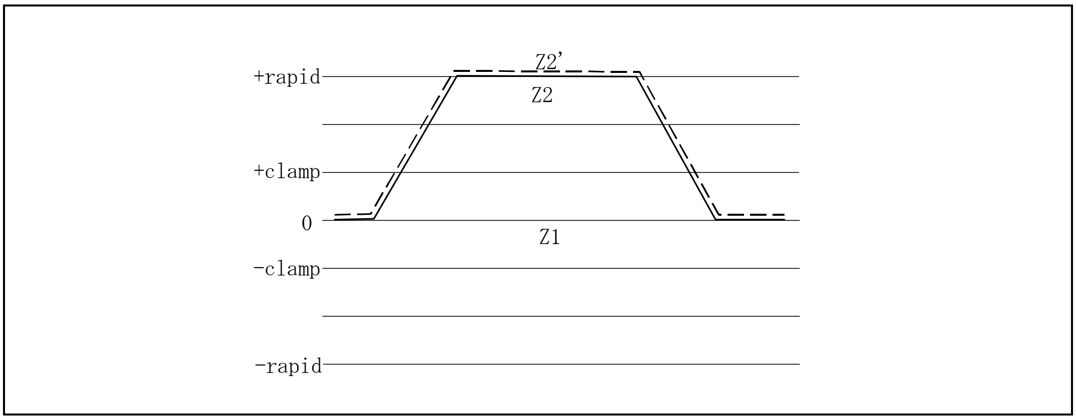
G126 Z2;	1) Wait for all axes in system containing Z3/Z2/Z1 axes to complete deceleration.
:	2) Change time constants for all axes in system containing Z3/Z2 axes to 2-axis superimposition time constants set in parameters. Change time constants for all axes in system containing Z1 axis to normal time constants set in parameters.
:	3) Z2, Z1 axis superimposition control ends, and Z3, Z2 axes 2-axis superimposition control state is entered.
G126 Z3;	4) Wait for all axes in system containing Z3/Z2 axes to complete deceleration.
:	5) Change time constants for all axes in system containing Z3/Z2 axes to normal time constants set in parameters.
:	6) Z3, Z2 axes 2-axis superimposition control ends.

(Note) Establish the workpiece coordinates for the superimposition axes (Z2, Z3) after starting superimposition using the workpiece coordinate system shift command (G92) in the machining program.

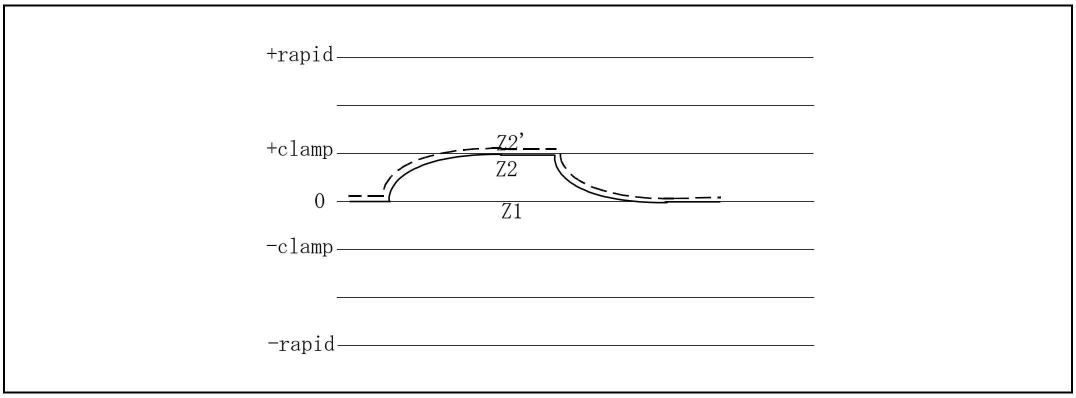
Composing superimposing axis movement

(Example) 2-axis superimposition with master axis: Z1, superimposing axis: Z2
Z1 in the figures shows the operation of only the master axis, Z2 indicates only the operation of the superimposing axis, and Z2' indicates ((master axis) + superimposing axis)).

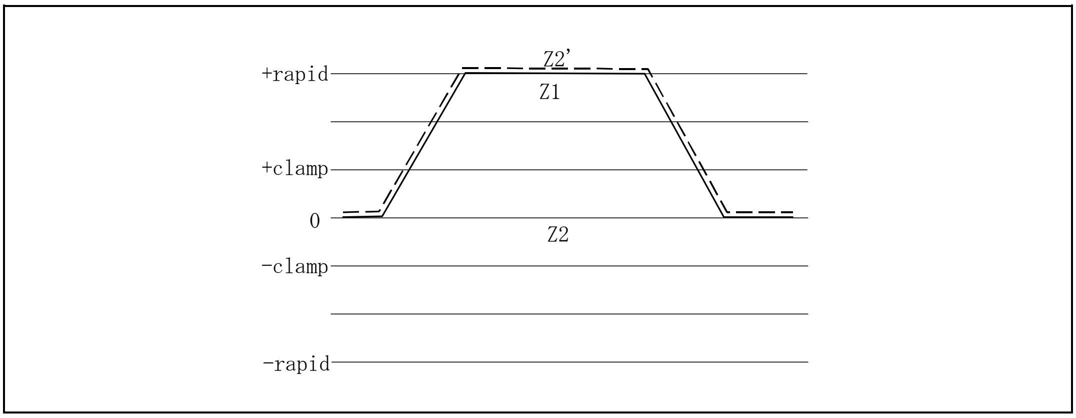
(1) Z1 stop, Z2 rapid traverse



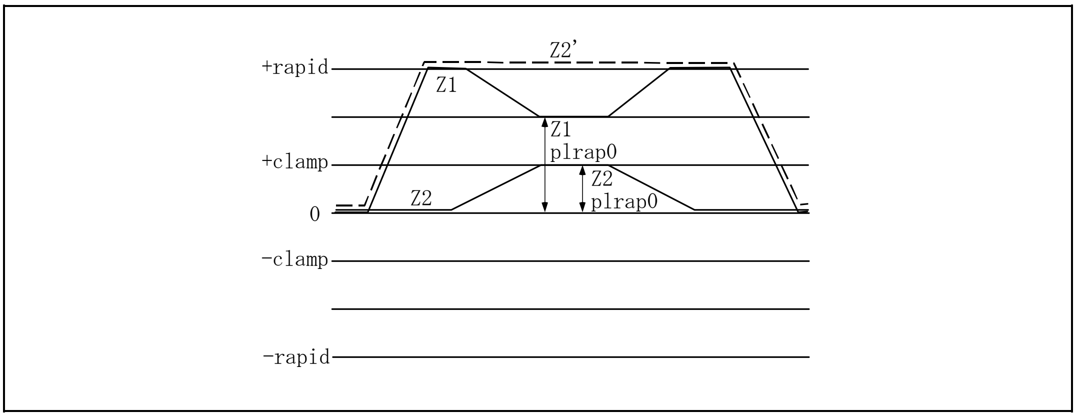
(2) Z1 stop, Z2 cutting feed



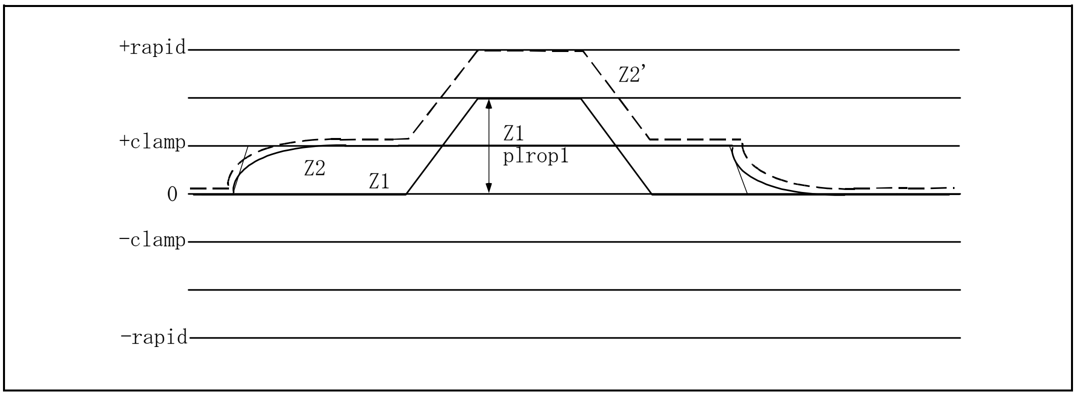
(3) Z1 rapid traverse, Z2 stop



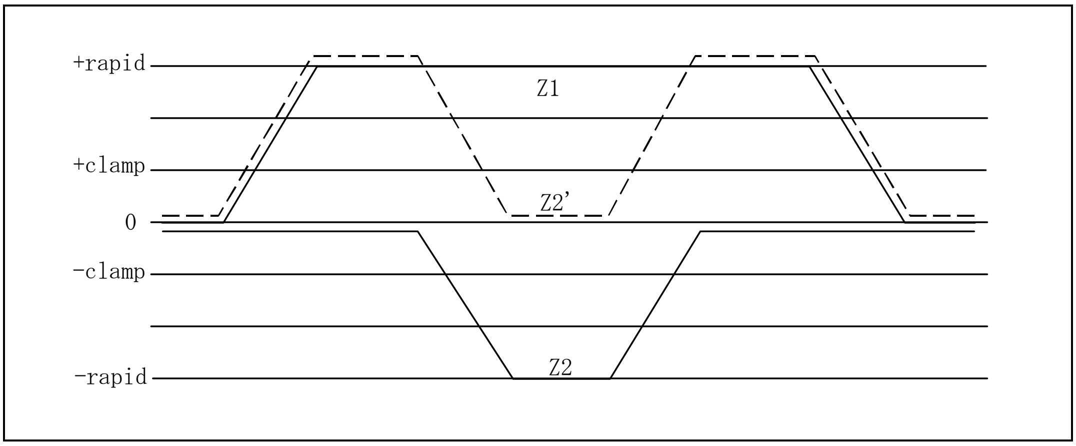
(4) Z1 rapid traverse, Z2 rapid traverse (same direction)



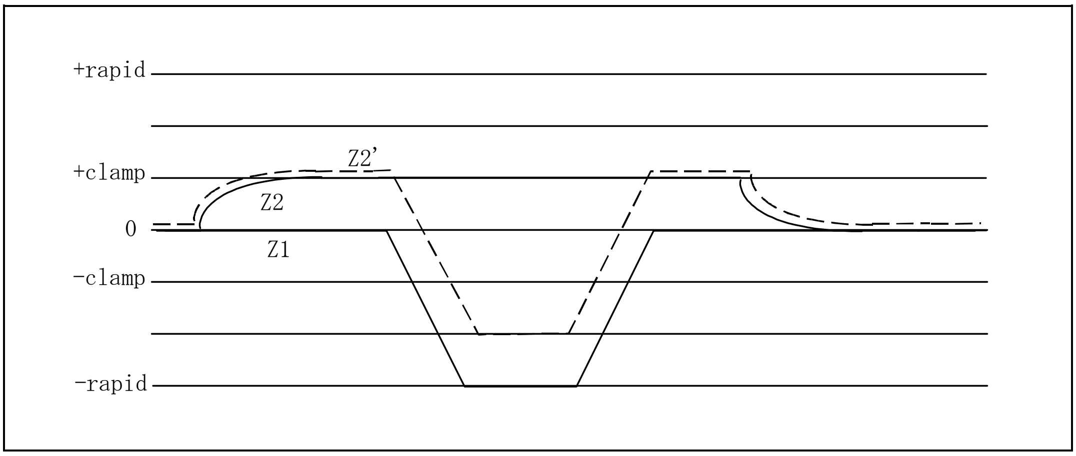
(5) Z1 rapid traverse, Z2 cutting feed (same direction)



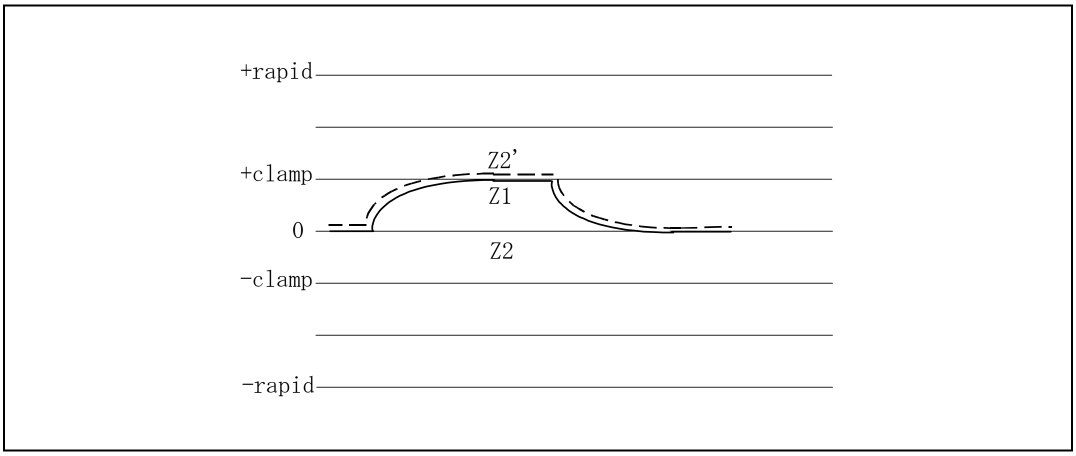
(6) Z1 rapid traverse, Z2 rapid traverse (different direction)



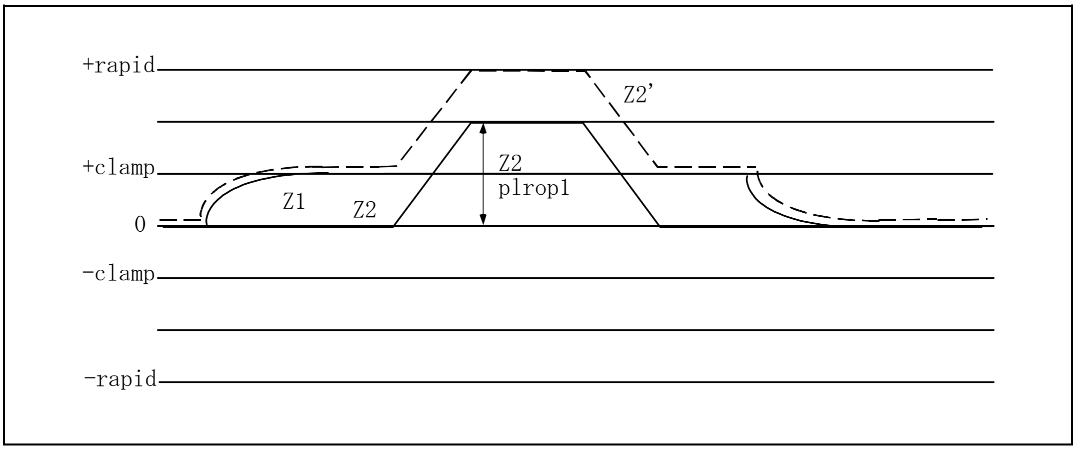
(7) Z1 rapid traverse, Z2 cutting feed (different direction)



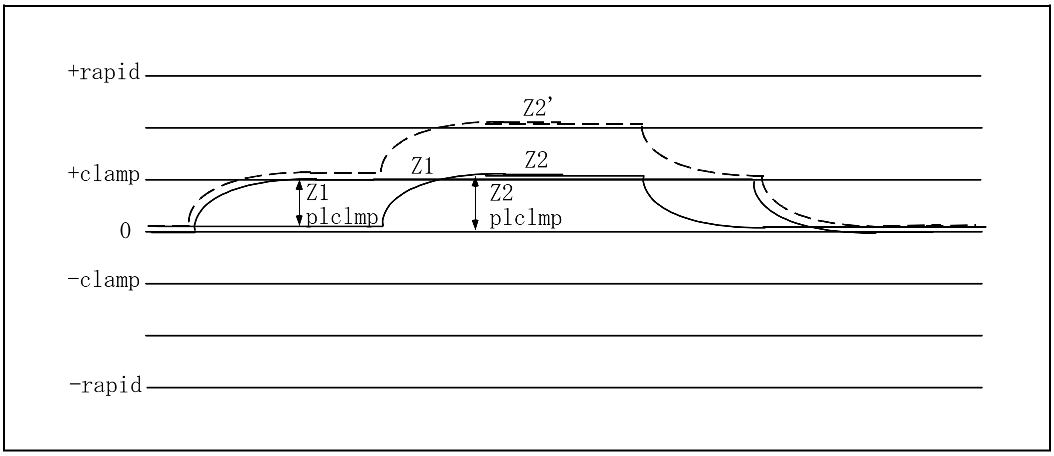
(8) Z1 cutting feed, Z2 stop



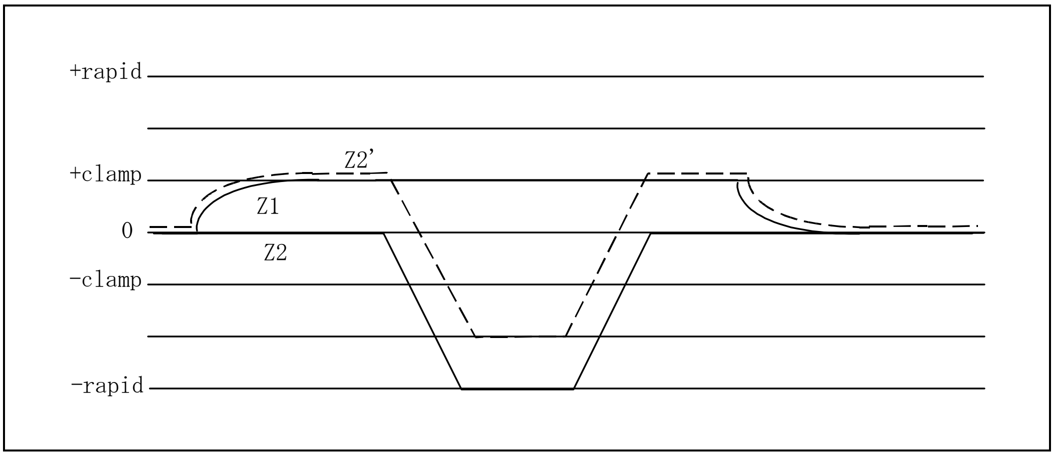
(9) Z1 cutting feed, Z2 rapid traverse (same direction)



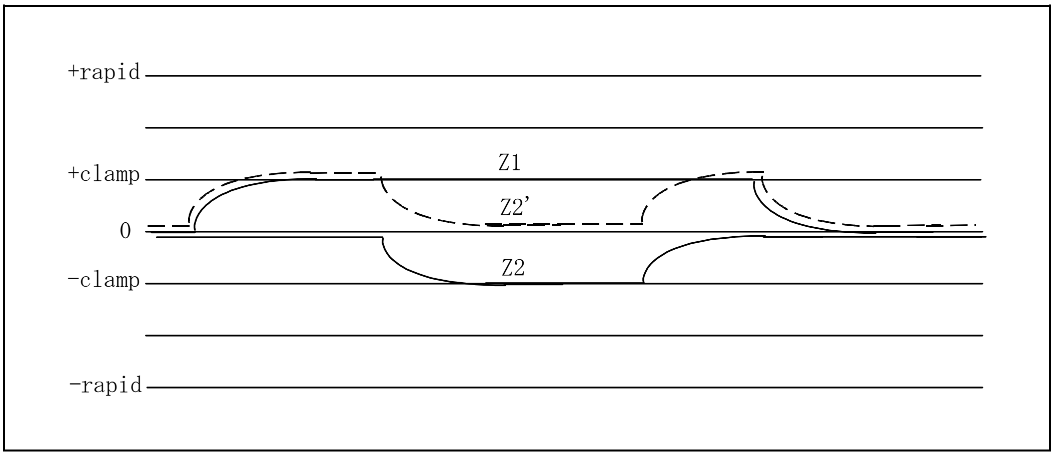
(10) Z1 cutting feed, Z2 cutting feed (same direction)



(11) Z1 cutting feed, Z2 rapid traverse (different direction)



(12) Z1 cutting feed, Z2 cutting feed (different direction)



Example of program with 2-axis superimposition control

(1) When issuing the command from a system containing the master axis

[System 1]	[System 2]
Machining program	Machining program
↓	↓
N10 !L1;	↓
N11 G126 Z2=Z1;	N15 !L1;
↓	↓

The Z2 axis is superimposed onto the Z1 axis by the command in the N11 block of system 1.

(2) When issuing the command from a system that contains the superimposing axis

[System 1]	[System 2]
Machining program	Machining program
↓	↓
N10 !L1;	N5 !L1;
N11 G04 X0;	N6 G126 Z2=Z1;
↓	↓

The Z2 axis is superimposed onto the Z1 axis by the command in the N6 block of system 2.

(3) When issuing the command from a system that does not contain either the superimposing axis or the master axis

[System 1]	[System 2]	[System 3]
Machining program	Machining program	Machining program
↓	↓	↓
N10 !2 !3L1;	N6 !1 !3L1;	N20 !1 !2L1;
N11 G04 X0;	↓	N21 G126 Z2=Z1;
↓	↓	↓

The Z2 axis is superimposed onto the Z1 axis by the command in the N21 block of system 3.

Precautions and restrictions

(1) Command the axis name set in the parameters (base specification parameter "axname2") for the master axis and superimposing axis.
If a name that is not set in the parameters is commanded, a program error (P32 Illegal address) will occur.

(2) The superimpose and superimpose cancel commands can be issued even from a system that does not contain the superimposing axis and master axis.

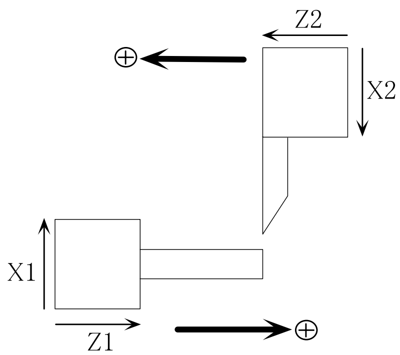
(3) The superimposition command is canceled when reset.

(4) When issuing the superimpose and superimpose cancel commands, the synchronization operation command for establishing the timing between the superimposing axis and master axis must be issued in the block before the superimpose and superimpose cancel command.

(5) The superimpose and superimpose cancel commands are valid when the master axis is stopped. Therefore, when issuing the superimpose and superimpose cancel commands from a system that does not contain the master axis, a dwell command should be issued in the block either before or after the block with the synchronization operation command for the system that contains the master axis. If the dwell command is not issued, the superimpose and superimpose cancel commands will be executed after the master axis has stopped.

(6) The relative polarities of the control axes are set in the "polar" machine parameter/axis specification parameter. Any axis may be used as the master axis for polarity setting.
For example, the parameters are set as follows when control axes Z1 and Z2 are related as shown on the below.

- Z1 axis: polar = 0 (+)
- Z2 axis: polar = 1 (–)



(7) If a rotation axis is commanded for the master axis or superimposing axis, a program error (P33 Format error) will occur.

(8) If a control axis superimposition command containing a master axis or superimposing axis (slave axis) is executed during control axis synchronization, an alarm (M01 operation error 1004) will occur.

(9) The superimposition command can be issued only for one pair of axes in the G126 block.
If two or more are commanded, a program error (P32 Illegal address) will occur.

(10) When superimpose commands are issued simultaneously from different systems, the command of the system with the highest number is given precedence.

(Example 1) Z3 is superimposed onto Z2, and Z2 is superimposed onto Z1.

[System 1]	**[System 2]**	**[System 3]**
Machining program	Machining program	Machining program
↓	↓	↓
!2 !3L1;	!1 !3L1;	!1 !2L1;
G126 Z2=Z1;	G126 Z3=Z2;	↓
↓	↓	↓

(Example 2) Z1 is superimposed onto Z2.

[System 1]	**[System 2]**	**[System 3]**
Machining program	Machining program	Machining program
↓	↓	↓
!2 !3L1;	!1 !3L1;	!1 !2L1;
G126 Z2=Z1;	G04 X0;	G126 Z1=Z2;
↓	↓	↓

(11) If one axis has been superimposed onto another axis when the superimpose cancel command has not been issued, the previous superimposition will be canceled.

(Example) Z2 is superimposed onto Z1 in the N10 block of system 1, the superimposition of Z2 onto Z1 in the N31 block of system 3 is canceled, and Z2 is superimposed onto Z3.

[System 1]	**[System 2]**	**[System 3]**
Machining program	Machining program	Machining program
↓	↓	↓
N10 !L1;	N5 !L1;	↓
N11 G126 Z2=Z1;	↓	↓
↓	↓	↓
N20 !1 !3L2;	N10 !1 !3L2;	N30 !1 !2L2;
↓	↓	N31 G126 Z2=Z3;
↓	↓	↓

(12) The superimpose command can be issued for up to two levels (superimposition for three axes).

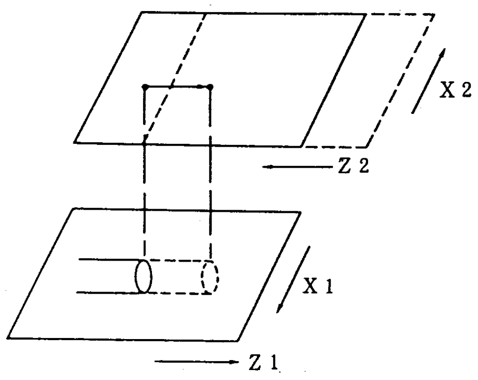
[System 1]	**[System 2]**	**[System 3]**
Machining program	Machining program	Machining program
↓	↓	↓
N5 !L1;	N8 !L1;	↓
N6 G04 X0;	N9 G126 Z2=Z1;	↓
↓	↓	↓
N10 !2 !3L2;	↓	N29 !1 !2L2;
N11 G04 X0;	N20 !1 !3L2;	N30 G126 Z3=Z2;
↓	N21 G04 X0;	↓
↓	↓	↓

(13) An alarm occurs if the issued axis does not exist as the basic definition axis or if the superimposing axis and master axis are not parallel axes (with the same name).

(14) An alarm occurs when executing a superimpose command that includes two axes during synchronization.

(15) An alarm occurs when the cross machining command is issued to the superimposing axis or master axis during superimposition.

(16) When the control axis superimpose command is issued, the coordinate system of the superimposing axis will move with the movement of the master axis.



Movement of Z1 master axis when Z2 is superimposed onto Z1, and movement of coordinate system of superimposing axis Z2.

Relation with other functions

(1) Commands not usable during control axis superimposition

If the following commands are issued to the master axis or superimposing axis during control axis superimposition, an alarm (M01 operation error 1003, 1004) will occur.

	Commands causing operation error 1003	Commands causing operation error 1004
Master axis	Dog-type zero point return (G28) Skip command (G31)	Control axis synchronization command (G125)
Superimposing axis	Zero point return command (G28 to G30) Skip command (G31) Machine coordinate system selection command (G53)	

(2) Hardware OT, soft limit and interference check alarms during control axis superimposition control

If a hardware OT, soft limit or interference check alarm occurs with the master axis or superimposing axis during control axis superimposition, the operation will differ according to the parameter (otsys) setting value.

(a) When otsys is 0

If a hardware OT, soft limit or interference check alarm occurs with the axis in the system containing the master axis and superimposing axis, the system containing the master axis and superimposing axis will stop.

(b) When otsys is 1

If a hardware OT, soft limit or interference check alarm occurs with any axis, all axes in the entire system will stop.

