MXTTPGE68

PROGRAMMING MANUAL

PUMA series

MX2000, MX2000L/T/LT/S/LS/LST

MX2500, MX2500L/T/LT/S/LS/LST

MX3000, MX3000S

TT1500MS/SY

TT1800MS/SY

TT2000SY

TT2500MS/SY

DOOSAN Infracore

Table of contents

Table of contents

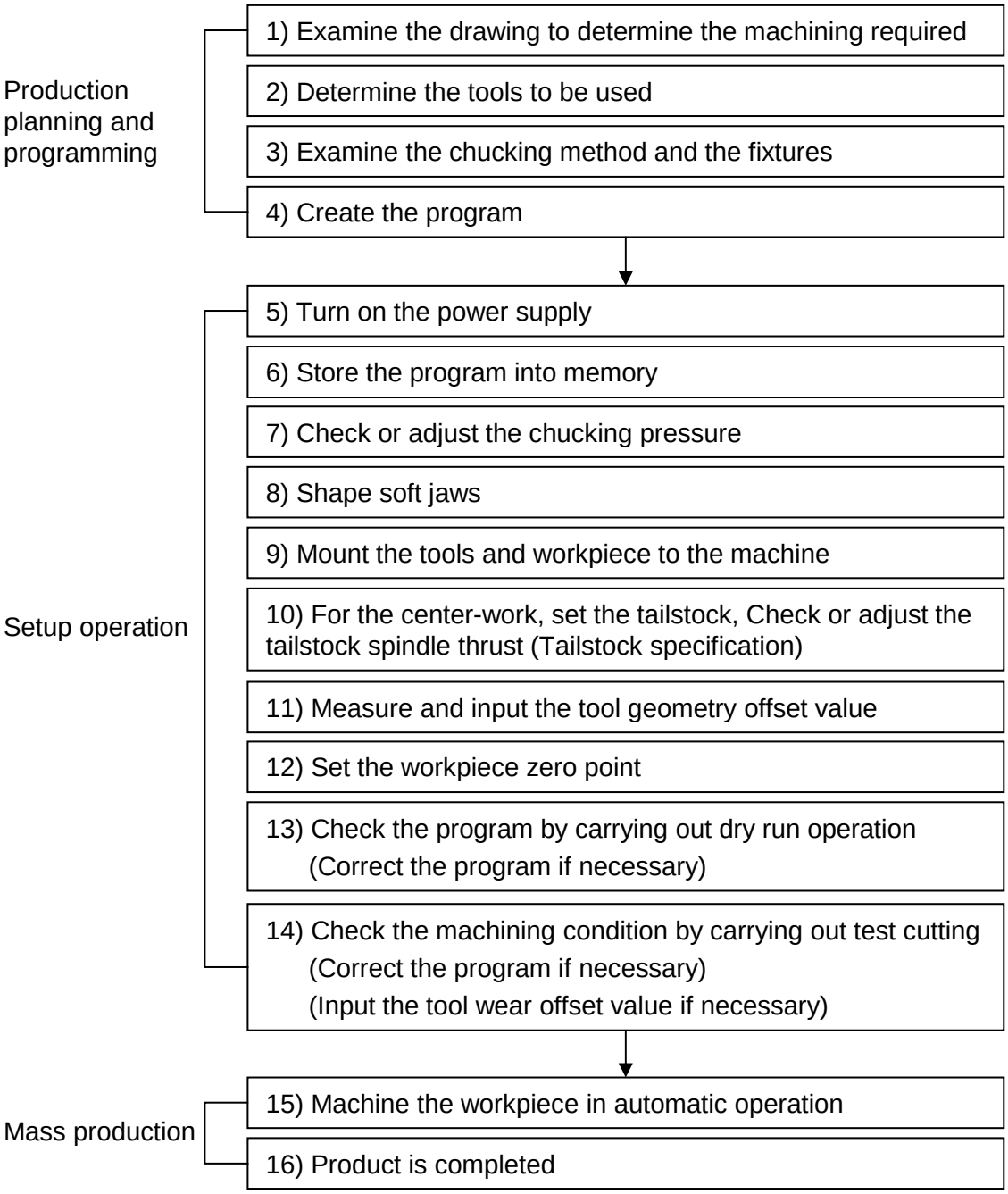
Table of contents

1. Before Programming

1.1 Flow until the product is completed

1) Flow of Operation

This section describes the flow of operation, including programming. Follow and understand the flow so that the operation can be performed smoothly.



1.2 Terms for programming

1) Program Number

Several programs can be stored in the NC memory.

Program numbers are used to keep multiple programs arranged in numerical order.

Program numbers appear at the beginning of a program stored in the memory.

A program number is set by inputting numbers four digits or less after the alphabet “O” Numbers from 1 to 9999 can be used. But, don’t use from O9000 to O9999. Because these numbers are reserved for Doosan macro program.

2) Sequence Number

The sequence number is used to search for or call the position that is being executed, or to facilitate finding the position you want to edit in the program easily.

The sequence number is expressed as a number of five digits or less (1 to 99999), following the letter “N”.

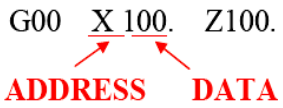
Generally, the sequence numbers are assigned to the part programs for individual cutting tools in the ascending order in the order the machining processes are executed.

3) Address

An address is expressed using letters of the alphabet.

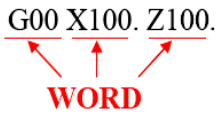
4) Data

The numbers (including the sign and decimal point) that follow the address are called the “data”.



5) Word

A word is the minimum unit for specifying functions. A word consists of an address and the data.



6) Block

A block is the minimum command unit necessary to operate a machine (including the NC unit). It is also the minimum unit used to create a part program. A block consists of words. On the program sheet, each on line corresponds to one block.

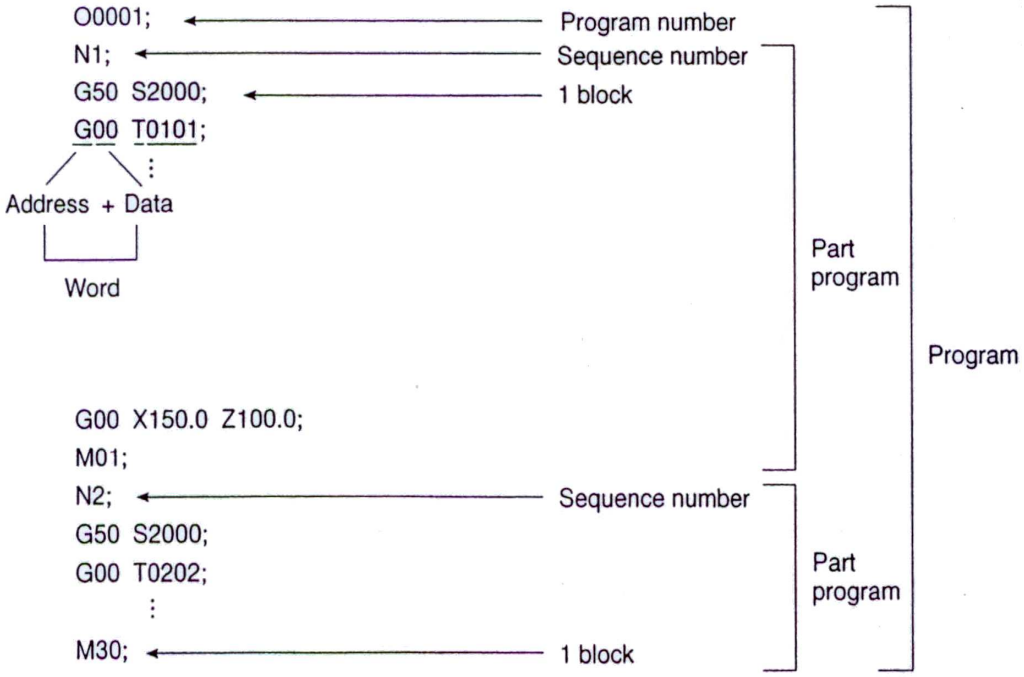
Specify the end of a block with [;].

One block

N	G	X Y	F	S	T	M	;
Sequence Auxiliary No.	Preparation function	Dimension word	Feed function	Spindle speed function	Tool function	Function auxiliary	EOB

7) Summary

A program consists of words, a combination of address and data, and of blocks, a combination of words, as shown below

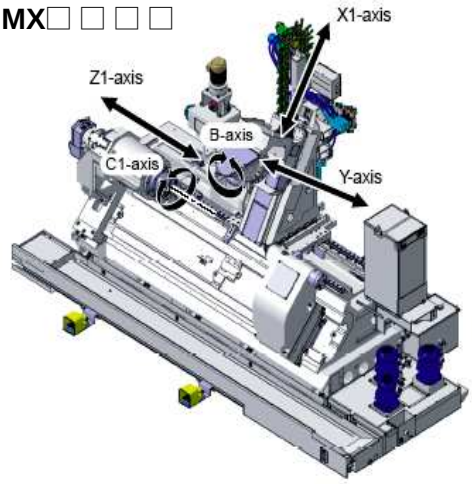


Function	Address	Meaning of address
Program number	O(EIA)/(ISO)	Program number
Block sequence number	N	Sequence number
Preparatory function	G	Sercifies a motion mode (Linear, Arc, etc)
Dimension word	X,Z	Command of moving position (absolute type) of each axis
	U,W	Instruction of moving distance and direction (increamental type)
	I,K	ingredient of each axis and chamfering volume of circulat center
	R	Radius of circle, corner R, edge R
Feed function	F,E	Designation of feedrate and thread lead
Auxiliary funcion	M	Command of ON/OFF for operation parts of machine
Spindle speed functin	S	Designation of speed of main spindle or rotation time of main spindle
Tool function	T	Designation of tool number and tool compensation number
Dwell	P,U,X	Designation of dwell time
Designation of program number	P	Designation of calling number of auxiliary program
Designation of sequence number	P,Q	Calling of compound repeat cycle, end number
Number of repetitions	L	Repeat time of auxiliary program
Parameters	A,D,I,K	Parameter at fixed cycle

