Changes for the Better

MITSUBISHI ELECTRIC

MITSUBISHI CNC

Programming Manual Lathe System (2/2)

M800/M80/C80 Series

IB-1501276(ENG)-D

Introduction

This manual describes how to carry out MITSUBISHI CNC programming.
Supported models are as follows:

Supported models	Abbreviations in this manual
M800W series	M800 series, M800, M8
M800S series	
M80W series	M80 series, M80, M8
M80 series	
C80 series	C80

This manual describes programming, therefore, read this manual thoroughly before using this NC system.
To ensure safe use of this NC system, thoroughly study the "Precautions for Safety" on the following page before using this NC system.
Be sure to always keep this manual on hand so that users can refer to it at any time.

Details described in this manual

The description concerning "Signals" in the main text refers to information transmission between a machine and PLC or between NC and PLC.
The method for controlling the signals (ON/OFF) differs depending on the machine. Refer to the manual issued by the machine tool builder (MTB).
Some parameters can be used by end-users and some parameters are set by the MTB according to the specifications.
End-users may not be able to set or change some of the parameters described as "... can be set with the parameter #XXXX" in the main text. Confirm the specifications for your machine with the manual issued by the MTB.

⚠ CAUTION

- ⚠ For items described as "Restrictions" or "Usable State" in this manual, the instruction manual issued by the machine tool builder (MTB) takes precedence over this manual.
- ⚠ Items not described in this manual must be interpreted as "not possible".
- ⚠ This manual is written on the assumption that all the applicable functions are included. Some of them, however, may not be available for your NC system. Refer to the specifications issued by the machine tool builder before use.
- ⚠ Refer to the Instruction Manual issued by the MTB for details regarding each machine tool.
- ⚠ Some screens and functions may differ depending on the NC system (or its version), and some functions may not be available. Please confirm the specifications before use.

General precautions
(1) Refer to the following documents for details handling

MITSUBISHI CNC M800/M80 Series Instruction Manual IB-1501274
MITSUBISHI CNC C80 Series Instruction Manual IB-1501453

(2) Refer to the following documents for details on programming

MITSUBISHI CNC M800/M80/C80 Series Programming Manual

Lathe System (1/2) IB-1501275
Lathe System (2/2) IB-1501276
Machining Center System (1/2) IB-1501277
Machining Center System (2/2) IB-1501278

Precautions for Safety

Always read the specifications issued by the machine tool builder, this manual, related manuals and attached documents before installation, operation, programming, maintenance or inspection to ensure correct use.
Understand this numerical controller, 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.
The following sings indicate prohibition and compulsory.

⃠	This sign indicates prohibited behavior (must not do). For example, indicates "Keep fire away".
●	This sign indicated a thing that is pompously (must do). For example, indicates "it must be grounded".

The meaning of each pictorial sing is as follows.

CAUTION	CAUTION rotated object	CAUTION HOT	Danger Electric shock risk	Danger explosive
Prohibited	Disassembly is prohibited	KEEP FIRE AWAY	General instruction	Earth ground

For Safe Use

Mitsubishi CNC is designed and manufactured solely for applications to machine tools to be used for industrial purposes.
Do not use this product in any applications other than those specified above, especially those which are substantially influential on the public interest or which are expected to have significant influence on human lives or properties.

⚠ DANGER

Not applicable in this manual.

⚠ WARNING

1. Items related to operation

⚠ If the operation start position is set in a block which is in the middle of the program and the program is started, the program before the set block is not executed. Please confirm that G and F modal and coordinate values are appropriate. If there are coordinate system shift commands or M, S, T and B commands before the block set as the start position, carry out the required commands using the MDI, etc. If the program is run from the set block without carrying out these operations, there is a danger of interference with the machine or of machine operation at an unexpected speed, which may result in breakage of tools or machine tool or may cause damage to the operators.

⚠ Under the constant surface speed control (during G96 modal), if the axis targeted for the constant surface speed control (normally X axis for a lathe) moves toward the spindle center, the spindle rotation speed will increase and may exceed the allowable speed of the workpiece or chuck, etc. In this case, the workpiece, etc. may jump out during machining, which may result in breakage of tools or machine tool or may cause damage to the operators.

⚠ 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 tool builder takes precedence over this manual.

⚠ Items not described in this manual must be interpreted as "not possible".

⚠ This manual is written on the assumption that all the applicable functions are included. Some of them, however, may not be available for your NC system.
Refer to the specifications issued by the machine tool builder before use.

⚠ Refer to the Instruction Manual issued by each machine tool builder for details on each machine tool.

⚠ Some screens and functions may differ depending on the NC system (or its version), and some functions may not be possible. Please confirm the specifications before use.

2. Items related to operation

⚠ Before starting actual machining, always carry out graphic check, dry run operation and single block operation to check the machining program, tool offset amount, workpiece compensation amount and etc.

⚠ If the workpiece coordinate system offset amount is changed during single block stop, the new setting will be valid from the next block.

⚠ Turn the mirror image ON and OFF at the mirror image center.

⚠ If the tool offset amount is changed during automatic operation (including during single block stop), it will be validated from the next block or blocks onwards.

⚠ Do not make the synchronized spindle rotation command OFF with one workpiece chucked by the reference spindle and synchronized spindle during the spindle synchronization.
Failure to observe this may cause the synchronized spindle stop, and hazardous situation.

3. Items related to programming

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

⚠ ";" "EOB" and "%" "EOR" are expressions used for explanation. The actual codes are: For ISO: "CR, LF", or "LF" and "%". Programs created on the Edit screen are stored in the NC memory in a "CR, LF" format, but programs created with external devices such as the FLD or RS-232C may be stored in an "LF" format.
The actual codes for EIA are: "EOB (End of Block)" and "EOR (End of Record)".

⚠ 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.

⃠ Do not change fixed cycle programs without the prior approval of the machine tool builder.

⚠ When programming the multi-part system, take special care to the movements of the programs for other part systems.

Disposal

(Note) This symbol mark is for EU countries only.
This symbol mark is according to the directive 2006/66/EC Article 20 Information for end-users and Annex II.

Your MITSUBISHI ELECTRIC product is designed and manufactured with high quality materials and components which can be recycled and/or reused.
This symbol means that batteries and accumulators, at their end-of-life, should be disposed of separately from your household waste.
If a chemical symbol is printed beneath the symbol shown above, this chemical symbol means that the battery or accumulator contains a heavy metal at a certain concentration. This will be indicated as follows:
Hg: mercury (0,0005%), Cd: cadmium (0,002%), Pb: lead (0,004%)
In the European Union there are separate collection systems for used batteries and accumulators.
Please, dispose of batteries and accumulators correctly at your local community waste collection/ recycling centre.

Please, help us to conserve the environment we live in!

Trademarks

MELDAS, MELSEC, EZSocket, EZMotion, iQ Platform, MELSEC iQ-R, MELSOFT, GOT, CC-Link, CC-Link/LT and CC-Link IE are either trademarks or registered trademarks of Mitsubishi Electric Corporation in Japan and/or other countries.

Ethernet is a registered trademark of Xerox Corporation in the United States and/or other countries.
Microsoft®, Windows®, SQL Server® and Access® are either trademarks or registered trademarks of Microsoft Corporation in the United States and/or other countries.
SD logo and SDHC logo are either registered trademarks or trademarks of LLC.
UNIX is a registered trademark of The Open Group in the United States and/or other countries.
Intel® and Pentium® are either trademarks or registered trademarks of Intel Corporation in the United States and/or other countries.
MODBUS® is either a trademark or a registered trademark of Schneider Electric USA, Inc. or the affiliated companies in Japan and/or other countries.
EtherNet/IP is a trademark of Open DeviceNet Vendor Association,Inc.
PROFIBUS-DP is a trademark of Profibus International.
Oracle® is a registered trademark of Oracle Corporation, the subsidiaries, or the affiliated companies in the United States and /or other countries.

Other company and product names that appear in this manual are trademarks or registered trademarks of the respective companies.

本製品の取扱いについて

(日本語 /Japanese)
本製品は工業用 (クラス A) 電磁環境適合機器です。販売者あるいは使用者はこの点に注意し、住商業環境以外での使用をお願いいたします。

Handling of our product

(English)
This is a class A product. In a domestic environment this product may cause radio interference in which case the user may be required to take adequate measures.

본 제품의 취급에 대해서

(한국어 /Korean)
이 기기는 업무용 (A 급) 전자파적합기기로서 판매자 또는 사용자는 이 점을 주의하시기 바라며 가정외의 지역에서 사용하는 것을 목적으로 합니다 .

Contents

Chapter 1 - 14 : Refer to Programming Manual (Lathe System) (1/2)
Chapter 15 and later : Refer to Programming Manual (Lathe System) (2/2)

15

Program Support Functions

15.1 Corner Chamfering I /Corner Rounding I

Function and purpose

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.
By setting the parameters, "I_", "K_", "C_" can be set for chamfering instead of ",C_", and "R_" can be set for rounding instead of ",R_". (This depends on the MTB specifications.)

15.1.1 Corner Chamfering I ; G01 X_ Z_ ,C_/I_/K_/C_

Function and purpose

This chamfers a corner by connecting the both side of the hypothetical corner which would appear as if chamfering is not performed, by the amount commanded by ",C_" (or "I_","K_","C_").

Command format

```
N100 G01 X__ Z__ C__ (or I__ / K__ / C__) ;
N200 G01  X__  Z__  ;
```

,C / I / K / C	Length up to chamfering starting point or end point from hypothetical corner

Corner chamfering is performed at the point where N100 and N200 intersect.

Detailed description

(1) The start point of the block following the corner chamfering is the hypothetical corner intersection point.

(2) If the parameter "#1272 ext08/bit6" is "0" in the MTB specifications, the ",C" command will be interpreted as a C command if there is no "," (comma).

(3) If there are multiple or duplicate corner chamfering commands in a same block, the last command will be valid.

(4) When both corner chamfering and corner rounding are commanded in the same block, the latter command will be valid.

(5) Tool compensation is calculated for the shape which has already been subjected to corner chamfering.

(6) When the block following a command with corner chamfering does not contain a linear command, a corner chamfering/corner rounding II command will be executed.

(7) Program error (P383) will occur when the movement amount in the corner chamfering block is less than the chamfering amount.

(8) Program error (P384) will occur when the movement amount in the block following the corner chamfering block is less than the chamfering amount.

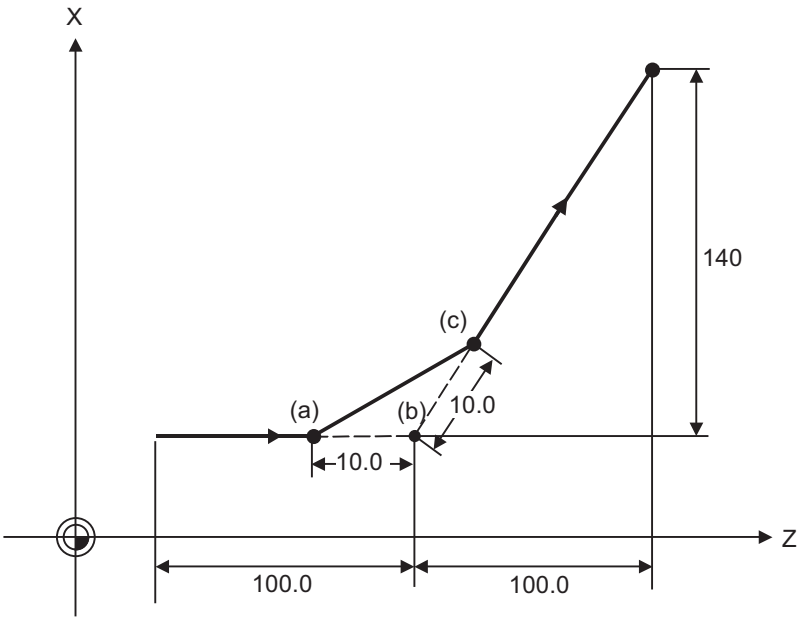
(9) Program error (P382) will occur when a movement command is not issued in the block following the corner chamfering I command.

(10) If "C" is used as the axis name or the 2nd miscellaneous function, corner chamfering cannot be commanded with "C".

(11) Corner chamfering cannot be commanded with "I" or "K" in a circular command block. "I" and "K" are the circular center commands.

Program example

G01 W100. ,C10. F100 ;
U280 W100. ;



(a) Chamfering start point (b) Hypothetical corner intersection point (c) Chamfering end point

Precautions

(1) Corner chamfering and corner rounding can be commanded with "I", "K", "R" only when the 1st block of the corner chamfering/corner rounding command is linear.

(2) Corner chamfering with "I", "K", and corner rounding with "R" can be commanded when the 1st block of the corner chamfering/corner rounding command is linear and the 2nd block is circular. "I", "K" in the 2nd block are the circular center commands.
N100 G01 Xx Zz Ii ; ..... Ii corner chamfering length
N200 G02 Xx Zz Ii Kk ; ..... Ii, Kk circular center command

(3) If ",C_", ",R_" or "I_", "K_", "C_", "R_" are commanded in a same block, ",C_", ",R_" will have the priority.

15.1.2 Corner Rounding I ; G01 X_ Z_ ,R_/R_

Function and purpose

This performs a corner rounding to the both side of the hypothetical corner which would appear as if chamfering is not performed, at the radius of the circular commanded with ",R_" (or "R_").

Command format

```
N100 G01 X__ Z__ ,R__ (or R__) ;
N200 G01  X__ Z__ ;
```

,R / R	Arc radius of corner rounding

Corner rounding is performed at the point where N100 and N200 intersect.

Detailed description

(1) The start point of the block following the corner rounding is the hypothetical corner intersection point.

(2) If the parameter "#1272 ext08/bit6" is "0" in the MTB specifications, the ",R" command will be interpreted as an R command if there is no "," (comma).

(3) When both corner chamfering and corner rounding are commanded in the same block, the latter command will be valid.

(4) Tool compensation is calculated for the shape which has already been subjected to corner rounding.

(5) When the block following a command with corner rounding does not contain a linear command, a corner chamfering/corner rounding II command will be executed.

(6) Program error (P383) will occur when the movement amount in the corner rounding block is less than the R value.

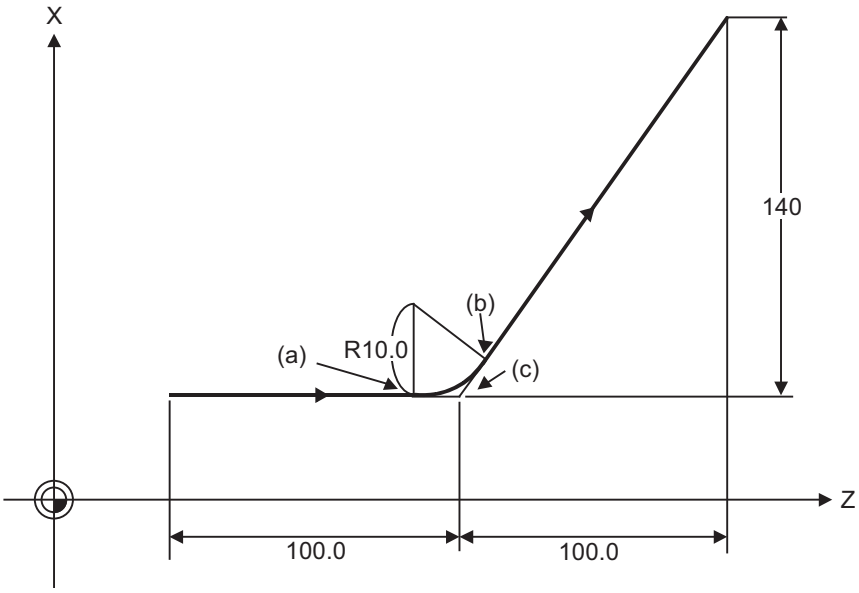
(7) Program error (P384) will occur when the movement amount is less than the R value in the block following the corner rounding.

(8) Program error (P382) will occur if a movement command is not issued in the block following the corner rounding.

(9) Corner rounding cannot be commanded with "R" in a circular command block. "R" is the circular radius command.

Program example

```
G01 W100. ,R10. F100 ;
U280 W100. ;
```



(a) Corner rounding start point (b) Corner rounding end point (c) Hypothetical corner intersection point

Precautions

(1) Corner chamfering and corner rounding can be commanded with "I", "K", "R" only when the 1st block of the corner chamfering/corner rounding command is linear.

(2) Corner chamfering with "I", "K", and corner rounding with "R" can be commanded when the 1st block of the corner chamfering/corner rounding command is linear and the 2nd block is circular. "I", "K" in the 2nd block are the circular center commands.
N100 G01 Xx Zz Ii ; ..... Ii corner chamfering length
N200 G02 Xx Zz Ii Kk ; ..... Ii, Kk circular center command

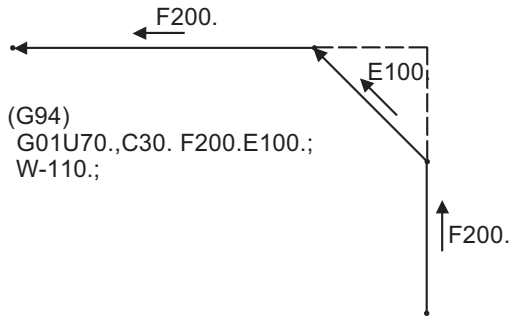
(3) If ",C_", ",R_" or "I_", "K_", "C_", "R_" are commanded in a same block, ",C_", ",R_" will have the priority.

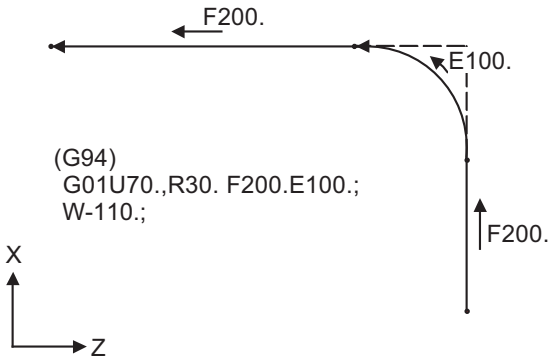
15.1.3 Corner Chamfering Expansion/Corner Rounding Expansion

Function and purpose

Using an E command, the feedrate can be designated for the corner chamfering and corner rounding section. In this way, the corner section can be cut into a correct shape.

Example

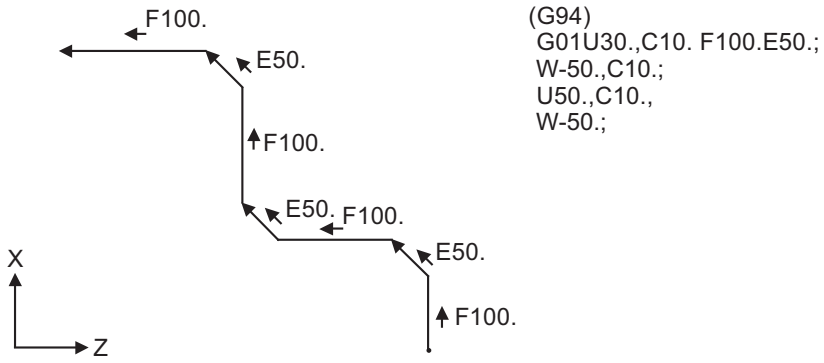




Detailed description

(1) The E command is modal. It is also valid for the feed in the next corner chamfering/corner rounding section.

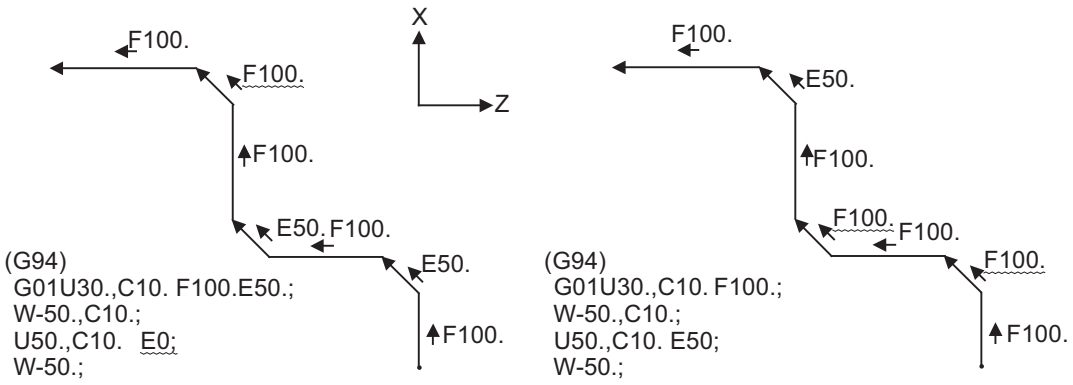
Example



(2) E command modal has separate asynchronous feedrate modal and synchronous feedrate modal functions. Which one is validated depends on the asynchronous/synchronous mode (G94/G95).

(3) When the E command is 0, or when there has not been an E command up to now, the corner chamfering/corner rounding section feedrate will be the same as the F command feedrate.

Example



(4) E command modal is not cleared even if the reset button is pressed.
It is cleared when the power is turned OFF. (In the same manner as F commands.)

(5) All E commands except those shown below are at the corner chamfering/corner rounding section feedrate.
- E commands during thread cutting modal
- E commands during thread cutting cycle modal

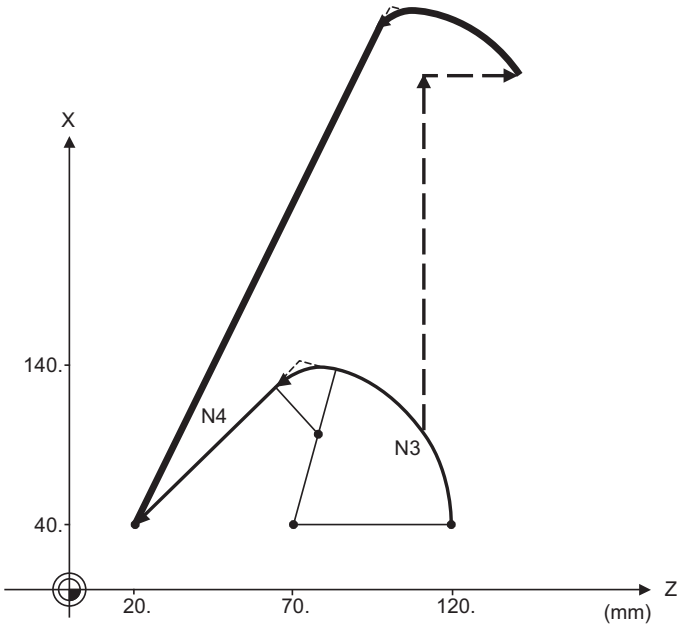
15.1.4 Interrupt during Corner Chamfering/Interrupt during Corner Rounding

Detailed description

(1)Shown below are the operations of manual interruption during corner chamfering or corner rounding.

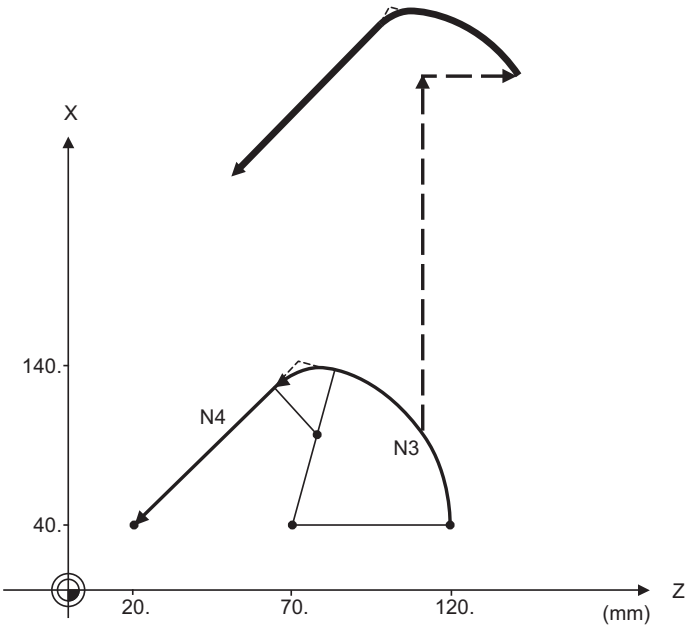
With an absolute value command and manual absolute switch ON.

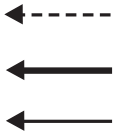
```
N1 G28 XZ;
N2 G00 X40. Z120.;
N3 G03 X140.Z70. K-50. ,R20. F100 ;
N4 G01 X40. Z20. ;
   :
```



With an incremental value command and manual absolute switch OFF

```
N1 G28 XZ;
N2 G00 U40. W120.;
N3 G03 U100. W-50. K-50. ,R20. F100 ;
N4 G01 U-100.W-50. ;
   :
```




Interrupt amount

Path in interrupt case

Path in non-interrupt case

(2)With a single block during corner chamfering or corner rounding, the tool stops after these operations areexecuted.

15.2 Corner Chamfering II /Corner Rounding II

Function and purpose

Corner chamfering and corner rounding can be performed by adding ",C" or ",R" to the end of the block which is commanded first among the block that forms a corner with continuous arbitrary angle lines or arcs.
By setting the parameters, "I_", "K_", "C_" can be set for chamfering instead of ",C_", and "R_" can be set for rounding instead of ",R_". Corner chamfering and corner rounding can both be commanded with an absolute value or incremental value.

15.2.1 Corner Chamfering II ; G01/G02/G03 X_ Z_ ,C_/I_/K_/C_

Function and purpose

The corner is chamfered by commanding ",C" (or "I_", "K_", "C_") in the 1st block of the two continuous blocks containing an arc. For an arc, this will be the chord length.

Command format

```
N100 G03 X__  Z__  I__  K__  ,C__ (or C__) ;
N200 G01  X__  Z__  ;
```

,C / C	Length up to chamfering starting point or end point from hypothetical corner

Corner chamfering is performed at the point where N100 and N200 intersect.

Detailed description

(1) If this function is commanded while the corner chamfer or corner rounding command is not defined in the specifications, it causes a program error (P381).

(2) The start point of the block following the corner chamfering is the hypothetical corner intersection point.

(3) If the parameter "#1272 ext08/bit6" is "0" in the MTB specifications, the ",C" command will be interpreted as a C command if there is no "," (comma).

(4) If there are multiple or duplicate corner chamfering commands in a same block, the last command will be valid.

(5) When both corner chamfering and corner rounding are commanded in the same block, the latter command will be valid.

(6) Tool compensation is calculated for the shape which has already been subjected to corner chamfering.

(7) Program error (P385) will occur when positioning or thread cutting is commanded in the corner chamfering command block or in the next block.

(8) Program error (P382) will occur when the block following corner chamfering contains a G command other than group 01 or another command.

(9) Program error (P383) will occur when the movement amount in the block, commanding corner chamfering, is less than the chamfering amount.

(10) Program error (P384) will occur when the movement amount is less than the chamfering amount in the block following the block commanding corner chamfering.

(11) Even if a diameter is commanded, it will be handled as a radial command value during corner chamfering.

(12) Program error (P382) will occur when a movement command is not issued in the block following the corner chamfering II command.

(13) If "C" is used as the axis name or the 2nd miscellaneous function, corner chamfering cannot be commanded with "C".

(14) Corner chamfering cannot be commanded with "I" or "K" in a circular command block. "I" and "K" are the circular center commands.

Program example

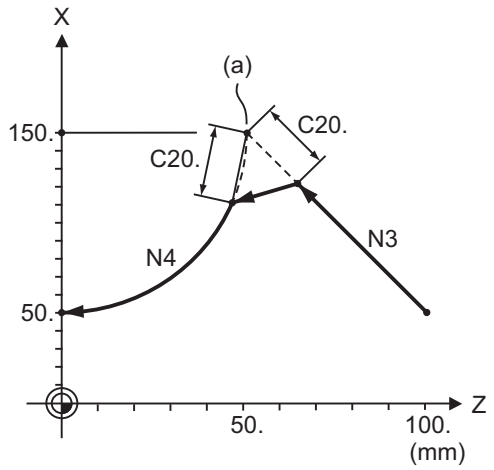
(1) Linear - arc

Absolute value command

```
N1 G28 X Z ;
N2 G00 X50. Z100. ;
N3 G01 X150. Z50. ,C20. F100 ;
N4 G02 X50. Z0 I0 K-50. ;
      :
```

Relative value command

```
N1 G28 X Z ;
N2 G00 U25. W100. ;
N3 G01 U50. W-50. ,C20. F100 ;
N4 G02 U-50. W-50. I0 K-50. ;
      :
```



(a) Hypothetical corner intersection point

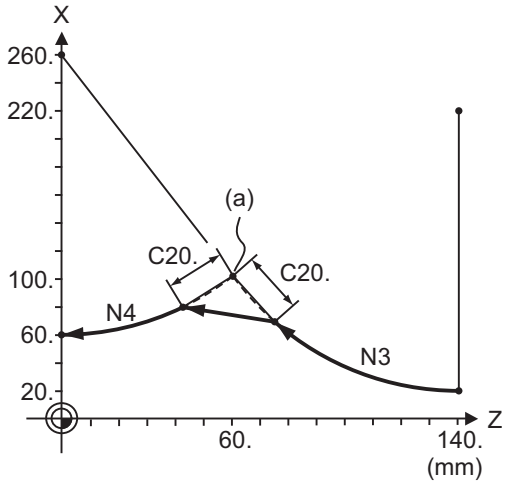
(2) Arc - arc

Absolute value command

```
N1 G28 X Z ;
N2 G00 X20. Z140. ;
N3 G02 X100. Z60. I100. K0. ,C20. F100 ;
N4 X60. Z0 I80. K-60. ;
      :
```

Relative value command

```
N1 G28 X Z ;
N2 G00 U10. W140. ;
N3 G02 U40. W-80. R100. ,C20. F100 ;
N4 U-20. W-60. I80. K-60. ;
      :
```



(a) Hypothetical corner intersection point

Precautions

(1) Corner chamfering and corner rounding can be commanded with "I", "K", "R" only when the 1st block of the corner chamfering/corner rounding command is linear.

(2) Corner chamfering with "I", "K", and corner rounding with "R" can be commanded when the 1st block of the corner chamfering/corner rounding command is linear and the 2nd block is circular. "I", "K" in the 2nd block are the circular center commands.
N100 G01 X__ Z__ I__ ; ..... I corner chamfering length
N200 G02 X__ Z__ I__ K__ ; ..... I,K circular center command

(3) If ",C_", ",R_" or "I_", "K_", "C_", "R_" are commanded in a same block, ",C_", ",R_" will have the priority.

15.2.2 Corner Rounding II ; G01/G02/G03 X_ Z_ ,R_/R_

Function and purpose

The corner is rounded by commanding ",R_" (or "R_") in the 1st block of the two continuous blocks containing an arc.

Command format

```
N100 G03  X__  Z__  I__  K__  ,R__ (or R__) ;
N200 G01  X__  Z__  ;
```

,R / R	Arc radius of corner rounding

Corner rounding is performed at the point where N100 and N200 intersect.

Detailed description

(1) If this function is commanded while the corner chamfer or corner rounding command is not defined in the specifications, it causes a program error (P381).

(2) The start point of the block following the corner rounding is the hypothetical corner intersection point.

(3) If the parameter "#1272 ext08/bit6" is "0" in the MTB specifications, the ",R" command will be interpreted as an R command if there is no "," (comma).

(4) When both corner chamfering and corner rounding are commanded in a same block, the latter command will be valid.

(5) Tool compensation is calculated for the shape which has already been subjected to corner rounding.

(6) Program error (P385) will occur when positioning or thread cutting is commanded in the corner rounding command block or in the next block.

(7) Program error (P382) will occur when the block following corner rounding contains a G command other than group 01 or another command.

(8) Program error (P383) will occur when the movement amount in the corner rounding block is less than the R value.

(9) Program error (P384) will occur when the movement amount is less than the R value in the block following the corner rounding.

(10) Even if a diameter is commanded, it will be handled as a radial command value during corner rounding.

(11) A program error (P382) will occur if a movement command is not issued in the block following corner rounding.

(12) Corner rounding cannot be commanded with "R" in a circular command block. "R" is the circular radius command.

Program example

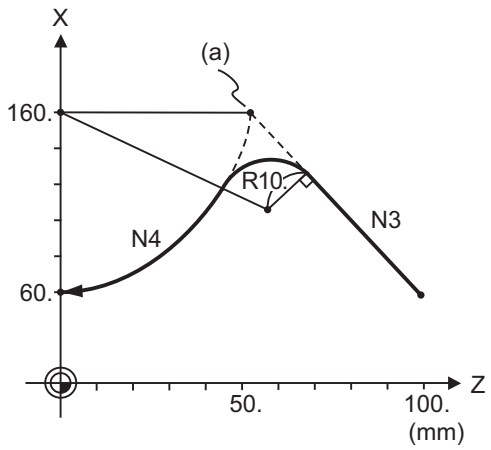
(1) Linear - arc

Absolute value command

```
N1 G28 X Z ;
N2 G00 X60.  Z100. ;
N3 G01 X160. Z50. ,R10. F100 ;
N4 G02 X60.  Z0 I0 K-50. ;
      :
```

Relative value command

```
N1 G28 X Z ;
N2 G00 U30. W100. ;
N3 G01 U50. W-50. ,R10. F100 ;
N4 G02 U-50. W-50. I0 K-50. ;
      :
```



(a) Hypothetical corner intersection point

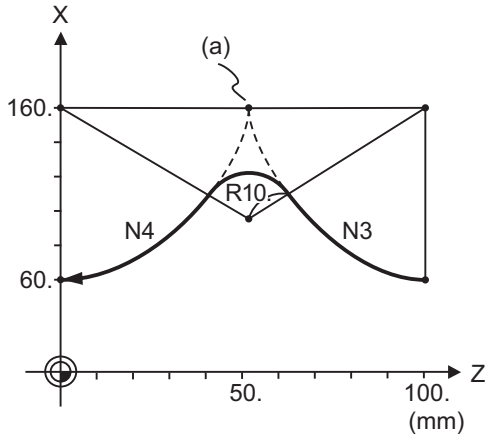
(2) Arc - arc

Absolute value command

```
N1 G28 X Z ;
N2 G00 X60. Z100. ;
N3 G02 X160. Z50. R60 ,R10. F100 ;
N4 X60. Z0  R50. ;
      :
```

Relative value command

```
N1 G28 X Z ;
N2 G00 U30. W100. ;
N3 G02 U50. W-50. I50. K0 ,R10. F100 ;
N4 U-50. W-50. I0. K-50. ;
      :
```



(a) Hypothetical corner intersection point

Precautions

(1) Corner chamfering and corner rounding can be commanded with "I", "K", "R" only when the 1st block of the corner chamfering/corner rounding command is linear.

(2) Corner chamfering with "I", "K", and corner rounding with "R" can be commanded when the 1st block of the corner chamfering/corner rounding command is linear and the 2nd block is circular. "I", "K" in the 2nd block are the circular center commands.

N100 G01 X__ Z__ I__ ; ..... I corner chamfering length

N200 G02 X__ Z__ I__ K__ ; ..... I,K circular center command

(3) If ",C_", ",R_" or "I_", "K_", "C_", "R_" are commanded in a same block, ",C_", ",R_" will have the priority.

15.2.3 Corner Chamfering Expansion/Corner Rounding Expansion

For details, refer to "Corner Chamfering I / Corner Rounding" and "Corner Chamfering Expansion / Corner Rounding Expansion".

15.2.4 Interrupt during Corner Chamfering/Interrupt during Corner Rounding

For details, refer to "Corner Chamfering I / Corner Rounding" and "Interrupt during Corner Chamfering Interrupt during / Corner Rounding".

15.3 Linear Angle Command ; G01 X_/Z_ A_/,A_

Function and purpose

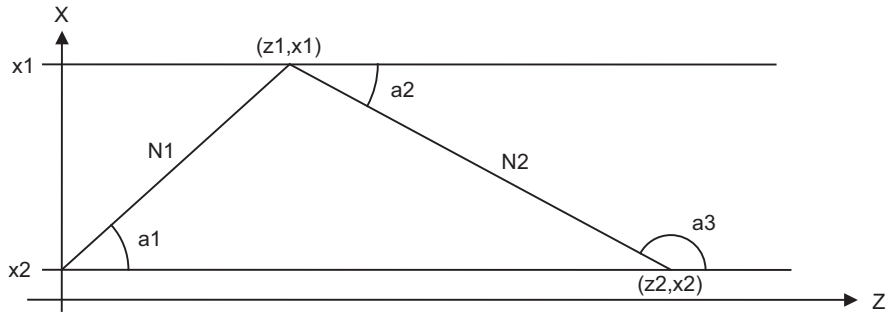
The end point coordinates are automatically calculated by commanding the linear angle and one of the end point coordinate axes.

Command format

```
N1 G01 Xx1(Zz1) Aa1;
N2 G01 Xx2(Zz2) A-a2; (A-a2 can also be set as Aa 3. )
```

```
N1 G01 Xx1(Zz1) ,Aa1;
N2 G01 Xx2(Zz2) ,A-a2;
```

This designates the angle and the X or Z axis coordinates.
Select the command plane with G17 to G19.



Detailed description

(1) As seen from the + direction of the horizontal axis of the selected plane, the counterclockwise (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) This function is valid only for the G01 command; it is not valid for other interpolation or positioning commands.

(7) The range of slope "a" is between -360.000 and 360.000.
When a value outside this range is commanded, it will be divided by 360 (degrees) and the remainder will be commanded.
(Example) If 400 is commanded, 40° (remainder of 400/360) will become the command angle.

(8) If an address A is used for the axis name or the 2nd miscellaneous function, use ",A" as the angle.

(9) If "A" and ",A" are commanded in a same block, ",A" will be interpreted as the angle.

15.4 Geometric ; G01 A_

Function and purpose

When it is difficult to calculate the intersection point of two straight lines in a continuous linear interpolation command, the end point of the first straight line will be automatically calculated inside the CNC and the movement command will be controlled, provided that the slope of the first straight line as well as the end point coordinates and slope of the second straight line are commanded.

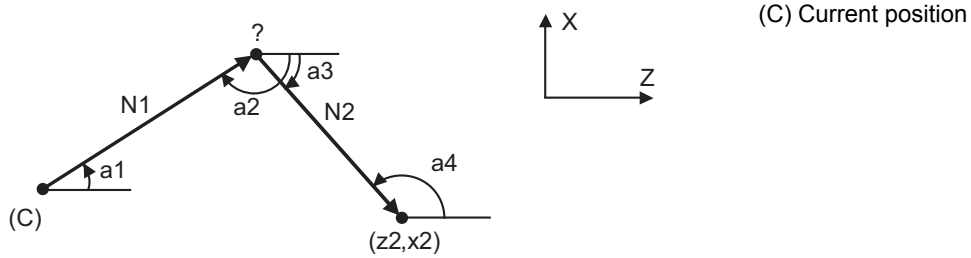
Note

(1) If the parameter "#1082 Geomet" is set to 0, geometric I will not function.

Command format

```
N1 G01 Aa1 (A-a2) Ff1;
N2 Xx2 Zz2 Aa4 (A-a3) Ff2;
```

Aa1, A-a2, A-a3, Aa4	Angle
Ff1, Ff2	Speed
Xx2, Zz2	Next block end point coordinates



Detailed description

(1) Program error (P396) will occur when the geometric command is not on the selected plane.

(2) The slope indicates the angle to the positive (+) direction of the horizontal axis for the selected plane. The counterclockwise (CCW) direction is indicated by a positive sign (+), and the clockwise (CW) direction by a negative sign (-).

(3) The range of slope "a" is between -360.000 and 360.000.
When a value outside this range is commanded, it will be divided by 360 (degrees) and the remainder will be commanded.
(Example) If 400 is commanded, 40° (remainder of 400/360) will become the command angle.

(4) The slope of the line can be commanded on either the start or end point side. Whether designated slope is the starting point or the end point will be automatically identified in NC.

(5) The end point coordinates of the second block should be commanded with absolute values. If incremental values are used, program error (P393) will occur.

(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 occur.

(8) Program error (P396) will occur when the plane is changed in the 1st block and 2nd block.

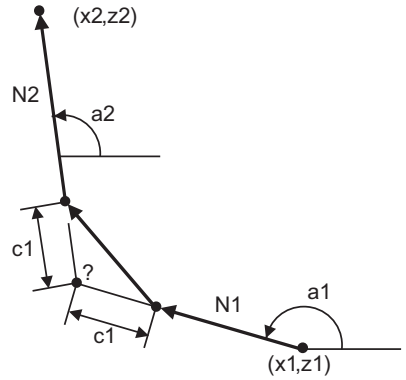
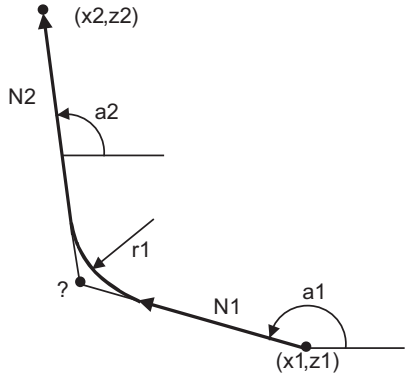
(9) This function is ignored when address A is used for the axis name or as the 2nd miscellaneous function.

(10) Single block stop is possible at the end point of the 1st block.

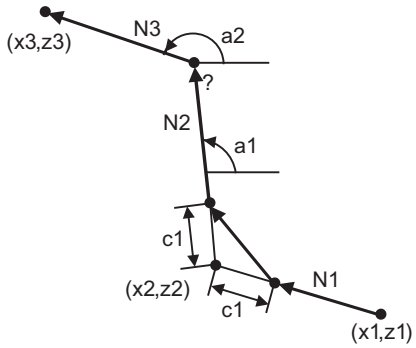
(11) Program error (P394) will occur when the 1st and 2nd blocks do not contain the G01 or G33 command.

Relationship with Other Functions

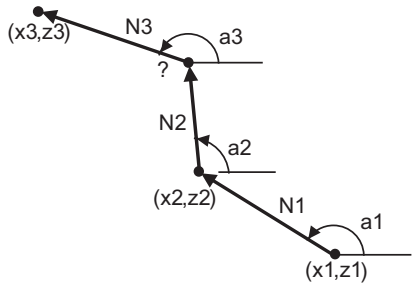
(1) Corner chamfering and corner rounding can be commanded after the angle command in the 1st block.

(Example 1) N1 Aa1 ,Cc1 ; N2 Xx2 Zz2 Aa2 ;	
(Example 2) N1 Aa1 ,Rr1 ; N2 Xx2 Zz2 Aa2 ;	

(2) The geometric command I can be issued after the corner chamfering or corner rounding command.

(Example 3) N1 Xx2 Zz2 ,Cc1 ; N2 Aa1 ; N3 Xx3 Zz3 Aa2 ;	

(3) The geometric command I can be issued after the linear angle command.

(Example 4) N1 Xx2 Aa1 ; N2 Aa2 ; N3 Xx3 Zz3 Aa3 ;	

15.5 Geometric IB

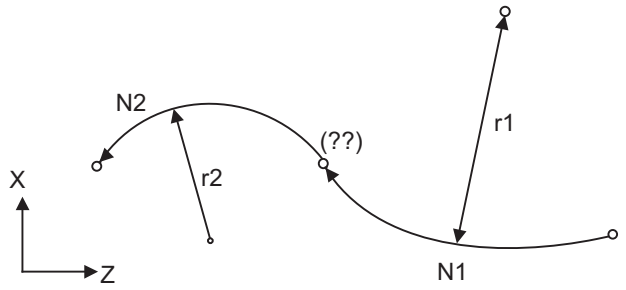
Function and purpose

With the geometric IB function, the contact and intersection are calculated by commanding an arc center point or linear angle in the movement commands of two continuous blocks (only blocks with arc commands), instead of commanding the first block end point.

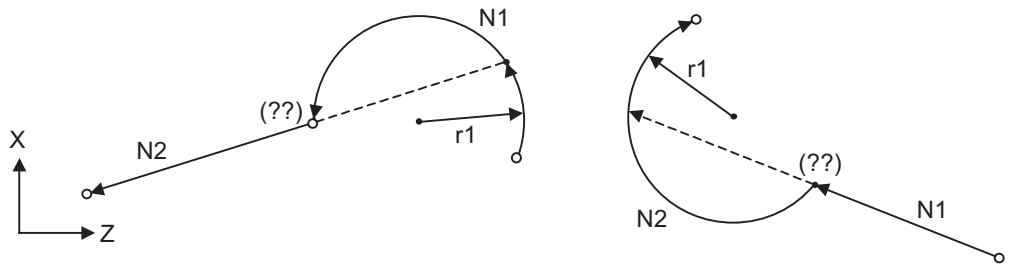
Note

(1) If the parameter (#1082 Geomet) is not set to 2, geometric IB will not function.

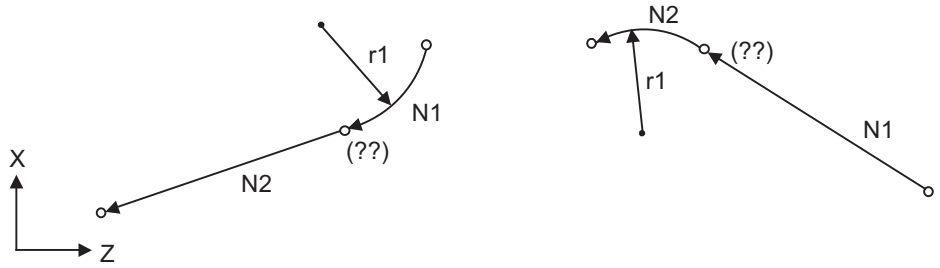
Two-arc contact



Linear - arc (arc - linear) intersection



Linear - arc (arc - linear) contact



15.5.1 Geometric IB (Automatic Calculation of Two-arc Contact) ; G02/G03 P_Q_ /R_

Function and purpose

When the contact of two continuous contacting arcs is not indicated in the drawing, it can be automatically calculated by commanding the 1st circular center coordinate value or radius, and the 2nd arc end point absolute value and center coordinate value or radius.

Command format

```
N1 G02(G03) Pp1 Qq1 Ff1;
N2 G03(G02) Xx2 Zz2 Pp2 Qq2 Ff2;
```

```
N1 G02(G03) Pp1 Qq1 Ff1;
N2 G03(G02) Xx2 Zz2 Rr2 Ff2;
```

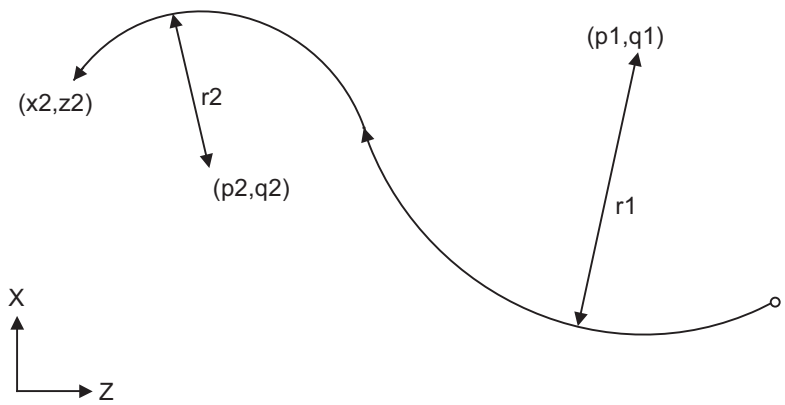
```
N1 G02(G03) Rr1 Ff1;
N2 G03(G02) Xx2 Zz2 Pp2 Qq2 Ff2;
```

P,Q	X and Z axes circular center coordinate absolute value (diameter/radius value command) The center address for the 3rd axis is commanded with A.
R	Arc radius (when a (-) sign is attached, the arc is judged to be 180° or more)

* I and K (X and Z axes arc center coordinate incremental value) commands can be issued instead of P and Q.

1st block arc : Radius command incremental amount from the start point to the center

2nd block arc : Radius command incremental amount from the end point to the center



Detailed description

(1) Program error (P393) will occur before the 1st block if the 2nd block is not a coordinate absolute value command.

(2) Program error (P398) will occur before the 1st block if there is no geometric IB specification.

(3) Program error (P395) will occur before the 1st block if there is no R (here, the 1st block is designated with P, Q (I, K)) or P, Q (I, K) designation in the 2nd block.

(4) Program error (P396) will occur before the 1st block if another plane selection command (G17 to G19) is issued in the 2nd block.

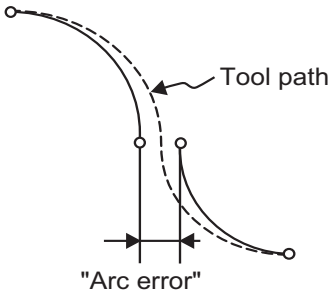
(5) Program error (P397) will occur before the 1st block if two arcs that do not contact are commanded.

(6) The contact calculation accuracy is ±1μm (fractions rounded up).

(7) Single block operation stops at the 1st block.

(8) When I or K is omitted, the values are regarded as I0 and K0. P and Q cannot be omitted.

(9) The error range in which the contact is obtained is set in parameter "#1084 RadErr".

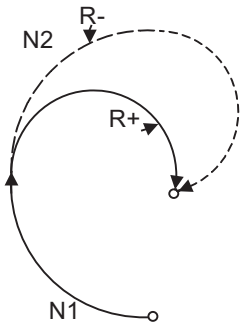


(10) For an arc block perfect circle command (arc block start point = arc block end point), the R designation arc command finishes immediately, and there is no operation. Thus, use a PQ (IK) designation arc command.

(11) G codes of the G modal group 1 in the 1st/2nd block can be omitted.

(12) Addresses being used as axis names cannot be used as command addresses for arc center coordinates or arc radius.

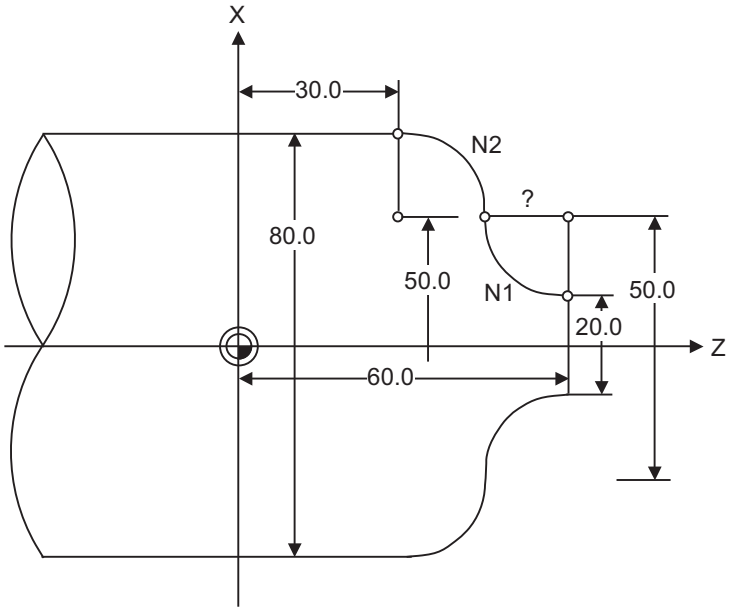
(13) When the 2nd block arc inscribes the 1st block arc and the 2nd block is an R designation arc, the R+ sign becomes the inward turning arc command, and the R- sign becomes the outward turning arc command.



(14) When the arc center of the 2nd block for geometric IB is commanded with IJK and the pitch is designated with address "P" or ",P", the helical interpolation is carried out for the arc of the 2nd block after geometric IB has been completed. Refer to "6.7 Helical Interpolation ; G17,G18,G19 and G02,G03" for details.

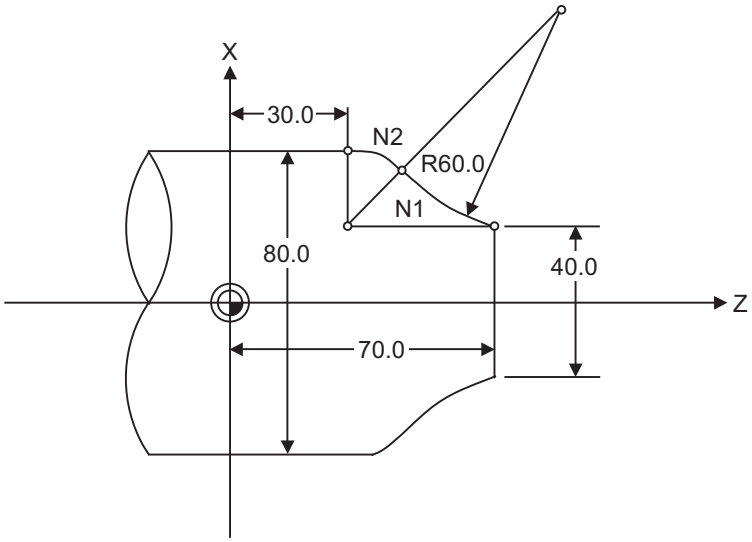
Program example

(1) **PQ, PQ command**



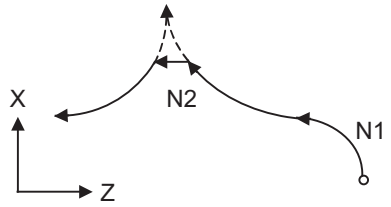
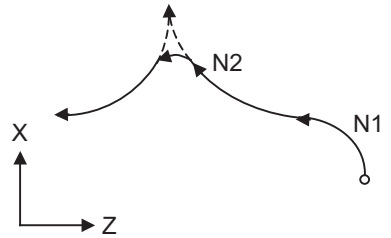
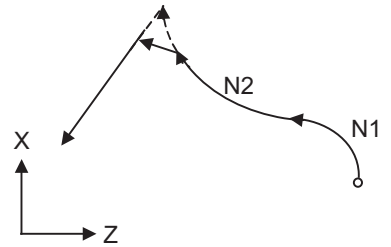
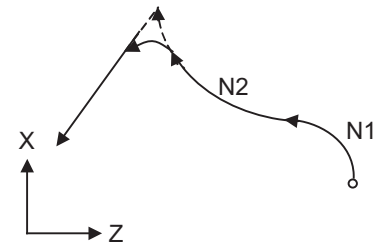

```
    G01 X20.0 Z60.0;
N1 G02 P50.0 Q60.0 F100;
N2 G03 X80.0 Z30.0 P50.0 Q30.0;
```
(mm)

(2) **PQ, R command**




```
    G01 X40.0 Z70.0 F100;
N1 G02 R60.0;
N2 G03 X80.0 Z30.0 P40.0 Q30.0;
```
(mm)

Relationship with Other Functions

Command	Movement of tool
Geometric IB + corner chamfering II N1 G09 P_ Q_ ; N2 G02 X_ Z_ R_ ,C_ ; G02 X_ Z_ R_ ;	
Geometric IB + corner R II N1 G03 P_ Q_ ; N2 G02 X_ Z_ R_ ,R_ ; G02 X_ Z_ R_ ;	
Geometric IB + corner rounding II N1 G03 P_ Q_ ; N2 G02 X_ Z_ R_ ,C_ ; G01 X_ Z_ ;	
Geometric IB + corner R II N1 G03 P_ Q_ ; N2 G02 X_ Z_ R_ ,R_ ; G01 X_ Z_ ;	

15.5.2 Geometric IB (Automatic Calculation of Linear - Arc Intersection) ; G01 A_ , G02/G03 P_Q_H_

Function and purpose

When the contact point of a shape in which contact between a line and an arc is not indicated in the drawing, it can be automatically calculated by commanding the following program.

Command format (For G18 plane)

```
N1 G01 Aa1 (A-a2) Ff1;
N2 G02(G03) Xx2 Zz2 Pp2 Qq2 Hh2 Ff2 ;
```

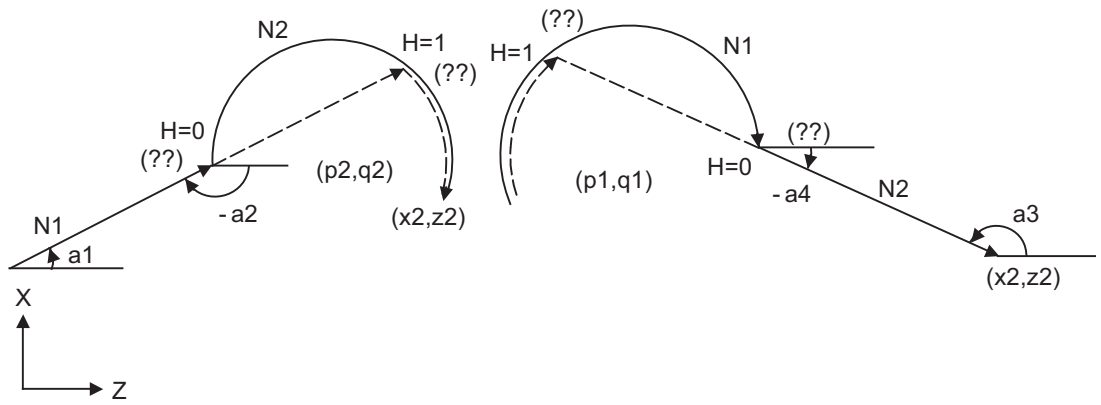
```
N1 G02(G03) Pp1 Qq1 Hh1 (,Hh1) Ff1 ;
N2 G01 Xx2 Zz2 Aa3 (A-a4) Ff2 ;
```

A	Linear angle (-360.000° to 360.000°)
P,Q	X and Z axes circular center coordinate absolute value (diameter/radius value command)The center address for the 3rd axis is commanded with A.
H (,H)	Selection of linear - arc intersection 0: Intersection of the shorter line 1: Intersection of the longer line

* I and K (X and Z axes arc center coordinate incremental value) commands can be issued instead of P and Q.

1st block arc : Radius command incremental amount from the start point to the center

2nd block arc : Radius command incremental amount from the end point to the center



Detailed description

(1) When the 2nd miscellaneous function address is A, the 2nd miscellaneous function is validated and this function is invalidated.

(2) Program error (P393) will occur before the 1st block if the 2nd block is not a coordinate absolute value command.

(3) Program error (P398) will occur before the 1st block if there is no geometric IB specification.

(4) In case of the 2nd block arc, a program error (P395) will occur before the 1st block if there is no P, Q (I, K) designation. A program error (P395) will also occur if there is no A designation for the line.

(5) Program error (P396) will occur before the 1st block if another plane selection command (G17 to G19) is issued in the 2nd block.

(6) Program error (P397) will occur before the 1st block if a straight line and arc that do not contact or intersect are commanded.

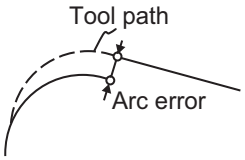
(7) Single block operation stops at the 1st block.

(8) When I or K is omitted, the values are regarded as I0 and K0. P and Q cannot be omitted.

(9) When H is omitted, the value is regarded as H0.

(10) The linear - arc contact is automatically calculated by designating R instead of P, Q (I, K).

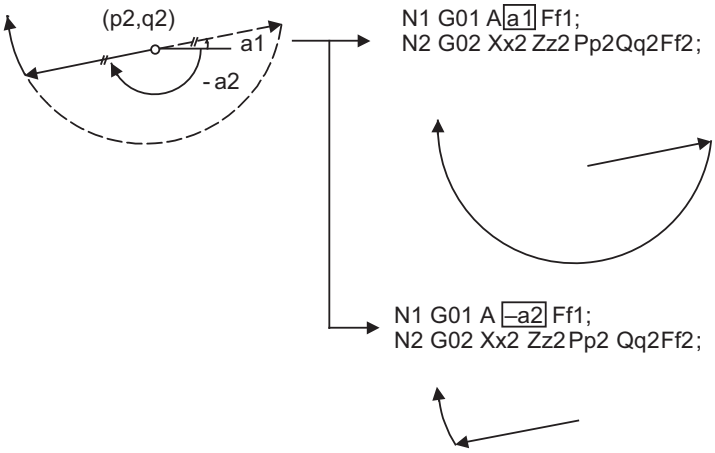
(11) The error range in which the intersect is obtained is set in parameter "#1084 RadErr".



(12) As seen from the + direction of the horizontal axis of the selected plane, the counterclockwise (CCW) direction is considered to be + and the clockwise direction (CW) -.

(13) The slope of the line can be commanded on either the start or end point side. Whether designated slope is the starting point or the end point will be automatically identified.

(14) When the distance to the intersection from the line and arc is same (as in the figure below), the control by address H (short/long distance selection) is invalidated. In this case, the judgment is carried out based on the angle of the line.



(15) The intersect calculation accuracy is ±1μm (fractions rounded up).

(16) In linear - arc intersections, the arc command can only be PQ (IK) command. When the arc block start point and arc block end point are the same point, the arc is a perfect circle.

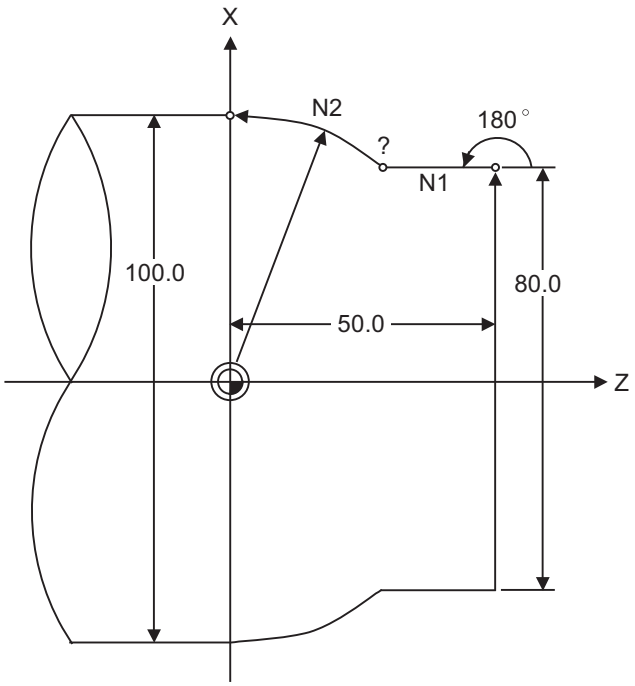
(17) G codes of the G modal group in the 1st block can be omitted.

(18) Addresses being used as axis names cannot be used as command addresses for angles, arc center coordinates or intersection selections.

(19) When geometric IB is commanded, two blocks are pre-read.

(20) When the arc center of the 2nd block for geometric IB is commanded with IJK and the pitch is designated with address "P" or ",P", the helical interpolation is carried out for the arc of the 2nd block after geometric IB has been completed. Refer to "6.7 Helical Interpolation ; G17,G18,G19 and G02,G03" for details.

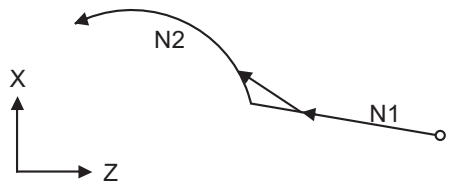
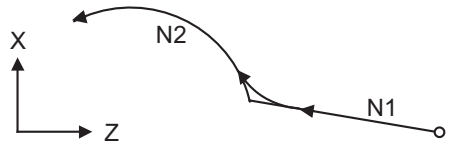
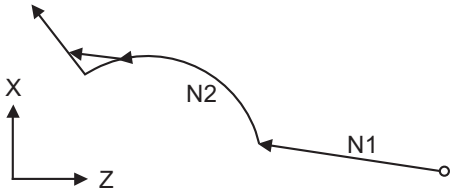
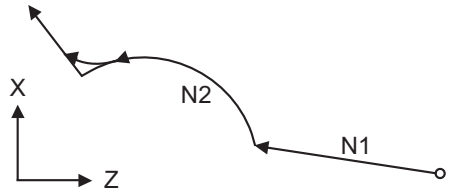
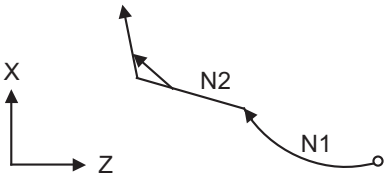
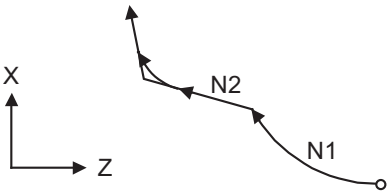
Program example




```
    G01 X80.0 Z50.0 F100;
N1 G01 A180.0;
N2 G03 X100.0 Z0 P0 Q0;
```

(mm)

Relationship with Other Functions

Command	Movement of tool
Geometric IB + corner chamfering II N1 G01 A_ ,C_ ; N2 G03 X_ Z_ P_ Q_ H_ ;	
Geometric IB + corner R II N1 G01 A_ ,R_ ; N2 G03 X_ Z_ P_ Q_ H_ ;	
Geometric IB + corner chamfering II N1 G01 A_ ; N2 G03 X_ Z_ P_ Q_ H_ ; G01 X_ Z_ ;	
Geometric IB + corner R II N1 G01 A_ ; N2 G03 X_ Z_ P_ Q_ H_ ; G01 X_ Z_ ;	
Geometric IB + corner chamfering II N1 G02 P_ Q_ H_ ; N2 G01 X_ Z_ A_ ,C_ ; G01 X_ Z_ ;	
Geometric IB + corner R II N1 G02 P_ Q_ H_ ; N2 G01 X_ Z_ A_ ,R_ ; G01 X_ Z_ ;	

15.5.3 Geometric IB (Automatic Calculation of Linear - Arc Intersection) ; G01 A_ , G02/G03 R_H_

Function and purpose

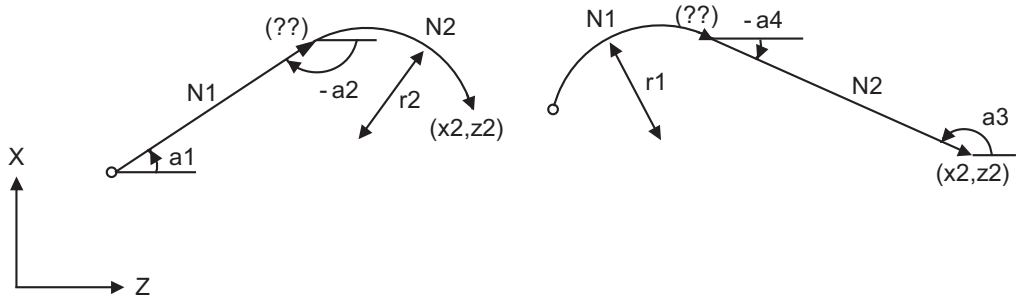
When the intersection of a shape in which a line and an arc intersect is not indicated in the drawing, it can be automatically calculated by commanding the following program.

Command format (For G18 plane)

```
N1 G01 Aa1 (A-a2) Ff1;
N2 G03(G02) Xx2 Zz2 Rr2 Ff2;
```

```
N1 G03(G02) Rr1 Ff1;
N2 G01 Xx2 Zz2 Aa3 (A-a4) Ff2 ;
```

A	Linear angle (-360.000° to 360.000°)
R	Circular radius



Detailed description

(1) When the 2nd miscellaneous function address is A, the 2nd miscellaneous function is validated and this function is invalidated.

(2) Program error (P393) will occur before the 1st block if the 2nd block is not a coordinate absolute value command.

(3) Program error (P398) will occur before the 1st block if there is no geometric IB specification.

(4) Program error (P396) will occur before the 1st block if another plane selection command (G17 to G19) is issued in the 2nd block.

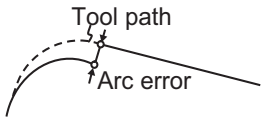
(5) A program error (P397) will occur before the 1st block if a straight line and arc that do not contact are commanded.

(6) In case of the 2nd block arc, a program error (P395) will occur before the 1st block if there is no R designation. A program error (P395) will also occur if there is no A designation for the line.

(7) Single block operation stops at the 1st block.

(8) The linear - arc contact is automatically calculated by designating R instead of P, Q (I, K).

(9) The error range in which the contact is obtained is set in parameter "#1084 RadErr".



(10) The line slope is the angle to the positive (+) direction of its horizontal axis. Counterclockwise (CCW) is positive (+). Clockwise (CW) is negative (-).

(11) The slope of the line can be commanded on either the start or end point side. Whether the commanded slope is on the start or end point side is identified automatically inside the NC unit.

(12) The intersect calculation accuracy is ±1μm (fractions rounded up).

(13) In linear - arc contact, the arc command can only be an R command. Thus, when the arc block start point = arc block end point, the arc command finishes immediately, and there will be no operation. (Perfect circle command is impossible.)

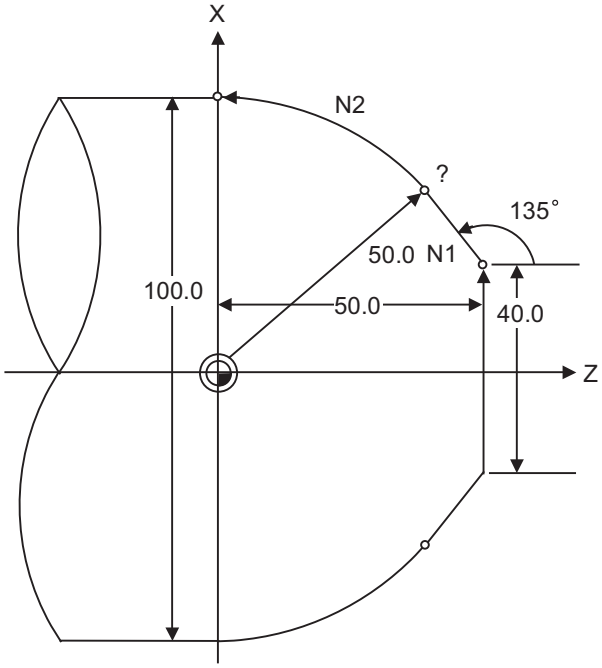
(14) G codes of the G modal group 1 in the 1st block can be omitted.

(15) Addresses being used as axis names cannot be used as command addresses for angles or arc radius.

(16) When geometric IB is commanded, two blocks are pre-read.

(17) When the arc center of the 2nd block for geometric IB is commanded with IJK and the pitch is designated with address "P" or ",P", the helical interpolation is carried out for the arc of the 2nd block after geometric IB has been completed. Refer to "6.7 Helical Interpolation ; G17,G18,G19 and G02,G03" for details.

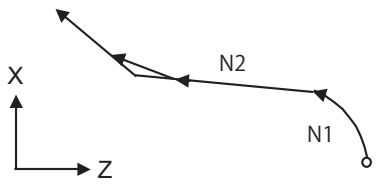
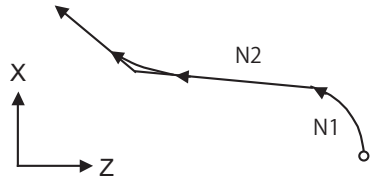
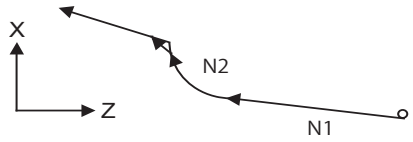
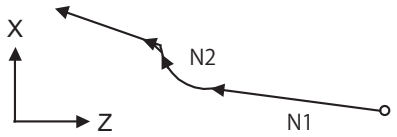
Program example




```
    G01 X40.0 Z50.0 F100;
N1 G01 A135.0;
N2 G03 X100.0 Z0.0 R50.0;
```

(mm)

Relationship with Other Functions

Command	Movement of tool
Geometric IB + corner chamfering N1 G03 R_ ; N2 G01 X_ Z_ A_ ,C_ ; G01 X_ Z_ ;	
Geometric IB + corner R N1 G03 R_ ; N2 G01 X_ Z_ A_ ,R_ ; G01 X_ Z_ ;	
Geometric IB + corner chamfering II N1 G01 A_ ; N2 G02 X_ Z_ R_ ,C_ ; G01 X_ Z_ ;	
Geometric IB + corner R II N1 G01 A_ ; N2 G02 X_ Z_ R_ ,R_ ; G01 X_ Z_ ;	

15.6 Manual Arbitrary Reverse Run Prohibition ; G127

Function and purpose

The manual arbitrary reverse run function controls the feedrate, which is under automatic operation in memory or MDI mode, in proportion to the manual feedrate by the jog or the rotation speed by the manual handle, and manually carries out the reverse run.
After the automatic operation has been stopped in a block, the reverse run can be carried out back through the blocks (up to 20 blocks) that were executed before the block. If necessary, it is possible to correct the program buffer and execute the fixed program after carrying out the reverse run up to the return position.

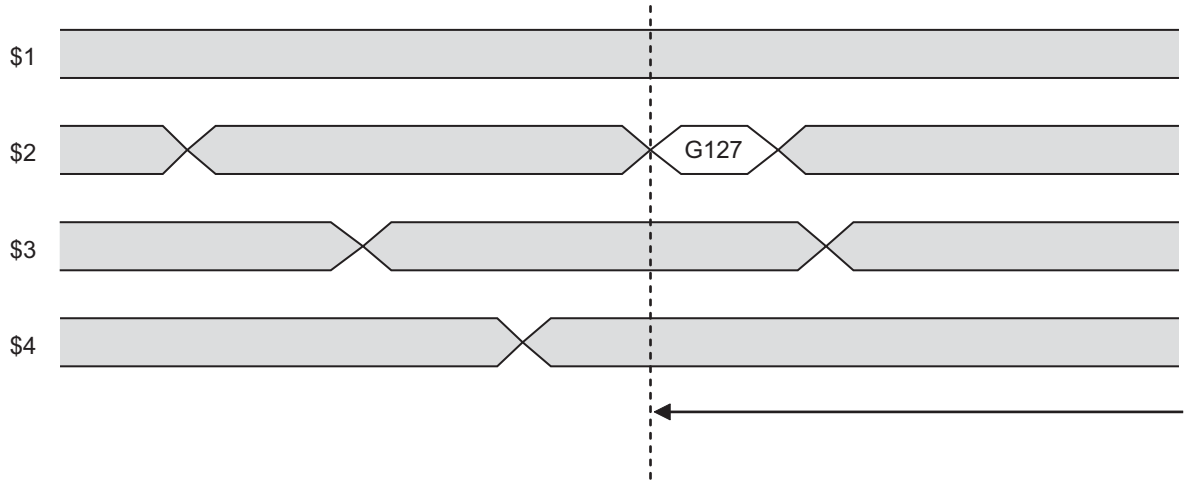
This function (G127) is available to prevent the program from backing to blocks before the commanded block when carrying out the manual arbitrary reverse run.
The detailed setting and operation vary depending on the machine specifications. Refer to the Instruction Manual issued by the MTB.
"Forward run" means to execute blocks in the same order as for the automatic operation.
"Reverse run" means to process the executed blocks backward.
Whether the reverse run is prohibited for each part system depends on the MTB specifications (system variable #3004). Refer to "List of System Variables" for details.

Command format

All part system reverse run prohibit command

```
G127 ;
```

This command disables the program from running reverse to blocks before G127. In part systems that do not have this command executed, the program cannot run reverse before the timing with G127 commanded in any part system even if a block is in process.
No commands in the machining program can be backed in the reverse run mode. For some G codes, the operation differs from the above. Refer to "Relationship with Other Functions".



The reverse run is disabled before the G127 block in the 2nd part system.
The reverse run is canceled in the middle of a block in part systems other than the 2nd part system.

Relationship with Other Functions

The following shows the relationship between the manual arbitrary reverse run command and G code.

The lathe-system G code is shown based on the G code system 3.

Symbol in "Reverse run" column	Operation
○ *1	Block with reverse run enabled
○ *2	Block with restricted-reverse run enabled Refer to the Remarks for restrictions.
Δ	Block with reverse run ignored. This block is ignored in both the forward and reverse run modes.
× *3	Block with reverse run prohibited. This is intended only for the command blocks.
× *4	Block with reverse run prohibited. The reverse run is also prohibited for all blocks after the mode has been switched by this block.
× *5	Prohibits the reverse run in all part systems.

G code (Lathe system)	Function name	Reverse run	Remarks
G00	Positioning	○ *1	-
G01	Linear interpolation	○ *1	-
G02	Circular interpolation CW	○ *1	-
G03	Circular interpolation CCW	○ *1	-
G02.3	Exponential interpolation CW	× *3	-
G03.3	Exponential interpolation CCW	× *3	-
G04	Dwell	○ *1	Dwell skip is invalid.
G05	High-speed high-accuracy control II / High-speed machining mode	× *4	-
G05.1	High-speed high-accuracy control I	× *4	-
G07.1 G107	Cylindrical interpolation	× *4	-
G08	High-accuracy control	× *4	-
G09	Exact stop check	○ *1	-
G10	Program data input (Parameter / Compensation amount) / Life management data registration	Δ	The reverse run is enabled, but data is not recovered.
G10.6	Tool retract command	× *3	-
G11	Program data input cancel / Life management data registration cancel	Δ	The reverse run is enabled, but data is not recovered.
G12.1 G112	Polar coordinate interpolation ON	× *4	-
G13.1 G113	Polar coordinate interpolation cancel	× *4	-
G12.1	Milling interpolation ON	× *4	-
G13.1	Milling interpolation cancel	× *4	-
G14	Balance cut OFF	○ *1	-
G15	Balance cut ON	○ *1	-
G16	Milling interpolation plane selection Y-Z cylindrical plane	× *3	The reverse run is prohibited because a command can only be issued during milling.
G17	X-Y plane selection	○ *2	Data is recovered using the modal information storage block.
G18	Z-X plane selection	○ *2	Data is recovered using the modal information storage block.

G code (Lathe system)	Function name	Reverse run	Remarks
G19	Y-Z plane selection	○ *2	Data is recovered using the modal information storage block.
G20	Inch command	○ *1	Switched with the movement command just after commanded.
G21	Metric command	○ *1	Switched with the movement command just after commanded.
G22	Barrier check ON	× *3	-
G23	Barrier check OFF	× *3	-
G22	Soft limit ON	× *3	-
G23	Soft limit OFF	× *3	-
G27	Reference position check	× *3	-
G28	Automatic reference position return	× *3	-
G29	Start position return	× *3	-
G30	2nd, 3rd and 4th reference position return	× *3	-
G30.1	Tool change position return 1	× *3	-
G30.2	Tool change position return 2	× *3	-
G30.3	Tool change position return 3	× *3	-
G30.4	Tool change position return 4	× *3	-
G30.5	Tool change position return 5	× *3	-
G31	Skip/Multi-step skip function 2	× *3	-
G31.1	Multi-step skip function 1-1	× *3	-
G31.2	Multi-step skip function 1-2	× *3	-
G31.3	Multi-step skip function 1-3	× *3	-
G33	Thread cutting	○ *2	Actual cutting mode available.
G34	Variable lead thread cutting	× *3	Actual cutting mode available.
G35	Circular thread cutting CW	× *3	Actual cutting mode available.
G36	Circular thread cutting CCW	× *3	Actual cutting mode available.
G37	Automatic tool length measurement	× *3	-
G40	Tool nose radius compensation cancel	○ *2	Data is recovered using the modal information storage block.
G41	Tool nose radius compensation left	○ *2	Data is recovered using the modal information storage block.
G42	Tool nose radius compensation right	○ *2	Data is recovered using the modal information storage block.
G43.1	1st spindle control mode	○ *2	Data is recovered using the modal information storage block.
G44.1	Selected spindle control mode	○ *2	Data is recovered using the modal information storage block.
G46	Tool nose radius compensation (automatic direction identification) ON	○ *2	Data is recovered using the modal information storage block.
G47.1	All spindles simultaneous control mode	○ *2	Data is recovered using the modal information storage block.
G92	Coordinate system setting / Spindle clamp speed setting	○ *1	-
G50.2 G250	Tool spindle synchronization IB mode cancel (Spindle - tool axis synchronization)	× *3	-
G51.2 G251	Tool spindle synchronization IB mode ON (Spindle - tool axis synchronization)	× *3	-
G52	Local coordinate system setting	○ *2	Data is recovered using the modal information storage block.

G code (Lathe system)	Function name	Reverse run	Remarks
G53	Machine coordinate system selection	○ *2	Data is recovered using the modal information storage block.
G54	Workpiece coordinate system 1 selection	○ *2	Data is recovered using the modal information storage block.
G55	Workpiece coordinate system 2 selection	○ *2	Data is recovered using the modal information storage block.
G56	Workpiece coordinate system 3 selection	○ *2	Data is recovered using the modal information storage block.
G57	Workpiece coordinate system 4 selection	○ *2	Data is recovered using the modal information storage block.
G58	Workpiece coordinate system 5 selection	○ *2	Data is recovered using the modal information storage block.
G59	Workpiece coordinate system 6 selection	○ *2	Data is recovered using the modal information storage block.
G54.1	Work coordinate system selection 48 sets expanded	○ *2	Data is recovered using the modal information storage block.
G61	Exact stop check mode	○ *1	-
G61.1	High-accuracy control	× *4	-
G62	Automatic corner override	○ *1	-
G63	Tapping mode	○ *1	-
G64	Cutting mode	○ *1	-
G65	Macro call Simple call	○ *1	-
G66	User macro Modal call A	○ *1	-
G66.1	User macro Modal call B	○ *1	-
G67	User macro Modal call cancel	○ *1	-
G68	Facing turret mirror image ON	× *3	-
G69	Mirror image for facing tool posts OFF	× *3	-
G68	Mirror image for facing tool posts ON or balance cut mode ON	× *3/ ○ *1	The reverse run is enabled if balance cut is designated.
G69	Mirror image for facing tool posts OFF or balance cut mode OFF	× *3/ ○ *1	The reverse run is enabled if balance cut is designated.
G68.1	Coordinate rotation by program ON	× *4	Speed control is required in the forward run mode.
G69.1	Coordinate rotation by program cancel	× *4	Speed control is required in the forward run mode.
G70	Finishing cycle	○ *1	Data is created for each movement block for the wrap operation.
G71	Longitudinal rough cutting cycle	○ *1	Data is created for each movement block for the wrap operation.
G72	Face rough cutting cycle	○ *1	Data is created for each movement block for the wrap operation.
G73	Formed material rough cutting cycle	○ *1	Data is created for each movement block for the wrap operation.
G74	Face cut-off Cycle	○ *1	Data is created for each movement block in the fixed cycle.
G75	Longitudinal cut-off cycle	○ *1	Data is created for each movement block in the fixed cycle.
G76	Compound thread cutting cycle	○ *2	The reverse run is enabled, but the synchronous feed is invalid. Actual cutting mode available.
G76.1	Multi-part system simultaneous thread cutting cycle I	○ *2	The reverse run is enabled, but the synchronous feed is invalid. Actual cutting mode available.

G code (Lathe system)	Function name	Reverse run	Remarks
G76.2	Two-part system simultaneous thread cutting cycle II	○ *2	The reverse run is enabled, but the synchronous feed is invalid. Actual cutting mode available.
G77	Longitudinal cutting fixed cycle	○ *1	Data is created for each movement block in the fixed cycle.
G78	Thread cutting fixed cycle	○ *2	The reverse run is enabled, but the synchronous feed is invalid. Actual cutting mode available.
G79	Face cutting fixed cycle	○ *1	Data is created for each movement block in the fixed cycle.
G80	Fixed cycle for drilling cancel	○ *1	-
G81	Fixed cycle (drill/spot drill)	○ *1	Data is created for each movement block in the fixed cycle.
G82	Fixed cycle (drill/counter boring)	○ *1	Data is created for each movement block in the fixed cycle.
G83	Deep hole drilling cycle (Z axis)	○ *1	Data is created for each movement block in the fixed cycle.
G83.1	Stepping cycle	○ *1	Data is created for each movement block in the fixed cycle.
G83.2	Deep hole drilling cycle 2	○ *1	Data is created for each movement block in the fixed cycle.
G84	Tap cycle (Z axis)	○ *2	The reverse run is enabled, but the synchronous feed is invalid. Actual cutting mode available.
G85	Boring cycle (Z axis)	○ *1	Data is created for each movement block in the fixed cycle.
G87	Deep hole drilling cycle (X axis)	○ *1	Data is created for each movement block in the fixed cycle.
G88	Tap cycle (X axis)	○ *2	The reverse run is enabled, but the synchronous feed is invalid. Actual cutting mode available.
G89	Boring cycle (X axis)	○ *1	Data is created for each movement block in the fixed cycle.
G84.1	Reverse tapping cycle (Z axis)	○ *2	The reverse run is enabled, but the synchronous feed is invalid. Actual cutting mode available.
G84.2	Synchronous tapping cycle	○ *2	The reverse run is enabled, but the synchronous feed is invalid. Actual cutting mode available.
G88.1	Reverse tapping cycle (X axis)	○ *2	The reverse run is enabled, but the synchronous feed is invalid. Actual cutting mode available.
G90	Absolute value command	○ *2	Switched with the movement command just after commanded.
G91	Incremental value command	○ *2	Switched with the movement command just after commanded.
G92	Coordinate system setting / Spindle clamp speed setting	○ *1	-
G92.1	Workpiece coordinate preset	○ *1	-
G94	Asynchronous feed (feed per minute)	○ *1	-
G95	Synchronous feed (feed per revolution)	○ *1	-
G96	Constant surface speed control ON	○ *2	Switched with the movement command just after commanded.
G97	Constant surface speed control OFF	○ *2	Switched with the movement command just after commanded.

G code (Lathe system)	Function name	Reverse run	Remarks
(G94)	Asynchronous feed (feed per minute)	○ *1	-
(G95)	Synchronous feed (feed per revolution)	○ *1	-
G98	Fixed cycle (Initial level return)	○ *1	-
G99	Fixed cycle R point level return	○ *1	-
G110	Mixed control I (cross control I)	× *3	-
G111	Axis name switch	× *3	-
G113	Tool spindle synchronization I cancel / Tool spindle synchronization IA (Spindle - spindle synchronization) mode cancel	× *3	-
G114.1	Spindle synchronization I	× *3	-
G114.2	Tool spindle synchronization IA (Spindle - spindle synchronization) mode ON	× *3	-
G114.3	Tool spindle synchronization II (Hobbing) ON	× *3	-
G115	Start point designation synchronization Type 1	○ *1	-
G116	Start point designation synchronization Type 2	○ *1	-
G117	M code output during axis traveling	× *3	-
G122	Sub part system control I	Δ	-
G125	Control axis synchronization between part systems	× *5	-
G126	Control Axis Superimposition	× *3	-
G140	Arbitrary axis exchange control	× *3	-
G141	Arbitrary axis exchange return	× *3	-
G144	Sub part system control II	Δ	-
G145	Wait for completion of sub part system OFF	Δ	-
G174	Simple tool center point control	× *1	-
G175	Simple tool center point control cancel	× *1	-
G176	Simple inclined surface control	× *1	-
M98	Subprogram call	○ *1	-

15.7 Data Input by Program

15.7.1 Parameter Input by Program; G10 L70, G11

Function and purpose

The parameters set from the setting and display unit can be changed in the machining programs.

G10 L70	For commanding data with decimal point, and character string data. The data's command range conforms to the parameter setting range described in Setup Manual.

Command format

Data setting start command

```
G10 L70 ;
P__  S__  A__  H□__ ;                Bit parameter
P__  S__  A__  D__  ;                Numerical value parameter
P__  S__  A__  <character string> ;  Character string parameter
```

P	Parameter No.
S	Part system No.
A	Axis No.
H	Bit type data
D	Numeric type data
character string	Character string data

Data setting end command

```
G11 ;
```

Note

(1) The sequence of addresses in a block must be as shown above.
When an address is commanded two or more times, the last command will be valid.

(2) The part system No. is set in the following manner. "1" for the 1st part system, "2" for 2nd part system, and so forth.
If the address S is omitted, the part system of the executing program will be applied.
As for the parameters common to part systems, the command of part system No. will be ignored.

(3) The axis No. is set in the following manner. "1" for 1st axis, "2" for 2nd axis, and so forth.
If the address A is omitted, the 1st axis will be applied.
As for the parameters common to axes, the command of axis No. will be ignored.

(4) Address H is commanded with the combination of setting data (0 or 1) and the bit designation □ (0 to 7).
Hd0: Sets the dth bit OFF. (d: 0 to 7)
Hd1: Sets the dth bit ON. (d: 0 to 7)

(5) Only the decimal number can be commanded with the address D.
The value that is smaller than the input setting increment (#1003 iunit) will be round off to the nearest increment.

(6) The character string must be put in angled brackets "<" and ">".
If these brackets are not provided, the program error (P33) will occur.
Up to 63 characters can be set.

(7) Issue "G10 L70" and "G11" commands in independent blocks. A program error (P421) will occur if not commanded in independent blocks.

(8) The parameter "#1078 Decimal pnt type 2" is disabled.

(9) The following data cannot be changed with the G10 L70 command:
Tool compensation data, workpiece coordinate data, PLC switch, and PLC axis parameter.

(10) The settings of the parameters with (PR) in the parameter list will be enabled after the power is turned OFF and ON.
Refer to the parameter list in your manual.

Precautions

Parameter update timing

The timing for updating the spindle parameter and the NC axis parameter settings depends on the MTB specifications (parameter "#1254 set26/bit3").

#1254 set26/bit3	Spindle parameter	NC axis parameter
Invalid	The program updates the parameter settings, waiting for "all axes smoothing zero" in all part systems.	
Valid	The program updates the parameter settings without waiting for "smoothing zero". (*1)	The program updates the parameter settings, waiting for "all axes smoothing zero" in control part systems. (*2)(*3)

(*1) The parameters of the target spindle are not updated while the functions below are active. The parameters are updated after the functions have been completed.
When the target spindle for which the functions below are being executed is the reference spindle for spindle synchronization or guide bushing spindle synchronization, the parameters of the synchronized spindle are not also updated.
- Synchronous tapping cycle
- Tool spindle synchronization IC (Spindle-NC axis, Polygon)
- Tool spindle synchronization II (Hobbing)
- The spindle for spindle position control is in C axis mode and the C axis is in motion.
- Synchronous tapping cycle for superimposed spindle in spindle superimposition control state

(*2) The program updates the exchange axis under the mixed control (cross axis control) or arbitrary axis exchange control, waiting for "all axes smoothing zero" in all part systems.

(*3) The program updates the synchronized axis or superimposed axis under the control axis superimposition, arbitrary axis superimposition, or control axis synchronization between part systems, waiting for "smoothing zero" of the reference axis and the synchronized axis or superimposed axis.
While the workpiece axis for the tool spindle synchronization II (hobbing) or the rotary axis is rotating in the spindle mode, the parameters of the target axis are not updated.

Program example

(1) For G10 L70

G10 L70 ;	
P6401 H71 ;	Sets "1" to "#6401 bit7".
P8204 S1 A2 D1.234 ;	Sets "1.234" to "#8204 of the 1st part system 2nd axis".
P8621 <X> ;	Sets "X" to "#8621".
G11 ;	

15.7.2 Compensation Data Input by Program ; G10 L2/L10/L11, G11

Function and purpose

The amount of tool compensation and workpiece offset can be set or changed by the G10 command. When the command is given with absolute values (X, Z and R), the offset is updated with the values. On the other hand, when the command is given with incremental values (U, W and C), the currently set offset plus the commanded offset serves as the new offset.

Command format

Workpiece coordinate system offset input (L2)

G10 L2 P__X__ (U__) Z__ (W__) ;

P	Compensation No.
X, Z	Compensation amount for each axis (absolute)
U, W	Compensation amount for each axis (incremental)

Extended workpiece coordinate offset amount setting (L20)

G10 L20 P__X__ (U__) Z__ (W__) ;

P	n No. of G54.n (1 to 48)
X, Z	Offset amount for each axis (absolute)
U, W	Offset amount for each axis (incremental)

External workpiece offset input (when L command is omitted)

G10 P0 X__ (U__) Z__ (W__) ;

P	Can be commanded when compensation No. is "0".
X, Z	Offset amount for each axis (absolute)
U, W	Offset amount for each axis (incremental)

Tool length compensation input (L10)

G10 L10 P__ X__ (U__) Z__ (W__) (Additional axis name)__ (2nd additional axis name)__ R__ (C__) Q__ ;

P	Compensation No.
X, Z	Compensation amount for each axis (absolute)
U, W	Compensation amount for each axis (incremental)
Additional axis name	Tool compensation amount for additional axes (absolute)
2nd additional axis name	Tool compensation amount for 2nd additional axis (absolute)
R	Nose R compensation amount (absolute)
C	Nose R compensation amount (incremental)
Q	Hypothetical tool nose point

Tool nose wear compensation input (L11)

G10 L11 P__ X__ (U__) Z__ (W__) (Additional axis name)__ (2nd additional axis name)__ R__ (C__) Q__ ;

P	Compensation No.
X, Z	Compensation amount for each axis (absolute)
U, W	Compensation amount for each axis (incremental)
Additional axis name	Tool compensation amount for additional axes (absolute)
2nd additional axis name	Tool compensation amount for 2nd additional axis (absolute)
R	Nose R compensation amount (absolute)
C	Nose R compensation amount (incremental)
Q	Hypothetical tool nose point

When there is no L command with tool length compensation input (L10) or tool nose wear compensation input (L11).

Tool length compensation input command	P = 10000 + Compensation No. ("10001" and later)
Tool nose wear compensation input command	P = Compensation No. ("1" and later)

Compensation input cancel

G11 ;

Detailed description

(1) The following table shows the compensation Nos. and the setting ranges of the hypothetical tool nose points.

Address	Meaning	Setting range		
		L2	L10	L11
P	Compensation No.	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 the L command is issued: 1 to maximum number of tool compensation sets ♦When the L command is not issued: 10001 and later (10000 + maximum number of tool compensation sets)	When the L command is/is not issued: 1 to maximum number of tool compensation sets
Q	Hypothetical tool nose point	-	0 to 9	

<Note>

♦The number of tool offset sets will differ according to the model and parameter, so check the specifications.

(2) The setting range for the compensation amount is given below.
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 compensation amount is the sum of the present setting value and command value.

Setting	Tool length compensation amount		Wear compensation amount	
	Metric system	Inch system	Metric system	Inch system
#1003=B	± 9999.999 (mm)	± 999.9999 (inch)	± 9999.999 (mm)	± 999.9999 (inch)
#1003=C	± 9999.9999 (mm)	± 999.99999 (inch)	± 9999.9999 (mm)	± 999.99999 (inch)
#1003=D	± 9999.99999 (mm)	± 999.999999 (inch)	± 9999.99999 (mm)	± 999.999999 (inch)
#1003=E	± 9999.999999 (mm)	± 999.9999999 (inch)	± 9999.999999 (mm)	± 999.9999999 (inch)

Precautions

(1) Compensation amount setting range check
The maximum value of the wear compensation amount and the maximum additional value for the wear compensation input check respectively take precedence for a single-time compensation amount in the maximum value and incremental value command of the wear compensation amount, and when an amount greater than these values has been commanded, program error (P35) will occur.

(2) G10 is an unmodal command and is valid only in the commanded block.

(3) Compensation input can be performed similarly for the additional axis but even when the C axis has been designated as the additional axis or the 2nd additional axis, address C is handled as an incremental command value of the nose R in the L10 or L11 command. The additional axis and the 2nd additional axis can only be compensated by the absolute value setting.

(4) If an illegal L No. and tool compensation No. are commanded, program errors (P172 and P170) will occur respectively.

(5) When the P command is omitted during workpiece coordinate system offset input (L2 or L20), it will be handled as the currently selected workpiece offset input.

(6) A program error (P35) will occur when the compensation 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 compensation input (X,U or Z, W) is commanded, the address which is input last is valid.

(8) A program error (P33) will occur when no input data is commanded for the G10L (2/10/11) command.

(9) Decimal point is valid for compensation amount.

(Example) When the "G10 L10 P3 Z50. ;" block is executed, the following data will be input:

[Tool length data]

#	Z
3	50.000

(10) When G40 to G42 and G10 are commanded in the same block, whether a program error (P45) occurs or G40 to G42 are ignored will depend on the MTB specifications (parameter "#1241 set13"/bit0).

(11) Do not command G10 in the same block as the fixed cycle and sub-program call command. This will cause malfunctioning and program errors.

(12) When the T command has been issued in the same block as G10 with a parameter "#1100 Tmove" set to "0", the compensation will be accomplished in the following block.

(13) For the multiple C axis system, both C axis workpiece offset are rewritten with the workpiece offset input.

(14) If not even one piece of input data is commanded when the G10P0 command (with L command omitted) is issued, it will be ignored.

15.7.3 Tool/Material Shape Input by Program ; G10 L100/L101, G11

Function and purpose

This function sets tool shape data of the Tool management screen and material shape data of the 3D solid program check (hereinafter referred to as “3D check”) using the machining program. Using this function saves the step to input tool/material shapes on the screen when the 3D check is executed.

Command format

Tool shape settings from the program

G10 L100 ;	**Data setting start command**
P_ T_ K_ D_ H_ I_ J_ C_ ;	**Data setting command**

P	Data No.	Specify the data No. on the tool management screen. (Cannot be omitted.) The maximum value of data No. varies depending on the number of tool management data sets.
T	Tool No.	Specify the tool No. (Cannot be omitted.) 0 to 99999999 When "0" is specified, all the tool shape data of data No. specified by address P will be "0". In this case, only the tool shape data is changed.
K	Type	Designate the tool type using a numerical value. [Mill tool] 1: Ball end mill 2: Flat end mill 3: Drill 4: Radius end mill 5: Chamfer 6: Tap 7: Face mill [Turning tool] 51: Turning 52: Slotting 53: Thread cutting 54: Turning drill 55: Turning tap
D	Shape data 1	Designate shape data of the tool. (Decimal point input enabled) The setting details of shape data differ depending on the tool type.
H	Shape data 2	
I	Shape data 3	Refer to the following "Correspondence between tool types and shape data" for the settings for each tool type.
J	Shape data 4	
C	Tool color	Specify the tool color. 1: Gray 2: Red 3: Yellow 4: Blue 5: Green 6: Light blue 7: Purple 8: Pink

G11;	**Data setting end command**

[Correspondence between tool types and shape data]

[Mill tool]

Shape data	Item by tool type						
	Ball end mill	Flat end mill	Drill	Radius end mill	Chamfer	Tap	Face mill
1	Tool length						
2	Tool radius (*1)						
3	-	-	Tool nose angle	Corner rounding	End angle	Pitch	Cutter length
4	-	-	-	-	End diameter	Thread diameter	Shank diameter
5	-	-	-	-	-	-	-
6	-	-	-	-	-	-	-

[Turning tool]

Shape data	Item by tool type				
	Turning	Slotting	Thread cutting	Turning drill	Turning tap
1	Tool length A				
2	Tool length B			Tool length B (*1)	
3	Tool nose radius	Tool nose radius	-	Tool nose angle	Pitch
4	Tool nose angle	Tool nose width	-	-	Thread diameter
5	Cutting edge angle	Max. slot depth	-	-	-
6	Tool width	Tool width	Tool width	-	-

(*1) When "#8968 Tool shape radius validity" is set to "0", input the diameter value. When it is set to "1", input the radius value.

Note

(1) Omitted addresses cannot be set.

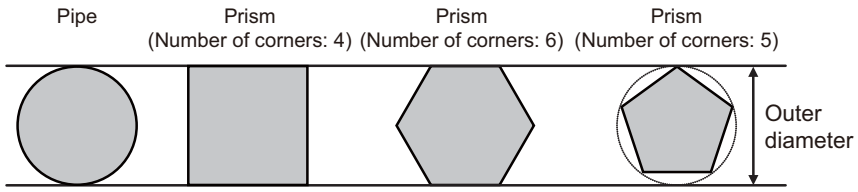
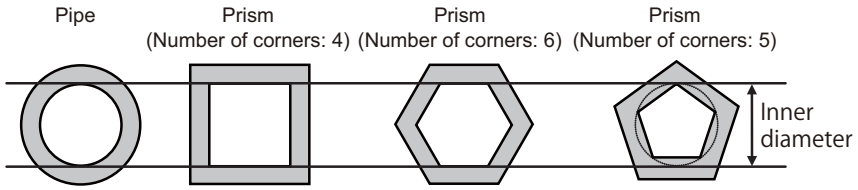
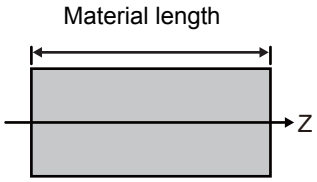
(2) If address "P" or "T" is omitted, a program error (P422) will occur.

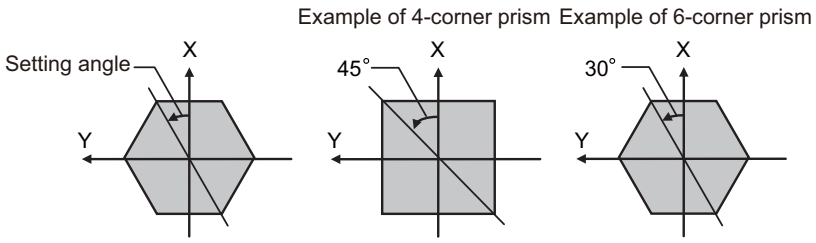
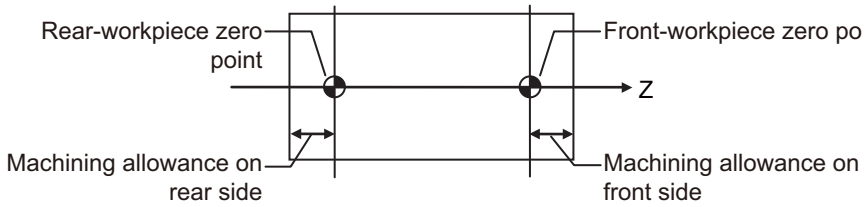
(3) For M80 Series, the tool shape data will be rewritten during the graphic check.

(4) For M800W and M800S Series, this change is only reflected on the graphic check drawing. The tool shape data is not rewritten.

Material shape settings from the program

G10 L101 ;	**Data setting start command**
C_ R_ D_ E_ L_ A_ K_ W_ F_ P_ H_ I_ J_ ;	**Data setting command**

C	Material shape	Specify the material shape. (Cannot be omitted.) 1: Pipe 2: Prism
R	Number of corners	Specify the number of corners when the material is in the shape of a prism. Setting range: 3 to 99 When the material is in the shape of a prism, the shape will be assumed to be a square if this setting is omitted. When the material is in the shape of a pipe, this setting is ignored even if the number of corners is entered.
D	External diameter of material	Specify the external diameter of the material. When the material is in the shape of a prism, specify the length between the opposite sides of the material if the number of corners is even, or the diameter of the circumscribed circle if the number of corners is odd. Setting range: 0.001 to 99999.999 mm, 0.0001 to 9999.9999 inch 
E	Internal diameter of material	Specify the internal diameter of the hope in the material. When the material is in the shape of a prism, specify the length between opposite sides of the hole if the number of corners is even, or the diameter of the circumscribed circle if the number of corners is odd. Setting range: 0.000 to 99999.999 mm, 0.0000 to 9999.9999 inch  If there is not a hole in the material, specify "0.000".
L	Material length	Specify the length of the material. Setting range: 0.001 to 99999.999 mm, 0.0001 to 9999.9999 inch 

A	Mounting angle	Specify the mounting angle of a prism. For the mounting angle, set the angle that is formed by the X axis and one angle in the material. Set CCW to the positive direction for the mounting angle. Setting range: 0.000 to 359.999°  This setting is valid only when material is in the shape of a prism. If the material is in the shape of a pipe, the mounting angle setting is ignored even if the mounting angle is entered.
K	Front face workpiece zero point position Z	Specify the machining allowances of the material face. The machining allowance is the distance on the Z axis between the face and the workpiece zero point. The machining allowance position from the material face is treated as the workpiece zero point. Setting range: -1, 0.000 to 99999.999 mm, -1, 0.0000 to 9999.9999 inch  [When the common workpiece zero point is used for machining of both the front and rear faces] Designate only the workpiece zero point for either the front or rear face, then set “-1” to the workpiece zero point that is not used. [When the common workpiece zero point is used individually for machining of the front and rear faces] Designate the individual workpiece zero point for both the front and rear faces. The rear workpiece zero point position Z can be set only when the spindle number on the rear face side is commanded. (This depends on the MTB specifications (parameter "#12628 sub sp no".) "-1" cannot be set for both workpiece zero points of the front and rear faces. (A setting error will occur.)
W	Rear workpiece zero point position Z	
F	Material color	Specify the material color, machining surface color, tap machining surface color, interference surface color, and cross-section surface color. 1: Gray 2: Red 3: Yellow 4: Blue 5: Green 6: Light blue 7: Purple 8: Pink
P	Machining surface color	
H	Machining surface color (Tap / Thread)	
I	Interference surface color	
J	Cross-section surface color	

G11;	**Data setting end command**

Note

(1) Omitted addresses cannot be set.

(2) If address "C" is omitted, a program error (P422) will occur.

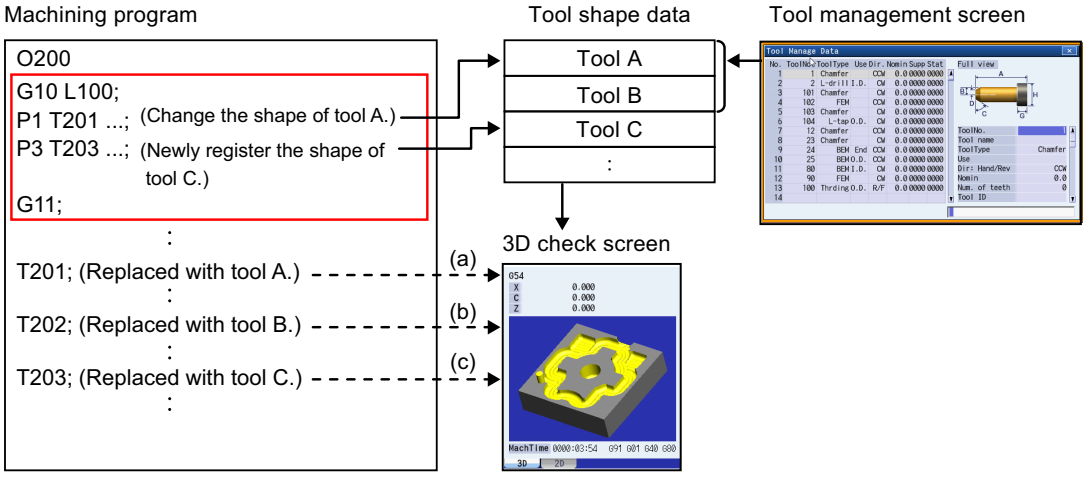
(3) For M80 Series, the workpiece shape data will be rewritten during the graphic check.

(4) For M800W and M800S Series, this change is only reflected on the graphic check drawing. The workpiece shape data is not rewritten.

Detailed description

Tool shape settings from the program

This function sets the tool shape of the Tool management screen from the machining program.
The 3D check switches the drawing of tools at the timing of a tool change command. Therefore, the machining program should be prepared to run a tool shape setting command prior to the tool change command being issued.



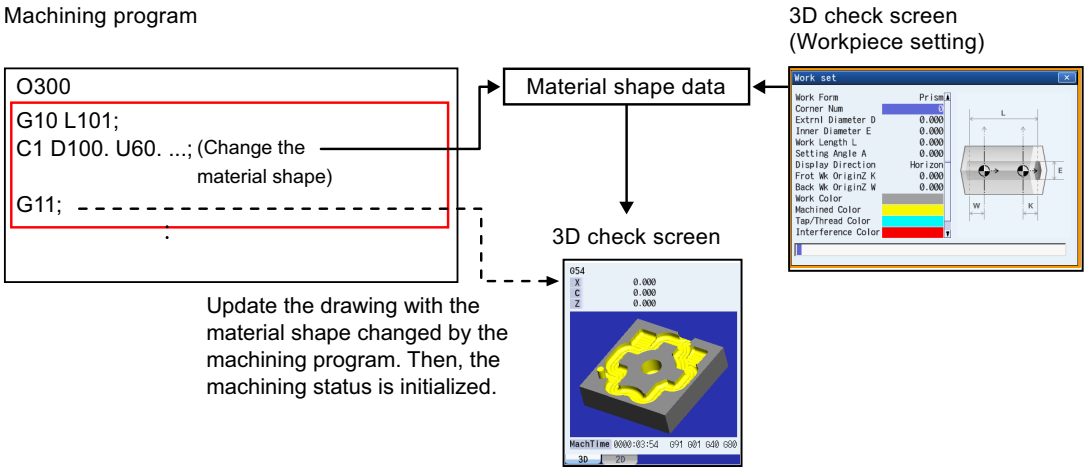
(a) The tool is drawn with the shape that has been changed by the machining program.

(b) The tool is drawn with a shape that has been registered on the tool management screen.

(c) The tool is drawn with a new shape that has been registered by the machining program.

Material shape settings from the program

This function sets a material shape on the 3D check screen from the machining program.
The 3D check sets the material shape using the data setting command of this function, and switches the drawing of materials at the timing of the subsequent data end command (G11). Then, the machined state is initialized, and the material is drawn in the raw state.



Program example

(1) Tool shape settings from the program

G10 L100 ;	
P1 T1 K3 D5. H20. I0 J0 C2 ;	Sets the data of data No. 1.
P2 T10 D10. ;	Sets the tool diameter of data No. 2 to "10.".
P8 T0 ;	Sets the tool shape data of data No. 8 to "0".
G11;	

(2) Material shape settings from the program

G10 L101 ;	
C1 D100. U60. R200. K10. W10.;	Material shape = pipe, material external diameter = 100 mm, material internal diameter = 60 mm, Material length = 200 mm, front face workpiece zero point position Z = 10 mm, Rear workpiece zero point position Z = 10 mm
G11;	

Precautions

(1) If the G10 or G11 command is not issued in an independent block, a program error (P422) will occur.

(2) If a block contains an address whose data is out of range, a program error (P35) will occur.

(3) If a block contains an illegal address, a program error (P32) will occur.

(4) The parameter "#1078 Decpt2" is valid for position commands (K address, W address).
Other command addresses comply with the minimum input unit ("#1015 cunit"). (Based on the MTB specifications.)

(5) The parameter "#8044 UNIT*10" is invalid.

(6) The command unit of parameters to be input in mm/inch can be switched by G20/G21.

15.8 Tool Life Management II ; G10 L3, G11

Function and purpose

Tool life management divides the tools being used into several groups, and manages the life (usage time, number of uses) of the tools in each group. When it comes to the end of life, a similar spare tool in the same group will be selected in order. This tool life management function with spare tools allows unmanned operation over a long time. The number of tools or groups for which the tool life management is available depends on the specifications.

(1) Number of tool life management sets	1-part system: Up to 1,000 sets (*1) Multi-part system: Allocated evenly to each part system. (*2)
(2) Number of groups	1-part system: Up to 1,000 sets Multi-part system: Allocated evenly to each part system.
(3) Group No.	1 to 9999
(4) Number of tools in one group	Up to 16 sets
(5) Life time	0 to 999999 minutes (approx. 16667 hours)
(6) Number of lives	0 to 999999 times

The tool life management data can be set from the NC program or from the Tool Life Management screen. Refer to the Instruction Manual for the method of setting from the Tool Life Management screen.
When using the NC program, register the data with the same method as programmable compensation data input.

(*1) The maximum number of tool life management sets per part system is 999.

(*2) For the even allocation, the remainder is allocated to the 1st part system.

Command format

Start of life management data registration

```
G10  L3 ;
P__ L__ N__R__ ;   (First group)
T__ ;
T__ ;
P__ L__ N__ ; (Next group)
T__ ;
T__ ;
```

P	Group No., (1 to 9999)
L	Life per tool (0 to 999999 minutes, or 0 to 999999 times)
N	Method (0: Time management, 1: Use count management)
R	Remaining life (0 to 999999 min. or 0 to 999999 times) <Note> ♦Only when the life prediction function is enabled ("#1277 ext13/bit1"="1"), the remaining life can be set with address R. The R command without the life prediction function causes the program error (P33).
T	Tool No. The spare tools are selected in the order of the tool Nos. registered here. (Tool No. 1 to 999999. Compensation No. 1 to 80) Tn follows the specifications.

End of life management data registration

G11 ;

Program example

(1) Format

: T□□□□99 ;	Starts use of □□□□ group tool.
: :	
T□□□□88 ; : :	Cancel □□□□ group tool compensation (Equivalent to TΔΔ00: ΔΔ is No. of the tool being used)
M02(M30) ;	Ends machining program.

(2) Actual example

: T0199 ;	Starts use of group 01 tool.
: :	
T0188 ; : :	Cancels group 01 tool compensation. If the No. of the tool being used is 17, this is equivalent to T1700.
T0609 ; : :	Selects tool No. 06 and compensation No. 09. * Life management is not carried out for tool 06.
T0600 ; : :	Cancels group 06 tool compensation.
T0299 ; : :	Starts use of group 02 tool.
T0199; :	Starts use of group 01 tool. If the selected tool has several compensation Nos., the second compensation No. will be selected.

<Note>

- When setting or changing tool life management data, be sure to select tool groups again with T□□□□99. When the data are set or changed with tools having been selected, if cutting is executed, the tool life may not be managed correctly.
- T□□□□99 command and T□□□□88 command may not be used depending on the MTB specifications. (Parameter “#1441 Tcode_Method_Chg” (T command method switching))
  The T command at this time acts as a T function. When using T□□□□99 command and T□□□□88 command as a command for the tool life management, be sure to set this parameter to "0".

Operation example

Example of tool selection operation (When one tool has several compensation Nos.)

(1) To use several compensation Nos. with one tool, select the next compensation No. for each T□□□□99 command.

(2) If T□□□□99 is commanded for more times than the number of registered compensation Nos., the last compensation No. will be selected, and the operation will continue. (Refer to following.)

Register to group 1	Program	Tool selection
T1701	T0199 ;	Equivalent to T1701
	:	Equivalent to T1702
T1702	T0199 ;	Equivalent to T1703
	:	Equivalent to T1703:
T1703	T0199 ;	
	:	
T2104	T0199 ;	
	:	
(Group 1)	:	: (Hereafter, same until tool 17 reaches the end of life.)

(3) If the above program is executed after resetting with M02/M30, or by resetting with external reset, the head compensation No will again be selected.

Precautions

(1) The tool life data is registered by executing the above program in the memory or MDI mode.

(2) When the above program is executed, all data (group No., tool No., life data) registered previously will be deleted. The registered data is held even if the power is turned OFF.

(3) The group No. designated with P does not have to be consecutive, but it should be set in ascending order if possible. Because Nos will be displayed in ascending order on the screen, this will make monitoring easier. The group No. cannot be commanded in duplicate.

(4) If the life data (L_) is omitted, the life data for that group will be "0". If N_ which specifies method is omitted, the method for that group depends on the MTB specifications (parameter "#1106 Tcount").

(5) If the remaining life data (R_) is omitted, the remaining life data for that group will be "0". When the remaining life data is "0", the "Life prediction" signal is not output for the designated tool. If N_ which specifies method is omitted, the method for that group depends on the MTB specifications (parameter "#1106 Tcount"). If the remaining life data (R_) is issued in larger value than life value (R_>L_), a program error (P35) will occur.
If the data L_ is omitted and the remaining life data (R_) is issued, a program error (P33) will occur.

(6) Programming with a sequence No. is not possible between G10 L3 and G11.

(7) If the usage data count valid signal (YC8A) is ON, G10 L3 cannot be commanded. (Program error (P177))
Refer to the specifications of your machine tool.

15.8.1 Counting the Tool Life

Function and purpose

The tool life can be counted with the time-count type or number of uses-count type. For the number of uses-count type, the type 2 may be employed for the count method and timing detection depending on the MTB specifications "#1277 ext13/bit0". Refer to the specifications of your machine tool.
If the usage data is equivalent to or more than the life data as a result of the count up, a spare tool in the group will be selected with the next group selection command (T□□□□99), and the newly selected tool will be counted.
When all tools in the group have reached their lives and a spare tool cannot be selected, the count will continue.

Detailed description

Counting the time of uses when the time-count type method is selected

In the cutting mode (G01, G02, G03, G31, G33, etc.), the length of time in which the tool is used is counted with 100 ms units.
The time is not counted during dwell, machine lock, miscellaneous function lock or dry run. Whether to count or not during single block depends on the MTB specifications (parameter "#1094 TI_SBK").

Note

- The maximum value for the life is 999999 minutes.
- The data is displayed in a minute unit on the Tool Life Management screen.

When the number of uses-count type method is selected: Type 1 (#1277 ext13/bit0: 0)

The number of uses is counted when the No. of the tool being used is changed with the tool selection command (T□□□□99) and when the program is in the cutting mode (excluding machine lock, miscellaneous function lock and dry run states).
The use is not counted if the cutting mode is not entered even once after the No. changes.
Whether to count or not during single block depends on the MTB specifications (parameter "#1094 TI_SBK").

Note

- The maximum value for the life is 999999 times.
- If only the compensation No. of the tool being used is changed, counting will not be conducted.

Example: When T code of tool being used is T12345678

T12345678
- Compensation No.: No count even when changed.
- Tool No.: Counted when changed.

<<Operation example 1>>

```
T0199                    (1)
  :
T0299
  :
T0199                    (2)
  :
T0299
  :
T0199                    (3)
```

Group 01 has been used three times.

<<Operation example 2>>

```
T0199                    (1)
  :
  :
T0299
  :
  :
T0199
```

Group 01 has been used once.

The number of uses is for one program execution. If the program is executed again after resetting, it will be counted.

When the number of uses-count type method is selected: Type 2 (#1277 ext13/bit0: 1)

(1) The groups, used for cutting between the start to reset of the machining program, is added with only "1". The count is made at the reset.

(2) If recount M is commanded, the group used up to that point will be added with "1" to the counter.

Note

- A count is not made in the machine lock, miscellaneous function lock or dry run states.
- Whether to count or not during single block depends on the MTB specifications (parameter "#1094 TI_SBK").
- The maximum value for the life is 999999 times.

Counting when one tool has several compensation Nos.

This function assigns independent usage data to each registered T number (Tool No. + Compensation No.). For a tool with several compensation Nos., counting of usage data is executed for each compensation No.
So, the sum of usage data for respective compensation Nos. is regarded as usage data of the tool and used for the tool life management. Accordingly, even if tool usage data for one # No. does not indicate that the tool reaches the end of its life, the tool status can be "2" (tool at the end of life) on the screen.
The sum of usage data for the tool currently selected is displayed in "Sum" field of <Tool in use> column on the Registered Group List screen.
The following are screen display examples when tools with several compensation Nos. are used.

Time-count type (Life: 100000 seconds)

#	Tool No.	Compensation No.	Use (s)	ST
1	101010	1	40000	2
2	101010	2	40000	2
3	101010	3	30000	2
4	202020	4	20000	1
5	202020	5	20000	1
6	202020	6	15000	1
7	303030	7	0	0

The sum of usage time for #1 to #3 indicates that tool 101010 comes to the end of its life.

Number of uses-count type (Life: 100000 times)

#	Tool No.	Compensation No.	Use (SET)	ST
1	101010	1	50000	2
2	101010	2	50000	2
3	101010	3	0	2
4	202020	4	40000	1
5	202020	5	40000	1
6	202020	6	0	1
7	303030	7	0	0

The sum of number of uses-count for #1 to #3 indicates that tool 101010 comes to the end of its life.

Re-selecting same tool group

When tool data of a tool with different compensation Nos. exists, the re-count M codes enable selecting the first compensation No.
The re-count M code depends on the MTB specifications (parameter "#1108 TlrectM").

[Example of re-count M]
When the re-count M code parameter "#1108 TlrectM" is "35":

(1) Tool selection when re-count M is not used

Register to group 1	Program	Tool selection	
T1701	T0199 ;	Equivalent to T1701.	
	:		
T1702	T0199 ;	Equivalent to T1702.	
	:		
T1703	T0199 ;	Equivalent to T1703.	
	:		
T2104	T0199 ;	Equivalent to T1703.	
	:		
	T0199 ;	Equivalent to T1703.	
	:	:	
	T0199 ;	Equivalent to T1703.	
(Group 1)		(Hereafter, the same until tool No. 17 reaches its life)	

(2) Tool selection when re-count M is used

Register to group 1	Program	Tool selection	
T1701	T0199 ;	Equivalent to T1701.	
	:		
T1702	T0199 ;	Equivalent to T1702.	
	:		
T1703	T0199 ;	Equivalent to T1703.	
	:		
T2104	M35 ;	Equivalent to T0188.	<Note> • Re-count M command (As MF is ON, FIN is returned)
	:		
	T0199 ;	Equivalent to T1701.	
	:		
	T0199 ;	Equivalent to T1702.	
	:		
	T0199 ;	Equivalent to T1703.	
(Group 1)		(Hereafter, the same until tool No. 17 reaches its life)	

Note

- In the case of "number of uses-count type" life management, the count is increased by one for the usage data when cutting is carried out after re-count M code is commanded.
  Therefore, in the case of the above example, the count is increased only by one in (1), but the count is increased by one again after M35 command in (2).
  In the case of "time-count type" life management, the length of time during cutting is counted, thus the re-count M code command does not affect the usage data.
- For the M code setting with "#1108 TlrectM", "0" is invalid, and 1, 2 and 30 are not allowed to use.

Operations when the end of tool life is reached

The tool status changes to "End of life" when the set tool usage time or uses-count is exceeded.
At this time, a tool life exceeding signal or tool group life exceeding signal is output according to the tool life management type.
Note that the signal is merely output and does not stop automatic operations of control units.
In addition, the tool life exceeding signal and tool group life exceeding signal can be temporarily canceled by using tool life exceeding temporary cancel signal or tool group life exceeding temporary cancel signal.

15.8.2 Allocation of The Number of Tool Life Management Sets to Part Systems

Function and purpose

The number of tool life management sets can be set per part system.
This function is divided into following methods and which one is used depends on the MTB specifications (parameters "#1439 Tlife-SysAssign", "#12055 Tol-lifenum").

Arbitrary allocation: Arbitrarily allocates the number of tool life management sets to each part system.

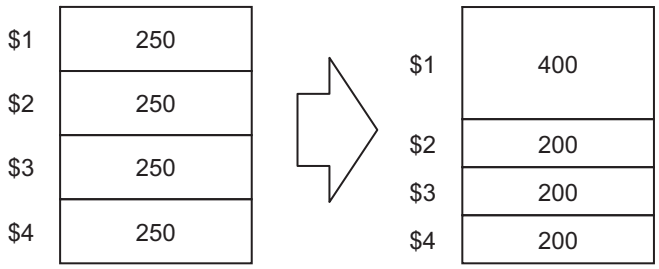
Fixed allocation: Automatically and evenly allocates the number of tool life management sets to each part system.

The arbitrary allocation enables the efficient allocation because when a certain part system needs only a small number of tool life management sets, the rest can be allocated to another part system. If an auxiliary-axis part system does not need the tool life management sets at all, the number of tool life management sets can be set to "0" for the auxiliary-axis part system.
Subsequent description is an example in the case where the number of tool life management sets in the system is 999 sets.

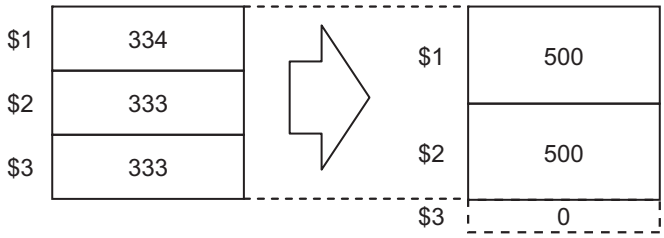
(1) Arbitrary allocation (with #1439=1)

The number of sets allocated to each part system depends on the MTB specifications (parameter "#12055 Tol-lifenum").
The following example shows the number of tool offset sets allocated when the lathe system is a 4-part system.

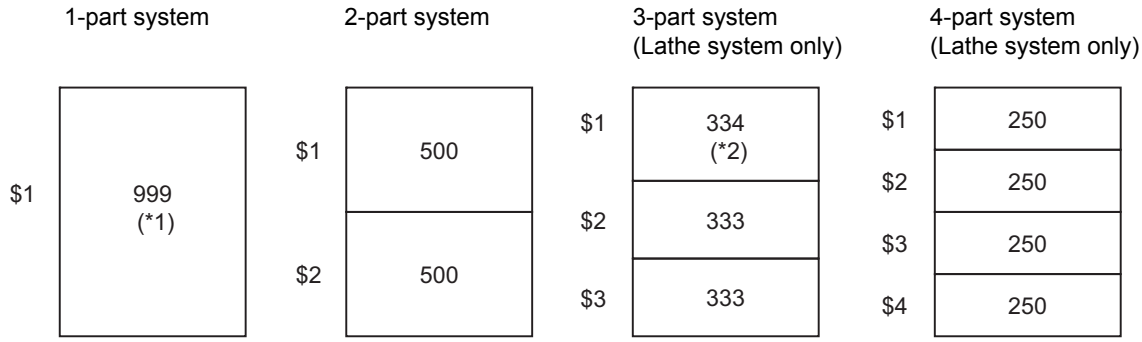
(a) When the number of tool life management sets is increased for the 1st part system ($1) of 4-part system



(b) When the number of tool life management sets is set to "0 sets" for the 3rd part system ($3) of 3-part system to use that part system as an auxiliary-axis part system



(2) Automatic and even allocation (with #1439=0)



(*1)The maximum number of tool life management sets per part system is 999.

(*2) If there is any remainder, the remainder is allocated to the 1st part system.

Precautions

(1) The maximum number of tool life management sets for 1-part system is 999.

(2) For 1-part system, up to the number of tool life management sets in the system is available regardless of the parameter setting.

(3) When the value of the parameter "#12055 Tol-lifenum" is equal to or lower than the number of tool life management sets in the system, the remainder is not allocated to any part system even if the specification allows arbitrary allocation.

(4) When the value of the parameter "#12055 Tol-lifenum" is equal to or lower than the number of tool life management sets in the system, system alarm (Y05) is generated even if the specification allows arbitrary allocation.

(5) Even if the specification allows arbitrary allocation, fixed allocation is applied if the parameter is "#12055 Tol-lifenum"= "0" for all part systems.

(6) When entering data into the tool life management file, if the number of tool life management data exceeds that of current tool life management sets, the excess tool life management data cannot be entered.

15.9 Axis Name Switch ; G111

Function and purpose

This function switches the commanded axis and the control axis.
When using a function, such as the hole drilling cycle (G88), that can be commanded to the limited axis this function can be used to give commands to axes that cannot be commanded with the normal command methods.

Command format

G111 Axis name 1 Axis name 2 ; ... Switching start command

Axis name 1/Axis name 2	The axis which operates after switching (The axis name set in the parameter "#1013 axname")

G111 ; ... Switching end command

Detailed description

(1) Axis name switch can be commanded simultaneously to several part systems.
While changing the axis name, G111 cannot be re-commanded.
If re-commanded, a program error (P411) will occur.
Command G111 in an independent block. If it is commanded in the same block with other G code, a program error (P33) will occur.

(2) G111 changes the command axis in the same part system.
During changing, "In axis name switch" signal turns ON.
If there is no commanded axis name, a program error (P32) will occur.

(3) The plane selection command modal (G17, G18, G19) will not change.
It automatically selects the plane which corresponds to the modal command when G111 is commanded.

(Example)

Basic system parameter

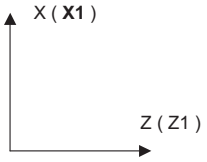
#1026	Basis axis I	X
#1027	Basic axis J	Y
#1028	Basic axis K	Z
#1029	Parallel axis I	
#1030	Parallel axis J	
#1031	Parallel axis K	

Basic axis specification parameter

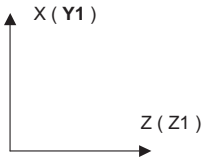
	<1>	<2>	<3>
#1013 axname	X	Y	Z
#1022 axname2	X1	Y1	Z1

(G18) G18 plane

:
:
:
:
:
:



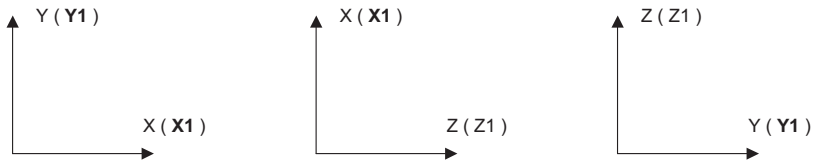
N10 G111 X Y ; X-Y axis switch
: G18 plane However, X command is applied to 2nd axis by
: G111 command.



(4) Plane selection can be commanded during the axis name switching.

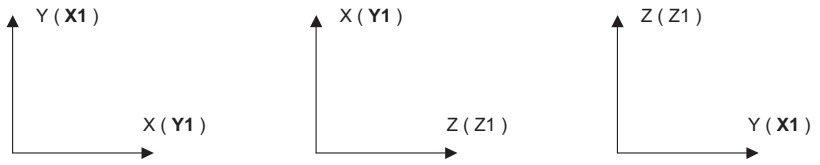
(Ex.1) Plane where no axis switching is carried out

G17 is commanded G18 is commanded G19 is commanded



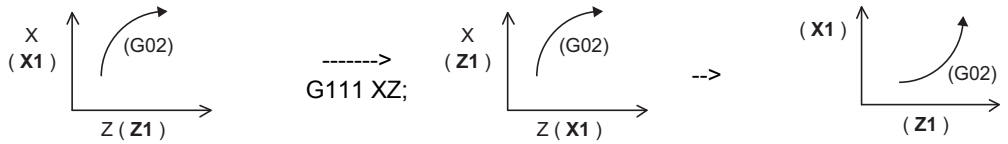
(Ex.2) Plane where axis name switching "G111 X Y ;" is carried out

G17 is commanded G18 is commanded G19 is commanded



(5) Circular interpolation, nose R and fixed cycle are carried out on the selected plane.

(Ex.) Circular interpolation after the axis name is switched



G02 command on the program G02 command after the axis name is switched Actual machine operation

(6) The plane selection will return to the state before G111 is commanded by canceling the axis name switching.

(7) Tool length compensation and tool nose wear compensation are applied to the axis before axis name switching. When X and Y are switched and then Y axis is moved by the X command, the compensation amount for Y axis is applied to Y axis and the compensation amount for X axis is not applied.

(Example)

[Compensation amount]

					X	Y	Z
T1103	->	Tool nose wear compensation No.	"03"	:	0.01	0.02	0.03
		Tool length compensation No.	"11"	:	0.1	0.2	0.3

[Program] [Machine coordinate value]

```
(G111)                                  <X1>    <Y1>    <Z1>
   :
N10 T1103 ;
N20 G90 G00 X10.0 Y20.0 Z30.0 ;  ---->  10.11   20.22   30.33
   :
N30 T0 ;
N40 G00 X0 Y0 Z0 ;
   :
N50 G111 X Y ;
N60 T1103 ;
N70 G00 X10.0 Y20.0 Z30.0 ;      ---->  20.11   10.22   30.33
```

(8) For the machine specification whose absolute/incremental is switched by address, when the axis name switching is carried out, likewise the absolute address will be switched.

(Example)

	Control X axis	Control Y axis
G111 ;	X , U , I	Y , V , J
G111 X Y ;	Y , V , J	X , U , I

However, an address which does not affect the axis movement, such as X address of dwell, does not switch.

(Example) G04 X2. ;
G111 X Y ;
G04 X2. ; <- X address of dwell does not switch to the command by Y.

(9) When an axis for which the axis name switch is carried out is a constant surface speed axis, the constant surface speed axis will be switched automatically.
When "G96 P_" is commanded, the axis No. is that of after the axis name is switched.
When the P command is omitted, the constant surface speed axis will follow the parameter "#1181 G96_ax".
However, when the constant surface speed control whose P command is omitted is commanded after the axis name is switched, the axis No., which is set by the parameter, becomes the axis No. after the axis name is switched.

(Example)
: <- Command "G96 P1" in this range -> X (X1) is the constant surface speed axis.
G111 X Y ;
: <- Command "G96 P1" in this range -> X (Y1) is the constant surface speed axis.

(10)The axis coordinate value and the tool length value which are read by variables are fixed regardless of the axis name switching.

Axis allocation by variables is not switched even during the axis name switching.

(Example)

	<1>	<2>	<3>
#1013 axname	X	Y	Z
#1022 axname2	X1	Y1	Z1

```
G111 ;
N10 G00 X100. Z200. Y300. ;
N20 #500 = #5021 ;      ←  Machine position coordinate value (100.) of <X1> is substituted into #500.
N30 #501 = #5022 ;      ←  Machine position coordinate value (200.) of <Z1> is substituted into #501.
N40 #502 = #5023 ;      ←  Machine position coordinate value (300.) of <Y1> is substituted into #502.
N50 G111 X Y ;
N60 #504 = #5021 ;      ←  Machine position coordinate value (100.) of <X1> is substituted into #504.
N70 #505 = #5022 ;      ←  Machine position coordinate value (200.) of <Z1> is substituted into #505.
N80 #506 = #5023 ;      ←  Machine position coordinate value (300.) of <Y1> is substituted into #506.
```

Program example

(Example)

```
G90 G00;
G111 X Y ;
G01 X100. ;        →  Y axis moves to 100.
G01 Y100. ;        →  X axis moves to 100.
G111 ;
G01 X0. ;          →  X axis moves to 0.
G01 Y0. ;          →  Y axis moves to 0.
```

Relation with other functions

Relation with fixed cycle

(1) Fixed cycle for drilling

Do not command the axis name switching during the fixed cycle for drilling.
If G111 is commanded during the fixed cycle for drilling, a program error (P411) will occur.

The following operation is carried out during the longitudinal hole drilling fixed cycle.
When "Longitudinal hole drilling axis selection" (YCD4) is turned ON, axis switching including Y axis becomes temporarily invalid during the longitudinal hole drilling fixed cycle (G87, G88, G89, G88.1).
When the longitudinal hole drilling fixed cycle is canceled, the switching becomes valid.
If the switching does not include Y axis, the switching is valid even during the longitudinal hole drilling fixed cycle.
"In axis name switch" signal will not turn OFF even when the switching is temporarily invalid.

(Ex.) Axis configuration X (1st axis), Z (2nd axis), Y (3rd axis), and C (4th axis)

```
G28 X Z Y C ;
G111 X Y ;
G87 R-5. X10. Z-30. Y-40.   ...  Switching X axis and Y axis becomes invalid during the longitudinal
F1000 ;                          hole drilling fixed cycle
Z-40 Y-50. ;
  :                              X: 1st axis
  :                              Z: 2nd axis
  :                              Y: 3rd axis
G80 ;
G01 X15. Z25. Y35. ;        ...  Switching X axis and Y axis becomes valid after canceling the the
  :                              longitudinal hole drilling fixed cycle
  :                              X: 3rd axis
  :                              Z: 2nd axis
  :                              Y: 1st axis
```

(2)Fixed cycle for turning machining

If G111 is commanded during the fixed cycle for turning machining (G77, G78, G79), the fixed cycle is canceled.

(3)Compound type fixed cycle for turning machining

Do not command the axis name switching for the finished shape program designated with G71, G72 or G73 block.
If G111 is commanded to the finished shape program, a program error (P411) will occur.

Relation with other functions

(1) Milling interpolation (G12.1/G13.1)
Do not command the axis name switching during the milling interpolation mode.
If G111 is commanded during the milling mode, a program error (P411) will occur.

(2) Constant surface speed control (G96, G97) (including clamp)
Do not command the axis name switching during the constant surface speed control mode.
If G111 is commanded during the constant surface speed control mode, a program error (P411) will occur.

(Example) Part system of axis No. (1) X, (2) Z, (3) Y and (4) C

```
G96 S300 P1
G111 X Y
G01 U-30.
```

The constant surface speed control is possible after the axis name switching.

(3) Nose R compensation (G41/G42/G40), nose R compensation automatic direction identification (G46/G40)
Do not command the axis name switching during the nose R compensation mode.
If G111 is commanded during the nose R compensation mode, a program error (P411) will occur.

(4) Workpiece coordinate selection (6 sets) (G54 to G59), external workpiece coordinate offset
Coordinate system offset amount is applied to the axis which is specified by the parameter regardless of the axis name switching.

(5) Plane selection
Axis name switching command does not change the plane selection's modal.
It automatically selects the plane which corresponds to the modal command when G111 is commanded.

(6) NC reset (Reset 1/2, reset & rewind)
When NC is reset, the axis name switching is automatically canceled.

(7) Mixed control (Cross axis control)
Do not command the axis name switching during the mixed control (cross axis control) mode.
If G111 is commanded during the mixed control (cross axis control) mode, a program error (P411) will occur.
If the mixed control (cross axis control) is commanded during axis name switch, the error "M01 operation error 1035" will occur.

(8) Polar coordinate interpolation
Do not command the axis name switching during the polar coordinate interpolation mode.
If G111 is commanded during polar interpolation mode, a program error (P411) will occur.

(9) Cylindrical interpolation
Do not command the axis name switching during the cylindrical interpolation mode.
If G111 is commanded during the cylindrical interpolation mode, a program error (P411) will occur.

(10) Workpiece offset input (G10 L2)
Do not command the workpiece offset input during G111 modal.
If the workpiece offset input is commanded during G111 modal, a program error (P421) will occur.

(11) Tool compensation input (G10 L10)
Do not command the tool compensation input during G111 modal.
If the tool compensation input is commanded during G111 modal, a program error (P421) will occur.

(12) Tool nose wear compensation input (G10 L11)
Do not command the tool nose wear compensation input during G111 modal.
If the tool nose wear compensation input is commanded during G111 modal, a program error (P421) will occur.

(13) Workpiece coordinate offset amount setting (G10 L20)
Do not set the workpiece coordinate offset amount during G111 modal.
If the workpiece coordinate offset amount is set during G111 modal, a program error (P421) will occur.

(14) Programmable current limitation (G10 L14)
Do not command the programmable current limitation during G111 modal.
If the programmable current limitation is commanded during G111 modal, a program error (P421) will occur.

(15)Workpiece coordinate preset
This function presets the workpiece coordinate system shifted with the program command or manual operation to the workpiece coordinate system which is offset by the workpiece coordinate offset amount from the machine zero point by the program command (G92.1/G50.3).

(16)Start point designation synchronization
When the axis name switching is carried out at the other part system after start point designation synchronization is commanded, the part systems can be waited at the axis before switching.
Command start point designation synchronization after the axis name switching completed.

Precautions

(1) Only two axes can be combined for the commanded axes when using the axis name switch command. A program error (P33) will occur if one independent axis or three or more axes are commanded.

(2) The settings such as the diameter value and radius value of the two axes used for axis name switch cannot be interchanged. Only the axis name is interchanged.

(3) Machining program, including the axis name switch, cannot perform program restart search.
If the axis name switch is commanded (G111) during the program restart search, a program error (P49) will occur.

(4) When address check is valid, do not command the axis name switch without the command value. Address check depends on the MTB specifications (parameter "#1227 aux11/bit4").
If the axis name switch is commanded without the command value, a program error (P33) will occur.

15.10 Mirror Image for Facing Tool Posts ; G68,G69

Function and purpose

In a machine in which the base turret and facing turret are integrated, this function is used to cut with the facing turret cutter using a program created with the base turret side.

The distance between the two turrets is set in the parameters beforehand.

Command format

G68 ; ... Mirror image for facing tool posts ON

G69 ; ... Mirror image for facing tool posts cancel

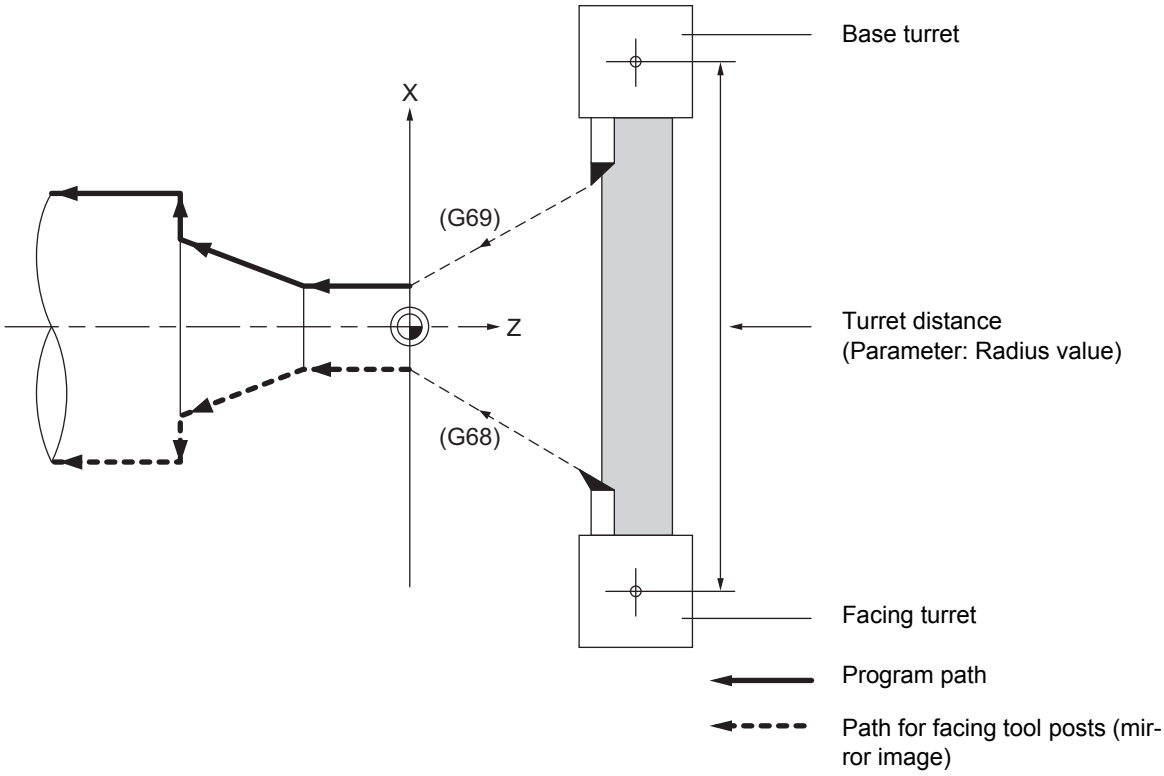
[T command mirror image for facing tool posts]
The mirror image for facing tool posts can be turned ON and OFF with the T command instead of the G68/G69 command.
The T command that validates or invalidates and executes a function depends on the MTB specifications (parameters shown below).

#	Item		Details	Setting range (unit)
1119	Tmiron	Select the T command mirror image for facing tool posts	Selects whether the mirror image for facing tool posts with the T command is valid or invalid.	0: Invalid 1: Valid
1203	TmirS1	Select turret as facing turret with T command	The turret selection for the T command mirror image for facing tool posts corresponding to tool No. 1 to 32 is set.	0 to FFFFFFFF
1204	TmirS2	Select turret as facing turret with T command	The turret selection for the T command mirror image for facing tool posts corresponding to tool No. 33 to 64 is set.	0 to FFFFFFFF

The operation is the same as for the mirror image for facing tool posts.

Detailed description

When G68 is commanded, the following program coordinate system is shifted to the facing turret side of the axis for which the mirror image for facing tool posts is valid (hereafter unless noted in particular, the X axis will be described as the axis for which mirror image for facing tool posts is valid). The axis movement direction is reversed from the program command. When G69 is commanded, the following program coordinate system will be returned to the base turret side.



The axis with the mirror image for facing tool posts valid depends on the MTB specifications (parameter "#1273 ext09/bit4").

(1) When "#1273 ext09 bit4" = 0
The mirror image for facing tool posts is valid for the 1st axis in the part system.

(2) When "#1273 ext09 bit4" = 1
Mirror image for facing tool posts is valid for the axis determined in the following manner by the plane selected when mirror image for facing tool posts is selected. Note that even if the plane is changed during mirror image for facing tool posts, the axis for which mirror image for facing tool posts is valid will not change.

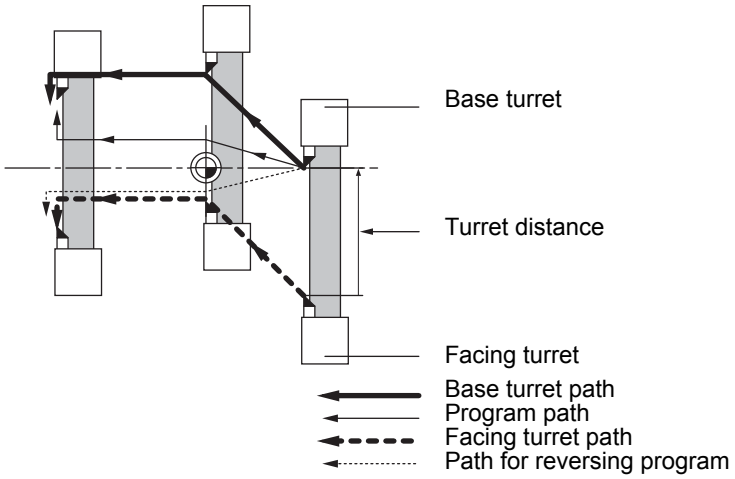
Plane selection	Axis for which mirror image for facing tool posts is valid
G17	J designated axis ("#1027 basic axis J" or "#1030 parallel axis J", which is an axis in plane configuration)
G18	I designated axis ("#1026 basic axis I" or "#1029 parallel axis I", which is an axis in plane configuration)
G19	K designated axis ("#1028 basic axis K" or "#1031 parallel axis K", which is an axis in plane configuration)

Absolute value/incremental value

(1) Absolute value command

The command position for the Z axis is reversed symmetrically, and the base turret moves to the position shifted by the distance between cutters.

```
T0101  ;
G00 X0. ;
G68 ;              Mirror image for facing tool posts ON
T0202  ;           Facing turret selection
G00 X10. Z0. ;
G01 Z-50. F400 ;
   X20. ;
```

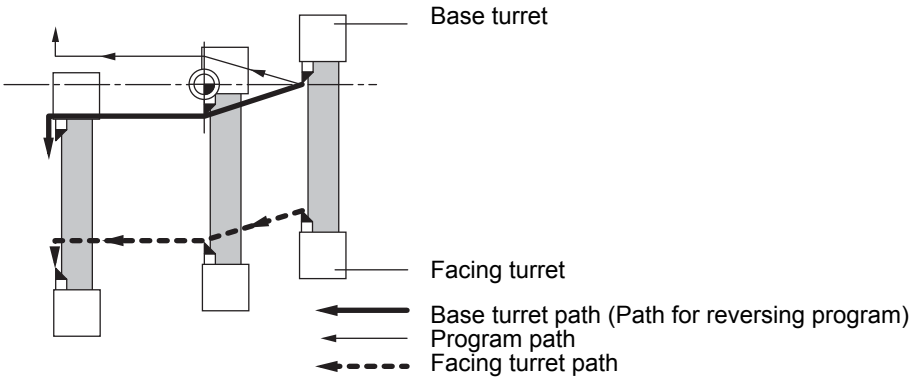


* This figure does not incorporate the tool length.

(2) Incremental value command

The commanded X axis direction reverses, and the base turret moves.

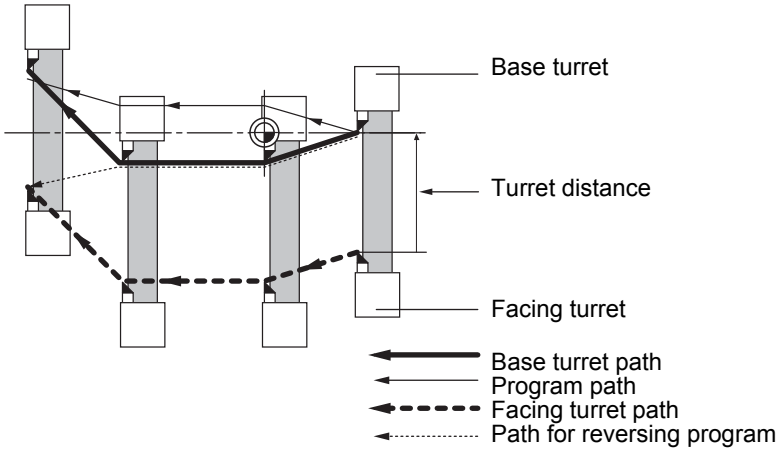
```
T0101  ;
G00 X0. ;
G68 ;              Mirror image for facing tool posts ON
T0202  ;           Facing turret selection
G00 U10. W-30. ;
G01 W-50. F400 ;
   U10. ;
```



* This figure does not incorporate the tool length.

(3) Changing from an incremental value command to an absolute value command
After changing to the absolute value, the same operation as “(1) Absolute value command” takes place.

```
T0101  ;
G00 X0. ;
G68 ;                Mirror image for facing tool posts ON
T0202  ;             Facing turret selection
G00 U10. W-30. ;
G01 W-50. F400 ;
   X20 Z-80. ;
```



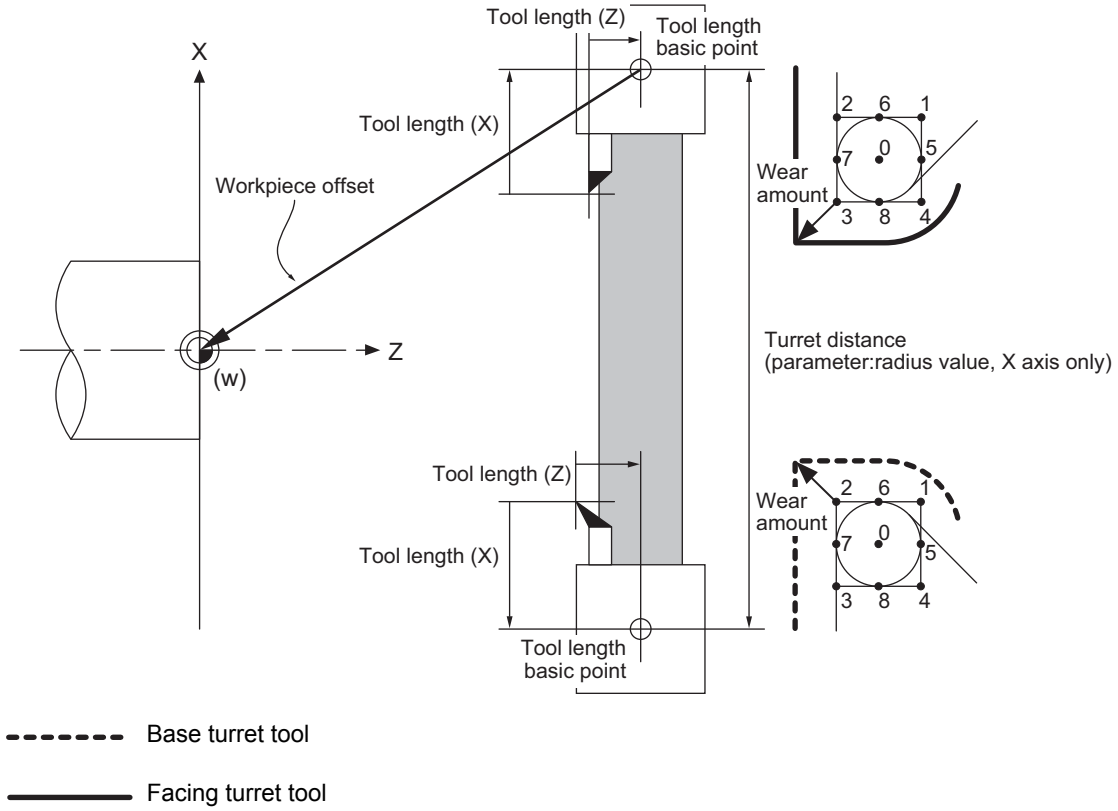
* This figure does not incorporate the tool length.

Displaying the workpiece coordinate values

The indication of the workpiece coordinate value for the axis with the mirror image for facing tool posts valid depends on the MTB specifications (parameter "#1273 ext09/bit3").

	#1273 ext09/bit3 = 0	#1273 ext09/bit3 = 1
Facing turret mirror image mode	The workpiece coordinate values are at the position where the program command value sign has been reversed, and increases/decreases in the same manner as the actual movement, that is, movement on the machine coordinate system. (Indicates coordinate values for facing turret side.)	The workpiece coordinate values are the same as the program command value, and increases/decreases opposite the actual movement, that is, movement on the machine coordinate system.
At startup	Workpiece coordinate value = Workpiece coordinate value before mirror image for facing tool posts starts - distance between cutters (#1202 mirofs)	Workpiece coordinate value = {Workpiece coordinate value before mirror image for facing tool posts starts - distance between cutters (#1202 mirofs)} × (-1)
When movement is commanded while valid	Workpiece coordinate value = Workpiece coordinate value before movement + movement amount of machine coordinate	Workpiece coordinate value = Workpiece coordinate value before movement + movement amount of machine coordinate
When canceled	Workpiece coordinate value = Workpiece coordinate value before mirror image for facing tool posts is canceled - distance between cutters (#1202 mirofs)	Workpiece coordinate value = {Workpiece coordinate value before mirror image for facing tool posts is canceled × (-1)} + distance between cutters (#1202 mirofs)

Tool compensation of facing turret



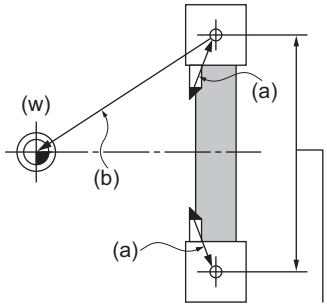
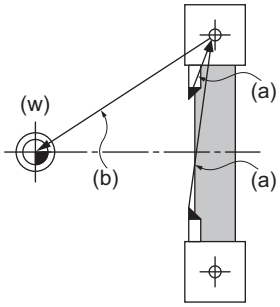
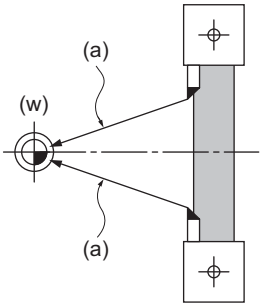
The original tool direction for the facing turret is shown in the drawing.

Whether the facing turret tool can be set to the same direction as the base turret tool depends on the MTB specifications (parameter "#1118 mirr_A").

(1) Tool length offset

The tool length offset amount is the length from the tool nose to the tool length basic point. This also applies for the facing turret. Note that the offset amount setting value differs according to the tool length basic point position as shown below.

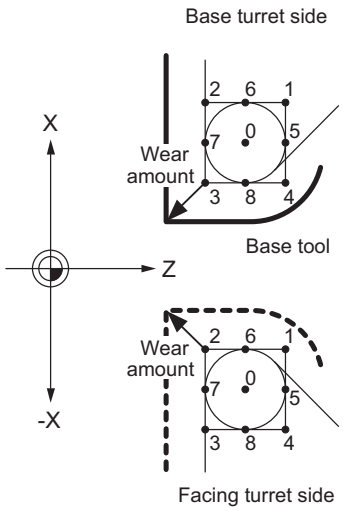
Tool length basic point and tool length offset

	Type A	Type B	Type C
Tool length basic point	Each turret basic point	Base turret basic point	Workpiece face center
Workpiece coordinate zero point (w)	Workpiece face center	Workpiece face center	Workpiece face center
Turret distance	Distance between basic points of both turrets (radius value)	0	0
Workpiece offset (b)	Workpiece coordinate zero point - base turret tool length basic point	Workpiece coordinate zero point - base turret tool length basic point	0
Tool length (a)	Tool length basic point - tool nose position	Tool length basic point - tool nose position	Tool length basic point - tool nose position
Outline drawing Upper: Base turret Lower: Facing turret			

The outline drawing in the table above shows the case when "#1118 mirr_A" is set to 0. When "#1118 mirr_A" is set to 1, the sign of the X axis tool length compensation amount for the facing turret will be reversed.

(2) Tool Nose Wear Compensation

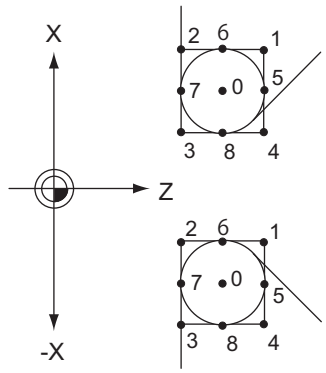
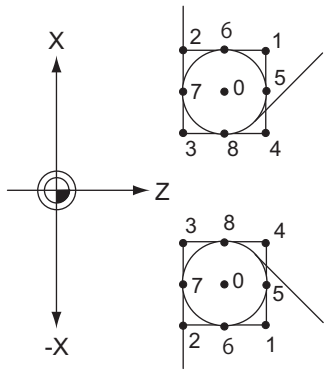
The tool nose wear compensation amount is the length from the current tool nose to the original tool nose. The original tool nose is the tool nose when the tool length offset value was set.

	The drawing shows the case when "#1118 mirr_A" is set to 0. When "#1118 mirr_A" is set to 1, the sign of the X axis tool nose wear compensation amount for the facing turret will be reversed.

(3) Tool nose point with nose R compensation

The tool nose point with nose R compensation is as follows.

Note that if the selected plane differs from when the mirror image for facing tool posts was started, this will be handled as "#1118 mirr_A" = 0 even if it is "#1118 mirr_A" = 1.

#1118 mirr_A = 0	#1118 mirr_A = 1
	

Upper: Base turret

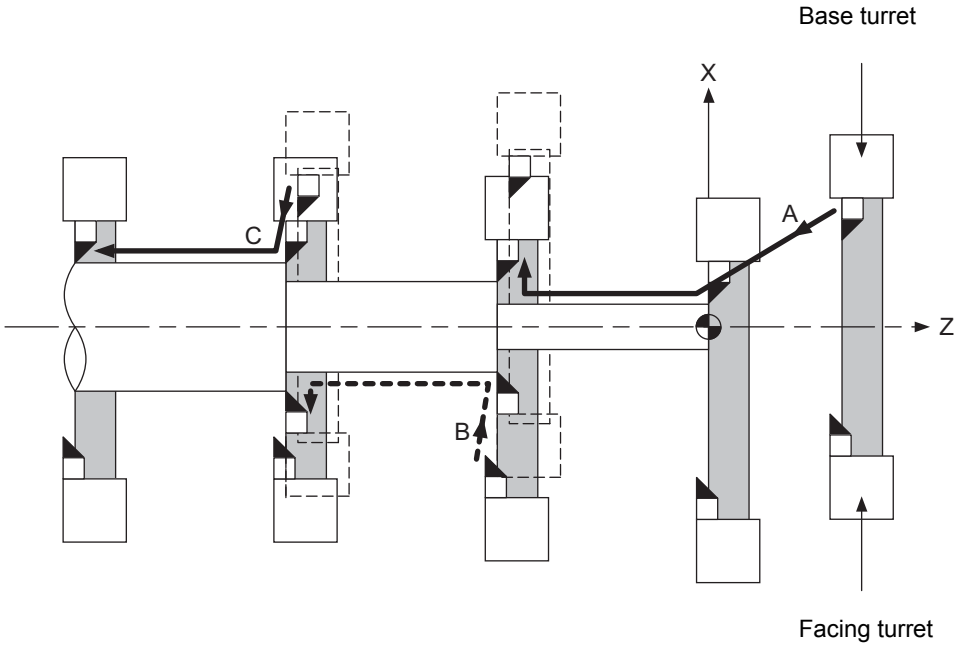
Lower: Facing turret

(4) Examples of setting the tool offset

	Type A				Type B				Type C			
	mirr_A = 0		mirr_A = 1		mirr_A = 0		mirr_A = 1		mirr_A = 0		mirr_A = 1	
	X	Z	X	Z	X	Z	X	Z	X	Z	X	Z
Workpiece offset	-100.	-150.	-100.	-150.	-100.	-150.	-100.	-150.	0.	0.	0.	0.
Turret distance (Setting value)	100.		100.		0.		0.		0.		0.	
Base turret tool length	40.	20.	40.	20.	40.	20.	40.	20.	40.	20.	40.	20.
Base turret tool wear amount	-4.	-2.	-4.	-2.	-4.	-2.	-4.	-2.	-4.	-2.	-4.	-2.
Base turret nose point	3		3		3		3		3		3	
Facing turret tool length	-30.	10.	30.	10.	170.	10.	-170.	10.	70.	-140.	-70.	-140.
Facing turret tool wear amount	2.	-1.	-2.	-1.	2.	-1.	-2.	-1.	2.	-1.	-2.	-1.
Facing turret nose point	2		3		2		3		2		3	

Program example

T0101 ; G00 X10. Z0. ; G01 Z-40. F400 ; X20. ;	Base turret selection	Machining with base turret .....A
G68 ; T0202 ; G00 X20. Z-40. ; G01 Z-80. F200 ; X30. ;	Mirror image for facing tool posts ON Facing turret selection	Machining with facing turret .....B
G69 ; T0101 ; G00 X30. Z-80. ; G01 Z-120. F400 ;	Mirror image for facing tool posts Cancel Base turret selection	Machining with base turret C



Relation with other functions

Reference position return (G28, G30)

Mirror image for facing tool posts will remain valid when moving to the intermediate point.
Mirror image for facing tool posts will be invalidated when moving past the intermediate point and during movement that ignores the intermediate point.
Mirror image for facing tool posts will be validated from the next block.

Machine coordinate system selection (G53)

Mirror image for facing tool posts will be invalidated during the movement.
Mirror image for facing tool posts will be validated from the next block.

Coordinate system setting (G92)

The following will take place on the axis for which mirror image for facing tool posts is valid.

Parameter		Details
#1273 ext09/bit3	0	The "command value × (-1)" coordinate system will be set as the coordinate system on the facing turret side.
	1	The commanded coordinate system will be set as the coordinate system on the facing turret side.

Reading the coordinate position with variables

The following will take place on the axis for which mirror image for facing tool posts is valid.

(1) When previous block is end point coordinate system (#5001, etc.)
The "end point coordinate values on the base turret side in the workpiece coordinate system" will be read in.

(2) For machine coordinate values (#5021, etc.)
The "coordinate values on the base turret side in the machine workpiece system" will be read in.

(3) For machine coordinate values (#5041, etc.)
The "workpiece coordinate value in the mirror image for facing tool posts" will be read in.

(4) For skip coordinate values (#5061, etc.)
If mirror image for facing tool posts is valid when skip is executed, the "workpiece coordinate values in the mirror image for facing tool posts" will be read in.

Resetting

(1) When parameter "#1210 RstGmd/bit14" = 0
The mirror image for facing tool posts will be canceled by reset.

(2) When parameter "#1210 RstGmd/bit14" = 1
The mirror image for facing tool posts state will be maintained even if it is reset.

External mirror image, parameter mirror image

(1) A program error (P371) will occur if mirror image for facing tool posts is applied on the axis during external mirror image or parameter mirror image.

(2) The operation error (M01 1036) alarm will occur if external mirror image or parameter mirror image is applied on the axis during the mirror image for facing tool posts.

Manual interruption

(1) When manual absolute is OFF
If manual interruption is applied on an axis for which mirror image for facing tool posts is valid, the mirror image will not be applied on the interrupt amount. The interrupt movement amount will not be added to the workpiece coordinate values.