17.12 Spindle Superimposition; G164, G113

Function and purpose

In a machine having two or more spindles, the commanded rotation speed for spindle B (superimposing spindle) is superimposed onto the rotation speed for spindle A (master spindle), and the spindle B (superimposing spindle) rotation speed is controlled. This function can be used even with the spindle type servo. Use this function when one spindle needs to be superimposed on another spindle's rotation and rotated. For example, use this to drill a hole in the center of the workpiece with a tool spindle while rotating the workpiece grasped by the first spindle.

Command format

(1) Spindle superimposition command

Designate the master spindle and superimposing spindle, and set the superimposing spindle in the spindle superimposition state in respect to the master spindle. In the spindle superimposition state, the superimposing spindle's actual rotation speed will be the rotation speed obtained by superimposing the commanded rotation speed onto the master spindle's rotation speed.

G164 H_ D_;	**Spindle superimposition control ON**
H	Master spindle selection
D	Superimposing spindle selection

(2) Spindle superimposition cancel command

The two spindles' synchronization and superimposition state are canceled with the spindle synchronization/superimposition command.

G113;	**Spindle superimposition control OFF**

Add-ress	Meaning of address	Command range (unit)	Remarks
H	Master spindle selection Command the number of the spindle to be used as the master spindle.	1 to 6 Command machine parameter Sname value	• If there is no command, a program error will occur. (P33 Format error). • If a spindle that does not exist is commanded, a program error will occur. (P35 Setting value range over)
D	Superimposing spindle selection Command the number of the spindle to be used as the superimposing spindle.	1 to 6 or –1 to –6 Command the machine parameter Sname value or the Sname value with – sign value	• If there is no command, a program error will occur. (P33 Format error). • The rotation direction of the superimposing spindle in respect to the master spindle is commanded with the D sign. • If a spindle that does not exist is commanded, a program error will occur. (P35 Setting value range over) • If the same spindle number as the master spindle number is commanded, a program error will occur. (P33 Format error)

(Note 1) If G164 is commanded to the spindle during the spindle synchronization (G114.n) mode or if G114.n is commanded to the spindle during the spindle superimposition mode, an operation error will occur. (M01 operation error 1005)

(Note 2) If a value exceeding the command range is commanded, a program error will occur. (P35 Setting value range over)

Rotation speed and rotation direction

(1) The rotation speed and rotation direction of the master spindle during the spindle superimposition control function follow the commands.
The rotation speed and rotation direction of the superimposing spindle takes into consideration the master spindle rotation speed, the rotation direction for the master spindle commanded from the program, and the rotation direction and rotation speed commanded for the superimposing spindle.

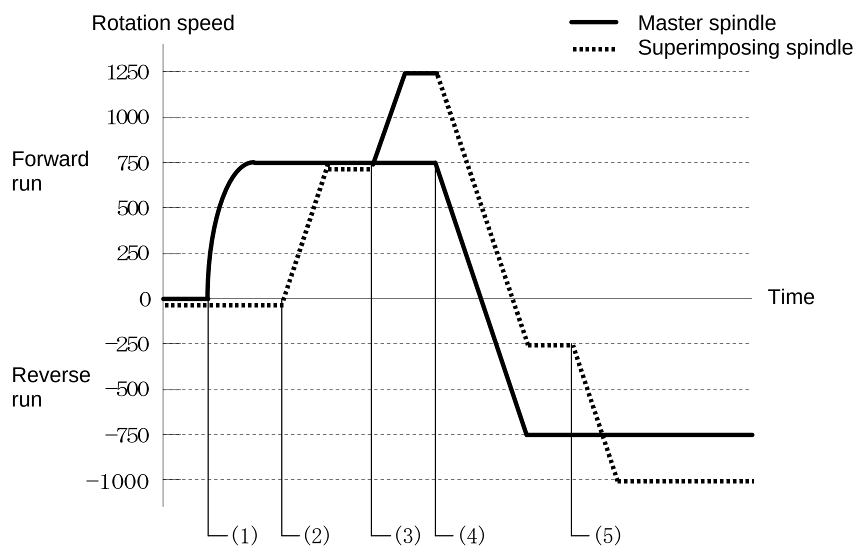
(2) The rotation speed and rotation direction for the master spindle and superimposing spindle can be changed during spindle superimposition control.

(3) The superimposing spindle rotation speed and rotation direction (signed rotation speed) are calculated with the following expression.

[Superimposition rotation speed] =
([D command sign] × [Master spindle command rotation direction] × [Master spindle command rotation speed]) + ([Superimposing spindle command rotation direction] × [Superimposing spindle command rotation speed])

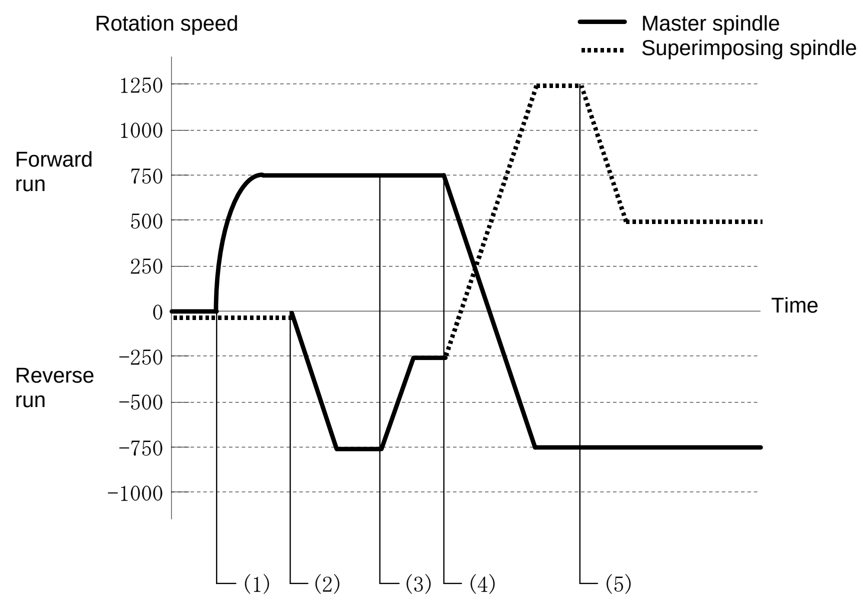
(4) Example of operation during forward run spindle superimposition control (G164 H_D_)

M**	· · · ·	Third spindle (superimposing spindle) forward run command (S3 = 0 state)	
:			
S1=750;	· · · ·	Command first spindle (master spindle) to 750r/min	
:			
M**;	· · · ·	First spindle (master spindle) forward run command	(1)
:			
G164 H1 D3;	· · · ·	Superimpose third spindle (superimposing spindle) on first spindle (master spindle) with forward run. (Superimposing spindle rotation already commanded)	(2)
:			
S3=500 ;	· · · ·	Command third spindle (superimposing spindle) to 500r/min	(3)
:			
M**;	· · · ·	First spindle (master spindle) reverse run command	(4)
:			
M** S3=250;	· · · ·	Command third spindle (superimposing spindle) to reverse run at 250r/min	(5)



(5) Example of operation during reverse run spindle superimposition control (G164 H_D_)

M**	· · · · Third spindle (superimposing spindle) forward run command (S3 = 0 state).	
:		
S1=750;	· · · · Command first spindle (master spindle) to 750r/min.	
:		
M**;	· · · · First spindle (master spindle) forward run command.	(1)
:		
G164 H1 D-3;	· · · · Superimpose third spindle (superimposing spindle) on first spindle (master spindle) with reverse run. (Superimposing spindle rotation already commanded).	(2)
:		
S3=500;	· · · · Command third spindle (superimposing spindle) to 500r/min.	(3)
:		
M**;	· · · · First spindle (master spindle) reverse run command.	(4)
:		
M** S3=250;	· · · · Command third spindle (superimposing spindle) to reverse run at 250r/min.	(5)



Explanation of operation for spindle superimposition control commands

(1) When rotation is not commanded for superimposing spindle

If a forward or reverse run command is not input for the superimposing spindle when spindle superimposition control is commanded, the superimposing spindle will not start rotating and will wait for superimposition.
When the forward or reverse run command is input while waiting for superimposition, the superimposing spindle will accelerate/decelerate to the rotation speed that considers the pre-commanded rotation speed for the master spindle and rotation direction commanded for the master spindle from the program, and the rotation direction and rotation speed commanded for the superimposing spindle, and will enter the spindle superimposition control state.
If the superimposing spindle forward run/reverse run command turns OFF during the spindle superimposition control state, the spindle superimposition control state will be canceled and the superimposing spindle will stop.

(2) When superimposing spindle is rotating

If the superimposing spindle is rotating at a random rotation speed when spindle superimposition control is commanded, the superimposing spindle will accelerate/decelerate to the rotation speed that considers the pre-commanded rotation speed for the master spindle and rotation direction commanded for the master spindle from the program, and the rotation direction and rotation speed commanded for the superimposing spindle, and will enter the spindle superimposition control state.

(3) When rotation is not commanded for master spindle

If a forward or reverse run command is not input for the master spindle when spindle superimposition control is commanded, the superimposing spindle will carry out normal operations.
When the forward or reverse run command is then input for the master spindle, the superimposing spindle will accelerate/decelerate to the rotation speed that considers the pre-commanded rotation speed for the master spindle and rotation direction commanded for the master spindle from the program, and the rotation direction and rotation speed commanded for the superimposing spindle, and will enter the spindle superimposition control state.

Rotation speed clamp

(1) Maximum rotation speed clamp

During spindle superimposition control, the master spindle will be clamped at the maximum rotation speed for the master spindle set in the parameters or the maximum rotation speed for the superimposing spindle, whichever is lower.
The superimposing spindle will be clamped if it exceeds the maximum rotation speed for the superimposing spindle.
In this case, the positional relation of the master spindle and superimposing spindle will deviate.

(2) Minimum rotation speed clamp

During spindle superimposition control, the superimposing spindle will not be clamped at the minimum rotation speed.
If a spindle rotating at the minimum rotation speed is designated as a superimposing spindle for spindle superimposition control, the minimum rotation speed clamp will be canceled, and the spindle will rotate at the commanded rotation speed.
When the superimposing spindle designation is canceled, the minimum rotation speed clamp will be validated.

17.12.1 Relation with other functions

Constant surface speed control during spindle superimposition

Constant surface speed control is valid within the following range for the master spindle and superimposing spindle during spindle superimposition control.

(1) Constant surface speed control for master spindle

Constant surface speed control can be commanded for the master spindle during spindle superimposition control.
Spindle superimposition control can be commanded while the master spindle is in constant surface speed control.

(2) Constant surface speed control for superimposing spindle

If constant surface speed control is commanded to a superimposing spindle during spindle superimposition control, the rotation speed after calculating the constant surface speed will be superimposed.
If spindle superimposition control is commanded while the superimposing spindle is in constant surface speed control, the rotation speed after calculating the constant surface speed will be superimposed.

(Note 1) The spindle clamp speed set with the G code for the superimposing spindle during spindle superimposition control is invalid.

(Note 2) If constant surface speed control is commanded to a spindle during the tap cycle or synchronous tap cycle, an operation error will occur, and the operation will stop. (M01 operation error 1027)

Synchronous tap cycle/tap cycle control

(1) Tap cycle or synchronous tap command for master spindle during spindle superimposition control

If tap cycle or synchronous tap is commanded to a master spindle during spindle superimposition control, an operation error will occur. (M01 operation error 1007)

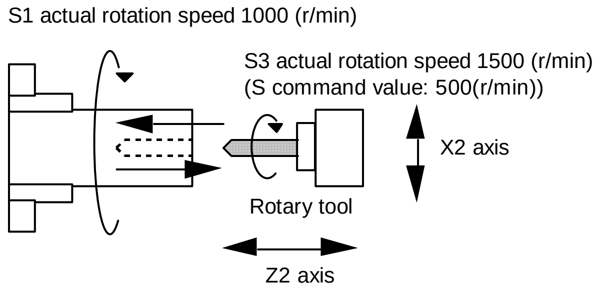
(2) Tap cycle or synchronous tap command for superimposing spindle during spindle superimposition control (differential tap)

The tap cycle or synchronous tap can be commanded to a superimposing spindle during spindle superimposition control.

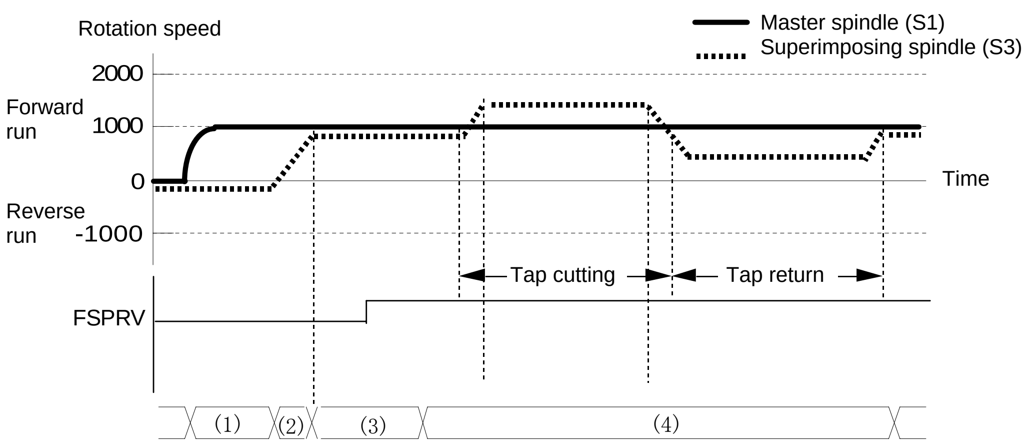
Example of operation

System 1	System 2 (control axes X: X2 axis, Z: Z2 axis)		
:	:		
S1=1000;	:		(1)
:	G164 H1 D3;	Spindle superimposition control ON	(2)
:	G0 Z-2.;		(3)
:	G84 X0. Z10. R1. F1. D3	Synchronous tap	(4)
:	S500 ,R1;		
:	:		
:	:		

By rotating the main spindle (S1) as the master spindle and the back tool spindle (S3) as the superimposing spindle, and carrying out tap machining by synchronously controlling the tap axis (Z2 axis) and tool spindle (S3) with the synchronous tap commands, differential tapping that tap machines with the back rotary tool while the main spindle is rotating can be carried out.



(Operation)



17.12.2 Precautions and restrictions

Restrictions for carrying out tap cycle or synchronous tap command to superimposing spindle

(1) If the spindle rotation speed is clamped when the tap cycle or synchronous tap command is issued during spindle superimposition control, an operation error will occur and the machining will stop. (M01 operation error 1108)

M3 S1=4000;	Spindle superimposition control ON
G164 H1 D3;	
G0 Z-2.;	
G84 X0. Z10. R1. F1. D3 S2000 ,R1;	Synchronous tap
G113;	Spindle superimposition control cancel

When the above commands are executed, the superimposition spindle's command rotation speed (command value after superimposition) will change from 2000r/min (during tap cutting) to 6000r/min (during tap return). If the superimposing spindle's clamp speed is 5000r/min, the superimposing spindle will be clamped at 5000r/min, so an operation error will occur before tap cutting is started. (M01 operation error 1108)

(2) If a command that changes the rotation speed is issued to the master spindle during the tap cycle or synchronous tap cycle of the superimposing spindle during spindle superimposition control, it will be ignored. The command will be validated after the tap cycle or synchronous tap cycle ends.
(3) If constant surface speed control is executed on a master spindle while the tap cycle or synchronous tap cycle is being executed with the superimposing spindle, an operation error will occur, and machining will stop. (M01 operation error 1109)
(4) If the tap cycle or synchronous tap cycle is commanded to the superimposing spindle while the master spindle is at the constant surface speed, an operation error will occur, and machining will stop. (M01 operation error 1109)
(5) If the synchronous tap command is issued when a rotation command is input to the master spindle/superimposing spindle while MST lock is valid, an operation error will occur, and machining will stop. If the synchronous tap command is issued when a rotation command is not input to the master spindle/superimposing spindle while MST lock is invalid, an operation error will occur, and machining will stop. (M01 operation error 1108)
Note that if the tap axis is not in the automatic machine lock state, an operation error will not occur.

General precautions and restrictions for spindle superimposition

(1) Spindle superimposition control will be canceled if emergency stop is applied.
(2) Orientation of the master spindle and superimposing spindle is not possible during the spindle superimposition control mode. Cancel the spindle superimposition control mode before starting orientation.
(3) The spindle gears cannot be changed for the master spindle or superimposing spindle during the spindle superimposition control mode. Cancel the spindle superimposition control mode before changing the spindle gears.
(4) Whether to use a D address sign in the G164 command (either forward run spindle superimposition or reverse run spindle superimposition), depends on the machine configuration (rotary tool installation direction, master spindle and superimposing spindle rotation direction, etc.)
(5) Take care to the rotation speed clamp when executing a command. When the rotation speed is clamped, the superimposing spindle cannot maintain the rotation speed difference commanded for the master spindle.
(6) Changes in the master spindle speed (S command, rotation command, override) are invalid while the superimposing spindle is tapping.
(7) The master spindle and superimposing spindle in spindle superimposition control cannot be set as the C axis with spindle/C axis control. If commanded, an operation error will occur. (M01 operation error 1026)
(8) The spindle cannot be set to the master spindle or superimposing spindle during the C axis mode with spindle/C axis control. If commanded, an operation error will occur. (M01 operation error 1026)
(9) The spindle rotation upper limit over/lower limit over is not checked for the master spindle or superimposing spindle during spindle superimposition.
(10) The command rotation speed display for the superimposing spindle during spindle superimposition will be the commanded rotation speed only for the superimposing spindle. The feedback rotation speed display will be the actual rotation speed of the superimposing spindle.
(11) If the forward run or reverse run command for the superimposing spindle turns OFF during spindle superimposing, the superimposition control state will be canceled and the superimposing spindle will stop.
If the spindle stop command turns ON or if 0 rotation is commanded, the superimposing spindle will rotate at the same rotation speed as the master spindle.
(12) If the synchronous tap command is issued with a system in constant surface speed, a program error will occur. (P182 Sync. Tap Error)
(13) If a constant surface speed command for a spindle in the tap cycle/synchronous tap cycle is commanded, or if a tap cycle/synchronous tap cycle command is issued to a spindle in constant surface speed control, an operation error will occur. (M01 operation error 1027)

17.13 2-System Simultaneous Thread-cutting Cycle

Function and purpose

The 2-system simultaneous thread-cutting cycle function allows system 1 and system 2 to perform thread-cutting simultaneously for the same spindle.
Featured in this cycle is the command (G76.1) for simultaneously cutting threads in two places, which is known as the "2-system simultaneous thread-cutting cycle I".

17.13.1 Parameter setting command

Command format

The various parameters for thread-cutting are set by commands.

G76 Pmra QΔd min Rd;

Address		Meaning
P	m	Number of cutting passes for finishing
	r	Chamfering amount This sets the chamfering width based on the thread lead 1 across a range from 0.0l to 9.9l with a 2-digit integer with the decimal point omitted.
	a	Tool nose angle (thread angle) This selects the angle from 80°, 60°, 55°, 30°, 29° or 0° and commands the value in two digits.
Q	Δdmin	Minimum cut amount When the calculated value of the cut amount for a single pass is less than "Δdmin", it is clamped by "Δdmin.".
R	d	Finishing allowance

Detailed description

(1) The data is set in setup parameters m: #8620, r: #8011, a: #8021, Δdmin: #8019 and d: #8018 for each system.

(2) Issue the command for each system.

17.13.2 2-system simultaneous thread-cutting cycle I

Command format

G76.1 X/U_ Z/W_ Ri Pk QΔd Fl;

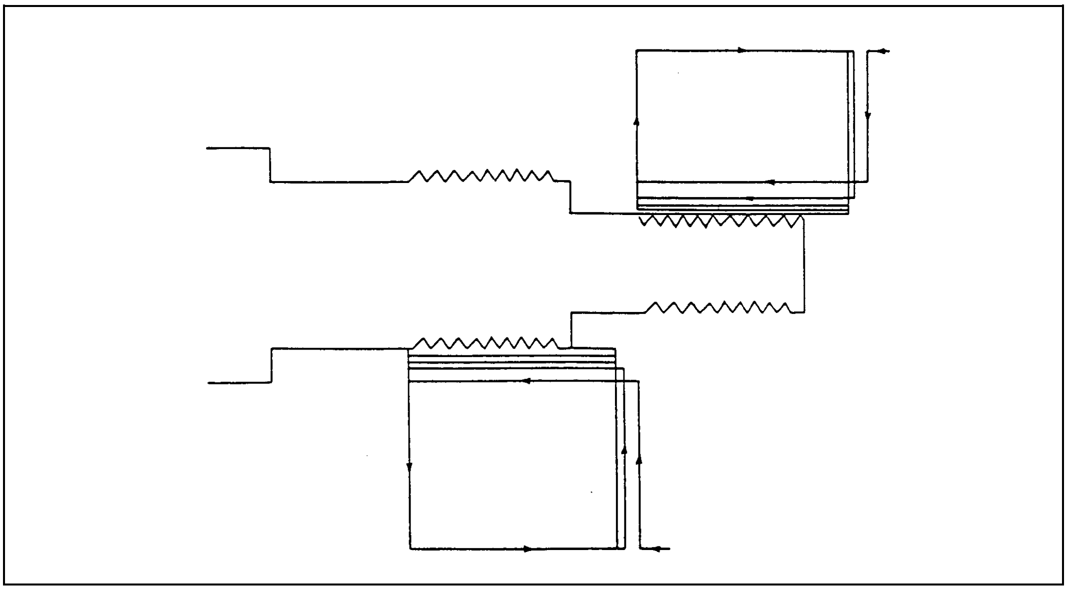
X/U	X-axis end point coordinates of thread section ... The X-axis coordinates of the end point at the thread section are commanded with absolute or incremental values.
Z/W	Z-axis end point coordinates of thread section ... The Z-axis coordinates of the end point at the thread section are commanded with absolute or incremental values.
i	Taper height component (radial value) at thread section ... A straight thread is created when i = 0.
k	Thread height ... The thread height is commanded with a positive radial value.
Δd	Cut amount ... The cut amount for the first cutting pass is commanded with a positive radial value.
l	Thread lead

Detailed description

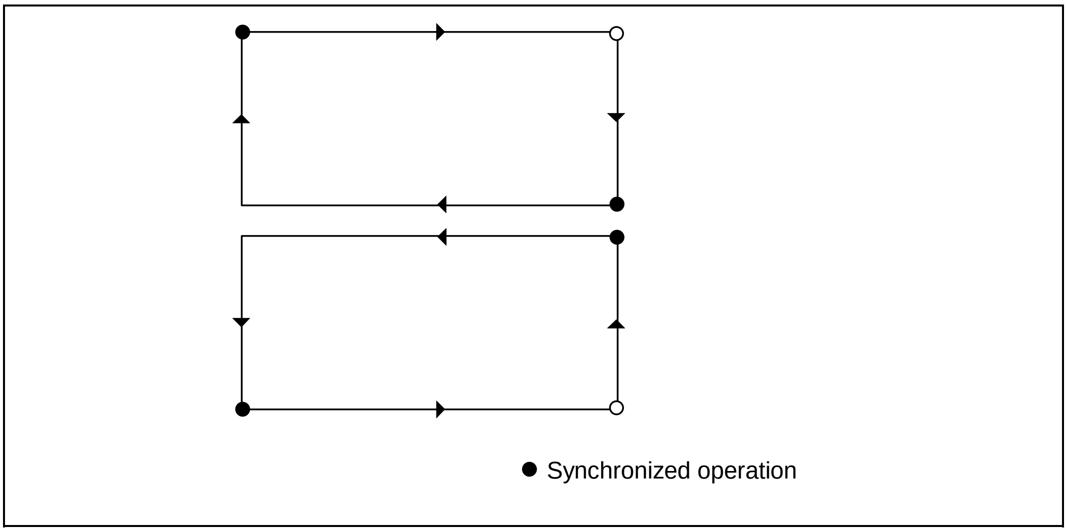
(1) When G76.1 is issued by system 1 and system 2, synchronized operation is done until the command is issued to another system. The thread-cutting cycle starts when the commands are aligned properly.

$1	$2
≀	≀
G00X_ Z_ ;	
G76.1··· ;	G00X_ Z_ ;
≀	G76.1··· ;
	≀

(2) In the G76.1 cycle, G76 is issued simultaneously by system 1 and system 2, and the thread is cut in synchronization at the start and end of thread-cutting.



(3) In one cycle, operation is synchronized at the start and end of the thread-cutting.



(4) The same precautions for thread-cutting command (G33), thread-cutting cycle (G78) and compound thread-cutting cycle (G76) apply to this cycle.

(5) As the threads are cut in two places by the G76.1 command, the various commands do not need to be the same. Each of them can be issued independently.

(6) Thread cutting controls the Z axis position while tracking the spindle encoder rotation. Thus, the relative relation of the spindle position detected by the spindle encoder and the Z axis will change with the following elements.

(a) Z axis feedrate (spindle rotation speed ∗ screw pitch)

(b) Cutting feed acceleration/deceleration time constant

(c) Position loop gain

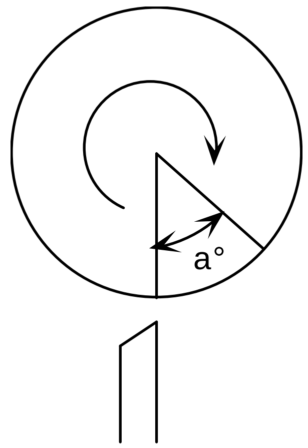
Thus, when cutting a multi-thread screw, the above conditions must be the same from the start to end of machining.

17.13.3 2-system simultaneous thread cutting cycle II

Command format

G76.2 X/U_ Z/W_ R<u>i</u> P<u>k</u> Q<u>Δd</u> A<u>a</u> F<u>1</u>;

(1) Thread cutting start shift angle

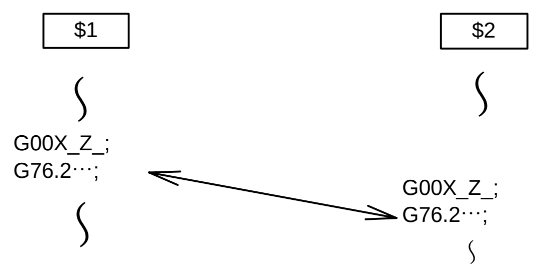


The thread cutting command starts movement after waiting for the spindle encoder's one rotation synchronization signal. However, the start point can be delayed by the amount equivalent to the thread cutting start angle.