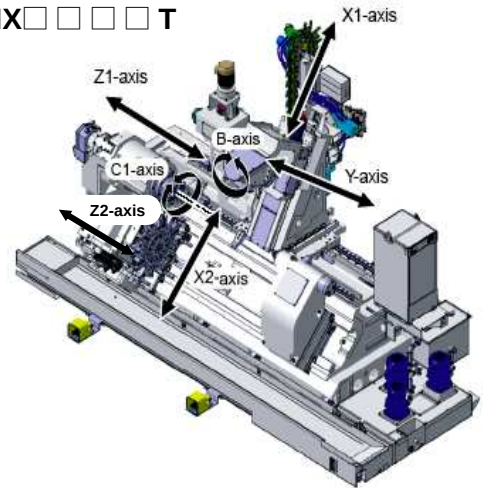
1.3 Axis control and direction

1.3.1 Concept of axis (MX series)

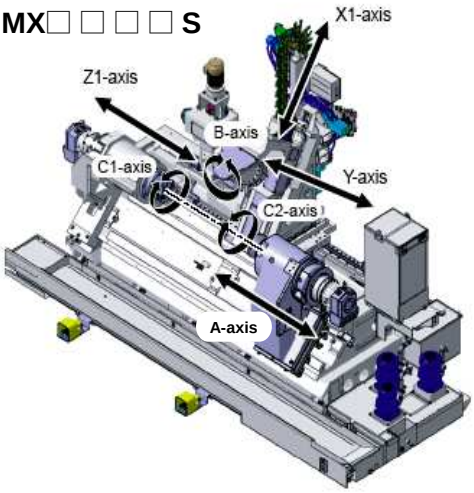
(1) MX□ □ □ □



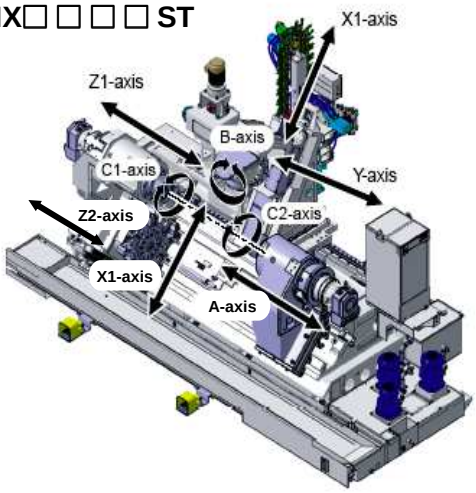
(2) MX□ □ □ □ T

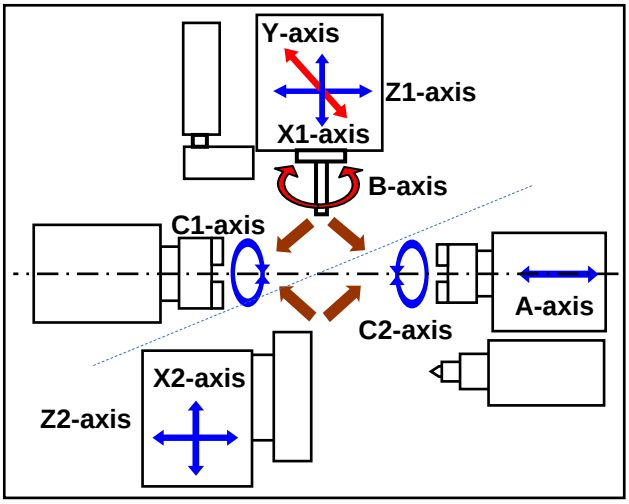


(3) MX□ □ □ □ S



(4) MX□ □ □ □ ST

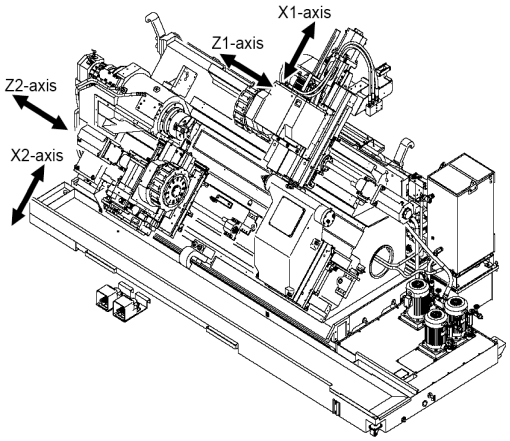




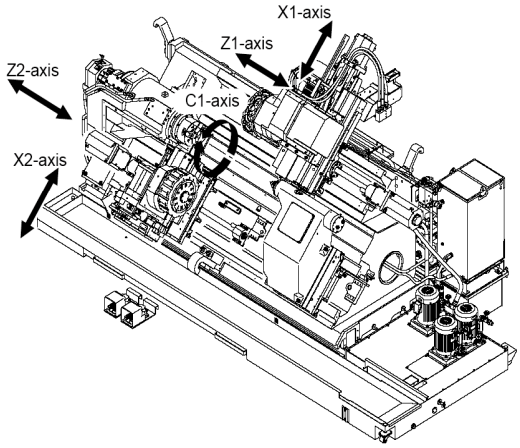
5 axes	7 axes	9 axes
X1,Z1,Y C1, B	X1,Z1,Y C1,B X2,Z2 or A,C2	X1,Z1,Y C1,B X2,Z2,C2,A
MX2000 MX2500	MX2000T MX2000S MX2500T MX2500S	MX2000ST MX2500ST

1.3.2 Concept of axis (TT series)

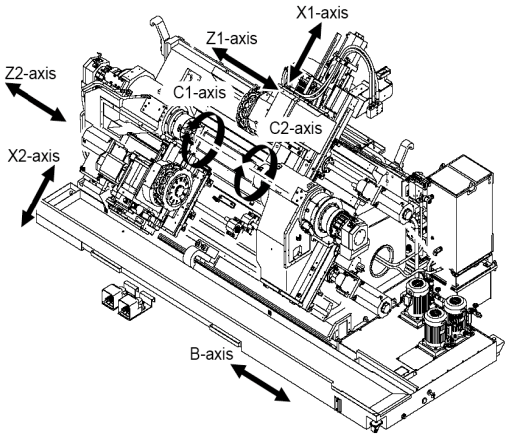
(1) TT□ □ □ □



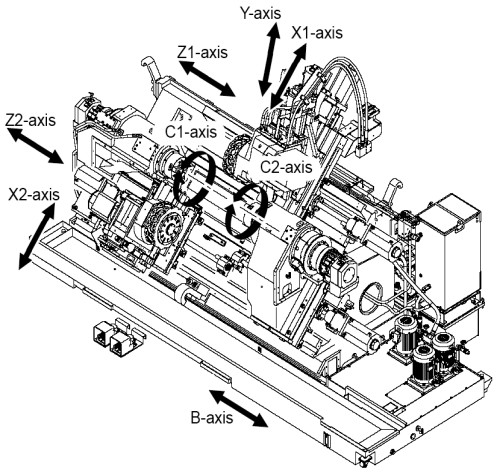
(2) TT□ □ □ □ M

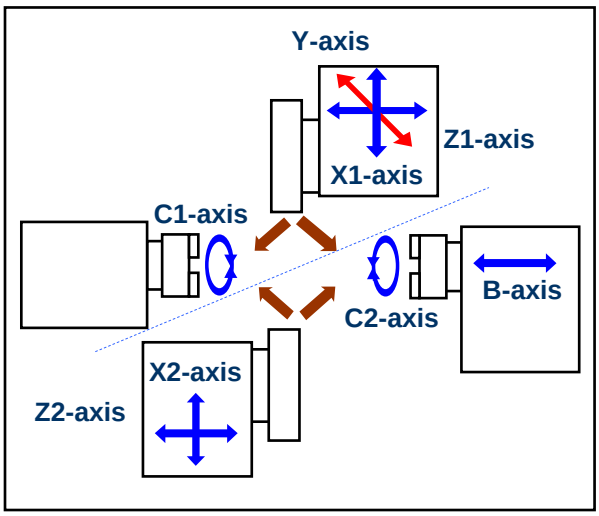


(3) TT□ □ □ □ MS



(4) TT□ □ □ □ SY

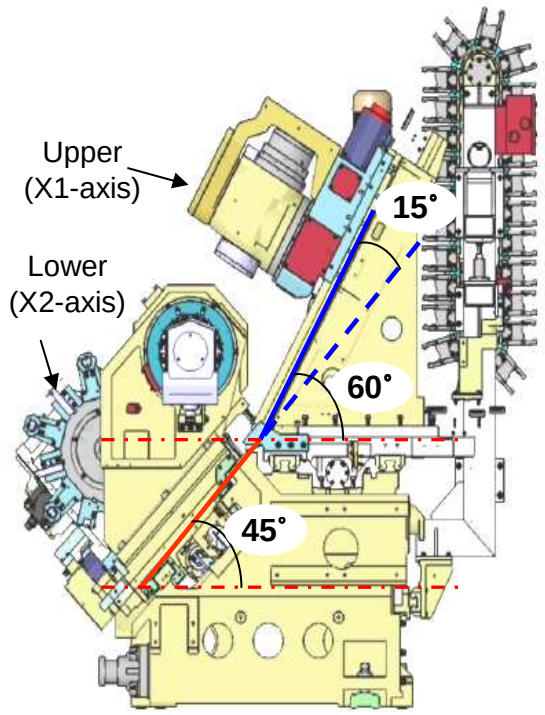




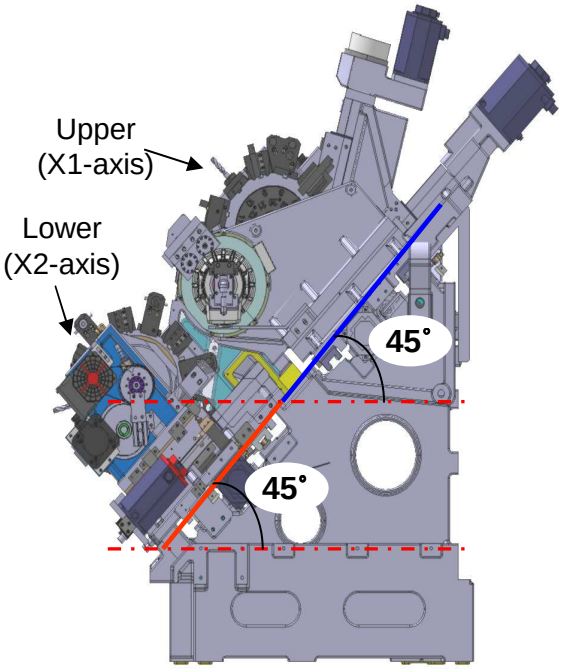
5 axes	7 axes	8 axes
X1,Z1,C1 X2,Z2 or B	X1,Z1,C1 X2,Z2,C2,B	X1,Z1,C1,Y X2,Z2,C2,B
TT2000M TT2000S TT2500M TT2500S	TT2000MS TT2500MS	TT2000SY TT2500SY