(2) When manual absolute is ON
If manual interruption is applied on an axis for which mirror image for facing tool posts is valid, the mirror image will not be applied on the interrupt amount. The interrupt movement amount will be added to the workpiece coordinate values.

Precautions

(1) The balance cut command (G68, G69) cannot be issued when the mirror image for facing tool posts specification is valid in the G code list 6 or 7. The G code list setting depends on the MTB specifications (parameter "#1037 cmdtyp").

(2) If mirror image for facing tool posts is turned ON for an axis in a part system having axes that are exchanged with complex control or in a part system to which the axis control is transferred, an operation error (M01 1035) will occur.
Note that mirror image for facing tool posts can be started after the axis is exchanged with complex control.

(3) A program error (P486) will occur if polar coordinate interpolation, cylindrical interpolation or milling interpolation is commanded while mirror image for facing tool posts is ON.

(4) A program error (P612) will occur if mirror image for facing tool posts is turned ON while the axis is moving with exponential interpolation.

(5) A program error (P371) will occur if the axis to which mirror image for facing tool posts is commanded is a rotation axis.

16

Multi-part System Control

16.1 Timing Synchronization Operation

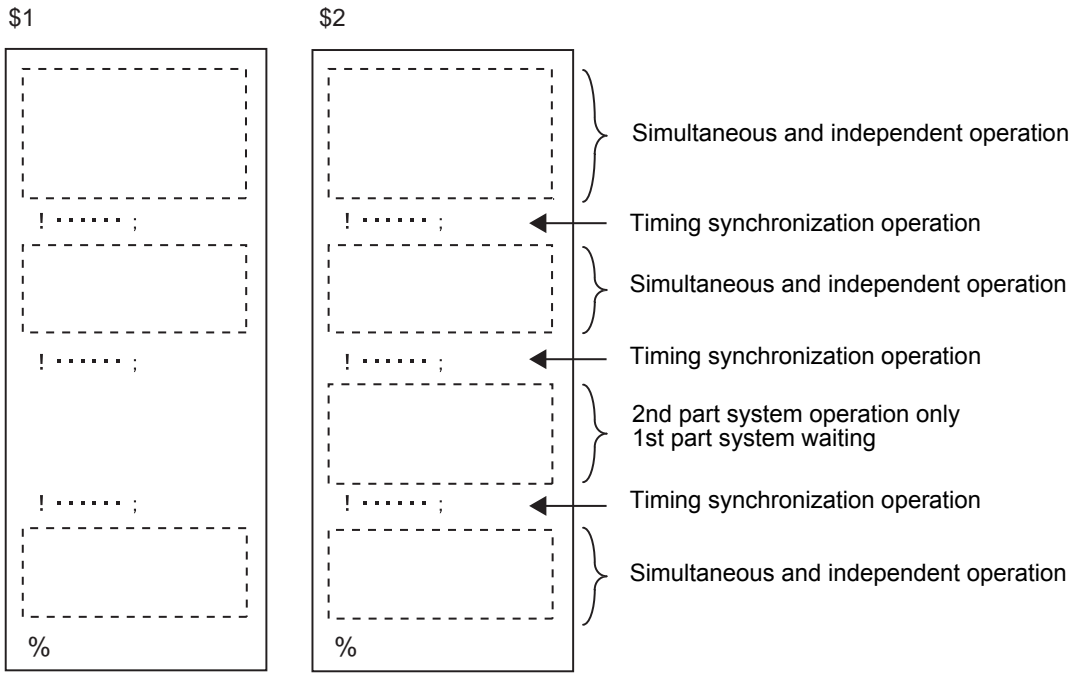
⚠ CAUTION

⚠ When programming a multi-part system, carefully observe the movements caused by other part systems' programs.

16.1.1 Timing Synchronization Operation (! code) !n (!m ...) L

Function and purpose

The multi-axis, multi-part system complex control CNC system can simultaneously run multiple machining programs independently. The synchronization-between-part systems function is used in cases when, at some particular point during operation, the operations of 1st and 2nd part systems are to be synchronized or in cases when the operation of only one part system is required.
When timing synchronization is executed in the 1st part system ($1) and the 2nd part system ($2), operations will be as follows.



Command format

!n (!m ...) L_ ; ... timing synchronization operation

!n, !m, ...	Timing synchronization operation (!) and part system No. (n:1 - number of part system that can be used) Follows the settings of the parameter "#19419 Timing sync system" if part system number is omitted.
L	Timing Synchronization Operation No. 0 to 9999

Also valid between three part systems and more.

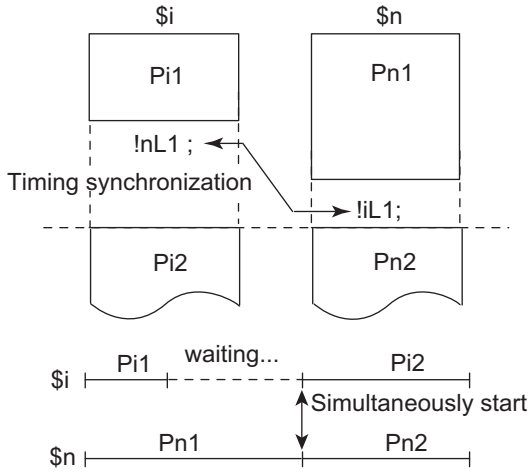
Detailed description

(1) Timing synchronization between part systems during automatic operation

If !n L__ is commanded from a part system (i), operation of the part system i program will wait until !i L_ is commanded from the part system n program.

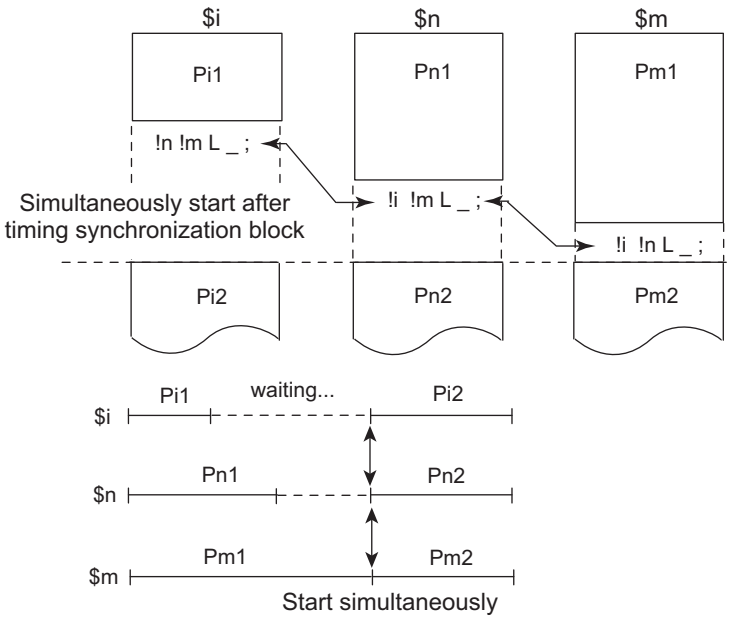
When !i L_ is commanded, the programs for the two part systems will start simultaneously.

(a) Timing synchronization between 2 part systems



(b) Timing synchronization between 3 part systems and more

Similarly with systems with 3 and more parts, when all part systems that are included in the timing synchronization operation reach timing synchronization block, these part systems start operating from the next block simultaneously.



(2) The timing synchronization operation is normally issued in a single block. However, if a movement command or M, S or T command is issued in the same block, whether to synchronize after the movement command or M, S or T command or to execute the movement command or M, S or T command after synchronization will depend on the MTB specifications (#1093 Wnvfin).

#1093 Wmvfin

0 : Wait before executing movement command.

1 : Wait after executing movement command.

(3) If there is no movement command in the same block as the timing synchronization operation, when the next block movement starts, synchronization may not be secured between the part systems. To synchronize the part systems when movement starts after waiting, issue the movement command in the same block as the timing synchronization operation.

(4) The L command is the timing synchronization identification No. The same Nos. are waited but when they are omitted, the Nos. are handled as L0.

(5) "SYN" will appear in the operation status section during timing synchronization operation. The timing synchronization operation signal will be output to the PLC I/F.

(6) In a timing synchronization operation, other part system to be waited for is specified but the own part system can be specified with the other part system.

(7) The timing synchronization operation of a specific part system can be ignored depending on the MTB specifications.
Operation will be determined by the combination of the timing synchronization operation ignore signal and parameter "#1279 ext15/bit0".
For setting combination, refer to "Time synchronization when timing synchronization ignore is set".
For the specifications of the machine you are using, see the instructions issued by the MTB.

Relation with other functions

Use timing synchronization operation between the part systems command and the balance cut command

If one part system is on standby for synchronization with the timing synchronization operation between part systems and the other part system command enters the synchronization standby state with the balance cut command (G15), both part systems will be in the standby state, and will not shift to the next block. Command so that timing synchronization with balance cut command (G15) and standby for waiting with the timing synchronization operation between part systems do not occur simultaneously.

Timing synchronization operation between part systems during balance cut mode

When the timing Synchronization operation between part systems is issued in balance cut mode, which is handled as an invalid command and will not be waited for.

Precautions

(1) When the M code can be used, both the M code and ! code can be used.

(2) While the timing synchronization operation M code is valid, if one part system is standing by with an M code, an alarm will occur if there is a ! code timing synchronization operation command in the other part system.

(3) While the timing synchronization operation M code is valid, if one part system is standing by with a ! code, an alarm will occur if there is an M code timing synchronization operation command in the other part system.

(4) The timing synchronization operation (! code, M code) in the machining program can be ignored with the timing synchronization operation ignore signal. (This depends on the MTB specifications.) Operation with a single part system is possible without deleting the timing synchronization operation (! code, M code) in the machining program.

(5) When macro interruption is carried out in a part system waiting, the part system can stop while waiting even if the conditions for time synchronization are met. In this case, you will be able to continue the program, ignoring the timing synchronization with timing synchronization operation ignore signal.
For details, contact the MTB.

(6) "SYN" will appear in the operation status section during timing synchronization operation.

16.1.2 Timing Synchronization Operation with Start Point Designated (Type 1) ; G115

Function and purpose

The part system can wait for the other part system to reach the start point before starting itself.
The start point can be set in the middle of a block.
Even when specifications are available for 3 part systems and more, G115 command cannot be used between 3 part systems. A program error (P33) will occur.

Command format

!n L__ G115 X__ Z__ C__ ;

!n	Timing synchronization operation (!) and part system No. (n:1 - number of part system that can be used) Part systems follow the settings of the parameter "#19419 Timing sync system" if the number is omitted.
L	Timing Synchronization Operation No. 0 to 9999 (It will be regarded as "L0" when omitted.)
G115	G command
X Z C	Start point (Command by axis and workpiece coordinate value)

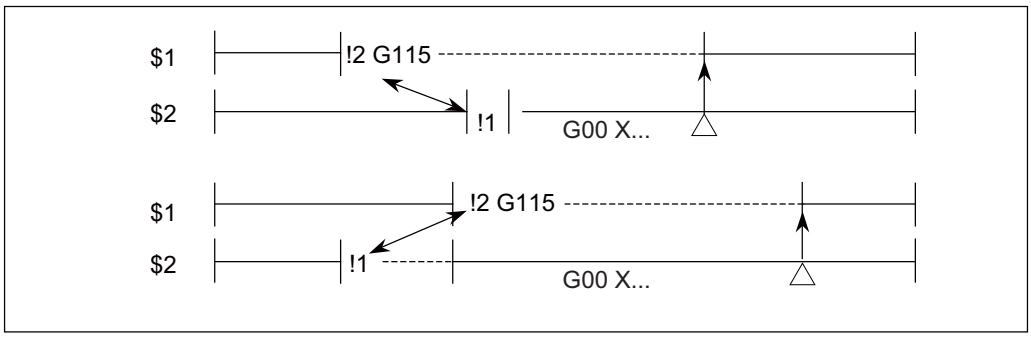
Detailed description

(1)Designate the start point using the workpiece coordinates of the other part system (ex. $2).

(2)The start point check is executed only for the axis designated by G115.
(Example) !L2 G115 X100. ;
Once the other part system reaches X100, the own part system (ex. $1) will start. The other axes are not checked.

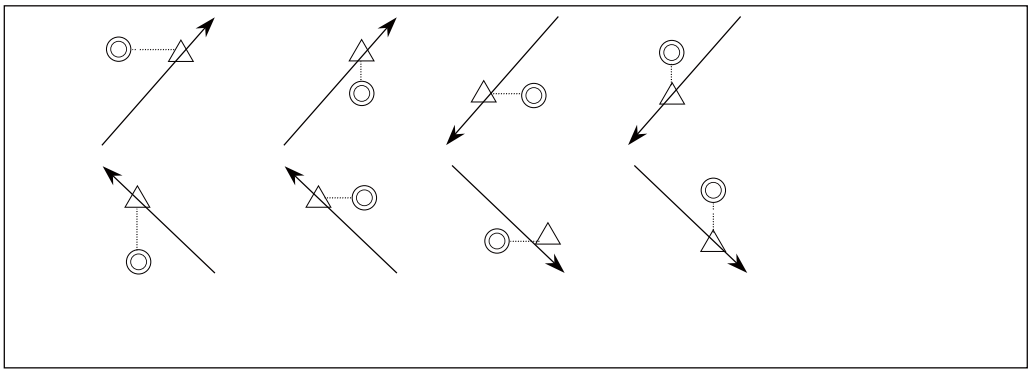
(3)The other part system starts first when timing synchronization operation is executed.

(4)The own part system waits for the other part system to move and reach the designated start point, and then starts.



Timing synchronization Designated start point

(5) When the start point designated by G115 is not on the next block movement path of the other part system, the own part system starts once all the designated axis of the other part system has reach the designated start point.



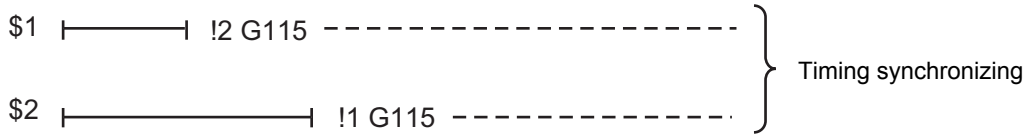
Movement Designated start point Actual start point

(6) After waiting, if the start point cannot be obtained with movement command of the other timing synchronization block, the operations depend on the MTB specifications (parameter "#1229 set01/bit5").

(a) When the parameter is ON
Wait till the own part system reaches the start point by moving after the next block.

(b) When the parameter is OFF
When the next block finishes moving, the own part system will start.

(7)The timing synchronization status continues when the G115 command has been duplicated between part systems. (Operations will not restart.)



(8) The single block stop function does not apply for the G115 block.

(9) A program error (P32) will occur if an address other than an axis is designated in G115 command block.

(10) In the timing synchronization operation, other part system to be waited for is specified but the own part system can be specified with the other part system.

(11) The timing synchronization operation of a specific part system can be ignored depending on the MTB specifications. Operation will be determined by the combination of the timing synchronization operation ignore signal (PLC signal) and parameter "#1279 ext15/bit0".
For setting combination, refer to "Time synchronization when timing synchronization ignore is set".
For the specifications of the machine you are using, see the instructions issued by the MTB.

Relation with other functions

Use the start point designation synchronization command and the balance cut command

If one part system is on standby for synchronization with the start point designation synchronization command and the other part system enters the synchronization standby state with the balance cut command (G15), both part systems will be in the standby state, and will not shift to the next block. Command so that timing synchronization with balance cut command (G15) and timing synchronization with the start point designation synchronization command do not occur simultaneously.

Start point designation synchronization command in balance cut mode

When the timing Start point designation timing synchronization command is issued in balance cut mode, it is handled as an invalid command and will not be waited for.

Precautions

(1) Parameter ”#1093 Wmvfin” that selects the timing of the timing synchronization operation and commands on the same block does not work for the start point command block (G115/G116). After synchronization. the start point check will be executed by G115/G116.

(2) Be careful about the timing when interrupting during the time synchronization of G115/G116. For example, assume interruption with the macro interrupt type 1 while a part system is waiting for time synchronization with G116. In this case, if there is a movement command or MSTB command in the interrupt program, the program will continue after the interrupt program completes without waiting for the start point.

(3) The L command is the timing synchronization identification No. The same Nos. are waited but when they are omitted, the Nos. are handled as L0.

16.1.3 Timing Synchronization Operation with Start Point Designated (Type 2) ; G116

Function and purpose

The own part system can make the other part system to wait until it reaches the start point.
The start point can be set in the middle of a block.
When specifications are available for 3 part systems and more, if G116 is commanded, the other designated part systems will start at the same time.

Command format

```
!n L__   G116   X__   Z__   C__ ;
```

!n	Timing synchronization operation (!) and part system No. (n:1 - number of part system that can be used) Part systems follow the settings of the parameter "#19419 Timing sync system" if the number is omitted.
L	Timing Synchronization Operation No. 0 to 9999 (It will be regarded as "L0" when omitted.)
G116	G command
X Z C	Start point (Command by axis and workpiece coordinate value)

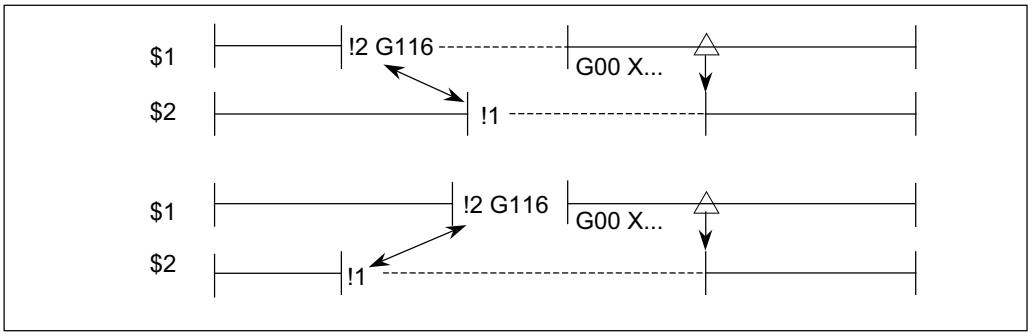
Detailed description

(1) Designate the start point using the workpiece coordinates of the own part system (ex. $1).

(2) The start point check is executed only for the axis designated by G116.
(Example) !L1 G116 X100. ;
Once the own part system reaches X100, the other part system (ex. $2) will start. The other axes are not checked.

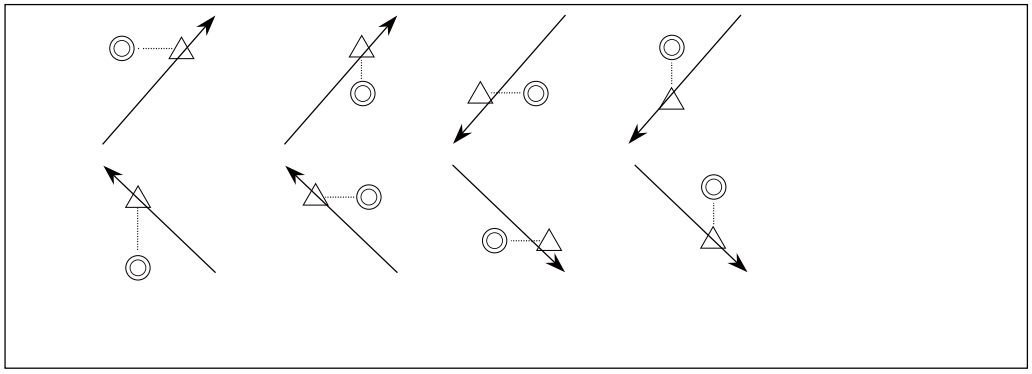
(3) The own part system starts first when timing synchronization operation is executed.

(4) The other part system waits for the own part system to move and reach the designated start point, and then starts.



Timing synchronization Designated start point

(5) When the start point designated by G116 is not on the next block movement path of own part system, the other part system starts once all the designated axes of the own part system has reach the designated start point.



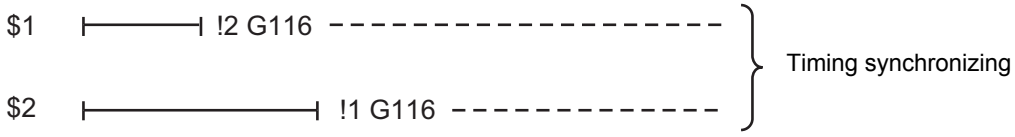
Movement Designated start point Actual start point

(6) If the start point cannot be obtained with the movement of the own part system to the next block, the operations depend on the MTB specifications (parameter "#1229 set01/bit5").

(a) When the parameter is ON
The own part system will have a program error (P511) before moving.

(b) When the parameter is OFF
When the next block finishes moving, the other part system will start.

(7) The timing synchronization status continues when the G116 command has been duplicated between part systems. (Operations will not restart.)



(8) When G116 is commanded between the 3 part systems, two of the other part systems will start at the same time.

(9) The single block stop function does not apply for the G116 block.

(10) A program error (P32) will occur if an address other than an axis is designated in G116 command block.

(11) In the timing synchronization operation, other part system to be waited for is specified but the own part system can be specified with the other part system.

(12) The timing synchronization operation of a specific part system can be ignored depending on the MTB specifications. Operation will be determined by the combination of the timing synchronization operation ignore signal (PLC signal) and parameter "#1279 ext15/bit0".
For setting combination, refer to "Time synchronization when timing synchronization ignore is set".
For the specifications of the machine you are using, see the instructions issued by the MTB.

Relation with other functions

Refer to "Start point designation timing synchronization (Type 1) ; G115".

Precautions

Refer to "Start point designation timing synchronization (Type 1) ; G115".

16.1.4 Timing Synchronization Operation Function Using M codes ; M***

Function and purpose

The timing synchronization operation function between part systems is conventionally commanded with the "!" code, but by using this function, the part systems can be waited with the M code commanded in the machining program. If the timing synchronization operation M code is commanded in either part system during automatic operation, the system will wait for the same M code to be commanded in the other part system before executing the next block. The timing synchronization operation M code is used to control the timing synchronization operation between the 1st part system and 2nd part system. Whether the timing synchronization operation M code can be used depends on the MTB specifications.

Command format

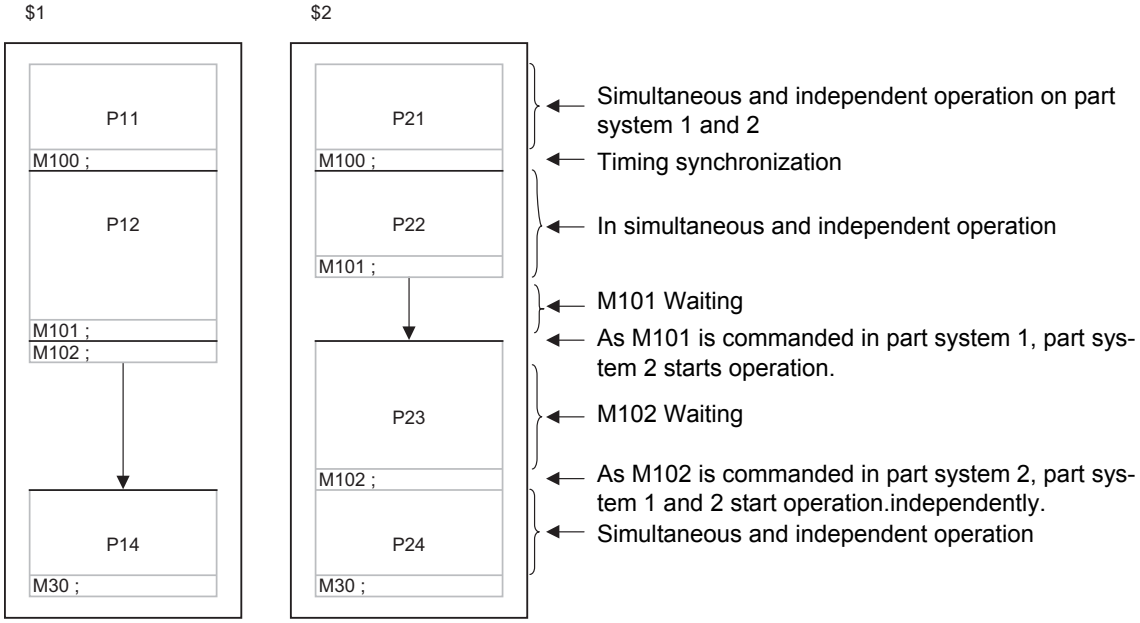
M* ;**

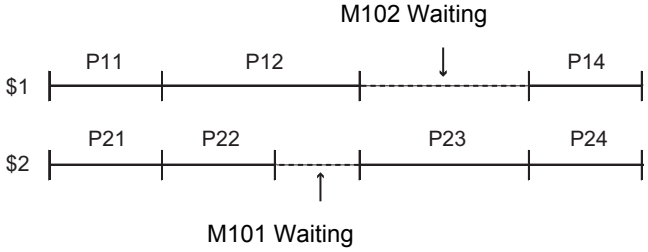
***	Timing synchronization operation M code

M code used for timing synchronization depends on the MTB specifications (parameter "#1310 WtMmin)", "#1311 WtMmax").

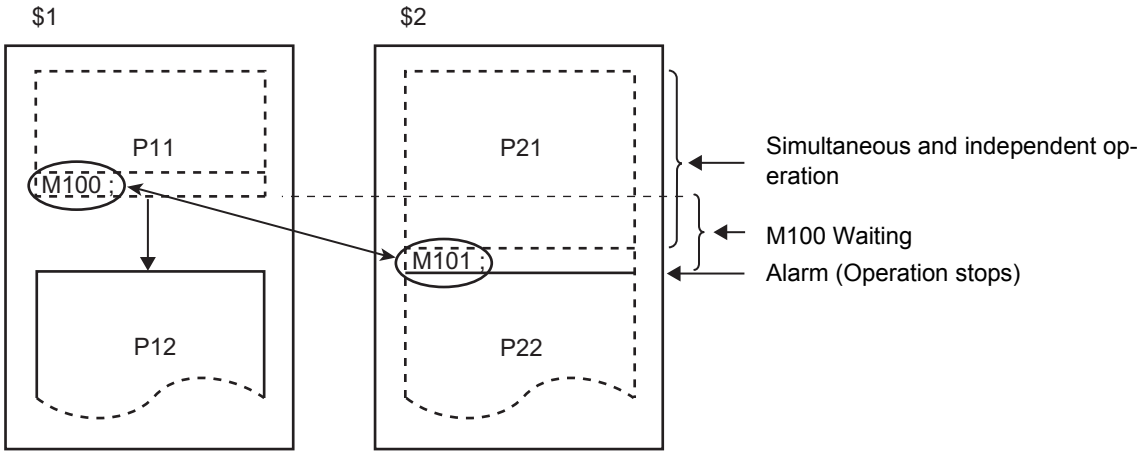
Detailed description

(1) When the timing synchronization operation M code is commanded in the machining program, the two part systems will be waited and operation will start in the commanded block. If the timing synchronization operation M code is commanded in either part system during automatic operation, the system will wait for the same M code to be commanded in the other part system before executing the next block.





(2) When the timing synchronization operation M code has been commanded in one part system, and the part system is standing by for waiting, an alarm will occur if a different M code is commanded in the other part system.



(3)The part systems are waited with the M code following the parameters below.
These settings depend on the MTB specifications. Refer to these settings. For details, refer to the specifications of your machine.

(a) M code range designation parameter (M code minimum value <= M code <= M code maximum value)

#	Item		Details	Setting range
1310	WtMmin	Timing synchronization M code ABS. MIN.	The minimum value of the M code. If the setting value is "0", the timing synchronization operation M code will be ignored.	0, 100 ~ 99999999
1311	WtMmax	Timing synchronization M code ABS. MAX.	The maximum value of the M code. If the setting value is "0", the timing synchronization operation M code will be ignored.	0, 100 ~ 99999999

This function is invalid if either parameter is set to "0".
The timing synchronization operation M code cannot be used if the M code maximum value is smaller than the minimum value.
When the timing synchronization operation M code is valid, both the M code and ! code can be used for timing synchronization operation.

(b) Timing synchronization operation method parameters

#	Item		Details	Setting range
1279 (PR)	ext15 (bit0)	Method for timing synchronization operation between part systems	Select an operation for timing synchronization operation between part systems. 0: If one of the part systems is not in automatic operation, ignore the timing synchronization operation and execute the next block. 1: Operate according to the timing synchronization operation ignore signal. If the timing synchronization operation ignore signal is "1", the timing synchronization operation will be ignored. If "0", the part systems will be waited.	0 / 1

Depending on the timing synchronization operation method selection parameter and timing synchronization operation ignore signal combination, the timing synchronization operation will be determined by the parameters, regardless of the command format ("!" code and M code).
This parameter requires the CNC to be turned OFF after the settings. Turn the power OFF and ON to enable the parameter settings.

#	Item		Details	Setting range
1093	Wmvfin	Method for timing synchronization operation between part systems	Parameter to designate the timing synchronization operation between part systems method when using multi-part systems. When there is a movement command in the timing synchronization operation (!, M) block: 0 : Wait before executing movement command. 1 : Wait after executing movement command.	0 / 1

Relation with other functions

Refer to "Timing Synchronization Operation (! code);!n (!m ...) L"

Precautions

For precautions for time synchronization, also refer to "Timing Synchronization (!code);!n (!m ...) L"

(1) When timing synchronization operation with the M code, always command the M code in an independent block.

(2) When standing by after commanding the timing synchronization operation M code in one part system, an alarm will occur if a different M code is commanded in the other part system. Operation will stop in both part systems.

(3) The timing synchronization operation (! code, M code) in the machining program can be ignored with the timing synchronization operation ignore signal. (This depends on the MTB specifications.) Operation with a single part system is possible without deleting the timing synchronization operation (! code, M code) in the machining program.

(4) Unlike other M codes, the timing synchronization operation M code does not output code signals and strobe signals.

(5) When the M code can be used, both the M code and ! code can be used.

(6) While the timing synchronization operation M code is valid, if one part system is standing by with an M code, an alarm will occur if there is a ! code timing synchronization operation command in the other part system.

(7) While the timing synchronization operation M code is valid, if one part system is standing by with a ! code, an alarm will occur if there is an M code timing synchronization operation command in the other part system.

(8) If there is a timing synchronization operation with M code after the 3rd part system, an alarm will occur.

(9) The G115 and G116 commands cannot be used when waiting with the M code.

(10) If the M code command Nos. are overlapped, the order of priority will be M code macro, M command synchronous tapping, timing synchronization operation M code and normal M code.

(11) When macro interruption is carried out in a part system waiting, the part system can stop while waiting even if the conditions for time synchronization are met. In this case, you will be able to continue the program, ignoring the timing synchronization with timing synchronization operation ignore signal. For details, contact the MTB.

(12) "SYN" will appear in the operation status section during timing synchronization operation.

16.1.5 Time Synchronization When Timing Synchronization Ignore Is Set

Function and purpose

Turning on the timing synchronization operation ignore signal makes it possible to ignore the timing synchronization operation of that part system.
With a 2-part system, if the timing synchronization operation ignore signal of the other part system is ON, timing synchronization is not executed. In the following section, a 3-part system is used as an example to make it easier to understand the functions.

This signal is also used in the following functions.

- Timing synchronization (! code, M code)
- Start point timing synchronization (G115, G116)
- Balance cut (G15) Lathe system only

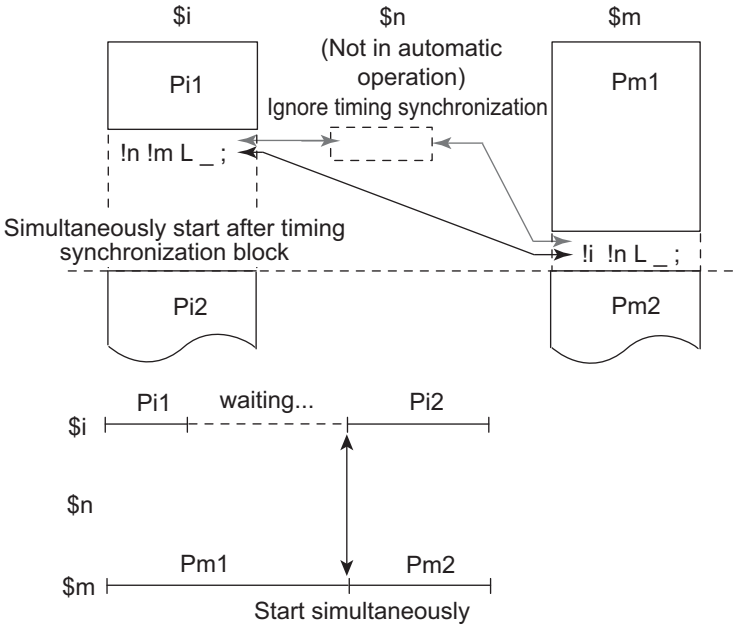
Note

(1) For sub part system control function, refer to "16.9 Sub Part System Control".

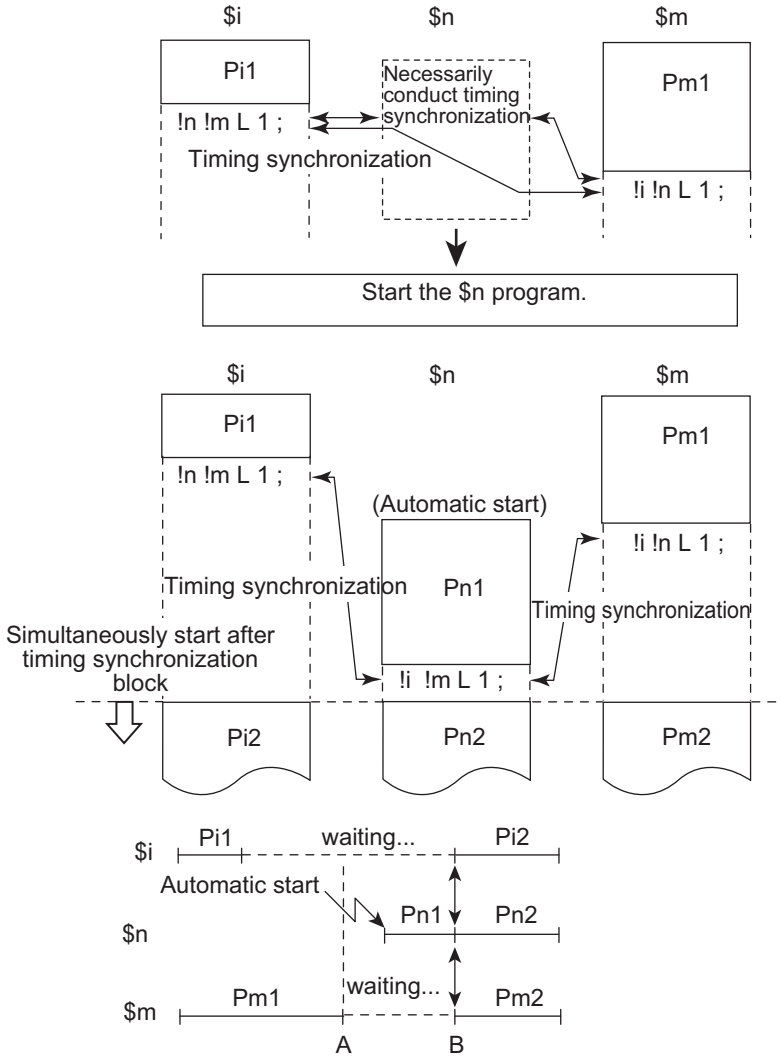
		Timing synchronization operation ignore signal (PLC signal)	
		OFF	**ON**
Parameter (#1279 ext15/bit0)	0	(1) Ignores the timing synchronization with a part system not in automatic operation	
	1	(2) Does not ignore the timing synchronization regardless of whether or not a part system is in automatic operation (the timing synchronization is executed until the conditions for timing synchronization are established.)	(3) Ignores the timing synchronization regardless of whether or not a part system is in automatic operation (ignores the timing synchronization command for the part system with the timing synchronization ignore signal ON and the timing synchronization operation for that part system)

The following operation diagram gives an example of ! code.

(1) A case that "Ignores the timing synchronization with a part system not in automatic operation"



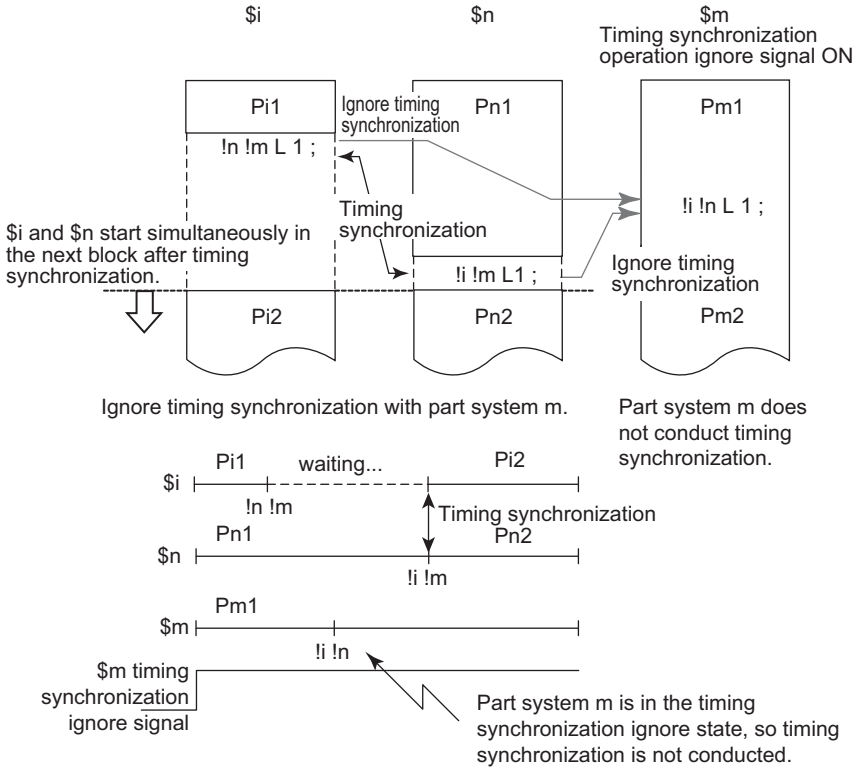
(2) A case that "Does not ignore the timing synchronization regardless of whether or not in automatic operation"



A: When timing synchronization operation between part systems (parameter "#1279 ext15/bit0" = 1), the timing synchronization status continues until the conditions for timing synchronization are established.

B: Part system n is automatically started. If the conditions for timing synchronization are established, the next block will start.

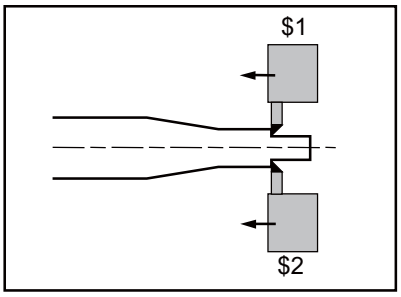
(3) A case that "Ignores the timing synchronization regardless of whether or not in automatic operation"



16.2 Balance Cut ; G15,G14

Function and purpose

The timing for starting the operation of the 1st part system turret and 2nd part system turret can be synchronized.



When workpiece that is relatively long and thin is machined on a lathe, deflection may result, making it impossible for the workpiece to be machined with any accuracy.
In this case, if the cutters are applied simultaneously from both sides of the workpiece and the workpiece is machined while synchronizing these (balance cut), the slack can be suppressed. This method has an additional advantage: since the workpiece is machined by two tools, the machining time is reduced.
With this function, the movement of two turrets belonging to different part systems can be completely synchronized, so the following type of machining can be carried out easily.
When commanding balance cut on any part system 2, designate the part system as G15 command block after ! code.

Command format

Balance cut command ON

```
!n L_ G15
```

!n	Part system number for commanding balance cut (n:0 - number of a part system that can be used) Follows the parameter "#19419 Timing sync system" if a part system number is omitted. (can be omitted)
L	Timing synchronization operation No. 0 to 9999 (can be omitted) It will be regarded as "L0" when omitted.

Balance cut command OFF

```
G14
```

Detailed description

(1) Command G15 and G14 in independent blocks.

(2) G15 and G14 commands are modals. In the CNC's initial state, the G14 balance cut command is OFF.

(3) When G15 is commanded, movement will standby until G14 is commanded or until the modal information is cleared by the reset signal. Whether to execute timing synchronization on cutting feed blocks or on all blocks depends on the MTB specifications (parameter "#1244 set16/bit2").

(4) If G15 or G14 is commanded in one part system, movement will not advance until the same G code is commanded in the other part system.

(5) When subprogram call, macro call or PLC interrupt is carried out, balance cut will apply to each command in a block that configures the subprogram.

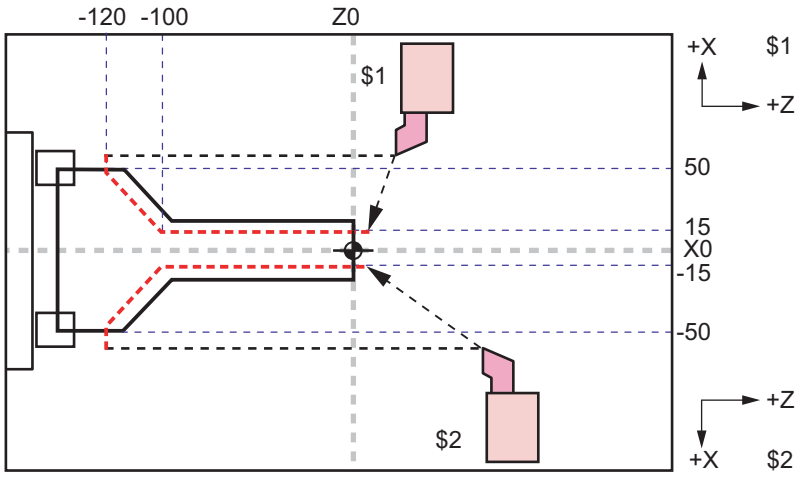
(6) After G14 is commanded, each part system will operate independently.

(7) The number of the other part system that is to be waited on is specified as the part system number, but the own part system can be specified with the other part system.

(8) The timing synchronization command with the balance cut on that part system can be ignored depending on the MTB specifications.
Operation will be determined by the combination of the timing synchronization operation ignore signal and parameter "#1279 ext15/bit0".
For setting combination, refer to "Time synchronization when timing synchronization ignore is set".
For the specifications of the machine you are using, see the instructions issued by the MTB.

Program example

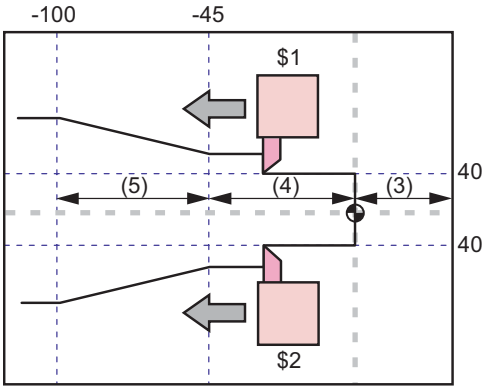
An example of a program for machining with a 1-spindle 2-turret CNC lathe while simultaneously applying the cutters from the top and bottom of a thin long workpiece using balance cut is shown below.



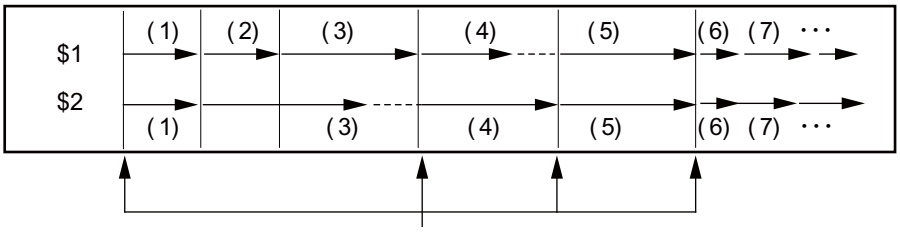
1st part system program ($1)	2nd part system program ($2)
G28 XZ ; S100 T0101 ; G15 ; G00 X15 Z3 ; G01 Z-100 F0.2 ; X50 Z-120 ; X52 ; G14 ; G28 XZ ; M30 ;	G28 XZ ; T0101 ; G15 ; G00 X15 Z3 ; G01 Z-100 F0.2 ; X50 Z-120 ; X52 ; G14 ; G28 XZ ; M30 ;

Operation example

The following is an operation example of balance cut on the 1st and the 2nd part systems as per the specification for executing timing synchronization on cutting feed blocks only.



<1st part system>	<2nd part system>	
:	:	
G15	G15	(1)
S200		(2)
G00 X40. Z-2.	G00 X40. Z-2.	(3)
G01 W47. F10.	G01 W47. F5.	(4)
G01 U40. W55.	G01 X80. Z100. F10.	(5)
G14	G14	(6)
G00 X100.	G00 X100.	(7)
:	:	



Timing synchronization section

(1) Balance cut is turned ON with the G15 command.

(2)(3) The S command and rapid traverse command are not waited, so the operation waits at the head of (4).

(4) The 1st part system will finish first, but since the next block is a cutting feed command, the operation will wait at the head of (5).

(5) Cutting will start with the 1st and 2nd part systems together.

(6) Balance cut is turned OFF with the G14 command.

(7) Each part system will operate independently after this.

Relationship with Other Functions

(1) A program error (P481) will occur if G15 or G14 is commanded in milling mode.

(2) A program error (P29) will occur if multi-part system simultaneous thread cutting (G76.1/G76.2) is commanded in balance cut mode.

Precautions

Synchronization during movement

This function simultaneously starts the block for both part systems. The following synchronization will change according to the movement amount and feedrate, etc., and thus cannot be guaranteed. To move in complete synchronization, the movement amount and feedrate must be set to the same values. Note that even if the previous conditions match, there may be cases when accurate machining, such as thread cutting, is not possible.

Number of blocks in balance cut mode

When commanding balance cut, make sure that the same total number of blocks to wait for are set between G15 and G14 on the own part system and on the other part system.
- When the target of timing synchronization is cutting feed blocks: Make sure that the same total number of cutting feed blocks are set between G15 and G14.
- When the target of timing synchronization is all blocks: Make sure that the same total number of all blocks are set between G15 and G14.

If G14 is commanded in one part system, the other part system will move to the block ahead without waiting.

<1st part system>	<2nd part system>	
:	:	
N20 G15	N20 G15	
N30 G00 X40. Z0.	N30 G00 X-40. Z250.	
N40 G01 W-30. F1000	N40 G01 W-130. F500	
N50 G01 U40. W-70.	N50 G01 X-80. Z50. F1000	
N60 G01 W-20.	**N60 G14**	← On the 2nd part system side, balance mode has been cancelled with G14. Therefore, N 60 block on the 1st part system side moves to the block ahead without waiting for the 2nd part system.
N70 G14	N70 S200	
N80 G01 X120. Z30.	N80 G00 X-100.	

Use with timing synchronization operation between part systems

If one part system is on standby for synchronization with the timing synchronization operation between part systems and the other part system enters the synchronization standby state with the G15 command, both part systems will be in the standby state, and will not shift to the next block. Command so that standby for waiting for G15 and standby for waiting with the timing synchronization operation between part systems do not occur simultaneously.

Timing synchronization operation between part systems during balance cut mode

When the timing Synchronization operation between part systems is issued during the balance cut mode, which is handled as a command of one block without movement and will not be waited for.

For one part system

A program error (P39) will occur if G15 or G14 is commanded in one part system.

Conditions for ignoring G15 and G14

If G14 is commanded when G15 is not commanded (when balance cut is OFF), the G14 block will be handled as one that has no process.

Macro interruption during timing synchronization (Type 1)

Do not carry out macro interruption (Type 1) in a part system waiting with G15 command. Doing so will result in the following operation.

Interrupted part systems:	After an interrupt program completes, timing synchronization with balance cut command G01 will not be executed.
Part systems not interrupted:	Stops at the G15 or G01 command block.

Macro interruption during timing synchronization (Type 2)

When carrying out macro interruption (Type 2) in a part system waiting with balance cut, attend to the contents of the interrupt program.
If the G01 command is in the interrupt program, the timing synchronization is executed in the block. Therefore, the block may be shifted slightly, compared to the case where there is no interruption.

Balance cut mode at resetting

Modals at resetting follow the modal code reset parameter "#1210 RstGmd".

16.3 Mixed Control

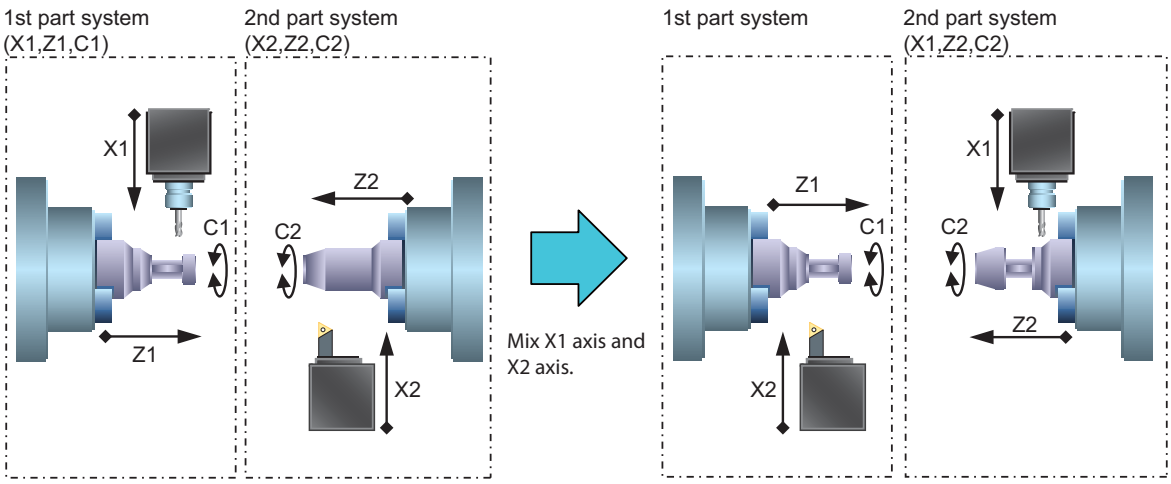
16.3.1 Cross Axis Control ;G110

Function and purpose

This function enables any axis to be replaced by another axis between part systems.
This makes it possible to perform operations which are not possible with regular axis configurations: for instance, tools which are provided only on part system 1 can be used for machining on part system 2.

This manual contains descriptions where part system Nos. have been added after the axis names in order to identify which part system the axes belong to (such as X1). In terms of designating the program commands, however, a single letter is used to designate the axis address just as before.

There are two types of mixed control (cross axis control) depending on the command method: Mixed control (cross axis control) I (G command), and mixed control (cross axis control) II (PLC signal). The method is chosen depending on the MTB specifications (parameter "#1280 ext16/bit4").



Command format

Mixed control (cross axis control) I

G110 Axis name1 Axis name2 Axis name3 ...;

Axis name1 Axis name2 Axis name3 ...	All axes which exist in the part system after commanded (axis name set in the parameter "#1022 axname2")

G110 needs to be commanded for a part system which occurs the existing axis switching (or move) by other part system's G110 command.

Detailed description

(1) 2-digit axis name
For two or more part systems, the same axis name may exist in each part system. To distinguish the name, display 2-digit axis name set by the parameter "#1022 axname2".

(2) Displaying the switched coordinate value
The basic specification parameter "#1280 ext16/bit2,bit6" can switch the axis name and coordinate value (such as the workpiece coordinate position, and the machine coordinate position).
This parameter can also set the command to move the axis (such as moving C axis of 1st part system to 2nd part system).

Following items are switched for display

Operation screen (except restart search)	Counter	Each counter(*), machine status animation
	Trace	Each counter(*), display mode, drawing area
	Coordinate system	Each coordinate system offset
Setup screen	Counter	Each counter(*)
	Coordinate system	Each coordinate system offset

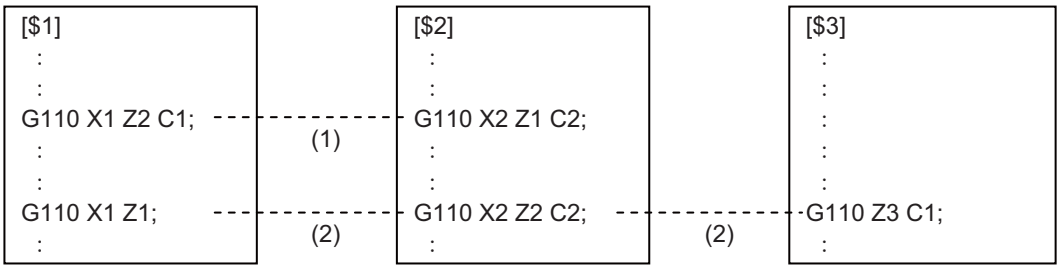
(*) The setting of the basic specification parameter "#1280 ext16/bit6" prevents the displays of machine position counters from being interchanged.

The following is displayed when X axis of 1st part system and X axis of 2nd part system are switched.

(3) G110 command carries out timing synchronization operation until it becomes mixing valid status by other part system's G110 command. On this occasion, "CRS" appears in the operation status of the operation screen to indicate that the machine is in timing synchronization operation by mixed control. After that, mixing completes when the part system to control against all axes to be mixed which relates to that part system, then "CRS" is cleared, and the machining program continues.

Program example

Initial axis configuration
1st part system: X1, Z1, C1
2nd part system : X2, Z2, C2
3rd part system: Z3



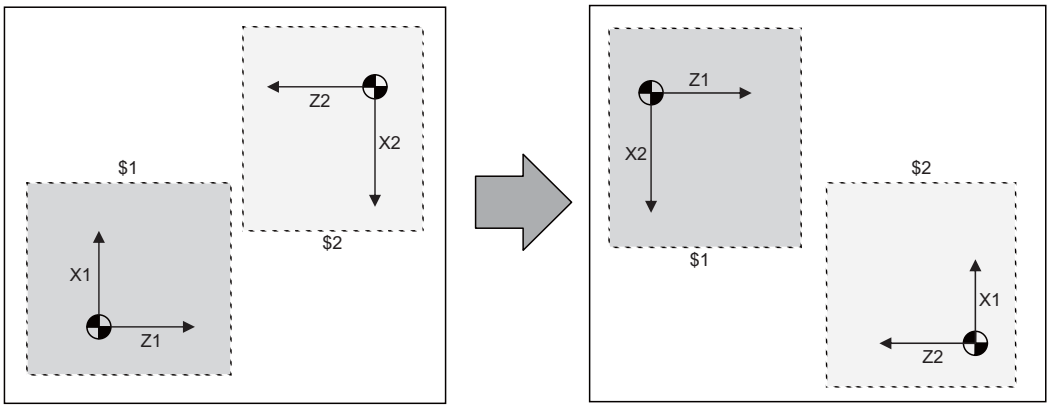
(1) 1st part system and 2nd part system execute timing synchronization operation with each other, and Z1 axis is switched with Z2 axis. Timing synchronization operation is not executed for 3rd part system as all axes of 1st part system and 2nd part system can mix due to 1st part system and 2nd part system's G110 commands.

(2) "1st part system and 2nd part system" and "1st part system and 3rd part system" execute timing synchronization operation with each other. Z1 axis and Z2 axis are replaced to restore the initial state, and move C1 axis to 3rd part system. On this occasion, if G110 command of 2nd part system is executed lastly, 3rd part system does not wait for 2nd part system's G command, and mix 1st part system and C1 axis first and then the program continues.

Relationship with other functions

(1) Coordinate system

The coordinate information, such as a reference position, machine coordinate zero point and workpiece coordinate zero point, is decided for each axis. Therefore, the coordinate systems are also switched when the axes are switched by the mixed control.



(2) Plane selection

For the part system where the axes are switched by mixed control, the plane is configured with the switched axis. On this occasion, an axis under the mixed control configures the plane with the axis address set by the parameter "#1023 crsadr".

(Example) Suppose the following is set:1st part system axis configuration (X1, Z1, C1, Y1) and 2nd part system axis configuration (X2, Z2, C2)

"#1023 crsadr" of Y1 axis =X

In the above example, the 2nd part system plane configuration is as follows:

Setting before mixed control		After mixed control
"#1026 Parallel axis I"=X "#1027 Parallel axis J"=X "#1028 Parallel axis K"=Z	→	I-J plane: Y1 axis - Y1 axis (*1) J-K plane: Y1 axis - Z1 axis K-I plane: Z1 axis - Y1 axis

Mixing X2 axis and Y1 axis (*1) This is insignificant as a plane like X2 axis.

When the other part system's axis is moved by the mixed control or when using the plane which includes the axis name never existed before, set the name of "#1023 crsadr" to the parallel axis parameter "#1029 Parallel axis I" to "#1031 Parallel axis K".

(Example) Suppose the following is set:1st part system axis configuration (X1, Z1, C1, Y1) and 2nd part system axis configuration (X2, Z2, C2)

"#1023 crsadr" of Y1 axis = Y

In the above example, when the mixing, such as moving Y axis to 2nd part system, is executed, to configure the plane with Y axis as J direction axis, set the 2nd part system plane configuration parameter as shown below.

"#1030 Parallel axis J"= Y

(3) Tool compensation

When tool compensation is carried out after the axis is switched by the mixed control, the tool compensation is applied to the mixed axis configuration. On this occasion, when two axes whose axes addresses are different are switched is different are switched, the compensation is carried out for the switched axis address.
For example, when Y axis (Y1) in 1st part system is mixed with X axis (X2) in 2nd part system, the compensation amount for X2 axis is applied to Y axis.

(4) Soft limit

Soft limit is carried out using the axis parameter setting value.
Soft limit I and II define the movement range of the axis, and it is valid even during the mixed control. Soft limit IB, IIB, and IC are the range set with two or more axes' parameters setting values, and the range during mixed control is set by the parameter of the then axis configuration.
Therefore, in across part systems where the zero points differ, the setting values of soft limit IB, IIB, and IC need to match for the mixed configuration axis to validate the same area as before mixed even during the mixed control.

(5) Chuck barrier/tailstock barrier
The chuck barrier/tailstock barrier is the tool nose point entry prohibited area of the tool, which is configured with the axis parameter setting value of the 1st axis and 2nd axis. In across part systems where the zero points differ, each parameter needs to be set again to validate the chuck barrier/tailstock barrier after the mixed control.

Precautions

Precautions when commanding

(1) Command G110 in an independent block. A program error (P33) will occur if not commanded in independent blocks.

(2) All axes that existed before switching in the G110 command need to be commanded by G110 in some part system.

(3) Pairing G110 command performs timing synchronization operation between the part systems.

(4) Mixed control by G110 command is only valid when the parameter "#1280 ext16/bit4" is set to "1". If G110 is commanded when "ext16/bit4" is set to "0", a program error (P610) will occur.

(5) Specify the axis name set in the parameter "#1022 axname2". If the axis name which is not set with axname2 is specified, a program error (P503) will occur.

(6) The axis name specified in G110 needs to be 2 digits. Set the parameter "#1022 axname2" with two digits.

(7) "In mixed control" signal is ON even during the mixed control by G110.

(8) When G110 is commanded and the command part system is in a state where the mixed control is disabled, a program error (P501) will occur.
When the mixed control is commanded in the part system where the mixed control is disabled, a program error (M01 1035) will occur.
Under the following circumstances, the mixed control is disabled:
- In tool nose radius compensation mode
- In polar coordinate interpolation
- In cylindrical interpolation mode
- In balance cut mode
- In fixed cycle machining mode
- In mirror image for facing tool posts
- In constant surface speed control mode
- In hobbing mode
- In axis name switch
- In interference check III alarm (Interference detection, entry to the interference alarm area, or entry to the interference warning area)

(9) The part system without axis cannot be created with the mixed control. If G110 command without axis name is given, a program error (P33) will occur.

(10) If the number of axes exceeds the maximum control axis for the part system in the G110 command, a program error (P503) will occur.

(11) If G110 is commanded to the part system whose parameter "#1501 polyax" is set other than "0", a program error (P503) will occur.

(12) If switching is commanded with G110 to the axis whose parameter "#2071 s_axis" is set to other than "0", a program error (P503) will occur.

(13) If switching is commanded with G110 to the axis whose parameter "#1205 G0bdcc" is set to other than "0", a program error (P503) will occur.

(14) If switching is commanded with G110 to the axis whose parameter "#1072 chop_ax" is set to other than "0", a program error (P503) will occur.

(15) If switching is commanded with the G110 command to the parameter which is set as a master axis or slave axis for the synchronous control, a program error (P503) will occur.

(16) Mixed control by G110 command is invalid for a graphic check.

(17) If a program restart is performed for G110 command, a program error (P49) will occur.

Other precautions

(1) A command which duplicates the existing axis and the command address by the mixed control cannot be executed. If this type of command is given, a program error (P11) will occur.

(2) Tool compensation amount holds the value before the mixed control even after the mixed control was carried out. Issue a tool compensation command again or issue a cancel command, if required. Tool compensation command during mixing compensates the tool with the axis configuration of the part system of mixing destination.

(3) 1st axis and 2nd axis of the tool length compensation and the tool nose wear compensation are fixed to 1st axis and 2nd axis of each part system respectively. Thus, if the mixed control which moves 1st axis and 2nd axis to the other part system, the target axes for the tool length compensation and the tool nose wear compensation are changed so that the normal compensation cannot be carried out.

(4) For mixing the diameter axis and radius axis, if the tool compensation is commanded to the radius axis during mixing, only half the value of tool compensation amount which is set to the mixing destination part system is applied. On the contrary, for the diameter axis, the double of the tool compensation amount which is set to the mixing destination part system is applied.

(5) Mixed control is canceled by emergency stop.

(6) Mixed control is canceled by reset. However, if the parameter "#1280 ext16/bit1" is set to "1", the mixed control state will continue when it is reset.

(7) PLC I/F (Interlock, machine lock, etc.) per axis during switching the axes with the mixed control uses the same device before switching. However, if the parameter "#1280 ext16/bit0" is set to "1", mirror image, machine lock and interlock signal use the device in the command part system side. For moving the axis not switching, PLC signal of the axis No. in the part system is used by following terms:
Set the last axis No. of the source part system for the moved axis.
Set the axis No. in ascending order of destination part system No. at the end of the destination part system for the axes moved from several part systems.
Set the axis No. in axis order of the source part system at the end of the destination part system for the several axes moved from the same part system.

(8) Mixed control is only valid during the automatic operation. If switching 1st part system axis and 2nd part system axis or restore the axes to their original state, both part systems must be in an automatic operation. If an attempt is made to mix axes of the part systems not in the automatic operation, an operation error (M01 1035) will occur.

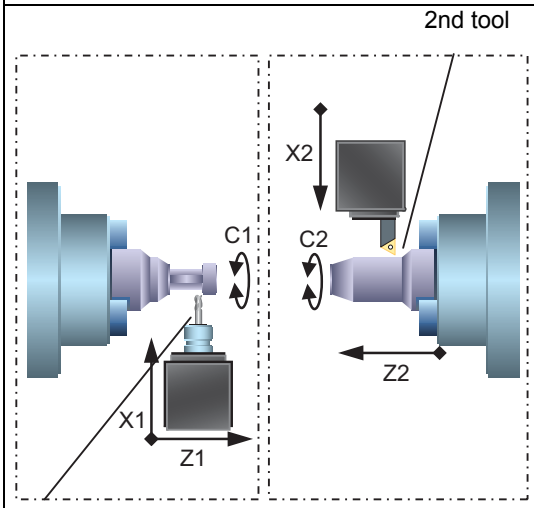
16.3.2 Arbitrary Axis Exchange ; G140, G141, G142

Function and purpose

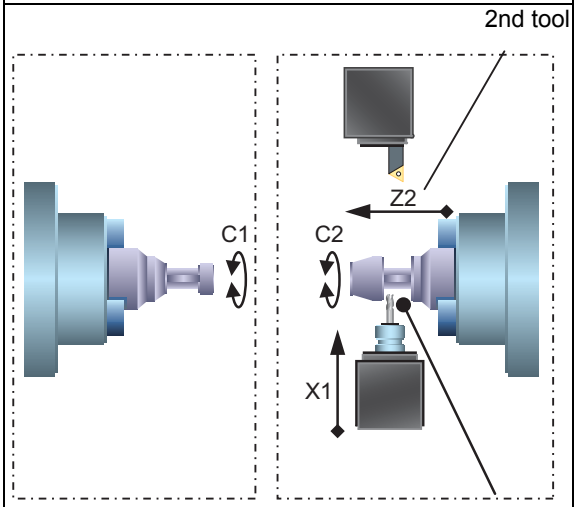
With this function, an arbitrary axis can be exchanged freely across part systems.
The machining can be freer in the multiple part systems by exchanging an axis that can be commanded for machining programs in each part system.
This makes it possible to perform operations which are not possible with regular axis configurations: for instance, tools which are provided only on the 1st part system can be used for machining on the 2nd part system.

(Example) Using X1 axis in part system 2

Before axis exchange	
1st part system (X1, Z1, C1) Machining with the 1st spindle	2nd part system (X2, Z2, C2) Machining with the 2nd spindle



After axis exchange	
1st part system (Z1, C1) Machining with the 1st spindle	2nd part system (X1, Z2, C2) Machining with the 2nd spindle



Machining program 1st part system		Machining program 2nd part system	
G140 X=X1 Z=Z1 C=C1; G00 X10.Z-0.1; G01 Z15. F1.5; : !2 L10;	Machining with the 1st spindle and the 1st tool <---------->	G140 X=X2 Z=Z2 C=C2; G00 X20. Z12.; G01 X8. F1.5; : !1 L10;	Machining with the 2nd spindle and the 2nd tool
	Timing synchronization operation		Machining with the 2nd spindle and the 2nd tool
		G00 Z-20.; G00 X3.; :	
!2 L20;	<---------->	!1L20;	
G140 X=X1 Z=Z1 C=C1;	Timing synchronization operation	G140 X=X2 Z=Z2 C=C2;	Machining with the 2nd spindle and the 2nd tool
G00 X30.;	Machining with the 1st spindle and the 1st tool	G00 X20. Z5.;	
G00 Z0.; :		G01 X15. F1.5; :	
M2;		M2;	

This chapter illustrates an example based on the placements of the basis axes below.

	X axis	Z axis	Y axis	C axis
1st part system ($1)	X1	Z1	Y1	-
2nd part system ($2)	X2	Z2	-	C2

Command format

When commanding the arbitrary axis exchange

```
G140 command address = axis address ... ;
```

Command Address	It is a command address used in a movement or other command after arbitrary axis exchange command (G140). Designate the command address with one alphabetical character set to parameters ("#12071 adr_abs[1]"to "#12078 adr_abs[8]") .
Axis address	Set the axis name for arbitrary axis exchange. Designate the command with two alphanumeric characters set to the parameter "#1022 axname2".

When returning the exchanged axis

```
G141;                    Arbitrary axis exchange return
```

Returns the control right of the axis, exchanged by the previous arbitrary axis exchange command (G140) in the commanded part system, to the state before the axis exchange.

```
G142;                    Reference axis arrange return
```

Returns the control right of the axis, exchanged by the arbitrary axis exchange command (G140) in the commanded part system, to the power-on state.

Detailed description

Arbitrary axis exchange command (G140)

There are two methods for axis exchange operations with arbitrary axis exchange command (G140). The methods for your machine depends on the MTB specifications (parameter "#1434 G140Type2").

Method	Operation
Method for exchanging all axes ("#1434 G140Type2" = 0)	Designates axes to be used in the part system with a command address. The command addresses axes that are not designated will be released as uncontrol axes.
Method for exchanging command axes ("#1434 G140Type2" = 1)	Designates axes to be used in the part system with a command address. The command addresses axes that are not designated will maintain the current state.

(1) Operation example of the method for exchanging all axes ("#1434 G140Type2"=0)

Below is the control axis of each part system when running the following machining programs (1st part system, 2nd part system)

$1 Machining program	$2 Machining program	Control axes $1 X	Control axes $1 Z	Control axes $1 Y	Control axes $2 X	Control axes $2 Z	Control axes $2 C	Uncontrol axes
G140 X=X1 Z=Z1 Y=Y1; (a) G00 X10.; G01 X5. F1; :	G140 X=X2 Z=Z2 C=C2; (d) G00 X20.; G01 X15. F2; :	X1	Z1	Y1	X2	Z2	C2	-
G140 X=X1 Z=Z2; (b) G00 Z25.; G01 X8. F2; :		X1	Z2	-	X2	-	C2	Z1,Y1
	G140 Z=Y1; (e) G00 Z10.; G01 Z8. F0.05; :				-	Y1	-	Z1,X2,C2
G140 X=X1 Z=Z1 Y=Y1; (c) G00 X20. Z15.; G01 X15. F5; :		X1	Z1	Y1	-	-	-	X2,Z2,C2
	G140 X=X2 Z=Z2 C=C2; (f) G00 X0; :				X2	Z2	C2	-

1st part system ($1)	(a),(c)	Declares the use of X1 axis, Z1 axis and Y1 axis.
	(b)	Declares the use of X1 axis and Z2 axis. The control right of Z2 axis shifts to the 2nd part system from the 1st part system. Z1 axis, exchanged for Y1 axis and Z2 axis which were not designated, will be an uncontrol axis.
2nd part system ($2)	(d),(f)	Declares the use of X2 axis, Z2 axis and C2 axis.
	(e)	Declares the use of Y1 axis. X2 axis and C2 axis which were not designated will be uncontrol axes.

(2) Operation example of the method for exchanging command axes ("#1434 G140Type2"=1)

Below is the control axis of each part system when running the following machining programs (1st part system, 2nd part system)

$1 Machining program	$2 Machining program	Control axes $1 X	Control axes $1 Z	Control axes $1 Y	Control axes $2 X	Control axes $2 Z	Control axes $2 C	Uncontrol axes
G140 X=X1 Z=Z1 Y=Y1; (a) G00 X10.; G01 X5. F1; :	G140 X=X2 Z=Z2 C=C2; (d) G00 X20.; G01 X15. F2; :	X1	Z1	Y1	X2	Z2	C2	-
G140 Z=Z2; (b) G00 Z25.; G01 X8. F2; :		X1	Z2	Y1	X2	-	C2	Z1
	G140 Z=Z1; (e) G00 Z10.; G01 Z8. F0.05; :				X2	Z1	C2	-
G140 Z=Z1 ; (c) G00 X20. Z15.; G01 X15. F5; :		X1	Z1	Y1	X2	-	C2	Z2
	G140 X=X2 Z=Z2 C=C2; (f) G00 X0; :				X2	Z2	C2	-

1st part system ($1)	(a)	Declares the use of X1 axis, Z1 axis and Y1 axis.
	(b)	Declares the use of Z2 axis. The control right of Z2 axis shifts to the 2nd part system from the 1st part system. Z1 axis which was exchanged for Z2 axis will be an uncontrol axis.
	(c)	Declares the use of Z1 axis. The control right of Z1 axis shifts to the 2nd part system from the 1st part system. Z2 axis which was exchanged for Z1 axis will be an uncontrol axis.
2nd part system ($2)	(d)	Declares the use of X2 axis, Z2 axis and C2 axis.
	(e)	Declares the use of Z1 axis.
	(f)	Declares the use of X2 axis, Z2 axis and C2 axis.

Arbitrary axis exchange return command (G141)

The arbitrary axis exchange return command (G141) returns the control right of the axis, exchanged by the previous arbitrary axis exchange command (G140) in the commanded part system, to the state before the axis exchange. However, it is the axis that remains an uncontrol axis by the arbitrary axis exchange return command (G140) that returns the control right to the part system which was commanded the arbitrary axis exchange return command (G141)

In other words, when the control right of an axis shifts by the arbitrary axis exchange command (G140) of other part systems, it is possible to return the control right of the axis to the state before the arbitrary axis exchange command (G140) by issuing the arbitrary axis exchange return command (G141) in the part system which was commanded the arbitrary axis exchange (G140). Therefore,if this command is used, it will not be necessary to issue the arbitrary axis exchange command (G140) in the part system to which the control right of the axis shifted by the arbitrary axis exchange command (G140) of other part systems.

(Example)

$1 Machining program		Control axes					Uncontrol axes
		$1			$2		
		X	Z	Y	X	Z	
:		X1	Z1	Y1	X2	Z2	-
G140 X=X2 Z=Z2 Y=Y1; : :	- (a) ->	X2	Z2	Y1	-	-	X1,Z1
G141; :	- (b) ->	X1	Z1	Y1	X2	Z2	-

(a)	X1, Z1 axis	Releases as uncontrol axis.
	Y1 axis	Declares the use in the 1st part system. (The control right does not move.)
	X2, Z2 axis	Declares the use in the 1st part system. (The control right moves to the 2nd part system from the 1st part system)
(b)	X1, Z1 axis	The control right returns to the 1st part system.
	Y1 axis	Stays with the 1st part system. (The control right does not move.)
	X2, Z2 axis	The control right returns to the 1st part system from the 2nd part system.

Reference axis arrange return command (G142)

Reference axis arrange return command (G142) returns the control right of the axis, exchanged by the arbitrary axis exchange command (G140) in the commanded part system, to the power-on state.
Executing the arbitrary axis exchange command (G140) multiple times may make it impossible to return the control right of the axis to the reference axis arrange with the arbitrary axis exchange return command (G141). If this happens, execute the reference axis arrange return command (G142) in the part system to be returned to the reference axis arrange, and the control right of the axis will return to the state when the power was turned on.
Therefore, if you use this command, you will not need to designate the reference axis with the arbitrary axis exchange command (G140).

(1) Basic operations of the reference axis arrange return command (G142)
Below is the control axis of each part system when running the following machining programs.

$1 Machining program	$2 Machining program	Control axes $1 X	Control axes $1 Z	Control axes $1 Y	Control axes $2 X	Control axes $2 Z	Control axes $2 C	Uncontrol axes
: !2 L10;	: !1 L10;	X1	Z1	Y1	X2	Z2	C2	-
G140 X=X1 Z=Z2 Y=Y1; (a) G00 X10.; G01 X5. F1; : !2 L20;	G140 Z=Z1; (d) G00 Z5.; : : !1 L20;	X1	Z2	Y1	-	Z1	-	X2,C2
G140 X=X2 Z=Z2 Y=C2; (b) G00 Z25.; G01 X8. F2; : !2 L30;	G140 X=X1 C=Y1; (e) G01 X15. F2; G00 C10. : !1 L30;	X2	Z2	C2	X1	-	Y1	Z1
G142; (c) G00 X20. Z15.; G01 X15. F5; :	G142; (f) G00 X10. C15. : :	X1	Z1	Y1	X2	Z2	C2	-

1st part system ($1)	(a)	Declares the use of X1 axis, Z2 axis and Y1 axis.
	(b)	Declares the use of X2 axis, Z2 axis and C2 axis. At this time, the command address of C2 axis is "Y".
	(c)	Returns the 1st part system to the reference axis arrange.
2nd part system ($2)	(d)	Declares the use of Z1 axis.
	(e)	Declares the use of X1 axis and Y1 axis. At this time, the command address of Y1 axis is "C".
	(f)	Returns the 2nd part system to the reference axis arrange.

(2) When the arbitrary axis exchange return command (G141) is executed after the reference axis arrange return command (G142)
When the arbitrary axis exchange return command (G141) is executed after the part system is returned to the reference axis arrange by the reference axis arrange return command (G142), the axis arrangement will return to the state that existed before the reference axis arrange return command (G142) was executed.

Unavailable state of axis exchange

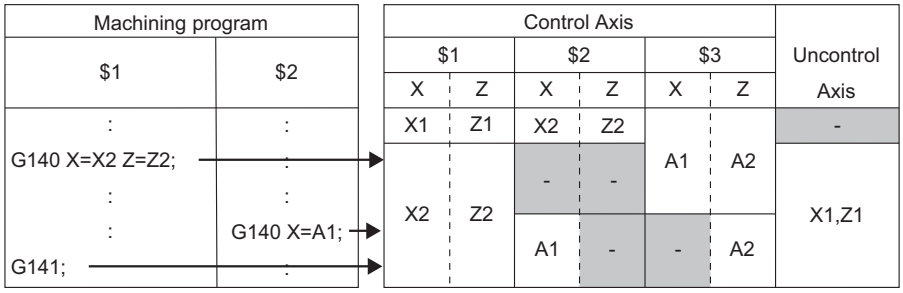
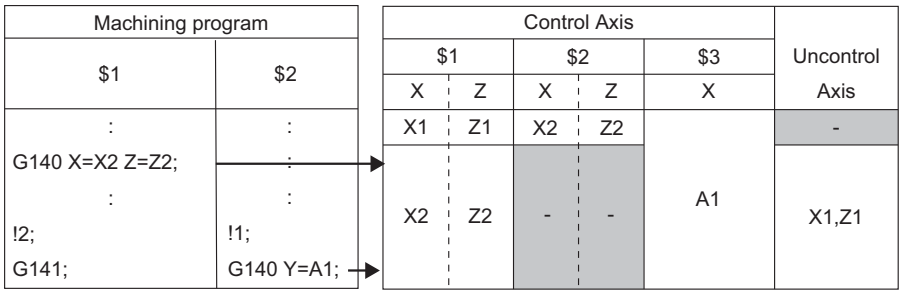
"Unavailable state of axis exchange" indicates a "condition in which a target axis for axis exchange is not available for exchange because the designated target axis for axis exchange is being used by other part systems or for other reasons" through the arbitrary axis exchange command (G140), the arbitrary axis exchange return command (G141), the reference axis arrange return command (G142).
When the conditions for unavailable state of axis exchange fall through, no axis exchange mode will be cancelled. It will be cancelled when a reset signal or emergency stop is entered.

The following is the conditions in which the target axis for axis exchange enters the unavailable state of axis exchange.

(1) Conditions under " the arbitrary axis exchange command (G140), the arbitrary axis exchange return command (G141), the reference axis arrange return command (G142)

Applied to	Conditions
Part system containing the target axis for axis exchange	Part system containing the target axis for axis exchange is in the following mode. •In cylindrical interpolation mode •In polar coordinate interpolation •In milling interpolation •In thread Cutting •In constant surface speed control •In tool nose radius compensation mode •In reference position return •In tool change position return •In NC reset •In feed hold •In axis name switch •In fixed cycle for drilling •In fixed cycle for turning machining •In compound type fixed cycle •In mirror image for facing tool posts •In coordinate rotation •In balance cut •In multi-part system simultaneous thread cutting cycle •In high-speed machining mode •In high-speed high-accuracy control mode •In high-accuracy control mode •In tolerance control mode •In simple inclined surface machining mode •In direct command mode •In Chuck/Tailstock barrier check •In inclined axis control mode •At activation of sub part system •Resetting sub part system •NC alarm 4 signal ON in manual operation mode •In interference check III alarm (Interference detection, entry to the interference alarm area, or entry to the interference warning area)
Target axis for axis exchange	Target axes for axis exchange are in the following conditions. •In movement (If there is remaining commands, the axis is assumed to be in movement even during the interlock). A movement command in the machine lock status is considered to be in movement.) •While a part system containing the target axis for axis exchange is executed, the next block of the machining program commands movement to the axis to be exchanged •Rotary axis in hobbing control •In C axis zero point return in the spindle/C axis function

(2) Conditions under the arbitrary axis exchange return command (G141)

Applied to	Conditions
Command part system and target axes for axis exchange	The other axis is allocated to the command address to which the control right of the axis is moved by the arbitrary axis exchange control command (G141). (Example)  As A1 axis is in the location to which X2 axis is returning,the unavailable state of axis exchange occurs.
Command part system	When the arbitrary axis exchange return (G141) is commanded, axis exchange is running in the part system to which the control right of axis returns through the arbitrary axis exchange command (G140), the arbitrary axis exchange return command (G141) or the reference axis arrange return command (G142). Part systems below are regarded as running. ♦Part system that executes the arbitrary axis exchange command (G140) after waiting for the completion of the axis movement. ♦Part system that waits until axis exchange is activated and executed the arbitrary axis exchange return command (G141). (Example)  As the arbitrary axis exchange command is being executed in the 2nd part system when the command is given to return X2.Z2 axis to the 2nd part system, the unavailable state of axis exchange occurs.

<table>
<tr><th>Applied to</th><th>Conditions</th></tr>
<tr><td rowspan="2">Target axis for axis exchange</td><td>After the axis was exchanged/acquired by the arbitrary axis exchange command (G140), the arbitrary axis exchange return command (G141) was executed with the axis pulled out by the arbitrary axis exchange command (G140) or the reference axis arrange return command (G142).
(Example)
<table>
<tr><th colspan="2">Machining program</th></tr>
<tr><th>$1</th><th>$3</th></tr>
<tr><td>:</td><td>:</td></tr>
<tr><td>G140 X=X2 Z=Z2;</td><td>:</td></tr>
<tr><td>:</td><td>G140 X=X2;</td></tr>
<tr><td>G141;</td><td>:</td></tr>
</table>
<table>
<tr><th colspan="5">Control Axis</th><th rowspan="3">Uncontrol Axis</th></tr>
<tr><th colspan="2">$1</th><th colspan="2">$2</th><th>$3</th></tr>
<tr><th>X</th><th>Z</th><th>X</th><th>Z</th><th>X</th></tr>
<tr><td>X1</td><td>Z1</td><td>X2</td><td>Z2</td><td rowspan="2">A1</td><td>-</td></tr>
<tr><td>X2</td><td>Z2</td><td rowspan="2">-</td><td rowspan="2">-</td><td>X1,Z1</td></tr>
<tr><td>-</td><td>Z2</td><td>X2</td><td>X1,Z1,A1</td></tr>
</table>
As X2 axis which was exchanged with the last arbitrary axis exchange return command (G141) belongs to the other part system (3rd part system), the unavailable state of axis exchange occurs.</td></tr>
<tr><td>The axis exchange return command (G141) was executed while the axis returning the control right to the arbitrary axis exchange return command (G141) system has the control right in other part systems.
(Example)
<table>
<tr><th colspan="2">Machining program</th></tr>
<tr><th>$1</th><th>$2</th></tr>
<tr><td>:</td><td>:</td></tr>
<tr><td>G140 X=X1;</td><td>:</td></tr>
<tr><td>:</td><td>:</td></tr>
<tr><td>:</td><td>G140 Z=Z1;</td></tr>
<tr><td>G141;</td><td>:</td></tr>
</table>
<table>
<tr><th colspan="4">Control Axis</th><th rowspan="3">Uncontrol Axis</th></tr>
<tr><th colspan="2">$1</th><th colspan="2">$2</th></tr>
<tr><th>X</th><th>Z</th><th>X</th><th>Z</th></tr>
<tr><td>X1</td><td>Z1</td><td rowspan="2">X2</td><td rowspan="2">Z2</td><td>-</td></tr>
<tr><td rowspan="2">X1</td><td rowspan="2">-</td><td>Z1</td></tr>
<tr><td>-</td><td>Z1</td><td>X2,Z2</td></tr>
</table>
As Z1 axis which returns the control right to the 1st part system is in control in the other part system (2nd part system), the unavailable state of axis exchange occurs.</td></tr>
</table>

Note

- For the unavailable state of axis exchange in manual operation, refer to " sub part system control function, refer to "Relation with other functions" and "Manual operation".

Operations in the unavailable state of axis exchange

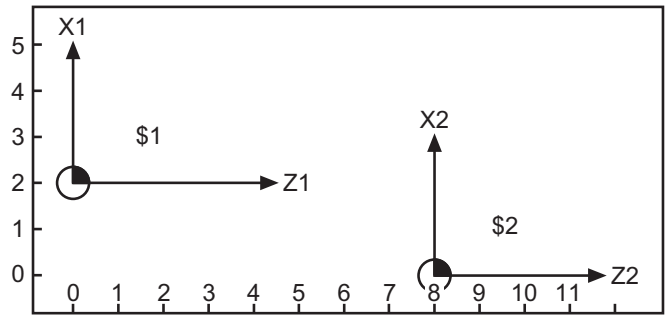
When the arbitrary axis exchange command (G140), the arbitrary axis exchange return command (G141), or the reference axis arrange return command (G142) is executed in the unavailable state of axis exchange, the operations will be as follows, depending on combination of the settings of the machining parameters and the MTB specifications (parameter. "#1433 G140TimeOut")

Base Specifications Parameter (Based on the MTB specifications.) #1432 Ax_Chg_Spec/bit0	Machining parameters #1433 G140TimeOut	Operation
ON	-	Operation error "M01 1101" will occur. When axis exchange is activated, the error will be canceled and the axis will be exchanged.
OFF	255	Waits until axis exchange is activated. When axis exchange is activated, the axis will be exchanged.
OFF	0 to 254(s)	When TIME-OUT SET has elapsed, the operation error (M01 1101) occurs. When axis exchange is activated, the error will be cancelled and the axis will be exchanged.

When the conditions for unavailable state of axis exchange fall through, no axis exchange mode will be cancelled. It will be canceled when a reset signal or emergency stop is entered.

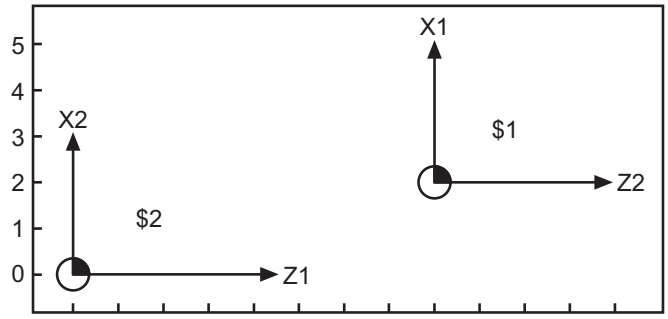
Caution to be observed on coordinate systems

The machine specific zero point and the reference point of each axis are not changed by the arbitrary axis exchange command (G140).
Coordinate system zero point of each axis does not change as the workpiece coordinate system/local coordinate system are based on he machine zero point.
However, when we look at these points on the X-Z plane as shown in the figure below, the coordinate system zero point of each part system is changed by the arbitrary axis exchange command (G140).



1st part system ($1):X1 Z1
2nd part system ($2):X2 Z2

Arbitrary axis exchange command
$1:G140 X=X1 Z=Z2 ;
$2:G140 X=X2 Z=Z1 ;
(Switching Z1 and Z2)



1st part system ($1):X1 Z2
2nd part system ($2):X2 Z1

Therefore, the workpiece coordinate system/local coordinate system for programming may be different from the coordinate system that programmers request with the arbitrary axis exchange command. In that case, execute the coordinate system command after the arbitrary axis exchange command (G140), and create a new coordinate system.

Program example

Example of the arbitrary axis exchange command (G140)

(1) Creating a machining program for a single part system only

$1 Machining program	
G140 X=X1 Z=Z1 Y=Y1; G00 X10.; G01 X3. F2; : G140 X=Y1 Z=Z1 Y=X1; G00 X10.; G01 X3. F2; : G140 X=X2 Z=Z2 C=C2; G00 Z25.; G00 X10.; G00 C20.; : M2;	Machining that uses the axis of the 1st part system Machining that uses the axis of the 1st part system (Exchanges the command addresses of X1 and Y1.) Machining that uses the axis of the 2nd part system

(2) Creating a machining program for multiple part systems

The arbitrary axis exchange control exchanges axes if the declared axis is available for exchange. As a result, it may lose the control right of the axis during machining, depending on the timing.

To avoid this, command the timing synchronization operation in the range where you do not want to lose the control right of the axis.

$1 Machining program		$2 Machining program	
G140 X=X1 Z=Z1 Y=Y1;	Acquires the control right of X1, Z1 Y1 axes	G140 X=X2 Z=Z2 C=C2;	Acquires the control right of X2, Z2 C2 axes
G00 X10.; G01 X3. F2.; : !2 L10;	Range that needs the control right of X1, Z1 Y1 axes <----------> Timing synchronization operation	G00 X5.; G01 X8. F0.5; : !1 L10;	Range that needs the control right of X2, Z2 C2 axes
G140 X=X2 Z=Z1 C=C2;	Acquires the control right of X2, Z1 C2 axes		
G00 C50.; G00 X15. Z30.; : !2 L20;	Range that needs the control right of X2, Z1 C2 axes <----------> Timing synchronization operation	: !1 L20;	
G00 X12. Z50.;	Range that needs the control right of X2 and Z1 axes	G140 C=C2;	Acquires the control right of C2 axis
G01 X5.5 F2.5; : : !2 L30;	<----------> Timing synchronization operation	G00 C180.; : : !1 L30;	Range that needs the control right of C2 axis
G00 X15.;		G140 X=X2;	Acquires the control right of X2 axis
G01 X6.5 F3.; : : !2 L40;	Range that needs the control right of X2 axis <----------> Timing synchronization operation	G00 X10.; G01 X3.5 F1.5; : !1 L40;	Range that needs the control right of X2 axis
M2;		G140 X=X2 Z=Z2 C=C2;	Acquires the control right of X2, Z2 C2 axes
		G00 Z25.; G01 X3.5 F1.5; : M2;	Range that needs the control right of X2, Z2 C2 axes

Example of the arbitrary axis exchange return command (G141)

(1) Using the arbitrary axis exchange return command (G141)

$1 Machining program		$2 Machining program	
G140 X=X1 Z=Z1 Y=Y1; G00 X10.; : :	Timing synchronization operation	G140 X=X2 Z=Z2 C=C2; : : :	Machining that uses the axis of the 2nd part system
!2 L10;	<---------->	!1 L10;	
G140 X=X2 Z=Z2 C=C2; G00 Z50.; G00 X12.; G01 X5.5 F2.5; : :	Machining that uses the axis of the 2nd part system	: : : : : :	
G141;	Returning the axis of the 2nd part system	:	
!2 L20;	<---------->	!1 L20;	
	Timing synchronization operation		
M2;		: G04 X1.; G00 Z25.; G00 X10.; G01 X3.5 F1.5; G00 C20. : M2;	Machining that uses the axis of the 2nd part system

Example of the reference axis arrange return command (G142)

(1) Using the reference axis arrange return command (G142)

$1 Machining program		$2 Machining program	
G140 X=X1 Z=Z1 Y=Y1; G00 X10.; :	Timing synchronization operation	G140 X=X2 Z=Z2 C=C2; : :	Machining that uses the axis of the 2nd part system
!2 L10;	<---------->	!1 L10;	
G140 X=X2 Z=Z2 C=C2; G00 Z50.; G00 X12.; G01 X5.5 F2.5; :	Machining that uses the axis of the 2nd part system		
!2 L20;	<---------->	!1 L20;	
G142;	Restoring the reference axis arrange	G142;	
G00 X10.; G00 Z25.; G01 X12.5 Z30. F2.5; : M2	Machining that uses the axis of the 1st part system	G00 X3.; G01 Z40. F3.; G00 X7.Z35.; : M2	Machining that uses the axis of the 2nd part system

Relationship with other functions

Plane selection

For the part system where the axes are switched by the arbitrary axis exchange command (G140), the arbitrary axis exchange return command (G141) or the reference axis arrange return command (G142), the plane is configured with the switched axis.
The axis being exchanged configures the plane at the axis addresses set in parameters ("#12071 adr_abs[1]"to "#12078 adr_abs[8]") .

#	Item	1st part system	2nd part system
1026	Basis axis I	X	X
1027	Basic axis J	Y	X
1028	Basic axis K	Z	Z
12071	adr_abs[1]	X	X
12072	adr_abs[2]	Z	Z
12073	adr_abs[3]	Y	Y
12074	adr_abs[4]	C	C

When the parameters are set as above, the axes will be configured as shown in the table blow.

	When the following command is issued in the 1st part system G140 X=Y1 Y=X2 Z=Z1	When the following command is issued in the 2nd part system G140 X=Y1 Y=X2 Z=Z1
I axis	Y1 axis	Y1 axis
J axis	X2 axis	Y1 axis
K axis	Z1 axis	Z1 axis
I-J plane	Y1-X2 axis	Y1-Y1 axis "this is insignificant as a plane:
J-K plane	X2-Z1 axis	Y1-Z1 axis
K-I plane	Z1-Y1 axis	Z1-Y1 axis

When you need to move the other part system's axis by the arbitrary axis exchange, and to use the plane with the built-in axis name which did not exist before, set the same axis address as the parameter ("#12071 adr_abs[1]" to "#12078 adr_abs[8]") for the parallel axis parameter (#1028 parallel axis I to #1030 parallel axis K).

Tool compensation

(1) When tool compensation is carried out after the axis is switched by the arbitrary axis exchange control, the tool compensation is applied to the mixed axis configuration. On this occasion, when two axes whose axes addresses are different are switched is different are switched, the compensation is carried out for the switched axis address.

(2) Tool compensation command during axis exchange compensates the tool with the axis configuration of the part system of mixing destination.

(3) 1st axis and 2nd axis of the tool length compensation and the tool nose wear compensation are fixed to 1st axis and 2nd axis of each part system respectively.

(4) Thus, if the arbitrary axis exchange command (G140) which moves the 1st axis and 2nd axis of each part system to other part systems is issued, the target axes for the tool length compensation and the tool nose wear compensation are changed, and the normal compensation cannot be carried out.
Execute the arbitrary axis exchange command (G140) so that axes that require offset will be assigned to the first axis and the second axis.

(5) For switching the diameter axis and the radius axis, if tool compensation is commanded to the axis during axis exchange, the normal compensation cannot be carried out because the compensation amount set for the diameter axis becomes valid for the position command of the radius axis, and the compensation amount set for the radius axis becomes valid for the position command of the diameter axis.

(6) Tool compensation amount holds the value before the arbitrary axis exchange even after the arbitrary axis exchange control is carried out. Issue a tool compensation command again or issue a cancel command, if required.

(7) Whether the tool length compensation/wear compensation is canceled after axis exchange depends on the MTB specifications (parameter "#1432 Ax_Chg_Spec/bit1").

Resetting

Depending on the MTB specifications (parameter "#1280 ext16/bit1"), the axis of the part system that was reset will be returned to the reference axis arrange or remain in the condition after the axis exchange.

(1) When the parameter is invalid ("#1280 ext16/bit1"= 0)

Return the axis of the part system that was reset to the reference axis arrange.
However, depending on the type (regular part system, sub part system control I, sub part system control II) of the part system, the axis arrange after resetting will be as follows.

Type of part system	Axis arrange after resetting
Regular part system	Reference axis arrange
Sub part system control I	Reference axis arrange
Sub part system control II	All axes Uncontrol Axes

(2) When the parameter is valid ("#1280 ext16/bit1"= 1)

Remains in the condition after the axis exchange.
However, depending on the type (regular part system, sub part system control I, sub part system control II) of the part system, the axis arrange after resetting will be as follows.

Type of part system	Axis arrange after resetting
Regular part system	Axis arrange after axis exchange
Sub part system control I	Axis arrange after axis exchange
Sub part system control II	All axes Uncontrol Axes

Manual operation

(1) Manual operation for the axis during axis exchange and the operations of manual interruption depends of the MTB specifications (parameter "#1435 crsman").

(a) When the parameter is invalid ("#1435 crsman"= 0)

Manual operation is possible only when the axis to be operated manually is in the reference axis arrange.

	1st part system	2nd part system
Reference axis arrange	X1 Z1 C1	X2 Z2
After the arbitrary axis exchange command	X2 Z1 C1	X1 Z2

In the case above, if manual operation is carried out for the axis in the 2nd part system after the axis exchange, Z2 axis in the reference axis arrange can be moved. However, X1 axis cannot be move as it is not in the reference axis arrange.
If the axis selected by manual operation is not in the reference axis arrange, the operation error (M01 1102) will occur. When the axis returns to the reference axis arrange, select it again.
The timing at which the operation error (M01 1102) occurs is as follows, depending on each manual operation mode.

Manual operation mode	Alarm generation timing
Jog	When the axis is selected
Handle	When the axis is selected and enabled
Arbitrary feed in manual mode 1	When the axis is selected and enabled

(b) When the parameter is valid ("#1435 crsman"= 1)

Even when the axis for manual operation is not in the reference axis arrange, manual operation can be carried out.
However, the method for selecting the axis to be moved by manual operation varies depending on each manual operation mode.

Manual operation mode	Part system selecting manual operation mode	Axis selection method
Jog	Part system containing the axis to be moved	Selects the feed axis in the reference axis arrange
Handle	Part system containing the axis to be moved	Selects the axis number in the reference axis arrange within the part system
Arbitrary feed in manual mode 1	Basic definition part system of the axis to be moved	Selects the axis number in the reference axis arrange within the part system

(2) When issuing the arbitrary axis exchange to the axis of the part system in manual operation mode

The arbitrary axis exchange command (G140), the arbitrary axis exchange return command (G141), the reference axis arrange return command (G142) can be issued to the axis in manual operation mode from other part systems.
However, if a command is issued in manual operation mode to a part system where the operation alarm is going off, it is impossible to exchange axes.

<Note>

- Even when the operation error has not occurred, axes cannot be exchanged when axis exchange is not available. Refer to "Unavailable state of axis exchange" for details of unavailability of axis exchange.

(3) When the manual automatic simultaneous valid axis is the axis to be exchanged in the part system

If a moving axis specified as the axis to be exchanged in the part system , axis exchange will be impossible and the operation error (M01 1101) will occur.
The operation error (M01 1101) that occurs in this unavailable state of axis exchange does not rely on the settings of the parameter ("#1432 Ax_Chg_Spec/bit1", "#1433 G140TimeOut") (This error always occurs.)
When the moving axis stops, the operation error (M01 1101) will be cancelled and axis exchange will be carried out.

Mixed control I/II

Mixed control I/II cannot be commanded in conjunction with the arbitrary axis exchange command (G140), the arbitrary axis exchange return command (G141) or the reference axis arrange return command (G142).
Whether to enable the mixed control or the arbitrary axis control depends on the MTB specifications (parameter "#1431 Ax_Chg)").
If the arbitrary axis exchange command (G140), the arbitrary axis exchange return command (G141) or the reference axis arrange return command (G142) is issued while the mixed control I/II is valid, the program error (P39) will occur.

Sub part system control I/II

(1) The axis arrange of the sub part system when turning on the power, when resetting, when issuing the reference axis arrange return command (G142) or when terminating the sub part system depends on the MTB specifications (parameter "#1280 ext16/bit1").

#1280 ext16/bit1	Conditions	Axis arrange of sub part system (Sub part system control I)	Axis arrange of sub part system (Sub part system control II)
0 (Restores the reference axis arrange)	At power ON	Reference axis arrange	All axes Uncontrol Axes
	When resetting	Reference axis arrange	All axes Uncontrol Axes
	When issuing the reference axis arrange return command (G142)	Reference axis arrange	Reference axis arrange
	When terminating the sub part system	Reference axis arrange	All axes Uncontrol Axes
1 (Does not restore the reference axis arrange)	At power ON	Reference axis arrange	All axes Uncontrol Axes
	When resetting	Axis arrange after axis exchange	All axes Uncontrol Axes
	When issuing the reference axis arrange return command (G142)	Reference axis arrange	Reference axis arrange
	When terminating the sub part system	Axis arrange after axis exchange	All axes Uncontrol Axes

(2) When the reference axis arrange return command (G142) is issued in the sub part system control II, the basic configuration axis of the sub part system control II will be the control axis.

(Example) The reference axis arrange return command (G142) is issued in the sub part system II, when the reference axis arrange is as follows

<Reference axis arrange>

	X axis	Z axis	Y axis
Sub part system 4 (sub part system)	X4	Z4	Y4

Sub part system 4 (sub part system) Machining program	Control axes			Uncontrol axes
	X	Z	Y	
(Activate sub part system II)	-	-	-	X4,Y4,Z4
G140 X=X1 Z=Z1 :	X1	Z1	-	
G142 :	X4	Z4	Y4	X1,Z1
M99 (terminate the sub part system II)	-	-	-	X1,Z1,X4,Z4,Y4

(3) When returning the axis to the sub part system control II with the arbitrary axis exchange return command (G141), the axis will become an uncontrol axis if the sub part system to which the axis is returned is out of operation.

Position switch

Position switch is set to the setting value of the reference axis arrange.
The axis for position switch will not change even if the axis arrange within the part system is changed with the arbitrary axis exchange command.

User macro

The following variables can read and write the date of each axis with the user macro. They are handled in the axis arrange after the axis exchange.

- Mirror image
- Position information
- Workpiece coordinate system offset
- Mirror image by external command
- Mirror image by parameter setting
- Programmable mirror image
- Coordinate rotation by program

Soft limit

Soft limit is carried out using axis parameter setting value.
Soft limit I and II define the movement range of the axis, and it is valid even during the arbitrary axis exchange control. Soft limit IB, IIB, and IC are the range set with two or more axes' parameters setting values, and the range during axis exchange is set by the parameter of the then axis configuration.
Therefore, in across part systems where the zero points differ, the setting values of Soft limit IB, IIB, and IC need to match so that an area same as that before the axis exchange will be enabled even during the axis exchange.

Parameter input by program command (G10)

Command the Parameter input by program command (G10) in the reference axis arrange mode.

Control axis synchronization (G125), Control axis superimposition (G126)

When the axis for axis exchange in the arbitrary axis exchange command is the synchronization axis in control axis synchronization (G125) or the superimposed axis in the control axis superimposition (G126), there is no movement command for the axis for axis exchange. If it is just following the reference axis, it will not be regarded as moving, and the axis exchange will be carried out.

Setting and display

The method of setting/displaying after the arbitrary axis exchange command (G140) is called "cross setting/display". As opposed to this, the method of setting/displaying as basic definition is called "basic definition setting/display". The method for your machine depends on the MTB specifications (parameter "#1280 ext16/bit2").

(1) 2-digit axis name
For two or more part systems, the same axis name may exist in each part system. To differentiate these axes, an axis name can be displayed in 2 digits (X1,Z2).
The axis name set in the parameter "#1022 axname2" will be displayed.

(2) Displaying the switched coordinate value
Whether or not the axis name or the coordinate value (workpiece coordinate zero points, machine coordinate system) is switched for display during arbitrary axis exchange depends on the MTB specifications.

Method	Operation
Cross setting/display	The axis name and the coordinate value are switched (moved) for display by the arbitrary axis switch control.
Basic definition setting/display	The axis name and the coordinate value are displayed in the reference axis arrange, without being switched (moved), by the arbitrary axis switch control.

The operation screen (except restart search), the setup screen and the graphic trace screen can be switched for display.
When the setup value is 0 in the arbitrary axis exchange control, the coordinate values in the part systems after axis exchange will be display in the order that the parameters were set ("#12071 adr_abs[1])" to "#12078 adr_abs [8]").
Uncontrol axes that were allocated by the arbitrary axis exchange command (G140) from the other part system will be displayed at the end of the part system.

(3) Graphic check screen
When performing a graphic check during the arbitrary axis exchange, the screen shows the paths of the axes, ignoring the commands in the arbitrary axis exchange command blocks.

Other functions

The following functions must be executed in reference axis arrange mode.
Do not designate any axis as a target for axis exchange if that axis is included in the part system in which any of following functions is being executed.
- Manual feedrate B surface speed control
- Arbitrary reverse run

Precautions and restrictions

Common precautions/restrictions for G140, G141 and G142

(1) When the arbitrary axis exchange command (G140), the arbitrary axis exchange return command (G141) or the reference axis arrange return command (G142) is issued in a part system in any of the following mode, the program error (P501) will occur.
- In cylindrical interpolation mode
- In polar coordinate interpolation
- In milling interpolation
- In constant surface speed control mode
- In polygon machining mode
- In hobbing mode
- In tool nose radius compensation mode
- In axis name switch
- In fixed cycle for drilling
- In fixed cycle for turning machining
- In compound type fixed cycle
- In mirror image for facing tool posts
- In balance cut mode
- In Chuck Barrier/Tailstock Barrier check
- In macro modal call (G66.1)

(2) When the arbitrary axis exchange command (G140), the arbitrary axis exchange return command (G141) or the reference axis arrange return command (G142) is issued to the following axes, the program error (P503) will occur.
- Parameters other than those on which acceleration /deceleration was set after G00 interpolation (parameter "#1205 G0bdcc" is other than "0")
- Chopping axis (parameter "#1072 chop_ax" is other than "0")
- Master axis or slave axis of synchronous control

(3) If the other G code is commanded in the same block as the arbitrary axis exchange command (G140), the arbitrary axis exchange return command (G141) or the reference axis arrange return command (G142), a program error (P34) will occur.

(4) If a command is issued to an axis that has lost the control right through the arbitrary axis exchange command of the other part system, the program error (P32) will occur.

(5) If an axis was extracted by the arbitrary axis exchange command (G140), the arbitrary axis exchange return command (G141) or the reference axis arrange return command (G142) from the other part system, the axis arrange will change to a state with the extracted axis filled.
The axis arrange also changes when the number of axes increase in the part system through the arbitrary axis exchange command (G140), the arbitrary axis exchange return command (G141) or the reference axis arrange return command (G142).
Therefore, when you use a function to be specified with the axis number in the part system (e.g. position information of user macro: #5001 - #5140+n) after issuing the arbitrary axis exchange command (G140), the arbitrary axis exchange return command (G141) or the reference axis arrange return command (G142), command with the changed axis arrange.

(6) If the arbitrary axis exchange command (G140), the arbitrary axis exchange return command (G141) or the reference axis arrange return command (G142) is issued while the high-speed machining mode II is valid, the program error (P34) will occur.
If the arbitrary axis exchange command (G140), the arbitrary axis exchange return command (G141) or the reference axis arrange return command (G142) is issued while any of the following functions is valid, a program error (P126) will occur.
- High-accuracy control
- High-speed high-accuracy control I
- High-speed high-accuracy control II
- Tolerance control

(7) After arbitrary axis exchange control is carried out, the number of axes that can be controlled with handle feed is limited to one per handle in each part system. If more than one axis are allocated to a handle, an operation error (M01 0106) will occur.

(8) The part system in which simple inclined surface machining is being carried out does not cancel mixed control regardless of the setting for the "#1280 ext16/bit1" parameter (Cancellation of mixed control by resetting) even if a reset operation that does not reset the modal ("#1151 rstint" = 0 and NC reset 1) is carried out. If an axis in a part system in which simple inclined surface machining is being carried out is specified as the axis to be exchanged in the part system, axis exchange will not be possible and an operation error (M01 1101) will occur regardless of whether the automatic operation mode has been established.

Precautions/restrictions specific to the arbitrary axis exchange command (G140)

(1) The command address of the arbitrary axis exchange command (G140) is set for each part system with parameters ("#12071 adr_abs[1]"to "#12078 adr_abs[8]"). When an address not in the parameters is commanded, the program error (P32) will occur.
The number of axes that can be commanded in each part system is the number of axes that can be set for these parameters.
When you use an axis that is not set in these parameters, allocate a command address in the arbitrary axis exchange command (G140) and then issue a command.

(2) If multiple axes are declared with the same command address, or if an axis is declared with multiple command addresses, the program error (P33) will occur.

(3) When the arbitrary axis exchange command (G140) is issued independently (without a command address), all the axes in the part system will become uncontrol axes.

(4) Note that when the arbitrary axis exchange command (G140) is issued from multiple part systems and the axes for axis exchange are overlapping, the operations will be as follows.

$1 Machining program	$2 Machining program		Control axes					
			$1			$2		
			X	Z	Y	X	Z	Y
: !2 L1;	: !1 L1;		X1	Z1	Y1	X2	Z2	Y2
G140 X=X1 Z=Z2 Y=Y1; : !2 L2;	G140 X=X1 Z=Z2 Y=Y2; : !1 L2;	-->	-	Z2 (*1)	Y1	X1 (*1)	-	Y2
; :	G140 X=X3 Z=Z2 Y=Y2; :	-->	-	Z1	Y1	X3 (*2)	Z2	Y2

*1: When the overlapping axes for axis exchange exist in the arbitrary axis exchange command (G140) part system, they will become control axes in the part system that does not exist in their own part system.

*2: When the overlapping axes for axis exchange do not exist in the arbitrary axis exchange command (G140) part system, they will become control axes in the part system with the larger part system number.

(5) If the arbitrary axis exchange command (G140) is issued in the method for exchanging command axes ("#1434 G140Type2"= 1) when an axis, which lost the control right of axis in the arbitrary axis exchange command (G140) from other part systems, exists, the operations will be as follows.

$1 Machining program	$2 Machining program	Control axes							Uncontrol axes
		$1				$2			
		X	Z	Y	H	X	Z	H	
: : : !2 L1;	: G140 H=H1; : !1 L1;	X1	Z1	Y1	H1	X2	Z2	H2	-
		X1	Z1	Y1	-	X2	Z2	H1	H2
G140 Z=Z2; G00 H100.; (*1) :	: : :	X1	Z2	Y1	-	X2	-	H1	Z1,H2

*1: The program error (P32) occurs in $1 (1st part system) as the control right of H1 axis was lost in the arbitrary axis exchange command (G140) in $2 (2nd part system).

Precautions/restrictions specific to the arbitrary axis exchange return command (G141)

(1) If an address other than N (sequence number) is commanded in the same block as for the arbitrary axis exchange return command (G141), a program error (P32) will occur.

(2) In a part system where the arbitrary axis exchange return command (G141) was executed, execute the arbitrary axis exchange command (G140) before executing the same command again. A program error (P33) will occur if G141 is commanded without having commanded G140.

(3) After turning the power on, and after resetting the reference axis arrange, if the arbitrary axis exchange return command (G141) is commanded in a part system where the arbitrary axis exchange command (G140) has never been executed, the command will be ignored.

(4) When the arbitrary axis exchange return command (G141) is executed after the reference axis arrange return command (G142), the axis arrange will be that before the reference axis arrange return command (G142) was executed.

Precautions/restrictions specific to the reference axis arrange return command (G142)

(1) If an address other than N (sequence number) is commanded in the same block as for the reference axis arrange return command (G142), a program error (P32) will occur.

16.4 Control Axis Superimposition

16.4.1 Control Axis Superimposition ; G126

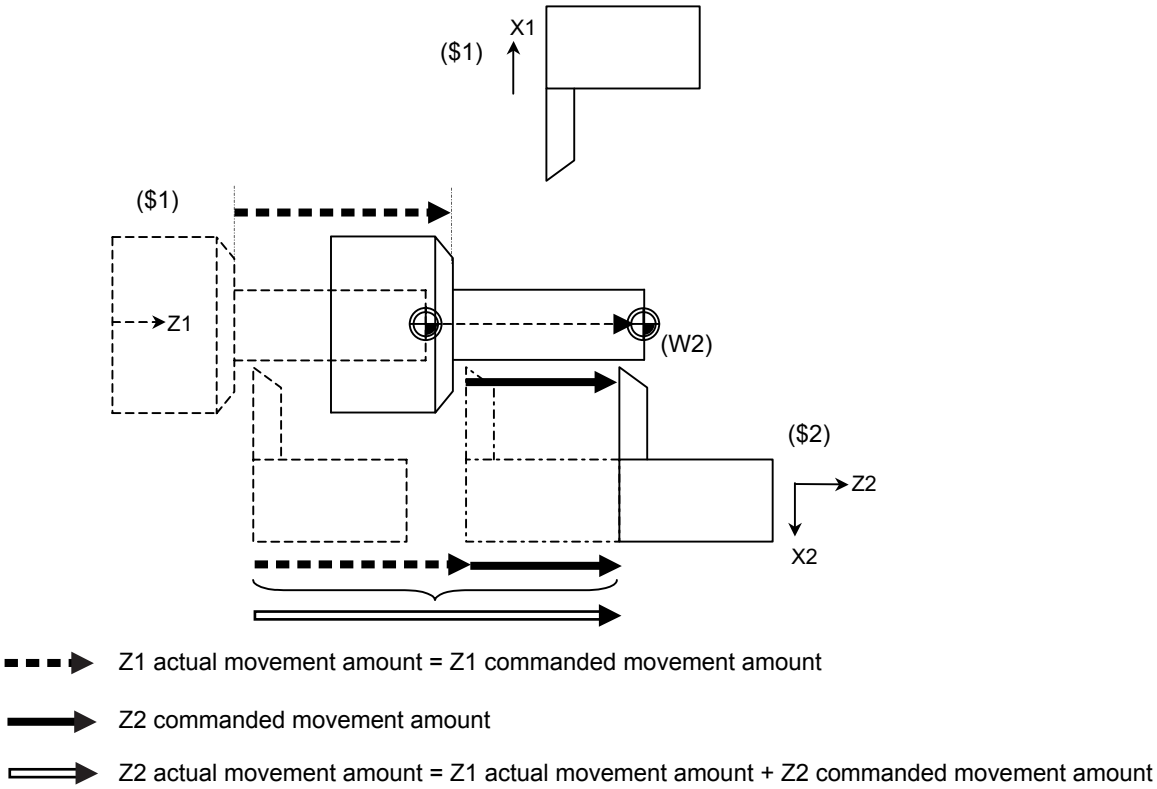
Function and purpose

This function enables superimposition on and control of an axis in a selected part system with an axis in another part system.
This is effective when machining by the 1st part system (X1, Z1) and the 2nd part system (X2, Z2) are executed simultaneously for the machine configuration where a workpiece moves by a movement command to the Z axis direction. This function eliminate the need of fixing the workpiece position for machining in the superimposed part system (the 2nd part system) and allows the simultaneous machining of the reference axis part system (the 1st part system) and the superimposed axis part system (the 2nd part system).
This function can control superimposition of up to 3 axes.
As long as the reference axis and the superimposed axis do not overlap, there is no restrictions on the number of sets.
There are two methods: commanding with PLC signal and commanding with G command. MTB specifications determine which is valid (parameter "#1280 ext16/bit7").
This section describes the methods of executing with G commands .
Terms in the explanation are defined as follows.

Reference axis	The basic in the control axis superimposition function (moves only by its own axis command)
Superimposed axis	The axis which moves including the reference axis movement in the control axis superimposition function (moves by reference axis or/and own axis command).
Superimposition of 2 axes	1 superimposed axis for a reference axis
3 axes tandem superimposition	The condition that an axis is the superimposed axis and becomes the reference axis for another superimposed axis at the same time. The first superimposed axis is referred to as the "1st superimposed axis" and the other axis as the "2nd superimposed axis".

While using the control axis superimposition function, the workpiece zero point of the superimposed axis moves according to the movement amount of the reference axis. Because the superimposed axis tries to maintain the position in the workpiece coordinate system, it moves as much as the movement amount of the reference axis.
As long as the reference axis and the superimposed axis do not overlap, the number of control axis superimposition set has no limitation.

(Example) Reference axis: Z1, superimposed axis: Z2.
The zero point in the figure indicates the 2nd part system workpiece coordinate.



Command format

Superimposition start

G126 Superimposed axis name = Reference axis name (,P__) ;

Superimposed axis name	The axis to be operated as superimposed axis (The axis name set in the parameter (#1022 axname2) (two characters))
Reference axis name	The axis to be operated as referenced axis (The axis name set in the parameter (#1022 axname2) (two characters)) If the symbol "-" is added in front of the reference axis name, the superimposed axis can be synchronized in the opposite direction of the reference axis.
P	Designation of the superimposed axis workpiece coordinate system (Set the workpiece coordinate zero point of the superimposed axis in the workpiece coordinate system of the reference axis) (Radius value) (mm/inch)(Decimal point command is possible.)
, (comma)	Used as delimiter when specifying the address after "superimposed axis name = reference name".

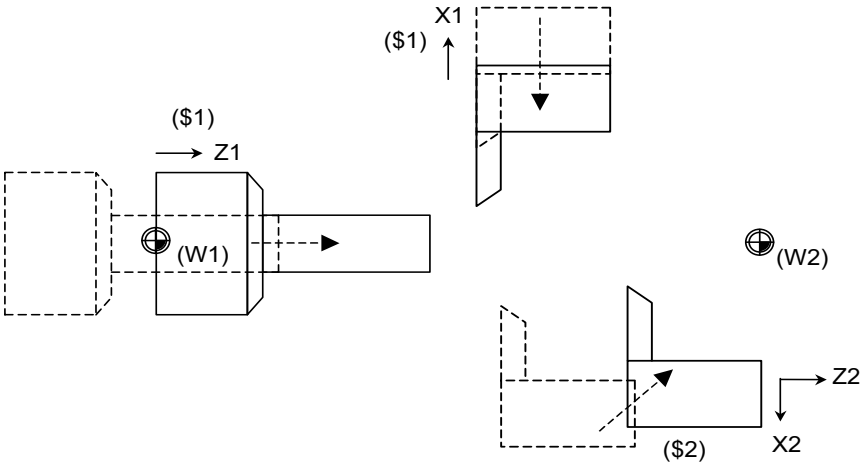
Superimposition end

G126 Superimposed axis name;

Superimposed axis name	The axis operating as superimposed axis (The axis name set in the parameter "#1022 axname2" (2 characters))

Detailed description

The machine configuration in the figure below describes each operation.
In the 1st part system (X1,Z1), the X axis command moves the tool and the Z axis command moves the workpiece.
In the 2nd part system (X2,Z2), the X and Z axes command moves the tool.
The workpiece coordinate zero point (W1, W2) in the figure indicates the Z axis workpiece coordinate zero point.
Reference axis: Z1, superimposed axis: Z2.



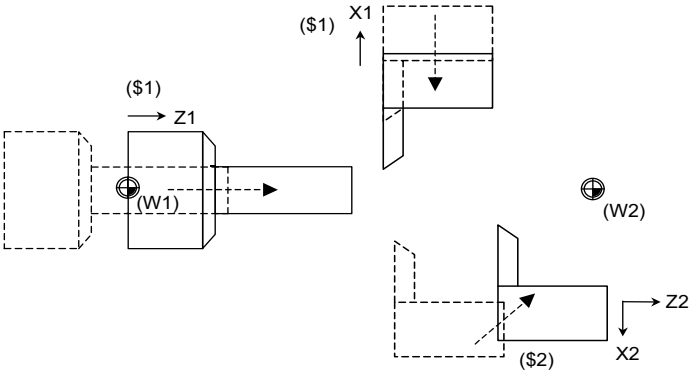
2-axis superimposition start operation

The following operation example explains the start command to superimpose Z2 axis onto Z1 axis.

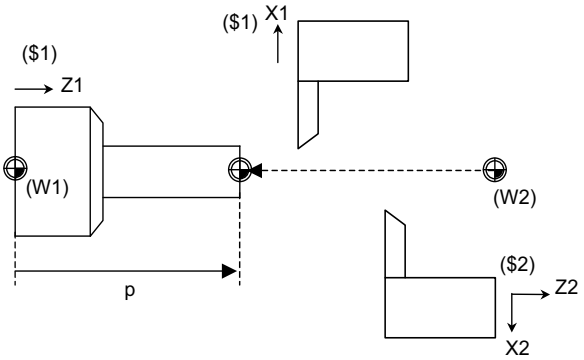
Command : G126 Z2 = Z1 ,Pp ;

The superimposition start command automatically executes the following operation.

(1) Wait for the acceleration/deceleration time constant of all axes including the reference axis/the superimposed axis.

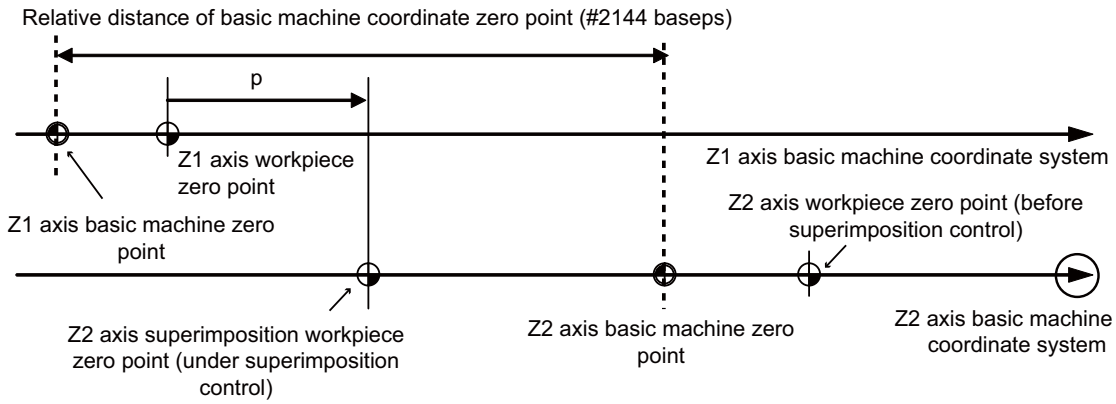


(2) Set the superimposed axis workpiece zero point by a P command, and the relative distance of Z1 axis (reference axis) and Z2 axis (superimposed axis). The relative distance and the setting depend on the MTB specifications (related parameters "#2144 baseps", "#2143 polar"). The superimposed axis will not move.

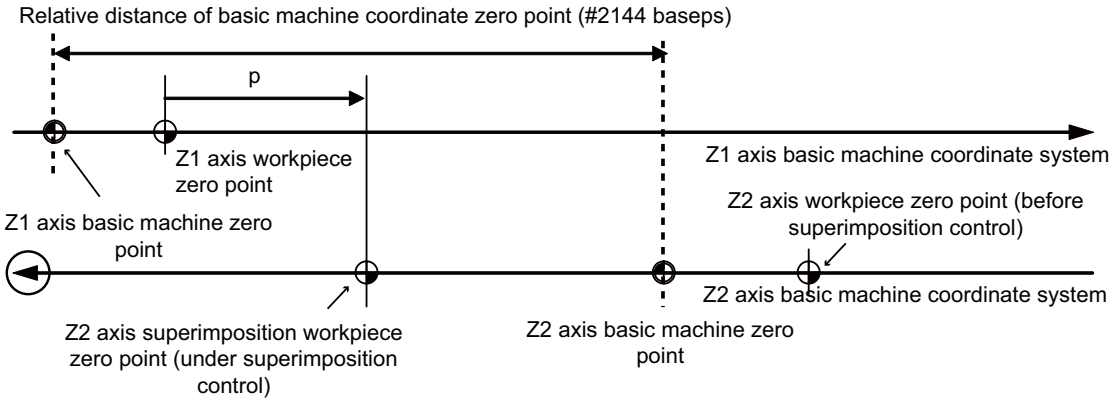


The following figure shows the relationship between the workpiece zero point of Z2 axis superimposition and the address P when the axis superimposition start is commanded in a case where (a) relative polarity is present and a case where (b) relative polarity is absent.

(a) Relative polarity (positive) (the value of "#2143 polar" is reference axis: 0, superimposed axis: 0)



(b) Relative polarity (negative) (the value of "#2143 polar" is reference axis: 0, superimposed axis: 1)



(3) Switch the time constant of all axes of the part system containing the reference axis and superimposed axis, to the time constant for the superimposition set in the MTB specifications (parameter #2092 - #2095).

(4) Start the superposition control of Z1 axis (reference axis) and Z2 axis (superimposed axis).
Parameters of the rapid traverse rate and the clamp rate vary depending on the MTB specifications (parameter "#2090, 2091).

Start operation for superimposition of 3 axes

The following operation example explains the start command for superimposing Z3 axis onto Z2 axis from the superimposition of 2 axes consisting of Z1 axis (reference axis) and Z2 axis (superimposed axis).

Command : G126 Z3=Z2 ,P_ ;

The superimposition start command automatically executes the following operation.

(1) Wait for the acceleration/deceleration time constant of all axes including Z3 axis (2nd superimposed axis), Z2 axis (1st superimposed axis) and Z1 axis (reference axis).

(2) Set the superimposed axis workpiece zero point by a P command.
The setting of the workpiece zero point can be carried out by shifting the G92 coordinate system. (Z2 axis (superimposed axis) will not move.)

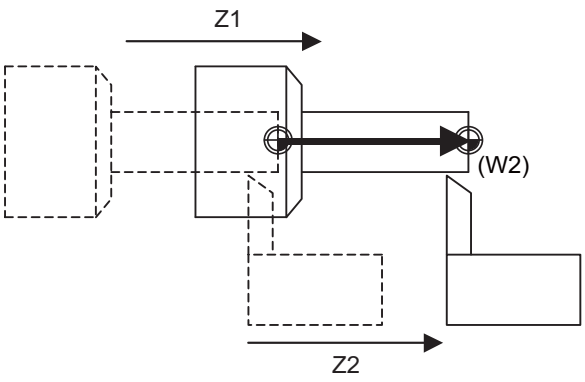
(3) Switch the time constant of all axes of the part system containing Z3 axis (2nd superimposed axis), Z2 axis (1st imposed axis) and Z1 (reference axis) , to the time constant for the superimposition set in the MTB specifications (parameter #2622 - #2625).

(4) Start the superposition control of Z3 axis (2nd superimposed axis)/Z2 axis (1st superimposed axis)/Z1 axis (reference axis).
Refer to the section of "Operation during superimposition" for details.
Parameters of the rapid traverse rate and the clamp rate vary depending on the MTB specifications (parameter "#2626 - ,#2630).

Operation during superimposition

[Workpiece coordinate system of the superimposed axis]

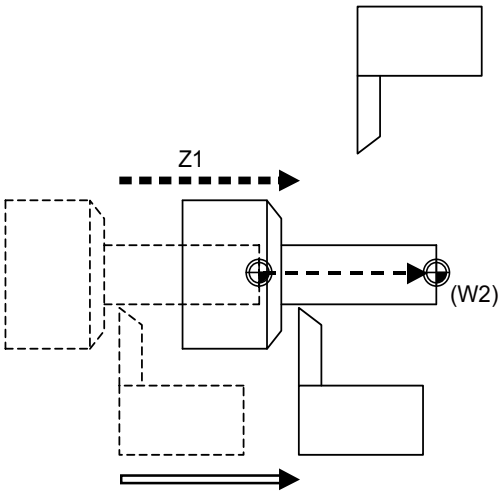
When the movement of reference axis is executed, the superimposed axis workpiece coordinate zero point moves according to the movement of the superimposed axis. The superimposed axis moves as much as the reference axis to maintain the workpiece position.
For the rapid traverse rate and the clamp rate, refer to the section of "Traverse rate of reference axis/superimposed axis (superimposition of 2 axes and 3 axes tandem superimposition)".



[Axis movement during the superimposition control]

(1) When commanding movement only to the reference axis

If a movement command is issued only to the reference axis and not to the superimposed axis when the control axis is superimposed, the superimposed axis moves as same as the reference axis. And the workpiece zero point of the superimposed axis moves as much as the movement amount of the reference axis.

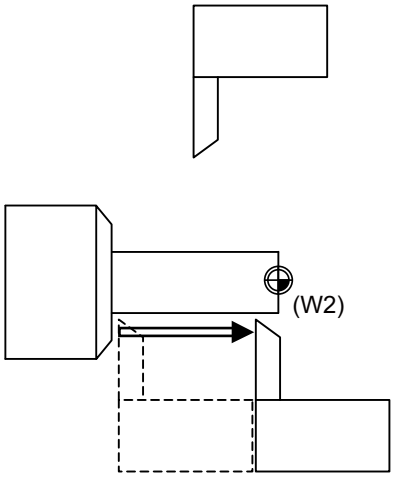


Z1 actual movement amount = Z1 commanded movement amount

Z2 actual movement amount = Z1 actual movement amount

(2) When commanding movement only to the superimposed axis

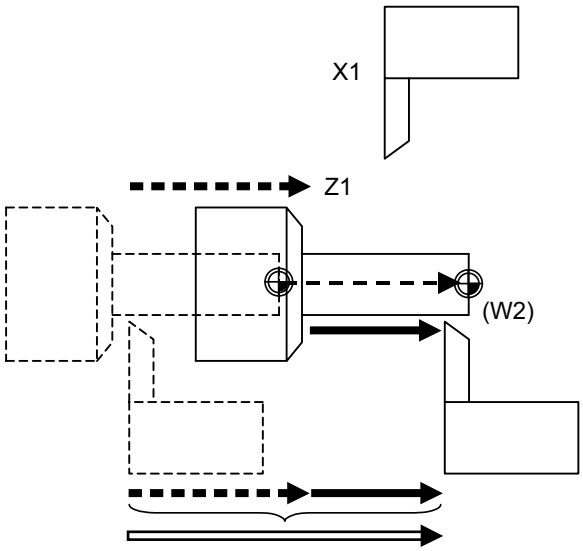
If a movement command is issued only to the superimposed axis and not to the reference axis when the control axis is superimposed, the superimposed axis actual movement amount equals to that of the reference axis. And the workpiece zero point of the superimposed axis will not move because the reference axis will not move.



Z2 actual movement amount = Z2 commanded movement amount

(3) When commanding movement to both the reference axis and superimposed axis

If a movement command is issued to both the reference axis and superimposed axis at the same time when the control axis is superimposed, the superimposed axis actual movement amount equals to "reference axis movement amount + superimposed axis commanded movement amount. And the relative movement amount of the superimposed axis to the workpiece equals the commanded movement amount of the superimposed axis. And the workpiece zero point of the superimposed axis moves as much as the movement amount of the reference axis.



Z1 actual movement amount = Z1 commanded movement amount

Z2 commanded movement amount

Z2 actual movement amount = Z1 actual movement amount + Z2 commanded movement amount

The feedrate of the reference axis and superimposed axis (superimposition of 2 axes and 3 axes tandem superimposition)

(1) Operation in the superimposition of 2 axes

If the movement command is issued to both the reference axis and the superimposed axis, the movement rate of the superimposed axis will be faster than when the movement is commanded only by the superimposed axis as long as the moving direction of the superimposed axis synchronized with the reference axis movement is the same as that commanded only by the superimposed axis.

In this case, in the normal clamp speed process, as the speed may become faster than that of the motor capacity, the rapid traverse rate and clamp rate of the reference axis and superimposed axis are calculated according to the following table.

Superimposed axis		Reference axis			
		Stop	Rapid traverse	Cutting feed	Thread cutting
Stop		Reference axis: Stop Superimposed axis: Stop	Reference axis: [#2001 rapid] Superimposed axis: Stop	Reference axis: [#2002 clamp] Superimposed axis: Stop	Reference axis: (*1) Superimposed axis: Stop
Rapid traverse	Move in the same direction as the reference axis	Reference axis: Stop Superimposed axis: [#2001 rapid]	Reference axis: [#2090 plrapid] Superimposed axis: [#2090 plrapid]	Reference axis: [#2002 clamp] Superimposed axis: [#2621 plrapid2]	Reference axis: (*1) Superimposed axis: [#2621 plrapid2]
	Move to the opposite direction of the reference axis		Reference axis: [#2001 rapid] Superimposed axis: [#2001 rapid]	Reference axis: [#2002 clamp] Superimposed axis: [#2001 rapid]	Reference axis: (*1) Superimposed axis: [#2001 rapid]
Cutting feed	Move in the same direction as the reference axis	Reference axis: Stop Superimposed axis: [#2002 clamp]	Reference axis: [#2621 plrapid2] Superimposed axis: [#2002 clamp]	Reference axis: [#2091 plclamp] Superimposed axis: [#2091 plclamp]	Reference axis: (*1) Superimposed axis: [#2091 plclamp]
	Move to the opposite direction of the reference axis		Reference axis: [#2001 rapid] Superimposed axis: [#2002 clamp]	Reference axis: [#2002 clamp] Superimposed axis: [#2002 clamp]	Reference axis: (*1) Superimposed axis: [#2002 clamp]
Thread cutting	Move in the same direction as the reference axis	Reference axis: Stop Superimposed axis: (*1)	Reference axis: [#2621 plrapid2] Superimposed axis: (*1)	Reference axis: [#2091 plclamp] Superimposed axis: (*1)	Reference axis: (*1) Superimposed axis: (*1)
	Move to the opposite direction of the reference axis		Reference axis: [#2001 rapid] Superimposed axis: (*1)	Reference axis: [#2002 clamp] Superimposed axis: (*1)	Reference axis: (*1) Superimposed axis: (*1)

(*1) If the spindle rotation speed is faster than [#2091 plclamp] when thread cutting is started, cutting will not start. (The operation error (M01 0107) will occur.)

(2) Operation in the 3 axes tandem superimposition

As with the case of superimposition of 2 axes, depending on the movement direction of the reference axis, the 1st imposed axis and the 2nd imposed axis, the feed rate may be faster than the rate of movement by the command of the 1st imposed axis or the 2nd imposed axis. Therefore, calculate the rapid traverse rate and the clamp rate of the reference axis, the 1st superimposed axis or the 2nd superimposed axis, following the table below.

(a) Rapid traverse rate and clamp rate of the reference axis

1st superimposed axis		2nd superimposed axis		Reference axis				
				Stop	Rapid traverse		Cutting feed	
					+	-	+	-
Stop		Stop		Stop	#2001 rapid	#2001 rapid	#2002 clamp	#2002 clamp
		Rapid traverse	+	Stop	#2090 plrapid	#2001 rapid	#2002 clamp	#2002 clamp
			-	Stop	#2001 rapid	#2090 plrapid	#2002 clamp	#2002 clamp
		Cutting feed (Thread cutting)	+	Stop	#2621 plrapid2	#2001 rapid	#2091 plclamp	#2002 clamp
			-	Stop	#2001 rapid	#2621 plrapid2	#2002 clamp	#2091 plclamp
Rapid traverse	+	Stop		Stop	#2090 plrapid	#2001 rapid	#2002 clamp	#2002 clamp
		Rapid traverse	+	Stop	#2626 pl3rapid	#2001 rapid	#2002 clamp	#2002 clamp
			-	Stop	#2090 plrapid	#2001 rapid	#2002 clamp	#2002 clamp
		Cutting feed (Thread cutting)	+	Stop	#2627 pl3rapid2	#2001 rapid	#2630 pl3clamp2	#2002 clamp
			-	Stop	#2090 plrapid	#2001 rapid	#2002 clamp	#2002 clamp
	-	Stop		Stop	#2001 rapid	#2090 plrapid	#2002 clamp	#2002 clamp
		Rapid traverse	+	Stop	#2001 rapid	#2090 plrapid	#2002 clamp	#2002 clamp
			-	Stop	#2001 rapid	#2626 pl3rapid	#2002 clamp	#2002 clamp
		Cutting feed (Thread cutting)	+	Stop	#2001 rapid	#2090 plrapid	#2002 clamp	#2002 clamp
			-	Stop	#2001 rapid	#2627 pl3rapid2	#2002 clamp	#2630 pl3clamp2
Cutting feed (Thread cutting)	+	Stop		Stop	#2621 plrapid2	#2001 rapid	#2091 plclamp	#2002 clamp
		Rapid traverse	+	Stop	#2627 pl3rapid2	#2001 rapid	#2630 pl3clamp2	#2002 clamp
			-	Stop	#2621 plrapid2	#2090 plrapid	#2091 plclamp	#2002 clamp
		Cutting feed (Thread cutting)	+	Stop	#2628 pl3rapid3	#2001 rapid	#2629 pl3clamp	#2002 clamp
			-	Stop	#2621 plrapid2	#2621 plrapid2	#2091 plclamp	#2091 plclamp
	-	Stop		Stop	#2001 rapid	#2621 plrapid2	#2002 clamp	#2091 plclamp
		Rapid traverse	+	Stop	#2090 plrapid	#2621 plrapid2	#2002 clamp	#2091 plclamp
			-	Stop	#2001 rapid	#2627 pl3rapid2	#2002 clamp	#2630 pl3clamp2
		Cutting feed (Thread cutting)	+	Stop	#2621 plrapid2	#2621 plrapid2	#2091 plclamp	#2091 plclamp
			-	Stop	#2001 rapid	#2628 pl3rapid3	#2002 clamp	#2629 pl3clamp

If the spindle rotation speed at thread cutting is faster than the clamp rate given in the MTB specifications, cutting will not start. (The operation error (M01 0107) will occur.)

(b) Rapid traverse rate and clamp rate of the 1st superimposed axis

Reference axis		2nd superimposed axis		1st superimposed axis				
				Stop	Rapid traverse		Cutting feed	
					+	-	+	-
Stop		Stop		Stop	#2001 rapid	#2001 rapid	#2002 clamp	#2002 clamp
		Rapid traverse	+	Stop	#2090 plrapid	#2001 rapid	#2002 clamp	#2002 clamp
			-	Stop	#2001 rapid	#2090 plrapid	#2002 clamp	#2002 clamp
		Cutting feed (Thread cutting)	+	Stop	#2621 plrapid2	#2001 rapid	#2091 plclamp	#2002 clamp
			-	Stop	#2001 rapid	#2621 plrapid2	#2002 clamp	#2091 plclamp
Rapid traverse	+	Stop		Stop	#2090 plrapid	#2001 rapid	#2002 clamp	#2002 clamp
		Rapid traverse	+	Stop	#2626 pl3rapid	#2001 rapid	#2002 clamp	#2002 clamp
			-	Stop	#2090 plrapid	#2001 rapid	#2002 clamp	#2002 clamp
		Cutting feed (Thread cutting)	+	Stop	#2627 pl3rapid2	#2001 rapid	#2630 pl3clamp2	#2002 clamp
			-	Stop	#2090 plrapid	#2001 rapid	#2002 clamp	#2002 clamp
	-	Stop		Stop	#2001 rapid	#2090 plrapid	#2002 clamp	#2002 clamp
		Rapid traverse	+	Stop	#2001 rapid	#2090 plrapid	#2002 clamp	#2002 clamp
			-	Stop	#2001 rapid	#2626 pl3rapid	#2002 clamp	#2002 clamp
		Cutting feed (Thread cutting)	+	Stop	#2001 rapid	#2090 plrapid	#2002 clamp	#2002 clamp
			-	Stop	#2001 rapid	#2627 pl3rapid2	#2002 clamp	#2630 pl3clamp2
Cutting feed (Thread cutting)	+	Stop		Stop	#2621 plrapid2	#2001 rapid	#2091 plclamp	#2002 clamp
		Rapid traverse	+	Stop	#2627 pl3rapid2	#2001 rapid	#2630 pl3clamp2	#2002 clamp
			-	Stop	#2621 plrapid2	#2090 plrapid	#2091 plclamp	#2002 clamp
		Cutting feed (Thread cutting)	+	Stop	#2628 pl3rapid3	#2001 rapid	#2629 pl3clamp	#2002 clamp
			-	Stop	#2621 plrapid2	#2621 plrapid2	#2091 plclamp	#2091 plclamp
	-	Stop		Stop	#2001 rapid	#2621 plrapid2	#2002 clamp	#2091 plclamp
		Rapid traverse	+	Stop	#2090 plrapid	#2621 plrapid2	#2002 clamp	#2091 plclamp
			-	Stop	#2001 rapid	#2627 pl3rapid2	#2002 clamp	#2630 pl3clamp2
		Cutting feed (Thread cutting)	+	Stop	#2621 plrapid2	#2621 plrapid2	#2091 plclamp	#2091 plclamp
			-	Stop	#2001 rapid	#2628 pl3rapid3	#2002 clamp	#2629 pl3clamp

If the spindle rotation speed at thread cutting is faster than the clamp rate given in the MTB specifications, cutting will not start. (The operation error (M01 0107) will occur.)

(c) Rapid traverse rate and clamp rate of the 2nd superimposed axis

Reference axis		1st superimposed axis		2nd superimposed axis				
				Stop	Rapid traverse		Cutting feed	
					+	-	+	-
Stop		Stop		Stop	#2001 rapid	#2001 rapid	#2002 clamp	#2002 clamp
		Rapid traverse	+	Stop	#2090 plrapid	#2001 rapid	#2002 clamp	#2002 clamp
			-	Stop	#2001 rapid	#2090 plrapid	#2002 clamp	#2002 clamp
		Cutting feed (Thread cutting)	+	Stop	#2621 plrapid2	#2001 rapid	#2091 plclamp	#2002 clamp
			-	Stop	#2001 rapid	#2621 plrapid2	#2002 clamp	#2091 plclamp
Rapid traverse	+	Stop		Stop	#2090 plrapid	#2001 rapid	#2002 clamp	#2002 clamp
		Rapid traverse	+	Stop	#2626 pl3rapid	#2001 rapid	#2002 clamp	#2002 clamp
			-	Stop	#2001 rapid	#2001 rapid	#2002 clamp	#2002 clamp
		Cutting feed (Thread cutting)	+	Stop	#2627 pl3rapid2	#2001 rapid	#2630 pl3clamp2	#2002 clamp
			-	Stop	#2090 plrapid	#2001 rapid	#2002 clamp	#2002 clamp
	-	Stop		Stop	#2001 rapid	#2090 plrapid	#2002 clamp	#2002 clamp
		Rapid traverse	+	Stop	#2001 rapid	#2001 rapid	#2002 clamp	#2002 clamp
			-	Stop	#2001 rapid	#2626 pl3rapid	#2002 clamp	#2002 clamp
		Cutting feed (Thread cutting)	+	Stop	#2001 rapid	#2090 plrapid	#2002 clamp	#2002 clamp
			-	Stop	#2001 rapid	#2627 pl3rapid2	#2002 clamp	#2630 pl3clamp2
Cutting feed (Thread cutting)	+	Stop		Stop	#2621 plrapid2	#2001 rapid	#2091 plclamp	#2002 clamp
		Rapid traverse	+	Stop	#2627 pl3rapid2	#2001 rapid	#2630 pl3clamp2	#2002 clamp
			-	Stop	#2001 rapid	#2090 plrapid	#2002 clamp	#2002 clamp
		Cutting feed (Thread cutting)	+	Stop	#2628 pl3rapid3	#2001 rapid	#2629 pl3clamp	#2002 clamp
			-	Stop	#2621 plrapid2	#2621 plrapid2	#2091 plclamp	#2091 plclamp
	-	Stop		Stop	#2001 rapid	#2621 plrapid2	#2002 clamp	#2091 plclamp
		Rapid traverse	+	Stop	#2090 plrapid	#2001 rapid	#2002 clamp	#2002 clamp
			-	Stop	#2001 rapid	#2627 pl3rapid2	#2002 clamp	#2630 pl3clamp2
		Cutting feed (Thread cutting)	+	Stop	#2621 plrapid2	#2621 plrapid2	#2091 plclamp	#2091 plclamp
			-	Stop	#2001 rapid	#2628 pl3rapid3	#2002 clamp	#2629 pl3clamp

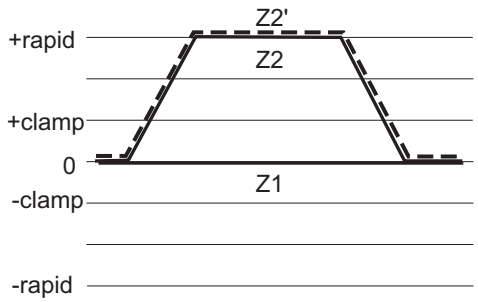
If the spindle rotation speed at thread cutting is faster than the clamp rate given in the MTB specifications, cutting will not start. (The operation error (M01 0107) will occur.)

Composition of axis movement for superimposed axis

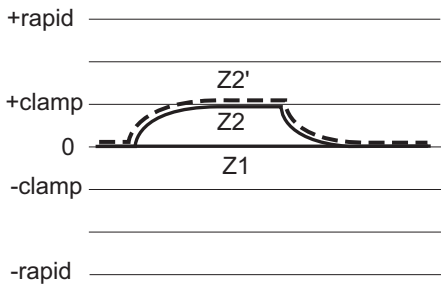
For details on "rapid" and "clamp" in the figure, refer to the section "The feedrate of the reference axis and superimposed axis".

(1) Superimposition of 2 axes (example of reference axis Z1 and superimposed axis Z2)
In the figure, Z1 shows the operation of the reference axis only, Z2 shows the operation of superimposed axis only, Z2' shows the operation example of ((reference axis) + (superimposed axis)).

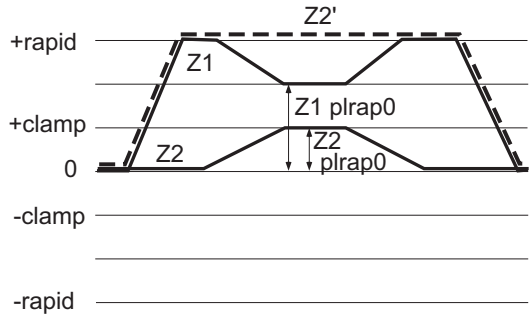
(a) Z1 Stop, Z2 Rapid traverse



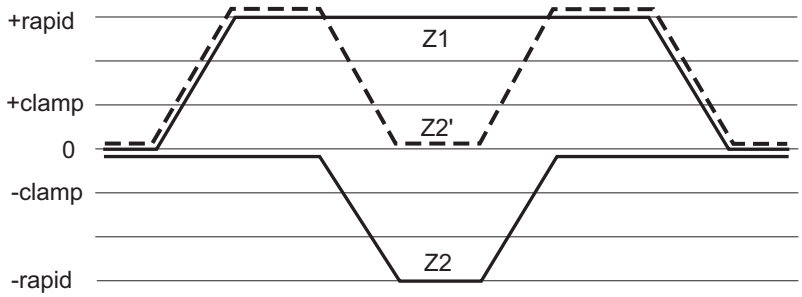
(b) Z1 Stop, Z2 Cutting feed



(c) Z1 Rapid traverse, Z2 (same direction)

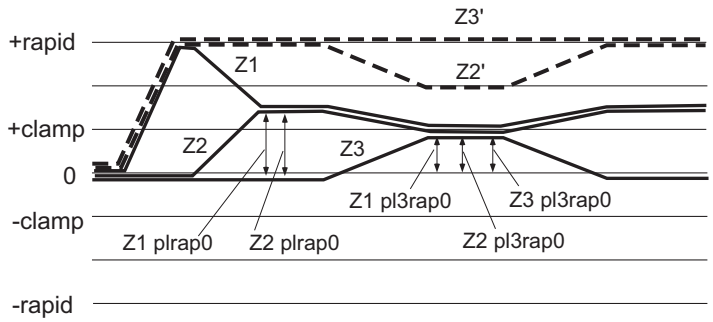


(d) Z1 Rapid traverse, Z2 (different direction)

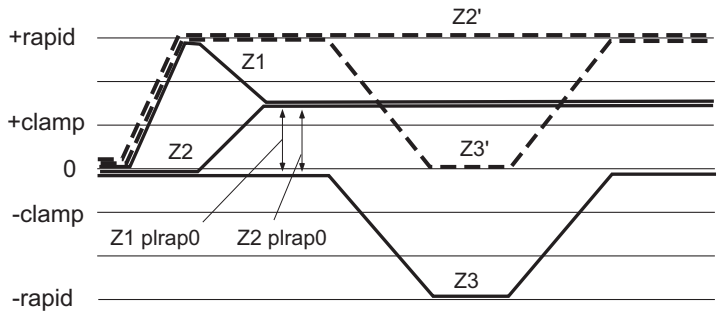


(2) 3 axes tandem superimposition (example of reference axis Z1 1st superimposed axis Z2, and 2nd superimposed axis Z3)
In the figure, Z1 shows the operation of the reference axis only, Z2 shows the operation of superimposed axis only, Z2' shows the operation example of ((reference axis) + (1st superimposed axis), Z3' shows the operation example of ((reference axis) + (1st superimposed axis) + (2nd superimposed axis)).

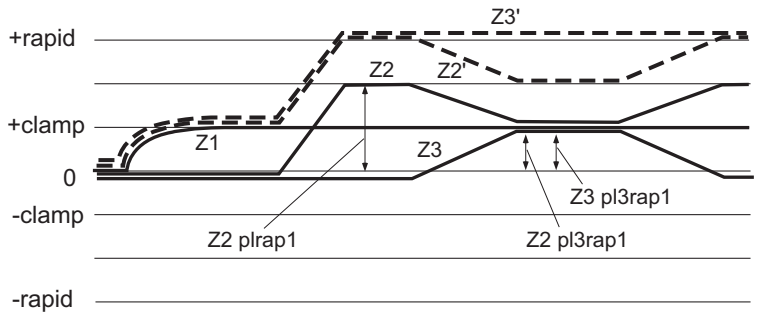
(a) Z1 Rapid traverse, Z2 Rapid traverse, Z3 Rapid Traverse (all axes in the same direction)



(b) Z1 Rapid traverse, Z2 Rapid traverse , Z3 Rapid traverse (only Z3 in the opposite direction)



(c) Z1 Rapid traverse, Z2 Rapid traverse, Z3 Rapid traverse (all axes in the same direction)



End of superimposition of 2 axes

This section shows the examples of executing an ending operation when the Z2 axis is superimposed onto Z1 axis.

Command: G126 Z2 ;

The superimposition end command automatically executes the following operation.

(1) After completing the currently processing block, the part system containing the reference axis and superimposed axis decelerates and stops.

(2) Switch the time constant of all axes of the part system containing the reference axis and superimposed axis, to the normal value (set in the MTB specifications).

(3) Return the superimposed axis workpiece coordinate zero point to where it was before the superimposition start command was issued. (The superimposed axis will not move.)

(4) The superposition control of 2 axes, Z1 axis (reference axis) and Z2 axis (superimposed axis), ends.

3 axes tandem superimposition ends

This section shows the example of executing an ending operation from the 3 axes tandem superimposition of Z1 axis (reference axis)/Z2 axis (1st superimposed axis)/Z3 axis (2nd superimposed axis).

Command : G126 Z2 ;

The superimposition end command automatically executes the following operation.

(1) Decelerates to a stop after executing a block processing the part system containing Z1 axis (reference axis)/Z2 axis (1st superimposed axis)/Z3 axis (2nd superimposed axis).

(2) Switch the time constant of all axes of the part system containing Z2 axis and Z3 axis into the time constant for the superimposition of 2 axes set in the parameter.
Switch the time constant of all axes of the part system containing Z1 axis into the normal time constant set in the parameter.

(3) Return Z2 axis workpiece coordinate zero point to where it was before the superimposition start command was issued. (The superimposed axis will not move.)

(4) First, superposition control of Z1/Z2 ends and tandem superimposition on Z1 axis/Z2 axis/Z3 axis will be eliminated.
(Superimposition of 2 axes is created with Z2 axis/Z3 axes.) For ending the superimposition of 2 axes, refer to the explanation of the superimposition of 2 axes.

Program example

Example of superimposition of 2 axis

(1) When commanding from the part system containing the reference axis

[1st part system]	[2nd part system]	Operation
:	:	
N10 !L1;	N20 !L1;	
N11 G126 Z2=Z1;	:	Start superimposing Z2 axis on Z1 axis
N12 !L2;	N21 !L2;	Superimposition of 2 axis with Z1 axis (reference axis)/Z2 axis (superimposed axis)
N13 G01 Z50.F100;	N22 G01 X-10. F100;	
N14 X4. Z4.;	N23 Z60.;	
:	:	
N15 !L3;	N24 !L3;	
N16 G126 Z2;	:	End of superimposition of Z2 axis
:	:	

(2) When commanding from a part system containing neither the superimposed axis nor reference axis

[1st part system]	[2nd part system]	Operation
:	:	
N10 !L1;	N20 !L1;	
:	N21 G126 Z2=Z1;	Start superimposing Z2 axis on Z1 axis
N12 !L2;	N22 !L2;	Superimposition of 2 axis with Z1 axis (reference axis)/Z2 axis (superimposed axis)
N13 G01 Z50.F100;	N22 G01 X-10. F100;	
N14 X4. Z4.;	N23 Z60.;	
:	:	
N15 !L3;	N24 !L3;	
:	N16 G126 Z2;	End of superimposition of Z2 axis
:	:	

(3) When commanding from a part system containing neither the superimposed axis nor reference axis

[1st part system]	[2nd part system]	[3rd part system]	Operation
:	:	:	
N10 !2!3L1;	N20 !1!3L1;	N30 !1!2L1;	
:	:	N31 G126 Z2=Z1;	Start superimposing Z2 axis on Z1 axis
N11 !2!3L2;	N21 !1!3L2;	N32 !1!2L2;	Superimposition of 2 axis with Z1 axis (reference axis)/Z2 axis (superimposed axis)
N12 G01 Z50. F100;	N22 G01 X-10. F100;	N33 G01 X10.Z50. F100;	
N13 X4.Z4.;	N23 Z60.;	N34 G00 X10.;	
:	:	:	
N14 !2!3L3;	N24 !1!3L3;	N35 !1!2L3;	
:	:	N36 G126 Z2;	End of superimposition of Z2 axis
:	:	:	

Example of superimposition of 3 axis

[1st part system]	[2nd part system]	[3rd part system]	Operation
:	:	:	
N10 !2L1;	N20 !1L1;	:	
:	N21 G126 Z2=Z1;	:	Z2 axis starts superimposing on Z1 axis
N11 !2L2;	N21 !1L2;	:	Superimposition of 2 axis with Z1 axis (reference axis)/Z2 axis (superimposed axis)
N12 G01 Z50. F100;	N22 G01 X-10. F100;	:	
N13 X4.Z4.;	N23 Z60.;	:	
:	:	:	
N14 !2!3L3;	N24 !1!3L3;	N31 !1!2L3;	
:	:	N32 G126 Z3=Z2;	Z3 axis starts superimposing on Z2 axis
:	:	N33 G01 X10.Z50. F100;	3 axes tandem superimposition of Z1 axis (reference axis)/Z2 axis (1st superimposed axis)/Z3 axis (2nd superimposed axis).
:	:	N34 G00 X10.;	
:	:	:	
N15 !2!3L4;	N25 !1!3L4;	N35 !1!2L4;	
:	:	N36 G126 Z3;	End of superimposition of Z3 axis
N16 !2!3L5;	N26 !1!3L5;	N37 !1!2L5;	Superimposition of 2 axis with Z1 axis (reference axis)/Z2 axis (superimposed axis)
:	:	:	
N17 !2!3L6;	N27 !1!3L6;	N38 !1!2L6;	
:	N28 G126 Z2;	:	End of superimposition of Z2 axis
:	:	:	

Relationship with Other Functions

Functions which cannot be used during control axis superimposition

(1) The operation error (M01 1003) will occur if the following commands are issued to the superimposed axis and reference axis during the control axis superimposition.

<Reference axis>

Function	G code
♦Zero point restoration command (The axis will move up to the intermediate point.) ♦Skip command ♦Tool change position return	G28 G31

<Superimposed axis>

Function	G code
♦Zero point restoration command (The axis will move up to the intermediate point.) ♦Skip command ♦Machine coordinate system selection command ♦Tool change position return	G28 - G30 G31 G53

(2) If any of the following commands is issued to the related axes of control axis superimposition, an operation error will occur.

♦Arbitrary axis superimposition command (M01 1004)
♦Auxiliary axis synchronization (M01 1004)
♦Synchronous control (M01 1036)
♦Control axis synchronization between part systems command (M01 1037)

(3) If any of the following commands is executed in the part system in which the related axes to control axis superimposition exists, the program error (P29) will occur.

♦High-speed machining mode II
♦High-speed high-accuracy control I
♦High-speed high-accuracy control II

Axes that cannot be specified as the related axes of control axis superimposition

(1) If an axis of the part system, for which either of the following functions is being executed, is specified as the related axis of control axis superimposition, these functions will be cancelled and the control axis superimposition will be enabled.
When the control axis superimposition ends, the function will resume.

♦High-speed machining mode II
♦High-accuracy control
♦High-speed High-accuracy control I/II

(2) If an axis of part system performing the following functions is specified as the related axis of control axis superimposition, the program error (P34) will occur.

♦Programmable coordinate rotation mode

(3) If any of the following axes is specified as the related axis of control axis superimposition, the operation error (M01 1004) will occur.

♦Master axis or slave axis of synchronous control
♦Reference axis or synchronized axis of control axis synchronization between part systems
♦Reference axis or superimposed axis of arbitrary axis superimposition command
♦Reference axis or synchronized axis being synchronized with auxiliary axis
♦Reference axis, 1st superimposed axis or 2nd superimposed axis of 3 axes tandem superimposition
♦Both reference axis and superimposed axis are reference axis and superimposed axis of superimposition
♦Milling interpolation axis
♦Axis in fixed cycle mode
♦Axis in mirror image for facing tool posts
♦Axis in the external mirror image
♦Axis in parameter mirror image
♦Axis with servo OFF in spindle position control

(4) If any of the following axes is specified as the related axis of control axis superimposition, the program error (P520) will occur.

- Basis axis of inclined axis control/selection axis of inclined axis control
  (axis whose parameter "#2071 s_axis" is set to a value other than "0" in the MTB specifications)
- Axis in a part system in which acceleration/deceleration before interpolation is selected
  (axis whose parameter "#1205 G0bdcc" (acceleration/deceleration before G0 interpolation) is set to a value other than "0" in the MTB specifications)

Precautions

Superimposition start command and precautions during operation

(1) The superimposition start command can be issued from a part system which does not contain the superimposed axis/reference axis.

(2) The axis name set in the parameter "#1022 axname2" must be used to issue the superimposition start command. A program error (P520) will occur if an axis which is not set in the parameter is commanded. Parameter "#1022 axname2" depends on the MTB specifications.

(3) Program error (P520) will occur if a same axis is designated as the superimposed axis and reference axis at the superimposition start command.

(4) Program error (P33) will occur if P address is written without comma at the superimposition start command.

(5) A program error (P33) will occur if G126 alone is issued.

(6) Program error (P520) will occur if a rotary axis is selected as the superimposed axis/reference axis.

(7) In G126 block, a superimposition command can be issued only to one set of axes. Program error (P33) will occur if more than 2 sets are commanded.

(8) Command this in an independent block.

(9) When P command is P0, the superimposed axis workpiece coordinate zero point overlaps with that of the reference axis.

(10) When there is no P command, the superimposed axis workpiece coordinate zero point will be same as before the superimposition start command.

(11) A program error (P35) will occur if a value exceeding the command range is commanded by P.

(12) If G126 is commanded where there is no specifications for control axis superimposition, a program error (P39) will occur.

(13) G126 command is enabled with the G code lists 2, 3, 4 and 5. If G126 is commanded with the G code 6 or 7, the program error (P34) will occur.

(14) A program error (P520) will occur if a rotary axis is selected for either the reference axis or the superimposed axis.

Precautions for superimposition end command

(1) The superimposition end command can be issued from a part system which does not contain the superimposed axis/reference axis.

(2) The axis name set in the parameter "#1022 axname2" must be used to issue the superimposition end command. A program error (P520) will occur if an axis which is not set in the parameter is commanded. Parameter "#1022 axname2" depends on the MTB specifications.

(3) The superimposition end command will be ignored, if an axis other than superimposed axes is commanded.

(4) A program error (P33) will occur if G126 alone is issued.

(5) In G126 block, a superimposition end command can be issued to only one superimposed axis. A program error (P33) will occur if more than one axis is commanded.

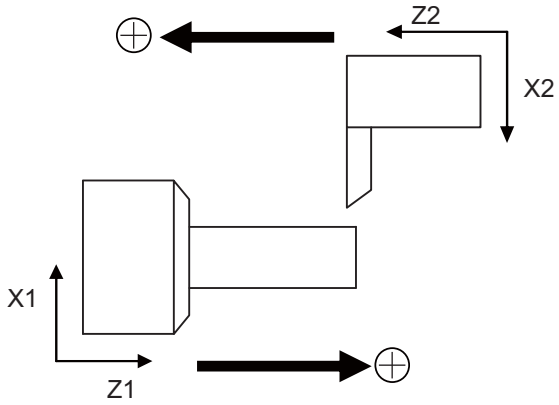
(6) Command this in an independent block.

Other precautions

(1) When being reset during the superimposition, the operation depends on the MTB specifications (parameter "#1280 ext16/bit3").

(2) A timing synchronization operation must be conducted in the block just before the superimposition start/end command, in order to stop the superimposed axis/reference axis and maintain timing between the superimposed axis/reference axis.

(3) Set the relative polarity of the control axes to the parameter (#2143 polar). Any axis can be set as the reference axis in the polarity setting.



(4) The operation error (1004) will occur if a superimposition start command is issued to a superimposed axis or reference axis with other axis before performing a superimposition end command.

(5) Do not use restart search function to commands in the control axis superimposition because the restart position cannot be guaranteed depending on the superimposition movement amount.

(6) The graphic trace in the control axis superimposition may draw a path different from the machining program path.

(7) When conducting an interlock to the superimposed axis, only the commands towards the superimposed axis will be interlocked. Movements accompanying the movement of the reference axis will not be interlocked.

(8) When the emergency stop occurs during the control axis superimposition, the control axis synchronization between part systems will be canceled.
The rapid traverse speed, cutting feedrate clamp speed, and time constant will return to the parameter values set by the MTB after the control axis synchronization between part systems is canceled due to the emergency stop.

(9) The superimposition will be canceled if a servo OFF signal is entered to the superimposed axis/reference axis in the control axis superimposition. The superimposed axis/reference axis must be stopped before entering a servo OFF signal. To confirm the axis stop, check if axis movement + / - signals are turned off.
The method to check the signal operating conditions depends on the MTB specifications.

(10) Start/end of control axis superimposition toward a part system containing the superimposed axis/ reference axis must be issued after all the axes are stopped. To stop all the axes, note that a manual command, chopping, and other factors which is not a machining program may cause an axis to move.

(11) A command deceleration check will be performed to a superimposed axis command even when an in-position check is valid.

(12) All axes in-position signal of the part system containing a superimposed axis will be turned on if the movements commanded by the part system are completed, even when the superimposed axis is moving according to the reference axis.

(13) The reset signals for the part systems containing the superimposed axis and reference axis must be input at the same time.

(14) If the address P of G126 is commanded from an axis which does not contain the superimposed axis, the setting of workpiece zero point by the address P will become valid from the block which is next to the block being executed in the part system containing the superimposed axis.

[1st part system]		[2nd part system]	
:		:	
:		:	
!L1 ;		!L1 ;	
G126 Z2=Z1 ,P20. ;	Superimposing operation		
!L2 ;		!L2 ;	Execute G126 while timing synchronization operation
G00 X46. Z2. ;	Superimposing operation	G01 X-10 F100 ;	From this block, the setting of the workpiece zero point becomes valid
G01 Z-50. F0.1 ;		Z60. ;	
X54. ;		:	
:		:	
:		:	
:		:	
!L3 ;		!L3 ;	
G126 Z2 ;	Superimposing operation		
!L4 ;		!L4 ;	Execute G126 while timing synchronization operation
:		G01 X-10 F100 ;	From this block, the workpiece zero point returns to the original point.
:		:	

(15) To cancel the superimposition, reset the G92 compensation amount of the superimposed axis to the amount before the superimposition start. The G92 compensation amount set by the G50 (G92) command during the superimposition will be invalidated when canceling the superimposition.

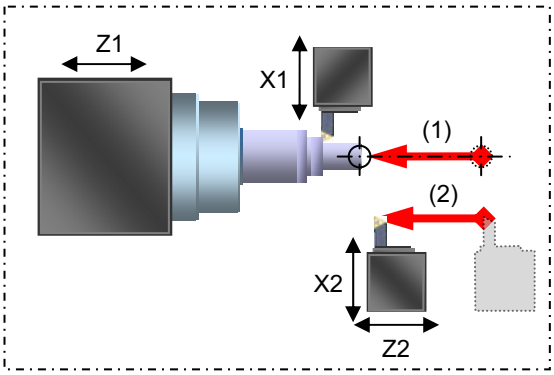
(16) If the following parameters have not been set or a value outside the setting range has been set for axes related to superimposition, an operation error (M01 1070) will occur.