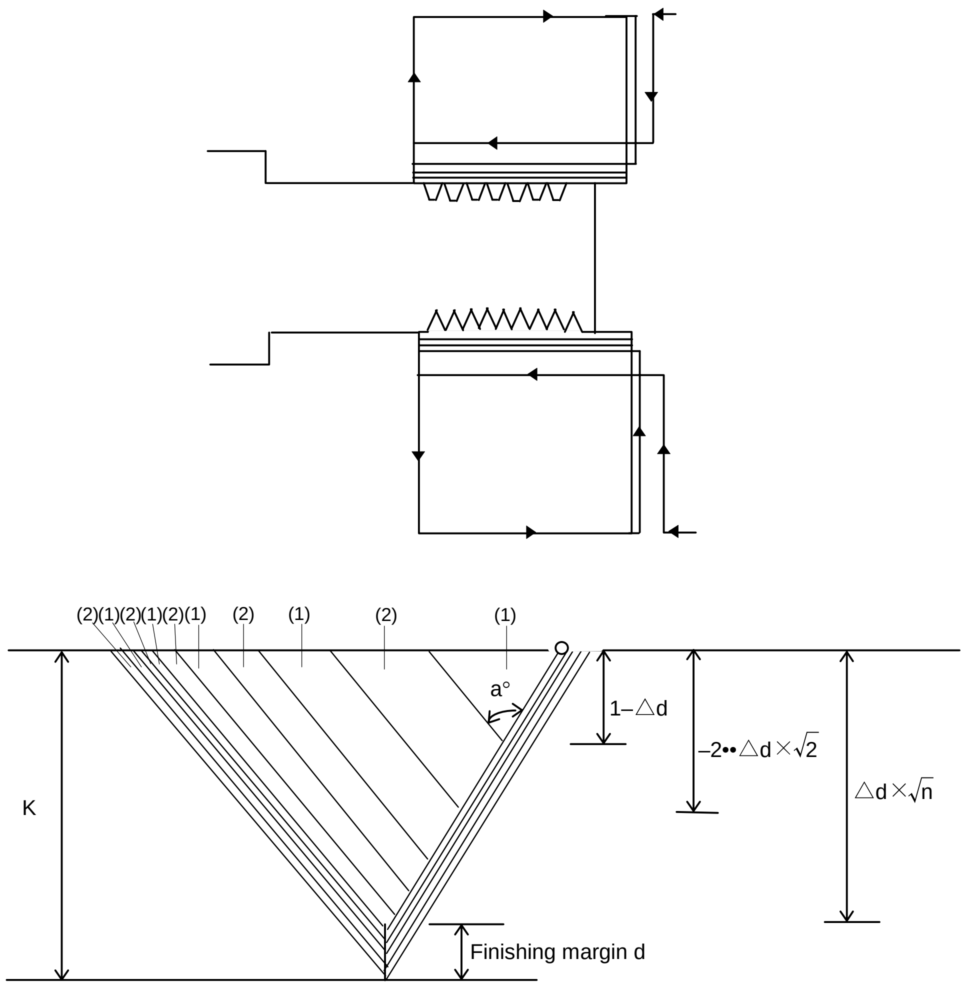
The meanings of the addresses other than A are the same as the 2-system simultaneous thread cutting cycle I.

Detailed description

(1) When G76.2 is issued by system 1 and system 2, synchronized operation is done until the command is issued to another system. The thread-cutting cycle starts when the commands are aligned properly.



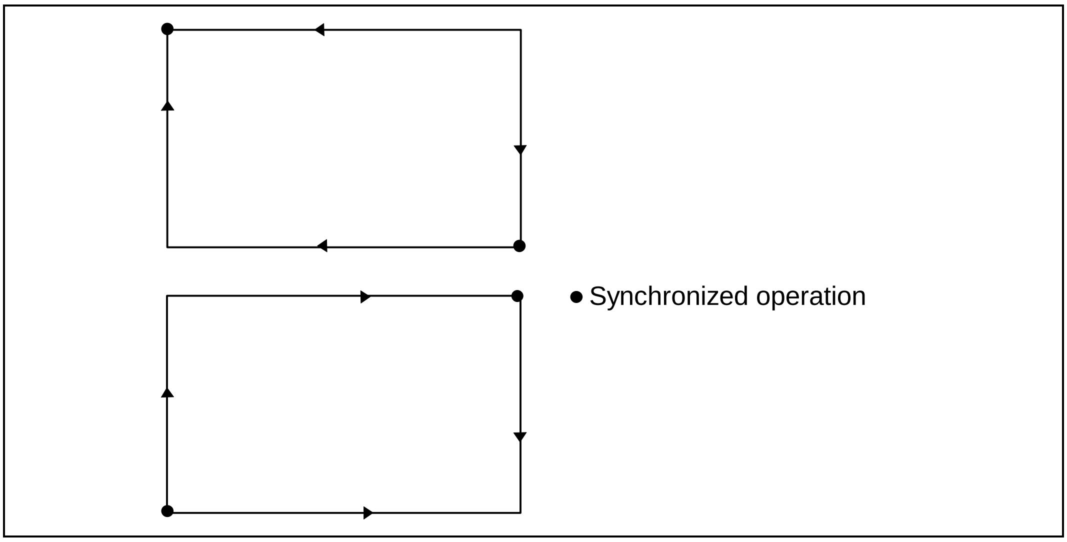
(2) G76.2 assumes the same thread cutting, and deeply cuts in with the cutting amount using system 1 and system 2 alternately.



(1) · · · Cutting with system 1

(2) · · · Cutting with system 2

(3) In one cycle, operation is synchronized at the start and end of the thread-cutting.



(4) The same precautions for thread-cutting command (G33), thread-cutting cycle (G78) and compound thread-cutting cycle (G76) apply to this cycle.

(5) G76.2 is the same thread cutting, so the various parameters, thread section, taper height, screw thread height, cutting amount and thread lead must be commanded to the same values for system 1 and system 2.
Note that the start shift angle can be commanded to match the thread cutting state.

(6) Thread cutting controls the Z axis position while tracking the spindle encoder rotation. Thus, the relative relation of the spindle position detected by the spindle encoder and the Z axis will change with the following elements.

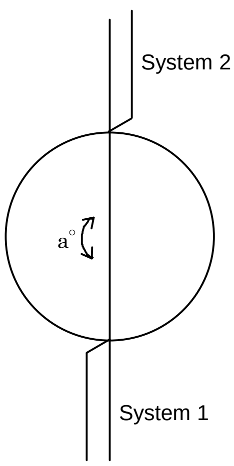
(a) Z axis feedrate (spindle rotation speed ∗ screw pitch)

(b) Cutting feed acceleration/deceleration time constant

(c) Position loop gain

Thus, with G76.2 which is same thread cutting, the parameters must be set so that the conditions are the same for system 1 and system 2.

(7) Thread cutting start shift angle command



As shown on the left, if the system 1 and system 2 blades oppose at 180°, the system 1 and system 2 thread cutting start shift angle difference is set to 180°.

Example)

$1	$2
G76.2X_ Z_ A0.;	G76.2X_ Z_ A180.;

(8) When G76.2 and G76.1 are commanded
The systems, in which each are commanded, will carry out the G76.1 and G76.2 movements. However, G76.2 system will assume that the other system is using G76.2 when cutting the threads, so the thread grooves will not be guaranteed.

18. OTHER MULTI-AXIS, MULTI-SYSTEM CONTROL FUNCTIONS

18.1 Miscellaneous Function Output during Axis Movement; G117

Function and purpose

This function controls the timing of the miscellaneous function to be output. The miscellaneous function is output when the position designated in axis movement is reached.

Command format

G117 X_ Z_ C_ □□□□ ;	
X Z C	Start point of operation
□□□□	Miscellaneous function

Detailed description

(1) This command is issued independently immediately before the block with the movement command that activates the miscellaneous function.

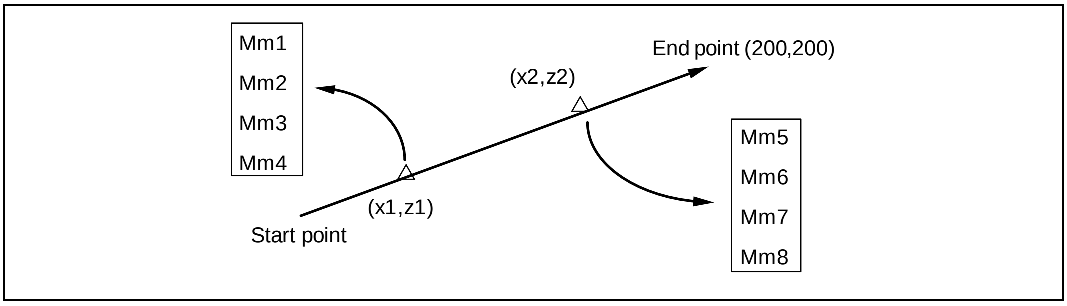
(2) Single block stop does not apply to this command.

(3) The maximum number of groups to which the miscellaneous functions in the G117 block can be issued is as follows:

M commands	: 4 sets
S commands	: 1 set each
T commands	: 1 set
2nd miscellaneous function	: 1 set

(4) This command can be issued in up to two consecutive blocks.
When issued in three or more consecutive blocks, the last two blocks will be valid.

(Example) G117 Xx1 Zz1 Mm1 Mm2 Mm3 Mm4;
G117 Xx2 Zz2 Mm5 Mm6 Mm7 Mm8;
G01 X200 Z200;
:



(5) When the operating start point commanded by G117 is not on the movement path, the miscellaneous function will be output once the movement has reached all the coordinate values of the operating start point. In addition, only the commanded axis is checked.

(Example) G117 X100. M××; M×× is output when X100. is reached.

(Note) The other axes are not subject to the check.

(6) The completion of the miscellaneous function in the previous group is checked at the operating start point, and the miscellaneous function of the next group is output. Thus, normal PLC interfacing is possible.

(7) An miscellaneous function issued in the same block as the block with the movement command is output before the movement and starts the movement. During movement, operation will not stop at the operating start point. However, at the end point of the block, the completion of all the miscellaneous functions is checked first, and then the execution of the next block is started.

(8) G117 should be issued in the sequence of operating start points. Program error occurs if the sequence of the operating start point is the reverse of the movements.
When operating start points coincide, the miscellaneous functions are output in the sequence in which they were issued.

(9) When an operating start point cannot be determined by the next block movement, the next operation is performed by the parameter.

Control parameter "#8042 Start point alarm"	Operation
ON	Program error occurs before movement
OFF	The functions are output when the next block movement is completed.

(10) The following tables show the combinations of (8) and (9).

G17 First block Second block	During intermediate point movement	Not during intermediate point movement
During intermediate point movement	Refer to (8).	Program error due to (8).
Not during intermediate point movement	Refer to (9) for second block.	Refer to (9). With output, the sequence of first block, second block is followed regardless of the sequence of the designated points.

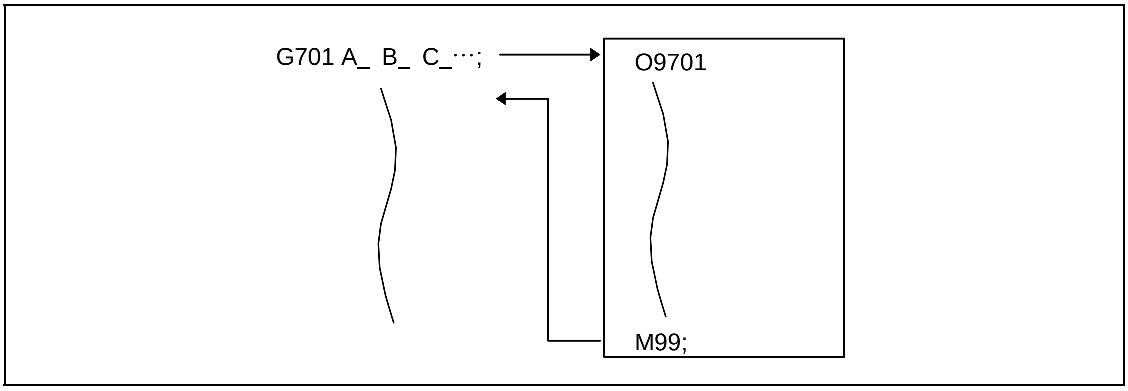
Precautions

(1) Command G117 in order of the operation start points. If the operation start point order is the opposite of the movement, a program error will occur.

18.2 G Code Macros

Function and purpose

G codes G200 to G999 can be used as user macro commands.
The NC operation is shifted to the determined program by these commands.



Command format

G □ ××;	
□	Value between 2 and 9

Detailed description

(1) The macro program numbers corresponding to the macro commands are determined by the last two digits of the G code and the value of the third digit set by the parameter.

(2) G macros return to the main program by the M99 command in the subprogram.

(3) The argument address, local variable and other macro limits comply with the macro call type of commands (M99, G65, G66, G66.1) set respectively by parameters.

18.3 Axis Name Change; G111

Function and purpose

This function is used to change the commanded axis name for two axes in the system.
When using a function in which the command axis is limited, such as the hole drilling cycle (G88), this function can be used to issue commands to axes that cannot be commanded with the normal command methods.

(Example)

The operation is as follows when the names of the X axis and Y axis in the system are changed with the axis name change command.

<Before axis name is changed>			**<After axis name is changed>**	
Command axis name	Control axes		Command axis name	Control axes
X	X axis	⇒	X	Y axis
Y	Y axis		Y	X axis

Command format

G111 □ Δ ;		**Axis name change ON**
□, Δ	Axis name	

The □ axis is changed with the Δ axis and controlled.

G111 ;	**Axis name change cancel**

The axis name change is canceled.

(Example of axis name change command and canceling)

:	
G111 XY;	Axis name change ON
G01 X100.;	The Y axis workpiece value moves to 100. with the X command.
G01 Y100.;	The X axis workpiece value moves to 100. with the Y command.
G111;	Axis name change cancel
G01 X0.;	The X axis workpiece value moves to 0.
G01 Y0.;	The Y axis workpiece value moves to 0.
:	

(Note) The axis name change command can be issued simultaneously to several systems. However, it cannot be commanded simultaneously to multiple sets within one system.

Explanation of operation

(1) G111 changes the command axes in the same system.
If an illegal axis name is designated, a program error (P32) will occur.

(2) The plane selection command (G17, G18, G19) modal will not change.
The plane for the plane selection modal is automatically determined when G111 is commanded.

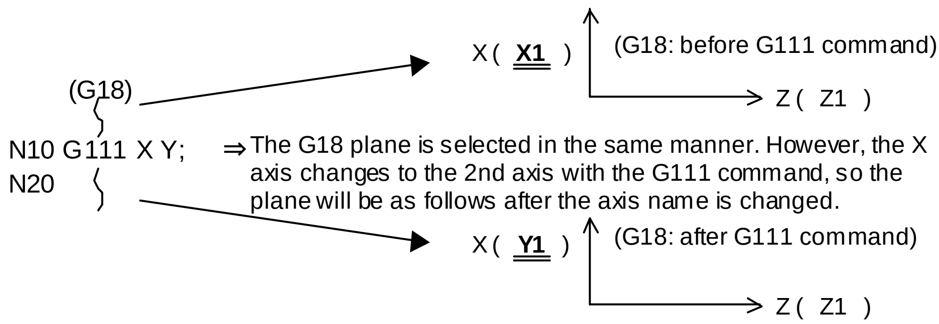
(Example)

Setup parameters
(user parameters)

#8001 Plane	<I> X
#8002	<J> Y
#8003	<K> Z
#8004 Parallel axis	<I>
#8005	<J>
#8006	<K>

Basic axis parameters
(machine parameters)

	<1>	<2>	<3>
#1001 axname	X	Y	Z
#1002 incax	U	V	W
#1009 dia	1	0	0

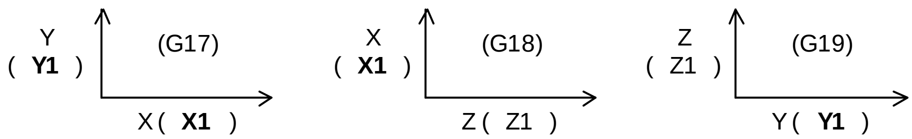


* The G111 command changes the control axis corresponding to the axis name. Thus, the G18 plane is the IK plane. In this example the plane is the XZ plane so, the control axis corresponding to the X axis name automatically changes from the 1st axis to the 2nd axis.

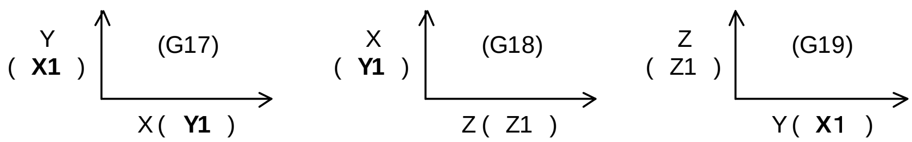
(3) The plane selection command can be commanded during axis name change.

(Example)

(a) Plane when axis name is not changed

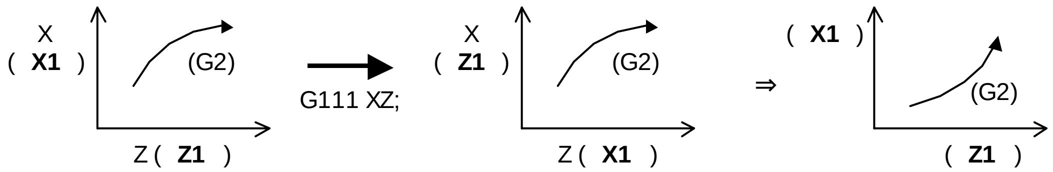


(b) Plane when axis name is changed (G111 XY;)



(4) The circular interpolation, nose R and fixed cycles are executed on the selected plane.

(Example) Circular interpolation after axis name is changed



G2 command in program | **G2 command after axis name is changed** | **Actual machine operation**

(5) When axis name change is canceled, the plane selection will return to the state before the axis name change command.

(6) The tool length and wear compensation are not affected by the axis name change.

(Example)

T 1 1 0 3

	X	Y	Z
Wear compensation (03)	0.01	0.02	0.03
Tool length compensation (11)	0.1	0.2	0.3

[Program]

```
(G111)
  ~
N10 T1103 ;
N20 G00 X10.0 Y20.0 Z30.0 ;
  ~
N30 T0 ;
N40 G00 X0 Y0 Z0 ;
  ~
N50 G111 X Y ;
N60 T1103 ;
N70 G00 X10.0 Y20.0 Z30.0 ;
```

[Machine coordinate value]

	<X1>	<Y1>	<Z1>
N20	10.11	20.22	30.33
N70	20.11	10.22	30.33

(7) If G111 is commanded during nose R compensation, a program error (P410) will occur.

(8) The incremental address and circular center designation address will also change when the axis name is changed.

(**Example**)

	X axis control	Y axis control
Command address during G111	X, U, I	X, V, J
Command address during G111 XY	Y, V, J	X, U, I

* Note that addresses not related to axis movement, such as the dwell X address, will not change.

```
G04 X2.   ;
G111 XY   ;
G04 X2.   ;    ← The dwell X address will not change with the Y command.
```

(9) If the axis for which the axis name change command has been executed is a constant surface speed axis, the constant surface speed axis will change automatically.
When G96 P_ is commanded, the axis number will be that after the axis name change.
If the P command is omitted, the constant surface speed axis will follow the basic system parameter "#1117 G96_ax". However, if a constant surface speed command for which the P command is omitted is commanded after axis name change, the axis number set in the parameters will be the axis number after axis name change.

(Example)

```
        ∫   ←――――  Command G96 P1 in this range ⇒ X (X1) is constant
                                                  surface speed axis
N10 G111 XY ;
N20
        ∫   ←――――  Command G96 P1 in this range ⇒ X (Y1) is constant
                                                  surface speed axis
```

* Note that a program error (P410) will occur if G111 is commanded during constant surface speed control.

(10) The axis coordinate value and tool length value read out with the macro system variables are fixed regardless of the axis name change.

(Example)

```
(G111) ∫
N10 G00 X100. Z200. Y300. ;
N20 #500 = #5021 ;   ←――――  <X1> machine position coordinate (100.) is substituted in #500
N30 #501 = #5022 ;   ←――――  <Z1> machine position coordinate (200.) is substituted in #501
N40 #502 = #5023 ;   ←――――  <Y1> machine position coordinate (300.) is substituted in #502
       ∫
N50 G111 X Y ;
       ∫
N60 G00 X-100. Z-200. Y-300. ;
N70 #504 = #5021 ;   ←――――  <X1> machine position coordinate (-300.) is substituted in #504
N80 #505 = #5022 ;   ←――――  <Z1> machine position coordinate (-200.) is substituted in #505
N90 #506 = #5023 ;   ←――――  <Y1> machine position coordinate (-100.) is substituted in #506
       ∫
```

* The assignment of the macro system variable axes will not change with the axis name change.

Relation with other functions

(1) Positioning (G00)
Positioning can be commanded to the axis name after axis name change.

(2) Linear interpolation (G01), circular interpolation (G02/G03), helical interpolation
Linear interpolation can be commanded to the axis name after axis name change.
Circular interpolation can be commanded to the axis name after axis name change.
Helical interpolation can be commanded to the axis name after axis name change.

(3) Milling interpolation (G12.1/G13.1)
Milling interpolation can be commanded to the axis name after axis name change.
Note that the axis name change command must not be executed during the milling interpolation mode.

(4) Thread cutting (lead/thread designation), continuous thread cutting, variable lead thread cutting (G34), circular thread cutting
Thread cutting can be commanded to the axis name after axis name change.

(5) Synchronous tapping (G84)
Synchronous tapping can be commanded to the axis name axis after axis name change.

(6) Constant surface speed control (G96, G97) (including clamp)
Always issue the constant surface speed control command while constant surface speed control is canceled.
A program error (P410) will occur if G111 is commanded during constant surface speed control.
If the constant surface speed control command is issued after axis name change, the axis name after the axis name change will be the constant surface speed control axis.

(Example) System with axis numbers (1) X (2) Z (3) Y and (4) C

G111 XY	(Axis name is changed with X and Y)
G96 S300 P1	
G01 U-30.	(X command: constant surface speed control matched to Y axis control)

(7) Nose R compensation (G41/G42/G40), nose R compensation direction automatic decision (G46/G40)
A program error (P410) will occur if G111 is commanded during nose R compensation.
If nose R compensation is commanded after axis name change, the command can be issued to the axis name after axis name change.

(8) Shape (tool length) compensation, wear compensation amount hold
Tool length compensation and wear compensation are applied on the axis before axis name change.

(9) Automatic coordinate system setting, machine coordinate system (G53), workpiece coordinate system selection (6 sets) (G54 to G59), external workpiece coordinate offset, rotation axis coordinate system
The coordinate system shift amount is the shift amount for the axis designated in the parameters regardless of the axis name change.

(10) Local coordinate system setting (G52), coordinate system setting (G50)
The local coordinate system setting and coordinate system setting command can be issued to the axis name after axis name change.

(11) Plane selection
The plane selection modal is not changed by the axis name change command.
The plane corresponding to the modal command is automatically selected when G111 is commanded.

(12) Automatic reference point return (G28)
If the automatic reference point return command (G28) is issued after the axis name change command, the intermediate point command will be issued with the axis name after axis name change.
Note that axis will return to the reference position set with the parameters for each axis regardless of the axis name change.

(13) Automatic return point designation (G29)
If the automatic return point designation (G29) is commanded after the axis name change command, the return position command will be issued with the axis name after axis name change.
The intermediate point command will be the intermediate point for the automatic reference point return command (G28) issued last.

(14) 2nd, 3rd, 4th reference point return (G30)
If the 2nd, 3rd or 4th reference point return command is issued after the axis name change command, the intermediate point command will be issued with the axis name after axis name change.
Note that the axis will return to the position set with the parameters for each axis regardless of the axis name change.

(15) Reference point check (G27)
If the reference point check command is issued after the axis name change command, the command will be issued with the axis name after axis name change.

(16) NC reset (Reset 1/2, reset & rewind)
The axis name change will not be canceled automatically when the NC is reset.
The axis name changed state is held until G111 (axis name change cancel command) is issued in the program.

(17) Hole drilling fixed cycle
The hole drilling fixed cycle can be commanded to the axis after axis name change.

(18) Linear angle command
The linear angle command can be issued to the axis after axis name change.

(19) Geometric command IA
The geometric command IA can be issued to the axis after axis name change.

(20) Start point designation wait (G115/G116)
The command axis address for start point wait is commanded with the client system's axis name. If the axis name has been changed in the client system, the command will be issued with the axis name after axis name change.

(21) Turning fixed cycle (G77/G78/G79)
The turning fixed cycle can be commanded for the axis after axis name change.

(22) Compound fixed cycle I (G70 to G73)
The compound fixed cycle can be commanded for the axis after axis name change.

(23) Cross machining command (G110)
Do not use the cross machining command together with this function.

(24) Miscellaneous function output during axis movement (G117)
The command axis address for the miscellaneous function output during axis movement is commanded with the axis name after axis name change.

(25) Control axis superimposition (G126)
If the control axis superimposition command (G126) is commanded for an axis commanded for axis name change, the control axis superimposition will be applied on the axis before axis name change.

(26) Control axis synchronization (G125)
If the control axis synchronization command (G125) is commanded for an axis commanded for axis name change, the control axis synchronization will be applied on the axis before axis name change.

(27) Emergency stop
The axis name change is not canceled automatically even after emergency stop is reset. The axis name change state is held until G111 is independently commanded in the program.

(28) Inclined coordinate rotation
Axis name change can be commanded after the G173 command to each axis after inclined coordinate rotation.
The G173 command after axis name change is valid to the address after changeover.

Precautions

(1) If an illegal axis name is designated for the axis name change command, a program error (P32) will occur.

(2) A program error (P410) will occur if G111 is commanded during the milling mode.

(3) A program error (P410) will occur if G111 is commanded during nose R.

(4) The command axis address for start point wait is commanded with the client system's axis name. If the axis name has been changed in the client system, the command will be issued with the axis name after axis name change.

(5) A program error (P410) will occur if G111 is commanded during constant surface speed control.
If constant surface speed control is issued after axis name change, the control will be applied on the axis name after axis name change.

(6) The intermediate point position when automatic reference point return command (G28) is commanded after axis name change, will be commanded with the axis name after axis name change.

(7) The return point commanded with G29 is the coordinate value designated with the axis name after axis name change.

(8) If 2nd, 3rd or 4th reference point return is commanded after axis name change, the command will be applied with the axis name after axis name change.
Note that the axis will return to the return point set with the parameters for each axis regardless of the axis name change.

(9) If the control axis superimposition command (G126) is commanded for an axis commanded for axis name change, the control axis superimposition will be applied on the axis before axis name change.

(Example of operation)

```
$1                        $2
G111 ZX
!2                        !1
G126 Z2 = Z1
G01 X-50. F500 ──────────▶ 1st system's Z axis moves with X command, and 1st system's
!2                        !1 Z axis movement is superimposed with that movement
```

(10) If the control axis synchronization command (G125) is commanded for an axis commanded for axis name change, the control axis synchronization will be applied on the axis before axis name change.

(11) Only two axes can be combined for the commanded axes when using the axis name change command. A program error (P32) will occur if one independent axis or more than three axes are commanded.

(12) The settings such as the diameter value and radius value of the two axes used for axis name change cannot be interchanged. Only the axis name is interchanged.

(13) The axis name change will not be canceled automatically when the NC is reset.
The axis name changed state is held until G111 (axis name change cancel command) is issued in the program.

(14) The axis name change is not canceled automatically even after emergency stop is reset. The axis name change state is held until G111 is independently commanded in the program.

(15) Do not command cross machining control command (G110) to the axis for which axis name change has been commanded. The "M01 operation error 1101" will occur if cross machining control command (G110), in which the axis configuration changes, is executed in a system for which the axis names have been changed, or if the axis name is changed in a system having an axis configuration that changes with the cross machining control command (G110).

⚠CAUTION

⚠ Do not issue another axis name change command before axis name change cancel is issued once axis name change is commanded.