1.3.3 Construction of MX & TT series Machine

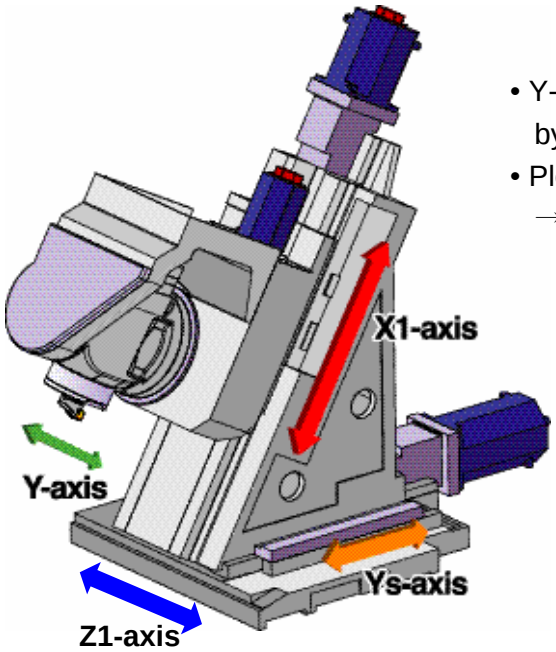
In MX series, Upper slant angle differs from Lower slant angle.
Please check 15˚ angle in programming



[MX series]



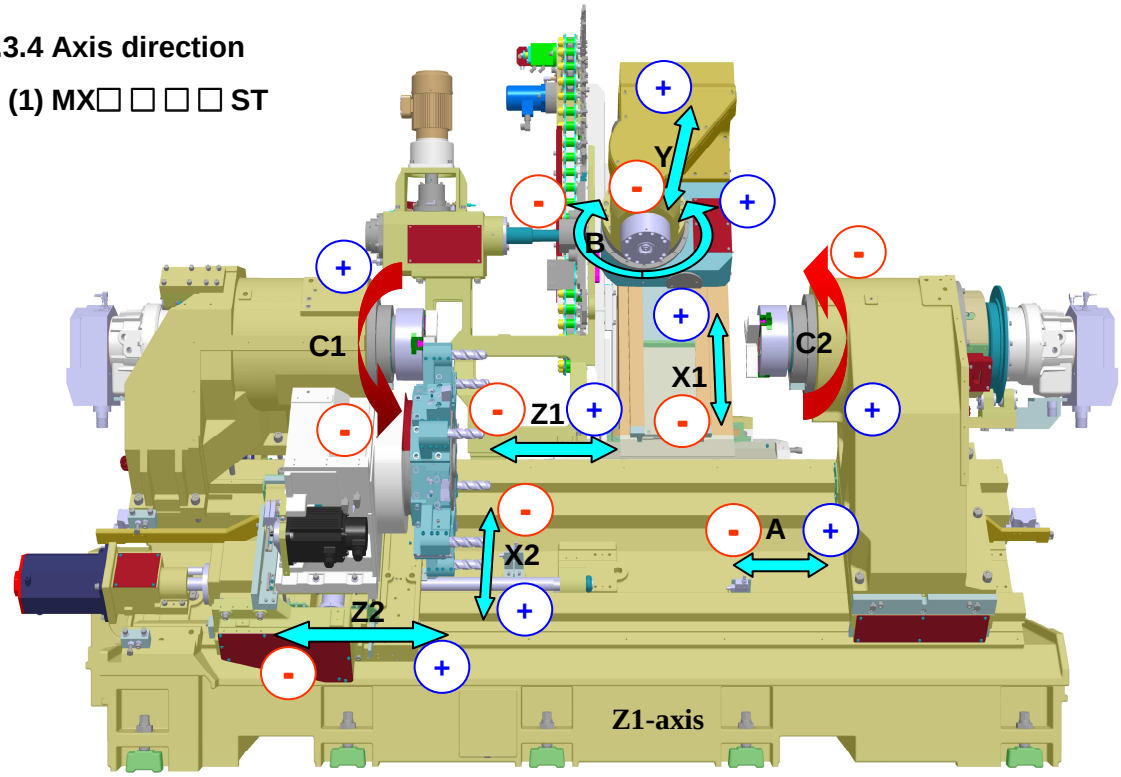
[TT series]

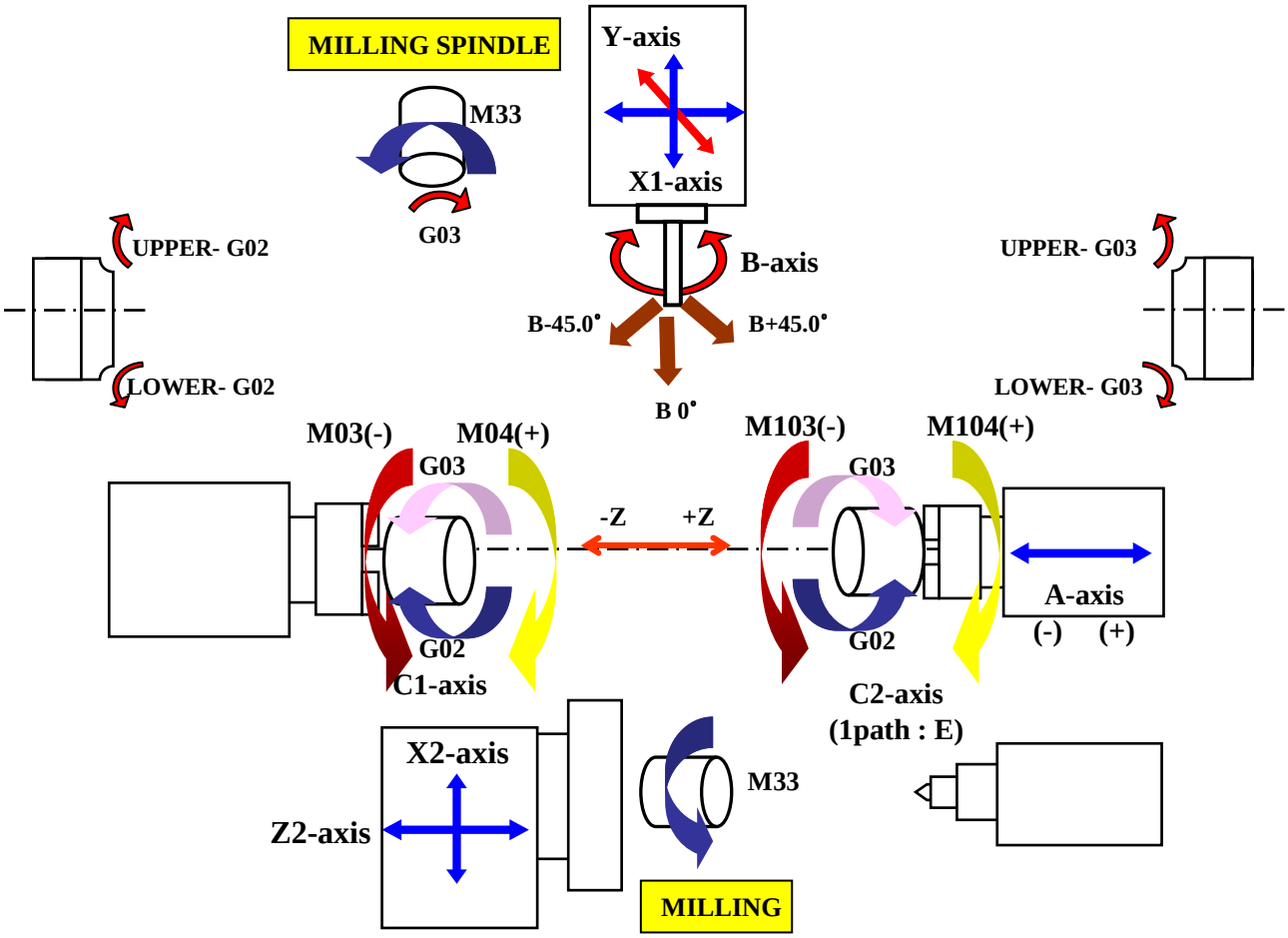


- Y-axis is composite axis controlled by X1 and Ys.
- Please check Upper G30 position
  → G30 position is Tool changing position