Cutting clamp speed	Rapid traverse rate
#2091 plclamp #2629 pl3clamp #2630 pl3clamp2	#2090 plrapid #2621 plrapid2 #2626 pl3rapid #2627 pl3rapid2 #2628 pl3rapid3

16.4.2 Arbitrary Axis Superimposition ; G156

Function and purpose

With this function, the arbitrary control axis in other part systems can be moved by superimposing on the movement command for the arbitrary control axis in own part system.
The superimposition control start command allows the workpiece coordinate system of the superimposed axis to switch for superimposition control, and the superimposed axis to move to the superimposition start position. (Refer to following figure.)
After the start command is issued, the superimposition control takes effect until the superimposition control end command is issued.
The superimposition control end command allows the workpiece coordinate system of the superimposed axis to return to the state before the superimposition control, that has been switched for superimposition control, and the superimposed axis to move to the superimposition end position.
This function can be used in multi-part systems of two or more part systems.



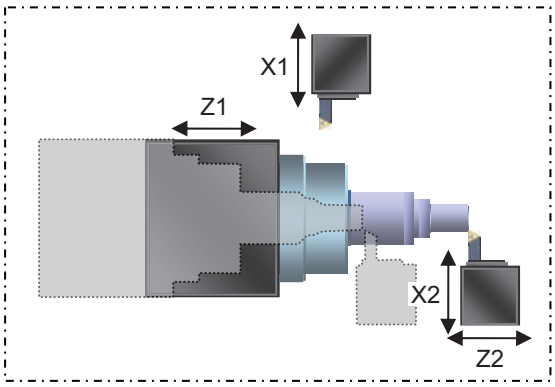
It is assumed that superimposition is performed in the following axis configuration.

- Z1: Reference axis
- Z2: Superimposed axis

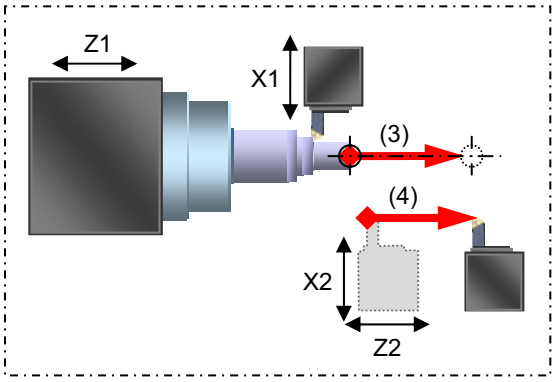
[Start of arbitrary axis superimposition control]
G156 Z2=Z1 ,P20. D10. R30.;

(1) Switching of workpiece zero point of superimposed axis (Z2)

(2) Movement of superimposed axis (Z2) to superimposition control start position



[During arbitrary axis superimposition control]
Z2 axis (superimposed axis) moves by superimposing on the Z1 axis (reference axis).



[End of arbitrary axis superimposition control]
G156 Z2 ,Q20.;

(3) Restore the workpiece zero point of the superimposed axis (Z2) to the position before the arbitrary axis superimposition.

(4) Move the superimposed axis (Z2) to the superimposition control end position.

The following describes the meanings of the terms used in this specifications.

Term	Meaning
Reference axis	The basic in the arbitrary axis superimposition function (moves only by its own axis command).
Superimposed axis	The axis which moves by the reference axis movement in the arbitrary axis superimposition (moves by reference axis command).
Related axis of arbitrary axis superimposition	The reference axis and superimposed axis of arbitrary axis superimposition.

Command format

Arbitrary axis superimposition start command

G156 Superimposed axis name = Reference axis name , P_ D_ R_ F_;

Superimposed axis name	The axis to be operated as superimposed axis (The axis name set in the parameter "#1022 axname2" (two characters))
Reference axis name	The axis to be operated as referenced axis (The axis name set in the parameter "#1022 axname2" (two characters)) If the symbol "-" is added in front of the reference axis name, the superimposed axis can be synchronized in the opposite direction of the reference axis.
, (comma)	Specify as delimiter when the address is needed to be assigned after "superimposed axis name = reference name".
P	Designation of the superimposed axis workpiece coordinate system
D	Superimposed axis workpiece coordinate system offset amount
R	Superimposition control start position
F	Moving speed to superimposition control start position

Command each address with a radius value.

If a value exceeding the command range is issued, a program error (P35) will occur.

Address	Meaning	Command range (unit)	Remarks
P	Designation of superimposed axis workpiece coordinate system	Range of coordinate position command (mm/inch) (Decimal point command is possible.)	♦Commands the workpiece coordinate system's origin of the superimposed axis with the workpiece coordinate system's position of the reference axis. ♦When there is no P command, it will be handled as P0. (The workpiece origins of the reference axis and the superimposed axis will coincide.)
D	Superimposed axis workpiece coordinate system offset amount	Range of coordinate position command (mm/inch) (Decimal point command is possible.)	♦Command the workpiece offset amount when the superimposed axis is used independently. ♦When there is no D command, it will be handled as D0.
R	Superimposition control start position	Range of coordinate position command (mm/inch) (Decimal point command is possible.)	♦Command the position of the superimposed axis when starting the superimposition control with the superimposed axis workpiece coordinate system after the arbitrary axis superimposition command. ♦If there is no address R command, the superimposition control is started from the position without moving the superimposed axis.
F	Moving speed to superimposition control start position	Range of speed command (mm/min, inch/min) (Decimal point command is possible.) (Asynchronous)	♦Command the moving speed of the superimposed axis to the superimposition control start position. ♦If there is no F command, the axis will move at the rapid traverse speed. ♦The F command is unmodal. ♦The F command is asynchronous feed (feed per minute) only. Even in the synchronous feed (feed per revolution) mode, this will be handled as asynchronous feed (feed per minute).

Arbitrary axis superimposition end command

G156 Superimposed axis name, Q/R_ F_;

Superimposed axis name	The axis to be operated as superimposed axis (The axis name set in the parameter "#1022 axname2" (two characters))
, (comma)	Specify as delimiter when the address is needed to be assigned after "superimposed axis name".
Q	Specify the position to end the superimposition control with a basic machine coordinate zero point.
R	Specify the position to end the superimposition control with a workpiece coordinate system.
F	Moving speed to superimposition control end position

Command each address with a radius value.

If a value exceeding the command range is issued, a program error (P35) will occur.

Address	Meaning	Command range (unit)	Remarks
Q	Designation of superimposed axis workpiece coordinate system	Range of coordinate position command (mm/inch) (Decimal point command is possible.)	♦Command the position of the superimposed axis when ending the superimposition control with a position on the basic machine coordinate system. ♦If there is neither address Q nor R command, the superimposition control is ended in the position without moving the superimposed axis. ♦Q takes precedence over R if Q command is issued simultaneously with R command.
R	Superimposition control start position	Range of coordinate position command (mm/inch) (Decimal point command is possible.)	♦Command the position of the superimposed axis when ending the superimposition control with the superimposed axis workpiece coordinate system before the arbitrary axis superimposition command. ♦If there is neither address Q nor R command, the superimposition control is ended in the position without moving the superimposed axis. ♦R takes precedence over Q if R command is issued simultaneously with Q command.
F	Moving speed to superimposition control end position	Range of speed command (mm/min, inch/min) (Decimal point command is possible.) (Asynchronous)	♦Command the moving speed of the superimposed axis to the superimposition control end position. ♦If there is no F command, the axis will move at the rapid traverse speed. ♦The F command is unmodal. ♦The F command is asynchronous feed (feed per minute) only. Even in the synchronous feed (feed per revolution) mode, this will be handled as asynchronous feed (feed per minute).

Detailed description

The following operation example explains the a case when superimposing Z2 axis (superimposed axis) onto Z1 axis (reference axis).

- Z1: Reference axis
- Z2: Superimposed axis

Operation of each axis when the arbitrary axis superimposition start is commanded

The operation of the superimposition start command differs according to the state of the related axis of arbitrary axis superimposition.
The superimposition start command automatically executes the following operation.

Command : G156 Z2=Z1 , Pp Dd Rr;

(1) If the reference axis and superimposed axis are stopped when the arbitrary axis superimposition start is commanded

(a) Set the superimposed axis workpiece zero point by a P command, D command, and the relative distance of the basic machine origin between the reference axis (Z1) and the superimposed axis (Z2) (parameter "#2144 baseps").

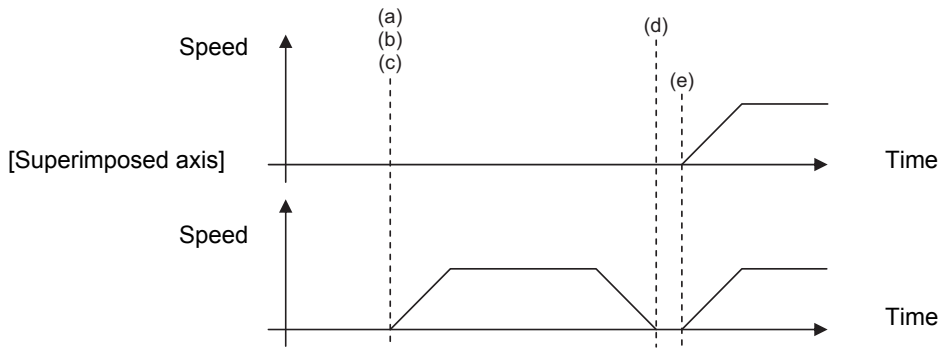
(b) Change the acceleration/deceleration time constants of the reference axis and superimposed axis to the time constants during the superimposition control ("#2092 plG0tL","#2093 plG0t1","#2094 plG1tL","#2095 plG1t1").

(c) The superimposed axis starts to move to the start position of arbitrary axis superimposition.

(d) The start operation of the arbitrary axis superimposition control will be completed when the superimposed axis reaches the start position of arbitrary axis superimposition.
The superimposed axis workpiece coordinate values switch for the arbitrary axis superimposition control at the start of the next block of the superimposed axis part system.

(e) The reference axis starts to move.

[Reference axis]



Movement to start position of arbitrary axis superimposition

<Note>

- The superimposed axis workpiece coordinate values (display) are switched for the arbitrary axis superimposition control at the start of the next block after the movement to the start position of arbitrary axis superimposition is complete. If there is no R command (movement to the start position of arbitrary axis superimposition), it is executed in the next block of the superimposed axis part system after the arbitrary axis superimposition control start command.

(2) If the reference axis is moving when the arbitrary axis superimposition start is commanded from the superimposed axis part system

(a) The operation will wait until smoothing for all axes of the reference axis part system reached zero.

(b) Set the superimposed axis workpiece zero point by a P command, D command, and the relative distance of the basic machine origin between the reference axis (Z1) and the superimposed axis (Z2) (parameter "#2144 baseps"). Change the acceleration/deceleration time constants of the reference axis and superimposed axis to the time constants during the superimposition control ("#2092 plG0tL","#2093 plG0t1","#2094 plG1tL","#2095 plG1t1"). At this time, the arbitrary axis superimposition control becomes valid, and the workpiece coordinate system of the superimposed axis will synchronize with the reference axis movement.

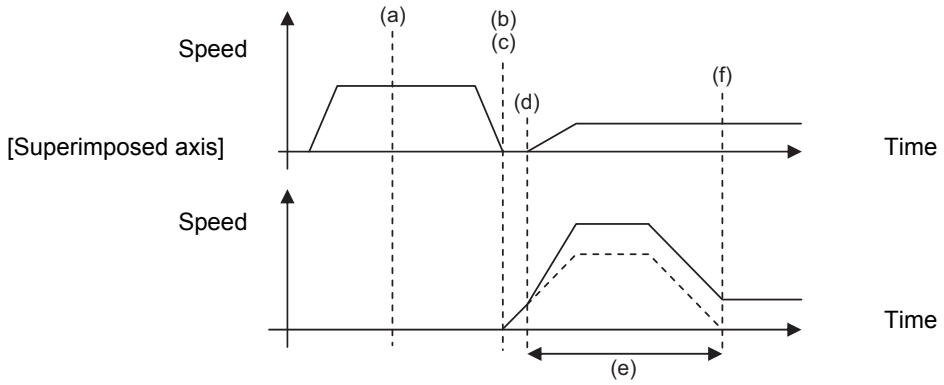
(c) The superimposed axis starts to move to the start position of arbitrary axis superimposition.

(d) The reference axis will start to move.

(e) If the reference axis starts to move before the superimposed axis reaches the start position of arbitrary axis superimposition, the superimposed axis moves by composing the movement amount synchronized with the reference axis and the movement amount to the start position of arbitrary axis superimposition.

(f) The start operation of arbitrary axis superimposition control will be completed when the superimposed axis reaches the start position of arbitrary axis superimposition.

[Reference axis]



The broken line portion indicates the movement amount to the start position of arbitrary axis superimposition.

(3) If the superimposed axis is moving when the arbitrary axis superimposition start is commanded from the reference axis part system

(a) The operation will wait until smoothing for all axes of the superimposed axis part system reached zero. (*1)

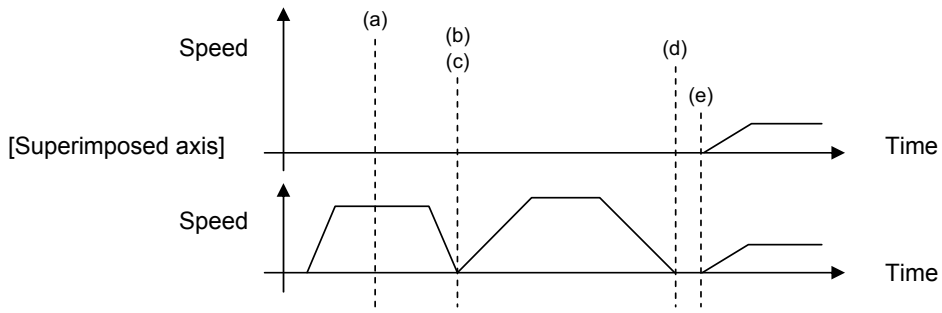
(b) Set the superimposed axis workpiece zero point by a P command, D command, and the relative distance of the basic machine origin between the reference axis (Z1) and the superimposed axis (Z2) (parameter "#2144 baseps").
Change the acceleration/deceleration time constants of the reference axis and superimposed axis to the time constants during the superimposition control ("#2092 plG0tL","#2093 plG0t1","#2094 plG1tL","#2095 plG1t1").

(c) The superimposed axis starts to move to the start position of arbitrary axis superimposition.

(d) The start operation of the arbitrary axis superimposition control will be completed when the superimposed axis reaches the start position of arbitrary axis superimposition. The superimposed axis workpiece coordinate values switch for the arbitrary axis superimposition control at the start of the next block of the superimposed axis part system.

(e) The reference axis starts to move.

[Reference axis]



(*1) Smoothing becomes zero even if the operation is stopped by feed hold, interlock, etc.

Workpiece coordinate and tool compensation when the arbitrary axis superimposition start is commanded

Command:G156 Z2=Z1 , Pp Dd Rr;

The following operation example explains the start command to superimpose Z2 axis onto Z1 axis while the relationship between the workpiece coordinate, tool compensation, and addresses P, D and R, when the arbitrary axis superimposition start is commanded is as shown below.

(1) The workpiece zero point of Z2 axis superimposition can be calculated with the following expression.

Workpiece zero point of Z2 axis superimposition = Workpiece zero point of Z1 axis (+ reference axis tool compensation amount tb) + p (P command value) + d (D command value)

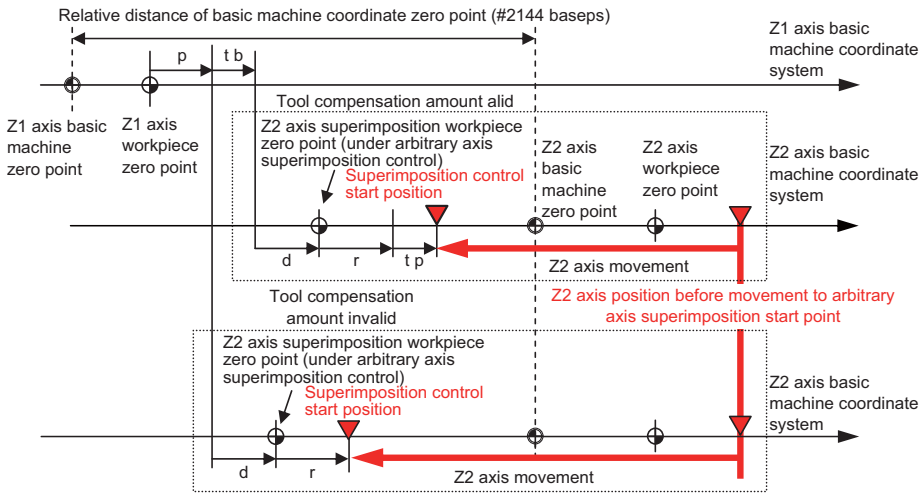
(2) The movement to the superimposition control start position is shifted by the amount commanded by R from the workpiece zero point of Z2 axis superimposition (+ superimposition tool compensation amount tp).

<Note>

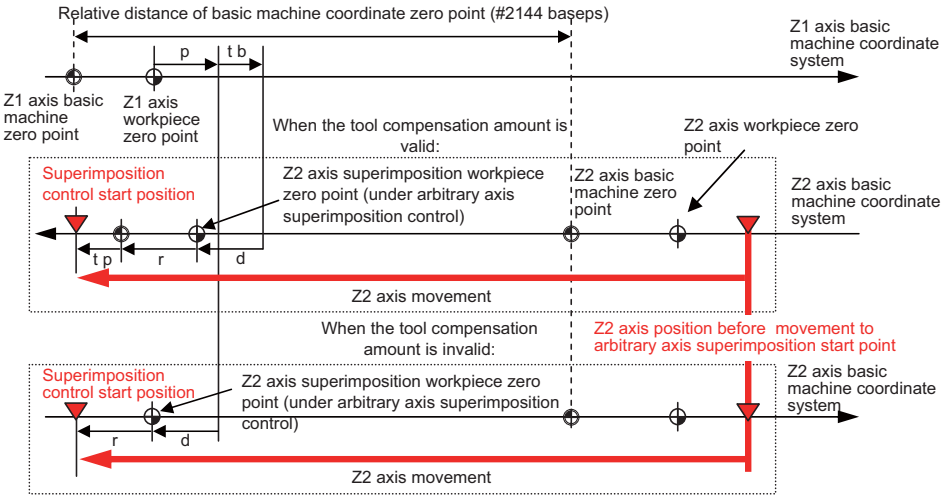
- Switching of the workpiece coordinate system and the validity of tool compensation with reference to the movement to the superimposition start position when the arbitrary axis superimposition control start is commanded depend on the MTB specifications (parameter "#1244 set16/bit3").
  "tb" (reference axis tool compensation amount) and "tp" (superimposed axis tool compensation amount) in above (1) and (2) are added when this parameter is set to enable tool compensation.

The following figure shows the relationship between the workpiece zero point of Z2 axis superimposition, superimposition control start position, tool compensation, addresses P, D and R at the time the arbitrary axis superimposition start is commanded, in a case where (a) relative polarity is present and a case where (b) relative polarity is absent.

(a) Relative polarity (positive) (the value of "#2143 polar" is reference axis: 0, superimposed axis: 0)

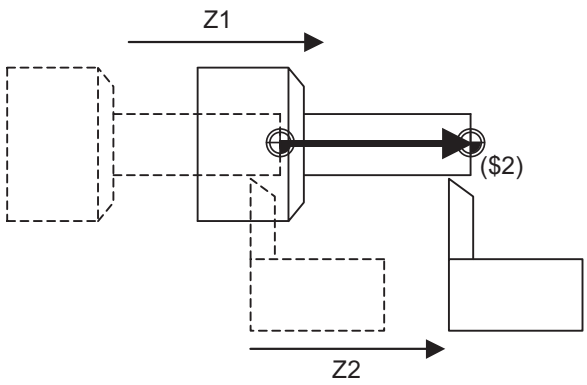


(b) Relative polarity (negative) (the value of "#2143 polar" is reference axis: 0, superimposed axis: 1)



Operation during arbitrary axis superimposition (workpiece coordinate system)

When the movement of reference axis is executed, the superimposed axis workpiece coordinate zero point moves according to the movement of the superimposed axis.
The superimposed axis moves as much as the reference axis to maintain the workpiece position.



Feedrate of the reference axis and superimposed axis

If the movement command is issued to both the reference axis and the superimposed axis, the movement rate of the superimposed axis will be faster than when the movement is commanded only by the superimposed axis as long as the moving direction of the superimposed axis synchronized with the reference axis movement is the same as that commanded only by the superimposed axis.

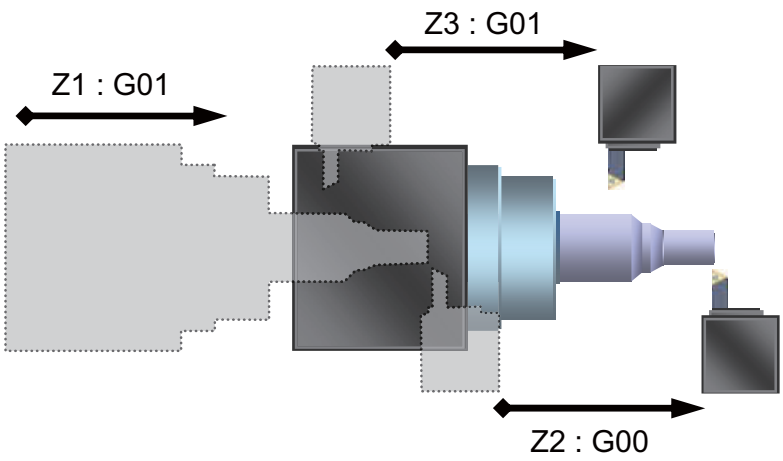
In this case, in the normal clamp speed process, as the speed may become faster than that of the motor capacity, the rapid traverse rate and clamp rate of the reference axis and superimposed axis are calculated according to the following table.

Superimposed axis		Reference axis			
		Stop	Rapid traverse	Cutting feed	Thread cutting
Stop		Reference axis: Stop Superimposed axis: Stop	Reference axis: [#2001 rapid] Superimposed axis: Stop	Reference axis: [#2002 clamp] Superimposed axis: Stop	Reference axis: (*1) Superimposed axis: Stop
Rapid traverse	Move in the same direction as the reference axis	Reference axis: Stop Superimposed axis: [#2001 rapid]	Reference axis: [#2090 plrapid] Superimposed axis: [#2090 plrapid]	Reference axis: [#2002 clamp] Superimposed axis: [#2621 plrapid2]	Reference axis: (*1) Superimposed axis: [#2621 plrapid2]
	Move to the opposite direction of the reference axis		Reference axis: [#2001 rapid] Superimposed axis: [#2001 rapid]	Reference axis: [#2002 clamp] Superimposed axis: [#2001 rapid]	Reference axis: (*1) Superimposed axis: [#2001 rapid]
Cutting feed	Move in the same direction as the reference axis	Reference axis: Stop Superimposed axis: [#2002 clamp]	Reference axis: [#2621 plrapid2] Superimposed axis: [#2002 clamp]	Reference axis: [#2091 plclamp] Superimposed axis: [#2091 plclamp]	Reference axis: (*1) Superimposed axis: [#2091 plclamp]
	Move to the opposite direction of the reference axis		Reference axis: [#2001 rapid] Superimposed axis: [#2002 clamp]	Reference axis: [#2002 clamp] Superimposed axis: [#2002 clamp]	Reference axis: (*1) Superimposed axis: [#2002 clamp]
Thread cutting	Move in the same direction as the reference axis	Reference axis: Stop Superimposed axis: (*1)	Reference axis: [#2621 plrapid2] Superimposed axis: (*1)	Reference axis: [#2091 plclamp] Superimposed axis: (*1)	Reference axis: (*1) Superimposed axis: (*1)
	Move to the opposite direction of the reference axis		Reference axis: [#2001 rapid] Superimposed axis: (*1)	Reference axis: [#2002 clamp] Superimposed axis: (*1)	Reference axis: (*1) Superimposed axis: (*1)

(*1) If the spindle rotation speed is faster than [#2091 plclamp] when thread cutting is started, cutting will not start. (The operation error (M01 0107) will occur.)

In case of a 2-axis superimposition (where there are two superimposed axes while there is one reference axis), the rapid traverse rate and clamp rate of the superimposed axis are calculated according to the following table.
The rapid traverse rate and clamp rate of the reference axis is calculated using either one of the smaller that are determined according to the above table, from the relationship between the reference axis and the superimposed axis of the respective sets of superimposition.

(Example) Clamp rate for 2-axis superimposition (where there are two superimposed axes (Z2 and Z3) while there is one reference axis (Z1))



Z1 reference axis: Cutting feed, Z2 superimposed axis: Cutting feed	→	The Z2 superimposed axis follows [#2091 plclamp].
Z1 reference axis: Cutting feed, Z3 superimposed axis: Rapid traverse	→	The Z3 superimposed axis follows [#2621 plrapid2].
Z1 reference axis: Cutting feed, Z2 superimposed axis: Cutting feed	→	The Z1 reference axis follows [#2091 plclamp].
Z1 reference axis: Cutting feed, Z3 superimposed axis: Rapid traverse	→	The Z1 reference axis will follow [#2002 clamp]. However, the reference axis Z1 will compare [#2091 plclamp] with clamp, and use either one of the smaller for calculation as the clamp value.

Composition of axis movement for superimposed axis

If the movement command is issued to both the reference axis and the superimposed axis, the movement rate of the superimposed axis will be faster than when the movement is commanded only by the superimposed axis as long as the moving direction of the superimposed axis synchronized with the reference axis movement is the same as that commanded only by the superimposed axis.
The superimposed axis will move at (the reference axis movement speed + the superimposed axis movement speed) during the arbitrary axis superimposition control.
For details, refer to the example of 2-axis superimposition in "Composition of axis movement for superimposed axis" of "Control axis superimposition;G126".

Operation of each axis when the arbitrary axis superimposition end is commanded

The following explains the end operation from the arbitrary axis superimposition state of Z1 axis (reference axis) and Z2 axis (superimposed axis).

Command : G156 Z2 , Qq (Rr) Ff;

(1) If the reference axis and superimposed axis are stopped when the arbitrary axis superimposition end is commanded

(a) Return to the normal the acceleration/deceleration time constant of the superimposed axis.

(b) Return the superimposed axis workpiece coordinate zero point to where it was before the superimposition command was issued.

(c) Move the superimposed axis to the arbitrary axis superimposition end position.

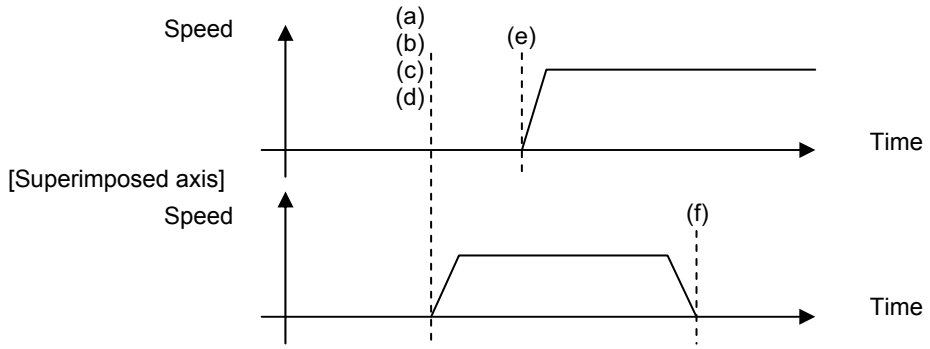
When R command is issued:	Move the axis to the position shifted by R command (+ tool compensation amount) from the workpiece coordinate zero point before the arbitrary axis superimposition command. (+ tool compensation amount) is added only if the tool compensation amount is valid ("#1244 set16/bit3" is "ON").
When Q command is issued:	Move the axis to the basic machine coordinate position commanded by Q. At this time, tool compensation is temporarily canceled.

(d) Return to the normal the acceleration/deceleration time constant of the reference axis.

(e) The reference axis starts to move.

(f) The end operation of arbitrary axis superimposition control will be completed when the superimposed axis reaches the end position of arbitrary axis superimposition.

[Reference axis]



A movement command can be issued to the reference axis while the superimposed axis is moving to the arbitrary axis superimposition end position.
Since the arbitrary axis superimposition end command has been executed, the superimposed axis will not superimpose onto the reference axis movement.

(2) If the reference axis is moving when the arbitrary axis superimposition end is commanded

(a) The operation will wait until smoothing for all axes of the reference axis part system reached zero. (*1)

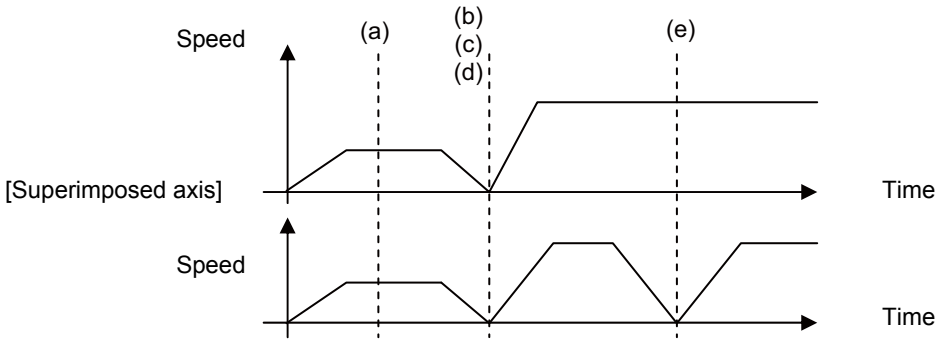
(b) Return to the normal the acceleration/deceleration time constant of the reference axis and superimposed axis.

(c) Return the superimposed axis workpiece coordinate zero point to where it was before the superimposition command was issued.

(d) Move the superimposed axis to the arbitrary axis superimposition end position.

(e) The end operation of arbitrary axis superimposition control will be completed when the superimposed axis reaches the end position of arbitrary axis superimposition.

[Reference axis]



(*1) Smoothing becomes zero even if the operation is stopped by feed hold, interlock, etc.

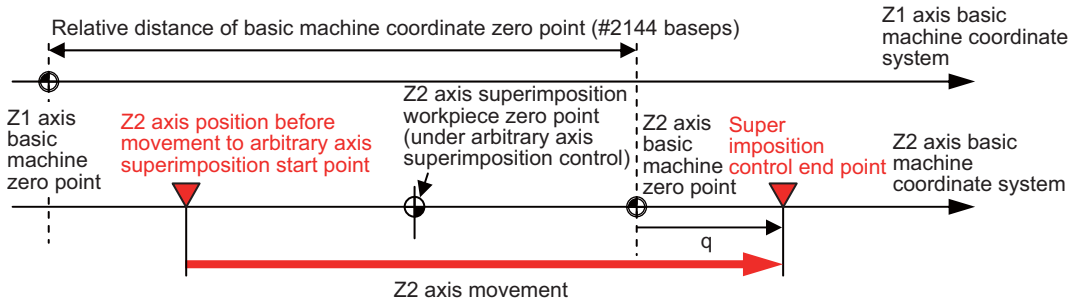
Workpiece coordinate and tool compensation when the arbitrary axis superimposition end is commanded

The following shows the relationship between the workpiece coordinate, tool compensation, and addresses Q and R, when the arbitrary axis superimposition end is commanded.
The following figure shows the relationship between the workpiece zero point of Z2 axis, superimposition control end position, tool compensation, addresses Q and R when the arbitrary axis superimposition end is commanded in a case of (1) Q command (the superimposition control end position is on the basic machine coordinate system) and a case of (2) R command (the superimposition control end position is on the workpiece coordinate system).

(1) Arbitrary axis superimposition end (Q command)
(Where the superimposition control end position is designated by a position on the basic machine coordinate system)

Command:G156 Z2 , Qq;

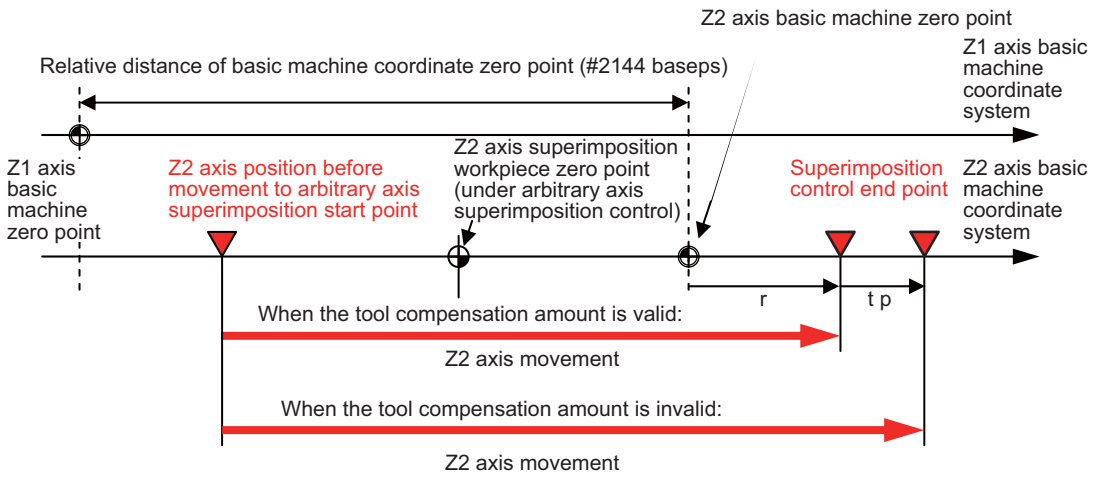
Move the axis to the basic machine coordinate position commanded by Q. At this time, tool compensation is temporarily canceled.



(2) Arbitrary axis superimposition end (R command)
(Where the superimposition control end position is designated by a position on the workpiece coordinate system)

Command:G156 Z2 , Rr;

The superimposition tool compensation amount is assumed as "tp".



The position is shifted by the amount of the position commanded by R (+ superimposed axis tool compensation amount tp).
The validity of tool compensation with reference to the movement to the superimposition end position when the arbitrary axis superimposition control end is commanded depends on the MTB specifications (parameter "#1244 set16/bit3").
The above "tp" value is added when this parameter is set to enable tool compensation.

Program example

(1) Arbitrary axis superimposition command of Z1 axis (reference axis) and Z2 axis (superimposed axis)

[1st part system]	[2nd part system]	Operation
:	:	
G00 Z25.;	:	The timing synchronization operation between part systems allows the reference axis and the superimposed axis to stop, and executes the arbitrary axis superimposition control start command.
	G156 Z2=Z1, P25. D5. R2.5 F1000;	Superimposes Z2 axis onto Z1 axis
!2 L2;	!1 L2;	
G01 Z22. F10;	G01 Z5. F100;	During superimposing operation
G01 X12. F1;	G01 X10. F1;	
:	:	
:	:	
:	:	
:	G156 Z2, Q15.;	Ends superimposition of Z2 axis
:	:	

(2) Arbitrary axis superimposition command of Z1 axis (reference axis), Z2 axis (superimposed axis 1) and Z3 axis (superimposed axis 2)

[1st part system]	[2nd part system]	[3rd part system]	Operation
:	:	:	
G00 Z25.;	:	:	
!2!3 L1;	!1!3 L1;	!1!2 L1;	The timing synchronization operation between part systems allows the reference axis and the superimposed axis to stop, and executes the arbitrary axis superimposition control start command.
	G156 Z2=Z1, P25. D5. R2.5 F500;		Superimposes Z2 axis onto Z1 axis
		G156 Z3=Z1, P20. D10. R5 F500;	Superimposes Z3 axis onto Z1 axis
!2!3 L2;	!1!3 L2;	!1!2 L2;	
G01 Z22. F10;	G01 Z5. F100;	G01 Z5. F100;	During superimposing operation
G01 X12. F1;	G01 X10. F1;	G01 X10. F1;	
:	:	G156 Z3, R15.;	Ends superimposition of Z3 axis At this time, superimposition of Z1 axis and Z2 axis will not end.
:	:	:	
:	:	:	
:	G156 Z2, Q15.;	:	Ends superimposition of Z2 axis
:	:	:	

Relationship with Other Functions

Commands that cannot be issued during the arbitrary axis superimposition control

(1) If any of the following commands is issued to the reference axis, an operation error (M01 1003) will occur.

- Zero point restoration command (The axis will move up to the intermediate point.)
- Skip command
- Tool change position return

(2) If any of the following commands is issued to the superimposed axis, an operation error (M01 1003) will occur.

- Zero point restoration command (The axis will move up to the intermediate point.)
- Skip command
- Machine coordinate system selection command
- Workpiece coordinate system selection command
- Tool change position return

(3) If any of the following commands is issued to the related axis of arbitrary axis superimposition, an operation error will occur.

Command	Operation error
Control Axis Superimposition	Operation error (M01 1004)
Auxiliary axis synchronization	
High-speed machining mode	
Synchronization control	Operation error (M01 1036)
Control axis synchronization between part systems	Operation error (M01 1037)

(4) If any of the following commands is executed in the part system in which the related axis of arbitrary axis superimposition exists, the program error (P29) will occur.

- High-speed machining mode II
- High-speed high-accuracy control I
- High-speed high-accuracy control II

Axes that cannot be specified as the related axis of arbitrary axis superimposition

(1) If an axis of the part system, for which either of the following functions is being executed, is specified as the related axis of arbitrary axis superimposition, these functions will be canceled temporarily, and the control axis superimposition will be enabled.
When the control axis superimposition ends, these functions will resume.

- High-speed machining control
- High-accuracy control
- High-speed high-accuracy control I, II

(2) If an axis, for which any of the following functions are being executed, is specified as the related axis of arbitrary axis superimposition, an operation error (M01 1004) will occur.

- Master axis or slave axis of synchronous control
- Reference axis or synchronized axis of control axis synchronization between part systems
- Reference axis, 1st superimposed axis or 2nd superimposed axis of 3 axes tandem superimposition
- Reference axis or superimposed axis of arbitrary axis superimposition command
- Reference axis or synchronized axis being synchronized with auxiliary axis
- Milling interpolation
- Fixed cycle mode
- Mirror image for facing tool posts
- Mirror image by external input
- Mirror image by parameter setting
- Axis with servo OFF in spindle position control
- Axis of the part system not in automatic operation

(3) If any of the following axes is specified as the related axis of arbitrary axis superimposition, a program error (P520) will occur.

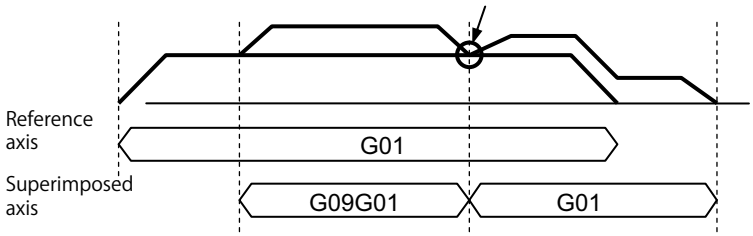
- Superimposed axis in arbitrary axis superimposition control
- The arbitrary axis superimposition command is executed to two axes being controlled in the same part system, by setting them as the reference axis and the superimposed axis.
- Basis axis of inclined axis control/selection axis of inclined axis control
  (Regardless of the inclined axis control valid signal (YC35), the axis whose parameter "#2071 s_axis Inclined axis selection" is not "0")
- Axis in selection part system for acceleration/deceleration before interpolation
  (An axis in the part system whose parameter "#1205 G0bdcc" Acceleration/deceleration before G0 interpolation is not "0")

(4) If an axis of the part system, for which the following function is executed, is specified as the related axis of arbitrary axis superimposition, a program error (P34) will occur.

- Programmable coordinate rotation mode

Other functions

Function name	Operation
Machine lock	When conducting a machine lock to the superimposed axis, only the commands towards the superimposed axis will be machine locked. Note that, movements accompanying the movement of the reference axis will not be machine locked but the machine will move.
Manual arbitrary reverse run	The G156 command cannot run reversed.
Stroke end	Stroke end, stored stroke limit, chuck/tailstock barrier check are checked in the machine position. If the stroke end, stored stroke limit, or chuck/tailstock barrier check alarm occurs to an axis of the part system containing the related axis of arbitrary axis superimposition, the part system containing the reference axis and superimposed axis will stop.
Stored stroke limit	
Chuck Barrier/Tailstock Barrier check	
Graphic check	Start or cancel of superimposition by the arbitrary axis superimposition command is invalid. The graphic trace in the arbitrary axis superimposition may draw a path different from the machining program path.

Function name	Operation
Program restart	Start or cancel of superimposition by the arbitrary axis superimposition command is invalid. Do not use restart search function to commands in the arbitrary axis superimposition because the restart position cannot be guaranteed depending on the superimposition movement amount.
Servo OFF	The superimposition will be canceled if a servo OFF signal is entered to the superimposed axis/reference axis in the control axis superimposition. Make sure that the superimposed axis/reference axis is stopped before entering a servo OFF signal. To confirm the axis stop, check if "axis movement + signal" and "axis movement - signal" are turned off.
Interlock	When conducting an interlock to the superimposed axis, only the commands towards the superimposed axis will be interlocked. Movements accompanying the movement of the reference axis will not be interlocked.
Arbitrary axis exchange control	Commands can be issued to the reference axis of arbitrary axis superimposition. If an axis, which has been set as the uncontrolled axis through the arbitrary axis exchange, becomes the superimposed axis of the arbitrary axis superimposition, an axis in a part system having uncontrolled axis will be used to check for axis stop at the start/end of superimposition. In addition, an axis in a part system, which contains an axis being moved to the start or end position of superimposition, cannot be exchanged as an arbitrary axis. If arbitrary axis exchange is commanded, an operation error (M01 1101) will occur.
Synchronous tapping cycle	When the synchronous tapping is performed with the superimposed axis, set the following values to the gain for the synchronous tapping and the normal gain. •#2203 SV003(PGN) = #2017 tap_g (#2249 SV049(PGN1sp)) •#2204 SV004(PGN2) = #2250 SV050(PGN2sp) •#2257 SV057(SHGC) = #2258 SV058(SHGCsp)
High-speed synchronous tapping	When the high-speed synchronous tapping is performed with the superimposed axis, an in-position check is performed at R point as in-position check cannot be canceled for traveling from the initial point to R point because of the data communication specification between NC and the drive unit.
Deceleration Check	Even if the servo in-position standby condition is applied by the deceleration check function, during the superimposition control, movement of the superimposed axis will be completed when the smoothing reached zero, and execution of the next block will start. Movement of the superimposed axis will be completed when the machine is put into the standby state for smoothing of superimposition component to reach zero. 
NC reset (Reset 1, reset 2 and reset & rewind)	When the system is reset during the arbitrary axis superimposition, select whether to cancel the arbitrary axis superimposition by the parameter "#1280 ext16/bit3". The rapid traverse speed, cutting feedrate clamp speed, and time constant will return to the values set in the parameters after the arbitrary axis superimposition is canceled due to the reset. Input reset to all part systems which are related to the superimposition at a time.
Emergency stop	When the emergency stop occurs during the arbitrary axis superimposition, the arbitrary axis superimposition will be canceled. The rapid traverse speed, cutting feedrate clamp speed, and time constant will return to the values set in the parameters after the arbitrary axis superimposition is canceled due to the emergency stop.

Precautions and restrictions

(1) Designate the superimposed axis/reference axis using the name set in the parameter "#1022 axname2".
A program error (P520) will occur if a name that is not set in the parameter is designated.
The axis name specified in G156 needs to be 2 digits.
(Parameter "#1022 axname2" setting depends on the MTB specifications.)

(2) The arbitrary axis superimposition start command can be issued from a part system that contains a superimposed axis or reference axis.
Otherwise, the operation error (M01 1103) will occur.

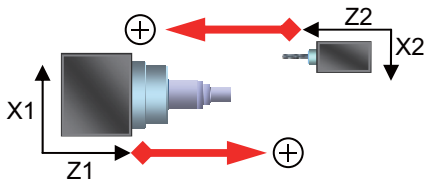
(3) A program error (P520) will occur if a rotary axis is selected for either the reference axis or the superimposed axis.

(4) If the local coordinate system shift command or the counter preset command is issued to the superimposed axis that is under arbitrary axis superimposition control, the preset amount is valid during arbitrary axis superimposition control. However, it does not affect the workpiece coordinate system before arbitrary axis superimposition control start is commanded, which is to be returned when the arbitrary axis superimposition control end command is issued.

(5) If multiple arbitrary axis superimposition commands are issued simultaneously from different part systems, priority is given to the command for which movement is to be started first. After the movement to the start or end position of arbitrary axis superimposition is complete, the other command will be executed.

(6) The arbitrary axis superimposition end command can be issued from a part system which is controlling the superimposed axis.
If the command is issued from another part system to the superimposed axis, an operation error (M01 1103) will occur.

(7) The relative polarity of the control axis depends on the MTB specifications (parameter "#2143 polar").
Any axis can be set as the reference axis in the polarity setting.
If the parameters are set as Z1 axis polar:0(+), and Z2 axis polar:1(-), control axes Z1 and Z2 are as follows.



(8) The time constants of G00 and G01, of each axis during the superimposition control, depend on the MTB specifications (parameter "#2092 plG0tL", "#2093 plG0t1", "#2094 plG1tL", "#2095 plG1t1").
If the parameter setting value is "0" or outside the setting range, the MCP alarm (Y51 0022 or Y51 0023) will occur.

(9) An axis in a part system, which contains an axis being moved to the start or end position of superimposition, cannot be exchanged as an arbitrary axis. If arbitrary axis exchange is commanded, an operation error (M01 1101) will occur.

(10) Issue the arbitrary axis superimposition end command to the superimposed axis. The arbitrary axis superimposition end command issued to an axis other than the superimposed axis is ignored.

16.5 Control Axis Synchronization between Part Systems ; G125

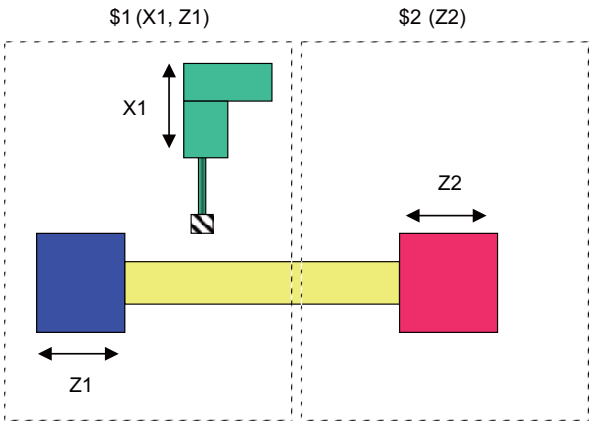
Function and purpose

Synchronization control enables an arbitrary control axis in the other part system to move in synchronization with the movement command assigned to an arbitrary control axis.
The direction in which the axis is to move synchronously can also be reversed using a parameter.

Reference axis	Axis to be commanded to move as the reference for synchronous movement.
Synchronization axis	Axis whose movement is synchronized with base axis.

There are two types of control axis synchronization between part systems depending on the command method: Control axis synchronization between part systems I (G command) and control axis synchronization between part systems II (PLC signal). The method is chosen depending on the MTB specifications (parameter "#1280 ext16/bit5").

The following is the example of machining while synchronizing Z1 axis with Z2 axis in 1st part system (X1, Z1) and 2nd part system (Z2).



While in synchronization, synchronized axis synchronizes and moves against the reference axis regardless of conducting automatic operation or manual operation.

Command format

Synchronization start

G125 Synchronized axis name = Reference axis name ;

Synchronized axis name	Synchronized axis name (Axis name set in the parameter "#1022 axname2")
Reference axis name	Reference axis name (Axis name set in the parameter "#1022 axname2)

Synchronization end

G125 Synchronized axis name ;

Synchronized axis name	Name of the synchronized axis to be ended (Axis name set in the parameter "#1022 axname2")

Detailed description

(1) When G125 is commanded, two part systems which include the reference axis and the synchronized axis wait for the completion of currently executing block. After confirming the smoothing for all axes reached zero for both part systems, start/end the synchronization.
In synchronous/superimposition control signal turns ON for the reference axis and the synchronized axis during synchronization.

(2) Set the synchronous direction with the parameter "#2087 syncnt/bit0". In synchronization start command, if a code "-" is added to the beginning of the reference axis name, the synchronous direction will be reversed.

(3) G125 can be commanded from any part system.

(4) By re-commanding the reference axis in synchronization as the reference axis, multiple synchronized axes are given to per reference axis.

Synchronization deviation detection

The synchronization deviation is detected during the control axis synchronization between part systems.

The deviation of the synchronized axis feedback value against the reference axis feedback value is calculated as the synchronization deviation amount.

The synchronization deviation amount is output to R register (R5076 to R5107) which correspond to each synchronized axis.

If the synchronization deviation amount exceeds the value of parameter "#2024 synerr", an operation error (M01 0051) will occur. Cancel the control axis synchronization between part systems of the error axis to clear the error. However, when the parameter "#2024 synerr" is set to "0", the range of the synchronization deviation amount will not be checked.

Program example

(1) When the number of part systems is two.

<Axis configuration>

1st part system: X1,Z1

2nd part system: X2,Z2

Below is the example of machining while executing the control axis synchronization between part systems.

Reference axis: Z1, synchronized axis: Z2

[1st part system]		**[2nd part system]**	
G28XZ;		G28XZ;	
G00 X100. Z100.;		G00 X60. Z50.;	
!L1;	(timing synchronization operation)	!L1;	(timing synchronization operation)
G125 Z2 = Z1;	(Start synchronizing Z2 to Z1)		
!L2;	(timing synchronization operation)	!L2;	(timing synchronization operation)
G01 X200. F500;		G01 X40. F300;	
G01 Z300.;		:	
G01 X100.;		:	
!L3;	(timing synchronization operation)	!L3;	(timing synchronization operation)
G125 Z2 ;	(End the synchronization between Z1 and Z2)		
!L4	(timing synchronization operation)	!L4	(timing synchronization operation)

(2) When number of part systems is three.

<Axis configuration> 1st part system: X1,Z1

2nd part system: X2,Z2

3rd part system: X3,Z3

Below is the example of machining while executing the control axis synchronization between part systems.

Reference axis: Z1, synchronized axis: Z2

[1st part system]		**[2nd part system]**		**[3rd part system]**	
:		**:**		**:**	
:		:		:	
!2!3 L1;	(timing synchronization operation)	!1!3 L1;	(timing synchronization operation)	!1!2 L1;	(timing synchronization operation)
				G125 Z2=Z1;	(Start synchronizing Z2 to Z1)
!2!3 L2;	(timing synchronization operation)	!1!3 L2;	(timing synchronization operation)	!1!2 L2;	(timing synchronization operation)
G01 Z50. F100;		G01 X-10. F100;		G01 Z50. F100;	
X4. Z4.;		:		G00 X10. Z20.;	
:		:		:	
:		:		:	
!2!3 L3;	(timing synchronization operation)	!1!3 L3;	(timing synchronization operation)	!1!2 L3;	(timing synchronization operation)
				G125 Z2 ;	(End the synchronization between Z1 and Z2)
!2!3 L4;	(timing synchronization operation)	!1!3 L4;	(timing synchronization operation)	!1!2 L4;	(timing synchronization operation)

Relationship with Other Functions

(1) Reference position return
When the reference position return is commanded to the reference axis, the synchronized axis only synchronizes with the reference axis movement and moves, and the reference position return is not carried out.

(2) Emergency stop/reset
When the emergency stop occurs during the control axis synchronization between part systems, the control axis synchronization between part systems will be canceled.
When being reset during the control axis synchronization between part systems, whether to cancel the control axis synchronization between part systems depends on the MTB specifications (parameter "#1280 ext16/bit3").
Input reset to all part systems which are related to the synchronization at a time.

(3) Stroke end, stored stroke limit, chuck barrier/tail stock barrier
When the reference axis enters stroke end (over travel), stored stroke limit, or chuck barrier/tail stock barrier, the synchronized axis synchronizes with the reference axis and stops. On this occasion, the movement command is enabled for other axes, including the synchronized axis of the part system.
When the synchronized axis enters stroke end (over travel), stored stroke limit, or chuck barrier/tail stock barrier, the reference axis will be stopped to keep the synchronization state. Furthermore, if the automatic operation is conducted, all axes, including the reference axis, will stop.

(4) Machine lock
When the reference axis is in automatic machine lock or manual machine lock, the synchronized axis is also in machine lock.

(5) Servo OFF
When the state of the reference axis is servo OFF, that of the synchronized axis is also servo OFF.
When the parameter "#1064 svof" is set to "1" (correct the error), do not make them servo OFF during synchronization. If they are servo OFF, the synchronous relationship between the reference axis and synchronized axis cannot be maintained by the tolerable correction operation.

(6) Inclined axis control
The control axis synchronization between part systems of the basic axis in the inclined axis control and the inclined axis are listed below.

The control axis synchronization between part systems for inclined axis ("#2071 s_axis=1")	The control axis synchronization between part systems for the reference axis ("#2071 s_axis=2") in the inclined axis control
× (*2)	○ (*3)

○: Available, ×: Not available

(*1) Above operations are determined only by the setting value of the parameter "#2071 s_axis" and the control axis synchronization between part systems command regardless of "inclined axis control valid" signal (YC35). This depends on the MTB specifications.

(*2) When the synchronization start command is given while the control axis synchronization between part systems is disabled, the operation differs depending on the control axis synchronization between part systems I or the control axis synchronization between part systems II. Refer to "Precautions" for details.

(*3) While the basic axis of the inclined axis control is in the control axis synchronization between part systems, the manual movement command and the automatic movement command cannot be given to the corresponding inclined axis. If the manual movement command or the automatic movement command is given to the inclined axis, the error "M01 operation error 0005" will occur.

Precautions

Precautions for starting/ending the synchronization

(1) Command G125 in an independent block. A program error (P33) will occur if not commanded in independent blocks.

(2) The control axis synchronization between part systems by G125 command is valid only when the parameter "#1280 ext16/bit5" is set to "1". If G125 is commanded when "ext16/bit5" is set to "0", a program error (P610) will occur.

(3) Specify the axis name set with the parameter "#1022 axname2". If the axis name which is not set with "axname2" is specified, a program error (P521) will occur.

(4) The axis name specified in G125 needs to be 2 digits. Set the parameter "#1022 axname2" with two digits.

(5) If an axis other than synchronized axis is specified at the synchronization end command, that command will be ignored.

(6) If the same axis name is given to the reference axis and the synchronized axis, a program error (P521) will occur.

(7) Two axes in the same part system cannot be commanded as a reference axis and synchronized axis. If this type of command is given, a program error (P521) will occur.

(8) Synchronization start/cancel by G125 command is invalid for graphic check and program restart.

(9) When an axis is set as "inclined axis (setting value: 1)" by the parameter "#2071 s_axis", it cannot be specified as a reference axis or synchronized axis. If the synchronization start command (G125 command) is given like this, a program error (P521) will occur.

(10) If G125 is commanded where there is no specifications for control axis synchronization between part systems, a program error (P39) will occur.

Other precautions

(1) When the control axis synchronization between part systems are started/canceled, the related two part systems wait for the completion of the currently executing block, and the synchronization starts/cancels after the smoothing for all axes reached zero for both part systems.

(2) As the state of two target part systems affects the control axis synchronization between part systems, be aware of the timing between the part systems. Therefore, command the timing synchronization operation before and after commanding the synchronization start/cancel.

(3) Do not change the related parameters during the control axis synchronization between part systems.

(4) The following parameters of two target axes for the control axis synchronization between part systems or two part systems which includes target axes need to match.

- Control unit (#1004 ctrl_unit)
- Input setting unit (#1003 iunit)
- Diameter specification axis (#1019 dia)
- Designation of the linear axis/linear type rotary axis/rotating type rotary axis (#1017 rot, #8213 Rotation axis type)
- Rapid traverse rate (#2001 rapid)
- Cutting clamp speed ("#2002 clamp)
- Acceleration/deceleration time constant (#2004 G0tL, #2005 G0t1, #2007 G1tL, #2008 G1t1)
- Acceleration and deceleration types (#2003 smgst)

As the acceleration/deceleration time constants between two part systems related to the synchronization need to match, the acceleration/deceleration time constant of the constant inclination acceleration/deceleration or the control axis superimposition cannot be used together with the function which calculates or switches automatically and this function.

(5) For the part systems which relate to the control axis synchronization between part systems, set "#1200 G0_acc", "#1201 G1_acc" and "#1205 G0bdcc" to "0".

(6) Regardless of conducting the automatic operation or the manual operation, the movement command cannot be issued for the synchronized axis during the control axis synchronization between part systems. If this type of command is given, the error "M01 operation error 1038" will occur.

(7) The slave axis during the synchronization control cannot be commanded as a reference or synchronized axis for the control axis synchronization between part systems. If this type of command is given, the error "M01 operation error 1037" will occur.

(8) The synchronized axis during the control axis synchronization between part systems cannot be commanded as a reference axis or synchronized axis. If this type of command is given, the error "M01 operation error 1037" will occur.

(9) The reference axis during the control axis synchronization between part systems cannot be commanded as a synchronized axis. If this type of command is given, the error "M01 operation error 1037" will occur. However, a command which has multiple synchronized axes to one reference axis is possible.

(10) The axis of the part system which includes the reference axis of the control axis superimposition or superimposition axis cannot be commanded as a reference axis or synchronized axis of the control axis synchronization between part systems. If this type of command is given, the error "M01 operation error 1037" will occur.

(11) The target control axis at the start of the control axis synchronization between part systems should be in a state where the reference position return after tuning the power ON is complete or the absolute position is established. If the axis whose zero point is not established is commanded, the error "M01 operation error 1037" will occur.

(12) The spindle/C axis whose servo is OFF cannot be commanded as a reference axis or synchronized axis. If this type of command is given, the error "M01 operation error 1037" will occur.

16.6 Multi-part System Simultaneous Thread Cutting Cycle

Function and purpose

Multi-part system simultaneous thread cutting allows multiple part systems to perform thread cutting simultaneously on one spindle.
Multi-part system simultaneous thread cutting has two commands; the command (G76.1) for simultaneously cutting threads in multiple places, which is known as "multi-part system simultaneous thread cutting cycle I", and the command (G76.2) for simultaneously cutting a thread by two part systems, which is known as "two-part system simultaneous thread cutting cycle II".
Chamfering is available for multi-part system simultaneous thread cutting cycle I and two-part system simultaneous thread cutting cycle II. Refer to "13.1.2 Thread Cutting Cycle ; G78" for details on chamfering.

16.6.1 Multi-part System Simultaneous Thread Cutting Parameter Setting Command ; G76

Command format

```
G76 Pmra QΔdmin Rd ;
```

Address		Meaning
P	m	Number of cutting passes for finishing Reversible parameter "#8058 G76 TIMES" is also available for setting.
	r	Chamfering amount Reversible parameter "#8014 CDZ-VALE (L system only)" is also available for setting. This sets the chamfering width based on thread lead l across a range from 0.0 to 9.9 with a 2-digit integer with the decimal point omitted. (00 to 99)
	a	Tool nose angle (thread angle) Reversible parameter "#8059 G76 ANGLE" is also available for setting. This selects the angle from 0° to 99° and commands the value with two digits.
Q	Δdmin	Minimum cut amount If the calculated cut amount is smaller than Δdmin, it is clamped by Δdmin. (When omitted, operations differ depending on #1222/bit4 setting.)
R	d	Finishing allowance Reversible parameter "#8057 G76 LAST-D" is also available for setting.

Parameters for thread cutting are set with this command.
Command in the block immediately before G76.1/G76.2 is commanded.

Note

(1) A reversible parameter enables to use parameter setting value without issuing a program command and also, the value can be changed by the program command.

Detailed description

(1) The data is set in machining parameters m: #8058, r: #8014, a: #8059 and d: #8057 for each part system.
(2) Issue the command for each part system.
(3) The parameter setting command can be omitted depending on the MTB specifications (parameter "#1222 aux06/bit5").
(4) If the parameter setting command is omitted, the parameter setting values are used from #8014, #8057, #8058, #8059 settings. The minimum cut-in amount (Δdmin) follows the #1222/bit4 setting.
(5) When "#1265 ext01/bit0" is "1", MITSUBISHI CNC special format will be applied. Therefore, program error (P33) will occur when the parameter setting commands. (The parameter setting depends on the MTB specifications.)

16.6.2 Multi-part System Simultaneous Thread Cutting Cycle I ; G76.1

Command format

```
G76.1  X/U__  Z/W__  R__  P__  Q__  J__ F__ ;
```

X/U	X-axis end point coordinates of thread section (absolute or incremental value)
Z/W	Z-axis end point coordinates of thread section (absolute or incremental value)
R	Taper height component (radius value) for thread A straight thread is created when "0" is set.
P	Thread height (positive radius value)
Q	1st cut amount (positive radius value)
J	Part system for simultaneous thread cutting (an integer that has 8 digits or less) (*1) (*2) (*3) (*4) (*5)
F	Thread lead

(*1) Using an integer that has 8 digits or less, command the numbers of the part systems on which simultaneous thread cutting is executed. (The part system where the command was issued can be omitted.)

Ones digit	Part system number for the 1st set
Tens digit	Part system number for the 2nd set
: Ten-millions digit:	: Part system number for the 8th set

(Example 1) To execute a thread cutting cycle on the 1st and 3rd part systems, command "J13" or "J31".

(Example 2) To execute a thread cutting cycle on the 1st, 3rd, and 4th part systems, command "J134", "J413", or "J341".

(*2) If the J address is omitted, the part system where G76.1 has been commanded or the part system that has been set in the parameter "#19419 Timing sync system" is determined as the part system on which the thread cutting cycle will be simultaneously executed.

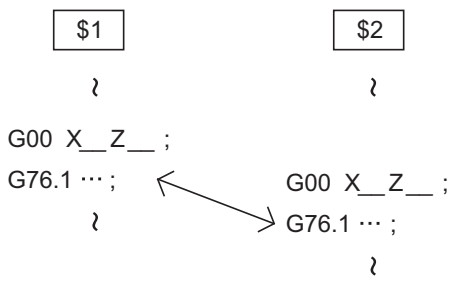
(*3) If the number of the part system where G76.1 has not been commanded is commanded, a timing synchronization status is established at the beginning of G76.1, and the thread cutting cycle will not be started.

(*4) If a part system that does not exist is commanded, a part system to be used in sub part system II is commanded, or the value set as a part system number is "0" or less, a program error (P35) will occur.

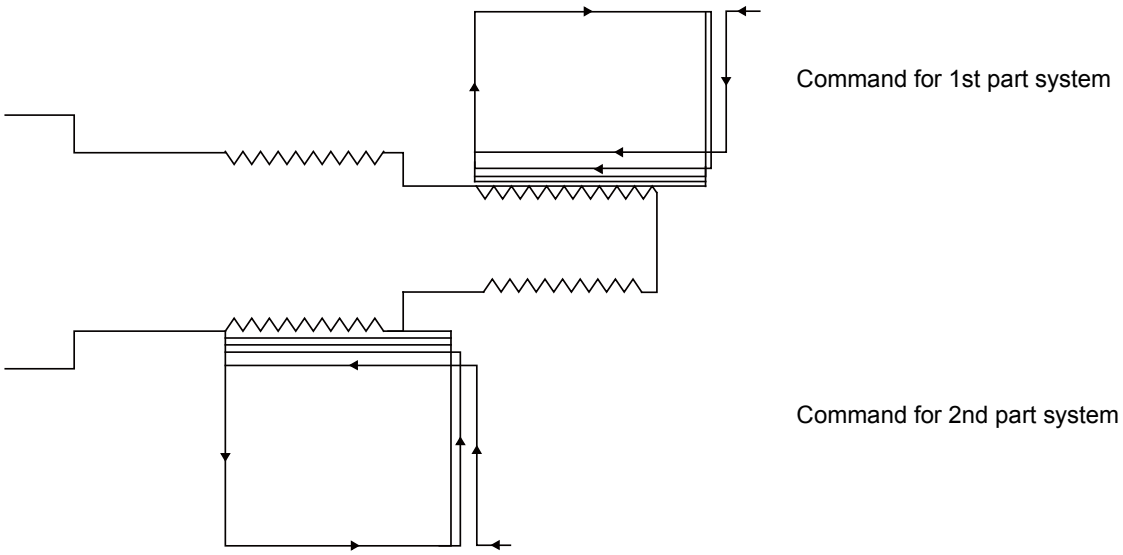
(*5) If the same part system is commanded more than once, a program error (P33) will occur.

Detailed description

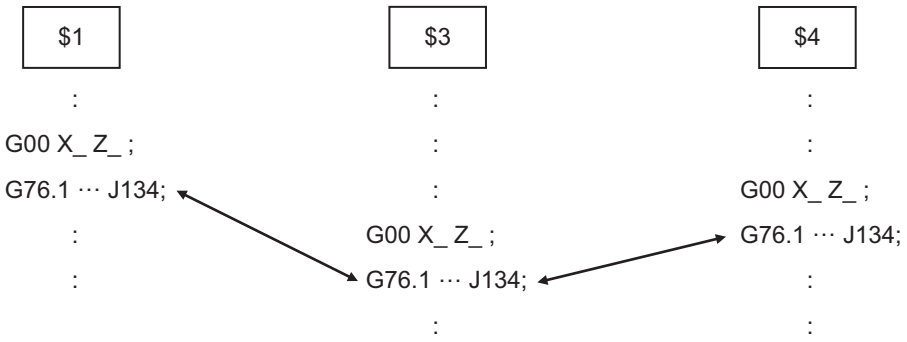
(1) When G76.1 is issued by multiple part systems, waiting is done until the G76.1 command is issued to another part system. The thread cutting cycle starts when the commands are aligned properly.
(Example 1) When the J address is omitted (parameter "#19419 Timing sync system" = 0)



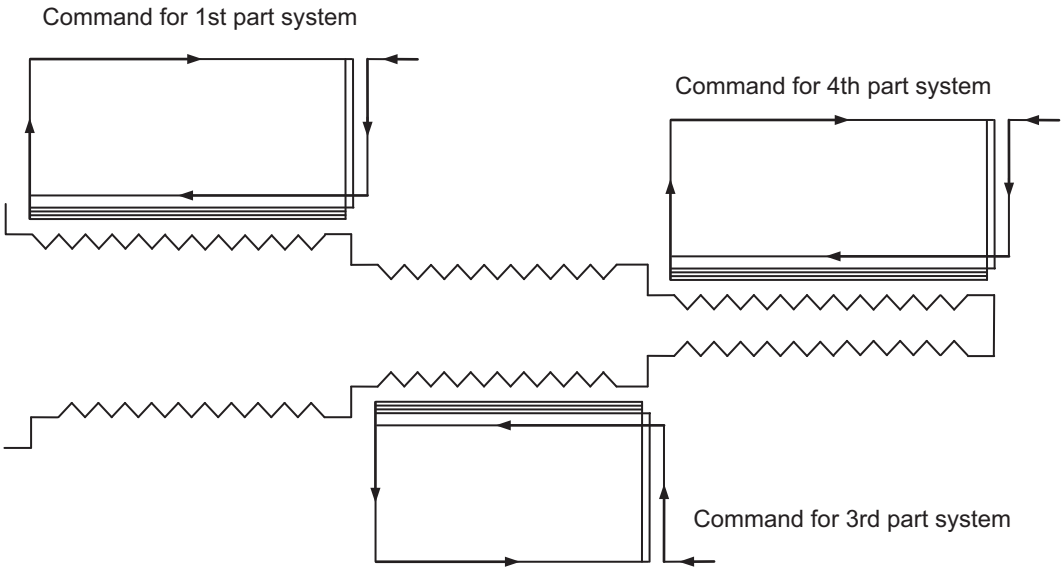
Thread cutting will start simultaneously after waiting for the 1st and 2nd part systems.



(Example 2) When "J134" is commanded for the 1st part system ($1), the 3rd part system ($3), and the 4th part system ($4)



Thread cutting will start simultaneously after waiting for the 1st, 3rd, and 4th part systems.



(2) In a multi-part system simultaneous thread cutting cycle, waiting is done at the start and end of the thread cutting process.
However, in multi-part system simultaneous thread cutting cycle I (G76.1), waiting in one cycle can be disabled depending on the MTB specifications (parameter "#1242 set14/bit0").

(3) The same precautions for thread cutting command (G33), thread cutting cycle (G78) and compound thread cutting cycle (G76) apply to this cycle.

(4) As the threads are cut in multiple places by the G76.1 command, the various commands do not need to be the same. Each of them can be issued independently.

(5) When G76.2 and G76.1 are commanded
The part systems, in which each are commanded, will carry out the G76.1 and G76.2 movements. However, the part system in which G76.2 is commanded will assume that the other part system is using G76.2 when cutting the threads, so the thread grooves will not be guaranteed.

(6) This function is unmodal. So it needs to be commanded each time.

(7) G code group 1 modal (G00, G01, G02, G03, G02.3, G03.3 etc.) is canceled temporarily during multi-part system simultaneous thread cutting.

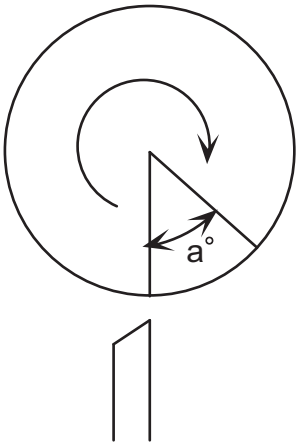
(8) Thread cutting start shift angle is not available in G76.1. A program error (P32) will occur if commanded.

16.6.3 Two-part System Simultaneous Thread Cutting Cycle II ; G76.2

Command format

```
G76.2  X/U__  Z/W__  R__  P__  Q__  Aa  F__ ;
```

(1) Thread cutting start shift angle



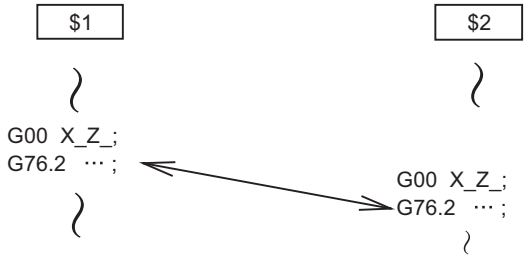
The thread cutting command starts movement after waiting for the spindle encoder's one rotation synchronization signal. However, the start point can be delayed by a degree amount.

a : Thread cutting start angle

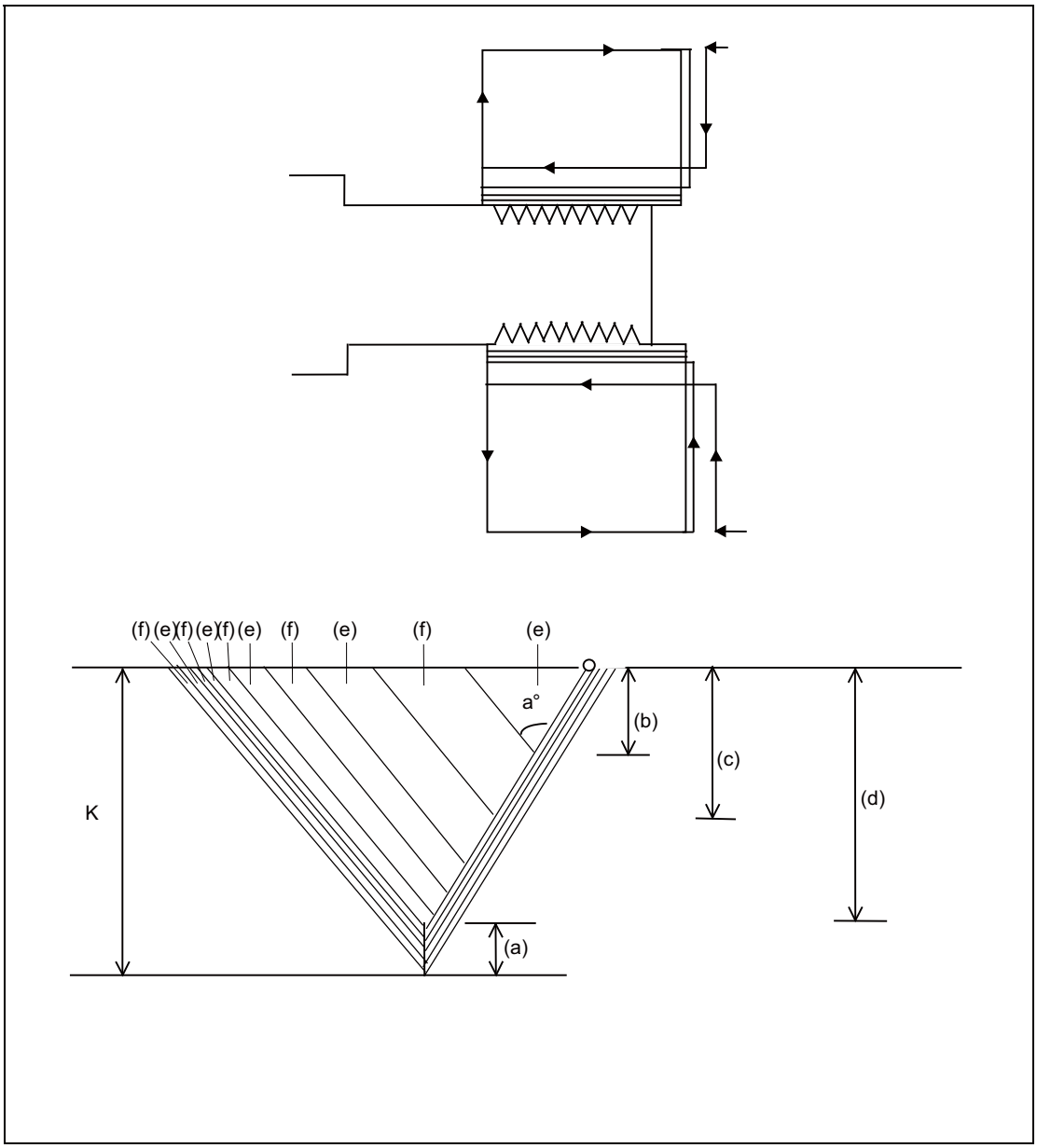
The meanings of the addresses other than A and J are the same as the multi-part system simultaneous thread cutting cycle I (G76.1).

Detailed description

(1) When G76.2 is issued by 1st part system and 2nd part system, waiting is done until the command is issued to another part system. The thread cutting cycle starts when the commands are aligned properly.

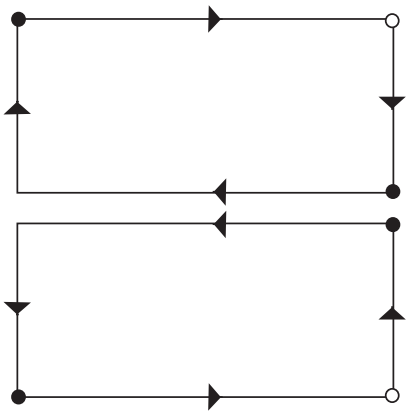


(2) G76.2 assumes the same thread cutting, and deeply cuts in with the cutting amount using 1st part system and 2nd part system alternately.



(a) Finishing allowance d
(b) Δd
(c) Δd ×√ 2
(d) Δd ×√ n
(e) Cutting with 1st part system
(f) Cutting with 2nd part system

(3) In one cycle, waiting is done at the start and end of the thread cutting.



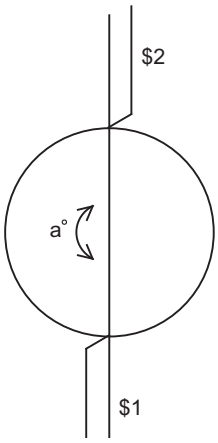
● Timing synchronization operation

(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 cuts the same thread, so the various parameters, thread section, taper height, screw thread height, cutting amount and thread lead must be commanded to the same values for the two part systems.
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 the 1st and 2nd part systems.

(7) Thread cutting start shift angle command



As shown on the left, if the 1st and 2nd part system blades oppose at 180 degrees, the 1st and 2nd part systems thread cutting start shift angle difference is set to 180 degrees.

(Example)

1st part system ($1)	2nd part system ($2)
:	:
G76.2 X_ Z_ A0.;	G76.2 X_ Z_ A180.;
:	:

(8) When G76.2 and G76.1 are commanded
The part systems, in which each are commanded, will carry out the G76.1 and G76.2 movements. However, the part system in which G76.2 is commanded will assume that the other part system is using G76.2 when cutting the threads, so the thread grooves will not be guaranteed.

(9) This function is unmodal. So it needs to be commanded each time.

(10) G group 1 modal (G00, G01, G02, G03, G02.3, G03.3 etc.) is canceled temporarily during two-part system simultaneous thread cutting.

(11) Reset simultaneously for 1st part system and 2nd part system during two-part system simultaneous thread cutting cycle II (G76.2).
If reset is individually executed for each part system, the part system is not in automatic operation. Thus, the other part system keeps moving according to the setting of "#1279 ext15/bit0".

16.7 Multi-part System Simultaneous Thread Cutting Cycle (MITSUBISHI CNC special format) ; G76.1,G76.2

Function and purpose

Multi-part system simultaneous thread cutting allows different part systems to perform thread cutting simultaneously on one spindle.
Multi-part system simultaneous thread cutting has two commands; the command (G76.1) for simultaneously cutting threads in multiple places, which is known as "multi-part system simultaneous thread cutting cycle I", and the command (G76.2) for simultaneously cutting a thread by two part systems, which is known as "two-part system simultaneous thread cutting cycle II".
When using the MITSUBISHI CNC special format (#1265 ext01/bit0 ON), some addresses differ from the normal format. This section gives details on the formats different from the normal ones.
Refer to section "16.6 Multi-part System Simultaneous Thread Cutting Cycle" for details on the multi-part system simultaneous thread cutting cycle.

Command format

```
G76.1 X/U__ Z/W__ I__ K__ D__ J__ F__ A__ ;
```

```
G76.2 X/U__ Z/W__ I__ K__ D__ Q__ F__ A__ ;
```

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 (radius value) at thread section A straight thread is created when I0 = 0.
K	Thread height This thread height is commanded with a positive radius value.
D	Cut amount The cut amount for the first cutting pass is commanded with a positive radius value.
J	Part system for simultaneous thread cutting (an integer that has 8 digits or less) (*1) (*2) (*3) (*4) (*5)
Q	Thread cutting start shift angle
F	Thread lead
A	Tool nose angle (thread angle)

(*1) Using an integer that has 8 digits or less, command the numbers of the part systems on which simultaneous thread cutting is executed. (The part system where the command was issued can be omitted.)

Ones digit	Part system number for the 1st set
Tens digit	Part system number for the 2nd set
: Ten-millions digit:	: Part system number for the 8th set

(Example 1) To execute a thread cutting cycle on the 1st and 3rd part systems, command "J13" or "J31".

(Example 2) To execute a thread cutting cycle on the 1st, 3rd, and 4th part systems, command "J134", "J413", or "J341".

(*2) If the J address is omitted, the part system where G76.1 has been commanded or the part system that has been set in the parameter "#19419 Timing sync system" is determined as the part system on which the thread cutting cycle will be simultaneously executed.

(*3) If the number of the part system where G76.1 has not been commanded is commanded, a timing synchronization status is established at the beginning of G76.1, and the thread cutting cycle will not be started.

(*4) If a part system that does not exist is commanded, a part system to be used in sub part system II is commanded, or the value set as a part system number is "0" or less, a program error (P35) will occur.

(*5) If the same part system is commanded more than once, a program error (P33) will occur.

Note

(1) This is 1 block command. It is not necessary to command "G76P__Q__R__" immediately before.

Detailed description

Comparison of normal format and MITSUBISHI CNC special format

Normal format	MITSUBISHI CNC Special format	Remarks
(1) G76 Pmra Q_ R_; (2) G76.1 X_ Z_ R_ P_ Q_ J_ F_; (2) G76.2 X_ Z_ R_ P_ Q_ A_ F_;	G76.1 X_ Z_ I_ K_ D_ J_ F_ A_; G76.2 X_ Z_ I_ K_ D_ F_ A_ Q_;	Command format
(2) X/U	X/U	X-axis end point coordinates of thread section
(2) Z/W	Z/W	Z-axis end point coordinates of thread section
(2) R	I	Taper height component at thread section
(2) P	K	Thread height
(2) Q	D	Cut amount
(2) J	J	Part system for simultaneous thread cutting
(2) A	Q	Thread cutting start shift angle
(2) F	F	Thread lead
(1) Q [When omitted: #1222/bit4]	None ["0" fixed]	Minimum cut amount
(1) Pa [Reversible parameter #8059]	A [Reversible parameter #8059]	Tool nose angle (thread angle)
(1) Pm [Reversible parameter #8058]	Parameter #8058	Number of cutting passes for finishing
(1) Pr [Reversible parameter #8014]	Parameter #8014	Chamfering amount
(1) R [Reversible parameter #8057]	Parameter #8057	Finishing allowance

Note

(1) A reversible parameter enables to use parameter setting value without issuing a program command and also, the value can be changed by the program command.

Relationship with Other Functions

The modal must be set as shown below when commanding G76.1/G76.2.

Function	G code
Cylindrical interpolation cancel	G07.1
Polar coordinate interpolation cancel	G13.1
Balance cut OFF	G14
Tool nose radius compensation cancel	G40
Mirror image by parameter setting	Cancel
Mirror image by external input	Cancel
Polygon machining mode cancel	G50.2
Constant surface speed control mode cancel	G97

Precautions

(1) The switch of MITSUBISHI CNC special format and normal format depends on the MTB specifications (parameter "#1265/bit0").

(2) When the MITSUBISHI CNC special format is selected, it is not necessary to command G76P__R__immediately before commanding G76.1/G76.2. A program error (P33) will occur if G76 P__ R__ or a normal format is commanded.

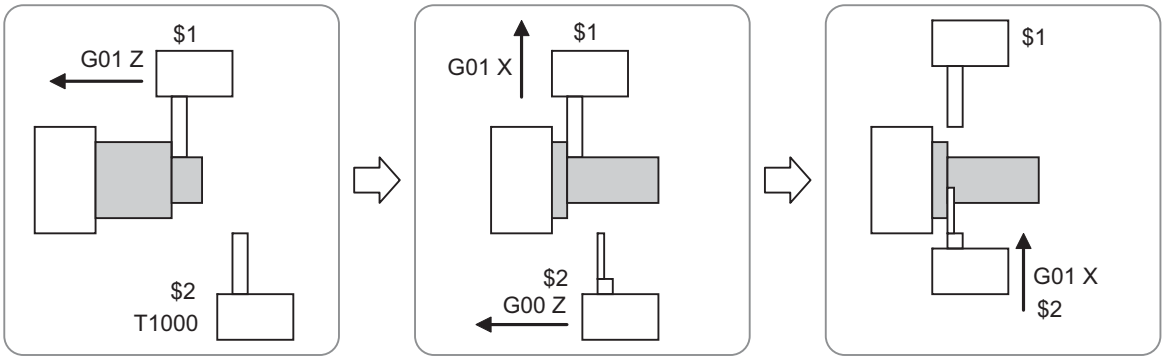
16.8 Synchronization between Part Systems

16.8.1 Dwell/Miscellaneous Function Time Override

Function and purpose

Override can be applied to dwell time and miscellaneous function finish wait time of all part systems. The synchronization between part systems can be maintained when the multiple machining programs are operated with override in the multiaxis and multi-part system mixed control CNC.
If a machining program that performs cutting-off machining in the 2nd part system ($2) after the completion of turning machining in the 1st part system ($1) is executed with override, the 1st and 2nd part systems will be out of synchronization and the machining may not be operated properly.
The validity of this function depends on the MTB specifications. Refer to your CNC specifications for details (parameter "#1436 mstsyn").

The differences on the start timing for each command are shown in (1) to (3) when executing the machining with the 1st part system ($1) and 2nd part system ($2) as shown in the figure below

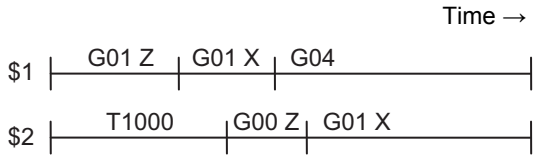


$1: Turning machining
$2: Tool exchange
(T1000)

$1: Turning machining
$2: Tool positioning

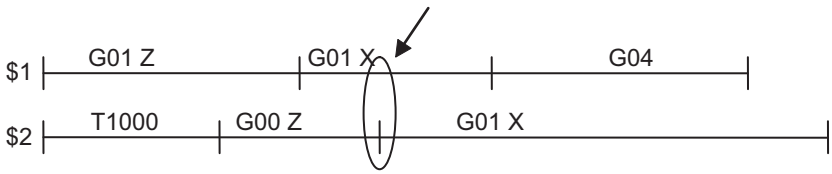
$1: Standby
$2: Cutting-off machining

(1) Override 100%



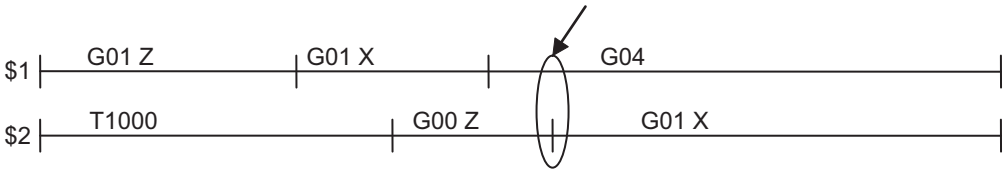
(2) Override 50% when this function is invalid

Only feed time doubled in the operation with override 50%.
Part systems are out of synchronization and the cutting-off machining starts during turning machining.



(3) Override 50% when this function is valid

Feed time, dwell time and miscellaneous function finish wait time double in the operation with override 50%.
The synchronization between part systems are maintained and the cutting-off machining stars after the completion of the turning machining.



Detailed description

Dwell override

Override can be applied to dwell time.

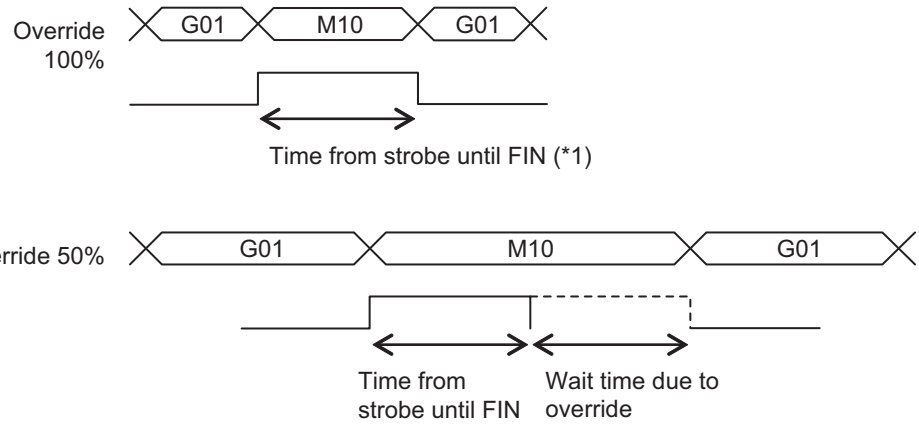
Dwell execution time = Dwell command time x (100/override value%)

(Example) Dwell time when override is applied to G04 X10.;

Override	Dwell time
100%	10.0 s
75%	13.3 s
50%	20.0 s
25%	40.0 s

Miscellaneous function time override

Apply override to miscellaneous function finish wait time to delay the start of the next block.



$$\text{Wait time due to override} = \text{Time from strobe until FIN} \times \frac{100}{\text{Override value}} - \text{Time from strobe until FIN}$$

(*1) Time from turning on of any of MF1 to 4, SF1 to 8, TF1 to 4 and BF1 to 4 signals until rising edge of FIN1 or falling edge of FIN2

Relationship with Other Functions

Dwell per revolution command

This function is invalid for the dwell per revolution command.

M code output during axis traveling

This function is also valid for the miscellaneous functions output by the M code output while axis is moving.

Miscellaneous functions multiple codes in 1 block

This function is valid if multiple miscellaneous functions are issued in one block. Override is applied to the time from when any strobe signal is turned on until the FIN signal.

Miscellaneous function lock

Miscellaneous functions are not output in the miscellaneous function lock state. This function is invalid.

Manual numerical value command

Override is not applied to the miscellaneous function time with the manual numerical value command. This function is invalid.

Synchronization between part systems OFF

This function can be disabled by the synchronization between part systems OFF function.
This function depends on the MTB specifications.

Program check operation

Override is set to 100% when the actual cutting mode is selected during the program check operation. This function is invalid.

Tapping cycle command

Override is cancelled during the tapping cycle. Therefore, override will not be applied to the hole bottom dwell function or miscellaneous function output (forward and reverse tool rotation). This function is invalid.

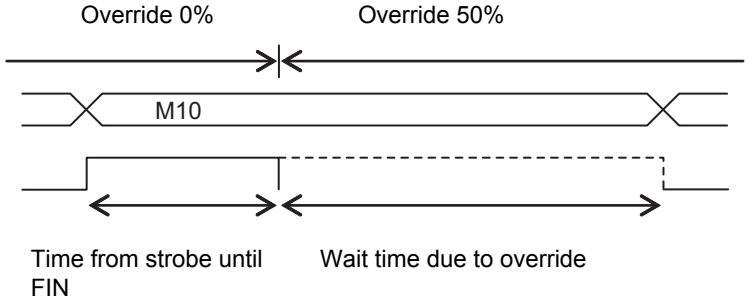
Precautions

(1) When operating the machine by applying override, set the cutting feed override and the rapid traverse override to the same rate on all part systems. Otherwise, part systems will be out of synchronization with one another.

(2) Setting the cutting feed override exceeding 100% will not shorten the miscellaneous function time. Moreover, part systems will be out of synchronization with one another. To maintain synchronization between part systems, the cutting feed override must be set to 100% or less.

(3) If the machine is operated by applying override, while a command to cancel override, such as tapping cycle and thread cutting, is issued, part systems will be out of synchronization with one another. To maintain synchronization between part systems, command the timing synchronization operation between part systems after a command to cancel override is issued.

(4) If cutting feed override is set to 0%, the dwell time and the miscellaneous function command will not be completed. If cutting feed override is set to 0% or more, the machine will wait for the wait time that is determined according to the cutting feed override value before starting the next block.



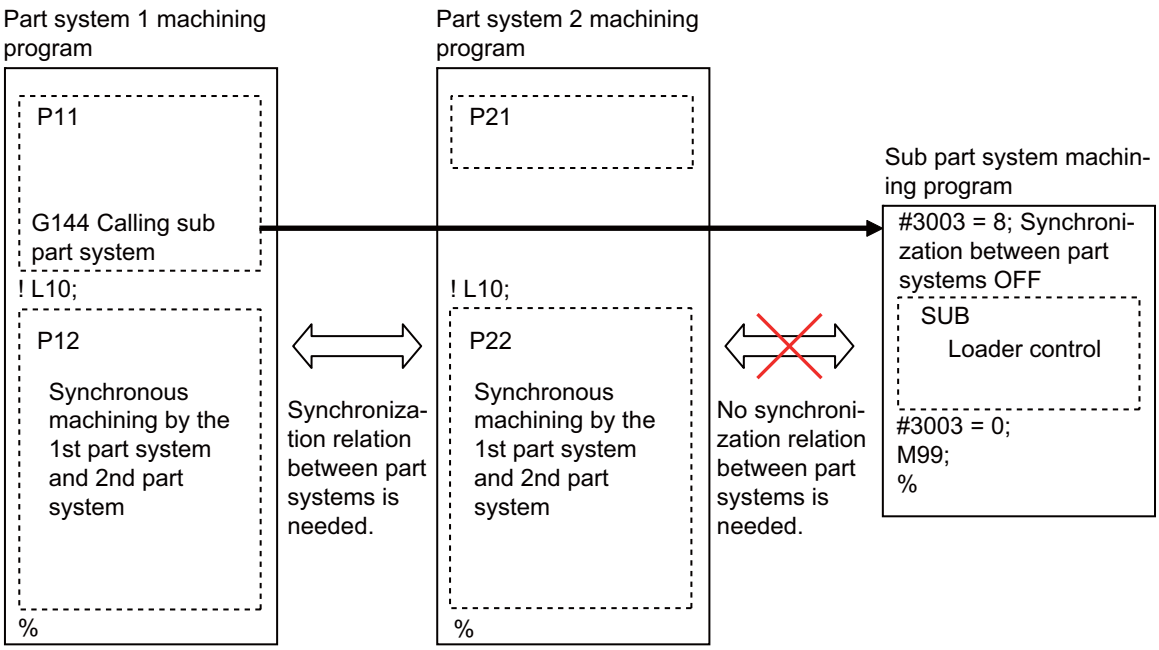
(5) If the automatic operation stops during the wait time due to override, the time period of automatic operation pause will not be included in the wait time. After the machine has been restarted, the machine will wait for the remaining wait time due to override, before starting the next block.

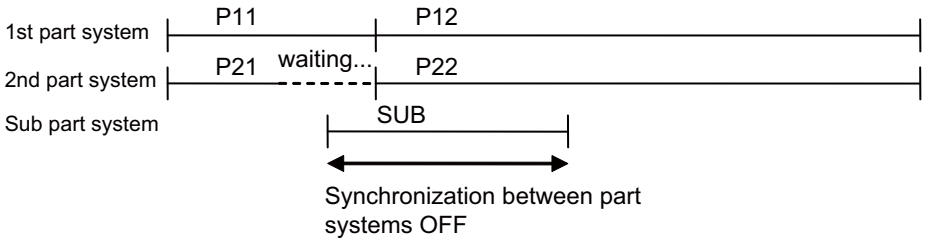
(Example) If the wait time due to override is set to 10.0 s, and the automatic operation stops after a lapse of 6.0 s, after the machine has been restarted, the machine will wait for 4.0 s before starting the next block.

16.8.2 Synchronization between Part Systems OFF

Function and purpose

To cancel synchronization with other part systems by single block operation with part systems synchronized, this function disables synchronization between part systems in a part of the machining program.
This function is mainly effective at blocking the automatic operation pause in only some part systems when the sub part system control II function is being used. This function is useful when loader control and ATC control, which are operated independently of machining, are performed with a sub program or sub part system control. (This function enables an operation with synchronization between part systems in machining part systems and an independent operation in sub part systems.)
Valid/invalid of this function can be switched with the system variable (#3003) or the PLC signal (based on the MTB specifications.)
The following explains the method to switch ON/OFF with the system variable.





The behaviors with synchronization between part systems OFF is as follows

Single block function with part systems synchronized:	- The pause is not caused by other part systems. - The other part systems do not pause with block stop.
Dwell/miscellaneous function time override:	- Override is invalid.

Detailed description

By substituting a value in the system variable #3003, the validity of each function can be selected.
Refer to the system variable list for details of each system variable.
Select Synchronization between part systems OFF in the system variable #3003/bit3.

Note

(1) Variable No. #3003 is set to zero by reset.

#3003/bit3:Synchronization between part systems OFF

The part system with "#3003/bit3" set to ON is not affected by the synchronization between part systems. At this time, the part systems, excluding that with "#3003/bit3" set to ON, are synchronized.

Synchronization between part systems	Part system with "#3003/bit3" set to ON	Part system with "#3003/bit3" set to OFF
Single block with part systems synchronized	Single block with part systems synchronized OFF When a part system has been stopped by single block stop, automatic operation will not stop in other part systems. When another part system has been stopped by single block stop, automatic operation will not stop in this part system.	Single block with part systems synchronized ON (*1) The Single block with part systems synchronized is valid between the part systems excluding that with "#3003/bit3" set to ON.
Dwell/Miscellaneous Function Time Override	Dwell/miscellaneous function time override OFF Override is not applied to the dwell time or miscellaneous function time.	Dwell/miscellaneous function time override ON (*2) Override is applied to the dwell time and miscellaneous function time.

(*1) The single block function with part systems synchronized must be enabled with the PLC signal.

(*2) The dwell/miscellaneous function time override function must be enabled with the parameter.

Relationship with Other Functions

Sub part system control II

When calling a sub part system, the sub part system will use the signals for cutting feed override, rapid traverse override, etc. of the main part system. However, the sub part system will use the signal for synchronization between part systems OFF of its own.
When the sub part system is complete, #3003 is cleared to 0.

Precautions

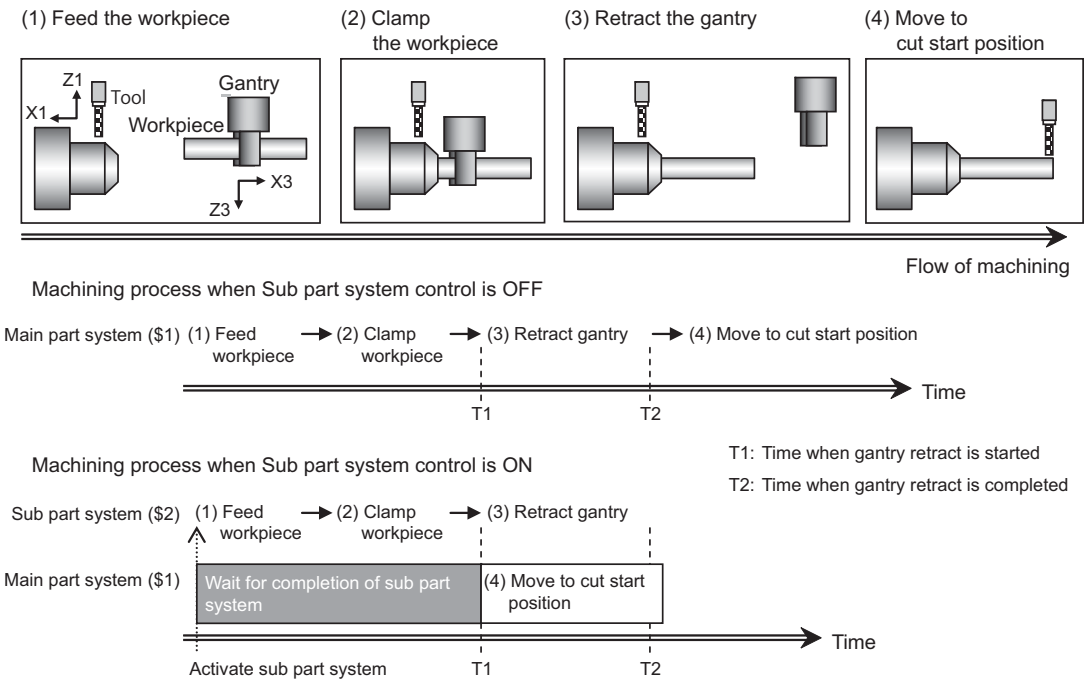
(1) If either of the system variable or PLC signal is turned ON, synchronization between part systems is invalid.

16.9 Sub Part System Control

16.9.1 Sub Part System Control I ; G122

Function and purpose

This function activates and operates any non-operating part system (sub part system) in the multi-part system. Sub part system control I can be used in the same manner as calling subprogram in a non-operating part system. An auxiliary axis machining program can be controlled in the sub part system by commanding Sub part system control I (G122) from the main part system.
In the usage example below, the tool positioning starts to the machining start point at the same time (time T1) as the start of gantry retract by using Sub part system control I (G145) in the flow from feeding the workpiece to moving to cut start position in order to reduce the cycle time.
Select whether main part system or sub part system for each part system in Sub part system control I. When using a part system as a sub part system, by setting the operation mode to "Sub part system I operation mode" with the PLC signal and commanding sub part system control I (G122) from an operating part system, it is possible to activate the part system in the sub part system I operation mode as a sub part system.



The following describes the meanings of the terms used in this chapter.

Term	Meaning
Main part system	Indicates a part system located on the uppermost stream side of a sub part system call flow.
Sub part system	Indicates a part system activated by the sub part system activation command.
Calling part system	Indicates a part system that issued the sub part system activation command.

The sub part system control I differs from the sub part system control II as follows.

Sub part system control I:	Main part system and sub part system depend on the MTB specifications, respectively.
Sub part system control II:	A part system dedicated for parallel processing is activated as a sub part system. In the program operation of a sub part system, the parameters per part system and the PLC signals follow the values of the main part system. (For some of the PLC signals, the device of the sub part system will be used. When an axis is needed to be controlled in sub part system, command the arbitrary axis exchange to give sub part system authorization to control the axis. Refer to the instruction manual issued by the MTB for details.) The availability of the arbitrary axis exchange function depends on the MTB specifications. Refer to your machine's specifications.

Enabling conditions

(1) This function can be used in multi-part systems of two or more part systems.

(2) In order to activate a sub part system using the sub part system control I command, the following conditions must be satisfied. There are conditions to enable functions that are only applicable to the M80 series.

[Condition 1]
This condition must only be satisfied for the M80 series.
The number of sub part systems has been set in the base common parameter "#1483 SBS1_sys num" (the number of part systems in sub part system I).

(a) Part systems as many as the number specified in #1483, counted from the end of the valid part system (the part system for which "#1001 SYS_ON" is set to "1"), will be reserved as sub part systems.

(b) If the number of sub part systems or main part systems exceeds the maximum number defined in the system specifications, an MCP alarm (Y51 1483) will occur.

(c) If the values set for "#1483 SBS1_sys num" and "#1474 SBS2_sys num" are both "1" or more, an MCP alarm (Y51 1483) will occur.

[Condition 2]
The identification No. (B command value) used to activate a sub part system has been set in the base common parameter "#12049 SBS_no" (sub part system I identification No.) for sub part systems.

(a) If an identification No. that is not set in the parameter "#12049 SBS_no" is specified when the sub part system control I command is issued, a program error (P650) (sub part system identification No. illegal) will occur.

[Condition 3]
The PLC signal SBSM (Sub part system I operation mode) of the sub part system is set to "1".

(a) In a part system operating the sub part system I operation mode, the operation mode appears as "SUB" in the part system display of the operation screen.

(b) If the sub part system control I command is issued to a part system that is not operating the sub part system I operation mode, an operation error (M01 1111) will occur. However, while the operation error (M01 1111) is occurring, the operation can be started by setting SBSM to "1".

Command format

Call sub part system

```
G122 A__P__Q__K__D__B__H__ (argument);
```

```
G122 <file name> P__Q__K__D__B__H__ (argument);
```

A	Program No. (1 to 99999999 or 100010000 to 199999998)
<File name>	File name of the program (up to 32 characters)
P	Start sequence number (Head of the program if omitted.)
Q	End sequence number (To end (M99) of the program if omitted.)
K	Number of repetitions (1 to 9999)
D	Synchronization control (0/1)
B	Sub part system identification No. (1 to 7)
H	Sub part system reset type (0/1)
Argument	Argument of a sub part system local variable (Setting rage of local variable (decimal point command is valid))

Complete sub part system

```
M99;  (command of a sub part system side)
```

Cancel the standby status for completion of sub part system

```
G145;  (command of a sub part system side that is issued when the D0 command is issued)
```

Note

(1) G145 is ignored in a sub part system activated in the parallel control method (D1 command).

Detailed description

This function can be used in multi-part systems of two or more part systems.
Main part system and sub part system are switched according to the MTB specifications.

Description of each address

Address	Meaning	Command range (unit)	Remarks
A	Program No.	1 to 99999999 or 100010000 to 199999998 (*1)	Program No. or file name of the machining program operating in the sub part system. •Programs in an external device cannot be designated. •If address A and <file name> are designated at the same time, precedence is given to address A. •If designation of the program is omitted, the machining program defined by the MTB will be used (parameter "#12050 SBS_pro").
<File name>	File name of the program	Up to 32 characters.	
P	Start sequence No.	1 to 99999999	Sequence No. to start the machining program operating in the sub part system. •If there is no command, the operation will start from the head of the machining program.
Q	End sequence No.	1 to 99999999	Sequence No. to end the machining program operating in the sub part system. •If there is no command, the program will run up to M99.
K	Number of repetitions	1 to 9999	The number of times to repeat the machining program for continuous operation in the sub part system. •If there is no command, the program will only run once. (No repetition)
D	Synchronization control	0 / 1	Validity of synchronous control 0: The next block is processed after the sub part system operation completes. 1: The next block is processed at the same time as the start of a sub part system operation. If there is no command, it is handled in the same manner as 0 is designated.
B	Sub part system identification No.	1 to 7	Identification No. used for timing synchronization with sub part system, etc. •The sub part system to be activated is designated by an identification No. The correspondence between identification No. and part system No. depends on the MTB specifications (parameter "#12049 SBS_no"). •If there is no command, it is handled in the same manner as 1 is designated.
H	Sub part system reset type (*2)	0 / 1	0: The G command modal is maintained by the reset when a sub part system is complete. 1: The G command modal is initialized by the reset when a sub part system is complete. •If there is no command, it is handled in the same manner as 0 is designated.
(Argument)	Argument of a sub part system local variable	Setting range of local variable (Decimal point command is possible.)	•Argument is passed to the sub part system as a local variable (level 0). However, addresses A, B, D, G, H, K, O, P, and Q cannot be used as an argument. •For the correspondence between address and variable number, refer to the following table.

(*1) When the parameter "#1253 set25/bit0" is set to "1", the command range is "100010000 to 199999989".

(*2) If a sub part system ends by M99 or the end sequence No., resetting processing is performed automatically in the sub part system.

Correspondence of argument designation address and variable number in sub part system

Argument designation address	Variable number in sub part system
A	-
B	-
C	#3
D	-
E	#8
F	#9
G	-
H	-
I	#4
J	#5
K	-
L	#12
M	#13

Argument designation address	Variable number in sub part system
N	#14
O	-
P	-
Q	-
R	#18
S	#19
T	#20
U	#21
V	#22
W	#23
X	#24
Y	#25
Z	#26

Note

(1) Addresses can be designated in an arbitrary order.

(2) Addresses which do not need to be designated can be omitted.

(3) Local variables in a sub part system are initialized every time the sub part system is activated. Default value is <empty>.

(4) To use local variables in a sub part system, user macros must be available. For the available functions of each model, refer to the list.

Operation mode of a sub part system

(1) The operation mode of sub part systems is used as "sub part system I operation mode". If the memory mode/MDI mode and the sub part system I operation mode are entered at the same time, the stop code (T01 0108) will be generated.

(2) In a part system operating the sub part system I operation mode, the operation mode appears as "SUB" in the part system display of the operation screen.
If an alarm or warning occurs in a sub part system, the part system No. appears as "SUB" in the alarm/warning message of the operation screen.

(3) If the sub part system control I command is issued to a part system that is not operating the sub part system I operation mode, an operation error (M01 1111) will occur.

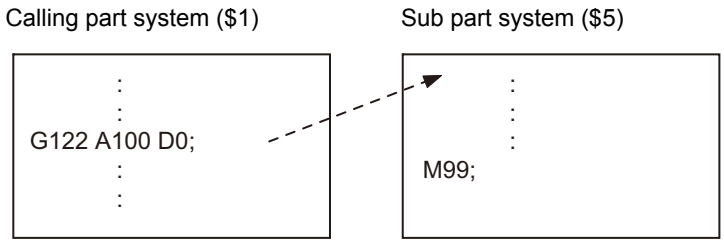
Activation part system of a sub part system

When issuing the sub part system control I command, designate the sub part system identification No. with command address B. (When there is no B command, it will be handled as the B1 command.) The sub part system identification No. and the sub part system No. to be called depend on the MTB specifications. (Parameter "#12049 SBS_no")

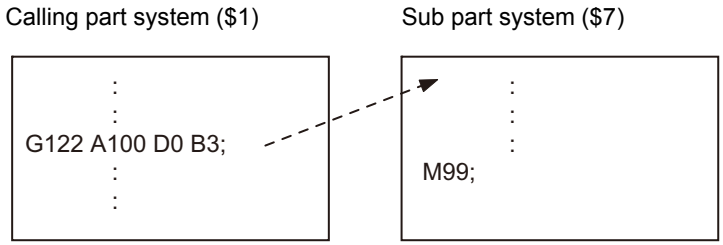
(Example 1) and (Example 2) show the operations when parameters are set as shown below.

#12049	SBS_no	Sub part system I identification No.	$1	$2	$3	$4	$5	$6	$7	$8
			0	0	0	0	1	2	3	4

(Example 1) If the B command is omitted, $5 corresponding to B1 will be activated.



(Example 2) Sub part system identification No. (the part system No. to be activated and correspondence) can be specified with the B command.

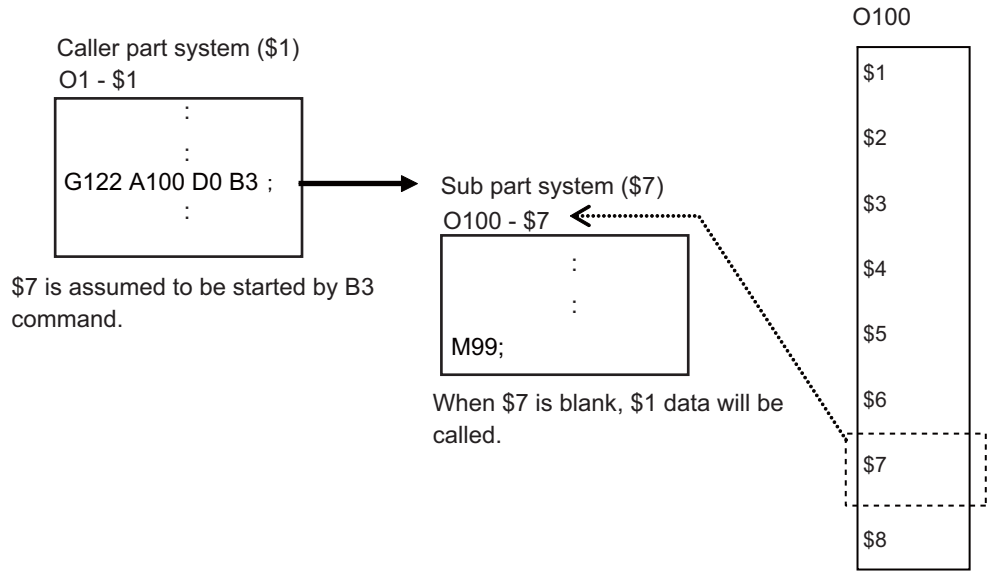


Operation program of a sub part system

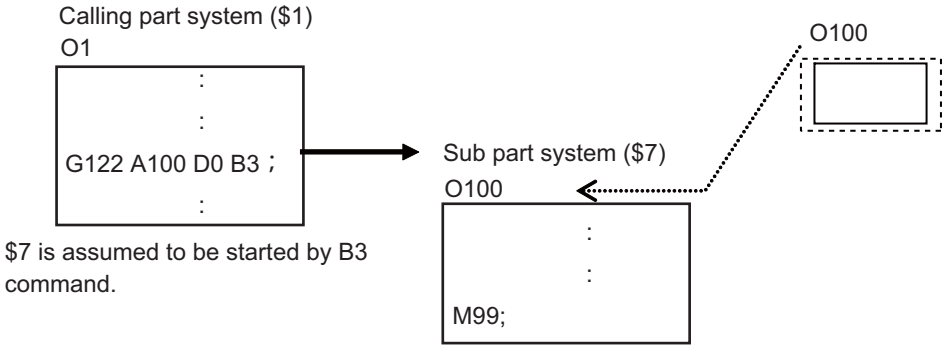
When issuing the sub part system control I command, designate the program No. or program name to be operated in the sub part system with command address A or <file name>. If designation of the program is omitted, the machining program set in parameter "#12050 SBS_pro" will be started.
If a machining program is managed for each part system, the program of the part system designated as a sub part system will be operated (*1). If a machining program is commonly managed between part systems, the designated program will be operated.

(*1) If the program of the part system No. for the sub part system is empty, the program of the 1st part system ($1) will be operated. If the program of the 1st part system is also empty, a program error (P461) will occur.

(1) If program is managed for each part system

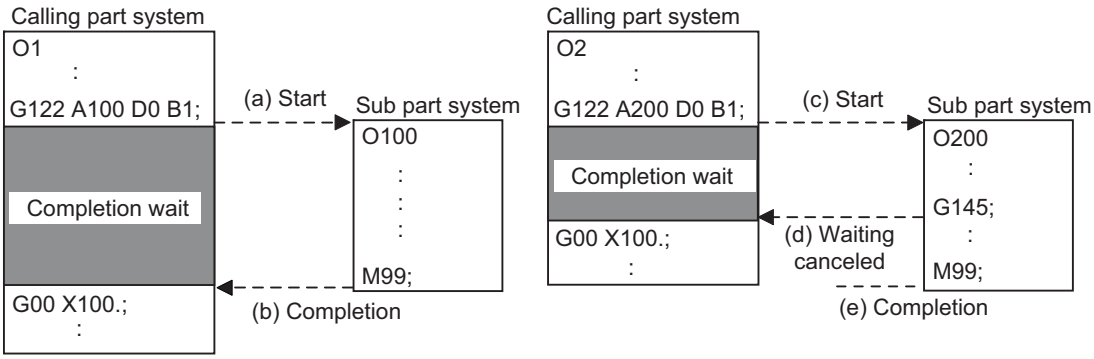


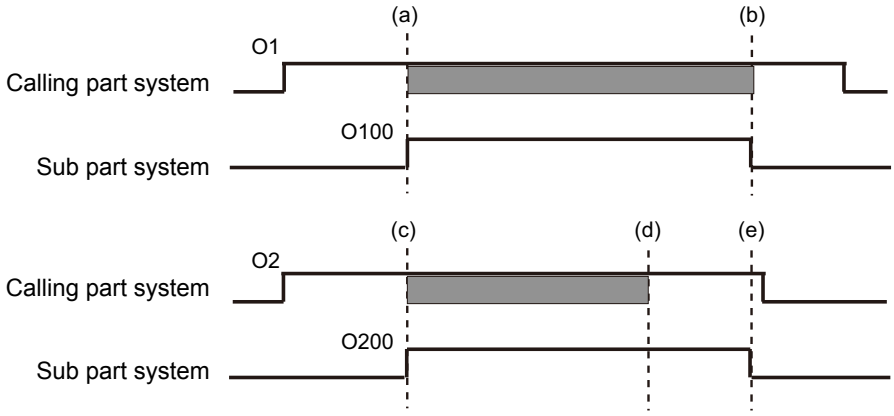
(2) If program is commonly managed between part systems



Sub part system activation with the completion wait method (D=0)

If "0" is designated for command address D when the sub part system control I command is issued, or if command address D is omitted, the calling part system will wait for the called sub part system to complete (to M99 or the end sequence No.) before starting the next block.
Meanwhile, if the completion wait cancel command (G145) is issued in a sub part system while the calling part system is in the sub part system completion standby state, the machine will shift to a parallel processing mode.
The following shows the operation and the activation timing of each part system.

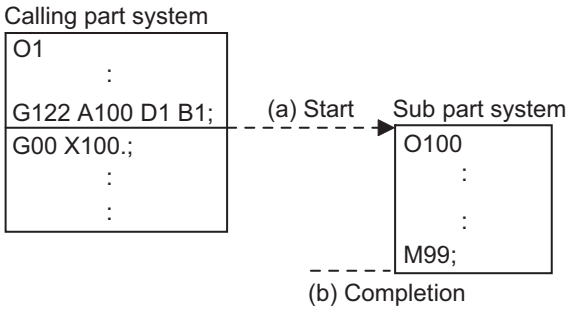


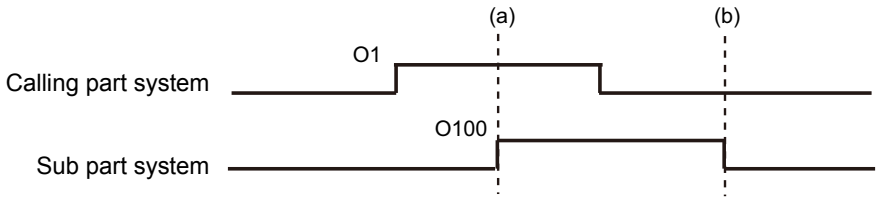


: Completion wait

Activation of a sub part system with parallel processing mode (D=1)

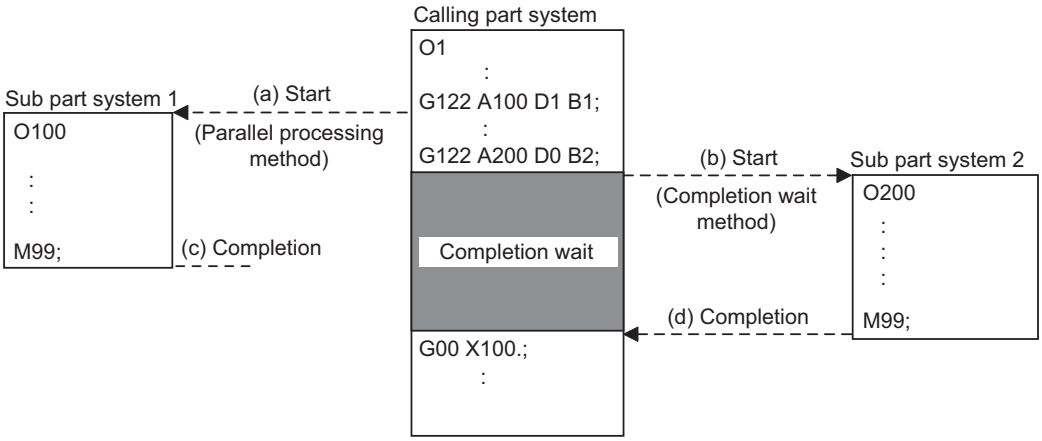
If "1" is designated for command address D when the sub part system control I command is issued, the following blocks of the calling part system and the first and the following blocks of the sub part system will be operated in parallel.
The following shows the operation and the activation timing of each part system.

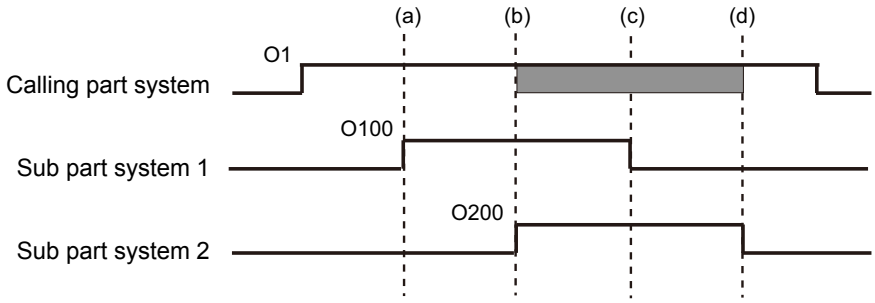




Activation of multiple sub part systems

Multiple sub part systems can be activated in parallel during separate processes by calling from a single part system.
The number of sub part systems to be processed simultaneously depends on the model.
The following shows the operation and the activation timing of each part system.

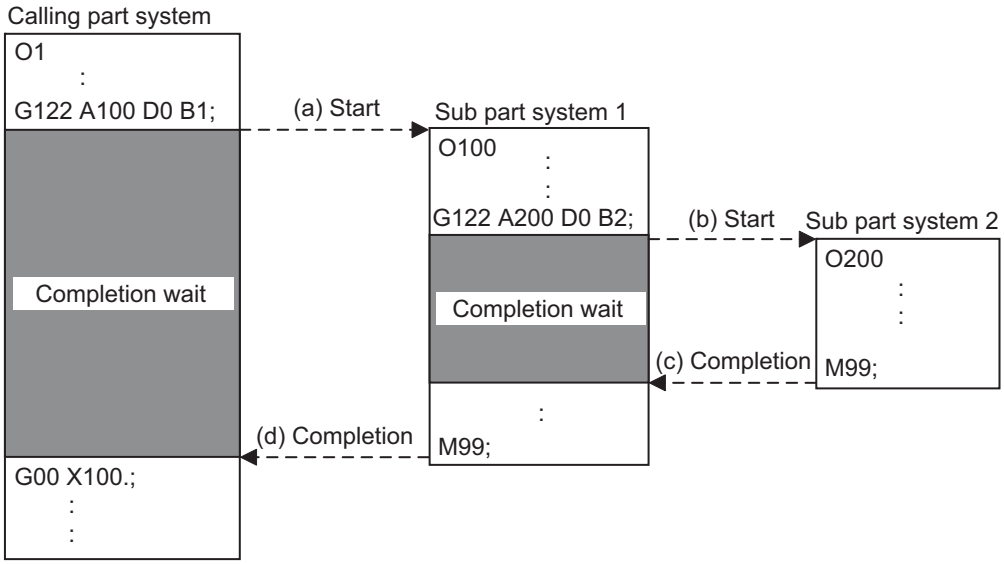


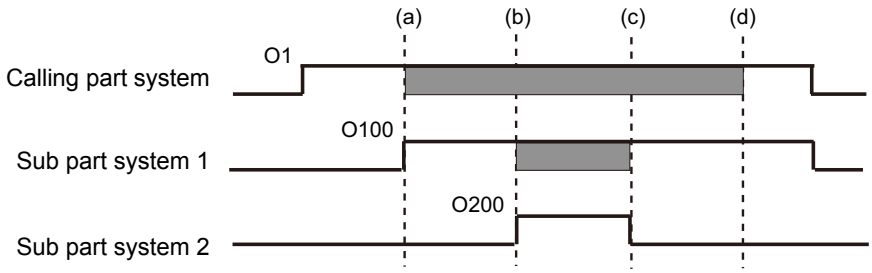


: Completion wait

Activate a sub part system from another sub part system

A sub part system can be activated from another sub part system.
The number of sub part systems to be processed simultaneously depends on the model.
The following shows the operation and the activation timing of each part system.

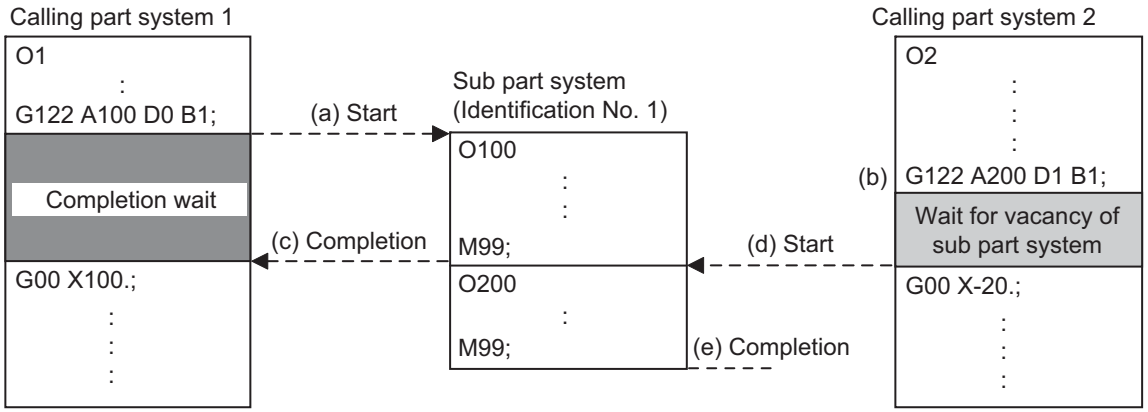


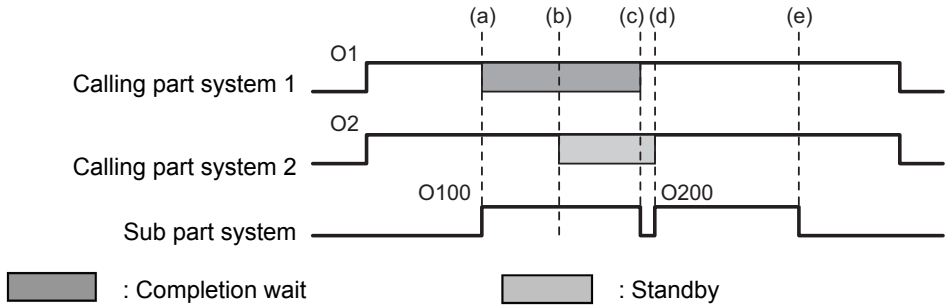


: Completion wait

Sub part system activation command to a sub part system being activated

If G122 is commanded while a sub part system is being activated, using the same identification No. (B command), the machine will wait for the earlier sub part system to complete activation, before activating the next sub part system.





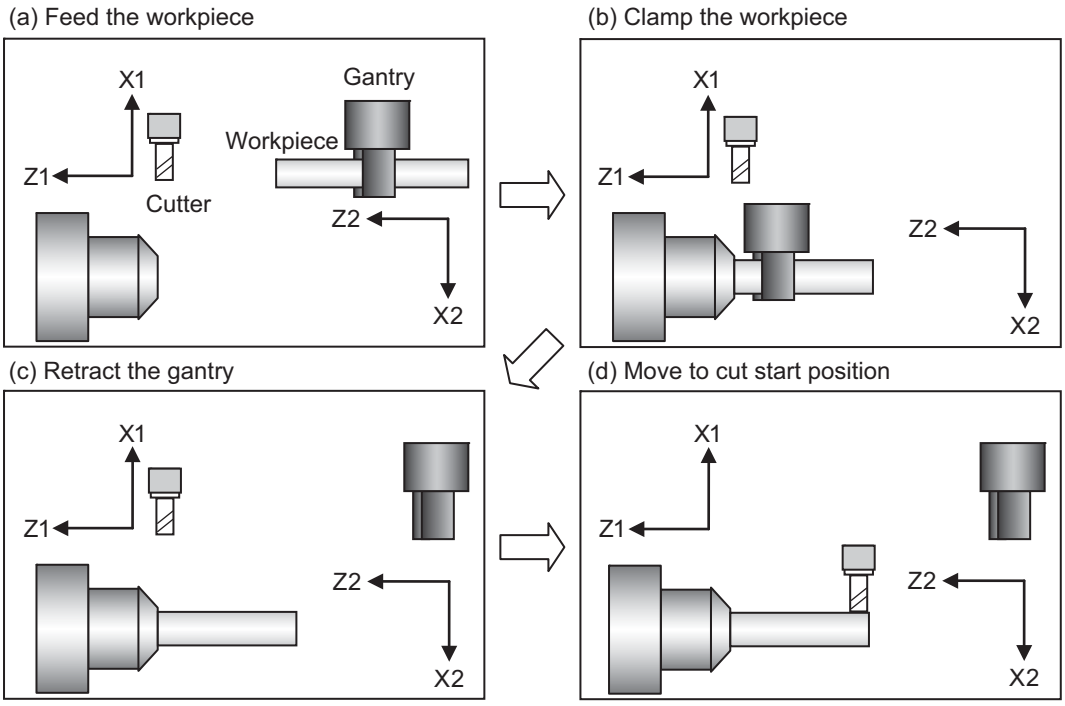
Operation example

In the following example, the machining start timing is accelerated by controlling auxiliary axis with a sub part system and operating the main part system and the sub part system in parallel. The tool positioning starts to the machining start point at the same time (time T1) as the start of gantry retract by using sub part system completion wait cancel command (G145) in the flow from mounting the workpiece to moving to cut start position, after feeding and mounting the workpiece with the gantry, in order to reduce the cycle time. (The machine configuration below is a sample only.)

[Axis configuration]

Main part system ($1) : X1 axis, Z1 axis => Tool
Sub part system ($2) : X2 axis, Z2 axis => Workpiece feed gantry

[Machining process]

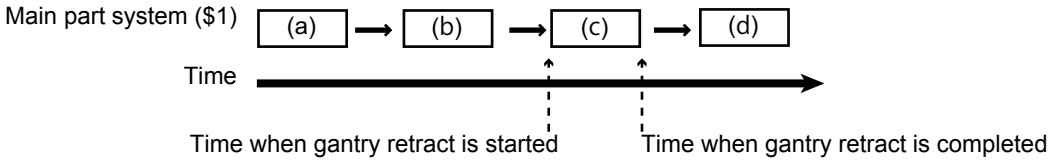


(1) Machining process when sub part system control is OFF

Main part system ($1)

```
O1
                    :
G140 X=X2 Z=Z2;     ... (a)
G00 X50.;
G00 Z20.;
M20;                ... (b)
G00 X0. Z0. ;       ... (c)
G141;
G00 X30. Z-15.;     ... (d)
G01 Z-20. F10.;
                    :
```

G140: Arbitrary axis exchange command (Lathe system only)
G141: Arbitrary axis exchange return command (Lathe system only)
M20 : M code of workpiece mounting



After the gantry is retracted, cut start position is determined.

(2) Machining process when sub part system control is ON

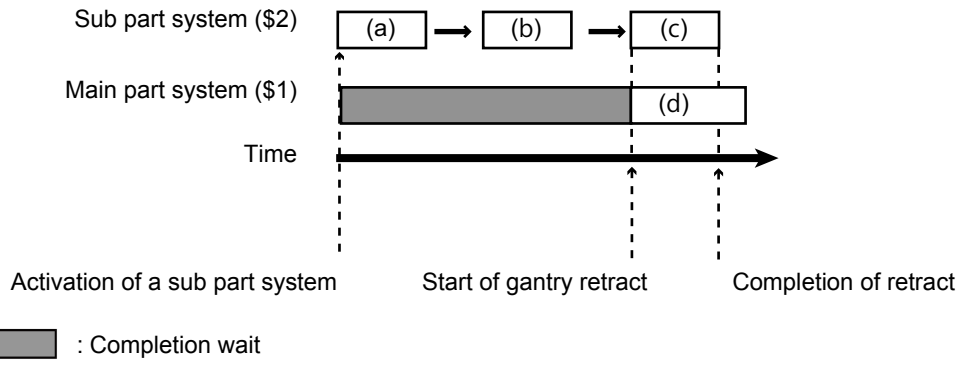
Main part system ($1)

```
O1
        :
        :
G122 A100 D0 B1;
G00 X30. Z-15.;     ... (d)
G01 Z-20. F10.;
        :
        :
```

Sub part system ($2)

```
O100
G00 X50.;           ... (a)
G00 Z20.;
M20;                ... (b)
G145;
G00 X0. Z0.;        ... (c)
        :
M99;
```

M20 : M code of workpiece mounting



Processes after "(c) Retract gantry" and "(d) Move to cut start position" will be operated in parallel.

Relationship with Other Functions

Timing synchronization with sub part system

While a sub part system is under control, timing synchronization between part systems can be issued with the "![Part system No.]" command. To synchronize timing between a main part system and a sub part system, or between sub part systems, it is also possible to designate a sub part system identification No. (B command) as shown below. However, the number of part systems that can be used is limited by the specifications.

![Sub part system identification No.]

For example, to synchronize timing with the calling part system, command "![0]". Note that, designate the calling part system with "![0]", not the main part system.
(Example 1) and (Example 2) shown below are examples of the timing synchronization operation between the main part system ($3), sub part system 1 ($5, identification No. 1), and sub part system 2 ($6, identification No. 2).

(Example 1) Timing synchronization by designating a part system No.

Main part system ($3)	Sub part system 1 ($5)	Sub part system 2 ($6)
: G122 A100 D1 B1; G122 A200 D1 B2; : !5!6; G00 X100.; : :	: : : : !3!6; : : :	: : : : !3!5; : : :
Timing synchronization with 5th and 6th part systems	Timing synchronization with 3rd and 6th part systems	Timing synchronization with 3rd and 5th part systems

(Example 2) Timing synchronization by designating a sub part system identification No.

Main part system ($3)	Sub part system 1 ($5, identification No. 1)	Sub part system 2 ($6, identification No. 2)
: G122 A100 D1 B1; G122 A200 D1 B2; : ![1]![2]; G00 X100.; :	: : : : ![0]![2]; : :	: : : : ![0]![1]; : :
Timing synchronization with the following part system • Sub part system of identification No. 1 • Sub part system of identification No. 2	Timing synchronization with the following part system • Calling part system ($3) • Sub part system of identification No. 2	Timing synchronization with the following part system • Calling part system ($3) • Sub part system of identification No. 1

Timing synchronization operation ignore signal

Whether to ignore the "![Sub part system identification No.]" command or not depends on the MTB specifications. (Settings of parameter "#1279 ext15/BIT0" and the following PLC signal)

<table>
<tr><th rowspan="2">#1279 ext15/BIT0</th><th rowspan="2">PLC signal for ignoring timing synchronization between part systems</th><th colspan="2">Operation</th></tr>
<tr><th>If the other part system is being activated as a sub part system</th><th>If the other part system is not being activated as a sub part system</th></tr>
<tr><td rowspan="2">0</td><td>ON</td><td rowspan="2">The timing synchronization operation is ignored when activation of a sub part system is completed for the other part system.</td><td rowspan="4">Program error (P35)</td></tr>
<tr><td>OFF</td></tr>
<tr><td rowspan="2">1</td><td>ON</td><td>Ignore the timing synchronization operation.</td></tr>
<tr><td>OFF</td><td>Execute the timing synchronization operation between part systems.</td></tr>
</table>

Arbitrary Axis Exchange Control

With the sub part system control I, axes that belong to the sub part system when the sub part system is activated can be controlled. To change the axis to be controlled, exchange axes (to transfer the control rights of the specified axis from other part systems to the own part system) with the arbitrary axis exchange return command (G140).

Tool Functions

If the tool No. is changed (T command) in the program run of a sub part system, the T code data will be changed for the sub part system only. The T code data will not be changed for the main part system or other sub part systems.

Tool compensation

When an axis in the main part system, for which the tool compensation has been commanded, is moved to a sub part system with the arbitrary axis exchange or other operation, the tool compensation will be maintained. Also, when an axis (*1) in a sub part system, for which tool compensation has been commanded, is moved to the main part system or another sub part system with the arbitrary axis exchange operation, tool compensation will be maintained.

(*1) If tools are managed for each part system, the offset data to be referenced when the tool compensation command is issued in a sub part system is used as setting values for the sub part system. (The setting value of the main part system will not be referenced.)

User macro

The sub part system control I command does not affect nesting in user macros and subprograms. It can be commanded from a subprogram nested at the deepest level.

Synchronization between part systems

Like the main part system, sub part systems are also affected by the synchronization function between part systems, such as single block operation with part systems synchronized, and dwell/miscellaneous function time override. Also, depending on the specifications, like the main part system, the synchronization function between part systems can be disabled for sub part systems.

Resetting

(1) If the NC reset signal is input to the main part system, the operation of the main part system will be reset and end immediately. However, the operation of sub part systems will continue. The reset operation of the sub part system follows the NC reset signal of the sub part system.

(2) If the NC reset signal is input to an operating sub part system, the operation of the sub part system will end immediately. Therefore, if the calling part system is in the sub part system completion standby state, the sub part system is reset, and at the same time, the calling part system cancels the standby state, and the following block will be executed.

Buffer correction

If both of the following conditions (1) and (2) are satisfied, the buffer correction is disabled. (The buffer correction window will not open even if the program correction key is pressed.)

(1) The next block is G122 command (including "macro statement + G122 command").

(2) The program designated by G122 is the same as that of the calling part system.

O100	
:	
G00 Z50.;	Buffer correction possible
G00 X100.;	Buffer correction impossible
G122 A100 P77 D0 B1;	Designated program is the program of its own part system (O100)
G00 Y30.;	Buffer correction possible
:	
N77 : M99;	Program operated in sub part system

Machining time computation

The completion wait time of the sub part system control I command (G122) will not be added to the machining time computation for the main part system.

Program restart

If the restart search from the block of the G122 command is attempted, a program error (P49) will occur.

Illegal modal of a sub part system control I command

If the sub part system control I (G122) is commanded during the following G command modal, a program error (P652) will occur.

- User macro modal call (G66, G66.1)
- Fixed cycle modal
- High-speed machining mode (G05P1, G05P2)
- High-speed high-accuracy mode (G05.1Q1, G05P10000)

Manual arbitrary reverse run

The sub part system control I (G122) is ignored at the reverse run or the forward run after the reverse run. Because the sub part systems are in a mode in which reverse run is prohibited, reverse run cannot be carried out in sub part systems.

Precautions

(1) The sub part system control I command (G122) is a G code that must be issued alone. If it is commanded in the same block together with another G code, a program error (P651) or (P32) will occur.

- If another G code is commanded prior to G122 (for example, G00 G122), a program error (P651) will occur.
- If another G code is commanded following G122 (for example, G122 G00), a program error (P32) will occur.

(2) While the sub part system I operation mode is in operation, even if the sub part system is not being activated, automatic operation cannot be started with the automatic operation start signal (ST). The stop code (0146) will be generated. However, when a sub part system is being activated, automatic operation is started with the automatic operation start signal (ST).

(3) If a sub part system identification No. of its own part system is designated for the B command with the sub part system control I command (G122), a program error (P650) will occur.

(4) The PLC signal of the sub part system references the state of the sub part system. (The signal state of the main part system will not be taken over.)

(5) Parameters per part system of the sub part system follow the setting in the sub part system. Therefore, parameters must also be set in the sub part system.

(6) If the sub part system completion wait cancel command (G145) is issued in the main part system, the program error (P34) will occur.

(7) Operation executed by M80 is as follows. These parameter settings depend on the MTB specifications.

- Activation of a sub part system is only possible in sub part systems that are reserved using the parameter "#1483 SBS1_sys num". If the sub part system activation command is issued to a main part system (*1), an operation error (M01 1111) will occur.
  (*1) This refers to a case in which the sub part system I operation mode is established (SBSM: ON) using the PLC signal before G122 is commanded.
- Operation searches cannot be carried out in sub part systems that are reserved using the parameter "#1483 SBS1_sys num".
- If the values set for the parameters "#1483 SBS1_sys num" and "#1474 SBS2_sys num" are both "1" or more, an MCP alarm (Y05 1483) will occur.

16.9.2 Sub Part System Control II ; G144

Function and purpose

This function activates a sub part system (called part system) by issuing the G144 command in an arbitrary part system (calling part system). Using this function, main part system and sub part system can be operated in parallel, enabling a reduction of the cycle time. Use this function to move the tool to the cutting start position, while positioning C axis, etc.
When the sub part system control II is commanded, non-operating sub part systems are activated through the automatic identification.
(When the sub part system control I is commanded, sub part systems should be designated in the program.)
For the difference between sub part system control I and sub part system control II , refer to "16.9.1 Sub Part System Control I ; G122".
The number of part systems that can be used is limited by the specifications.

The following describes the meanings of the terms used in this chapter.

Term	Meaning
Main part system	Indicates a part system located on the uppermost stream side of a sub part system call flow.
Sub part system	Indicates a part system activated by the sub part system activation command. (Normal automatic operation cannot be performed.)
Calling part system	Indicates a part system that issued the sub part system activation command.

Command format

Call sub part system

```
G144 A__P__Q__K__D__B__H__ (argument);
G144 <file name> P__Q__K__D__B__H__ (argument);
```

Complete sub part system

```
M99;  (command of a sub part system side)
```

Cancel the standby status for completion of sub part system

```
G145;   (command of a sub part system side that is issued when the D0 command is issued)
```

Note

(1) G145 is ignored in a sub part system activated in the parallel control method (D1 command).

A	Program No. (1 to 99999999 or 100010000 to 199999998)
<File name>	File name of the program (up to 32 characters)
P	Start sequence number (Head of the program if omitted.)
Q	End sequence number (To end (M99) of the program if omitted.)
K	Number of repetitions (1 to 9999)
D	Synchronization control (0/1)
B	Sub part system identification No. (1 to 9999)
H	Sub part system designation (1 to 8)
Argument	Argument of a sub part system local variable (Setting rage of local variable (decimal point command is valid))

Detailed description

This function can be used in multi-part systems of two or more part systems.

Description of each address

Address	Meaning	Command range (unit)	Remarks
A	Program No.	1 to 99999999 or 100010000 to 199999998 (*1)	Program No. or file name of the machining program operating in the sub part system. •Programs in external device cannot be designated. •If address A and <file name> are designated at the same time, precedence is given to address A. •If designation of the program with A command or <file name> is omitted, the same program operating in the calling part system will be used for the program in sub part systems. Only in this case, the start sequence No. must be designated. A program error (P33) will occur if there is no designation.
<File name>	File name of the program	Up to 32 characters.	
P	Start sequence No.	1 to 99999999	Sequence No. to start the machining program operating in the sub part system. •If there is no command, the operation will start from the head of the machining program.
Q	End sequence No.	1 to 99999999	Sequence No. to end the machining program operating in the sub part system. •If there is no command, the program will run up to M99.
K	Number of repetitions	1 to 9999	The number of times to repeat the machining program for continuous operation in the sub part system. •If there is no command, the program will run only once. (No repetition)
D	Synchronization control	0 / 1	Validity of synchronous control 0: The next block is processed after the sub part system operation completes. 1: The next block is processed at the same time as the start of a sub part system operation. If there is no command, it is handled in the same manner as "0" is designated.
B	Sub part system identification No.	1 to 9999	Identification No. used for timing synchronization with sub part system, etc. •The sub part system to be activated is designated by an identification No. •A program error (P33) will occur if there is no command.
H	Sub part system designation	1 to 8	•Designate the part system number to be operated as the sub part system. Only in this case, a part system that is not designated as a sub part system in the parameter "#1474 SBS2_sys num" (the number of part systems in sub part system II) can be operated as a sub part system. •If not commanded, the bootable one of the sub part systems designated in the parameter "#1474 SBS2_sys num" (the number of part systems in sub part system II) is selected automatically.
(Argument)	Argument of a sub part system local variable	Setting range of local variable (Decimal point command is possible.)	•Argument is passed to the sub part system as a local variable (level 0). However, addresses A, B, D, G, H, K, O, P, and Q cannot be used as an argument. •For the correspondence between address and variable number, refer to the following table.

(*1) When the parameter "#1253 set25/bit0" is set to "1", the command range is "100010000 to 199999989".

Correspondence of argument designation address and variable number in sub part system

Argument designation address	Variable number in sub part system
A	-
B	-
C	#3
D	-
E	#8
F	#9
G	-
H	-
I	#4
J	#5
K	-
L	#12
M	#13

Argument designation address	Variable number in sub part system
N	#14
O	-
P	-
Q	-
R	#18
S	#19
T	#20
U	#21
V	#22
W	#23
X	#24
Y	#25
Z	#26

Note

(1) Addresses can be designated in an arbitrary order.

(2) Addresses which do not need to be designated can be omitted.

(3) Local variables in a sub part system are initialized every time the sub part system is activated. Default value is <empty>.

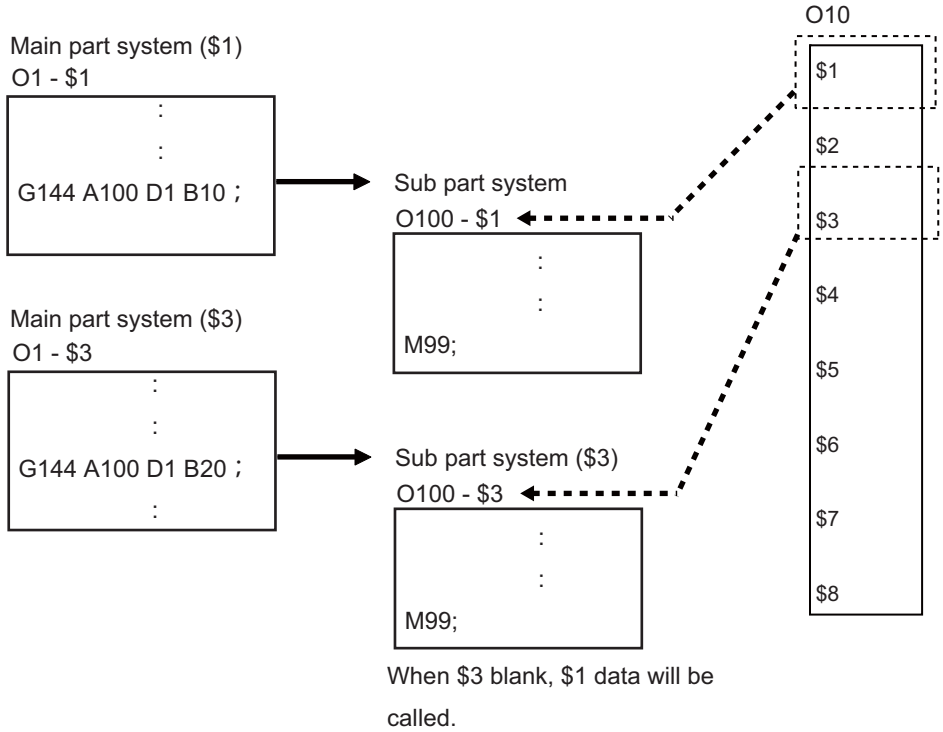
(4) To use local variables in a sub part system, user macros must be available. For the available functions of each model, refer to the list.

Operation program of a sub part system

When issuing the sub part system control II command, designate the program No. or program name to be operated in the sub part system with command address A or <file name>. If designation of the program is omitted, the same program operating in the calling part system will be used for the program in sub part systems. (In this case, the start sequence No. must be designated.)
If a machining program is managed for each part system, the program of the main part system will be operated (*1).
If a machining program is commonly managed between part systems, the designated program will be operated.

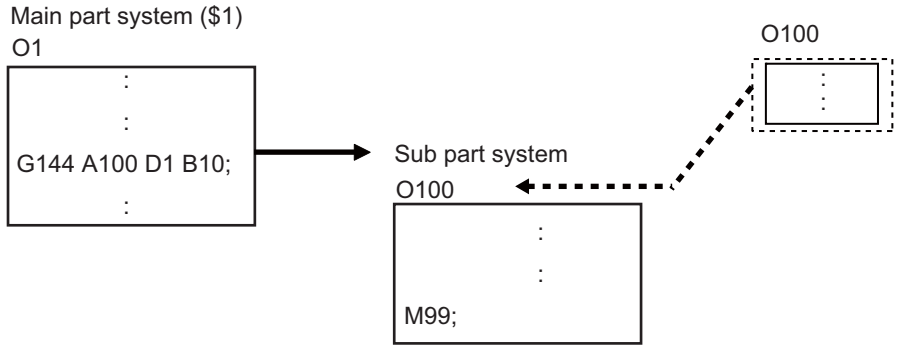
(*1) If the program of the part system No. for the main part system is empty, the program of the 1st part system ($1) will be operated. If the program of the 1st part system is also empty, a program error (P461) will occur.

(1) If program is managed for each part system



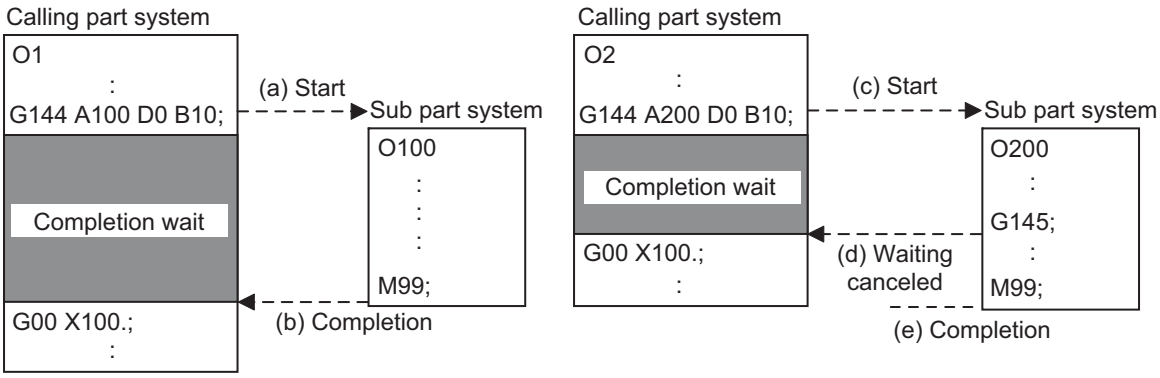
When $3 blank, $1 data will be called.

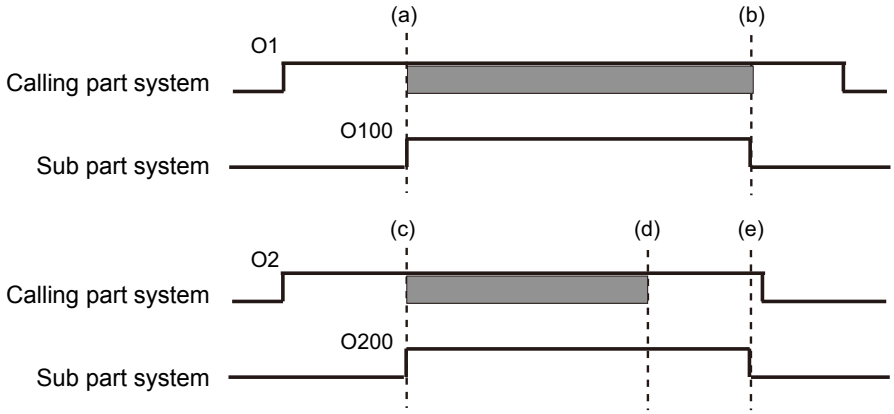
(2) If program is commonly managed between part systems



Sub part system activation with the completion wait method (D=0)

If "0" is designated for command address D when the sub part system control II command is issued, or if command address D is omitted, the calling part system will wait for the called sub part system to complete (to M99 or the end sequence No.) before starting the next block.
Meanwhile, if the completion wait cancel command (G145) is issued in a sub part system while the calling part system is in the sub part system completion standby state, the machine will shift to a parallel processing mode.
The following shows the operation and the activation timing of each part system.

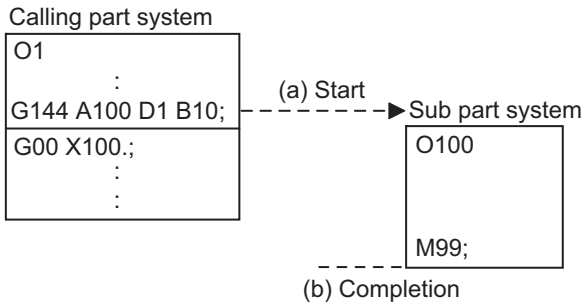


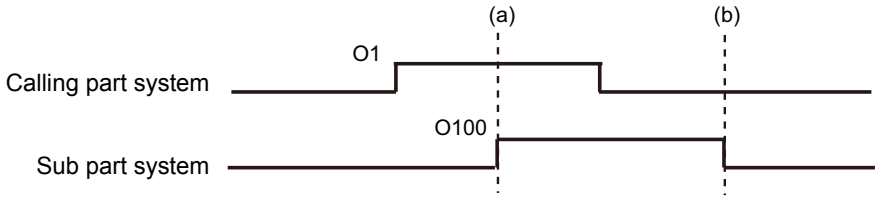


: Completion wait

Activation of a sub part system with parallel processing mode (D=1)

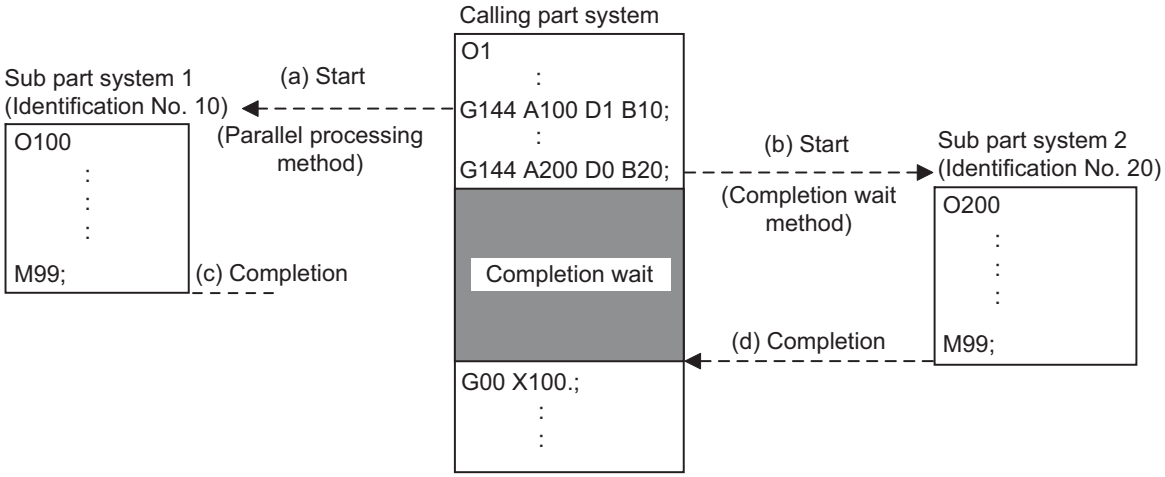
If "1" is designated for command address D when the sub part system control II command is issued, the following blocks of the calling part system and the first and the following blocks of the sub part system will be operated in parallel.
The following shows the operation and the activation timing of each part system.

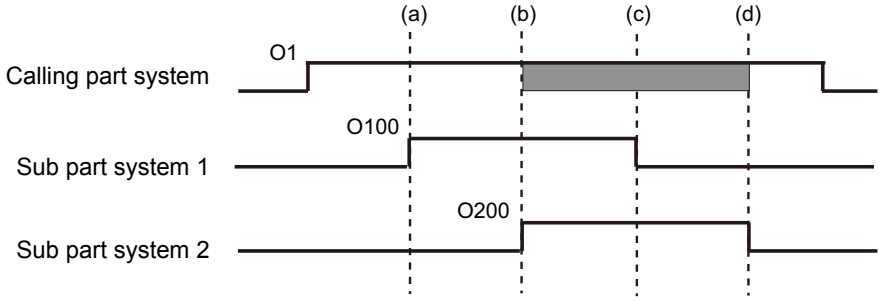




Activation of multiple sub part systems

Multiple sub part systems can be activated in parallel during separate processes by calling from a single part system.
The number of sub part systems to be processed simultaneously depends on the model.
The following shows the operation and the activation timing of each part system.

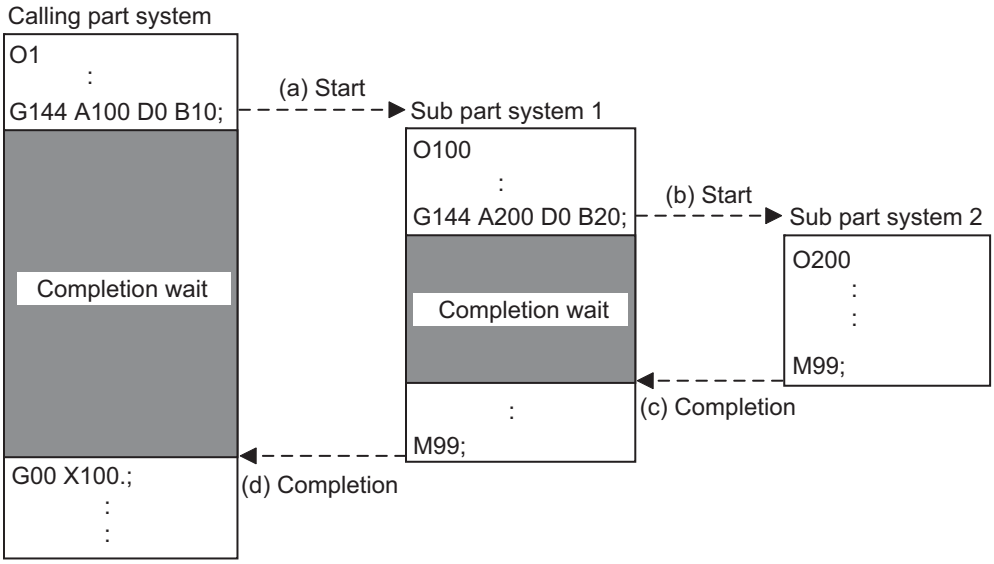


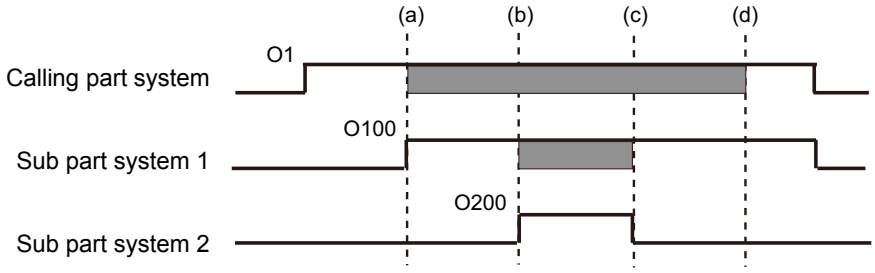


: Completion wait

Activate a sub part system from another sub part system

A sub part system can be activated from another sub part system.
The number of sub part systems to be processed simultaneously depends on the model.
The following shows the operation and the activation timing of each part system.

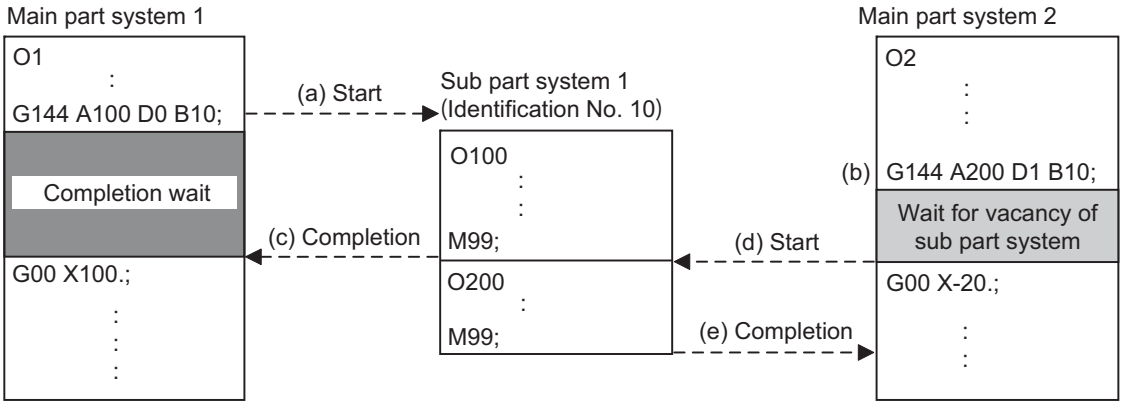


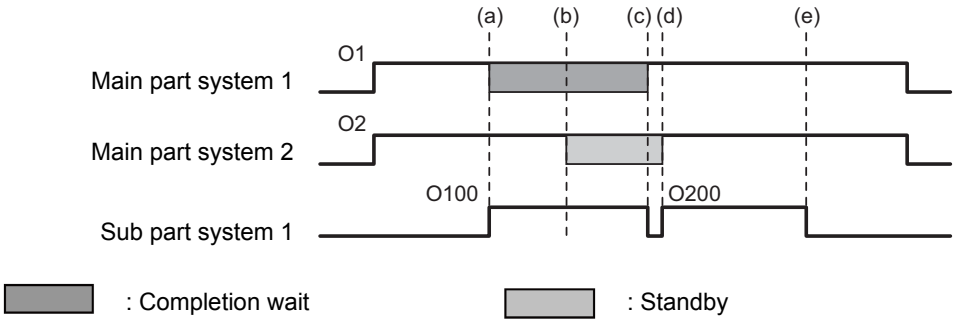


: Completion wait

Sub part system activation command to a sub part system being activated

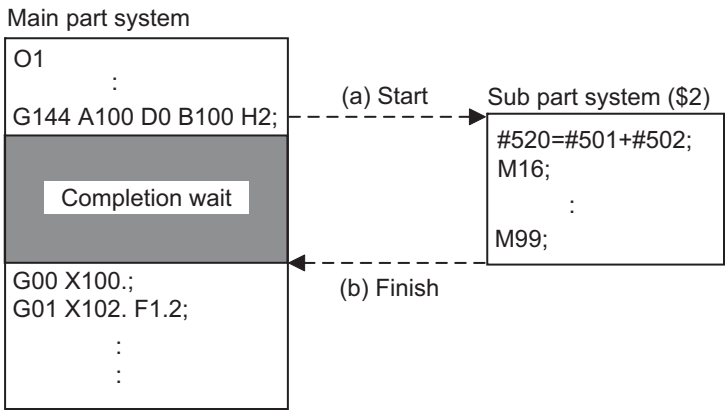
If G144 is commanded while a sub part system is being activated, using the same identification No. (B command), the machine will wait for the earlier sub part system to complete activation, before activating the next sub part system.
The following shows the operation and the activation timing of each part system.





Using a normal part system as a sub part system

To designate the part system to be operated as a sub part system, designate the part system number with the H command.
Only in this case, a part system that is not designated as a sub part system in the parameter can be operated as a sub part system.
If the designated part system is in the automatic or manual operation mode, an operation error (M01 1112) will occur.



Operation example

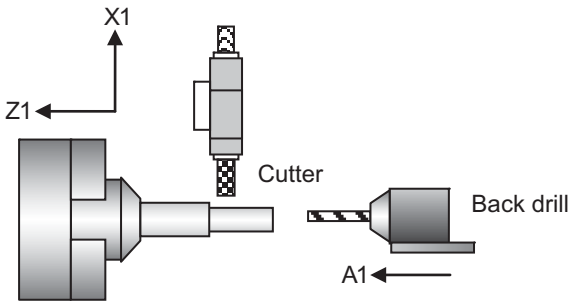
In the following example, multiple machining operations can be performed simultaneously by controlling some of the axes in the main part system with a sub part system and operating the main part system and the sub part system in parallel. When performing the back drill machining (machining 1) and the cutting by tool (machining 2), machining 1 and 2 are operated in order only with main part system, however machining 1 and 2 can be operated simultaneously by using the sub part system in addition to the main part system , resulting in a shorter cycle time. (The time when machining 2 is completed is shorter.)

[Axis configuration of main part system ($1)]

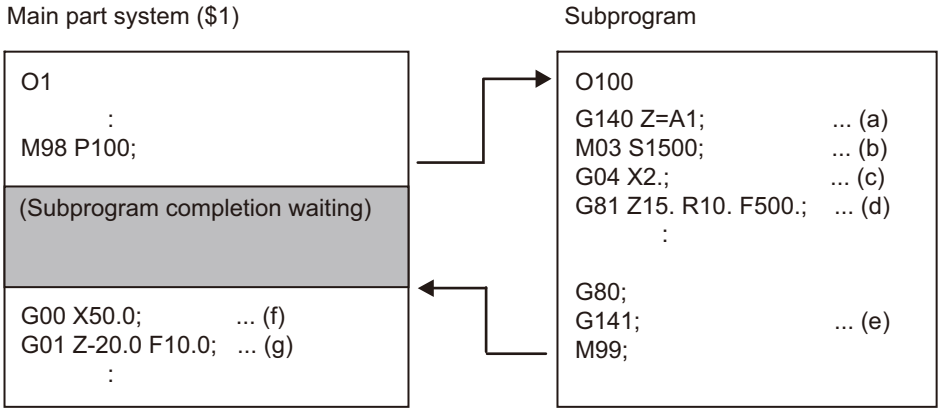
Turret	:X1 axis, Z1 axis
Back drill	:A1 axis

[Machining process]

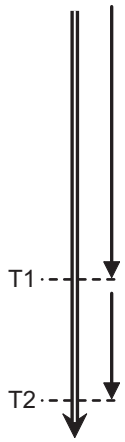
O1	: Main machining program
O100	: Back drill machining program



(1) Machining process when sub part system control is OFF



G140: Arbitrary axis exchange command (Lathe system only)
G141: Arbitrary axis exchange return command (Lathe system only)
G81 : Fixed cycle command

	(a) Set auxiliary axis to command address Z (b) Spindle rotation for drilling (c) Waiting for top spindle speed (d) Drilling (e) Axis exchange return
	(f) Cutting start positioning of turret (g) Cutting by tool

After the back drill machining is complete, cutting by tool is executed.