APPENDIX 1 LIST OF FUNCTION CODES

Function code ISO	NC unit recognition	CRT display	Setting display unit key-in	Stored in memory	Internal NC unit function
0 ~ 9	Yes	Displayed	Key-in	Stored	Numerical data
A ~ Z	Yes	Displayed	Key-in	Stored	Addresses
+	Yes	Displayed	Key-in	Stored	Sign, variable operator (+)
–	Yes	Displayed	Key-in	Stored	Sign, variable operator (–)
.	Yes	Displayed	Key-in	Stored	Decimal point
,	Yes	Displayed	Key-in	Stored	Address function selection
/	Yes	Displayed	Key-in	Stored	Block delete (optional block skip) Variable operator (+)
%	Yes	Displayed (%)	No key-in (automatically inserted)	Stored	End of record (tape storage end) Rewind start & stop during tape search
LF/NL	Yes	Displayed (;)	Key-in ;/EOB	Stored	End of block
(	Yes	Displayed (;)	Key-in ;/EOB	Stored	Control out (comment start)
)	Yes	Displayed (;)	Key-in ;/EOB	Stored	Control out (comment end)
:	Yes	Displayed (;)	No key-in	Stored	Program number address (instead of O)
#	Yes	Displayed (;)	Key-in	Stored	Variable number
*	Yes	Displayed (;)	Key-in	Stored	Variable operator (×)
=	Yes	Displayed (;)	Key-in	Stored	Variable definition
[	Yes	Displayed (;)	Key-in	Stored	Variable operator
]	Yes	Displayed (;)	Key-in	Stored	Variable operator
!	Yes	Displayed (;)	Key-in	Stored	Synchronized operation
$	Yes	Displayed (;)	Key-in	Stored	System selection
BS	No	Blank	No key-in	Stored	
HT	No	Blank	No key-in	Stored	
SP	No	Blank	Key-in	Stored	
CR	No	Blank	No key-in	Stored	
DEL	No	No displayed	No key-in	Not stored	
NULL	No	No displayed	No key-in	Not stored	
(DEL)	No	No displayed	No key-in	Not stored	
Any other the above	No	**(Note 2)**	No key-in	Stored	

(Note 1) Codes not listed in the above table are stored on tape, but an error will result during operation if they are not comments.

(Note 2) This denotes characters (including blanks) which are stored inside the NC unit and which correspond to the command codes. "@" is not displayed.

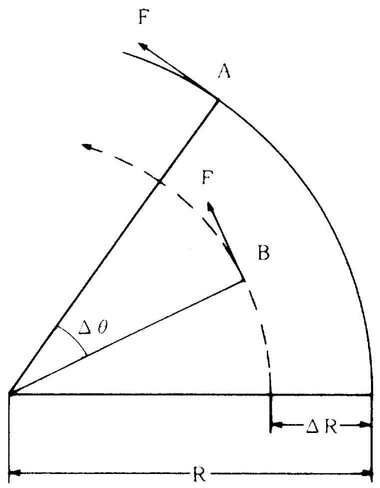
APPENDIX 2 LIST OF COMMAND VALUES AND SETTING RANGES

	Linear axis		Rotation axis
	Input unit (mm)	**Input unit (inch)**	**Degree (°)**
Minimum input setting unit	0.0001/0.001/0.01	0.00001/0.0001/0.001	0.001/0.01
Maximum stroke (value with machine coordinate system)	±9999.9999mm ±99999.999mm ±99999.99mm	±999.99999inch ±9999.9999inch ±9999.999inch	±9999.9999° ±99999.999° ±99999.99°
Maximum command value	±9999.9999mm ±99999.999mm ±99999.99mm	±999.99999inch ±9999.9999inch ±9999.999inch	±9999.9999° ±99999.999° ±99999.99°
2nd zero point offset (value with machine coordinate system)	±9999.9999mm ±99999.999mm ±99999.99mm	±999.99999inch ±9999.9999inch ±9999.999inch	±9999.9999° ±99999.999° ±99999.99°
Tool offset amount (tool length)	±99.9999mm ±999.999mm ±999.99mm	±9.99999inch ±99.9999inch ±99.999inch	
Tool offset amount (wear)	±9.9999mm ±99.999mm ±99.99mm	±0.99999inch ±9.9999inch ±9.999inch	
Incremental feed amount	0.0001mm/pitch 0.001mm/pitch	0.00001inch/pitch 0.0001inch/pitch	0.0001°/pitch 0.001°/pitch
Handle feed amount	0.0001mm/pitch 0.001mm/pitch	0.00001inch/pitch 0.0001inch/pitch	0.0001°/pitch 0.001°/pitch
Soft limit range (value with machine coordinate system)	–9999.9999mm to +9999.9999mm –99999.999mm to +99999.999mm	–999.99999inch to +999.99999inch –9999.9999inch to +9999.9999inch	1 to 359.999°
Dwell time	0 to 99999.999s 0 to 99999rev	0 to 99999.999s 0 to 99999rev	
Backlash compensation amount	0 to ± 9999pulses	0 to ± 9999pulses	0 to ± 9999pulses
Pitch error compensation	0 to ± 9999pulses	0 to ± 9999pulses	0 to ± 9999pulses
Dry run speed	0 to 3600mm/min	0 to 360inch	0 to 3600°/min
Manual jog rapid traverse	0 to 280000mm/min	0 to 11023inch/min	0 to 280000°/min
Thread lead	0.0001 to 99.999999mm 0.001 to 999.99999mm	0.00001 to 9.9999999inch/rev 0.0001 to 99.999999inch/rev	
Synchronous feed	0.0001 to 99.9999mm/rev 0.001 to 999.999mm/rev	0.00001 to 9.99999inch/rev 0.0001 to 99.9999inch/rev	

APPENDIX 3 CIRCULAR CUTTING RADIUS ERROR

Function and purpose

When circular cutting is performed, an error is caused between the command coordinate and the tracking coordinate due to the tracking delay in the smoothing circuit and servo system, and the workpiece ends up smaller due to the commanded radius.



A	: Command coordinate	
B	: Tracking coordinate	
R	: Command radius	(mm)
ΔR	: Radius error	(mm)
Δθ	: Angle error	(rad)
F	: Cutting feedrate	(m/min)

The ΔR radius error and Δθ angle error are calculated from the following formula.

$$\Delta R = \frac{1}{2R} \cdot (Ts^2 + Tp^2) \cdot \frac{F \times 10^3}{60})^2 \qquad \text{(mm)}$$

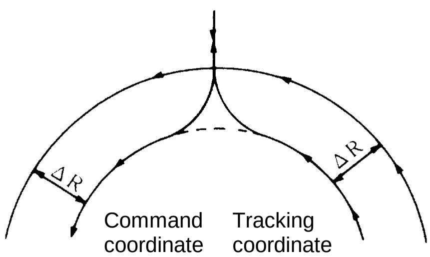
$$\Delta\theta = \tan^{-1} \left(Ts \cdot \frac{F}{R} \right) + \tan^{-1} \left(Tp \cdot \frac{F}{R} \right) \quad \text{(rad)}$$

Ts : 0.03s for semi-closed loop system
Tp : 0.04s for closed loop system

Precautions

(1) When the ΔR radius error applying with circular cutting does not come within the allowable value proceed to reduce the cutting feedrate F, set Ts to a lower value or review the program.

(2) In the steady state ΔR is constant. However, it is not constant with command start and stop transitions. Under command start and stop conditions, therefore, the tracking coordinate should be as shown in the figure below.



APPENDIX 4 STANDARD FIXED CYCLE SUBPROGRAMS

⃠ **Do not change the fixed cycle program without prior consent from the machine manufacturer.**

G37 (O370)	Automatic tool length measurement

```
G31 Z#5 F#3 ;
IF[ROUND [ABS[#2–[##10 ∗ #11]]]GT #8] GOTO 1 ;
IF[ROUND [##10 ∗ #11] EQ #4] GOTO1 ;
##9 = ##10 – #2 / #11 + ##9 ;
#3003 = #1 ;
N2 ;
M99 ;
N1 #3901 = 126 ;
```

G74 (O740)	Face cutting-off cycle

```
G.1 ;
IF[ABS[#2]GT 0]GOTO 10 ;
#14 = 1 ;
N10 #13 = #3 ;
IF[#15 NE 0]GOTO 11 ;
#13 = #3 – #5 ;
N11 #16 = 0 ;
DO 1 ;
#10 = 0 ;
#11 = #4 ;
DO 2 ;
#10 = #10 + #4 ;
IF[ABS[#10]GE[ABS[#1]]]GOTO 1 ;
G01 X #11 ;
G00 X #6    ;
#11 = #4 – #6 ;
END 2 ;
N1 G01 X #1–#10+#11 ;
IF[#15 EQ 0]GOTO 20 ;
IF[#16 EQ 0]GOTO 21 ;
N20 G00 Y #5 ;
N21 #16 = 1 ;
G00 X -#1 ;
IF[#14] GOTO3 ;
#12 = #12 + #3 ;
IF[ABS[#12]LT[ABS[#2]]]GOTO 2 ;
#14 = 1 ;
#13 = #2 – #12 + #13 ;
N2 G00 Y #13 ;
#13 = #3 – #5 ;
END 1 ;
N3 G00 Y –#2-#5 ;
M99 ;
%
```

APPENDIX 4 STANDARD FIXED CYCLE SUBPROGRAMS

G75 (O750)	Longitudinal cutting-off cycle

```
G.1 ;
IF[ABS[#1]GT 0]GOTO 10 ;
#14 = 1 ;
N10 #13 = #4 ;
IF[#15 NE 0]GOTO 11 ;
#13 = #4 – #5 ;
N11 #16 = 0 ;
DO 1 ;
#10 = 0 ;
#11 = #3 ;
DO 2 ;
#10 = #10 + #3 ;
IF[ABS[#10]GE]ABS[#2]]]GOTO 1 ;
G01 Y #11 ;
G00 Y #6    ;
#11 = #3 – #6 ;
END 2 ;
N1 G01 Y #2–#10+#11 ;
IF[#15 EQ 0]GOTO 20 ;
IF[#16 EQ 0]GOTO 21 ;
N20 G00 X #5 ;
N21 #16 = 1 ;
G00 Y –#2 ;
IF[#14] GOTO3 ;
#12 = #12 + #4 ;
IF[ABS[#12]LT[ABS[#1]]]GOTO 2 ;
#14 = 1 ;
#13 = #1 – #12 + #13 ;
N2 G00 X #13 ;
#13 = #4 – #5 ;
END 1 ;
N3 G00 X –#1–#5 ;
M99 ;
%
```

G76 (O760)	Compound thread cutting cycle

```
G.1 ;
#12 = 1 ;
#13 = #9 ;
IF[ABS[#13]GE[ABS[#8]]]GOTO 1 ;
#16 = 1 ;
#13 = #8 ;
N1 #11 = #13 ;
IF[ABS[#11]LT[ABS[#4 – #5]]]GOTO 2 ;
#11 = #4 – #5 ;
#14 = 1 ;
N2 #17 = #11 ;
#18 = ROUND[[#4 – #11 – #5]∗ #17]    ;
IF[[#18 XOR #1]GE 0]GOTO 10 ;
#18 = –#18 ;
N10 #19 = #18 ;
#10 = ROUND[[#11 + #5]∗ #7];
```

```
IF[[#10 XOR #1]GE 0]GOTO 20 ;
#10 = –#10 ;
N20 G00 X #10 ;
#20 = #10 ;
DO 1 ;
#15 = ROUND[#10 ∗ #3/#1];
G00 Y #2+#3–#4–#15+#11 ;
G33 X #1–#10–#18    Y –#3+#15 ;
G00 Y –#2+#4-#11 ;
IF[#14 GT 0]GOTO 3 ;
IF[#16 GT 0]GOTO 7 ;
#12 = #12 + 1 ;
#13 = ROUND[#9 ∗ SQRT[#12]] ;
IF[ABS[#13 – #11]GE[ABS[#8]]]GOTO 8 ;
#16 = 1 ;
N7 #13 = #11 + #8 ;
N8 #11 = #13 ;
IF[ABS[#11[LT[ABS[#4 – #5]]]GOTO 9 ;
#11 = #4 – #5 ;
#14 = 1 ;
N9 #10 = ROUND[[#17 – #11]∗#7] ;
IF[[#10 XOR #1]GE 0]GOTO 6 ;
#10 = –#10 ;
N6 #10 = #10 + #20 ;
G00 X –#1+#10+#18 ;
IF[#14 LT 0]GOTO 11 ;
#18 = 0 ;
GOTO 12 ;
N11 #18 = #19 – #10 + #20 ;
N12 END 1 ;
N3 IF[ABS[#6]LT 1]GOTO 5 ;
#14 = 0 ;
#13 = 0 ;
DO 2 ;
IF[#14 GT 0]GOTO 5 ;
#13 = #13 + #6 ;
IF[ABS[#13]LT[ABS[#5]]]GOTO 4 ;
#13 = #5 ;
#14 = 1 ;
N4 G00 X #10–#1 ;
G00 Y #2+#3–#4+#13–#15+#11 ;
G33 X #1–#10    Y –#3+#15 ;
G00 Y –#2+#4–#13–#11 ;
END 2 ;
N5 G00 X–#1 ;
M99 ;
```

APPENDIX 4 STANDARD FIXED CYCLE SUBPROGRAMS

G76 (O760)	Compound thread cutting cycle I (2-system)

```
G.1 ;
N761 !L10 ;
#12 = 1 ;
#13 = #9 ;
IF[ABS[#13]GE[ABS[#8]]]GOTO 1 ;
#16 = 1 ;
#13 = #8 ;
N1 #11 = #13 ;
IF[ABS[#11]LT[ABS[#4 – #5]]]GOTO 2 ;
#11 = #4 – #5 ;
#14 = 1 ;
N2 #17 = #11 ;
#18 = ROUND[[#4 – #11 – #5]∗ #7] ;
IF[[#18 XOR #1]GE 0]GOTO 10 ;
#18 = –#18 ;
N10 #19 = #18 ;
#10 = ROUND[[#11 + #5]∗ #7];
IF[[#10 XOR #1]GE 0]GOTO 20 ;
#10 = –#10 ;
N20 G00 X #10 ;
#20 = #10 ;
DO 1 ;
#15 = ROUND[#10 ∗ #3/#1];
G00 Y #2+#3–#4–#15+#11 ;
!L11 ;
G33 X #1–#10–#18 Y –#3+#15 ;
G00 Y –#2+#4–#11 ;
!L12 ;
IF[#14 GT 0]GOTO 3 ;
IF[#16 GT 0]GOTO 7 ;
#12 = #12 + 1 ;
#13 = ROUND[#9 ∗ SQRT[#12]] ;
IF[ABS[#13 – #11]GE[ABS[#8]]]GOTO 8 ;
#16 = 1 ;
N7 #13 = #11 + #8 ;
N8 #11 = #13 ;
IF[ABS[#11]LT[ABS[#4 – #5]]]GOTO 9 ;
#11 = #4 – #5 ;
#14 = 1 ;
N9 #10 = ROUND[[#17 – #11]∗ #7] ;
IF[[#10 XOR #1]GE 0]GOTO 6 ;
#10 = –#10 ;
N6 #10 = #10 + #20 ;
G00 X –#1+#10+#18 ;
IF[#14 LT 0]GOTO 11 ;
#18 = 0 ;
GOTO 12 ;
N11 #18 = #19 – #10 + #20 ;
N12 END 1 ;
N3 IF[ABS[#6]LT 1]GOTO 5 ;
#14 = 0 ;
#13 = 0 ;
```

```
DO 2 ;
IF[#14 GT 0]GOTO 5 ;
#13 = #13 + #6 ;
IF[ABS[#13]LT[ABS[#5]]]GOTO 4 ;
#13 = #5 ;
#14 = 1 ;
N4 G00 X #10–#1 ;
G00 Y #2+#3–#4+#13–#15+#11 ;
!L11 ;
G33 X #1–#10   Y –#3+#15 ;
G00 Y –#2+#4–#13–#11 ;
!L12 ;
END 2 ;
N5 G00 X –#1 ;
M99 ;
%
```

G77 (O770)	Longitudinal cutting cycle

```
G.1 ;
IF[[#1 EQ 0]OR[#2 EQ 0]]GOTO 1 ;
Y #2+#7 ;
G01 X #1   Y –#7 ;
Y –#2 ;
G00 X –#1 ;
N1 M99 ;
```

G78 (O780)	Thread cutting cycle

```
G.1 ;
IF[[#1 EQ 0]OR[#2 EQ 0]]GOTO 1 ;
Y #2+#7 ;
G33 X #1   Y –#7   F #9   E #10;
G00 Y –#2 ;
X –#1 ;
N1 M99 ;
```

G79 (O790)	Face cutting cycle

```
G.1 ;
IF[[#1 EQ 0]OR[#2 EQ 0]]GOTO 1 ;
Y #1+#7 ;
G01 X –#7   Y #2 ;
X –#1 ;
G00 Y –#2 ;
N1 M99 ;
```

APPENDIX 4 STANDARD FIXED CYCLE SUBPROGRAMS

G83, G87 (O830)	Deep hole drilling cycle B

```
G.1 ;
IF[#30]GOTO 2 ;
M #24 ;
#29 = #11    #28 = 0 ;
Z #2 ;
#2 = ##5    #3003 = #8 OR 1 ;
DO 1 ;
#28 = #28 – #11    #26 = –#28 –#29 ;
Z #26 ;
IF[ABS[#28]GE[ABS[#3]]]GOTO 1 ;
G01 Z #29 ;
G00 Z #28 ;
#29 = #11+#14 ;
END 1 ;
N1 G01 Z #3–#26 ;
G04 P#4 ;
#3003 = #8 ;
G00 Z –#3-#2 ;
IF[#24 EQ #0] GOTO 2 ;
M #24+1 ;
G04 P #21 ;
N2 M99 ;
```

G83, G87 (O831)	Deep hole drilling cycle A

```
G.1 ;
IF[#30]GOTO 2 ;
M #24 ;
#29 = 0    #28 = #11 ;
Z #2 ;
#2 = ##5    #3003 = #8 OR 1 ;
DO 1 ;
#29 = #29 + #11 ;
IF[ABS[#29]GE[ABS[#3]]]GOTO 1 ;
G01 Z #28 ;
G00 Z –#14 ;
#28 = #11+#14 ;
END 1 ;
N1 G01 Z #3–#29+#28 ;
G04 P #4 ;
#3003 = #8 ;
G00 Z –#3-#2 ;
IF[#24 EQ #0] GOTO 2 ;
M #24+1 ;
G04 P #21 ;
N2 M99 ;
```

APPENDIX 4 STANDARD FIXED CYCLE SUBPROGRAMS

G83.2 (O832)	Deep hole drilling cycle 2

```
G.1 ;
IF[#30]GOTO 3 ;
#3003 = #8 OR 1 ;
#29 = #12    #28 = 0 ;
G00 Z #2 ;
IF[#12 NE #0]GOTO 1 ;
IF[#11 EQ #0]GOTO 2 ;
N1 #28 = #28 – #12    #26 = –#28 –#29 ;
IF[ABS[#28]GE[ABS[#3]]]GOTO 2 ;
G01 Z #12 ;
G04 P #4 ;
G00 Z #28–#2 ;
G04 P #13 ;
#29 = #11+#15 ;
DO 1 ;
#28 = #28 – #11    #26 = –#28 –#29 ;
G00 Z #26 + #2 ;
IF[ABS[#28]GE[ABS[#3]]]GOTO 2 ;
G01 Z #29 ;
G04 P #4 ;
G00 Z #28–#2 ;
G04 P #13 ;
END 1 ;
N2 G01 Z #3–#26 ;
G04 P #4 ;
#3003 = #8 ;
G00 Z –#3–#2 ;
N3 M99 ;
```

G84, G88 (O840)	Tapping cycle

```
G.1 ;
IF[#30]GOTO 2 ;
M #24 ;
Z #2 ;
#2 = ##5    #3003 = #8 OR 1    #3004 = #9 OR 3;
G01 Z #3    F #22 ;
G04 P #4 ;
M #6 ;
#3900 = 1 ;
G01 Z –#3    F #23 ;
#3004 = #9 ;
M #7 ;
#3003 = #8 ;
IF[#24 EQ #0]GOTO 1 ;
M #24 + 1 ;
G04 P #21 ;
N1 G00 Z –#2 ;
N2 M99 ;
```

G85, G89 (O850)	Boring cycle

```
G.1 ;
IF[#30]GOTO 2 ;
M #24 ;
Z #2 ;
#2 = ##5    #3003 = #8 OR 1 ;
G01 Z #3 ;
G04 P #4 ;
#3003 = #8 ;
Z –#3    F #23 ;
F #22 ;
IF[#24 EQ #0]GOTO 1 ;
M #24 + 1 ;
G04 P #21 ;
N1 G00 Z –#2 ;
N2 M99 ;
```

APPENDIX 5 LIST OF VARIABLE NUMBERS

○ : Can be read or written
× : Cannot be written
– : Unrelated to reading or writing

<table>
<tr><th>Variable number(#)</th><th>Description</th><th>Read</th><th>Write</th></tr>
<tr><td>0</td><td>(blank)</td><td>○</td><td>×</td></tr>
<tr><td>1 ~ 32</td><td>Local variables
Argument designation I
<table>
<tr><td>#</td><td>1</td><td>2</td><td>3</td><td>4</td><td>5</td><td>6</td><td>7</td><td>8</td><td>9</td><td>10</td><td>11</td><td>12</td><td>13</td><td>14–26</td><td>27–32</td></tr>
<tr><td>Argument address</td><td>A</td><td>B</td><td>C</td><td>I</td><td>J</td><td>K</td><td>D</td><td>E</td><td>F</td><td>G</td><td>H</td><td>L</td><td>M</td><td>N–Z</td><td></td></tr>
</table>
Argument designation II
<table>
<tr><td>#</td><td>1</td><td>2</td><td>3</td><td>4</td><td>5</td><td>6</td><td>7</td><td>8</td><td>9</td><td>10–27</td><td>28</td><td>29</td><td>30</td><td>31</td><td>32</td></tr>
<tr><td>Argument address</td><td>A</td><td>B</td><td>C</td><td>I1</td><td>J1</td><td>K1</td><td>I2</td><td>J2</td><td>K2</td><td>I3–K8</td><td>I9</td><td>J9</td><td>K9</td><td>I10</td><td>J10</td></tr>
</table></td><td>○</td><td>○</td></tr>
<tr><td>100 ~ 149
500 ~ 549</td><td>Common variables
Type A: Common for systems 500~549
For each system 100~149
Type B: Common for systems 500~599
For each system 100~199</td><td>○</td><td>○</td></tr>
<tr><td>1000~1031
1032
1033
1034
1035</td><td>Macro interface inputs
<table>
<tr><th>#</th><th>Contents</th></tr>
<tr><td>1000~1015</td><td>Register R72 bit 0 ~ Register R72 bit 5</td></tr>
<tr><td>1016~1031</td><td>Register R73 bit 0 ~ Register R73 bit 5</td></tr>
</table>
<table>
<tr><th>#</th><th>Contents</th></tr>
<tr><td>1032</td><td>Register R72, R73</td></tr>
<tr><td>1033</td><td>Register R74, R75</td></tr>
<tr><td>1034</td><td>Register R76, R77</td></tr>
<tr><td>1035</td><td>Register R78, R79</td></tr>
</table></td><td rowspan="2">○</td><td rowspan="2">×</td></tr>
<tr><td>1400~1527</td><td>The macro interface expansion function is required. Refer to "Appendix 8. Macro Interface Expansion" for details.</td></tr>
<tr><td>1100~1131
1132
1133
1134
1135</td><td>Macro interface outputs
<table>
<tr><th>#</th><th>Contents</th></tr>
<tr><td>1100~1115</td><td>Register R172 bit 0 ~ Register R172 bit 5</td></tr>
<tr><td>1116~1131</td><td>Register R173 bit 0 ~ Register R173 bit 5</td></tr>
</table>
<table>
<tr><th>#</th><th>Contents</th></tr>
<tr><td>1132</td><td>Register R172, R173</td></tr>
<tr><td>1133</td><td>Register R174, R175</td></tr>
<tr><td>1134</td><td>Register R176, R177</td></tr>
<tr><td>1135</td><td>Register R178, R179</td></tr>
</table></td><td rowspan="2">○</td><td rowspan="2">○</td></tr>
<tr><td>1600~1727</td><td>The macro interface expansion function is required. Refer to "Appendix 8. Macro Interface Expansion" for details.</td></tr>
<tr><td>3000</td><td>NC alarm (alarm display with macro program)</td><td>–</td><td>–</td></tr>
<tr><td>3001</td><td>Integrated time 1</td><td>○</td><td>○</td></tr>
<tr><td>3002</td><td>Integrated time 2</td><td>○</td><td>○</td></tr>
<tr><td>3003</td><td>Single block stop, miscellaneous function finish signal waiting control</td><td>○</td><td>○</td></tr>
<tr><td>3004</td><td>Feed hold, feedrate override, G09 valid/invalid</td><td>○</td><td>○</td></tr>
<tr><td>3006</td><td>Message display/stop (message display with macro programs, block stop)</td><td>–</td><td>–</td></tr>
<tr><td>3007</td><td>Mirror image</td><td>○</td><td>×</td></tr>
</table>

(Note) The variables corresponding to the additional specifications can be used only when those specifications have been provided.

APPENDIX 5 LIST OF VARIABLE NUMBERS

○ : Can be read or written
× : Cannot be written
– : Unrelated to reading or writing

<table>
<tr><th>Variable number(#)</th><th>Description</th><th>Read</th><th>Write</th></tr>
<tr><td>4001~4320</td><td>Modal area
<table>
<tr><th>Modal information</th><th>Pre-read</th><th>In-execution</th></tr>
<tr><td>G codes (group 1) ~ G code (group 21)</td><td>4001 ~ 4021</td><td>4201 ~ 4221</td></tr>
<tr><td>F code</td><td>4109</td><td>4309</td></tr>
</table>
<table>
<tr><th>Modal information</th><th>Pre-read</th><th>In-execution</th></tr>
<tr><td>M codes</td><td>4113</td><td>4313</td></tr>
<tr><td>N codes</td><td>4114</td><td>4314</td></tr>
<tr><td>O codes</td><td>4115</td><td>4315</td></tr>
<tr><td>S codes</td><td>4119</td><td>4319</td></tr>
<tr><td>T codes</td><td>4120</td><td>4320</td></tr>
</table>
</td><td>○</td><td>×</td></tr>
<tr><td>5001~5103</td><td>Position information
<table>
<tr><th>Axis / Position information</th><th>1st axis</th><th>2nd axis</th><th>3rd axis</th></tr>
<tr><td>Block end point</td><td>5001</td><td>5002</td><td>5003</td></tr>
<tr><td>Machine coordinate</td><td>5021</td><td>5022</td><td>5023</td></tr>
<tr><td>Workpiece coordinate</td><td>5041</td><td>5042</td><td>5043</td></tr>
<tr><td>Skip coordinate value</td><td>5061</td><td>5062</td><td>5063</td></tr>
<tr><td>Tool position offset amount</td><td>5081</td><td>5082</td><td>5083</td></tr>
<tr><td>Servo deviation amount</td><td>5101</td><td>5102</td><td>5103</td></tr>
</table>
</td><td>○</td><td>×</td></tr>
<tr><td>5201~5323</td><td>Workpiece coordinate offset
<table>
<tr><th>Axis / Workpiece</th><th>1st axis</th><th>2nd axis</th><th>3rd axis</th></tr>
<tr><td>EXT</td><td>5201</td><td>5202</td><td>5203</td></tr>
<tr><td>G54</td><td>5221</td><td>5222</td><td>5223</td></tr>
<tr><td>G55</td><td>5241</td><td>5242</td><td>5243</td></tr>
<tr><td>G56</td><td>5261</td><td>5262</td><td>5263</td></tr>
<tr><td>G57</td><td>5281</td><td>5282</td><td>5283</td></tr>
<tr><td>G58</td><td>5301</td><td>5302</td><td>5303</td></tr>
<tr><td>G59</td><td>5321</td><td>5322</td><td>5323</td></tr>
</table>
</td><td>○</td><td>○</td></tr>
<tr><td>10001~18040
2001~2940</td><td>Tool offset
<table>
<tr><td>1st axis tool length offset amount</td><td>10001 ~ 10040</td><td>2001 ~ 2940</td></tr>
<tr><td>1st axis wear offset amount</td><td>11001 ~ 11040</td><td>2701 ~ 2740</td></tr>
<tr><td>3rd axis tool length offset amount</td><td>12001 ~ 12040</td><td>–</td></tr>
<tr><td>3rd axis wear offset amount</td><td>13001 ~ 13040</td><td>–</td></tr>
<tr><td>2nd axis tool length offset amount</td><td>14001 ~ 14040</td><td>2101 ~ 2140</td></tr>
<tr><td>2nd axis wear offset amount</td><td>15001 ~ 15040</td><td>2801 ~ 2840</td></tr>
<tr><td>Nose R compensation amount</td><td>16001 ~ 16040</td><td>2201 ~ 2240</td></tr>
<tr><td>Nose R wear compensation amount</td><td>17001 ~ 17040</td><td>2901 ~ 2940</td></tr>
<tr><td>Tool nose points</td><td>18001 ~ 18040</td><td>2301 ~ 2340</td></tr>
</table>
</td><td>○</td><td>○</td></tr>
</table>

(Note) The variables corresponding to the additional specifications can be used only when those specifications have been provided.