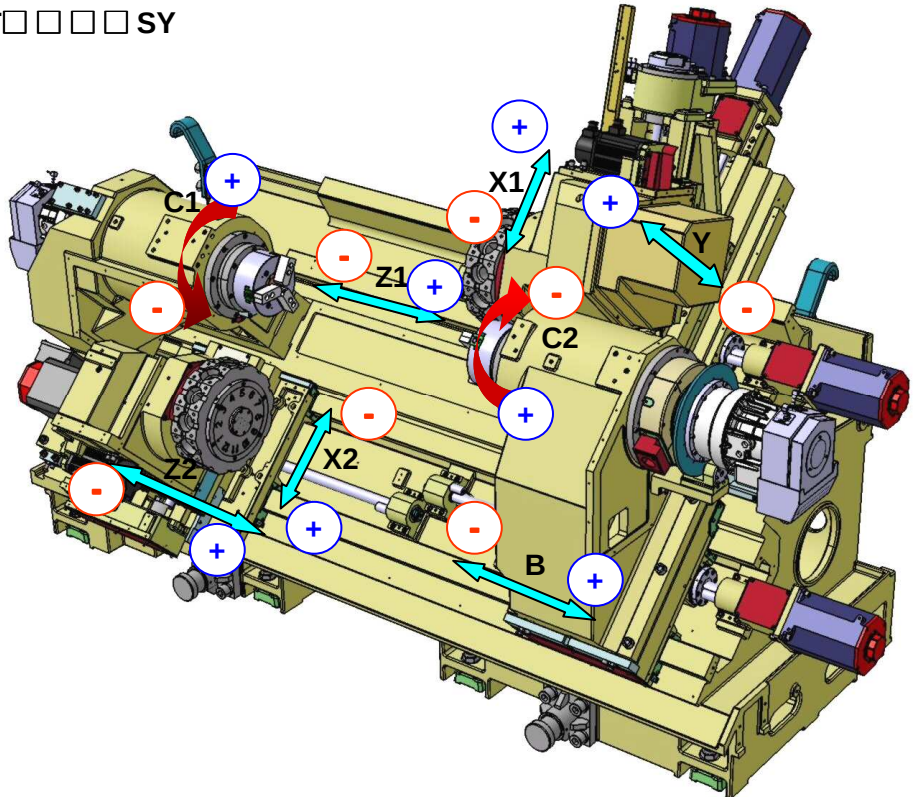
1.3.4 Axis direction

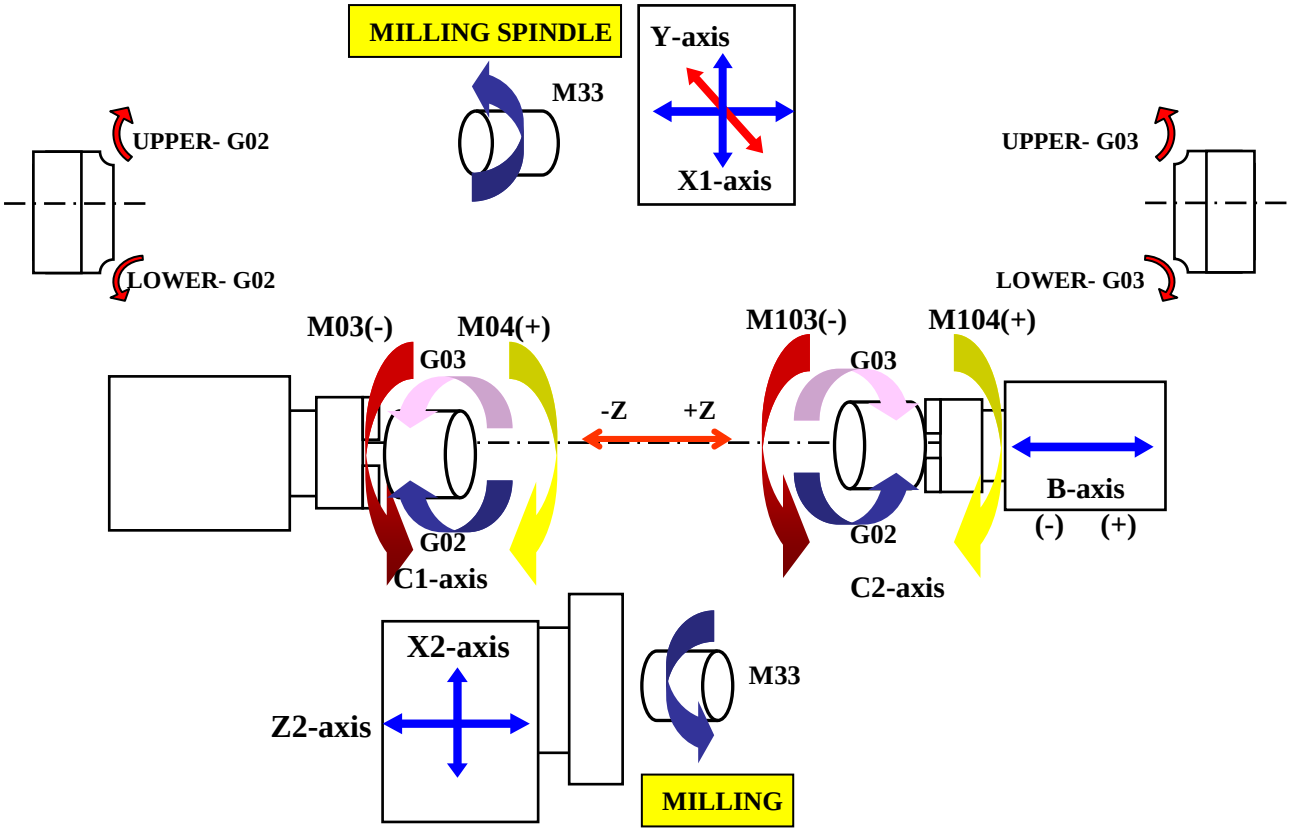
(1) MX☐☐☐☐ST



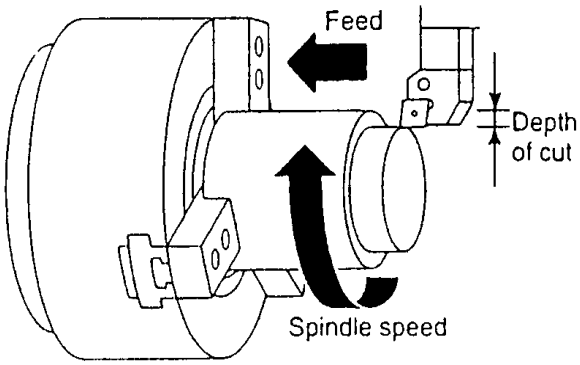


(2) TT□□□□SY


C1
X1
Y
Z1
C2
X2
Z2
B


MILLING SPINDLE
M33
Y-axis
X1-axis
UPPER- G02
LOWER- G02
UPPER- G03
LOWER- G03
M03(-)
M04(+)
G03
G02
C1-axis
-Z
+Z
M103(-)
M104(+)
G03
G02
C2-axis
B-axis
(-)
(+)
X2-axis
Z2-axis
M33
MILLING

1.4 Specifying the cutting conditions



The cutting conditions that are set when programming have a great influence on the machining efficiency and accuracy. These conditions must be checked carefully.

The following four cutting conditions are necessary for machining the workpiece.

1) Spindle speed (rpm), cutting speed (surface speed) (m/min)

The spindle speed or cutting speed is specified directly following address S (S function).

G97 S400 ····· Spindle speed 400 rpm

G96 S200 ····· Cutting speed 200 m/min

2) Cutting feedrate (mm/rev) (mm/min)

Feedrate is specified directly following address F (F function).

G99 F0.1 ····· Feedrate per spindle revolution, 0.1 mm/rev

G98 F100 ····· Feedrate per minute, 100 mm/min

3) Depth of cut

There is no special function used to specify the depth of cut.

Depth of cut is specified using tool movement along the X or Z-axis.

* For the following cycles, depth of cut may be specified using an address.

- Multiple repetitive cycles
- Hole machining canned cycles for the MC type and Y-axis specification machines.

For details of multiple repetitive cycles and hole machining canned cycles.

1.5 Functions (G,M,S,F,T)

A program is created using alphabets which show functions, and numerical values.
The G,M,S,F and T functions represent the main functions.

Details of each function are described in Next chapters.

The following table gives an overview of functions

Code	Functions
G code	Specifies the machining method in each block of a program or movement along an axis. Proceeding from this command, the NC prepares for movement in each block. For this reason, the G function is called a preparatory function. *Example : G00 ... ... Rapid traverse of axes
M code	Is called the miscellaneous function and works as the function to support the functions called by the G code. It specifies ON/OFF control of machine operations, including program stop, coolant discharge or stop, and spindle rotation or stop etc. *Example : M08 ..... Coolant discharge M09 ..... Coolant stop
S code	Specifies the spindle speed and the cutting speed.
F code	Specifies the feedrate of the tool.
T code	Specifies the tool number and the tool length offset number.

1.6 Basic pattern of program

When creating a part program for each tool (Turning cutting tool, Milling cutting tool etc.), the following basic patterns are used.

1.6.1 Turning Programming

O0001 (UPPER MILL)	O0001 (LOWER TURN)	Program No.
N1 (O.D TURN)	N1 (O.D TURN)	Sequence No.
T01000		Call T01- tool (only Upper)
G28 U0 V0 W0	G28 U0 W0	Return Ref. position
G99 G80 G40	G99 G80 G40	G99(mm/rev),G80/G40(cancel)
G54	G55	Work coordinate system
M6 T01000	T01001	Tool change & Tool offset
G28 U0 W0 V0		Return Ref. position after ATC
M5	M105	2-axis (Turn) mode select
M62	M162	Change High-Winding
T01001		Tool offset (only Upper)
(G400 B-45.0 J1. R2.)		G400 function (only Upper)
G50 S1500	G50 S1500	Setup Max. Spindle speed
G96 S250 M3	G96 S250 M103	CW Spindle rotation (CSS mode)
G0 X150.0 Z5.0 M8	G0 X150.0 Z5.0 M8	:
:	:	:
:	:	Machining program
:	:	:
G0 U2.0 W2.0 M9	G0 U2.0 W2.0 M9	:
G28 U0 V0 W0 M5	G28 U0 W0 M105	Ret. Ref. position & Stop Spindle
M01	M01	Optional stop
M30	M30	End of program

■ Spindle Selection M-code

* M05 : Stop L-spindle + Select L-spindle (2-axis mode)
* M105 : Stop R-spindle + Select R-spindle (2-axis mode)
* M35 : Stop Mill-spindle + Select C1-axis (3-axis mode)
* M135 : Stop Rev.spindle + Select C2-axis (3-axis mode)