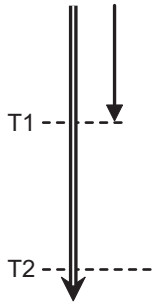
(2) Machining process when sub part system control is ON

Main part system ($1)

```
O1
        :
G144 A100 D1 B10;
G00 X50.0;              ... (a)
G01 Z-20.0 F10.0;       ... (b)
        :
```

Sub part system

```
O100
G140 Z=A1;              ... (a)
M03 S1500;              ... (b)
G04 X2. ;               ... (c)
G81 Z15. R10. F500. ;   ... (d)
G80;
G141;                   ... (e)
M99;
```



Main part system ($1)	Sub part system
Cutting start positioning of turret	(a) Set auxiliary axis to command address Z
	(b) Spindle rotation for drilling
Cutting by tool	(c) Waiting for top spindle speed
	(d) Drilling
	(e) Axis exchange return

Back drill machining and cutting by tool are executed by parallel operation.

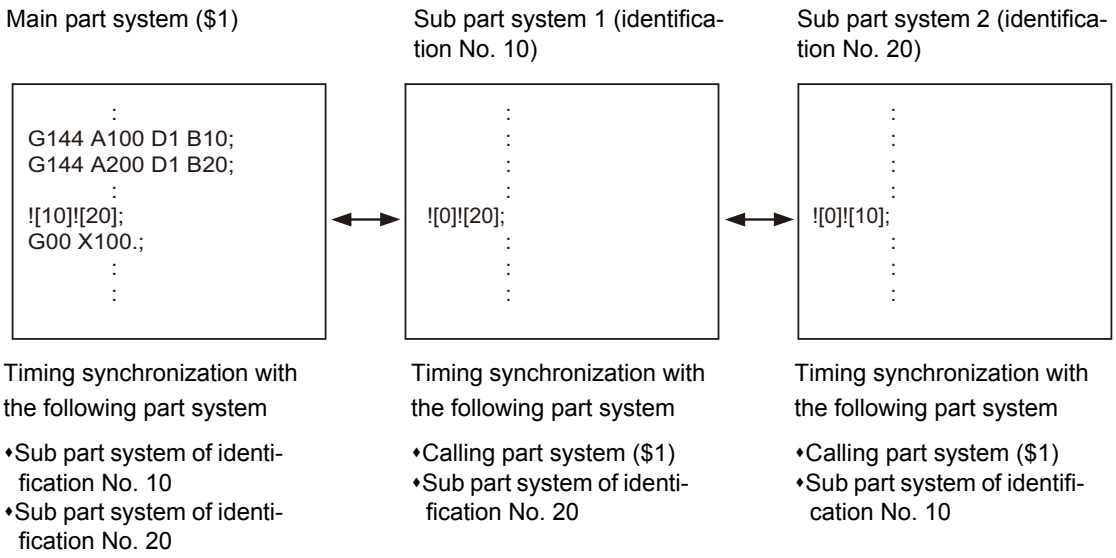
Relationship with Other Functions

Timing synchronization with sub part system

While a sub part system is under control, timing synchronization between part systems can be issued with the "![Part system No.]" command. To synchronize timing between a main part system and a sub part system, or between sub part systems, it is also possible to designate a sub part system identification No. (B command) as shown below. However, the number of part systems that can be used is limited by the specifications.

![Sub part system identification No.]

For example, to synchronize timing with the calling part system, command "![0]". Note that, designate the calling part system with "![0]", not the main part system.
The following is an example of the timing synchronization operation between the main part system ($1), sub part system 1 (identification No. 10), and sub part system 2 (identification No. 20).



Timing synchronization operation ignore signal

Whether to ignore the "![Sub part system identification No.]" command or not depends on the MTB specifications. (Settings of parameter "#1279 ext15/bit0" and the following PLC signal)

#1279 ext15/bit0	PLC signal for ignoring timing synchronization between part systems	Operation	
		If the other part system is being activated as a sub part system	**If the other part system is not being activated as a sub part system**
0	ON	If the timing synchronization with the calling part systems is not commanded in the other part system, the timing synchronization operation will be ignored when the operation is completed for the other part system.	Program error (P35)
	OFF		
1	ON	Ignore the timing synchronization operation.	
	OFF	Execute the timing synchronization operation between part systems.	

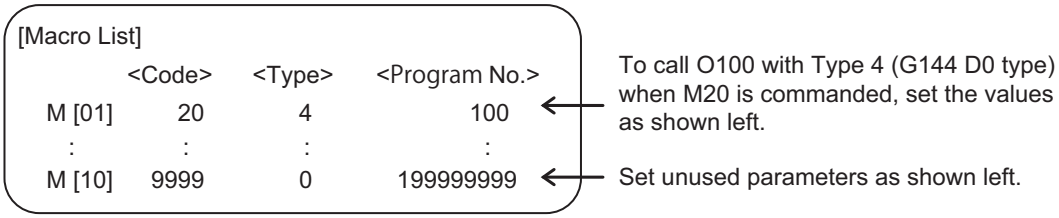
Screen display of a sub part system

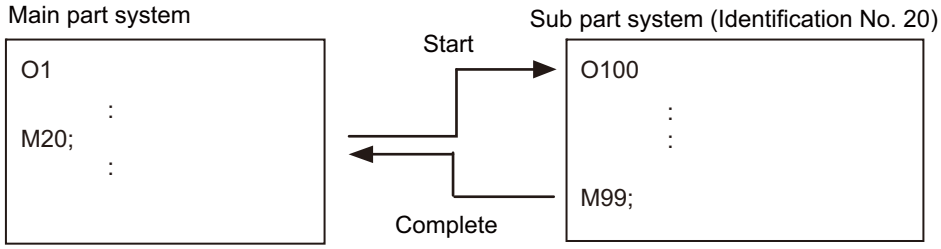
When the number of main part systems is designated in the parameter "#11055 Disp. sysno" (the number of part systems to be displayed), sub part systems are not displayed. (This parameter setting value depends on the MTB specifications.)

Sub part system control II type of miscellaneous command macro

By setting the <type> of macro list parameter, the calling method can be selected. If the calling method (types 4, and 5) with sub part system control II is selected, the value of M code will be used as a sub part system identification No. (B command value).
In the macro calling with types 4 and 5, the default value of local variable #13 (variable number corresponding to argument M) is <empty>. (The M code value will not be passed.)

Calling type	Calling method
0	Equivalent call to M98 PΔΔΔΔ;
1	Equivalent call to G65 PΔΔΔΔ;
2	Equivalent call to G66 PΔΔΔΔ;
3	Equivalent call to G66.1 PΔΔΔΔ
4	Equivalent call to G144 D0; (completion wait method)
5	Equivalent call to G144 D1; (parallel processing method)
Others	Equivalent call to M98 PΔΔΔΔ;





M20 = G144 A100 D0 B20

Tool compensation

When an axis in the main part system, for which the tool compensation has been commanded, is moved to a sub part system with the arbitrary axis exchange or other operation, the tool compensation will be maintained. Also, when an axis (*1) in a sub part system, for which tool compensation has been commanded, is moved to the main part system or another sub part system with the arbitrary axis exchange operation, tool compensation will be maintained.

(*1) If tools are managed for each part system, the offset data to be referenced when the tool compensation command is issued in a sub part system is used as setting values for the main part system.

Arbitrary Axis Exchange Control

In the sub part system control II, the just started sub part system has no axis. To control the axis in a sub part system, carry out axis exchange (to transfer the control rights of the specified axis from other part systems to the own part system) with the arbitrary axis exchange return command (G140).

User macro

The sub part system control II command does not affect nesting in user macros and subprograms. It can be commanded from a subprogram nested at the deepest level.

Synchronization between part systems

Like the main part system, sub part systems are also affected by the synchronization function between part systems, such as single block operation with part systems synchronized, and dwell/miscellaneous function time override. Also, depending on the specifications, like the main part system, the synchronization function between part systems can be disabled for sub part systems.

Resetting

The reset operation of sub part system control II to be carried out when the NC reset 1 signal (NRST1), NC reset 2 signal (NRST2), or reset & rewind signal (RRW) is input to the main part system depends on the MTB specifications (parameter "#1437 SBS2_Spec/bit1").

#1437 SBS2_Spec/bit1	Reset operation
0	The operations of the main and sub part systems are immediately reset and ended even if a sub part system is executing a command. (These reset signals will be ignored if they are input to a sub part system.)
1	Only the main part system is reset. (Sub part systems are not reset.) To reset a sub part system, input these reset signals to the sub part system.

Buffer correction

If both of the following conditions (1) and (2) are satisfied, the buffer correction is disabled. (The buffer correction window will not open even if the program correction key is pressed.)

(1) The next block is G144 command (including "macro statement + G144 command").

(2) The program designated by G144 is the same as that of the calling part system.

O100	
:	
G00 Z50.;	Buffer correction possible
G00 X100.;	Buffer correction impossible
G144 A100 P77 D0 B1;	Designated program is the program of its own part system (O100)
G00 Y30.;	Buffer correction possible
:	
N77 : M99;	Program operated in sub part system

Also, the buffer correction is not available for the program operating in a sub part system.

Illegal modal of a sub part system control II command

If the sub part system control II (G144) is commanded during the following G command modal, a program error (P652) will occur.

- User macro modal call (G66, G66.1)
- Fixed cycle modal
- High-speed machining mode (G05P1, G05P2)
- High-speed high-accuracy mode (G05.1Q1, G05P10000)

Functions that cannot be used in sub part system

If any of the following G codes is commanded in a sub part system, a program error (P653) will occur.

- Program data input command (G10)
- Automatic tool length measurement (G37)
- Workpiece coordinate system selection (G54 G59, G54.1)
- Barrier ON command (G22)

Barrier check is not available for the axis used in sub part system, even if the barrier has been enabled in the main part system.

Precautions

(1) The sub part system control II command (G144) is a G code that must be issued alone in a single block. If another G code is commanded prior to G144, a program error (P651) will occur. If another G code is commanded following G144, a program error (P32) will occur.

(2) G command modals for the sub part system are initialized at activation.

(3) If the sub part system completion wait cancel command (G145) is issued in the main part system, the program error (P34) will occur.

(4) If the number of sub part systems reserved using the parameter "#1474 SBS2_sys num" (the number of part systems in sub part system II), or the number of main part systems (*1) exceeds the maximum number defined in the system specifications, an MCP alarm (Y05 1474) will occur.
(*1) Indicates the number of part systems that is determined by subtracting the number of sub part systems from the number of part systems for which "#1001 SYS_ON" is set to "1".

(5) In M80, an MCP alarm (Y05 1483) will occur if the values set for the parameters "#1483 SBS1_sys num" and "#1474 SBS2_sys num" are both "1" or more. These parameter settings depend on the MTB specifications.

(6) When the sub part system control II specifications are invalid, an MCP alarm (Y05 1474) will occur if the value set for the parameter "#1474 SBS2_sys num" is "1" or more.

17

High-speed High-accuracy Control

17.1 High-speed Machining Mode

17.1.1 High-speed Machining Mode I, II ; G05 P1, G05 P2

Function and purpose

This function runs a machining program for which a freely curved surface has been approximated by fine segments at high speed.
A higher fine segment processing capability leads to a faster cutting speed, resulting in a shorter cycle time and a better machining surface quality.
The high-speed high-accuracy control I/II enable not only the high-speed machining mode but also the high-accuracy control mode. Use the high-speed high-accuracy control I/II for machining which needs to make an edge at a corner or reduce an error from an inner route of curved shape.
This function can be used simultaneously for up to two part systems depending on the MTB specifications.
kBPM, the unit for the fine segment processing capability, is an abbreviation of "kilo blocks per minute" and refers to the number of machining program blocks that can be processed per minute.
In the main text, the axis address refers to the address of an axis that exits on the machine.
It corresponds to the address designated in the parameters "#1013 axname" and "#1014 incax".
These parameter settings depend on the MTB specifications.

For one part system

G01 block fine segment capacity for 1mm segment (unit: kBPM)

Mode	Command	Maximum feedrate when 1mm segment G01 block is executed (kBPM)		
		M850 / M830	M80	
			Type A	Type B
High-speed machining mode I	G05 P1	33.7	33.7	-
High-speed machining mode II	G05 P2	168	67.5	-

Note

(1) The above performance applies under the following conditions.

- 6-axis system (including spindle) or less
- 1-part system
- 3 axes or less commanded simultaneously in G01
- The block containing only the axis name and movement amount (Macro and variable command are not included.)
- In the "G61.1" high-accuracy control mode or cutting mode (G64)
- During tool nose radius compensation cancel (G40) (only in the high-speed machining mode II)
- The parameter "#1259 set31/bit1" is set to "1".

(The number of machining blocks per unit time is set to "low-speed mode".)
When the above conditions are not satisfied, the given feedrate may not be secured.

(2) The performance in the table may vary depending on the combination with other function.

Multi-part system (high-speed machining mode II)

G01 block fine segment capacity for 1mm segment (unit: kBPM)

		Maximum feedrate when 1mm segment G01 block is executed (kBPM)		
		M850 / M830	M80	
			Type A	Type B
1-part system	1 part systems	168	67.5	- (*2)
2-part system	1 part system only	100	67.5	- (*2)
	Two part systems simultaneously	67.5	33.7	- (*2)
4-part system Up to 16 axes	1 part system only	33.7	- (*1)	- (*2)
	Two part systems simultaneously	33.7	- (*1)	- (*2)
5 part systems or more or 17 axes or more	1 part system only	16.8	- (*1)	- (*2)
	Two part systems simultaneously	16.8	- (*1)	- (*2)

(*1) This system cannot be used for this model.

(*2) There are no high-speed machining mode II specifications.

Note

(1) The above performance applies under the following conditions.

- 3 axes commanded simultaneously in G01
- The block containing only the axis name and movement amount (Macro and variable command are not included.)
- Tool nose radius compensation cancel (G40) mode
- The parameter "#1259 set31/bit1" is set to "1".

(The number of machining blocks per unit time is set to "low-speed mode".)

When the above conditions are not satisfied, the given feedrate in the table may not be secured.

(2) The performance in the table may vary depending on the combination with other function.

(3) The number of part systems and axes that can be used depends on the specifications of your machine tool.

Command format

High-speed machining mode I ON

```
G05 P1 ;
```

High-speed machining mode II ON

```
G05 P2 ;
```

High-speed machining mode I/II OFF

```
G05 P0 ;
```

In addition to the G05 P0 command, the high-speed machining mode I is canceled when the high-speed machining mode II (G05 P2) is commanded.
In reverse, the high-speed machining mode II is canceled when the high-speed machining mode I (G05 P1) is commanded.

Command G05 in an independent block. A program error (P33) will occur if a movement or other command is additionally issued in a G05 command block. A program error (P33) will also occur if there is no P command in a G05 command.
In addition to cancel the high-speed machining mode II, a G05 P0 command is also used to cancel the high-speed high-accuracy control II.
Refer to "17.3 High-speed High-accuracy Control" for details.

Detailed description

(1) The override, maximum cutting speed clamp, single block operation, dry run, manual interruption and graphic trace and high-accuracy control mode are valid even during the high-speed machining mode I/II.
For a part system that uses the high-speed machining mode II, "1" must be set for the parameter "#8040 High-SpeedAcc". By default, the high-speed machining mode II can only be used in the first part system.

(2) When using the high-speed machining mode II, setting to eliminate the speed fluctuation at the seams between the arc and the straight line, or between arcs depends on the MTB specifications (parameter "#1572 Cirorp/bit1").

(3) Combination with high-accuracy control
The high-speed machining mode and high-accuracy control can be used simultaneously by taking the following steps:
(a) Set "1" for the parameter "#8040 High-SpeedAcc".
(b) Command "G05 P2" and "G08 P1" or "G61.1" from the machining program.
The parameter "#8040 High-SpeedAcc" can be set to "1" for up to two part systems. If "0" is set for all part systems, the first and second part systems can use the high-speed machining mode and high-accuracy control simultaneously.
Also refer to the following for the description of each function:

- High-accuracy control: "17.2 High-accuracy Control"
- Simultaneous usage of the high-speed machining mode and high-accuracy control: "17.3 High-speed High-accuracy Control"

(4) While high-speed machining mode II is valid, the following variable commands or operation commands can be designated following the axis address. When other variable commands or operation commands are issued, high-speed machining mode II is canceled temporarily.

(a) Referencing common variables or local variables

Common variables or local variables can be referenced (example: X#500, Y#1, Z##100, A#[#101], etc.).

(b) Four basic arithmetic rule

Four basic arithmetic rule (+, -, *, /) operations are available, and also the operation priority can be designated using parentheses ( ) ([#500+1.0]*#501, etc.).

Program example

High-speed machining mode I

G28 X0. Y0. Z0. ;	
G91 G00 X-100. Y-100. ;	
G01 F10000 ;	
G05 P1 ;	High-speed machining mode I ON
:	
X0.1 Y0.01 ;	
X0.1 Y0.02 ;	
X0.1 Y0.03 ;	
:	
G05 P0 ;	High-speed machining mode I OFF
M30 ;	

Note

(1) When using the incremental value command (G91), set the G code type to "3".

Relationship with other functions

Relationship between the high-speed machining mode II and G code functions

Column A: Operation when the additional function is commanded while the high-speed machining mode II is enabled
Column B: Operation when the high-speed machining mode II (G05P2) is commanded while the additional function is enabled
○: The high-speed machining mode II and the additional function are both enabled
Δ: The high-speed machining mode II is temporarily canceled, while the additional function is enabled
x: Alarm generation (the text in parentheses refers to the number of the program error to be generated.)
-: No combination
□: Others

Group	G code type						Additional function	A	B
	2	3	4	5	6	7			
0	G04	G04	G04	G04	G04	G04	Dwell	Δ	-
	G05 P0	G05 P0	G05 P0	G05 P0	G05 P0	G05 P0	High-speed machining mode II OFF High-speed high-accuracy control II OFF	□ (*1)	□ (*2)
	G05 P2	G05 P2	G05 P2	G05 P2	G05 P2	G05 P2	High-speed machining mode II ON	□ (*3)	□ (*3)
	G05 P10000	G05 P10000	G05 P10000	G05 P10000	G05 P10000	G05 P10000	High-speed high-accuracy control II ON	□ (*2)	□ (*2)
	G05.1 Q0	G05.1 Q0	G05.1 Q0	G05.1 Q0	G05.1 Q0	G05.1 Q0	High-speed high-accuracy control I OFF Spline interpolation OFF	□ (*3)	□ (*2)
	G05.1 Q1	G05.1 Q1	G05.1 Q1	G05.1 Q1	G05.1 Q1	G05.1 Q1	High-speed high-accuracy control I ON	□ (*2)	□ (*2)
	G08 P0	G08 P0	G08 P0	G08 P0	G08 P0	G08 P0	High-accuracy control OFF	□ (*3)	□ (*2)
	G08 P1	G08 P1	G08 P1	G08 P1	G08 P1	G08 P1	High-accuracy control ON	□ (*4)	□ (*4)
	G09	G09	G09	G09	G09	G09	Exact stop check	Δ	-
	G10 I_J_ G10 K_	G10 I_J_ G10 K_	G10 I_J_ G10 K_	G10 I_J_ G10 K_	G10 I_J_ G10 K_	G10 I_J_ G10 K_	Parameter coordinate rotation input	Δ	-
	G10 L2	G10 L2	G10 L2	G10 L2	G10 L2	G10 L2	Compensation data input by program	Δ	-
	G10 L70 G10 L50	G10 L70 G10 L50	G10 L70 G10 L50	G10 L70 G10 L50	G10 L70 G10 L50	G10 L70 G10 L50	Parameter input by program	Δ	-
	-	-	-	-	G22	G22	Soft limit ON	○	○
	-	-	-	-	G23	G23	Soft limit OFF	○	○
	G27	G27	G27	G27	G27	G27	Reference position check	Δ	-
	G28	G28	G28	G28	G28	G28	Reference position return	Δ	-
	G29	G29	G29	G29	G29	G29	Start position return	Δ	-

Group	G code type						Additional function	A	B
	2	3	4	5	6	7			
0	G30	G30	G30	G30	G30	G30	2nd to 4th reference position return	Δ	-
	G30.1	G30.1	G30.1	G30.1	G30.1	G30.1	Tool change position return 1	Δ	-
	G30.2 - G30.5	G30.2 - G30.5	G30.2 - G30.5	G30.2 - G30.5	-	-	Tool change position return 2 to 5	Δ	-
	G31	G31	G31	G31	G31	G31	Skip Multiple-step skip 2	Δ	-
	G31.1 - G31.3	G31.1 - G31.3	G31.1 - G31.3	G31.1 - G31.3	G31.1 - G31.3	G31.1 - G31.3	Multi-step skip 1 to 3	Δ	-
	G37	G37	G37	G37	G36 G37 G37.1 G37.2	G36 G37 G37.1 G37.2	Automatic tool length measurement	Δ	-
	G50	G92	G50	G92	G50	G92	Spindle clamp speed setting Coordinate system setting	Δ	-
	-	-	-	-	G50.2 G250	G50.2 G250	Polygon machining mode OFF	○	○
	-	-	-	-	G51.2 G251	G51.2 G251	Polygon machining mode ON	Δ	Δ
	G50.3	G92.1	G50.3	G92.1	G50.3	G92.1	Workpiece coordinate system preset	Δ	-
	G52	G52	G52	G52	G52	G52	Local coordinate system setting	Δ	-
	G53	G53	G53	G53	G53	G53	Machine coordinate system selection	Δ	-
	G65	G65	G65	G65	G65	G65	User macro simple call	□ (*5)	□ (*6)
	G110	G110	G110	G110	G110	G110	Mixed control I (cross control)	○	○
	G111	G111	G111	G111	G111	G111	Axis name switch	○	○
	G113	G113	G113	G113	-	-	Spindle synchronization control OFF	○	○
	G114.1	G114.1	G114.1	G114.1	G114.1	G114.1	Spindle synchronization control ON	Δ	Δ
	G114.2	G114.2	G114.2	G114.2	G114.2	G114.2	Tool spindle synchronization I (polygon machining mode)	Δ	Δ
	G114.3	G114.3	G114.3	G114.3	G114.3	G114.3	Tool spindle synchronization II (hobbing mode)	Δ	Δ
	G115	G115	G115	G115	G115	G115	Start point timing synchronization	Δ	-
	G116	G116	G116	G116	G116	G116			
	G117	G117	G117	G117	G117	G117	M code output during axis traveling	Δ	-
	G122	G122	G122	G122	G122	G122	Sub part system control I	× (P652)	□ (*7)

Group	G code type						Additional function	A	B
	2	3	4	5	6	7			
0	G125	G125	G125	G125	G125	G125	Control axis synchronization between part systems	○	○
	G126	G126	G126	G126	G126	G126	Control Axis Superimposition	Δ	× (P29)
	G140	G140	G140	G140	G140	G140	Arbitrary axis exchange control	× (P34)	○
	G141	G141	G141	G141	G141	G141	Arbitrary axis exchange return	× (P34)	-
	G142	G142	G142	G142	G142	G142	Reference axis arrange return	× (P34)	-
	G144	G144	G144	G144	G144	G144	Sub part system control II	× (P652)	□ (*7)
1	G00	G00	G00	G00	G00	G00	Positioning	Δ	Δ
	G01	G01	G01	G01	G01	G01	Linear interpolation	○	○
	G02 G03	G02 G03	G02 G03	G02 G03	G02 G03	G02 G03	Circular interpolation	○	○
	G02.3 G03.3	G02.3 G03.3	G02.3 G03.3	G02.3 G03.3	G02.3 G03.3	G02.3 G03.3	Exponential interpolation	Δ	Δ
	G32	G33	G32	G33	G32	G33	Thread cutting	Δ	Δ
	G34	G34	G34	G34	G34	G34	Variable lead thread cutting	Δ	Δ
	G35 G36	G35 G36	G35 G36	G35 G36	G35 G36	G35 G36	Circular thread cutting	Δ	Δ
2	G16	G16	G16	G16	-	-	Milling interpolation Plane selection Y-Z cylindrical plane	× (P34)	× (P34)
	G17 G18 G19	G17 G18 G19	G17 G18 G19	G17 G18 G19	G17 G18 G19	G17 G18 G19	Plane selection	○	○
3	G190	G90	G190	G90	G190	G90	Absolute value command	○	○
	G191	G91	G191	G91	G191	G91	Incremental value command	○	○
4	G22	G22	G22	G22	-	-	Barrier check ON	Δ	Δ
	G23	G23	G23	G23	-	-	Barrier check OFF	○	○
5	G98	G94	G98	G94	G98	G94	Asynchronous feed (feed per minute)	○	○
	G99	G95	G99	G95	G99	G95	Synchronous feed (feed per revolution)	Δ	Δ
6	G20	G20	G20	G20	G20	G20	Inch command	○	○
	G21	G21	G21	G21	G21	G21	Metric command	○	○
7	G40	G40	G40	G40	G40	G40	Tool nose radius compensation OFF	○	○
	G41 G42	G41 G42	G41 G42	G41 G42	G41 G42	G41 G42	Tool nose radius compensation ON	○	○
	G46	G46	G46	G46	G46	G46	Tool nose radius compensation ON (automatic direction identification)	○	○

Group	G code type						Additional function	A	B
	2	3	4	5	6	7			
9	G80	G80	G80	G80	G80	G80	Fixed cycle cancel	○	○
	G80 Other than	G80 Other than	G80 Other than	G80 Other than	G80 Other than	G80 Other than	Fixed cycle	Δ	Δ
10	-	G98	-	G98	-	G98	Fixed cycle Initial level return	○	○
	-	G99	-	G99	-	G99	Fixed cycle R point level return	○	○
12	G54 - G59 G54.1	G54 - G59 G54.1	G54 - G59 G54.1	G54 - G59 G54.1	G54 - G59 G54.1	G54 - G59 G54.1	Workpiece coordinate system selection	○	○
13	G61	G61	G61	G61	G61	G61	Exact stop check mode	Δ	Δ
	G61.1	G61.1	G61.1	G61.1	G61.1	G61.1	High-accuracy control	○	○
	G62	G62	G62	G62	G62	G62	Automatic corner override	Δ	Δ
	G63	G63	G63	G63	G63	G63	Tapping mode	Δ	Δ
	G64	G64	G64	G64	G64	G64	Cutting mode	○	○
14	G66 G66.1	G66 G66.1	G66 G66.1	G66 G66.1	G66 G66.1	G66 G66.1	User macro modal call	Δ	Δ
	G67	G67	G67	G67	G67	G67	User macro modal call cancel	○	○
15	G68	G68	G68	G68	-	-	Mirror image for facing tool posts ON	× (P29)	× (P29)
	G69	G69	G69	G69	-	-	Mirror image for facing tool posts OFF	○	○
	-	-	-	-	G68	G68	Mirror image for facing tool posts ON	× (P29)	× (P29)
							Balance cut ON	× (P29)	× (P29)
	-	-	-	-	G69	G69	Mirror image for facing tool posts OFF	○	○
							Balance cut OFF	○	○
16	G68.1	G68.1	G68.1	G68.1	G68.1	G68.1	Coordinate rotation by program ON	Δ	Δ
	G69.1	G69.1	G69.1	G69.1	G69.1	G69.1	Coordinate rotation by program OFF	○	○
17	G96	G96	G96	G96	G96	G96	Constant surface speed control ON	○	○
	G97	G97	G97	G97	G97	G97	Constant surface speed control OFF	○	○
18	G14	G14	G14	G14	-	-	Balance cut OFF	○	○
	G15	G15	G15	G15	-	-	Balance cut ON	× (P29)	× (P29)

Group	G code type						Additional function	A	B
	2	3	4	5	6	7			
19	G12.1	G12.1	G12.1	G12.1	-	-	Milling interpolation ON	× (P34)	× (P481)
	G13.1	G13.1	G13.1	G13.1	-	-	Milling interpolation OFF	○	○
	-	-	-	-	G07.1 G107	G07.1 G107	Cylindrical interpolation	× (P34)	× (P481)
	-	-	-	-	G12.1 G112	G12.1 G112	Polar coordinate interpolation ON	× (P34)	× (P481)
	-	-	-	-	G13.1 G113	G13.1 G113	Polar coordinate interpolation OFF	○	○
20	G43.1	G43.1	G43.1	G43.1	G43.1	G43.1	1st spindle control mode	○	○
	G44.1	G44.1	G44.1	G44.1	G44.1	G44.1	2nd spindle control mode	○	○
	G47.1	G47.1	G47.1	G47.1	G47.1	G47.1	Two spindles simultaneous control mode	○	○
24	G188	G188	G188	G188	G188	G188	Dynamic M/L program changeover ON	○	○
	G189	G189	G189	G189	G189	G189	Dynamic M/L program changeover OFF	○	○

(*1) Disables the high-speed machining mode II.

(*2) Enables the high-speed machining mode II.

(*3) High-speed machining mode II continues.

(*4) Enables the high-speed machining mode II and high-accuracy control.

(*5) Enables the high-speed machining mode II in a macro program.

(*6) Enables the high-speed machining mode II if G05P2 is commanded in a macro program.

(*7) Enables the high-speed machining mode II if G05P2 is commanded in a sub part system.

Relationship between the high-speed machining mode II and functions other than G codes

Column A: Operation when the additional function is commanded while the high-speed machining mode II is enabled
Column B: Operation when the high-speed machining mode II (G05P2) is commanded while the additional function is enabled
○: The high-speed machining mode II and the additional function are both enabled
Δ: The high-speed machining mode II is temporarily canceled, while the additional function is enabled
x: Alarm generation (the text in parentheses refers to the number of the program error to be generated.)
-: No combination
□: Others

Additional function	A	B
SSS ON	-	○
Mirror image by parameter setting ON	-	Δ
PLC mirror image ON	-	Δ
Subprogram call (M98)	□ (*8)	□ (*9)
Timing synchronization between part systems	□ (*10)	-
Inclined axis control	-	○
T code offset	○	○
T code mirror image for facing tool posts	× (P29)	× (P29)
MTB macro	□ (*11)	□ (*12)
Macro interruption	□ (*13)	□ (*14)
Corner chamfering/Corner R	Δ	-
Linear angle command	Δ	-
Geometric command	Δ	-
Chopping	○	○
Optional block skip	○	-

(*8) Enables the high-speed machining mode II in a subprogram.

(*9) Enables the high-speed machining mode II if G05P2 is commanded in a subprogram.

(*10) Enables timing synchronization.

(*11) Enables the high-speed machining mode II in a MTB program.

(*12) Enables the high-speed machining mode II if G05P2 is commanded in a MTB program.

(*13) Enables the high-speed machining mode II in an interrupt program.

(*14) Enables the high-speed machining mode II if G05P2 is commanded in an interrupt program.

Precautions

(1) If "G05 P1(P2)" is commanded when the high-speed machining mode I/(II) specifications are not provided, a program error (P39) will occur.

(2) The automatic operation process has priority in high-speed machining mode I/II, and as a result, the screen display may slow down.

(3) The speed will decelerate once at the G05 command block, so turn ON and OFF when the tool separates from the workpiece.

(4) When carrying out operations in high-speed machining mode I/II by communication or tape mode, the machining speed may be suppressed depending on the program transmission speed limit.

(5) Command G05 in an independent block.

(6) A decimal point is invalid for the P address in the G05 command block.

(7) The P addresses, which are valid in the G05 command block, are P0, P1 and P2 only.
If other P addresses are commanded, a program error (P35) will occur.
If there is no P command, a program error (P33) will occur.

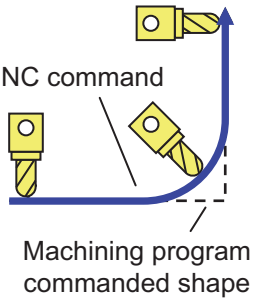
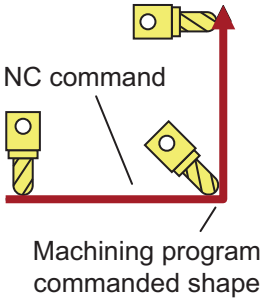
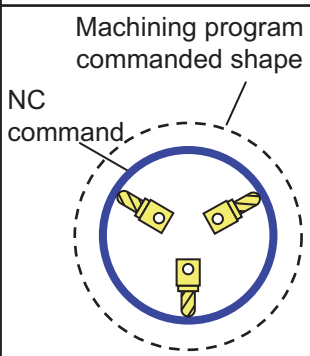
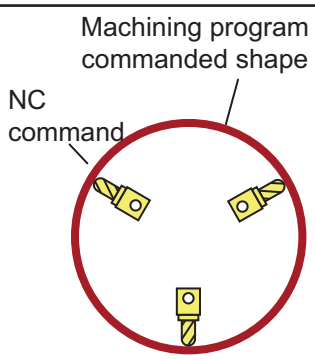
(8) The machining speed may be suppressed depending on the number of characters in one block.

17.2 High-accuracy Control

17.2.1 High-accuracy Control ; G61.1, G08

Function and purpose

Machining errors caused by delays in control systems can be inhibited. This function is useful for machining which needs to make an edge at a corner or reduce an error from an inner route of curved shape. In high-accuracy control, acceleration/deceleration is performed not to cause machining error by pre-reading blocks and acceleration/deceleration is automatically performed according to a machining shape so that the machining error is inhibited with minimizing an extension of machining time.

	High-accuracy control OFF	High-accuracy control ON
Corner shape		
Curve shape		

Commands to enable high-accuracy control are as follows:

- High-accuracy control command (G08P1/G61.1)
- High-speed high-accuracy control I command (G05.1Q1)
- High-speed high-accuracy control II command (G05P10000)

This function uses the following functions to minimize the increase in machining time while reducing the shape error.

(1) Acceleration/deceleration before interpolation

(2) Optimum speed control

(3) Vector accuracy interpolation

(4) Feed forward

(5) S-pattern filter control

In the main text, the axis address refers to the address of an axis that exits on the machine.
It corresponds to the address designated in the parameters "#1013 axname" and "#1014 incax".
These parameter settings depend on the MTB specifications.

Command format

High-accuracy control valid

```
G61.1 ;
or, G08 P1;
```

High-accuracy control invalid

```
G08 P0 ;
or, G command in G code group 13 except G61.1
```

High-accuracy control can be canceled with either command regardless of the command that has enabled the control.

Note

(1) After "G08 P1" is commanded, G code group 13 is automatically switched to the G61.1 modal.
If the high-accuracy control mode is canceled by the "G08 P0" command, G code group 0 is switched to the "G08P0" modal and G code group 13 becomes the "commanded mode".

Detailed description

(1) Feedrate command F is clamped with the "#2110 Clamp (H-precision)" (Cutting feed clamp speed for high-accuracy control mode) set with parameter.

(2) Rapid traverse rate enables "#2109 Rapid(H-precision)" (Rapid traverse rate during high-accuracy control mode) set by the parameter.

(3) When the "#2109 Rapid(H-precision)" is set to "0", the movement follows "#2001 rapid" (rapid traverse rate) set by the parameter. Also, when "#2110 Clamp (H-precision)" is set to "0", the speed will be clamped with "#2002 clamp" (Cutting clamp speed) set with parameter.

(4) The modal holding state of the high-accuracy control mode depends on the MTB specifications (combination of the parameters "#1151 rstint" (reset initial) and "#1148 I_G611" (initial high-accuracy)).

Parameter		Default state	Resetting		
Reset initial (#1151)	**Initial high-accuracy (#1148)**	**Power ON**	**Reset 1**	**Reset 2**	**Reset & rewind**
OFF	**OFF**	OFF	Hold	OFF	
ON			OFF		
OFF	**ON**	ON	Hold	ON	
ON			ON		

Parameter		Emergency stop	Emergency stop cancel
Reset initial (#1151)	**Initial high-accuracy (#1148)**	**Emergency stop switch or external emergency stop**	**Emergency stop switch or external emergency stop**
OFF	**OFF**	Hold	Hold
ON			OFF
OFF	**ON**	Hold	Hold
ON			ON

Parameter		Block interruption	Block stop	NC alarm	OT
Reset initial (#1151)	**Initial high-accuracy (#1148)**	Mode changeover (automatic/manual) or feed hold	Single block	Servo alarm	H/W OT
OFF	**OFF**	Hold			
ON					
OFF	**ON**				
ON					

Hold: Modal hold

ON: Switches to the high-accuracy control mode

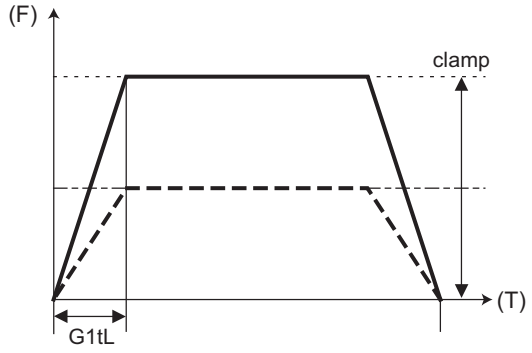
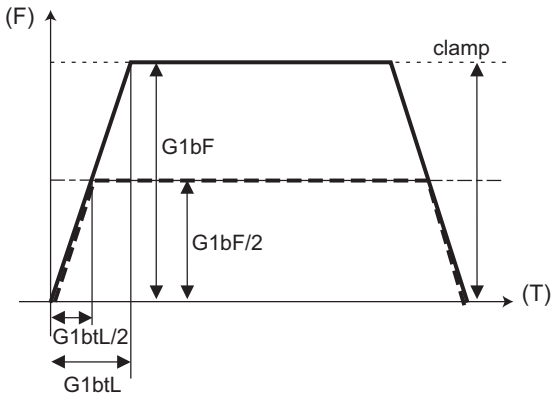
As for G61.1, the mode is switched to the high-accuracy mode, even if the other modes (G61 to G64) are valid.

OFF: The status of the high-accuracy control mode is OFF.

Acceleration/deceleration before interpolation

Acceleration/deceleration control is carried out for the movement commands to suppress the impact and to smooth out the velocity waveform when the machine starts or stops moving. However, if high-accuracy control is disabled, the corners at the block seams are rounded, and path errors occur regarding the command shape because acceleration/deceleration is performed after interpolation.
In the high-accuracy control function mode, acceleration/deceleration is carried out before interpolation to solve the above problems. This acceleration/deceleration before interpolation enables machining with a faithful path to the commanded shape of the machining program.
Furthermore, the acceleration/deceleration time can be reduced because the constant inclination acceleration/deceleration is performed for the acceleration/deceleration before interpolation.

(1) Basic patterns of acceleration/deceleration control in linear interpolation commands

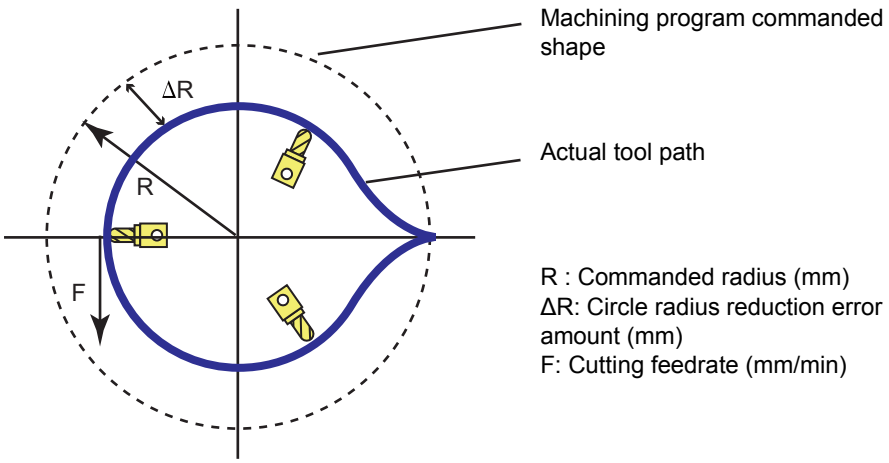
	Acceleration/deceleration waveform pattern	
Normal mode		(a) Because of the acceleration/deceleration that controls the acceleration/deceleration time to achieve the commanded speed at a constant level (constant time constant acceleration/deceleration), the acceleration/deceleration becomes more gentle as the command speed becomes slower (the acceleration/deceleration time does not change). (b) The time to achieve the commanded speed (G1tL) can be set independently for each axis. Note, however, that an arc shape will be distorted if the time constant differs among the base axes. G1tL: G1 time constant (linear) (MTB-specified parameter #2007)
High-accuracy control mode	 (F) Combined speed (T) Time	(a) Because of the acceleration/deceleration that controls the acceleration/deceleration time to achieve the maximum speed (G1bF) set by a parameter at a constant level (constant inclination type linear acceleration/deceleration), the acceleration/deceleration time is reduced as the command speed becomes slower. (b) Only one acceleration/deceleration time constant (common for each axis) exists in a system. G1bF: Maximum speed (MTB-specified parameter #1206) G1btL: Time constant (MTB-specified parameter #1207) <Note> (1) G1bF and G1btL are values for specifying the inclination of the acceleration/deceleration time. The actual cutting feed maximum speed is clamped by the "#2002 clamp" value.

(2) Path control in circular interpolation commands

When commanding circular interpolation with the conventional post-interpolation acceleration/deceleration control method, the path itself that is output from the NC to the servo runs further inside the commanded path, and the circle radius becomes smaller than that of the commanded circle. This is due to the influence of the smoothing course droop amount for NC internal acceleration/deceleration.

With the pre-interpolation acceleration/deceleration control method, the path error is eliminated and a circular path faithful to the command results, because interpolation is carried out after the acceleration/deceleration control. Note that the tracking lag due to the position loop control in the servo system is not the target here.

The following shows a comparison of the circle radius reduction error amounts for the conventional post-interpolation acceleration/deceleration control and pre-interpolation acceleration/deceleration control in the high-accuracy control mode.



If an arc is commanded by a machining program as shown above, the error ΔR occurs for the commanded shape on the actual tool path. In the normal mode (acceleration/deceleration after interpolation), ΔR is caused by acceleration/deceleration of NC and lag of servo system. High-accuracy control (acceleration/deceleration before interpolation), however, can eliminate errors caused by acceleration/deceleration of NC. By additionally using the feed forward control, it is also possible to reduce errors caused by lag of servo system.

The compensation amount of the circle radius reduction error (ΔR) is theoretically calculated as shown in the following table.

Post-interpolation acceleration/deceleration control (normal mode)	Pre-interpolation acceleration/deceleration control (high-accuracy control mode)
Linear acceleration/deceleration $\Delta R = \frac{1}{2R}\left(\frac{1}{12}Ts^2 + Tp^2\right)\left(\frac{F}{60}\right)^2$ Exponential function acceleration/deceleration $\Delta R = \frac{1}{2R}\left(Ts^2 + Tp^2\right)\left(\frac{F}{60}\right)^2$	Linear acceleration/deceleration $\Delta R = \frac{1}{2R}\left\{Tp^2\left(1 - Kf^2\right)\right\}\left(\frac{F}{60}\right)^2$ (a) Because the item Ts can be ignored by using the pre-interpolation acceleration/deceleration control method, the radius reduction error amount can be reduced. (b) Item Tp can be negated by making Kf = 1.

Ts: Acceleration/deceleration time constant in the NC (s)
Tp: Servo system position loop time constant (s) (inverse number to "#2203 PGN1")
Kf: Feed forward coefficient
Kf = fwd_g / 1000 (fwd_g: #2010 Feed forward gain)

Optimum speed control

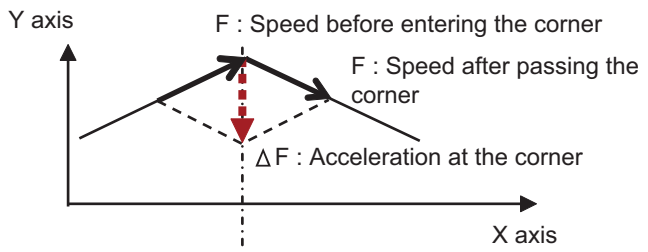
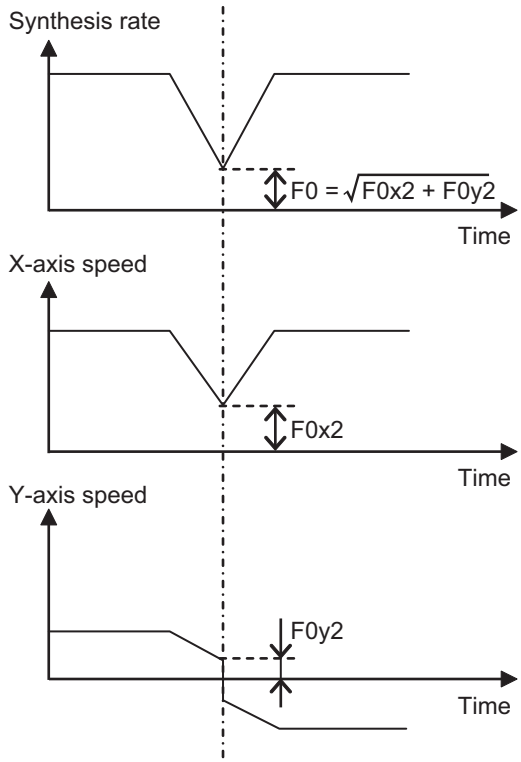
When the moving direction is changed on the corner, arc, etc., acceleration corresponding to the amount of change and the feedrate is generated. When the acceleration is large, there is a possibility of machine vibration and it may leave stripes on the machining surface.
In the high-accuracy control mode, the deceleration control (optimum speed control) is performed to keep the generated acceleration under the allowance that has been designed with the parameter so that the problem mentioned above can be solved. The optimum speed control suppresses the machine vibration and enables highly accurate machining while minimizing the extension of cycle time.

Corner deceleration
: Consists of optimum corner deceleration and tolerable acceleration control for each axis.

Arc speed clamp
: Controls deceleration so that the combined acceleration on an arc is kept below the tolerable acceleration common to all axes. This can suppress path errors (circle radius reduction error amount) on an arc to a certain level.

(1) Optimum corner deceleration

Highly accurate edge machining can be achieved by controlling deceleration so that the combined acceleration at the seam between blocks is kept under the tolerable acceleration common to all axes, which is determined by "#1206 G1bF (maximum speed)", "#1207 G1btL (time constant)", and accuracy coefficient. When entering in a corner, optimum speed for the corner (optimum corner speed) is calculated from the angle with the next block (corner angle) and the tolerable acceleration common to all axes. The machine decelerates to the speed in advance, and then accelerates back to the command speed after passing the corner.

	When a corner with the corner angle θ is passed at speed F, the acceleration ΔF occurs according to θ and F.
	The corner speed F is controlled so that ΔF generated above does not exceed the tolerable acceleration common to all axes. The speed pattern is as shown on the left.

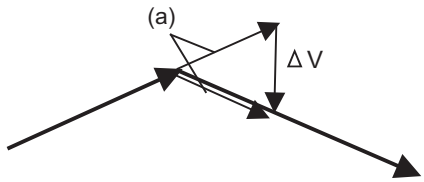
Optimum corner deceleration is not carried out when blocks are smoothly connected, because deceleration is not necessary. The criteria for whether the connection is smooth or not can be designated by the machining parameter "#8020 DCC ANGLE". If the corner angle is equal to or less than the corner deceleration angle, the connection is judged to be smooth and optimum corner deceleration is not carried out.
The edge accuracy can be further improved by setting a greater accuracy coefficient. A greater accuracy coefficient, however, reduces the optimum corner speed, which may increase the cycle time. Setting a negative accuracy coefficient can increase the optimum corner speed and reduce the cycle time.
As shown below, different accuracy coefficients can be used depending on the parameter "#8021 COMP_CHANGE", and the tolerable acceleration common to all axes can be obtained with the following formula:

#8021 COMP CHANGE	Accuracy coefficient used
0	#8019 R COMP
1	#8022 CORNER COMP

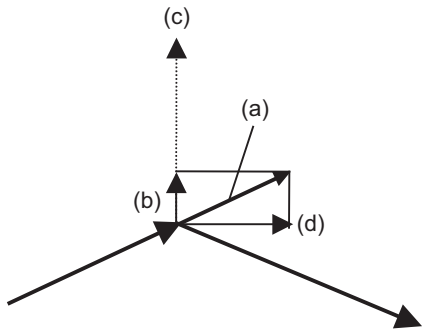
$$\text{Tolerable acceleration for all axes (mm/s}^2\text{)} = \frac{\text{G1bF(mm/min)}}{\text{G1btL(ms)}} * 60 * 1000 * \frac{100 - \text{R COMP}}{100}$$

The corner speed V0 can be maintained at more than a certain speed so that the corner speed does not drop too far.
Set "#2096 crncsp (corner deceleration minimum speed)" for each axis, and make a combined speed so that the moving axis does not exceed this setting.

Speed is not clamped



Speed is clamped



(a) Corner deceleration speed
(b) Clamp value according to X axis
(c) Y axis setting value
(d) X axis setting value

Note that the speed is controlled with the optimum corner deceleration speed in the following cases.

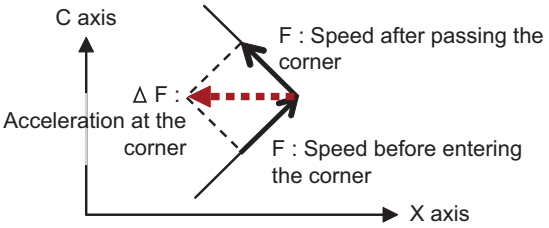
- When the combined corner deceleration speed is equal to or less than the optimum corner deceleration speed
- When the corner deceleration minimum speed parameter setting for the moving axes is set to "0" for even one axis.

(2) Tolerable acceleration control for each axis (optimum acceleration control)

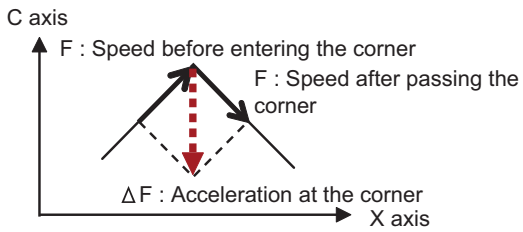
The acceleration to be generated at a seem between blocks is evaluated for each axis to control deceleration so that the seam is passed at the optimum speed. This enables highly accurate edge machining.

The optimum deceleration speed is calculated so that the acceleration of each axis to be generated at the seam is equal to or less than the tolerable acceleration for each axis, which is determined by "#2157 G1bFx" (maximum speed for each axis), "#2158 G1btLx" (time constant for each axis), and the accuracy coefficient. The machine decelerates to the speed in advance, and then accelerates back to the command speed after passing the corner. This control enables deceleration at an appropriate speed for the characteristics of each axis even when machine vibrations may easily occur due to a low tolerable acceleration for a specific axis (rotary axis). This means that the deceleration speed can be raised at a corner where acceleration is generated only for an axis with a high tolerable acceleration, leading to a reduced cycle time.

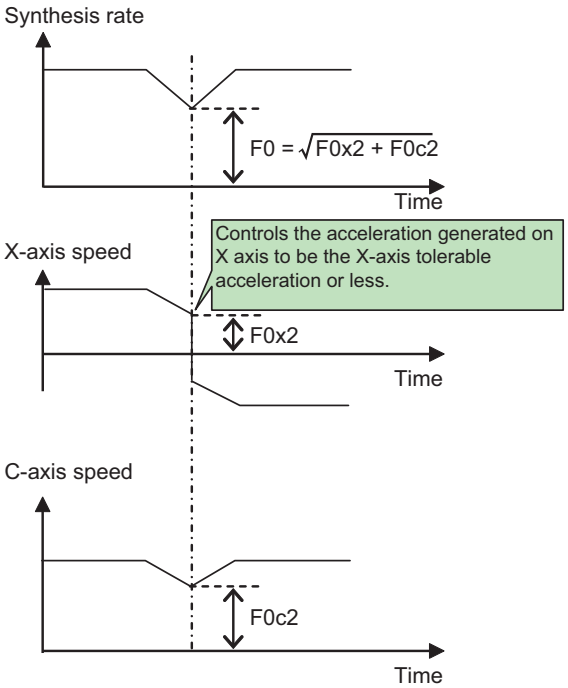
If acceleration is generated for the X axis (linear axis) as shown in Figure (a) below or for the C axis (rotary axis) as shown in Figure (b), the corner speed F is controlled so that the acceleration to be generated at the X or C axis does not exceed the tolerable acceleration for the X or C axis, respectively. If the tolerable acceleration for the X axis is higher than that for the C axis, a higher deceleration speed can be used for a path where acceleration is generated only for the X axis than where acceleration is generated only for the C axis. In this case, the speed patterns are as shown in Figures (c) and (d) below:



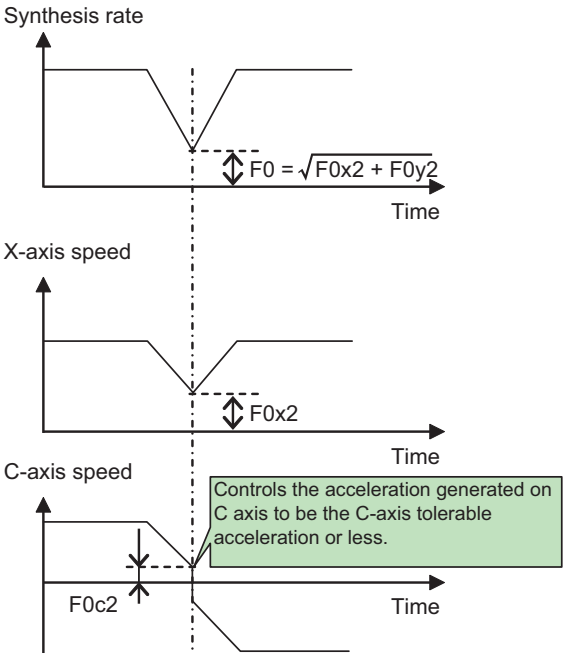
(a) Corner shape which generates the acceleration on X axis (linear axis)



(b) Corner shape which generates the acceleration on C axis (rotary axis)



(c) Speed pattern which generates the acceleration on X axis (linear axis)



(d) Speed pattern which generates the acceleration on C axis (rotary axis)

Deceleration is not carried out when blocks are smoothly connected (when the acceleration to be generated for each axis is equal to or lower than the tolerable acceleration for each axis).
The edge accuracy can be further improved by setting a greater accuracy coefficient. A greater accuracy coefficient, however, reduces the optimum corner speed, which may increase the cycle time. Setting a negative accuracy coefficient can increase the optimum corner speed and reduce the cycle time.
As shown below, different accuracy coefficients can be used depending on the parameter "#8021 COMP_CHANGE". Also, the tolerable acceleration can be adjusted for each axis using "#2159 compx" (accuracy coefficient for each axis), and the tolerable acceleration for each axis can be obtained with the following formula. It is necessary, however, to set the same tolerable acceleration for all base axes because an arc shape is distorted if it differs among them. If G1bFx is 0 (not set), the tolerable acceleration is calculated using "#2001 rapid" (rapid traverse rate). And if G1btLx is 0 (not set), the tolerable acceleration is calculated using "#2004 G0tL" (G0 time constant (linear)).
If G1bFx and G1btLx are 0 for all base axes, the tolerable accelerations for the base axes are unified to the lowest one.

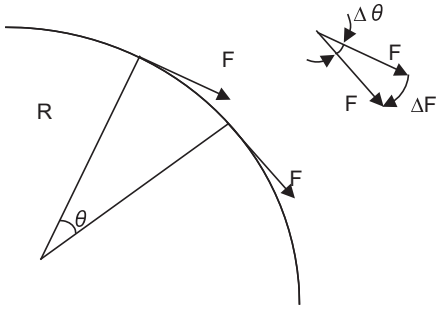
#8021 COMP CHANGE	Accuracy coefficient used
0	#8019 R COMP
1	#8022 CORNER COMP

$$\text{Tolerable acceleration for each axes (mm/s}^2\text{)} = \frac{\text{G1bFx(mm/min)}}{\text{G1btLx(ms)}} * 60 * 1000 * \frac{100 - \text{R COMP}}{100} * \frac{100 - \text{compx}}{100}$$

(3) Arc speed clamp

During circular interpolation, even when moving at a constant speed, acceleration is generated as the advance direction constantly changes. When the arc radius is large enough in relation to the commanded speed, control is carried out at the commanded speed. However, when the arc radius is relatively small, the speed is clamped so that the generated acceleration does not exceed the tolerable acceleration/deceleration speed before interpolation, calculated with the parameters.

This allows arc cutting to be carried out at an optimum speed for the arc radius.
The figure below shows the acceleration ΔF (mm/s²) for movement at the constant speed F (mm/min) on an arc shape with the radius R (mm). Here, the arc clamp speed F' (mm/min) that makes the acceleration ΔF lower than the tolerable acceleration common to all axes Ac (mm/s²) can be obtained with the following formula:



F : Commanded speed (mm/min)
R : Commanded arc radius (mm)
Δθ : Angle change per interpolation unit
ΔF : Speed change per interpolation unit

The tool is fed with the arc clamp speed F' so that ΔF does not exceed the tolerable acceleration common to all axes Ac (mm/s²).

$$F' \leqq \sqrt{R*Ac*60}$$

$$\Delta F' = \frac{G1bF(mm/min)}{G1btL(ms)}$$

When the above F' expression is substituted with F in the expression for the maximum logical arc radius reduction error amount ΔR, explained in the section "Pre-interpolation acceleration/deceleration", the commanded radius R is eliminated, and ΔR does not rely on R.
Here, Tp is the servo system position loop time constant (s) and Kf is the feed forward coefficient.
Tp is the inverse number to "#2203 PGN1" (position loop gain) (Tp = 1 / PGN1) and Kf is a ratio of "#2010 fws_g" (feed forward gain) (Kf = fwd_g / 100), both of which depend on the MTB specifications.

$$\Delta R = \frac{1}{2R}\left\{Tp^2\left(1 - Kf^2\right)\right\}\left(\frac{F}{60}\right)^2$$

$$= \frac{AC}{2}\left\{Tp^2\left(1 - Kf^2\right)\right\}$$

ΔR : Arc radius reduction error amount
Tp : Position loop gain time constant of servo system
Kf : Feed forward coefficient
F : Cutting feedrate

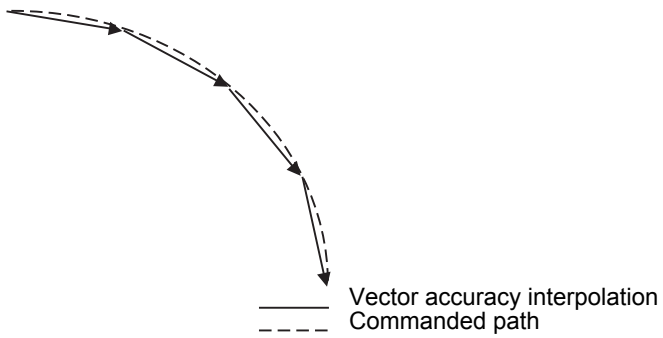
In other words, with an arc command to be clamped at the arc clamp speed, in logical terms regardless of the commanded radius R, machining can be carried out with a radius reduction error amount within a constant value. The roundness can be further improved by setting a greater accuracy coefficient. A greater accuracy coefficient, however, reduces the arc clamp speed, which may increase the cycle time. Setting a negative accuracy coefficient can increase the arc clamp speed and reduce the cycle time.
As shown below, different accuracy coefficients can be used depending on the parameter "#8021 COMP_CHANGE", and the tolerable acceleration common to all axes can be obtained with the following formula:

#8021 COMP CHANGE	Accuracy coefficient used
0	#8019 R COMP
1	#8023 CURVE COMP

$$\text{Tolerable acceleration for all axes (mm/s}^2\text{)} = \frac{G1bF(mm/min)}{G1btL(ms)} * 60 * 1000 * \frac{100 - R\ COMP}{100}$$

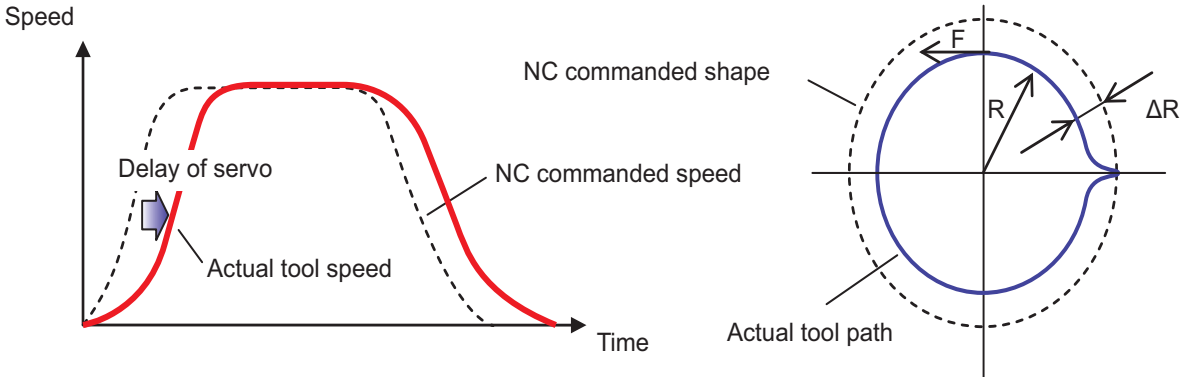
Vector accuracy interpolation

When a fine segment is commanded and the angle between the blocks is extremely small (when not using optimum corner deceleration), interpolation can be carried out more smoothly using the vector accuracy interpolation.

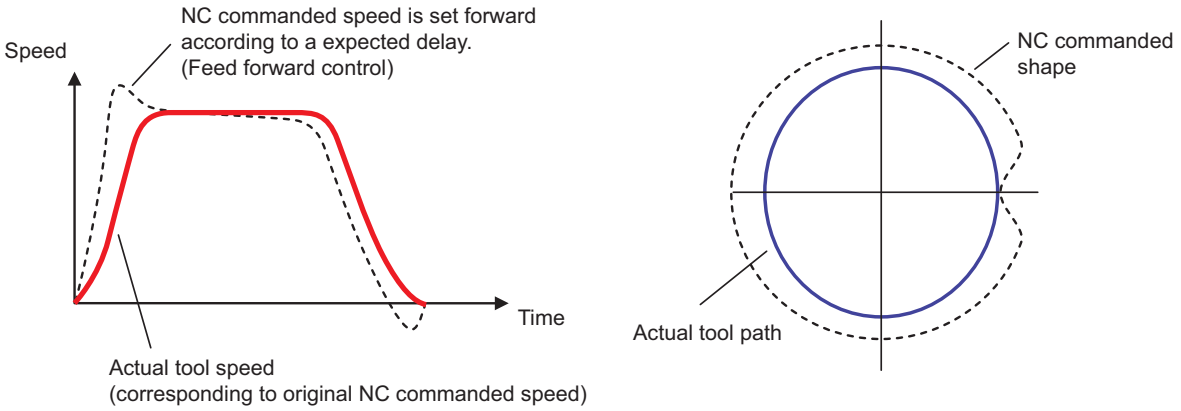


Feed forward control

This function reduces path errors caused by delay of servo systems. Path errors caused by acceleration/deceleration of NC can be eliminated by acceleration/deceleration before interpolation, however errors caused by delay of servo systems cannot be eliminated by acceleration/deceleration before interpolation. Therefore, when the arc shape of radius R (mm) is machined at speed F (mm/min) as the figure (a)below, for instance, the lag time occurs between the NC commanded speed and the actual tool speed in amount of the servo system time constant and the path error ΔR (mm) occurs. Feed forward control generates the command value taking the delay of servo systems as shown in figure (b)below so that the path error caused by delay of servo systems can be inhibited.



(a) NC command and actual tool movement during Feed forward control OFF



(b) NC command and actual tool movement during Feed forward control ON

$$\Delta R = \frac{1}{2R}\left\{Tp^2\left(1 - Kf^2\right)\right\}\left(\frac{F}{60}\right)^2$$

Here, Tp is the servo system position loop time constant (s) and Kf is the feed forward coefficient. Tp is the inverse number to "#2203 PGN1" (position loop gain) (Tp = 1 / PGN1) and Kf is a ratio of "#2010 fws_g" (feed forward gain) (Kf = fwd_g / 100), both of which depend on the MTB specifications.

Combination with the smooth high gain (SHG) control function

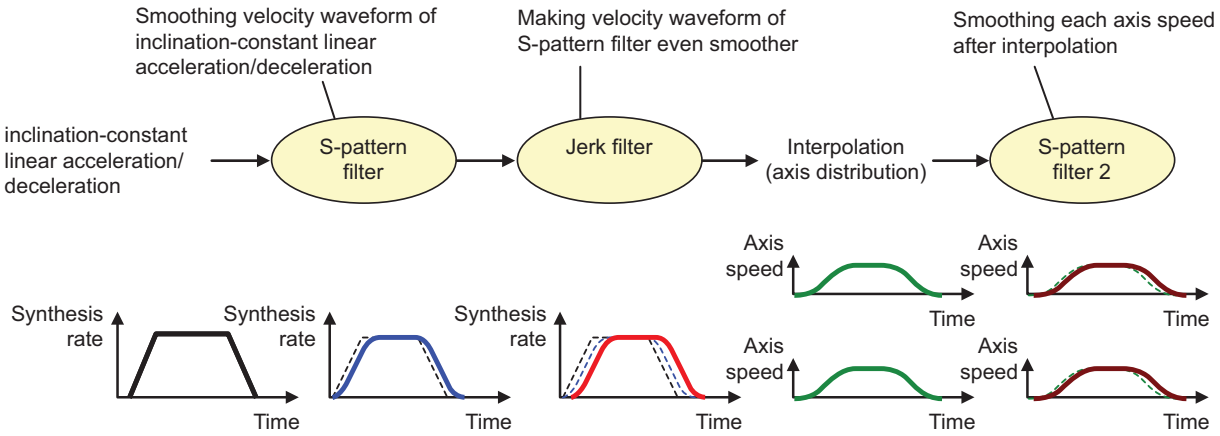
Feed forward control can inhibit path errors more effectively by increasing the feed forward coefficient. In some cases, however, the coefficient cannot be increased because a greater coefficient may cause machine vibrations. In this case, use this function together with the smooth high gain (SHG) control function to stably compensate path errors caused by lag of servo system.
To enable the SHG control, it is also necessary to set "#2204 PGN2" (position loop gain 2) and "#2257 SHGC SHG" (control gain) in addition to "#2203 PGN1" (position loop gain 1), all of which depend on the MTB specifications. By enabling the SHG control, it is possible to inhibit path errors, for example, for an arc shape equivalently as with conventional control (SHG control OFF) using the equivalent feed forward gain fwd_g as shown in the following formula. This means that setting fwd_g = 50 (%) for the SHG control is as effective as setting fwd_g = 100 (%) for conventional control in inhibiting path errors.

$$fwd\_g' = 100\sqrt{1 - \left\{1 - \left(\frac{fwd\_g}{100}\right)^2\right\}\left(\frac{1}{2}\right)}$$

S-pattern filter control

S-pattern filter (soft acceleration/deceleration filter) is the function that inhibits the machine vibration by smoothing a velocity waveform. There are following types of S-pattern filters:

- G01/G00 S-pattern filter
- G01/G00 jerk filter
- S-pattern filter 2



(1) G01/G00 S-pattern filter

This function inhibits the machine vibration by smoothing a velocity waveform generated by inclination-constant linear acceleration/deceleration.

Inclination-constant linear acceleration/deceleration generates continuous velocity waveforms, but makes the acceleration discontinuous. As a result, machine vibrations may easily occur when there are discontinuities in acceleration, which may cause scratches or streaks on the machining surface. The S-pattern filter can make the velocity waveform even smoother and eliminate acceleration discontinuities to inhibit machine vibrations. The S-pattern filter does not impair machining accuracy because it makes the combined speed smoother before interpolation. A greater S-pattern filter time constant, however, may increases the cycle time.

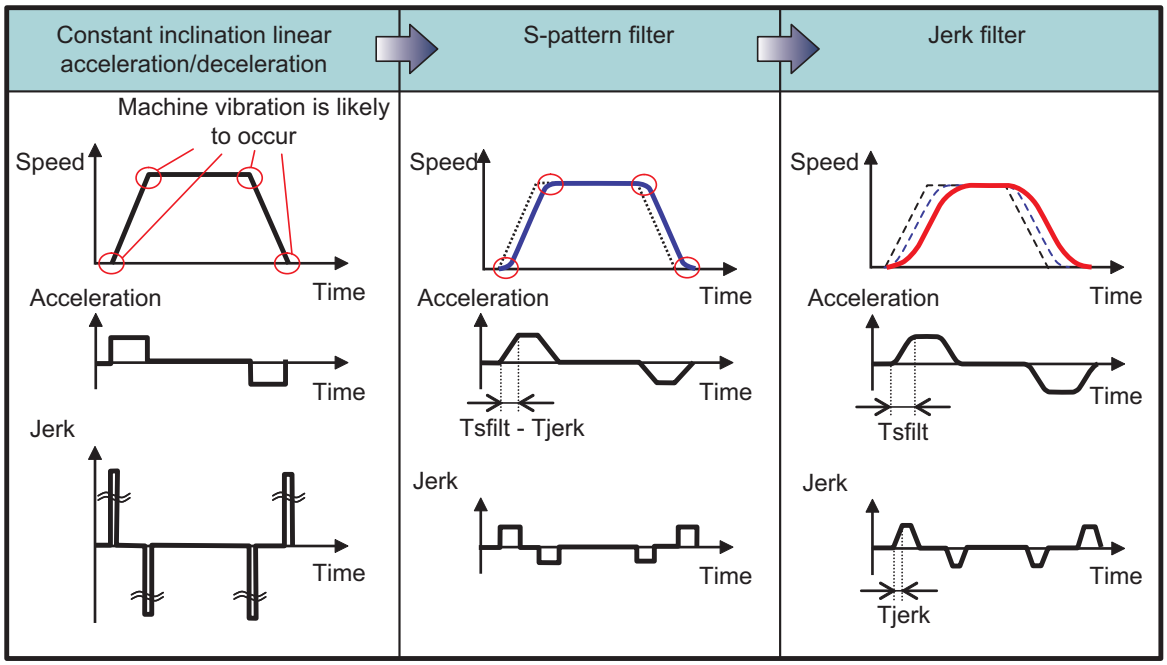
To the S-pattern filter time constant, "#1568 SfiltG1" is applied during cutting feed (G01) or "#1569 SfiltG0" during rapid traverse (G00), each of which can be set in the range of 0 to 200 (ms).

(2) G01/G00 jerk filter

The jerk filter function inhibits machine vibrations by eliminating jerk discontinuities when the S-pattern filter alone cannot inhibit such vibrations.

Through the S-pattern filter, continuous velocity waveforms can be obtained up to acceleration, but jerk discontinuities remain. The jerk filter further filters the velocity waveform smoothed by the S-pattern filter to smooth jerk as well to inhibit machine vibrations. The jerk filter does not impair machining accuracy because it makes the combined speed smoother before interpolation.

To the jerk filter time constant, "#12051 Jerk_filtG1" is applied during cutting feed (G01) or "#12052 Jerk_filtG0" during rapid traverse (G00), each of which can be set in the range of 0 to 50 (ms). Even if a jerk filter time constant is set, the S-pattern filter time constant is the time to achieve the target acceleration. As a result, the time constant for S-pattern filter processing is "S-pattern filter time constant" - "Jerk filter time constant". If the jerk filter time constant is greater than the S-pattern filter time constant, an MCP alarm (Y51 0030) will occur.



Tsfilt: S-pattern filter time constant
Tjerk: Jerk filter time constant

(3) S-pattern filter 2

This function inhibits machine vibrations by smoothing slight speed fluctuation caused when the combined speed is distributed to each axis element.

S-pattern filter 2 can inhibit machine vibrations by smoothing slight speed fluctuation on each axis. The function, however, may impair machining accuracy because it filters each axis speed after interpolation. A greater S-pattern filter 2 time constant, however, may increases the cycle time.

To the S-pattern filter 2 time constant, "#1570 Sfilt2" is applied, which can be set in the range of 0 to 200 (ms).

(4) How to adjust parameters

(a) The table below shows typical initial values for each filter time constant. If your machine's natural angular frequency fn (Hz) is known, vibrations can be inhibited effectively by setting the vibration period Tn (ms) obtained with the following formula for the S-pattern filter time constant:

$$Tn = \frac{1000}{fn} \text{ (ms)}$$

S-pattern filter (SfiltG1/SfiltG0)	Jerk filter (Jerk_filtG1/Jerk_filtG0)	S-pattern filter (Sfilt2)
50ms	0ms	10ms

(b) If vibrations cannot be inhibited properly with the above initial values, increase the S-pattern filter time constant. Or, decrease the S-pattern filter time constant to reduce the cycle time.

(c) If vibrations occur at a corner or other section and stripes remain on the machining surface even after the S-pattern filter time constant is increased, increase the S-pattern filter 2 time constant. The maximum S-pattern filter 2 time constant, however, should be 20 to 25 ms because a greater S-pattern filter 2 time constant may impair machining accuracy.

(d) If high-frequency machine vibrations remain even after the S-pattern filter/S-pattern filter 2 are applied, set the jerk filter time constant.

If a shorter cycle time has a priority over the machining accuracy, it is possible to inhibit vibrations at a corner by reducing the corner accuracy coefficient to increase the corner deceleration speed and increasing the S-pattern filter 2 time constant.

Relationship with other functions

(1) The modal must be set as shown below when commanding G08 P1/G61.1.

Function	G code
Cylindrical interpolation cancel (*1)	G07.1
Polar coordinate interpolation cancel (*1)	G15
Tool radius compensation mode cancel	G40
Tool length compensation cancel	G49
Mirror image with settings	Cancel
Mirror image with signals	Cancel
No macro modal call	G67
Feed per revolution cancel	G94
Constant surface speed control mode cancel	G97
Interruption type macro mode cancel	M97

(*1) These functions can be commanded if the tolerable acceleration control for each axis (optimum acceleration control) specifications are valid.

(2) A program error will occur if high-accuracy control is commanded in the following modes.
- During milling -> Program error (P481)
- During cylindrical interpolation -> Program error (P481) (*2)
- During polar coordinate interpolation -> Program error (P481) (*2)

(3) A program error (P29) will occur if the following commands are issued during the high-accuracy control mode.
- Milling
- Cylindrical interpolation (*2)
- Polar coordinate interpolation (*2)

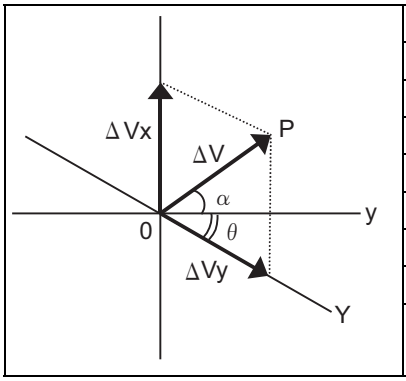
(*2) An error will not occur if the specifications of the tolerable acceleration control for each axis (optimum acceleration control) are valid.

(4) High-accuracy control is temporarily disabled in the following modals:
- During a thread cutting cycle
- Tool spindle synchronization IC (spindle-NC axis polygon machining)
- Tool spindle synchronization II (hobbing)
- Balance cut

Inclined axis control

The acceleration to be used in high-accuracy control is determined by the values set for parameters "#1206 G1bF" and "#1207 G1btL" (depending on the MTB specifications). However, this acceleration is based on the program coordinates (orthogonal coordinates: X-y coordinates). This means that, if inclined axis control is enabled, the acceleration of the actual axes (base/inclined axes: X-Y axes) exceeds the acceleration determined by #1206 and #1207. Therefore, #1206 and #1207 must be set by taking the mechanically allowable acceleration of the actual axes into consideration.

The following shows the relationship between acceleration (ΔV) in an orthogonal coordinate system and that of the actual axes (ΔVx/ΔVy).
For #1206 and #1207, set values determined by taking "ΔV*(1/cosθ)" into consideration.



X :	Actual X axis
Y :	Actual Y axis
y :	Virtual Y (program coordinates (orthogonal coordinates))
θ :	Angle of inclination
ΔV :	Acceleration determined by "#1206 G1bF" and "#1207 G1btL"
ΔVx :	Acceleration of the actual X axis
ΔVy :	Acceleration of the actual Y axis
ΔVx = ΔV*sin(θ + α) / cosθ	
ΔVy = ΔV*cosα / cosθ	

As suggested by the above equations, the maximum acceleration of the actual X axis (ΔVx) and the maximum acceleration of the actual Y axis (ΔVy) respectively take the maximum value when the following equations are true:
(ΔVx): sin(θ+α)=|1|
(ΔVy): cosα=|1|

(Example) When parameter values set on program coordinates (orthogonal coordinates) are used in inclined axis control at an angle of inclination of 60 degrees
Setting values on orthogonal coordinates: #1206 =10000, #1207= 100
Acceleration on orthogonal coordinates (ΔV) = "#1206 G1bF" / "#1207 G1btL"
Maximum acceleration of the actual axes (ΔVx or ΔVy) = ΔV*(1/cos60) = 2*ΔV
In this case, the actual acceleration is twice the setting value.

By using either of the following means, you can prevent the acceleration of the actual axes from exceeding the original setting values (acceleration on the orthogonal coordinates):

(1) Reduce the setting value for "#1206 G1bF".
#1206 = 5000, #1207 = 100

(2) Increase the setting value for "#1207 G1btL".
#1206 = 10000, #1207 = 200

Arbitrary axis exchange/mixed control (cross axis control)

A program error (P126) will occur if the arbitrary axis exchange command is issued during high-accuracy control. However, the high-accuracy control command can be issued after arbitrary axis exchange is carried out. Furthermore, the mixed control command can be issued during high-accuracy control, and the high-accuracy control command can be issued during mixed control.

Synchronous control/control axis synchronization between part systems

High-accuracy control can be commanded during synchronous control or control axis synchronization between part systems. Control axis synchronization between part systems can also be commanded during high-accuracy control. However, deceleration stop is applied to part systems that contain axes subjected to the control axis synchronization between part systems.

Sub part system control

High-accuracy control can be commanded for part systems that have been called by sub part system control. However, high-accuracy control must be commanded from a sub part system because sub part systems do not inherit the modal information of the calling part system.

Control axis superimposition control

During control axis superimposition, high-accuracy control is temporarily disabled in part systems that contain a reference axis (an axis that is only moved by its own commands) and a superimposed axis (an axis that is moved by both its own commands and commands from the reference axis). Therefore, when the Z2 axis (2nd part system) is superimposed on the Z1 axis (1st part system) as shown in the following example, the high-accuracy control of both the 1st and 2nd part systems is temporarily disabled during superimposition control.

[1st part system]	[2nd part system]	Operation
: ! L1;	: ! L1;	
G126 Z2=Z1;	:	The Z2 axis is superimposed on the Z1 axis.
! L2;	! L2;	High-accuracy control of both the 1st and 2nd part systems is temporarily disabled when superimposition is being carried out.
X4. Z4.;	Z60.;	
:	:	
! L3;	! L3;	
G126 Z2 ;	G126 Z2 ;	Ends superimposition of Z2 axis
! L4	! L4	

Operation when high-accuracy control-related G commands are combined

The table below shows operations when following high-accuracy control-related commands are combined:

G61.1, G8P1 : High-accuracy control
G64 : Cutting mode
G61 : Exact stop check mode
G62 : Automatic corner override
G63 : Tapping mode
G08P0 : High-accuracy control cancel (cutting mode)
G05.1Q1 : High-speed high-accuracy control I
G05P2 : High-speed machining mode II
G05P10000 : High-speed high-accuracy control II

A	B	Operation when B is commanded during A command
G61.1/G08P1	G61.1	Continues high-accuracy control.
	G61, G62, G63, G64	Cancels high-accuracy control and operates in the commanded mode.
	G8P1	Continues high-accuracy control.
	G8P0	Cancels high-accuracy control. (Changes G code group 13 to G64.)
	G05.1Q1	Operates in the high-speed high-accuracy control I mode.
	G05P2	Operates in high-accuracy control + high-speed machining mode II.
	G05P10000	Operates in the high-speed high-accuracy control II mode.
G61.2	G61.1	Operates in the high-accuracy control mode.
	G61, G62, G63, G64	Operates in the commanded mode.
	G08P1	Operates in the high-accuracy control mode.
	G08P0	A program error (P29) will occur.
	G05.1Q1	A program error (P29) will occur.
	G05P10000	A program error (P29) will occur.
G05.1Q1	G61.1	Continues the high-speed high-accuracy control I mode.
	G64	Continues the high-speed high-accuracy control I mode.
	G61, G62, G63	Operates in the high-speed high-accuracy control I + commanded mode.
	G08P1	Continues the high-speed high-accuracy control I mode.
	G08P0	Continues the high-speed high-accuracy control I mode.
	G05.1Q1	Continues the high-speed high-accuracy control I mode.
	G05P2	Operates in the high-speed machining mode II.
	G05P10000	A program error (P34) will occur.
G05P10000	G61.1	Continues the high-speed high-accuracy control II mode.
	G64	Continues the high-speed high-accuracy control II mode.
	G61, G62, G63	Operates in the high-speed high-accuracy control II + commanded mode.
	G08P1	Continues the high-speed high-accuracy control II mode.
	G08P0	Continues the high-speed high-accuracy control II mode.
	G05.1Q1	A program error (P34) will occur.
	G05P2	Operates in the high-speed machining mode II.
	G05P10000	Continues the high-speed high-accuracy control II mode.

Precautions

(1) The "high-accuracy control" specifications are required to use this function
If G61.1 is commanded when there are no specifications, a program error (P123) will occur.

(2) The high-accuracy control function is internally enabled by the high-speed high-accuracy I/II (G5.1Q1/G5P10000) command. If the high-speed high-accuracy I/II is commanded in the high-accuracy control mode, the high-speed high-accuracy I/II mode is enabled. Then, if the high-speed high-accuracy I/II mode is canceled, the high-accuracy control mode is restored.

(3) In the high-accuracy control mode, feedrate command F is clamped with the "#2110 Clamp (H-precision)" (Cutting feed clamp speed for high-accuracy control mode) set with parameter. When the cutting feed clamp speed for the high-accuracy control mode is 0, however, it is clamped with the "#2002 clamp" cutting clamp speed set by the parameter.

(4) In the high-accuracy control mode, rapid traverse rate conforms to "#2109 Rapid(H-precision)" (Rapid traverse rate during high-accuracy control mode) set by the parameter. When the rapid traverse rate during the high-accuracy control mode is set to "0", however, the movement follows "#2001 rapid" set by the parameter.

(5) If the specifications for the multi-part system simultaneous high-accuracy control are not provided, the "#1205 G0bdcc" (G0 pre-interpolation) can be used with only one part system.
If the 2nd or later part system is set to the G0 pre-interpolation acceleration/deceleration, an MCP alarm (Y51 0017) will occur.

(6) "#1568 SfiltG1", "#1569 SfiltG0" and "#1570 Sfilt2" cannot be changed from the screen during program mode.
If these parameters is changed by "parameter input by program", these parameters become valid from the next block.

(7) If Reset or Emergency signal is input during axis travel, it takes a time equal to the time constant to recover from the reset or emergency stop state.

(8) When there are the high-accuracy control time constant expansion specifications, the sampling buffer area may be smaller.

(9) The high-accuracy control time constant expansion specifications can only be used for a 1-part system. In a multi-part system, the high-accuracy control time constant expansion specifications are disabled even when they are set to ON.

(10) For a part system where high-accuracy control is to be commanded, set the number of axes in the part system to 8 or less. If high-accuracy control is commanded for a part system that has 9 or more axes, an operation error (M01 0135) will occur. The error will not occur, however, if the number of axes in the part system excluding the master axis/slave axis is 8 or less during the synchronous control/control axis synchronization between part systems.

(11) Even if the parameter "#1210 RstGmd" (modal G code reset setting) is set to "not to initialize group 13 at reset", group 13 is initialized according to the setting of "#1148 I_G611" (Initial hi-precis) if it is enabled. To retain group 13 at reset, set "#1148 I_G611" to "0".
These parameters depend on the MTB specifications.

(12) If the parameter "#1205 G0bdcc" (G0 acceleration/deceleration before interpolation) is set to "1", the value set with the parameter "#2224 SV024" (in-position detection width) will be used as the in-position width. The setting of the parameter "#2077 G0inps" (G0 in-position width) and the programmable in-position check with ",I" address are disabled.

17.2.2 SSS Control

Function and purpose

Machining programs that approximate a freely curved surface with fine segments are run at high speed and with high-level accuracy. This function enables machining with less scratches and streaks on the cutting surface compared to the conventional high-accuracy control function.
With conventional high-accuracy control, the angle between two blocks is compared with the corner deceleration angle to determine whether to execute corner deceleration between the blocks. This can cause the speed to suddenly change between the blocks with an angle close to the corner deceleration angle, resulting in scratches or streaks.

The SSS (Super Smooth Surface) control uses information on not only the angle but also global paths between two blocks to provide optimum speed control that is not significantly affected by minute stepping or waviness. The favorable effects of this control include a reduction in the number of scratches or streaks on cutting surfaces.
The SSS control has the following features:

(1) This function is effective at machining smooth-shaped dies using a fine segment program.

(2) This function provides speed control that is not susceptible to errors in paths.

(3) Even if corner deceleration is not required, the speed is clamped if the predicted acceleration is high.
(The clamp speed can be adjusted using the parameter "#8092 ClampCoeff".)

The length of the path direction recognized with SSS control can be adjusted with the machining parameter "#8091 reference length". The range is increased as the setting value increases, and the effect of the error is reduced.
If the multi-part system simultaneous high-accuracy specification is provided, up to two part systems can be used at the same time.

Note

(1) The use of this function requires the following functions, in addition to the SSS control specifications. Make sure that these specifications are enabled before using this function.
High-accuracy control (G61.1/G08P1)
High-speed high-accuracy control I (G05.1 Q1)
High-speed high-accuracy control II (G05 P10000)

Detailed description

When the parameters are set as below, each of the following high-accuracy control commands is activated under SSS control.

<Parameter>
"#8090 SSS ON" ON

<Command format of the modes activated under SSS control>"

[High-accuracy control]	
G61.1 ; or G08P1; G08P0; or, G command in group 13 except G61.1	High-accuracy control ON High-accuracy control OFF

[High-speed high-accuracy control I]	
G05.1 Q1 ; G05.1 Q0 ;	High-speed high-accuracy control I ON High-speed high-accuracy control I OFF

[High-speed high-accuracy control II]	
G05 P10000 ; G05 P0 ;	High-speed high-accuracy control II ON High-speed high-accuracy control II OFF

"SSS" is displayed on the modal display screen under SSS control.
However "SSS" is not displayed when a command being executed is out of the scope of SSS control.

Adjustment of accuracy coefficient

The clamp speed at a corner and arc can be adjusted using "#8022 CORNER COMP" and "#8023 CURVE COMP" (If "#8021 COMP_CHANGE" is set to "0", use "#8019 R COMP" to adjust the clamp speed at a corner and arc). When "#8096 Deceler. coeff. ON" is set to "1", "#8097 Corner decel coeff" and "#8098 Arc clamp spd coef" become valid during SSS control. Using these parameters, you can use different corner deceleration speeds and clamp speeds at arcs according to whether or not the SSS control is enabled.
For parameters #8097 and #8098, respectively, set a percentage ratio to the level of the relevant speed that is applied when the SSS control is disabled.

Parameter	Item to be adjusted
#8097 Corner decel coeff	Corner deceleration speed to be applied when the SSS control is enabled
#8098 Arc clamp spd coef	Arc clamp speed to be applied when the SSS control is enabled

(Example) When "#8097 Corner decel coeff" is set to 200 (%), the corner deceleration speed that is applied when the SSS control is enabled becomes twice the corner deceleration speed that is applied when the SSS control is disabled.

When setting the parameters, adjust the values within the range in which the machine does not vibrate.

Parameter standard values

The standard values of the parameters related to SSS control are shown below.

(1) User parameters

#	Item	Standard value
8090	SSS ON	1
8091	StdLength	1.000
8092	ClampCoeff	1
8093	StepLeng	0.005
8094	DccWaitAdd	0
8096	Deceler. coeff. ON	1
8097	Corner decel coeff	300
8098	Arc clamp spd coef	100
8019	R COMP	0
8020	DCC ANGLE	10
8021	COMP CHANGE	1
8022	CORNER COMP	0
8023	CURVE COMP	-20
8034	AccClampt ON	0
8036	CordecJudge	0
8037	CorJudgeL	0

<Note>

♦Reference items for adjusting the parameter

The relationship between each parameter, accuracy and speed is shown below.
The accuracy and speed required for machining can be adjusted with these settings.
When setting the parameters, adjust the values within the range in which the machine does not vibrate.

Parameter	Adjustment target	Effect
#8022 CORNER COMP	Accuracy at corner section	Large setting = Accuracy increases, speed drops
#8023 CURVE COMP	Accuracy at curve section	Large setting = Accuracy increases, speed drops
#8092 ClampCoeff	Accuracy at curve section	Large setting = Accuracy drops, speed increases <Note> ♦Usually use the standard value and adjust with "#8023".

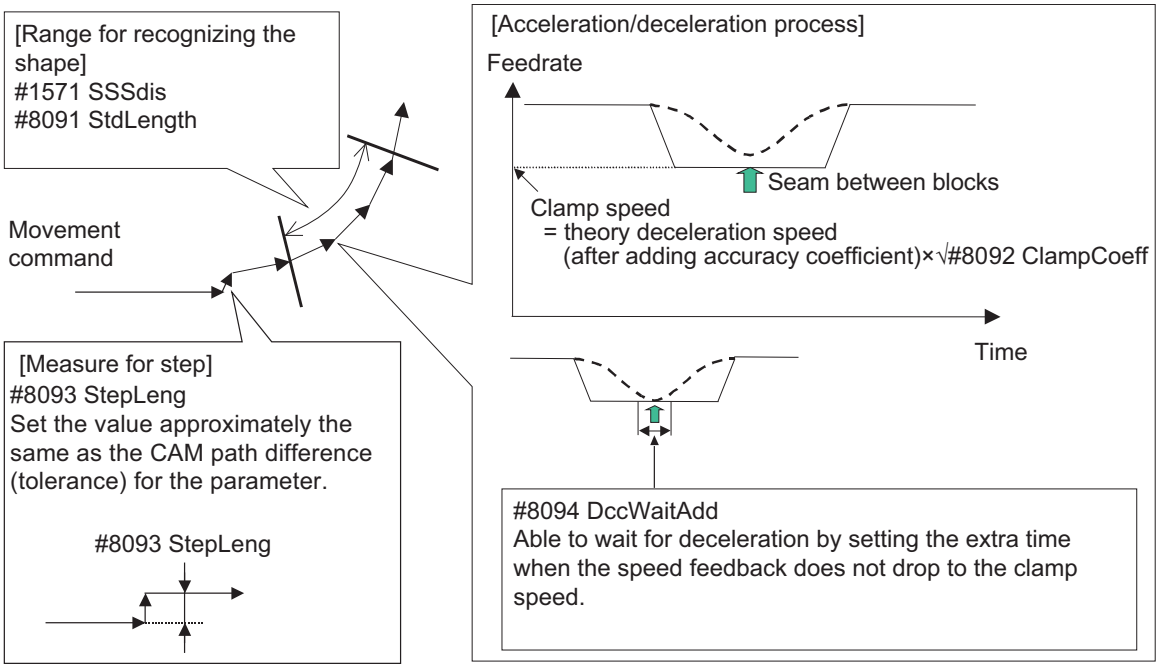
(2) Basic specification parameters (depend on the MTB specifications)

#	Item		Standard value
1148	I_G611	Initial high-accuracy	0
1205	G0bdcc	G0 before interpolation	-
1206	G1bf	Acceleration/deceleration before interpolation Maximum speed	-
1207	G1btL	Acceleration/deceleration before interpolation Time constant	-
1571	SSSdis	SSS control adjustment coefficient fixed value selection	0
1572	Cirorp	Arc command overlap	0
1568	SfiltG1	G1 soft acceleration/deceleration filter	0
1569	SfiltG0	G0 soft acceleration/deceleration filter	0
1570	Sfilt2	Soft acceleration/deceleration filter 2	0

(3) Axis specification parameters (depend on the MTB specifications)

#	Item		Standard value
2010	fwd_g	Feed forward gain	70
2068	G0fwdg	G00 feed forward gain	70
2096	crncsp	Minimum corner deceleration speed	0

SSS control parameter



Precautions

(1) Pre-reading is executed during SSS control, so a program error could occur before the block containing the error is executed.

(2) Buffer correction is not guaranteed during SSS control.

(3) If automatic/manual simultaneous or automatic handle feed interrupt are used during SSS control, the machining accuracy will not be guaranteed.

(4) If a fine arc command is issued during SSS control, it may take longer to machine.

(5) The same path as single block operation will be used during graphic check.

(6) The line under the cutting feedrate and arc command block are subjected to the speed control in the SSS control. The command blocks that are not subjected to speed control, decelerate first and automatically switch the SSS control ON and OFF.

(7) SSS control is temporally disabled in the following modal:

- Polar coordinate interpolation
- Cylindrical interpolation
- User macro interruption enable (M96)
- Feed per revolution (synchronous feed)
- Constant surface speed control
- Fixed cycle
- Automatic tool length measurement

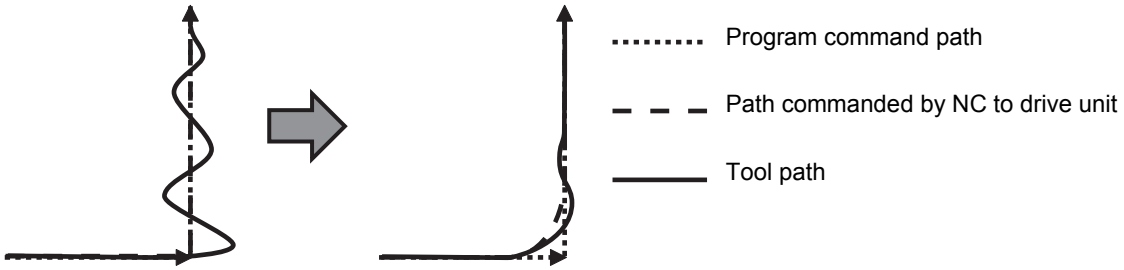
(8) There are some restrictions for each high-accuracy control. Refer to each section for restrictions.

- "17.2 High-accuracy Control"
- "17.3 High-speed High-accuracy Control"

17.2.3 Tolerance Control

Function and purpose

This function obtains the optimum clamp speed for corners or curves based on the designated tolerance to perform operations. It also ensures smooth passing within the tolerance range in corner sections, which suppresses machine vibrations. This means that the clamp speed can be increased to reduce the cycle time.
This function allows the machine to operate with the optimum tool path and speed, simply by specifying the tolerance, so an operator can easily carry out high quality machining.
The tolerance refers to the allowable error amount between the path commanded in the machining program and the path output by NC.
The validity of this function depends on the MTB specifications. This function also requires the SSS control specifications because it can only be used under SSS control.



Tolerance control: Invalid　　Tolerance control: Valid

This function is enabled when the following conditions are satisfied:

(1) The tolerance control specification is valid. (Based on the MTB specifications.)
(2) The parameter "#8090 SSS ON" is set to "1".
(3) The parameter "#12066 Tolerance ctrl ON" is set to "1". (*1)(*2)
(4) Either high-accuracy control (G61.1/G08P1) or high-speed high-accuracy control I/II (G05.1Q1/G05P10000) is valid.

(*1) Even if conditions (1) and (3) are satisfied, an operation error (M01 0139) will occur and the cycle start cannot be performed automatically if the parameter "#8090 SSS ON" is set to "0". In this case, enable SSS control and reset the alarm to start the cycle automatically.

(*2) A setting error will occur if "1" is set when this specification is invalid.

Command format

Set the tolerance with the parameter "#2659 tolerance" or the ",K" address following the G code (G61.1 command). When the setting value is "0", this function runs with "0.01(mm)".

Tolerance specification

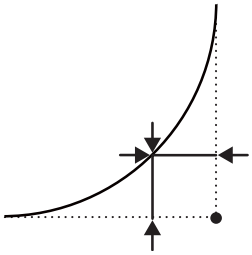
G61.1 or G61.4 ,K__ ;

,K	Tolerance (mm)

- The range of the command value is 0.000 to 100.000. If a value exceeding the range is commanded, a program error (P35) will occur.
- The tolerance designated by ",K" is applied to all axes in the part system.
- When "0" is designated or ",K" is omitted, the program runs based on the value of the parameter "#2659 tolerance".
- The tolerance designated by ",K" is not held after reset. Therefore, if ",K" is not designated in the G61.1 command after reset, the axis runs based on the value of the parameter "#2659 tolerance".

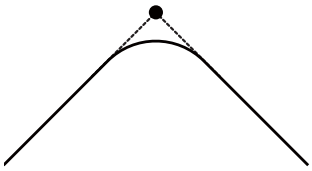
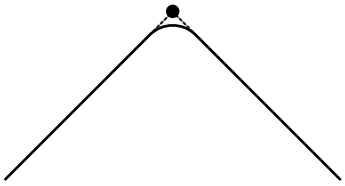
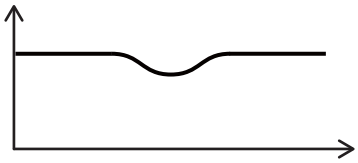
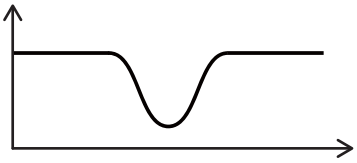
Detailed description

The axis moves in the designated tolerance range during tolerance control.
The tolerance on the corner shape is as shown on the right.



Speed control

The clamp speed is obtained from the tolerance in the corner or curve section during tolerance control.
As the designated tolerance is lower, the axis speed decelerates.

	Tolerance: High	Tolerance: Low
Command path		
Synthesis rate		

Parameters valid during tolerance control

The parameters valid and invalid during tolerance control are as follows. Some parameters depend on the MTB specifications.

(1) Valid parameters

No.	Parameter name	Supplements
1206	G1bF	When combining with the tolerable acceleration control for each axis, specify parameters "#2157 G1bFx" and "#2158 G1btLx".
1207	G1btL	
1568	SfiltG1	
12051	Jerk_filtG1	
2659	tolerance	

(2) Invalid parameters (Parameters with no setting required)

No.	Parameter name	Supplements
1570	Sfilt2	Ignored even if the value is entered.
2159	compx	Ignored even if the value is entered. The clamp speed is obtained from the tolerance during tolerance control; therefore, parameters for adjusting the clamp speed are not required.
8019	R COMP	
8020	DCC ANGLE	
8021	COMP CHANGE	
8022	CORNER COMP	
8023	CURVE COMP	
8096	Deceler. coeff. ON	
8097	Corner decel coeff	
8098	Arc clamp spd coef	

Program example

: G91 ;	
G61.1 ,K0.02;	Designate tolerance 0.02 (mm).
G01 X0.1 Z0.1 F1000 ; X0.1 Z-0.2 ; Y0.1 ;	Tolerance: 0.02 (mm)
G61.1 ,K0;	Designate tolerance 0 (mm).
X-0.1 Z-0.05 ; X-0.1 Z-0.3 ;	Tolerance: Follows parameter "#2659 tolerance".
G64 ; :	

Precautions

(1) While tolerance control is valid, tolerance control may be canceled temporarily depending on some commands. If tolerance control is canceled temporarily, the axis moves to the commanded position without taking an inner route in a corner section. After this, when a temporary cancel cause is removed, tolerance control restarts. The temporary cancel conditions are as follows.

(a) Modal in which the group 1 command is not G01 (linear interpolation) or G02/G03 (circular interpolation).

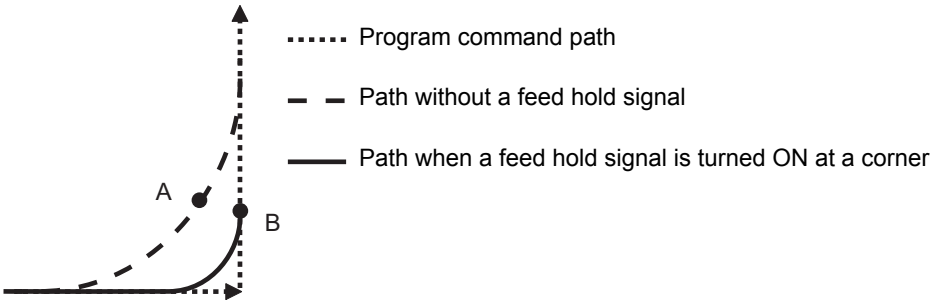
(b) Under single block operation

(c) Modal in which SSS control is disabled temporarily (Modal shown below)

Polar coordinate interpolation
Cylindrical interpolation
User macro interruption enable (M96)
Feed per revolution (synchronous feed)
Constant surface speed control
Fixed cycle
Automatic tool length measurement
Exponential interpolation

(2) The stored stroke limit's prohibited range is determined based on the program command path. As a result, machining may not be stopped even if the command moved inward by tolerance control enters the prohibited range.

(3) If a feed hold signal is turned ON at a corner, machining stops on the program command path. This means that it does not stop at point A in the figure below but at point B.



17.2.4 Initial High-accuracy Control

If "#1148 I_G611" (Initial high-accuracy) is set by the MTB specifications, high-accuracy control-related functions can be enabled when the power is turned ON.
At power ON, the modes set by this parameter are enabled, but each mode can be changed to a different one by commanding as follows in the machining program.

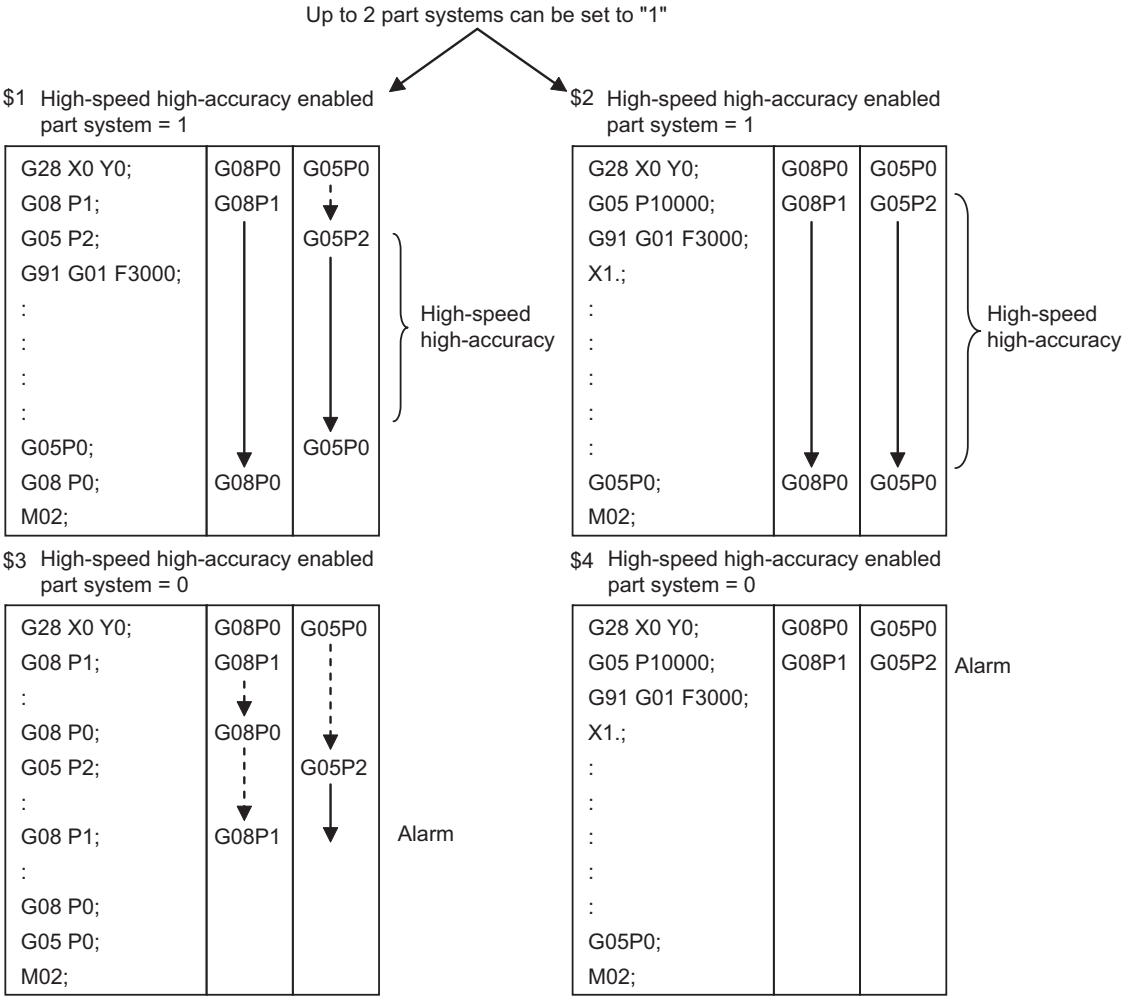
#1148 setting value	Modes enabled at power ON
0	G08P0/G64 (cutting mode) command
1	G08P1/G61.1 (high-accuracy control mode) command
2	G05.1Q1 (high-speed high-accuracy control I mode) command
3	G05P10000 (high-speed high-accuracy control II mode) command

It is impossible, however, to shift to the high-speed high-accuracy control II mode during the high-speed high-accuracy control I. Likewise, it is also impossible to shift to the high-speed high-accuracy control I mode during the high-speed high-accuracy control II.
To shift to either mode, cancel the current high-speed high-accuracy control mode using "G05.1 Q0" or "G05 P0" first and then command the target mode.
If any function set by this parameter is not included in your machine's specifications, an available high-accuracy function with a number smaller than the parameter setting is enabled.

17.2.5 Multi-part System Simultaneous High-accuracy

Function and purpose

High-accuracy control and high-speed machining mode are available respectively in all part systems, however, the simultaneous usage of high-accuracy control and high-speed machining mode (including High-speed high-accuracy control I/II) are available only in part systems which are limited by the parameter "#8040 High-SpeedAcc". While high-accuracy control and high-speed machining mode are available simultaneously in a part system where this parameter is set to "1", a program error (P129) will occur in those where the parameter is set to "0" when commanded. Also, for part systems where "#8040 High-SpeedAcc" is set to "0", "#1148 I_G611" must be set to "0" (Cutting mode when the power is turned ON) or "1" (High-accuracy control mode when the power is turned ON). If the parameter "#1148 I_G611" is set to a value other than "0" and "1", the parameter is regarded as being set to "1".
Note that up to two part systems can be set to use high-accuracy control and high-speed machining mode simultaneously. If three or more part systems are set as such, an MCP alarm (Y51 0032) will occur.
If the parameter "#8040 High-SpeedAcc" is set to "0" for all part systems, the simultaneous usage of high-accuracy control and high-speed machining mode is available in the 1st and 2nd part systems.



(Note) It is limited also in G61.1 command.

Refer to the following chapters for details of each high-accuracy control.
"17.2 High-accuracy Control"
"17.3 High-speed High-accuracy Control"

Detailed description

When "#1148 I_G611" (Initial hi-precis) is enabled, the initial modal state after power ON will be the high-accuracy control mode. Refer to "17.2.4 Initial High-accuracy Control" for details.
In this case, the high-accuracy control mode is enabled if the multi-part system simultaneous high-accuracy specification is provided. Otherwise, the 1st part system enters the high-accuracy control mode, but the 2nd part system enters the cutting mode.

17.3 High-speed High-accuracy Control

It depends on the MTB specifications whether the modal state at power ON is high-speed high-accuracy control I, II, III, or OFF.
It also depends on the specifications whether to hold the modal state at reset.
Refer to your machine's specifications.
In the main text, the axis address refers to the address of an axis that exits on the machine.
It corresponds to the address designated in the parameters "#1013 axname" and "#1014 incax".
These parameter settings depend on the MTB specifications.

17.3.1 High-speed High-accuracy Control I, II ; G05.1 Q1/Q0, G05 P10000/P0

Function and purpose

This function runs a machining program that approximates a freely curved surface with fine segments at high speed and with high-level accuracy. This is effective in increasing the speed of machining dies of a freely curved surface. This function is useful for machining which needs to make an edge at a corner or reduce an error from an inner route of curved shape.
A higher fine segment processing capability leads to a faster cutting speed, resulting in a shorter cycle time and a better machining surface quality, kBPM, the unit for the fine segment processing capability, is an abbreviation of "kilo blocks per minute" and refers to the number of machining program blocks that can be processed per minute.
In the main text, the axis address refers to the address of an axis that exits on the machine.
It corresponds to the address designated in the parameters "#1013 axname" and "#1014 incax".
These parameter settings depend on the MTB specifications.

Fine segment capacity for 1-part system

G01 block fine segment capacity for 1mm segment (unit: kBPM)

The performance below applies under the following conditions.

- 6-axis system (including spindle) or less
- 1-part system
- 3 axes or less commanded simultaneously in G01
- The block containing only the axis name and movement amount (Macro and variable command are not included.)
- During tool nose radius compensation cancel (G40)
- The parameter "#1259 set31/bit1" is set to "1". (The number of machining blocks per unit time is set to "low-speed mode".)

When the above conditions are not satisfied, the given feedrate may not be secured.

	Fine segment capacity			Restriction in the program
	M850 / M830	M80		
		Type A	Type B	
High-speed high-accuracy function I mode	67.5	33.7	-	Yes
High-speed high-accuracy function II mode	168 (*1)	67.5	-	Yes

(*1) In the network connection, the value described in the above table may not be guaranteed depending on the state.

Fine segment capacity for multi-part system

G01 block fine segment capacity for 1mm segment (unit: kBPM)

The fine segment processing capability below applies under the following conditions.

- 3 axes or less commanded simultaneously in G01
- The block containing only the axis name and axis movement amount (Macro and variable command are not included.)
- Tool radius compensation OFF (G40)
- The parameter "#1259 set31/bit1" is set to "1".
  (The number of machining blocks per unit time is set to for "low-speed mode".)

When the above conditions are not satisfied, the given feedrate may not be secured.

(1) High-speed high-accuracy control I

Number of part systems/number of axes	Number of part systems	M850 / M830	M80	
	(#8040=1)		Type A	Type B
1-part system	1 part systems	67.5	33.7	- (*2)
2-part system	1 part systems	67.5	33.7	- (*2)
	2 part systems	33.7	33.7	- (*2)
4-part system	1 part systems	33.7	- (*1)	- (*2)
Up to 16 axes	2 part systems	33.7	- (*1)	- (*2)
5 part systems or more or 17 axes or more	1 part systems	16.8	- (*1)	- (*2)
	2 part systems	16.8	- (*1)	- (*2)

(2) High-speed high-accuracy control II

Number of part systems/number of axes	Number of part systems	M850 / M830	M80	
	(#8040=1)		Type A	Type B
1-part system	1 part systems	168 (*3)	67.5	- (*2)
2-part system	1 part systems	100	67.5	- (*2)
	2 part systems	67.5	67.5	- (*2)
4-part system	1 part systems	33.7	- (*2)	- (*2)
Up to 16 axes	2 part systems	33.7	- (*2)	- (*2)
5 part systems or more or 17 axes or more	1 part systems	16.8	- (*2)	- (*2)
	2 part systems	16.8	- (*2)	- (*2)

(*1) This system cannot be used for this model.

(*2) There are no high-speed high-accuracy control specifications.

(*3) 100 kBPM for a time constant expansion system.
(The time constant expansion system is available when its specifications are enabled and it is a 1-part system.)

High-speed high-accuracy control simultaneously for two part systems

High-speed high-accuracy control I/II can be used simultaneously in up to two part systems.
High-speed high-accuracy control I/II can be used in a part system where "1" is set for the parameter "#8040 High-SpeedAcc". A program error occurs (P129) if this is commanded for a part system where "0" is set for the parameter.

If the parameter "#8040 High-SpeedAcc" is set to "0" for all part systems, only the first part system is handled as the one with the parameter set to "1". Also, a part system where the parameter "#1148 Initial hi-precis" is set to "2" to "4" is handled as the one with the parameter "#8040 High-SpeedAcc" set to "1".
The parameter "#8040 High-SpeedAcc" can be set to "1" for up to two part systems. If 3 or more part systems are set to "1", an MCP alarm (Y51 0032) occurs. When "1" is set for two part systems, the fine segment processing capability decreases compared to when "1" is set only for one part system.

Command format

G05.1 Q1 ;	**High-speed high-accuracy control I ON**

G05.1 Q0 ;	**High-speed high-accuracy control I OFF**

G05 P10000 ;	**High-speed high-accuracy control II ON**

G05 P0 ;	**High-speed high-accuracy control II OFF**

Note

(1) The high-speed high-accuracy mode I and II cannot be used at the same time.

(2) G05.1 Q1 (high-speed high-accuracy mode I) and G05 P10000 (high-speed high-accuracy mode II) are valid when the specifications are available regardless of the parameter "#1267 ext03/bit0" setting.

Detailed description

(1) The high-speed high-accuracy control I / II can be used during tape, MDI, SD card or memory modes.

(2) The override, maximum cutting speed clamp, single block operation, dry run, handle interrupt and graphic trace are valid even during the high-speed high-accuracy control I / II modal.

(3) The machining speed may drop depending on the number of characters in one block.

(4) The high-speed high-accuracy control I / II function automatically turns the high-accuracy control mode ON.
For high-accuracy control function, refer to "17.3 High-speed High-accuracy Control".

(5) Turn the tool nose radius compensation command ON and OFF during the high-speed high-accuracy control I / II mode.
If the high-speed high-accuracy control I / II mode is turned OFF without turning the tool nose radius compensation OFF, program error (P34) will occur.

(6) Turn the high-speed high-accuracy control I / II mode OFF before commanding data other than those that can be commanded.

(7) When using the high-speed high-accuracy control II mode, it is necessary to set the parameter "#1572 Cirorp" to eliminate the speed fluctuation at the seams between arc and straight line or arc and arc. This parameter, however, depends on the MTB specifications.

(8) Feedrate command F is clamped with the "#2110 Clamp (H-precision)" (Cutting feed clamp speed for high-accuracy control mode) set with parameter.

(9) Rapid traverse rate enables "#2109 Rapid(H-precision)" (Rapid traverse rate during high-accuracy control mode) set by the parameter.

(10) When the "#2109 Rapid(H-precision)" is set to "0", the movement follows "#2001 rapid" (rapid traverse rate) set by the parameter. Also, when "#2110 Clamp (H-precision)" is set to "0", the speed will be clamped with "#2002 clamp" (Cutting clamp speed) set with parameter.

Enabling conditions

To enable each high-speed high-accuracy control function, it is necessary to satisfy the following conditions respectively:

(1) The specification of each function is valid.

(2) Each function is in a valid modal state. (Refer to "Relationship with Other Functions")

(3) Each function is enabled by one of the following procedures:

- Command each in the machining program.
- Set each for the parameter "#1148 Initial hi-precis". (The modal at power ON corresponds to each high-speed high-accuracy control function.)

	#1148 setting
High-speed high-accuracy control I	2
High-speed high-accuracy control II	3

Relationship with other functions

Relationship between the high-speed high-accuracy control I and other functions

(1) Relationship between the high-speed high-accuracy control I and G code functions
Column A: Operation when the additional function is commanded while the high-speed high-accuracy control I is enabled
Column B: Operation when the high-speed high-accuracy control I (G05.1Q1) is commanded while the additional function is enabled
○: The high-speed high-accuracy control I and the additional function are both enabled
Δ: The high-speed high-accuracy control I is temporarily canceled, while the additional function is enabled
X: Alarm generation (the text in parentheses refers to the number of the program error to be generated.)
-: No combination
□: Others

Group	G code (G code type: 3)	Additional function	A	B
0	G04	Dwell	Δ	-
	G05 P0	High-speed machining mode II OFF High-speed high-accuracy control II OFF High-speed high-accuracy control III OFF	× (P34)	□ (*2)
	G05 P2	High-speed machining mode II ON	□ (*4)	□ (*2)
	G05 P10000	High-speed high-accuracy control II ON	× (P34)	× (P34)
	G05.1 Q0	High-speed high-accuracy control I OFF Spline interpolation OFF	□ (*1)	□ (*2)
	G05.1 Q1	High-speed high-accuracy control I ON	○	○
	G08 P0	High-accuracy control OFF	□ (*3)	□ (*2)
	G08 P1	High-accuracy control ON	□ (*3)	□ (*2)
	G09	Exact stop check	Δ	-
	G10 I_J_ G10 K_	Parameter coordinate rotation input	Δ	-
	G10 L2	Compensation data input by program	Δ	-
	G10 L70 G10 L50	Parameter input by program	Δ	-
	-(G22(*))	Soft limit ON	○	○
	-(G23(*))	Soft limit OFF	○	○
	G27	Reference position check	Δ	-
	G28	Reference position return	Δ	-
	G29	Start position return	Δ	-
	G30	2nd to 4th reference position return	Δ	-
	G30.1	Tool change position return 1	Δ	-
	G30.2 - G30.5	Tool change position return 2 to 5	Δ	-
	G31	Skip Multi-step skip 2	Δ	-
	G31.1 - G31.3	Multi-step skip 1 to 3	Δ	-
	G37	Automatic tool length measurement	Δ	-

Group	G code (G code type: 3)	Additional function	A	B
0	G92	Spindle clamp speed setting Coordinate system setting	Δ	-
	-(G50.2,G250(*))	Polygon machining mode OFF	○	○
	-(G51.2,G251(*))	Polygon machining mode ON	Δ	Δ
	G92.1	Workpiece coordinate system preset	Δ	-
	G52	Local coordinate system setting	Δ	-
	G53	Machine coordinate system selection	Δ	-
	G65	User macro call	□ (*5)	□ (*6)
	G110	Mixed control I (cross control)	○	○
	G111	Axis name switch	○	○
	G113	Spindle synchronization control OFF	○	○
	G114.1	Spindle synchronization control ON	Δ	Δ
	G114.2	Tool spindle synchronization I (polygon machining mode)	Δ	Δ
	G114.3	Tool spindle synchronization II (hobbing mode)	Δ	Δ
	G115 G116	Start point timing synchronization	Δ	-
	G117	M code output during axis traveling	Δ	-
	G122	Sub part system control I	× (P652)	□ (*7)
	G125	Control axis synchronization between part systems	○	○
	G126	Control Axis Superimposition	Δ	× (P29)
	G140	Arbitrary axis exchange control	× (P126)	○
	G141	Arbitrary axis exchange return	× (P126)	-
	G142	Reference axis arrange return	× (P126)	-
	G144	Sub part system control II	× (P652)	□ (*7)
1	G00	Positioning	Δ	Δ
	G01	Linear interpolation	○	○
	G02 G03	Circular interpolation	□ When SSS is enabled: ○ When SSS is disabled: Δ	□ When SSS is enabled: ○ When SSS is disabled: Δ
	G02.3 G03.3	Exponential interpolation	Δ	Δ
	G33	Thread cutting	Δ	Δ
	G34	Variable lead thread cutting	Δ	Δ
	G35 G36	Circular thread cutting	Δ	Δ

Group	G code (G code type: 3)	Additional function	A	B
2	G16	Milling interpolation Plane selection Y-Z cylindrical plane	× (P34)	× (P34)
	G17/G18/G19	Plane selection	○	○
3	G90	Absolute value command	○	○
	G91	Incremental value command	○	○
4	G22	Barrier check ON	○	○
	G23	Barrier check OFF	○	○
	-(G22(*))	Soft limit ON	○	○
	-(G23(*))	Soft limit OFF	○	○
5	G94	Asynchronous feed (feed per minute)	○	○
	G95	Synchronous feed (feed per revolution)	○	○
6	G20	Inch command	○	○
	G21	Metric command	○	○
7	G40	Tool nose radius compensation OFF	○	○
	G41 G42	Tool nose radius compensation ON	○	× (P29)
	G46	Tool nose radius compensation ON (automatic direction identification)	○	× (P29)
9	G80	Fixed cycle cancel	○	○
	Other than G80	Fixed cycle	Δ	Δ
10	G98	Fixed cycle (Initial level return)	○	○
	G99	Fixed cycle (R point level return)	○	○
12	G54-G59, G54.1	Workpiece coordinate system selection	○	○
13	G61	Exact stop	□	□
		check mode	(*8)	(*9)
	G61.1	High-accuracy control	□ (*3)	□ (*2)
	G62	Automatic corner	□	□
		Override	(*3)	(*2)
	G63	Tapping mode	□ (*3)	□ (*2)
	G64	Cutting mode	□ (*3)	□ (*2)
14	G66 G66.1	User macro modal call	□ (*5)	□ (*6)
	G67	User macro modal call cancel	○	○
15	G68	Facing turret mirror image ON	× (P29)	× (P29)
	G69	Mirror image for facing tool posts OFF	○	○
	-(G68(*))	Facing turret mirror image ON	× (P29)	× (P29)
		Balance cut ON	× (P29)	× (P29)
	-(G69(*))	Mirror image for facing tool posts OFF	○	○
		Balance cut OFF	○	○

Group	G code (G code type: 3)	Additional function	A	B
16	G68.1	Coordinate rotation by program ON	○	× (P34)
	G69.1	Coordinate rotation by program OFF	○	○
17	G96	Constant surface speed control ON	○	○
	G97	Constant surface speed control OFF	○	○
18	G14	Balance cut OFF	○	○
	G15	Balance cut ON	× (P29)	× (P29)
19	G12.1	Milling interpolation ON	× (P485)	Δ
	G13.1	Milling interpolation OFF	○	○
	-(G07.1,G107(*))	Cylindrical interpolation	× (P485)	Δ
	-(G12.1,G112(*))	Polar coordinate interpolation ON	× (P485)	Δ
	-(G13.1,G113(*))	Polar coordinate interpolation OFF	○	○
20	G43.1	1st spindle control mode	○	○
	G44.1	2nd spindle control mode	○	○
	G47.1	Two spindles simultaneous control mode	○	○
24	G188	Dynamic M/L program changeover ON	○	○
	G189	Dynamic M/L program changeover OFF	○	○

(*) Only applies to G code lists 6 and 7

(*1) Disables the high-speed high-accuracy control I.

(*2) Enables the high-speed high-accuracy control I.

(*3) High-speed high-accuracy control I continues.

(*4) Enables the high-speed machining mode II.

(*5) Enables the high-speed high-accuracy control I in a macro program.

(*6) Enables the high-speed high-accuracy control I if G05.1Q1 is commanded in a macro program.

(*7) Enables the high-speed high-accuracy control I if G05.1Q1 is commanded in a sub part system.

(*8) Enables the exact stop check mode.

(*9) Exact stop check mode continues.

(2) Relationship between the high-speed high-accuracy control I and functions other than G codes

Column A: Operation when the additional function is commanded while the high-speed high-accuracy control I is enabled

Column B: Operation when the high-speed high-accuracy control I (G05.1Q1) is commanded while the additional function is enabled

○: The high-speed high-accuracy control I and the additional function are both enabled

Δ: The high-speed high-accuracy control I is temporarily canceled, while the additional function is enabled

X: Alarm generation (the text in parentheses refers to the number of the program error to be generated.)

-: No combination

□: Others

Additional function	A	B
SSS ON	-	○
Mirror image by parameter setting ON	-	× (P34)
PLC mirror image ON	-	× (P34)
Subprogram call (M98)	□ (*10)	□ (*11)
Timing synchronization between part systems	□ (*12)	-
Inclined axis control	-	○
T code offset	○	○
T code mirror image for facing tool posts	× (P29)	× (P29)
MTB macro	□ (*13)	□ (*14)
Macro interruption	□ (*15)	□ (*16)
PLC interruption	□ (*15)	□ (*16)
Corner chamfering/Corner R	Δ	-
Linear angle command	○	-
Geometric command	○	-
Chopping	○	○
Optional block skip	○	-

(*10) Enables the high-speed high-accuracy control I in a subprogram.

(*11) Enables the high-speed high-accuracy control I if G05.1Q1 is commanded in a subprogram.

(*12) Enables timing synchronization.

(*13) Enables the high-speed high-accuracy control I in a MTB program.

(*14) Enables the high-speed high-accuracy control I if G05.1Q1 is commanded in a MTB program.

(*15) Enables the high-speed high-accuracy control I in an interrupt program.

(*16) Enables the high-speed high-accuracy control I if G05.1Q1 is commanded in an interrupt program.

Relationship between the high-speed high-accuracy control II and other functions

(1) Relationship between the high-speed high-accuracy control II and G code functions
Column A: Operation when the additional function is commanded while the high-speed high-accuracy control II is enabled
Column B: Operation when the high-speed high-accuracy control II (G05P10000) is commanded while the additional function is enabled
○: The high-speed high-accuracy control II and the additional function are both enabled
Δ: The high-speed high-accuracy control II is temporarily canceled, while the additional function is enabled
X: Alarm generation (the text in parentheses refers to the number of the program error to be generated.)
-: No combination
□: Others

Group	G code (G code type: 3)	Additional function	A	B
0	G04	Dwell	Δ	-
	G05 P0	High-speed machining mode II OFF High-speed high-accuracy control II OFF High-speed high-accuracy control III OFF	□ (*1)	□ (*2)
	G05 P2	High-speed machining mode II ON	□ (*4)	□ (*2)
	G05 P10000	High-speed high-accuracy control II ON	□ (*3)	□ (*3)
	G05.1 Q0	High-speed high-accuracy control I OFF Spline interpolation OFF	□ (*3)	□ (*2)
	G05.1 Q1	High-speed high-accuracy control I ON	× (P34)	× (P34)
	G08 P0	High-accuracy control OFF	□ (*3)	□ (*2)
	G08 P1	High-accuracy control ON	□ (*3)	□ (*2)
	G09	Exact stop check	Δ	-
	G10 I_J_ G10 K_	Parameter coordinate rotation input	Δ	-
	G10 L2	Compensation data input by program	Δ	-
	G10 L70 G10 L50	Parameter input by program	Δ	-
	-(G22(*))	Soft limit ON	○	○
	-(G23(*))	Soft limit OFF	○	○
	G27	Reference position check	Δ	-
	G28	Reference position return	Δ	-
	G29	Start position return	Δ	-
	G30	2nd to 4th reference position return	Δ	-
	G30.1	Tool change position return 1	Δ	-
	G30.2 - G30.5	Tool change position return 2 to 5	Δ	-
	G31	Skip Multiple-step skip 2	Δ	-
	G31.1 - G31.3	Multi-step skip 1 to 3	Δ	-
	G37	Automatic tool length measurement	Δ	-
	G92	Spindle clamp speed setting Coordinate system setting	Δ	-
	-(G50.2,G250(*))	Polygon machining mode OFF	○	○
	-(G51.2,G251(*))	Polygon machining mode ON	Δ	Δ

Group	G code (G code type: 3)	Additional function	A	B
0	G92.1	Workpiece coordinate preset	Δ	-
	G52	Local coordinate system setting	Δ	-
	G53	Machine coordinate system selection	Δ	-
	G65	User macro simple call	□ (*5)	□ (*6)
	G110	Mixed control I (cross control)	○	○
	G111	Axis name switch	○	○
	G113	Spindle synchronization control OFF	○	○
	G114.1	Spindle synchronization control ON	Δ	Δ
	G114.2	Tool spindle synchronization I (polygon machining mode)	Δ	Δ
	G114.3	Tool spindle synchronization II (hobbing mode)	Δ	Δ
	G115 G116	Start point timing synchronization	Δ	-
	G117	M code output during axis traveling	Δ	-
	G122	Sub part system control I	× (P652)	□ (*7)
	G125	Control axis synchronization between part systems	○	○
	G126	Control Axis Superimposition	Δ	× (P29)
	G140	Arbitrary axis exchange control	× (P126)	○
	G141	Arbitrary axis exchange return	× (P126)	-
	G142	Reference axis arrange return	× (P126)	-
	G144	Sub part system control II	× (P652)	□ (*7)
1	G00	Positioning	Δ	Δ
	G01	Linear interpolation	○	○
	G02 G03	Circular interpolation	○	○
	G02.3 G03.3	Exponential interpolation	Δ	Δ
	G33	Thread cutting	Δ	Δ
	G34	Variable lead thread cutting	Δ	Δ
	G35 G36	Circular thread cutting	Δ	Δ
2	G16	Milling interpolation Plane selection Y-Z cylindrical plane	× (P34)	× (P34)
	G17/G18/G19	Plane selection	○	○
3	G90	Absolute value command	○	○
	G91	Incremental value command	○	○
4	G22	Barrier check ON	Δ	Δ
	G23	Barrier check OFF	○	○

Group	G code (G code type: 3)	Additional function	A	B
5	G94	Asynchronous feed (feed per minute)	○	○
	G95	Synchronous feed (feed per revolution)	Δ	Δ
6	G20	Inch command	○	○
	G21	Metric command	○	○
7	G40	Tool nose radius compensation OFF	○	○
	G41 G42	Tool nose radius compensation ON	○	○
	G46	Tool nose radius compensation ON (automatic direction identification)	○	○
9	G80	Fixed cycle cancel	○	○
	Other than G80	Fixed cycle	Δ	Δ
10	G98	Fixed cycle (Initial level return)	○	○
	G99	Fixed cycle (R point level return)	○	○
12	G54 - G59 G54.1	Workpiece coordinate system selection	○	○
13	G61	Exact stop check mode	Δ	Δ
	G61.1	High-accuracy control	□ (*3)	□ (*2)
	G62	Automatic corner override	Δ	Δ
	G63	Tapping mode	Δ	Δ
	G64	Cutting mode	□ (*3)	□ (*2)
14	G66 G66.1	User macro modal call	Δ	Δ
	G67	User macro modal call cancel	○	○
15	G68	Facing turret mirror image ON	× (P29)	× (P29)
	G69	Mirror image for facing tool posts OFF	○	○
	-(G68(*))	Facing turret mirror image ON	× (P29)	× (P29)
		Balance cut ON	× (P29)	× (P29)
	-(G69(*))	Mirror image for facing tool posts OFF	○	○
		Balance cut OFF	○	○
16	G68.1	Coordinate rotation by program ON	Δ	Δ
	G69.1	Coordinate rotation by program OFF	○	○
17	G96	Constant surface speed control ON	○	○
	G97	Constant surface speed control OFF	○	○
18	G14	Balance cut OFF	○	○
	G15	Balance cut ON	× (P29)	× (P29)
19	G12.1	Milling interpolation ON	Δ	× (P481)
	G13.1	Milling interpolation OFF	○	○
	-(G07.1,G107(*))	Cylindrical interpolation	× (P34)	× (P481)
	-(G12.1,G112(*))	Polar coordinate interpolation ON	× (P34)	× (P481)
	-(G13.1,G113(*))	Polar coordinate interpolation OFF	○	○

Group	G code (G code type: 3)	Additional function	A	B
20	G43.1	1st spindle control mode	○	○
	G44.1	2nd spindle control mode	○	○
	G47.1	Two spindles simultaneous control mode	○	○
24	G188	Dynamic M/L program changeover ON	○	○
	G189	Dynamic M/L program changeover OFF	○	○

(*) Only applies to G code lists 6 and 7

(*1) Disables the high-speed high-accuracy control II.

(*2) Enables the high-speed high-accuracy control II.

(*3) High-speed high-accuracy control II continues.

(*4) Enables the high-speed machining mode II.

(*5) Enables the high-speed high-accuracy control II in a macro program.

(*6) Enables the high-speed high-accuracy control II if G05P10000 is commanded in a macro program.

(*7) A program error (P653) will occur if G05P10000 is commanded in a sub part system.

(*8) Enables the exact stop check mode.

(*9) Exact stop check mode continues.

(2) Relationship between the high-speed high-accuracy control II and functions other than G codes
Column A: Operation when the additional function is commanded while the high-speed high-accuracy control II is enabled
Column B: Operation when the high-speed high-accuracy control II (G05P10000) is commanded while the additional function is enabled
○: The high-speed high-accuracy control II and the additional function are both enabled
Δ: The high-speed high-accuracy control II is temporarily canceled, while the additional function is enabled
X: Alarm generation (the text in parentheses refers to the number of the program error to be generated.)
-: No combination
□: Others

Additional function	A	B
SSS ON	-	○
Mirror image by parameter setting ON	-	Δ
PLC mirror image ON	-	Δ
Subprogram call (M98)	□ (*10)	□ (*11)
Timing synchronization between part systems	□ (*12)	-
Inclined axis control	-	○
T code offset	○	○
T code mirror image for facing tool posts	× (P29)	× (P29)
MTB macro	□ (*13)	□ (*14)
Macro interruption	□ (*15)	□ (*16)
PLC interruption	□ (*15)	□ (*16)
Corner chamfering/Corner R	Δ	-
Linear angle command	Δ	-
Geometric command	Δ	-
Chopping	○	○
Optional block skip	○	-

(*10) Enables the high-speed high-accuracy control II in a subprogram.

(*11) Enables the high-speed high-accuracy control II if G05P10000 is commanded in a subprogram.

(*12) Enables timing synchronization.

(*13) Enables the high-speed high-accuracy control II in a MTB program.

(*14) Enables the high-speed high-accuracy control II if G05P10000 is commanded in a MTB program.

(*15) Enables the high-speed high-accuracy control II in an interrupt program.

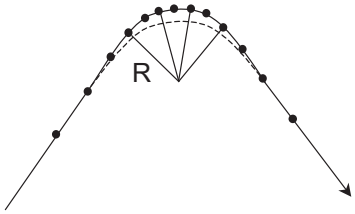
(*16) Enables the high-speed high-accuracy control II if G05P10000 is commanded in an interrupt program.

17.3.2 Acceleration Clamp Speed

Function and purpose

This function is an additional function when the high-speed high-accuracy control II mode is ON
The cutting feed clamp speed during the high-speed high-accuracy control II / III mode, when the following parameter is set to "1", is clamped so that the acceleration generated by each block movement does not exceed the tolerable value. This function clamps the speed optimally even at a section where "angle change at each block is small but entire curvature is large" such as shown below.
The tolerable acceleration value is calculated from the parameter "#1206 G1bF" and "#1207 G1btL" setting values.
(Tolerable acceleration = #1206/#1207)

Related parameter		Details
#8034	AccClampt ON	0 : Clamp the cutting speed with parameter "#2002 clamp" (*1) or the corner deceleration function. 1: Cutting speed clamp determined by acceleration reference is also executed.



Speed control by curvature

If the tool moves along the large curvature section without deceleration, a large acceleration is generated resulting in a path error by curving inward.

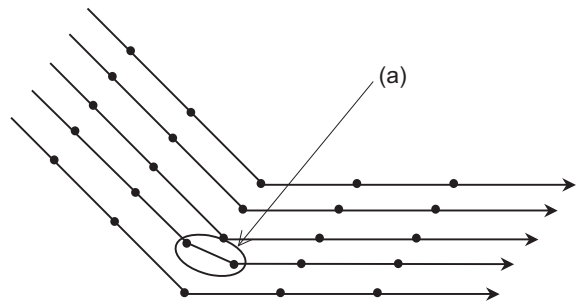
(*1) When a speed is set in "#2109 Clamp(H-precision)", clamp is executed at that speed. When the setting value is "0", clamp is executed with "#2002 clamp".

17.3.3 Corner Deceleration in High-speed Mode

Function and purpose

This function is an additional function when high-speed high-accuracy control II mode is ON.
During high-accuracy control, if the angle between the adjacent blocks in the machining program is large, this function, conventionally, automatically decelerates the machining so that the acceleration generated when passing through the corner is maintained within the tolerable value.
If a fine block is inserted at the corner section in the machining program generated with the CAM, etc., the corner passing speed will not match the periphery. This can affect the machining surface.
In the corner deceleration in the high-speed mode, even when this type of fine block is inserted, the corner will be judged from a vantage point by setting the below parameter.
The fine block is excluded at the judgment of an angle, but is not excluded from the actual movement command.

Related parameter		Details
#8036	CordecJudge	0 : Judge the corner from the angle of the neighboring block. 1 : Judge the corner from the angle of the neighboring block, excluding the minute blocks.
#8027	CorJudgeL	Exclude shorter block than this setting value.



High-speed mode corner deceleration

(a) When"#8036 CordecJudge" is set to "1", corner deceleration is realized without an influence of fine blocks.

17.3.4 Precautions on High-speed High-accuracy Control

Precautions

Common precautions on high-speed high-accuracy control I and II

(1) The machining speed may drop depending on the number of characters in one block.
If any of the above is commanded when the corresponding specification is not available on the machine, a program error (P39) will occur.

(2) Feedrate command F is clamped with the "#2110 Clamp (H-precision)" (Cutting feed clamp speed for high-accuracy control mode) set with parameter.

(3) The rapid traverse rate conforms to "#2109 Rapid(H-precision)" (Rapid traverse rate during high-accuracy control mode) set by the parameter.

(4) When "#2109 Rapid(H-precision)" (high-accuracy control mode rapid traverse rate) is set to "0", however, the movement follows "#2001 rapid" (Rapid traverse rate) set with the parameter. Also, when "#2110 Clamp (H-precision)" (Cutting feed clamp speed for high-accuracy control mode) is set to "0", the speed will be clamped with "#2002 clamp" (Cutting clamp speed) set with parameter.

(5) The automatic operation processing has priority in the high-speed high-accuracy control I/II modal, so the screen display, etc., may be delayed.

(6) The speed will decelerate once at the high-speed high-accuracy control I command (G05.1 Q1), high-speed high-accuracy control I OFF (G05.1 Q0), high-speed high-accuracy control II command (G05P10000), and high-speed high-accuracy control II OFF command (G05P0), so turn ON and OFF when the tool separates from the workpiece.

(7) When carrying out high-speed high-accuracy control I/II operation during tape mode, the machining speed may be suppressed depending on the program transmission speed and the number of characters in one block.

(8) If "#1205 G0bdcc" (G0 acceleration/deceleration before interpolation) is set to "1", the value set with the parameter "#2224 SV024" (in-position detection width) will be used as the in-position width. "#2077 G0inps" (G0 in-position width) and the ",I" command (programmable in-position check) are disabled.

Precautions on high-speed high-accuracy control I

(1) Command G05.1Q0 after turning the nose R compensation OFF. If G05.1Q0 is commanded without turning the nose R compensation OFF, the program error (P29) will occur.

(2) G05.1Q1 and G05.1Q0 are independent commands. If a sequence number other than "N" is commanded, the program error (P33) will occur.

(3) The program error (P33) will occur if the G05.1 command block does not contain a Q command.

(4) If the high-speed high-accuracy control I command is issued in the high-speed high-accuracy control II modal, a program error (P34) will occur.

Precautions on high-speed high-accuracy control II

(1) While high-speed high-accuracy control II is valid, the following variable commands or operation commands can be designated following the axis address. When other variable commands or operation commands are issued, high-speed high-accuracy control II is canceled temporarily.

(a) Referencing common variables or local variables

Common variables or local variables can be referenced (example: X#500, Y#1, Z##100, A#[#101], etc.).

(b) Four basic arithmetic rule

Four basic arithmetic rule (+, -, *, /) operations are available, and also the operation priority can be designated using parentheses ( ) ([#500 + #501] * #502, etc.).

(2) G05P10000 and G05P0 are independent commands. If a sequence number other than "N" is commanded, the program error (P33) will occur.

(3) The program error (P33) will occur if the G05 command block does not contain a P command.

(4) Fairing function is valid for the continuous linear command G01. Fairing is not possible in the case below.

(5) When using the high-speed high-accuracy control II mode, set parameter "#1572 Cirorp/Bit0" to "1" to eliminate the speed fluctuation at the seams between the arc and the straight line, or between arcs.

(6) A program error (P33) will occur if the geometric command is issued during the high-speed high-accuracy control II.

(7) If the high-speed high-accuracy control II command is issued in the high-speed high-accuracy control I modal, a program error (P34) will occur.

17.4 Machining Condition Selection I ; G120.1,G121

Function and purpose

After initializing the machining condition parameter groups with the machining condition selection I function, the machining condition parameter groups can be switched by G code command.
Switching is also possible on the machining condition selection screen. In that case, however, the machining conditions selected on the screen are applied to all part systems.

Command format

G120.1 Pp Qq ; ... Machining condition selection I

P	Machining purpose 0: Reference parameter 1: Usage 1 2: Usage 2 3: Usage 3
Q	Condition 1: Condition 1 2: Condition 2 3: Condition 3 When omitted, Q1 will be applied

G121; ... Machining condition selection I cancel

Detailed description

(1) G120.1 and G121 commands are unmodal commands of G code group 0.

(2) Switching of the machining condition parameter group using the G120.1 or G121 command is applied only to the commanded part system.

(3) Command G120.1 and G121 in an independent block. If not, a program error (P33) will occur.

(4) Address P in G120.1 command cannot be omitted. If omitted, a program error (P33) will occur.

(5) Address Q in G120.1 command can be omitted. If omitted, it will be handled as "Q1 (condition 1)" is commanded.

(6) When address P and Q in G120.1 command is commanded with a decimal point, the digit after the decimal point is ignored.

(7) If other than "0 to 3" is set to address P in G120.1 command or other than "1 to 3" is set to address Q, a program error (P35) will occur.

(8) When address P is set to "0" and address Q is omitted or set between "1" and "3" in G120.1 command, it will be switched to the reference parameter.

(9) It will be switched to the machining condition parameter group selected in "Machining cond" screen by G121 command.

(10) When the emergency stop and reset (reset 1, reset 2, and reset & rewind) are performed while running the machining program whose machining condition parameter group is switched by G120.1 command, it will be switched to the selected condition parameter group machining in "Machining cond".

(11) Because the parameters are switched after being decelerated by G120.1 and G121 commands, the workpiece may be damaged. Make sure to keep the tool away from the workpiece when commanding G120.1 and G121.

(12) When the machining condition parameter group is switched by G120.1 command more than once, the parameter group commanded last becomes valid.

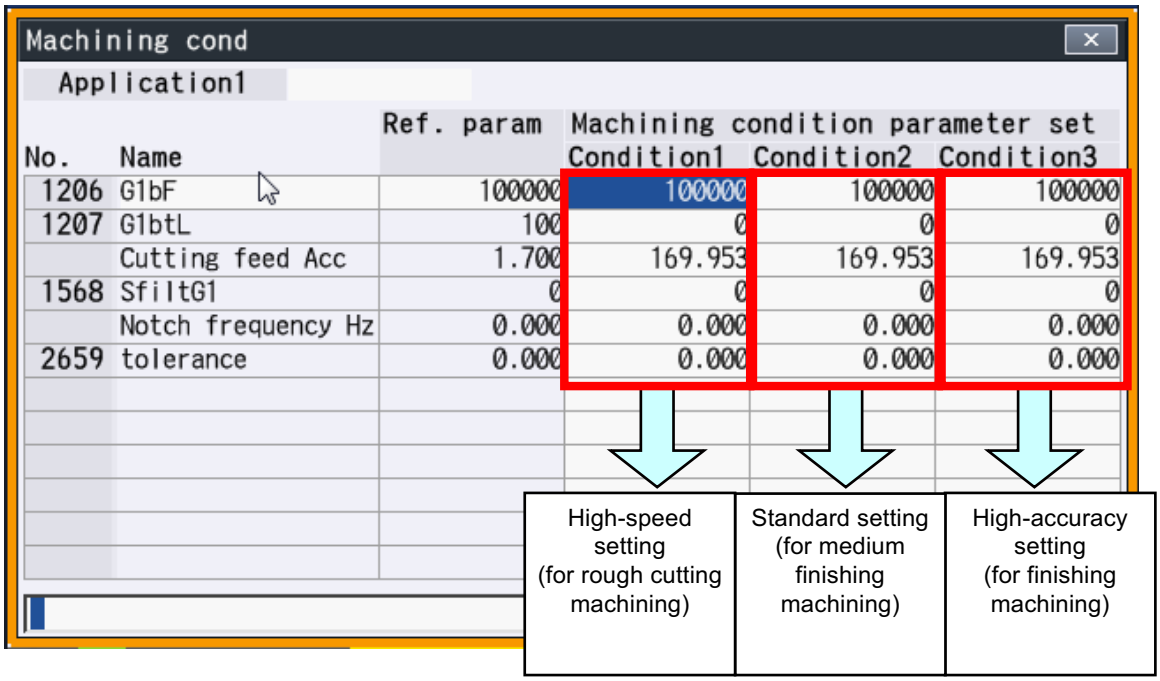
(13) It is switched to the selected machining condition parameter group in the "Machining cond" screen by program end (M02 and M30).

(14) If G120.1 and G121 are commanded without initializing the machining condition parameter group, a program error (P128) will occur.

(15) A program error (P49) will occur if restart search is attempted for a block after G120.1 or G121.

Program example

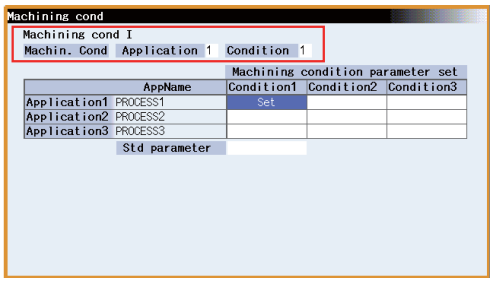
"Machining cond" (setting) screen
The machining condition parameter groups are switched by the setting whether the tolerance control is valid or invalid.



(1) When "machin usage 1" and "condition 1" from the machining condition parameter group are selected in "Machining cond" (selecting) screen before running the program.
(The following machining programs are assuming "I" is X axis, "J" is Z axis and "K" is Y axis.)

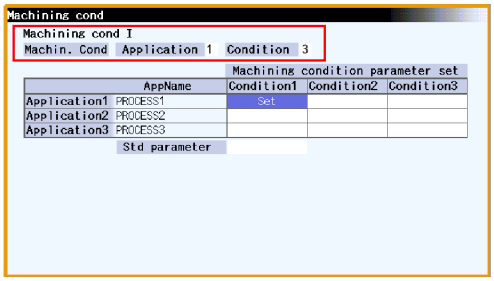
```
N1      G28 U0;
N2      G28 W0 V0;
N3      G54 G00 Z2. Y2.;
N4      T0101;
N5      X20.;
N6      M33 S10000;
N7      G01 X10. F3000;
N8      F2000;
N9      G05 P10000;
N10     G01 Z2.099 Y1.99;
N11     Z2.199;
:
N1499   G05 P0;
N1500   G28 U0;
N1501   G28 W0 V0;
N1502   M35;
N1503   G120.1 P1 Q3;
N1504   G54 G00 Z2. Y2.;
N1505   T0101;
N1506   X20.;
N1507   M33 S10000;
N1508   G01 X7. F3000;
N1509   F1200;
N1510   G05 P10000;
N1511   G01 Z2.099 Y1.99;
N1512   Z2.199;
:
N2999   G05 P0;
N3000   G28 U0;
N3001   G28 W0 V0;
N3002   M35;
N3003   M30;
```

Operate with the machining condition parameter group (machining usage 1/condition 1)

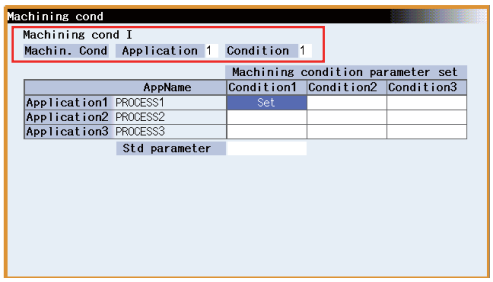


The machining condition parameter groups are switched.

Operate with the machining condition parameter group (machining usage 1/condition 3)



Return to the selected machining condition parameter group in "Machining cond" (selecting) screen at the program end.



Relation with other functions

(1) G code modals which cause the program error when commanding G120.1 and G121 are listed below.

G code	Function	Program error when G120.1 and G121 are commanded
G02.3, G03.3	Exponential interpolation	P128
G07.1	Cylindrical interpolation (only 6 and 7 in G code list)	P128
G12.1	Polar coordinate interpolation (only 6 and 7 in G code list)	P128
G10	Parameter input by program	P421
	Tool compensation input by program	
G33	Thread cutting	P128
G41, G42	Nose R compensation	P128
G66, G66.1	User macro (modal call A, B)	P128
G73/G74/G75/G76/G76.1/G76.2/G80/G81/G82/G83/G83.1/G83.2/G84/G84.1/G84.2/G85/G87/G88/G88.1/G89	Fixed cycle	P33(When G120.1 command is issued)
		P128(When G121 command is issued)

Precautions

(1) Because the parameters are switched after being decelerated once G120.1 or G121 is commanded, the workpiece may be damaged. Make sure to keep the tool away from the workpiece when commanding G120.1 and G121.

(2) For the parameters "#8033 Fairing ON" and "#8090 SSS ON", the switched machining condition parameter group is effective only after it has been switched on the machining condition selection screen.

(3) It is switched to the reference parameter by turning the power ON again.

(4) The machining condition parameter cannot be switched on the "Machining cond" (selecting) screen and cannot be set on the "Machining cond" (setting) screen during the automatic operation.

(5) When the machining condition parameter group is switched by the G120.1 command in the machining program during displaying the "Machining cond" (selecting) screen, the selected machining condition parameter being displayed will not be switched unless the display screen is transited to the other screen once.

(6) When G120.1 and G121 are commanded, parameters are switched when smoothing for NC axes in all part systems become "0".

(7) The machining condition parameter group neither set the parameter setting from the program by G10 command nor read the parameters by system variables (from #100000).

(8) When the machining condition parameter group is switched, the same values are used for all NC axes which belong to the switched part system to the parameter "#2010 Feed forward gain".

18

Advanced Multi-Spindle Control Function

18.1 Spindle Synchronization

Function and purpose

In a machine having two or more spindles, this function controls the rotation speed and phase of one spindle (reference spindle) in synchronization with the rotation of the other spindle (synchronized spindle).
This function provides, for example, an effect that re-grasps the workpiece grasped by the 1st spindle to the 2nd spindle while maintaining the rotation speed of the 1st spindle to reduce the cycle time by the 1st spindle deceleration time and the 2nd spindle acceleration time in the next process during re-grasping.
Furthermore, this function carries out turning or phase control while grasping both edges of a longer workpiece using the 1st and 2nd spindles, preventing a twist or bow from occurring in the workpiece under machining and enabling the machining accuracy.
The spindle synchronous multi-step acceleration/deceleration of the reference spindle is applied to the acceleration/deceleration of the spindle-synchronization relation spindle under spindle synchronization.
This function is available when the G code list is set to 2, 3, 4, or 5 in the MTB specifications (parameter "#1037 cmdtyp").
The following control methods are available. Which mode is valid depends on the MTB specifications (parameter "#1300 ext36/bit7"). This section describes spindle synchronization control I that is executed with G commands.

The spindle synchronization I
The designation of the synchronized spindle and start/stop of the synchronization are executed by commanding G codes in the machining program.

The spindle synchronization II
The selections of the synchronized spindle and synchronization start, etc., are controlled from PLC based on the MTB specifications. Refer to the instruction manual issued by the MTB for details.

Common setting for the spindle synchronization control I and II

When the spindle synchronization control is carried out, the followings must be set.

- Chuck close
- Error temporary cancel
- Phase monitor
- Multi-speed acceleration/deceleration

For details, refer to "18.1.2 Precautions for Using Spindle Synchronization Control".

18.1.1 Spindle Synchronization I; G114.1

Function and purpose

With the spindle synchronization I, the designation of the spindle and start/stop of the synchronization are executed by commanding G codes in the machining program.
The other spindle synchronization functions (Spindle synchronization I/ Spindle synchronization II/ Tool spindle synchronization IA/ Tool spindle synchronization IB/ Tool spindle synchronization II/ Spindle superimposition control) cannot be commanded while this function is running. Also, this function cannot be commanded while the above functions are being executed. In these cases an operation error (M01 1005) will occur.
However, when the Multiple spindle synchronization set control function is enabled, multiple spindle synchronization functions can be commanded simultaneously. The validity of this function depends on the MTB specifications (parameter "#1440 multi_sp_syn"). For details, refer to "18.5 Multiple Spindle Synchronization Set Control".

Command format

Spindle synchronization control start command

```
G114.1 H__ D__ R__ A__ ;
```

H	Reference spindle selection
D	Synchronized spindle selection
R	Synchronized spindle phase shift amount
A	Spindle synchronization acceleration/deceleration time constant

Spindle synchronization control ON (G114.1) command designates the reference spindle and synchronized spindle, and synchronizes the two designated spindles. By commanding the synchronized spindle phase shift amount, the phases of the reference spindle and synchronized spindle can be aligned.

Address	Meaning	Command range (unit)	Remarks
H	Specifies the reference spindle. Select the number or name of the spindle to be used as the reference spindle from the two spindles to be synchronized. (*1)	For spindle number: 1 to n (n: Maximum number of available spindles) For spindle name: 1 to 9	♦If a value exceeding the command range or spindle No. without specifications is commanded, a program error (P35) will occur. ♦If there is no command, a program error (P33) will occur. ♦If an analog-connected spindle is commanded, a program error (P700) will occur. (*2)
D	Specifies the synchronous spindle. Select the number or name of the spindle to be synchronized with the reference spindle from the two spindles to be synchronized. (*1)	For spindle number: 1 to n or -1 to -n (n: Maximum number of available spindles) For spindle name: 1 to 9 or -1 to -9	♦If a value exceeding the command range is commanded, a program error (P35) will occur. ♦If there is no command, a program error (P33) will occur. ♦If the same spindle as that commanded for the reference spindle selection is designated, a program error (P33) will occur. ♦The rotation direction of the synchronized spindle in respect to the reference spindle is commanded with the D sign. ♦If an analog-connected spindle is commanded, a program error (P700) will occur. (*2)

Address	Meaning	Command range (unit)	Remarks
R	Spindle synchronization phase shift amount Set the shift amount from the synchronous spindle's reference position (one rotation signal).	0 to 359.999 (°) or 0 to 359999 (°* 10^{-3})	♦If a value exceeding the command range is commanded, a program error (P35) will occur. ♦The commanded shift amount will be effective in the clockwise direction of the reference spindle. ♦Minimum resolution of commanded shift amount Semi-close case (Gear ratio: 1:1 only) 360/4096 [°] Full-close case (360/4096) * K [°] (K: Gear ratio of spindle and encoder) ♦If there is no R command, phase alignment will not be carried out.
A	Spindle synchronization acceleration/deceleration time constant Command the acceleration/deceleration time constant for when the spindle synchronization command rotation speed changes. (Command this to accelerate or decelerate at a speed slower than the time constant set in the parameters.)	0.001 to 9.999 (s) or1 to 9999 (ms)	♦If a value exceeding the command range is commanded, a program error (P35) will occur. ♦If the commanded value is smaller than the acceleration/deceleration time constant set with the parameters, the value set in the parameters will be applied.

(*1) There are two spindle command methods: spindle number method and spindle name method.
Command with the spindle names, only when all spindles are set with the spindle name parameter (#3077 Sname) (from 1 to 9). For others, command with the spindle number. These settings depend on the MTB specifications.

(*2) The available spindle type and connection method depend on the specifications of your machine tool.

Canceling Spindle synchronization

G113 ;	**(When the spindle synchronization status of all sets is canceled)**

G113 D__ ;	**(When the specifications for multiple sets of spindle synchronizations is valid)**

Note

(1) An axis that involves any travel cannot be put in the same block as the Spindle synchronization cancel command. Doing so causes the program error (P33) when the cancel command is issued, which causes automatic operation to pause.

Spindle synchronization cancel (G113) cancels the synchronous state of the two spindles rotating in synchronization with the spindle synchronization command.

Address	Meaning	Command range (unit)	Remarks
D	Select a synchronous spindle. Commands the number of the spindle to be synchronized with the reference spindle from the two spindles to be synchronized.	For spindle number: 1 to n (n: Maximum number of available spindles) For spindle name: 1 to 9	♦If a value exceeding the command range is commanded, a program error (P35) will occur. ♦If there is no command, a program error (P33) will occur. ♦If the same spindle as that commanded for the reference spindle selection is designated, a program error (P33) will occur. ♦If a spindle that is not under spindle synchronization control is commanded, it causes an operation error (M01 1005). ♦If the D address is omitted, it causes a cancelation for the synchronization status of all spindle synchronization controls that are in process.

Detailed description

Rotation speed and rotation direction

(1) The rotation speed and rotation direction of the reference spindle and synchronized spindle during spindle synchronization are the rotation speed and rotation direction commanded for the reference spindle. Note that the rotation direction of the synchronized spindle can be reversed from the reference spindle through the program.

(2) The reference spindle's rotation speed and rotation direction can be changed during spindle synchronization.

(3) If spindle stop is commanded for the synchronized spindle during spindle synchronization, the synchronized spindle rotation will stop.

(4) The rotation speed command (S command) and constant surface speed control are invalid for the synchronized spindle during spindle synchronization. Note that the modal is updated, so these will be validated when the spindle synchronization is canceled.

(5) The constant surface speed can be controlled by issuing a command to the reference spindle even during spindle synchronization.

Rotation synchronization

(1) When rotation synchronization control (command with no R address) is commanded with the G114.1 command, the synchronized spindle rotating at an arbitrary rotation speed will accelerate or decelerate to the rotation speed commanded beforehand for the reference spindle, and will enter the rotation synchronization state.

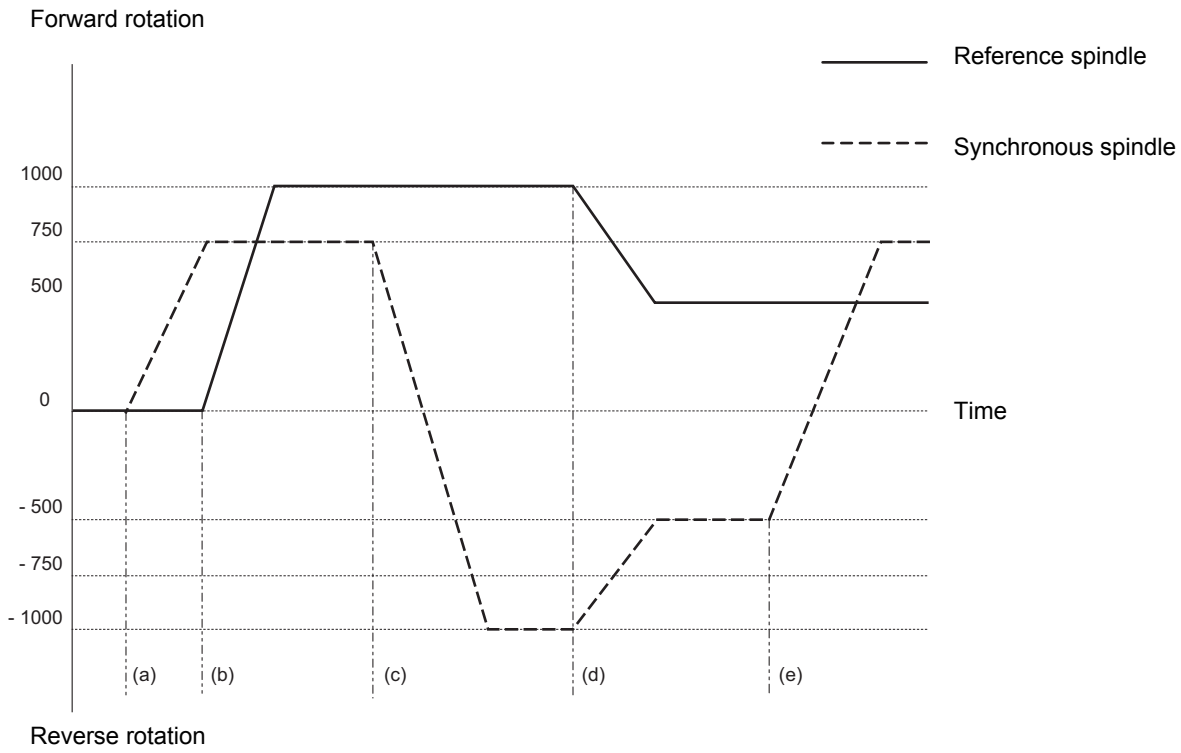
(2) If the reference spindle's commanded rotation speed is changed during the rotation synchronization state, acceleration/deceleration will be carried out while maintaining the synchronization state following the spindle acceleration/deceleration time constants set in the parameters, and the commanded rotation speed will be achieved.

(3) In the rotation synchronization state, the reference spindle can be controlled at a constant surface speed even when two spindles are grasping one workpiece.

(4) The following type of operation will take place.

M23 S2=750 ;	Forward rotate 2nd spindle (synchronized spindle) at 750 r/min (speed command) (a)
:	
M03 S1=1000 ;	Forward rotate 1st spindle (reference spindle) at 1000 r/min (speed command) (b)
:	
G114.1 H1 D-2. ;	Synchronize 2nd spindle (synchronized spindle) to 1st spindle (reference spindle) with reverse run. (c)
:	
S1=500 ;	Change 1st spindle (reference spindle) rotation speed to 500 r/min. (d)
:	
G113;	Cancel the spindle synchronization. (e)

<Operation>



Phase synchronization

(1) When phase synchronization (command with R address) is commanded with the G114.1 command, the synchronized spindle rotating at an arbitrary rotation speed will accelerate or decelerate to the rotation speed commanded beforehand for the reference spindle, and will enter the rotation synchronization state. Then, the phase is aligned so that the rotation phase commanded with the R address is reached, and the phase synchronization state is entered.

(2) If the reference spindle's commanded rotation speed is changed during the phase synchronization state, acceleration/deceleration will be carried out while maintaining the synchronization state following the spindle acceleration/deceleration time constants set in the parameters, and the commanded rotation speed will be achieved.

(3) In the phase synchronization state, the reference spindle can be controlled at the constant surface speed even when two spindles are grasping one workpiece.