APPENDIX 6 CORRESPONDENCE TABLE OF PROGRAM PARAMETER INPUT N NUMBERS

<How to read the table>

Screen title

6.1.1 Control parameter

#	Parameter	P	A	N	Data type	Setting range	(Unit)	Remarks
8101	G00 dry run	1	×	916	H3	0 to 1		

Setting range
(Refer to the Parameters Manual for details of parameter data.)

Data type

N number

Axis specification
- ○ · · · An axis number is specified.
- × · · · No axis number is specified.

P number

Parameter name

Setting number on screen

(Note 1) The units indicated in the tables are the minimum input setting unit of the parameter data.
Set the correct data for the "interpolation unit", "output unit" and "speed unit".

APPENDIX 6 CORRESPONDENCE TABLE OF PROGRAM PARAMETER INPUT N NUMBERS

6.1.1 Control parameter

#	Parameter	P	A	N	Data type	Setting range	(Unit)	Remarks
8101	G00 dry run	1	×	916	H3	0 to 1		
8102	Macro single	1	×	914	H6	0 to 1		
8103	Middle point ignore	1	×	914	H5	0 to 1		
8104								
8105	Machine lock rapid	1	×	914	H4	0 to 1		
8106	ABS/INC Addr.	1	×	914	H7	0 to 1		
8107	G04 time fixed	1	×	913	H2	0 to 1		
8108	Rad compen intrf byp	1	×	913	H5	0 to 1		
8109		1	×	912	H0	0 to 1		
8110	Decimal point type 2	1	×	912	H5	0 to 1		
8111								
8112								
8113	G0 interpolation OFF	1	×	912	H6	0 to 1		
8114	Precision thrd cut E	1	×	912	H7	0 to 1		
8115	Radius compen type B	1	×	913	H4	0 to 1		
8116	Ext deceleration OFF	1	×	915	H1	0 to 1		
8117	Initial inch*	1	×	912	H4	0 to 1		
8118	Initial absolute val	1	×	914	H2	0 to 1		
8119	Initial synchr feed	1	×	914	H1	0 to 1		
8120	Init cnst prphl spd	1	×	914	H0	0 to 1		
8121	Initial Z-X plane	1	×	913	H0	0 to 1		
8122	Initial Y-Z plane	1	×	913	H1	0 to 1		
8123	Initial G00	1	×	915	H3	0 to 1		
8124								
8125	G83/G87 rapid	1	×	913	H6	0 to 1		
8126	Fixed cycle modal	1	×	915	H0	0 to 1		
8127	Lathe cycle mode	1	×	913	H7	0 to 1		
8128								
8129	Synchronous tapping	1	×	915	H6	0 to 1		
8130	T-life manage valid	1	×	915	H7	0 to 1		
8131								
8132								

APPENDIX 6 CORRESPONDENCE TABLE OF PROGRAM PARAMETER INPUT N NUMBERS

#	Parameter	P	A	N	Data type	Setting range	(Unit)	Remarks
8133	G code type 1	1	×	912	H2	0 to 1		
8134	G code type 2	1	×	912	H3	0 to 1		
8135	G code type 3	1	×	912	H1	0 to 1		
8136	Interrupt amt reset	1	×	916	H6	0 to 1		
8137	G46 no reverse error	1	×	916	H7	0 to 1		
8138								
8139								
8140	Edit lock B	1	×	917	H2	0 to 1		
8141								
8142	Start point alarm	1	×	918	H4	0 to 1		
8143								
8144	Milling G16	1	×	918	H2	0 to 1		
8145	Milling G19	1	×	918	H3	0 to 1		
8146								
8147								
8148								
8149	Superposition compensation Z2/Z1	1	×	918	H5	0 to 1		
8150	Superposition compensation Z3/Z1	1	×	918	H6	0 to 1		
8151	Superposition compensation Z3/Z2	1	×	918	H7	0 to 1		
8152								
8153								
8154								
8155								
8156								
8157								
8158	Tool set type 2	1	×	919	H5	0 to 1		
8159	Screen copy	1	×	921	H5	0 to 1		Not used.
8160								
8161								
8162								
8163								
8164								

6.1.2 Axis parameter

#	Parameter	P	A	N	Data type	Setting range	(Unit)	Remarks
8201	Mirror image	2	○	640	H0	0 to 1		
8202	Automatic dog type	2	○	641	H0	0 to 1		
8203	Manual dog type	2	○	641	H1	0 to 1		
8204	Axis removal	2	○	641	H6	0 to 1		
8205								
8206								
8207	Soft limit invalid	2	○	640	H2	0 to 1		
8208	Soft limit (–)	2	○	632	L	0 to ±99999999 ×2	Interpola-tion units	0 to ±99999.999(mm)
8209	Soft limit (+)	2	○	628	L	0 to ±99999999 ×2	Interpola-tion units	0 to ±99999.999(mm)
8210								

APPENDIX 6 CORRESPONDENCE TABLE OF PROGRAM PARAMETER INPUT N NUMBERS

6.1.3 Setup parameter

#	Parameter	P	A	N	Data type	Setting range	(Unit)	Remarks
8001	Plane <I>	13	×	298	D	ASCII		Axis address
8002	<J>	13	×	301	D	ASCII		Axis address
8003	<K>	13	×	304	D	ASCII		Axis address
8004	Aux-plane <I>	13	×	299	D	ASCII		Axis address
8005	<J>	13	×	302	D	ASCII		Axis address
8006	<K>	13	×	305	D	ASCII		Axis address
8007								
8008								
8009								
8010	G02/03 Error	13	×	280	L	0 to 100×2	Interpolation units	0 to 0.100(mm)
8011	Chamfer value	13	×	307	D	0 to 127	0.1 lead	
8012	Chamfer angle	13	×	297	D	0 to 89	°	
8013	G71 Minimum thick	13	×	320	L	0 to 99999×2	Interpolation units	0 to 99.999(mm)
8014	Delta-D	13	×	324	L	0 to 99999×2	Interpolation units	0 to 99.999(mm)
8015	Pull up	13	×	328	L	0 to 99999×2	Interpolation units	0 to 99.999(mm)
8016	Thick	13	×	344	L	0 to 99999×2	Interpolation units	0 to 99.999(mm)
8017	G74 Retract	13	×	332	L	0 to 99999×2	Interpolation units	0 to 99.999(mm)
8018	G76 Finishing	13	×	336	L	0 to 99999×2	Interpolation units	0 to 99.999(mm)
8019	Minimum thick	13	×	340	L	0 to 99999×2	Interpolation units	0 to 99.999(mm)
8020	Times	13	×	358	D	0 to 99	times	
8021	Angle	13	×	359	D	0 to 99	times	
8022	G71 Pocket	13	×	415	H0	0 to 1		
8023	G73 Cut X	13	×	348	L	0 to 99999×2	Interpolation units	0 to 99.999(mm)
8024	Cut Z	13	×	352	L	0 to 99999×2	Interpolation units	0 to 99.999(mm)
8025	Times	13	×	356	S	0 to 9999	times	
8026	G83 Retract	13	×	276	L	0 to 99999	Command unit	0 to 99.999(mm)
8027								
8028								
8029								
8030								
8031	Tool wear max	13	×	400	L	0 to 99999×2	Interpolation units	0 to 99.999(mm)
8032	inc max	13	×	404	L	0 to 99999×2	Interpolation units	0 to 99.999(mm)
8033	Auto TLM speed	13	×	316	L	1 to 60000	Interpolation units	1 to 60000(mm/min)
8034	zone r	13	×	308	L	0 to 99999999×2	Interpolation units	0 to 99999.999(mm)
8035	zone d	13	×	312	L	0 to 99999999×2	Interpolation units	0 to 99999.999(mm)

APPENDIX 6 CORRESPONDENCE TABLE OF PROGRAM PARAMETER INPUT N NUMBERS

#	Parameter	P	A	N	Data type	Setting range	(Unit)	Remarks
8051	Constant speed	1	×	976	S	1 to 60000		
8052	Interval	1	×	978	D	0 to 99	0.1s	0 to 9.9(s)
8053	Control*	1	×	928	D	0 to FF (Hexadecimal number)		
8054								
8055	Scrn saver time-out	1	×	1013	D	0 to 60	1min	0 to 60(min)
8056	Intrf byps time-out	1	×	1014	D	0 to 255	1s	0 to 255(s)
8057	Corner check angle	1	×	1015	D	0 to 180	°	
8058	Corner check width	1	×	1020	L	0 to 99999×2	Interpola-tion units	0 to 99.999(mm)
8059	Angle (G1 -> G0)	1	×	1028	D	0 to 180	°	

6.1.4 Setup parameter 2

#	Parameter	P	A	N	Data type	Setting range	(Unit)	Remarks
8301	X	13	×	224	L	±99999999×2	Interpola-tion units	±99999.999(mm)
8302	Z	13	×	248	L	±99999999×2	Interpola-tion units	±99999.999(mm)
8303	X	13	×	228	L	±99999999×2	Interpola-tion units	±99999.999(mm)
8304	Z	13	×	252	L	±99999999×2	Interpola-tion units	±99999.999(mm)
8305	X	13	×	232	L	±99999999×2	Interpola-tion units	±99999.999(mm)
8306	Z	13	×	256	L	±99999999×2	Interpola-tion units	±99999.999(mm)
8307	X	13	×	236	L	±99999999×2	Interpola-tion units	±99999.999(mm)
8308	Z	13	×	260	L	±99999999×2	Interpola-tion units	±99999.999(mm)
8309	X	13	×	240	L	±99999999×2	Interpola-tion units	±99999.999(mm)
8310	Z	13	×	284	L	±99999999×2	Interpola-tion units	±99999.999(mm)
8311	X	13	×	244	L	±99999999×2	Interpola-tion units	±99999.999(mm)
8312	Z	13	×	288	L	±99999999×2	Interpola-tion units	±99999.999(mm)

APPENDIX 6 CORRESPONDENCE TABLE OF PROGRAM PARAMETER INPUT N NUMBERS

6.2.1 Base axis parameter

#	Parameter	P	A	N	Data type	Setting range	(Unit)	Remarks
1001	axname	2	○	50	D	ASCII		Axis address
1002	incax	2	○	51	D	ASCII		Axis address
1003	cunit	2	○	48	S	1,10	1: 0.1µm 10: 1µm	
1004	sp_ax	2	○	68	H0	0 to 1		
1005	iout	2	○	69	H1	0 to 1		
1006	rot	2	○	69	H4	0 to 1		
1007	ccw	2	○	69	H5	0 to 1		
1008	svof	2	○	69	H3	0 to 1		
1009	dia	2	○	68	H5	0 to 1		
1010	polar	2	○	68	H3	0 to 1		
1011	abson							Data setting not possible
1012	intabs	2	○	68	H6	0 to 1		
1013	axname2	2	○	52	S	ASCII (2 characters)		
1014	cros_$	2	○	56	S	0 to FF (Hexadecimal number)		
1015	axoff	2	○	69	H7	0 to 1		
1016	mcp_no	2	○	55	D	11 to 17, 21 to 27, 31 to 37, 41 to 47, 51 to 57, 61 to 67 (Hexadecimal number)		

6.2.2 Base system parameter

#	Parameter	P	A	N	Data type	Setting range	(Unit)	Remarks
1101	Mflg	13	×	32	D	1 to 4		
1102	Mbin	13	×	33	D	–1 to 1		
1103	Sfig	13	×	34	D	1 to 6		
1104	Sbin	13	×	35	D	–1 to 1		
1105	Tfig	13	×	36	D	1		
1106	Tbin	13	×	37	D	–1 to 1		
1107	M2fig	13	×	38	D	1		
1108	M2bin	13	×	39	D	–1 to 1		
1109	M2name	13	×	40	D	A,B,C (ASCII)		
1110	skip_F	13	×	8	L	1 to 480000	Speed unit	1 to 480000(mm/min)
1111	skip_C	13	×	41	D	0 to 7		
1112	extdcc	13	×	0	L	1 to 480000	Speed unit	1 to 480000(mm/min)
1113	tapovr							Not used.
1114	thr_SF	13	×	28	L	1 to 99999999, 0	Speed unit	1 to 99999999 (0.001mm/rev) 0: Cutting feed cramp rate
1115	tap_tl							Not used.
1116	dwlskp	13	×	45	D	0 to 7		
1117	G96_ax	13	×	46	D	0 to 5		
1118	clmp_M	13	×	64	L	0 to 99999999		
1119	clmp_D	13	×	68	L	0 to 99999999	0.001s	0.000 to 99999.999(s)
1120	origin	13	×	77	H0	0 to 1		
1121								
1122	mirofs	13	×	4	L	0 to 99999999×2	Interpola-tion unit	0 to 99999.999(mm)
1123	TmirS1	13	×	144	L	0 to FFFFFFFF		(Hexadecimal number)
1124	TmirS2	13	×	148	L	0 to FFFFFFFF		(Hexadecimal number)
1125	mill_ax	13	×	72	D	0 to 4		
1126	mill_C	13	×	73	D	0 to 1		
1127								
1128								
1129								
1130								
1131	Sselect	13	×	47	D	0 to 1		
1132								
1133	adr_abs[1]	13	×	80	D	ASCII		
1134	adr_abs[2]	13	×	81	D	ASCII		
1135	adr_abs[3]	13	×	82	D	ASCII		
1136	adr_abs[4]	13	×	83	D	ASCII		
1137	adr_abs[5]	13	×	84	D	ASCII		
1138	adr_inc[1]	13	×	85	D	ASCII		
1139	adr_inc[2]	13	×	86	D	ASCII		
1140	adr_inc[3]	13	×	87	D	ASCII		
1141	adr_inc[4]	13	×	88	D	ASCII		
1142	adr_inc[5]	13	×	89	D	ASCII		

APPENDIX 6 CORRESPONDENCE TABLE OF PROGRAM PARAMETER INPUT N NUMBERS

#	Parameter	P	A	N	Data type	Setting range	(Unit)	Remarks
1143	base_ax[1]	13	×	90	S	ASCII (2 chracters)		
1144	base_ax[2]	13	×	92	S	ASCII (2 characters)		
1145	base_ax[3]	13	×	94	S	ASCII (2 characters)		
1146	base_ax[4]	13	×	96	S	ASCII (2 characters)		
1147	base_ax[5]	13	×	98	S	ASCII (2 characters)		
1148								
1149	real_I	13	×	100	S	ASCII		
1150	real_J	13	×	102	S	ASCII		
1151	real_K	13	×	104	S	ASCII		

6.2.3 Base common parameter

#	Parameter	P	A	N	Data type	Setting range	(Unit)	Remarks
1301	Mmac	1	×	208	H1	0 to 1		
1302	Smac	1	×	208	H2	0 to 1		
1303	Tmac	1	×	208	H3	0 to 1		
1304	M2mac	1	×	208	H4	0 to 1		
1305	M_inch	1	×	208	H7	0 to 1		
1306	fix_P	1	×	466	H0	0 to 1		
1307	edlk_C	1	×	210	H1	0 to 1		
1308	Pinc	1	×	209	H3	0 to 1		
1309	DPRINT	1	×	213	H2	0 to 1		
1310	ofsfix	1	×	213	H3	0 to 1		
1311	Tmiron	1	×	213	H7	0 to 1		
1312	G96_G0	1	×	212	H0	0 to 1		
1313	radius	1	×	208	H0	0 to 1		
1314	T1digt	1	×	208	H5	0 to 1		
1315	TLno.	1	×	208	H6	0 to 1		
1316	Treset	1	×	212	H1	0 to 1		
1317	Tmove	1	×	212	H2	0 to 1		
1318	rstint	1	×	212	H4	0 to 1		
1319	I_abs	1	×	212	H3	0 to 1		
1320	H_acdc	1	×	212	H5	0 to 1		
1321	G30SL	1	×	212	H6	0 to 1		
1322	inpos	1	×	212	H7	0 to 1		
1323	tlm	1	×	213	H1	0 to 1		
1324	lang	1	×	200	D	0 to 1		
1325	mirr_A	1	×	213	H0	0 to 1		
1326								
1327	sp_1	1	×	528	D	0 to FF		(Hexadecimal number)
1328	2	1	×	529	D	0 to FF		(Hexadecimal number)
1329	3	1	×	532	S	0 to FFFF		(Hexadecimal number)
1330	4	1	×	534	S	0 to 255		
1331	5	1	×	536	S	0 to 100		0 to 100(r/min)
1332	sp_6	1	×	538	D	0 to FF		(Hexadecimal number)
1333	mstsyn	1	×	210	H5	0 to 1		
1334	Tcom	1	×	210	H3	0 to 1		
1335	syncch	1	×	210	H2	0 to 1		
1336	dspax	1	×	256	L	0 to 7FFFFFFF		(Hexadecimal number)
1337	crsman	1	×	210	H4	0 to 1		
1338	otsys	1	×	210	H6	0 to 1		
1339								
1340	TGSmax	1	×	197	D			
1341	H1_pno	1	×	201	D	0 to FF		(Hexadecimal number)
1342	H2_pno	1	×	202	D	0 to FF		(Hexadecimal number)
1343	H3_pno	1	×	203	D	0 to FF		(Hexadecimal number)
1344	statio	1	×	416	D	1 to 7		Not used.
1345	size-i	1	×	417	D	0 to 32		Not used.

APPENDIX 6 CORRESPONDENCE TABLE OF PROGRAM PARAMETER INPUT N NUMBERS

#	Parameter	P	A	N	Data type	Setting range	(Unit)	Remarks
1346	size-o	1	×	418	D	0 to 32		Not used.
1347	length	1	×	400	D	0 to 3		Not used.
1348	b-rate	1	×	401	D	0 to 6		Not used.
1349	s-bit	1	×	402	D	0 to 3		Not used.
1350	parity	1	×	403	D	0 to 1		Not used.
1351	even	1	×	404	D	0 to 1		Not used.
1352	Tout-i	1	×	406	S	0 to 999	0.1s	Not used.
1353	Tout-o	1	×	408	S	0 to 999	0.1s	Not used.
1354	siobus	1	×	393	D	0 to 2		Not used.
1355	cmacdb	1	×	486	D	0 to FF (Hexadecimal number)		Not used.
1356	GBtest	1	×	470	H0	0 to 1		
1357	COOL.t	1	×	487	D	0 to 255	1min	
1358	SBSsys	1	×	539	D	0 to FF (Hexadecimal number)		
1359	SP2name	1	×	489	D	1 to 9		
1401	M_type	1	×	260	D	0 to 1		
1402	S_mode	1	×	261	D	0 to 1		
1403	T_mode	1	×	262	D	0 to 1		
1404	M2_mode	1	×	263	D	0 to 1		
1405								
1406								
1407								
1408								
1409								
1410								
1411	M031-000	1	×	264	L	0 to FFFFFFFF		(Hexadecimal number)
1412	M063-032	1	×	268	L	0 to FFFFFFFF		(Hexadecimal number)
1413	M095-064	1	×	272	L	0 to FFFFFFFF		(Hexadecimal number)
1414	M127-096	1	×	276	L	0 to FFFFFFFF		(Hexadecimal number)
1415	M159-128	1	×	280	L	0 to FFFFFFFF		(Hexadecimal number)
1416	M191-160	1	×	284	L	0 to FFFFFFFF		(Hexadecimal number)
1417	M223-192	1	×	288	L	0 to FFFFFFFF		(Hexadecimal number)
1418	M255-224	1	×	292	L	0 to FFFFFFFF		(Hexadecimal number)

6.2.4 Axis specification parameter

#	Parameter	P	A	N	Data type	Setting range	(Unit)	Remarks
2001	rapid	2	○	8	L	1 to 480000	Speed unit	1 to 480000(mm/min)
2002	clamp	2	○	16	L	1 to 480000	Speed unit	1 to 480000(mm/min)
2003	smgst	2	○	72	S	0 to FFFF		(Hexadecimal number)
2004	G0tL	2	○	74	S	1 to 1500	1ms	
2005	G0t1	2	○	78	S	1 to 5000	1ms	
2006	G0t2	2	○	80	S	1 to 5000	1ms	
2007	G1tL	2	○	76	S	1 to 1500	1ms	
2008	G1t1	2	○	82	S	1 to 5000	1ms	
2009	G1t2	2	○	84	S	1 to 5000	1ms	
2010	OTtm	2	○	86	S	1 to 32767	1ms	
2011	G0back	2	○	96	S	±9999	Command unit/2	
2012	G1back	2	○	98	S	±9999	Command unit/2	
2013	OT -	2	○	44	L	±99999999×2	Interpola-tion unit	±99999.999(mm)
2014	OT +	2	○	40	L	±99999999×2	Interpola-tion unit	±99999.999(mm)
2015	tlml -	2	○	356	L	±99999999×2	Interpola-tion unit	±99999.999(mm)
2016	tlml +	2	○	352	L	±99999999×2	Interpola-tion unit	±99999.999(mm)
2017	pG0t	2	○	102	S	1 to 1500	1ms	
2018	PG1t	2	○	104	S	1 to 5000	1ms	
2019	tap_g							Data setting not possible
2020								
2021	plrap0	2	○	360	L	1 to 480000	Speed unit	1 to 480000(mm/min)
2022	plrap1	2	○	364	L	1 to 480000	Speed unit	1 to 480000(mm/min)
2023	plclmp	2	○	368	L	1 to 480000	Speed unit	1 to 480000(mm/min)
2024	offset							Data setting not possible
2025	G0fwdg	2	○	348	S	0 to 200		0 to 200(%)
2026	fwd_g	2	○	350	S	0 to 200		0 to 200(%)
2027	vir_ax	2	○	69	H6	0 to 1		
2028	ref-	2	○	340	L	0 to 179999×2	Interpola-tion unit	0 to 179.999(mm)
2029	ref+	2	○	336	L	0 to 179999×2	Interpola-tion unit	0 to 179.999(mm)
2030	spx_1	2	○	608	D	0 to FF		(Hexadecimal number)
2031	2							Not used.
2032	3							Not used.
2033	baseps	2	○	344	L	±99999999×2	Interpola-tion unit	±99999.999(mm)
2034	m_clamp	2	○	392	L	1 to 480000	Speed unit	1 to 480000 (mm/min)
2035	pG0t3	2	○	108	S	1 to 1500	1ms	
2036	pG1t3	2	○	110	S	1 to 5000	1ms	
2037	pl3rap0	2	○	372	L	1 to 480000	Speed unit	1 to 480000 (mm/min)
2038	pl3rap1	2	○	376	L	1 to 480000	Speed unit	1 to 480000 (mm/min)
2039	pl3rap2	2	○	380	L	1 to 480000	Speed unit	1 to 480000 (mm/min)
2040	pl3clmp0	2	○	384	L	1 to 480000	Speed unit	1 to 480000 (mm/min)
2041	pl3clmp1	2	○	388	L	1 to 480000	Speed unit	1 to 480000 (mm/min)
2042	handle	2	○	71	H4	0 to 1		
2043	tlml_1	2	○	396	L	±99999999×2	Interpola-tion unit	±99999.999 (mm)

6.2.5 Zero point return parameter

#	Parameter	P	A	N	Data type	Setting range	(Unit)	Remarks
2101	G28rap	2	○	12	L	1 to 480000	Speed unit	1 to 480000(mm/min)
2102	G28crp	2	○	88	S	1 to 480000	Speed unit	1 to 480000(mm/min)
2103	G28sft	2	○	94	S	0 to 65535	1µm	
2104	grspc	2	○	92	S	0 to 32767		
2105	grmask	2	○	90	S	0 to 65535	1µm	
2106								
2107	dir(-)	2	○	69	H2	0 to 1		
2108	noref	2	○	68	H2	0 to 1		
2109	Z_pulse	2	○	68	H1	0 to 1		
2110	nochk	2	○	69	H0	0 to 1		
2111								
2112								
2113	#1_rfp	2	○	24	L	±99999999×2	Interpola-tion unit	±99999.999(mm)
2114	#2_rfp	2	○	28	L	±99999999×2	Interpola-tion unit	±99999.999(mm)
2115	#3_rfp	2	○	32	L	±99999999×2	Interpola-tion unit	±99999.999(mm)
2116	#4_rfp	2	○	36	L	±99999999×2	Interpola-tion unit	±99999.999(mm)

6.2.6 Absolute position set

#	Parameter	P	A	N	Data type	Setting range	(Unit)	Remarks
	ABS POSITION							Data setting not possible
1201	SET							Data setting not possible
1202	base							Data setting not possible
1203	slip							Data setting not possible
1204	G28max	2	○	292	S	0 to 65535		
1205	no stopper	2	○	284	H2	0 to 1		
1206	abs.ILP±%							Data setting not possible
1207	OD							Data setting not possible
1208	ref.pnt.typ	2	○	285	D	0 to 1		
1209	approach	2	○	304	L	0 to 99999999×2	Interpolation unit	0 to 99999.999(mm)

6.2.7 Position switch

#	Parameter	P	A	N	Data type	Setting range	(Unit)	Remarks
7501	axis	16	×	16	D	ASCII		Axis name
7502	dog1			0	L	±99999999×2	Interpolation unit	±99999.999(mm)
7503	dog2			4	L	±99999999×2	Interpolation unit	±99999.999(mm)
7511	axis	16	×	36	D	ASCII		Axis name
7512	dog1			20	L	±99999999×2	Interpolation unit	±99999.999(mm)
7513	dog2			24	L	±99999999×2	Interpolation unit	±99999.999(mm)
7521	axis	16	×	56	D	ASCII		Axis name
7522	dog1			40	L	±99999999×2	Interpolation unit	±99999.999(mm)
7523	dog2			44	L	±99999999×2	Interpolation unit	±99999.999(mm)
7531	axis	16	×	76	D	ASCII		Axis name
7532	dog1			60	L	±99999999×2	Interpolation unit	±99999.999(mm)
7533	dog2			64	L	±99999999×2	Interpolation unit	±99999.999(mm)
7541	axis	16	×	96	D	ASCII		Axis name
7542	dog1			80	L	±99999999×2	Interpolation unit	±99999.999(mm)
7543	dog2			84	L	±99999999×2	Interpolation unit	±99999.999(mm)
7551	axis	16	×	116	D	ASCII		Axis name
7552	dog1			100	L	±99999999×2	Interpolation unit	±99999.999(mm)
7553	dog2			104	L	±99999999×2	Interpolation unit	±99999.999(mm)
7561	axis	16	×	136	D	ASCII		Axis name
7562	dog1			120	L	±99999999×2	Interpolation unit	±99999.999(mm)
7563	dog2			124	L	±99999999×2	Interpolation unit	±99999.999(mm)
7571	axis	16	×	156	D	ASCII		Axis name
7572	dog1			140	L	±99999999×2	Interpolation unit	±99999.999(mm)
7573	dog2			144	L	±99999999×2	Interpolation unit	±99999.999(mm)