1.6.2 Milling Programming

O0002 (UPPER MILL)	O0002 (LOWER MILL)	Program No.
N1 (D10 DRILLING)	N1 (D10 DRILLING)	Sequence No.
T02000		Call T01- tool (only Upper)
G28 U0 V0 W0	G28 U0 W0	Return Ref. position
G98 G80 G40	G99 G80 G40	G99(mm/rev),G80/G40(cancel)
G54	G55	Work coordinate system
M6 T02000	T01001	Tool change & Tool offset
G28 U0 W0 V0		Return Ref. position after ATC
M5	M105	2-axis (turning) mode select
M61	M161	Change Low-Winding
M35	M135	3-axis (Milling) mode select
T02002		Tool offset (only Upper)
(G400 B-90.0 J0.)		G400 function (only Upper)
G97 S1500 M33	G97 S1500 M33	CW Rev.spindle rotation
G0 X150.0 Z5.0 M8	G0 X150.0 Z5.0 M8	:
:	:	:
:	:	Machining program
:	:	:
G0 U2.0 W2.0 M9	G0 U2.0 W2.0 M9	:
G28 U0 V0 W0 M35	G28 U0 W0 M135	Ret. Ref. position & Stop Spindle
M01	M01	Optional stop
M30	M30	End of program

■ Spindle Winding Selection M-code (Must be command on 2-axis mode)

* M61 : Select Low-Winding for L-spindle
* M62 : Select High-Winding for L-spindle
* M161 : Select Low-Winding for R-spindle
* M162 : Select High-Winding for R-spindle

■ Spindle Selection & Rotation Command (Details of function are describe in Chapter 4)

(1) M35→ M33S_ → M35
(2) M05→ M03S_ →M05
(3) M135 → M211 → M33S_ →M135 → M212
(4) M105 →M103S_ →M105
(5) M135 →M33S_ →M135
(6) M105 → M103S_ →M105
(7) M35 →M211 →M33S_ →M35 →M212
(8) M05 →M03S_ →M05

1.7 Cautions for creating a program

1) Program number

This manual describes all program numbers in a four digit number. However, it is not necessary to write or enter a program number in a four digit number. A program number specified in less than four digit number is recognized and displayed in a four digit number after it is input to the NC. If “O1” is entered, for example, it is recognized and displayed as “O0001”.

2) Signs and Symbols

A program is expressed in a combination of alphabetic letters, positive/negative(+/-) signs, and numbers containing a decimal point. In addition to these, the end of block symbol “;” and the block skip symbol “/” are used.

3) Inputting a Decimal point

For an NC, it is possible to use a decimal point to enter numerical values. A decimal point can be used to express the numerical values that have the unit of “distance”, ”angle”, ”time”, or “speed”.

The addresses which allow the use of a decimal point are indicated below.

Distance or angle : X, Y, Z, C, U, V, W, H, I, J, K, R, B
Time : U, X, P
Feedrate : F

“mm” setting (specified by **G21**)
X1.0 ····· X1 mm
X1 ····· X0.001 mm or X0.0001mm
(if a decimal point is not entered, it is assumed that the value is specified in the until of lease input increment.)

“inch” setting (specified by **G20**)
X1.0 ····· X1 inch
X1 ····· X0.0001 inch or X0.00001inch
(if a decimal point is not entered, it is assumed that the value is specified in the until of lease input increment.)

(1) To call for dwell for 1second, specify as
G04 U1.0 (X1.0 or P1000)

(2) In program, or in a block, it is allowed to specify the commands with and without a decimal point.
X1000 Z23.7; or X10.0 Z22359;

(3) N3401#0(DPI) ; When a decimal point is omitted in an address that can include a decimal point.
0 : The least input increment is assumed.
1 : The unit of mm, inches, or second is assumed.

2. Preparatory Function (G function)

2. Preparatory Function (G function)

2.1 Preparatory Function (G function)

A number following address G determines the meaning of the command for the concerned block.

G codes are divided into the following two types.

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

For example, G00 and G01 are both modal codes, that is, they are G codes in the group other than group 00.

G01 X_ Z_ ;
X_ ;
Z_ ; ····· G01 is effective in this range.
G00 X_ Z_ ;

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

2.2 G function Explanations

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

2.3 G code list for T- series

TABLE (1/3)

G code			Group	Function
A	B	C		
G00	G00	G00	01	Positioning (Rapid traverse)
G01	G01	G01		Linear interpolation (Cutting feed)
G02	G02	G02		Circular interpolation CW or helical interpolation CW
G03	G03	G03		Circular interpolation CCW or helical interpolation CCW
G04	G04	G04	00	Dwell
G05	G05	G05		High-speed cycle cutting
G07	G07	G07		Hypothetical axis interpolation
G07.1 (G107)	G07.1 (G107)	G07.1 (G107)		Cylinderical interpolation
G08	G08	G08		Look-ahead control
G10	G10	G10		Programmable data input
G10.6	G10.6	G10.6		Tool retract and return
G11	G11	G11		Programmable data input mode cancel
G12.1 (G112)	G12.1 (G112)	G12.1 (G112)	21	Polar coordinate interpolation mode
G13.1 (G113)	G13.1 (G113)	G13.1 (G113)		Polar coordinate interpolation cancel mode
G17	G17	G17	16	XpYp plane selection
G18	G18	G18		ZpXp plane selection
G19	G19	G19		YpZp plane selection
G20	G20	G70	06	Input in inch
G21	G21	G71		Input in mm
G22	G22	G22	09	Stored stroke check function on
G23	G23	G23		Stored stroke check function off
G25	G25	G25	08	Spindle speed fluctuation detection off
G26	G26	G26		Spindle speed fluctuation detection on
G27	G27	G27	00	Reference to reference position
G28	G28	G28		Return to reference position
G30	G30	G30		2nd, 3rd and 4th reference position return
G30.1	G30.1	G30.1		Floating reference point return
G31	G31	G31		Skip function
G32	G33	G33	01	Thread cutting
G34	G34	G34		Variable-lead thread cutting
G35	G35	G35		Circular threading CW
G36	G36	G36		Circular threading CCW (When the bit 3(G36) of parameter No.3405 is set to 1)

2. Preparatory Function (G function)

TABLE (2/3)

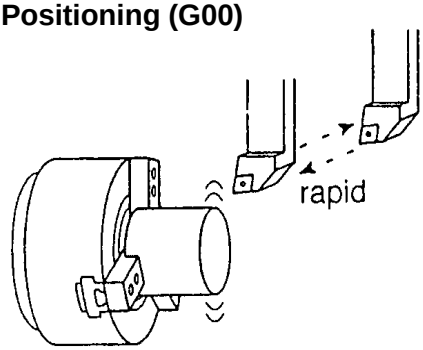
G code			Group	Function
A	B	C		
G36	G36	G36	00	Automatic tool compensation X (When the bit 3(G36) of parameter No.3405 is set to 0.
G37	G37	G37		Automatic tool compesation Z
G37.1	G37.1	G37.1		Automatic tool compesation X
G37.2	G37.2	G37.2		Automatic tool compesation Z
G39	G39	G39		Corner circular interpolation
G40	G40	G40	07	Too nose radius compensation cancel
G41	G41	G41		Too nose radius compensation left
G42	G42	G42		Too nose radius compensation right
G50	G92	G92	00	Coordinate system setting or max. spindle speed setting
G50.3	G92.1	G92.1		Workpiece coordiate system preset
G50.2 (G250	G50.2 (G250	G50.2 (G250	20	Polygonal turning cancel
G51.2 (G251)	G51.2 (G251)	G51.2 (G251)		Polygonal turning
G52	G52	G52	00	Local coordinate system setting
G53	G53	G53		Machine coordinate system setting
G54	G54	G54	14	Workpiece coordiate system 1 selection
G55	G55	G55		Workpiece coordiate system 2 selection
G56	G56	G56		Workpiece coordiate system 3 selection
G57	G57	G57		Workpiece coordiate system 4 selection
G58	G58	G58		Workpiece coordiate system 5 selection
G59	G59	G59		Workpiece coordiate system 6 selection
G60	G60	G60	00	Spindle direction positioning
G65	G65	G65		Macro calling
G66	G66	G66	12	Macro modal call
G67	G67	G67		Macro modal call cancel
G68	G68	G68	04	Mirror image for double turrets on or balance cut mode
G68.1	G68.1	G68.1	17	Coordinate system rotation start or three-dimensional coordinate system conversion mode on
G69	G69	G69	04	Mirror image for double turrets off or balance cut mode cancel
G69.1	G69.1	G69.1	17	Coordinate system rotation cancel or three-dimensional coordinate system conversion mode off

2. Preparatory Function (G function)

TABLE (3/3)

G code			Group	Function
A	B	C		
G70	G70	G72	00	Finishing cycle
G71	G71	G73		Stock removal in turning
G72	G72	G74		Stock removal in facing
G73	G73	G75		Pattern repeating
G74	G74	G76		End face peck drilling
G75	G75	G77		Outer diameter/internal diameter drilling
G76	G76	G78		Multiple threading cycle
G71	G71	G72	01	Traverse grinding cycle (for grinding machine)
G72	G72	G73		Traverse direct constant-dimension grinding cycle (for grinding machine)
G73	G73	G74		Oscilation grinding cycle (for grinding machine)
G74	G74	G75		Oscilation direct constant-dimension grinding cycle (for grinding machine)
◤G80	◤G80	◤G80	10	Canned cycle for drilling cancel
G83	G83	G83		Cycle for face drilling
G84	G84	G84		Cycle for face tapping
G85	G85	G85		Cycle for face boring
G87	G87	G87		Cycle for side drilling
G88	G88	G88		Cycle for side tapping
G89	G89	G89		Cycle for side boring
G90	G77	G20	01	Outer diameter/internal diameter cutting cycle
G92	G78	G21		Thread cutting cycle
G94	G79	G24		Endface turning cycle
G96	G96	G96	02	Constant surface speed control
◤G97	◤G97	◤G97		Constant surface speed control cancel
G98	G94	G94	05	Per minute feed
◤G99	◤G95	◤G95		Per rotation feed
-	◤G90	◤G90	03	Absolute programming
-	G91	G91		Incremetal programming
-	G98	G98	11	Return to initial level
-	G99	G99		Return to R point level
G100	G100	G100	00	B axis control-Program registration completion
G101	G101	G101		B axis control-First program registration start
G102	G102	G102		B axis control-Second program registration start
G103	G103	G103		B axis control-Third program registration start
G110	G110	G110		B axis control-One motion operation programming