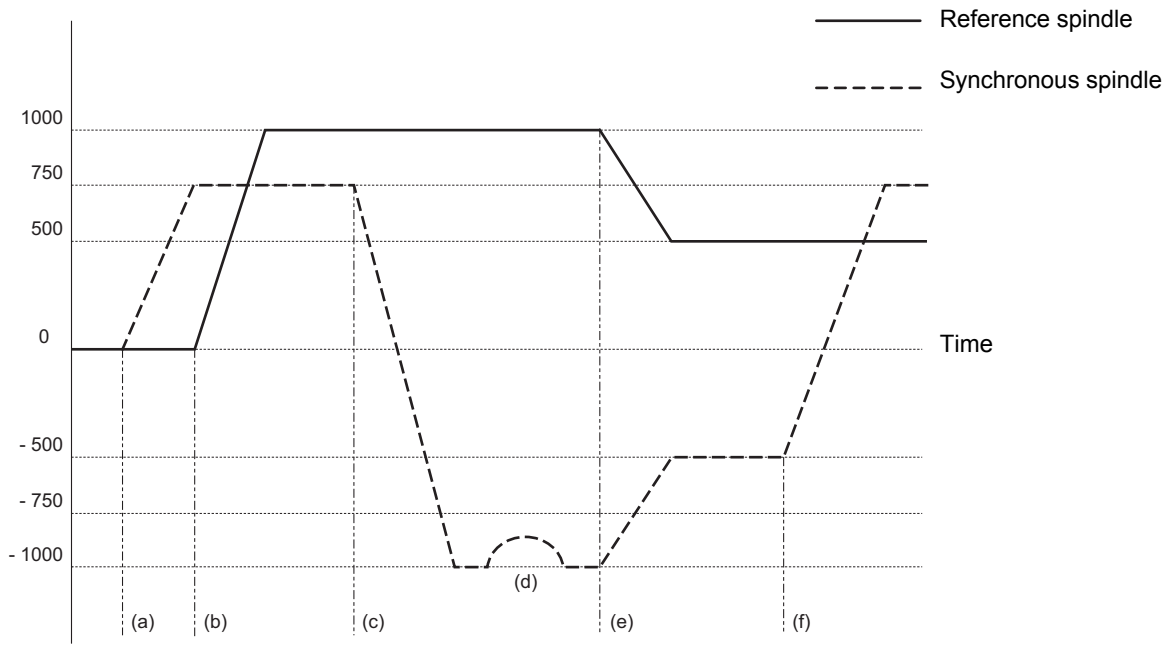
(4) The following type of operation will take place.

M23 S2=750 ;	Forward rotate 2nd spindle (synchronous spindle) at 750 r/min (speed command). (a)
:	
M03 S1=1000 ;	Forward rotate 1st spindle (reference spindle) at 1000 r/min (speed command) (b)
:	
G114.1 H1 D-2 R0;	Synchronize the 2nd spindle (synchronous spindle) with the 1st spindle (reference spindle) by reverse run. (c) Shift phase of synchronous spindle by R command value. (d)
:	
S1=500 ;	Change 1st spindle (reference spindle) rotation speed to 500 r/min. (e)
:	
G113;	Cancel the spindle synchronization. (f)

<Operation>

Rotation speed (r/min)

Forward rotation



Reverse rotation

<Note>

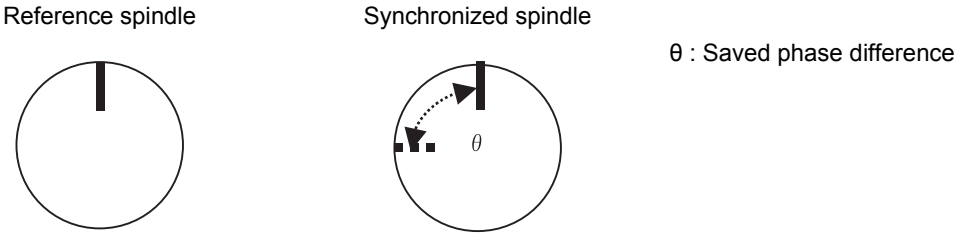
- Acceleration/Deceleration at the phase synchronization is the step synchronization method when "#3130 syn_spec/bit1" = "0", the multi-step acceleration/deceleration method when "#3130 syn_spec/bit1" = "1".

Spindle synchronization phase shift amount calculation function

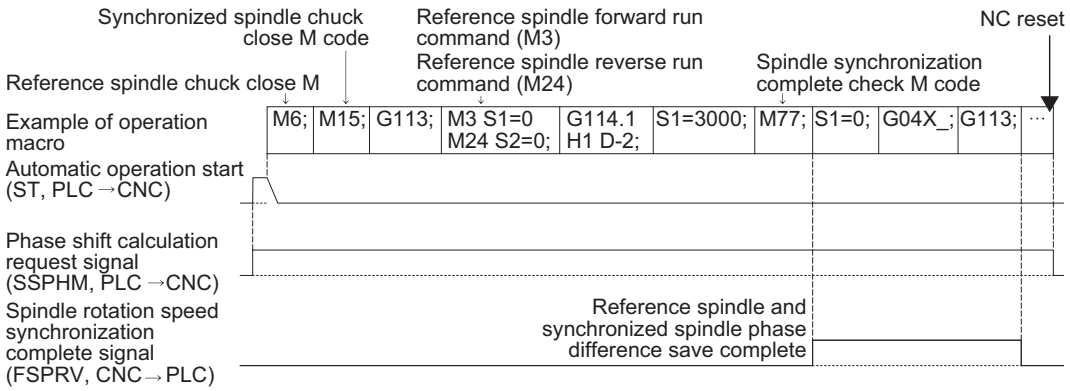
The spindle phase shift amount calculation function obtains and saves the phase difference of the reference spindle and synchronized spindle by turning the PLC signal ON when the phase synchronization command is executed. When the phase is positioned to the automatically saved phase difference before executing the phase synchronization control command, phases can be aligned easier when re-grasping profile materials.

[Saving the reference spindle and synchronized spindle phase difference]

(1) Set a profile material in the reference spindle.
(2) Set the profile material in the synchronized spindle.
(3) Turn the phase shift calculation request signal (SSPHM) ON.
(4) Input a rotation command, with 0 speed, for the reference spindle and synchronized spindle.
<Example> M3 S1=0 M24 S2 = 0;
(5) Execute the rotation synchronization signal (with no R address command).
<Example> G114.1 H1 D-2;
(6) Rotate the reference spindle at the speed actually used when re-grasping.
<Example> S1 = 3000;
(7) Check that the phase difference has been saved by looking at the spindle speed synchronization complete signal.
(8) Stop both spindles.
(9) Turn the phase shift calculation request signal OFF.



<Example of operation>

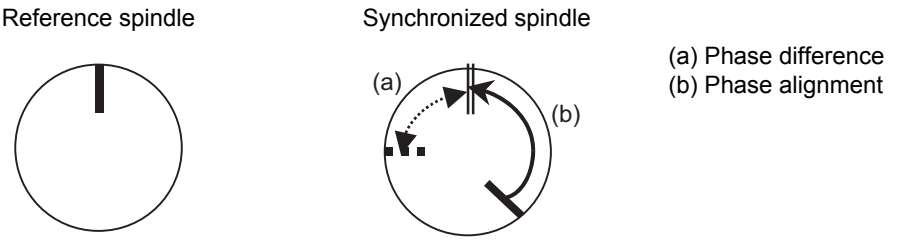


[Automatic phase alignment of reference spindle and synchronized spindle]

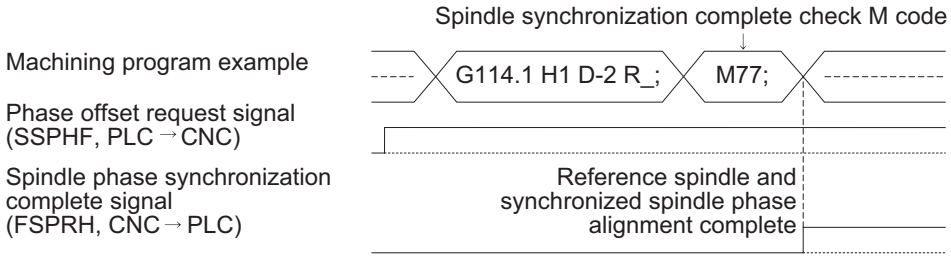
(1) Turn the phase offset request signal ON.

(2) Issue the phase synchronization command (with R command).
<Example> G114.1 H1 D-2 R0;

(3) The phase is aligned by offsetting the phase synchronization command by the phase difference obtained with the spindle synchronization phase shift calculation function. The state in which the synchronized spindle phase shift amount designation R value is 0 is the same as the reference state (state obtained with phase shift calculation request signal).



<Example of operation>



Multi-step acceleration/deceleration

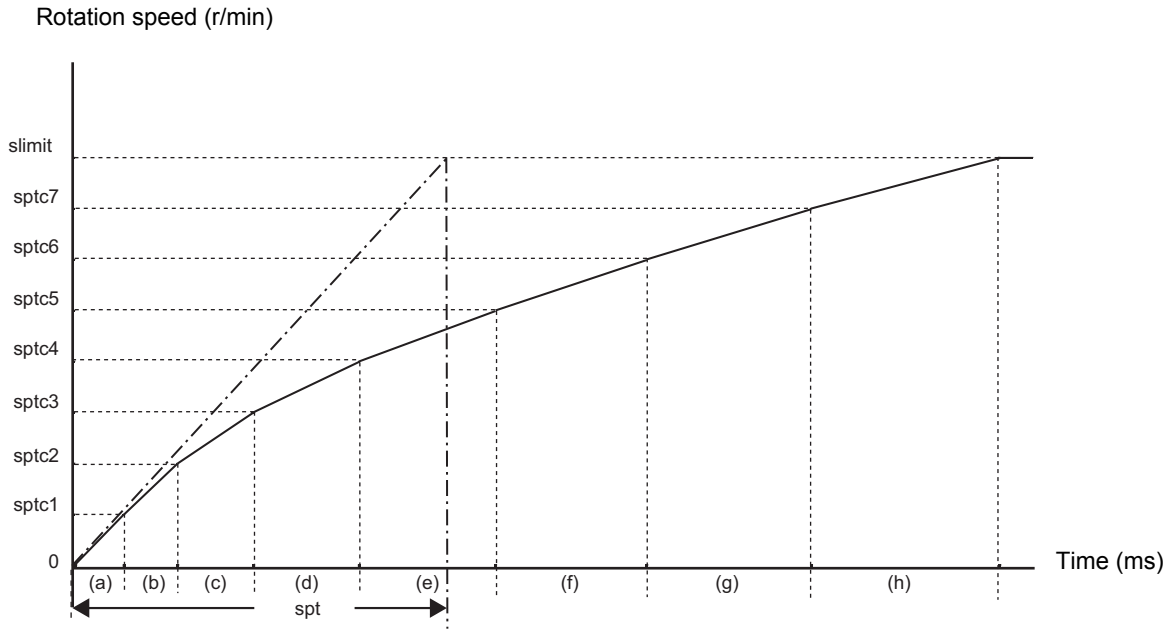
Acceleration/deceleration time constants for up to eight steps can be selected according to the spindle rotation speed for the acceleration/deceleration during spindle synchronization.
The acceleration/deceleration in each step is as follows.

Time required from minimum rotation speed to maximum rotation speed in each step
= [Time constant without multi-step acceleration/deceleration] * [magnification of time constant in each step] * [Rate of rotation speed width in each step respect to rotation speed width up to limit rotation speed]

Note

(1) When the "A" address is designated at G114.1 command, the time is obtained with the "A" address instead of "spt" in the formula below.

Time required to rotate to sptc 1 set rotation speed from stopped state (a)
= spt * sptc1 / slimit
Time required to reach sptc 2 set rotation speed from sptc 1 (b)
= spt * spdiv1 * (sptc2 - sptc1) / slimit
Time required to reach sptc 3 set rotation speed from sptc 2 (c)
= spt * spdiv2 * (sptc3 - sptc2) / slimit
Time required to reach sptc 4 set rotation speed from sptc 3 (d)
= spt * spdiv3 * (sptc4 - sptc3) / slimit
Time required to reach sptc 5 set rotation speed from sptc 4 (e)
= spt * spdiv4 * (sptc5 - sptc4) / slimit
Time required to reach sptc 6 set rotation speed from sptc 5 (f)
= spt * spdiv5 * (sptc6 - sptc5) / slimit
Time required to reach sptc 7 set rotation speed from sptc 6 (g)
= spt * spdiv6 * (sptc7 - sptc6) / slimit
Time required to reach slimit set rotation speed from sptc 7 (h)
= spt * spdiv7 * (slimit - sptc7) / slimit



To decrease the number of acceleration/deceleration steps during spindle synchronization, set one of the following for the unnecessary step.

- Magnification for time constant changeover speed (spdiv7 to spdiv1) = 0 (or 1)
- Spindle synchronous multi-step acceleration/deceleration changeover speed (sptc7 to sptc1) = Limit rotation speed (slimit) or higher

Relationship with other functions

Spindle orientation signal (ORC)

- The spindle orientation is carried out with the spindle orientation command for the reference spindle while the spindle synchronization status remains kept. The spindle orientation command is ignored for the synchronized spindle. The multi-step orientation command or indexing command is also enabled.
- The spindle orientation command (ORC) for the reference spindle in the C axis mode is ignored during spindle synchronization. Also, the spindle position control command (C axis mode switch command) under spindle orientation is ignored.

The spindle orientation signal operation depends on the MTB specifications. Refer to the instruction manual issued by the MTB for details.

Switching the spindle gear

- Gear switching is enabled while the reference spindle is in the spindle mode.
- Gear switching is disabled while the reference spindle is in the C axis mode or in process of spindle orientation. Switching to the C axis mode or the spindle orientation cannot be carried out during gear switching. After gear switching has been completed, the mode is switched to the C axis mode.

Spindle superimposition control

- If the reference spindle under spindle superimposition or the superimposed spindle is switched to the C axis mode, it causes an operation error (M01 1026).
- Also, if the spindle superimposition command is issued to the spindle in the C axis mode, it causes an operation error (M01 1026).
- The alarm can be reset by canceling the spindle superimposition cancel command or the C axis mode.

Tool spindle synchronization IA/IB (spindle-spindle polygon)

- If the rotary tool axis (spindle) under tool spindle synchronization control or the workpiece axis (spindle) is switched to the C axis mode, it causes an operation error (M01 1026).
- Also, if the rotary tool axis command or workpiece axis command is issued to the spindle in the C axis mode, it causes an operation error (M01 1026). The alarm can be reset by canceling the tool spindle synchronization cancel command or the C axis mode.

Tool spindle synchronization IC (spindle-NC axis polygon)

- If the workpiece axis (spindle) under tool spindle synchronization control is switched to the C axis mode, it causes an operation error (M01 1026).
- Also, if the workpiece axis command of tool spindle synchronization IC is issued to the reference spindle in the C axis mode, it causes an operation error (M01 1026). The alarm can be reset by canceling the tool spindle synchronization cancel command or the C axis mode.
- If necessary, the C axis under spindle synchronization control can be set to a tool axis (NC axis). Make sure to switch to the C axis mode before issuing the tool spindle synchronization IC command.

Spindle override

- The reference position return operation at C axis mode switch command for the reference spindle and the spindle override in the C axis mode are invalid.
- The cutting feed override or rapid traverse override is valid in the C axis mode.
- The spindle override is invalid while the reference spindle is in process of spindle orientation or spindle indexing.

Guide bushing spindle synchronization

- The spindle synchronization control (for both the reference and synchronized spindles) using the reference spindle under guide bushing spindle synchronization is enabled.
  However, the spindle position control, spindle orientation control, spindle forward run indexing, or spindle reverse run indexing is enabled for the reference spindle under spindle synchronization control only when the reference spindle under guide bushing spindle synchronization is commanded to the reference spindle under spindle synchronization control.
  (If the reference spindle under guide bushing spindle synchronization is commanded to the synchronized spindle under spindle synchronization control, it causes an operation error (M01 1026) or operation error (M01 1005)).

Spindle clamp speed setting

- The maximum clamp rotation speed specified with the address S following G92 is valid for the reference or synchronized spindle.
- The minimum clamp rotation speed specified with the address Q following G92 is valid for the reference spindle, but invalid for the synchronized spindle. If the rotating spindle is set to the synchronized spindle at the minimum clamp rotation speed, the minimum rotation speed clamp is canceled, and the spindle rotates at the commanded rotation speed. If the spindle synchronization control state is canceled, the minimum clamp rotation speed is enabled.

Synchronous tapping cycle

- The synchronous tap spindle cannot be commanded as the reference or synchronous spindle of the spindle synchronization I. The operation error (M01 1007) will occur, which causes automatic operation to pause.
- You cannot command a synchronous tapping that uses the reference spindle or synchronous spindle of spindle synchronization I. The operation error (M01 1139) will occur, which causes automatic operation to pause.

Precautions

(1) The spindle rotating with spindle synchronization control will stop when emergency stop is applied.

(2) The rotation speed clamp during spindle synchronization will follow the smaller clamp value set for the reference spindle or synchronized spindle.

(3) Orientation of the reference spindle and synchronized spindle is not possible during the spindle synchronization mode. To carry out orientation, cancel the spindle synchronization mode first.

(4) The rotation speed command (S command) is invalid for the synchronized spindle during the spindle synchronization mode. Note that the modal will be updated, so this command will be validated when spindle synchronization is canceled.

(5) The constant surface speed control is invalid for the synchronized spindle during the spindle synchronization mode. Note that the modal will be updated, so this will be validated when spindle synchronization is canceled.

(6) The rotation speed command (S command) and constant surface speed control for the synchronized spindle will be validated when spindle synchronization is canceled. Thus, the synchronized spindle may carry out different operations when this control is canceled.

(7) If the phase difference is not obtained with the phase shift calculation request signal and the phase synchronization command is executed by turning the phase offset request signal ON, the phase shift amount will not be calculated correctly.

(8) The spindle Z phase encoder position parameter "#3035 sppst" is invalid when using the spindle synchronization phase shift amount calculation function. (It is ignored.)
The spindle Z phase encoder position parameter "#3035 sppst" is valid when the phase offset request signal is OFF.

(9) If the phase synchronization command (command with R address) is issued while the phase shift calculation request signal is ON, an operation error (1106) will occur.

(10) If the phase shift calculation request signal is ON and the reference spindle or synchronized spindle is rotation while rotation synchronization is commanded, an operation error (1106) will occur.

(11) If the phase synchronization command R0 (<Ex.> G114.1 H1 D-2 R0) is commanded while the phase offset request signal is ON, the reference spindle and synchronized spindle phases will be aligned to the phase difference of the reference spindle and synchronized spindle saved in the NC memory.

(12) If a value other than the phase synchronization command R0 (<Ex.> G114.1 H1 D-2 R100) is commanded while the phase offset request signal is ON, the phase difference obtained by adding the value commanded with the R address command to the phase difference of the reference spindle and synchronized spindle saved in the NC memory will be used to align the reference spindle and synchronized spindle.

(13) The phase offset request signal will be ignored when the phase shift calculation request signal is ON.

(14) The phase difference of the reference spindle and synchronous spindle saved in the NC is valid only when the phase shift calculation signal is ON and for the combination of the reference spindle selection (H_) and synchronous spindle selection (D_) commanded with the rotation synchronization command (no R address). For example, if the reference spindle and synchronized spindle phase difference is saved as "G114.1 H1 D-2 ;", the saved phase difference will be valid only when the phase offset request signal is ON and "G114.1 H1 D-2 R*** ;" is commanded. If "G114.1 H2 D-1 R*** ;" is commanded in this case, the phase shift amount will not be calculated correctly.

(15) The reference spindle and synchronized spindle phase difference saved in the NC is held until the next spindle synchronous phase shift calculation (rotation synchronization command is completed with phase shift calculation request signal ON).

(16) When the spindle synchronization commands are being issued with the PLC I/F method (#1300 ext36/bit7 OFF), a program error (P610) will occur if the spindle synchronization is commanded with G114.1/G113.

(17) Always set the "Chuck close". If the "Chuck close" is not set, an excessive load may be applied on the machine or an alarm may occur.

(18) When the spindle-mode servo is used for the reference spindle or the synchronized spindle, the spindle parameter "#13003 SP003" (PGS) and spindle-mode servo parameter "#52203 SV003" (PGN) must be set to the same value between the reference and synchronized spindles. (These settings depend on the MTB specifications.)

Cautions on programming

(1) To enter the rotation synchronization mode while the reference spindle and synchronized spindle are chucking the same workpiece, turn the reference spindle and synchronized spindle rotation commands ON before turning the spindle synchronization mode ON.

$1 (1st part system)		$2 (2nd part system)	
:		:	
M6;	1st spindle chuck close	:	
:		M25 S2=0;	2nd spindle stops at S=0
:		:	
!2;		!1;	Timing synchronization operation between part systems
M5 S1=0;	1st spindle stops at S=0	M15;	2nd spindle chuck close
:		M24;	2nd spindle rotation command ON
M3;	1st spindle rotation command ON	:	
!2;		!1;	Timing synchronization operation between part systems
:		G114.1 H1 D-2;	Rotation synchronization mode ON
:		:	
S1=1500;	Synchronous rotation at S=1500	:	
:		:	
S1=0;	Both spindles stop		
G113	Synchronization mode cancel		

(2) To chuck the same workpiece with the reference spindle and synchronized spindle in the phase synchronization mode, align the phases before chucking.

$1 (1st part system)		$2 (2nd part system)	
:		:	
M6;	1st spindle chuck close	:	
:		:	
M3 S1=1500;	1st spindle rotation command ON	:	
:		G114.1 H1 D-2 R0;	Phase synchronization mode ON
:		:	
:		M24;	2nd spindle rotation command ON
:		:	
:		M15;	2nd spindle chuck close (*1)
:		:	

(*1) Close the chuck after confirming that the spindle phase synchronization complete signal has turned ON (phase alignment complete).

⚠ CAUTION

⚠ **Do not make the synchronized spindle rotation command OFF with one workpiece chucked by the reference spindle and synchronized spindle during the spindle synchronization mode. Failure to observe this may cause the synchronized spindle stop, and hazardous situation.**

18.1.2 Precautions for Using Spindle Synchronization Control

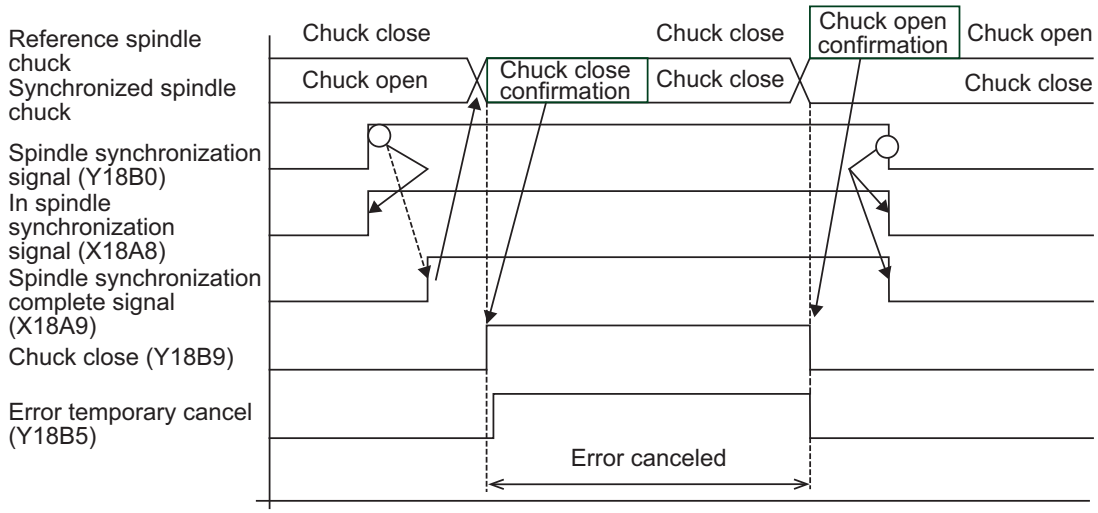
Precautions

Some PLC signals must be set when spindle synchronization control I or II is used. If these signals are not set, an excessive load or an alarm may occur. Refer to the instruction manual issued by the MTB for details. In this section, each function and the signal are explained.

Chuck close signal

The synchronized spindle side carries out droop compensation while the chuck is opened, and aligns itself with the reference spindle. However, when the chuck is closed, the droop compensation is added, and the synchronization error with the base increases. Droop compensation is prevented with the chuck close signal and the position where the chuck is grasped is maintained with position compensation.

Device No.	Signal name	Abbreviation	Description
Y18B9	Chuck close	-	This turns ON when the chuck is closed. When this signal turns ON, the compensation between the reference spindle and synchronized spindle will change from droop compensation to position compensation.
X18AC	Chuck close confirmation	-	This turns ON when the chuck close signal is received during the spindle synchronization mode.



Note

(1) Use the error temporary cancel only when there is still an error between the spindle and synchronization with the chuck close signal.

Error temporary cancel function

When spindle synchronization is carried out while grasping the workpiece with the reference spindle and rotating, if the chuck is closed to grasp the workpiece with the synchronized spindle, the speed will fluctuate due to external factors and an error will occur. If spindle synchronization is continued without compensating this error, the workpiece will twist.
This torsion can be prevented by temporarily canceling this error.

Device No.	Signal name	Abbreviation	Description
Y18B5	Error temporary cancel	SPDRP0	The error is canceled when this signal is ON.

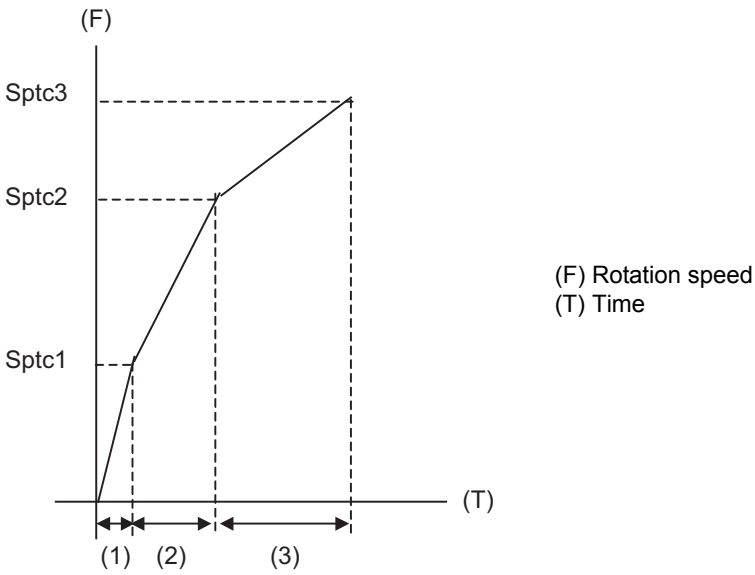
Phase error monitor

The phase error can be monitored during spindle phase synchronization.

Device No.	Signal name	Abbreviation	Description
R6519	Phase error monitor	-	The phase error during spindle phase synchronization control is output as a pulse unit.
R6520	Phase error monitor (lower limit value)	-	The lower limit value of the phase error during spindle phase synchronization control is output as a pulse unit.
R6521	Phase error monitor (upper limit value)	-	The upper limit value of the phase error during spindle phase synchronization control is output as a pulse unit.

Multi-step acceleration/deceleration

Up to eight steps of acceleration/deceleration time constants for spindle synchronization can be selected according to the spindle rotation speed.



(1) Time required from stopped state to sptc1 setting rotation speed
spt * (sptc1/maximum rotation speed)

(2) Time required from sptc1 to sptc2 setting rotation speed
sptc * ((sptc2-sptc1)/maximum rotation speed) *spdiv1

(3) Time required from sptc2 to sptc3 setting rotation speed
spt * ((sptc3-sptc2)/maximum rotation speed) *spdiv2

18.1.3 Spindle Position Control under Spindle Synchronization Control

Function and purpose

This function enables the spindle position control by the reference spindle under spindle synchronization control. The reference spindle can be controlled as the rotary axis while the spindle synchronization status remains kept, and also positioning or interpolation with another servo axis is enabled by issuing the position command (movement command) in the same way as for the servo axis.
There are two methods: PLC signal method and program command method to switch the spindle and rotary axis during spindle synchronization control. The method that is applied for switching depends on the MTB specifications (parameter "#3129 cax_spec/bit0").
This section describes the program command method.
In this manual, the state to control an axis as a spindle is referred to as "spindle mode", and the state to control an axis as a rotary axis as "C axis mode".
For details on the spindle position control function, refer to "10.5 Spindle Position Control (Spindle/C Axis Control)".
This section also describes considerations on spindle position control under spindle synchronization, and the status of various PLC signals and restrictions. The status, control method, and operation of the PLC signal depend on the MTB specifications.

The spindle position control under spindle synchronization has the following features.
All the parameters below depend on the MTB specifications, so they cannot be changed by the user.

(1) During spindle synchronization, the gain of the reference spindle or synchronous spindle is designated by the spindle parameter #13003 SP003 (PGS position loop gain spindle synchronization). After the spindle has been switched to the C axis mode, the gain is not switched to the value designated by the parameter #13002 SP002 (PGN position loop gain interpolation mode).
(2) When the reference spindle is set to the gain switching valid state (#3129 cax_spec/bit4=1), the part system that contains the reference spindle in the C axis mode switches the position loop gain of all axes, excluding the C axis, in the part system to the spindle synchronization loop gain (#2249 SV049, #2250 SV050, or #2258 SV058).
(3) Parameters such as the time constant under C axis control are executed based on the C axis setting.

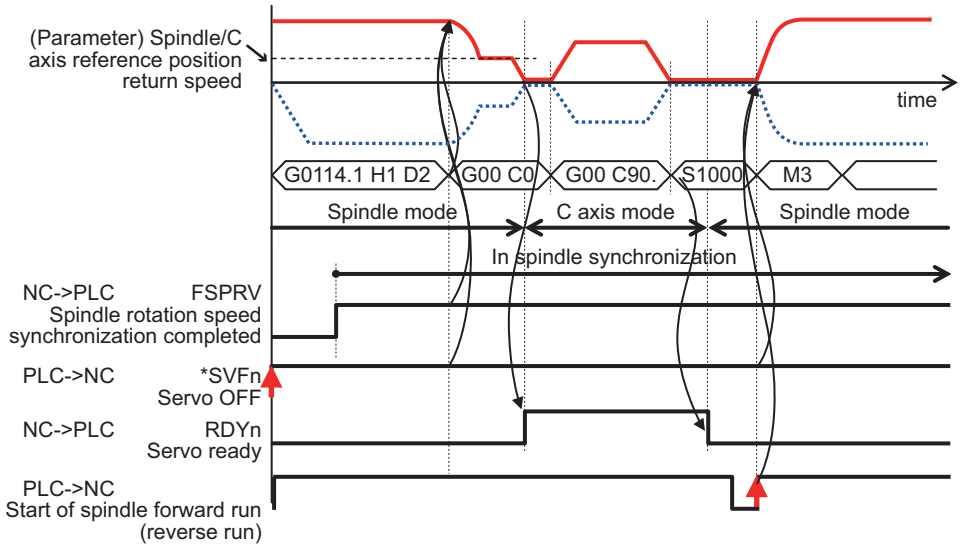
Detailed description

Program command method

The machining program switches the reference spindle to the C axis mode with G00 command, and to the spindle mode with S command. The C axis servo OFF signal (*SVFn) must be kept ON while the program command method is selected.

When the program command method is selected, switching operation is performed only with the reference position return type.

The following shows the C axis switching sequence under spindle synchronization.



<Spindle control mode under spindle synchronization to C axis control mode switching>

(1) If "G00 C_" is commanded on the program when the reference spindle under spindle synchronization is in the spindle mode, the spindle is directly located at the position commanded by the reference spindle while the synchronous status remains kept.

(2) Only the G00 command is valid to switch the mode. If the C axis movement is commanded with the G code other than G00, it causes a program error (P430).

(3) The spindle position control axis must be commanded with the absolute value address or absolute value command (G90). If the incremental value address or incremental value command (G91) is used, it causes a program error (P32).

(4) In the switching specifications, only the reference position return type (equivalent to "#3106 zrn_typ BIT8" = 0) is valid, and the direction to return from the rotation mode to the reference position follows the rotation direction (equivalent to "#3106 zrn_typ BITB" = 1). The direction to return from the spindle stop state to the reference position ("#3106zrn_typ/BITA-BIT9") and interpolation mode selection ("#3016 zrn_typ BITD/BITE") follow the appropriate parameters.

[C axis mode switching conditions]

(1) The C axis servo OFF signal (*SVFn) of the reference spindle is set to ON when switching is commanded.

<Switching from C axis control mode under spindle synchronization to spindle control mode>

(1) The switching is performed with the spindle forward run signal (SRN) on or the spindle reverse run signal (SRI) on and the S command.

(2) The switching is performed with the startup of the spindle forward run signal (SRN) or the spindle reverse run signal (SRI).

[Spindle mode switching condition]

(1) The C axis servo OFF signal (*SVFn) of the reference spindle is set to ON and the C axis selection signal (CMD) is set to OFF when switching is commanded.

<Operation when the servo OFF signal (*SVFn) of the reference spindle is set off>

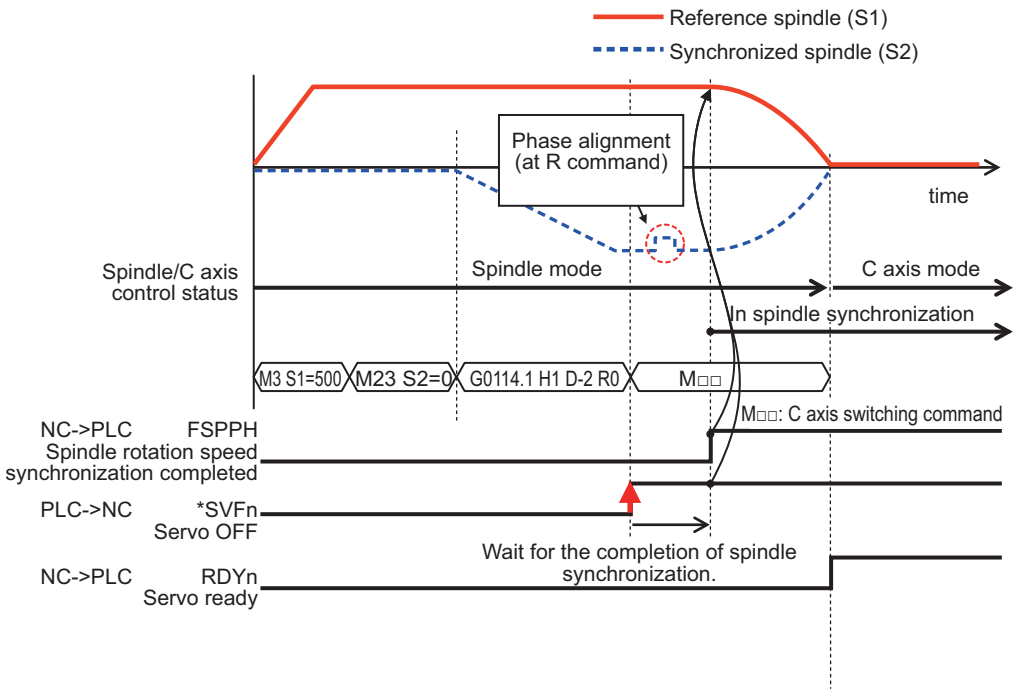
(1) The mode cannot be switched from the spindle mode to the C axis mode or from the C axis mode to the spindle mode.

(2) In the spindle mode, the axis does not run as a spindle even if the forward run command (SRN) or reverse run command (SRI) is executed.

(3) In the C axis mode, an operation error (M01 0005) occurs if the movement command is executed. In the servo OFF mode, switching follows the setting of the spindle specification parameter "#1064 svof " (error correction).

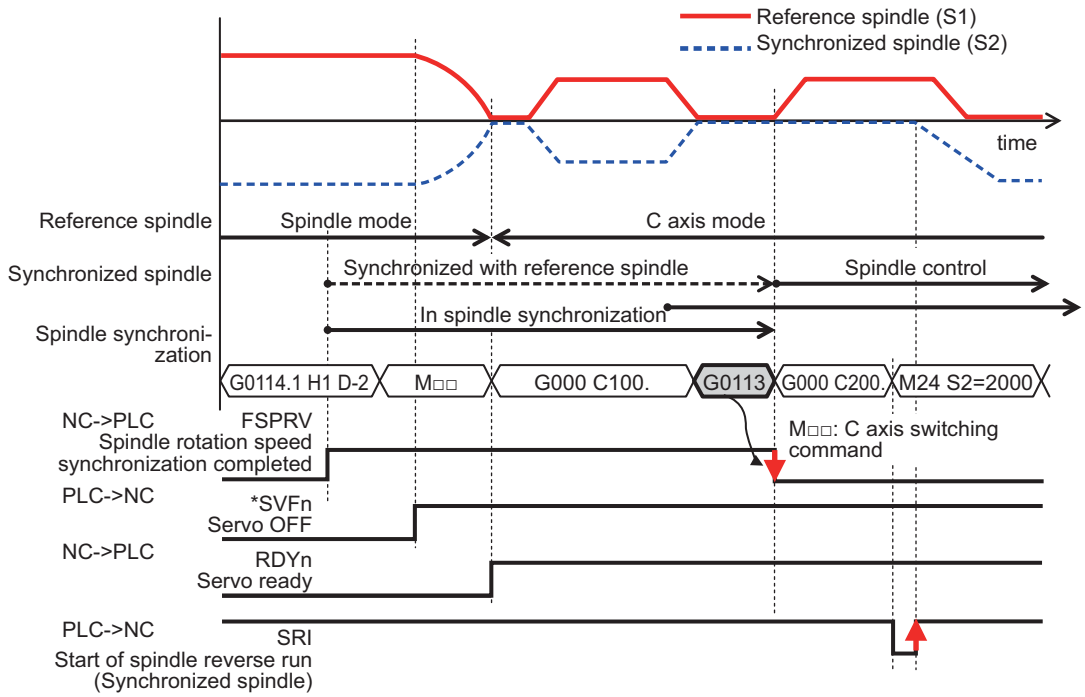
C axis mode selection command in spindle synchronization incomplete state

If the C axis mode selection command is issued until the spindle synchronization completion signal (spindle rotation speed synchronization completion (FSPRV) for rotation synchronization and spindle phase synchronization completion (FSPPH) for phase synchronization (FSPPH)) is set on after the spindle synchronization command has been issued, the mode is switch to the C axis mode after the spindle synchronization completion signal has been set on. The following shows the C axis mode selection command from after the spindle synchronization command has been issued to before the spindle synchronization is completed.



Spindle synchronization cancel in C axis mode

The spindle synchronization is canceled while the reference spindle remains set in the C axis mode by issuing the spindle synchronization cancel command in the C axis mode or axis stop state.
The following shows the spindle synchronization cancel operation in the C axis mode.



(1) If the spindle is stopped by feed hold or cutting override zero during C axis movement, the spindle synchronization control is canceled with the spindle synchronization cancel command.

(2) If the C axis is in process of movement when the spindle synchronization cancel command is issued, it causes an operation error (M01 1135), and the spindle synchronization cancel operation is not completed. When the movement of the C axis is completed and the C axis is set to the smoothing zero, the operation error is canceled, and the spindle synchronization cancel operation is completed.

(3) After the spindle synchronization has been canceled, the gain remains set to the spindle synchronization gain (SP003) for the reference spindle under C axis control, and to the spindle gain (SP001) for the synchronized spindle.

Spindle synchronization command using the spindle in the C axis mode as the reference spindle

(1) When a spindle that is not in the spindle synchronization state is set in the C axis control mode, if the spindle synchronization command is issued using the spindle as the reference spindle, it causes an operation error (M01 1026).

Relationship with other functions

Function that can be commanded by the spindle position control under spindle synchronization control

When the reference spindle is in the C axis control state, the synchronized spindle synchronizes with the reference spindle in the spindle control state without being setting in the C axis control state.

Contents of command	Operation
Milling interpolation (G12.1/G13.1)	• The reference spindle in the C axis mode can be used as the rotary axis for milling interpolation.
C axis selection command OFF	• The reference spindle is switched to the C axis mode, but the synchronized spindle is kept in the synchronous state.
Spindle synchronization cancel command (G113)	• Cancels the spindle synchronization control. • If the spindle synchronization cancel command is issued during movement of the C axis, it causes an operation error (M01 1135), and the spindle synchronization is not canceled. When the operation error is canceled after the C axis has stopped, the spindle synchronization control is also canceled.
Tool spindle synchronization II (Hobbing) (G114.3)	• When the Multiple spindle synchronization set control function is enabled, the reference spindle in C axis mode can be specified as the workpiece axis of Tool spindle synchronization II. For details, refer to "18.5 Multiple Spindle Synchronization Set Control". • Also, if the reference spindle in C axis mode is specified as the hob axis (spindle) of tool spindle synchronization II, an operation error (M01 1005) will occur.
Mixed synchronization control (G110) Arbitrary axis exchange control (G140)	• The spindle set to the spindle/C axis can be exchanged even when it is in the C axis mode or spindle mode. • When the C axis is designated as the spindle with gain switching valid (parameter "#3129 cax_spec/bit4") in the MTB specifications, the gain of all servo axes in the destination part system is switched by exchanging the C axis. When the spindle/C axis with gain switching valid is cleared from the part system, in which the gain of all the servo axes is switched, to another part system, the gain of all servo axes in the part system is returned.
Control axis synchronization between part systems (G125)	• The control axis synchronization between part systems (reference spindle or synchronized spindle) can be commanded to the reference spindle in the C axis mode. • If the reference spindle is commanded as the axis related to the control axis synchronization between part systems in the spindle mode, it causes an operation error (M01 1037). • When the reference spindle in the C axis mode is set as the axis related to the control axis synchronization between part systems, the synchronization relation cannot be maintained if the mode is switched from the C axis mode to the spindle mode. Do not switch from the C axis mode to the spindle mode.
Control axis superimposition (G126) Arbitrary axis superimposition (G156)	• The control axis superimposition or arbitrary axis superimposition can be commanded to the reference spindle in the C axis mode. • The control axis superimposition or arbitrary axis superimposition is canceled by commanding the switching from the C axis mode to the spindle mode while the reference spindle is used as the control axis superimposition or arbitrary axis superimposition related axis in the C axis mode. When the [program command method] is selected to switch between the C axis mode and spindle mode, the control axis superimposition or arbitrary axis superimposition is canceled by setting the servo OFF signal (*SVFn) off. • If the reference spindle is commanded as the superimposition-related axis (reference axis or superimposed axis) in the spindle mode, it causes an operation error (M01 1004).
Emergency stop	• When the spindle is set in the emergency stop state, the spindle synchronization control is canceled immediately. • If emergency stop occurs in the C axis mode, the reference or synchronized spindle decelerates and stops based on the parameter (spindle parameter "#13056 SP056 EMGt " deceleration time constant at emergency stop) that is set to each spindle.

Cautions on spindle position control under spindle synchronization control

(1) The upper limit of the rotation speed in the C axis mode is controlled by the rapid traverse rate (#2001 rapid) or cutting feed clamp speed (#2002 clamp) of the C axis set to the axis specification parameter; therefore, the rotation speed of the spindle motor may exceed the highest spindle rotation speed depending on the setting of the C axis rapid traverse rate or clamp speed.
The rapid traverse speed (#2001 rapid) and cutting feed clamp speed (#2002 clamp) of the C axis must not be below the highest spindle rotation speed (#3001 slimt1 to #3004 slimt4) of the reference or synchronized spindle (which depends on the MTB specifications).

C axis rapid traverse rate (°/min) <= (Highest spindle rotation speed (r/min) × 360(°))

(2) If the spindle in the C axis mode is set to the reference or synchronized spindle, it causes an operation error (M01 1026). When the control mode at power-on is set to the C axis mode, switch it to the spindle mode once to command the spindle synchronization, and switch it to the C axis mode again.
The control mode at power-on depends on the MTB specifications (parameter "#3129 cax_spec/bit2").

(3) If the C axis servo OFF signal (*SVFn) of the synchronous spindle is set to ON during spindle synchronization, it causes an operation error (M01 1026).

18.2 Tool Spindle Synchronization I

18.2.1 Tool Spindle Synchronization IA (Spindle-Spindle, Polygon) ; G114.2

Function and purpose

In a machine having a rotary tool and having a spindle controlled between spindles as the workpiece axis, spindle-spindle polygon machining can be carried out by controlling the workpiece axis rotation in synchronization with the rotation of the rotary tool axis.
The spindle and rotary tool axis can be controlled with MDS-*-SP.
The other spindle synchronization functions (Spindle synchronization I/ Spindle synchronization II/ Tool spindle synchronization IA/ Tool spindle synchronization IB/ Tool spindle synchronization II/ Spindle superimposition control) cannot be commanded while this function is running. Also, this function cannot be commanded while the above functions are being executed. In these cases an operation error (M01 1005) will occur.
However, when the Multiple spindle synchronization set control function is enabled, multiple spindle synchronization functions can be commanded simultaneously. The validity of this function depends on the MTB specifications (parameter "#1440 multi_sp_syn"). For details, refer to "18.5 Multiple Spindle Synchronization Set Control".

Command format

Tool spindle synchronization IA (Spindle-spindle, polygon mode) ON

```
G114.2 H__ D__ E__ L__ R__ ;
```

H	Rotary tool axis selection (Reference spindle)
D	Workpiece axis selection (Synchronized spindle)
E	Rotary tool axis rotation ratio designation
L	Workpiece axis rotation ratio designation
R	Synchronized spindle phase shift amount

Tool spindle synchronization IA (Spindle-spindle, polygon mode) OFF

```
G113 ;
```

Note

(1) An axis address that involves any travel cannot be put in the same block as for the tool spindle synchronization IA cancel command. Doing so causes the program error (P33) to occur when cancel command is issued, which causes automatic operation to pause.

Explanation of address

Tool spindle synchronization IA ON (G114.2) 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 (Number of the rotary tool gear teeth and workpiece corners) of the two designated spindles (spindle and spindle). Tool spindle synchronization IA OFF (G113) cancels the synchronous state of rotating two spindles by the spindle synchronization command.

Address	Meaning	Command range (unit)	Remarks
H	Select a rotary tool. Command the spindle number or spindle name of the rotary tool axis from multiple spindles. (*1)	Spindle number: 1 to n (n: Maximum number of available spindles) Spindle name: 1 to 9	♦If a value exceeding the command range is commanded, a program error (P35) will occur. ♦If there is no command, a program error (P33) will occur. ♦If the same value as the D command is commanded, a program error (P33) will occur. ♦If an analog-connected spindle is selected, a program error (P700) will occur. (*2)
D	Select a workpiece axis. Commands the spindle number or spindle name of the workpiece axis of two spindles. (*1)	Spindle number: 1 to n or -1 to -n (n: Maximum number of available spindles) Spindle name: 1 to 9 or -1 to -9	♦If a value exceeding the command range is commanded, a program error (P35) will occur. ♦If there is no command, a program error (P33) will occur. ♦The rotation direction of the workpiece axis in respect to the rotary tool axis is commanded with the D sign. ♦If the same value as the H command is commanded, a program error (P33) will occur. ♦If an analog-connected spindle is selected, a program error (P700) will occur. (*2)
E	Specify the rotary tool axis rotation ratio. Set the rotation ratio (Number of rotary tool gear teeth) of the rotary tool axis.	1 to 999	♦If a value exceeding the command range is commanded, a program error (P35) will occur. ♦If there is no command, the rotation ratio will be interpreted as 1.
L	Specify the workpiece axis rotation ratio. Set the rotation ratio (number of workpiece corners) of the workpiece axis.	1 to 999	♦If a value exceeding the command range is commanded, a program error (P35) will occur. ♦If there is no command, the rotation ratio will be interpreted as 1.
R	Specify the spindle synchronization phase shift amount. Set the shift amount from the synchronous spindle's reference position (one rotation signal).	0 to 359.999 (°)	♦If a value exceeding the command range is commanded, a program error (P35) will occur. ♦The commanded shift amount will be applied in the clockwise direction in respect to the spindle. ♦Minimum resolution of commanded shift amount Semi-close case 360/4096 [°] Full-close case (360/4096) * K [°] (K: Gear ratio of spindle and encoder) ♦If there is no R command, phase alignment will not be carried out.

(*1) There are two spindle command methods: spindle number method and spindle name method.
Command with the spindle names, only when all spindles are set with the spindle name parameter (#3077 Sname) (from 1 to 9). For others, command with the spindle number. These settings depend on the MTB specifications.

(*2) The available spindle type and connection method depend on the specifications of your machine tool.

Detailed description

Rotary axis and rotation direction

The rotary tool axis and workpiece axis rotation speed and rotation direction during tool spindle synchronization IA command are as follows.

(1) The rotation speed and rotation direction of the 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 number of the rotary tool gear teeth 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)
E: Rotary tool axis rotation ratio (Number of rotary tool gear teeth)
L: Workpiece axis rotation ratio (Number 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 IA is commanded, the relation of the rotary tool axis and workpiece axis rotation is held in all automatic or manual operation modes until spindle synchronization cancel (G113) is commanded, the spindle synchronization cancel signal is input, or reset (reset 1, reset 2, reset & rewind) is executed when "#1239 set11/bit3" is set to "1".
Even during feed hold, the rotary tool axis and workpiece axis synchronization state is held.

Spindle operation for spindle-spindle polygon

(1) Even if the forward run command and reverse run command are not issued to the workpiece axis when the tool spindle synchronization IA mode is commanded, the workpiece axis starts rotation synchronously with the rotary tool axis if the rotation command is issued to the rotary tool axis.

(2) The rotation command (S command) and constant surface speed control are invalid in respect to the workpiece axis during the tool spindle synchronization IA mode. Note that the modal will be updated, so these will be effective after the spindle synchronization is canceled.

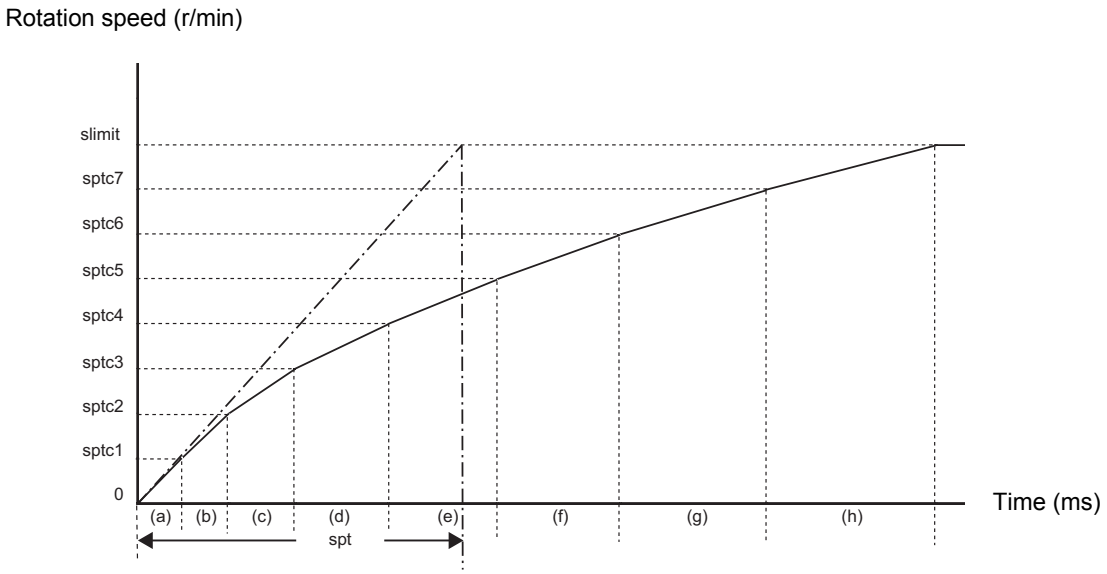
(3) If the rotation speed commanded to the workpiece axis exceeds the maximum rotation speed of the rotary tool axis or the maximum clamp speed designated by address S following G92, the workpiece axis rotation speed is clamped to prevent the rotation speed from exceeding those maximum speeds.

Multi-step acceleration/deceleration control

(1) Acceleration/deceleration time constants for up to eight steps can be selected according to the spindle rotation speed for the acceleration/deceleration during spindle-spindle polygon machining.
The acceleration/deceleration in each step is as follows.

Time required from minimum rotation speed to maximum rotation speed in each step
= [Time constant without multi-step acceleration/deceleration] * [magnification of time constant in each step]
* [Rate of rotation speed width in each step respect to rotation speed width up to limit rotation speed]

Time required to rotate to sptc 1 set rotation speed from stopped state (a)
= spt * sptc1 / slimit
Time required to reach sptc 2 set rotation speed from sptc 1 (b)
= spt * spdiv1 * (sptc2 - sptc1) / slimit
Time required to reach sptc 3 set rotation speed from sptc 2 (c)
= spt * spdiv2 * (sptc3 - sptc2) / slimit
Time required to reach sptc 4 set rotation speed from sptc 3 (d)
= spt * spdiv3 * (sptc4 - sptc3) / slimit
Time required to reach sptc 5 set rotation speed from sptc 4 (e)
= spt * spdiv4 * (sptc5 - sptc4) / slimit
Time required to reach sptc 6 set rotation speed from sptc 5 (f)
= spt * spdiv5 * (sptc6 - sptc5) / slimit
Time required to reach sptc 7 set rotation speed from sptc 6 (g)
= spt * spdiv6 * (sptc7 - sptc6) / slimit
Time required to reach slimit set rotation speed from sptc 7 (h)
= spt * spdiv7 * (slimit - sptc7) / slimit



To decrease the number of acceleration/deceleration steps, set one of the followings for the unnecessary step.
- Magnification for time constant changeover speed (spdiv7 to spdiv1) = 0 (or 1)
- Spindle synchronous multi-step acceleration/deceleration changeover speed (sptc7 to sptc1) = Limit rotation speed (slimit) or higher

(2) The rotary tool axis accelerates/decelerates linearly according to the spindle synchronous acceleration/deceleration time constant (spt) setting value of the spindle selected as the rotary tool axis and workpiece axis, whichever is larger.

(3) If the rotary tool axis command rotation speed is changed during spindle synchronization, the axis will accelerate/decelerate to the commanded rotation speed according to the spindle acceleration/deceleration set in the parameters while maintaining the synchronized state.

Phase alignment control

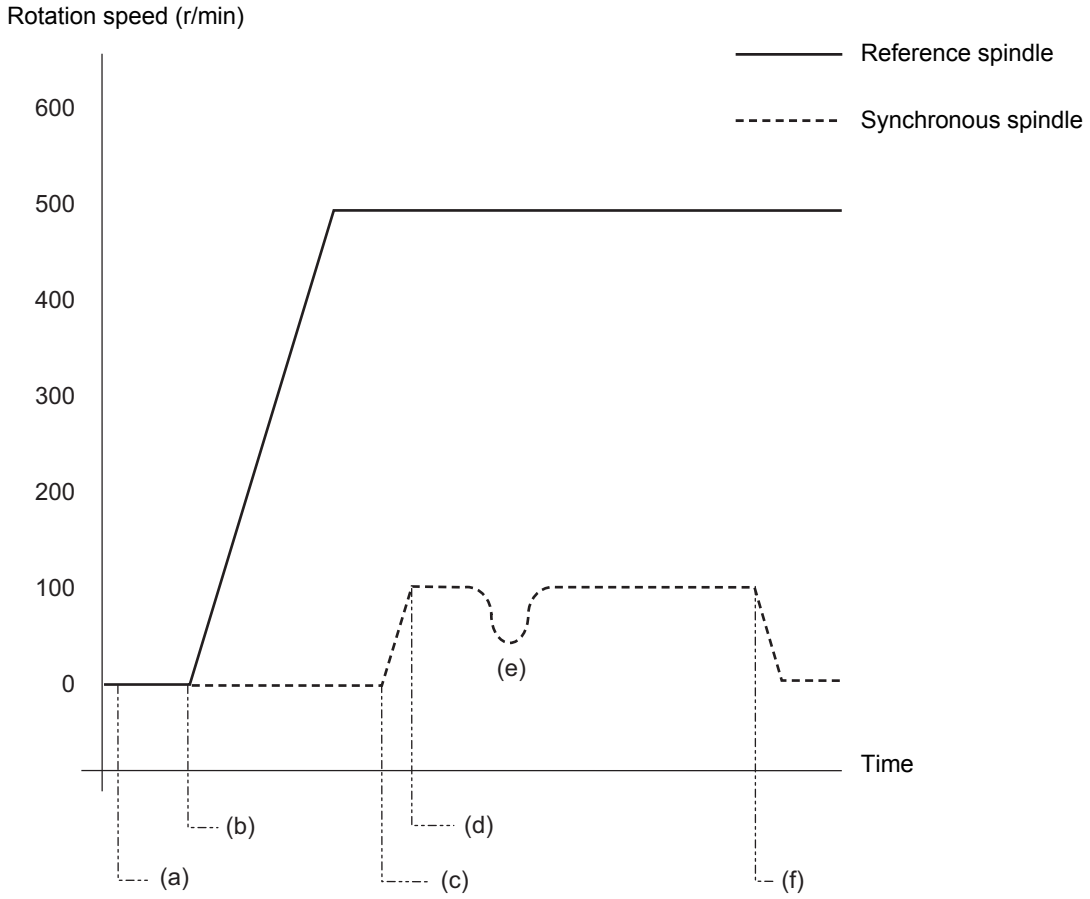
(1) If the tool spindle synchronization IA command (with R designation) is commanded with the G114.2 command, the synchronized spindle rotating at an arbitrary rotation speed will accelerate or decelerate to the rotation speed following the reference spindle and synchronized spindle rotation ratio command, and the spindle synchronization state will be entered. The spindles will then enter the spindle synchronization state. After that, the phases will be aligned to realize the rotation phase commanded with the R address.

(2) The spindle synchronization phase shift amount is commanded as the shift amount from the synchronized spindle's (workpiece axis) reference position (one rotation signal). There is not the shift amount in respect to the reference spindle (rotary tool axis).

(3) The following type of operation will take place.

T1100;	Rotary tool selection
M03 S1=0;	Carries out forward run (speed command) 1st spindle (synchronized spindle) (a)
:	
M43 S4=500;	Carries out forward run (speed command) 4th spindle (reference spindle) (b)
:	
G114.2 H4 D1 E1 L5 R0;	Forward runs the 1st spindle (synchronous spindle) and synchronizes with the 4th spindle (reference spindle). (c)
:	The 1st spindle (synchronous spindle) synchronizes with the 4th spindle. (d)
	Shift synchronous spindle's phase by R command value. (e)
:	
G113;	Cancel the tool spindle synchronization IA. (f)

<Operation>



Program example

```
:
:
M03 S1=0 ;                  1st spindle forward run
T1100;                      Rotary tool selection
M43 S4=500 ;                4th spindle forward run
G00 X40.Z-5 ;
:
:
G114.2 H4 D1 E1 L10 R0 ;    Tool Spindle Synchronization IA (Spindle-Spindle, Polygon mode) ON
                              Rotary tool axis: 4th spindle
                              Workpiece axis: 1st axis
                              Number of rotary tool gear teeth: 1
                              Rotation ratio: Number of workpiece corners: 10
                              Spindle synchronization phase shift amount: 0°
                                  S1 starts rotating by forward run in synchronization with S4.
                                  The phase is aligned with shift amount 0°.
                                  The S1 rotation speed is 50 r/min (S2:S1 = 10:1).

G95;                         Synchronous feed mode selection

G00 X18 ;                   1st cut in
G01 Z20.F0.1 ;              Z axis feedrate is 0.1 mm per workpiece axis rotation.
G00 X40 ;                   If synchronization is not completed, wait to start cutting feed.
Z-5. ;
:
:
G00 X14 ;                   Final cut in
G01 Z20.F0.1 ;              Z axis feedrate is 0.1 mm per workpiece axis rotation.
G00 X40 ;
Z-5. ;

G113 ;                      Spindle synchronization cancel

M45;                        4th spindle stop
M05;                        1st spindle stop
:
```

Precautions

Precautions for programming

(1) If S is commanded in the same block as G114.2, the synchronization speed will be created at the previous S command until the S command ends, so the spindle speed may fluctuate momentarily. Thus, do not command S in the same block if possible.

(2) Always command G114.2 in an independent block.

(3) The tool spindle synchronization I (spindle-spindle, polygon) mode cannot be commanded during the spindle synchronization mode commanded with G114.*. An operation error (M01 1005) will occur.

(4) If spindle-spindle polygon machining is commanded while the phase shift calculation request signal SSPHM is ON, an operation error (M01 1106) will occur.

(5) Tool spindle synchronization IA (G114.2) cannot be executed using the spindle which is used in the synchronous tapping. An operation error (M01 1007) will occur.
Also, the synchronous tapping cannot be commanded using the spindle which is used in G114.2 command. An operation error (M01 1139) will occur.

(6) When the spindle/C-axis is used for the spindle-spindle polygon machining cannot be executed by designating the C axis mode spindle with the G114.2 command. An operation error (M01 1026) will occur.

(7) After G114.2 is commanded, the cutting feed block will not start until synchronization is established. Operation will stop with an operation error (M01 1033).

Restrictions regarding phase alignment control

(1) Make sure that the rotation ratio of spindle (and rotary tool axis spindle) actual rotation speed and encoder rotation speed has the following relation.
Spindle rotation speed/encoder rotation speed = n ("n" is an integer of 1 or more)
If this relationship is not established, the encoder's reference position will not stay at a constant position on the spindle, and thus the phase (position) will deviate with each phase alignment command.
Note that even in this case, as shown below, if the number of rotary tool gear teeth (Number of workpiece corners) is equivalent to the rotation ratio, the blade and workpiece phase (position) will not deviate.
(Rotary tool axis spindle rotation speed * Number of rotary tool gear teeth) /encoder rotation speed = n
("n" is an integer of 1 or more)

(2) During phase alignment control, phase alignment is carried out following each spindle encoder's reference position.
So if the positional relation of the workpiece and reference position (rotary tool and reference position) deviates when the power is turned OFF/ON or when the tool is changed, etc., the phase will deviate.

18.2.2 Tool Spindle Synchronization IB (Spindle-Spindle, Polygon) ; G51.2/G50.2 or G251/G250 (only 6 and 7 in G code list)

Function and purpose

In a machine containing a workpiece axis and that has a spindle controlled as the rotary tool axis, spindle-spindle polygon machining can be carried out by controlling the rotary tool axis rotation in synchronization with the rotation of the workpiece axis.
Tool spindle synchronization IB and tool spindle synchronization IC are switched depending on the setting of the parameter (#1501).

#1501 polyax =	0	: Tool spindle synchronization IB
	Other than 0	: Tool spindle synchronization IC

The workpiece axis and rotary tool axis can be controlled with MDS-*-SP.

This function is valid when the G code system is 6 or 7.

The other spindle synchronization functions (Spindle synchronization I/ Spindle synchronization II/ Tool spindle synchronization IA/ Tool spindle synchronization IB/ Tool spindle synchronization II/ Spindle superimposition control) cannot be commanded while this function is running. Also, this function cannot be commanded while the above functions are being executed. An operation error (M01 1005) will occur.
However, when the Multiple spindle synchronization set control function is enabled, multiple spindle synchronization functions can be commanded simultaneously. The validity of this function depends on the MTB specifications (parameter "#1440 multi_sp_syn"). For details, refer to "18.5 Multiple Spindle Synchronization Set Control".

Command format

Tool spindle synchronization IB (Spindle-spindle, polygon mode) ON (or G251)

```
G51.2 H__D__P__Q__R__ ;
```

H	Workpiece axis selection (Reference spindle)
D	Rotary tool axis selection (Synchronized spindle)
P	Workpiece axis rotation ratio designation
Q	Rotary tool axis rotation ratio designation
R	Synchronized spindle phase shift amount

Tool spindle synchronization IB (Spindle-spindle, polygon mode) OFF (or G250)

```
G50.2 ;
```

Note

(1) An axis address that involves any travel cannot be put in the same block as the tool spindle synchronization IB mode cancel command. Doing so causes the program error (P33) to occur when the cancel command is issued, which causes automatic operation to pause.

Tool spindle synchronization IB ON (G51.2) 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 (Number of the rotary tool gear teeth and workpiece corners) of the two designated spindle (spindle and spindle).
Tool spindle synchronization IB OFF (G50.2) command cancels the synchronous state of rotating two spindles by the tool spindle synchronization command.
The spindle-spindle polygon machining mode is also canceled in the following cases.
- Power OFF
- Emergency stop
- Reset (reset 1, reset 2, reset & rewind)
  (only when #1239 set11/bit3 = 1)
- Spindle-spindle polygon machining cancel signal

Address	Meaning	Command range (unit)	Remarks
H	Select a workpiece axis. Command the spindle number of the workpiece axis. (*1)	Spindle number: 1 to n (n: Maximum number of available spindles) Spindle name: 1 to 9	•If a value exceeding the command range is commanded, a program error (P35) will occur. • If the same value as the D command is commanded, a program error (P33) will occur. •If an analog-connected spindle is selected, a program error (P33) will occur. (*2) •If this option is omitted, the spindle number or spindle name specified in the parameter is designated.
D	Select a rotary tool axis. Command the spindle number of the rotary tool axis. (*1)	Spindle number: 1 to n (n: Maximum number of available spindles) Spindle name: 1 to 9	•If a value exceeding the command range is commanded, a program error (P35) will occur. • If the same value as the H command is commanded, a program error (P33) will occur. •If an analog-connected spindle is selected, a program error (P33) will occur. (*2) •If this option is omitted, the spindle number or spindle name specified in the parameter is designated.
P	Specify the workpiece axis rotation ratio. Set the rotation ratio (number of workpiece corners) of the workpiece axis.	1 to 999	•If a value exceeding the command range is commanded, a program error (P35) will occur.
Q	Specify the rotary tool axis rotation ratio. Command the rotary tool axis rotation ratio (number of tool teeth).	1 to 999 -1 to -999	•If a value exceeding the command range is commanded, a program error (P35) will occur. •If a negative sign is commanded, the rotary tool axis will rotate in the direction opposite to the workpiece axis.
R	Specify the spindle synchronization phase shift amount. Command the shift amount designation from the reference position (one rotation signal) of the rotary tool axis spindle.	0 to 359.999 (°)	•If a value exceeding the command range is commanded, a program error (P35) will occur. •The commanded shift amount will be applied in the clockwise direction in respect to the spindle. •Minimum resolution of commanded shift amount Semi-close case 360/4096 [°] Full-close case (360/4096) * K [°] (K: Gear ratio of spindle and encoder) •If there is no R command, the phase will be handled as R0. (only when #1239 set11/bit4 = 0)

(*1) There are two spindle command methods: spindle number method and spindle name method.
Command with the spindle names, only when all spindles are set with the spindle name parameter (#3077 Sname) (from 1 to 9). For others, command with the spindle number. These settings depend on the MTB specifications.

(*2) The available spindle type and connection method depend on the specifications of your machine tool.

Detailed description

Rotary axis and rotation direction

The workpiece axis and rotary tool axis rotation speed and rotation direction during spindle-spindle polygon machining are as follows.

(1) The workpiece axis rotation speed and rotation direction are rotation speed commanded with the S command and the rotation direction commanded with the M command, etc., for the spindle selected as the workpiece axis.

(2) The rotary tool axis rotation speed is determined by the number of the rotary tool teeth and number of workpiece corners commanded with G51.2.

$$Sw = Sh * \frac{Q}{P}$$

Sw: Rotary tool axis rotation speed (r/min)
Sh: Workpiece axis rotation speed (r/min)
P: Workpiece axis rotation ratio (Number of workpiece corners)
Q: Rotary tool axis rotation ratio (Number of rotary tool gear teeth)

(3) The rotary tool axis rotation direction is determined by the sign of the rotary tool axis selection Q commanded with G51.2.
In other words, if the Q sign is "+", the rotary tool axis will rotate in the same direction as the workpiece axis. If the Q sign is "-", the rotary tool axis will rotate in the reverse direction of workpiece axis.

(4) After tool spindle synchronization IB (G51.2) is commanded, the relationship between the workpiece axis and rotary tool axis rotation is held until tool spindle synchronization IB cancel (G50.2) is commanded, the spindle-spindle polygon machining cancel signal is input, or until the reset or emergency stop signal is input.
Even at feed hold, the workpiece axis and rotary tool axis synchronization states are held.

Operation for polygon machining with rotary tool axis

(1) When the tool spindle synchronization IB mode is commanded, even if neither the forward run nor reverse run command is input for the rotary tool axis, the rotary tool axis will start rotating.

(2) If spindle stop is commanded to a rotary tool axis during the tool spindle synchronization IB mode (when the spindle stop signal is ON), the rotary tool axis will stop rotating even if the workpiece axis is rotating.

(3) The rotation command (S command) and constant surface speed control are invalid in respect to the rotary tool axis during the tool spindle synchronization IB mode. Note that the modal is updated, so these will be validated when the spindle-spindle polygon machining is canceled.

(4) If the rotation speed commanded to the workpiece axis exceeds the maximum rotation speed of the rotary tool axis or the maximum clamp speed designated by address S following G92, the workpiece axis rotation speed is clamped to prevent the rotation speed from exceeding those maximum speeds.

Acceleration/deceleration control

(1) Acceleration/deceleration of the workpiece axis will be carried out linearly according to the spindle synchronization acceleration/deceleration time constant (spt) of the spindle selected as the workpiece axis.

(2) By setting the spindle synchronization multi-speed acceleration/deceleration time constant changeover speed (spdct1 to 7) and the scale for the time constant changeover speed (spddiv1 to 7), the acceleration/deceleration time can be changed in up to eight steps.

(3) If the workpiece axis command rotation speed is changed during spindle synchronization state, the commanded speed will be reached by accelerating or decelerating according to the spindle acceleration/deceleration set in the parameters while maintaining the synchronized state.

Phase alignment control

(1) If the tool spindle synchronization IB command (R = 0 with no R command) is commanded with G51.2, the workpiece axis spindle rotating at an arbitrary rotation speed will accelerate/decelerate to the rotation speed following the rotation ratio command of the workpiece axis spindle and the spindle synchronization state will be entered. The spindles will then enter the spindle synchronization state. After that, the phases will be aligned to realize the rotation phase commanded with the R address.

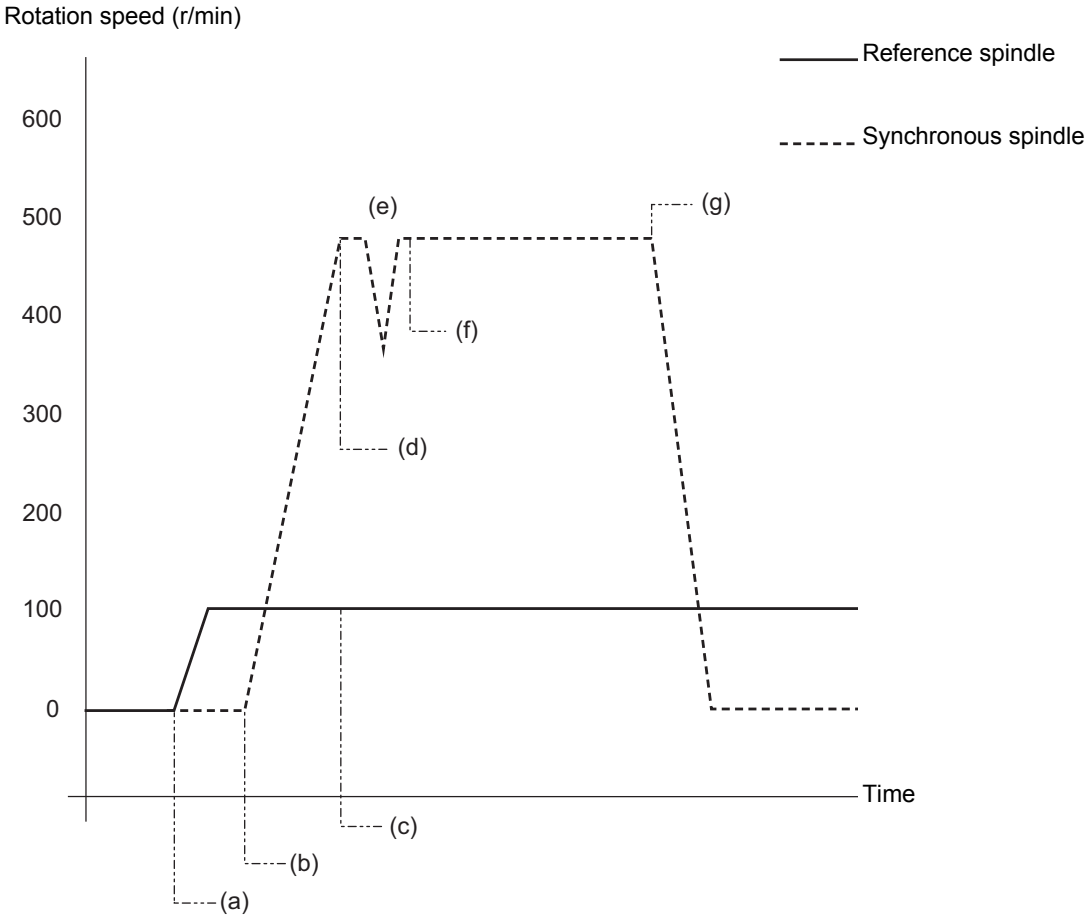
(2) The spindle synchronization phase shift amount is commanded the shift amount from the rotary tool axis spindle's reference position (one rotation signal). This is not the shift amount for the workpiece axis.

(3) The following type of operation will take place.

<Program example 1> With phase error (#1239 set11/bit4 = 0)

```
  :
T1100;                  Rotary tool selection
M03 S100 ;              Carries out forward run (speed command) 1st spindle (workpiece axis) (a)
  :
G51.2  H1 D3 P1 Q5 R0;  Forward run the 3rd spindle (rotary tool axis) using the tool spindle synchronization
  :                     IB command.   (b)
                        Synchronize the 3rd spindle with the 1st spindle (workpiece axis (spindle)). (c) (d)
                        Shift synchronous spindle's phase by R command value.  (e)
  :                     (Synchronization is completed.)  (f)
G50.2;                  Cancel the tool spindle synchronization IB.   (g)
```

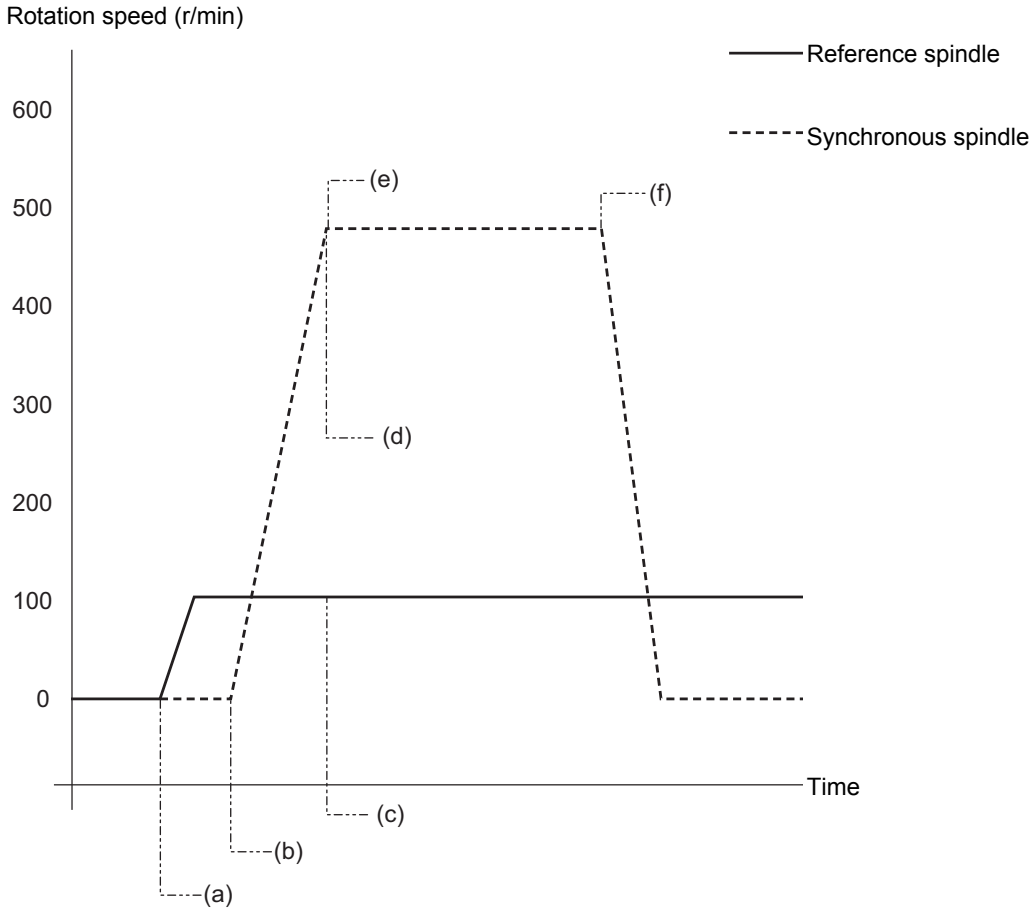
<Operation>



<Program example 2> No phase error (#1239 set11/bit4 = 1)

```
  :
  T1100;                Rotary tool selection
  M03 S100 ;            Forward run (speed command) the 1st spindle (workpiece axis).   (a)
  :
  G51.2  H1 D3 P1 Q5 ;  Forward run the 3rd spindle (rotary tool axis) using the tool spindle synchronization
  :                     IB command.   (b)
                        Synchronize the 3rd spindle with the 1st spindle (workpiece axis (spindle)).   ...(c) (d)
  :                     (Synchronization is completed.)  (e)
  G50.2;                Cancel the tool spindle synchronization IB.   (f)
```

<Operation>



Program example

```
:
:
T1100;                  Rotary tool selection
M03 S500 ;              1st spindle forward run
G00 X40.Z-5 ;

G51.2 H1 D3 P1 Q3 R0 ;  Tool spindle synchronization IB mode ON
                          Select 1st spindle as workpiece axis and 3rd spindle as rotary tool axis.
                          Designate rotation ratio as one workpiece corner and three rotary tool teeth.
                          Designate rotary tool axis spindle phase shift amount as 0°.
                              Start synchronous rotation of S3 to S1 with forward run.
                              The phase is aligned with shift amount 0°.
                              S3 rotation speed is 1500 r/min (S1:S3 = 1:3)
G95;                      Synchronous feed mode selection

G00 X18 ;               If synchronization is not completed, wait to start cutting feed.
G01 Z20.F0.1 ;          First cut
G00 X40 ;
  Z-5. ;
:
:
:
G00 X14 ;               Final cut
G01 Z20.F0.1 ;
G00 X40 ;
  Z-5. ;

G50.2 ;                 Tool spindle synchronization IB cancel
                        3rd spindle stop

M05 ;                   1st spindle stop
:
:
```

Precautions

Restrictions regarding phase alignment control

(1) Make sure that the spindle (and workpiece axis spindle) actual rotation speed and encoder rotation speed's rotation ratio has the following relation.
Spindle rotation speed/encoder rotation speed = n ("n" is an integer of 1 or more)
If this relationship is not established, the encoder's reference position will not stay at a constant position on the spindle, and thus the phase (position) will deviate with each phase alignment command.
Note that even in this case, as shown below, if the number of workpiece corners (number of rotary tool teeth) corresponds to the rotation ratio, the phase (position) of the blade and workpiece will not deviate.
(Workpiece axis spindle rotation speed * Number of workpiece teeth) /encoder rotation speed = n (n is an integer of 1 or more)

(2) During phase alignment control, the phase are aligned to the reference position of each spindle's encoder. Thus, if the position relation of the workpiece and reference position (workpiece and reference position) deviates when the power is turned ON/OFF or the tool is replaced, etc., the phase will deviate.

Precautions for programming

(1) Always command G51.2 and G50.2 in an independent block.

(2) The R command can be omitted when entering the tool spindle synchronization IB mode, but the P and Q commands must always be issued. A program error (P33) will occur if there are not the P and Q commands.

(3) To change the P, Q or R modal value in the tool spindle synchronization IB mode, command G51.2 again. In this case, R can be commanded independently. However, if either P or Q is also changed, always command P and Q again.

(4) Commands can be issued to each part system, but two or more part systems cannot be used simultaneously. The part system commanded first will be valid, and an operation error (M01 1005) will occur for that commanded last.

(5) The spindle No. designated in the parameters will be used if D_H_ is omitted from the G51.2 command.

(6) A program error (P610) will occur if the workpiece axis No. (#1518) and rotary tool axis No. (#1519) are the same as the value set in the parameters. A program error (P33) will occur if the spindle is connected in analog mode. (These parameters depend on the MTB specifications.)

(7) After G51.2 is commanded, the cutting feed block will not start until synchronization is established. (An operation error (M01 1033) will occur, and the program stops.)

(8) Tool spindle synchronization IB (G51.2) cannot be executed using the spindle which is used in the synchronous tapping. An operation error (M01 1007) will occur. Also, the synchronous tapping cannot be commanded using a spindle which is used in G51.2 command. An operation error (M01 1139) will occur.

(9) If the rotary tool axis number or workpiece axis number is changed in the tool spindle synchronization IB mode, a program error (P33) will occur.

18.2.3 Tool Spindle Synchronization IC (Spindle-NC Axis, Polygon) ; G51.2/G50.2 or G251/G250 (only 6 and 7 in G code list)

Function and purpose

This function carries out polygon machining by controlling the workpiece (spindle) and tool axis (NC servo axis) to rotate in synchronization at the commanded ratio.
Tool spindle synchronization IB and tool spindle synchronization IC are switched depending on the setting of the parameter (#1501).

Command format

Tool Spindle Synchronization IC (Spindle-NC Axis, Polygon mode) ON (or G251)

```
G51.2 P__ Q__ ;
```

P,Q	Spindle and rotary tool axis rotation ratio (P__:Q:__) P : Spindle Q : Rotary tool axis Command range : Integer value between 1 and 9, -1 and -9 Rotation direction : Designate with a sign ("+" Forward rotation, "-" Reverse rotation)

Tool Spindle Synchronization IC (Spindle-NC Axis, Polygon mode) OFF (or G250)

```
G50.2 ;
```

In addition to the G50.2 command, the tool spindle synchronization IC mode is also canceled in the following cases.
- Power OFF
- Emergency stop
- Reset (Reset 1/2, reset & rewind)

Detailed description

Details of Operation

S1000 ;	The spindle rotation speed (workpiece rotation speed) is commanded.
G51.2 P1 Q2 ; (Cutting into workpiece)	The polygon machining mode is entered with the G51.2 command. The spindle and rotary tool axis start rotating, and control is applied so that the spindle rotation speed and tool axis rotation speed are the commanded ratio (P:Q).
:	
G50.2;	The polygon machining mode between the spindle and rotary tool axis is canceled by the G50.2 command, and the spindle and rotary tool axis rotation stop.

The rotary tool axis is designated with the base specification parameter "#1501 polyax".

Rotation direction

(1) The spindle rotation direction during the polygon machining mode is determined by the P command sign and the spindle parameter "#3052 spplr (Spindle motor spindle relative polarity)".

P command sign	#3052 spplr	Rotation direction
(+)	0000	CW
(+)	0001	CCW
(-)	0000	CCW
(-)	0001	CW

(2) The rotation direction of the rotary tool axis during the polygon machining mode is determined by the Q command sign and the base specifications parameters "#1018 CCW".

Q command sign	#1018 CCW	Rotation direction
(+)	0	CW
(+)	1	CCW
(-)	0	CCW
(-)	1	CW

Program example

N10 G00 X100. Z20. ;	Positioning
N20 S1000 ;	Spindle (workpiece) rotation speed command
N30 G51.2 P1 Q2 ;	Spindle/tool axis rotation start (Spindle rotation speed 1000 [r/min], tool axis rotation speed 2000 [r/min])
N40 G01 X80. F10. ;	X axis cut in
N50 G04 X2. ;	Dwell
N60 G00 X100. ;	X axis retract
N70 G50.2 ;	Spindle/tool axis rotation stop

Note

(1) Always command G51.2 and G50.2 in independent blocks.

Precautions

(1) The "spindle synchronization (polygon)" specifications must be provided to use this function.
If G51.2 or G50.2 is commanded without the specifications, the program error (P39) will occur.

(2) Always command G51.2 and G50.2 in independent blocks.
- If the G51.2 (G50.2) command and group 0 G code are commanded in the same block, the G code commanded last in the block will have the priority.
- If the G51.2 (G50.2) command and G code other than a group 0 code are commanded in the same block, the program error (P33) will occur.

(3) While in the polygon machining mode, a movement command cannot be issued in the machining program for a servo axis set as the rotary tool axis.
If a movement command is issued to the rotary tool axis during the polygon machining mode, the program error (P32) will occur.

(4) The servo axis set as the rotary tool axis can be used as a feed axis in modes other than the polygon machining mode.

(5) The following functions are invalid for the rotary tool axis during the polygon machining mode.
- Override
- Feed hold
- Stored stroke limit

(6) The spindle rotation speed can be changed with the S command even during the polygon machining mode.
The spindle override and spindle rotation speed clamp are also valid.
If the spindle rotation speed is changed, the rotary tool axis rotation speed will also change so that the spindle and rotary tool axis established the P:Q ratio.

(7) The forward run/reverse run commands are invalid for the spindle when the polygon machining mode is in effect.

(8) If the feedrate for the rotary tool axis exceeds the rapid traverse rate (axis specifications parameters "#2001 rapid") when the polygon machining mode is in effect, the speed will be clamped at the rapid traverse rate. If the rotary tool axis is clamped at the rapid traverse rate, the spindle speed will also be set to lower than the command speed so that the spindle and rotary tool axis establish the P:Q ratio.

(9) The position loop gain for the rotary tool axis will be the value set in the axis specifications parameters "#2017 tap_g" during the polygon machining mode. The position loop gain for the spindle will be the spindle parameters "#13002 PGN" setting value.

(10) The following functions cannot be used simultaneously with polygon machining.
- Synchronous tap
- Thread cutting

(11) If an axis other than the rotary tool axis reaches the stroke end during the polygon machining mode, the axis other than the rotary tool axis will stop moving, but the rotary tool axis and spindle rotation will not stop.

(12) If the rotary tool axis reaches the stroke end during the polygon machining mode, the rotary tool axis and spindle rotation will stop, and the movement of axes other than the rotary tool axis will also stop.

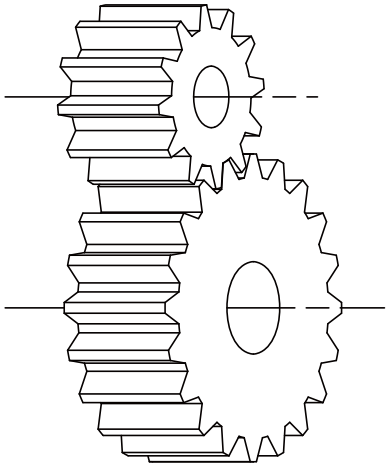
(13) If the spindle specifications parameter "#3106 zrn_typ/bit4" is set to "0", the polygon machining will start after the spindle returns to the zero point. (This parameter setting depends on the MTB specifications.)

18.3 Tool Spindle Synchronization II

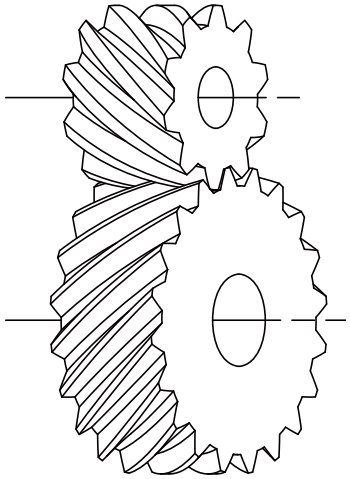
18.3.1 Tool Spindle Synchronization II (Hobbing) ; G114.3/G113

Function and purpose

This function is to cut the gear with hob (hob cutter).
A spur gear can be machined by synchronizing and rotating the hob axis and the workpiece axis in a constant ratio.
A helical gear can be machined by compensating the workpiece axis according to the gear torsion angle for the Z axis movement.

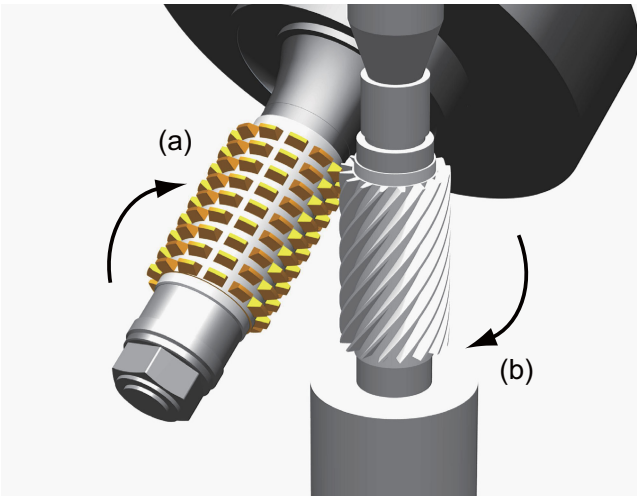


Spur gears



Helical gears

By synchronizing and rotating the hob axis and the workpiece axis in a constant rotation ratio, a gear is machined so that the cutter is engaged with gear.



(a) Hob
(b) Gear

In this manual, the hob axis and the workpiece axis are defined as follows:

Hob axis : Rotary tool axis on which a hob is mounted
Workpiece axis : Rotary axis on which a workpiece is mounted
Hob threads : Number of the screw paths created by cutter part on hob. Usually this is 1 row.

The other spindle synchronization functions (Spindle synchronization I/ Spindle synchronization II/ Tool spindle synchronization IA/ Tool spindle synchronization IB/ Tool spindle synchronization II/ Spindle superimposition control) cannot be commanded while this function is running. Also, this function cannot be commanded while the above functions are being executed. In these cases an operation error (M01 1005) will occur.
However, when the Multiple spindle synchronization set control function is enabled, multiple spindle synchronization functions can be commanded simultaneously. The validity of this function depends on the MTB specifications (parameter "#1440 multi_sp_syn"). For details, refer to "18.5 Multiple Spindle Synchronization Set Control".

Command format

Tool spindle synchronization II (hobbing) ON (for spur gear)

```
G114.3 H__ D__ E__ L__ R__ ;
```

```
G81.4 L_ R_ A_;          (For G code lists 6 and 7)
```

H	Hob axis selection
D	Workpiece axis selection
E(L)	Hob axis rotation ratio designation
L(R)	Workpiece axis rotation ratio designation
R(A)	Workpiece axis phase shift amount

Tool spindle synchronization II (hobbing) ON (for helical gear)

```
G114.3 H__ D__ E__ L__ P__ Q__ R__ ;
```

```
G81.4 L__ R__ P__ Q__ A__;          (For G code lists 6 and 7)
```

H	Hob axis selection
D	Workpiece axis selection
E(L)	Hob axis rotation ratio designation
L(R)	Workpiece axis rotation ratio designation
P	Gear torsion angle designation
Q	Module or diametral pitch designation
R(A)	Workpiece axis phase shift amount

Tool spindle synchronization II (hobbing) OFF

```
G113 ;
```

```
G80.4;          (For G code lists 6 and 7)
```

Note

(1) An axis address that involves any travel cannot be put in the same block as the tool spindle synchronization II cancel command. Doing so causes the program error (P33) to occur when the cancel command is issued, which causes automatic operation to pause.

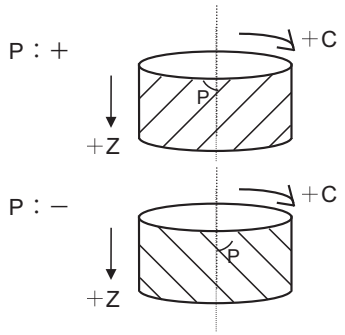
Detailed explanation of format

Tool spindle synchronization II ON (G114.3 for spur gear) is set as the hobbing mode for the spur gears that synchronously rotates two axes at different speeds by designating the hob axis and workpiece axis and designating the rotation ratio (number of hob threads and number of gear teeth) for the two designated axes.

Tool spindle synchronization II ON (G114.3 for helical gears) is set as the hobbing mode for the helical gears by designating the gear torsion angle and module or diametral pitch.

Tool spindle synchronization II OFF (G113) cancels the synchronous state of the hob axis and workpiece axis rotating in synchronization with the tool spindle synchronization II (hobbing) command.

Address	Meaning	Command range (unit)	Remarks
H	Select a hob axis. Command the spindle number of the hob axis. (*1)	Spindle No.: 1 to n (n: Maximum number of available spindles) Spindle name: 1 to 9	♦If there is no command, a program error (P33) will occur. ♦If an analog-connected spindle is selected, a program error (P33) will occur. ♦If disconnected spindle No. is designated, a program error (P35) will occur.
D	Select a workpiece axis. Command the rotation number of the workpiece axis.	-9 to -1, 1 to 9 ±1 to 8: Axis No. (in part system) ±9: C axis	♦If there is no command, a program error (P33) will occur. ♦The rotation direction of the workpiece axis in respect to the hob axis is commanded with the D sign. ♦If the D sign is "+", the workpiece axis will rotate in the forward direction when the hob axis rotates in the forward direction. If the D sign is "-", the workpiece axis will rotate in the reverse direction when the hob axis rotates in the forward direction. ♦If the axis specified as the workpiece axis is not a rotation axis, a program error (P33) will occur. ♦If C axis is selected when there is no C axis, a program error (P33) will occur.
E(L)	Specify the hob axis rotation ratio. Command the hob axis rotation ratio (hob 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). (*2)
L(R)	Specify a workpiece axis rotation ratio. Command 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. (For G code lists 6 and 7) ♦If "L0" is commanded, the workpiece axis will stop (synchronized with the Z axis for a helical gear). (*6) ♦The rotation direction of the workpiece axis in respect to the hob axis is commanded with the L sign. If the L sign is "+", the workpiece axis will rotate in the forward direction when the hob axis rotates in the forward direction. If the L sign is "-", the workpiece axis will rotate in the reverse direction when the hob axis rotates in the forward direction.
R(A)	Workpiece axis phase shift amount Command the amount to shift from the workpiece axis reference position to synchronize with the hob axis reference position.	0 to 359999 (0 to 359.999°) Decimal point input possible (*3)	♦The commanded shift amount will be applied in the workpiece axis counter's positive direction. ♦If there is no R command, phase alignment will not be carried out.

Address	Meaning	Command range (unit)	Remarks
P	Specify the gear torsion angle. Command the torsion angle for the helical gear.	-89000 to 89000 (-89.000 to 89.000) Decimal point input possible (*4)	♦If there is no P command, or if P0 is commanded, a spur gear will be machined. ♦To move the Z axis in the plus direction after entering the hobbing mode, command the direction that the workpiece axis is twisted. P sign: when it is +, + direction P sign: when it is -, - direction 
Q	Specify a module. Command the normal module for helical gear. When inch input, command the diametral pitch.	Metric input Module designation 100 to 25000 0.1 to 25. (0.1 to 25 mm) Inch input Diametral pitch designation 1000 to 250000 0.1 to 25. (0.1 to 25 $inch^{-1}$) Decimal point input possible (*5)	♦If there is no Q command for helical gear (when P is designated), a program error (P33) will occur. ♦For spur gear (when P is not designated, or P0 is commanded), the Q command will be ignored.

(*1) If a value exceeding the command range is commanded, a program error (P35) will occur.

(*2) 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.).

(*3) The range which can be set depends on the input setting unit (parameter "#1003 iunit").
(Example) When the input setting unit is 0.000001°, the range is 0 to 359.999999°.

(*4) If the decimal point input is OFF, the available setting range varies according to the input setting unit (parameter "#1003 iunit"). When the input setting unit is 0.000001°, the range is -89000000 to 89000000.

(*5) If the decimal point input is OFF, the available setting range varies according to the input setting unit (parameter "#1003 iunit"). When the input setting unit is 0.000001°, the range is as follows.
100000 to 250000000 in metric system
1000000 to 2500000000 in inch system

(*6) When address L = 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

(1) For G81.4 command, the hob axis (spindle No. or spindle name) and workpiece axis (NC axis No.) depend on the MTB specifications (parameter "#1595 hobm" and "#1596 hobs").

Detailed description

Rotation ratio change during tool spindle synchronization II (hobbing) mode

The rotation ratio and the number of hob threads can be changed without stopping the hob axis or the workpiece axis during tool spindle synchronization II (hobbing) mode.

G114.3 E__ L__ P__ Q__ ;	**Rotation ratio change**
G81.4 L__ R__ P__ Q__ ;	**Rotation ratio change (For G code lists 6 and 7)**

E(L)	Hob axis rotation ratio designation
L(R)	Workpiece axis rotation ratio designation
P	Gear torsion angle designation
Q	Module or diametral pitch designation

(1) Each address can be omitted in the G114.3 command in the tool spindle synchronization II (hobbing) mode. If the address is omitted, modal value of the last command is used.
(Example) When changing only the workpiece axis rotation ratio (the modal value of the previous command is used for items other than the workpiece axis rotation ratio.)
G114.3 L50 ;

(2) If the followings are issued, a program error (P33) will occur.

(a) When R command (workpiece axis phase shift amount) is issued.

(b) When the hob axis number and workpiece axis number are changed (for G code lists 6 and 7).

(c) When other than 0 is commanded by E command in E=0 state, or 0 is commanded by E command in E≠0 state.

(3) The workpiece axis rotation speed may be changed by rotation ratio. At this time, the acceleration/deceleration time constant follows the hobbing workpiece axis time constant (parameter "#2195 hob_tL").

(4) Spindle rotation speed synchronization completion signal is turned OFF by changing the rotation ratio. This signal is turned ON when the workpiece axis rotation speed reaches the prescribed range for hob axis rotation speed after completing the rotation ratio change.

(5) The hob axis rotation speed cannot be changed while the rotation ratio is changed (during workpiece axis acceleration/deceleration). If the rotation command is issued for hob axis during the rotation ratio change, the commanded rotation speed is applied after completing the rotation ratio speed change.

(6) The helical gear machining by Z axis movement does not be executed while the rotation ratio is changed (during workpiece axis acceleration/deceleration).
The helical gears machining is executed after completing the rotation ratio change.

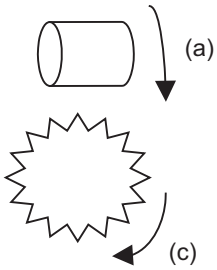
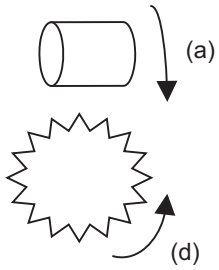
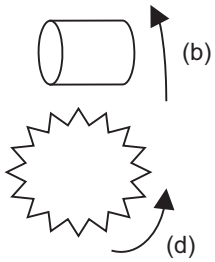
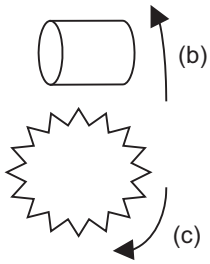
(7) The phase of hob axis and workpiece axis during rotation ratio changing (during workpiece axis acceleration/deceleration) or after changing is not warrantable. A phase cannot be aligned with gears machining of the last command.

(8) The "Hob axis delay (advance) monitoring", "Compensation control by workpiece axis" and "The workpiece axis feed forward control" are invalid while the rotation ratio is changed (during workpiece axis acceleration/deceleration).
These functions are valid after finishing the rotation ratio change.

Rotation speed and rotation direction

The rotation speed and rotation direction of the hob axis and workpiece axis during tool spindle synchronization II (hobbing) are as follows.

(1) The rotation speed and rotation direction of hob 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 hob axis.

When the sign of D command is +	When the sign of D command is -
	
	

(a) Hob axis: Forward rotation
(b) Hob axis: Reverse rotation
(c) Workpiece axis: + direction
(d) Workpiece axis: - direction

<Note>

- The rotation directions (+/- directions) of the workpiece axis shown above refer to the forward and reverse directions of the machine coordinate system of the workpiece axis.

(2) The workpiece axis rotation speed is determined by the hob threads specified using the hobbing mode command and the number of gear teeth.

$$Sw = Sh * \frac{E}{L}$$

Sw : Workpiece axis rotation speed (r/min)
Sh: Hob axis rotation speed (r/min)
E: Hob axis rotation ratio (number of hob threads)
L: Workpiece axis rotation ratio (number of gear teeth)

(3) The workpiece axis rotation direction varies depending on the G code system.
[When the G code system 2, 3, 4, or 5 is set]
The workpiece axis rotation direction is determined by the sign of the workpiece axis selection "D" commanded with the hobbing mode command.
In other words, when the "D" sign is "+", the workpiece axis will rotate in the same direction as the hob axis, and when the "D" sign is "-", the workpiece axis will rotate in the direction opposite to the hob axis.
[When the G code system 6 or 7 is set]
The workpiece axis rotation direction is determined by the sign of the hob axis rotation ratio "L" commanded with the hobbing mode command.
In other words, when the "L" sign is "+", the workpiece axis will rotate in the same direction as the hob axis, and when the "L" sign is "-", the workpiece axis will rotate in the direction opposite to the hob axis.

(4) After Tool spindle synchronization II (hobbing) is commanded, the relationship between the hob 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 reset or feed hold, the hob axis and workpiece axis synchronization state is held.

Workpiece axis control

(1) When the hobbing mode is commanded during hob axis rotation, the workpiece axis accelerates up to the speed required in synchronization with the hob axis according to the hobbing workpiece axis time constant (#2195 hob_tL) using the constant inclination acceleration/deceleration control. Then rotates in synchronization with hob axis.

(2) The axis selection signal and in axis motion signal of the workpiece axis are not output during the tool spindle synchronization II (hobbing) mode.

(3) If a manual movement command is issued to the workpiece axis during the tool spindle synchronization II (hobbing) mode, the manual movement will be superimposed on the workpiece axis movement with tool spindle synchronization. In this case, the axis selection signal and in axis motion signal of workpiece axis will be output.
Note that, if the movement command is issued in the manual reference position return mode, an operation error (0005) occurs.
An automatic movement command can be issued to the workpiece axis during the tool spindle synchronization II (hobbing) mode. Refer to "(2) Command compensation" in "Compensation control by workpiece axis" for details of the command to the workpiece axis.

(4) During the tool spindle synchronization II (hobbing) mode, the operations in respect to the input signals of external deceleration, interlock and machine lock for workpiece axis are as follows.

	Interlock	Machine lock	External deceleration
Movement by the hobbing function	Invalid	Invalid	Invalid
Movement by manual command	Valid for manual interlock	Valid for manual machine lock	Valid
Automatic compensation by incremental command	Valid for automatic interlock	Valid for automatic machine lock	Valid

(5) If a servo OFF signal is input for the workpiece axis during the Tool spindle synchronization II (hobbing) mode, the tool spindle synchronization II (hobbing) is canceled because synchronization cannot be maintained.

(6) The workpiece axis rotation speed is determined according to the hob axis rotation speed, so designate the hob axis rotation speed so that the workpiece axis cutting clamp speed is not exceeded.

(7) The C axis counter on each screen will be updated as shown below during the tool spindle synchronization II (hobbing) mode.
(a) When the workpiece 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 the workpiece axis is a linear-type rotation axis (all coordinate values linear type)
The axis will rotate in the 360° range including the machine coordinate position and workpiece coordinate position when hobbing starts.
(c) When the workpiece axis is a linear-type rotation axis (workpiece coordinate values linear type)
The axis will rotate in the 360° range including the workpiece coordinate position when hobbing starts.
(Example)

Coordinate value when the hobbing starts		Rotation range	
125.000	(°)	0.000 to 359.999	(°)
750.500	(°)	720.000 to 1079.999	(°)
-252.200	(°)	-360.000 to -0.001	(°)

(8) If the hobbing command is issued before the workpiece axis completes zero point return, a program error (P430) will occur.

Acceleration/deceleration control

(1) The hob axis will carry out multi-step acceleration/deceleration with the spindle synchronization acceleration/deceleration time constant (spt) set for the spindle selected as the hob axis.

Phase alignment control (Machine configuration that the phase alignment is possible)

To carry out phase alignment during hobbing, the spindle detector commanded to the hob axis must have a Z phase and satisfy the following conditions.

Control system	Gear ratio conditions
Semi-closed control	Spindle side gear : motor side gear = 1 : 1
Full-closed control	Spindle end : encoder end = 1 : 1
Control system	Gear ratio conditions

Phase alignment control (Operation when the zero point of hob axis is not established)

When the zero point of hob axis is not established by the hob axis rotation after turning the power ON or the spindle gear changeover, carry out phase alignment by following operations.

(The zero point of the hob axis is established within the range of (a) - (b) in the figure.)

(1) When tool spindle synchronization II (with R command) is specified using the hobbing mode command , the rotation axis commanded as the workpiece axis will enter the spindle synchronization II (hobbing) control state.

(2) The hob axis will start rotation at the Z phase detection speed (parameter "#3109 zdetspd") set in the parameters with the first S command issued for the hob axis after the hobbing control state is entered.
At this time, the workpiece axis will reach the rotation speed following the rotation ratio command for the hob axis and workpiece axis.
If this command rotation speed is 0 (r/min), the hob axis will not start rotating, and instead will wait for the next S command.

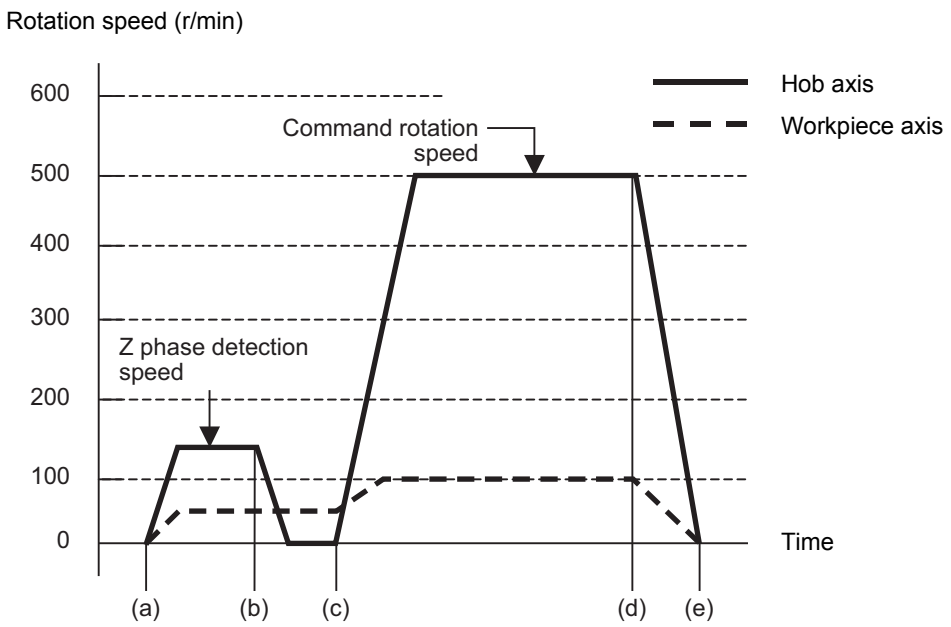
(3) The hob axis and workpiece axis phases will be aligned in this state.

(4) After the phases are aligned, the hob 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 hob axis rotation speed allowing for the hob axis and workpiece axis rotation ratio, and will enter the synchronized state.

(5) Below is the operation example:

```
Txx00;                  Select a rotary axis
M83 S4=0;               Forward run the fourth spindle (hob axis). (Rotation speed is 0) (a)
:
G114.3 H4 D9 E1 L5 R0;  Hobbing mode ON (phase alignment with the phase difference of zero) (b)
S4=500;                 Rotate the fourth spindle (hob axis) at 500 r/min. (c)
:
M85;                    Stop the fourth spindle. (d)
G113;                   Hobbing mode OFF (e)
```

<Operation>



Phase alignment control (Operation when the zero point of hob axis is established)

When the zero point of hob axis has already been established, the zero point of hob axis establishment is omitted. Thus, the process finishes fast compared with the case that the zero point of hob axis is not established.

(1) When tool spindle synchronization II (with R command) is specified using the hobbing mode command , the rotation axis commanded as the workpiece axis will enter the spindle synchronization II (hobbing) control state.

(2) The hob axis rotation speed follows the Z phase detection speed (parameter "#3109 zdetspd") set in the parameters with the first S command issued for the hob axis after entering the hobbing control state.
If this command rotation speed is 0 (r/min), the workpiece axis will not start rotating, and instead will wait for the next S command.

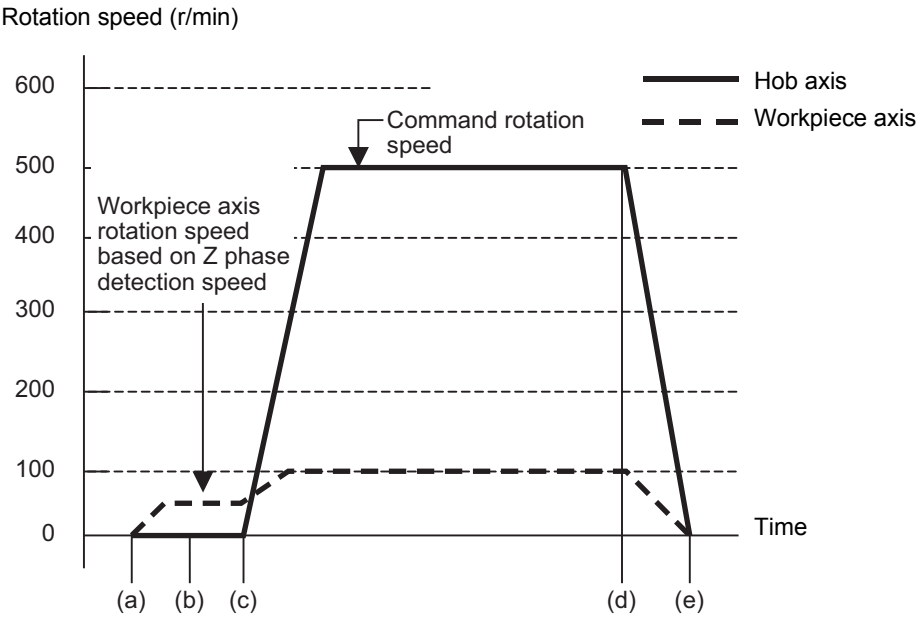
(3) Phase alignment is carried out when the hob axis is stopped and the workpiece axis is rotating.

(4) After the phases are aligned, the hob 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 hob axis rotation speed allowing for the hob axis and workpiece axis rotation ratio, and will enter the synchronized state.

(5) Below is the operation example:

```
Txx00;                  Select a rotary axis
M83 S4=0;               Forward run the fourth spindle (hob axis). (Rotation speed is 0) (a)
:
G114.3 H4 D9 E1 L5 R0;  Hobbing mode ON (phase alignment with the phase difference of zero) (b)
S4=500;                 Rotate the fourth spindle (hob axis) at 500 r/min. (c)
:
M85;                    Stop the fourth spindle. (d)
G113;                   Hobbing mode OFF (e)
```

<Operation>



Compensation control by workpiece axis

(1) Automatic compensation

The workpiece axis is controlled while constantly allowing for hob 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.

When the amount of the compensation added to the workpiece axis by hobbing conditions etc. changes greatly and rapidly, a servo alarm might occur for the workpiece axis. In that case, with the compensation amount through the primary delay filter, this enables the compensation amount fluctuation to further smoothen. However, the more widely the primary delay time constant is set, the more the effect of the compensation decreases, so the effect of the workpiece accuracy might not improve.

[Spindle NC parameter] (Machine parameter)

#3130 syn_spec/bit0 Tool spindle synchronization II (hobbing) automatic compensation selection

OFF: No compensation

ON: Hob axis delay (advance) is compensated with workpiece axis.

#3134 sphtc Tool spindle synchronization II (hobbing) automatic compensation primary delay time constant

0: Primary delay filter control invalid

1 to 32768: Primary delay filter time constant Setting unit (ms)

(2) Command compensation

Errors in the cutting workpiece shape caused by insufficient machine rigidity, etc., are compensated for with the workpiece axis command in the machining program.

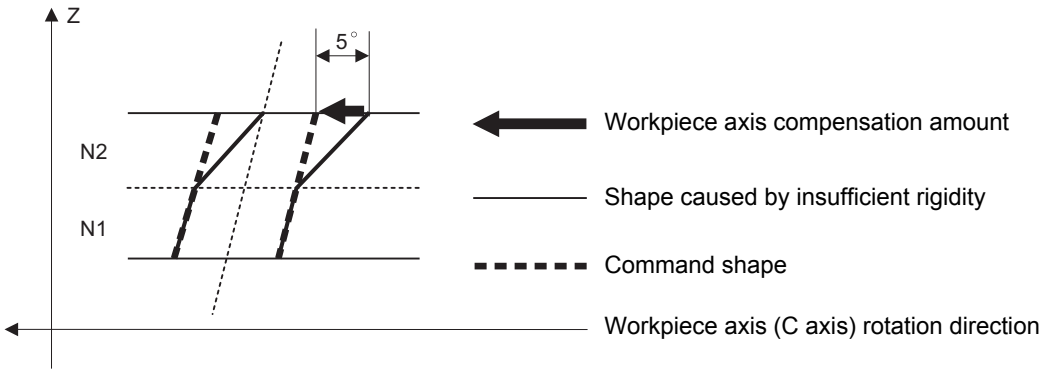
(a) Command the workpiece axis compensation amount as an incremental value.

(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) When the movement command is issued with absolute value for the workpiece axis during the Tool spindle synchronization II (hobbing) mode, a program error (P32) will occur.

<Program example>

```
G114.3 H1 D9 E1 L10 P30. Q100. ;
S1 = 100;
G94;
N1 G01 Z20 F10;
N2 G91 G01 Z20. C5.;                <- Workpiece axis compensation amount
```



Feedforward control during tool spindle synchronization II (hob machining) mode

A feedforward control can be issued for the hob axis and the workpiece axis during the tool spindle synchronization II (hobbing) mode.

(1) The hob axis feedforward control is controlled according to hob axis feedforward gain (#3135 sfwd_g).

(2) The workpiece axis feedforward control is controlled according to hob axis feedforward gain (parameter "#3135 sfwd_g") for the workpiece axis rotation contents of the hob axis rotation. The feedforward control is controlled according to workpiece axis feedforward gain (#2155 hob_fwd_g) for the helical compensation of the Z axis movement.

Tool spindle synchronization II (hobbing) command during hob axis rotation

The hobbing mode command and hobbing cancel mode command can be issued during hob axis rotation.

(1) When the hobbing mode command is issued during hob axis rotation, the rotary axis designated as workpiece axis accelerates up to the speed according to the hob and workpiece axes' rotation ratio command. This acceleration follows the hobbing workpiece axis time constant (#2195 hob_tL) and uses the constant inclination acceleration/deceleration control. If the setting of hobbing workpiece axis time constant is outside the setting range, set the maximum value in the range.

(2) After the workpiece axis finishes acceleration, phase alignment is carried out between hob and workpiece axes if the hobbing mode command contains R command.

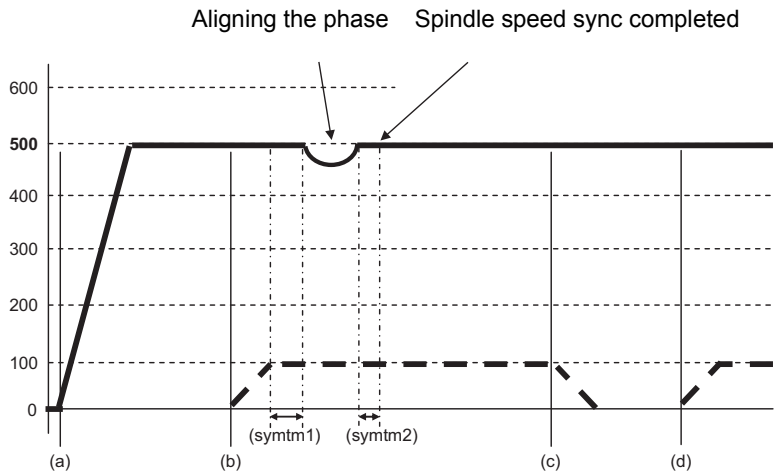
(3) Synchronization is established after phase alignment is completed.

(4) If the hobbing cancel mode command is issued during hob axis rotation, the workpiece axis decelerates and stops. This deceleration follows the hobbing workpiece axis time constant (#2195 hob_tL), and uses the constant inclination acceleration/deceleration. If the setting of hobbing workpiece axis time constant is outside the setting range, set the maximum value in the range.

(5) Below is the operation example:

```
Txx00;                  Select a rotary axis
M** S4=500;             Forward run the 4th spindle (hob spindle) (at 500 r/min). (a)
:
G114.3 H4 D9 E1 L5 R0;  Hobbing mode ON (phase alignment with the phase difference of zero) (b)
:
G113;                   Hobbing mode OFF (c)
:
G114.3 H4 D9 E1 L5 R0   Hobbing mode ON (phase alignment with the phase difference of zero) (d)
:
```

<Operation>



Hob axis

Workpiece axis

(symtm1) Phase sync start confirmation time

(symtm2) Phase sync end confirmation time

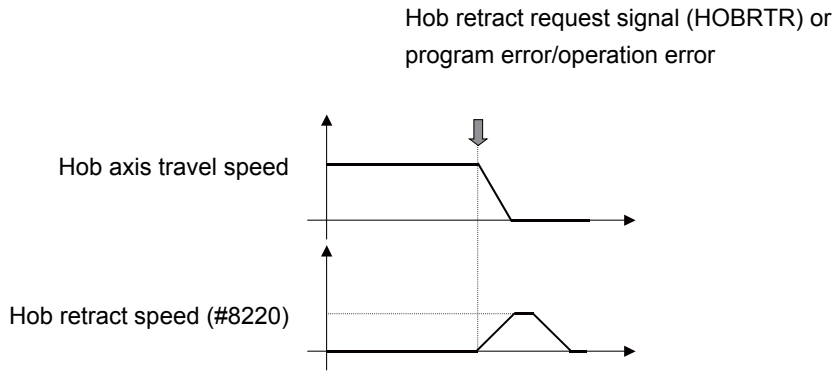
The acceleration/deceleration speed at phase alignment follows the setting of the parameter "#3130 sync_spec/bit1".

0: Step alignment method

1: Multi-step acceleration/deceleration method

Retract during Hobbing Mode

Operating retract during hobbing mode prevents a destruction of the work when hobbing is interrupted.
When Hob retract request signal is input during hobbing mode, the control can retract the axis designated by parameter. Retract operation can also be carried out when a program error or operation error occurs.



(1) The retract operation is carried out when Hob retract request signal (YCDE) is turned ON during hobbing mode. However, the retract operation is not performed while the hob retract complete signal is ON.
Also, after the operation has been completed, the tool is separated from the workpiece by the retract amount in hobbing mode; therefore, hob cutting is not performed properly.

(2) Retract operation can also be carried out when a program error or operation error occurs during hobbing mode. Specify whether to enable or disable a retract by alarm with the parameter "#19406" (Hob retract ON at alarm). However, if the parameter #19406 (Hob retract ON at alarm) is enabled, a retract by alarm is not carried out when the hob alarm retract inhibit signal (YCDF) is turned ON.

(3) Retract is carried out in automatic operation mode. But when it is in automatic mode, retract can also be performed when not in automatic operation. (The hob axis and workpiece axis do not stop.)

(4) After the retract operation is completed, automatic operation pause.
When performing retract operation in automatic operation, retract operation can be interrupted by turning ON Automatic operation stop signal.
But when performing retract in a mode other than automatic operation, retract will not be interrupted by turning ON Automatic operation stop signal.
If the retract operation is interrupted by automatic operation pause or by switching the operation mode (automatic to manual), retract will not be resumed even when you activate automatically after the interruption. However, if a new retract factor occurs after automatic operation has been started, the retract operation is performed by the specified amount.

(5) No retract operation is performed during manual operation mode.

(6) The movement amount in the retract operation is determined by either the parameter "#8219 Hob retract amt 1" or "#8220 Hob retract amt 2", which is specified by the hob retract amount selection signal (YB20).
If the retract amount of all axes are set to "0", retract operation and automatic operation pause are not carried out.

(7) The parameters "#8219 Hob retract amt 1" and "#8220 Hob retract amt 2" are handled as radius values.

(8) Retract speed is set for each axis in the parameter "#8221 Hob retract speed".

(9) The In hob retract signal (XCAE) is ON during the retract operation. This signal is turned ON by either a retract triggered by an alarm or a retract triggered by the hob retract request signal.

(10) When the retract operation is completed, Hob retract complete signal (XCAF) turns ON. This signal is turned ON by either a retract triggered by an alarm or a retract triggered by the hob retract request signal.

(11) When retract is performed, acceleration and deceleration are carried out based on the travel command's acceleration/deceleration mode. However, when the parameter "#19407 Hob ret ac/dc OFF" is set to "1", step-wise acceleration/deceleration is carried out.
When the parameter "#19407 Hob ret ac/dc OFF" is set to "1", and retract speed is relatively fast, a servo alarm (excessive error etc.) may occur.

(12) Retract is not carried out for axes that are in movement.

(13) If mirror image is set for the retract axis, mirror image will be reflected on the retract operation. Therefore, the retract is carried out in the direction opposite to the setting.

(14) Machine lock is enabled for the retract axis.

(15) Automatic interlock is enabled for the retract axis. When not in automatic operation, automatic interlock is enabled for retract axis.

(16) Cutting feed override or rapid traverse override is not reflected on the retract axis.

(17) External deceleration is enabled for the retract axis.

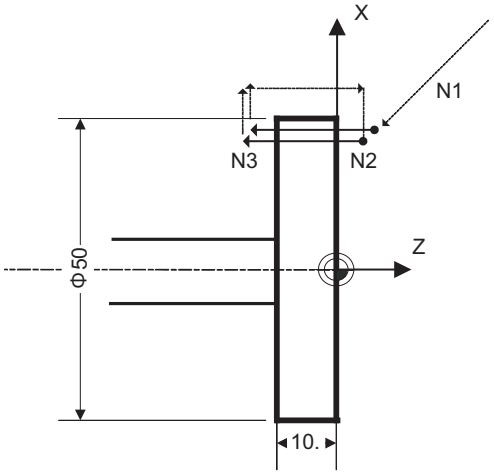
(18) Dry run is disabled for the retract axis.

(19) Pre-interpolation acceleration/deceleration is disabled for a retract axis. Post-interpolation acceleration/deceleration is applied instead.

(20) Retract can be performed for an axis that is under synchronization control. Retraction for a master axis causes the slave axis to move.

(21) Retract can be performed for an axis that is under inclined axis control. In accordance with the inclined axis' movement, the reference axis moves by the compensation amount.

Program example



(1) Spur gear machining without phase alignment

```
          :
          :
        T1100;                  Rotary tool selection
        M43 S4=0;               S4 spindle stop (Rotation command forward rotation ON)
N1      G00 X48. Z5.;
          :
          :
        G114.3 H4 D9 E3 L10:    Tool spindle synchronization II (hobbing) mode ON
                                Hob axis: S4 spindle, workpiece axis: C axis
                                Hob threads: 3, 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 : 3).

        M80;                    Spindle rotation speed synchronization completion confirmation
                                (When the M code of spindle synchronization completion confirmation is set
                                to "M80")
        G94;                    Asynchronous feed mode selection

N2      G01 Z-15. F10;          First cut
        G00 X54.;
            Z5.:
        X46.;
N3      G01 Z-15. F10;          Final cut
        G00 X54.;

        S4=0;                   S4 spindle stop
        G113;                   Tool spindle synchronization II (hobbing) mode OFF
        :
```

(2) Helical gear machining with phase alignment

```
        :
        :
        T1100;                          Rotary tool selection
        M43 S4=0;                       S4 spindle stop (Rotation command forward rotation ON)
N1      G00 X48. Z5.;
        :
        :
        G114.3 H4 D9 E3 L10 P30. Q2000  Tool spindle synchronization II (hobbing) mode ON
        R0.;                            Hob axis: S4 spindle, Workpiece axis: C axis
                                        Hob threads: 3, Rotation ratio: 10 teeth
                                        Torsion angle : 30°, Module : 2 mm
                                        Phase alignment at phase difference 0
        S4=500;                         C axis starts forward run synchronized with S4.
                                        The C axis rotation speed is 150r/min (S4 : C = 10 : 3).

        M80;                            Spindle rotation speed synchronization completion confirmation
                                        (When the M code of spindle synchronization completion confir-
                                        mation is set to "M80")
        G94;                            Asynchronous feed mode selection

N2      G01 Z-15. F10;                  First cut
        G00 X54.;
            Z5.:
        X46.;
N3      G01 Z-15. F10;                  Final cut
        G00 X54.;


        S4=0;                           S4 spindle stop
        G113;                           Tool spindle synchronization II (hobbing) mode OFF
        :
```

(3) When hobbing several times successively with the hob axis rotated
Spur gear machining example with phase alignment

```
T1100;                      Rotary tool selection
M43 S4=0;                    S4 spindle stop (Rotation command forward rotation ON)
G00 X48.Z5.;
:
:
G114.3 H4 D9 E3 L10 R0;     Tool spindle synchronization II (hobbing) mode ON
                            Hob axis:S4 spindle, Workpiece axis: C axis,
                            Hob threads:3, 10 teethe
                            Phase alignment at phase difference 0
S4=500;                     C axis starts forward run synchronized with S4.
                            The C axis rotation speed is 150r/min (S4 : C = 10 : 3).

M80;                        Spindle rotation speed synchronization completion confirmation
                            (When the M code of spindle synchronization completion confir-
                            mation is set to "M80")
G94;                        Asynchronous feed mode selection

G01 Z-15.F10;               First cut
G00 X54.;
    Z5.;
     X46.;
:
G113;                       Tool spindle synchronization II (hobbing) mode OFF
:
G114.3 H4 D9 E3 L10 R0;     Tool spindle synchronization II (hobbing) mode ON
M**;                        Spindle speed synchronization completion confirmation
G01 Z-15.F10;               Nth cut
G00 X54.;
  Z5.;
  X44.;
:
G113;                       Tool spindle synchronization II (hobbing) mode OFF
:
G114.3 H4 D9 E3 L10 R0;     Tool spindle synchronization II (hobbing) mode ON
M80;                        Spindle rotation speed synchronization completion confirmation
G01 Z20. F10;               Final cut
G00 X40.;
   Z-5.;
:
G113;                       Tool spindle synchronization II (hobbing) mode OFF
:
```

Relationship with other functions

Synchronous tapping cycle

(1) The synchronous tapping spindle cannot be commanded as the hob spindle of tool spindle synchronization II (hobbing).
The operation error (M01 1007) will occur, which causes automatic operation to pause.

(2) The synchronous tapping cannot be commanded using the hob axis in tool spindle synchronization II (hobbing).
The operation error (M01 1139) will occur, which causes automatic operation to pause.

Spindle synchronization I, Spindle synchronization II, Tool spindle synchronization IA and IB (spindle-spindle, polygon), or Spindle superimposition control

(1) The Tool spindle synchronization II (hobbing) mode cannot be commanded during Spindle synchronization I, spindle synchronization II, Tool spindle synchronization IA, IB (Spindle-Spindle, Polygon), or Spindle superimposition control ON. An operation error (M01 1005) will occur.

(2) Spindle synchronization I, Spindle synchronization II, Tool spindle synchronization IA, IB (Spindle-Spindle, Polygon), or Spindle superimposition control cannot be commanded during Tool spindle synchronization II (hobbing) mode. An operation error (M01 1005) will occur.

NC reset, Emergency Stop

(1) Even if the reset signal is input, the tool spindle synchronization II (hobbing) maintains synchronization. However, the synchronization is canceled at emergency stop.

Mixed control (Cross axis control)

(1) The mixed control functions cannot be commanded in the part system where the the tool spindle synchronization II (hobbing) is being performed. A program error (P501) will occur. However, hobbing that uses the axis subject to mixed control is possible.

(2) When the mixed control is commanded in the part system where the hobbing has been commanded, the operation error (M01 1035) will occur.

Arbitrary axis exchange command (G140), arbitrary axis exchange return command (G141), reference axis arrange return command (G142)

(1) The arbitrary axis exchange command (G140), arbitrary axis exchange return command (G141) or reference axis arrange return command (G142) cannot be issued in the part system where the tool spindle synchronization II (hobbing) is being performed. A program error (P501) will occur. However, hobbing that uses the axis in arbitrary axis exchange mode is possible.

(2) The workpiece axis of tool spindle synchronization II (hobbing) cannot be commanded as the axis exchange target. Doing so triggers the arbitrary axis exchange disable state.

Door interlock I / Door interlock II

(1) If door interlock I or door interlock II is turned ON during the tool spindle synchronization II (hobbing) mode, tool spindle synchronization II (hobbing) will be canceled because synchronization cannot be maintained.

Precautions and restrictions

(1) When cutting helical gear, correct cutting feed will not be possible in the synchronous feed mode, so always cut in the asynchronous feed mode.

(2) To carry out phase alignment when machining a helical gear, phase alignment will not be carried out correctly if the Z axis is moving, so always carry out phase alignment control when the Z axis is stopped.

(3) The linear-type rotation axis for the absolute position system cannot be used as the hobbing workpiece axis. If used, the absolute position detection alarm (Z70 0002) will occur after the power was turned OFF and ON.

(4) If hobbing control is carried out using the linear-type rotation axis as the hob axis, the current value will be illegal when the hobbing is canceled. In this case, preset the counter after canceling hob machining.

(5) If the hob axis rotation ratio is set to "0", phase alignment will not be carried out. Even if the workpiece axis phase shift amount is commanded, it is ignored.

(6) When "Hob axis delay (advance) allowable angle" (parameter "#3133 spherr") is "0", "Hob axis delay excess" (X18B3) is not output.

(7) During acceleration/deceleration of hob axis, "Hob axis delay angle" (R6516) and "Maximum hob axis delay angle" (R6517) are not updated.

(8) When the maximum hob axis delay (advance) angle (R6516) exceeds the hob axis delay (advance) allowable angle (parameter "#3133 spherr"), CNC only outputs the delay excess signal (X18B3). For information about how to take an action, contact the MTB representative.

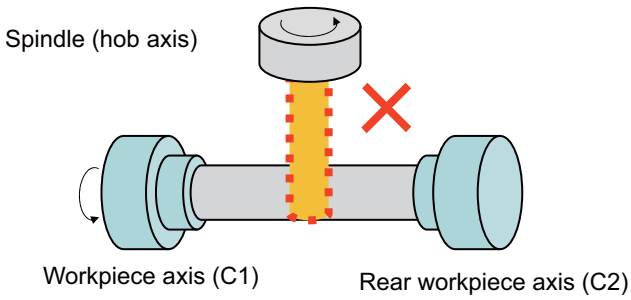
(9) Issue the commands for the workpiece axis from the machining program as a G00 incremental value or G01 incremental value. When an absolute value is commanded, program error (P32) will occur.

(10) Always set the position loop gain of the hob axis and the workpiece axis to the same value. If different values are set, the machining accuracy is not warrantable.

- Hob axis: #13003 SP003, #13036 SP036/bit4
- Workpiece axis (servo axis): #2203 SV003 , #2204 SV004, #2257 SV057
- Workpiece axis (spindle/C axis): #13002 SP002, #13035 SP035/bitC

(11) Do not command hobbing (tool spindle synchronization II) during synchronous control of the workpiece axis (C1 axis) and rear workpiece axis (C2 axis). When hobbing is commanded during synchronous control, rear workpiece axis (C2 axis) and workpiece axis (C1 axis) will not operate in synchronization (because this does not trigger an alarm), and this may twist the workpiece.

[Hobbing model that applies a synchronous control between the workpiece C1 axis and C2 axis]



<Note>

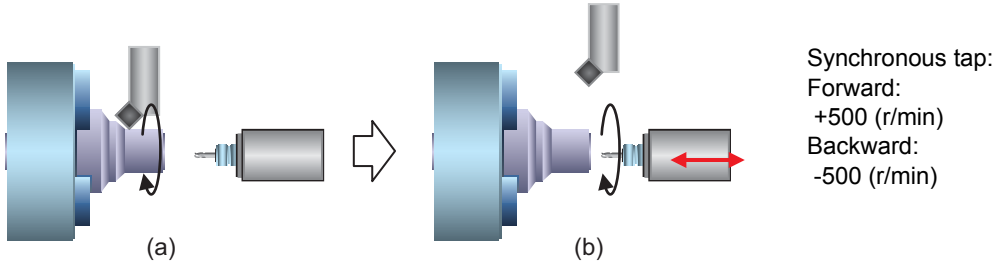
- If hobbing is commanded during synchronous control, only operations for the spindle (hob axis) and workpiece axis (C1 axis) can be carried out.

18.4 Spindle Superimposition ; G164, G113

Function and purpose

Spindles are controlled by superimposing the rotation speed of one spindle on the rotary speed of other spindle. Use this function when the tool spindle needs to be rotated with the superimposed speed on the spindle rotation speed.
If this function is used when tapping is performed to the center of a workpiece clamped to a spindle during its rotation, for instance, the tapping can be performed in the center of the workpiece (shown in Figure (b) below) during turning machining (shown in Figure (a) below) so that the cycle time can be reduced.

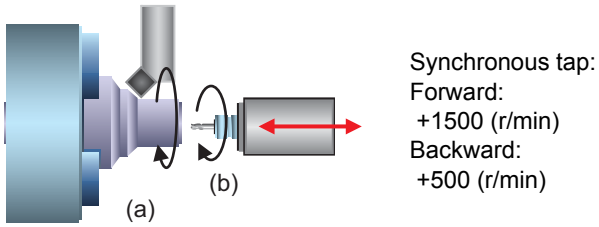
(1) Conventional method



S1 = 1000 (r/min)
Normal turning machining

S1 = 0 (r/min)
The rotation of the front workpiece is stopped, and then the tapping is started at the front center.

(2) Using the spindle superimposition control



S1 = 1000 (r/min)
Turning (a) and tapping at the front center (b) are performed simultaneously

The following describes the meanings of the terms used in this manual.

Term	Meaning
Command rotation speed	Rotation speed that is commanded for the reference spindle or superimposed spindle
Superimposed-spindle rotation speed	Spindle end rotation speed of the superimposed spindle. (including the rotation speed of the reference spindle)
Spindle superimposition control state	State in which the reference or superimposed spindle is in synchronous control.
Differential velocity tap	Synchronous tap cycle under superimposed-spindle control (tap machining by superimposed spindle.)

Command with spindle superimposition and other spindle synchronization functions combined

The other spindle synchronization functions (Spindle synchronization I/ Spindle synchronization II/ Tool spindle synchronization IA/ Tool spindle synchronization IB/ Tool spindle synchronization II/ Spindle superimposition control) cannot be commanded while this function is running. Also, this function cannot be commanded while the above functions are being executed. In these cases an operation error (M01 1005) will occur.
However, when the Multiple spindle synchronization set control function is enabled, multiple spindle synchronization functions can be commanded simultaneously. The validity of this function depends on the MTB specifications (parameter "#1440 multi_sp_syn"). For details, refer to "18.5 Multiple Spindle Synchronization Set Control".

Command format

Spindle superimposition valid command

```
G164 H__D__;
```

H	Reference spindle selection (Command the spindle number or spindle name.) (*1) [Setting range] Spindle No.: 1 to n (n: Maximum number of available spindles) Spindle name: 1 to 9
D	Superimposed spindle selection (Command the spindle number or spindle name.) (*1) Command the rotation direction for the reference spindle using the sign of the D address value. [Setting range] Spindle No.: 1 to n or -1 to -n (n: Maximum number of available spindles) Spindle name: 1 to 9 or -1 to -9

(*1) There are two spindle command methods: spindle number method and spindle name method.
Command with the spindle names, only when all spindles are set with the spindle name parameter (#3077 Sname) (from 1 to 9). For others, command with the spindle number. These settings depend on the MTB specifications.

Note

(1) If the spindle that is not located at the H or D address is commanded, it causes a program error (P35).
(2) If the analog spindle is set to the H or D address, it causes a program error (P700).
(3) If the H or D address is not commanded, it causes a program error (P33).

Spindle superimposition control cancel command

```
G113;
```

Note

(1) Canceled by the PLC signal or emergency stop. The operation of the PLC signal depends on the MTB specifications.
(2) An axis address that involves any travel cannot be put in the same block as the spindle superimposition cancel command. Doing so causes the program error (P33) when the cancel command is issued, which causes automatic operation to pause.

Detailed description

Superimposed-spindle operation

(1) The spindle is set in the spindle superimposition mode with the spindle superimposition control command (G164). (The PLC signal is set on.)

(2) The rotation speed for the reference spindle or superimposed spindle can be changed during spindle superimposition.

(3) The rotation speed and rotation direction of the reference spindle are set to those commanded for the reference spindle.

(4) If the forward run or reverse run command is input to the superimposed spindle, the spindle accelerates or decelerates up to the rotation speed superimposed with the reference spindle, and it is placed into the spindle superimposition control state after the superimposition speed has been reached. (The PLC signal is set on.) Check that the spindle is set in the spindle superimposition control state, and start machining. The method for checking the state of the PLC signal depends on the MTB specifications.

(5) If the speed of the superimposed spindle is changed in the spindle superimposition control state, the superimposed spindle accelerates or decelerates up to the rotation speed superimposed with the reference spindle.

(6) The rotation speed of the superimposed spindle is obtained in the following formula.

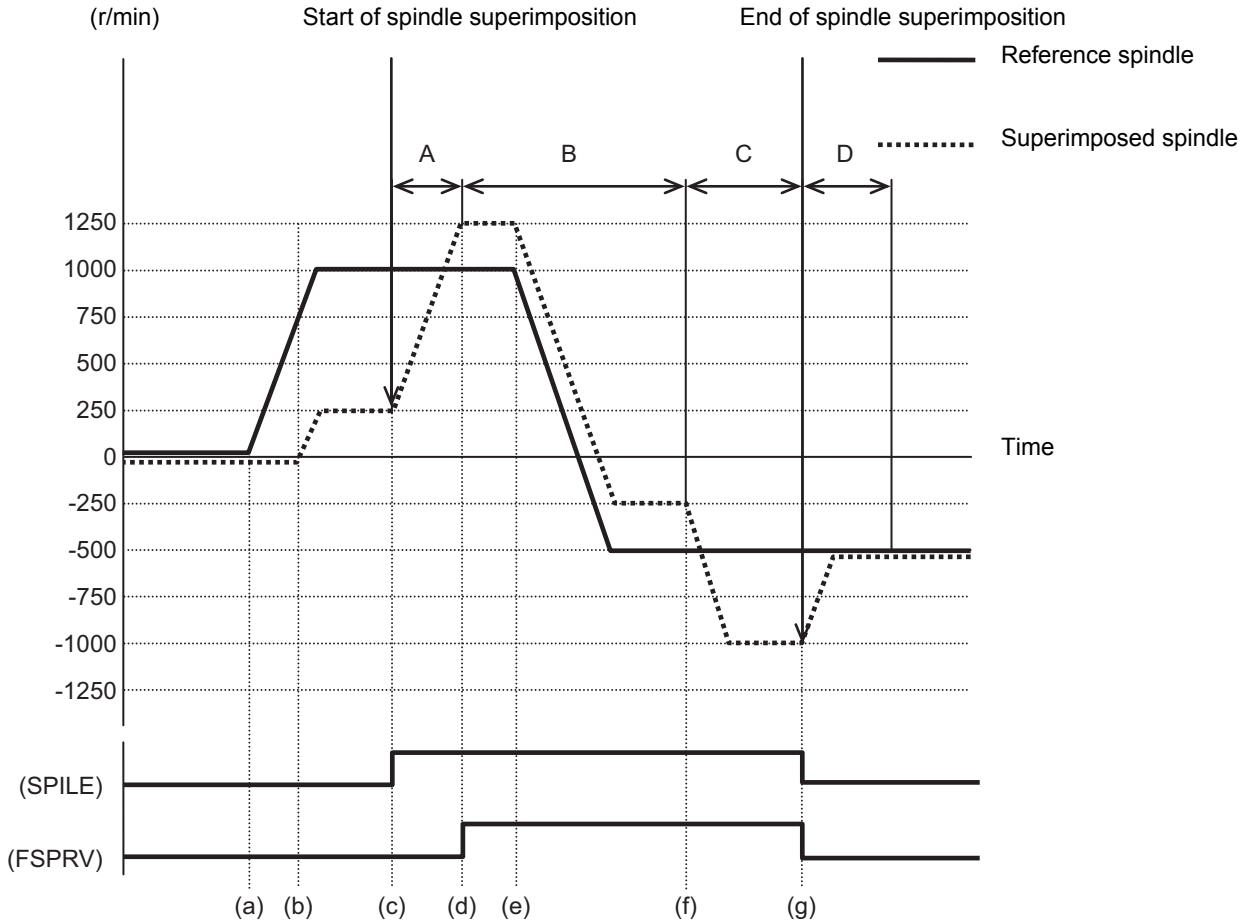
[Superimposed rotation speed] = ([Sign of D command] x [Rotation direction of reference spindle command] x [Rotation speed of reference spindle command]) + ([Rotation direction of superimposed spindle command] x [Rotation speed of superimposed spindle command])

(7) If the spindle superimposition cancel command (G113) is issued, the spindle superimposition control state or spindle superimposition mode is canceled. (The PLC signal is set to OFF.)

(8) The program example below shows the operation of the reference spindle under spindle superimposition and superimposed spindle.

<Program example>

M03 S1=1000	Command 1000 (r/min) to the 1st spindle with the forward run command. (a)
:	
M33 S3=250	Command 250 (r/min) to the 3rd spindle with the forward run command. (b)
:	
G164H1D3	Superimpose the 3rd spindle (superimposed spindle) on the 1st spindle (reference spindle) with the forward run command. (c)
Mxxxx	Check that spindle superimposition is completed. (d)
M4 S1=500	Command 500 (r/min) to the 1st spindle (reference spindle) with the reverse run command. (e)
:	
M34 S3=500	Command 500 (r/min) to the 3rd spindle (superimposed spindle) with the reverse run command. (f)
:	
G113	Cancel the spindle superimposition control. (g)
:	



SPILE (Under spindle superimposition control mode): Setting ON outputs the PLC signal indicating that the spindle is in the spindle superimposition control mode.
FSPRV (Spindle rotation speed synchronization completion): Setting ON outputs the PLC signal indicating that the spindle superimposition has been completed.

(a) The rotation speed of the superimposed spindle in the superimposition mode between intervals A and C is obtained in the following formula.

[Superimposed rotation speed] = ([Sign of D command] x [Rotation direction of reference spindle command] x [Rotation speed of reference spindle command]) + ([Rotation direction of superimposed spindle command] x [Rotation speed of superimposed spindle command])

(b) If the spindle superimposition is commanded, the superimposed spindle accelerates in the speed mode up to the rotation speed that is obtained in the formula shown in item (1). (Interval A)

(c) In interval B, the rotation speed command of the reference spindle is changed from 1000 (r/min) to -500 (r/min), and the superimposed spindle also accelerates/decelerates synchronously with the acceleration/deceleration of the reference spindle. The rotation speed of the superimposed spindle is changed from 1250(r/min) to -250(r/min).

(d) In the superimposition mode, the reference spindle or superimposed spindle accelerates/decelerates (between intervals B and C) as shown below.
First, each spindle synchronization time constant parameter (#3049 spt) is compared.
The spindle is accelerated/decelerated based on the spindle synchronization multi-step acceleration/deceleration speed parameter settings (combination of "#3049 spt" and "#3054 sptc1" to "#3060 sptc7" and combination of "#3061 spdiv1" to "#3067 spdiv7") of the time constant that is longer. These parameters are all determined in the MTB specifications.

(e) The spindle superimposition mode signal is set on with the spindle superimposition command (G164). This is set off with the cancel command.

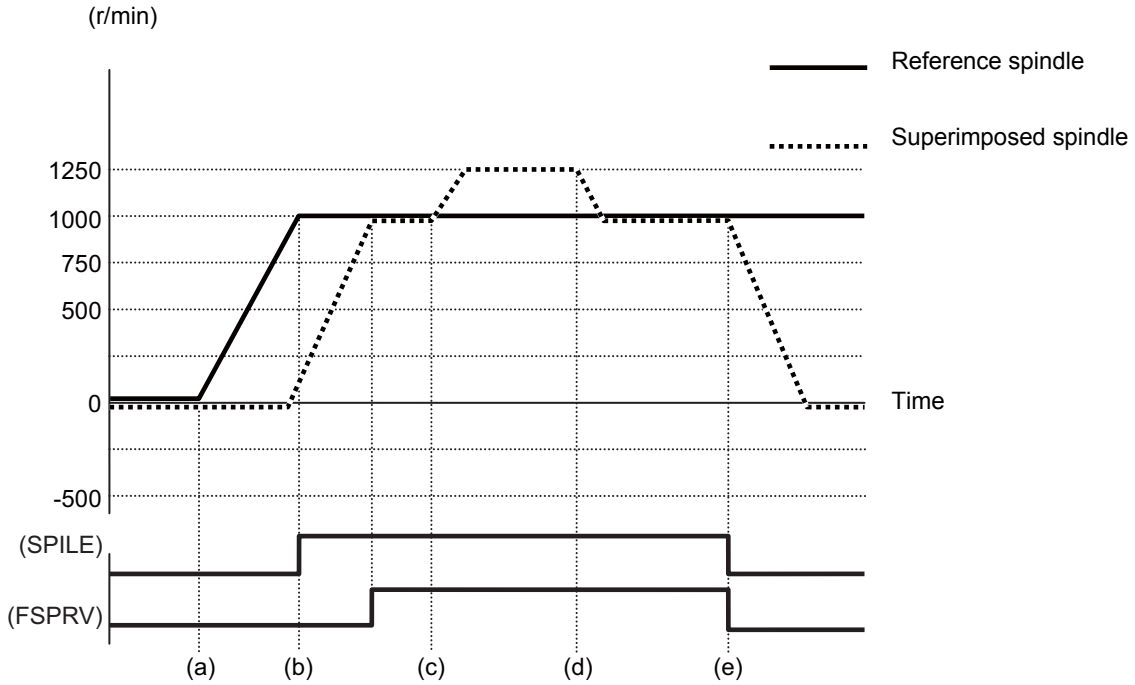
(f) The spindle rotation speed synchronization completion signal is set to ON when each of the reference and superimposed spindles reaches the commanded rotation speed in the spindle superimposition mode. Be sure to check that this signal is set on before starting machining. This is set off with the cancel command. The operation of the PLC signal depends on the MTB specifications.

(g) After the spindle superimposition is canceled (interval D), the superimposed spindle decelerates in the speed mode, and finally reaches the commanded rotation speed.

(9) If the forward run command or reverse run command is not input to the superimposed spindle when the spindle superimposition is commanded, the superimposed spindle is set in the servo ON state, and accelerates up to the rotation speed of the reference spindle to be set in the spindle superimposition control state (in the same way as when the rotation speed 0 (r/min) is commanded to the superimposed spindle).
If the forward run command or reverse run command is input in this state, the rotation speed is set to the value based on the "rotation speed of the reference spindle", "rotation direction of the reference spindle by the D command", and "rotation speed commanded to the superimposed spindle" that are commanded in advance. If the forward run command or reverse run command of the superimposed spindle in the spindle superimposition state is set off, the spindle accelerates/decelerates up to the rotation speed of the reference spindle (in the same way as when the rotation speed 0 (r/min) is commanded to the superimposed spindle).

<Program example>

```
Sl=1000;          Command 1000r/min to the 1st spindle (reference spindle).
    :
M03;              1st spindle (reference spindle) forward run command (a)
    :
G164H1 D3 ;       Superimpose the 3rd spindle (superimposed spindle) on the 1st spindle (reference spindle) with the forward run command. (b)
    :
S3=250;           Command 250r/min to the 3rd spindle (superimposed spindle).
    :
M33;              3rd spindle (superimposed spindle) forward run command (c)
    :
M35;              3rd spindle (superimposed spindle) forward run command OFF (d)
    :
G113;             Cancel the spindle superimposition. (e)
```

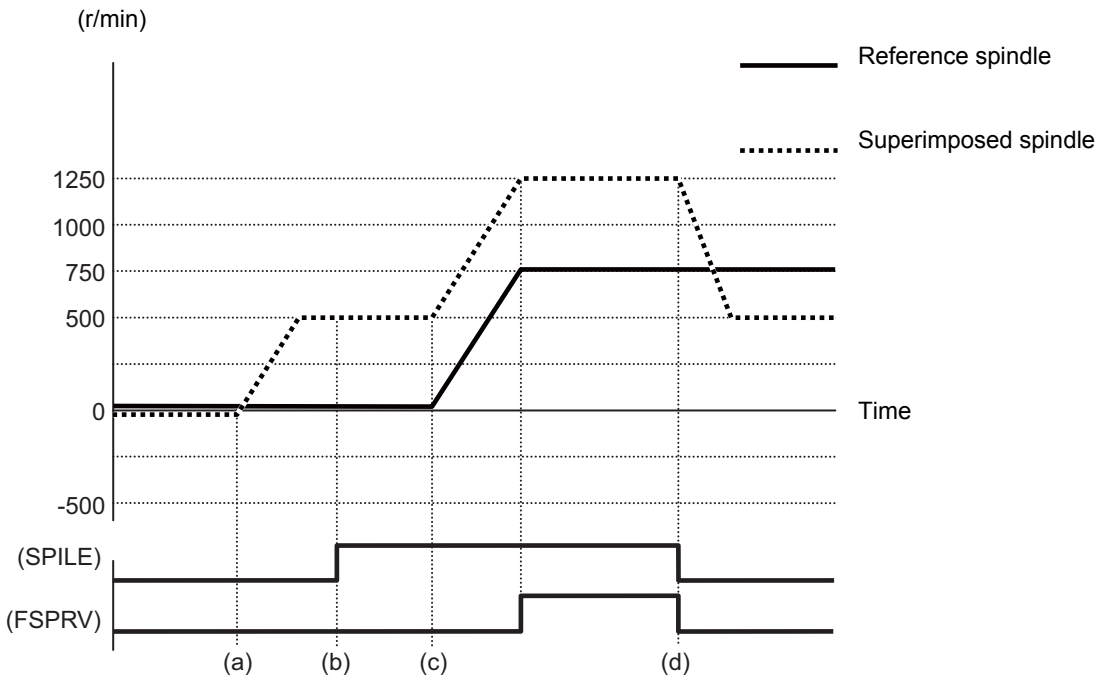


SPILE (Under spindle superimposition control mode): Setting ON outputs the PLC signal indicating that the spindle is in the spindle superimposition control mode.
FSPRV (Spindle rotation speed synchronization completion): Setting ON outputs the PLC signal indicating that the spindle superimposition has been completed.

(10) If neither the forward run command nor the reverse run command is input to the reference spindle when the spindle superimposition control is commanded, the superimposed spindle runs normally. After this, if the forward run command or reverse run command is input to the reference spindle, the rotation speed accelerates/decelerates up to the rotation speed based on the "rotation speed of the reference spindle", "rotation direction of the reference spindle by the D command", and "rotation speed commanded to the superimposed spindle" that are commanded in advance. Then, it is set into the spindle superimposition state.

<Program example>

S3=500;	Command 500r/min to the 3rd spindle (superimposed spindle).
:	
M33;	3rd spindle (superimposed spindle) forward run command (a)
:	
G164H1 D3 ;	Superimpose the 3rd spindle (superimposed spindle) on the 1st spindle (reference spindle) with the forward run command. (b)
:	
S1=750;	Command 750r/min to the 1st spindle (reference spindle).
:	
M03;	1st spindle (reference spindle) forward run command (c)
:	
G113;	Cancel the spindle superimposition. (d)



SPILE (Under spindle superimposition control mode): Setting ON outputs the PLC signal indicating that the spindle is in the spindle superimposition control mode.
FSPRV (Spindle rotation speed synchronization completion): Setting ON outputs the PLC signal indicating that the spindle superimposition has been completed.

Acceleration/deceleration under spindle superimposition

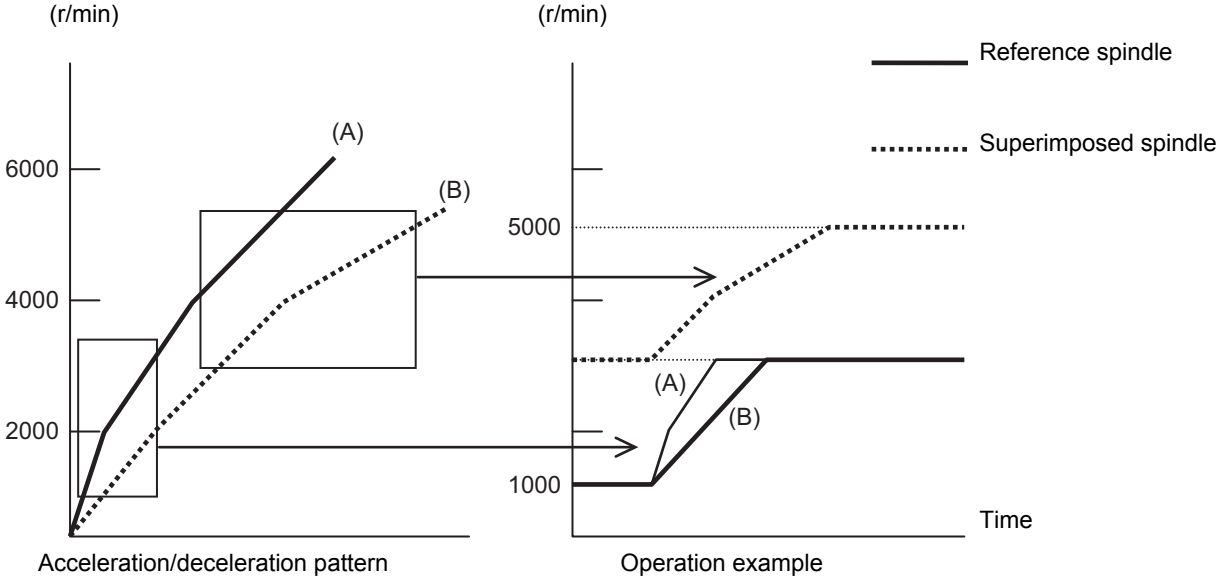
(1) Acceleration/deceleration of reference spindle or superimposed spindle under spindle superimposition

When the reference spindle or superimposed spindle accelerates or decelerates during spindle superimposition, the superimposed spindle also accelerates or decelerates synchronously with the acceleration/deceleration of the reference spindle. Therefore, select a shelving acceleration/deceleration pattern of the reference spindle or superimposed spindle acceleration/deceleration patterns in order to perform acceleration/deceleration. This is also applied to the acceleration/deceleration pattern particular to the superimposed spindle.

<Example of operation>

The following shows an operation example when the rotation speed commanded to the reference spindle is changed to 3000 (r/min) while the reference spindle and superimposed spindle set to the acceleration/deceleration pattern shown below is rotating at 1000 (r/min) for the reference spindle and at 3000 (r/min) for the superimposed spindle in the spindle superimposition state.

The reference spindle is set to the acceleration/deceleration pattern (A), but the superimposed spindle is set to the shelving acceleration/deceleration pattern; therefore, select the acceleration/deceleration pattern (B) of the superimposed spindle to perform acceleration/deceleration.



(2) When the acceleration/deceleration speed is superimposed between the reference and superimposed spindles:

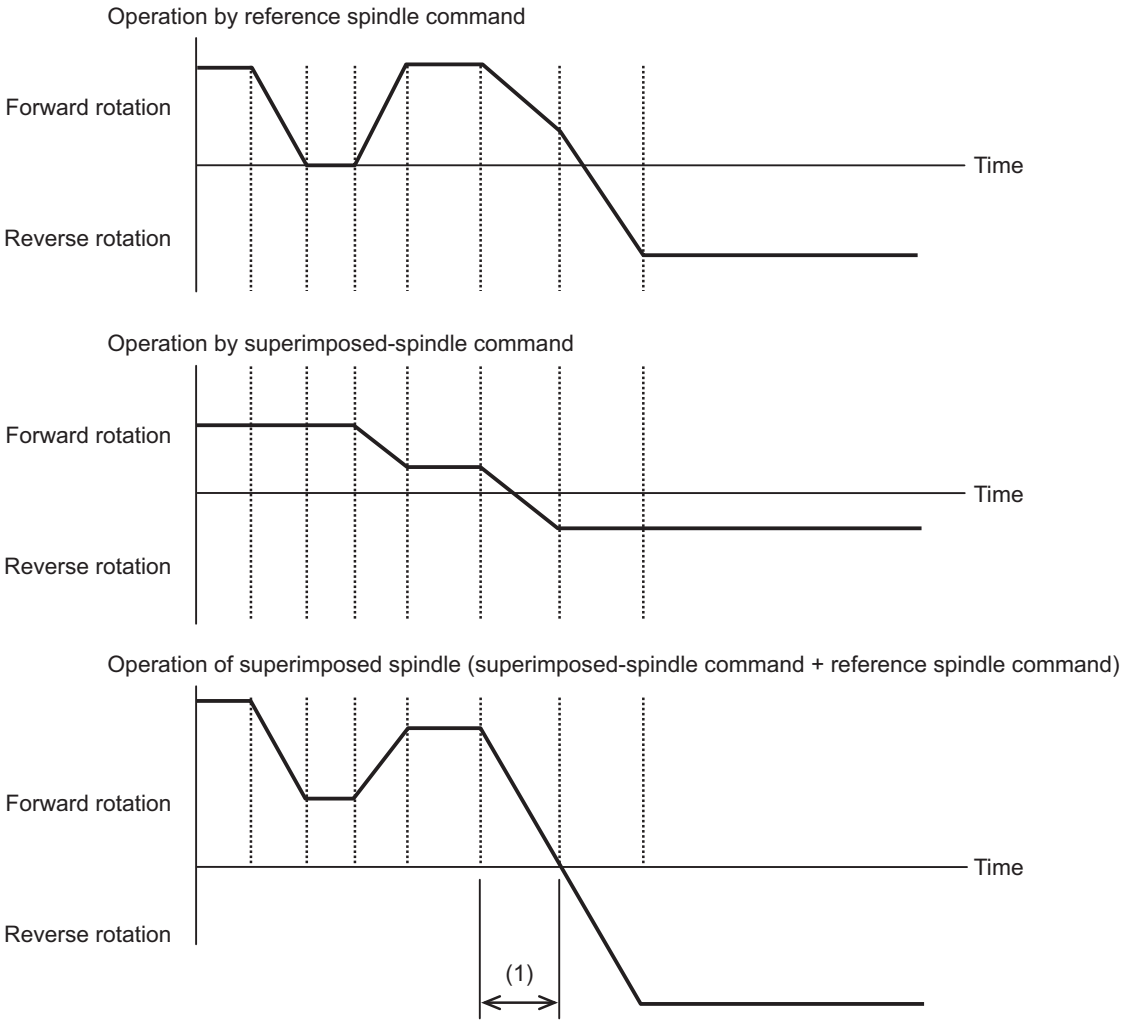
If the command speed or rotation direction of the reference spindle or superimposed spindle is changed in the spindle superimposition state, the spindle accelerates/decelerates based on the spindle synchronization acceleration/deceleration speed time constant or the time constant of the doubled spindle synchronization acceleration/deceleration speed time constant that is specified in the parameter.

The table below shows which time constant is to be used.

When based on the spindle synchronization acceleration/deceleration speed time constant, "x1" is shown. When based on the doubled spindle synchronization acceleration/deceleration speed time constant, "x2" is shown.

These parameter settings depend on the MTB specifications.

			Reference spindle				
			At acceleration		At deceleration		In normal mode
			Forward rotation	Reverse rotation	Forward rotation	Reverse rotation	
Superimposed spindle	Forward rotation	At acceleration in forward run	x2	x1	x1	x2	x1
		At acceleration in reverse run	x1	x2	x2	x1	x1
		At deceleration in forward run	x1	x2	x2	x1	x1
		At deceleration in reverse run	x2	x1	x1	x2	x1
		In normal mode	x1	x1	x1	x1	x1
	Reverse rotation	At acceleration in forward run	x1	x2	x2	x1	x1
		At acceleration in reverse run	x2	x1	x1	x2	x1
		At deceleration in forward run	x2	x1	x1	x2	x1
		At deceleration in reverse run	x1	x2	x2	x1	x1
		In normal mode	x1	x1	x1	x1	x1



In interval "A", the forward-run deceleration of the reference spindle and the forward- and reverse-run deceleration of the superimposed spindle are carried out frequently; therefore, the acceleration/deceleration time constant is doubled for both the reference and superimposed spindles.

Rotation speed clamp

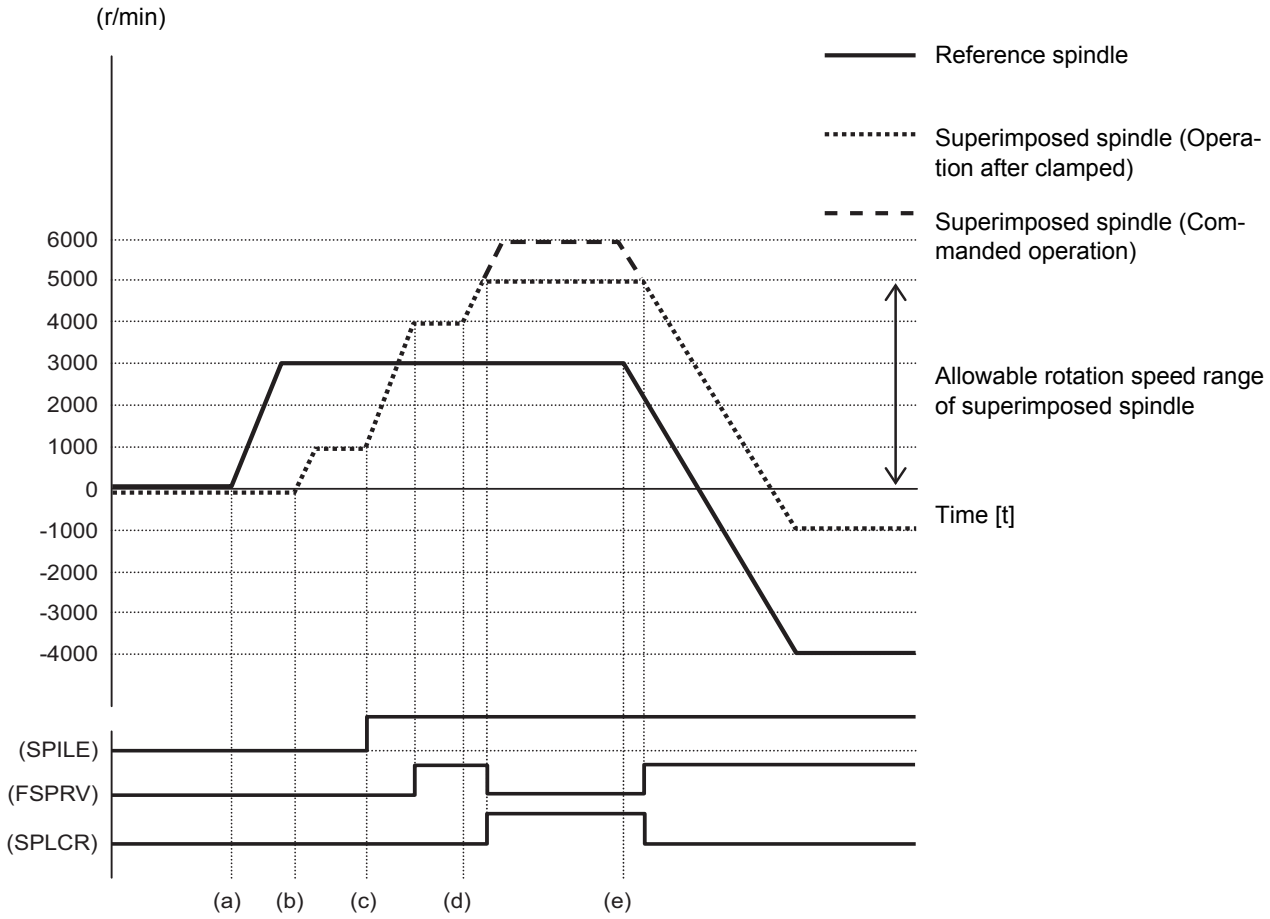
(1) Maximum rotation speed clamp

In the spindle superimposition state, the reference spindle is clamped at the maximum rotation speed of the lower rotation speed of the maximum rotation speed of the reference spindle and the maximum rotation speed of the superimposed spindle that are designated in parameters. The superimposed spindle is clamped if the maximum rotation speed of the superimposed spindle is exceeded. In this case, a displacement occurs between the reference and superimposed spindles.
If either the reference spindle or the superimposed spindle is clamped, the state shifts to under spindle superimposition clamp. (The PLC signal is set on.)