6.2.8 Servo parameter

#	Parameter	P	A	N	Data type	Setting range	(Unit)	Remarks
2201 to 2264	SV001 to SV064	2	○	112+ 2n	S	Refer to Parameter Manual.		n: 0 to 63 (integer) n = # numbers – 2201

6.2.9 Machine error compensation

#	Parameter	P	A	N	Data type	Setting range	(Unit)	Remarks
4001	cmpax	3	○	1	S	0 to FFFF (Hexadecimal number)		00 00(H) │ └ Lower: System no-1 └───Upper: Axis no-1 **Ex.)** 2nd axis (0102H) of 3rd system.
4002	drcax	3	○	2	S	0 to FFFF (Hexadecimal number)		
4003	rdvno	3	○	3	S	0 to 256		
4004	mdvno	3	○	4	S	0 to 256		
4005	pdvno	3	○	5	S	0 to 256		
4006	sc	3	○	6	S	0 to 99		
4007	spcdv	3	○	7	L	0 to 9999999×2	Interpolation unit	0 to 9999999

6.2.10 Machine compensation data

#	Parameter	P	A	N	Data type	Setting range	(Unit)	Remarks
4301 to 5836	Machine compensation data	4	×	1 to 1536	D	–128 to 127		N numbers = # numbers－4300

APPENDIX 6 CORRESPONDENCE TABLE OF PROGRAM PARAMETER INPUT N NUMBERS

6.2.11 Macro list

#	Parameter	P	A	N	Data type	Setting range	(Unit)	Remarks
7201	G[01] <Code>	1	×	4	S	1 to 255		
	<Type>	1	×	6	D	0 to 3		
	<Program No.>	1	×	0	L	1 to 99999999		
7211	G[02] <Code>	1	×	12	S	1 to 255		
	<Type>	1	×	14	D	0 to 3		
	<Program No.>	1	×	8	L	1 to 99999999		
7221	G[03] <Code>	1	×	20	S	1 to 255		
	<Type>	1	×	22	D	0 to 3		
	<Program No.>	1	×	16	L	1 to 99999999		
7231	G[04] <Code>	1	×	28	S	1 to 255		
	<Type>	1	×	30	D	0 to 3		
	<Program No.>	1	×	24	L	1 to 99999999		
7241	G[05] <Code>	1	×	36	S	1 to 255		
	<Type>	1	×	38	D	0 to 3		
	<Program No.>	1	×	32	L	1 to 99999999		
7251	G[06] <Code>	1	×	44	S	1 to 255		
	<Type>	1	×	46	D	0 to 3		
	<Program No.>	1	×	40	L	1 to 99999999		
7261	G[07] <Code>	1	×	52	S	1 to 255		
	<Type>	1	×	54	D	0 to 3		
	<Program No.>	1	×	48	L	1 to 99999999		
7271	G[08] <Code>	1	×	60	S	1 to 255		
	<Type>	1	×	62	D	0 to 3		
	<Program No.>	1	×	56	L	1 to 99999999		
7281	G[09] <Code>	1	×	68	S	1 to 255		
	<Type>	1	×	70	D	0 to 3		
	<Program No.>	1	×	64	L	1 to 99999999		
7291	G[10] <Code>	1	×	76	S	1 to 255		
	<Type>	1	×	78	D	0 to 3		
	<Program No.>	1	×	72	L	1 to 99999999		
7401	G200 <Type>	1	×	81	D	0 to 3		
	<Program No.>	1	×	80	D	0 to 9		
7411	G300 <Type>	1	×	83	D	0 to 3		
	<Program No.>	1	×	82	D	0 to 9		
7421	G400 <Type>	1	×	85	D	0 to 3		
	<Program No.>	1	×	84	D	0 to 9		

APPENDIX 6 CORRESPONDENCE TABLE OF PROGRAM PARAMETER INPUT N NUMBERS

#	Parameter	P	A	N	Data type	Setting range	(Unit)	Remarks
7431	G500 <Type>	1	×	87	D	0 to 3		
	<Program No.>	1	×	86	D	0 to 9		
7441	G600 <Type>	1	×	89	D	0 to 3		
	<Program No.>	1	×	88	D	0 to 9		
7451	G700 <Type>	1	×	91	D	0 to 3		
	<Program No.>	1	×	90	D	0 to 9		
7461	G800 <Type>	1	×	93	D	0 to 3		
	<Program No.>	1	×	92	D	0 to 9		
7471	G900 <Type>	1	×	95	D	0 to 3		
	<Program No.>	1	×	94	D	0 to 9		
7481	Pcint							
	<Program No.>	1	×	191	D	0 to 9		
7001	M[01] <Code>	1	×	100	S	1 to 9999		
	<Type>	1	×	102	D	0 to 5		
	<Program No.>	1	×	96	L	1 to 99999999		
7011	M[02] <Code>	1	×	108	S	1 to 9999		
	<Type>	1	×	110	D	0 to 5		
	<Program No.>	1	×	104	L	1 to 99999999		
7021	M[03] <Code>	1	×	116	S	1 to 9999		
	<Type>	1	×	118	D	0 to 5		
	<Program No.>	1	×	112	L	1 to 99999999		
7031	M[04] <Code>	1	×	124	S	1 to 9999		
	<Type>	1	×	126	D	0 to 5		
	<Program No.>	1	×	120	L	1 to 99999999		
7041	M[05] <Code>	1	×	132	S	1 to 9999		
	<Type>	1	×	134	D	0 to 5		
	<Program No.>	1	×	128	L	1 to 99999999		
7051	M[06] <Code>	1	×	140	S	1 to 9999		
	<Type>	1	×	142	D	0 to 5		
	<Program No.>	1	×	136	L	1 to 99999999		
7061	M[07] <Code>	1	×	148	S	1 to 9999		
	<Type>	1	×	150	D	0 to 5		
	<Program No.>	1	×	144	L	1 to 99999999		
7071	M[08] <Code>	1	×	156	S	1 to 9999		
	<Type>	1	×	158	D	0 to 5		
	<Program No.>	1	×	152	L	1 to 99999999		
7081	M[09] <Code>	1	×	164	S	1 to 9999		
	<Type>	1	×	166	D	0 to 5		
	<Program No.>	1	×	160	L	1 to 99999999		
7091	M[10] <Code>	1	×	172	S	1 to 9999		
	<Type>	1	×	174	D	0 to 5		
	<Program No.>	1	×	168	L	1 to 99999999		
7102	M2mac <Type>	1	×	190	D	0 to 3		
	<Program No.>	1	×	184	L	1 to 99999999		
7302	Smac <Type>	1	×	188	D	0 to 3		
	<Program No.>	1	×	176	L	1 to 99999999		
7312	Tmac <Type>	1	×	189	D	0 to 7		
	<Program No.>	1	×	180	L	1 to 99999999		

APPENDIX 6 CORRESPONDENCE TABLE OF PROGRAM PARAMETER INPUT N NUMBERS

#	Parameter	P	A	N	Data type	Setting range	(Unit)	Remarks
10001	M[11] <Code>	1	×	1060	S	1 to 9999		
	<Type>	1	×	1062	D	0 to 5		
	<Program No.>	1	×	1056	L	1 to 99999999		
10011	M[12] <Code>	1	×	1068	S	1 to 9999		
	<Type>	1	×	1070	D	0 to 5		
	<Program No.>	1	×	1064	L	1 to 99999999		
10021	M[13] <Code>	1	×	1076	S	1 to 9999		
	<Type>	1	×	1078	D	0 to 5		
	<Program No.>	1	×	1072	L	1 to 99999999		
10031	M[14] <Code>	1	×	1084	S	1 to 9999		
	<Type>	1	×	1086	D	0 to 5		
	<Program No.>	1	×	1080	L	1 to 99999999		
10041	M[15] <Code>	1	×	1092	S	1 to 9999		
	<Type>	1	×	1094	D	0 to 5		
	<Program No.>	1	×	1088	L	1 to 99999999		
10051	M[16] <Code>	1	×	1100	S	1 to 9999		
	<Type>	1	×	1102	D	0 to 5		
	<Program No.>	1	×	1096	L	1 to 99999999		
10061	M[17] <Code>	1	×	1108	S	1 to 9999		
	<Type>	1	×	1110	D	0 to 5		
	<Program No.>	1	×	1104	L	1 to 99999999		
10071	M[18] <Code>	1	×	1116	S	1 to 9999		
	<Type>	1	×	1118	D	0 to 5		
	<Program No.>	1	×	1112	L	1 to 99999999		
10081	M[19] <Code>	1	×	1124	S	1 to 9999		
	<Type>	1	×	1126	D	0 to 5		
	<Program No.>	1	×	1120	L	1 to 99999999		
10091	M[20] <Code>	1	×	1132	S	1 to 9999		
	<Type>	1	×	1134	D	0 to 5		
	<Program No.>	1	×	1128	L	1 to 99999999		

APPENDIX 6 CORRESPONDENCE TABLE OF PROGRAM PARAMETER INPUT N NUMBERS

#	Parameter	P	A	N	Data type	Setting range	(Unit)	Remarks
10101	M[21] <Code>	1	×	1140	S	1 to 9999		
	<Type>	1	×	1142	D	0 to 5		
	<Program No.>	1	×	1136	L	1 to 99999999		
10111	M[22] <Code>	1	×	1148	S	1 to 9999		
	<Type>	1	×	1150	D	0 to 5		
	<Program No.>	1	×	1144	L	1 to 99999999		
10121	M[23] <Code>	1	×	1156	S	1 to 9999		
	<Type>	1	×	1158	D	0 to 5		
	<Program No.>	1	×	1152	L	1 to 99999999		
10131	M[24] <Code>	1	×	1164	S	1 to 9999		
	<Type>	1	×	1166	D	0 to 5		
	<Program No.>	1	×	1160	L	1 to 99999999		
10141	M[25] <Code>	1	×	1172	S	1 to 9999		
	<Type>	1	×	1174	D	0 to 5		
	<Program No.>	1	×	1168	L	1 to 99999999		
10151	M[26] <Code>	1	×	1180	S	1 to 9999		
	<Type>	1	×	1182	D	0 to 5		
	<Program No.>	1	×	1176	L	1 to 99999999		
10161	M[27] <Code>	1	×	1188	S	1 to 9999		
	<Type>	1	×	1190	D	0 to 5		
	<Program No.>	1	×	1184	L	1 to 99999999		
10171	M[28] <Code>	1	×	1196	S	1 to 9999		
	<Type>	1	×	1198	D	0 to 5		
	<Program No.>	1	×	1192	L	1 to 99999999		
10181	M[29] <Code>	1	×	1204	S	1 to 9999		
	<Type>	1	×	1206	D	0 to 5		
	<Program No.>	1	×	1200	L	1 to 99999999		
10191	M[30] <Code>	1	×	1212	S	1 to 9999		
	<Type>	1	×	1214	D	0 to 5		
	<Program No.>	1	×	1208	L	1 to 99999999		

APPENDIX 6 CORRESPONDENCE TABLE OF PROGRAM PARAMETER INPUT N NUMBERS

#	Parameter	P	A	N	Data type	Setting range	(Unit)	Remarks
10201	M[31] <Code>	1	×	1220	S	1 to 9999		
	<Type>	1	×	1222	D	0 to 5		
	<Program No.>	1	×	1216	L	1 to 99999999		
10211	M[32] <Code>	1	×	1228	S	1 to 9999		
	<Type>	1	×	1230	D	0 to 5		
	<Program No.>	1	×	1224	L	1 to 99999999		
10221	M[33] <Code>	1	×	1236	S	1 to 9999		
	<Type>	1	×	1238	D	0 to 5		
	<Program No.>	1	×	1232	L	1 to 99999999		
10231	M[34] <Code>	1	×	1244	S	1 to 9999		
	<Type>	1	×	1246	D	0 to 5		
	<Program No.>	1	×	1240	L	1 to 99999999		
10241	M[35] <Code>	1	×	1252	S	1 to 9999		
	<Type>	1	×	1254	D	0 to 5		
	<Program No.>	1	×	1248	L	1 to 99999999		
10251	M[36] <Code>	1	×	1260	S	1 to 9999		
	<Type>	1	×	1262	D	0 to 5		
	<Program No.>	1	×	1256	L	1 to 99999999		
10261	M[37] <Code>	1	×	1268	S	1 to 9999		
	<Type>	1	×	1270	D	0 to 5		
	<Program No.>	1	×	1264	L	1 to 99999999		
10271	M[38] <Code>	1	×	1276	S	1 to 9999		
	<Type>	1	×	1278	D	0 to 5		
	<Program No.>	1	×	1272	L	1 to 99999999		
10281	M[39] <Code>	1	×	1284	S	1 to 9999		
	<Type>	1	×	1286	D	0 to 5		
	<Program No.>	1	×	1280	L	1 to 99999999		
10291	M[40] <Code>	1	×	1292	S	1 to 9999		
	<Type>	1	×	1294	D	0 to 5		
	<Program No.>	1	×	1288	L	1 to 99999999		

6.2.12 Spindle NC parameter

#	Parameter	P	A	N	Data type	Setting range	(Unit)	Remarks
3001	slimt 1	21	○	784	L	0 to 99999		
3002	2	21	○	788	L	0 to 99999		
3003	3	21	○	792	L	0 to 99999		
3004	4	21	○	796	L	0 to 99999		
3005	smax 1	21	○	768	L	0 to 99999		
3006	2	21	○	772	L	0 to 99999		
3007	3	21	○	776	L	0 to 99999		
3008	4	21	○	780	L	0 to 99999		
3009	ssift 1	21	○	816	S	0 to 32767		
3010	2	21	○	818	S	0 to 32767		
3011	3	21	○	820	S	0 to 32767		
3012	4	21	○	822	S	0 to 32767		
3013	stap 1	21	○	800	L	0 to 99999		
3014	2	21	○	804	L	0 to 99999		
3015	3	21	○	808	L	0 to 99999		
3016	4	21	○	812	L	0 to 99999		
3017	stapt 1	21	○	824	S	1 to 5000		
3018	2	21	○	826	S	1 to 5000		
3019	3	21	○	828	S	1 to 5000		
3020	4	21	○	830	S	1 to 5000		
3021	sori	21	○	848	S	0 to 32767		
3022	sgear	21	○	853	D	0 to 3		
3023	smini	21	○	850	S	0 to 32767		
3024	serr	21	○	854	D	0 to 99		
3025	sname	21	○	855	D	0 to 9		
3026	sprcmm	21	○	860	L	0 to 999999		
3027	senc_pno	21	○	868	D	0 to 2		
3028	sana_pno	21	○	866	D	0 to 2		
3029	spflg	21	○	869	D	0 to FF		(Hexadecimal number)
3030	senc_no	21	○	867	D	0 to FF		(Hexadecimal number)
3031	sana_no	21	○	865	D	0 to FF		(Hexadecimal number)
3032	smcp_no	21	○	857	D	0 to FF		(Hexadecimal number)

APPENDIX 6 CORRESPONDENCE TABLE OF PROGRAM PARAMETER INPUT N NUMBERS

#	Parameter	P	A	N	Data type	Setting range	(Unit)	Remarks
3033	spt	21	○	858	S	0 to 9999		
3034	sprlv	21	○	832	S	0 to 4095		
3035	spplv	21	○	834	S	0 to 4095		
3036	sptc1	21	○	836	L	0 to 99999		
3037	sptc2	21	○	840	L	0 to 99999		
3038	spdiv1	21	○	844	D	0 to 127		
3039	spdiv2	21	○	845	D	0 to 127		
3040	spplr	21	○	846	H0	0 to 1		
3041	sppst	21	○	872	L	0 to 359999	1/1000°	0.000 to 359.999(°)
3042	GBsp	21	○	901	D	0 to 2		
3043	sptc3	21	○	876	L	0 to 99999		0 to 99999(r/min)
3044	sptc4	21	○	880	L	0 to 99999		0 to 99999(r/min)
3045	sptc5	21	○	884	L	0 to 99999		0 to 99999(r/min)
3046	sptc6	21	○	888	L	0 to 99999		0 to 99999(r/min)
3047	sptc7	21	○	892	L	0 to 99999		0 to 99999(r/min)
3048	spdiv3	21	○	896	D	0 to 127		
3049	spdiv4	21	○	897	D	0 to 127		
3050	spdiv5	21	○	898	D	0 to 127		
3051	spdiv6	21	○	899	D	0 to 127		
3052	spdiv7	21	○	900	D	0 to 127		

6.2.13 Spindle parameter

#	Parameter	P	A	N	Data type	Setting range	(Unit)	Remarks
3201 to 3584	SP001 to SP384	21	○	2n	S	Refer to Parameter Manual.		n: 0 to 383 (integer) n=#number – 3201

6.2.14 Spindle type servo parameter

#	Parameter	P	A	N	Data type	Setting range	(Unit)	Remarks
3601 to 3664	SV001 to SV064	21	○	1024 +2n	S	Refer to Parameter Manual.		n: 0 to 63 (integer) n=#number – 3601
	(P number for M500L compatibility)	17	×	2n	S	Refer to Parameter Manual.		n: 0 to 63, 80 to 143, 160 to 223, 240 to 303, 320 to 383, 400 to 463 (integer) n=(#number – 3601) +(spindle number – 1)×80 The spindle numbers correspond to <1> to <6> on the screen.

6.2.15 PLC constant

#	Parameter	P	A	N	Data type	Setting range	(Unit)	Remarks
6301 to 6348	PLC constant	5	×	1 to 48	L	±99999999		N number=#number – 6300

6.2.16 PLC timer

#	Parameter	P	A	N	Data type	Setting range	(Unit)	Remarks
6000 to 6015	10ms add timer	6	×	0 to 15	S	0 to 32767	0.01s	N number=#number – 6000
6016～6095	100ms add timer	6	×	16 to 95	S	0 to 32767	0.1s	
6096～6103	100ms accumulating timer	6	×	96 to 103	S	0 to 32767	0.1s	

6.2.17 PLC counter

#	Parameter	P	A	N	Data type	Setting range	(Unit)	Remarks
6200 to 6223	PLC constant	7	×	0 to 23	S	0 to 32767		N number=#number – 6200

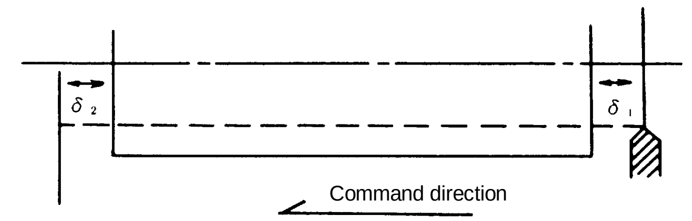
6.2.18 Bit selection

#	Parameter	P	A	N	Data type	Setting range	(Unit)	Remarks
6401 to 6496	Bit selection	8	×	1 to 96	D or H0 to 7	0 to FF (Hexadecimal number) 0 to 256 (Decimal number) 0 to 1		N number=#number – 6400 #6449 to 6496 are used by the machine manufacturer and Mitsubishi, so the contents are fixed.

APPENDIX 7 SUPPLEMENTARY DETAILS ON INCOMPLETE THREAD AREAS ARISING DURING THREAD CUTTING

Function and purpose

The delay caused by the automatic acceleration/deceleration and delay caused by the position loop in the servo system create an illegal pitch near the start and end points of thread cutting. When programming steps must be taken to assign thread cutting commands which include a margin for the approach distance δ_1 and for the length of the area δ_2 where the thread is incomplete during chamfering, as shown in the figure below.



δ_1 : Approach distance
δ_2 : Area where thread is incomplete during chamfering

Approach distance [δ_1]

(1) When Ts is not equal to zero:

$$\delta_1 = \frac{F}{60} t1 - \frac{F}{60} \left(Ts + Tp - \frac{Tp^2 e^{-\frac{t1}{Tp}} - Ts^2 e^{-\frac{t1}{Tp}}}{Tp - Ts} \right) \quad \text{(mm)}$$

Where F : Thread cutting speed (mm/min)
Ts : Acceleration/deceleration time constant (s)
Tp : Position loop time constant (s)
t1 : Time taken until pitch error reaches allowable limit "a" (s)

If "p" is the pitch and "ΔP" is the pitch error, then allowable limit "a" will be:

$$a = \frac{1}{Tp - Ts} \left(Tpe^{-\frac{t1}{Tp}} - Tse^{-\frac{t1}{Ts}} \right)$$

(2) When Ts is equal to zero

$$\delta_1 = \frac{F}{60} t1 - \frac{F}{60} \left(Tp + Tpe^{\frac{T1}{Tp}} \right) \quad \text{(mm)}$$

$$a = e^{-\frac{t1}{Tp}}$$

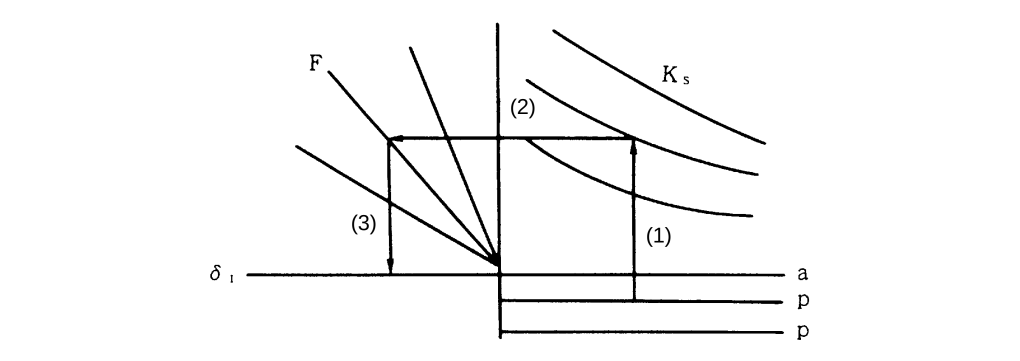
APPENDIX 7 SUPPLEMENTARY DETAILS ON INCOMPLETE THREAD AREAS ARISING DURING THREAD CUTTING

Since the calculation of approach distance δ_1 is a complicated procedure, δ_1 is normally determined from the chart on the next page. This chart is used as follows.

(a) Find the position on the p axis scale represents by the thread grade and pitch [P], and follow the perpendicular (1) drawn upward to find the point where it intersects with the curve of acceleration/deceleration time constant [Ts].

(b) Follow horizontal line (2) and find where it intersects with the thread cutting speed [F].

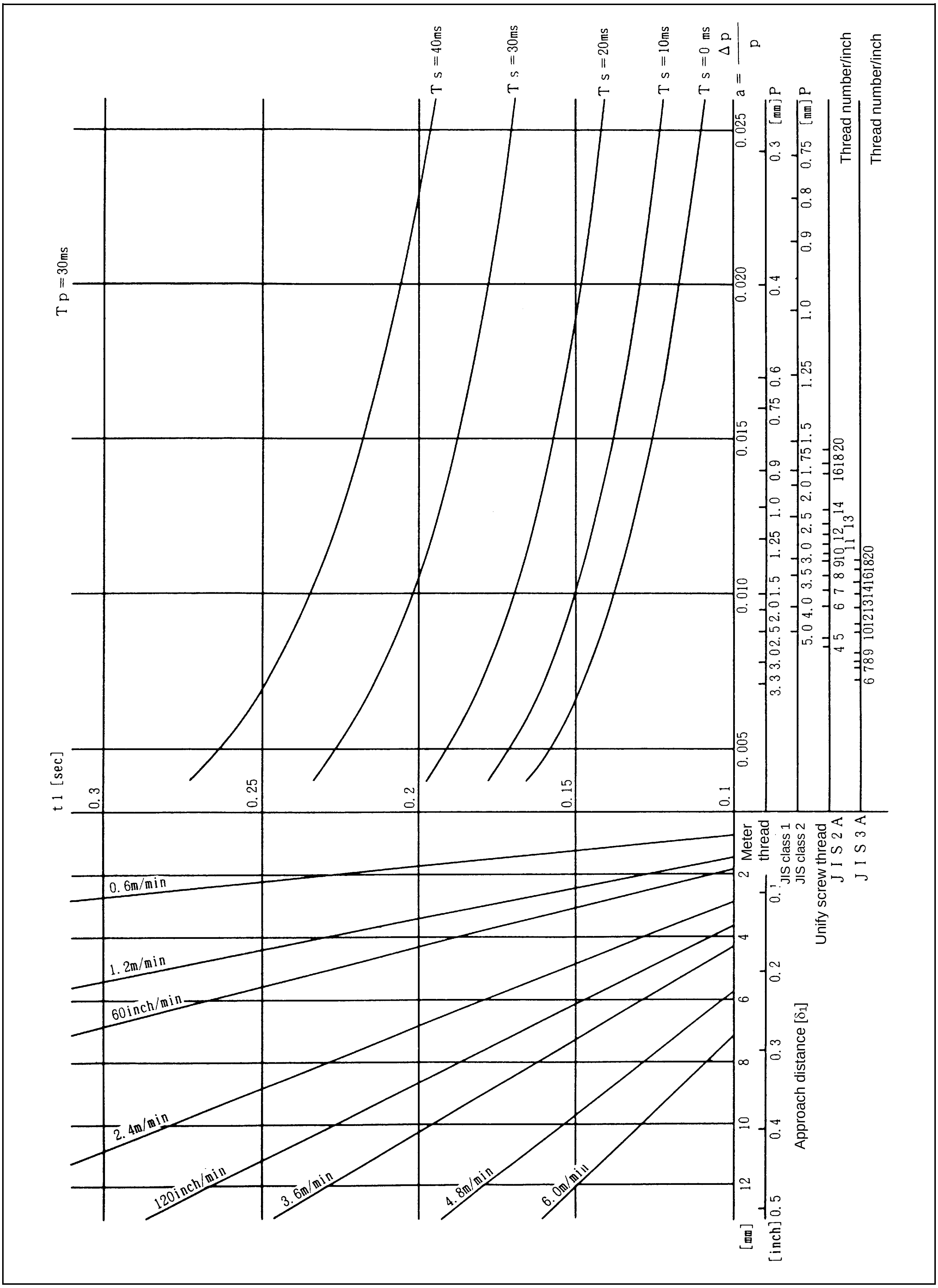
(c) Follow perpendicular (3) and find approach distance [δ_1] on the scale at the point where it intersects with the δ_1 axis.



(Note 1) The chart on the next page applies when the position loop time constant Tp is 30ms.

APPENDIX 7 SUPPLEMENTARY DETAILS ON INCOMPLETE THREAD AREAS ARISING DURING THREAD CUTTING

Approach distance δ_1 calculation chart



APPENDIX 7 SUPPLEMENTARY DETAILS ON INCOMPLETE THREAD AREAS ARISING DURING THREAD CUTTING

Length of area where thread is incomplete during chamfering [δ_2]

$$\delta_2 = (Ts + Tp)\frac{F}{60} \quad \text{(mm)}$$

Where F : Thread cutting speed (mm/min)
Ts : Acceleration/deceleration time constant (s)
Tp : Position loop time constant (s)

(Note 2) When proceeding with chamfering during a thread cutting cycle, the length of the area where the thread is incomplete is equivalent to the value produced by adding δ_2 determined by the above formula to the chamfering pitch set by parameter.