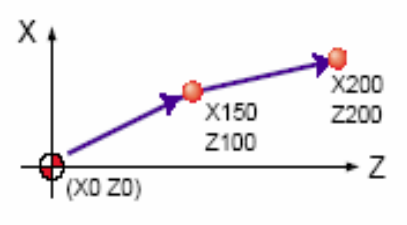
2.4 Positioning (G00)



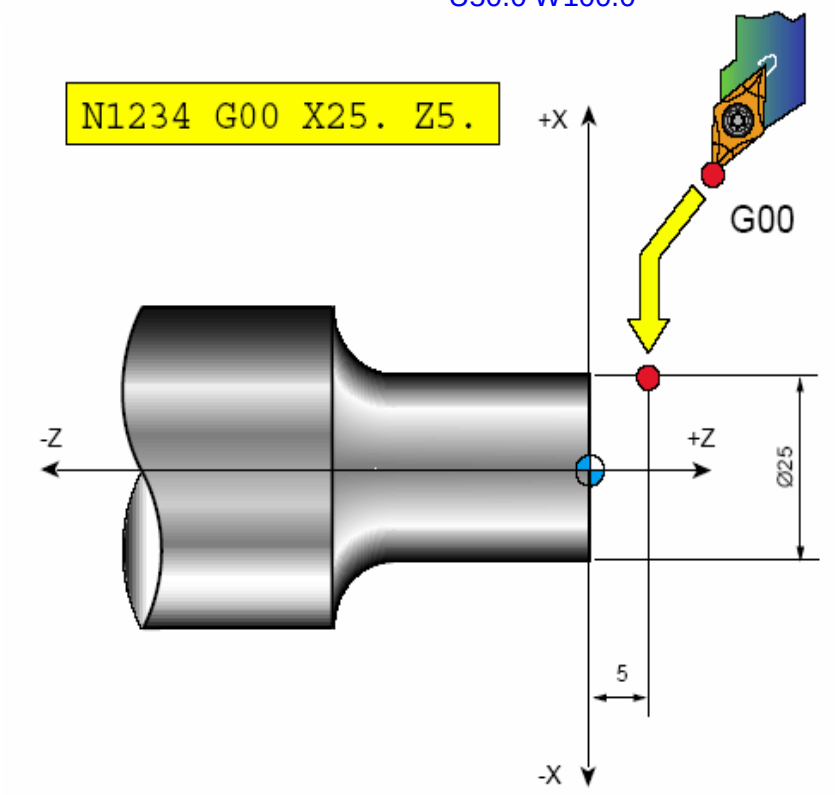
By specifying the G00 command, all axis movement commands are executed at the rapid traverse rate. In other words, the cutting tool is positioned at the programmed target point at a rapid traverse rate.

G00 X(U)_ Z(W)_ ;

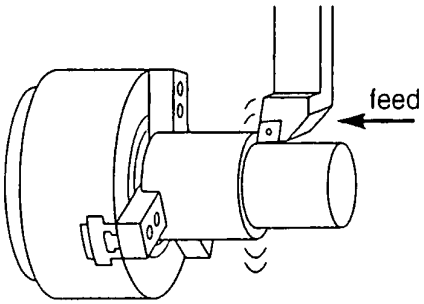
* G00 ····· Calls positioning at a rapid traverse rate.
* X, Z ····· Specifies the positioning target point at a rapid traverse rate.
* U, W ····· Specifies the positioning target point.
  The coordinates are specified in incremental values in reference to the present position.



G00 X150.0 Z100.0
X200.0 Z200.0
or
G00 U150.0 W100.0
U50.0 W100.0



2.5 Linear interpolation (G01)



By specifying the G01 command, the cutting tool is moved along a straight line to cut a workpiece.

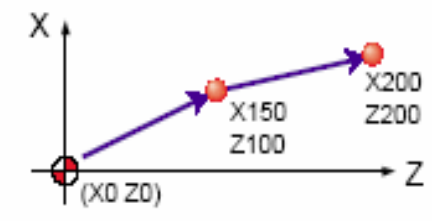
The feedrate is specified with an F code by the distance the cutting tool should be moved while the spindle rotates one turn, or the distance to be moved in a minute.

G01 X(U)_ Z(W)_ F_ ;

* G01 ····· Calls the linear interpolation mode.
* X, Z ····· Specifies the cutting target point. (absolute values)
* U, W ····· Specifies the cutting target point. (incremental value)
* F ····· Specifies the feedrate in ordinary control.

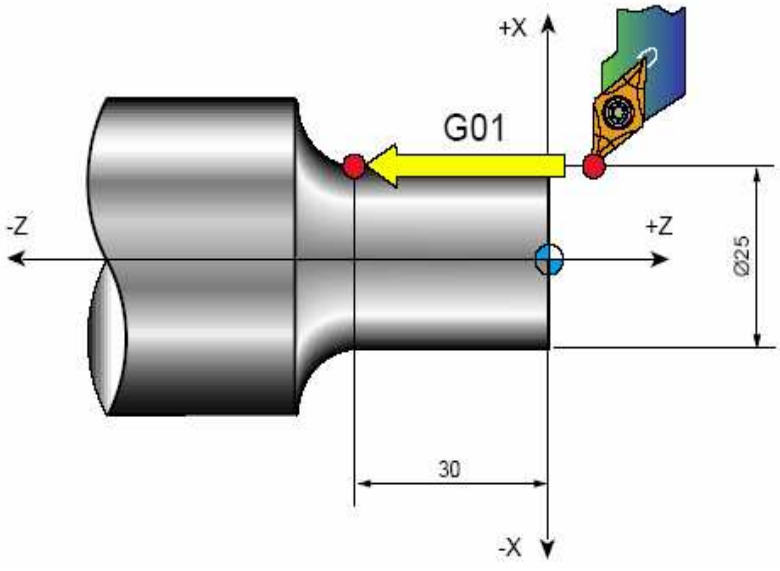
in the G99 mode, the feedrate is specified in “mm/rev”. (F0.2 = 0.2mm/rev)

in the G98 mode, the feedrate is specified in “mm/min”. (F200 = 200mm/min)

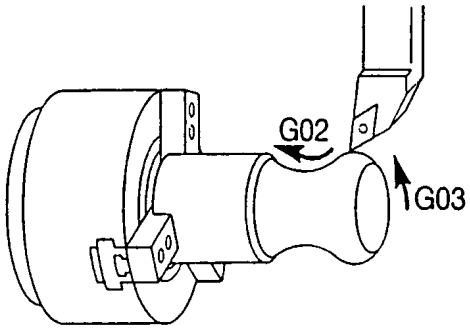


G01 X150.0 Z100.0 F0.2
Z200.0 Z200.0
or
G01 U150.0 W100.0 F0.2
U50.0 W100.0

```
N1234 G01 X25. Z-30. F0.2
```



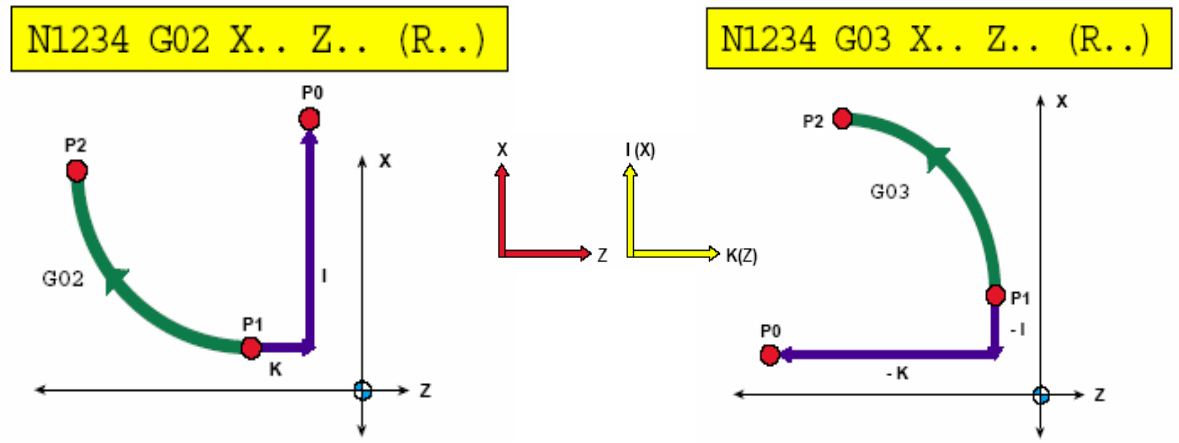
2.6 Circular interpolation (G02/G03)



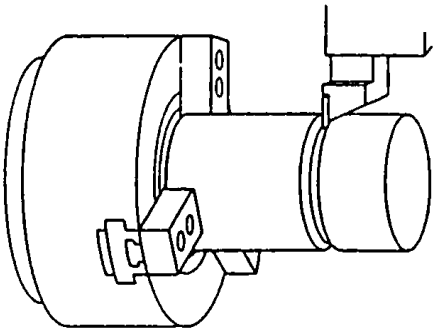
By specifying the G02,G03 command, the cutting tool is moved along an arc to cut a workpiece.

G02(G03) X(U)_ Z(W)_ R_ F_ ;
G02(G03) X(U)_ Z(W)_ I_ K_ F_ ;

Conditions		Instruction	Meaning	
			Right hand coordinate	Left hand coordinate
1	Rotation direction	G02	CW	CCW
		G03	CCW	CW
2	Location of end point	X,Z	Location X,Z of commanded point from coordinate	
	Distance to the end point	U,W	Distance from start point to commanded point	
3	Distance between start point and the ceter point	I,K	Distance from start point to the center of and arc with sign, radius value (I always designates the raduis)	
4	Arc radius with no sign radius of circumference	R	Radius of circumference	



2.7 Dwell (G04)



The G04 command is used to suspend program execution during automatic operation for the period specified in the program.