After the spindle superimposition has been commanded, a clamp is triggered if the maximum rotation speed of the superimposed spindle is exceeded before the spindle is placed into the spindle superimposition state. In this case, a displacement also occurs between the reference and superimposed spindles, and the state shifts to under spindle superimposition clamp. (The PLC signal is set on.)

<Program example>

S1=3000;	Command 3000r/min to the 1st spindle (reference spindle).
:	
M03;	1st spindle (reference spindle) forward run command (a)
:	
S3=1000;	Command 1000r/min to the 3rd spindle (superimposed spindle).
:	
M33;	3rd spindle (superimposed spindle) forward run command (b)
:	
G164 H1 D3;	Superimpose the 3rd spindle (superimposed spindle) on the 1st spindle (reference spindle) with the forward run command. (c)
:	
S3=3000;	Command 3000r/min to the 3rd spindle (superimposed spindle). (d)
M4 S1=4000;	Command 4000r/min to the 1st spindle (reference spindle). (e)



(2) Minimum rotation speed clamp

In the spindle superimposition state, the minimum rotation speed clamp is not carried out for the superimposed spindle.

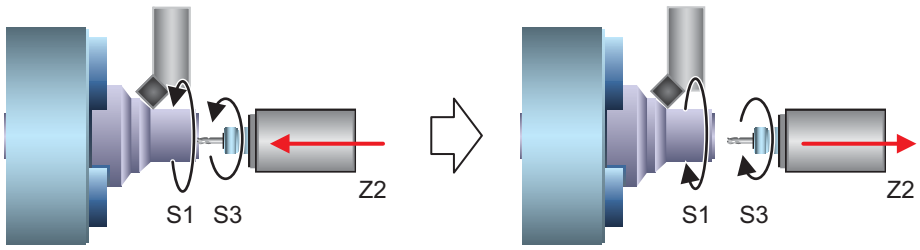
If the rotating spindle is set to the superimposed spindle under spindle superimposition at the minimum rotation speed, the minimum rotation speed clamp is canceled, and the spindle rotates at the commanded rotation speed. When the spindle is released from the superimposed spindle, its minimum rotation speed clamp is enabled again.

Relationship with other functions

Synchronous tapping cycle

(1) If the synchronous tap cycle is commanded to the reference spindle in the spindle superimposition state, it causes an operation error (M01 1131).

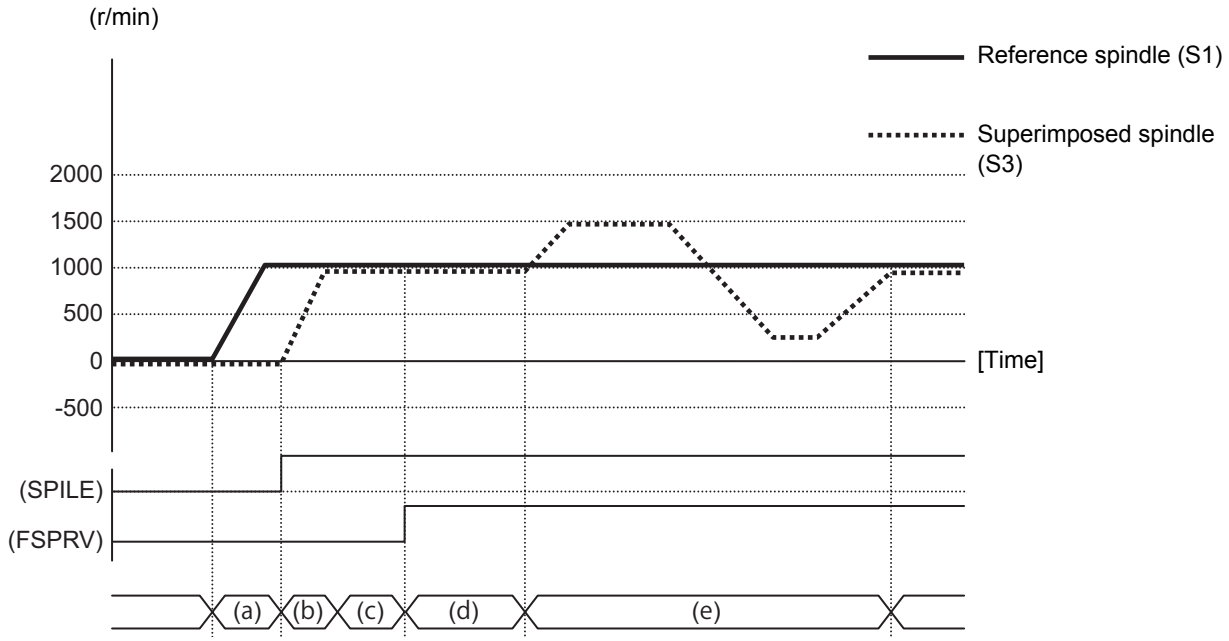
(2) The synchronous tap cycle command is enabled for the superimposed spindle under the spindle superimposition control state. (This is hereinafter referred to as "differential velocity tap".)
The example below shows a differential velocity tapping case in which tapping is carried out by the rotary tool while the main spindle is turning. The tap axis (Z2) and tool spindle (S3) are synchronized with the synchronous tap cycle command to carry out tapping by using the main spindle (S1) as the reference spindle and rotating the tool spindle (S3) as the superimposed spindle.



S1 = 1000 (r/min)
Synchronous tap S3
Forward: +1500 (r/min)
Differential speed between S1 and S3: 500 (r/min)

S1 = 1000 (r/min)
Synchronous tap S3
Backward: +250 (r/min)
Differential speed between S1 and S3: 750 (r/min)

$1	$2	
:	:	
M03 S1=1000;	:	[$1]S1=1000r/min Forward run command ... (a)
:	G164 H1 D3;	[$2] Spindle superimposition control (b) S1: Reference spindle, S3: Superimposed spindle
:	M***;	[$2] Spindle superimposition completed (c)
:	G0Z-2;	[$2] Z2 axis positioning (d)
:	G98 G84 X0. Z10. R2. F1. P0.5 S500 ,S750 ,R1;	[$2] Synchronous tap command (Forward tapping, Pitch: 1mm, Forward speed: 500r/min, Backward speed: 750r/min) (e)



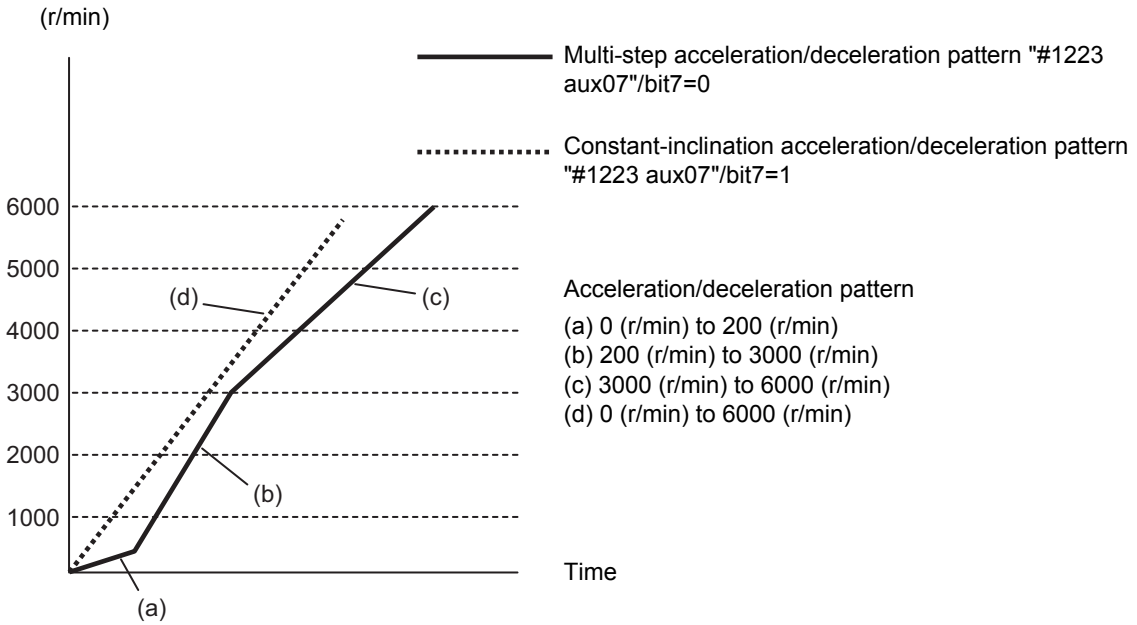
(3) In the spindle superimposition control state, the acceleration/deceleration of the differential velocity tap is performed with the synchronous tap acceleration/deceleration pattern (constant inclination or multi-step acceleration/deceleration) of the superimposed spindle side in the area of (commanded rotation speed of reference spindle) + (commanded rotation speed for synchronous tap of superimposed spindle). There are two types of acceleration/deceleration patterns: constant inclination and multi-step acceleration/deceleration. The pattern to be used depends on the MTB specifications. (Parameter "#1223 aux07 bit7")

<Example of operation>

The example below shows how to select the synchronous tapping time constant when the synchronous tap acceleration/deceleration time constant is designated as shown below.

When the rotation speed command of the differential velocity tap is set to 500(r/min) while the reference spindle is rotating at 3000 (r/min), the rotation speed of the differential velocity tap spindle is set to 2500(r/min) to 3500(r/min).

Types of acceleration/deceleration patterns	Rotation speed area	Pattern to be used
Multi-step acceleration/deceleration	2500(r/min) to 3000(r/min)	(b) shown below
	3000(r/min) to 3500(r/min)	(c) shown below
Constant-inclination acceleration/deceleration	(d) shown below	



(4) The spindle is decelerated and stopped even when the differential velocity tap starting operation is set as the reference position return. These settings depend on the MTB specifications (parameter "#3106 zrn_typ/bit4).

(5) The normal synchronous tap cycle uses the interpolation mode (#13002 SP002) as the spindle position loop gain, but the spindle synchronous position loop gain (#13003 SP003) is used during differential velocity tapping. Therefore, the feed axis position loop gain at differential velocity tapping is automatically switched to the spindle synchronous position loop gain.
However, it differential velocity tapping is carried out in combination with the control axis superimposition, a greater synchronization error occurs if the gain of the control axis / superimposed axis (reference or superimposed axis) is not the same as that of the synchronous tap spindle.
Therefore, the axis servo gain (#2017 tap_g) of the reference axis in the control axis superimposition mode and the superimposed spindle must be set to the same value as that of the interpolation mode position loop gain (#13002 SP002) and spindle synchronous position loop gain (#13003 SP003) of the superimposed spindle. These parameter settings vary depending on the MTB specifications. Refer to the instruction manual issued by the MTB for details.

(6) A command (S command, spindle override, rotation command OFF, or spindle stop), which changes the rotation speed of the reference spindle during differential velocity tapping, is ignored even if issued. Such a command is enabled after differential velocity tapping has been completed. While a command is ignored, the speed change invalid signal (SPNCH) is output.

(7) If the rotation speed of the superimposed spindle is clamped when the differential velocity tap command is issued, it causes an operation error (M01 1132), and machining stops.

(8) If the constant surface speed control is commanded to the reference or superimposed spindle during differential velocity tapping, it causes an operation error (M01 1133), and machining stops.

(9) If the differential velocity tap is commanded to the reference or superimposed spindle during constant surface speed control, it causes an operation error (M01 1133), and machining stops.

(10) If the synchronous tapping spindle is commanded as the reference or superimposed spindle of the spindle superimposition control, an operation error (MOI 1007) will occur.

Spindle override (SP1 to SP4)

(1) If the spindle override of the reference spindle is changed in the spindle superimposition control state, the spindle runs in the same way as when the commanded rotation speed of the reference spindle is changed.

(2) If the spindle override of the superimposed spindle is changed in the spindle superimposition control state, the spindle runs in the same way as when the commanded rotation speed of the superimposed spindle is changed. When the override is set to 0%, the spindle runs in the same way as when the rotation speed is set to 0 (r/min).

Spindle position control (Spindle/C axis control)

The C axis control is not applicable to the reference or superimposed spindle in the spindle superimposition mode. If the C axis control is commanded, it causes an operation error (M01 1026). Also, if the spindle superimposition control for the reference or superimposed spindle is commanded to the spindle in the C axis mode, it causes an operation error (M01 1026).
Before commanding the C axis control, cancel the spindle superimposition mode.

Constant surface speed control

(1) The constant surface speed control command is enabled for the reference spindle in the spindle superimposition control state.
The spindle superimposition command can be issued while the reference spindle is in process of constant surface speed control.

(2) If the constant surface speed control is commanded to the superimposed spindle in the spindle superimposition control state, the spindle is superimposed by the rotation speed that is calculated with the constant surface speed control formula. When the spindle superimposition is commanded to the superimposed spindle during constant surface speed control, the spindle is superimposed by the rotation speed that is calculated with the constant surface speed control formula.

(3) If the constant surface speed control is commanded to the reference spindle or superimposed spindle during differential velocity tapping, it causes an operation error (M01 1133), and machining stops.

Spindle clamp speed setting

(1) Maximum clamp rotation speed (G92 S command)
The maximum clamp rotation speed specified with the address S following G92 is valid for the reference spindle or superimposed spindle.
The maximum clamp rotation speed of the superimposed spindle is obtained by adding the rotation speed commanded to the reference spindle to that commanded to the superimposed spindle. If the rotation speed of the reference spindle or superimposed spindle exceeds the rotation speed at which either the reference spindle or the superimposed spindle is clamped at a maximum , the spindle superimposition clamping signal is set to ON.

(2) Maximum clamp rotation speed (G92 Q command)
The minimum clamp rotation speed specified with the address Q following G92 is valid for the reference spindle, but invalid for the superimposed spindle.
If the rotating spindle is set to the superimposed spindle at the minimum clamp rotation speed, the minimum rotation speed clamp is canceled, and the spindle rotates at the commanded rotation speed. If the spindle superimposition control state is canceled, the minimum clamp rotation speed is enabled.

Feed per revolution (synchronous feed)

(1) Feed per revolution is set to the feedrate per spindle revolution (mm/rev) of the differential velocity rotation speed for the superimposed spindle. This enables a tap cycle, etc. with floating tap chuck in the spindle superimposition state.

(2) When the feed per revolution is set to the superimposed spindle, the superimposition state must not be canceled during feed per revolution.
If it is canceled, the spindle rotation speed changes from the differential velocity rotation speed to the superimposed-spindle rotation speed; therefore, the feed axis speed varies, and the machining surface may be damaged.

Spindle synchronization, Tool spindle synchronization I (polygon), and Tool spindle synchronization II (hobbing)

(1) If the Spindle superimposition control is commanded to the Spindle under spindle synchronization I or II, Tool spindle synchronization I (A/B/C), or Tool spindle synchronization II, it causes an operation error (M01 1005).

(2) If the spindle synchronization I or II, tool spindle synchronization I (A/B/C), or tool spindle synchronization II is commanded to the reference spindle or superimposed spindle in the spindle superimposition control state, it causes an operation error (M01 1005).

High-speed synchronous tapping

When the high-speed synchronous tapping is enabled in the specifications, the normal synchronous tapping is applied during spindle superimposition. The high-speed synchronous tapping depends on the MTB specifications (parameter "#1281 ext17/bit5").

Tapping return

When differential velocity tapping is canceled by emergency stop or reset, whether tapping return is to be enabled depends on the MTB specifications. Refer to the instruction manual issued by the MTB for details.
If reset or emergency stop is performed during differential velocity tapping, the differential velocity state cannot be maintained, causing the screw section to be cut off. The tap tool can be removed from the workpiece, but the workpiece will become defective.

Spindle stop signal

If the spindle stop signal is input to the reference spindle in the spindle superimposition state, the spindle runs in the same way as when the spindle stop or rotation speed 0 (r/min) is commanded to the reference spindle.
If the spindle stop signal is input to the superimposed spindle, the spindle runs in the same way as when the spindle stop or rotation speed 0 (r/min) is commanded to the superimposed spindle. (The operation with a signal depends on the MTB specifications.)

Spindle orientation signal

The spindle orientation is not applicable to the reference or superimposed spindle in the spindle superimposition mode.
If the orientation is commanded, it causes an operation error (M01 1025). Also, if the spindle superimposition control for the reference or superimposed spindle is commanded to the spindle under orientation, it causes an operation error (M01 1025).
Before commanding the orientation, cancel the spindle superimposition mode.

Switching the spindle gear

The spindle gear switching is not applicable to the reference or superimposed spindle in the spindle superimposition control mode.
Before commanding the spindle superimposition, switch the gear mode.
The spindle superimposition state or each PLC signal of spindle gear shift command 1 or 2 is kept in the state in which the spindle superimposition commanded.

Zero speed signal

The superimposed-spindle zero speed signal is et on when the motor rotation speed for (rotation speed commanded to the reference spindle) + (rotation speed commanded to the superimposed spindle) falls below the speed specified in the spindle parameter "#13027 SP027".

Spindle speed reach signal

The superimposed-spindle speed reach signal is output when the speed reaches (rotation speed commanded to the reference spindle) + (rotation speed commanded to the superimposed spindle).

Spindle forward running signal / spindle reverse running signal

For the superimposed-spindle forward running / reverse running signal, the forward running signal is set on if the motor rotation direction for (rotation speed commanded to the reference spindle) + (rotation speed commanded to the superimposed spindle) is set to CCW, and the reverse running signal is set on if it is set to CW.

Precautions

(1) The spindle synchronous position loop gain parameter "#13003 SP003" and SHG setting must be the same between the reference and superimposed spindles. These are determined in the MTB specifications.

(2) Be careful of the rotation speed clamp when issuing the command. The rotation speed difference commanded to the reference or superimposed spindle cannot be maintained while the rotation speed remains clamped.

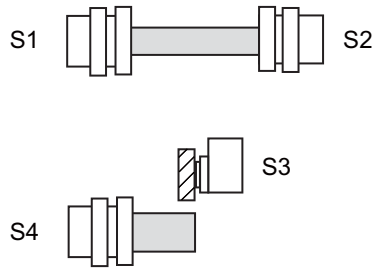
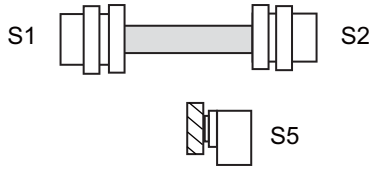
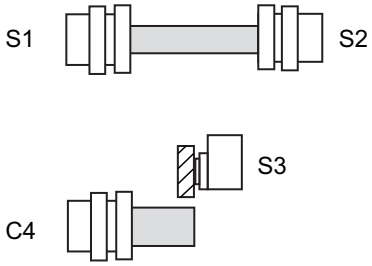
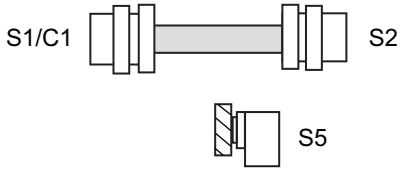
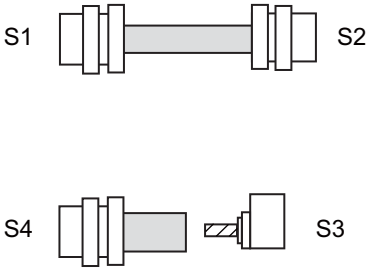
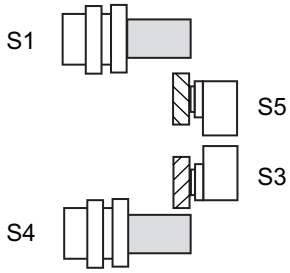
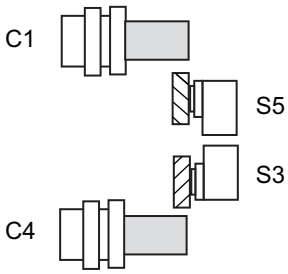
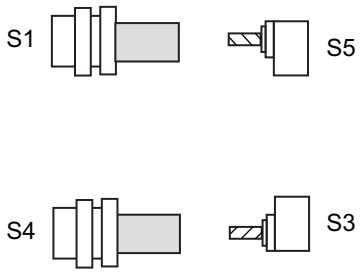
(3) The rotation speed commanded to the superimposed spindle side in the spindle superimposition control state is indicated by the rotation speed commanded only to the superimposed spindle. The feedback rotation speed is indicated by the actual rotation speed of the superimposed spindle.

(4) Before starting machining in the spindle superimposition control state, be sure to check that the spindle rotation speed synchronization completed signal (FSPRV) is output after the spindle superimposition has been commanded. (The signal check method depends on the MTB specifications.)

18.5 Multiple Spindle Synchronization Set Control

Function and purpose

Multiple spindle synchronization set control is a function designed to execute Spindle synchronization I, Tool spindle synchronization IA/IB (Spindle-Spindle, Polygon), Tool spindle synchronization II (hobbing) and Spindle superimposition control simultaneously for multiple sets of spindles.
This function enables machining using a combination of the following functions.

♦Spindle synchronization I command (S1-S2) ♦Tool spindle synchronization IA command (S3-S4) 	♦Spindle synchronization I command (S1-S2) ♦Tool spindle synchronization IA command (S1-S5) 
♦Spindle synchronization I command (S1-S2) ♦Tool spindle synchronization II command (S3-C4) 	♦Spindle/C axis synchronization control command (S1/C1-S2) ♦Tool spindle synchronization II command (S5-C1) 
♦Spindle synchronization I command (S1-S2) ♦Spindle superimposition command (S4-S3) 	♦Tool spindle synchronization IA command (S5-S1) ♦Tool spindle synchronization IA command (S3-S4) 
♦Tool spindle synchronization II command (S5-C1) ♦Tool spindle synchronization II command (S3-C4) 	♦Spindle superimposition command (S1-S5) ♦Spindle superimposition command (S4-S3) 

List of available combinations

G114.1: Spindle synchronization I command
G114.2: Tool spindle synchronization IA command
G51.2: Tool spindle synchronization IB command
G114.3: Tool spindle synchronization II command
G164: Spindle superimposition control
G84: Synchronous tap

[Ref] indicates "reference axis" or "reference spindle".
[Sync] indicates "synchronous axis" or "synchronous spindle".

Spindle synchronization command for the 2nd set			Spindle synchronization command for the 1st set		
	[Ref]	[Sync]	G114.1 S1 [Ref] - S2 [Sync]	G114.2 S1 [Ref] - S2 [Sync]	G51.2 S1 [Ref] - S2 [Sync]
G114.1	**S3**	**S4**	○	○	○
	S1	**S4**	○	○	○
	S3	**S2**	x (*2)	x (*2)	x (*2)
	S2	**S3**	x (*3)	x (*3)	x (*3)
	S3	**S1**	x (*3)	x (*3)	x (*3)
G114.2	**S3**	**S4**	○	○	○
	S1	**S4**	○	○	x (*1)
	S3	**S2**	x (*2)	x (*2)	x (*2)
	S2	**S3**	x (*3)	x (*3)	x (*3)
	S3	**S1**	x (*3)	x (*3)	x (*3)
G51.2	**S3**	**S4**	○	○	○
	S1	**S4**	○	○	○
	S3	**S2**	x (*2)	x (*2)	x (*2)
	S2	**S3**	x (*3)	x (*3)	x (*3)
	S3	**S1**	x (*3)	x (*3)	x (*3)
G114.3	**S3**	**C4**	○	○	○
	S1	**C4**	x (*1)	x (*1)	x (*1)
	S3	**C1**	○	○	○
G164	**S3**	**S4**	○	○	○
	S1	**S4**	x (*1)	x (*1)	x (*1)
	S3	**S2**	x (*2)	x (*2)	x (*2)
	S2	**S3**	x (*3)	x (*3)	x (*3)
	S3	**S1**	x (*3)	x (*3)	x (*3)
G84	**S1**		x (*4)	x (*4)	x (*4)
	S2		x (*4)	x (*4)	x (*4)
	S3		○	○	○

Spindle synchronization command for the 2nd set			Spindle synchronization command for the 1st set		
	[Ref]	[Sync]	G114.3 S1 [Ref] - C1 [Sync]	G164 S1 [Ref] - S2 [Sync]	G84 S1
G114.1	S3	S4	○	○	○
	S1	S4	x (*1)	x (*1)	x (*4)
	S3	S2	○	x (*2)	○
	S2	S3	○	x (*3)	○
	S3	S1	○	x (*3)	x (*4)
G114.2	S3	S4	○	○	○
	S1	S4	x (*1)	x (*1)	x (*4)
	S3	S2	○	x (*2)	○
	S2	S3	○	x (*3)	×
	S3	S1	○	x (*3)	x (*4)
G51.2	S3	S4	○	○	○
	S1	S4	x (*1)	x (*1)	x (*4)
	S3	S2	○	x (*2)	○
	S2	S3	○	x (*3)	×
	S3	S1	○	x (*3)	x (*4)
G114.3	S3	C4	○	○	○
	S1	C4	x (*1)	x (*1)	x (*1)
	S3	C1	x (*2)	○	○
G164	S3	S4	○	○	○
	S1	S4	x (*1)	x (*1)	x (*4)
	S3	S2	x (*2)	x (*2)	○
	S2	S3	x (*3)	x (*3)	○
	S3	S1	○	x (*3)	○
G84	S1		x (*4)	x (*4)	x (*4)
	S2		○	x (*4)	○
	S3		○	○	○

(*1) The reference spindle of the 1st set overlaps with that of the 2nd set.

(*2) The synchronous spindle of the 1st set overlaps with that of the 2nd set.

(*3) The reference spindle of the 1st set overlaps with the synchronous spindle of the 2nd set. Or the synchronous spindle of the 1st set overlaps with the reference spindle of the 2nd set.

(*4) The reference spindle or synchronous spindle of the 1st or 2nd set overlaps with the synchronous tap spindle.

Enabling conditions

(1) This function can be applied to a lathe system equipped with two or more spindles.

(2) The validity of this function depends on the MTB specifications (parameter "#1440 multi_sp_syn").
If this function is invalid, multiple spindle synchronization commands cannot be issued. (If two or more spindle synchronization commands are issued, the operation error (M01 1005) occurs, which causes automatic operation to pause.)
Note that guide bushing spindle synchronization is not included in the multiple spindle synchronization sets.

Command format

For details about the command format to start each function, refer to the relevant function's section.

Canceling spindle synchronization

(1) Command to cancel all synchronization modes
Cancels all the synchronized sets. The format varies depending on the MTB specifications (parameter "#1242 set14/bit6").

[When "#1242 set14 bit6" = 0]

G113 H0 ;	Cancels all the active tool spindle synchronization II (hobbing) commands.

G113 D0 ;	Cancels all the active Spindle synchronization I, Tool spindle synchronization IA (Spindle-Spindle, Polygon), and spindle superimposition commands.

Note

- If one set of spindle synchronization is active, G113 (without H_/D_) is able to cancel the spindle synchronization control. However, if two or more sets of Spindle synchronization are active, the command causes the operation error (M01 1135) to occur.

[When "#1242 set14 bit6" = 1]

G113 ;

Note

- If "G113 H0;" or "G113 D0;" is issued, the program error (P35) occurs.

(2) Spindle synchronization I, Tool spindle synchronization IA (Spindle-Spindle, Polygon), or Spindle superimposition control cancel command

G113 D__ ;

Address	Meaning	Command range	Remarks
D	Cancel target synchronized or superimposed spindle Specify the number or name of the spindle that serves as the synchronous spindle in the Spindle synchronization I or Tool spindle synchronization IA, or as the superimposed spindle in the Spindle superimposition control mode. (*1)	Spindle No.: 1 to number of spindles Spindle name: 1 to 9	♦If the command range is exceeded, the program error (P35) will occur. ♦If you specify a non-existent spindle, the program error (P35) occurs. ♦If you specify a spindle that is not under synchronization, the operation error (M01 005) occurs. ♦If there is no address D, the synchronization status of all spindle synch sets is cancelled. ♦If the address H is issued simultaneously with the address D, the program error (P33) will occur.

(*1) There are two types of spindle designation methods: Spindle number method and spindle name method. When any name (1 to 9) is set to the spindle name parameter "#3077 Sname" of all the spindles, the spindle name method takes effect. In other cases, the spindle number is used.

(*2) Spindle synchronization I, Tool spindle synchronization IA (Spindle-Spindle, Polygon), or spindle superimposition control can be canceled with the G113D_command regardless of whether the multiple spindle synchronization set control is valid or invalid.

(3) Tool spindle synchronization II (hobbing) cancel command (when the G code system of the program is 2, 3, 4, or 5):

```
G113 H_;
```

Address	Meaning	Command range	Remarks
H	Reference spindle to be canceled Specify the number or name of the spindle that serves as the reference spindle in the tool spindle synchronization II (hobbing) mode. (*1)	Spindle No.: 1 to number of spindles Spindle name: 1 to 9	♦ If the command range is exceeded, the program error (P35) will occur. ♦ If you specify a non-existent spindle, the program error (P35) occurs. ♦ If you specify a spindle that is not under synchronization, the operation error (M01 1005) occurs. ♦ If the address H is not issued, the synchronization status of all the spindle synchronization sets is canceled. ♦ If the address D is issued simultaneously with the address H, the program error (P33) will occur.

(*1) There are two types of spindle designation methods: Spindle number method and spindle name method. When any name (1 to 9) is set to the spindle name parameter "#3077 Sname" of all the spindles, the spindle name method takes effect. In other cases, the spindle number is used.

(*2) Tool spindle synchronization II (hobbing) can be canceled with the G113D_ command regardless of whether the multiple spindle synchronization set control is valid or invalid.

(4) Tool spindle synchronization II (hobbing) cancel command (when the G code system of the program is 6 or 7)
Cancel tool spindle synchronization II (hobbing) of the specified part system.

```
G80.4;
```

(5) Tool spindle synchronization IB (Spindle-Spindle, Polygon) cancel command
Cancel tool spindle synchronization IB (Spindle-Spindle, Polygon) of the specified part system.

```
G50.2;
```

Operation example

The function combination table in the operation example shows each function as follows.

G114.1: Spindle synchronization I command
G114.2: Tool spindle synchronization IA command
G51.2: Tool spindle synchronization IB command
G114.3: Tool spindle synchronization II command
G164: Spindle superimposition control
G84: Synchronous tap

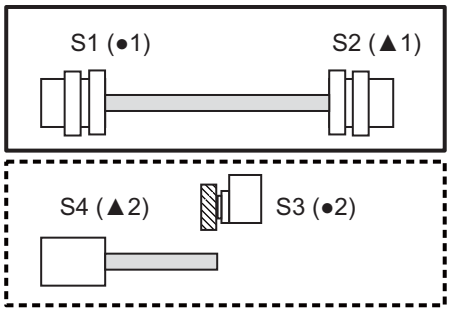
(1) If there is no overlap among the spindles of each spindle synchronization set or the workpiece spindle of tool spindle synchronization II (hobbing), the multiple synchronization sets can be controlled at the same time.

Example: Combinations when the commanded reference spindle or synchronous spindle is not included in other control sets

Spindle synchronization command for the 2nd set	Spindle synchronization command for the 1st set				
	G114.1 (S1-S2)	G114.2 (S1-S2)	G51.2 (S1-S2)	G114.3 (S1-C1)	G164 (S1-S2)
G114.1 (S3-S4)	○	○	○	○	○
G114.2 (S3-S4)	○	○	○	○	○
G51.2 (S3-S4)	○	○	○	○	○
G114.3 (S3-C4)	○	○	○	○	○
G164 (S3-S4)	○	○	○	○	○

S1, S2, S3, S4: S command name C1, C3: Name of rotary axis

<Program example (1)-1: When Spindle synchronization I [S1-S2] and Tool spindle synchronization IA [S3-S4] are commanded simultaneously>



1st set: Spindle synchronization I command [S1(●1) - S2(▲1)]

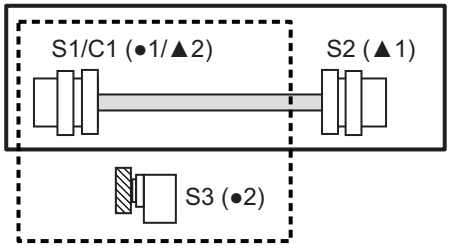
2nd set: Tool spindle synchronization IA command [S3(●2) - S4(▲2)]

●: Reference axis ▲: Synchronized axis or superimposed axis

The number following each symbol indicates which set is specified. For example, "●1" indicates the reference axis of the 1st set.

1st part system		2nd part system	
S1=1000 ;	Commands 1000 r/min to the 1st spindle.	:	
M13;	Issues forward-run command to S1	:	
S2=1500 ;	Issues 1500r/min command to S2		
M24 ;	Issues reverse-run command to S2		
G04X2.;			
G114.1H1D-2 ; :	1st set: Spindle synchronization I command Both the 1st and 2nd spindles start synchronization at 1000 r/min.		
:		S3=1500;	Issues 1500r/min command to S3
		M33;	Issues forward-run command to S3
		G04X2.;	
		G114.2H3D-4E4L5; :	2nd set: Tool spindle synch IA ON Start a synchronization at the rotation speed shown below. 3rd spindle: 1500 r/min 4th spindle: 1200 r/min
G113D2; :	1st set: Spindle synch control OFF	:	
		G113D4; :	Cancels tool spindle synchronization IA in the 2nd set.

<Program example (1)-2: When spindle synchronization C axis control [S1/C1-S2] and tool spindle synchronization II [S3-C1] are commanded simultaneously>



1st set: Spindle synchronization C axis command [S1/C1(●1) - S2(▲1)]

2nd set: Tool spindle synchronization II command [S3(●2) - S1/C1(▲2)]

●: Reference axis ▲: Synchronized axis or superimposed axis

The number following each symbol indicates which set is specified. For example, "●1" indicates the reference axis of the 1st set.

1st part system	
S1=1000 ;	Commands 1000 r/min to the 1st spindle.
M13 ;	Issues forward-run command to S1
S2=1500 ;	Issues 1500r/min command to S2
M24 ;	Issues reverse-run command to S2
G04X2.;	
G114.1H1D-2 ;	1st set: Spindle synchronization I command Both the 1st and 2nd spindles start synchronization at 1000 r/min.
:	
Mxxx ;	Changes S1 to C axis mode.
:	
S3=1500 ;	Issues 1500r/min command to S3
M33 ;	Issues forward-run command to S3
G4X2.;	
G114.3H3D9;	2nd set: Tool spindle synchronization II command The 1st, 2nd, and 3rd spindles start synchronization at 1500 r/min.
:	
:	
G113D3;	Cancels the tool spindle synchronization II command in the 2nd set.
:	
Mxx;	Changes S1 to spindle mode.
:	
G113;	Cancels the Spindle synchronization I command in the 1st set.
:	

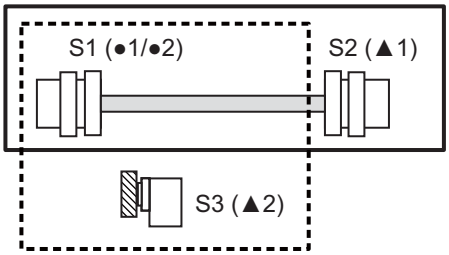
(2) If the reference spindle of one spindle synchronization set overlaps with that of another synchronization set, the operation error (M01 1005) occurs. Note, however, that the reference spindle of Spindle synchronization or Tool spindle synchronization IA (Spindle-Spindle, Polygon) is allowed to be used as the reference spindle of Spindle synchronization or Tool spindle synchronization IA (Spindle-Spindle, Polygon) of another synchronization set.

Example: Combinations when the reference spindle of the 2nd set is controlled as the reference spindle of another set (When S1 is duplicated as the reference spindle)

Spindle synchronization command for the 2nd set	Spindle synchronization command for the 1st set				
	G114.1 (S1-S2)	G114.2 (S1-S2)	G51.2 (S1-S2)	G114.3 (S1-C1)	G164 (S1-S2)
G114.1 (S1-S3)	○	○	×	×	×
G114.2 (S1-S3)	○	○	×	×	×
G51.2 (S1-S3)	×	×	×	×	×
G114.3 (S1-C4)	×	×	×	×	×
G164 (S1-S3)	×	×	×	×	×

S1, S2, S3, S4: S command name C1, C3: Name of rotary axis

<Program example (2)-1: When Spindle synchronization I [S1-S2] and tool spindle synchronization IA [S1-S3] are commanded simultaneously>



1st set: Spindle synchronization I command [S1(●1) - S2(▲1)]

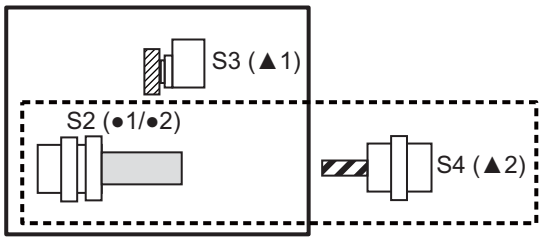
2nd set: Tool spindle synchronization IA command [S1(●2) - S3(▲2)]

●: Reference axis ▲: Synchronized axis or superimposed axis

The number following each symbol indicates which set is specified. For example, "●1" indicates the reference axis of the 1st set.

1st part system		2nd part system	
S1=1000 ;	Commands 1000 r/min to the 1st spindle.	:	
M13;	Issues forward-run command to S1		
S2=1500 ;	Issues 1500r/min command to S2		
M24 ;	Issues reverse-run command to S2		
G04X2.;			
G114.1H1D-2 ; :	1st set: Spindle synchronization I command Both the 1st and 2nd spindles start synchronization at 1000 r/min.		
:		S3=1500	Issues 1500r/min command to S3
		M33 ;	Issues forward-run command to S3
		G04X2.;	
		G114.2H3D-4E4L5; :	2nd set: Tool spindle synch IA ON Start a synchronization at the rotation speed shown below. 1st spindle: 1000 r/min 3rd spindle: 1250 r/min
G113D2;	Cancels the Spindle synchronization I command in the 1st set.	:	
:			
		G113D3; :	2nd set: Tool spindle synch IA OFF

<Program example (2)-2: When tool spindle synchronization IA [S3-S2] and spindle superimposition control command [S2-S4] are commanded simultaneously>



1st set: Tool spindle synchronization IA command [S2(●1) - S3(▲1)]

2nd set: Spindle superimposition control command [S2(●2) - S4(▲2)]

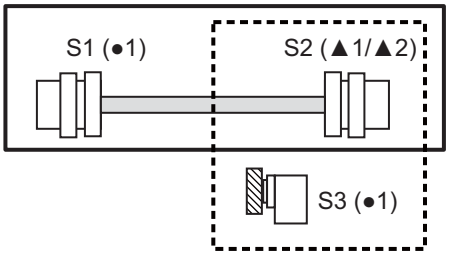
●: Reference axis ▲: Synchronized axis or superimposed axis

The number following each symbol indicates which set is specified. For example, "●1" indicates the reference axis of the 1st set.

1st part system		2nd part system	
S2=1000 ;	Commands 1000 r/min to the 2nd spindle.	:	
M23;	Commands forward running to the 2nd spindle.		
G04X2.;			
G114.2H2D-3 ;	1st set: Tool spindle synchronization IA command		
:	Both the 2nd and 3rd spindles start synchronization at 1000 r/min.		
		S4=1000 ;	Commands 1000 r/min to the 4th spindle.
		M44 ;	Commands reverse running to the 4th spindle.
		G04X2.;	
		G164H2D-4; :	2nd set: Spindle superimposition control command The operation error (M01 1005) will occur (automatic operation pause).

(3) If the synchronous spindles (or workpiece spindle for Tool spindle synchronization II (hobbing)) of two spindle synchronization sets overlap with each other, the operation error (M01 1005) occurs.

<Program example (3)-1: When Spindle synchronization I [S1-S2] and tool spindle synchronization IA [S3-S2] are commanded simultaneously>



1st set: Spindle synchronization I command [S1(●1) - S2(▲1)]

2nd set: Tool spindle synchronization IA command [S3(●2) - S2(▲2)]

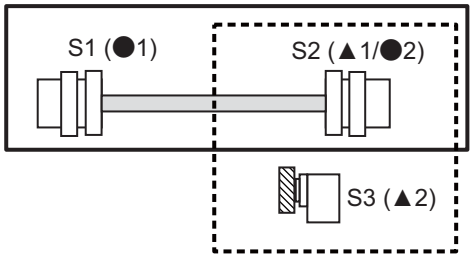
●: Reference axis ▲: Synchronized axis or superimposed axis

The number following each symbol indicates which set is specified. For example, "●1" indicates the reference axis of the 1st set.

1st part system		2nd part system	
S1=1000 ;	Commands 1000 r/min to the 1st spindle.	:	
M13;	Issues forward-run command to S1		
S2=1500 ;	Issues 1500r/min command to S2		
M24 ;	Issues reverse-run command to S2		
G04X2.;			
G114.1H1D-2 ; :	1st set: Spindle synchronization I command Both the 1st and 2nd spindles start synchronization at 1000 r/min.	:	
:		:	
		S3=1500	Issues 1500r/min command to S3
		M33 ;	Issues forward-run command to S3
		G04X2.;	
		G114.2H3D-4E4L5; :	2nd set: Tool spindle synchronization IA command The operation error (M01 1005) will occur (automatic operation pause).

(4) If the reference spindle of one spindle synchronization set overlaps with the synchronous spindle of another synchronization set, the operation error (M01 1005) occurs.

<Program example (4)-1: When Spindle synchronization I [S1-S2] and Tool spindle synchronization IA [S2-S3] are commanded simultaneously>



1st set: Spindle synchronization I command [S1(●1) - S2(▲1)]

2nd set: Tool spindle synchronization IA command [S2(●2) - S3(▲2)]

●: Reference axis ▲: Synchronized axis or superimposed axis

The number following each symbol indicates which set is specified. For example, "●1" indicates the reference axis of the 1st set.

1st part system		2nd part system	
S1=1000 ;	Commands 1000 r/min to the 1st spindle.	:	
M13;	Issues forward-run command to S1		
S2=1500 ;	Issues 1500r/min command to S2		
M24	Issues reverse-run command to S2		
G04X2.;			
G114.1H1D-2 ; :	1st set: Spindle synchronization I command Both the 1st and 2nd spindles start synchronization at 1000 r/min.		
		:	
		S3=1500	Issues 1500r/min command to S3
		M33 ;	Issues forward-run command to S3
		G04X2.;	
		G114.2H3D-4E4L5; :	2nd set: Tool spindle synchronization IA command The operation error (M01 1005) will occur (automatic operation pause).

Relationship with other functions

Spindle synchronization II

Spindle synchronization II does not support the multiple spindle synchronization set control. Thus, output the PLC output signal of the 1st spindle regardless of whether the multiple spindle synchronization set control is valid or invalid.

Tool spindle synchronization IC (Spindle-NC axis, Polygon)

(1) The reference spindle of tool spindle synchronization IC cannot be used as the reference or synchronized (or superimposed) spindle of Spindle synchronization I, Tool spindle synchronization IA, IB (Spindle-Spindle, Polygon), Tool spindle synchronization II (hobbing) or Spindle superimposition control.
The operation error (M01 1005) will occur, which causes automatic operation to pause.

(2) The rotary tool axis of tool spindle synchronization IC cannot be used as the workpiece axis of tool spindle synchronization II (hobbing).
The operation error (M01 1005) will occur, which causes automatic operation to pause.

Synchronous tapping cycle

In the following cases, the operation error will occur, which causes automatic operation to pause.

(1) A synchronous tap spindle cannot be commanded as the reference spindle of Spindle synchronization I, Tool spindle synchronization IA, IB (Spindle-Spindle, Polygon), Tool spindle synchronization II (hobbing) or Spindle superimposition control. (Operation error (M01 1007))

(2) A synchronous tap spindle cannot be commanded as the synchronous spindle of Spindle synchronization I, Tool spindle synchronization IA, IB (Spindle-Spindle, Polygon) or Spindle superimposition control. (Operation error (M01 1007))

(3) You cannot command a synchronous tapping that uses the reference spindle of Spindle synchronization I, Tool spindle synchronization IA, IB (Spindle-Spindle, Polygon) or Tool spindle synchronization II (hobbing). (Operation error (M01 1139))

(4) You cannot command a synchronous tapping that uses the reference spindle of Spindle superimposition control. (Operation error (M01 1131))

(5) You cannot command a synchronous tap that uses the synchronous spindle of spindle synchronization I or Tool spindle synchronization IA, IB (Spindle-Spindle, Polygon). (Operation error (M01 1139))

Precautions

(1) If any of the following commands are issued to a spindle for which cancel operation is being processed by the G113 command or the spindle synchronization/superimposition cancel signal (SPSYC), the operation error (M01 1005) occurs.
- Spindle synchronization
- Tool spindle synchronization IA (Spindle-Spindle, Polygon)
- Tool spindle synchronization II (hobbing)
- Spindle superimposition control

Note that the commanded spindle synchronization operation starts after the cancel process is completed.

(2) If the cancel command (G113 D_ , G113 H_) are issued to a spindle that is not under synchronous control, the operation error (M01 1005) will occur.

(3) If the spindle synchronization/superimposition cancel signal (SPSYC) turns ON for a spindle that is not under synchronization control, the control ignores the cancel signal.

(4) Do not issue two or more tool spindle synchronization II (hobbing) commands to one part system. Doing so causes the program error (P33) when the tool spindle synchronization II command in the 2nd set is issued, which causes automatic operation to pause.

(5) Do not issue two or more tool spindle synchronization IB (Spindle-Spindle, Polygon) commands to one part system. Doing so causes the program error (P33) when the tool spindle synchronization IB command in the 2nd set is issued, which causes automatic operation to pause.

(6) Do not put an axis address that causes any travel in the same block as the spindle synchronization cancel command (G113/G80.4/G50.2). Doing so causes the program error (P33) when the Spindle synchronization cancel command is issued, which causes automatic operation to pause.

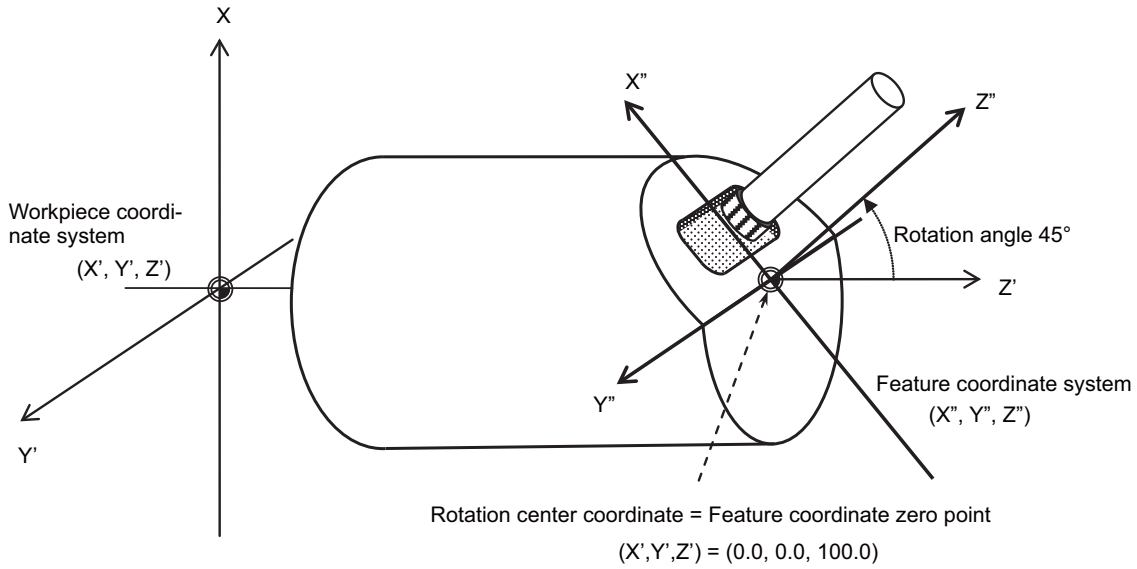
19

Advanced Machining Control

19.1 Simple Inclined Surface Machining

Function and purpose

In a lathe with three orthogonal axes and one rotary axis on the tool side, this function enables simultaneous 4-axis control milling on an inclined surface along workpiece end face.
To facilitate machining on an inclined surface, this function uses two commands: Simple inclined surface control (G176) and Simple tool center point control (G174) as follows. When Simple tool center point control (G174) is given after Simple inclined surface control (G176), the direction of tool length is compensated in accordance with the tool angle.
For a machine configured with a turret, this function allows you to use any tool on the turret to carry out simple inclined surface machining.



If simple inclined surface control (G176) or simple tool center point control (G174) is commanded while this function is not defined in the specifications, it causes a program error (P959).

Simple inclined surface control (G176)

G176 enables a new coordinate system (called "feature coordinate system") to be defined. Feature coordinate system is obtained by rotating and parallel translating a plane using the X, Y and Z axes configuring the workpiece coordinate system as a rotary axis.
Consequently, there is no need to calculate the coordinate position in accordance with the inclined surface angle. This simplifies creation of a machining program. Refer to "19.1.1 Simple Inclined Surface Control ; G176".

Simple tool center point control (G174)

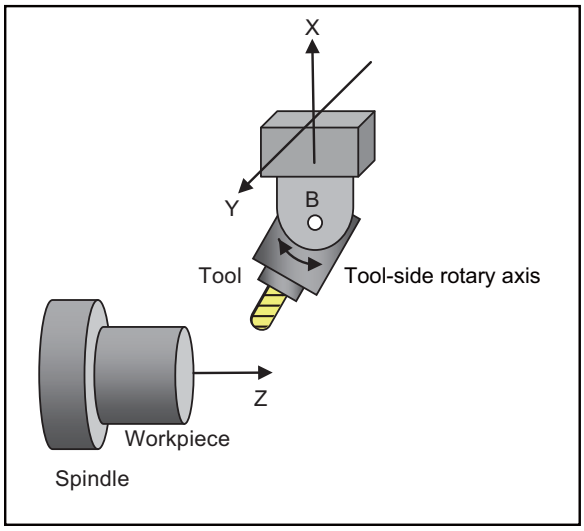
G174 allows a tool length offset in the tool axis direction even when the tool axis direction is not parallel to the orthogonal coordinate system after rotation of the rotary axis.
Consequently, the nose of the tool mounted on the rotary axis can be kept in the programmed position at all times. This simplifies creation of a complicated-shape machining program. Refer to "19.1.2 Simple Tool Center Point Control ; G174".

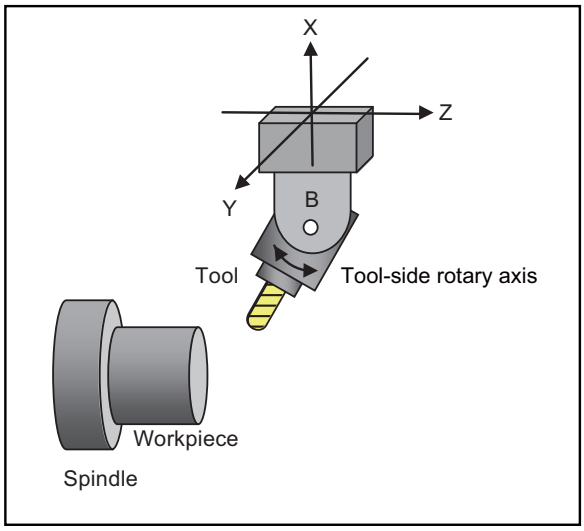
Detailed description

Supporting axis configuration

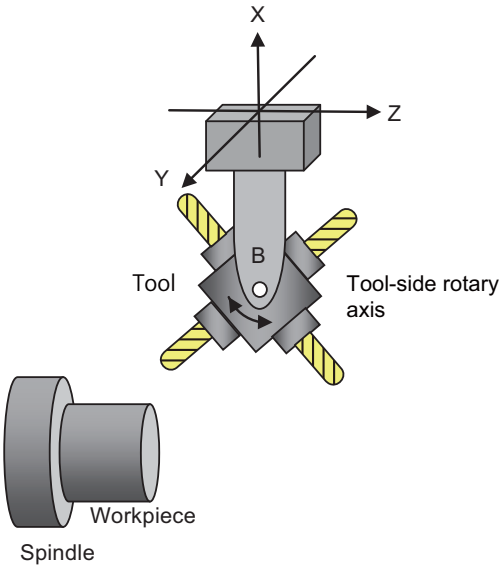
This function is available for the machines equipped with the following axis configurations.

[Work-moving-type tool tilt]	[Work-stationary-type tool tilt]
Tool head side: Rotary axis X-Y: Orthogonal axis Z : Spindle	Tool head side: Rotary axis XZY: Orthogonal axis





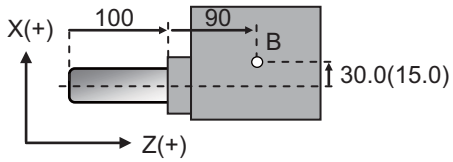
[Turret-type tool tilt]
Tool head side: Rotary axis X-Y: Orthogonal axis Any tool on the turret can be used.



Tool length offset in simple inclined surface machining

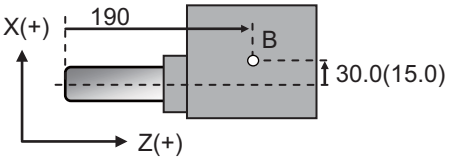
(1) Tool length offset setting when simple inclined surface machining (G174, G176) is active
While simple inclined surface machining is ON, tool offset can be applied in the feature coordinate system axis direction.
Tool offset is set while the tool for simple inclined surface machining is facing Z(-) of the machine coordinate system.
For tool offset, specify the vector from the tool tip to the B axis rotation center.
The offset setting methods are different between when the rotation center offset (rotary axis configuration parameters "#7934 COFST2H" to "#7936 COFST1T") is used together or not.

(a) When using rotation center offset together
Specify the vector from the tool tip to the tool mount position in tool offset, and the vector from the tool mount position to the rotation center position in rotation center offset ("#7934 COFST2H" to "#7936 COFST1T").



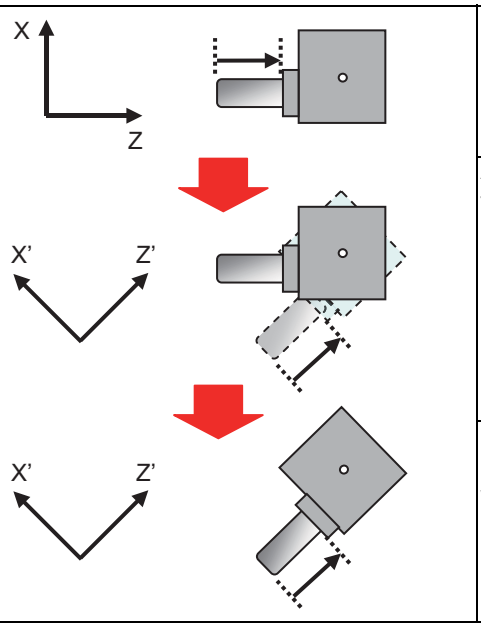
Tool offset	X: 0.0 Y: 0.0 **Z: 100.0**
Rotation center offset (radius value setting)	**X: 15.0** Y: 0.0 **Z: 90.0**

(b) When not using rotation center offset together
Specify the length from the tool tip to the rotation center position in tool offset.



Tool offset	**X: 30.0 (15.0)** Y: 0.0 **Z: 190.0**
Rotation center offset (radius value setting)	X: 0.0 Y: 0.0 Z: 0.0

(2) Tool length direction when simple inclined surface machining control (G176) is used as a single command (when not combined with simple tool center point control (G174))
When Simple inclined surface control (G176) is commanded, the tool length offset is applied in the feature coordinate system direction (Z' direction in the following illustration).



Before G176 is issued, tool length offset is applied in machine coordinate direction. (Tool tip position = Programmed command position)
After G176 has been issued, tool length offset is applied in feature coordinate direction. (Tool tip position and programmed command position are almost the same.)
By turning B axis, the tool is directed to the feature coordinate system direction, and so the tool's direction coincides with the tool length offset direction. (Tool tip position = Programmed command position)

Combination with arbitrary axis exchange function

When using the simple inclined surface machining in combination with an arbitrary axis exchange (G140) command, you need to set the rotary axis configuration parameters using the 2nd axis name. Set the parameter "#1450 5axis_Spec/bit0" to "1" (setting by the 2nd axis name), and assign the axis configuration for executing simple inclined surface machining to the rotary axis configuration parameter (#7900 or later) using the 2nd axis name (example: A1, B2).

If the simple inclined surface control (G176) or simple tool center point control (G174) is commanded after the arbitrary axis exchange has been completed while the parameter "#1450 5axis_Spec/bit0" is not designated, a program error (P952) or (P941) will occur.

You can set the configurations up to the number of valid part systems (up to four part systems) in the rotary axis configuration parameter. With multiple configurations set, you can perform simple inclined surface machining in different axis configurations.

(1) Application of rotary axis configuration parameters

Simple inclined surface machining can be performed using the axis configuration in the part system with axis exchange completed by applying the rotary axis configuration parameter in the configuration in which all axes included in the part system are set.

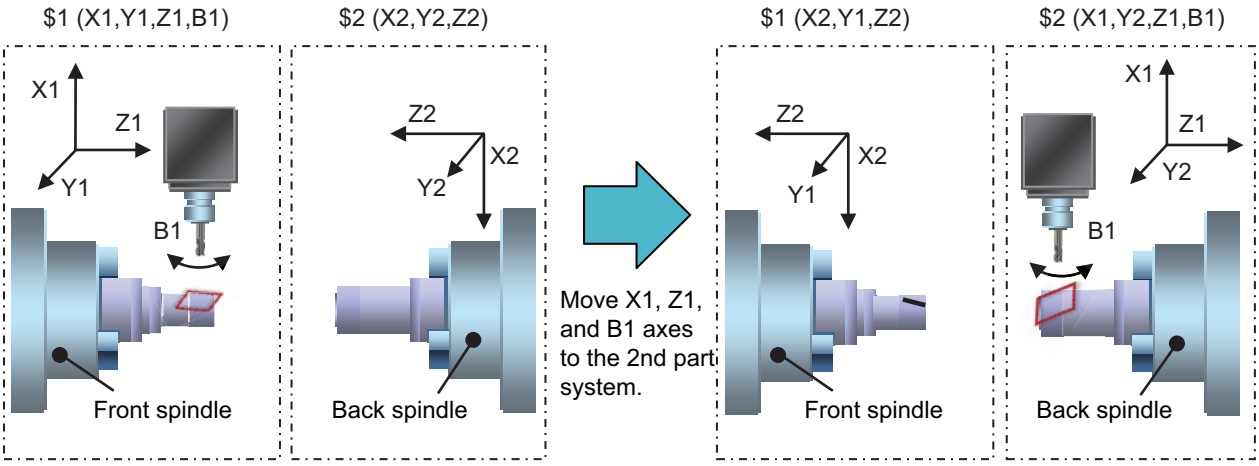
(2) Methods to apply the rotary axis configuration parameters

The following two methods are provided to apply rotary axis configuration parameters. Which method is to be applied depends on the MTB specifications (parameter "#1450 5axis_Spec/bit2" (application of rotary axis configuration parameters)).

Automatic selection method	When the power is turned ON, NC is reset, or the arbitrary axis exchange control (G140/G141/G142) is commanded, the system searches for the configuration in which all the axes of the command's part system are set in the order of the designated rotary axis configuration parameters: self part system, $1, $2, and after. The system applies the rotary axis configuration parameters of the configuration that first matches the designated conditions. This method is available when the rotary axis configuration parameters that match the axis configuration for executing simple inclined surface machining is determined uniquely.
PLC designation method	Designate the rotary axis configuration parameters to be applied to the axis configuration for executing simple inclined surface machining using the R register, and enable it with the M command or T command. This method is available when designating the parameters to be applied.

(3) Use example

(a) Case to first perform the inclined surface machining in the part system 1 ($1), and change the part system of X1, Z1, and B1 axes to the part system 2 ($2), and then perform the inclined surface machining in $2 using the changed axes.



[$1]		[$2]	
G28 X0. Y0. Z0. B0.; T1010 ;		G28 X0 Y0. Z0. B0.;	
G176 X0. Z0. D-60.; G174 G00 B-60.; G01 X0. Y0. Z10. F1000; G175; G69.1;	Execute simple inclined surface machining command, referencing the rotary axis configuration parameters of configuration 1.		
!2 L10;	<-Timing synchronization->	!1 L10;	
G140 X=X2 Y=Y1 Z=Z2 B=B2;	Use X1, Z1, and B1 axes in $2.	G140 X=X1 Y=Y2 Z=Z1 B=B1 ;	Search the configuration of the rotary axis configuration parameter. Apply the configuration 3 parameter which matches first.
		T2020; G176 Y0. Z0. D30.; G174; G01 Y10. Z5. F1000 G175; G69.1;	Execute simple inclined surface machining, referencing configuration 3 of the rotary axis configuration parameters.
!2 L20	<-Timing synchronization->	!1 L20	
G142;	Return from axis exchange	G142;	
M2;		M2;	

[Parameter configuration example (For 4-axis tool tilt)]

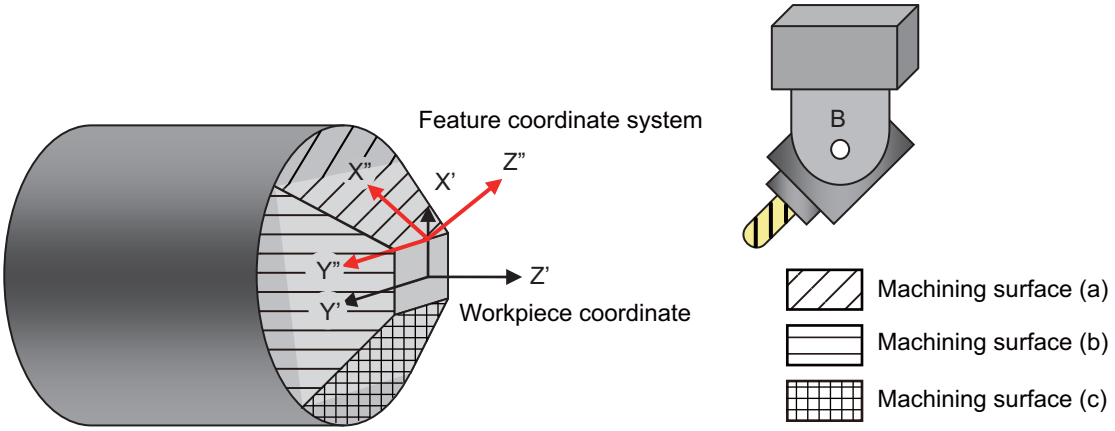
Rotary axis configuration parameter		Configuration 1	Configuration 2	Configuration 3	Configuration 4
#7900	RCDAX_I	X1	X2	X1	X2
#7901	RCDAX_J	Y1	Y2	Y2	Y1
#7902	RCDAX_K	Z1	Z2	Z1	Z2
#7930	SLCT_T2	2	2	2	2
#7932	ROTAXT2	B1	B2	B1	B2

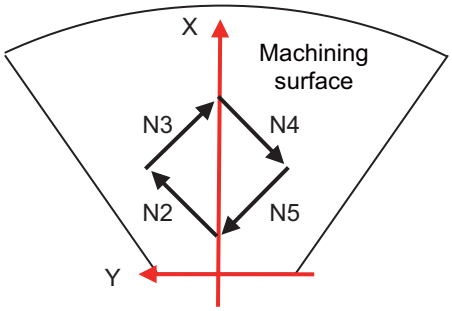
<Note>

- If the simple inclined surface machining command after the arbitrary axis exchange is to be drawn with the 2D graphic check, the machining path will be drawn but the arbitrary axis exchange command will be ignored.
- To manually take out the tool used by the simple inclined surface machining control, use 3-dimensional manual feed.

Operation example

The following shows the machining program intended to cut grooves in the shape as illustrated above on an inclined surface of cylinder as illustrated in the figure below. The main program uses an inclined surface machining command to define a feature coordinate system and index the rotary axis, while the sub program (O100) executes grooving.





Subprogram O100 (Grooving)

```
N1 G90 G00 X20. Y0. ;
N2 G01 X40. Y10. F1000 ;
N3 G01 X60. Y0. ;
N4 G01 X40. Y-10. ;
N5 G01 X20. Y0. ;
N6 G00 X0. Y0. ;
M99 ;
```

:	Positioned in the tool exchange position
T0101;	Tool exchange (Tool No. and offset No. command)
G176, G174;	During modal, the tool offset in Z axis direction is used as the tool length offset.
G90 G00 X0 Y0 Z0 C0.;	Index Surface (a)
G176 X10. Z0. D45.;	Simple inclined surface machining control command Define the feature coordinate system by using (X',Y',Z')=(10.0,0.0,0.0) of the workpiece coordinate system as the feature coordinate origin, and turning the coordinate system 45° around X and Z axes.
G00 X0. Y0. Z0. B45. ;	Position the tool tip to be on (X',Y',Z')=(0.0,0.0,0.0) of the feature coordinate system with the B axis head inclined by 45°.
M98 P100 ;	Execute milling on Surface (a) (Subprogram O100) Control the XYZ machine position so that the tool tip point passes through the command path on the feature coordinate system.
G00 C90.;	Index Surface (b)
M98 P100 ;	Execute milling on Surface (b) (Subprogram O100)
:	Ditto on Surfaces (c) and (d).
G69.1;	Simple inclined surface control cancel The feature coordinate system is canceled. No axis movement is caused by the offset cancel. The axes retain their current positions.
G00 X100. Y0. Z150. B0.;	Since the feature coordinate system has been canceled, the programmed position is relative to the workpiece coordinate system. Tool offset is in Z axis direction, regardless of the B axis head angle. (Normal tool compensation)
:	:

Relationship with other functions

Commands given during simple inclined surface machining mode

The commands marked with "○" in the table are able to be commanded during the simple inclined surface control (G176) or simple tool center point control (G174) modal state. If any other command is issued during the simple inclined surface control (G176) modal state, the program error (P951) occurs. If any other command is issued during the simple tool center point control (G174) modal state, the program error (P942) occurs. These alarms are cancelled by NC reset.

Group	G code lists				Function	G176	G174
	2	3	4	5			
0	G04				Dwell	○	○
	G05 P0, P1, P2, P10000				High-speed machining mode, High-speed high-accuracy control II	○	○
	G05.1 Q0, Q1				High-speed high-accuracy control I	○	P34
	G08 P1				High-accuracy control	○	○
	G09				Exact stop	○	○
	G10, G11				Parameter input by program, Compensation data input by program	○	P942
	G27				Reference position check	P951	P942
	G28				Automatic reference position return	○	P942
	G29				Start position return	○	P942
	G30				2nd to 4th reference position return	○	P942
	G30.1				Tool change position return 1	○	P942
	G30.2-G30.5				Tool change position return 2 to 5	○	P942
	G31				Skip/Speed change skip (*2)	P951	P942
	G31.1-G31.3				Multi-step skip (*2)	P951	P942
	G37		G36/G37		Automatic tool length measurement	P951	P942
	G50	G92	G50	G92	Spindle clamp speed setting Coordinate system setting	P951	P942
	G52				Local coordinate system setting	P951	P942
	G53				Machine coordinate system selection	○	P942
	G65				Macro simple call	○	○
	G110				Mixed control (cross axis control) I	P951	P942
	G111				Axis name switch	P951	P942
	G113, G114.1-G114.3, G164				Spindle synchronization, Spindle superimposition command, Cancel	P951	P942
	G115, G116				Start point designation synchronization (*3)	P951	P942
	G122, G144, G145				Sub part system control I/II, Sub part system completion standby	P951	P942
	G126				Control Axis Superimposition	P951	P942
	G156				Arbitrary Axis Superimposition	P951	P942
1	G00, G01				Positioning, Linear Interpolation	○	○
	G02, G03				Circular interpolation, Helical interpolation	○	(*1)
	G32	G33	G32	G33	Thread cutting	P951	P942
	G34, G35, G36				Variable lead thread cutting, circular screw	P951	P942
2	G17, G18, 19				Plane selection	○	○
3	G190	G90	G190	G90	Absolute value command	○	○
	G191	G91	G191	G91	Incremental value command		
4	G22				Barrier check	P951	P942
	G23				Barrier check cancel	P951	P942

Group	G code lists				Function	G176	G174
	2	3	4	5			
5	G98	G94	G98	G94	Feed per minute	○	○
	G99	G95	G99	G95	Feed per revolution	○	P942
6	G20, G21				Inch command, Metric command	P951	P942
7	G40, G41, G42				Tool nose radius compensation, Tool radius compensation (tool nose point 0 is used)	○	P942
8	G174				Simple tool center point control	○	○
	G175				Simple tool center point control cancel	○	○
9	G70, G71, G72, G73, G74, G75, G76, G76.1, G76.2				Compound type fixed cycle for turning machining	P951	P942
	G79	G83.2	G79	G83.2	Deep hole drilling cycle 2	○	○
	G81, G82, G83, G83.1, G85, G87, G89				Fixed cycle for drilling/Fixed cycle for drilling (MITSUBISHI CNC special format) (*) Excluding the tapping cycle.	○	○
	G84, G88, G84.1, G84.2, G88.1				Tapping cycle (X/Z)/ Reverse tapping cycle (X/Z)/ Synchronous tapping cycle	○	P942
	G90 G92 G94	G77 G78 G79	G90 G92 G94	G77 G78 G79	Longitudinal cutting fixed cycle Thread cutting fixed cycle Face cutting fixed cycle	P951	P942
10	-	G98 G99	-	G98 G99	Hole drilling cycle I point return, R point return	○	○
12	G54-G59, G54.1				Workpiece coordinate system selection 1 to 6, Extended workpiece coordinate system selection	P951	P942
13	G61				Exact stop mode	○	○
	G61.1				High-accuracy mode	○	○
	G62				Automatic corner override	○	P942
	G64				Cutting mode	○	○
14	G66, G66.1, G67				Macro modal call	○	○
15	G68, G69				Mirror image for facing tool posts (*4)	○	○
16	G68.1				Coordinate rotation by program	P951	P942
	G69.1				Inclined surface machining cancel	○	○
	G176				Simple inclined surface control	P951	P942
17	G96, G97				Constant surface speed control	○	○
18	G14, G15				Balance cut	○	○
19	G12.1, G13.1				Milling interpolation	P951	P942
20	G43.1, G44.1, G47.1				1st spindle control mode, 2nd spindle control mode, multiple-spindle simultaneous control mode	○	○
	M98, M99				Subprogram call	○	○
	F				Feedrate command	○	○
	M, S, B				M, S, B command	○	○
	T				Tool No., compensation amount command	○	P942
	Macro instruction				Local variable, Common variable, Arithmetic commands (such as four basic arithmetic rule, trigonometric functions, square root), Control commands (IF-GOTO- and WHILE-DO-)	○	○

(*1) Refer to "Circular interpolation, helical interpolation (G02, G03)" of the "Relationship with other functions".

(*2) Only the three orthogonal axes designated by the rotary axis configuration parameter can be commanded. If used for a rotary axis, the program error (P951) occurs.

(*3) If you use start point designation timing synchronization during G176 or G174 modal status, specify the timing synchronization position with respect to the feature coordinate system.

(*4) Do not issue this G code command under simple inclined surface control or simple tool center point control.

Modes where simple inclined surface machining is able to be commanded

During the modal status marked with "○" below, simple inclined surface control (G176) and simple tool center point control (G174) commands can be issued.
In any mode other than them, simple inclined surface control command (G176) causes the program error (P952), or simple tool center point control command (G174) causes the program error (P941). These alarms are cancelled by NC reset.

Group	G code lists				Function	G176	G174
	2	3	4	5			
0	G05 P0, P1, P2, P10000				High-speed machining mode, High-speed high-accuracy control II	○	○
	G05.1 Q0, Q1				High-speed high-accuracy control I	○	○
	G52				Local coordinate system setting	P952	P941
	G110				Mixed control (cross axis control) I	P952	P941
	G111				Axis name switch	P952	P941
	G113, G114.1-G114.3, G164				Spindle synchronization, Spindle superimposition command, Cancel (*1)	○	○
	G115, G116				Start point designation synchronization (*1)	○	○
	G122, G144, G145				Sub part system control I/II, Sub part system completion standby	○	○
	G126				Control Axis Superimposition	○	○
	G156				Arbitrary Axis Superimposition	○	○
1	G00, G01				Positioning, Linear Interpolation	○	○
	G02, G03				Circular interpolation, Helical interpolation	P952	P941
	G32	G33	G32	G33	Thread cutting	P952	P941
	G34				Variable lead thread cutting	P952	P941
	G35, G36				Circular screw		
2	G17, G18, 19				Plane selection	○	○
3	G190	G90	G190	G90	Absolute value command	○	○
	G191	G91	G191	G91	Incremental value command		
4	G22				Barrier check	P952	P941
	G23				Barrier check cancel	○	○
5	G98	G94	G98	G94	Feed per minute	○	○
	G99	G95	G99	G95	Feed per revolution	○	P941
6	G20, G21				Inch command, Metric command	○	○
7	G41, G42				Tool nose radius compensation, Tool radius compensation (tool nose point 0 is used)	○	○
	G40				Tool nose radius compensation, tool radius compensation cancel	○	○
8	G174				Simple tool center point control	P952	○
	G175				Simple tool center point control cancel	○	○
9	G70, G71, G72, G73, G74, G75, G76, G76.1, G76.2				Compound type fixed cycle for turning machining	P952	P941
	G79	G83.2	G79	G83.2	Deep hole drilling cycle 2	○	○
	G80				Fixed cycle cancel	○	○
	G81, G82, G83, G83.1, G85, G87, G88, G89, G84.1, G84.2, G88.1				Fixed cycle for drilling/Fixed cycle for drilling (MITSUBISHI CNC special format)	P952	P941
	G90	G77	G90	G77	Longitudinal cutting fixed cycle	P952	P941
	G92	G78	G92	G78	Thread cutting fixed cycle		
	G94	G79	G94	G79	Face cutting fixed cycle		
10	-	G98 G99	-	G98 G99	Hole drilling cycle I point return, R point return	○	○

Group	G code lists				Function	G176	G174
	2	3	4	5			
12	G54-G59, G54.1				Workpiece coordinate system selection 1 to 6, Extended workpiece coordinate system selection	P952	P941
13	G61				Exact stop mode	○	○
	G61.1				High-accuracy mode	○	○
	G62				Automatic corner override	P952	P941
	G64				Cutting mode	○	○
14	G67				Macro modal callCancel	○	○
15	G68				Mirror image for facing tool posts	○	○
	G69				Mirror image for facing tool posts Cancel	○	○
16	G68.1				Coordinate rotation by program	P952	P941
	G69.1				Coordinate rotation by program cancel Inclined surface machining cancel	○	○
	G176				Simple inclined surface control	P952	○
17	G96, G97				Constant surface speed control	P952	P941
18	G14, G15				Balance cut	○	○
19	G12.1				Milling interpolation	P952	P941
	G13.1				Milling interpolation cancel	○	○
20	G43.1, G44.1, G47.1				1st spindle control mode, 2nd spindle control mode, multiple-spindle simultaneous control mode	○	○

(*1) Do not issue the command under Inclined surface control or tool center point control.

Arbitrary axis exchange (G140, G141, G142)

You can perform simple inclined surface control (G176) or simple tool center point control (G174) by using the axis that was exchanged with the arbitrary axis exchange command.
However, if you attempt to take an axis of a part system in simple inclined surface control (G176) or simple tool center point control (G174) modal out of any other part system using an axis exchange command, the operation error (M01 1101) occurs. These alarms are cancelled by NC reset. If an arbitrary axis exchange command is issued during simple inclined surface control (G176) or simple tool center point control (G174) mode, the program error (P951) or (P942) occurs.

Reset, Emergency stop

NC reset can cancel simple inclined surface control (G176) and simple tool center point control (G174). However, if the modal state is not initialized by reset, the modal information of simple inclined surface control (G176) is retained, but that of simple tool center point control (G174) is canceled. (The modal retention reset setting depends on the MTB specifications (parameter "#1151 rstint").) Whether the modal reset is either retained (NC reset 1) or not (NC reset 2) when the emergency stop is canceled depends on the MTB specifications (parameter "#1282 ext18/bit6").

Circular interpolation, helical interpolation (G02, G03)

Issuing a circular command during simple tool center point control (G174) causes the program error (P942). However, a circular interpolation and helical interpolation are enabled under the following conditions.

(1) When simple inclined surface control (G176) is not active, and B axis (the tool head rotation axis) angle is at 0° (machine coordinate system)

(2) When simple inclined surface control (G176) is active, and B axis (the rotation angle of the tool head) is at an angle that makes the tool direction coincide with a line perpendicular to the inclined surface.
The angles of B axis and inclined surface are both clamped in the range of 0° to 360°.

Reference position return command (G28, G30)

If G28 or G30 is issued during simple inclined surface control (G176), the control is performed relative to the feature coordinate system up to the intermediate point, and then relative to the machine coordinate system after the point.

Reading position information (system variables)

During simple inclined surface control (G176), the coordinates that are relative to the feature coordinate system are set to the system variables for position information (#5001 to #5100+n (excluding #5021 to #5021+n)). However, the coordinates that are relative to the machine coordinate system are set to the variables #5021 to #5021+n (machine coordinate values) even during simple inclined surface control.

Tool No./Tool Compensation No. (T Code)

If simple tool center point control (G174) command is issued without any T code, the program error (P941) occurs.
If a T command is issued during simple tool center point control (G174) mode, the program error (P942) results.
These alarms are cancelled by NC reset.
Make sure to issue the T command and select the tool length before executing simple inclined surface machining.

Restart Search

If you execute restart search during simple inclined surface control (G176) or simple tool center point control (G174) mode, the program error (P49) occurs.

Linear angle command, Figure rotation command

If a linear angle command or figure rotation command is executed during simple inclined surface control mode (G176), the program error (P951) occurs.

Manual arbitrary reverse run

Reverse run is disabled for the simple inclined surface control start or cancel command (G176 or G69.1) or for the simple tool center point control start or cancel command (G174 or G175). It is not possible to go back to a block that precedes the command block.
Reverse run is enabled during simple inclined surface control (G176) or simple tool center point control (G174) mode. A programmed path can be reversed.

Manual speed command

Reverse run is enabled during simple inclined surface control (G176) or simple tool center point control (G174) mode. A programmed path can be reversed.

Inclined Axis Control

When the inclined axis control is active (YC35:ON), do not issue a command that causes the inclined axis to move during simple inclined surface control (G176) or simple tool center point control (G174).

Actual feedrate display

During the simple tool center point control (G174) mode, the screen indicates the travel speed of the tool tip position.

Cutting feed / Rapid traverse override

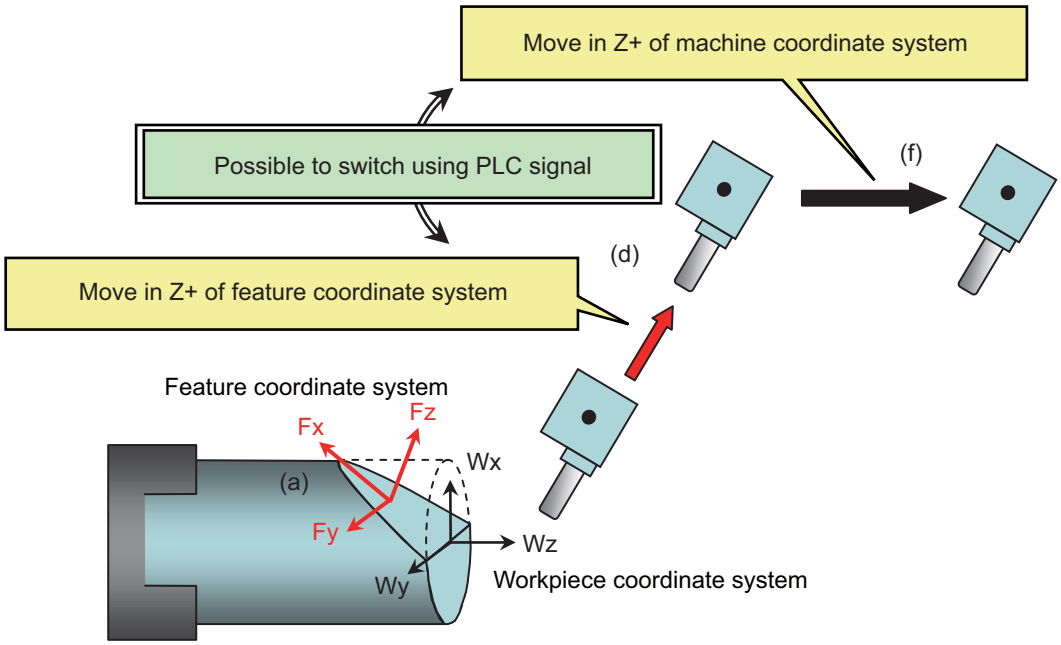
During the simple tool center point control (G174) mode, the override is applied to the feedrate on the tool tip point.
If speed clamp is ON, the override is applied to the clamp speed.

Corner chamfering/Corner R

If the corner chamfering/corner R command is issued during simple tool center point control (G174) mode, simple tool center point control (G174) will be valid for the path after corner chamfering/corner R.
In the block for simple tool center point control (G174) command, do not command corner chamfering/corner R.

3-dimensional manual feed

(1) Manual feed in feature coordinate system
When the parameter "#7912 NO_MANUAL" is "0", and you perform manual feed in the simple inclined surface control modal status, the feed can be done in the feature coordinate axis direction. In addition, when you use the PLC output signals YD1A (3-dimensional manual feed [JOG,INC] in feature coordinate system) and YD1D (3-dimensional manual feed [1st handle] in feature coordinate system), you can switch the axial direction between the feature coordinate and machine coordinate systems.



(a) Activate memory/MDI mode and command simple inclined surface control (G176) through a program.

(b) Use Reset 1 ("#1151rstint" = 0) to reset the NC with the feature coordinate system retained.

(c) Change the mode to handle, and select the feature coordinate system (turn ON the PLC output signal YD1D).

(d) Select the Z+ direction of the handle.
-> The axis moves in Z+ direction of the feature coordinate system.

(e) Select the machine coordinate system (turn OFF the PLC output signal YD1D).

(f) Select the Z+ direction of the handle.
-> Axis moves in Z+ of the machine coordinate system.

At this time, the NC retains the feature coordinate system. Thus, by selecting the feature coordinate system again (turning ON the PLC signal YD1D), you can move the axis in the feature coordinate axis direction. If you wish to cancel the feature coordinate system, switch the mode to memory/MDI, and issue the Inclined surface control cancel command (G69.1) through a program.

(2) Counter display during manual feed
[Inclined surface coordinate position counter]
The machine position on the feature coordinate system is displayed in the inclined surface coordinate position counter.
If inclined surface control cancel command is invalid, the machine position on the feature coordinate system is displayed in the inclined surface coordinate position counter.
However, while the manual absolute signal is OFF, the inclined surface coordinate position counter is not updated.

Manual absolute signal	Inclined surface machining command	Selected coordinate system	Inclined surface coordinate position counter
ON	ON	Feature coordinate system	The machine position is displayed on the feature coordinate system.
		Other than feature coordinate system	
	OFF	Feature coordinate system	The machine position is displayed on the workpiece coordinate system.
		Other than feature coordinate system	
OFF	ON/OFF	-	Not updated

(3) [Manual interruption amount counter]
If simple inclined surface control is valid, the manual interruption amount on the feature coordinate system is displayed in the manual interruption amount counter.
If simple inclined surface control is invalid, the manual interruption amount on the machine coordinate system is displayed in the manual interruption amount counter.
However, while the manual absolute signal is ON, the manual interruption amount counter is not updated.

Manual absolute signal	Inclined surface machining command	Selected coordinate system	Manual interruption amount counter
ON	ON/OFF	-	Not updated
	ON	-	The interruption amount in the feature coordinate system is displayed.
OFF	OFF	Machine coordinate system	The interruption amount in the machine coordinate system is displayed.

Precautions and restrictions

(1) During simple inclined surface control or simple tool center point control, even if G00 non-interpolation is set, the axis moves linearly (interpolation type) to the programmed position. (Parameter "#1086 G0Intp")

(2) Indirect interpolation type is applied regardless of the parameter setting. (Parameter "#7910 SLCT_INT_MODE")

(3) Make sure to specify the name of the three orthogonal axes set in the rotary axis configuration parameters ("#7900 RCDAX_I" to "#7902 RCDAX_K") if the axes are present in the part system.

X, Y and Z axes are present	#7900=X, #7901=Y, #7902=Z
X and Y axes are present	#7900=X, #7901="0", #7902=Z

(4) When a tool is to be installed in the left-hand orthogonal coordinate system, the target rotation direction parameters ("#7923 DIR_T1", "#7933 DIR_T2", "#7943 DIR_W1", and "#7953 DIR_W2") must be set to CCW.
In case of the work-stationary-type tool tilt for the simple inclined surface machining, set "#7933" to CCW.

(5) When simple inclined surface control (G176) and simple tool center point control (G174) are used together, always command so that the start and cancel of simple inclined surface control are nested within the start and cancel of simple tool center point control start or cancel. If simple inclined surface control start or cancel is commanded during simple tool center point control, it causes a program error (P952).

(During simple inclined surface control)	G176 X_Z_D_; :	Simple inclined surface machining control command
	G174 Rr; : (During simple tool center point control)	Simple tool center point control start
	G175;	Simple tool center point control cancel
	: G69.1;	Simple inclined surface control cancel

(6) Always command simple tool center point control start or cancel command (G174) while the tool is facing the Z axis (- direction). If simple tool center point control is commanded while the tool is facing any other directions than the Z axis (- direction), it is not possible to execute compensation correctly.

(7) During simple inclined surface control or simple tool center point control, SSS control is temporarily disabled.

(8) During simple tool center point control, if a travel command is issued to any axis not specified for the rotary axis configuration parameters, the program error (P942) occurs.

(9) The part system in which simple inclined surface machining or Inclined surface machining is being carried out does not cancel mixed control regardless of the setting for the "#1280 ext16/bit1" parameter (cancellation of mixed control by resetting) even if a reset operation that does not reset the modal (parameter "#1151 rstint" = 0 and NC reset 1) is carried out.
If an axis in a system in which simple inclined surface machining is being carried out is specified as the axis to be exchanged in the part system, axis exchange will not be possible and an operation error (M01 1101) will occur regardless of whether the automatic operation mode has been established.

19.1.1 Simple Inclined Surface Control ; G176

Function and purpose

Refer to "19.1 Simple Inclined Surface Machining".

Command format

Simple inclined surface control start

G176 X_ Z_ D_;	**Rotates K-I plane about the axis parallel to the J axis.**

G176 X_ Y_ D_;	**Rotates I-J plane about the axis parallel to the K axis.**

G176 Y_ Z_ D_;	**Rotates J-K plane about the axis parallel to the I axis.**

X/Y/Z	Feature coordinate system's origin (Rotation center position)
D	Rotation angle

Note

(1) The command is enabled when the G code system of the program is 2, 3, 4 or 5 (#1037 cmdtyp=3, 4, 5 or 6).
If the command is issued while the G code system is other than 2, 3, 4 or 5, the program error (P34) occurs.

(2) The command belongs to G code group 16 modal.

(3) G176 appears on the modal status screen during simple inclined surface control (G176).

(4) Make sure to command G176 as a single command in a block. If any other G code is included in the G176 command block, the program error (P954) occurs.

(5) If G176 is issued during the G176 modal status, the program error (P951) occurs.

(6) If the addresses of the axis specified by the parameters "#7900 RCDAX_I", "#7901 RCDAX_J", or "#7902 RCDAX_K", or any other addresses than address D are commanded, the program error (P954) occurs.

(7) If the value of address D is outside the command range, the program error (P35) occurs.

Simple inclined surface control cancel

G69.1 ;

Note

(1) Make sure to put G69.1 as a single command in a block.
If any other G code or travel command is included in the G69.1 command block, the program error (P954) occurs.

(2) If a cancel command is issued during any modal state other than G176, the cancel command is ignored.

(3) G176 (single command) can also cancel the G176 modal status.

(4) When not in simple inclined surface control (G176), "G69.1" appears on the modal status screen.

Detailed description

Detailed address setting

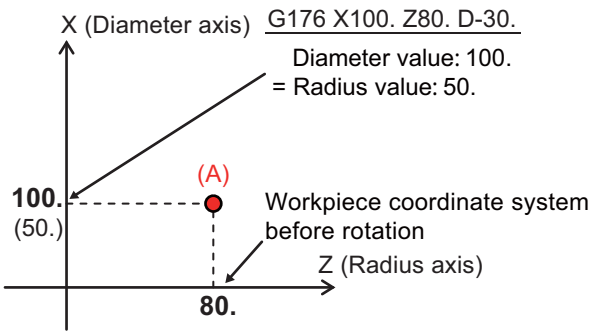
Address	Meaning	Command range (unit)	Remarks
X/Y/Z	Feature coordinate system's origin (Rotation center position)	-99999.999 to 99999.999	Specify the origin of the feature coordinate system that is viewed from the workpiece coordinate system, using the coordinates of the two axes that configure the plane to rotate. •Specify the position from the workpiece coordinate origin using an absolute value. •The origin position for simple inclined surface control is specified with an absolute value, irrespective of G90/G91. •Use a command address of the axis specified by the parameters "#7900 RCDAX_I", "#7901 RCDAX_J", or "#7902 RCDAX_K" for axis command. •Use an absolute value name for axis command. A program error (P954) will occur if an incremental value is used. •It is not possible to omit the rotation center coordinates. •Whether to use a diameter or radius to specify the origin of coordinate rotation depends on the MTB specifications (parameter "#1009 dia").
D	Rotation angle	-359.999 to 359.999 (°)	Specify the angle by which to rotate the coordinate plane. •Rotation angle (D) is always an absolute value, irrespective of G90 or G91. •It is not possible to omit the rotation angle. •If the rotation of the coordinate system is CCW when the rotation center is viewed from the positive direction of each rotation center axis, the rotation is regarded as positive.

Operation example

Define the coordinate system after a workpiece coordinate system is rotated

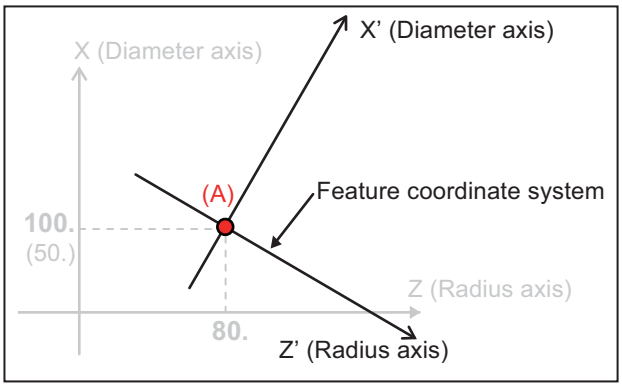
(Example) When the command "G176 X100 Z80. D-30." is issued.

(1) Issue G176 to rotate the workpiece coordinate system by any desired angle with any desired position in the workpiece coordinate system as the rotation center, and define the feature coordinate system.



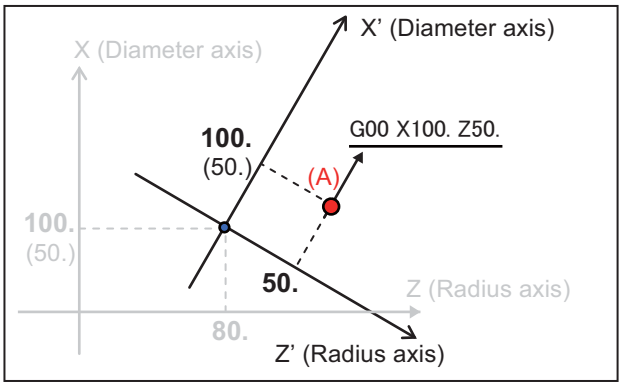
Whether to use a diameter or radius to specify the origin of coordinate rotation (X/Y/Z) depends on the MTB specifications (parameter "#1009 dia").

(2) Rotate the workpiece coordinate system with the specified coordinates as the workpiece origin to define the feature coordinate system. At this time no axes move.



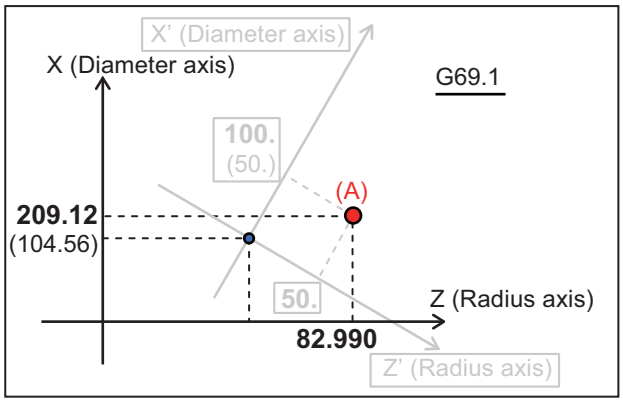
After coordinate system rotation, the position programmed in G176 block serves as the rotation coordinate origin. (Point (A) in the figure)

(3) During the simple inclined surface control mode, the control calculates the machine position so that it will be relative to the feature coordinate system.



A position command given during the simple inclined surface control mode specifies a position in feature coordinate system.
(Point (A) in the left-side figure indicates the position of "X100(50). Z50." on the feature coordinate system.)
If a travel command is given to a diameter designation axis during the simple inclined surface control mode, use a diameter value for the travel command.

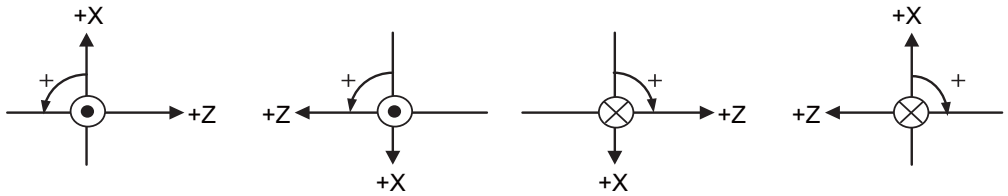
(4) Use either G69.1 or G176 (single command) to cancel the simple inclined surface control modal status. At this time, no axes move, but the setting of the feature coordinate system is cancelled and the original workpiece coordinate system becomes active.



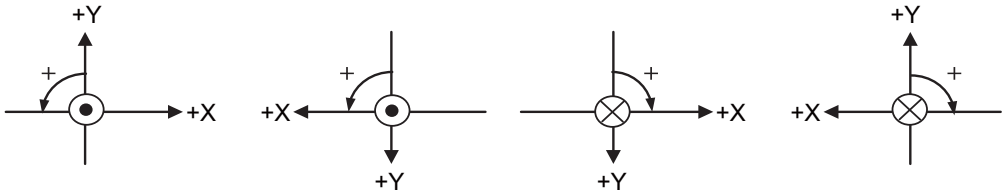
Point (A) in the left-side figure indicates the position of "X173.30 Z223.20" on the workpiece coordinate system before rotation.

Define the rotation direction by simple inclined surface control (G176)

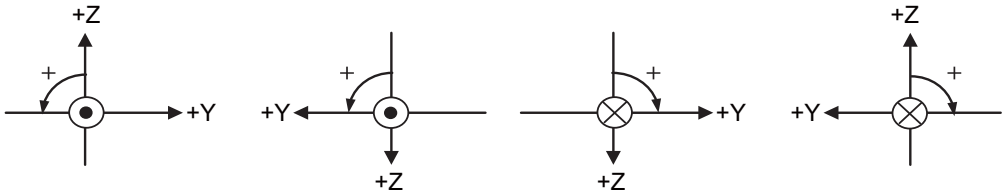
[When rotating ZX plane around J axis]



[When rotating XY plane around Z axis]



[When rotating YZ plane around X axis]



⊙ :The forward direction of the rotary axis is on the front side.

⊗ :The forward direction of the rotary axis is on the back side.

↷ :"D_" command direction

Relationship with other functions

Refer to "19.1 Simple Inclined Surface Machining".

19.1.2 Simple Tool Center Point Control ; G174

Function and purpose

Refer to "19.1 Simple Inclined Surface Machining".

Command format

Simple tool center point control start

G174 R__ ;

R	Tool axis rotation angle compensation amount

Note

(1) Simple tool center point control is started by assuming that the tool is facing the Z axis (- direction) with the angle of B axis at the start command.

(2) Always issue this command while the tool used for simple tool center point control is facing the Z axis (- direction) of the machine coordinate system.

(3) The command is enabled when the G code system of the program is 2, 3, 4 or 5 (#1037 cmdtyp=3, 4, 5 or 6). If the command is issued while the G code system is other than 2, 3, 4 or 5, the program error (P34) occurs.

(4) The command belongs to G code group 8 modal.

(5) G174 appears on the modal status screen during simple tool center point control (G174).

(6) If G174 is issued without any tool compensation command (T code), the program error (P941) occurs.

(7) A G174 command issued during the G174 modal state is ignored.

(8) If an axis travel command is given along with G174, the command follows either the G00 or G01 modal state. It is possible to issue G00 or G01 in the same block.
However, if any travel command other than G00 and G01 is given, the program error (P941) occurs.

(9) If no axis travel command is given along with G174, the axes will not move (No startup operation). At the first axis travel command after G174, the startup operation is carried out.

(10) If an axis travel command is given simultaneously with G174, the tool moves so that the tool tip position coincides with the programmed position (Startup operation).

(11) Tool axis rotation angle compensation amount is able to be specified with the address R of the G174 block, while parameter "#1450 5axis_Spec/bit1" is 0 (Using G174 tool axis rotation angle as compensation amount). (This setting depends on the MTB specifications.)

(12) If the value of address R is outside the command range, a program error (P35) will occur.

Simple tool center point control cancel command

G175;

Note

(1) The command is enabled when the G code system of the program is 2, 3, 4 or 5 (#1037 cmdtyp=3, 4, 5 or 6). If the command is issued while the G code system is other than 2, 3, 4 or 5, the program error (P34) occurs.

(2) A G175 command issued during the G175 modal state is ignored.

(3) If an axis travel command is included in G175 block, a program error (P33) will occur.

(4) "G175" appears on the modal status screen when not in simple tool center point control (G174).

Detailed description

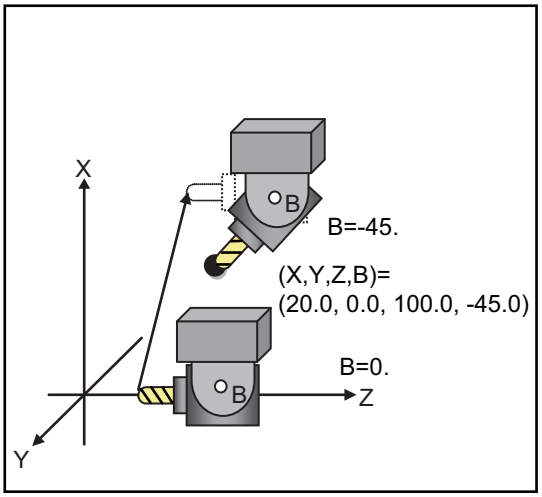
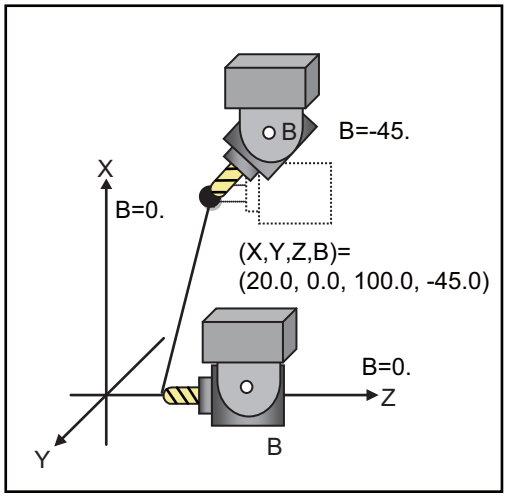
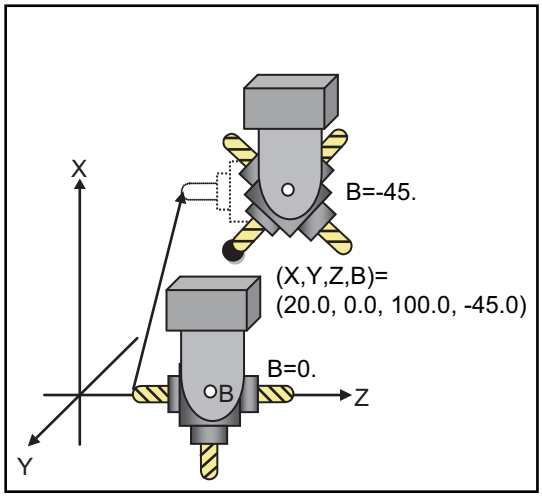
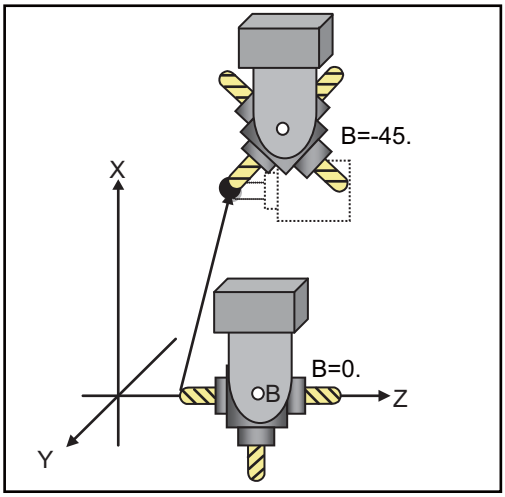
Detailed address setting

Address	Meaning	Command range (unit)	Remarks
R	Tool axis rotation angle compensation amount	-359.999 to 359.999 (°)	Compensates the rotation angle reference position with respect to the tool-side rotary axis. ♦Use the address R to specify the tool offset angle relative to the Z axis (- direction) of the machine coordinate system. For details, refer to "How to apply tool axis rotation angle compensation amount". When parameter "#7233 DIR_T2" (rotation direction of the tool-side rotary axis) is "0", the CW direction is treated as positive. When this parameter is "1", the CCW direction is treated as positive. This setting depends on the MTB specifications. ♦Address R can be omitted. If address R is omitted, it will be handled as "R0.".

Start simple tool center point control (startup operation)

(1) When no axis travel command is included in the G174 command block
If no axis travel command is given along with G174, the axes will not move. If a first travel command is given after G174, the tool moves so that the tool tip coincides with the programmed position.

(2) When an axis travel command is included in the G174 command block
If a travel command is given along with G174, the tool moves so that the tool tip coincides with the programmed position.

	(1) When no axis travel command is included in the G174 command block	**(2) When an axis travel command is included in the G174 command block**
	G00 X150. Y0. Z20. B-45.;	G174 X150. Y0. Z20. B-45.;
non-turret type		
Turret type		

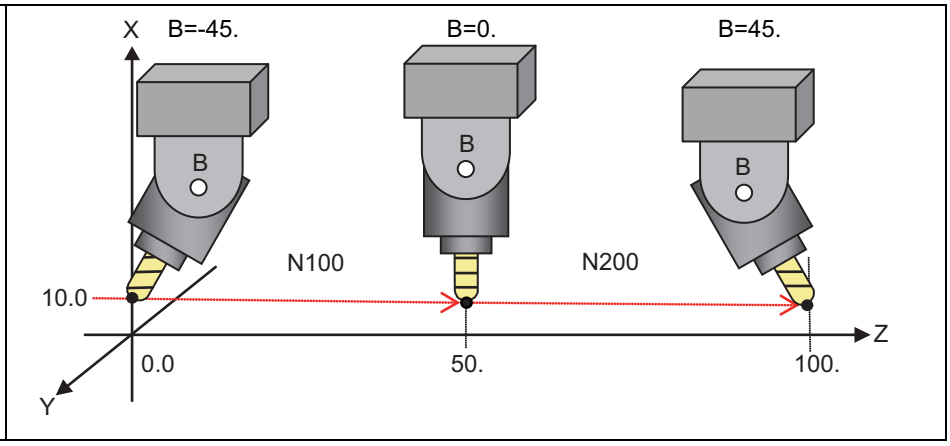
Cancel operation

Cancel operation for the tool offset is not carried out as a result of the cancel command.

Operation during the modal state

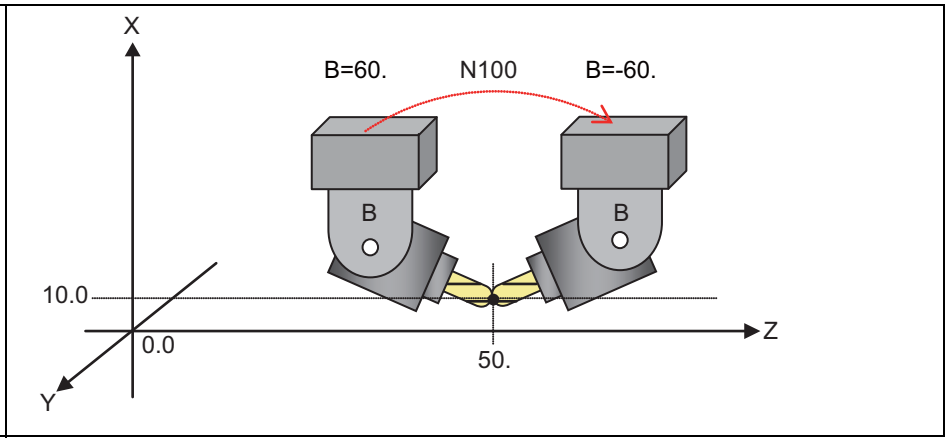
(1) When travel commands are given to orthogonal and rotary axes
The tool tip moves along the programmed path. (Refer to following figure.)

```
G174;
G0 X10. Y0. Z0. B-45.;
N100 G01 Z50. B0. F1000;
N200 Z100. B45.;
```



(2) When a travel command is given to a rotary axis only
The tool tip position does not move. The orthogonal axes are controlled so that the tool rotates around the tool tip position. (Refer to following figure.)

```
G174;
G0 X10. Y0. Z50. B60.;
N100 G01 B-60. F1000;
```



(3) If a travel command is given to any axis not specified for the rotary axis configuration parameters, the program error (P942) occurs.

How to apply tool axis rotation angle compensation amount

If you want to perform simple tool center point control (G174) using a tool that is not parallel to Z(-) of the machine coordinate system while the tool axis rotation angle is at 0 degree, G174 can be implemented by using the offset angle (tool axis rotation angle compensation amount).
This is effective, for instance, when you use any tool mounted on a turret for machining.

There are two types of methods to apply the tool axis rotation angle compensation amount: "Offset angle designation method" and "Rotary axis position automatic-read method". The method is chosen depending on the MTB specifications (parameter "#1450 5axis_Spec/bit1").

(1) Offset angle designation method ("#1450 5axis_Spec/bit1" is 0)
Use the R register and the address R of the G174 block to specify the tool's angle relative to the reference position rotation angle (Z (-) direction of the machine coordinate system) while B axis is at 0 degree. During the G174 modal state, compensation is performed using the total of the angle specified in the address R and that of the R register.

(a) Use the address R of the G174 block for the tool angle offset command

- Set "0" in the R registers (R2634 to R2635), and then issue the command "G174 R90." in order that you can use Tool "A" in machining during the G174 modal state. (Refer to (A) in the following figure.)
- If the parameter "#7233 DIR_T2" is "0" (CW direction), issue the command "G174 R270." (Refer to (B) in the following figure.)

(b) Use the R registers for the tool angle offset command

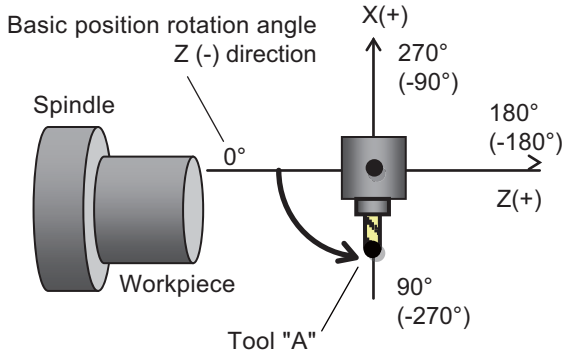
- Set "90000" in the R registers (R2634 to R2635), and then issue the command "G174 (R0)" in order that you can use Tool "A" in machining during the G174 modal state. (Refer to (A) in the following figure.)
- The machine behaves the same when the address R is omitted in the G174 block.
- If "#7233 DIR_T2" is "0" (CW direction), set "270000" in the R registers (R2634 to R2635) and issue G174 command. (Refer to (B) in the following figure.)

(c) It is also possible to specify the rotary angle's reference position in the R register, and then specify the offset angle of the tool from the reference tool using the address R.

- Set "30000", an offset to the reference tool's reference position, in the R registers (R2634 to R2635). After that, use the address R of the G174 block to specify 0, 60, 120, 180, 240 or 300 degrees so that you can use any desired tool on the hexagon turret to implement simple tool center point control during the G174 modal state. (Refer to (C) in the following figure.)

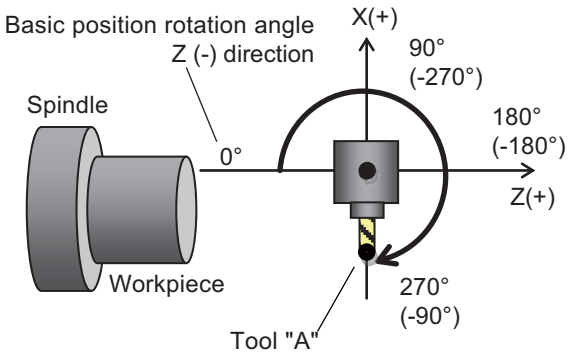
(A) Offset angle setting (CCW)

B axis 0°(At CCW setting)



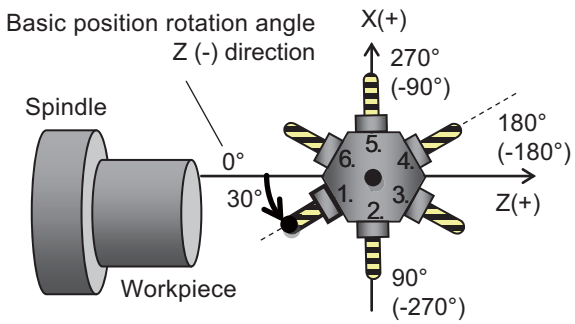
(B) Offset angle setting (CW)

B axis 0°(At CW setting)



(C) Offset angle setting (CCW, hexagon turret)

B axis 0°(At CCW setting)



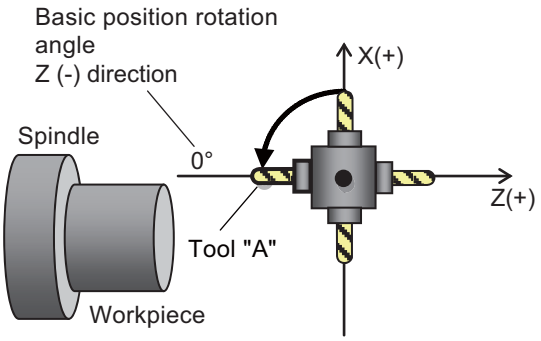
<Note>

- In the address R of the G174 block and the R register, specify the tool angle relative to the reference position rotation angle when B axis is at 0 degree. Thus, even when the tool you use is not at 0 degree of B axis as a result of the B axis rotation, make sure to set the offset angle for B axis at 0 degree.

(2) Rotary axis position automatic-read method ("#1450 5axis_Spec/bit1" is "1")
The tool-side rotary axis position when simple tool center point control (G174) command is issued is treated as the tool axis rotation angle compensation amount.
Direct the tool you use toward the reference position rotation angle (Z(-) of the machine coordinate system) before issuing a G174 command.

(Example) If you use a tool (Tool "A") that is at 90 (-270) degrees relative to the reference position rotation angle while "#7233 DIR_T2" is "1" (CCW direction), use the following machining program.

T1010 ;	
G00 X0. Z0. B90.	Direct the tool (Tool "A") toward the Z (-) direction of machine coordinate system (Refer to following figure.)
G92 B0.	Coordinate system setting
G174 ;	Simple tool center point control command
G01 X100. B45. F1000 ;	
:	
G175;	Simple tool center point control cancel



Rotate the target tool by the basic position rotation angle, then issue G174 command.

<Note>
- If you issue a G174 command after cancelling the G174 modal state, you need to rotate the tool you use to the reference position rotation angle before issuing the G174 command.

Feedrate during simple tool center point control

During simple tool center point control (G174), the feedrate is controlled so that the tool tip moves at a programmed speed.

20

Coordinate System Setting Functions

20.1 Coordinate Words and Control Axes

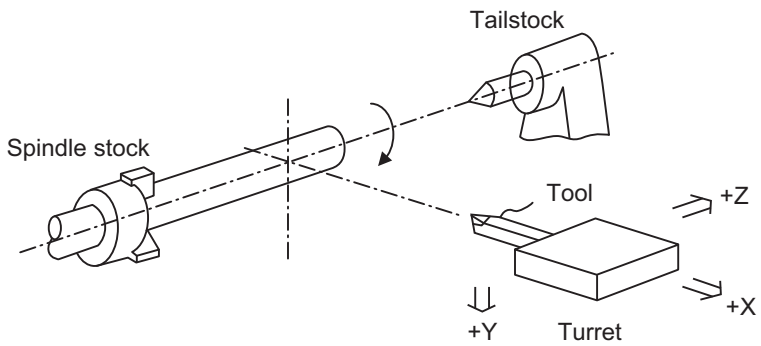
Function and purpose

In the case of a lathe, axis names (coordinate words) and directions are defined as follows.

The axis at right angles to the spindle
Axis name: X axis

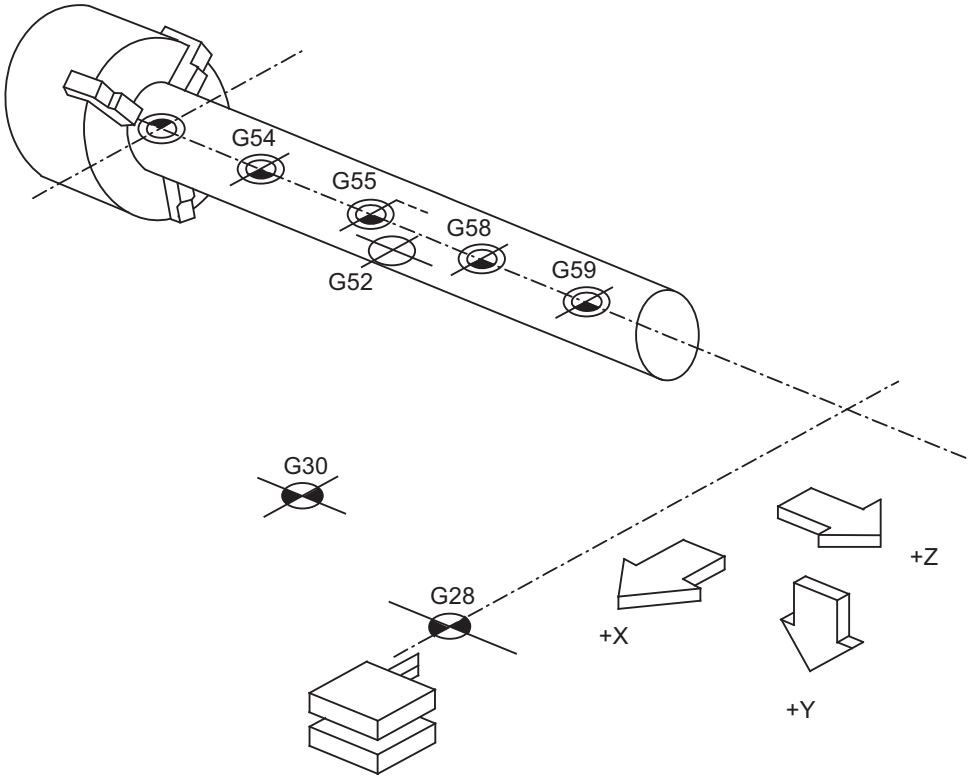
The axis parallel to the spindle
Axis name: Z axis

Coordinate axes and polarities



Since coordinates based on the right hand rule are used with a lathe, in the above figure, the positive direction of the Y axis which is at right angles to the X-Z plane is downward.
Note that a circular on the X-Z plane is expressed as clockwise or counterclockwise as seen from the forward direction of the Y axis.
(Refer to "Circular Interpolation; G02, G03".)

Relationship between coordinates



Reference position

Basic machine coordinate

Workpiece coordinate zero point

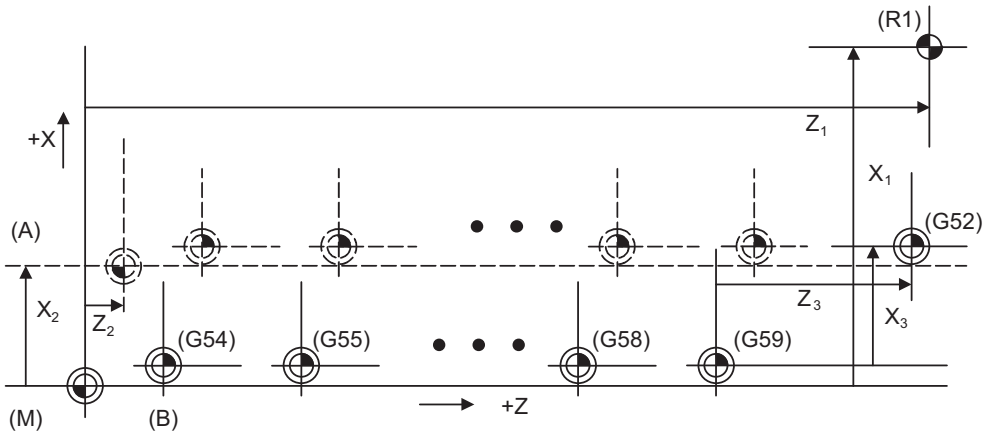
Local coordinate zero point

20.2 Types of Coordinate Systems

20.2.1 Basic Machine, Workpiece and Local Coordinate Systems

Function and purpose

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 basic 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 position 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 position is brought to the position specified by the parameter from the basic machine coordinate zero point (machine zero point).



(A) Hypothetical machine coordinate system (G92 shift)
(B) Machine zero point
(G54) Workpiece coordinate system 1
(G55) Workpiece coordinate system 2
(G58) Workpiece coordinate system 5
(G59) Workpiece coordinate system 6
(G52) Local coordinate system
(R1) 1st reference position
(M) Basic machine coordinate system

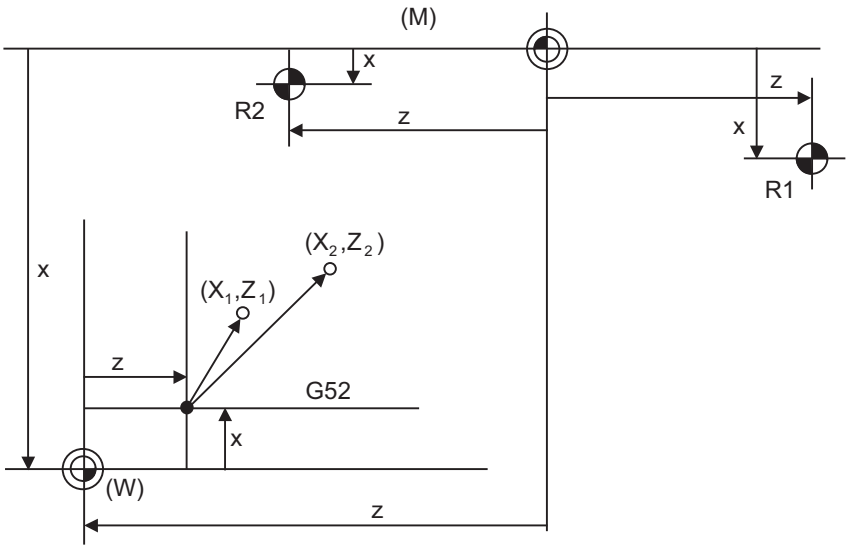
The local coordinate systems (G52) are valid on the coordinate systems designated by workpiece coordinate systems 1 to 6.
The hypothetical machine coordinate system can be set on the basic machine coordinate system using a G92 command. At this time, the workpiece coordinate system 1 to 6 is also simultaneously shifted.

Also refer to "Coordinate Systems and Coordinate Zero Point symbols".

20.2.2 Machine Zero Point and 2nd Reference Position (Zero point)

Function and purpose

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.
2nd reference position (zero point) relates to the position of the coordinates which have been set beforehand by parameter from the zero point of the basic machine coordinate system.

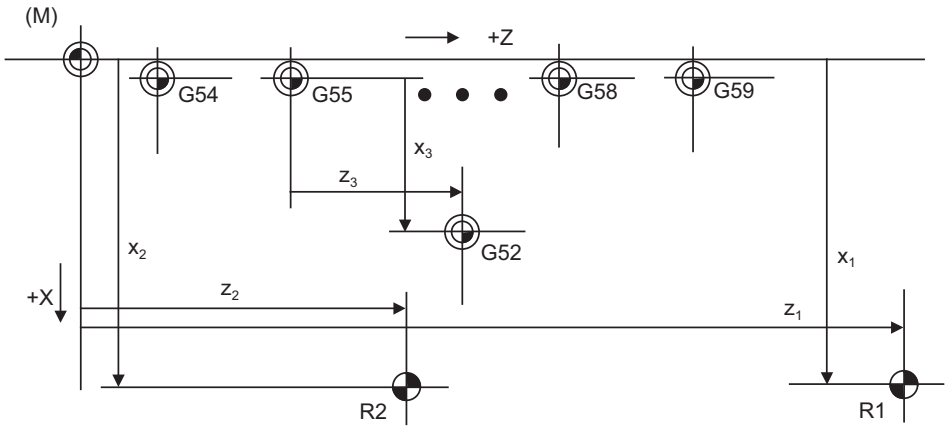


(M) Basic machine coordinate system
(G52) Local coordinate system
(W) Workpiece coordinate systems (G54 to G59)
(R1) 1st reference position
(R2) 2nd reference position

20.2.3 Automatic Coordinate System Setting

Function and purpose

This function creates each coordinate system according to the parameter values input beforehand from the setting and display unit when the first manual reference position return or the reference position is reached with the dog-type reference position return when the NC power is turned ON.
The actual machining program is programmed over the coordinate systems which have been set above.



(M) Basic machine coordinate system
(G52) Local coordinate system
(G54) Workpiece coordinate system 1
(G55) Workpiece coordinate system 2
(G58) Workpiece coordinate system 5
(G59) Workpiece coordinate system 6
(R1) 1st reference position
(R2) 2nd reference position

Detailed description

(1) The coordinate systems created by this function are as follow:
- Basic machine coordinate system
- Workpiece coordinate systems (G54 to G59)

(2) The parameters related to the coordinate system all provide the distance from the zero point of the basic machine coordinate system. Therefore, after deciding at which position the first reference position should be set in the basic machine coordinate system and then set the zero point positions of the workpiece coordinate systems.

(3) When the automatic coordinate system setting function is executed, shifting of the workpiece coordinate system with G92, setting of the local coordinate system with G52, shifting of the workpiece coordinate system with origin set, and shifting of the workpiece coordinate system with manual interrupt will be canceled.

(4) The dog-type reference position return will be executed when the first time manual reference position return or the first time automatic reference position return is executed after the power has been turned ON. It will be also executed when the dog-type is selected by the parameter for the manual reference position return or the automatic reference position return for the second time onwards.

⚠ CAUTION

⚠ If the workpiece coordinate offset amount is changed during automatic operation (including during single block operation), it will be validated from the next block or after multiple blocks of the command.

20.2.4 Coordinate System for Rotary Axis

Function and purpose

The axis designated as the rotary axis with the parameters is controlled with the rotary axis' coordinate system.
The rotary axis includes the rotating type (short-cut valid/invalid) and linear type (workpiece coordinate position linear type and all coordinate position linear type).
The workpiece coordinate position range is 0 to 359.999° for the rotating type, and 0 to ± 99999.999° for the linear type.
The machine coordinate value and relative position differ according to the parameters.
The rotary axis is commanded with a degree (°) unit regardless of the inch or metric designation.
The rotary axis type can be set with the parameter "#8213 rotation axis type" for each axis.

<table>
<tr><th rowspan="3"></th><th colspan="4">Rotary axis</th><th rowspan="3">Linear axis</th></tr>
<tr><th colspan="2">Rotating type rotary axis</th><th colspan="2">Linear type rotary axis</th></tr>
<tr><th>Short-cut invalid</th><th>Short-cut valid</th><th>Workpiece coordinate position linear type</th><th>All-coordinate position linear type</th></tr>
<tr><td>#8213 setting value</td><td>0</td><td>1</td><td>2</td><td>3</td><td>-</td></tr>
<tr><td>Workpiece coordinate position</td><td colspan="2">Displayed in the range of 0° to 359.999°.</td><td colspan="3">Displayed in the range of 0° to ± 99999.999°.</td></tr>
<tr><td>Machine coordinate position/relative position</td><td colspan="3">Displayed in the range of 0° to 359.999°.</td><td colspan="2">Displayed in the range of 0° to ± 99999.999°.</td></tr>
<tr><td>ABS command</td><td>The incremental amount from the end point to the current position is divided by 360 degrees, and the axis moves by the remainder amount according to the sign.</td><td>Moves with a short-cut to the end point.</td><td colspan="3">In the same manner as the normal linear axis, it moves according to the sign by the amount obtained by subtracting the current position from the end point (without rounding up to 360 degrees).</td></tr>
<tr><td>INC command</td><td colspan="5">Moves in the direction of the commanded sign by the commanded incremental amount starting at the current position.</td></tr>
<tr><td rowspan="2">Reference position return</td><td colspan="5">Depends on the absolute command or the incremental command during the movement to the intermediate point.</td></tr>
<tr><td colspan="3">Returns with movement within 360 degrees.</td><td colspan="2">Moves and returns in the R point direction for the difference from the current position to the R point.</td></tr>
</table>

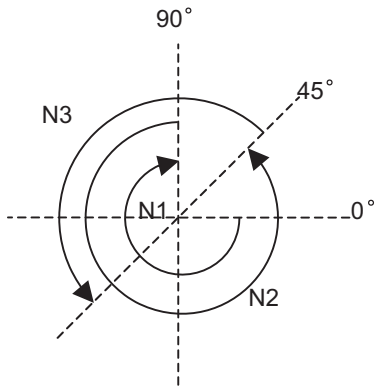
Operation example

Examples of differences in the operation and counter displays according to the type of rotation coordinate are given below.
(The workpiece offset is set as 0°.)

Rotary type (short-cut invalid)

(1)The machine coordinate position, workpiece coordinate position and relative position are displayed in the range of 0° to 359.999°.

(2)For the absolute position command, the axis moves according to the sign by the remainder amount obtained by dividing by 360°.

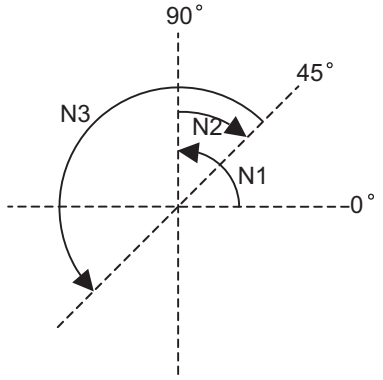


Program	Workpiece	Machine
G28 C0.		
N1 G90 C-270.	90.000	90.000
N2 C405.	45.000	45.000
N3 G91 C180.	225.000	225.000

Rotary type (short-cut valid)

(1)The machine coordinate position, workpiece coordinate position and relative position are displayed in the range of 0° to 359.999°.

(2)For the absolute position command, the axis rotates to the direction having less amount of movement to the end point.



Program	Workpiece	Machine
G28 C0.		
N1 G90 C-270.	90.000	90.000
N2 C405.	45.000	45.000
N3 G91 C180.	225.000	225.000

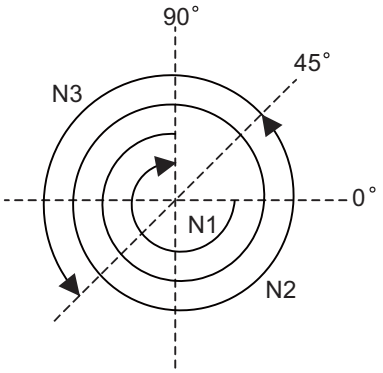
Linear type (workpiece coordinate position linear type)

(1) The coordinate position counter other than the workpiece coordinate position is displayed in the range of 0° to 359.999°.
The workpiece coordinate position is displayed in the range of 0 to ±99999.999°.

(2) The movement is the same as the linear axis.

(3) During reference position return, the axis moves in the same manner as the linear axis until the intermediate point is reached. The axis returns with a rotation within 360° from the intermediate point to the reference position.

(4) During absolute position detection, even if the workpiece coordinate position is not within the range of 0 to 359.999°, the system will start up in the range of 0 to 359.999° when the power is turned ON again.



Program	Workpiece	Machine	Relative position
G28 C0.			
N1 G90 C-270.	-270.000	90.000	90.000
N2 C405.	405.000	45.000	45.000
N3 G91 C180.	585.000	225.000	225.000
	After the power is turned ON again		
	Workpiece	**Machine**	
	225.000	225.000	

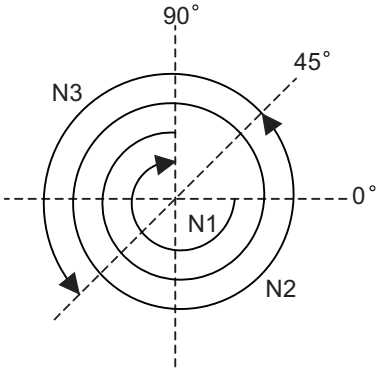
Linear type (all coordinate position linear type)

(1) The workpiece coordinate position counter is displayed in the range of 0 to ±99999.999°.

(2) The movement is the same as the linear axis.

(3) During reference position return, the axis moves in the same manner as the linear axis until the intermediate point is reached.
The axis rotates by the difference from the intermediate point to the reference position and returns to the reference position.

(4) During absolute position detection, the system starts up at the position where the power was turned OFF when the power is turned ON again.



Program	Workpiece	Machine	Relative position
G28 C0.			
N1 G90 C-270.	-270.000	-270.000	-270.000
N2 C405.	405.000	405.000	405.000
N3 G91 C180.	585.000	585.000	585.000
	After the power is turned ON again		
	Workpiece	**Machine**	
	585.000	585.000	

20.3 Basic Machine Coordinate System Selection ; G53

Function and purpose

The tool is moved to the position commanded on the basic machine coordinate system, using the G53 command, the feed mode command (G01 or G00) and the coordinate command that follow them.

Command format

```
G53 G00 X__ Z__ α__;
```

```
G53 G00 U__ W__ β__;
```

α	Additional axis
β	Incremental command 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 position (zero point) return position, which is determined by the automatic or manual reference position (zero point) return.

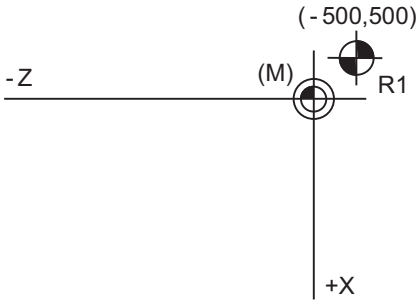
(2) The basic machine coordinate system is not changed by the G92 command.

(3) The G53 command is valid only in the designated block.

(4) When an incremental value is issued (U, W, β), the axis will move with the incremental value within the selected coordinate system.

(5) The 1st reference position coordinate value indicates the distance from the basic machine coordinate system zero point to the reference position (zero point) return position.

(6) The G53 command will move with cutting feedrate or rapid traverse following command modal.



(M) Basic machine coordinate system
(R1) 1st reference position

1st reference position coordinate value: X = -500 and Z=+500

(7) If the G53 command and G28 command (reference position return) are issued in the same block, the command issued last will be valid.

(8) Depending on the MTB specifications (bit5 of parameter "#1253 set"), all the movement commands in the G53 command block may run in rapid traverse.

(a) If the movement methods of the G53 command block follow command modal

[Example where the G53 block is executed during G01 modal]

Program	G group 01 modal	Cutting or rapid traverse
N01 G01 X100. Z100. F1000;	G01	Cutting
N02 G53 X200. Z200.;	G01	Cutting
N03 X300. Z300.;	G01	Cutting

[Example where the G53 block is executed during G00 modal]

Program	G group 01 modal	Cutting or rapid traverse
N01 G00 X100. Z100.;	G00	Rapid traverse
N02 G53 X200. Z200.;	G00	Rapid traverse
N03 X300. Z300.;	G00	Rapid traverse

(b) If all the movement methods of the G53 command block are rapid traverse.

[Example where the G53 block is executed during G01 modal]

Program	G group 01 modal	Cutting or rapid traverse
N01 G01 X100. Z100. F1000;	G01	Cutting
N02 G53 X200. Z200.;	G01	Rapid traverse
N03 X300. Z300.;	G01	Cutting

The G group 01 modal does not change in the G53 command block, but the operation changes to rapid traverse.

Relationship with Other Functions

(1) Milling interpolation
When the G53 command is issued in the compensation mode, the compensation vectors are temporarily eliminated and then, compensation mode will automatically return.
In this case, the compensation is not canceled, and the tool goes directly from the intersection point vector to the point without vectors, in other words, to the programmed command point. When returning to the compensation mode, it goes directly to the intersection point.

(2) Tool Compensation Functions
When the G53 command is issued, the tool compensation amount for the axis with a movement command will be temporarily canceled.

(3) Nose R compensation
When the compensation direction is reversed during nose R compensation, a program error (P157) will occur except when it is commanded in G00 block. Even if directions differ between before and after the G53 block, an error will not occur because compensation is temporarily canceled. A parameter can be set to move the tool in the same compensation direction.

(4) Fixed cycle for turning machining
The group modal of fixed cycle for turning machining will be canceled by the G53 command.

(5) Mirror image for facing tool posts
Mirror image for facing tool posts will be invalidated during the movement of G53. Mirror image for facing tool posts will be validated from the next block.

(6) Machine coordinate system selection and feedrate designation
If an ",F" command is specified when there are no specifications for the feedrate specified for G53, a program error (P39) will occur.

(7) Polar coordinate interpolation
Do not issue the G53 command when the polar coordinate interpolation mode is active.

(8) High-speed High-accuracy Control
A program error will occur if the G53 command is issued when the high-speed high-accuracy control II mode is active.

Precautions

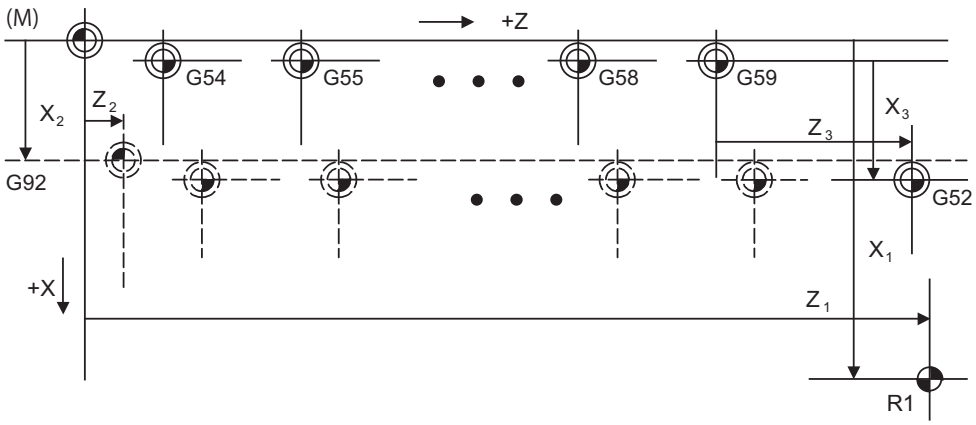
(1) In a machine with specifications that define all movement commands of the G53 command block run in rapid traverse, the block will run in rapid traverse even if the G53 and G01 commands are issued to that same block. However, the G group 01 modal switches so the movement changes to cutting feed for the following and subsequent blocks.
[Example where the G53 and G01 commands are issued to the same block]

Program	G group 01 modal	Cutting or rapid traverse
N01 G00 X100. Z100.;	G00	Rapid traverse
N02 G53 G01 X200. Z200. F1000;	G01	Rapid traverse
N03 X300. Z300.;	G01	Cutting

20.4 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 G92 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.



(M) Basic machine coordinate system
(G52) Local coordinate system
(G92) Hypothetical machine coordinate system (shifted by G92)
(R1) 1st reference position
(G54) Workpiece coordinate system 1
(G55) Workpiece coordinate system 2
(G58) Workpiece coordinate system 5
(G59) Workpiece coordinate system 6

Command format

```
G92 Xx2 Zz2 αα2;
```

α	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 rotation speed is set. (Refer to the section on setting the spindle clamp rotation speed.)

Precautions

(1) If the parameter "#1279 ext15/bit5" is set to "1", the coordinate systems setting (G92) shift amount is cleared when the axis reaches to the manual reference position.

20.5 Local Coordinate System Setting ; G52

Function and purpose

The local coordinate systems can be set on the G54 through 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  X__  Z__ ;
```

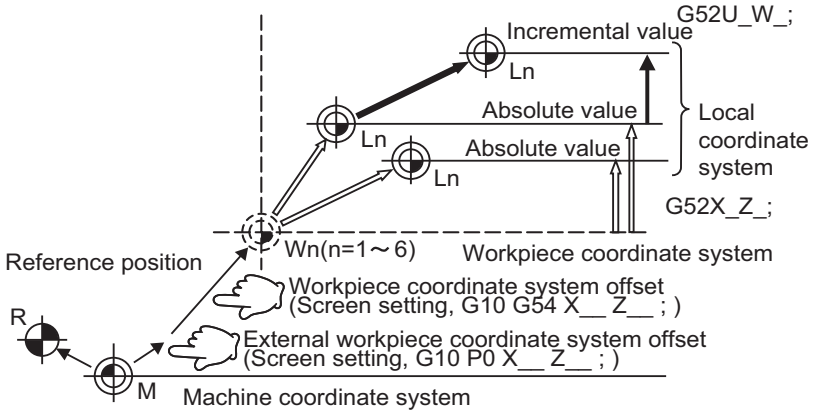
Detailed description

(1) The G52 command is valid until a new G52 command is issued, and the tool does not move. This command, G52, 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 will be cleared by the dog-type manual reference (zero) point return or reference (zero) point return performed after the power has been switched ON.

(3) The local coordinate system is canceled by (G54 to G59) G52 X0 Z0;.

(4) Coordinate commands in the absolute value cause the tool to move to the local coordinate system position.



Note

(1) If the program is executed repeatedly, the workpiece coordinate system will deviate each time. Thus, when the program is completed, the reference position return operation must be commanded.

20.6 Workpiece Coordinate System Setting and Offset ; G54 to G59 (G54.1)

Function and purpose

(1) The workpiece coordinate systems facilitate the programming on the workpiece, serving the reference position of the machining workpiece as the zero point.

(2) These commands enable the tool to move to the positions in the workpiece coordinate system. There are extended workpiece coordinate systems (G54.1) in addition to 6 workpiece coordinate systems, which are used by the programmer for programming (G54 to G59). The number of the extended workpiece coordinate systems varies depending on the MTB specifications.

(3) Among the workpiece coordinate systems currently selected by these commands, any workpiece coordinate system with coordinates that have been commanded by the current position of the tool is reset. (The "current position of the tool" includes the compensation amounts for tool nose R, tool length and compensation.)

(4) A hypothetical machine coordinate system with coordinates that have been commanded by the current position of the tool is set by these commands.
(The "current position of the tool" includes the compensation amounts for tool nose R, tool length and compensation.) (G54,G92)

Command format

Workpiece coordinate system

G54 to G59	**Workpiece coordinate system selection**

(G54 to G59) G92 X__ Z__ α__ ;	**Set workpiece coordinate system**

α	Additional axis

Extended workpiece coordinate system

G54.1 Pn ;	**Extended workpiece coordinate system selection (P1 to P48)**

G54.1 Pn ; G92 X__(U__) Z__(W__) ;	**Extended workpiece coordinate system setting (P1 to P48)**

Note

(1) The maximum number of coordinate systems depends on the specifications.

(2) Depending on the MTB specifications, G54Pn can be used as an extended workpiece coordinate system selection command (parameter "#1274 ext10/bit5").

G10 L20 Pn X__(U__) Z__(W__);	**Extended workpiece coordinate system offset amount setting (P1 to P48)**

When the extended workpiece coordinate offset amount of the workpiece coordinate system currently selected is rewritten

G10 G54.1 Pn X__(U__) Z__(W__) ;	**Extended workpiece coordinate system offset amount setting (P1 to P48)**

When the extended workpiece coordinate system is selected, and the offset amount is rewritten

Detailed description

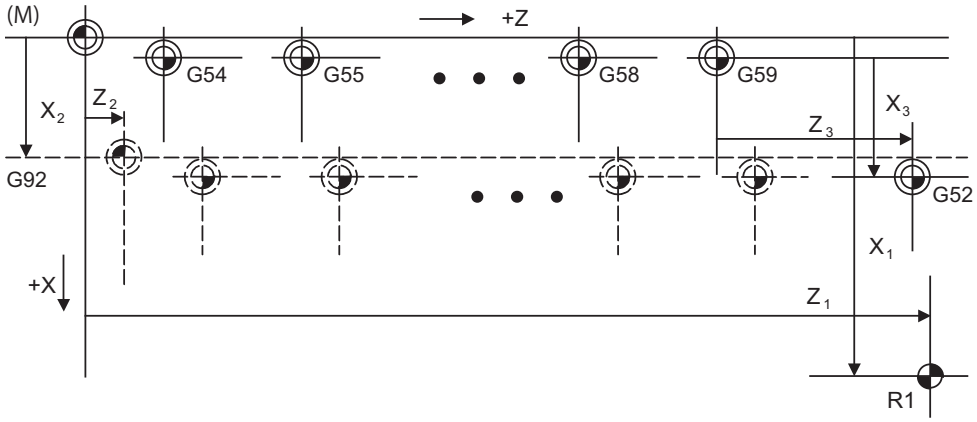
(1) With any of the G54 through G59 commands, the nose radius compensation amounts for the commanded axes will not be canceled even if workpiece coordinate system selection is commanded.

(2) The G54 workpiece coordinate system is selected when the power is turned ON.

(3) Commands G54 through G59 are modal commands (group 12).

(4) The coordinate system will move with G92 in a workpiece coordinate system.

(5) The offset setting amount in a workpiece coordinate system denotes the distance from the basic machine coordinate system zero point.



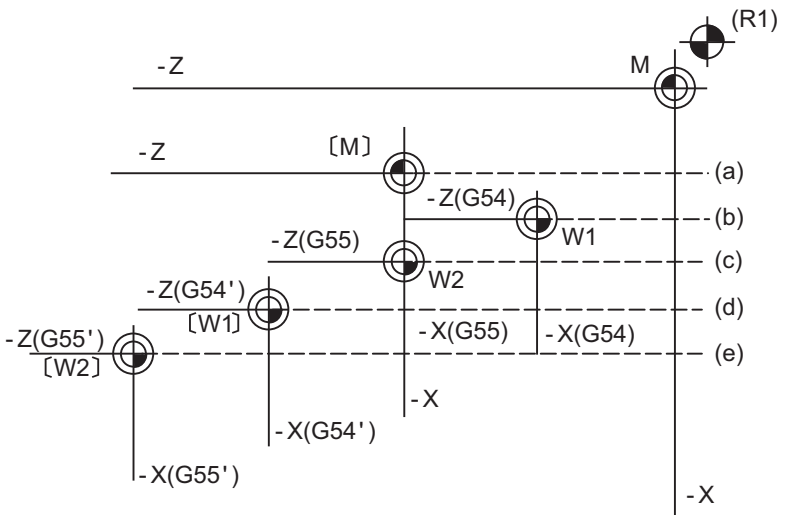
(M) Basic machine coordinate system
(G52) Local coordinate system
(G92) Hypothetical machine coordinate system (shifted by G92)
(R1) 1st reference position
(G54) Workpiece coordinate system 1
(G55) Workpiece coordinate system 2
(G58) Workpiece coordinate system 5
(G59) Workpiece coordinate system 6

(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.)
[Handling when L or P is omitted]

G10 L2 Pn Xx Zz ;	n=0 : Set the offset amount in the external workpiece coordinate system. n=1 to 6 : Set the offset amount in the designated workpiece coordinate system. Others : The program error (P35) will occur.
G10 L2 Xx Zz ;	Set the offset amount in the currently selected workpiece coordinate system. When in G54.1 modal, the program error (P33) will occur.
G10 L20 Pn Xx Zz ;	n=1 to n : Set the offset amount in the designated workpiece coordinate system. (The number of extended workpiece coordinate systems under the specifications) Others : Program error (P35) will occur.
G10 L20 Xx Zz ;	Set the offset amount in the currently selected workpiece coordinate system. When in G54 to G59 modal, the program error (P33) will occur.
G10 Pn Xx Zz ; G10 Xx Zz ; G10 G54.1 Xx Yy Zz ;	L10 (tool offset) will be judged if there is no L value.

(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) A hypothetical machine coordinate system is formed at the position that deviates from the new workpiece reference position (zero point) by an amount equivalent to the workpiece coordinate system offset amount.



(R1) Reference position 1	(a) Hypothetical machine coordinate system based on G92
(b) Old workpiece 1 (G54) coordinate system	(c) Old workpiece 2 (G55) coordinate system
(d) New workpiece 1 (G54) coordinate system	(e) New workpiece 2 (G55) coordinate system

After the power has been switched on, the hypothetical machine coordinate system is matched with the basic machine coordinate system by the first automatic (G28) or manual reference position (zero point) return.

(9) By setting the hypothetical machine coordinate system, the new workpiece coordinate system will be set at a position that deviates from that hypothetical machine coordinate system by an amount equivalent to the workpiece coordinate system offset amount.

(10) When the first automatic (G28) or manual reference position (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.

(11) If G54 X- ; is commanded after the reference position return (both automatic or manual) executed after the power is turned ON, a program error (P62) will occur. (A speed command is required as the movement will be controlled with the G01 speed.)

(12) Do not command a G code for which a P code is used in the same block as G54.1 or G10L20. If a G code is commanded, a P code is used for a prior G command or the program error occurs (P33).

(13) If there are no specifications for the extended workpiece coordinate system selection, a program error (P35) will occur when the G54.1 command is executed.

(14) If there are no specifications for the extended workpiece coordinate system selection, the program error (P172) will occur when the G10 L20 command is executed.

(15) A new workpiece coordinate system P1 can be set by commanding G92 in the G54.1 P1 mode. However, the workpiece coordinate system of the other workpiece coordinate systems G54 to G59, G54.1, and P2 to P48 will move in parallel with it, and a new workpiece coordinate system will be set.

(16) The offset amount of the extended workpiece coordinate system is assigned to system variables #7001 onwards.

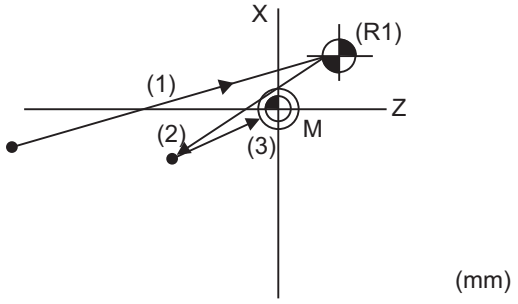
⚠ CAUTION

⚠ **If the workpiece coordinate system offset amount is changed during single block stop, the new setting will be valid from the next block.**

Program example

(Example 1)

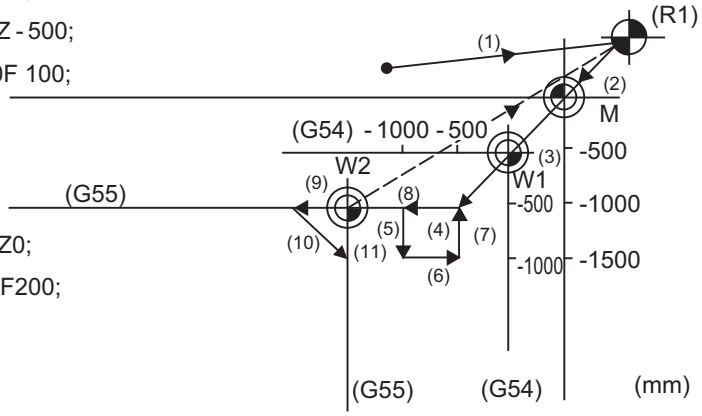
(1) G28 X0Z0;
(2) G53 X-500 Z-1000;
(3) G53 X0Z0;



When the coordinate value of the 1st reference position (R1) is zero, the basic machine coordinate system zero point (M) and reference position (zero point) return position (#1) will coincide.

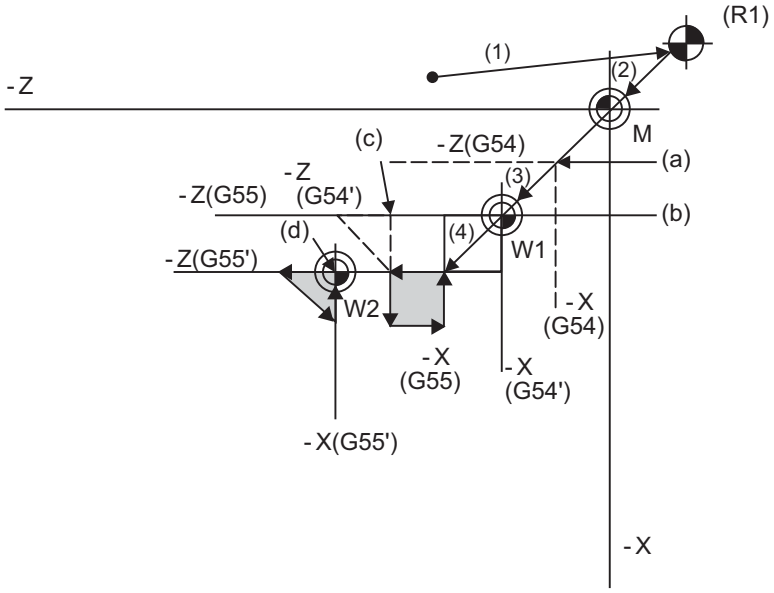
(Example 2)

(1) G28X0Z0;
(2) G00G53X0Z0;
(3) G54X-500Z-500;
(4) G01W-500F 100;
(5) U-500;
(6) W+500;
(7) U+500;
(8) G00G55X0Z0;
(9) G01Z-500 F200;
(10) X-500 Z0;
(11) G28X0Z0;



(Example 3) When workpiece coordinate system G54 (-500, -500) has deviated in Example 2. (It is assumed that (3) to (10) in Example 2 have been entered in subprogram 1111.)

(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.
(5) M98 P1111 ;	



(a) Old G54 coordinate system
(b) New G54 coordinate system
(c) Old G55 coordinate system
(d) New G55 coordinate system
(R1) Reference position return position

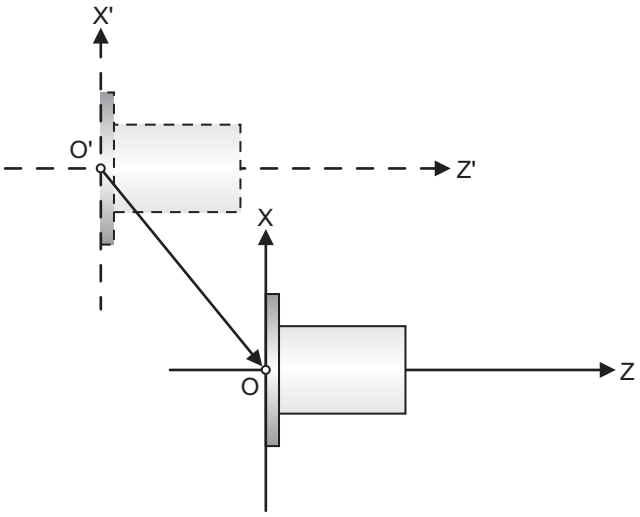
Note

(1) The workpiece coordinate system will deviate each time steps (3) to (5) shown in the above figure are repeated. The reference position return (G28) command should therefore be issued upon completion of the program.

20.7 Workpiece Coordinate System Shift; G54 to G59 (G54.1)

Function and purpose

This function allows you to shift the workpiece coordinate system.
There may be a case in which the workpiece coordinate system assumed when the machining program is created does not match the coordinate system that is actually programmed or programmed by the automatic coordinate system setting. The function allows you to machine parts without changing the machining program by shifting from the measured coordinate system to the coordinate system that is assumed when the program is created. (In the figure below, amoun of shift from O' to O is specified as the workpiece coordinate system shift amount.)
In addition, workpieces can be machined using the same program even if their shapes are changed.



X' - Z' : Measured coordinate system
X - Z : Coordinate system established when the program is created

The validity of the functions depends on the MTB specifications.
This can be programmed via the screen. However, this section describes how to program them in a machining program.

Command format

G10 P0 X__ (U__) Z__ (W__); ... Input of workpiece coordinate system shift amount (no L command)

G10 L10 P0 X__ (U__) Z__ (W__); ... Input of workpiece coordinate system shift amount (L10)

X	X axis compensation amount (absolute)
U	X axis compensation amount (incremental)
Z	Z axis compensation amount (absolute)
W	Z axis compensation amount (incremental)

(1) The G10 command is unmodal. When executing the G10 command continuously, it must always be command for each block separately.

(2) X, Z and U, W can be programmed in one block when using G10. When an address that commands the same compensation input (X,U or Z, W) is commanded, the address which is input last is valid.

(Example) When "G10 P0 Z10. W50" is executed, the workpiece shift amount for Z axis is set to "50.0".

(3) If "G10 P0" is commanded when the workpiece coordinate system shift is invalid, it is input as an external workpiece offset.

(4) If "G10 L10 P0" is commanded when the workpiece coordinate system shift is invalid, the program number becomes illegal and a program error (P170) occurs.

(5) When the workpiece coordinate offset measurement specifications are effective, you can automatically set the measured workpiece shift amount, rather than manually input workpiece coordinate system shift amount. Refer to the Instruction Manual (IB-1501260) for details on the measuring method.

Relation with other functions

(1) If the external workpiece coordinate offset has been set, the amount of workpiece coordinate system shift from the external workpiece offset position is set.

(2) If parameter "#8716 Ext/Workpiece coordinate system shift share" is set to "1", the display of the external workpiece coordinate offset changes to the workpiece coordinate system shift amount. The validity of this parameter depends on the MTB specifications. (Parameter "#11056 Work shift invld")

20.8 Workpiece Coordinate System Preset ; G92.1

Function and purpose

This function presets the workpiece coordinate system shifted with the program command during manual operation to the workpiece coordinate system offset from the machine zero point by the workpiece coordinate offset amount by the program command G92.1 (G50.3).
The workpiece coordinate system, which is set when the following type of operation or program command is executed, will be shifted from the machine coordinate system.

- When manual interrupt is executed while manual absolute is OFF
- When movement command is issued in machine lock state
- When axis is moved with handle interrupt
- When operation is carried out with mirror image
- When local coordinate system is set with G52
- Shifting the workpiece coordinate system with G92

This function presets the shifted workpiece coordinate system to the workpiece coordinate system offset from the machine zero point by the workpiece coordinate offset amount. This takes place in the same manner as manual reference position return. Whether to preset the relative coordinate depends on the MTB specifications (parameter "#1228 aux12/bit6").

Command format

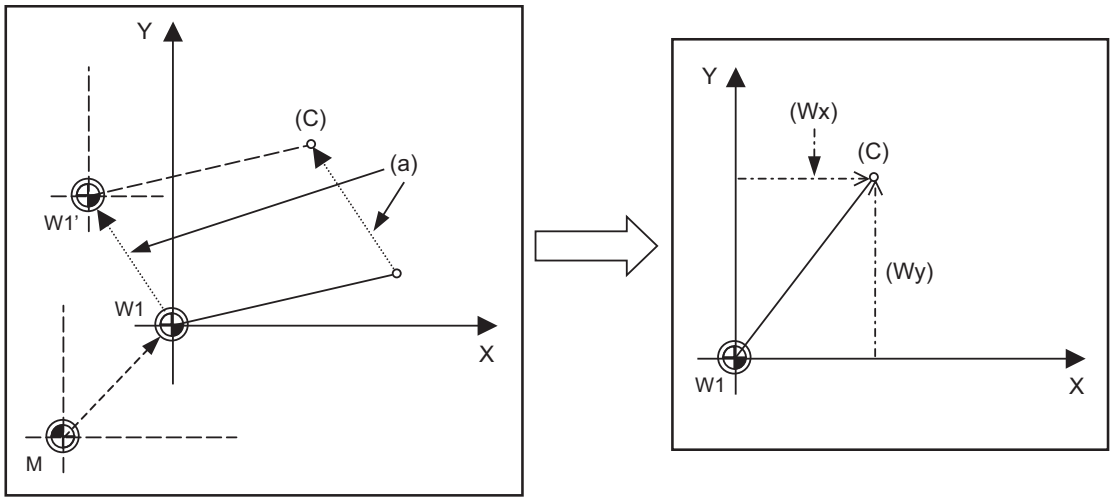
```
G92.1    X0. Y0. Z0. α0;
(G50.3)
```

α0	Additional axis

(1) Command the address of the axis to be preset. The axis will not be preset unless commanded.
(2) A program error (P35) will occur if a value other than "0" is commanded.
(3) Depending on the G code list, the G code will be "G50.3".
(4) Command G92.1 (50.3) in an independent block.
(5) Whether to conduct an error check when the coordinate system preset command (G92.1 or G50.3) is independently issued depends on the MTB specifications (parameter "#1242 set14/bit1").

Detailed description

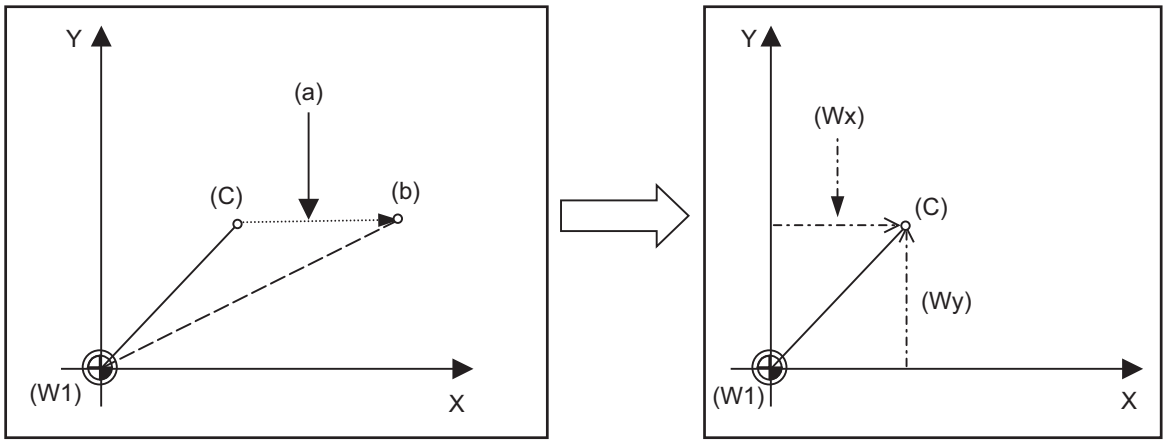
(1) When the manual operation is carried out when the manual absolute is set to OFF, or if the axis is moved with handle interrupt.



(a) Manual movement amount
(C) Current position
(Wx) Workpiece coordinate x after preset
(Wy) Workpiece coordinate y after preset

If manual operation is carried out when manual absolute is set to OFF, or if the axis is moved with handle interrupt, the workpiece coordinate system will be shifted by the manual movement amount.
This function returns the shifted workpiece coordinate zero point W1' to the original workpiece coordinate zero point W1, and sets the distance from W1 to the current position as the workpiece coordinate system's current position.

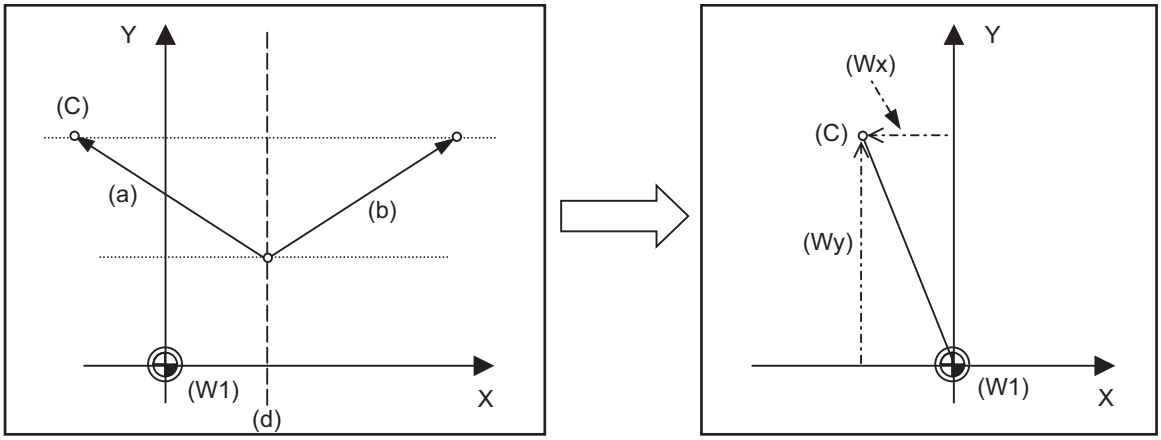
(2) When movement command is issued in machine lock state



(a) Movement amount during machine lock
(b) Workpiece coordinate system coordinate value
(Wx) Workpiece coordinate x after preset
(Wy) Workpiece coordinate y after preset
(C) Current position
(W1) Workpiece coordinate zero point

If the movement command is issued in the machine lock state, the current position will not move, and only the workpiece coordinates will move.
This function returns the moved workpiece coordinates to the original current position, and sets the distance from W1 to the current position as the workpiece coordinate system's current position.

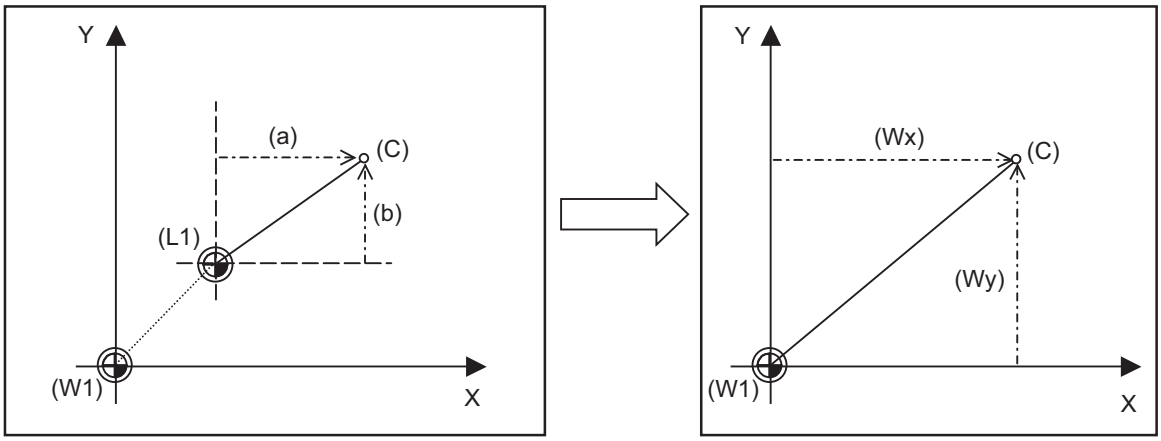
(3) When operation is carried out with mirror image



(a) Actual operation
(b) Program command
(C) Current position
(d) Mirror image center
(Wx) Workpiece coordinate x after preset
(Wy) Workpiece coordinate y after preset
(W1) Workpiece coordinate zero point

If operation is carried out with mirror image, only the NC internal coordinates are used as the program command coordinates. The other coordinates are the current position coordinates.
This function sets the NC internal coordinates as the current position coordinates.

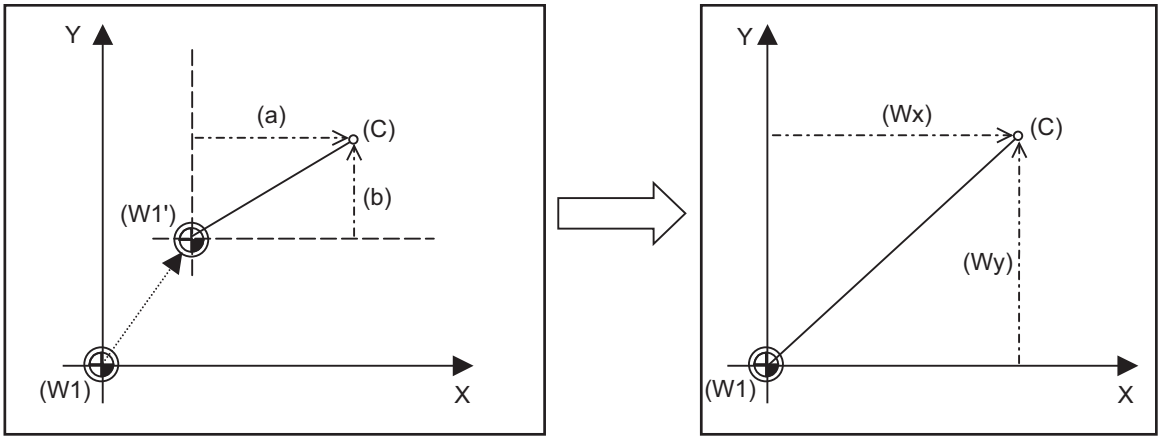
(4) Setting local coordinate system with G52



(a) Local coordinates x
(b) Local coordinates y
(Wx) Workpiece coordinate x after preset
(Wy) Workpiece coordinate y after preset
(C) Current position
(L1) Local coordinate zero point
(W1) Workpiece coordinate zero point

The local coordinate system is set with the G52 command, and the program commands, etc., are issued with the local coordinate system.
With this function, the set local coordinate system is canceled, and the program commands, etc., use the workpiece coordinate system which has W1 as the zero point. The canceled local coordinate system is only the selected workpiece coordinate system.

(5) Shifting the workpiece coordinate system with G92



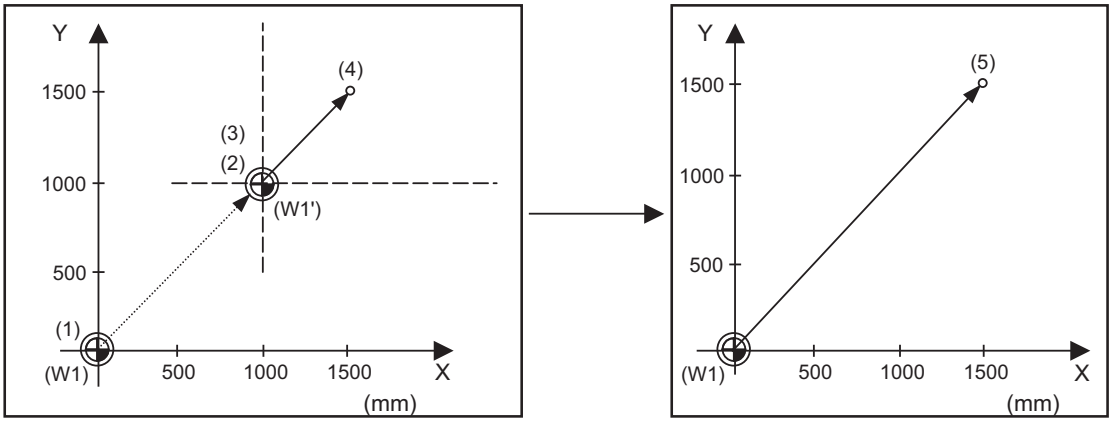
(a) Local coordinates x
(b) Local coordinates y
(Wx) Workpiece coordinate x after preset
(Wy) Workpiece coordinate y after preset
(C) Current position
(W1) Workpiece coordinate zero point
(W1') Workpiece zero point after G92 command

The workpiece coordinate system shifts with the G92 command, and the distance between W1' and the current position is set as the current position of the workpiece coordinate system.

This function returns the shifted workpiece coordinate zero point to W1, and sets the distance from W1 to the current position as the workpiece coordinate system's present position. This is valid for all workpiece coordinate systems.

Program example

The workpiece coordinate system shifted with G92 is preset with G92.1.



(W1) Workpiece coordinate zero point
(W1') Workpiece zero point after G92 command

(Example)

```
G28 X0 Y0 ;          ... (1)
G00 G90 X1. Y1. ;    ... (2)
G92 X0 Y0            ... (3)
G00 X500 Y500 ;      ... (4)
G92.1 X0 Y0 ;        ... (5)
```

Relationship with other functions

Tool No./Tool Compensation No. (T Code)/tool length compensation

If the error check is enabled when the workpiece coordinate system preset is independently commanded (*1), command all the tool compensation axes when commanding "G92.1" during the tool compensation. When commanding "G92.1" during the tool length compensation, designate the tool length compensation axis.
If those axes are not commanded, a program error (P29) will occur.
(*1) The setting depends on the MTB specifications (parameter "#1242 set14/bit1").

When the parameter "#1100 Tmove" is set to "1" or "2", it is assumed that the tool compensation is in process until the movement command is issued to the tool compensation axis after T0101 (tool compensation execution) and T0000 (tool compensation cancellation) are commanded.

Tool nose radius compensation / Tool radius compensation

Cancel the tool nose radius compensation or the tool radius compensation, and command the workpiece coordinate system preset (G92.1). When the workpiece coordinate system preset (G92.1) is commanded during the tool nose radius compensation or the tool radius compensation, a program error (P29) will occur if none of the tool compensation axes are commanded.

Other G code commands

If the workpiece coordinate system preset (G92.1) is commanded during the modal shown below, a program error (P34) will occur.

(1) Milling interpolation

(2) Coordinate rotation by program

Precautions

(1) Cancel tool length compensation, tool nose radius compensation, and tool length compensation when using this function. If this function is executed without canceling them, the workpiece coordinates will be at a position obtained by subtracting the workpiece coordinate offset amount from the machine value. Thus, the compensation vector will be temporarily canceled.

(2) This function cannot be executed while the program is being resumed.

20.9 Coordinate Rotation by Program ; G68.1/G69.1

Function and purpose

When machining a complicated shape located in a rotated position in respect to the coordinate system, this function enables to machine the rotated shape with the program for the shape before rotation on the local coordinate system and with the rotation angle designated by the program coordinate rotation command.
This function rotates the coordinate system and realizes mainly drilling and tapping.

Command format

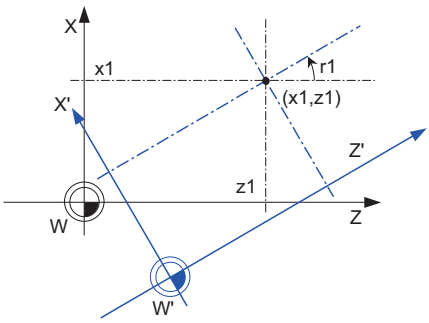
G68.1 X__ Z__ R__ ; ... Coordinate rotation ON

X,Z	Coordinate values of the rotation center An axis corresponding to the plane selected from the rotation center coordinates X, Y and Z.
R	Rotation angle Designate the angle from -360° to 360° in least input increments. The counterclockwise direction on the selected plane is + direction.

G69.1 ; ... Coordinate rotation cancel

Select the command plane with G17 to G19.

Command) G68.1 Xx1 Zz1 Rr1 ;



(1) Command the rotation center coordinates (x1, z1) with an absolute value.

(2) Rotate the coordinate counterclockwise by the angle designated in rotation angle r1.

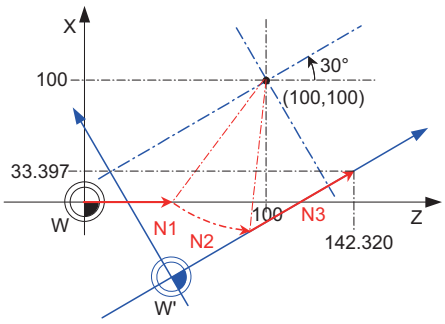
(3) When the minimum setting unit for r1 is 0.001deg, the setting range of rotation angle is -360.000 to 360.000.
When a value out of the range is commanded, a remainder of the value divided by 360° is commanded.
(Ex.) When 400 is commanded, the remainder of 400 minus 360, which is 40, is the commanded angle.

(4) The counter is indicated as the point on the coordinate system prior to rotation.

W: Local coordinate system before rotation
W' : Rotated local coordinate system
r1: Rotation angle
(x1, z1) Rotation center

The following is the example of relationship of program command position and the displayed position.

Program example) N1 G00 Z50.N2 G68.1
X100. Z100. R30.;
N3 G00 Z120. ;



(1) The program command performs positioning on the local coordinates after the rotation.

(2) The counter display shows the point after the coordinate rotation on the coordinate system before rotation. In this example, the position display when the N3 block is finished is:
X 33.397
Z 142.320

(3) G68.1 command does not carry out the actual movement. Therefore, in this example, it moves linearly from the end point of N1 to the end point of N3.

W: Local coordinate system before rotation

W' : Rotated local coordinate system

Detailed description

(1) G68.1 and G69.1 are the G code of group 16.

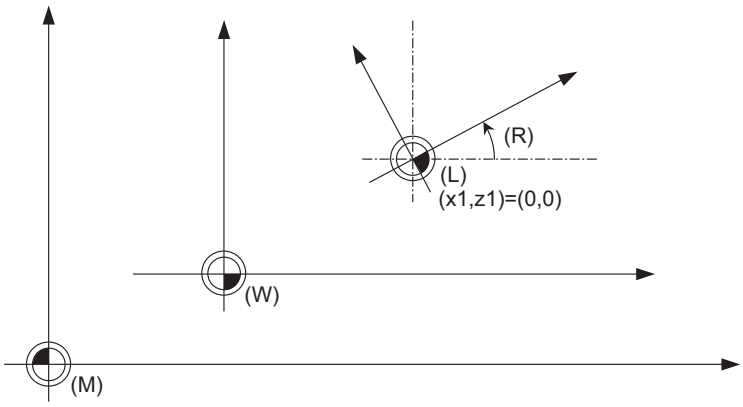
(2) Command the rotation center coordinate (x1, z1) with an absolute value. Even if commanded with an incremental address, it will not be handled as an incremental value.

(3) If the rotation center coordinates (x1, z1) are omitted, the position where the G68 command was executed will be the rotation center.

(4) The rotation angle R is commanded with an absolute value. However, it can be commanded with an incremental value if the parameter "#8082 G68.1 R INC" is set.

(5) The performance when the rotation angle R is omitted depends on the setting of parameter "#1270 ext06/bit5":
0: Use the previously commanded value (modal value).
1: Use the set value in "#8081 Gcode Rotat".
If the coordinate rotation mode is canceled by G69.1 command, the modal value will be cleared. If G68.1 is commanded after G69.1 was commanded, the rotation angle becomes 0° by omitting R.
The setting value of the parameter is an absolute value regardless of the setting of the parameter "#8082 G68.1 R INC".

(6)The program coordinate rotation is a function used on the local coordinate system. The relation of the rotated coordinate system, workpiece coordinate system and basic machine coordinate system is shown below.



(R) Rotation angle
(L) Local coordinate system
(W) Workpiece coordinate system
(M) Basic machine coordinate system

(7)The coordinate rotation command during coordinate rotation is processed as the changes of center coordinates and rotation angle.

(8) If commanding G68.1 without the coordinate rotation specification, a program error (P260) will occur. If commanding G69.1 without the coordinate rotation specification, a program error (P39) will occur.

(9) Program error (P111) will occur if the plane selection code is commanded during the coordinate rotation mode. The arc command (G02, G03) can not be executed with G17 plane (XY plane) designated during coordinate rotation in the G18 plane (ZX plane).

(10) The program coordinate rotation function is valid only in the automatic operation mode.

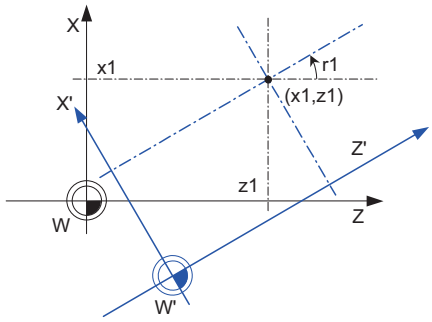
(11) G68.1 is displayed on the modal information screen during the coordinate rotation mode. When the mode is canceled, the display changes to G69.1. (The modal value is not displayed for the rotation angle command R.)

Coordinate rotation command during coordinate rotation

The coordinate rotation command during coordinate rotation is processed as the changes of center coordinates and rotation angle.

(1) For absolute command
Command) G68.1 Xx1 Zz1 Rr1 ;
G68.1 Xx2 Zz2 Rr2 ;

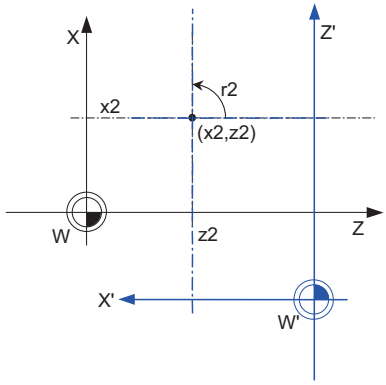
1) G68.1 Xx1 Zz1 Rr1 ;



With spinning around on the center coordinate of the rotation (x1, z1), the rotation takes place in the counterclockwise direction by the angle designated in rotation angle r1.

W: Local coordinate system before rotation
W': Local coordinate system after the rotation
r1: Rotation angle
(x1, z1) Rotation center

2) G68.1 Xx2 Zz2 Rr2 ;



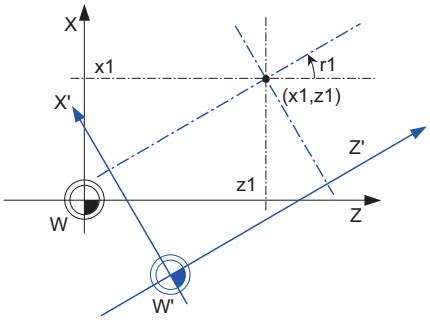
The center coordinate of the rotation switches from (x1, z1) to (x2, z2), and the rotation angle is cleared once. Then the rotation takes place in the counterclockwise direction by the angle designated with r2

W: Local coordinate system before rotation
W': Local coordinate system after the rotation
r2 : Rotation angle
(x2, z2) Rotation center

(2) For incremental command

Command) G68.1 Xx1 Zz1 Rr1 ;
G68.1 Ux2 Uz2 Rr2 ;

1) G68.1 Xx1 Zz1 Rr1 ;



With spinning around on the center coordinate of the rotation (x1, z1), the rotation takes place in the counterclockwise direction by the angle designated in rotation angle r1.

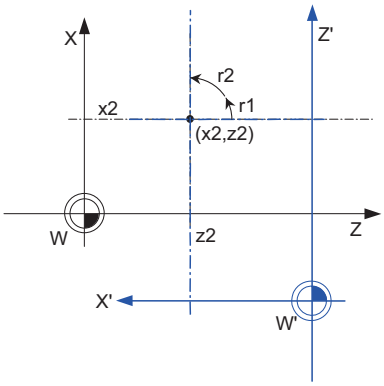
W: Local coordinate system before rotation

W': Local coordinate system after the rotation

r1: Rotation angle

(x1, z1) Rotation center

2) G68.1 Ux2 Uz2 Rr2 ;



The center coordinate of the rotation switches from (x1, z1) to (x2, z2).
Even if the rotation center coordinate command is the incremental value command, it is handled as the absolute value.

The rotation takes place in the counterclockwise direction by the angle rotated at r1 and another angle commanded at r2.

W: Local coordinate system before rotation

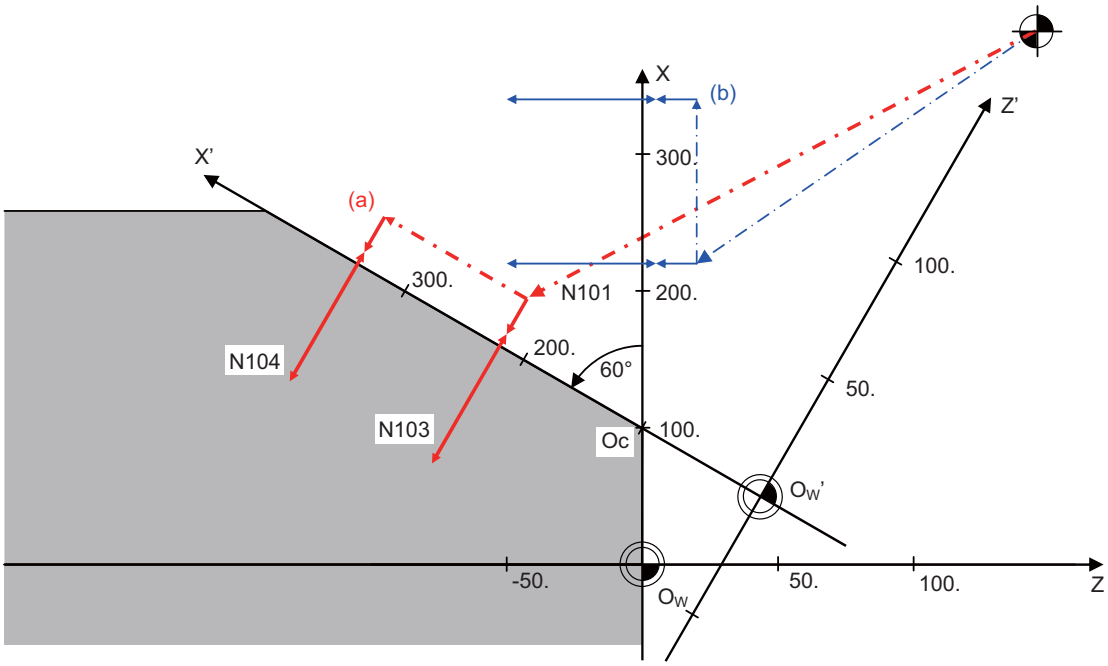
W': Local coordinate system after the rotation

r1,r2 : Rotation angle

(x2, z2) Rotation center

Program example

Program coordinate rotation by absolute command



(Oc) Rotation center

(Ow) Workpiece coordinate zero point before rotation (Ow') Workpiece coordinate zero point after rotation

(a) Subprogram path after rotation (b) Subprogram path before rotation

Main program

```
N01 G97 G18;                    Z-X Plane selection
N02 G91 G28 X0. Z0.;
N03 G54;
N04 G90 T1010;
N05 G68.1 X100. Z0 R60.;        Coordinate rotation ON
N06 M98 H101;                   Subprogram execution
N07 G69.1;                      Coordinate rotation cancel
N08 M02;                        End
```

Subprogram (Shape programmed with coordinate system before rotation)

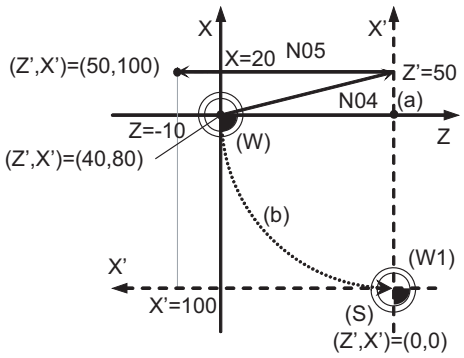
```
N101 G00 X220. Z20.;
N102 G94 S2=1000 M3;            2nd spindle (tool spindle) forward
N103 G98 G83 Z-50. R-15. Q-10.
F100;
N104 X340. ;
N105 G80;
N106 S2=0 M5;                   2nd spindle stop
N107 M99;
```

Operation when only one axis was commanded by the first movement command

Command basically two axes in the rotation plane by an absolute value immediately after the coordinate rotation command.
When commanding one axis only, the following two kinds of operations can be selected by the parameter "#19003 PRG coord rot type".

(1) When "#19003 PRG coord rot type" is "1", the operation is the same as when "N04" is "X50.Z0.". The end point is calculated on the assumption that the start point rotates along with the coordinates' rotation.

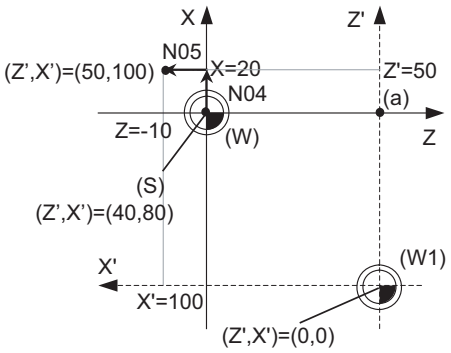
```
N01 G18 G91 G28 X0.
Z0.;
N02 G90 G92 G53 X0.
Z0.;
N03 G68.1 X0. Z40. R90.;  Coordinate rotation ON
N04 Z50.;
N05 X100.;
N06 G69.1;                Coordinate rotation cancel
N07 M02;                  End
```



→ Machine movement path
(S) Start point
(a) Center of rotation
(b) The start point is rotated virtually
(W) Local coordinate system before rotation
(W') Local coordinate system after rotation

(2) When "#19003 PRG coord rot type" is "0", only axis commanded in N04 (Z' Axis) is moved. The start point does not rotate along with the coordinate rotation; therefore the end position is calculated based on the current position on local coordinate system before rotation.