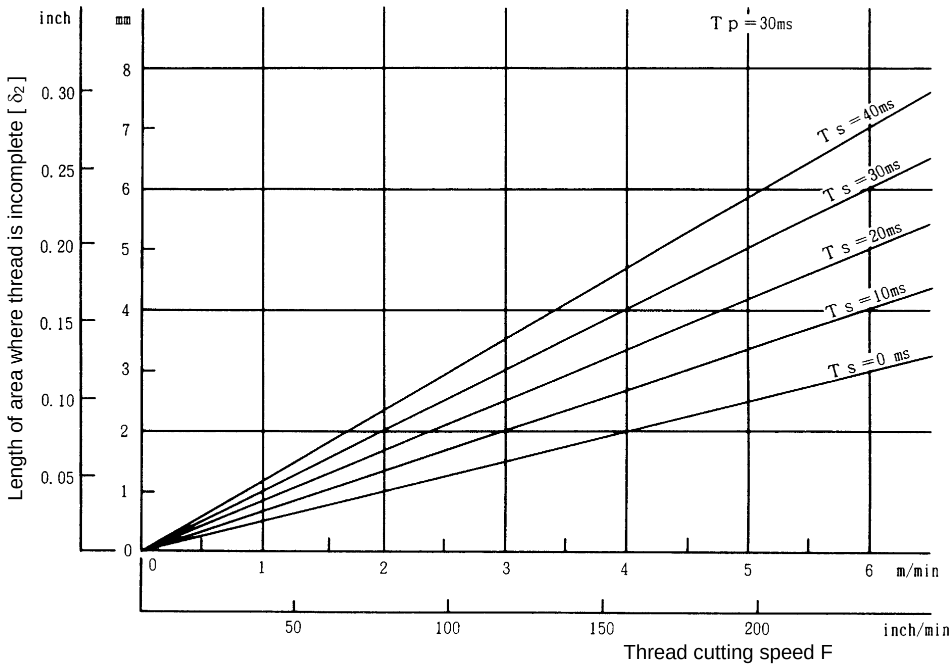


Chart to calculate length of area where thread is incomplete [δ_2]

APPENDIX 8 MACRO INTERFACE EXPANSION

Function and purpose

By using the macro interface input/output function, data can be input and output between the machining program and user PLC.
The number of macro interface input/output points will be expanded as shown below when the macro interface expansion is used.

<Number of points for input>

128 points (of which, one-point read is 32 points) → 512 points (of which, one-point read is 160 points)

<Number of points for output>

128 points (of which, one-point write is 32 points) → 512 points (of which, one-point write is 160 points)

Refer to the tables on the following pages for the number of each point used for input/output.

8.1 Macro Interface Input

Function and purpose

The macro interface input signals are used to find the user PLC status with the machining program.
The machining program can read the details written into R72 to R79 and R1820 to R1843 with the user PLC using the variables #1000 to #1035, #1216 to #1227 and #1400 to #1527.
Variables include those that read one point (1 bit) of the R register, and those that read two R registers (32 points). The variable value read with the R register one-point read variable can either be 1 or 0 (1 = contact closed/0 = contact open).

List of variables

<One-point read variables>

Variable number	Number of points	R register	BIT	Variable number	Number of points	R register	BIT	Variable number	Number of points	R register	BIT
# 1000	1	R 72	BIT 0	# 1400	1	R 1820	BIT 0	# 1432	1	R 1822	BIT 0
# 1001	1	R 72	BIT 1	# 1401	1	R 1820	BIT 1	# 1433	1	R 1822	BIT 1
# 1002	1	R 72	BIT 2	# 1402	1	R 1820	BIT 2	# 1434	1	R 1822	BIT 2
# 1003	1	R 72	BIT 3	# 1403	1	R 1820	BIT 3	# 1435	1	R 1822	BIT 3
# 1004	1	R 72	BIT 4	# 1404	1	R 1820	BIT 4	# 1436	1	R 1822	BIT 4
# 1005	1	R 72	BIT 5	# 1405	1	R 1820	BIT 5	# 1437	1	R 1822	BIT 5
# 1006	1	R 72	BIT 6	# 1406	1	R 1820	BIT 6	# 1438	1	R 1822	BIT 6
# 1007	1	R 72	BIT 7	# 1407	1	R 1820	BIT 7	# 1439	1	R 1822	BIT 7
# 1008	1	R 72	BIT 8	# 1408	1	R 1820	BIT 8	# 1440	1	R 1822	BIT 8
# 1009	1	R 72	BIT 9	# 1409	1	R 1820	BIT 9	# 1441	1	R 1822	BIT 9
# 1010	1	R 72	BIT A	# 1410	1	R 1820	BIT A	# 1442	1	R 1822	BIT A
# 1011	1	R 72	BIT B	# 1411	1	R 1820	BIT B	# 1443	1	R 1822	BIT B
# 1012	1	R 72	BIT C	# 1412	1	R 1820	BIT C	# 1444	1	R 1822	BIT C
# 1013	1	R 72	BIT D	# 1413	1	R 1820	BIT D	# 1445	1	R 1822	BIT D
# 1014	1	R 72	BIT E	# 1414	1	R 1820	BIT E	# 1446	1	R 1822	BIT E
# 1015	1	R 72	BIT F	# 1415	1	R 1820	BIT F	# 1447	1	R 1822	BIT F
# 1016	1	R 73	BIT 0	# 1416	1	R 1821	BIT 0	# 1448	1	R 1823	BIT 0
# 1017	1	R 73	BIT 1	# 1417	1	R 1821	BIT 1	# 1449	1	R 1823	BIT 1
# 1018	1	R 73	BIT 2	# 1418	1	R 1821	BIT 2	# 1450	1	R 1823	BIT 2
# 1019	1	R 73	BIT 3	# 1419	1	R 1821	BIT 3	# 1451	1	R 1823	BIT 3
# 1020	1	R 73	BIT 4	# 1420	1	R 1821	BIT 4	# 1452	1	R 1823	BIT 4
# 1021	1	R 73	BIT 5	# 1421	1	R 1821	BIT 5	# 1453	1	R 1823	BIT 5
# 1022	1	R 73	BIT 6	# 1422	1	R 1821	BIT 6	# 1454	1	R 1823	BIT 6
# 1023	1	R 73	BIT 7	# 1423	1	R 1821	BIT 7	# 1455	1	R 1823	BIT 7
# 1024	1	R 73	BIT 8	# 1424	1	R 1821	BIT 8	# 1456	1	R 1823	BIT 8
# 1025	1	R 73	BIT 9	# 1425	1	R 1821	BIT 9	# 1457	1	R 1823	BIT 9
# 1026	1	R 73	BIT A	# 1426	1	R 1821	BIT A	# 1458	1	R 1823	BIT A
# 1027	1	R 73	BIT B	# 1427	1	R 1821	BIT B	# 1459	1	R 1823	BIT B
# 1028	1	R 73	BIT C	# 1428	1	R 1821	BIT C	# 1460	1	R 1823	BIT C
# 1029	1	R 73	BIT D	# 1429	1	R 1821	BIT D	# 1461	1	R 1823	BIT D
# 1030	1	R 73	BIT E	# 1430	1	R 1821	BIT E	# 1462	1	R 1823	BIT E
# 1031	1	R 73	BIT F	# 1431	1	R 1821	BIT F	# 1463	1	R 1823	BIT F

Variable number	Number of points	R register	BIT	Variable number	Number of points	R register	BIT
# 1464	1	R 1824	BIT 0	# 1496	1	R 1826	BIT 0
# 1465	1	R 1824	BIT 1	# 1497	1	R 1826	BIT 1
# 1466	1	R 1824	BIT 2	# 1498	1	R 1826	BIT 2
# 1467	1	R 1824	BIT 3	# 1499	1	R 1826	BIT 3
# 1468	1	R 1824	BIT 4	# 1500	1	R 1826	BIT 4
# 1469	1	R 1824	BIT 5	# 1501	1	R 1826	BIT 5
# 1470	1	R 1824	BIT 6	# 1502	1	R 1826	BIT 6
# 1471	1	R 1824	BIT 7	# 1503	1	R 1826	BIT 7
# 1472	1	R 1824	BIT 8	# 1504	1	R 1826	BIT 8
# 1473	1	R 1824	BIT 9	# 1505	1	R 1826	BIT 9
# 1474	1	R 1824	BIT A	# 1506	1	R 1826	BIT A
# 1475	1	R 1824	BIT B	# 1507	1	R 1826	BIT B
# 1476	1	R 1824	BIT C	# 1508	1	R 1826	BIT C
# 1477	1	R 1824	BIT D	# 1509	1	R 1826	BIT D
# 1478	1	R 1824	BIT E	# 1510	1	R 1826	BIT E
# 1479	1	R 1824	BIT F	# 1511	1	R 1826	BIT F
# 1480	1	R 1825	BIT 0	# 1512	1	R 1827	BIT 0
# 1481	1	R 1825	BIT 1	# 1513	1	R 1827	BIT 1
# 1482	1	R 1825	BIT 2	# 1514	1	R 1827	BIT 2
# 1483	1	R 1825	BIT 3	# 1515	1	R 1827	BIT 3
# 1484	1	R 1825	BIT 4	# 1516	1	R 1827	BIT 4
# 1485	1	R 1825	BIT 5	# 1517	1	R 1827	BIT 5
# 1486	1	R 1825	BIT 6	# 1518	1	R 1827	BIT 6
# 1487	1	R 1825	BIT 7	# 1519	1	R 1827	BIT 7
# 1488	1	R 1825	BIT 8	# 1520	1	R 1827	BIT 8
# 1489	1	R 1825	BIT 9	# 1521	1	R 1827	BIT 9
# 1490	1	R 1825	BIT A	# 1522	1	R 1827	BIT A
# 1491	1	R 1825	BIT B	# 1523	1	R 1827	BIT B
# 1492	1	R 1825	BIT C	# 1524	1	R 1827	BIT C
# 1493	1	R 1825	BIT D	# 1525	1	R 1827	BIT D
# 1494	1	R 1825	BIT E	# 1526	1	R 1827	BIT E
# 1495	1	R 1825	BIT F	# 1527	1	R 1827	BIT F

The expanded section is #1400 to #1527.

<32-point read variables>

Variable number	Number of points	R register	Variable number	Number of points	R register
# 1032	32	R 72/73	# 1220	32	R 1828/1829
# 1033	32	R 74/75	# 1221	32	R 1830/1831
# 1034	32	R 76/77	# 1222	32	R 1832/1833
# 1035	32	R 78/79	# 1223	32	R 1834/1835
# 1216	32	R 1820/1821	# 1224	32	R 1836/1837
# 1217	32	R 1822/1823	# 1225	32	R 1838/1839
# 1218	32	R 1824/1825	# 1226	32	R 1840/1841
# 1219	32	R 1826/1827	# 1227	32	R 1842/1843

The expanded section is #1216 to #1227.

8.2 Macro Interface Output

Function and purpose

The macro interface output signals are used to find the user macro status with the user pLC.
The details written into #1100 to #1135, #1316 to #1327 and #1600 to #1727 with the machining program can be read with R172 to R179 and R1844 to R1867 R registers by the user PLC.
Variables include those that write to one point (1 bit) of the R register, and those that write to two R registers (32 points).
The variables and R registers used for macro interface output can also be used for macro interface input.

List of variables

<One-point write variables>

Variable number	Number of points	R register	BIT	Variable number	Number of points	R register	BIT	Variable number	Number of points	R register	BIT
# 1100	1	R 172	BIT 0	# 1600	1	R 1844	BIT 0	# 1632	1	R 1846	BIT 0
# 1101	1	R 172	BIT 1	# 1601	1	R 1844	BIT 1	# 1633	1	R 1846	BIT 1
# 1102	1	R 172	BIT 2	# 1602	1	R 1844	BIT 2	# 1634	1	R 1846	BIT 2
# 1103	1	R 172	BIT 3	# 1603	1	R 1844	BIT 3	# 1635	1	R 1846	BIT 3
# 1104	1	R 172	BIT 4	# 1604	1	R 1844	BIT 4	# 1636	1	R 1846	BIT 4
# 1105	1	R 172	BIT 5	# 1605	1	R 1844	BIT 5	# 1637	1	R 1846	BIT 5
# 1106	1	R 172	BIT 6	# 1606	1	R 1844	BIT 6	# 1638	1	R 1846	BIT 6
# 1107	1	R 172	BIT 7	# 1607	1	R 1844	BIT 7	# 1639	1	R 1846	BIT 7
# 1108	1	R 172	BIT 8	# 1608	1	R 1844	BIT 8	# 1640	1	R 1846	BIT 8
# 1109	1	R 172	BIT 9	# 1609	1	R 1844	BIT 9	# 1641	1	R 1846	BIT 9
# 1110	1	R 172	BIT A	# 1610	1	R 1844	BIT A	# 1642	1	R 1846	BIT A
# 1111	1	R 172	BIT B	# 1611	1	R 1844	BIT B	# 1643	1	R 1846	BIT B
# 1112	1	R 172	BIT C	# 1612	1	R 1844	BIT C	# 1644	1	R 1846	BIT C
# 1113	1	R 172	BIT D	# 1613	1	R 1844	BIT D	# 1645	1	R 1846	BIT D
# 1114	1	R 172	BIT E	# 1614	1	R 1844	BIT E	# 1646	1	R 1846	BIT E
# 1115	1	R 172	BIT F	# 1615	1	R 1844	BIT F	# 1647	1	R 1846	BIT F
# 1116	1	R 173	BIT 0	# 1616	1	R 1845	BIT 0	# 1648	1	R 1847	BIT 0
# 1117	1	R 173	BIT 1	# 1617	1	R 1845	BIT 1	# 1649	1	R 1847	BIT 1
# 1118	1	R 173	BIT 2	# 1618	1	R 1845	BIT 2	# 1650	1	R 1847	BIT 2
# 1119	1	R 173	BIT 3	# 1619	1	R 1845	BIT 3	# 1651	1	R 1847	BIT 3
# 1120	1	R 173	BIT 4	# 1620	1	R 1845	BIT 4	# 1652	1	R 1847	BIT 4
# 1121	1	R 173	BIT 5	# 1621	1	R 1845	BIT 5	# 1653	1	R 1847	BIT 5
# 1122	1	R 173	BIT 6	# 1622	1	R 1845	BIT 6	# 1654	1	R 1847	BIT 6
# 1123	1	R 173	BIT 7	# 1623	1	R 1845	BIT 7	# 1655	1	R 1847	BIT 7
# 1124	1	R 173	BIT 8	# 1624	1	R 1845	BIT 8	# 1656	1	R 1847	BIT 8
# 1125	1	R 173	BIT 9	# 1625	1	R 1845	BIT 9	# 1657	1	R 1847	BIT 9
# 1126	1	R 173	BIT A	# 1626	1	R 1845	BIT A	# 1658	1	R 1847	BIT A
# 1127	1	R 173	BIT B	# 1627	1	R 1845	BIT B	# 1659	1	R 1847	BIT B
# 1128	1	R 173	BIT C	# 1628	1	R 1845	BIT C	# 1660	1	R 1847	BIT C
# 1129	1	R 173	BIT D	# 1629	1	R 1845	BIT D	# 1661	1	R 1847	BIT D
# 1130	1	R 173	BIT E	# 1630	1	R 1845	BIT E	# 1662	1	R 1847	BIT E
# 1131	1	R 173	BIT F	# 1631	1	R 1845	BIT F	# 1663	1	R 1847	BIT F

<32-point write variables>

Variable number	Number of points	R register	BIT	Variable number	Number of points	R register	BIT
# 1664	1	R 1848	BIT 0	# 1696	1	R 1850	BIT 0
# 1665	1	R 1848	BIT 1	# 1697	1	R 1850	BIT 1
# 1666	1	R 1848	BIT 2	# 1698	1	R 1850	BIT 2
# 1667	1	R 1848	BIT 3	# 1699	1	R 1850	BIT 3
# 1668	1	R 1848	BIT 4	# 1700	1	R 1850	BIT 4
# 1669	1	R 1848	BIT 5	# 1701	1	R 1850	BIT 5
# 1670	1	R 1848	BIT 6	# 1702	1	R 1850	BIT 6
# 1671	1	R 1848	BIT 7	# 1703	1	R 1850	BIT 7
# 1672	1	R 1848	BIT 8	# 1704	1	R 1850	BIT 8
# 1673	1	R 1848	BIT 9	# 1705	1	R 1850	BIT 9
# 1674	1	R 1848	BIT A	# 1706	1	R 1850	BIT A
# 1675	1	R 1848	BIT B	# 1707	1	R 1850	BIT B
# 1676	1	R 1848	BIT C	# 1708	1	R 1850	BIT C
# 1677	1	R 1848	BIT D	# 1709	1	R 1850	BIT D
# 1678	1	R 1848	BIT E	# 1710	1	R 1850	BIT E
# 1679	1	R 1848	BIT F	# 1711	1	R 1850	BIT F
# 1680	1	R 1849	BIT 0	# 1712	1	R 1851	BIT 0
# 1681	1	R 1849	BIT 1	# 1713	1	R 1851	BIT 1
# 1682	1	R 1849	BIT 2	# 1714	1	R 1851	BIT 2
# 1683	1	R 1849	BIT 3	# 1715	1	R 1851	BIT 3
# 1684	1	R 1849	BIT 4	# 1716	1	R 1851	BIT 4
# 1685	1	R 1849	BIT 5	# 1717	1	R 1851	BIT 5
# 1686	1	R 1849	BIT 6	# 1718	1	R 1851	BIT 6
# 1687	1	R 1849	BIT 7	# 1719	1	R 1851	BIT 7
# 1688	1	R 1849	BIT 8	# 1720	1	R 1851	BIT 8
# 1689	1	R 1849	BIT 9	# 1721	1	R 1851	BIT 9
# 1690	1	R 1849	BIT A	# 1722	1	R 1851	BIT A
# 1691	1	R 1849	BIT B	# 1723	1	R 1851	BIT B
# 1692	1	R 1849	BIT C	# 1724	1	R 1851	BIT C
# 1693	1	R 1849	BIT D	# 1725	1	R 1851	BIT D
# 1694	1	R 1849	BIT E	# 1726	1	R 1851	BIT E
# 1695	1	R 1849	BIT F	# 1727	1	R 1851	BIT F

The expanded section is #1600 to #1727.

<32-point write variables>

Variable number	Number of points	R register	Variable number	Number of points	R register
# 1132	32	R 172/173	# 1320	32	R 1852/1853
# 1133	32	R 174/175	# 1321	32	R 1854/1855
# 1134	32	R 176/177	# 1322	32	R 1856/1857
# 1135	32	R 178/179	# 1323	32	R 1858/1859
# 1316	32	R 1844/1845	# 1324	32	R 1860/1861
# 1317	32	R 1846/1847	# 1325	32	R 1862/1863
# 1318	32	R 1848/1849	# 1326	32	R 1864/1865
# 1319	32	R 1850/1851	# 1327	32	R 1866/1867

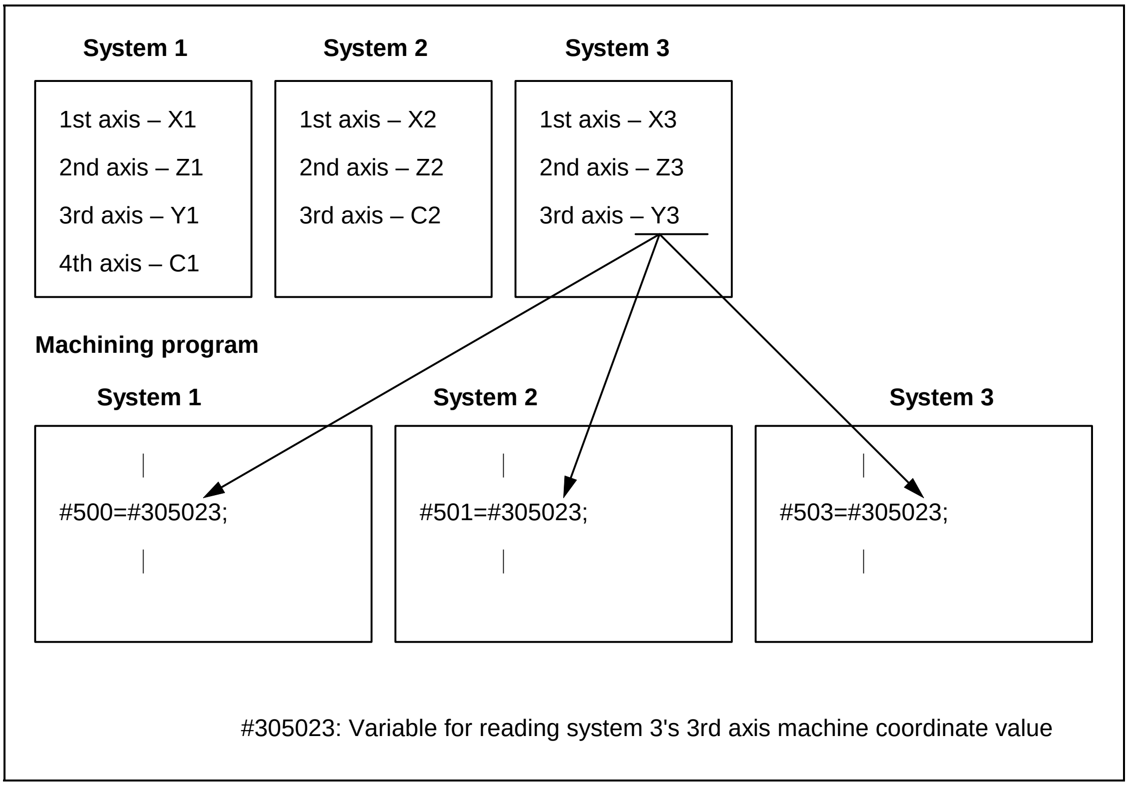
The expanded section is #1316 to #1327.

APPENDIX 9 SYSTEM COMMON POSITION INFORMATION RETRIEVING VARIABLES

Function and purpose

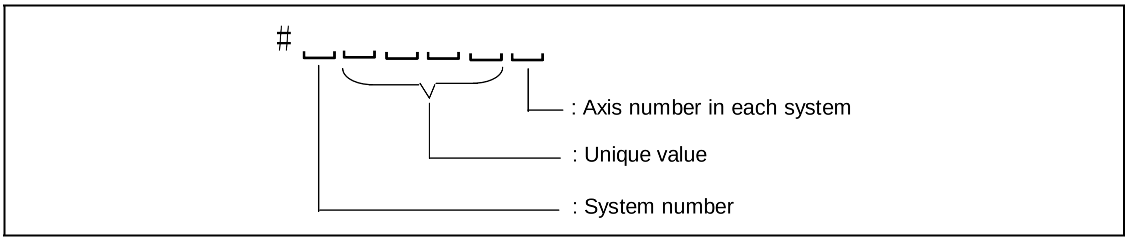
The system common position information retrieving variables can read the machine coordinate values and workpiece coordinate values, etc., from any system.

(Example) To read system 3's 3rd axis machine coordinate value



Variable number

Each variable number is decided following the rules below.



The last digit of the variable number corresponds to the control axis number.
The head digit corresponds to the system number.

APPENDIX 9 SYSTEM COMMON POSITION INFORMATION RETRIEVING VARIABLES

(1) Machine coordinate values

Axis No.	System							
	System 1	System 2	System 3	System 4	System 5	System 6	System 7	System 8
1	#105021	#205021	#305021	#405021	#505021	#605021	#705021	#805021
2	#105022	#205022	#305022	#405022	#505022	#605022	#705022	#805022
3	#105023	#205023	#305023	#405023	--	--	--	--
4	#105024	#205024	#305024	#405024	--	--	--	--
5	#105025	#205025	--	--	--	--	--	--

(2) Workpiece coordinate values

Axis No.	System							
	System 1	System 2	System 3	System 4	System 5	System 6	System 7	System 8
1	#105041	#205041	#305041	#405041	#505041	#605041	#705041	#805041
2	#105042	#205042	#305042	#405042	#505042	#605042	#705042	#805042
3	#105043	#205043	#305043	#405043	--	--	--	--
4	#105044	#205044	#305044	#405044	--	--	--	--
5	#105045	#205045	--	--	--	--	--	--

(3) Tool offset amount

Axis No.	System							
	System 1	System 2	System 3	System 4	System 5	System 6	System 7	System 8
1	#105081	#205081	#305081	#405081	#505081	#605081	#705081	#805081
2	#105082	#205082	#305082	#405082	#505082	#605082	#705082	#805082
3	#105083	#205083	#305083	#405083	--	--	--	--
4	#105084	#205084	#305084	#405084	--	--	--	--
5	#105085	#205085	--	--	--	--	--	--

Precautions and restrictions

(1) These variables read each data following the basic definition axis.

(2) If an undefined axis is designated, the error "P241 No Variable No." will occur.

(3) The coordinate values are read at the commanded timing. If the axis is moving, the machine coordinate values and workpiece coordinate values during the movement will be read. Thus, depending on the conditions such as override, etc., and the system load state, the coordinate values read for each cycle may differ even within the same machining program.

(4) The tool offset amount is temporarily canceled during reference point return and machine coordinate system selection (G53). Thus, the read coordinate values may differ before and after machine coordinate system selection is executed. The coordinate values will also differ before and after workpiece coordinate system setting, local coordinate system setting, external workpiece coordinate system setting, coordinate system setting or tool offset (T function) is executed, so the coordinate values read out before and after the above function is executed will differ.

(5) If the axis coordinate values are read during mirror image, coordinate values read out and allowing for the mirror image must be used.

(6) Even if the inch command or metric command is executed in the machining program, the coordinate value will be read with the original unit system. (When initial inch is invalid, the metric system is applied, so even if inch is commanded with the machining program, the coordinate values will be read out with the metric system. When initial inch is valid, the inch system is applied, so even if metric is commanded with the machining program, the coordinate values will be read out with the inch system.)

(7) Data cannot be written into these variables. If written in, nothing will be executed, and the system will proceed to the next block.

APPENDIX 10 LIST OF ALARMS

These alarms are generated during automatic operation. In the main, they denote errors in the creation of the machining programs and also program errors applying when programs matching the NC specifications have not been prepared.

Error No.	Message	Details	Remedy
P10	No. of simultaneous axes over	• The number of axis addresses commanded in the same block is greater than the number provided for by the specifications.	• Divide the alarm block commands into two. • Check the specifications.
P11	Illegal axis address	• The axis address names in the program commands and the axis address names set in the parameters do not match.	• Correct the axis names in the program.
P20	Division error	• An axis has been commanded for which division cannot be made by the command units.	• Check the program.
P32	Illegal address	• An address not contained in the specifications has been used.	• Check and correct the addresses in the program. • Check the specifications.
P33	Format error	• The command format in the program is not correct.	• Check the program.
P34	Illegal G code	• A G code not contained in the specifications has been commanded.	• Check and correct the G code addresses in the program.
P35	Setting value range over	• The setting range of the addresses has been exceeded.	• Check the program.
P36	Program end error	• "EOR" has been read during memory operation.	• Enter M02 or M30 at the end of the program. • Enter M99 at the end of the subprogram.
P37	Prog. No. and sequence No. zero	• A zero has been designated for the program number or for the sequence number	• The program numbers which can be designated range from 1 to 99999999. • The sequence numbers which can be designated range from 1 to 99999.
P38	No spec: Optional block skip	• A command with /2 to /9 has been issued.	• Check the program. (The command cannot be issued with /2 to /9.)
P39	No specifications	• The command issued is not included in the specifications.	• Check the specifications.
P50	No spec: Inch/mm	• A command for inch/millimeter conversion has been assigned using a G code though there is no such G code specification.	• Check the specifications.
P60	Compensation length over	• The commanded movement distance is too great (it exceeds 2^{31}).	• Check the value of each address in the program.