This function is called the dwell function, and is used in operation such as the grooving operation.

If dwell is specified at the bottom of the groove, the tool stops moving. The spindle keeps rotating while the tool stays at the bottom of the groove.

By rotating the spindle one turn while locating the tool at the bottom of the groove, the groove profile accuracy is improved and uncut portion is eliminated. The dwell function is also used for opening and closing the chuck when the machine is equipped with the air blow, the bar feeder or the loader.

G04 X_ ;

G04 U_ ;

G04 P_ ;

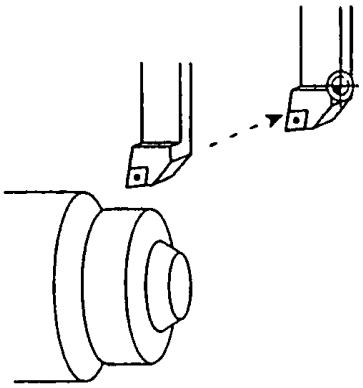
* G04 ····· Calls the dwell function
* X, U ····· Specifies the period in which the program execution is suspended. The dwell period should be specified in units of seconds with a decimal point.

  X1.0(U1.0) = 1sec

  X1 (U1) = 0.001sec
* P ····· Specifies the period in which the program execution is suspended. The dwell period should be specified in units of 0.001 second without a decimal point.

  P1000 = 1sec

2.8 Reference point return (G28/G30)



The G04 command is used to suspend program execution during automatic operation for the period specified in the program.

This function is called the dwell function, and is used in operation such as the grooving operation.

If dwell is specified at the bottom of the groove, the tool stops moving. The spindle keeps rotating while the tool stays at the bottom of the groove.

By rotating the spindle one turn while locating the tool at the bottom of the groove, the groove profile accuracy is improved and uncut portion is eliminated. The dwell function is also used for opening and closing the chuck when the machine is equipped with the air blow, the bar feeder or the loader.

G27 IP_ ; ····· Reference return check

G28 IP_ ; ····· Reference position return

G30 P2 IP_ ; ····· 2ND reference position return (P2 can be omitted.)

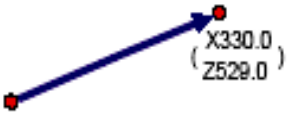
G30 P3 IP_ ; ····· 3rd reference position return

G30 P4 IP_ ; ····· 4th reference position return

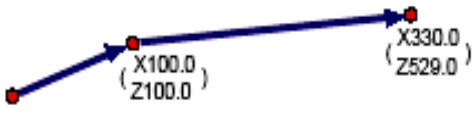
* IP : Command specifying the intermediate position
(Absolute/Incremental command)

[Example] When PARAMETER N1240(X) is 330000
N1240(Z) is 529000

G28 U0 W0



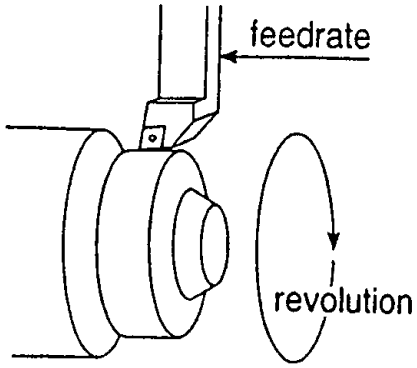
G27 X100.0 Z100.0



(1) When the G28 command is specified, the tool offset and the automatic tool nose R offset functions are canceled.

(2) PARAMETER N1240 (1ST Reference point)
PARAMETER N1241 (2ND Reference point)

2.9 Setting feedrate units (G98/G99)



Axis feedrate units are determined by specifying the following two G codes:

G98 ; ····· Specifies the feedrate per minute (mm/min).

The G98 command calls the mode in which axis feedrates specified by F codes are interpreted in units of mm per minute. This mode is used when the bar feeder, the pull-out finger, or the rotary tool is used.

G99 ; ····· Specifies the feedrate per revolution (mm/rev)

The G99 command calls the mode in which the axis feedrates specified by F codes are interpreted in units of mm per revolution. This mode is used for general turning operations such as O.D cutting, I.D cutting and thread cutting.

(1) The G98 and G99 commands are modal.
Therefore, once the G98 command is specified, it remains valid until the G99 command is specified.

(2) When the power is turned on, the G99 mode (feedrate per revolution) is set.

(3) In the G98 mode, the turret moves at the feedrate specified by the F code even when the spindle is not rotating. Make sure that the cutting tool will not strike the workpiece, etc., since this could damage the machine.

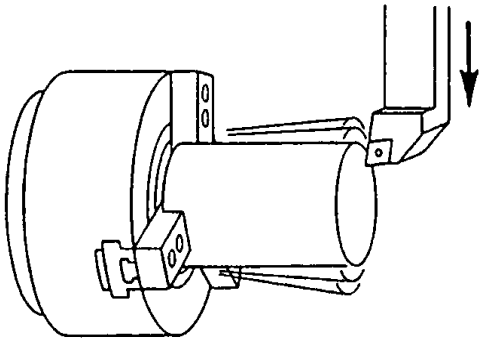
G98;
G01 Z_ F100.0; ····· The cutting tool moves at a rate of 100mm/min even when the spindle is not rotating.

3. Spindle Rotation command (S function)

3. Spindle Rotation Command

3.1 Max Speed Limits setting and Coordinate system setting (G50)

3.1.1 Setting the Spindle Speed Limit



The spindle speed limit for an automatic operation is set the G50 command.

If the programmed spindle speed is faster than the limit value set in the G50 block, actual spindle speed is clamped at the set limit speed.

G50 S_ ;

* G50 ····· Calls the mode to specify the spindle speed limit for automatic operation.

* S ····· Specifies spindle speed limit (rpm)

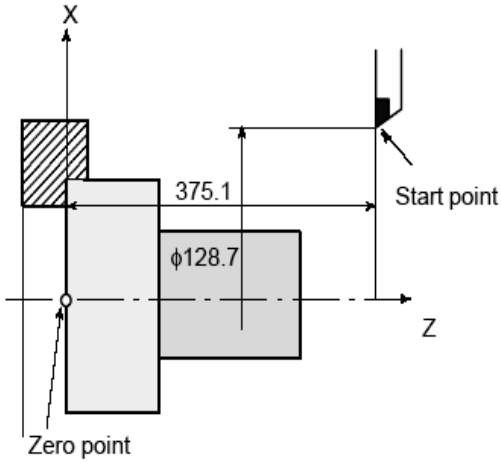
3.1.2 Setting the Coordinate system

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

Example 1

Setting the coordinate system by the

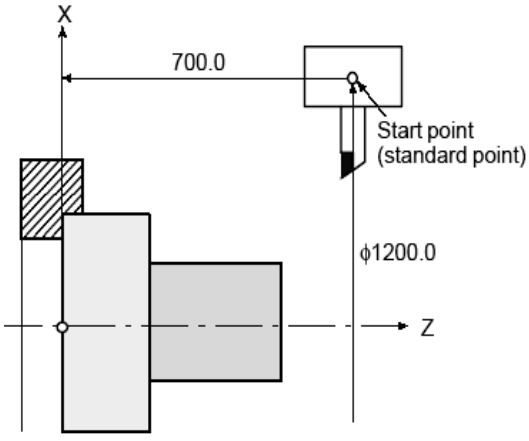
G50 X128.7 Z375.1



Example 2

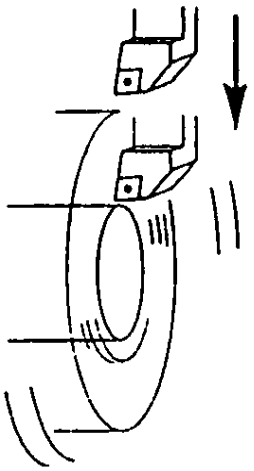
Setting the coordinate system by the

G50 X1200.0 Z700.0



3.2 Spindle speed control (G96/G97)

3.2.1 Constant Surface Speed Control “CSS” (G96)



The G96 command is used to maintain surface speed constant at the specified value.

The surface speed is also called the cutting speed.

It indicates the distance the cutting tool moves along the workpiece surface per minute.

When the surface speed is specified with the G96 command, the spindle speed is automatically controlled to maintain the surface speed constant as the cutting diameter varies. This mode is used to maintain the cutting conditions constant.

For example, if the cutting speed (V) is specified at 100m/min to cut a 30mm diameter (D) workpiece, the spindle speed (N) is calculated as indicated below.

$$N = \frac{1000V}{\pi \cdot D}$$

$$N = \frac{1000 \times 100}{3.14 \times 30}$$

$$\fallingdotseq 1061 \text{ rpm}$$

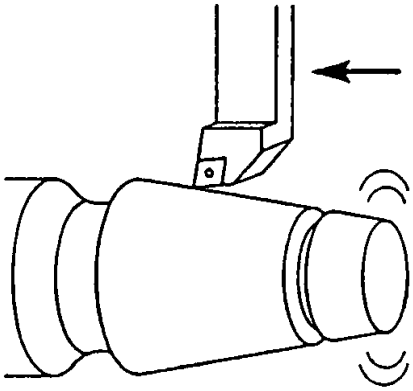
Therefore, the spindle rotates at 1061 rpm.

Generally, the standard cutting speed is determined according to the material of the workpiece and the cutting tool, the workpiece shape, and the chucking method.

G96 S_ M03(M04);

* G96 ····· Calls the constant surface speed control mode.
* S ····· Specifies the cutting speed (m/min)
* M03(M04) ····· Specifies spindle rotation in the normal (reverse) direction

3.2.2 Constant spindle speed Control (G97)



The G97 command is used to call the mode in which a constant spindle speed is maintained.

During automatic operation, the spindle rotates at the programmed speed.

$$N = \frac{1000V}{\pi \cdot D}$$

N: Spindle speed (rpm)

V: Cutting speed (m/min)

D: Cutting diameter (mm)

π: Circumference constant

The G97 command must be specified for thread cutting operation and turning drilling operation at the center of workpiece end face.