```
N01 G18 G91 G28 X0.
Z0.;
N02 G90 G92 G53 G0 X0.
Z0.;
N03 G68.1 X0. Z40. R90.;  Coordinate rotation ON
N04 Z50.;
N05 X100.;
N06 G69.1;                Coordinate rotation cancel
N07 M02;                  End
```



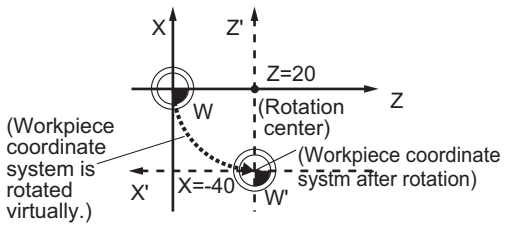
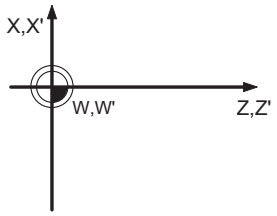
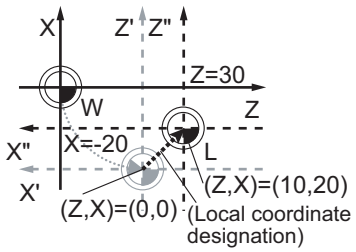
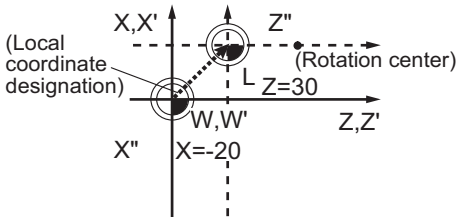
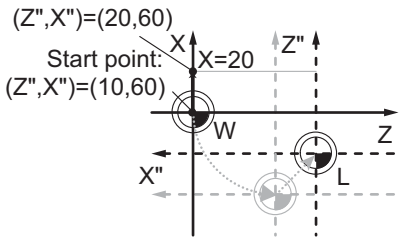
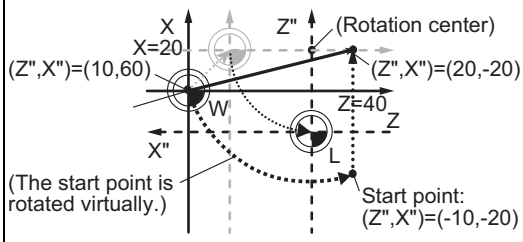
→ Machine movement path
(S) Start point
(a) Center of rotation
(W) Local coordinate system before rotation
(W') Local coordinate system after rotation

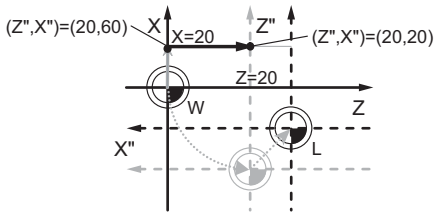
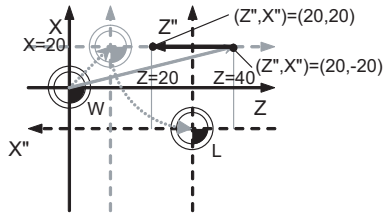
Local coordinate designation during program coordinate rotation

(1) When "#19003 PRG coord rot type" is "0", the position commanded on the rotated coordinate system is set as the local coordinate zero point.

(2) When "#19003 PRG coord rot type" is "1", the position commanded on the coordinate system before it is rotated, is set as the local coordinate zero point and the local coordinate will be rotated.

```
N01 G18 G91 G28 X0. Z0.;
N02 G90 G92 G53 G0 X0. Z0.;
N03 G68.1 X0. Z20. R90.;        Coordinate rotation ON
N04 G52 X20. Z10.;              Local coordinate setting
N05 Z20.;
N06 X20.;
N07 G69.1 ;                     Coordinate rotation cancel      W: Workpiece coordinate system
N08 M02 ;                       End                             L : Local coordinate system
```

	(1) Operation of #19003 = 0	(2) Operation of #19003 = 1
N03		
	Workpiece coordinate system is rotated virtually.	Workpiece coordinate system is not rotated.
N04		
	The workpiece coordinate zero point after rotation is considered as (Z,X)=(0,0). The position after shifted by 10 in the X and 20 in the Y direction is set as the local coordinate zero point. The direction of the shift is not the direction of Z' and X'.	Designate the local coordinate system on the workpiece coordinate system.
N05		
	The commanded axis moves on the rotation coordinate system. Axis without movement command does not move.	The commanded axis moves on the rotation coordinate system. Axis without movement command moves to the position on rotation coordinate system.

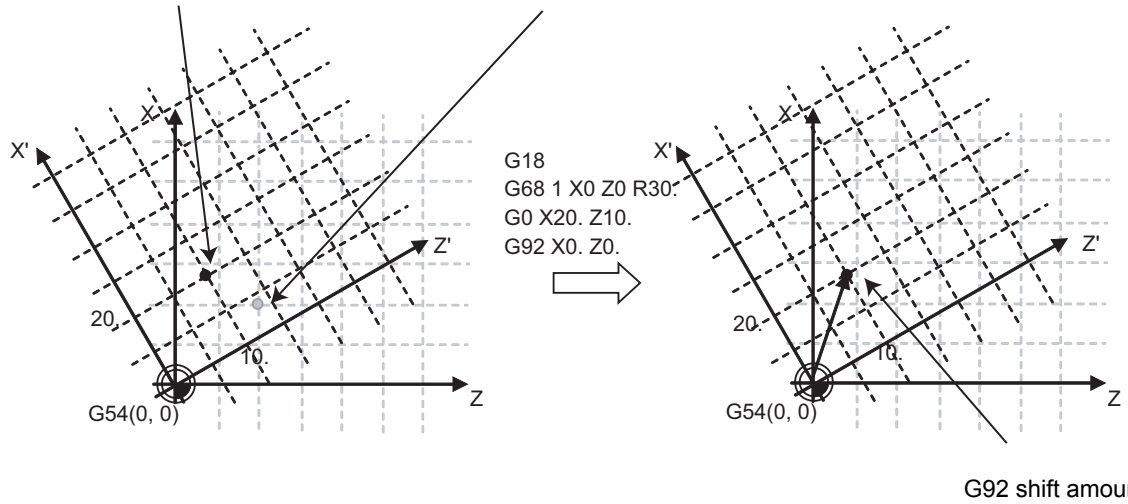
N06	 (Z",X")=(20,60) X X=20 Z" (Z",X")=(20,20) Z=20 W Z X" L	 X X=20 Z" (Z",X")=(20,20) (Z",X")=(20,-20) Z=20 Z=40 W Z X" L

Coordinate system designation during program coordinate rotation

When the coordinate system setting (G92) is executed during program coordinate rotation (G68.1), this program operates same as "Local coordinate designation during program coordinate rotation".

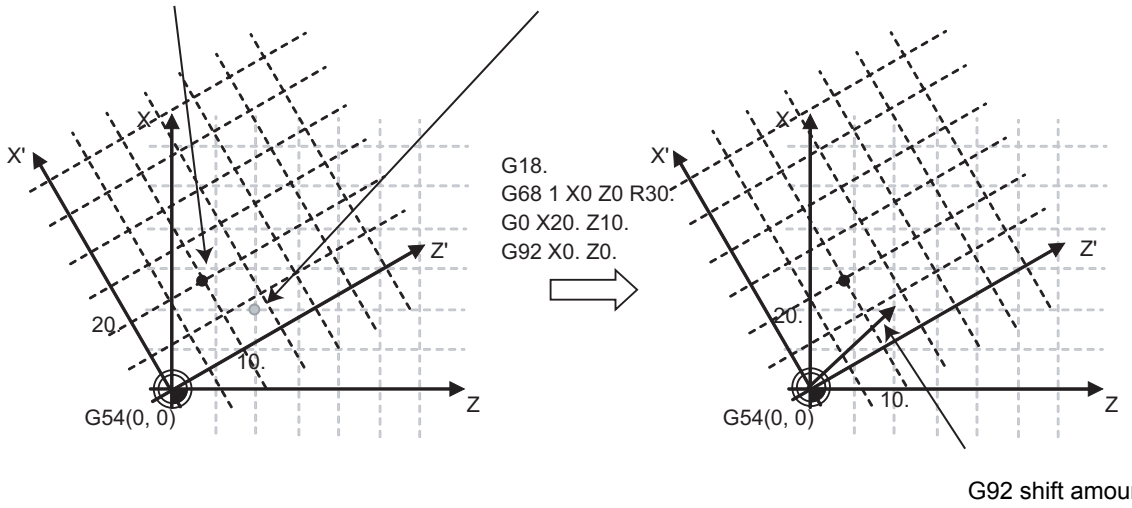
(1) When "#19003 PRG coord rot type" is "0", the position is preset to the current position commanded on the rotated coordinate system.
(Ex.) Setting on the coordinate system (Z' -X') after rotation

Position after rotation Commanded position



(2) When "#19003 PRG coord rot type" is "1", the position is preset to the current position commanded on the coordinate system before rotation. The coordinate system is rotated after the position is commanded.
(Ex.) Setting on the coordinate system (Z-X) after rotation

Position after rotation Commanded position



Note

(1) When "#19003 PRG coord rot type" is "1"and the coordinate system setting (G92) is executed during coordinate rotation mode, the rotation center of the program coordinate rotation is not shifted.
(It stays at the same position in respect to the basic machine coordinate system.)

Operation when circular interpolation is commanded after commanding rotation

Basically, command two axes in the rotation plane by positioning/linear interpolation with an absolute value immediately after the coordinate rotation command.

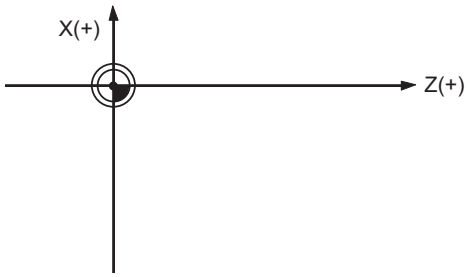
When commanding the circular interpolation, command the movement by positioning/linear interpolation to the start point even if the axis does not move.

When the circular interpolation is commanded straight after the coordinate rotation command, the following operation takes place depending on the setting of the parameter "#19003 PRG coord rot type".

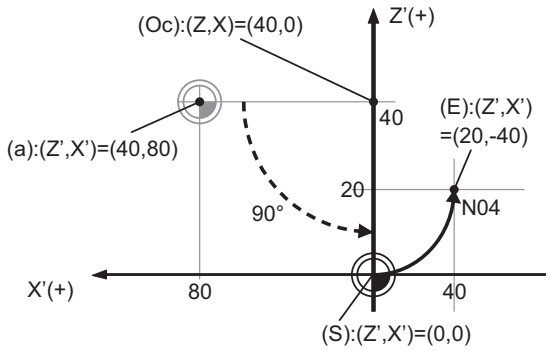
(1) When "#19003 PRG coord rot type" is "1", the start point position which was assumed to be rotated with the coordinate rotation and the actual axis position are positioned in a different position in respect to the basic machine coordinate system. Consequently, the axis cannot move along with the circular path between the start point and the end point, and a program error (P70) will occur.

```
N01 G18 G91 G28 X0. Z0.;
N02 G90 G92 G53 G0 X0. Z0.;
N03 G68.1 X0. Z40. R90.;    Coordinate rotation ON
N04 G03 X-40. Z20. R20. F500;
N05 G00 X50.;
N06 G69.1 ;                 Coordinate rotation cancel
N07 M02 ;                   End
```

[Local coordinate system before rotation]



[Local coordinate system after rotation]

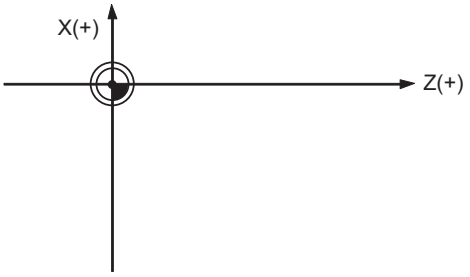


(a) Actual axis position
(Oc) Rotation center
(S) Start point (*1)
(E) End point
(*1) Start point which was assumed to be rotated with the coordinate rotation

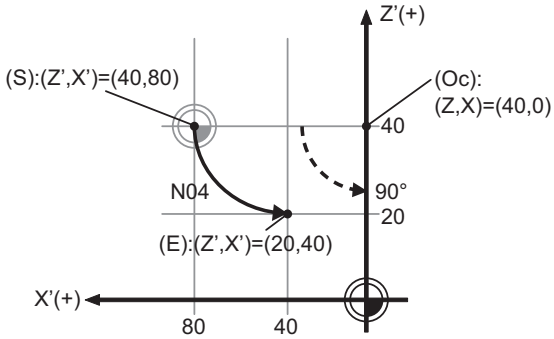
(2) When "#19003 PRG coord rot type" is "0", the circular interpolation start point does not rotate with the coordinate rotation. It remains in the same position before the coordinate rotation in respect to the basic machine coordinate system. Consequently, the axis operates the circular interpolation from this start point to the end point.

N 01 G18 G91 G28 X0. Z0.;	
N 02 G90 G92 G53 G0 X0. Z0.;	
N 03 G68.1 X0. Z40. R90.;	Coordinate rotation ON
N 04 G03 X40. Z20. R20. F500;	
N 05 G00 X50.;	
N06 G69.1 ;	Coordinate rotation cancel
N07 M02 ;	End

[Local coordinate system before rotation]



[Local coordinate system after rotation]



(a) Actual axis position (Oc) Rotation center
(S) Start point (E) End point

Relationship with Other Functions

(1) The tool compensation during the coordinate rotation mode is carried out in the local coordinate system after the coordinate rotation.

(2) During the coordinate rotation mode, the mirror image performs reverse processing of the point after the coordinate rotation is inverted in the coordinate system before the coordinate rotation.

(3) For all position display, points after the coordinate rotation are displayed in the coordinate system before the rotation.

(4) Similarly for the system variable of the coordinate value, points after the coordinate rotation are displayed in the coordinate system before the rotation.

(5) The coordinates can also be rotated for the parallel axis. Select the plane that contains the parallel axis before issuing the G68.1 command. (The plane which contains the parallel axis cannot be selected in the same block as the G68.1 command.)

(6) Program error (P485) will occur if pole coordinate interpolation/milling interpolation is commanded during the coordinate rotation mode.

(7) Program error (P481) will occur if the coordinate rotation is commanded during the polar interpolation/milling interpolation mode.

(8) Program error (P485) will occur if cylindrical interpolation is commanded during the coordinate rotation mode.

(9) Program error (P481) will occur if coordinate rotation is commanded during the cylindrical interpolation mode.

(10) Program error (P34) will occur if the workpiece coordinate system preset (G92.1) is commanded during the coordinate rotation mode.

(11) A system variable can read the skip coordinate value when the skip command is issued during the coordinate rotation. The system variable reads the workpiece coordinate value after the coordinate conversion.
When the multiple axes move for one axis movement command, the skip coordinate value is read by the multiple axes.

(12) Program error (P261) will occur if the coordinate rotation and other G code are commanded in the same block. However, the type of program error differs depending on the combination of the coordinate rotation command and the G code listed below.

G code commanded with the coordinate rotation command in the same block	
Control axis synchronization between part systems start/end Control axis superimposition start/end	Program error (P33)
Mixed control (Cross axis control)	Program error (P503)

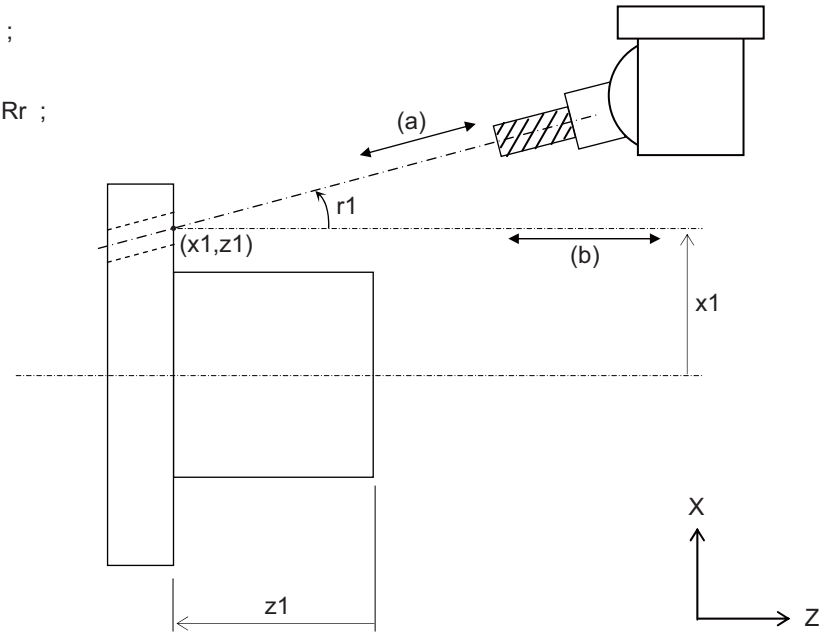
(13) Program error (P34) will occur if the following functions are commanded during coordinate rotation:
- Mixed control (Cross axis control)
- Axis name switch
- Control axis synchronization between part systems start/end
- Control axis superimposition start/end
- Thread cutting
- Variable lead thread cutting
- Circular thread cutting
- Fixed cycle for turning machining
- Compound type fixed cycle for turning machining
- Multi-part system simultaneous thread cutting cycle
- User macro modal call B
- Mirror image for facing tool posts
- Balance cut

(14) Program error (P111) will occur if the following functions are commanded during coordinate rotation:
- Plane selection (X-Y, Z-X, Y-Z)
- Milling interpolation plane selection Y-Z cylindrical plane

(15) Program error (P262) will occur if the coordinate rotation is commanded during the following G code modal:
- Thread cutting
- Variable lead thread cutting
- Circular thread cutting
- Fixed cycle for turning machining
- Compound type fixed cycle for turning machining
- Multi-part system simultaneous thread cutting cycle
- Fixed cycle for drilling
- User macro
- User macro modal call A, B
- Mirror image for facing tool posts
- Balance cut

(16) Tapping cycle can be carried out during the coordinate rotation mode. This enables the tapping diagonally. Refer to "Face tapping cycle (longitudinal tapping cycle);G84(G88)" for details on tapping cycle.

```
G68.1 Xx1 Zz1 Rr1 ;
      •
      •
G84 Zz2 Rr2 Ff Ss ,Rr ;
      •
      •
```



(a): Actual movement direction
(b): Movement direction with program command
r1: Rotation angle
(x1, z1): Rotation center

(16-1) Feedrate/pitch command (F command)
- The F command value specified with the machining program is as follows:
  Asynchronous tap: Feedrate toward the tap cutting direction (diagonally)
  Synchronous tap: Pitch toward the tap cutting direction (diagonally)

(16-2) Programmable in-position check
- In-position check takes place on two axes as two axes move during tapping diagonally.
- Each axis is checked, and the in-position check is complete when both of two axes come into the commanded in-position width.

(16-3) Tapping retract
- Tapping return can be carried out by the tapping return signal (1st part system: YC5C, 2nd part system: YD9C) for diagonal cutting.
- Tap cutting axes (two axes moved during tap cutting) move toward the initial point.
- For multiple spindle control II ("#1300 ext36/bit0" is set to "1"), select the spindle when the tapping cycle is interrupted before turning ON the tapping return signal.
  The error "M01 operation error 1032" will occur if the tapping return is carried out while a different spindle is being selected.

(16-4) Servo gain during the synchronous tapping cycle

- When diagonal synchronous tapping is carried out, the setting value of "#2017 tap_g" is the servo gain of two axes which move during tap cutting.

(16-5) Specify boring axis (#1080 Dril_Z)

- When "#1080 Dril_Z" is set to "1", the tapping cycle during coordinate rotation mode can be carried out only when G17 plane is being selected.
  Program error (P111) will occur if the tapping cycle is commanded while G18 or G19 is being selected.

  <Note>

  - "#1080 Dril_Z" is valid when MITSUBISHI CNC special format ("#1265 ext01/bit2" is set to "1") is applied.

(16-6) Precautions for synchronous tapping cycle

- Set the same value for the servo gain ("#2017 tap_g") of two axes which move during tap cutting.
- Do not issue a command which moves the inclined axis during tap cutting.

(17) Program error (P612) will occur when the exponential interpolation is commanded during program coordinate rotation.

Precautions

(1) Command G68.1 in an independent block. Program error (P261) will occur if not commanded in independent blocks.

(2) If an axis which is not in the selected plane is commanded to the coordinate value of the rotation center, axes in other than the selected plane moves to the position which is specified by the last G01 modal.

(3) Always command an absolute value for the movement command immediately after G68.1 and G69.1.
If an increment value is commanded, it may not move to the intended position.
Also command it together with the axis address on the selected plane (for G18 plane, Z-X).
If it is omitted, that axis is handled as "no movement command".

(4) If the manual absolute is ON and interrupted the coordinate rotary axis, then, do not use automatic operation for the following absolute value command.

(5) The intermediate point during reference position return is the position after the coordinates are rotated.

(6) If the workpiece coordinate system offset amount is changed during the coordinate rotation mode, the rotation center for the program coordinate rotation will be shifted. (The center will follow the coordinate system.)

(7) If workpiece coordinate system setting (G92) is executed during the coordinate rotation mode, the rotation center for program coordinate rotation will not shift. (The same position in respect to the basic machine coordinate system)

(8) If the workpiece coordinates are changed during the coordinate rotation mode (ex. from G54 to G55), the rotation center of the program coordinate rotation will be the position on the coordinate system which the command was issued. (The same position in respect to the basic machine coordinate system)

(9) If coordinate rotation is executed to the G00 command for only one axis during the coordinate rotation mode, two axes will move. If the parameter "#1086 G0Intp" is set to "1", the interpolation is carried out.

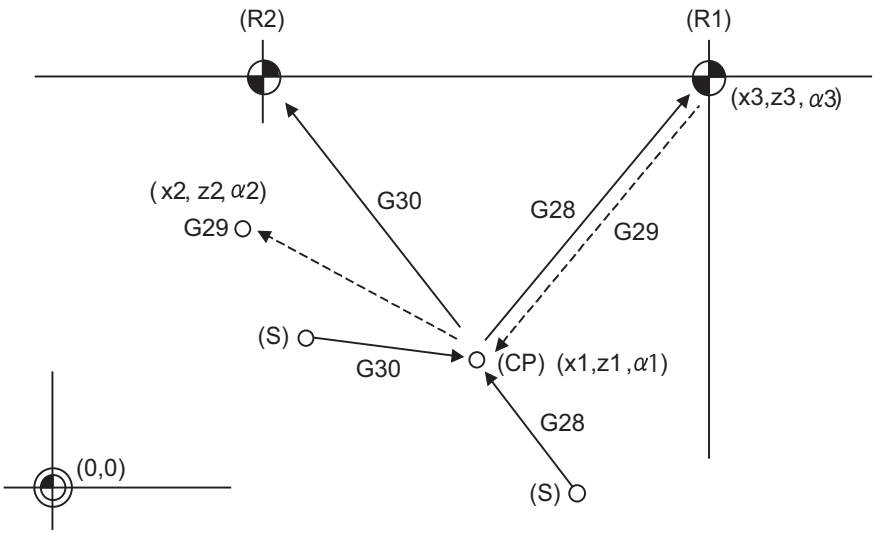
(10) If a synchronous tapping is commanded during the coordinate rotation mode in inclined axis control, a normal synchronous tapping is applied even if the high-speed synchronous tapping function is enabled. (The availability of the high-speed synchronous tapping function depends on the MTB specifications (parameter "#1281 ext17/bit5").

(11) During the coordinate rotation by program and machine lock on a axis of vertical/horizontal axes of selected plane, normal synchronous tapping is applied even if high-speed synchronous tapping function is enabled. (The availability of the high-speed synchronous tapping function and coordinate system rotation depend on the MTB specifications (parameter "#1281 ext17/bit5").

20.10 Reference Position (Zero Point) Return ; G28,G29

Function and purpose

After the commanded axes have been positioned by G0, they are returned respectively at rapid traverse to the first reference position when G28 is commanded.
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.



(R1) 1st reference position
(R2) 2nd reference position
(S) Start point
(CP) Intermediate point

Command format

G28 Xx1 Zz1 αα1; ... Automatic reference position return

X, Z, α	Coordinate value of the intermediate point (α is an additional axis)

G29 Xx2 Zz2 αα2; ... Start point return

X, Z, α	Coordinate value of the end point (α is an additional axis)

Detailed description

(1)The G28 command is equivalent to the following:
G00 Xx1 Zz1 αα1 ;G00 Xx3 Zz3 αα3 ;
where x3, z3 and α3 are the coordinate values of the reference position which are set in parameters "#2037 G53ofs" for the distance from the basic machine coordinate system zero point as specified by the MTB.

(2)After the power has been switched on, the axes which have not been subject to manual reference position return 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 position which was stored at the first time.

(3)When reference position return is completed, the zero point arrival output signal is output and also #1 appears at the axis name line on the setting and display unit screen.

(4)The G29 command is equivalent to the following:
G00 Xx1 Zz1 αα1 ;
G00 Xx2 Zz2 αα2 ;
The rapid traverse (non-interpolation type) independent for each axis takes place.
In this case, x1, z1 and α1 are the coordinate value of the G28 or G30 intermediate point.

(5)Program error (P430) occurs when G29 is executed without executing automatic reference position (zero point) return (G28) after the power has been turned ON.

(6)The intermediate point coordinate values (x1, z1, α1) 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 position return unless it is already canceled, and the intermediate point will be the compensated position.

(9)The intermediate point can be ignored by parameter "#1091 Ignore middle point" setting.

(10)Control from the intermediate point to the reference position is ignored for reference position return in the machine lock status. When the designated axis reaches as far as the intermediate point, the next block will be executed.

(11)Mirror image is valid from the start point to the intermediate point during reference position 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 position and the tool will move to the reference position.

(12)Do not issue T commands in a block where G29 exits. Program error (P29) occurs if the workpiece compensation amounts differ between the current and the previous blocks.

(13)When G28/G29/G30 is commanded in single block mode, if "#1279 ext15/bit6 Enable single block stop at middle point" is set to "1", single block stop at middle point will be performed; single block stop at middle point will not be performed if set to "0".

(14)If the mode is switched to MDI mode or reference position return mode while in a single block stop at the interim position, an operation error (M01 0013) occurs.

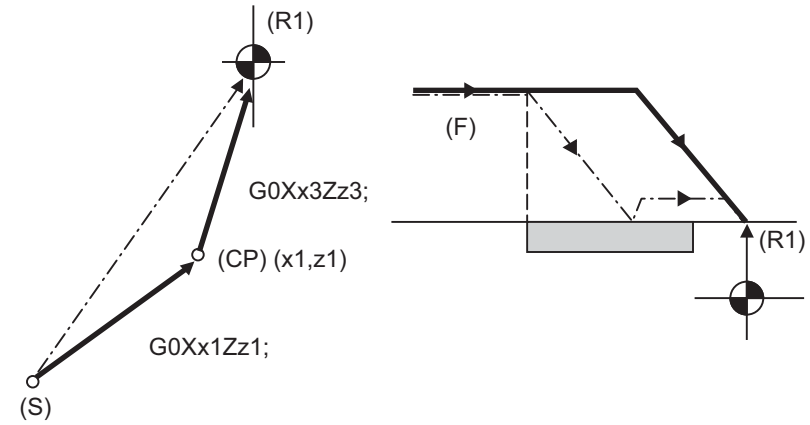
(15)If the NC is reset while in a single block stop at middle point, the interim position for G29 start position return will not be updated.

(16)If a miscellaneous function is commanded in the same block as G28/G29, the miscellaneous function completion waiting point will be the end of commanded movement, instead of the interim position.

(17) If the PLC interrupt operation is operated while in a single block stop at the interim position, an operation error (M01 0129) occurs.

Program example

(Example 1) G28 Xx1 Zz1 ;



1st operation after power has been turned ON

2nd and subsequent operations

Near-point dog

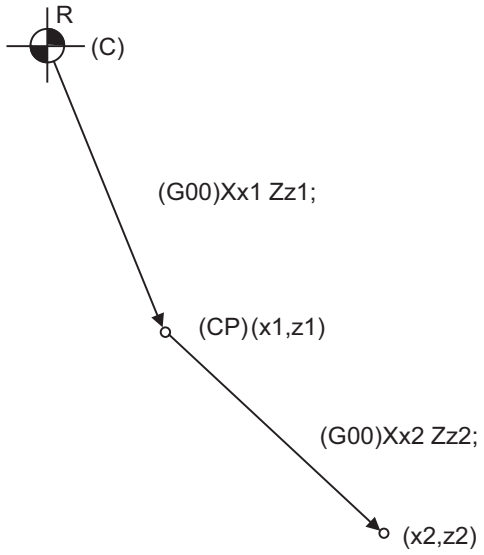
(S) Return start position

(CP) Interim point

(R1) Reference position (#1)

(F) Rapid traverse rate

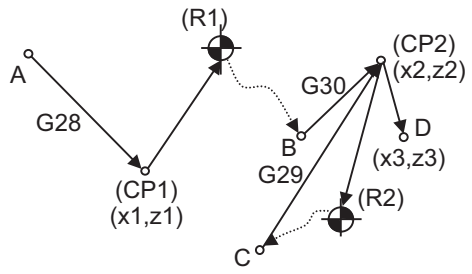
(Example 2) G29 Xx2, Zz2 ;



(C) Current position

(CP) G28, G30 Intermediate point

(Example 3) G28 Xx1 Zz1 ;
: (From point A to 1st reference position)
:
G30 Xx2 Zz2 ;
: (From point B to 2nd reference position)
:
G29 Xx3 Zz3 ;
(From point C to point D)

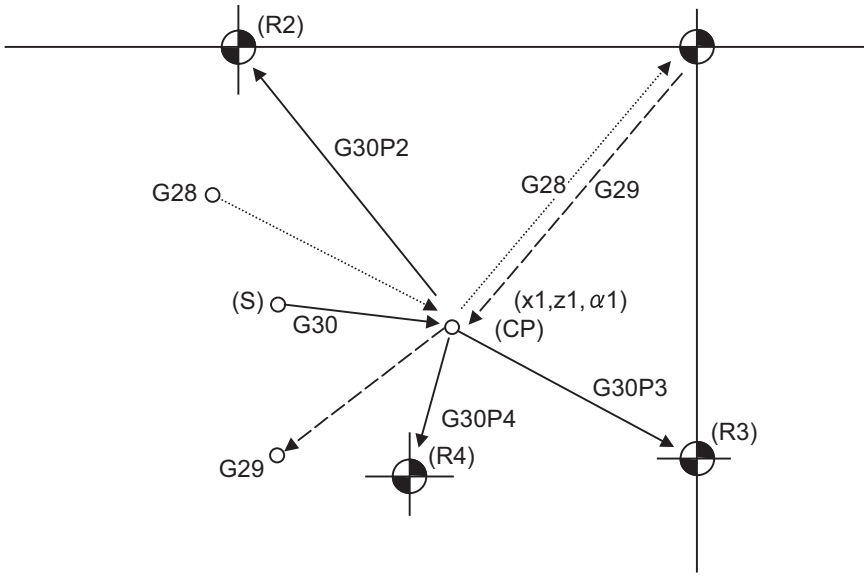


(CP1) Old intermediate point
(R1) Reference position (#1)
(CP2) New intermediate point
(R2) 2nd reference position (#2)

20.11 2nd, 3rd, and 4th Reference Position (Zero Point) Return ; G30

Function and purpose

The tool can return to the second, third, or fourth reference position by specifying G30 P2 (P3 or P4).



(S) Start point
(R2) 2nd reference position
(R4) 4th reference position
(CP) Intermediate point
(R3) 3rd reference position

Command format

G30 P2(P3,P4)Xx1 Zz1 αα1;

X, Y, Z, α	Coordinate value of the intermediate point (α is an additional axis)
P	Reference position No. P2: 2nd reference position return P3: 3rd reference position return P4: 4th reference position return

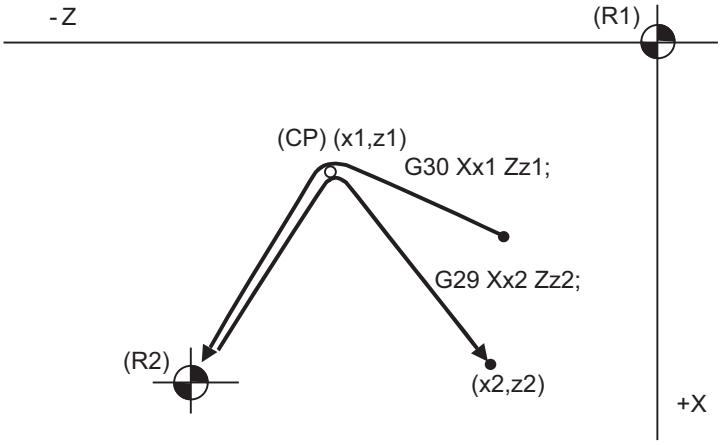
Detailed description

(1)The 2nd, 3rd, or 4th reference position return is specified by P2, P3, or P4.
A command without P or with other designation method will return the tool to the 2nd reference position.

(2) In the 2nd, 3rd, or 4th reference position return mode, as in the 1st reference position return mode, the tool returns to the 2nd, 3rd, or 4th reference position via the intermediate point specified by G30.

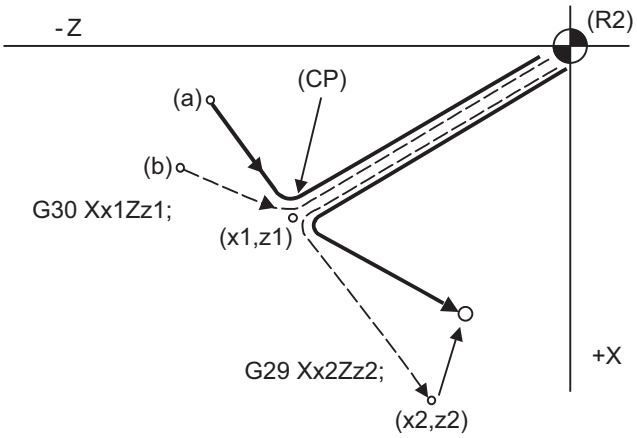
(3) The 2nd, 3rd, and 4th reference position coordinates refer to the positions specific to the machine, and these can be checked with the setting and display unit.

(4) If G29 is commanded after completion of returning to the 2nd, 3rd, and 4th reference position, the intermediate position used last is used as the intermediate position for returning by G29.



(CP) Intermediate point
(R1) 1st reference position (R2) 2nd reference position

(5) With reference position return on a plane during compensation, the tool moves without nose R compensation (zero compensation) from the intermediate point as far as the reference position. With a subsequent G29 command, the tool move without nose R compensation from the reference position to the intermediate point and it moves with such compensation until the G29 command from the intermediate point.

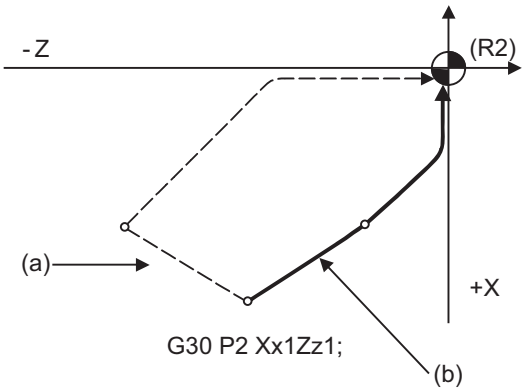


(a) Tool nose center path (b) Program path
(CP) Intermediate point
(R1) 1st reference position (R2) 2nd reference position

(6) The tool length offset amount for the axis involved is temporarily canceled after the 2nd, 3rd and 4th reference position return.

(7) With second, third and fourth reference position returns in the machine lock status, control from the intermediate point to the reference position will be ignored. When the designated axis reaches as far as the intermediate point, the next block will be executed.

(8) With second, third and fourth reference position returns 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 position and the tool moves to the reference position.



(a) X-axis mirror image
(b) No mirror image
(R2) 2nd reference position

(9)If the 2nd, 3rd or 4th reference position is changed while G30 zero point return operation is in pause due to an interlock, "M01 Operation Error" occurs.

(10)When G28/G29/G30 is commanded in single block mode, if "#1279 ext15/bit6 Enable single block stop at middle point" is set to "1", single block stop at middle point will be performed; single block stop at middle point will not be performed if set to "0".

(11)If the mode is switched to MDI mode or reference position return mode while in a single block stop at the interim position, an operation error (M01 0013) occurs.

(12)If the NC is reset while in a single block stop at middle point, the interim position for G29 start position return will not be updated.

(13)If a miscellaneous function is commanded in the same block as G30, the miscellaneous function completion waiting point will be the end of commanded movement, instead of the interim position.

(14) If the PLC interrupt operation is operated while in a single block stop at the interim position, an operation error (M01 0129) occurs.

20.12 Tool Change Position Return ; G30.1 - G30.5

Function and purpose

By specifying the tool change position in a parameter "#8206 tool change" and also specifying a tool change position return command in a machining program, the tool can be changed at the most appropriate position.
The axes that are going to return to the tool change position and the order in which the axes begin to return can be changed by commands.

Command format

Tool change position return

```
G30.n ;
```

n = 1 to 5: Specify the axes that return to the tool change position and the order in which they return.

Detailed description

Commands and return order are given below.

Command	Return order
G30.1	X axis only (-> additional axis)
G30.2	Z axis only (-> additional axis)
G30.3	X axis -> Z axis (-> additional axis)
G30.4	Z axis -> X axis (-> additional axis)
G30.5	X axis - Z axis (-> additional axis)

<Note>

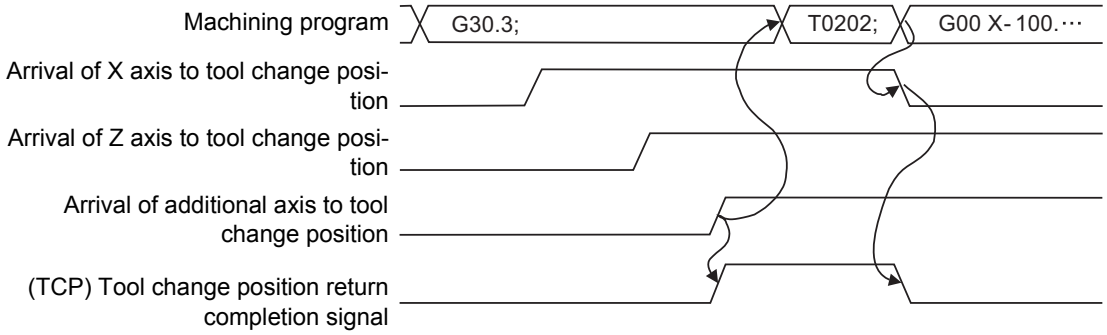
- An arrow (->) indicates the order of axes that begin to return. An period (-) indicates that the axes begin to return simultaneously. (Example: "Z axis -> X axis" indicates that the Z axis returns to the tool change position, then the X axis does.)

(1) Whether the tool exchange position return for the additional axis is enabled or disabled depends on the MTB specifications (parameter "#1092 Tchg_A").
For the order for returning to the tool change position, the axes return after the standard axis completes the return to the tool change position (refer to above table). For specifications having two additional axes, the two additional axes simultaneously return to the tool change position after the standard axis has finished its return to the tool change position.
The additional axis alone cannot return to the tool change position.

(2) If the axis address is commanded in the same block as the tool change position return command, a program error (P33) will occur.

(3) When additional axes have also completed their tool change position return commanded by G30.n, the tool change position return completion signal TCP (XC93) turns ON. When an axis out of those having returned to the tool change position by a G30.n command leaves the tool change position, the TCP signal is turned OFF. (With a G30.3 command, for example, the TCP signal is turned ON when the Z axis has reached the tool change position after the X axis did (after the additional axis did if additional axis tool change position return is valid)). The TCP signal is then turned OFF when the X or Z axis leaves the position. If tool change position return for additional axes is ON with parameter "#1092 Tchg_A", the TCP signal is turned ON when the additional axis or axes have reached the tool change position after the standard axes did. It is then turned OFF when one of the X, Z, and additional axes leaves the position.)

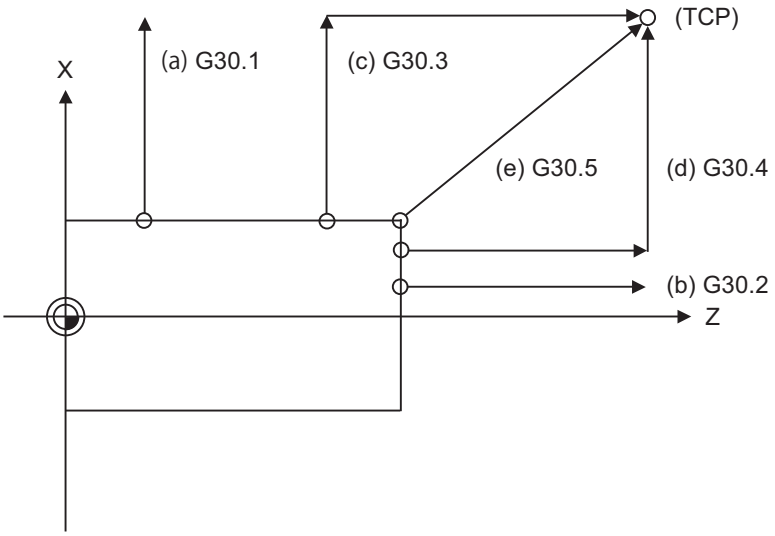
[TCP signal output timing chart] (G30.3 command with tool change position return for additional axes set ON)



(4) Tool compensation data such as tool length offset and tool nose wear compensation are temporarily canceled by the tool change position return command. The machine moves to the tool change position set in the parameters, but because the tool compensation amount is stored in the memory, it moves by the next movement command to a position with the tool compensation applied.

(5) This command is executed by dividing blocks for every axis. If this command is issued during single-block operation, therefore, a block stop occurs each time one axis returns to the tool change position. To make the next axis tool change position return, therefore, a cycle start needs to be specified.

Operation example



TCP : Tool change position

(a) G30.1 command: X axis only returns to the tool change position. (If the tool change position return is validated for the additional axis, the additional axis also returns to the tool change position after the X axis reaches the tool change position.)

(b) G30.2 command: Z axis only returns to the tool change position. (If the tool change position return is validated for the additional axis, the additional axis also returns to the tool change position after the Z axis reaches the tool change position.)

(c) G30.3 command : X axis returns to the tool change position, then Z axis does the same thing. (If the tool change position return is validated for the additional axis, the additional axis also returns to the tool change position after the X and Z axes reached the tool change position.)

(d) G30.4 command : Z axis returns to the tool change position, then X axis does the same thing. (If the tool change position return is validated for the additional axis, the additional axis also returns to the tool change position after the Z and X axes reache the tool change position.)

(e) G30.5 command : X and Z axes return to the tool change position simultaneously. (If the tool change position return is validated for the additional axis, the additional axis also returns to the tool change position after the Z and X axes reached the tool change position.)

20.13 Reference Position 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 position, it outputs the reference position 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 position and return to the 1st reference position, it is possible to check whether the tool has returned to the reference position after the program has been run.

Command format

G27 X__ Z__ α__ P__ ; ... Check command

X Z α	Return control axis
P	Check No. P1: 1st reference position check P2: 2nd reference position check P3: 3rd reference position check P4: 4th reference position check

Detailed description

(1) If the P command has been omitted, the 1st reference position will be checked.

(2) The number of axes whose reference positions can be checked simultaneously depends on the number of axes which can be controlled simultaneously.

(3) An alarm will occur if the reference position is not reached after the command is completed.

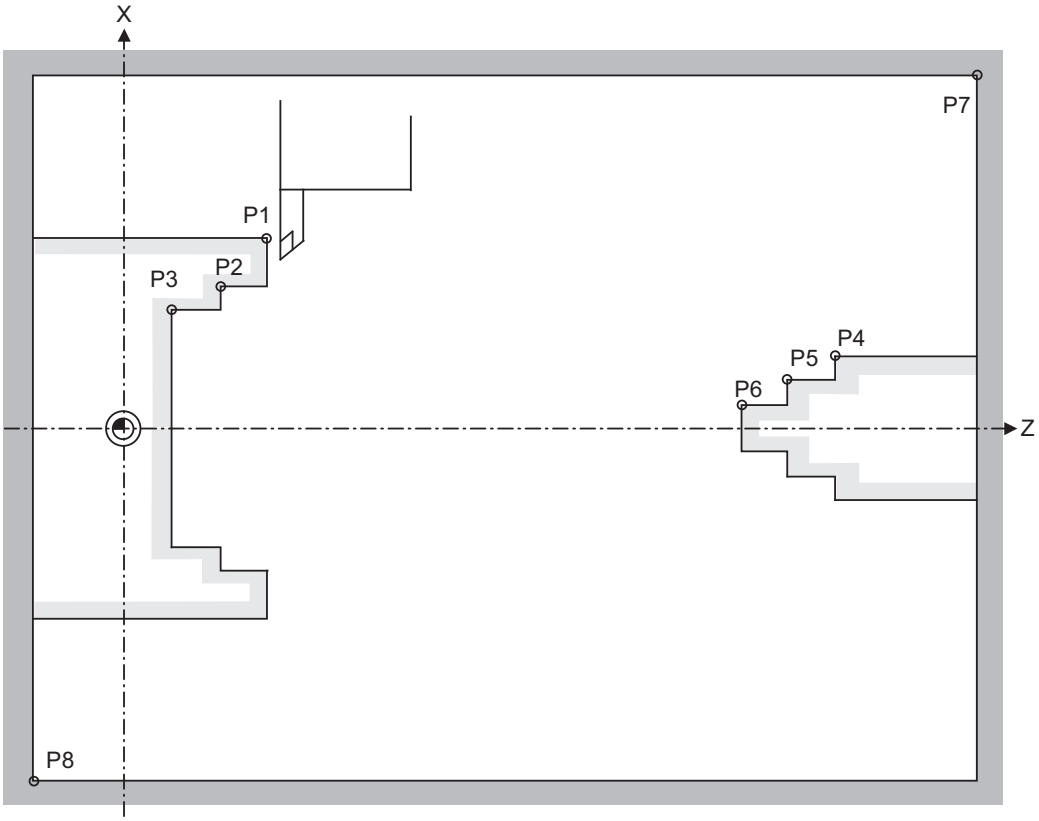
21

Protection Function

21.1 Chuck Barrier/Tailstock Barrier ; G22,G23

Function and purpose

By limiting the tool nose movement range, the chuck barrier and tailstock barrier prevent collision with the chuck and tailstock due to programming errors. If movement is commanded which exceeds the region set by the parameters, the tool will automatically stop at the barrier boundary.



P1,P2,P3 : Chuck barrier
P4,P5,P6 : Tailstock barrier
P7,P8 : Stored stroke limit

Command format

G22 ; ... Barriers valid

G23 ; ... Barriers invalid

Command G22 and G23 in independent blocks.

Detailed description

(1) An alarm will appear at the same time as the machine stops because it was about to exceed the set region. Reset to cancel this alarm.

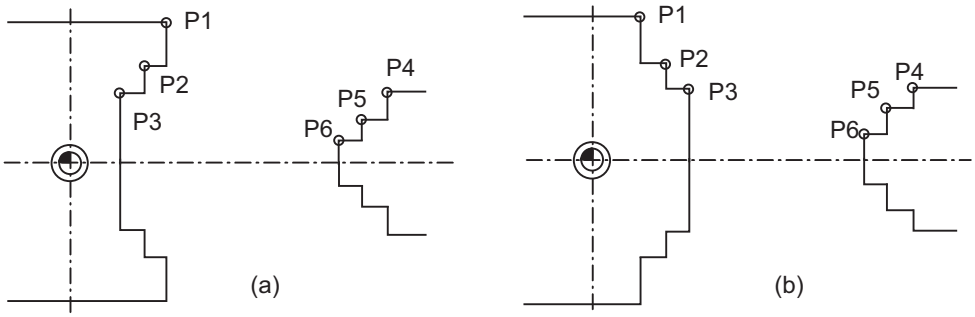
(2) This function is also valid during machine lock.

(3) This function is validated when all axes in which chuck barrier and tailstock barrier are set have finished their reference position returns.

(4) When there is a stored stroke check function, and the stored stroke limit region is set, the chuck barrier/tailstock barrier function is validated simultaneously with the stored stroke check function.

(5) A detailed barrier can be set on the left and right by using the PLC signal. This depends on the MTB specifications.

Setting when using G22 and G23



(1) Three points can be input as parameters for both the chuck barrier and tailstock barrier. Set them in the machine coordinate system.
Points P1, P2 and P3 (parameters "#8301 P1" to "#8303 P3") are for the chuck barrier. Points P4, P5 and P6 (parameters "#8304 P4" to "#8306 P6") are for the tailstock barrier.

(2) The barrier region should be a symmetric shape regarding the Z axis. When the X axis coordinates of barrier point P_ are a negative value, reverse the sign to the positive side, then convert and check.
The absolute value of each barrier point's X axis coordinates must be set as follows.
P1 >= P2 >= P3, P4 >= P5 >= P6(Note that the Z axis coordinates do not have to follow this setting.)

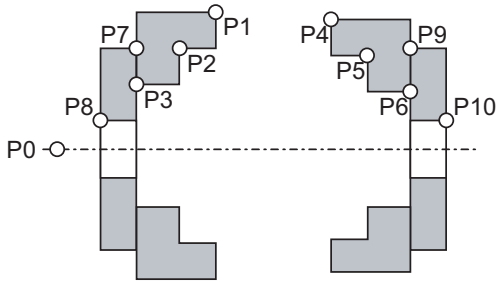
Setting when using the PLC external signal input

Set the coordinate values for each point between "#8300 P0" to "#8314 P10".
P0 is the basic X coordinate of the chuck and tailstock barrier. Set the workpiece center coordinate in the basic machine coordinate system as a radius value.
The coordinate value from the workpiece center (P0) is set as a radius value for the P1 to P10 X axes. The Z axis is set as a basic machine coordinate system coordinate.
The barrier region is symmetrical in respect to P0.
The X axis coordinate of each point must be set in the following manner.
P1 >= P2 >= P3, P4 >= P5 >= P6,
P7 >= P8, P9 >= P10
If the P8 Z axis coordinate must be less than P1 to P3, and the P10 Z axis coordinate must be more than P4 to P6.

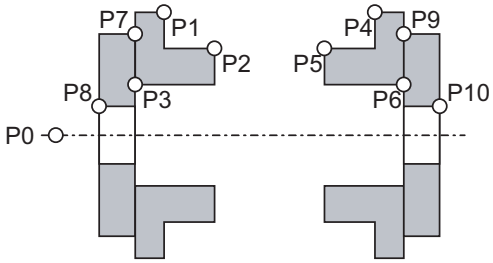
The positions of the each point between P0 and P10 in the chuck/tailstock are as follow.

(1) Setting the chuck

[For external claw]



[For internal claw]

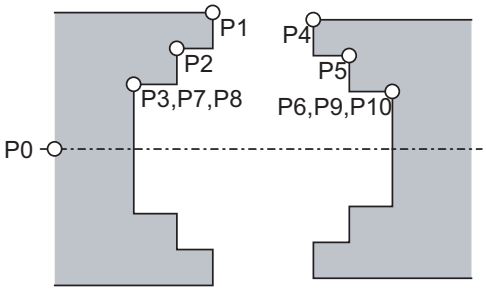


P0 : Workpiece rotation center coordinate
P1,P2,P3 : Chuck section (left)
P4,P5,P6 : Chuck section (right)
P7,P8 : Spindle section (left)
P9,P10 : Spindle section (right)

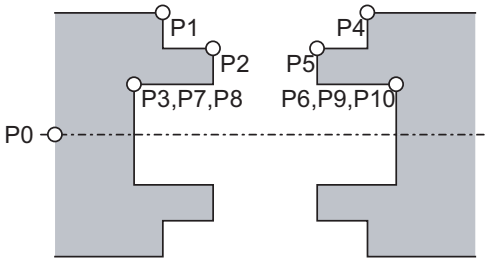
- Set the X axis coordinates so that P1 > P2 > P3 is established.
- The P3 and P7 in Z coordinates and the P6 and P9 in Z coordinates shall be equal.
- If there is no spindle inner diameter, set the P8 X coordinate to the same value as P0.

If a spindle section is not to be set, set the P3, P7 and P8 values to the same values, and the P6, P9 and P10 values to the same values. The barrier range will be as shown below in this case.

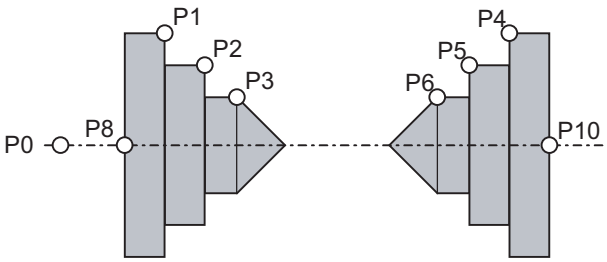
[For external claw]



[For internal claw]



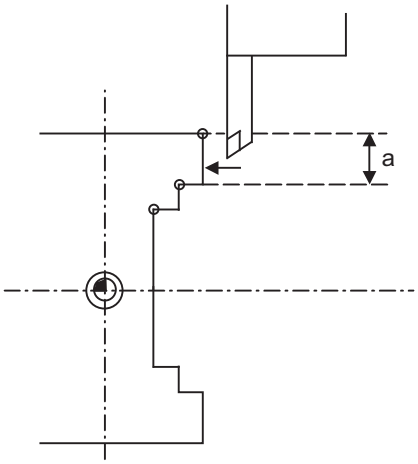
(2) Setting the tailstock

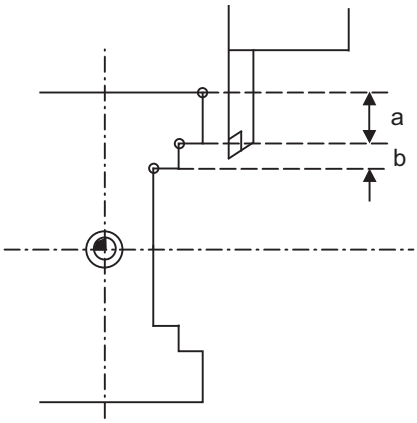


- P7 and P9 are not used.
- The tailstock end section angle is set with the parameters.
  "#8318 STOCK ANGLE (L)" "#8319 STOCK ANGLE (R)"
- If "0" is set, the angle will be 90°(default).
- Set the P8 X coordinate to the same value as P0. (No spindle inner diameter)

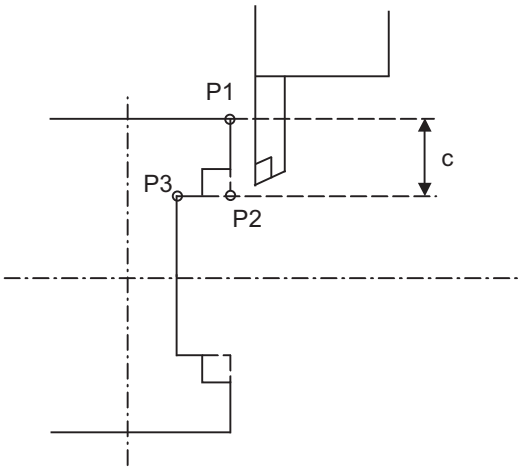
Precautions

(1) There is only one checkpoint from the tool regarding the chuck barrier/tailstock barrier. Therefore, the following cautions must be observed.
In the following examples, when the barrier points are set to be checked by the hypothetical tool nose point and the tool moves in the direction of the arrow in the drawing, the following situation may occur. In Example 1, there is a checkpoint in the range "a", so the tool will automatically stop at the barrier boundary. However, in Example 2 there is a checkpoint in the range "b", so the chuck and tool may collide in the range "a".
(Example 1) (Example 2)





To avoid this, Example 3 is given. In this example, if the barrier points P1, P2 and P3 are set and the checkpoint is set in range "c", the tool can be stopped at the barrier boundary.
(Example 3)



(2) When the tool enters the barrier region and an alarm occurs, the tool may move in the opposite direction from which it came, once the alarm is canceled by resetting.

(3) There is no barrier region for axes without a reference position return function. Thus, there is no barrier alarm for that axis.

(4) When the tool enters a canceled barrier region, and that barrier is then validated, an alarm will occur immediately if the tool is moved.
In this case, after canceling the alarm with reset and then invalidate the barrier (G23) before escaping or change the value set for each barrier point.

(5) The soft limit is valid even if the barrier is invalid (G23).

21.2 Stored Stroke Limit ; G22,G23

Function and purpose

The stored stroke limit is a function that can prevent collision of the tools by setting the tool entry prohibited ranges. The stored stroke limit II function can change the tool entry prohibited ranges (parameter) and command its function's ON/OFF by the program.

Command format

Stored stroke limit II function (parameter change, function ON (all axes))

```
G22 X__ Z__ C__ I__ J__ K__ ;
```

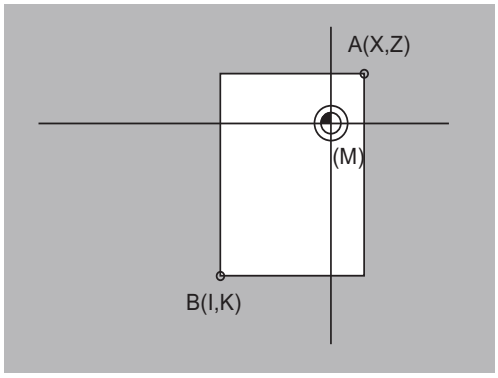
Stored stroke limit II function OFF (all axes)

```
G23;
```

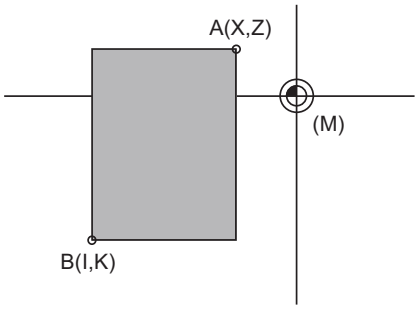
X, Z, C (1st to 3rd axis name)	Stored stroke limit + side designation
I, J, K (Plane selection axis name)	Stored stroke limit - side designation

Detailed description

Command G22 and G23 in independent blocks.
Whether to set the prohibited range on the inside or outside follows the parameter settings (#8204 or #8205).



[When prohibited range is on outside]



[When prohibited range is on inside]

Prohibited range

(M) Basic machine coordinate system

Precautions

(1) If the maximum value and minimum value of the stored stroke limit's prohibited range are set to the same value, the following will occur.
(a) When the maximum value and minimum value are set to 0, if the outside is the prohibited range, the entire range will be prohibited. If the inside is the prohibited range, the entire range will be the moveable range.
(b) If data other than 0 is set for the maximum value and minimum value, the entire range will be the moveable range. It depends on the MTB specifications (parameter "#1037 cmdtyp").

(2) The settings (G22, G23) with the stored stroke limit II program command are valid at the G code list 6 and 7. It depends on the MTB specifications (parameter "#1037 cmdtyp").

(3) G22 and G23 are unmodal, and are valid only in the commanded block.

21.3 Enable Interfering Object Selection Data; G186

Function and purpose

Sixteen interfering objects to be checked in the interference check III are preset by the MTB (R register or system variables).
The interfering object selection is enabled by the "Interference check III: Enable interfering object selection data" signal (Y769) or the "Enable interfering object selection data" command (G186) after the target interfering object has been selected.
When the "Interference check III mode" signal (Y76A) is set to ON after the interfering object selection has been enabled, the interference check starts.

Refer to the PLC Interface Manual (IB-1501258) for the R register and commands issued by the PLC device.
For details on the system variables, refer to "23.24 System Variables (Interfering Object Selection)".
This section describes the "Enable interfering object selection data" command (G186).

Command format

"Enable interfering object selection data" command

```
G186;
```

Detailed description

Consistency check between interfering object definition and interfering object selection

(1) When the "Enable interfering object selection data" command (G186) or the "interference check III: Enable interfering object selection data" signal is set to ON, the consistency between the interfering object definition and interfering object selection is checked.

(2) If the consistency check causes an operation error, all axes in all part systems will stop.
An operation error can be remedied by redefining the interfering object data (*1) or resetting all part systems (except for sub part system 2).

(*1) After correcting the interfering object data, issue the "Enable interfering object selection data" signal or "Enable interfering object selection data" command (G186).

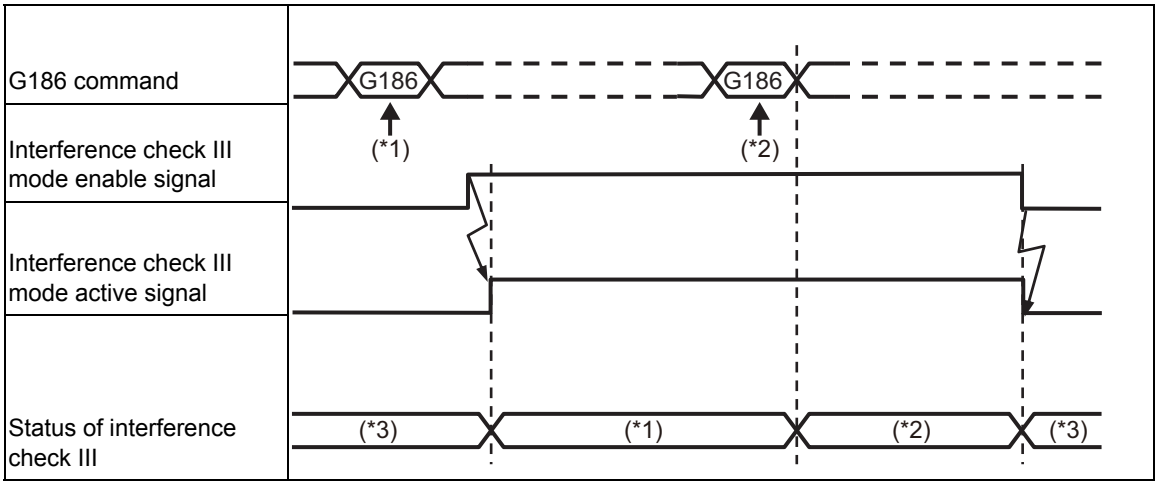
(3) The manual operation and automatic operation are not available until all the part systems (except for subpart system 2) are reset.

(4) In the case the alarm occurs due to the consistency check, the interfering data will not be updated. For the interference check between interfering objects, the interfering data enabled last time is continuously used.

Interference check III mode enable command

While the interference check III mode signal is set to ON after the "Enable interfering object selection data" signal or the "Enable interfering object selection data" command (G186) has been executed, the interference between interfering objects is checked. While the interference check III is being executed, the interference check III mode active signal is turned ON.
After the NC power is turned ON, if the interference check III mode signal is turned ON without executing the "Enable interfering object selection data" signal or the "Enable interfering object selection data" command (G186) even once, an operation error (M03 1001) will occur.

Timing chart at execution of G186



(*1) The first interfering data pattern is set by the G186 command.
The interference check III function executes check processing based on the first data pattern setting.

(*2) The second interfering data pattern is set by the G186 command.
The interference check III function executes check processing based on the second data pattern setting.

(*3) The interference check III function is not executed.

Relationship with other functions

Manual arbitrary reverse run

The program cannot be run backward prior to the "Enable interfering object selection data" command (G186).

Arbitrary reverse run

If the "Enable interfering object selection data" command (G186) is run backward, the interference data at the reverse run is enabled, instead of returning to the interference data at forward run.

Precautions

(1) The high-speed high-accuracy control function (high-speed machining mode, high-accuracy control, spline interpolation, etc.) generates a path appropriate for the tolerance amount to execute a machining program commanded with fine segments at high speed and smoothly. Thus, a difference arises between the path on which the interference check III is performed and the path on which the tool actually passes. When using the interference check III together with the high-speed high-accuracy control (high-speed machining mode, high-accuracy control, spline interpolation, etc.), define an interfering object (solid) with the clearance amount to suit the path difference that occurs depending on the tolerance amount.

(2) The axis that is stopped when an operation error (M03 0001) or (M03 0002) is detected depends on the MTB specifications (parameter "#1444 otsys" (OT all-part-system stop enable/disable selection).
When "0" is set, all the axes in the part system which controls the axes set to "interfering object I/J/K control axis" and "I/J/K rotary control axis" in the interfering object definition will stop.
When "1" is set, all axes in all part systems will stop.

(3) If an operation error (M03 0002) is detected between the fixed interfering objects (*1), an alarm will be output to part system 1.
(*1) These refer to the interfering objects for which "interfering object I/J/K control axis" and "I/J/K rotary control axis" are not set in the interfering object definition.

(4) If you perform the interference check III during the high-speed simple program check, an operation error (M03 0001) may occur at a position different from the actual operation.

(5) If multiple interfering objects including the rotary axis setting are set as one interfering object using the interfering check III: designation of disabled interference object, only the interfering object in which a rotary axis is set will be in rotating operation, checking the interference between the interfering objects.

(6) If an operation error (M03 0001) occurs, cancel the alarm by moving the interfering object to the retracting direction with the linear axis.

(7) The PLC axis is not available for the interference check III. However, it is available when NC axis/auxiliary axis switching is enabled.

(8) In the interference check III, the interference is checked in 0.1μm units regardless of the control unit.

(9) At the occurrence of the operation error (M03 0001), all the axes in the part system in which the alarm has occurred will stop. If the entry to the interference alarm area is not detected by the subsequent axis travel command (manual operation/automatic operation), the operation error (M03 0001) will be cancelled and the axes will travel. Depending on the relative positional relation between the interfering objects or the feedrate of axes, the axis can travel further to the interfering direction from the stopped position (a direction to which the interfering objects interfere).
Even if the axis moves toward the interfering direction, it will stop before entering the interference alarm area.

22

Measurement Support Functions

22.1 Automatic Tool Length Measurement ; G37

Function and purpose

These functions issue the command values from the measuring start position as far as the measurement position, 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 compensation of the present wear compensation amount.

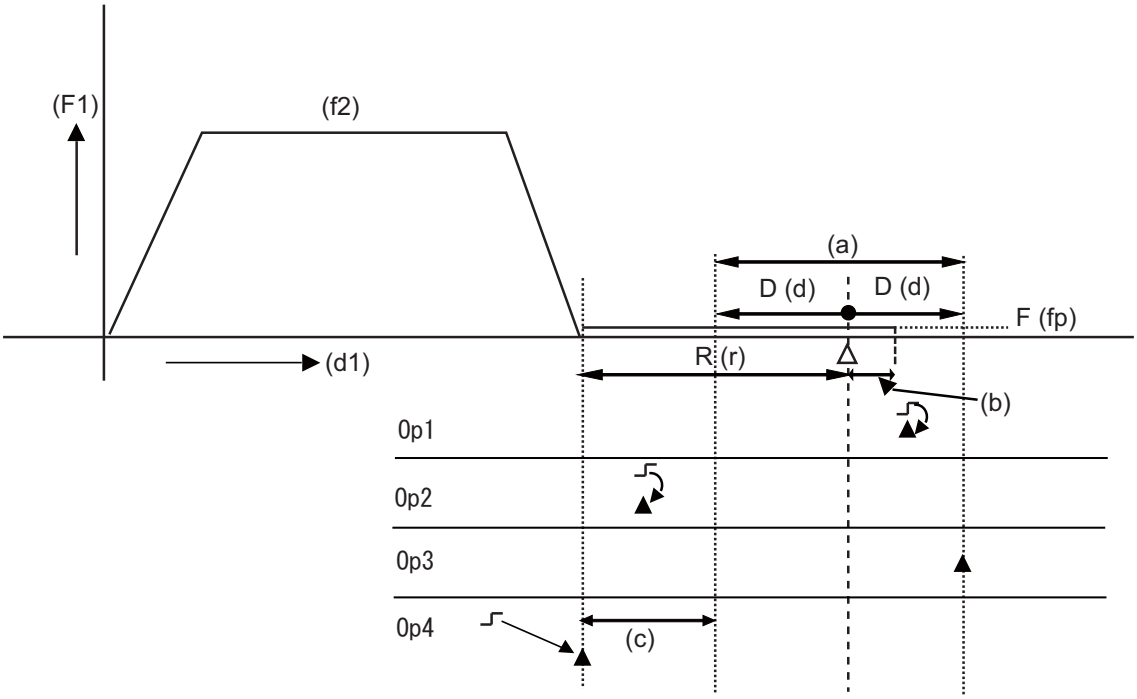
Command format

G37 α__ R__ D__ F__ ; ... Automatic tool length measurement command

α	Measuring axis address and coordinates of the measurement position .....X,Z
R	This commands the distance between the measurement position and point where the movement is to start at the measuring speed. (Radius value fixed, incremental value)
D	This commands the range within which the tool is to stop. (Radius 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> ("AUTO TLM" on machining parameter screen)- #8004 SPEED (measuring feedrate): 0 to 60000 [mm/min] - #8005 ZONE r: 0 to 99999.999 [mm] - #8006 ZONE d: 0 to 99999.999 [mm]

Detailed description

(1) Operation with G37 command



Op1 : Normal completion as it is measurement within the allowable range.

Op2 : Alarm stop (P607) as it is outside of the measurement allowable range.

Op3 : Alarm stop (P607) as the sensor is not detected.

Op4 : Alarm stop (P607) as it is outside of the measurement allowable range. However if there is no (c) area, normal completion will occur.

(a) Measurement allowable range (b) Compensation amount (d1) Distance

(F1) Speed (f2) Feedrate (d) Measurement range

(r) Deceleration range

△ Measuring position ▲ Stop point ⎍ Sensor output

(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 axis will move at the synchronous feedrate [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 delay at the PLC side, the delay and fluctuations in the sensor signal processing range from 0 to 0.2ms.
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) occurs if G37 is commanded when the automatic tool length measurement function is not provided.

(2)Program error (P604) will occur when no axis has been commanded in the G37 block or when two or more axes have been commanded.

(3)Program error (P605) will occur when the T code is commanded in the G37 block. If the last one digit or last two digits is 0, the (4) error will occur.

(4)Program error (P606) will occur when T code is not commanded prior to the G37 block. If the last one digit or last two digits is 0, the (P606) error will occur even if T is commanded.

(5)Program error (P607) will occur 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, if the sensor signal stays ON and there is no range of (b) in Op3 exemplified in Detailed description, it will be judged as normal measurement.

(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:
| Measurement point start point | > R command or parameter r > D command or parameter d

(8) When the D address and parameter d in (7) above are zero, the operation will be completed normally only when the commanded measurement point and sensor signal detection point coincide. Otherwise, program error (P607) will occur.

(9) When the R and D addresses as well as parameters r and d in (7) above are all zero, program error (P607) will occur regardless of whether the sensor signal is present or not after the tool has been positioned at the commanded measurement point.

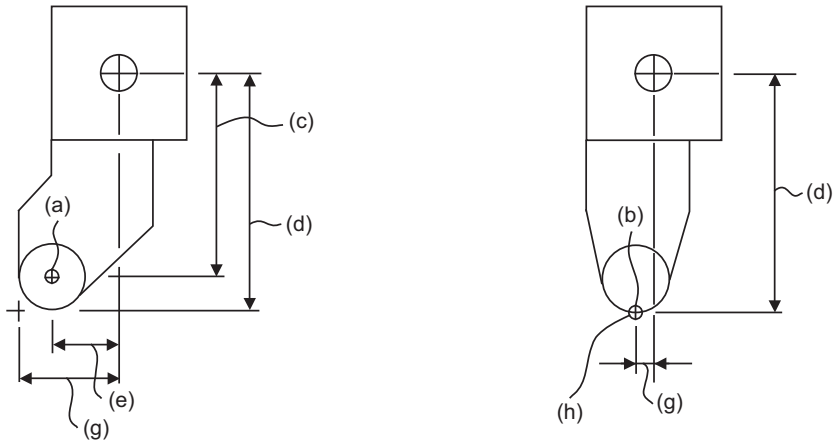
(10) When the measurement allowable range is larger than the measurement command distance, it becomes the measurement allowable range for all axes.

(11) When the measurement speed movement distance is larger than the measurement command distance, all axes move at the measurement speed.

(12) When the measurement allowable range is larger than the measurement speed movement distance, the axis moves in the measurement allowable range at the measurement speed.

(13)Always cancel nose R compensation before commanding G37.

(14) Calculate the tool length compensation without regard for the nose R value and tool nose point No. even if the nose R compensation option is specified.
To set the tool nose point No. to 0, subtract the nose R value from the measured tool length offset amount.
When the tool nose point No. (tool nose shape) is 5, 6, 7, or 8, measure the tool length at the tool tip.



(a) Tool nose point 0 (b) Tool nose point 8
(c) X axis tool length offset value with nose R value subtracted
(d) Measured X axis tool length offset amount
(e) Z axis tool length offset value with nose R value subtracted
(g) Measured Z axis tool length offset amount (h) Tip of tool nose

22.2 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 coordinate value is read, the remaining distance is discarded and the command in the following block is executed.

Command format

G31 X/U__ Z/W__ R__ F__ ;

X, Z, U, W	Axis coordinate value; they are commanded with the absolute or the incremental values.
R	Acceleration/deceleration command R0: Acceleration/deceleration time constant=0 (No automatic acceleration/deceleration after interpolation.) R1: Acceleration/deceleration time constant valid. Accelerate/decelerate with the time constant set in the parameters "#2102 skip_tL" and "#2103 skip_t1". When omitted, R0 will be applied.
F	Feedrate (mm/min)

Detailed description

(1) If an F command is programmed in the same block as G31, the commanded speed is set as the skip speed.
If an F 1-digit feed command is issued to program the feedrate, F 1-digit feed is disabled.
Note that, in the following cases, the skip speed and operations depend on the MTB specifications (parameter "#12022 skipF_spec/bit2").

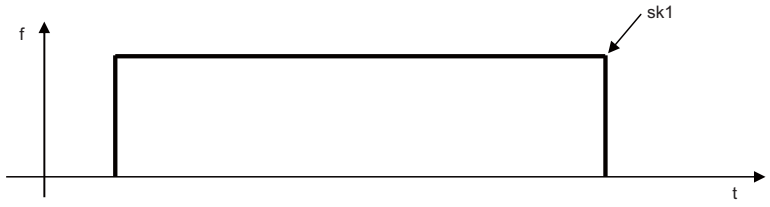
	#12022/bit2 = 0	#12022/bit2 = 1
Skip speed if an F command is not programmed in the G31 block	The value of parameter "#1174 skip_F" is used as the skip speed.	The skip speed is determined based on the modality of F when G31 is executed.
	A program error (P603) will occur if the value of parameter "#1174 skip_F" is "0".	A program error (P62) will occur if the value of F modality is "0".
Mode of commanded speed	Only feed per minute mode is available. Feed per minute mode is enabled even in feed per revolution mode.	Follows the mode (Feed per minute/ Feed per revolution) that is active when G31 is executed.
Modality of F command	The F modal is not updated even if the G31 block contains an F command.	The F modal that is updated by an F command in the G31 block varies depending on the mode (Feed per minute/Feed per revolution) that is active when G31 is executed.

(2) The maximum speed of G31 command is determined by the machine specification.

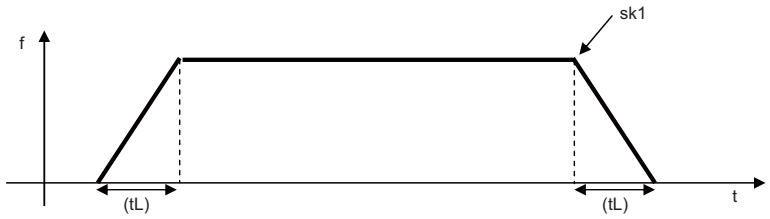
(3) When R0 is commanded or the R command is omitted, the step acceleration/deceleration will be applied to G31 block after the interpolation without performing the automatic acceleration/deceleration.
When R1 is commanded, the automatic acceleration/deceleration will be performed according to the cutting feed acceleration/deceleration mode set by the parameter "#2003 smgst" with the time constant set by the parameter "#2102 skip_tL" and "#2103 skip_t1". Even if G1 constant inclination acceleration/deceleration (the parameter "#1201 G1_acc" is set to "1") is valid, the time constant acceleration and deceleration will be performed.

(4) When the R1 is commanded with the acceleration and deceleration command, the automatic acceleration and deceleration will be performed after the interpolation even if the skip single is input. Note that if the value of the parameter "#2102 skip_tL" and "#2103 skip_t1" are large, the movement will not stop immediately.

Acceleration/deceleration when R0 is commanded or R is omitted



Acceleration/deceleration when R1 is commanded



(sk1) Skip signal (tL) Skip time constant

(5) Command the acceleration/deceleration command (R0/R1) whenever G31 is commanded. If R0/R1 has not been commanded, or anything other than R0/R1 has been commanded, the acceleration/deceleration time constant is assumed to "0" (R0), and automatic acceleration/deceleration after interpolation will not be performed.

(6) When G31 is commanded, the stop conditions (feed hold, interlock, override zero and stroke end) are valid. External deceleration is also valid. For the validity of the following functions, confirm to the MTB specifications.
- Cutting feed override (parameter "#12022 skipF_spec/bit0")
- Dry run (parameter "#12022 skipF_spec/bit1")

(7) The G31 command is unmodal and it needs to be commanded each time.

(8) If the skip command is input at the start of the G31 command, the G31 command will be completed immediately. When a skip signal has not been input until the completion of the G31 block, the G31 command will also be completed upon completion of the movement commands.

(9) When the G31 command is issued during nose radius compensation, the program error (P608) will occur.

(10) When there is no F command in the G31 command and the parameter speed is also zero, the program error (P603) will occur.

(11) With machine lock or with the Z axis cancel switch ON when only the Z axis is commanded, the skip signal will be ignored and execution will continue as far as the end of the block.

Readout of skip coordinates

The coordinate positions for which the skip signal is input are stored in the system variables #5061 (1st axis) to #506n (n-th axis), so these can be used in the user macros.

```
:
G00  X-100. ;
G31 X-200. F60  ;     (Skip command)
#101=#5061 ;          Skip signal input coordinate position (workpiece coordinate system) is readout to #101.
:
```

Note

(1) When the parameter "#1366 skipExTyp (Multi-part system simultaneous skip command)" is set to "1", the skip coordinate value will be "0", even if G31 command is given in the first part system or G31 command is given in only one of the multiple part system.

G31 coasting

The amount of coasting from when the skip signal is input during the G31 command until the machine stops differs according to the parameter "#1174 skip_F" or F command in G31.
The time to start deceleration to stop after responding to the skip signal is short, so the machine can be stopped precisely with a small coasting amount. The coasting amount can be calculated from the following formula.

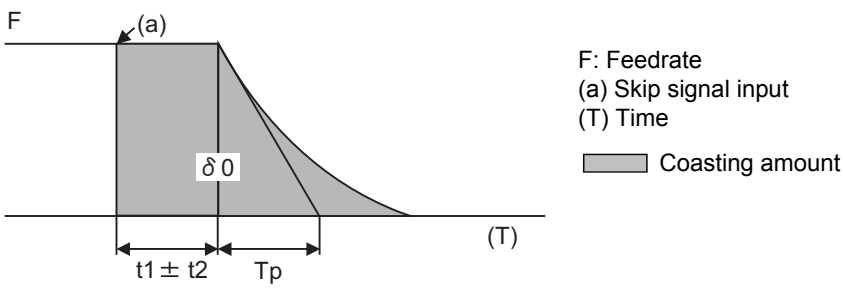
$$\delta 0 = \frac{F}{60} \times Tp + \frac{F}{60} \times (t1 \pm t2)$$

$$= \underbrace{\frac{F}{60} \times (Tp+t1)}_{\delta 1} \underbrace{\pm \frac{F}{60} \times t2}_{\delta 2}$$

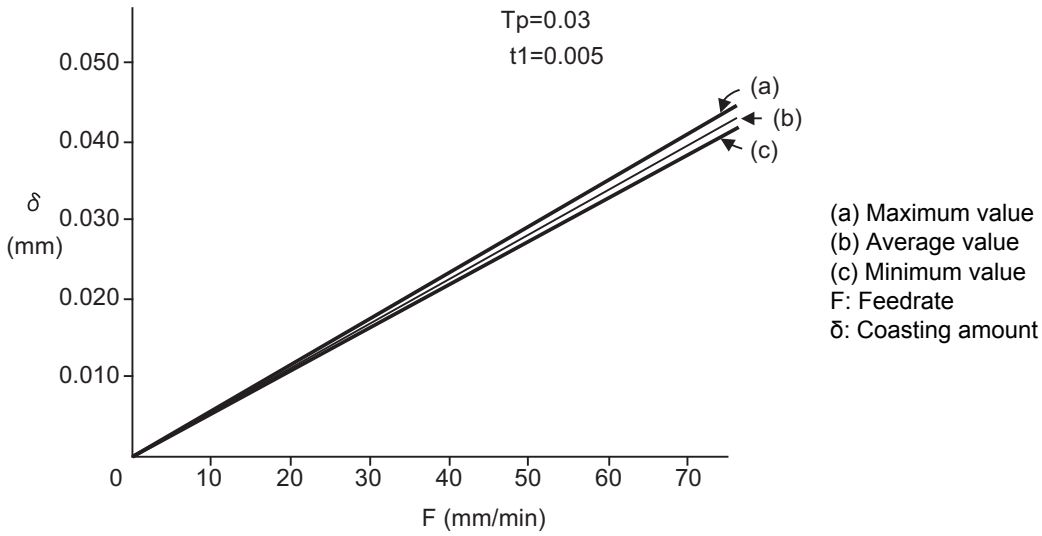
δ0 : Coasting amount (mm)
F : G31 skip speed (mm/min)
Tp : Position loop time constant (s) = (position loop gain)$^{-1}$
t1 : Response delay time (s) = (time taken from the detection to the arrival of the skip signal at the controller via PC)
t2 : Response error time 0.001 (s)

When G31 is used for calculation, the value calculated from the section indicated by δ1 in the above equation can be compensated for, however, δ2 results in calculation error.

Stop pattern with skip signal input is shown below.



The relationship between the coasting amount and speed when Tp is 30ms and t1 is 5ms is shown in the following figure.

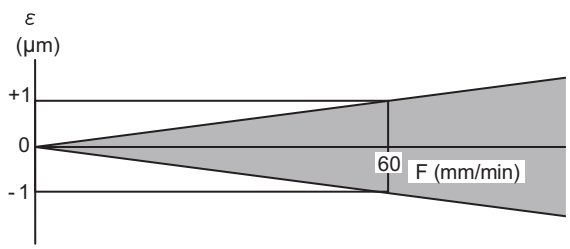


Readout error of skip coordinates mm

(1) Skip signal input coordinate readout

The coasting amount based on the position loop time constant Tp and cutting feed time constant Ts is not included in the skip signal input coordinate values.

Therefore, the workpiece coordinate values applying when the skip signal is input can be readout within the error range in the following formula as the skip signal input coordinate values. However, coasting based on response delay time t1 results in a measurement error and so compensation must be provided.



ε : Readout error
ε=±(F/60)×t 2
F : Feedrate
t2 : Response error time 0.001 (s)

Measurement value

Readout error of skip signal input coordinates

Readout error with a 60mm/min feedrate is as shown below and the measurement value is within readout error range of ±1μm:

ε= ± (60/60) x 0.001 = ± 0.001 (mm)

(2) Readout of other coordinates

The readout coordinate values include the coasting amount. Therefore, when coordinate values at the time of skip signal input is required, reference should be made to the section on the G31 coasting amount to compensate the coordinate value. As in the case of (1), the coasting amount based on the delay error time t2 cannot be calculated, and this generates a measuring error.

Examples of compensating for coasting

(1) Compensating for skip signal input coordinates

```
:
G31 X100.F100 ;            Skip command
G04 ;                      Machine stop check
#101=#5061 ;               Skip signal input coordinate readout
#102=#110*#111/60 ;        Coasting based on response delay time
#105=#101-#102 ;           Skip signal input coordinates
:
#110 = Skip feedrate:
#111 = Response delay time t1;
```

(2) Compensating for workpiece coordinates

```
:
G31 X100.F100 ;            Skip command
G04 ;                      Machine stop check
#101=#5061 ;               Skip signal input coordinate readout
#102=#110*#111/60 ;        Coasting based on response delay time
#103=#110*#112/60 ;        Coasting based on position loop time constant
#105=#101-#102-#103 ;      Skip signal input coordinates
:
#110 = Skip feedrate:
#111 = Response delay time t1;
#112 = Position loop time constant Tp ;
```

Operation when the skip command is executed on multiple part systems at the same time

The operation resulting from the G31 command executed simultaneously on multiple part systems depends on the MTB specifications (parameter "#1366 skipExTyp").

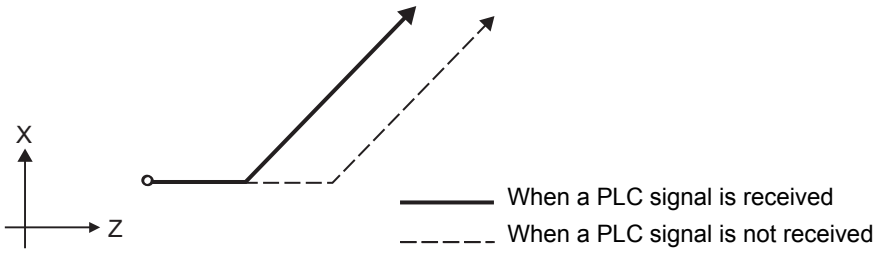
#1366	Operation
0	When any part system is executing the G31 command, the G31 command issued for other part systems is subjected to a block interlock state, and such G31 command will be executed after the current G31 command execution is completed. (No error is displayed.) In a single block operation, for example, where the G31 block is started in multiple part systems at the same time, it is executed in the smallest part system first.
1	The G31 command is executed on multiple part systems at the same time. However, the skip coordinate position is not read and is set to "0" in all part systems. (*1)

(*1) The skip coordinate position is also set to "0" when the G31 command is executed on a single part system. Furthermore, it is set to "0" when the G31 command is executed on one part system in a multiple part system configuration.
When the G31 command is used for measuring purposes, "#1366 skipExTyp" must be "0".

Operation example

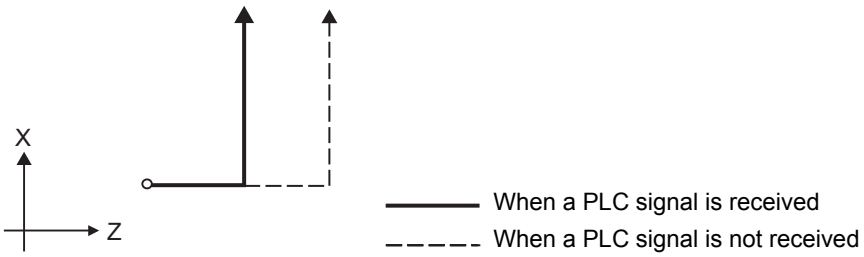
(Example 1) When the next block is an incremental value command

```
G31 Z1000 F100;
G01 U2000 W1000;
```



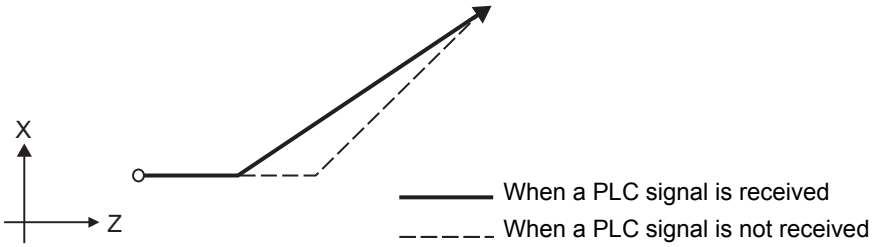
(Example 2) When the next block is a one-axis movement command with absolute values

```
G31 Z1000 F100;
G01 X1000;
```



(Example 3) When the next block is a two-axis movement command with absolute values

```
G31 Z1000 F100;
G01 X1000 Z2000;
```



22.3 Multi-step Skip Function 1 ; G31.n, G04

Function and purpose

The setting of combinations of skip signals to be input enables skipping under various conditions. The actual skip operation is the same as G31.
The G commands which can specify skipping are G31.1, G31.2, G31.3, and G04, and the correspondence between the G commands and skip signals and settings for each parameter depend on the MTB specifications.

Command format

```
G31.1 X__ Z__ α__ R__ F__ ;
```

X Z α	Target coordinates
R	Acceleration/deceleration command R0: Acceleration/deceleration time constant=0 (No automatic acceleration/deceleration after interpolation.) R1: Acceleration/deceleration time constant valid. Accelerate/decelerate with the time constant set in the parameters "#2102 skip_tL" and "#2103 skip_t1". When omitted, R0 will be applied.
F	Feedrate (mm/min)

Same with G31.2 and G31.3; Ff is not required with G04.
As with the G31 command, this command executes linear interpolation and when the preset skip signal conditions have been met, the machine is stopped, the remaining commands are canceled, and the next block is executed.

Detailed description

(1) The skip speed is specified by program command or parameter. Feedrate G31.1 set with the parameter corresponds to "#1176 skip1f", G31.2 corresponds to "#1178 skip2f", G31.3 corresponds to "#1180 skip3f", and G04 corresponds to "#1173 dwlskp". Note that the F modal is not updated in each case.

(2) A command is skipped if it meets the specified skip signal condition.

(3) The feedrates corresponding to the G31.1, G31.2, and G31.3 commands can be set by parameters.

(4) The skip conditions (logical sum of skip signals that have been set) corresponding to the G31.1, G31.2, G31.3 and G04 commands can be set by parameters.

Parameter setting	Valid skip signal		
	1	2	3
1	○		
2		○	
3	○	○	
4			○
5	○		○
6		○	○
7	○	○	○

(5) Details other than the above are the same as those on G31 (Skip function).

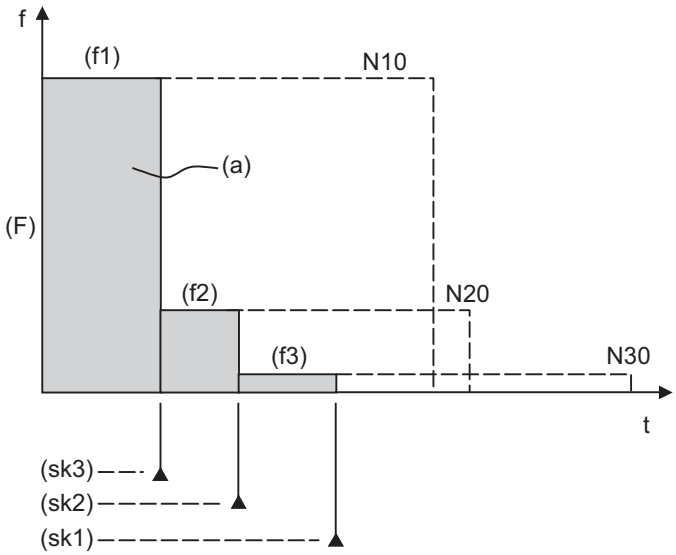
Operation example

(1) The multi-step skip function enables the following control, thereby improving measurement accuracy and shortening the time required for measurement.

[Parameter settings]

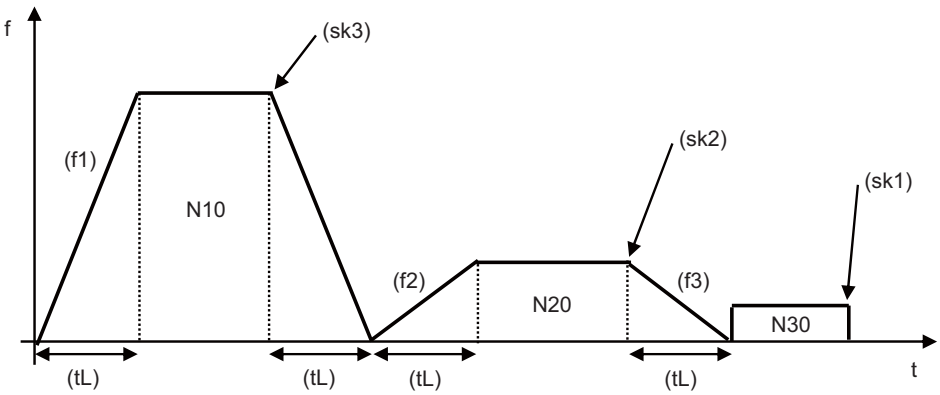
Skip condition	Skip speed
G31.1 :7	20.0 mm/min (f1)
G31.2 :3	5.0 mm/min (f2)
G31.3 :1	1.0 mm/min (f3)

[Program example]
N10 G31.1 X200.0 ;
N20 G31.2 X40.0 ;
N30 G31.3 X1.0 ;



(a) Measurement distance
(F) Skip speed
(sk1) Input of skip signal 1
(sk2) Input of skip signal 2
(sk3) Input of skip signal 3

<Note>
- If skip signal 1 is input before skip signal 2 in the above operation, N20 is skipped at that point and N30 is also ignored.



(sk1) Skip signal (tL) Skip time constant

(2) If a skip signal with the condition set during G04 (dwell) is input, the remaining dwell time is canceled and the following block is executed.

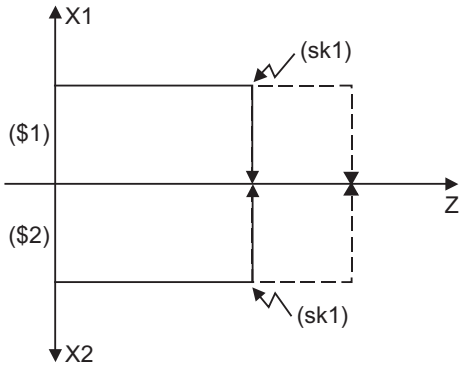
22.4 Multi-step Skip Function 2 ; G31 P

Function and purpose

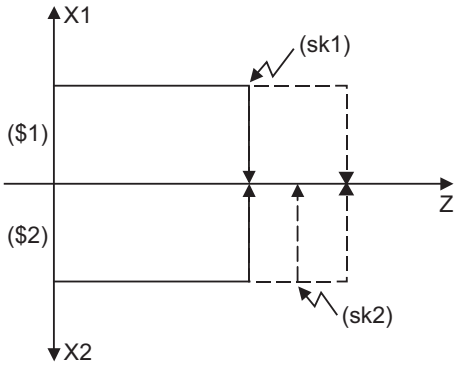
During linear interpolation by the skip command (G31), operation can be skipped according to the conditions of the skip signal parameter Pp.

If multi-step skip commands are issued simultaneously in different part systems as shown in the left figure, both part systems perform skip operation simultaneously if the input skip signals are the same, or they perform skip operation separately if the input skip signals are different as shown in the right figure. The skip operation is the same as a normal

skip command (G31 without P command).



[Same skip signals input in both 1st and 2nd part systems]



[Different skip signals input in 1st and 2nd part systems]

($1) 1st part system
($2) 2nd part system
(sk1) Skip signal 1
(sk2) Skip signal 2

If the skip condition specified by the parameter "#1173 dwlskp" (indicating external skip signals 1 to 4) is met during execution of a dwell command (G04), the remaining dwell time is canceled and the following block is executed.

Command format

G31 X__ Z__ α__ P__ R__ F__ ;

X Z α	Target coordinates
P	Skip signal command
R	Acceleration/deceleration command R0: Acceleration/deceleration time constant=0 (No automatic acceleration/deceleration after interpolation.) R1: Acceleration/deceleration time constant valid. Accelerate/decelerate with the time constant set with the parameters "#2102 skip_tL" and "#2103 skip_t1". R0 is applied when it is omitted.
F	Feedrate (mm/min)

Detailed description

(1) The skip speed is specified by program command or parameter. The feedrate by the parameter is set by "#1174 skip_F". Note that the F modal is not updated in each case.

(2)The skip signal is specified by skip signal command p. The command range of "p" is from 1 to 255. If outside the range is commanded, program error (P35) will occur.

Skip signal command P	Valid skip signal							
	8	7	6	5	4	3	2	1
1								○
2							○	
3							○	○
4						○		
5						○		○
6						○	○	
7						○	○	○
8					○			
: : :								
253	○	○	○	○	○	○		○
254	○	○	○	○	○	○	○	
255	○	○	○	○	○	○	○	○

(3) The specified skip signal command is a logical sum of the skip signals.
(Example) G31 X100. P5 F100 ;
Operation is skipped if skip signal 1 or 3 is input.

(4) If skip signal parameter Pp is not specified, it works as a skip function (G31), not as a multi-step skip function. If speed parameter Ff is not specified, the skip speed set by the parameter "#1174 skip_F" will apply.

[Relations between skip and multi-step skip]

Skip specifications	×		○	
	Condition	Speed	Condition	Speed
G31 X100 ; (Without P and F)	Program error (P601)		Skip 1	#1174 skip_F
G31 X100 P5 ; (Without F)	Program error (P602)		Command value	#1174 skip_F
G31 X100 F100 ; (Without P)	Program error (P601)		Skip 1	Command value
G31 X100 P5 F100 ;	Program error (P602)		Command value	Command value

(5) If skip specification is effective and P is specified as an axis address, skip signal parameter P will be given a priority. The axis address "P" will be ignored.
(Example) G31 X100. P500 F100 ;
This is regarded as a skip signal. (The program error (P35) will occur.)

(6) Other than above, the same detailed description as "Skip function; G31" applies.

22.5 Speed Change Skip ; G31 Fn

Function and purpose

When the skip signal is detected during linear interpolation by the skip command (G31), the feedrate is changed.

Command format

G31 X__ (Y__) Z__ α__ R__ F__ F1 = __ ... Fn = __ ; ("n" is the skip signal 1 to 8) ... Skip command

X, (Y,) Z, α	Target coordinates
R	Acceleration/deceleration command R0: Acceleration/deceleration time constant=0 When the movement is stopped by the skip signal detection, the step stop will occur. R1: Acceleration/deceleration time constant valid. When the movement is stopped by the skip signal detection, it will decelerate with the time constant set in the parameter "#2102 skip_tL" and "#2103 skip_t1". When omitted, R0 will be applied.
F	Feedrate when starting the cutting feed (mm/min)
Fn=	Feedrate after detecting the skip signal (mm/min) Fn = 0: Movement stop Fn ≠ 0 :Changing the feedrate to fn F1 = Feedrate after inputting the skip signal 1 : F8 = Feedrate after inputting the skip signal 8

Detailed description

(1) When the skip signal for which the feedrate fn≠0 is commanded, the speed is changed to the command speed corresponding to the skip signal.

(2) When the skip signal for which the feedrate fn=0 is commanded, the movement is stopped. If R0 is commanded or R command is omitted, the skip stop will occur when the movement is stopped by the skip signal detection without performing the automatic acceleration/deceleration by the skip time constant.
When R1 is commanded, the automatic acceleration/deceleration will be performed with the skip time constant after the interpolation even if the movement is stopped by the skip signal detection. Note that if the value of the parameter "#2102 skip_tL" and #2103 skip_t1" are large, it will not stop immediately.
After the movement is stopped, the remaining movement commands are canceled and the following block will be executed.

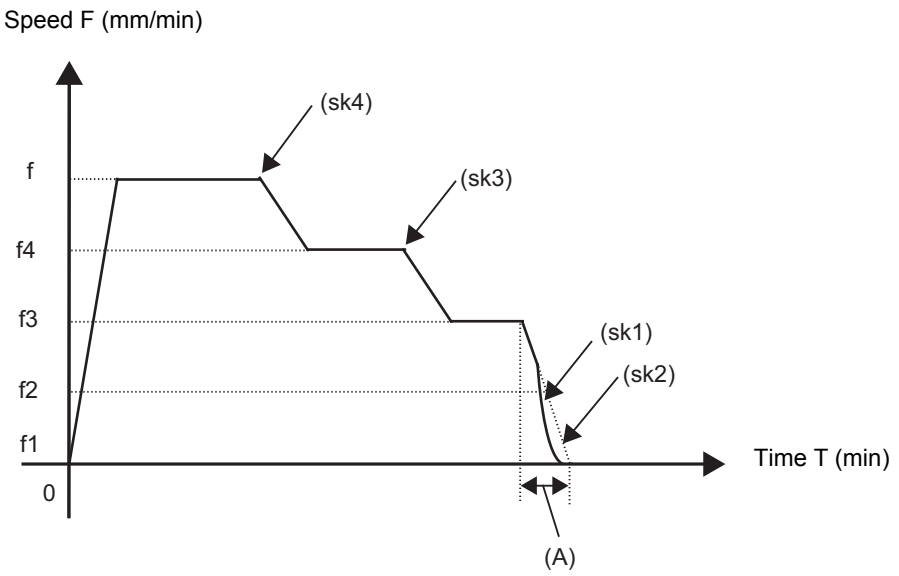
(3) When a skip signal has not been input until the completion of the G31 block, the G31 command will also be completed upon completion of the movement commands.

(4) When the skip return is valid, the return operation by the skip signal detection is executed after the movement is stopped.

(5) Even if G1 constant inclination acceleration/deceleration (#1201 G1_acc) is valid, the speed change skip will be the operation of the time constant acceleration and deceleration.

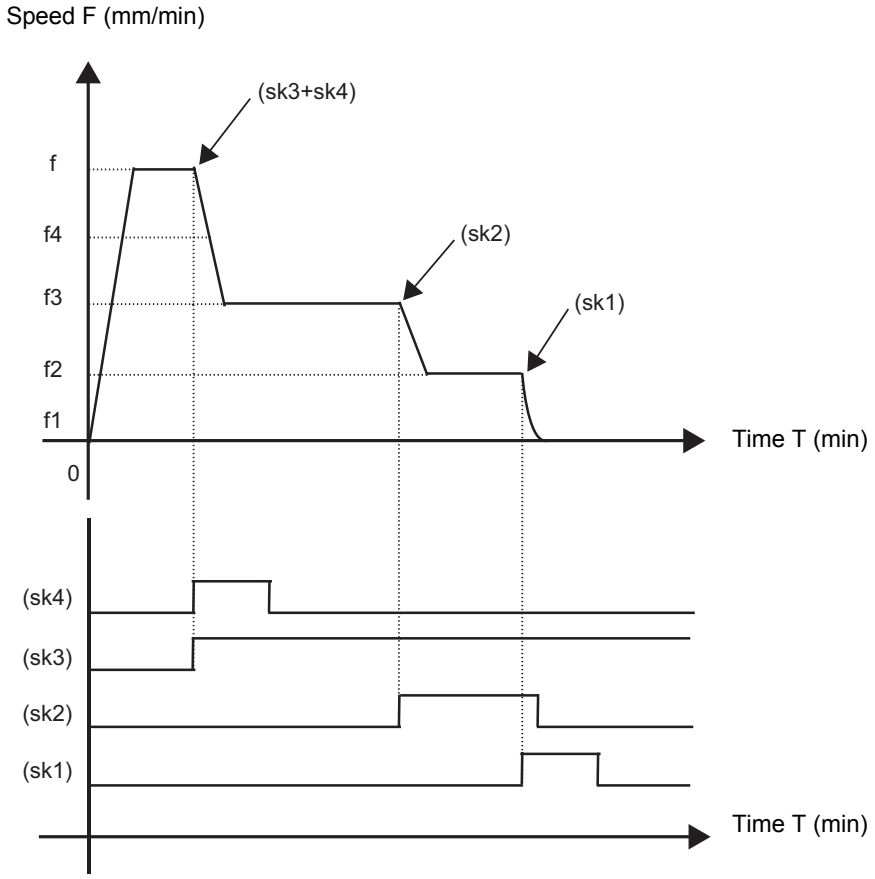
(6) When the feedrate command (Fn=fn) is not specified after detecting the skip signal, the normal G31 skip operation will be applied.

(7) If a skip signal (one of sk1 to sk4) are input during the deceleration (area (A) in the figure) after a move command has finished:
(a) A skip signal (sk2 in the figure) for changing speed is ignored.
(b) A skip signal (sk1 in the figure) for stopping the movement is executed and the speed is set to "0".



(8) The skip signal without commanding the feedrate in the program will be ignored.

(9) The speed change or the movement stop is performed when detecting the rising edge of the skip signal. Note that if several rising edges are input at 3.5ms intervals or less, they may be judged as the simultaneous input. When they are judged as the simultaneous input, the smaller value will be valid.
Shown below are changes in time (T) and speed (F) when skip signals, 1 (sk1) to 4 (sk4), are input.



(10) If the G31 block is started with the skip signal input, that signal is considered to rise at the same time as the block starts.

(11) If the skip signals for changing the speed and for stopping the movement are simultaneously input, the skip signal for stopping the movement will be valid regardless of the size of the number.

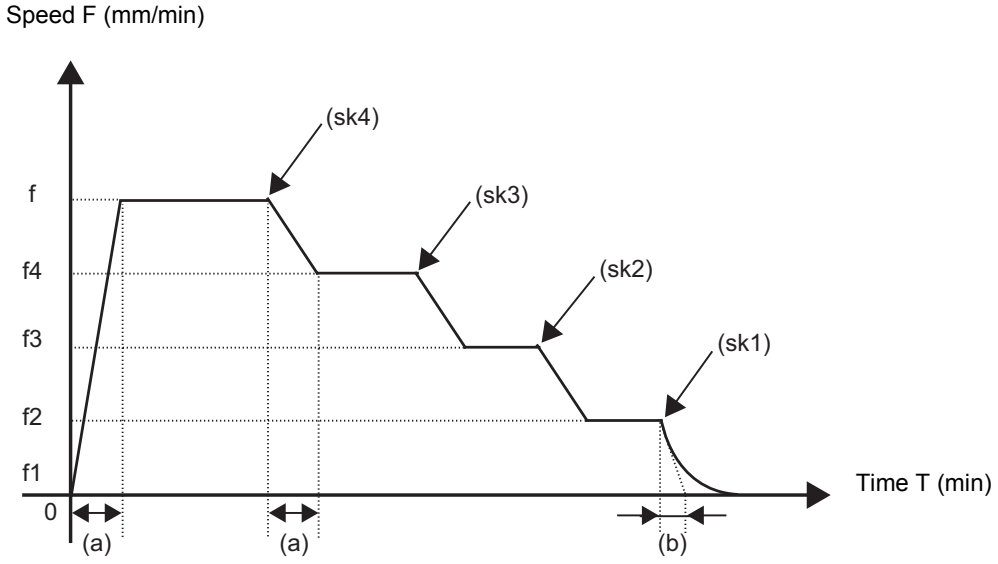
(12) If the skip time constant "#2102 skip_tL" is illegal, an MCP alarm (Y51 15) will occur. If the "#2103 skip_t1" is illegal, an MCP alarm (Y51 16) will occur.

(13) Other than above, the same detailed description as "Skip function; G31" applies.

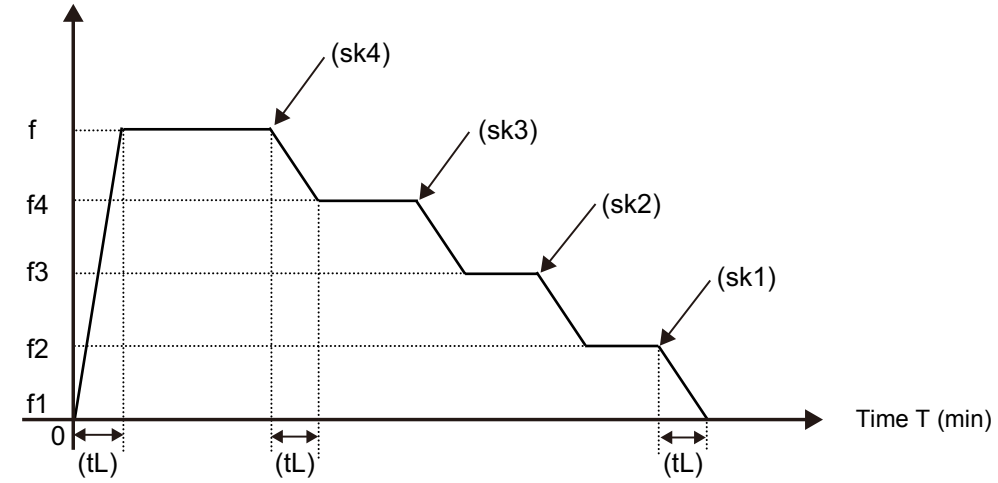
Operation example

The following shows the operations when a skip time constant and skip signals, 1 (sk1) to 4 (sk4), are input.

(1) Example of when R is not commanded
Skip time constant ((a) in the figure) and position loop time constant ((b) in the figure)
G31 X100. Ff F1=0 F2=f2 F3=f3 F4=f4 ;



(2) Example of when R1 is commanded
Skip time constant ((tL) in the figure)
G31 X100. R1 Ff F1=0 F2=f2 F3=f3 F4=f4;

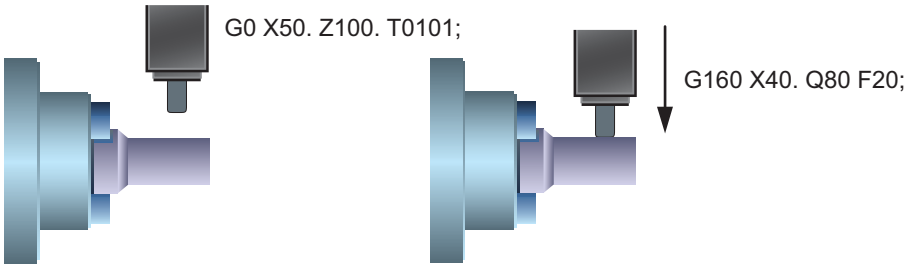


22.6 Torque Limitation Skip ; G160

Function and purpose

Axis movement is performed in the torque limited status, and the axis movement command is suspended to proceed to the next block when the current command value reaches the designated torque skip value and the torque skip turns ON. In addition to the torque, the droop value can be add to the condition of the skip ON (Droop skip).
This function enables measurement without a sensor.

Workpiece radius measurement tool



:	
G00 X50. Z100. T0101;	Tool selection for measurement
G160 X40. Q80 F20;	Torque skip command
#100=#5061;	Coordinate position (workpiece value) read
:	

Command format

Torque limitation skip

G160 X/U/Z/W/α__ Q__ D__ F__ ;

The G160 command is unmodal (group 00). When executing the G160 command continuously, it must always be command for each block separately.

X/U/Z/W/α	Axis address and coordinate value command (mm/inch) (Decimal point command is possible)
Q	Torque skip value (0 to 500 (%))
D	Droop skip value (0 to 99999.999 mm, 0 to 9999.9999 inch)
F	Skip speed Set it in the range of feedrates. (mm/min, inch/min, mm/rev, inch/rev)

Note

(1) Designate an axis that exits in the part system for the axis address. If an axis that does not exist in the part system, a program error (P32) will occur.

(2) Only one axis can be commanded with the axis address. If no axis is specified or if two or more axes are specified in the same block, a program error (P595) will occur.

(3) For spindle/C axis (C axis command), a Q command is specified with 121 to 500 %, the axis is clamped at 120%.

(4) If a Q command is omitted, torque skip function is performed as specified by the MTB (parameter shown below).
NC axis (servo axis): SV014 ILMTsp (current limit value in special control)
Spindle/C axis (C axis command):
- For the normal spindle, SP065 TLM1 (torque limit 1)
- For spindle-mode servo, SV014 ILMTsp (current limit value in special control)

(5) If D command is omitted, a skip operation is performed using the torque skip value only.

(6) D command must be programmed within the excessive error width shown below.
NC axis (servo axis): SV023 OD1 (detected excessive error width when servo is on)
Spindle/C axis: SP023 OD1 (detected excessive error width (interpolation mode))

(7) If an F command is omitted, the feedrate depends on the MTB specifications (parameter "#1174 skip_F").

(8) A program error (P603) will occur if the skip speed in F command is 0.

Detailed description

Acceleration/deceleration when G160 is commanded

- Follow the acceleration/deceleration pattern for linear interpolation (G01).
- Even if G01 constant inclination acceleration/deceleration is valid, the time constant acceleration and deceleration will be performed.

Skip speed

If F command is programmed in the same block as G160, the commanded speed is set as the skip speed.

If an F 1-digit feed command is issued to program the feedrate, F 1-digit feed is disabled.

Note that, in the following cases, the skip speed and operations depend on the MTB specifications (parameter "#12022 skipF_spec/bit2").

	#12022/bit2 = 0	#12022/bit2 = 1
Skip speed if F command is not programmed in the G160 block	The value of parameter "#1174 skip_F" is used as the skip speed.	The skip speed is determined based on the modality of F when G160 is executed.
	A program error (P603) will occur if the value of parameter "#1174 skip_F" is "0".	A program error (P62) will occur if the value of F modality is "0".
Mode of commanded speed	Only feed per minute mode is available. Feed per minute mode is enabled even in feed per revolution mode.	Follows the mode (Feed per minute/Feed per revolution) that is active when G160 is executed.
Modality of F command	F modal is not updated even if the G160 block contains an F command.	The F modal that is updated by F command in the G160 block varies depending on the mode (Feed per minute/Feed per revolution) that is active when G160 is executed.

Control signals regarding speed control and stop

(1) For the validity of the following various functions, refer to the MTB specifications.
- Cutting feed override valid/invalid (parameter "#12022 skipF_spec/bit0")
- Dry run valid/invalid (parameter "#12022 skipF_spec/bit1")

(2) An operation error (M01 0102) occurs if 0% cutting feed override is performed when cutting feed override is invalid.

(3) The stop conditions (feed hold, interlock, override zero and stroke end) and external deceleration are valid when torque limitation skip is used.

(4) The machine lock signal is valid. (The counter is updated until the program reaches the end point of the block.)

Processing when the torque skip turns on

(1) If the current value for the specified axis exceeds the torque skip value, the torque limit is reached and droop exceeds the droop skip value, the torque skip turns on. If there is no D command, the torque skip turns on when the torque limit is reached.

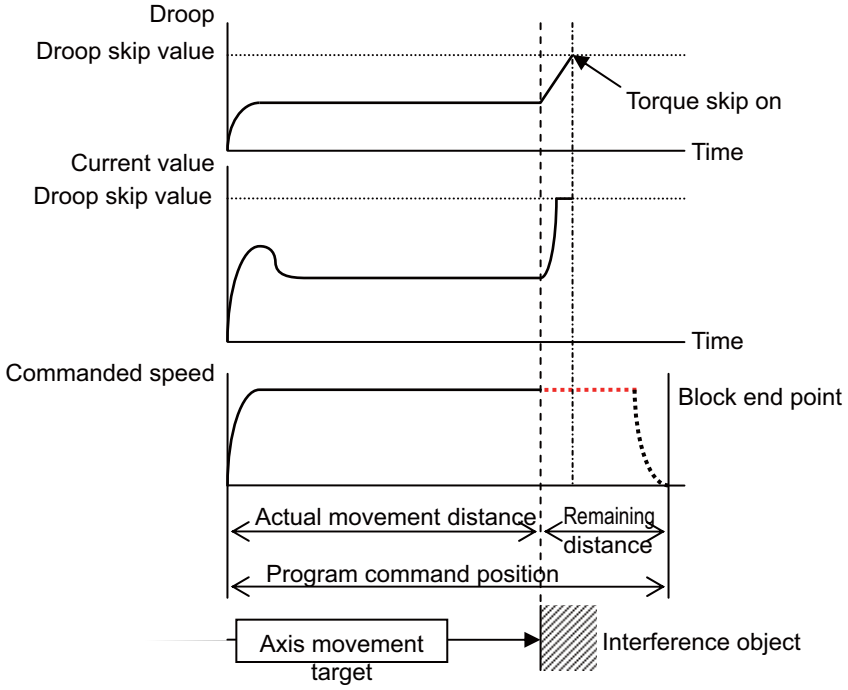
(2) The current position when the torque skip turns on is regarded as the block end point and the remaining distance (command value - actual movement distance) is discarded.

Completion of skip command

(1) If the torque skip turns on during G160 command, the program completes the current block before moving on to the next block.

(2) If the torque skip does not turn on until G160 command reaches the end point, the skip command completes at the end of the block and then the program moves on to the next block.

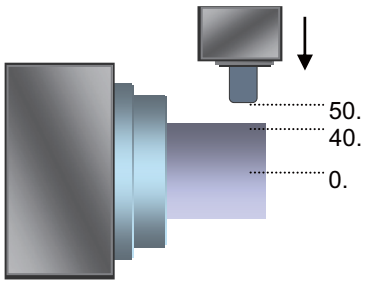
(3) Set the skip coordinate values (workpiece coordinate values) to system variables (#5061 and onwards). When the tool has moved to the end point, set the end point position.



Program example

:	:
G00 X50. Z100. T0101;	Tool selection for measurement
G160 X40. Q80 F20;	Torque skip command
#100=#5061;	Completion of skip command Coordinate position (workpiece value) read
:	

Workpiece radius measurement tool



Relationship with Other Functions

Manual arbitrary reverse run

The skip speed is controlled with the manual arbitrary reverse run speed. Torque skip command block cannot be executed in the reverse run.

Manual interruption

When a manual interrupt is applied during execution of torque skip, calculate the position shifted by the amount of the manual interruption as the skip position.

Skip variables

The torque skip position is common to skip variables (#5061 and onwards) for G31 skip function.

Geometric, Corner Rounding, Corner Chamfering

Geometric, Corner Rounding, and Corner Chamfering are not available for torque skip blocks. Program error (P595) will occur.

Torque limit

Torque skip command, if executed on the axis to which torque limits are applied, is based on the torque skip value in the G160 command.

Functions for which torque skip command is not available

Torque skip command (G160) cannot be commanded when any of the following functions is in use. (An error will occur.)

Function name	Error
Nose R Compensation (G40, G41, G42, G46)	Program error (P608)
Milling Interpolation (G12.1)	Program error (P481)
Synchronous control (G114.1)	Program error (P595)
High-speed high-accuracy control (G05.1/G05)	Program error (P34)
Axis under inclined axis control	Program error (P595)
Axis during the control axis synchronization between part systems	Operation error (M01 1038)
Axes in the control axis superimposition and arbitrary axis superimposition	Operation error (M01 1003)

Precautions

(1) Decreasing the torque limit value may cause a torque limit to be applied during acceleration/deceleration.

(2) When the reset button is pressed while torque skip is active, an axis moving with G160 stops. After the axis has stopped, the original torque is restored.

(3) Writing parameters via a PLC or other host controller during execution of torque skip causes the torque limit value to be the setting value of servo parameter SV014, possibly causing it to be no longer correct torque skip value. (The PLC signal operations and setting values of the servo parameters are based on the MTB specifications.)

(4) When using D command (droop skip value), command a value that does not exceed the excessive error width.

(5) After a torque skip, the droop is canceled.

(6) The droop is displayed in interpolation increments on the drive monitor. They are different from the command increments of D command.

22.7 Programmable Current Limitation ; G10 L14 ;

Function and purpose

This function allows the current limit value of the NC axis to be changed to a desired value in the program, and is used for the workpiece stopper, etc. "#2214 SVO14(current limit value in special control)" can be changed.
The commanded current limit value is designated with a ratio of the limit current to the rated current.

Command format

G10 L14 Xn ;

L14	Current limit value setting (+ side/- side)
X	Axis address
n	Current limit value (%) Setting range: 1 to 999

Precautions

(1) If the current limit value is reached when the current limit is valid, the current limit reached signal is output.

(2) The following two modes can be used with external signals as the operation after the current limit is reached. The external signal determines which mode applies.

[Normal mode]
The movement command is executed in the current state.
During automatic operation, the movement command is executed until the end, and then move to the next block with the droops still accumulated.

[Interlock mode]
During the occurrence of the droops, it enters to the internal interlock state and the next movement will not be carried out.
During automatic operation, the operation stops at the corresponding block, and the next block is not moved to.
During manual operation, the following same direction commands are ignored.

(3) The position droop generated by the current limit can be canceled when the current limit changeover signal of external signals is canceled. (Note that the axis must not be moving.)

(4) The setting range of the current limit value is 1% to 999%. Commands that exceed this range will cause a program error (P35).

(5) If a decimal point is designated with the G10 command, only the integer will be valid.
Example) G10 L14 X10.123 ; The current limit value will be set to 10%.

(6) For the axis name "C", the current limit value cannot be set from the program (G10 command).
To set from the program, set the axis address with an incremental axis name, or set the axis name to one other than "C".

23

System Variables

23.1 System Variables List

The M800/M80/C80 series provides the following system variables.
Note that the available types and numbers vary depending on the machine specifications and whether the machine is intended for use by a user or MTB.

○: Available
-: Unavailable

No.	Data type or use	Reading	Setting	Section
#1000 - #1035, #1200 - #1295	Signal input from PLC to NC	○ (*1)	-	23.20
#1100 - #1135, #1300 - #1395	Signal output from NC to PLC	○ (*1)	○ (*1)	23.21
#2001 - #2000+n #2101 - #2100+n #2201 - #2200+n #2301 - #2300+n #2701 - #2700+n #2801 - #2800+n #2901 - #2900+n	Tool offset data Also refer to "#10001" and later.	○	○	23.6
#2501, #2601	External workpiece coordinate offset / Workpiece coordinate system shift	○	○	23.10
#3000	Used to forcibly set to the alarm mode. Designate the number and message.	-	○	23.12
#3001, #3002	Cumulative time (integrating time)	○	-	23.14
#3001, #3002, #3011, #3012	Time read variables	○	○	23.15
#3003	♦Inhibition of single block stop ♦Inhibition of miscellaneous function finish signal waiting ♦Prohibition of program check reverse run ♦Synchronization between part systems OFF	○	○	23.16
#3004	♦Automatic operation pause OFF ♦Cutting override OFF ♦G09 check OFF ♦Dry run invalid	○	○	
#3006	Used to display and stop a message.	-	○	23.13
#3007	Mirror image	○	-	23.18
#3901, #3902	Number of machining processes / Maximum number of machining processes	○	○	23.17
#4001 - #4021 #4201 - #4221	G command modal information	○	-	23.2
#4101 - #4120 #4301 - #4320	Non-G command modal information	○	-	23.3
#4401 - #4421 #4507 - #4520	Modal information at macro interruption	○	-	23.4
#5001 - #5140+n	Position information ♦End point coordinate position of the previous block ♦Machine coordinate position ♦Workpiece coordinate position ♦Skip coordinate position ♦Servo deviation amount ♦Macro interruption stop block coordinate position	○	-	23.11
#5201 - #532n	External workpiece coordinate offset data	○	○	23.8
#7001 - #7948	Extended workpiece coordinate offset data	○	○	23.9

No.	Data type or use	Reading	Setting	Section
#10001 - #10000+n #11001 - #11000+n #12001 - #12000+n #13001 - #13000+n #14001 - #14000+n #15001 - #15000+n #16001 - #16000+n #17001 - #17000+n #18001 - #18000+n #28001 - #28000+n #29001 - #29000+n	Tool offset data Also refer to "#2001" and later.	○	○	23.6
#40000 - #40097	Specification of the selected interfering object and interfering model coordinate system offset	○	○	23.24
#50000 - #50749 #51000 - #51749 #52000 - #52749	Data of user backup area for R device	○ (*1)	○ (*1)	23.22
#50000 - #51199	ZR device access variables (C80 series only)	○	○	23.25
#60000 - #63016	Tool life management	○	○ (*2)	23.7
#68000 - #68003	Tool management	○	○ (*2)	23.5
#68011 - #68023	Basic information	○	○ (*2)	
#68031 - #68040	Shape information	○	○	
#68051 - #68054	Cutting conditions	○	○	
#68061 - #68072	Additional information	○	○	
#68081 - #68088	Tool life	○	○	
#68101 - #68113	Compensation amount	○	○	
#100000	Parameter No. designation	-	○	23.19
#100001	Part system No. designation	-	○	
#100002	Axis No./spindle No. designation	-	○	
#100010	Parameter value read	○	-	
#100100	Device type designation	-	○	23.23
#100101	Device No. designation	-	○	
#100102	Number of read bytes designation	-	○	
#100103	Read bit designation	-	○	
#100110	Reading PLC data	○	-	

(*1) Only for MTB. This cannot be designated by the user.

(*2) Some numbers are not available depending on the contents.

23.2 System Variables (G Command Modal)

Detailed description

Using variable Nos. #4001 to #4021, it is possible to read the modal commands which have been issued in previous blocks.
Similarly, it is possible to read the modals in the block being executed with variable Nos. #4201 to #4221.

Variable No.		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 : G46
#4008	#4208	No variable No.	
#4009	#4209	Fixed cycle	G80 : 80, G70-G79 : 70-79, G83-G85 : 83-85, G83.2 : 83.2, G87-G89 : 87-89
#4010	#4210	Return level	G98 : 98, G99 : 99
#4011	#4211		
#4012	#4212	Workpiece coordinate system	G54-G59 : 54-59, G54.1:54.1
#4013	#4213	Acceleration/deceleration	G61-G64 : 61-64, G61.1 : 61.1
#4014	#4214	Macro modal call	G66 : 66, G66.1 : 66.1, G67 : 67
#4015	#4215		
#4016	#4216	No variable No.	
#4017	#4217	Constant surface speed	G96 : 96, G97 : 97
#4018	#4218	Balance cut	G14 : 14, G15 : 15
#4019	#4219		
#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
#  = #4201 ;   = ->Group 01  G modal (active) #2 = 0.0
G#1 W#24 ;
M99 ;
%
```

23.3 System Variables (Non-G Command Modal)

Detailed description

Using variable Nos. #4101 to #4120, it is possible to read the modal commands which have been issued in previous blocks.
Similarly, it is possible to read the modals in the block being executed with variable Nos. #4301 to #4320.

Variable No.		Modal information	Variable No.		Modal information
Pre-read block	Execution block		Pre-read	Execution	
#4101	#4301		#4111	#4311	Tool length, position compensation number H
#4102	#4302	2nd miscellaneous function B	#4112	#4312	
#4103	#4303		#4113	#4313	Miscellaneous function M
#4104	#4304		#4114	#4314	Sequence number N
#4105	#4305		#4115	#4315	Program number O (*1)
#4106	#4306		#4116	#4316	
#4107	#4307		#4117	#4317	
#4108	#4308		#4118	#4318	
#4109	#4309	Feedrate F	#4119	#4319	Spindle function S
#4110	#4310		#4120	#4320	Tool function T (*2)
			#4130	#4330	Extended workpiece coordinate system No. P

(*1) Programs are registered as files. When the program No. (file name) is read with #4115, #4315, the character string will be converted to a value.
(Example 1)
The file name "123" is the character string 0×31, 0×32, 0×33,
so the value will be (0×31-0×30)*100 + (0×32-0×30)*10 + (0×33-0×30) = 123.0.
Note that if the file name contains characters other than numbers, it will be "blank".
(Example 2)If the file name is "123ABC", it contains characters other than numbers, so the result will be "blank".

(*2) The tool number is read with omitted by the number of digits in the compensation number that is designated in the parameter "#1097 T1digt".

23.4 System Variables (Modal Information at Macro Interruption)

Detailed description

Modal information when control passes to the user macro interruption program can be known by reading system variables #4401 to #4520.
The unit specified with a command applies.

System variable	Modal information	
#4401 : #4421	G code (group01) : G code (group21)	Some groups are not used.
#4507	D code	
#4509	F code	
#4511	H code	
#4513	M code	
#4514	Sequence number N	
#4515	Program number O (*1)	
#4519	S code	
#4520	T code	

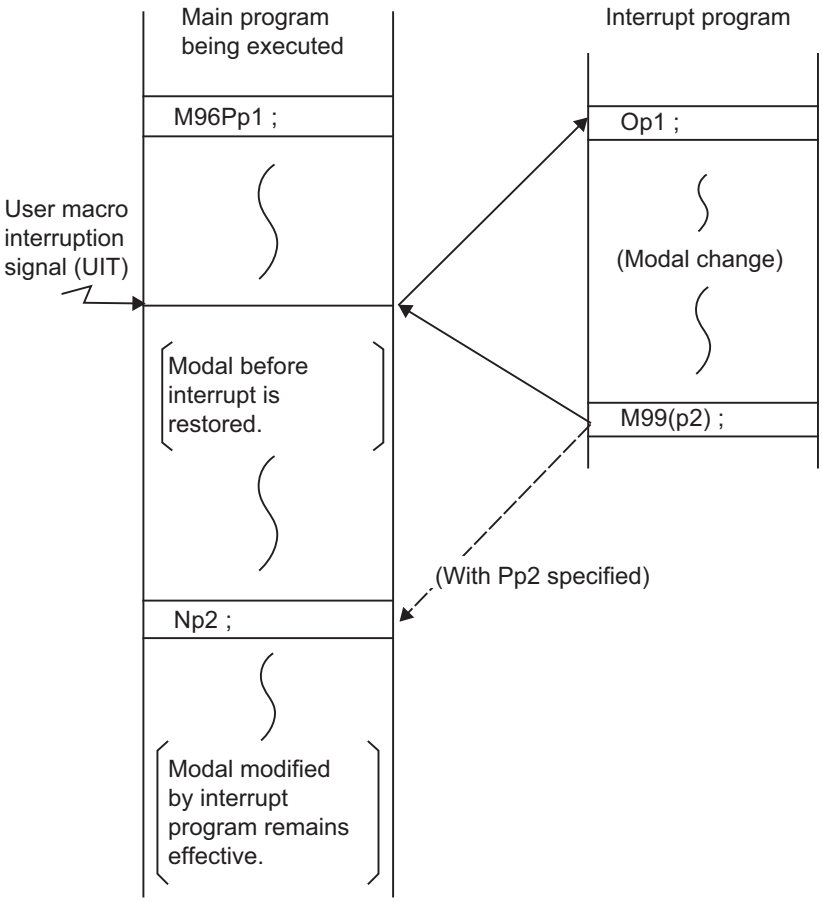
The above system variables are available only in the user macro interrupt program.
If they are used in other programs, program error (P241) will occur.

(*1) Programs are registered as files. When the program No. (file name) is read with #4515, the character string will be converted to a value.
(Example 1)
The file name "123" is the character string 0×31, 0×32, 0×33, so the value will be (0×31-0×30)*100 + (0×32-0×30)*10 + (0×33-0×30) = 123.0.
Note that if the file name contains characters other than numbers, it will be "blank".
(Example 2)
If the file name is "123ABC", it contains characters other than numbers, so the result will be "blank".

Modal information affected by user macro interruption

If modal information is changed by the interrupt program, it is handled as follows after control returns from the interrupt program to the main program.

Returning with M99;	The change of modal information by the interrupt program is invalidated and the original modal information is restored. With interrupt type 1, however, if the interrupt program contains a move or miscellaneous function (MSTB) command, the original modal information is not restored.
Returning with M99P__ ;	The original modal information is updated by the change in the interrupt program even after returning to the main program. This is the same as in returning with M99P__; from a program called by M98, etc.



Modal information affected by user macro interruption

23.5 System Variables (Tool Information)

Tool management (#68000 - #68003)

<table>
<tr><th>Variable No.</th><th colspan="2">Item / Description</th><th>Data range</th><th>Attribute</th></tr>
<tr><td>#68000</td><td>Tool designation method</td><td>Method to designate the tool to be read or written
1: In-use tool designation
2: Tool number designation
3: Tool management screen registration number designation</td><td>1 to 3</td><td>-/W</td></tr>
<tr><td>#68001</td><td>Tool selection No.</td><td>Designate the tool selection number that matches the setting of "#68000".
<table>
<tr><th>#68000</th><th>Details of "#68001"</th><th>Data range</th></tr>
<tr><td>1</td><td>ATC magazine number
(Used only when the ATC is added.)</td><td>0 to 5</td></tr>
<tr><td>2</td><td>Tool No. (T No.)
(Tool No. and compensation No. for lathe system)</td><td>1 to 99999999</td></tr>
<tr><td>3</td><td>Tool management screen registration number</td><td>1 to Number of managed tools</td></tr>
</table></td><td>Refer to the "Description" column.</td><td>-/W</td></tr>
<tr><td>#68003</td><td>Top vacant registration number on tool management screen</td><td>The tool number indicates a vacant line number.
0: No vacant registration number
1 to 999: Vacant registration number</td><td>0 to 999</td><td>R/-</td></tr>
</table>

- If you command to read data to a write only variable or write to a read only variable, a program error (P241) will occur.
- If a value exceeding the allowable range is issued, a program error (P35) will occur.

(1) Tool designation method (#68000), Tool selection number (#68001)

Substitute a value to the parameters "#68000" and "#68001" to designate the tool to be read and written with the parameters "#68011" to "#68111".

The tool designation methods are classified into three types as shown below.

Tool designation method	Details	"#68000" setting value	"#68001" setting value
In-use tool designation	Reads or writes tool management data of the tool in use.	1	ATC magazine No.
Tool number designation	Reads or writes tool management data designated with the tool number.	2	Tool No. (T No.)
Tool management screen registration number designation	Reads or writes tool management data designated with the registration number.	3	Tool management screen registration number

(a) In-use tool designation (#68000=1)

For the in-use tool, when the R register is checked in the order from "1" to "3" shown below, if the value designated in the R register is other than "0", it is judged to be the in-use tool number.

- Tool life management spindle tool number in machining center system (R12200: 1st part system to R12270: 8th part system)
- ATC spindle tool number (R10620: magazine 1 to R10660: magazine 5)
- T code data (R536)

"#68001" designates the ATC magazine number.

If ATC is not used, this item does not need to be designated.

The "#68001" setting value has the meanings shown below.

"#68001" setting value	Meaning
"0" or no "#680001" command	Magazine 1
1 to 5	Magazine 1 to magazine 5

Note

- The in-use tool is determined when "#68000=1" or "#68001" is commanded.
  To designate the tool which is exchanged after the in-use tool has been determined as an in-use tool, command "#68000=1" or "#68001" again.

(b) Tool number designation (#68000=2)

"#68001" designates the tool number.

In the lathe system, designate the T code (tool number and tool compensation number).

(c) Tool management screen registration number designation (#68000=3)

"#68001" designates the tool management screen registration number (line number).

Note

- If "#68000" is commanded multiple times, the last designation method will be valid.
- "#68000" and "#68001" are valid until they are reset. When the power is turned ON or when the system is reset, "0" is set.
- When #68000 is 2, and when there are multiple tools which have the same tool number and the same tool compensation number as the ones designated by "#68001", the tool that has been found first will be selected.
- A program error (P245) will occur when:

  "#68000" is not designated;

  "#68000=1 ;" is commanded while the in-use tool number is set to "0";

  "#68000=1 ;" is commanded while the in-use tool number is not registered on the tool management screen;

  "#68000=2 ;" is commanded while a read/write command is issued using "#68011" to "#68111" without commanding "#68001";

  a tool not registered on the tool management screen with "#68011" is designated during the "#68000=2 ;" command;

  a write command is issued with "#68011" during the "#68000=2 ;" command;

  "#68000=3 ;" is commanded while a read/write command is issued using "#68011" to "#68111" without commanding "#68001";

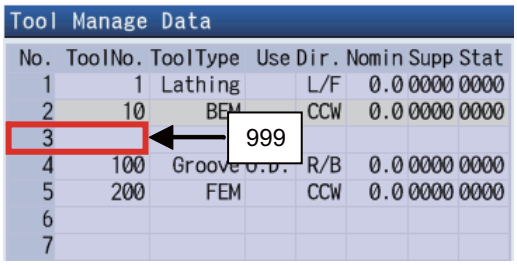
  "#68001=0 ;" is commanded.

(2) Top vacant registration number on tool management screen (#68003)

Designating this value reads the top vacant registration number on the tool management screen with "#68003".

Use example:

Follow the procedure below to measure the compensation amount with the measurement macro, etc. and search for and register a vacant registration number when registering a new tool.



[Measurement macro program]

:

#68000 = 3 ;

:	Measurement
#68001=#68003 ;	Searches for a vacant registration number (No.3 in the example above), and designates registration number 3.
#68011=999 ;	Sets "999" to the tool management data "tool number" of tool management screen registration number 3.

Note

- If no vacant registration number is found because all numbers are registered, "0" is set when "#68003" is read out.
  When "#68001=#68003;", "#68001" is set to "0", and a program error (P245) will occur.

Basic information ("#68011" to "#68023")

Variable No.	Item / Description		Data range	Attri-bute
#68011	Tool No.		0 to 99999999	R/W
#68012	Name		Eight one-byte alphanu-meric characters	R/W
#68013	Type	0: No setting 1: Ball end mill 2: Flat end mill 3: Drill 4: Radius end mill 5: Chamfering 6: Tapping 7: Face mill 51: Turning 52: Slotting 53: Thread cutting 54: Turning drill 55: Turning tap	0 to 7, 51 to 55	R/W
#68014	Usage	0: No setting 1: External diameter 2: Internal diameter 3: Face	0 to 3	R/W
#68015	Direction: hand/ro-tation	<Mill tool, turning drill, turning tap> 0: CW 1: CCW 2: CW 3 :CW <Turning, slotting, thread cutting> 0: Right hand / Front 1: Left hand / Front 2: Right hand / Rear 3: Left hand / Rear	0 to 3	R/W
#68016	Call		0.0 to 999.9 (mm) 0.00 to 99.99 (inch)	R/W
#68017	Number of blades		0 to 9	R/W
#68018	Tool ID		Eight one-byte alphanu-meric characters	R/W
#68019	Supplementary information		0 to 65535	R/W
#68020	Conditions		0 to 65535	R/-
#68021	Mounting angle		0.0 to 359.999 (degree)	R/W
#68023	Comb-shaped cutter offset J		±9999.999 (mm) ±999.9999 (inch)	R/W

♦If a value exceeding the allowable range is issued, a program error (P35) will occur.

(1) Tool No. ("#68011")

The registered tool cannot be registered. If a tool is registered, the operation will be performed as shown below.

Type		Operation performed when a registered tool is designated
Machining center system	Life management I	Program error (P245)
	Life management II	Program error (P245)
	Life management III	Program error (P245)
	The life management specifications are invalid.	Program error (P245)
Lathe system	Life management I	Program error (P245)
	Life management II	Can be registered.
	The life management specifications are invalid.	Can be registered.

Example: When an attempt is made to change tool management data "tool number" of No.3 (3rd line) from "11" to "1" in life management II of the machining center system, the setting is as follows.

Tool Manage Data

No.	ToolNo.	ToolType	Use	Dir.	Nomin	Supp	Stat
1	1	FEM		CCW	0.0	0000	0000
2	2	BEM		CCW	0.0	0000	0000
3	11	L-drill		CW	0.0	0000	0000
4	12	Lathing	O.D.	R/F	0.0	0000	0000

#68000=3 Tool management screen registration number designation

#68001=3 Designates No. 3 (3rd line)

#68013=1 Tool No. 1 is already registered with No. 1 (1st line), causing a program error (P245).

(2) Tool name ("#68012"), Tool ID ("#68018"), Material ("#68053")

(a) Read

Reads data only with the variable No. designation of the DPRNT command.

Example 1: DPRNT [#68012] ; The tool name is read.

Example 2: #100=#68012 ; A program error (P243) will occur.

(b) Write

A string can be designated by enclosing it in parentheses ( ).

Example 1: #68012=(M-TOOL1) ; Data is written up to the number of valid characters, and the rest is ignored.

Example 2: #68012=#0 ; A string is cleared by writing "null" characters.

Example 3: #68012= M-TOOL1 ; If parentheses are omitted, a program error will occur.

(3) Type ("#68013") to tool nose point P ("#68111")

A program error will occur in the following case.

Operation	Operation result
Type ("#68013") to tool nose point P ("#68111") is read or written for the registration number with the tool number unspecified.	Program error (P245)

(4) Compensation amount ("#68103" to "#68111")

A program error will occur in the following case.