Error No.	Message	Details	Remedy
P62	No F command	• A cutting feedrate command or thread lead command has not been issued.	• When the power is turned ON, G01 is set as the movement modal command. Assign the feedrate using an F command. • Designate F with a thread lead command.
P70	Arc radius error	• The start and end points of the arc and the arc center are not correct.	• Check the address values designated for the program's start point, end point and the arc center. • Check the plus and minus directions of the address value.
P71	Arc center error	• The center of the arc is not found during R-designated circular interpolation.	• Check the address values in the program.
P72	No spec: Herical cutting	• A helical interpolation command was issued though such specifications do not exist.	• Check the specifications. • Check the arc plane.
P73			
P80			
P90	No spec: Thread cutting	• A thread-cutting command has been assigned though such specifications do not exist.	• Check the specifications.
P91	No spec: Vrbl lead thread (G34)	• A variable thread-cutting command has been issued though such specifications do not exist.	• Check the specifications.
P92	No spec: arc thread cutting	• The arc thread cutting command was issued though such specifications do not exist.	• Check the specifications.
P93	Illegal pitch value	• The thread lead (thread pitch) is not correct when thread cutting is commanded.	• Set the thread lead command properly in the thread cutting command.
P100			
P110			
P112	Plane selected while R compen	• A plane selection command (G17, G18, G19) has been assigned during a tool radius compensation command and nose R compensa-tion command (G41, G42, G46).	• Assign the plane selection command after canceling the tool radius compensation command and nose R compensation command (by issuing the G40 command).
P113	Illegal plane select	• The circular command axis and selected plane do not match.	• Assign the circular command with the proper plane selection.
P120	No spec: Synchronous feed	• A synchronous feed command has been assigned though such specifications do not exist.	• Check the synchronous feed command specifications. • Change the synchronous feed command (G95) into a per-minute feed command (G94). (The F command value must also be changed.)
P121			

Error No.	Message	Details	Remedy
P130	2nd M function code illegal	• The address specified by parameter is other than A, B or C. • The name is duplicated with the axis name.	• Check the parameter's 2nd miscellaneous function address. • Designate an address different from the axis address.
P131	No spec: Cnst perphrl ctrl G96	• A constant surface speed command (G96) has been assigned even though such specifications do not exist.	• Check the specifications. • Change the constant surface speed command (G96) into a speed command (G97).
P133	Illegal P-No.: G96	• An illegal constant surface speed control axis has been designated. • An axis No. that does not exist in the command system was commanded during constant surface speed control.	• Check the parameter and program designation for the constant surface speed control axis.
P140			
P141			
P142			
P150	No spec: Nose R compensation	• The tool radius compensation and nose R compensation (G41, G42, G46) were issued even though the tool radius compensation and nose R compensation specifications are not provided.	• Check the tool radius compensation and nose R compensation specifications.
P151	Radius compen during arc mode	• A compensation command (G40, G41, G42, G46) has been assigned in the circular mode (G02, G03).	• Assign a rapid traverse command (G00) or linear command (G01) in the compensation command block or cancel block. (Set the modal to linear interpolation.)
P152	No intersection	• The intersection point compensation vector is not found when a tool radius compensation or nose R compensation command (G41, G42, G46) has been executed.	• Check the program.
P153	Compensation interference	• An interference error is occurred when a tool radius compensation and nose R compensation command (G41, G42, G46) is executed.	• Check the program.
P154			
P155	Fixed cyc exec during compen	• A fixed cycle command was assigned in the radius compensation mode.	• The radius compensation mode is established when a fixed cycle command is executed and so the radius compensation cancel command (G40) should be assigned.
P156	R compen direction not defined	• When the G46 nose R compensation is started, the movement vector has an undefined compensation direction.	• Change to a movement vector whose compensation direction is defined. • Change to a tool with a different tool nose point number.

Error No.	Message	Details	Remedy
P157	R compen direction changed	• The compensation direction is reversed during G46 nose R compensation.	• Even if the compensation direction is reversed, change to a more suitable G command (G00, G28, G30, G33, G53). • Change to a tool with a different tool nose point number. • Set the G46 no reverse error parameter to ON.
P158	Illegal tip point	• The tool nose point number is illegal (any number except 1 to 8) during G46 nose R compensation.	• Change to the correct tool nose point number.
P159	Cmnd invalid during R compen	• A command disable command has been issued in nose R compensation (G41, G42, or G46) mode.	• Cancel nose R compensation before the block where an error occurs.
P170	No offset number	• There is no compensation number (T○○) command when a compensation command (G41, G42, G46) is assigned. • The compensation number is greater than the number of sets in the specifications.	• Add the compensation number command to the compensation command block. • Check the number of compensation number sets and correct the command so that it has a compensation number within the number of compensation sets.
P171	No spec: G10 option	• A G10 command has been assigned even though such specifications do not exist.	• Check the specifications.
P172	G10 L number error	• The L address command is not correct when the G10 command is assigned.	• Check the G10 command address L number and command the proper number.
P173	G10 P number error	• A compensation number outside the number of sets in the specifications has been assigned for the compensation number command when the G10 command is assigned.	• First check the number of compensation sets and then set the address P designation to within that number.
P180	No fixed cycle	• A fixed cycle (G81 to G89) command has been assigned though such specifications do not exist.	• Check the specifications. • Correct the program.
P181	No spindle command (Tap cycle)	• The spindle speed command has not been assigned when a hole drilling fixed cycle command is assigned.	• Assign the spindle speed command (S) with the G84 or G88 hole drilling fixed cycle command.
P182	Synchronous tap error	• Connecting is not possible with the spindle unit. • A synchronous tap command was issued for a system in the constant surface speed control.	• Check the connection with the spindle unit. • Check whether the spindle encoder is present or not. • Check the program.
P183	No pitch/thread number	• The pitch or thread number command is not present in the tap cycle of the hole drilling fixed cycle command.	• Assign the pitch or thread number command using the F or E command.

APPENDIX 10 LIST OF ALARMS

Error No.	Message	Details	Remedy
P184	Pitch/thread number error	• An incorrect pitch or thread number command has been assigned in the tap cycle of the hole drilling fixed cycle command.	• Check the pitch or thread number.
P190	No spec: Cutting cycle	• A turning cycle command has been assigned though the turning cycle specifications are not provided.	• Check the specifications. • Delete the turning cycle command.
P191	Taper length error	• There is an error in the taper length command when the turning cycle command is assigned.	• Set the setting value of R in the turning cycle command to less than the movement amount of the axis.
P192	Chamfering error	• The chamfering conducted during the thread cutting cycle is illegal.	• Set a chamfering amount which does not protrude from the cycle.
P200	No spec: MRC cycle	• A compound fixed cycle command (G70 to G73) has been assigned even though the compound fixed cycle specifications are not provided.	• Check the specifications.
P201	Program error (MRC)	• One or more of the following commands are present in the subprogram which has been called by the compound fixed cycle: 1. Reference point return commands (G27, G28, G30), 2. Thread cutting (G33), 3. Fixed cycle, 4. Skip function (G31). • The first movement block of the finish shape program in compound fixed cycle contains a circular command.	• Delete the following G codes from the subprogram which has been called by the compound fixed cycle (G70 to G76) : G27, G28, G30, G31. G33, G code in fixed cycle. • Delete G2 and G3 from the first movement block of the finish shape program in compound fixed cycle.
P202	Block over (MRC)	• The number of blocks in the shape program which has been called by the compound fixed cycle exceeds 50.	• Make the number of blocks in the shape program which has been called by the compound fixed cycle (G70 to G73) less than 50.
P203	D cmnd figure error (MRC)	• The shape program of the compound fixed cycle (G70 to G73) does not give the shape which can be cut properly.	• Check the shape program in the compound fixed cycle (G70 to G73).
P204	E cmnd fixed cycle error (MRC)	• The command value in the compound fixed cycle (G70 to G76) is not correct.	• Check the command value in the compound fixed cycle (G70 to G76).
P210	No spec: Pattern cycle	• A compound fixed cycle command has been assigned though the compound fixed cycle (G74 to G76) specifications are not provided.	• Check the specifications.
P230	Sub-program nesting over	• The number of times subprograms have been called in sequence from subprograms has exceeded 8.	• Check the number of subprogram calls and correct the program so that it does not exceed 8.

Error No.	Message	Details	Remedy
P231	No sequence No.	• The sequence number commanded by GOTO has not been set when calling out a subprogram or when returning to the main program from a subprogram.	• Enter the sequence number in the appropriate block.
P232	No program No.	• The subprogram has not been registered when it is called.	• Register the subprogram.
P240	No spec: Variable command	• A variable command has been assigned though such specifications (#○○) do not exist.	• Check the specifications. • Delete the variable command.
P241	No variable No.	• The commanded variable number is higher than the variable numbers in the specifications.	• Check the specifications. • Check the program variable numbers.
P242	" = " not defined at vrble set	• " = " has not been commanded when a variable is defined.	• Set " = " into the program variable definition.
P243	Can't use variables	• Even though the hypothetical coordinates were set, the basic coordinate system position information was read during end point synchronization.	• Review the program.
P250			
P251			
P252			
P260			
P270	No spec: User macro	• A macro specification command has been assigned though such specifications do not exist.	• Check the specifications. • Delete the macro command.
P271	No spec: Macro interrupt	• A macro interrupt command has been issued though it is not included in the specifications.	• Check the specifications.
P272	NC and macro texts in a block	• An NC statement and a macro statement exist in the same block.	• Check the program, and program so that the NC statement and macro statement are in separate blocks.
P273	Macro call nesting over	• The maximum number of macro call nesting levels have been exceeded.	• Check the program and correct so that the macro calls do not exceed the number of levels provided for by the specifications.
P275	Macro argument over	• There are too many argument sets in macro call argument type II.	• Check the program.
P276	Illegal G67 command	• A G67 command has been assigned though it is not during the G66 command modal.	• Check the program. • The G67 command serves to cancel the call and so the G66 command is assigned before this command.
P277	Macro alarm message	• An alarm command has been issued in user macro.	• Check the user macro program.
P280	Brackets [ ] nesting over	• More than 5 bracket " [" or "] " parentheses have been used in a block.	• Check the program and correct so that the number of " [" or "] " parentheses does not exceed 5 brackets.

APPENDIX 10 LIST OF ALARMS

Error No.	Message	Details	Remedy
P281	Brackets [ ] not paired	• The number of the " [" or "] " parentheses commanded in a block does not match.	• Check the program and correct so that the numbers of " [" and "] " parentheses are paired off properly.
P282	Calculation impossible	• An operation formula is not correct.	• Check the program and correct the operation formula.
P283	Divided by zero	• The division denominator is zero.	• Check the program and correct so that the denominator for division in the operation formula is not zero.
P284	Integer value overflow	• In the process of operation the integer value has exceeded -2^{31} ($2^{31}-1$).	• Check the operation formula in the program and correct so that the value of the integers after the operation does not exceed -2^{31}.
P285	Float value overflow	• There is a variable data overflow.	• Check the variable data in the program.
P290	IF sentence error	• There is an error in the IF [<conditional formula>] GOTO□ statement.	• Check the program.
P291	WHILE sentence error	• There is an error in the WHILE [< conditional formula>] DO□ to END□ statement.	• Check the program.
P292	SETVN sentence error	• The variable name setting or SETVN□ statement is incorrect.	• Check the program. • Make sure that the variable name of the SETVN statement has 7 or fewer characters.
P293	DO-END nesting over	• The numbers of □'s in the DO□ and END□ of the WHILE [<conditional formula>] DO□ to END□ statement (nesting levels) has been exceeded 27.	• Check the program and correct so that the nesting levels of DO to END statements does not exceed 27.
P294	DO and END not paired	• The DO and END are not paired off properly.	• Check the program and correct so that the DO and END are paired off properly.
P300	Variable name illegal	• A variable name has not been commanded properly.	• Correct the program so that the variable name is correct. • Check the variable names in the program and correct them.
P301	Variable name duplicated	• A variable name has been duplicated.	• Correct the program so that the variable names are not duplicated.
P340	Coordinate parameter illegal	• Even though there is a tool offset amount, the coordinate system setting command (G92) was issued immediately after the double-turret mirror image mode change command was issued.	• Review the machining program. • Turn the setup parameter #8053 CONTROL BIT6 ON.
P370	No spec: Mirr image dubl turet	• An opposite tool rest mirror image command (G68) has been assigned though the double-turret mirror image specifications are not provided.	• Check the specifications.

APPENDIX 10 LIST OF ALARMS

Error No.	Message	Details	Remedy
P380	No spec: Corner R/C	• A corner chamfering (C) or corner rounding (R) command has been assigned though the corner chamfering and corner rounding I and II specifications do not exist.	• Check the specifications. • Remove "corner R" and "corner C" from the program.
P381	No spec: Arc R/C	• A corner chamfering (C) or corner rounding (R) command has been assigned in a circular interpolation block though the corner chamfering and corner rounding II specifications do not exist.	• Check the specifications.
P382	No corner movement	• There is no movement command in the block following the corner chamfering/rounding.	• Assign the block following the corner R/C command in the block with movement.
P383	Corner movement short	• The movement distance is shorter than the corner R/C command when such a command is assigned.	• Since the movement distance is shorter than the corner R/C, the corner R/C should be reduced to less than the movement distance.
P384	Corner next movement short	• With a corner R/C command, the movement distance in the following block is shorter than the corner R/C.	• Since the movement distance in the following block is shorter than the corner R/C, the corner R/C should be made less than the movement distance.
P385	Corner during G0/G33	• G0 or G33 is contained in the block following the corner R/C.	• Check the program.
P390	No spec: Geometric	• A geometric command has been issued while geometric I is not included in the specifications.	• Check the specifications.
P391	No spec: Geometric arc	• A geometric command has been issued while geometric II is not included in the specifications.	• Check the specifications.
P392	Angle<1 degree (GEOMT)	• The difference in the angle between geometric lines is less than 1 degree.	• Correct the geometric angle.
P393	Inc value in 2nd block (GEOMT)	• The commands in the second geometric block have been assigned as incremental values.	• Assign the commands in the second geometric block as absolute values.
P394	No linear move command (GEOMT)	• There is no linear command in the second geometric block.	• Assign the linear command (G01) in the second block.
P395	Illegal address (GEOMT)	• The geometric format is incorrect.	• Check the program.
P396	Plane selected in GEOMT ctrl	• A plane selection command was assigned in the geometric command.	• Select the plane before issuing the geometric command.
P397	Arc error (GEOMT)	• The arc end point did not contact or intersect with the next block's start point during geometric IB, II.	• Check the previous and next commands including the geometric arc command.
P398	No spec: Geometric 1B	• The geometric command was issued when the geometric IB specifications were not available.	• Check the specifications.

APPENDIX 10 LIST OF ALARMS

Error No.	Message	Details	Remedy
P399	Direction error (GEOMT)	• The monotone is not incremented or decremented in geometric II.	• Check the program so that the monotone is incremented or decremented.
P410	No spec: Address convertor	• The specifications for converting absolute/incremental axis addresses do not exist.	• Check the specifications.
P420	No spec: Parameter input	• A parameter input command has been assigned though such specifications do not exist.	• Check the specifications.
P421	Parameter input error	• The commanded parameter number and setting data is illegal. • An illegal G command address has been assigned in the parameter input mode. • A parameter input command has been assigned during a fixed cycle modal or nose R compensation.	• Check the program.
P430	Zero return not completed	• A movement command except reference point return has been assigned for an axis which has not returned to the reference point.	• Execute the reference point return manually.
P431	No spec: 2, 3, 4th ref-point ret	• A 2nd, 3rd or 4th reference point return command has been assigned though such specifications do not exist.	• Check the specifications.
P432	No spec: G29	• A start position return command (G29) has been assigned though such specifications do not exist.	• Check the specifications.
P433	No spec: G27	• A zero point check command (G27) has been assigned though such specifications do not exist.	• Check the specifications.
P434	Compare error	• There is an axis which does not return to the zero point position when the zero point check command (G27) is executed.	• Check the program.
P435	G27 and M commands in a block	• An M independent command has been assigned simultaneously in the G27 command block.	• An M independent command cannot be assigned in the G27 command block and so the G27 command and M independent command should be divided into separate blocks.
P436	G29 and M commands in a block	• An M independent command has been assigned simultaneously in the G29 command block.	• An M independent command cannot be assigned in the G29 command block and so the G29 command and M independent command should be divided into separate blocks.
P450	No spec: Chuck barrier	• A chuck barrier valid command (G22) has been assigned though the chuck barrier specifications are not provided.	• Check the specifications.
P459	External I/O timeout	• A timeout error occurred in the external input/output device.	• Check the external input/output device. • Check the timeout time parameter.

APPENDIX 10 LIST OF ALARMS

Error No.	Message	Details	Remedy
P460	Tape I/O error	• An error has occurred in the tape reader or in the printer during macro printing.	• Check the power and cables for the connected units. • Check the Input/output parameters.
P461	No file data	• The machining program file cannot be read or the file cannot be found.	• Review the machining program. • The program saved in the memory may be damaged. Output all of the necessary data, such as the machining programs, tool data and workpiece offset data, to an external device, and then format.
P462	Computer link error	• A communication error occurred in the computer link.	• Reset the system.
P480	No spec: Milling	• No milling function specifications are found.	• Check the specifications.
P481	Illegal G code (mill)	• An invalid G code was commanded in the milling mode.	• Remove the illegal G command.
P482	Illegal axis (mill)	• A rotation axis command was issued in the milling mode. • Alternatively, milling was performed though an illegal value was set for the milling axis number.	• Delete the rotation axis command. • Check the milling axis number.
P483	Cancel shift X (mill)	• The milling mode was turned ON before the tool offset operation was completed or in the offset temporary cancel state.	• Review the program. • Complete tool offset before entering the milling mode.
P484	ZRN not completed (mill)	• A movement command was issued in milling mode to the axis that had not completed return to the reference point.	• Manually return the axis to the reference point.
P485	Illegal modal (mill)	• The system entered in milling mode during nose R compensation or constant surface speed control. • A T command was issued in milling mode. • The milling mode was changed to the turning mode during tool radius compensation.	• Before issuing G12.1, issue G40 (nose R compensation cancel) or G97 (constant surface speed cancel). • Before issuing G12.1, issue a T command. • Before issuing G13.1, issue G40 (tool radius compensation cancel).
P500	No spec: Cross machining (G110)	• A cross machining command (G110) was issued though it was not included in the specifications.	• Check the specifications.
P501	Cross (G110) impossible	• A cross machining command (G110) was issued in nose R, milling, balance cut, fixed cycle, or double-turret mirror image mode.	• Check the program.
P502	Illegal G110 $No.	• An illegal axis system number was commanded.	• Correct the program address.

APPENDIX 10 LIST OF ALARMS

Error No.	Message	Details	Remedy
P503	Illegal G110 axis	• The commanded axis was not found. • Too many axes were commanded. • The commanded axis cannot be controlled by the commanded system.	• Correct the program address.
P510	Illegal G128/G129 axis	• During the axis movement synchronous superimposition command (G128, G129), the synchronous superimposing axis is not between the start position and the end position of synchronous superimposition. • A rotation axis was commanded for the synchronous superimposing axis or master axis.	• Check the program.
P511	Illegal $-command	• The axis movement synchronous superimposition command (G128, G129) was issued in a system that does not contain a synchronous superimposing axis.	• Check the program.
P520	No TGSET value	• A position control command (G132) or position control variable skip command (G133) was issued before a set number specification (TGSET[ ]) command.	• Check the program.
P521	Illegal G code (G130)	• One of G130 to G133 was issued during nose R compensation (G41, G42, or G46), thread cutting (G33), fixed cycle (G70 to G79, or G81 to G89), or milling (G12.1) modal.	• Check the program.
P600	No spec: Auto TLM	• An automatic tool length measurement command (G37) has been assigned through such specifications do not exist.	• Check the specifications.
P601	No spec: Skip	• A skip command (G31/G160) has been assigned though such specifications do not exist.	• Check the specifications.
P602	No spec: Multi skip	• A multiple skip command (G31.1, G31.2, G31.3) was assigned though no such command exists in the specifications.	• Check the specifications.
P603	Skip speed 0	• The skip speed is zero.	• Command the skip speed.
P604	G37 illegal axis	• The axis has not been commanded in the automatic tool length measurement block or, alternatively, two or more axes have been commanded.	• Command only one axis.
P605	H and G37 commands in a block	• The T code is in the same block as the automatic tool length measurement command.	• Assign the T command before the block which contains the automatic tool length measurement command.

Error No.	Message	Details	Remedy
P606	H command not found before G37	• The T code has still not been commanded for automatic tool length measurement.	• Assign the T command before the block which contains the automatic tool length measurement command.
P607	Signal turned illegally by G37	• The measurement position arrival signal has been set ON before the area commanded by the parameter deceleration area "d" or D command. • Alternatively, the signal was not set ON until the end.	• Check the program.
P608	Skip during radius compen	• The skip command was assigned during a radius compensation or nose R compensation command.	• Assign the radius compensation cancel or nose R compensation cancel command (G40) or remove the skip command.
P609	Illegal skip axis	• An axis was not commanded in the G160 block, or two or more axes were commanded.	• Command only one axis.
P700	No B, N number	• When using the end point synchronization function, the designated block's identification number was not found at the end point block.	• Check the program.
P710	Inclined ax ctrl mode illegal	• The G170/G171 command was issued in the milling mode, nose R mode, mirror image, compound fixed cycle or constant surface speed control mode.	• Check the program.
P711	Coordinate conver prohibited	• The hypothetical coordinate setting was commanded in a mode in which the hypothetical coordinates cannot be set. • Inclined coordinate rotation was commanded in a mode in which the inclined coordinate rotation cannot be commanded.	The hypothetical coordinate setting and the hypothetical coordinate cancel command were executed in the following modes. Cancel the mode. • Mirror image is applied on the imaginary coordinate target axis. • The fixed cycle mode is active. • The milling mode is active. • The nose R compensation mode is active. The inclined coordinate rotation command and inclined coordinate rotation cancel command were executed in the following modes. Cancel the mode. • Mirror image is applied on the inclined coordinate rotation target axis. • The fixed cycle mode is active. • The milling mode is active. • The nose R compensation mode is active.

Error No.	Message	Details	Remedy
P712	Invalid axis of coord conver	• An illegal G code was commanded during hypothetical coordinate setting. • An illegal command was issued during the inclined coordinate rotation mode.	One of the following commands was executed during the hypothetical coordinate setting. • Automatic reference point return (G28/G29) • 2nd, 3rd, 4th reference point return (G30) • Reference point check (G27) • Random axis exchange (G140) A command that changes the hypothetical coordinate target axis to a non-control axis was executed. • Current value sampling (G159) • Torque skip (G160) • Collision detection (G161) One of the following commands was executed during the inclined coordinate rotation mode. • Milling command (G12.1) • Thread cutting command • Hobbing command • Automatic reference point return (G28/G29) • 2nd, 3rd, 4th reference point return (G30) • Reference point check (G27) • Tool position return command • Mirror image command • Compound type turning cycle (G70 to G76) • Skip command (G31) • Random axis exchange (G140) A command that changes the inclined coordinate rotation target axis to a non-control axis was executed. • Current value sampling (G159) • Torque skip (G160) • Collision detection (G161)
P718	TDO illegal startup	• An arc command was issued for the tool axis direction tool length offset startup command.	• An arc command was issued for the tool axis direction tool length offset startup command. Change the arc command to a linear command.
P719	TDO ctrl axis illegal	• An illegal value was set for the tool axis direction tool length offset target axis.	• Check the plane selection parameters #8001 (I axis) and #8003 (K axis). • Confirm that the tool axis direction tool length offset parameter #1182 (bt_axname) is a rotary axis.

Error No.	Message	Details	Remedy
P720	TDO command prohibited	• Tool axis direction tool length offset was commanded in an illegal mode.	The tool axis direction tool length offset start command or cancel command was executed in one of the following modes. Cancel the mode. • Mirror image is applied on the tool axis direction tool length offset target axis. • The milling mode is active. • The nose R compensation mode is active. • The thread cutting mode is active. • The end point synchronization mode is active. • The tool axis direction tool length offset mode is active. • The tool offset amount is not commanded. Command G174 after the tool offset command (T command). • The tool-spindle synchronization II (hobbing) mode is active.
P721	Illegal command in TDO mode	• An illegal command was executed during the tool axis direction tool length offset modal.	One of the following commands was executed during the tool axis direction tool length offset modal. • Milling command (G12.1) • Thread cutting command • Automatic reference point return command (G28/G29) • 2nd, 3rd, 4th reference point return command (G30) • Reference point check command (G27) • Mirror image command • Compound type turning cycle (G70 to G76) • Skip command (G31) • Inclined coordinate rotation command (G173) • Automatic tool length measurement (G37) • Double-turret mirror image command • T command double-turret mirror image command • Tool-spindle synchronization II (hobbing) command
P980	Illegal HSPRGM call	• During subprogram call involving nesting, the execution format data machine maker macro and EIA machine maker macro were called out alternately.	• Review the machine maker macros.

Error No.	Message	Details	Remedy
P981	Illegal HSPRGM program	• A G code incompatible with execution type data was commanded. • The return destination sequence No. was commanded with the program return command M99 called out from the execution type data. • Execution type data was executed when the G code series was set to a value other than 1.	• Review the machine maker macros. • Review the G code series setting. (The execution type data can be used only with the G code series 1.)
P982	HSPRGM conversion incomplete	• The machine maker macro conversion process has not been completed.	• Start the EIA → execution type data conversion process.
P990	PREPRO error	• Combining commands that required pre-reading (nose R compensation, corner chamfering/corner rounding (R), geometric I, geometric IB, and compound fixed cycle commands) resulted in eight or more pre-read blocks.	• Reduce the number of commands that require pre-reading or delete such commands.

Revision History

Date of revision	Manual No.	Revision details
Oct. 2000	BNP-B2232A	First edition created.
Jan. 2004	BNP-B2232C	(1) The following explanations were added. • Circular thread cutting • Synchronous tapping cycle; return override, spindle zero point return method • System variables: #5081 to 5083 tool compensation amount • Axis name change • Inclined coordinate rotation control • G0/G53 speed designated positioning • Tool-spindle synchronization I, II E command range expansion (10 to 999) (2) Other mistakes were corrected.

Notice

Every effort has been made to keep up with software and hardware revisions in the contents described in this manual. However, please understand that in some unavoidable cases simultaneous revision is not possible.

Please contact your Mitsubishi Electric dealer with any questions or comments regarding the use of this product.

Duplication Prohibited

MITSUBISHI ELECTRIC CORPORATION
HEAD OFFICE : MITSUBISHI DENKI BLDG., 2-2-3, MARUNOUCHI, CHIYODA-KU, TOKYO 100-8310, JAPAN

MODEL	M600L Series
MODEL CODE	008-021
Manual No.	BNP-B2232C(ENG)

Specifications subject to change without notice.
(0401)MEE
Printed in Japan on recycled paper.