Operation	Operation result
The compensation amount ("#68103" to "#68111") is read or written for the tool with the compensation number unspecified.	Program error (P170)

(5) Tool life ("#68082" to "#68086")
A program error will occur in the following case.

Operation	Operation result
The tool life ("#68082" to "#68086") is read or written for the tool with the tool life group number unspecified in tool life management I and II of the machining center system, or in tool life management II of the lathe system.	Program error (P179)

Shape information ("#68031" to "#68040")

Variable No.	Item / Description		Data range	Attri-bute
#68031 to #68039	Tool shapes A to I		Length: 0 to 9999.999 (mm) 0 to 999.9999 (inch) Angle: 0 to 180.000 (degree)	R/W
#68040	Tool color	1: Gray 2: Red 3: Yellow 4: Blue 5: Green 6: Light blue 7: Purple 8: Pink	1 to 8	R/W

♦If a value exceeding the allowable range is issued, a program error (P35) will occur.

Cutting conditions ("#68051" to "#68054")

Variable No.	Item / Description	Data range	Attri-bute
#68051	Spindle rotation speed S	0 to 99999999	R/W
#68052	Feedrate F	0 to 1000000 (mm/min) 0 to 100000 (inch/min)	R/W
#68053	Material	Four one-byte alphanu-meric characters	R/W
#68054	Coolant M code	0 to 99999999	R/W

♦If a value exceeding the allowable range is issued, a program error (P35) will occur.

Additional information ("#68061" to "#68072")

Variable No.	Item / Description	Data range	Attri-bute
#68061 to #68066	Customize 1 to 6	±999999999 (*1)	R/W
#68067 to #68072	Customize 7 to 12	±9999.999 (*1)	R/W

(*1) For customize data 1 to 12, the data range varies depending on the data format.

♦If a value exceeding the allowable range is issued, a program error (P35) will occur.

Tool life ("#68081" to "#68088")

Variable No.	Item / Description		Attribute
	Life management I	Life management II	
#68081	(Not used)	Group No. (0 to 9999)	R/W
#68082	Status A (0 to 2)	Status (0 to 3)	R/W
#68083	(Not used)	Method (0 to 1)	R/W
#68084	Status B (0 to 99)	(Not used)	R/W
#68085	Life time (0 to 5999 min.)	Life time / Number of uses until life limit (0 to 999999 min. / 0 to 999999 sets)	R/W
#68086	Usage time (0 to 5999 min.)	Usage time / Number of uses (0 to 999999 min. / 0 to 999999 sets)	R/W
#68087	Number of lives (0 to 65000 sets)	(Not used)	R/W
#68088	Number of uses (0 to 65000 sets)	(Not used)	R/W

- If an unused variable is commanded, a program error (P241) will occur.
- If a value exceeding the allowable range is issued, a program error (P35) will occur.

Compensation amount ("#68101" to "#68113")

Variable No.	Item / Description			Attri-bute
	Compensation type I	Compensation type II	Compensation type III	
#68101	No. H (0 to number of tool offset sets)	No. H (0 to number of tool offset sets)	Tool length compensation No. (0 to number of tool offset sets)	R/W
#68102	(Not used)	No. D (0 to number of tool offset sets)	Wear compensation No. (0 to number of tool offset sets)	R/W
#68103	Tool length (±9999.999999 (mm) ±999.9999999 (inch))	Length dimension (±9999.999999 (mm) ±999.9999999 (inch))	Tool length X (±9999.999999 (mm) ±999.9999999 (inch))	R/W
#68104	(Not used)	(Not used)	Tool length Z (±9999.999999 (mm) ±999.9999999 (inch))	R/W
#68105	(Not used)	(Not used)	Additional axis tool length (±9999.999999 (mm) ±999.9999999 (inch))	R/W
#68106	(Not used)	Length wear (±9999.999999 (mm) ±999.9999999 (inch))	Wear X (±9999.999999 (mm) ±999.9999999 (inch))	R/W
#68107	(Not used)	(Not used)	Wear Z (±9999.999999 (mm) ±999.9999999 (inch))	R/W
#68108	(Not used)	(Not used)	Additional axis wear (±9999.999999 (mm) ±999.9999999 (inch))	R/W
#68109	(Not used)	Radius dimension (±9999.999999 (mm) ±999.9999999 (inch))	Tool nose radius (±9999.999999 (mm) ±999.9999999 (inch))	R/W
#68110	(Not used)	Radius wear (±9999.999999 (mm) ±999.9999999 (inch))	Radius wear (±9999.999999 (mm) ±999.9999999 (inch))	R/W
#68111	(Not used)	(Not used)	Tool nose point P (0 to 9)	R/W
#68112	(Not used)	(Not used)	2nd additional axis tool length (±9999.999999 (mm) ±999.9999999 (inch))	R/W
#68113	(Not used)	(Not used)	2nd additional axis wear (±9999.999999 (mm) ±999.9999999 (inch))	R/W

♦If an unused variable is commanded, a program error (P241) will occur.
♦If a value exceeding the allowable range is issued, a program error (P35) will occur.

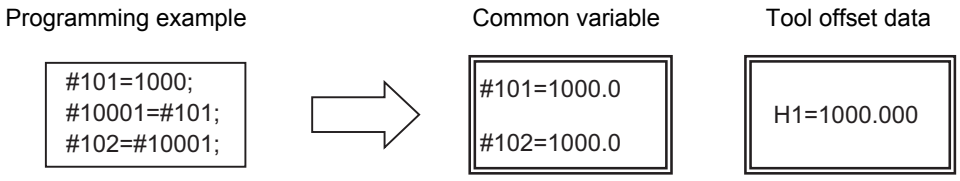
23.6 System Variables (Tool Offset)

Detailed description

This function reads or sets tool offset data using a variable No.

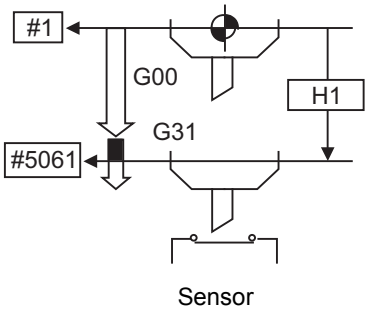
Variable number range			Description
	#1120 TofVal = 0	#1120 TofVal = 1	
#10001 to #10000+n	#2001 to #2000+n	#2701 to #2700+n	X shape compensation amount
#11001 to #11000+n	#2701 to #2700+n	#2001 to #2000+n	X wear compensation amount
#12001 to #12000+n			Additional axis shape compensation amount
#13001 to #13000+n			Additional axis wear compensation amount
#28001 to #28000+n			Tool length compensation amount for the 2nd additional axis
#29001 to #29000+n			Tool wear compensation amount for the 2nd additional axis
#14001 to #14000+n	#2101 to #2100+n	#2801 to #2800+n	Z shape compensation amount
#15001 to #15000+n	#2801 to #2800+n	#2101 to #2100+n	Z wear compensation amount
#16001 to #16000+n	#2201 to #2200+n	#2901 to #2900+n	R shape compensation amount
#17001 to #17000+n	#2901 to #2900+n	#2201 to #2200+n	R wear compensation amount
#18001 to #18000+n	#2301 to #2300+n		Nose compensation amount

"n" in the table corresponds to the tool No. Maximum "n" value is the number of tool compensation sets.
The #10000s and #2000s are equivalent functions.
The tool offset data has a decimal point in the same way as other variables.
If "#10001=1000;" is programmed, "1000.000" is set to tool offset data.
The additional axes' tool compensation can be used for only either the 3rd axis or 4th axis.
Whether to use either one depends on the MTB specifications (parameter "#1520 Tchg34)").
The variable No. corresponding to the #2000s' shape/wear compensation amount can be changed with the parameter "#1120 TofVal".



Example 1: Tool offset data measurement example

G28X0 T0101 ;	Reference position return
M06;	Tool change (T0101)
#1=#5001;	Start point memory
G00 X-200 ;	Rapid traverse to safe position
G31 X-50.F100 ;	Skip measurement
#10001=#5061-#1;	Measurement distance calculation and tool offset data set



Note

(1) In (Example 1), no consideration is given to the delay in the skip sensor signal.
#5001 is the X axis start point position and #5061 indicates the position at which the skip signal is input while G31 is being executed in the X axis skip coordinates.

(2) When multi-part system
There are cases when each part system has tool data, and cases when the part systems share common tool data. This can be selected with parameter (#1051 MemTol).
Parameter #1051 MemTol
0: Tool data is provided for each part system.
1: Common tool data is shared by part systems.
When the tool data is shared by the part systems, the read amount and substitution value for all part systems' tool compensation variables will be the same (when the same variable No. is commanded).

(3) Whether to enable or disable the tool compensation for the 2nd additional axis function and which axis is to be used as the 2nd additional axis vary depending on the MTB specifications (parameters "#12103 2nd add T-ofs ON" and "#12104 2nd add T-ofs set").

23.7 System Variables (Tool Life Management)

Detailed description

Definition of variable Nos.

(1) Group number designation
#60000
Assign the value to this variable No. to designate the group number of the tool life management data to be read with parameters "#60001" to "#63016". If a group No. is not designated, the data of the group registered first is read. This is valid until reset.

(2) Tool life management system variable No. (Read)
#60001 to #63016
| a | b | c | d | e |

| a | : "6" Fix (Tool life management)

| b | c | : Details of data classification

Data class	Details	Remarks
00	For control	Refer by data types
05	Group No.	Refer by registration No.
10	Tool No.	Refer by registration No.
15	Method	Refer by registration No.
20	Status	Refer by registration No.
25	Life time/No. of uses	Refer by registration No.
30	Usage time/No. of uses	Refer by registration No.

The group No., method, and life data are common for the group.

| d | e | : Registration No. or data type

Registration No.

1 to 16

Data type

Type	Details
1	Number of registered tools
2	Life current value
3	Tool selection No.
4	Number of remaining registered tools
5	Execution signal
6	Cutting time cumulative value (min)
7	Life end signal
8	Life prediction signal

List of variables

Variable No.	Item	Type	Details	Data range
60001	Number of registered tools	Common to system	Total number of tools registered in each group.	0 to 80
60002	Life current value	For each group (*1)	Usage time / Number of uses of in-use tool Usage data of the in-use tool (total of usage data for each compensation number when multiple compensation numbers are used for the tool)	0 to 999999 min. 0 to 999999 sets
60003	Tool selection No.		In-use tool registration number Registration number of the selected tool in the designated group (the first tool of ST:1 if the selected tool is not registered, the first tool of ST:0 if ST:1 is not defined, or the last tool if all tools have reached the end of their lives)	0 to 16
60004	Number of remaining registered tools		Total number of "usable" tools in the group Number of tools registered in designated group, whose ST is 0: Not used.	0 to 16
60005	Execution signal		"1" when this group is used in the program being executed. "1" when the tool in the designated group is selected.	0/1
60006	Cutting time cumulative value (min)		Indicates the time that this group is used in the program being executed.	
60007	Life end signal		"1" when lives of all tools in this group have expired. "1" when all registered tools in the designated group reach the end of their lives.	0/1
60008	Life prediction signal		"1" when a new tool is selected with the next command in this group. "1" when there are no tools in use (ST: 1) while there is an unused tool (ST: 0) in the designated group.	0/1
60500 +***	Group No.	For each group or registration number (*2)	This group's No.	1 to 9999
61000 +***	Tool No.		Tool No. and compensation No. of the designated tool Tool No. + compensation No. (When tool No. = 22 and compensation No. = 01, 2201=899H)	0 to 9999
61500 +***	Method		Whether to manage this group's life as time or number of lives. 0 : Time, 1: Number of lives	0/1
62000 +***	Status		Tool usage state 0 : Unused tool 1 : In-use tool 2 : Normal life tool 3 : Tool skip tool	0 to 3
62500 +***	Life time/No. of uses		This group's tool life value	0 to 999999 min. 0 to 999999 sets
63000 +***	Usage time/No. of uses			0 to 999999 min. 0 to 999999 sets

(*1) Designate group number "#60000".

(*2) Designate group number "#60000" / registration number***.
However, group number / method / life is data common to groups.

Program example

(1) Normal commands

#101 = #60001 ;	Reads the number of registered tools.
#102 = #60002 ;	Reads the life current value.
#103 = #60003 ;	Reads the tool selection No.
#60000 = 10 ;	Designates the group No. of the life data to be read. Designated group No. is valid until reset.
#104 = #60004 ;	Reads the remaining number of registered tools in group 10.
#105 = #60005 ;	Reads the signal being executed in group 10.
#111 = #61001 ;	Reads the group 10, #1 tool No.
#112 = #62001 ;	Reads the group 10, #1 status.
#113 = #61002 ;	Reads the group 10, #2 tool No.
%	

(2) When the group number is not designated:

#104 = #60004 ;	Reads the remaining number of registered tools in the first registered group.
#111 = #61001 ;	Reads the #1 tool No. in the first registered group.
%	

(3) When an unregistered group number is designated (group 9999 does not exist):

#60000 = 9999 ;	Designates the group No.
#104 = #60004 ;	#104 = -1.

(4) When an unused registration number is designated (15 tools for group 10):

#60000 = 10 ;	Designates the group No.
#111 = #61016 ;	#111 = -1.

(5) When a registration number not defined in the specifications is designated:

#60000 = 10 ;	
#111 = #61017 ;	Program error (P241)

(6) When tool life management data is registered with the G10 command after a group number has been designated:

#60000 = 10 ;	Designates the group No.
G10 L3 ;	Starts the life management data registration. The group 10 life data is registered through the commands from G10 to G11.
P10 LLn NNn ;	10 is the group No., Ln is the life per tool, Nn is the method.
TTn ;	"Tn" is the tool No.
:	
G11;	Registers data in group 10 with the G10 command.
#111 = #61001 ;	Reads the group 10, #1 tool No.
G10 L3 ;	Starts the life management data registration. The life data other than group 10 is registered from G10 to G11.
P1 LLn NNn ;	1 is the group No., "Ln" is the life per tool, "Nn" is the method.
TTn ;	"Tn" is the tool No.
:	
G11;	Registers the life data with the G10 command. (The registered data is deleted.)
#111 = #61001 ;	Group 10 does not exist. #111 = -1.

Precautions

(1) If the tool life management system variable is commanded without designating a group No., the data of the group registered at the head of the registered data will be read.

(2) If a non-registered group No. is designated and the tool life management system variable is commanded, "-1" will be read as the data.

(3) If an unused registration No. tool life management system variable is commanded, "-1" will be read as the data.

(4) Once commanded, the group No. is valid until NC reset.

(5) When the tool life management I specifications are provided, the tool life management system variable specifications are not provided. A program error (P241) will occur if commanded.

23.8 System Variables (Workpiece Coordinate Offset)

Detailed description

By using variable Nos #5201 to #532n, it is possible to read out the workpiece coordinate system compensation data or to substitute values.

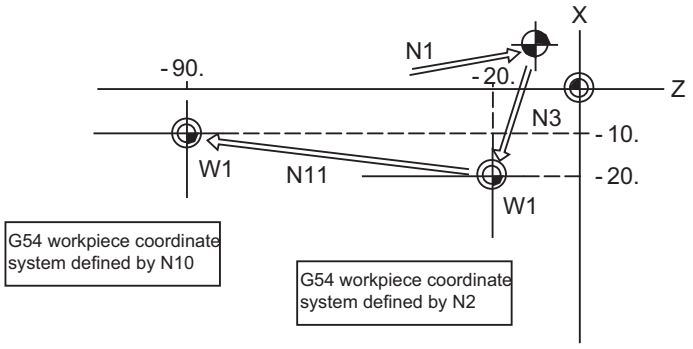
Note

(1) The number of controllable axes varies depending on the specifications.
The last digit of the variable No. corresponds to the control axis No.

Coordinate name	1st axis	2nd axis	3rd axis	4th axis	.....	nth axis	Remarks
External workpiece offset	#5201	#5202	#5203	#5204	.....	#520n	External workpiece offset specifications are required.
G54	#5221	#5222	#5223	#5224	.....	#522n	Workpiece coordinate system offset specifications are required.
G55	#5241	#5242	#5243	#5244	.....	#524n	
G56	#5261	#5262	#5263	#5264	.....	#526n	
G57	#5281	#5282	#5283	#5284	.....	#528n	
G58	#5301	#5302	#5303	#5304	.....	#530n	
G59	#5321	#5322	#5323	#5324	.....	#532n	

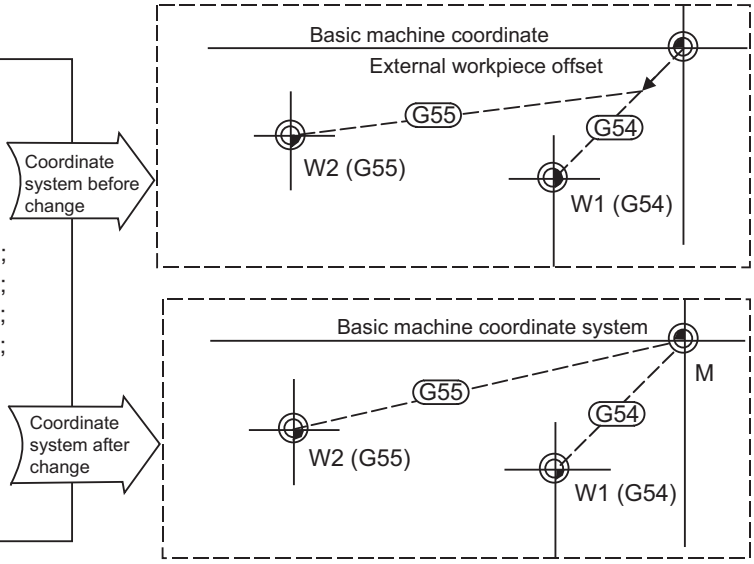
(Example 1)

```
N1 G28 X0  Z0 ;
N2 #5221=-20. #5222= - 20. ;
N3 G00 G54 X0 Z0 ;
      ~
N10 #5221= -10. #5222= - 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 compensation values are added to the workpiece coordinate (G54, G55) system compensation values without changing the position of the workpiece coordinate systems.

23.9 System Variables (Extended Workpiece Coordinate Offset)

Detailed description

#7001 to #794n (48-set specification)

By using variable Nos #7001 to #794n, it is possible to read out the extended workpiece coordinate system compensation data or to substitute values.

Note

(1) The system variables #7001 to #794n are available up to the valid number of sets. The last digit of the variable No. corresponds to the control axis No.

Table 1 of syst to em variables for extended workpiece coordinate system compensation (n=1 to 8)

	1st axis to nth axis
P1	#7001 to #700n
P2	#7021 to #702n
P3	#7041 to #704n
P4	#7061 to #706n
P5	#7081 to #708n
P6	#7101 to #710n
P7	#7121 to #712n
P8	#7141 to #714n
P9	#7161 to #716n
P10	#7181 to #718n
P11	#7201 to #720n
P12	#7221 to #722n
P13	#7241 to #724n
P14	#7261 to #726n
P15	#7281 to #728n
P16	#7301 to #730n
P17	#7321 to #732n
P18	#7341 to #734n
P19	#7361 to #736n
P20	#7381 to #738n
P21	#7401 to #740n
P22	#7421 to #742n
P23	#7441 to #744n
P24	#7461 to #746n

	1st axis to nth axis
P25	#7481 to #748n
P26	#7501 to #750n
P27	#7521 to #752n
P28	#7541 to #754n
P29	#7561 to #756n
P30	#7581 to #758n
P31	#7601 to #760n
P32	#7621 to #762n
P33	#7641 to #764n
P34	#7661 to #766n
P35	#7681 to #768n
P36	#7701 to #770n
P37	#7721 to #772n
P38	#7741 to #774n
P39	#7761 to #776n
P40	#7781 to #778n
P41	#7801 to #780n
P42	#7821 to #782n
P43	#7841 to #784n
P44	#7861 to #786n
P45	#7881 to #788n
P46	#7901 to #790n
P47	#7921 to #792n
P48	#7941 to #794n

23.10 System Variables (External Workpiece Coordinate Offset / Workpiece Coordinate System Shift)

Detailed description

External workpiece coordinate system offset

The workpiece coordinate system compensation amount can be read using variables #2501 and #2601.
By substituting a value in these variable Nos., the workpiece coordinate system compensation amount can be changed.

System variable No.	External workpiece coordinate system offset amount
#2501	1st axis
#2601	2nd axis

Workpiece coordinate system shift

When the workpiece coordinate system shift function is enabled, "#2501" and "#2601" are used to acquire or set the workpiece coordinate system shift amount (parameter "#11056 Workshift invalid").

System variable No.	Workpiece coordinate system shift amount
#2501	1st axis
#2601	2nd axis

23.11 System Variables (Position Information)

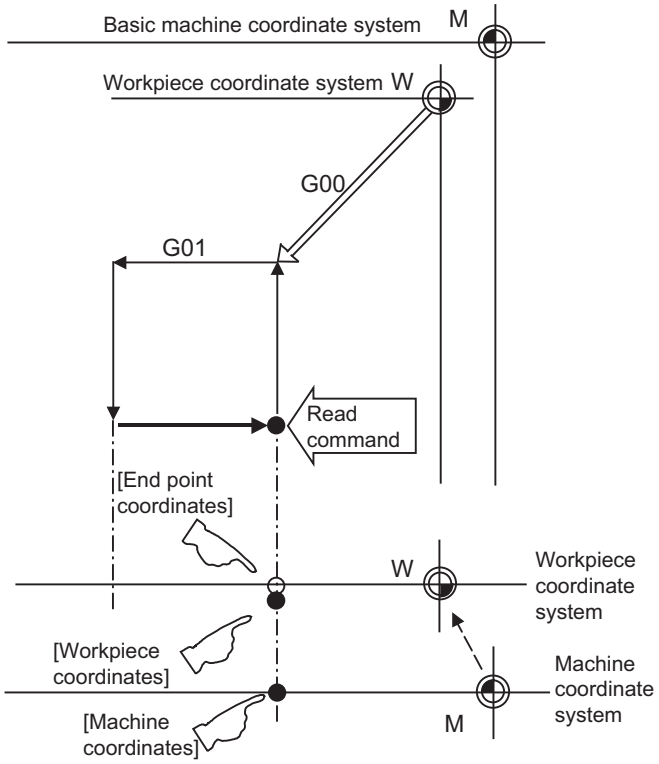
Detailed description

Using variable Nos. #5001 to #5140+n, it is possible to read the end point coordinates, machine coordinates, workpiece coordinates, skip coordinates, and servo deviation amounts in the last block.

Position information	Axis No.						Reading during movement
	1	2	3	4	...	n	
End point coordinate of the last block	#5001	#5002	#5003	#5004	...	#5000+n	Valid
Machine coordinate	#5021	#5022	#5023	#5024	...	#5020+n	Invalid
Workpiece coordinate	#5041	#5042	#5043	#5044	...	#5040+n	Invalid
Skip coordinate	#5061	#5062	#5063	#5064	...	#5060+n	Valid
Servo deviation amount	#5101	#5102	#5103	#5104	...	#5100+n	Valid
Macro interruption stop Start point coordinates	#5121	#5122	#5123	#5124	...	#5120+n	Valid
Macro interruption stop End point coordinates	#5141	#5142	#5143	#5144	...	#5140+n	Valid

Note

- The number of axes which can be controlled differs according to the specifications.
  The last digit of the variable No. corresponds to the control axis No.

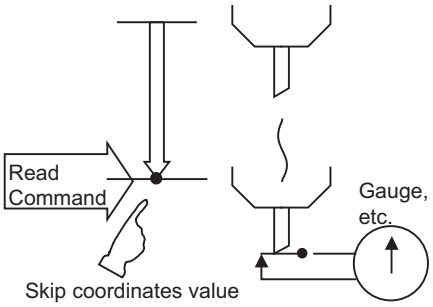


- During Simple inclined surface control (G176), the coordinates that are relative to the feature coordinate system are set to the system variables for position information (#5001 to #5100+n (excluding #5021 to #5021+n)). The coordinates that are relative to the machine coordinate system are set to the variables #5021 to #5021+n (machine coordinate values) even during Simple inclined surface control.

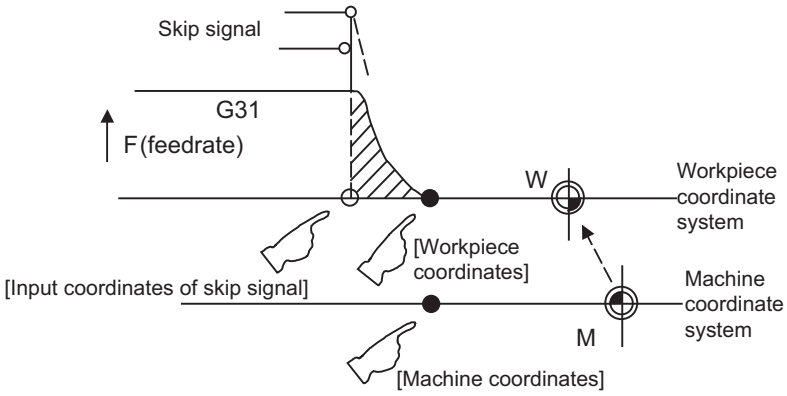
(1) The positions of the end point coordinates and skip coordinates are positions in the workpiece coordinate system.

(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 the workpiece coordinates.

(3) The skip coordinates indicates the position where the skip signal is turned ON in the G31 block. If the skip signal does not turn ON. they will be the end point position.
(For further details, refer to the section on Automatic Tool Length Measurement.)



(4) The end point coordinates indicate the tool nose position regardless of the tool compensation and other such factors. On the other hand, the machine coordinates, workpiece coordinates and skip coordinates indicate the tool reference point position with consideration given to tool compensation.



For "●", check stop and then proceed to read.
For "○", reading is possible during movement.

Note

- The skip signal input coordinates value is the position in the workpiece coordinate system. The coordinate value in variable Nos. #5061 to #5060+n memorize the moments when the skip input signal during movement was input and so they can be read at any subsequent time.
  For details, refer to "22.2 Skip Function ; G31".
- When the parameter "#1366 skipExTyp" (Multi-part system simultaneous skip command) is set to "1", the skip coordinate value will be "0", even if G31 command is given in one-part system or G31 command is given in only one of the multiple part systems.

23.12 System Variables (Alarm)

Detailed description

The NC unit can be forcibly set to the alarm state by using variable No. #3000.

#3000= 70 (CALL #PROGRAMMER #TEL #530) ;

70	Alarm No.
CALL #PROGRAMMER #TEL #530	Alarm message

Any alarm number from 1 to 9999 can be specified.
The alarm message must be written in 31 or less characters.
NC alarm 3 signal (program error) is output.
The "P277: MACRO ALM MESG" appears in the <ALARM> column on "DIAG 1." screen and the alarm message " (CALL #PROGRAMMER #TEL #530)" and the alarm No. (70) will appear in the <Operator massage>.

Example of program (alarm when #1 = 0)

```
    ≀
IF[#1 NE 0]GOTO 100 ;
#3000=70 ( CALL #PROGRAMMER #TEL #530 );

N100
    ≀
```

Stops with NC alarm

```
<ALARM> DIAG 1.

  P277: MACRO ALM MESG

<Operator message>

   CALL #PROGRAMMER #TEL #530 70
```

Note

(1) If zero or any number greater than 9999 is specified for the alarm No., the number will be invalid and it will not display. However, the operation will be in the alarm status, and the specified alarm message will appear.

(2) Specify the alarm message by enclosing it in round parentheses after the alarm number. If there is any character string between the number and the alarm message enclosed in round parentheses, the alarm message will be invalid and it will not display. However, the operation will be in the alarm status, and the specified alarm No. will appear.

(3) When 32 or more characters are specified for the alarm message, characters after the 32nd character will not display.

(4) Spaces included in an alarm message character string are ignored, and will not display. To split the character string insert a character such as "." (period).

23.13 System Variables (Message Display and Stop)

Detailed description

By using variable No. #3006, the operation stops after the previous block is executed and, if message display data is commanded, the corresponding message and the stop No. will be indicated on the operator message area.

#3006 = 1(TAKE FIVE);

1 to 9999	Stop No. (When Nos. other than 1 - 9999 are set, the command will be invalidated.)
TAKE FIVE	Message (Nothing will be displayed if no message is designated.)

The message should be written in 31 or less characters and should be enclosed by round parentheses.

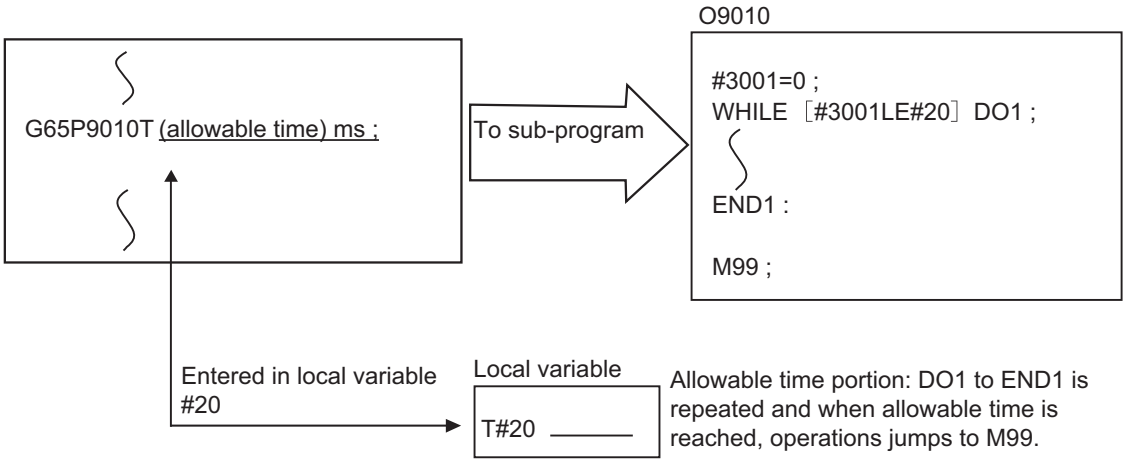
23.14 System Variables (Cumulative Time)

Detailed description

The integrating time during the power is turned ON or the automatic start is running, can be read or values can be substituted by using variable Nos. #3001 and #3002.

Type	Variable No.	Unit	Contents when power is switched on	Initialization of contents	Count condition
Power-on	3001	1ms	Same as when power is switched off	Substitute values to variables	At all times while power is ON
Automatic start	3002				In-automatic start

The cumulative time is reset to "0" at approximately 2.44×10^{11}ms (approximately 7.7 years).



23.15 System Variables (Time Read Variables)

Detailed description

The following operations can be carried out using the system variable extension for the user macro time.

(1) By adding time information system variable #3011 and #3012, the current date (#3011) and current time (#3012) can be read and written.

(2) By adding parameter #1273/bit1, the unit (millisecond unit/hour unit) of the system variable "#3002" (cumulative time during automatic start) can be changed.

Variable No.	Details
#3001	The cumulative time during power ON can be read and the value can be substituted. The unit is millisecond.
#3002	The cumulative time during automatic start can be read and the value can be substituted. The unit can be changed between millisecond and hour with parameter #1273/bit1.
#3011	The current date can be read and written. YYYY/MM/DD is read as a YYYYMMDD value. If a value "YYYYMMDD" is written, it is set to YY/MM/DD (the year is indicated by the last two digits). Command range for year/month/day setting: Year (YYYY): 2000 to 2099 Month (MM): 1 to 12 Day (DD): 1 to maximum number of days in one month
#3012	The current time can be read and written. HH/MM/SS is read as a value "HHMMSS". When a value "HHMMSS" is written in, it will be set as HH/MM/SS. Command range for time setting: Hour (HH): 0 to 23 (24-hour system) Minute (MM): 0 to 59 Second (SS): 0 to 59

(3) The cumulative time is reset to "0" at approximately 2.44×10^{11}ms (approximately 7.7 years).

(4) If a negative value or a value exceeding 244335917226 milliseconds (67871.08811851 hours for #3002 time designation) is set for the cumulative time, a program error (P35) will occur.

(5) If a value exceeding the command range is set for the date or time, a program error (P35) will occur.

(6) Always set the month/date/hour/minute/second as a two-digit value when setting the date and time.
If the value only has one digit, always add 0.
(February 14, 2001 => #3011= 20010214 ;, etc.)

Program example

Example of use (#3011, #3012)

(Example 1) To read the current date (February 14, 2001) in common variable #100
#100 = #3011 ; (20010214 is inserted in #100)

(Example 2) To write current time (18 hours, 13 minutes, 6 seconds) into system variable #3012
#3012 = 181306 ; (The command value cumulative time #2: time is set to 18:13:06.)

(Example 3) By setting the following program example, the machining start/end time (year/month/date/hour/minute/second) can be viewed.

```
#100=#3011 ; => Machining start year/month/date
#101=#3012 ; => Machining start hour/minute/second
G28 X0 Y0 Z0 ;

G92 ;
G0 X50. ;
    :
    :
    :
#102=#3011 ;    => Machining end year/month/date
#103=#3012 ;    => Machining end hour/minute/second
M30 ;
```

Precautions

Limits and precautions for using time reading variable

(1) #3011 reads the date as an eight-digit value, so the difference between the two dates read in will not be the difference of days.

(2) #3012 reads the time as a six-digit value, so the difference between the two times read in will not be the difference of hours.

23.16 System Variables (Machining Information)

Detailed description

Contents of variable No. "#3003"

By substituting the values below in variable No. #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 (FIN) signal.

#3003/bit	Function	Set to "1"	Set to "0"
0	Inhibition of single block stop	Inhibits stop.	Does not inhibit stop.
1	Inhibition of miscellaneous function complete signal waiting	Does not wait for the signal.	Waits for the signal.
2	Prohibition of program check reverse run	Prohibits reverse run.	Allows reverse run.
3	Synchronization between part systems OFF	Invalid	Valid
4	(Not used)	-	-
5	(Not used)	-	-
6	(Not used)	-	-
7	(Not used)	-	-

Note

(1) Variable No. #3003 is set to "0" by reset.

(2) For details on the "Synchronization between part systems" function, refer to "16.8 Synchronization between Part Systems".

Contents of variable No. "#3004"

By substituting the values below in variable No. #3004, it is possible to make the feed hold, feedrate override and G09 functions either valid or invalid in the subsequent blocks.

#3004/bit	Function	Set to "1"	Set to "0"
0	Automatic operation pause OFF	Invalid	Valid
1	Cutting override OFF	Invalid	Valid
2	G09 check OFF	Invalid	Valid
3	(Not used)	-	-
4	Dry run invalid	Invalid	Valid
5	(Not used)	-	-
6	(Not used)	-	-
7	(Not used)	-	-

Note

(1) Variable No. #3004 is set to "0" by reset.

(2) The functions are valid when the above bits are "0", and invalid when they are "1".

(3) When the feed hold is set to invalid with #3004, the following will occur when the feed hold switch is pressed.

- During thread cutting, block stop will be carried out at the end of the next block of the block where thread cutting is completed.
- During tapping with tap cycle, block stop will be carried out after the operation of R point return.
- In the case other than above, block stop will be carried out after the termination of the currently executing block.

23.17 System Variables (Number of Workpiece Machining Times)

Detailed description

The number of workpiece machining times can be read using variables #3901 and #3902.
By substituting a value in these variable Nos., the number of workpiece machining times can be changed.

Variable No.	Type	Data setting range
#3901	Number of workpiece machining times	0 to 999999
#3902	Maximum workpiece value	

Note

(1) The number of workpiece machining times must be a positive value.

23.18 System Variables (Mirror Image)

Detailed description

By reading variable No. #3007, it is possible to ascertain the status of mirror image of the each axis at the point.
The axis corresponds to each bit of "#3007" as shown below.
0: Mirror image invalid
1: Mirror image valid
The number of axes varies depending on your machine's specifications.

#3007

Bit	15	14	13	12	11	10	9	8	7	6	5	4	3	2	1	0
nth axis									8	7	6	5	4	3	2	1

23.19 System Variables (Parameter Reading)

Function and purpose

Parameter data can be read in with the system variables.

Variable No.	Application
#100000	Parameter No. designation
#100001	Part system No. designation
#100002	Axis No./spindle No. designation
#100010	Parameter value read

Detailed description

The parameter values are read in with the following four blocks using these four system variables.

#100000 = 1001 ;	Designates the parameter No.
#100001 = 1 ;	Designates the part system No.
#100002 = 1 ;	Designates the axis No./spindle No.
#100 = #100010;	Reads the parameter value.

Parameter No. designation (#100000)

The parameter to be read in is designated by substituting the parameter No. in this system variable.
If the parameters are read without designating this No., the parameters will be read in the same manner as if the minimum parameter No. (#1) is designated. Once designated, the setting is held until the parameter No. is designated again or until it is reset.
A program error (P39) will occur if a nonexistent parameter No. is set.

Part system No. designation (#100001)

(1) System variable for part system No. designation
The part system No. of the parameter to be read in is designated by substituting an index value for this system variable. This designation will be ignored when reading in parameters that are not in a specific part system.
If the parameters are read without designating this No., the parameters will be read in the same manner as if the index value 0 (part system in running program) is designated. Once designated, the setting is held until the part system No. is designated again or until it is reset.
A program error (P39) will occur if a nonexistent part system No. is set.

(2) Index values

Index values	Parameters per part system
0	Running part system
1	1st part system
2	2nd part system
3	-
:	-
9	-
10	PLC axis

Axis No. /spindle No. designation (#100002)

(1) System variable for axis or spindle No. designation
The axis or spindle No. of the parameter to be read in is designated by substituting an index value for this system variable. This designation will be ignored when reading in parameters that are neither for a specific axis nor spindle.
The axis parameter index value is the value set in the part system designated with #100001.
Thus, when reading parameters that are not in the designated part system, the part system No. must be designated again.
The spindle parameter's index value is not affected by the part system designation.
If the parameters are read without designating this number, the parameters will be read in the same manner as when the index value 1 (1st axis/1st spindle in the designated part system) is designated. Once designated, the setting is held until the index value is designated again or until it is reset.
A program error (P39) will occur if a nonexistent axis/spindle No. is set.

(2) Index values

Index values	Axis parameter	Spindle parameter
1	1st axis	1st spindle
2	2nd axis	2nd spindle
3	3rd axis	3rd spindle
4	4th axis	4th spindle
5	5th axis	-
6	6th axis	-

Reading the parameters (#100010)

The designated parameter data is read with this system variable.
Data to be read as follows, depending on the parameter type.

Type	Read in data
Numeric value	The values displayed on the Parameter screen are output.
Text	ASCII codes are converted into decimal values.

Program example

(1) To read the parameter "#1002 axisno (number of axes)" for each part system:

#100000 = 1002 ;	Designates [#1002].
#100001 = 1 ;	Designates [1st part system].
#101 = #100010;	Reads the number of axes in 1st part system.
#100000 = 1002 ;	Designates [#1002]. (can be omitted since parameter No. is same)
#100001 = 2 ;	Designates [2nd part system].
#102 = #100010;	Reads the number of axes in 2nd part system.
#100001 = 5 ;	Designates [5th part system]. (The program error (P39) will occur.)
#100001 = 10 ;	Designates [PLC axis].
#110 = #100010;	Reads the number of PLC axes.

(2) To read the axis parameter "#2037 G53ofs (#1 reference position)":

[Conditions]	1 part systems		2 part systems	
	<1st axis>	<2nd axis>	<1st axis>	<2nd axis>
#2037 G53ofs	100.000	200.000	300.000	400.000

[1st part system program]

#100002 = 1 ;	Designates [1st axis].
#100000 = 2037 ;	Designates [#2037].
#101 = #100010;	Reads the [#1 reference point] for the 1st axis. (#101=100.000)
#100002 = 2 ;	Designates [2nd axis].
#102 = #100010;	Reads the [#2 reference point] for the 1st axis. (#102=200.000)
#100001 = 2 ;	Designates [2nd part system].
#100002 = 1 ;	Designates [1st axis].
#201 = #100010;	Reads the [#2 reference position] for the 1st axis in the 1st part system. (#201=300.000)

[2nd part system program]

#100002 = 1 ;	Designates [1st axis].
#100000 = 2037 ;	Designates [#2037].
#101 = #100010;	Reads the [#1 reference point] for the 1st axis. (#101=300.000)
#100002 = 2 ;	Designates [2nd axis].
#102 = #100010;	Reads the [#2 reference point] for the 1st axis. (#102=400.000)
#100001 = 1 ;	Designates [1st part system].
#100002 = 1 ;	Designates [1st axis].
#201 = #100010;	Reads the [#1 reference position] for the 1st axis in the 1st part system. (#201=100.000)

(3) To read the parameter for each part system, axis, or spindle:

#100002 = 1 ;	Designates [1st spindle].
#100000 = 3001 ;	Designates [#3001].
#101 = #100010;	Reads the [#3001 slimt1 (Number of limit rotation gears 00)] for 1st spindle.
#100000 = 3002 ;	Designates [#3002].
#102 = #100010;	Reads the [#3002 slimt2 (Number of limit rotation gears 01)] for 1st spindle.
#100002 = 2 ;	Designates [2nd spindle].
#100000 = 3001 ;	Designates [#3001].
#201 = #100010 ;	Reads the [#3001 slimt1 (Number of limit rotation gears 00)] for 2nd spindle.
#100000 = 3002 ;	Designates [#3002].
#202 = #100010;	Reads the [#3002 slimt2 (Number of limit rotation gears 01)] for 2nd spindle.

(4) To read the text type parameter "#1169 system name" (part system name):

[Conditions]	<1st part system>	<2nd part system>
#1169 system name	SYS1	SYS2

#100000 = 1169 ;	Designates #1169.
#100001 = 1 ;	Designates 1st part system.
#101 = #100010;	This will be #101 = 1398362929 (0x53595331).

Precautions

(1) The number of part systems, axes and spindles is set at the maximum number specified by the model.

(2) The inch/metric changeover function for the setting and display is valid for the readout data.

(3) The machining condition parameter group cannot set the parameters from the program using the G10 command, and cannot read the parameters using the system variables ("#100000" and later).

23.20 System Variables (Macro Interface Input (PLC -> NC))

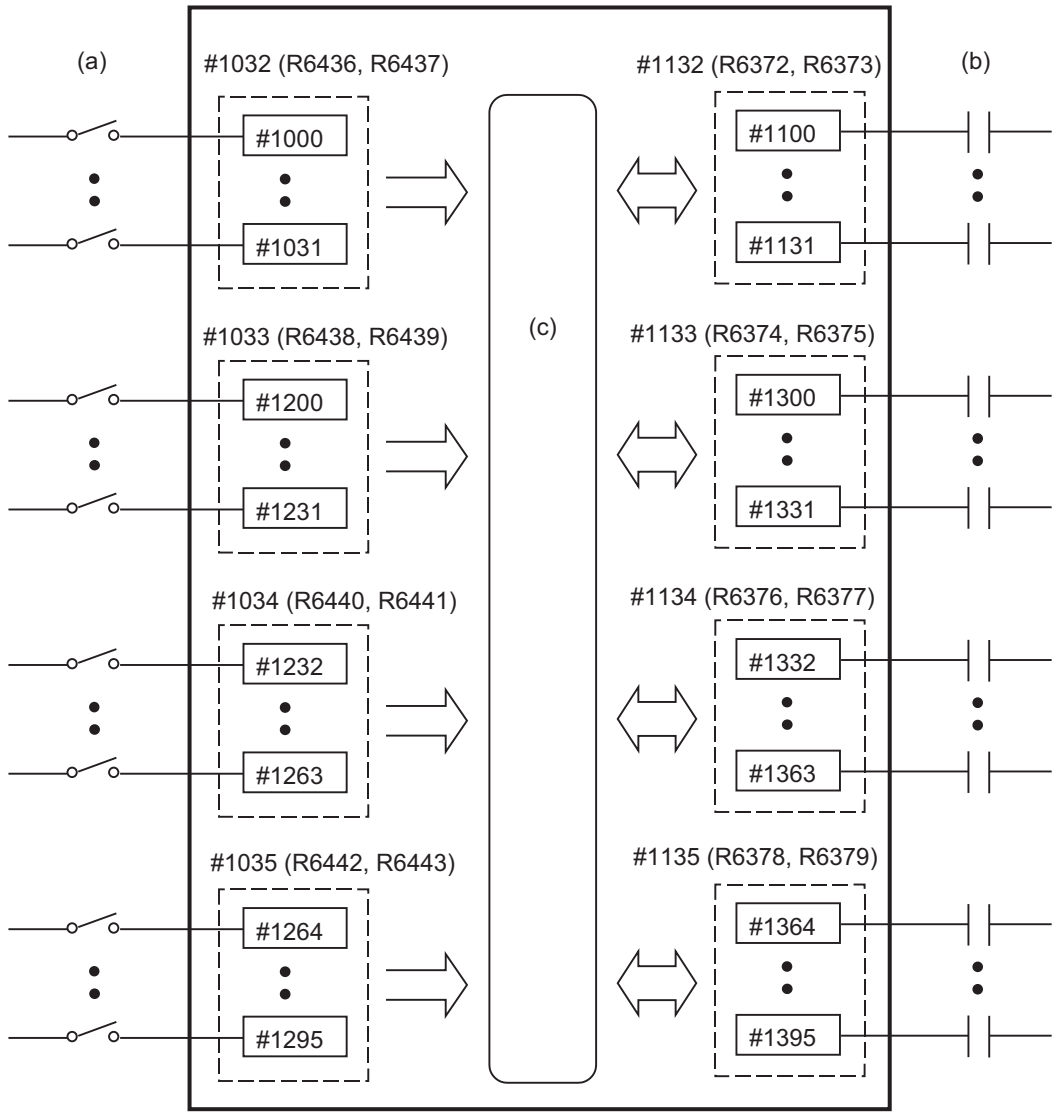
Function and purpose

The status of the interface input signals can be ascertained by reading out the values of variable Nos. #1000 to #1035, #1200 to #1295.

Note

- The interface output signals can be sent by substituting values in variable Nos. #1100 to #1135, #1300 to #1395. (For details of the system variables for the output signals, refer to "23.21 System Variables (Macro Interface Output (NC -> PLC))".)

Example of 1st part system



(a) Input signal (b) Output signal (c) Macro instructions

Detailed description

Variable Nos. #1000 to #1035, #1200 to #1295 are for readout only, and nothing can be placed in the left side member of their operation formula.
Input here refers to input to the NC.
Whether it is per part system or common between part systems depends on the MTB specifications (parameter "#1230 set02/bit07").

Data unit (32 bits)

All the input signals from #1000 to #1031 can be read at once by reading out the value of variable No. #1032.
The input signals from #1200 to #1231, #1232 to #1263, and #1264 to #1295 can be read by reading out the values of variable Nos. #1033 to #1035.
The data of the 1st part system ($1) to the 8th part system ($8) is as follows.

System variable	No. of points	Interface input signal							
		$1	$2	$3	$4	$5	$6	$7	$8
#1032	32	R6436, R6437	R6444, R6445	R6452, R6453	R6460, R6461	R6468, R6469	R6476, R6477	R6484, R6485	R6492, R6493
#1033	32	R6438, R6439	R6446, R6447	R6454, R6455	R6462, R6463	R6470, R6471	R6478, R6479	R6486, R6487	R6494, R6495
#1034	32	R6440, R6441	R6448, R6449	R6456, R6457	R6464, R6465	R6472, R6473	R6480, R6481	R6488, R6489	R6496, R6497
#1035	32	R6442, R6443	R6450, R6451	R6458, R6459	R6466, R6467	R6474, R6475	R6482, R6483	R6490, R6491	R6498, R6499

Bit unit

The input signal has only two values: "0" and "1".

Part system	$1	$2	$3	$4	$5	$6	$7	$8
R device	R6436-R6443	R6444-R6451	R6452-R6459	R6460-R6467	R6468-R6475	R6476-R6483	R6484-R6491	R6492-R6499

If the value is common between part systems, refer to the column of the 1st part system ($1).

System variable	No. of points	Interface input signal register							
		$1	$2	$3	$4	$5	$6	$7	$8
#1000	1	R6436/bit0	R6444/bit0	R6452/bit0	R6460/bit0	R6468/bit0	R6476/bit0	R6484/bit0	R6492/bit0
#1001	1	bit1	bit1	bit1	bit1	bit1	bit1	bit1	bit1
#1002	1	bit2	bit2	bit2	bit2	bit2	bit2	bit2	bit2
#1003	1	bit3	bit3	bit3	bit3	bit3	bit3	bit3	bit3
#1004	1	bit4	bit4	bit4	bit4	bit4	bit4	bit4	bit4
#1005	1	bit5	bit5	bit5	bit5	bit5	bit5	bit5	bit5
#1006	1	bit6	bit6	bit6	bit6	bit6	bit6	bit6	bit6
#1007	1	bit7	bit7	bit7	bit7	bit7	bit7	bit7	bit7
#1008	1	bit8	bit8	bit8	bit8	bit8	bit8	bit8	bit8
#1009	1	bit9	bit9	bit9	bit9	bit9	bit9	bit9	bit9
#1010	1	bit10	bit10	bit10	bit10	bit10	bit10	bit10	bit10
#1011	1	bit11	bit11	bit11	bit11	bit11	bit11	bit11	bit11
#1012	1	bit12	bit12	bit12	bit12	bit12	bit12	bit12	bit12
#1013	1	bit13	bit13	bit13	bit13	bit13	bit13	bit13	bit13
#1014	1	bit14	bit14	bit14	bit14	bit14	bit14	bit14	bit14
#1015	1	bit15	bit15	bit15	bit15	bit15	bit15	bit15	bit15
#1016	1	R6437/bit0	R6445/bit0	R6453/bit0	R6461/bit0	R6469/bit0	R6477/bit0	R6485/bit0	R6493/bit0
#1017	1	bit1	bit1	bit1	bit1	bit1	bit1	bit1	bit1
#1018	1	bit2	bit2	bit2	bit2	bit2	bit2	bit2	bit2
#1019	1	bit3	bit3	bit3	bit3	bit3	bit3	bit3	bit3
#1020	1	bit4	bit4	bit4	bit4	bit4	bit4	bit4	bit4
#1021	1	bit5	bit5	bit5	bit5	bit5	bit5	bit5	bit5
#1022	1	bit6	bit6	bit6	bit6	bit6	bit6	bit6	bit6
#1023	1	bit7	bit7	bit7	bit7	bit7	bit7	bit7	bit7
#1024	1	bit8	bit8	bit8	bit8	bit8	bit8	bit8	bit8
#1025	1	bit9	bit9	bit9	bit9	bit9	bit9	bit9	bit9
#1026	1	bit10	bit10	bit10	bit10	bit10	bit10	bit10	bit10
#1027	1	bit11	bit11	bit11	bit11	bit11	bit11	bit11	bit11
#1028	1	bit12	bit12	bit12	bit12	bit12	bit12	bit12	bit12
#1029	1	bit13	bit13	bit13	bit13	bit13	bit13	bit13	bit13
#1030	1	bit14	bit14	bit14	bit14	bit14	bit14	bit14	bit14
#1031	1	bit15	bit15	bit15	bit15	bit15	bit15	bit15	bit15

System variable	No. of points	Interface input signal register							
		$1	$2	$3	$4	$5	$6	$7	$8
#1200	1	R6438/ bit0	R6446/ bit0	R6454/ bit0	R6462/ bit0	R6470/ bit0	R6478/ bit0	R6486/ bit0	R6494/ bit0
#1201	1	bit1	bit1	bit1	bit1	bit1	bit1	bit1	bit1
#1202	1	bit2	bit2	bit2	bit2	bit2	bit2	bit2	bit2
#1203	1	bit3	bit3	bit3	bit3	bit3	bit3	bit3	bit3
#1204	1	bit4	bit4	bit4	bit4	bit4	bit4	bit4	bit4
#1205	1	bit5	bit5	bit5	bit5	bit5	bit5	bit5	bit5
#1206	1	bit6	bit6	bit6	bit6	bit6	bit6	bit6	bit6
#1207	1	bit7	bit7	bit7	bit7	bit7	bit7	bit7	bit7
#1208	1	bit8	bit8	bit8	bit8	bit8	bit8	bit8	bit8
#1209	1	bit9	bit9	bit9	bit9	bit9	bit9	bit9	bit9
#1210	1	bit10	bit10	bit10	bit10	bit10	bit10	bit10	bit10
#1211	1	bit11	bit11	bit11	bit11	bit11	bit11	bit11	bit11
#1212	1	bit12	bit12	bit12	bit12	bit12	bit12	bit12	bit12
#1213	1	bit13	bit13	bit13	bit13	bit13	bit13	bit13	bit13
#1214	1	bit14	bit14	bit14	bit14	bit14	bit14	bit14	bit14
#1215	1	bit15	bit15	bit15	bit15	bit15	bit15	bit15	bit15
#1216	1	R6439/ bit0	R6447/ bit0	R6455/ bit0	R6463/ bit0	R6471/ bit0	R6479/ bit0	R6487/ bit0	R6495/ bit0
#1217	1	bit1	bit1	bit1	bit1	bit1	bit1	bit1	bit1
#1218	1	bit2	bit2	bit2	bit2	bit2	bit2	bit2	bit2
#1219	1	bit3	bit3	bit3	bit3	bit3	bit3	bit3	bit3
#1220	1	bit4	bit4	bit4	bit4	bit4	bit4	bit4	bit4
#1221	1	bit5	bit5	bit5	bit5	bit5	bit5	bit5	bit5
#1222	1	bit6	bit6	bit6	bit6	bit6	bit6	bit6	bit6
#1223	1	bit7	bit7	bit7	bit7	bit7	bit7	bit7	bit7
#1224	1	bit8	bit8	bit8	bit8	bit8	bit8	bit8	bit8
#1225	1	bit9	bit9	bit9	bit9	bit9	bit9	bit9	bit9
#1226	1	bit10	bit10	bit10	bit10	bit10	bit10	bit10	bit10
#1227	1	bit11	bit11	bit11	bit11	bit11	bit11	bit11	bit11
#1228	1	bit12	bit12	bit12	bit12	bit12	bit12	bit12	bit12
#1229	1	bit13	bit13	bit13	bit13	bit13	bit13	bit13	bit13
#1230	1	bit14	bit14	bit14	bit14	bit14	bit14	bit14	bit14
#1231	1	bit15	bit15	bit15	bit15	bit15	bit15	bit15	bit15

System variable	No. of points	Interface input signal register							
		$1	$2	$3	$4	$5	$6	$7	$8
#1232	1	R6440/ bit0	R6440/ bit0	R6448/ bit0	R6456/ bit0	R6472/ bit0	R6480/ bit0	R6488/ bit0	R6496/ bit0
#1233	1	bit1	bit1	bit1	bit1	bit1	bit1	bit1	bit1
#1234	1	bit2	bit2	bit2	bit2	bit2	bit2	bit2	bit2
#1235	1	bit3	bit3	bit3	bit3	bit3	bit3	bit3	bit3
#1236	1	bit4	bit4	bit4	bit4	bit4	bit4	bit4	bit4
#1237	1	bit5	bit5	bit5	bit5	bit5	bit5	bit5	bit5
#1238	1	bit6	bit6	bit6	bit6	bit6	bit6	bit6	bit6
#1239	1	bit7	bit7	bit7	bit7	bit7	bit7	bit7	bit7
#1240	1	bit8	bit8	bit8	bit8	bit8	bit8	bit8	bit8
#1241	1	bit9	bit9	bit9	bit9	bit9	bit9	bit9	bit9
#1242	1	bit10	bit10	bit10	bit10	bit10	bit10	bit10	bit10
#1243	1	bit11	bit11	bit11	bit11	bit11	bit11	bit11	bit11
#1244	1	bit12	bit12	bit12	bit12	bit12	bit12	bit12	bit12
#1245	1	bit13	bit13	bit13	bit13	bit13	bit13	bit13	bit13
#1246	1	bit14	bit14	bit14	bit14	bit14	bit14	bit14	bit14
#1247	1	bit15	bit15	bit15	bit15	bit15	bit15	bit15	bit15
#1248	1	R6441/ bit0	R6441/ bit0	R6449/ bit0	R6457/ bit0	R6473/ bit0	R6481/ bit0	R6489/ bit0	R6497/ bit0
#1249	1	bit1	bit1	bit1	bit1	bit1	bit1	bit1	bit1
#1250	1	bit2	bit2	bit2	bit2	bit2	bit2	bit2	bit2
#1251	1	bit3	bit3	bit3	bit3	bit3	bit3	bit3	bit3
#1252	1	bit4	bit4	bit4	bit4	bit4	bit4	bit4	bit4
#1253	1	bit5	bit5	bit5	bit5	bit5	bit5	bit5	bit5
#1254	1	bit6	bit6	bit6	bit6	bit6	bit6	bit6	bit6
#1255	1	bit7	bit7	bit7	bit7	bit7	bit7	bit7	bit7
#1256	1	bit8	bit8	bit8	bit8	bit8	bit8	bit8	bit8
#1257	1	bit9	bit9	bit9	bit9	bit9	bit9	bit9	bit9
#1258	1	bit10	bit10	bit10	bit10	bit10	bit10	bit10	bit10
#1259	1	bit11	bit11	bit11	bit11	bit11	bit11	bit11	bit11
#1260	1	bit12	bit12	bit12	bit12	bit12	bit12	bit12	bit12
#1261	1	bit13	bit13	bit13	bit13	bit13	bit13	bit13	bit13
#1262	1	bit14	bit14	bit14	bit14	bit14	bit14	bit14	bit14
#1263	1	bit15	bit15	bit15	bit15	bit15	bit15	bit15	bit15

System variable	No. of points	Interface input signal register							
		$1	$2	$3	$4	$5	$6	$7	$8
#1264	1	R6442/ bit0	R6450/ bit0	R6458/ bit0	R6466/ bit0	R6474/ bit0	R6482/ bit0	R6490/ bit0	R6498/ bit0
#1265	1	bit1	bit1	bit1	bit1	bit1	bit1	bit1	bit1
#1266	1	bit2	bit2	bit2	bit2	bit2	bit2	bit2	bit2
#1267	1	bit3	bit3	bit3	bit3	bit3	bit3	bit3	bit3
#1268	1	bit4	bit4	bit4	bit4	bit4	bit4	bit4	bit4
#1269	1	bit5	bit5	bit5	bit5	bit5	bit5	bit5	bit5
#1270	1	bit6	bit6	bit6	bit6	bit6	bit6	bit6	bit6
#1271	1	bit7	bit7	bit7	bit7	bit7	bit7	bit7	bit7
#1272	1	bit8	bit8	bit8	bit8	bit8	bit8	bit8	bit8
#1273	1	bit9	bit9	bit9	bit9	bit9	bit9	bit9	bit9
#1274	1	bit10	bit10	bit10	bit10	bit10	bit10	bit10	bit10
#1275	1	bit11	bit11	bit11	bit11	bit11	bit11	bit11	bit11
#1276	1	bit12	bit12	bit12	bit12	bit12	bit12	bit12	bit12
#1277	1	bit13	bit13	bit13	bit13	bit13	bit13	bit13	bit13
#1278	1	bit14	bit14	bit14	bit14	bit14	bit14	bit14	bit14
#1279	1	bit15	bit15	bit15	bit15	bit15	bit15	bit15	bit15
#1280	1	R6443/ bit0	R6451/ bit0	R6459/ bit0	R6467/ bit0	R6475/ bit0	R6483/ bit0	R6491/ bit0	R6499/ bit0
#1281	1	bit1	bit1	bit1	bit1	bit1	bit1	bit1	bit1
#1282	1	bit2	bit2	bit2	bit2	bit2	bit2	bit2	bit2
#1283	1	bit3	bit3	bit3	bit3	bit3	bit3	bit3	bit3
#1284	1	bit4	bit4	bit4	bit4	bit4	bit4	bit4	bit4
#1285	1	bit5	bit5	bit5	bit5	bit5	bit5	bit5	bit5
#1286	1	bit6	bit6	bit6	bit6	bit6	bit6	bit6	bit6
#1287	1	bit7	bit7	bit7	bit7	bit7	bit7	bit7	bit7
#1288	1	bit8	bit8	bit8	bit8	bit8	bit8	bit8	bit8
#1289	1	bit9	bit9	bit9	bit9	bit9	bit9	bit9	bit9
#1290	1	bit10	bit10	bit10	bit10	bit10	bit10	bit10	bit10
#1291	1	bit11	bit11	bit11	bit11	bit11	bit11	bit11	bit11
#1292	1	bit12	bit12	bit12	bit12	bit12	bit12	bit12	bit12
#1293	1	bit13	bit13	bit13	bit13	bit13	bit13	bit13	bit13
#1294	1	bit14	bit14	bit14	bit14	bit14	bit14	bit14	bit14
#1295	1	bit15	bit15	bit15	bit15	bit15	bit15	bit15	bit15

23.21 System Variables (Macro Interface Output (NC -> PLC))

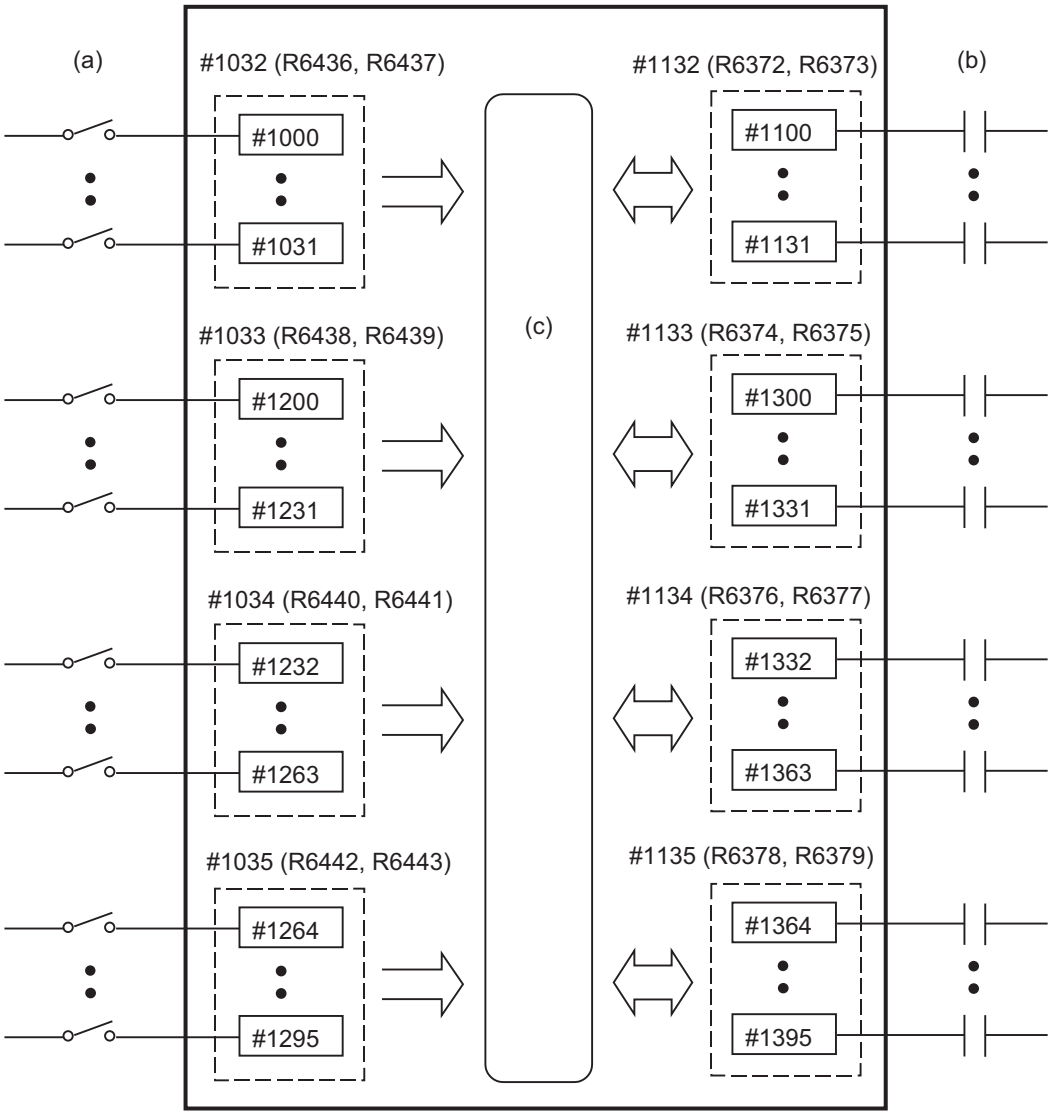
Function and purpose

The interface output signals can be sent by substituting values in variable Nos. #1100 to #1135, #1300 to #1395.

Note

- The status of the interface input signals can be ascertained by reading out the values of variable Nos. #1000 to #1035, #1200 to #1295. (For details of the system variables for the output signals, refer to "23.20 System Variables (Macro Interface Input (PLC -> NC))".)

Example of 1st part system



(a) Input signal (b) Output signal (c) Macro instructions

Detailed description

The status of the writing and output signals can be read in order to compensate the #1100 to #1135, #1300 to #1395 output signals.
Output here refers to the output from the NC side.
Whether it is per part system or common between part systems depends on the MTB specifications (parameter "#1230 set02/bit07").

Note

(1) The last values of the system variables #1100 to #1135, #1300 to #1395 sent are retained as 1 or 0. (They are not cleared even by resetting.)

(2) The following applies when any number except 1 or 0 is substituted into #1100 to #1131, #1300 to #1395.
<Blank> is treated as 0. All values other than <blank> or "0" are treated as 1.
Any value less than 0.00000001 is indefinite.

Data unit (32 bits)

All the output Nos. from #1100 to #1131 can be sent at once by substituting a value in variable No. #1132.
The output signals from #1300 to #1331, #1332 to #1363, and #1364 to #1395 can be sent by substituting a value in variable Nos. #1133 to #1135. (2^0 to 2^{31})
The data of the 1st part system ($1) to the 8th part system ($8) is as follows.

System variable	No. of points	Interface output signal							
		$1	$2	$3	$4	$5	$6	$7	$8
#1132	32	R6372, R6373	R6380, R6381	R6388, R6389	R6396, R6397	R6404, R6405	R6412, R6413	R6420, R6421	R6428, R6429
#1133	32	R6374, R6375	R6382, R6383	R6390, R6391	R6398, R6399	R6406, R6407	R6414, R6415	R6422, R6423	R6430, R6431
#1134	32	R6376, R6377	R6384, R6385	R6392, R6393	R6400, R6401	R6408, R6409	R6416, R6417	R6424, R6425	R6432, R6433
#1135	32	R6378, R6379	R6386, R6387	R6394, R6395	R6402, R6403	R6410, R6411	R6418, R6419	R6426, R6427	R6434, R6435

Bit unit

The output signal has only two values: "0" and "1".

Part system	$1	$2	$3	$4	$5	$6	$7	$8
R device	R6372-R6379	R6380-R6387	R6388-R6395	R6396-R6403	R6404-R6411	R6412-R6419	R6420-R6427	R6428-R6435

If the value is common between part systems, refer to the column of the 1st part system ($1).

System variable	No. of points	Interface output signal register							
		$1	$2	$3	$4	$5	$6	$7	$8
#1100	1	R6372/bit0	R6380/bit0	R6388/bit0	R6396/bit0	R6404/bit0	R6412/bit0	R6420/bit0	R6428/bit0
#1101	1	bit1	bit1	bit1	bit1	bit1	bit1	bit1	bit1
#1102	1	bit2	bit2	bit2	bit2	bit2	bit2	bit2	bit2
#1103	1	bit3	bit3	bit3	bit3	bit3	bit3	bit3	bit3
#1104	1	bit4	bit4	bit4	bit4	bit4	bit4	bit4	bit4
#1105	1	bit5	bit5	bit5	bit5	bit5	bit5	bit5	bit5
#1106	1	bit6	bit6	bit6	bit6	bit6	bit6	bit6	bit6
#1107	1	bit7	bit7	bit7	bit7	bit7	bit7	bit7	bit7
#1108	1	bit8	bit8	bit8	bit8	bit8	bit8	bit8	bit8
#1109	1	bit9	bit9	bit9	bit9	bit9	bit9	bit9	bit9
#1110	1	bit10	bit10	bit10	bit10	bit10	bit10	bit10	bit10
#1111	1	bit11	bit11	bit11	bit11	bit11	bit11	bit11	bit11
#1112	1	bit12	bit12	bit12	bit12	bit12	bit12	bit12	bit12
#1113	1	bit13	bit13	bit13	bit13	bit13	bit13	bit13	bit13
#1114	1	bit14	bit14	bit14	bit14	bit14	bit14	bit14	bit14
#1115	1	bit15	bit15	bit15	bit15	bit15	bit15	bit15	bit15
#1116	1	R6373/bit0	R6381/bit0	R6389/bit0	R6397/bit0	R6405/bit0	R6413/bit0	R6421/bit0	R6429/bit0
#1117	1	bit1	bit1	bit1	bit1	bit1	bit1	bit1	bit1
#1118	1	bit2	bit2	bit2	bit2	bit2	bit2	bit2	bit2
#1119	1	bit3	bit3	bit3	bit3	bit3	bit3	bit3	bit3
#1120	1	bit4	bit4	bit4	bit4	bit4	bit4	bit4	bit4
#1121	1	bit5	bit5	bit5	bit5	bit5	bit5	bit5	bit5
#1122	1	bit6	bit6	bit6	bit6	bit6	bit6	bit6	bit6
#1123	1	bit7	bit7	bit7	bit7	bit7	bit7	bit7	bit7
#1124	1	bit8	bit8	bit8	bit8	bit8	bit8	bit8	bit8
#1125	1	bit9	bit9	bit9	bit9	bit9	bit9	bit9	bit9
#1126	1	bit10	bit10	bit10	bit10	bit10	bit10	bit10	bit10
#1127	1	bit11	bit11	bit11	bit11	bit11	bit11	bit11	bit11
#1128	1	bit12	bit12	bit12	bit12	bit12	bit12	bit12	bit12
#1129	1	bit13	bit13	bit13	bit13	bit13	bit13	bit13	bit13
#1130	1	bit14	bit14	bit14	bit14	bit14	bit14	bit14	bit14
#1131	1	bit15	bit15	bit15	bit15	bit15	bit15	bit15	bit15

System variable	No. of points	Interface output signal register							
		$1	$2	$3	$4	$5	$6	$7	$8
#1300	1	R6374/ bit0	R6382/ bit0	R6390/ bit0	R6398/ bit0	R6406/ bit0	R6414/ bit0	R6422/ bit0	R6430/ bit0
#1301	1	bit1	bit1	bit1	bit1	bit1	bit1	bit1	bit1
#1302	1	bit2	bit2	bit2	bit2	bit2	bit2	bit2	bit2
#1303	1	bit3	bit3	bit3	bit3	bit3	bit3	bit3	bit3
#1304	1	bit4	bit4	bit4	bit4	bit4	bit4	bit4	bit4
#1305	1	bit5	bit5	bit5	bit5	bit5	bit5	bit5	bit5
#1306	1	bit6	bit6	bit6	bit6	bit6	bit6	bit6	bit6
#1307	1	bit7	bit7	bit7	bit7	bit7	bit7	bit7	bit7
#1308	1	bit8	bit8	bit8	bit8	bit8	bit8	bit8	bit8
#1309	1	bit9	bit9	bit9	bit9	bit9	bit9	bit9	bit9
#1310	1	bit10	bit10	bit10	bit10	bit10	bit10	bit10	bit10
#1311	1	bit11	bit11	bit11	bit11	bit11	bit11	bit11	bit11
#1312	1	bit12	bit12	bit12	bit12	bit12	bit12	bit12	bit12
#1313	1	bit13	bit13	bit13	bit13	bit13	bit13	bit13	bit13
#1314	1	bit14	bit14	bit14	bit14	bit14	bit14	bit14	bit14
#1315	1	bit15	bit15	bit15	bit15	bit15	bit15	bit15	bit15
#1316	1	R6375/ bit0	R6383/ bit0	R6391/ bit0	R6399/ bit0	R6407/ bit0	R6415/ bit0	R6423/ bit0	R6431/ bit0
#1317	1	bit1	bit1	bit1	bit1	bit1	bit1	bit1	bit1
#1318	1	bit2	bit2	bit2	bit2	bit2	bit2	bit2	bit2
#1319	1	bit3	bit3	bit3	bit3	bit3	bit3	bit3	bit3
#1320	1	bit4	bit4	bit4	bit4	bit4	bit4	bit4	bit4
#1321	1	bit5	bit5	bit5	bit5	bit5	bit5	bit5	bit5
#1322	1	bit6	bit6	bit6	bit6	bit6	bit6	bit6	bit6
#1323	1	bit7	bit7	bit7	bit7	bit7	bit7	bit7	bit7
#1324	1	bit8	bit8	bit8	bit8	bit8	bit8	bit8	bit8
#1325	1	bit9	bit9	bit9	bit9	bit9	bit9	bit9	bit9
#1326	1	bit10	bit10	bit10	bit10	bit10	bit10	bit10	bit10
#1327	1	bit11	bit11	bit11	bit11	bit11	bit11	bit11	bit11
#1328	1	bit12	bit12	bit12	bit12	bit12	bit12	bit12	bit12
#1329	1	bit13	bit13	bit13	bit13	bit13	bit13	bit13	bit13
#1330	1	bit14	bit14	bit14	bit14	bit14	bit14	bit14	bit14
#1331	1	bit15	bit15	bit15	bit15	bit15	bit15	bit15	bit15

System variable	No. of points	Interface output signal register							
		$1	$2	$3	$4	$5	$6	$7	$8
#1332	1	R6376/ bit0	R6384/ bit0	R6392/ bit0	R6400/ bit0	R6408/ bit0	R6416/ bit0	R6424/ bit0	R6432/ bit0
#1333	1	bit1	bit1	bit1	bit1	bit1	bit1	bit1	bit1
#1334	1	bit2	bit2	bit2	bit2	bit2	bit2	bit2	bit2
#1335	1	bit3	bit3	bit3	bit3	bit3	bit3	bit3	bit3
#1336	1	bit4	bit4	bit4	bit4	bit4	bit4	bit4	bit4
#1337	1	bit5	bit5	bit5	bit5	bit5	bit5	bit5	bit5
#1338	1	bit6	bit6	bit6	bit6	bit6	bit6	bit6	bit6
#1339	1	bit7	bit7	bit7	bit7	bit7	bit7	bit7	bit7
#1340	1	bit8	bit8	bit8	bit8	bit8	bit8	bit8	bit8
#1341	1	bit9	bit9	bit9	bit9	bit9	bit9	bit9	bit9
#1342	1	bit10	bit10	bit10	bit10	bit10	bit10	bit10	bit10
#1343	1	bit11	bit11	bit11	bit11	bit11	bit11	bit11	bit11
#1344	1	bit12	bit12	bit12	bit12	bit12	bit12	bit12	bit12
#1345	1	bit13	bit13	bit13	bit13	bit13	bit13	bit13	bit13
#1346	1	bit14	bit14	bit14	bit14	bit14	bit14	bit14	bit14
#1347	1	bit15	bit15	bit15	bit15	bit15	bit15	bit15	bit15
#1348	1	R6377/ bit0	R6385/ bit0	R6393/ bit0	R6401/ bit0	R6409/ bit0	R6417/ bit0	R6425/ bit0	R6433/ bit0
#1349	1	bit1	bit1	bit1	bit1	bit1	bit1	bit1	bit1
#1350	1	bit2	bit2	bit2	bit2	bit2	bit2	bit2	bit2
#1351	1	bit3	bit3	bit3	bit3	bit3	bit3	bit3	bit3
#1352	1	bit4	bit4	bit4	bit4	bit4	bit4	bit4	bit4
#1353	1	bit5	bit5	bit5	bit5	bit5	bit5	bit5	bit5
#1354	1	bit6	bit6	bit6	bit6	bit6	bit6	bit6	bit6
#1355	1	bit7	bit7	bit7	bit7	bit7	bit7	bit7	bit7
#1356	1	bit8	bit8	bit8	bit8	bit8	bit8	bit8	bit8
#1357	1	bit9	bit9	bit9	bit9	bit9	bit9	bit9	bit9
#1358	1	bit10	bit10	bit10	bit10	bit10	bit10	bit10	bit10
#1359	1	bit11	bit11	bit11	bit11	bit11	bit11	bit11	bit11
#1360	1	bit12	bit12	bit12	bit12	bit12	bit12	bit12	bit12
#1361	1	bit13	bit13	bit13	bit13	bit13	bit13	bit13	bit13
#1362	1	bit14	bit14	bit14	bit14	bit14	bit14	bit14	bit14
#1363	1	bit15	bit15	bit15	bit15	bit15	bit15	bit15	bit15

System variable	No. of points	Interface output signal register							
		$1	$2	$3	$4	$5	$6	$7	$8
#1364	1	R6378/ bit0	R6386/ bit0	R6394/ bit0	R6402/ bit0	R6410/ bit0	R6418/ bit0	R6426/ bit0	R6434/ bit0
#1365	1	bit1	bit1	bit1	bit1	bit1	bit1	bit1	bit1
#1366	1	bit2	bit2	bit2	bit2	bit2	bit2	bit2	bit2
#1367	1	bit3	bit3	bit3	bit3	bit3	bit3	bit3	bit3
#1368	1	bit4	bit4	bit4	bit4	bit4	bit4	bit4	bit4
#1369	1	bit5	bit5	bit5	bit5	bit5	bit5	bit5	bit5
#1370	1	bit6	bit6	bit6	bit6	bit6	bit6	bit6	bit6
#1371	1	bit7	bit7	bit7	bit7	bit7	bit7	bit7	bit7
#1372	1	bit8	bit8	bit8	bit8	bit8	bit8	bit8	bit8
#1373	1	bit9	bit9	bit9	bit9	bit9	bit9	bit9	bit9
#1374	1	bit10	bit10	bit10	bit10	bit10	bit10	bit10	bit10
#1375	1	bit11	bit11	bit11	bit11	bit11	bit11	bit11	bit11
#1376	1	bit12	bit12	bit12	bit12	bit12	bit12	bit12	bit12
#1377	1	bit13	bit13	bit13	bit13	bit13	bit13	bit13	bit13
#1378	1	bit14	bit14	bit14	bit14	bit14	bit14	bit14	bit14
#1379	1	bit15	bit15	bit15	bit15	bit15	bit15	bit15	bit15
#1380	1	R6379/ bit0	R6387/ bit0	R6395/ bit0	R6403/ bit0	R6411/ bit0	R6419/ bit0	R6427/ bit0	R6435/ bit0
#1381	1	bit1	bit1	bit1	bit1	bit1	bit1	bit1	bit1
#1382	1	bit2	bit2	bit2	bit2	bit2	bit2	bit2	bit2
#1383	1	bit3	bit3	bit3	bit3	bit3	bit3	bit3	bit3
#1384	1	bit4	bit4	bit4	bit4	bit4	bit4	bit4	bit4
#1385	1	bit5	bit5	bit5	bit5	bit5	bit5	bit5	bit5
#1386	1	bit6	bit6	bit6	bit6	bit6	bit6	bit6	bit6
#1387	1	bit7	bit7	bit7	bit7	bit7	bit7	bit7	bit7
#1388	1	bit8	bit8	bit8	bit8	bit8	bit8	bit8	bit8
#1389	1	bit9	bit9	bit9	bit9	bit9	bit9	bit9	bit9
#1390	1	bit10	bit10	bit10	bit10	bit10	bit10	bit10	bit10
#1391	1	bit11	bit11	bit11	bit11	bit11	bit11	bit11	bit11
#1392	1	bit12	bit12	bit12	bit12	bit12	bit12	bit12	bit12
#1393	1	bit13	bit13	bit13	bit13	bit13	bit13	bit13	bit13
#1394	1	bit14	bit14	bit14	bit14	bit14	bit14	bit14	bit14
#1395	1	bit15	bit15	bit15	bit15	bit15	bit15	bit15	bit15

23.22 System Variables (R Device Access Variables)

Function and purpose

By using variable Nos. #50000 to #50749, #51000 to #51749, #52000 to #52749, it is possible to read data (R8300 to R9799, R18300 to R19799, R28300 to R29799) and substitute value in the R device user backup area.

Variable No.	R device	
#50000	R8300, R8301	User backup area (1500 points)
#50001	R8302, R8303	
:		
#50749	R9798, R9799	

Variable No.	R device	
#51000	R18300, R18301	User backup area (1500 points)
#51001	R18302, R18303	
:		
#51749	R19798, R19799	

Variable No.	R device	
#52000	R28300, R28301	User backup area (1500 points)
#52001	R28302, R28303	
:		
#52749	R29798, R29799	

Detailed description

These variables read and write the two words of R device.
Data range of these variables is -2147483648 to 2147483647.
Depending on the setting of the PLC bit selection parameter "#6455/ bit0 to 2", these variables can be changed between decimal point valid or invalid for each user backup area.
The position of the decimal point when decimal point valid is selected, varies according to the parameters "#1003 iunit" (inupt setting unit) and "#1041 I_inch" (initial inch). (This depends on the MTB specifications.)

#1041 I_inch	#1003 iunit			
	B	C	D	E
0: Metric	Three digits after the decimal point	Four digits after the decimal point	Five digits after the decimal point	Six digits after the decimal point
1: Inch	Four digits after the decimal point	Five digits after the decimal point	Six digits after the decimal point	Seven digits after the decimal point

These variables are retained even when the power is off.
These are common among part systems.

Access from a machining program to R device

[Reading variables]

When the variable #50000 is used in a machining program as shown below, the data set in device R8300 and R8301 will be referred.

	Device	Value	#50000
G0 X#50000 ; ↑ R8300,R8301	**R8301**	0x0001	0x1e240 (Hex.) = 123456 (Decimal)
	R8300	0xe240	

(1) When decimal point invalid is selected:

Regardless of the setting of the parameter "#1003 iunit" (input setting unit) and "#1041 I_inch" (initial inch), the data set in the R device will be the command value.
In case of the above example, the command value will be "X123456.".

(2) When decimal point valid is selected:

The data set in the R device will be read as a data with a decimal point.
The position of the decimal point will be as follows, according to the settings of the parameters "#1003 iunit" (inupt setting unit) and "#1041 I_inch" (initial inch).

#1041 I_inch	#1003 iunit			
	B	C	D	E
0: Metric	X123.456	X12.3456	X1.23456	X0.123456
1: Inch	X12.3456	X1.23456	X0.123456	X0.0123456

[Substitution into variables]

When substituting a value to the variable #50001 in a machining program as shown below, data will be set in the device R8302 and R8303.

#50001 = 123 ;
→ R8302,R8303

(1) When decimal point invalid is selected:

Regardless of the setting of the parameter "#1003 iunit" (inupt setting unit) and "#1041 I_inch" (initial inch), substituted value will be set in the R device.

#50001	Device	Value
123 (Decimal) = 0x7b (Hex.)	**R8303**	0x0000
	R8302	0x007b

When a value with a decimal point is substituted to a variable like "#50001 = 123.456 ;", the numbers after the decimal point will be truncated and "123" will be set.

(2) When decimal point valid is selected:

According to the settings of the parameter "#1003 iunit" (inupt setting unit) and "#1041 I_inch" (initial inch), values which are shifted for the number of decimals will be set in the R device, as shown below.

#1041 I_inch		0: Metric			
#1003 iunit		B	C	D	E
#50001		123000 (Decimal) = 0x1e078 (Hex.)	1230000 (Decimal) = 0x12c4b0 (Hex.)	12300000 (Decimal) = 0xbbaee0 (Hex.)	123000000 (Decimal) = 0x754d4c0 (Hex.)
Device	R8303	0x0001	0x0012	0x00bb	0x0754
	R8302	0xe078	0xc4b0	0xaee0	0xd4c0

#1041 I_inch		1: Inch			
#1003 iunit		B	C	D	E
#50001		1230000 (Decimal) = 0x12c4b0 (Hex.)	12300000 (Decimal) = 0xbbaee0 (Hex.)	123000000 (Decimal) = 0x754d4c0 (Hex.)	1230000000 (Decimal) = 0x49504f80 (Hex.)
Device	R8303	0x0012	0x00bb	0x0754	0x4950
	R8302	0xc4b0	0xaee0	0xd4c0	0x4f80

When a value with a decimal point is substituted to a variable like "#50001 = 123.456 ;", the value will directly be set.

#1041 I_inch		0: Metric			
#1003 iunit		B	C	D	E
#50001		123456 (Decimal) = 0x1e240 (Hex.)	1234560 (Decimal) = 0x12d680 (Hex.)	12345600 (Decimal) = 0xbc6100 (Hex.)	123456000 (Decimal) = 0x75bca00 (Hex.)
Device	R8303	0x0001	0x0012	0x00bc	0x075b
	R8302	0xe240	0xd680	0x6100	0xca00

#1041 I_inch		1: Inch			
#1003 iunit		B	C	D	E
#50001		1234560 (Decimal) = 0x12d680 (Hex.)	12345600 (Decimal) = 0xbc6100 (Hex.)	123456000 (Decimal) = 0x75bca00 (Hex.)	1234560000 (Decimal) = 0x4995e400 (Hex.)
Device	R8303	0x0012	0x00bc	0x075b	0x4998
	R8302	0xd680	0x6100	0xca00	0xe400

If the number of decimals of the substituted data exceeds the number of significant figures, the value will be rounded off to the number of significant figures and will be set.

When "#50001 = 123.4567899 ;".

#1041 I_inch		0: Metric			
#1003 iunit		B	C	D	E
#50001		123457 (Decimal) = 0x1e241 (Hex.)	1234568 (Decimal) = 0x12d688 (Hex.)	12345679 (Decimal) = 0xbc614f (Hex.)	123456790 (Decimal) = 0x75bcd16 (Hex.)
Device	R8303	0x0001	0x0012	0x00bc	0x075b
	R8302	0xe241	0xd688	0x614f	0xcd16

#1041 I_inch		1: Inch			
#1003 iunit		B	C	D	E
#50001		1234568 (Decimal) = 0x12d688 (Hex.)	12345679 (Decimal) = 0xbc614f (Hex.)	123456790 (Decimal) = 0x75bcd16 (Hex.)	1234567899 (Decimal) = 0x499602db (Hex.)
Device	R8303	0x0012	0x00bc	0x075b	0x4996
	R8302	0xd688	0x614f	0xcd16	0x02db

Use of R device access variables in control command

These variables can be used in control command.
However, note that the variable value and the condition of true/false differ between decimal point valid variables and invalid variables.

```
IF [#50003 EQ 1] GOTO 30 ;
G00 X100  ;
N30
```

(1) When decimal point invalid is selected:

Regardless of the setting of the parameter "#1003 iunit" (input setting unit) and "#1041 I_inch" (initial inch), R device value of #50003 whose condition is true, will be "1".

#50003	Device	Value
1 (Decimal) = 0x01 (Hex.)	**R8307**	0x0000
	R8306	0x0001

(2) When decimal point valid is selected:

The condition is true when #50003 is "1". So the R device value of #50003 will be as follows depending on the setting of the parameter "#1003 iunit" (inupt setting unit) and "#1041 I_inch" (initial inch).

#1041 I_inch		**0: Metric**			
#1003 iunit		**B**	**C**	**D**	**E**
#50003		1000 (Decimal) = 0x3e8 (Hex.)	10000 (Decimal) = 0x2710 (Hex.)	100000 (Decimal) = 0x186a0 (Hex.)	1000000 (Decimal) = 0xf4240 (Hex.)
Device	R8307	0x0000	0x0000	0x0001	0x000f
	R8306	0x03e8	0x2710	0x86a0	0x4240

#1041 I_inch		**1: Inch**			
#1003 iunit		**B**	**C**	**D**	**E**
#50003		10000 (Decimal) = 0x2710 (Hex.)	100000 (Decimal) = 0x186a0 (Hex.)	1000000 (Decimal) = 0xf4240 (Hex.)	10000000 (Decimal) = 0x989680 (Hex.)
Device	R8307	0x0000	0x0001	0x000f	0x0098
	R8306	0x2710	0x86a0	0x4240	0x9680

Substitution between R device access variable and other variables.

[Substitution into R device access variables]

Common variables and coordinates variables can be substituted to the R device access variables.

(Example 1) Common variables

```
#101 = -123.456 ;
#50004 = #101 ;
```

(Example 2) #5063 : Skip coordinates #5063

```
#50004 = #5063 ;
```

(1) When decimal point invalid is selected:

Regardless of the settings of the parameter "#1003 iunit" (inupt setting unit) and "#1041 I_inch" (initial inch), the value which is rounded off will be set.
When the common variable and coordinate variable in the above example are "-123.456":

#50004	Device	Value
-123 (Decimal) = 0xffffff85 (Hex.)	**R8309**	0xffff
	R8308	0x0085

(2) When decimal point valid is selected:

Substitution will be as follows according to the settings of the parameter "#1003 iunit" (inupt setting unit) and "#1041 I_inch" (initial inch).

#1041 I_inch		**0: Metric**			
#1003 iunit		**B**	**C**	**D**	**E**
#50004		-123456 (Decimal) = 0xfffe1dc0 (Hex.)	-1234560 (Decimal) = 0xffed2980 (Hex.)	-12345600 (Decimal) = 0xff439f00 (Hex.)	-123456000 (Decimal) = 0xf8a43600 (Hex.)
Device	R8309	0xfffe	0xffed	0xff43	0xf8a4
	R8308	0x1dc0	0x2980	0x9f00	0x3600

#1041 I_inch		**1: Inch**			
#1003 iunit		**B**	**C**	**D**	**E**
#50004		-1234560 (Decimal) = 0xffed2980 (Hex.)	-12345600 (Decimal) = 0xff439f00 (Hex.)	-123456000 (Decimal) = 0xf8a43600 (Hex.)	-1234560000 (Decimal) = 0xb66a1c00 (Hex.)
Device	R8309	0xffed	0xff43	0xf8a4	0xb66a
	R8308	0x2980	0x9f00	0x3600	0x1c00

[Substitution of R device access variables]

```
#50005 = 123.456789 ;
#102 = #50005 ;
```

(1) When decimal point invalid is selected:

Regardless of the settings of the parameter "#1003 iunit" (inupt setting unit) and "#1041 I_inch" (initial inch), #102 will be "123".

(2) When decimal point valid is selected:

Substitution will be as follows according to the settings of the parameter "#1003 iunit" (inupt setting unit) and "#1041 I_inch" (initial inch).

#1041 I_inch	0: Metric			
#1003 iunit	B	C	D	E
#102	123.4570	123.4568	123.4568	123.4568

#1041 I_inch	1: Inch			
#1003 iunit	B	C	D	E
#102	123.4568	123.4568	123.4568	123.4568

Precautions

(1) The position of a decimal point changes depending on the settings of the parameter "#1003 iunit" (input setting unit) and "#1041 I_inch" (initial inch). Fix the decimal point position while considering these parameter settings when setting a number to an R device.

(2) These variables do not handle <Blank>. If #0<Blank> is substituted, it will be converted into "0".
Therefore, when comparing this variable after substituting #0<Blank> and #0<Blank> with a conditional expression (EQ), it will not be formed.

(3) If a value exceeding the allowable range is substituted into this variable, a program error (P35) will occur.

(4) When these values are used as decimal point invalid, the settings of "#1078 Decpt2" (Decimal point type 2) and "#8044 UNIT*10" will not be applied.

(5) When a graphic is being checked, writing into R device will not be executed even if a value is substituted into these variables.
For reading of these variables (reference to the R device value) during a graphic check, "0" is always read.

23.23 System Variables (PLC Data Reading)

Function and purpose

PLC data can be read in with the system variables.

Variable No.	Application
#100100	Device type designation
#100101	Device No. designation
#100102	Number of read bytes designation
#100103	Read bit designation
#100110	Reading PLC data

Note

(1) These can be used only with some models.

(2) The readable devices are limited.

Detailed description

The PLC data is read in with the following five blocks using these five system variables.

#100100 = 1 ; Designates the device type.

#100101 = 0 ; Designates the device No.

#100102 = 1 ; Designates the number of bytes.

#100103 = 2 ; Designates the bit. (Valid only when reading word device bits.)

#100=#100110; Reads in the PLC data.

Device designation (#100100)

(1) System variable for device designation

The type of device to be read in can be designated by substituting the device designation value in this system variable.

If the data is read without designating this variable, the data will be read in the same manner as when the minimum value (0: M device) of the device designation value is designated. Once designated, the setting is held until the device is designated again or until it is reset.

A program error (P39) will occur if a nonexistent device is set.

(2) Device designation value

[M800/M80 series]

Device designation value	Device	Unit	Device No.
0	M	Bit	M0 to M61439
1	D	Word	D0 to D4095
2	C	Bit	C0 to C511
4	X (*1)	Bit	X0 to X1FFF
5	Y (*1)	Bit	Y0 to Y1FFF
6	R	Word	R0 to R32767
7	T	Bit	T0 to T2047
9	SM	Bit	SM0 to SM2047

Device designation value	Device	Unit	Device No.
10	F	Bit	F0 to F2047
13	L	Bit	L0 to L1023
18	V	Bit	V0 to V511
19	ST	Bit	ST0 to ST127
20	SD	Word	SD0 to SD2047
21	SB (*1)	Bit	SB0 to SB3FF
22	SW (*1)	Word	SW0 to SW3FF
23	B (*1)	Bit	B0 to BDFFF
24	W (*1)	Word	W0 to W2FFF

[C80 series]

Device designation value	Device	Unit	Device No.
0	M	Bit	M0 to M61439
1	D	Word	D0 to D8191
2	C	Bit	C0 to C511
4	X (*1) (*2)	Bit	X0 to X1FFF
5	Y (*1) (*2)	Bit	Y0 to Y1FFF
6	R (*2)	Word	R0 to R32767
7	T	Bit	T0 to T2047
9	SM (*2)	Bit	SM0 to SM4095

Device designation value	Device	Unit	Device No.
10	F	Bit	F0 to F2047
13	L	Bit	L0 to L1023
18	V	Bit	V0 to V511
19	ST	Bit	ST0 to STI27
20	SD	Word	SD0 to SD4095
21	SB (*1)	Bit	SB0 to SB3FF
22	SW (*1)	Word	SW0 to SW1023
23	B (*1)	Bit	B0 to BDFFF
24	W (*1)	Word	W0 to W2FFF

The unit indicates the amount of data per device No. "Word" is 16 bits, and "Bit" is one bit.

(*1) Device of which the device number is indicated in hexadecimal notation.

(*2) The device marked by an asterisk (*) in the Device column has the determined use; therefore, do not use the undefined device number even for a vacant device.

Device No. designation (#100101)

The device to be read in is designated by substituting the device No. in this system variable.
Convert a device expressed as a hexadecimal into a decimal when designating.
If the data is read without designating this number, the data will be read in the same manner as when the minimum device No. (0) is designated. Once designated, the setting is held until the device No. is designated again or until it is reset.
A program error (P39) will occur if a nonexistent device No. is set.

Number of bytes designation (#100102)

(1) System variable for number of bytes designation
The reading size is designated by substituting the number of bytes designation value in this system variable.
If the data is read without designating this number, the data will be read in the same manner as when the minimum device designation value (0: M device) is designated. Once designated, the setting is held until the number of bytes is designated again or until it is reset.
A program error (P39) will occur if a number of bytes that does not exist in the specifications is set.

(2) Number of bytes designation value

Number of bytes designation value	Read in data			Operation	
	Size	Sign	Range	Word device	Bit device
0	1 bit	-	0 to 1	The number of bits designated is read in.	The bits for the designated device No. are read in.
1	1 bytes	No	0 to 255	The low-order byte is read in.	8 bits are read in from the designated device No.
101		Yes	-128 to 127		
2	2 bytes	No	0 to 65535	Two bytes are read in.	16 bits are read in from the designated device No.
102		Yes	-32768 to 32767		
4	4 bytes	No	0 to 4294967295	The designated device (L) and next device (H) are read in.	32 bits are read in from the designated device No.
104		Yes	-2147483648 to 2147483647		

0 to 4 are designated without a sign, and 101 to 104 are designated with a sign.

Bit designation (#100103)

(1) System variable for bit designation
The bit to be read in is designated by substituting the bit designation value in this system variable.
This designation is valid only when reading the bits for a 16-bit device, and is invalid for the others.
If the data is read without designating this number, the data will be read in the same manner as if the minimum bit designation value (0: bit 0) is designated. Once designated, the setting is held until the bit is designated again or until it is reset.
A program error (P39) will occur if a nonexistent bit is set.

(2) Bit designation value

Bit designation value	Read in bit
0	Bit 0
1	Bit 1
:	:
15	Bit 15

Reading PLC data (#100110)

The data for the designated device is read in with this system variable.
Refer to the table for number of bytes designation for details on the range of data read in.

Program example

(1) To read a bit device

#100100 = 0 ;	Designates [M device].
#100101 = 0 ;	Designates [Device No. 0].
#100102 = 0 ;	Designates [Bit].
#100 = #100110;	Reads M0 (one bit).
#100102 = 1 ;	Designates [1 byte].
#101 = #100110;	Reads M0 to M7 (8 bits). (If M7 to M0 is 0001 0010, this will be #102 = 18 (0x12).)
#100102 = 102 ;	Designates [Signed two bytes].
#102 = #100110;	Reads M0 to M15 (16 bits). (If M15 to M0 is 1111 1110 1101 1100, this will be #102 = -292 (0xFEDC).)
#100102 = 4 ;	Designates [4 byte].
#104 = #100110;	Reads M0 to M31 (32 bits). (If M31 to M0 is 0001 0010 0011 0100 0101 0110 0111 1000, #104 = 305419896 (0x12345678).)

(2) To read a word device

#100100 = 1 ;	Designates [D device].
#100101 = 0 ;	Designates [Device No. 0].
#100102 = 0 ;	Designates [Bit].
#100103 = 1 ;	Designates [Bit 1].
#100 = #100110;	Reads the D0 bit 1. (If D0 = 0x0102, #101 =1.)
#100102 = 1 ;	Designates [1 byte].
#101 = #100110;	Reads the low-order byte of D0. (If D0 = 0x0102, #101 =2.)
#100102 = 2 ;	Designates [2 byte].
#102 = #100110;	Reads D0. (If D0 = 0x0102, #102 =258.)
#100102 = 104 ;	Designates [Signed four bytes].
#104 = #100110;	Reads D0 and D1. (If D0 = 0xFFFE and D1 = 0xFFFF, #104 =-2.)

Precautions

(1) As the PLC data is read asynchronously from the ladder execution, the data is not necessarily the one which was gained when the program was executed. Be careful when reading devices which are changing.

(2) If reading of a nonexistent device is attempted by designating the device No. and number of bytes, the 0 value will be read in only for the nonexistent section.

(3) When "1" is set to the parameter "#1316 CrossCom", #100100 to #100110 cannot be used as system variables to read PLC data.

23.24 System Variables (Interfering Object Selection)

Detailed description

Select 16 interfering objects to use in the interference check III with system variables or R register.
Refer to the "PLC Interface Manual" (IB-1501272) for the R register.
When selecting an interfering object, specify the specification of the selected interfering object and interfering model coordinate system offset 1.
The write command to the system variables (#40000 to #40097) is possible only in the machine tool builder macro programs (O100010000 to O199999998).

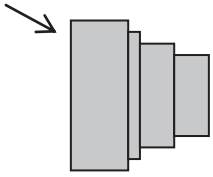
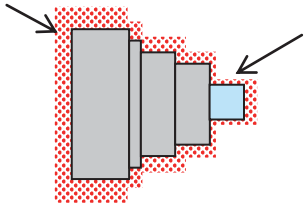
System variable	R register	Item	Details	Setting range (unit) Upper: System variable Lower: R register
#40000	R20304	Interfering object enable/disable designation	Set enable/disable for each interfering object. Bit designation (0: enable 1: disable)	0 to 65535 (decimal)
			bit0: Disable 1st interfering object : bitF: Disable 16th interfering object	0x0000 to 0xFFFF (hexadecimal)
#40001	R20305	preliminary		0
				0
40002	R20306	1st interfering object selection	Select interfering object definition No. to use.	0 to 128 (0: not selected)
				0 to 128 (0: not selected)
#40003	R20307	1st interfering object specification	In the configured solid specification of the interfering object definition, specify alarm area/warning area/solid setting invalid of the solid in which switching method is selected.	0 to 3
			0, 1: Alarm area 2: Warning area 3: Solid setting invalid	0 to 3
#40004	R20308 (L)	1st interfering model coordinate system I axis offset 1	Set the interfering model coordinate system offset with a radius value. (I axis direction) (*1)	-99999.999 to 99999.999 (mm) (radius value)
	R20309 (H)			
#40005	R20310 (L)	1st interfering model coordinate system J axis offset 1	Set the interfering model coordinate system offset with a radius value. (J axis direction) (*1)	
	R20311 (H)			-99999999 to 99999999 (μm) (radius value)
#40006	R20312 (L)	1st interfering model coordinate system K axis offset 1	Set the interfering model coordinate system offset with a radius value. (K axis direction) (*1)	
	R20313 (H)			
:	:			
#40077	R20426	16th interfering object selection	Same as above	Same as above
#40078	R20427	16th interfering object specification selection	Same as above	Same as above
#40079	R20428 (L)	16th interfering model coordinate system I axis offset 1	Same as above	Same as above
	R20429 (H)			
#40080	R20430 (L)	16th interfering model coordinate system J axis offset 1	Same as above	Same as above
	R20431 (H)			

System variable	R register	Item	Details	Setting range (unit) Upper: System variable Lower: R register
#40081	R20432 (L) R20433 (H)	16th interfering model coordinate system K axis offset 1	Same as above	Same as above
#40082	R20434	1st interfering object Interference check III: Specifying disabled interfering object	Select an interfering object that you do not check the interference with the 1st interfering object. bit0: Disable 1st interfering object (inaction data) bit1: Disable 2nd interfering object : bitF: Disable 16th interfering object	0 to 65535 (decimal) 0x0000 to 0xFFFF (hexadecimal)
#40083	R204325	2nd interfering object Interference check III: Specifying disabled interfering object	Select an interfering object that you do not check the interference with the 2nd interfering object. bit0: Disable 1st interfering object bit1: Disable 2nd interfering object (inaction data) : bitF: Disable 16th interfering object	0 to 65535 (decimal) 0x0000 to 0xFFFF (hexadecimal)
:	:			
#40097	R20449	16th interfering object Interference check III: Specifying disabled interfering object	Select an interfering object that you do not check the interference with the 16th interfering object. bit0: Disable 1st interfering object bit1: Disable 2nd interfering object : bitF: Disable 16th interfering object (inaction data)	0 to 65535 (decimal) 0x0000 to 0xFFFF (hexadecimal)

(*1) The interfering model coordinate system offset is the sum of the interfering model coordinate system offsets 1 and 2.

Interference check III: designation of disabled interference object

(Example) In the case that you do not check the interference between the 1st interfering object and the 2nd interfering object
"R20434 (#40082): 0x0002 (disable 2nd interfering object)" or "R20435 (#40083): 0x0001 (disable 1st interfering object)"
Since each interfering object is designated to perform the interference check, the setting of the interference check III specifying disabled interference object is repeated, but if either one is on disabled setting, the interference check is not performed.

Back side of spindle part (without a workpiece)	Back side of spindle part (with a workpiece)
1st interfering object (back side of spindle part) 	1st interfering object (back side of spindle part) 2nd interfering object (workpiece part)  By specifying the interference III disabled between the back side of spindle part (1st interfering object) and workpiece part (2nd interfering object), these 2 parts are treated as one interfering object.

Precautions

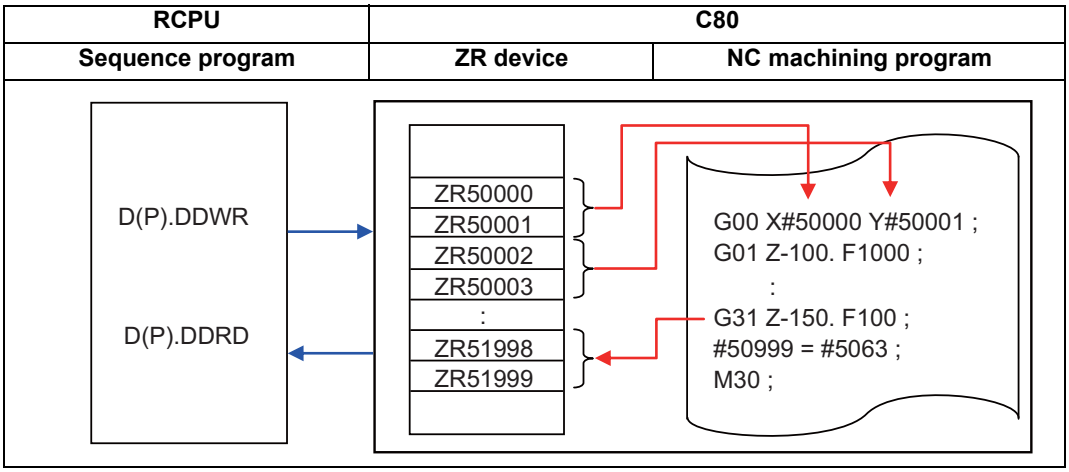
(1) When the interfering object selection is input with the system variables, the system variable in which the command range integer is set in R register with the value after the decimal point being ignored.
(a) When any value out of the setting range is input in #40000 to #40097, the low-order 16 bits of the input value are set in R register.
(b) When "#0" <empty> is input in #40000 to #40097, "0" is set in R register.

(2) If you have made a write command to the system variable (#40000 to #40097) in a program except for the machine tool builder macro program, the program error (P241) occurs.

23.25 System Variables (ZR Device Access Variables) [C80]

Detailed description

System variables that can read and write data from and to the ZR device are provided by 2,250 sets (#50000 to #52749).
Data can be read and written between the NC machining program and RCPU sequence program by using the ZR device as shown below.
How to handle the ZR device in the RCPU sequence program depends on the MTB specifications.
Refer to the PLC Interface Manual (IB-1501258) for the DDWR/DDRD command.



Number of variable sets

The table below shows a list of variables specific to C80.
A ZR device access variable is based on long-type data, and a ZR device on word-type data.
Therefore, when this variable is read or a value is substituted to this variable, it reads and writes two words of the ZR device. The correspondence between the ZR device access variable numbers and ZR device numbers is shown below.

Variable No. (2,250 sets)		Corresponding ZR device (4,500 units)
#50000 - #50749	#50000	ZR50000, ZR50001
	#50001	ZR50002, ZR50003
	#50002	ZR50004, ZR50005
	:	:
	#50000+n	ZR50000+2n, ZR50000+2n+1
	:	:
	#52749	ZR55498, ZR55499
#51000 - #51749	#51000	ZR52000, ZR52001
	:	:
	#51749	ZR53498, ZR53499
#52000 - #52749	#52000	ZR54000, ZR54001
	:	:
	#52749	ZR55498, ZR55499

(1) The data range of these variables is -2147483648 to 2147483647.
(2) The ZR device is backed up even when the power is turned OFF; therefore, the value is maintained after the power has been turned ON again.
(3) Whether this variable is used with the decimal point invalid or valid can be selected for each user backup area according to the MTB specifications (parameter "#6455 bit0 -bit2").

(4) When "decimal point valid" is selected, the position of the decimal point depends on the MTB specifications (parameters "#1003 iunit" (input setting unit) and "#1041 I_inch" (initial inch)).
Therefore, to set a numeric value to a ZR device, consider the position of the decimal point according to these parameters.
The table below shows the number of digits that is valid after the decimal point.

#1041 I_inch	#1003 iunit			
	B	C	D	E
Metric	3 digits	4 digits	5 digits	6 digits
Inch	4 digits	5 digits	6 digits	7 digits

24

Appx.1: Fixed Cycles

[G37(O100000370) Automatic tool length measurement]

```
G31 Z#5 F#3 ;
IF[ ROUND[ ABS[#2-[##10*#11-#12]]] GT#8]G
OTO1 ;
IF[ ROUND[##10*#11-#12] EQ#4] GOTO1 ;
##9=##10-#12/#11-#2/#11+##9 ;
#3003=#1 ;
N2 ;
M99 ;
N1#3901=126 ;
```

[G74(O100000740) End face cutoff cycle]

```
G.1 ;
IF[ ABS[#2] GT0] GOTO10 ;
#14=1 ;
N10#13=#3 ;
IF[#15NE0] GOTO11 ;
#13=#3-#5 ;
N11#16=0 ;
DO1 ;
#10=0 ;
#11=#4 ;
DO2 ;
#10=#10+#4 ;
IF[ ABS[#10] GE[ ABS[#1]]] GOTO1 ;
G01 X#11 ;
G00 X#6 ;
#11=#4-#6 ;
END2 ;
N1 G01 X#1-#10+#11 ;
IF[#15 EQ0] GOTO20 ;
IF[#16 EQ0] GOTO21 ;
N20 G00 Y#5 ;
N21#16=1 ;
G00 X-#1 ;
IF[#14] GOTO3 ;
#12=#12+#3 ;
IF[ ABS[#12] LT[ ABS[#2]]] GOTO2 ;
#14=1 ;
#13=#2-#12+#13 ;
N2 G00Y#13 ;
#13=#3-#5 ;
END1 ;
N3 G00 Y-#2-#5 ;
M99 ;
```

[G75(O100000750) Longitudinal cutting cycle]

```
G.1 ;
IF[ ABS[#2] GT0] GOTO10 ;

#14=1 ;
N10#13=#3 ;
IF[#15NE0] GOTO11 ;
#13=#3-#5 ;
N11#16=0 ;
DO1 ;
#10=0 ;
#11=#4 ;
DO2 ;
#10=#10+#4 ;
IF[ ABS[#10] GE[ ABS[#1]]] GOTO1 ;
G01 X#11 ;
G00 X#6 ;
#11=#4-#6 ;
END2 ;
N1 G01 X#1-#10+#11 ;
IF[#15 EQ0] GOTO20 ;
IF[#16 EQ0] GOTO21 ;
N20 G00 Y#5 ;
N21#16=1 ;
G00 X-#1 ;
IF[#14] GOTO3 ;
#12=#12+#3 ;
IF[ ABS[#12] LT[ ABS[#2]]] GOTO2 ;
#14=1 ;
#13=#2-#12+#13 ;
N2 G00Y#13 ;
#13=#3-#5 ;
END1 ;
N3 G00 Y-#2-#5 ;
M99 ;
```

[G76(O100000760) Compound thread cutting cycle]

```
G.1 ;
#12=1 ;
#13=#9 ;
IF[ ABS[#13] GE[ ABS[#8]]] GOTO1 ;
#16=1 ;
#13=#8 ;
N1#11=#13 ;
IF[ ABS[#11] LT[ ABS[#4-#5]]] GOTO2 ;
#11=#4-#5 ;
#14=1 ;
N2#17=#11 ;
#10= ROUND[[#11+#5]*#7] ;
IF[[#10 XOR#1] GE0] GOTO20 ;

#10=-#10 ;N20 G00X#10 ;
#55=#10 ;
DO1 ;
#15= ROUND[#10*#3/#1] ;
N90#40=90#41=#5001#42=#5002 ;
G00 Y#2+#3-#4-#15+#11 ;
G33 X#1-#10 Y-#3+#15 Q#22 M96.101 P1000 D3
;
G00 Y-#2+#4-#11 M97.101 ;
IF[#14 GT0] GOTO3 ;
IF[#16 GT0] GOTO7 ;
#12=#12+1 ;
#13= ROUND[#9* SQRT[#12]] ;
IF[ ABS[#13-#11] GE[ ABS[#8]]]GOTO8 ;
#16=1 ;
N7#13=#11+#8 ;
N8#11=#13 ;
IF[ ABS[#11] LT[ ABS[#4-#5]]] GOTO9 ;
#11=#4-#5 ;
#14=1 ;
N9#10= ROUND[[#17-#11]*#7] ;
IF[[#10 XOR#1] GE0] GOTO6 ;
#10=-#10 ;
N6#10=#10+#55 ;
G00X-#1+#10 ;
N12 END1 ;
N3IF[ ABS[#6] LT1] GOTO5 ;
#14=0 ;
#13=0 ;
DO2 ;
IF[#14 GT0] GOTO5 ;
#13=#13+#6 ;
IF[ ABS[#13] LT[ ABS[#5]]] GOTO4 ;
#13=#5 ;
#14=1 ;
N4 G00 X#10-#1 ;
N91#40=91#41=#5001#42=#5002 ;
G00 Y#2+#3-#4+#13-#15+#11 ;
G33 X#1-#10 Y-#3+#15 Q#22 M96.101 P1000 D3
;
G00 Y-#2+#4-#13-#11 M97.101 ;
END2 ;
N5 G00X-#1 ;
M99 ;
```

[G76.1(O100000761) Two-part system simultaneous compound thread cutting cycle]

```
G.1 ;
N761! L10 ;
#12=1 ;
#13=#9 ;
IF[ ABS[#13] GE[ ABS[#8]]] GOTO1 ;
#16=1 ;
#13=#8 ;
N1#11=#13 ;
IF[ ABS[#11] LT[ ABS[#4-#5]]] GOTO2 ;
#11=#4-#5 ;
#14=1 ;
N2#17=#11 ;
#10= ROUND[[#11+#5]*#7] ;
IF[[#10 XOR#1] GE0] GOTO3 ;
#10=-#10 ;
N3 G00X#10 ;
#55=#10 ;
DO1 ;
#15= ROUND[#10*#3/#1] ;
N90#40=90#41=#5001#42=#5002 ;
G00 Y#2+#3-#4-#15+#11 ;
! L11 ;
G33 X#1-#10 Y-#3+#15 M96.101 P1000 D3 ;
G00 Y-#2+#4-#11 M97.101 ;
! L12 ;
IF[#14 GT0] GOTO4 ;
IF[#16 GT0] GOTO11 ;
#12=#12+1 ;
#13= ROUND[#9* SQRT[#12]] ;
IF[ ABS[#13-#11] GE[ ABS[#8]]] GOTO12 ;
#16=1 ;
N11#13=#11+#8 ;
N12#11=#13 ;
IF[ ABS[#11] LT[ ABS[#4-#5]]] GOTO13 ;
#11=#4-#5 ;
#14=1 ;
N13#10= ROUND[[#17-#11]*#7] ;
IF[[#10 XOR#1] GE0] GOTO14 ;
#10=-#10 ;
N14#10=#10+#55 ;
G00 X-#1+#10 ;
N15 END1 ;
N4 IF[ ABS[#6] LT1] GOTO5 ;
#14=0 ;
#13=0 ;
DO2 ;
IF[#14 GT0] GOTO5 ;
#13=#13+#6 ;
IF[ ABS[#13] LT[ ABS[#5]]] GOTO21 ;
#13=#5 ;
#14=1 ;
N21 G00 X#10-#1 ;
N91#40=91#41=#5001#42=#5002 ;
G00 Y#2+#3-#4+#13-#15+#11 ;
! L11 ;
G33 X#1-#10 Y-#3+#15 M96.101 P1000 D3 ;
G00 Y-#2+#4-#13-#11 M97.101 ;
! L12 ;
END2 ;
N5 G00 X-#1 ;
M99 ;
```

[G76.2(O100000762) Two-part system simultaneous compound thread cutting cycle]

```
G.1 ;
N762! L10 ;
#12=1 ;
#13=#9 ;
IF[ ABS[#13] GE[ ABS[#8]]] GOTO1 ;
#16=1 ;
#13=#8 ;
N1#11=#13 ;
IF[ ABS[#11] LT[ ABS[#4-#5]]] GOTO2 ;
#11=#4-#5 ;
#14=1 ;
N2#17=#11 ;
#10= ROUND[[#11+#5]*#7] ;
IF[[#10 XOR#1] GE0] GOTO3 ;
#10=-#10 ;
N3 IF[#27 NE1] GOTO4 ;
G00 X#10 ;
N4#55=#10 ;
#28=1 ;
DO1 ;
#15= ROUND[#10*#3/#1] ;
#29=#28 MOD2 ;
IF[[#27 EQ1] AND[#29 EQ0]] GOTO11 ;
IF[[#27 EQ2] AND[#29 EQ1]] GOTO11 ;
N90#40=90#41=#5001#42=#5002 ;
G00 Y#2+#3-#4-#15+#11 ;
! L11 ;
G33 X#1-#10 Y-#3+#15 M96.101 P1000 D3 ;
G00 Y-#2+#4-#11 M97.101 ;
! L12 ;
N11 IF[#14 GT0] GOTO5 ;
IF[#16 GT0] GOTO12 ;
#12=#12+1 ;
#13= ROUND[#9* SQRT[#12]] ;
IF[ ABS[#13-#11] GE[ ABS[#8]]] GOTO13 ;
#16=1 ;
N12#13=#11+#8 ;
N13#11=#13 ;
IF[ ABS[#11] LT[ ABS[#4-#5]]] GOTO14 ;
#11=#4-#5 ;
#14=1 ;
N14#10= ROUND[[#17-#11]*#7] ;
IF[[#10 XOR#1] GE0] GOTO15 ;
#10=-#10 ;
N15#10=#10+#55 ;
IF[[#27 EQ1] AND[#29 EQ1]] GOTO17 ;
IF[[#27 EQ2] AND[#29 EQ0]] GOTO17 ;
IF[[#27 EQ2] AND[#28 EQ1]] GOTO16 ;
G00 X-#1+#10 ;
GOTO17 ;
N16 G00 X#10 ;
N17#28=#28+1 ;
END1 ;
N5 IF[ ABS[#6] LT1] GOTO6 ;
#14=0 ;
#13=0 ;
DO2 ;
IF[#14 GT0] GOTO6 ;
#13=#13+#6 ;
IF[ ABS[#13] LT[ ABS[#5]]] GOTO21 ;
#13=#5 ;
#14=1 ;
N21#29=#28 MOD2 ;
IF[[#27 EQ1] AND[#29 EQ1]] GOTO23 ;
IF[[#27 EQ2] AND[#29 EQ0]] GOTO23 ;
IF[[#27 EQ2] AND[#28 EQ1]] GOTO22 ;
G00 X#10-#1 ;
GOTO91 ;
N22 G00X#10 ;
N91#40=91#41=#5001#42=#5002 ;
G00 Y#2+#3-#4+#13-#15+#11 ;
! L11 ;
G33 X#1-#10 Y-#3+#15 M96.101 P1000 D3 ;
G00 Y-#2+#4-#13-#11 M97.101 ;
! L12 ;
N23#28=#28+1 ;
END2 ;
N6 G00 X-#1 ;
M99 ;
```

[G77(O100000770) Longitudinal cutting cycle]

```
G.1 ;
IF[[#1 EQ0] OR[#2 EQ0]] GOTO1 ;
Y#2+#7 ;
G1 X#1 Y-#7 ;
Y-#2 ;
G0 X-#1 ;
N1 M99 ;
```

[G78(O100000780) Thread cutting cycle]

```
G.1 ;
IF[[#1 EQ0] OR[#2 EQ0]] GOTO1 ;
N90#40=90#41=#5001#42=#5002 ;
Y#2+#7 ;
G33 X#1 Y-#7 F#9 E#10 Q#20 M96.101 P1000
D3 ;
G0 Y-#2 M97.101 ;
X-#1 ;
N1 M99 ;
```

[G79(O100000790) End face cutoff cycle]

```
G.1 ;
IF[[#1 EQ0] OR[#2 EQ0]] GOTO1 ;
X#1+#7 ;
G1 X-#7 Y#2 ;
X-#1 ;
G0 Y-#2 ;

N1 M99 ;
```

[G83, G87(O100000830) Deep hole drilling cycle B]

```
G.1 ;
IF[#30] GOTO2 ;
M#24 ;
#29=#11#28=0 ;
Z#2 ;
#2=##5#3003=#8 OR1 ;
DO1 ;
#28=#28-#11#26=-#28-#29 ;
Z#26 ;
IF[ ABS[#28] GE[ ABS[#3]]] GOTO1 ;
G1 Z#29 ;
G0 Z#28 ;
#29=#11+#14 ;
END1 ;

N1 G1 Z#3-#26 ;
G4 P#4 ;
#3003=#8 ;
G0 Z-#3-#2 ;
IF[#24 EQ#0] GOTO2 ;
M#24+1 ;
G4 P#56 ;
N2 M99 ;
```

[G83, G87(O100000831) Deep hole drilling cycle A]

```
G.1 ;
IF[#30] GOTO2 ;
M#24 ;
#29=0#28=#11 ;
Z#2 ;
#2=##5#3003=#8OR1 ;
DO1 ;
#29=#29+#11 ;
IF[ ABS[#29] GE[ ABS[#3]]] GOTO1 ;
G1 Z#28 ;
G0 Z-#14 ;
#28=#11+#14 ;
END1 ;
N1 G1 Z#3-#29+#28 ;
G4 P#4 ;
#3003=#8 ;
G0 Z-#3-#2 ;
IF[#24 EQ#0] GOTO2 ;
M#24+1 ;
G4 P#56 ;
N2 M99 ;
```

[G83.2(O100000832) Deep hole drilling cycle 2]

```
G.1 ;
IF[#30] GOTO3 ;
#3003=#8 OR1 ;
#29=#12#28=0#26=0 ;
G0 Z#2 ;
IF[#12 NE#0] GOTO1 ;
IF[#11 EQ#0] GOTO2 ;
N1#28=#28-#12#26=-#28-#29 ;
IF[ ABS[#28] GE[ ABS[#3]]] GOTO2 ;
G1 Z#12 ;
G4 P#4 ;
G0 Z#28-#2 ;
G4 P#13 ;
#29=#11+#15 ;
DO1 ;
#28=#28-#11#26=-#28-#29 ;
G0 Z#26+#2 ;
IF[ ABS[#28] GE[ ABS[#3]]] GOTO2 ;
G1 Z#29 ;
G4 P#4 ;
G0 Z#28-#2 ;
G4 P#13 ;
END1 ;
N2 G1 Z#3-#26 ;
G4 P#4 ;
#3003=#8 ;
G0 Z-#3-#2 ;
N3 M99 ;
```

[G84, G88(O100000840) Tap cycle]

```
G.1 ;
IF[#30] GOTO2 ;
M#24 ;
Z#2 ;
#2=##5#3003=#8 OR1#3004=#9 OR3 ;
G1 Z#3 ;
G4 P#4 ;
M#53 ;
#3900=1 ;
G1 Z-#3 ;
#3004=#9 ;
M#54 ;
#3003=#8 ;
IF[#24 EQ#0] GOTO1 ;
M#24+1 ;
G4 P#56 ;
N1 G0 Z-#2 ;
N2 M99 ;
```

[G85, G89(O100000850) Boring cycle]

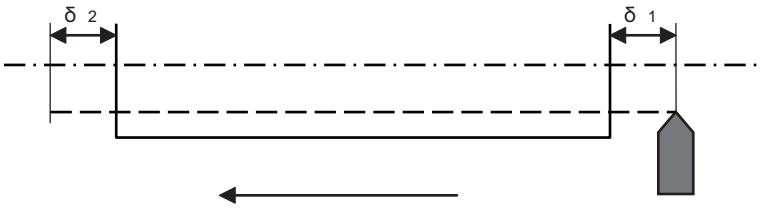
```
G.1 ;
IF[#30] GOTO2 ;
M#24 ;
Z#2 ;
#2=##5#3003=#8 OR1 ;
G1 Z#3 ;
G4 P#4 ;
#3003=#8 ;
Z-#3 F#23 ;
F#22 ;
IF[#24 EQ#0] GOTO1 ;
M#24+1 ;
G4 P#56 ;
N1 G0 Z-#2 ;
N2 M99 ;
```

25

Appx.2: Supplementary Explanation for Incomplete Thread Area of Thread Control

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 1 and for the length of the area 2 where the thread is incomplete during chamfering, as shown in the figure below.



Command direction

δ1 : Approach distance

δ2 : Area where thread is incomplete during chamfering

Detailed description

Approach distance [δ1]

(1) When Ts is not equal to zero (Ts ≠ 0)

$$\delta_1 = \frac{F}{60} t1 - \frac{F}{60}\left(Ts + Tp - \frac{Tp^2 e^{-\frac{t1}{Tp}} - Ts^2 e^{-\frac{t1}{Tp}}}{Tp - Ts}\right) \text{ (mm)}$$

Where F : Thread cutting speed (mm/min)

t1 : Time taken until pitch error reaches allowable limit "a" (s)

Ts : Acceleration/deceleration time constant (s)

Tp : Position loop time constant (s) = 1/Position loop gain(PGN1)

During the SHG control, a position loop gain should be calculated by multiplying √ 2.

If "p" is the pitch and "ΔP" is the pitch error, then allowable limit "a" will be:

$$a = \frac{1}{Tp - Ts}\left(Tp\, e^{-\frac{t1}{Tp}} - Ts\, e^{-\frac{t1}{Ts}}\right)$$

(2) When Ts is equal to zero (Ts = 0)

$$\delta_1 = \frac{F}{60} t1 - \frac{F}{60}\left(Tp - Tp\, e^{\frac{T1}{Tp}} \right) \quad (mm)$$

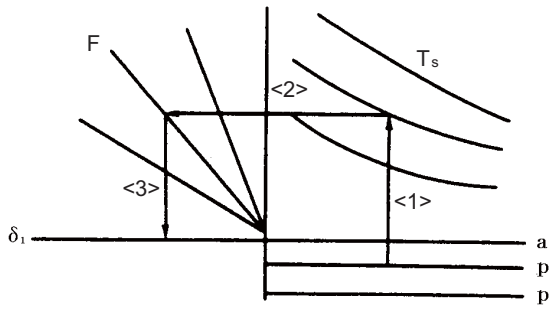
$$a = e^{-\frac{t1}{Tp}}$$

Since the calculation of approach distance δ1 is a complicated procedure, δ1 is normally determined from the chart on the next page. This chart is used as follows.

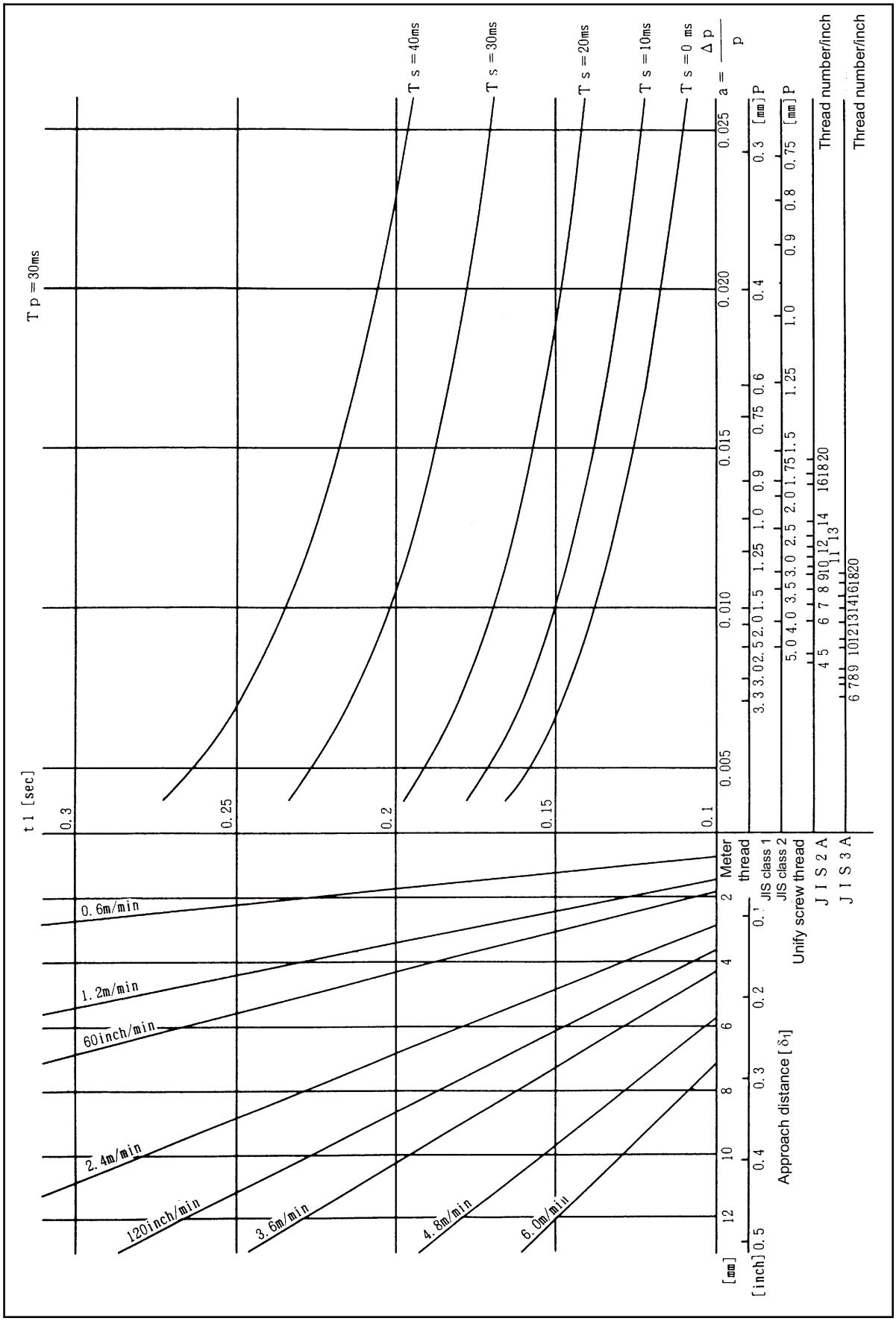
(a)Find the position on the p axis scale represents by the thread grade and pitch [P], and follow the perpendicular <1> drawn upward to find the point where it intersects with the curve of acceleration/ deceleration time constant [Ts].

(b) Follow horizontal line <2> and find where it intersects with the thread cutting speed [F].

(c) Follow perpendicular <3> and find approach distance [δ1] on the scale at the point where it intersects with the δ1 axis.



Approach distance δ1 calculation chart (when position loop gain = 33)



Length of area where thread is incomplete during chamfering [δ2]

$$\delta_2 = (T_s + T_p)\frac{F}{60} \text{ (mm)}$$

Where F : Thread cutting speed (mm/min)
Ts : Acceleration/deceleration time constant (s)
Tp : Position loop time constant (s) = 1/ position loop gain (PGN1)
During the SHG control, a position loop gain should be calculated by multiplying √ 2.

Note

(1) 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 δ2 determined by the above formula to the chamfering pitch set by parameter.

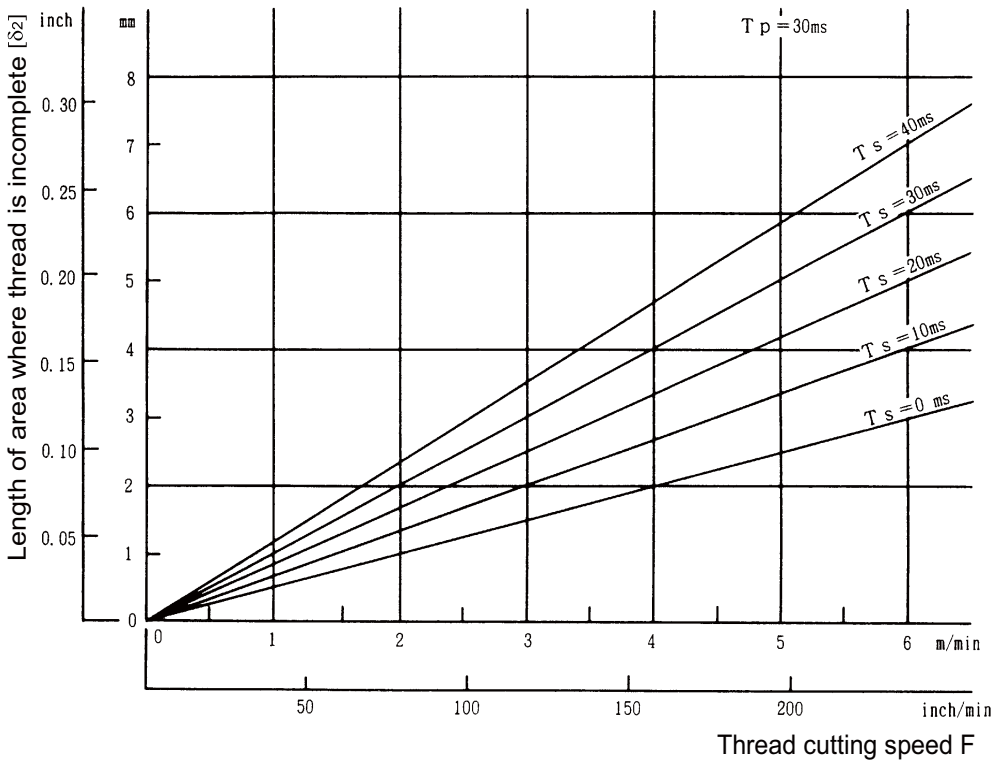


Chart to calculate length of area where thread is incomplete [δ2]

26

Appx.3: Parameter Input by Program N No. (G10 L50, G11)

Function and purpose

The parameters set from the setting and display unit can be changed in the machining programs.

Note

(1) G10 L50 is for specific models only (no specification for M800 Series).

Command format

Data setting command

```
G10 L50 ;
P_ N_ H □ _ ;
P_ A_ N_ D_ ;
P_ A_ N_ S_ ;
P__ A axis No. N__ L__ ;
```

P	Major classification No.
N	Data No.
A	Axis No.
H	Bit type data
D	Byte type data
S	S word type data
L	2 word type data

Data setting mode cancel (data setting completed)

```
G11;
```

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

P__N__H □ __ ;	Bit-type parameter
P__N__D__ ;	Byte-type parameter
P__N__S__ ;	Word-type parameter
P__N__L__ ;	2-word-type parameter

With axis-independent data

P__A__N__H □ __ ;	Bit-type parameter
P__A__N__D__ ;	Byte-type parameter
P__A__N__S__ ;	Word-type parameter
P__A__N__L__ ;	2-word-type parameter

Note

(1) The sequence of addresses in a block must be as shown above.

(2) For a bit type parameter, the data type will be H □ (□ is a value between 0 and 7).

(3) The axis No. is set in the following manner: 1st axis is 1, 2nd axis is 2, and so forth.
When using multi-part systems, the 1st axis in each part system is set as 1, the 2nd axis is set as 2, and so forth.

(4) Command G10 L50, G11 in independent blocks. A program error (P33, P421) will occur if not commanded in independent blocks.

Detailed description

(1) The units in the table indicate the minimum setting units for the parameter data.

(2) The setting ranges given in the table are the setting ranges on the screen. Designate parameters related to the length by doubling the input setting unit.
(Example 1) If the input setting unit in the metric system is "B" (0.001mm), set 30mm in the parameter.
L60000
(Example 2) If the input setting unit in the inch system is "B" (0.0001inch), set 5inch in the parameter.
L100000

(3) The binary type parameters must be converted into byte type data, and commanded with a decimal data following address D.
(Example 1) Binary data
01010101B = 55H = 85D : Command 85
(Example 2) ASCII code
"M" = 01001101B = 4DH = 77D : Command 77

(B indicates Binary, H indicates Hexadecimal, and D indicates Decimal.)

(4) In a system with a section No. (P) (in case of G10 L50), the section No. is assigned according to the function increment after No. 20. Thus, the No. may be duplicated with the parameters that can be set with other section Nos.

[Machining parameters]

#	Item	P	A	N	Data type	Setting range	Unit
8007	Override	11	-	1010	D	0 - 100	%
8008	MAX ANGLE	11	-	756	L	0 - 180	°
8009	DSC.ZONE	11	-	760	L	0 - 199999998	0.5μm
8010	ABS. MAX.	11	-	776	L	0 - 199998	0.5μm
8011	INC. MAX.	11	-	780	L	0 - 199998	0.5μm
8013	G83 n	11	-	832	L	0 - 199999998	0.5μm
8014	CDZ-VALE	11	-	1012	D	0 - 127	0.1 lead
8015	CDZ-ANGLE	11	-	1011	D	0 - 89	°
8016	G71 MINIMUM	11	-	788	L	0 - 199998	0.5μm
8017	G71 DELTA-D	11	-	792	L	0 - 199998	0.5μm
8051	G71 THICK	11	-	784	L	0 - 199999998	0.5μm
8052	G71 PULL UP	11	-	796	L	0 - 199999998	0.5μm
8053	G73 U	11	-	800	L	±199999998	0.5μm
8054	G73 W	11	-	804	L	±199999998	0.5μm
8055	G73 R	11	-	808	L	0 - 99999	
8056	G74 RETRACT	11	-	820	L	0 - 199998	0.5μm
8057	G76 LAST-D	11	-	824	L	0 - 199998	0.5μm
8058	G76 TIMES	11	-	997	D	0 - 99	
8059	G76 ANGLE	11	-	998	D	0 - 99	

[Axis parameters]							
#	**Item**	**P**	**A**	**N**	**Data type**	**Setting range**	**Unit**
8202	OT-CHECK OFF	2	No.	897	H2	0 - 1	
8204	OT-CHECK-N	2	No.	916	L	±199999998	0.5μm
8205	OT-CHECK-P	2	No.	912	L	±199999998	0.5μm

[Barrier data]							
#	**Item**	**P**	**A**	**N**	**Data type**	**Setting range**	**Unit**
8300	(P0) X	11	-	1128	L	±199999998	0.5μm
8301	(P1) X	11	No.	1136	L	±199999998	0.5μm
	Z	11	No.	1160	L	±199999998	0.5μm
8302	(P2) X	11	No.	1140	L	±199999998	0.5μm
	Z	11	No.	1164	L	±199999998	0.5μm
8303	(P3) X	11	No.	1144	L	±199999998	0.5μm
	Z	11	No.	1168	L	±199999998	0.5μm
8304	(P4) X	11	No.	1148	L	±199999998	0.5μm
	Z	11	No.	1172	L	±199999998	0.5μm
8305	(P5) X	11	No.	1152	L	±199999998	0.5μm
	Z	11	No.	1176	L	±199999998	0.5μm
8306	(P6) X	11	No.	1156	L	±199999998	0.5μm
	Z	11	No.	1180	L	±199999998	0.5μm

[BASE SPEC PARAM]							
#	**Item**	**P**	**A**	**N**	**Data type**	**Setting range**	**Unit**
1595	hobm	11	-	595	D	1 to the number of spindles	
1596	hobs	11	-	596	D	1 to the number of the axis in the part system	

[AXIS SPEC PARAM]							
#	**Item**	**P**	**A**	**N**	**Data type**	**Setting range**	**Unit**
2592	Extstnum	2	No.	2592	S	4101 - 5508	
2593	Extset	2	No.	2593	D	0 - 32	
2594	Extsc	2	No.	2594	D	0 - 99	

Program example

```
G10 L50 ;
P11 N1010 D30 ; parameter "#8007 OVERRIDE" to "30"
G11 ;
```

Index

Refer to Programming Manual (Lathe System) (1/2) for Chapter 14 and previous chapters (page 522 and before).

Refer to Programming Manual (Lathe System) (2/2) for Chapter 15 and succeeding chapters (page 523 and later).

H

I

T

U

V

W

Revision History

Date of revision	Manual No.	Revision details
Apr. 2015	IB(NA)1501275-A IB(NA)1501276-A	First edition created.
Sep. 2015	IB(NA)1501275-B IB(NA)1501276-B	The descriptions of M800 Series/M80 Series were revised in response to S/W version A4. The following chapters were added. •6.6.6 Thread Cutting Override •6.6.7 Variable Feed Thread Cutting •7.15.2 Inner Arc Override •15.7.3 Tool/Material Shape Input by Program; G10 L100/L101, G11 The following chapters were revised. •3.4 G Codes •5.4 Decimal Point Input •7.3 F1-digit Feed •7.13 Deceleration Check •7.15 Automatic Corner Override •10.4 Multiple-spindle Control •13.5 Fixed Cycle for Drilling •14.4 Macro Call Instructions •15.7.2 Compensation Data Input by Program; G10 L2/L10/L11, G11 •16.4.2 Arbitrary Axis Superimposition; G156 •16.9 Sub Part System Control •17.1 High-speed Machining Mode •17.2 High-accuracy Control •17.3 High-speed High-accuracy Control •18.1 Spindle Synchronization •18.2 Tool Spindle Synchronization I •18.4 Spindle Superimposition; G164, G113 •19.3 Basic Machine Coordinate System Selection; G53 •19.6 Workpiece Coordinate System Setting and Offset; G54 to G59 (G54.1) The following chapters were moved. •Parameter Input by Program; G10 L70/L100, G11 (15.6 -> 15.7.1) •Compensation Data Input by Program; G10 L2/L10/L11, G11 (12.6 -> 15.7.2) •Tool Life Management II; G10 L3, G11 (12.7 -> 15.8) Other contents were added/revised/deleted according to the specifications.
Apr. 2016	IB(NA)1501275-C IB(NA)1501276-C	The descriptions of M800 Series/M80 Series were revised in response to S/W version B2. The following chapters were added. •12.1.5 Tool Compensation for 2nd Additional Axis •18.5 Multiple Spindle Synchronization Set Control •19. Advanced Machining Control •19.1 Simple Inclined Surface Machining •23. System Variables (Continue to the next page)

Date of revision	Manual No.	Revision details
		(Continued from the previous page) The following chapters were revised. •Introduction •3.4 G Codes •5.4 Decimal Point Input •6.3 Circular Interpolation; G02/G03 •6.4 R Specification Circular Interpolation; G02, G03 •6.6 Thread Cutting •6.7 Helical Interpolation; G17, G18, G19, and G02, G03 •7.3 F1-digit Feed •7.8 Rapid Traverse Constant Inclination Acceleration/Deceleration •7.14 Rapid Traverse Block Overlap; G0.5 P1 •9.3 Index Table Indexing •10.4 Multiple-spindle Control •10.5 Spindle Position Control (Spindle/C Axis Control) •12.1 Tool Compensation •13.5 Fixed Cycle for Drilling •14.1 Subprogram Control; M98, M99, M198 •14.2 Variable Commands •14.4 Macro Call Instructions •14.6 User Macro Commands •15.5 Geometric IB •15.6 Manual Arbitrary Reverse Run Prohibition; G127 •15.7 Data Input by Program •16.3 Mixed Control •16.4 Control Axis Superimposition •16.6 Multi-part System Simultaneous Thread Cutting Cycle •16.7 Multi-part System Simultaneous Thread Cutting Cycle (MITSUBISHI CNC Special Format); G76.1, G76.2 •16.9 Sub Part System Control •17.1 High-speed Machining Mode •17.2 High-accuracy Control •17.3 High-speed High-accuracy Control •18.1 Spindle Synchronization •18.2 Tool Spindle Synchronization I •18.3 Tool Spindle Synchronization II •18.4 Spindle Superimposition; G164, G113 •20.6 Workpiece Coordinate System Setting and Offset; G54 to G59 (G54.1) •22.2 Skip Function; G31 Other mistakes were corrected.
Sep. 2016	IB(NA)1501275-D IB(NA)1501276-D	The descriptions were revised in response to S/W version C1 of M800 Series/M80 Series. The descriptions were revised in response to S/W version A1 of C80 Series. The following chapters were added. •6.6.9 Thread Cutting Start Shift Angle Operation Switching •6.6.10 Thread Cutting Feed Forward •21.3 Enable Interfering Object Selection Data; G186 •23.24 System Variables (Interfering Object Selection) •23.25 System Variables (ZR Device Access Variables) [C80] The following chapters were revised. •Introduction •Precautions for Safety •3.2 File Format •3.4 G Codes •6.8 Milling Interpolation; G12.1 (Continue to the next page)

Date of revision	Manual No.	Revision details
		♦6.9 Cylindrical Interpolation; G07.1 (Only 6 and 7 in G Code List) ♦6.10 Polar Coordinate Interpolation; G12.1,G13.1/G112,G113 (Only 6 and 7 in G Code List) ♦7.4 Feed Per Minute/Feed Per Revolution (Asynchronous Feed/Synchronous Feed); G94, G95 ♦13.1 Fixed Cycle for Turning Machining ♦13.2 Fixed Cycles for Turning Machining (MITSUBISHI CNC special format); G77, G78, G79 ♦13.5 Fixed Cycle for Drilling ♦14.1 Subprogram Control; M98, M99, M198 ♦14.6 User Macro Commands ♦15.7.1 Parameter Input by Program; G10 L70, G11 ♦16.3 Mixed Control ♦16.9 Sub Part System Control ♦17.1 High-speed Machining Mode ♦17.3 High-speed High-accuracy Control ♦18.5 Multiple Spindle Synchronization Set Control ♦19.1 Simple Inclined Surface Machining ♦20.8 Workpiece Coordinate System Preset; G92.1 ♦22.6 Torque Limitation Skip; G160 ♦23.1 System Variable List ♦23.5 System Variables (Tool Information) ♦23.15 System Variables (Time Read Variables) ♦23.22 System Variables (R Device Access Variables) Other mistakes were corrected.

M800/M80/C80 Series Manual List

These contents are described in the presupposition that all functions of M800/M80/C80 Series are available.
Some functions or screens may not be available depending on the machine or specifications set by MTB. (Confirm the specifications before use.)
The manuals issued by MTB take precedence over these manuals.

Manual	IB No.	Purpose and Contents
M800/M80 Series Instruction Manual	IB-1501274	- Operation guide for NC - Explanation for screen operation, etc.
C80 Series Instruction Manual	IB-1501453	- Operation guide for NC - Explanation for screen operation, etc.
M800/M80/C80 Series Programming Manual (Lathe System) (1/2)	IB-1501275	- G code programming for lathe system - Basic functions, etc.
M800/M80/C80 Series Programming Manual (Lathe System) (2/2)	IB-1501276	- G code programming for lathe system - Functions for multi-part system, high-accuracy function, etc.
M800/M80/C80 Series Programming Manual (Machining Center System) (1/2)	IB-1501277	- G code programming for machining center system - Basic functions, etc.
M800/M80/C80 Series Programming Manual (Machining Center System) (2/2)	IB-1501278	- G code programming for machining center system - Functions for multi-part system, high-accuracy function, etc.
M800/M80/C80 Series Alarm/Parameter Manual	IB-1501279	- Alarms - Parameters

Manuals for MTBs (NC)

Manual	IB No.	Purpose and Contents
M800/M80/C80 Series Specifications Manual	IB-1501267	- Model selection - Specifications of hardware unit - Outline of various functions
M800W/M80W Series Connection and Setup Manual	IB-1501268	- Detailed specifications of hardware unit - Installation, connection, wiring, setup (startup/adjustment)
M800S/M80 Series Connection and Setup Manual	IB-1501269	- Detailed specifications of hardware unit - Installation, connection, wiring, setup (startup/adjustment)
C80 Series Connection and Setup Manual	IB-1501452	- Detailed specifications of hardware unit - Installation, connection, wiring, setup (startup/adjustment)
M800/M80 Series PLC Development Manual	IB-1501270	- Electrical design - I/O relation (assignment, setting, connection), field network - Development environment (PLC on-board, peripheral development environment), etc.
M800/M80 Series PLC Programming Manual	IB-1501271	- Electrical design - Sequence programming - PLC support functions, etc.
M800/M80/C80 Series PLC Interface Manual	IB-1501272	- Electrical design - Interface signals between NC and PLC
M800/M80 Series Maintenance Manual	IB-1501273	- Cleaning and replacement for each unit - Other items related to maintenance
C80 Series Maintenance Manual	IB-1501454	- Cleaning and replacement for each unit - Other items related to maintenance

Manuals for MTBs (drive section)

Manual	IB No.	Contents
MDS-E/EH Series Specifications Manual	IB-1501226	- Specifications for power supply regeneration type
MDS-E/EH Series Instruction Manual	IB-1501229	- Instruction for power supply regeneration type
MDS-EJ/EJH Series Specifications Manual	IB-1501232	- Specifications for regenerative resistor type
MDS-EJ/EJH Series Instruction Manual	IB-1501235	- Instruction for regenerative resistor type
MDS-EM/EMH Series Specifications Manual	IB-1501238	- Specifications for multi-hybrid, power supply regeneration type
MDS-EM/EMH Series Instruction Manual	IB-1501241	- Instruction for multi-hybrid, power supply regeneration type
DATA BOOK	IB-1501252	- Specifications of servo drive unit, spindle drive unit, motor, etc.

Global Service Network

AMERICA

MITSUBISHI ELECTRIC AUTOMATION INC. (AMERICA FA CENTER)

Central Region Service Center (Chicago)
500 CORPORATE WOODS PARKWAY, VERNON HILLS, ILLINOIS 60061, U.S.A.
TEL: +1-847-478-2500 / FAX: +1-847-478-2650
Minneapolis, MN Service Satellite
Detroit, MI Service Satellite
Grand Rapids, MI Service Satellite
Lima, OH Service Satellite
Cleveland, OH Service Satellite
Indianapolis, IN Service Satellite
St. Louis, MO Service Satellite

South/East Region Service Center (Georgia)
1845 SATTELITE BOULEVARD STE. 450, DULUTH, GEORGIA 30097, U.S.A.
TEL +1-678-258-4529 / FAX +1-678-258-4519
Charleston, SC Service Satellite
Charlotte, NC Service Satellite
Raleigh, NC Service Satellite
Dallas, TX Service Satellite
Houston, TX Service Satellite
Hartford, CT Service Satellite
Knoxville, TN Service Satellite
Nashville, TN Service Satellite
Baltimore, MD Service Satellite
Pittsburg, PA Service Satellite
Allentown, PA Service Satellite
Syracuse, NY Service Satellite
Tampa, FL Service Satellite
Lafayette, LA Service Satellite

Western Region Service Center (California)
5900-B KATELLA AVE. - 5900-A KATELLA AVE. CYPRESS, CALIFORNIA 90630, U.S.A.
TEL: +1-714-699-2625 / FAX: +1-847-478-2650
San Francisco, CA Service Satellite
Seattle, WA Service Satellite

Canada Region Service Center (Tronto)
4299 14TH AVENUE MARKHAM, ONTARIO L3R OJ2, CANADA
TEL: +1-905-754-3805 / FAX: +1-905-475-7935
Edmonton, AB Service Satellite
Montreal, QC Service Satellite

Mexico Region Service Center (Queretaro)
Parque Tecnológico Innovación Querétaro, Lateral Carretera Estatal 431, Km 2+200, Lote 91 Modulos 1 y 2
Hacienda la Machorra, CP 76246, El Marqués, Querétaro, México
TEL: +52-442-153 4250
Monterrey, NL Service Satellite
Mexico City, DF Service Satellite

BRAZIL

MELCO CNC do Brasil Comércio e Serviços Ltda.

Brazil Region Service Center
AV. GISELE CONSTANTINO,1578, PARQUE BELA VISTA, VOTORANTIM-SP, BRAZIL CEP:18.110-650
TEL: +55-15-3023-9000
JOVIMAQ – Joinville, SC Service Satellite
MAQSERVICE – Canoas, RS Service Satellite

EUROPE

MITSUBISHI ELECTRIC EUROPE B.V.

European Service Headquarter (Dusseldorf, GERMANY)
Mitsubishi-Electric-Platz 1 40882 RATINGEN, GERMANY
TEL: +49-2102-486-1850 / FAX: +49-2102-486-5910

South Germany Service Center (Stuttgart)
KURZE STRASSE. 40, 70794 FILDERSTADT-BONLANDEN, GERMANY
TEL: + 49-711-770598-123 / FAX: +49-711-770598-141

France Service Center (Paris)
25, BOULEVARD DES BOUVETS, 92741 NANTERRE CEDEX FRANCE
TEL: +33-1-41-02-83-13 / FAX: +33-1-49-01-07-25

France Service Satellite (Lyon)
120, ALLEE JACQUES MONOD 69800 SAINT PRIEST FRANCE
TEL: +33-1-41-02-83-13 / FAX: +33-1-49-01-07-25

Italy Service Center (Milan)
VIALE COLLEONI, 7 - CENTRO DIREZIONALE COLLEONI PALAZZO SIRIO INGRESSO 1,
20864 AGRATE BRIANZA (MB), ITALY
TEL: +39-039-6053-342 / FAX: +39-039-6053-206

Italy Service Satellite (Padova)
VIA G. SAVELLI, 24 - 35129 PADOVA, ITALY
TEL: +39-039-6053-342 / FAX: +39-039-6053-206

U.K. Service Center
TRAVELLERS LANE, HATFIELD, HERTFORDSHIRE, AL10 8XB, U.K.
TEL: +49-2102-486-1850 / FAX: +49-2102-486-5910

Spain Service Center
CTRA. DE RUBI, 76-80-APDO. 420, 08173 SAINT CUGAT DEL VALLES, BARCELONA SPAIN
TEL: +34-935-65-2236 / FAX: +34-935-89-1579

Poland Service Center
UL.KRAKOWSKA 50, 32-083 BALICE, POLAND
TEL: +48-12-347-6500 / FAX: +48-12-630-4701

Hungary Service Center
MADARASZ VIKTOR 47-49 , BUDAPEST XIII; HUNGARY
TEL: +48-12-347-6500 / FAX: +48-12-630-4701

Turkey Service Center
MITSUBISHI ELECTRIC TURKEY A.Ş
SERIFALI MAHALLESI NUTUK SOKAK. NO.5 34775
UMRANIYE, ISTANBUL, TURKEY
TEL: +90-216-526-3990 / FAX: +90-216-526-3995

Czech Republic Service Center
AutoCont Control Systems s.r.o (Service Partner)
KAFKOVA 1853/3, 702 00 OSTRAVA 2, CZECH REPUBLIC
TEL: +420-59-5691-185 / FAX: +420-59-5691-199

Russia Service Center
NC-TECH (Service Partner)
213, B.NOVODMITROVSKAYA STR., 14/2, 127015 MOSCOW, RUSSIA
TEL: +7-495-748-0191 / FAX: +7-495-748-0192

Sweden Service Center
HAMMARBACKEN 14, P.O.BOX 750 SE-19127, SOLLENTUNA, SWEDEN
TEL: +46-8-6251000 / FAX: +46-8-966877

Bulgaria Service Center
AKHNATON Ltd. (Service Partner)
4 ANDREJ LJAPCHEV BLVD. POB 21, BG-1756 SOFIA, BULGARIA
TEL: +359-2-8176009 / FAX: +359-2-9744061

Ukraine Service Center (Kharkov)
CSC Automation Ltd. (Service Partner)
APTEKARSKIY PEREULOK 9-A, OFFICE 3, 61001 KHARKOV, UKRAINE
TEL: +380-57-732-7774 / FAX: +380-57-731-8721

Belarus Service Center
TECHNIKON Ltd. (Service Partner)
NEZAVISIMOSTI PR.177, 220125 MINSK, BELARUS
TEL: +375-17-393-1177 / FAX: +375-17-393-0081

South Africa Service Center
MOTIONTRONIX (Service Partner)
P.O. BOX 9234, EDLEEN, KEMPTON PARK GAUTENG, 1625, SOUTH AFRICA
TEL: +27-11-394-8512 / FAX: +27-11-394-8513

ASEAN

MITSUBISHI ELECTRIC ASIA PTE. LTD. (ASEAN FA CENTER)
Singapore Service Center
307 ALEXANDRA ROAD #05-01/02 MITSUBISHI ELECTRIC BUILDING SINGAPORE 159943
TEL: +65-6473-2308 / FAX: +65-6476-7439

Philippines Service Center
Flexible (Service Partner)
UNIT NO.411, ALABAMG CORPORATE CENTER KM 25. WEST SERVICE ROAD
SOUTH SUPERHIGHWAY, ALABAMG MUNTINLUPA METRO MANILA, PHILIPPINES 1771
TEL: +63-2-807-2416 / FAX: +63-2-807-2417

VIETNAM

MITSUBISHI ELECTRIC VIETNAM CO.,LTD
Vietnam Ho Chi Minh Service Center
UNIT 01-04, 10TH FLOOR, VINCOM CENTER 72 LE THANH TON STREET, DISTRICT 1,
HO CHI MINH CITY, VIETNAM
TEL: +84-8-3910 5945 / FAX: +84-8-3910 5946

Vietnam Hanoi Service Center
6TH FLOOR, DETECH TOWER, 8 TON THAT THUYET STREET, MY DINH 2 WARD,
NAM TU LIEM DISTRICT, HA NOI CITY, VIETNAM
TEL: +84-4-3937-8075 / FAX: +84-4-3937-8076

INDONESIA

PT. MITSUBISHI ELECTRIC INDONESIA
Indonesia Service Center (Cikarang)
JL. KENARI RAYA BLOK G2-07A, DELTA SILICON 5, LIPPO CIKARANG - BEKASI 17550, INDONESIA
TEL: +62-21-2961-7797 / FAX: +62-21-2961-7794

MALAYSIA

MITSUBISHI ELECTRIC SALES MALAYSIA SDN. BHD.
Malaysia Service Center (Kuala Lumpur Service Center)
LOT 11, JALAN 219, P.O BOX 1036, 46860 PETALING JAYA, SELANGOR DARUL EHSAN. MALAYSIA
TEL: +60-3-7960-2628 / FAX: +60-3-7960-2629
Johor Bahru Service satellite

THAILAND

MITSUBISHI ELECTRIC FACTORY AUTOMATION (THAILAND) CO.,LTD
Thailand Service Center
12TH FLOOR, SV.CITY BUILDING, OFFICE TOWER 1, NO. 896/19 AND 20 RAMA 3 ROAD,
KWAENG BANGPONGPANG, KHET YANNAWA, BANGKOK 10120,THAILAND
TEL: +66-2-682-6522 / FAX: +66-2-682-6020

INDIA

MITSUBISHI ELECTRIC INDIA PVT., LTD.
CNC Technical Center (Bangalore)
PLOT NO. 56, 4TH MAIN ROAD, PEENYA PHASE 3,
PEENYA INDUSTRIAL AREA, BANGALORE 560058, KARNATAKA, INDIA
TEL : +91-80-4655-2121 FAX : +91-80-4655-2147
Chennai Service Satellite
Coimbatore Service Satellite
Hyderabad Service Satellite

North India Service Center (Gurgaon)
2ND FLOOR, TOWER A&B, DLF CYBER GREENS, DLF CYBER CITY,
DLF PHASE-III, GURGAON- 122 002, HARYANA, INDIA
TEL : +91-124-4630 300 FAX : +91-124-4630 399
Ludhiana Satellite
Panth Nagar Service Satellite
Delhi Service Satellite
Jamshedpur Service Satellite

West India Service Center (Pune)
EMERALD HOUSE, EL-3, J BLOCK, M.I.D.C., BHOSARI, PUNE - 411026, MAHARASHTRA, INDIA
TEL : +91-20-2710 2000 FAX : +91-20-2710 2100
Kolhapur Service Satellite
Aurangabad Service Satellite
Mumbai Service Satellite

West India Service Center (Ahmedabad)
UNIT NO: B/4, 3RD FLOOR, SAFAL PROFITAIRE, PRAHALADNAGAR CORPORATE ROAD,
PRAHALADNAGAR SATELLITE, AHMEDABAD – 380015, GUJRAT, INDIA
TEL : +91-265-2314699
Rajkot Service Satellite

CHINA

MITSUBISHI ELECTRIC AUTOMATION (CHINA) LTD. (CHINA FA CENTER)
China Shanghai Service Center
1-3,5-10,18-23/F, NO.1386 HONG QIAO ROAD, CHANG NING QU,
SHANGHAI 200336, CHINA
TEL: +86-21-2322-3030 / FAX: +86-21-2322-3000*8422
China Ningbo Service Partner
China Wuxi Service Partner
China Jinan Service Partner
China Hangzhou Service Partner

China Beijing Service Center
9/F, OFFICE TOWER 1, HENDERSON CENTER, 18 JIANGUOMENNEI DAJIE,
DONGCHENG DISTRICT, BEIJING 100005, CHINA
TEL: +86-10-6518-8830 / FAX: +86-10-6518-8030
China Beijing Service Partner

China Tianjin Service Center
UNIT 2003, TIANJIN CITY TOWER, NO 35 YOUYI ROAD, HEXI DISTRICT,
TIANJIN 300061, CHINA
TEL: +86-22-2813-1015 / FAX: +86-22-2813-1017

China Chengdu Service Center
1501-1503,15F,GUANG-HUA CENTRE BUILDING-C,NO.98 NORTH GUANG HUA 3th RD,
CHENGDU,610000,CHINA
TEL: +86-28-8446-8030 / FAX: +86-28-8446-8630

China Shenzhen Service Center
ROOM 2512-2516, 25/F., GREAT CHINA INTERNATIONAL EXCHANGE SQUARE, JINTIAN RD.S.,
FUTIAN DISTRICT, SHENZHEN 518034, CHINA
TEL: +86-755-2399-8272 / FAX: +86-755-8229-3686
China Xiamen Service Partner
China DongGuang Service Partner

China Dalian Service Center
DONGBEI 3-5, DALIAN ECONOMIC & TECHNICAL DEVELOPMENTZONE, LIAONING PROVINCE,
116600, CHINA
TEL: +86-411-8765-5951 / FAX: +86-411-8765-5952

KOREA

MITSUBISHI ELECTRIC AUTOMATION KOREA CO., LTD. (KOREA FA CENTER)
Korea Service Center
8F GANGSEO HANGANG XI-TOWER A, 401 YANGCHEON-RO, GANGSEO-GU,
SEOUL 07528 KOREA
TEL: +82-2-3660-9609 / FAX: +82-2-3664-8668
Korea Daegu Service Satellite

TAIWAN

MITSUBISHI ELECTRIC TAIWAN CO., LTD. (TAIWAN FA CENTER)
Taiwan Taichung Service Center
NO.8-1, INDUSTRIAL 16TH RD., TAICHUNG INDUSTRIAL PARK, SITUN DIST.,
TAICHUNG CITY 40768, TAIWAN
TEL: +886-4-2359-0688 / FAX: +886-4-2359-0689

Taiwan Taipei Service Center
10F, NO.88, SEC.6, CHUNG-SHAN N. RD., SHI LIN DIST., TAIPEI CITY 11155, TAIWAN
TEL: +886-2-2833-5430 / FAX: +886-2-2833-5433

Taiwan Tainan Service Center
11F-1., NO.30, ZHONGZHENG S. ROAD, YONGKANG DISTRICT, TAINAN CITY 71067, TAIWAN
TEL: +886-6-252-5030 / FAX: +886-6-252-5031

OCEANIA

MITSUBISHI ELECTRIC AUSTRALIA PTY. LTD.
Oceania Service Center
348 VICTORIA ROAD, RYDALMERE, N.S.W. 2116 AUSTRALIA
TEL: +61-2-9684-7269/ FAX: +61-2-9684-7245

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.

MITSUBISHI CNC

MITSUBISHI ELECTRIC CORPORATION
HEAD OFFICE : TOKYO BLDG.,2-7-3 MARUNOUCHI,CHIYODA-KU,TOKYO 100-8310,JAPAN

MODEL	M800/M80/C80 Series
MODEL CODE	100-441
Manual No.	IB-1501276

IB(NA)1501276(ENG)-D(1609)MEE
Made in JAPAN