G97 S_ M03(M04);

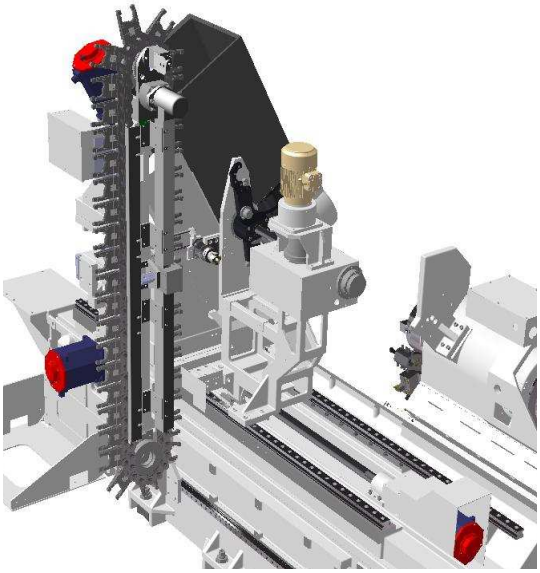
* G97 ····· Calls the constant spindle speed control mode.
* S ····· Specifies the spindle speed (rpm)
* M03(M04) ····· Specifies spindle rotation in the normal (reverse) direction

4. Tool changing and Tool offset

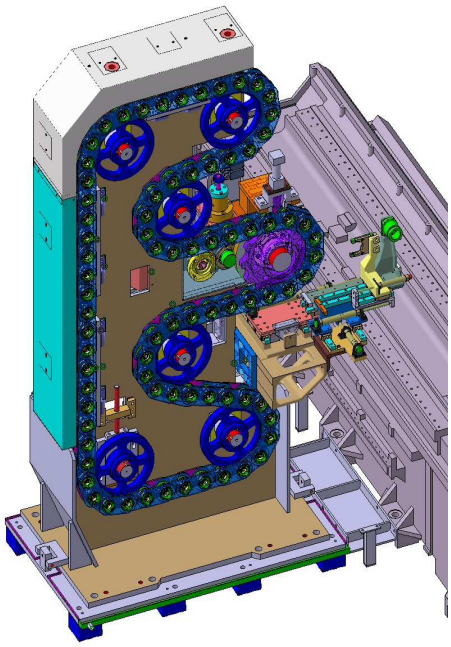
4. Tool changing and Tool offset

4.1 Tool change method

4.1.1 Automatic tool changing in MX upper unit



[24,40 tools]



[80 tools]

```
T01000 ····· (Tool call)
M06 T01000 ····· (Tool change command)
T02000 ····· (Next tool call)
T01001 ····· (Tool offset)
G28 U0 V0 (W0)
…
cutting
  …
N2
T02000
M06 T02000
```

4.1.2 Turret type (Lower of MX / Upper and Lower of TT) → 12 tools

1) **T0101** ····· (Tool index & Tool offset)

TT series : Standard in Upper and Lower unit

MX series : User option in Lower unit

→ N5040#1bit = 1 and N3032=4 (4digits) T□□△△

2) **T01001** ····· (Tool index & Tool offset)

MX series : Standard in Lower unit

→ N5040#1bit = 0 and N3032=5 (5digits) T□□△△△

4.1.3 Manual tool changing in MX Upper unit

[Tool change in front]

Select Upper unit

(Main OP)

Select MDI mode

(Main OP)

Execute Program

(G490 S0)

Select JOG mode

(Main OP)

Open the Front door

Tool Unclamp/Clamp

(Over 2sec.)

[Tool change in rear]

Select JOG mode

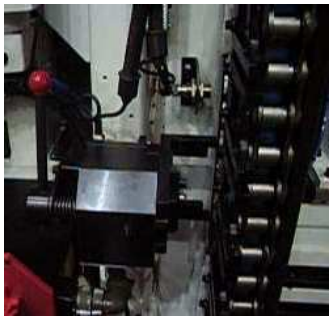
(Main OP)

Select Manual mode

(ATC OP)

Check Waiting pot. (initial condition)

Execute Magazine CW/CCW (ATC OP)

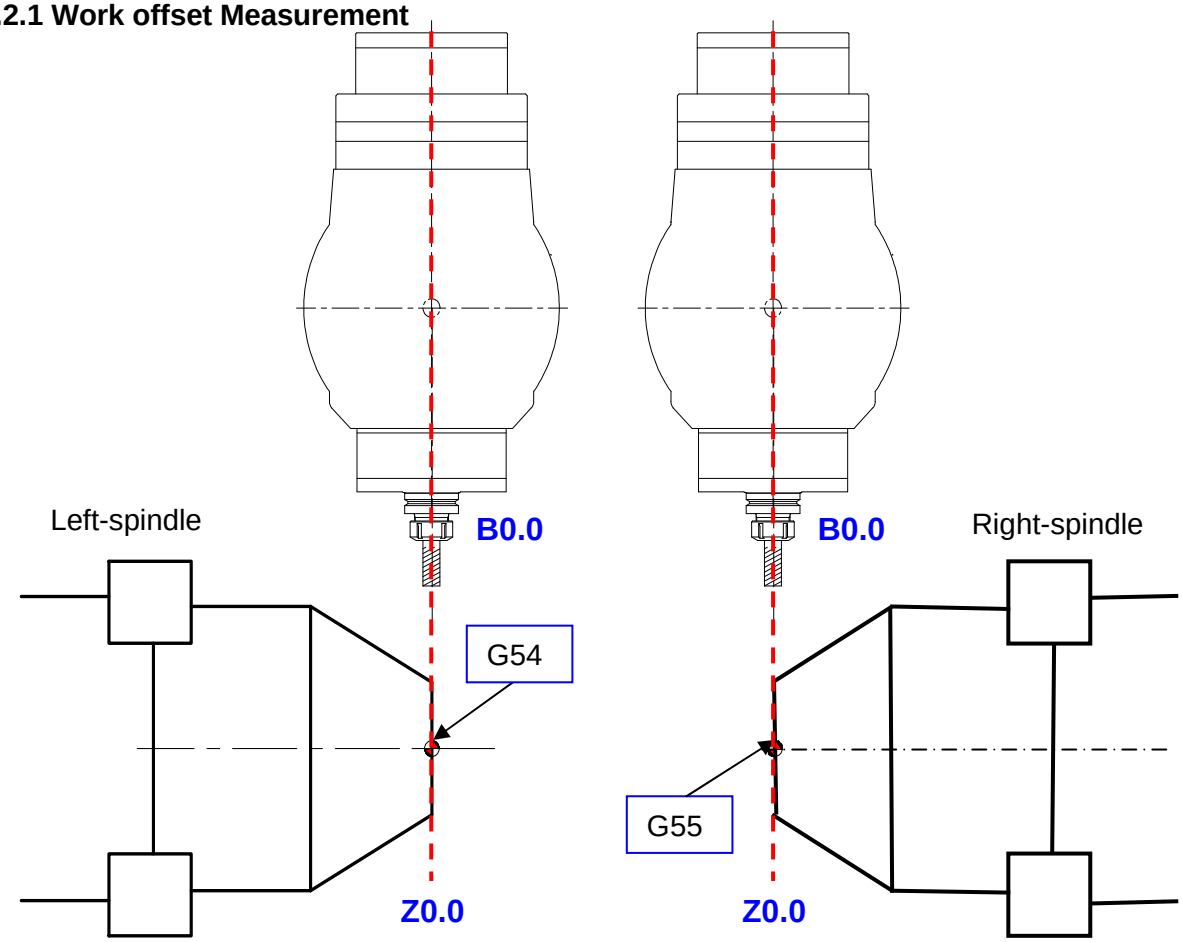


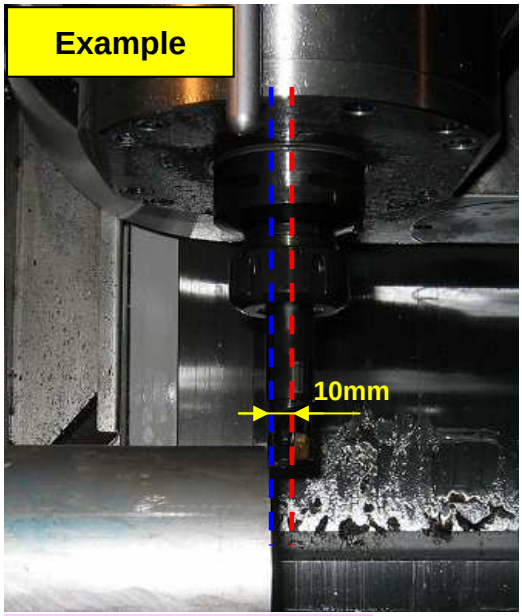
Tool loading device

(Check key hole)

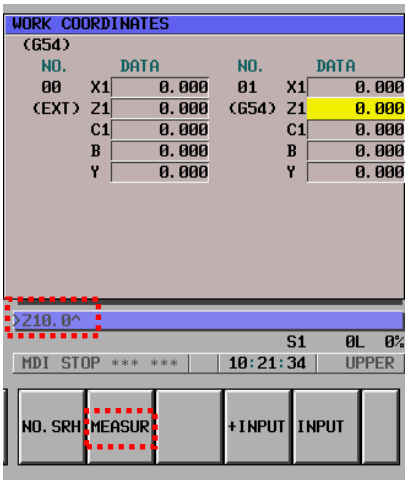
4.2 Tool offset

4.2.1 Work offset Measurement





Tool : Ø20 mm (End mill)



1) MDI mode ; execute G54
2) Touch side face of Milling tool
3) Input “Z10.0” → MEASUR

4.2.2 Tool offset Measurement

1) The method of tool offset measurement with Q-setter

[TYPE 1 : Turning tool]

Measuring X/Z offset value
for Constant B-axis angle (**S0**)

[TYPE 2 : Turning tool]

Measuring X offset value
for G400 function (**B0, S0**)

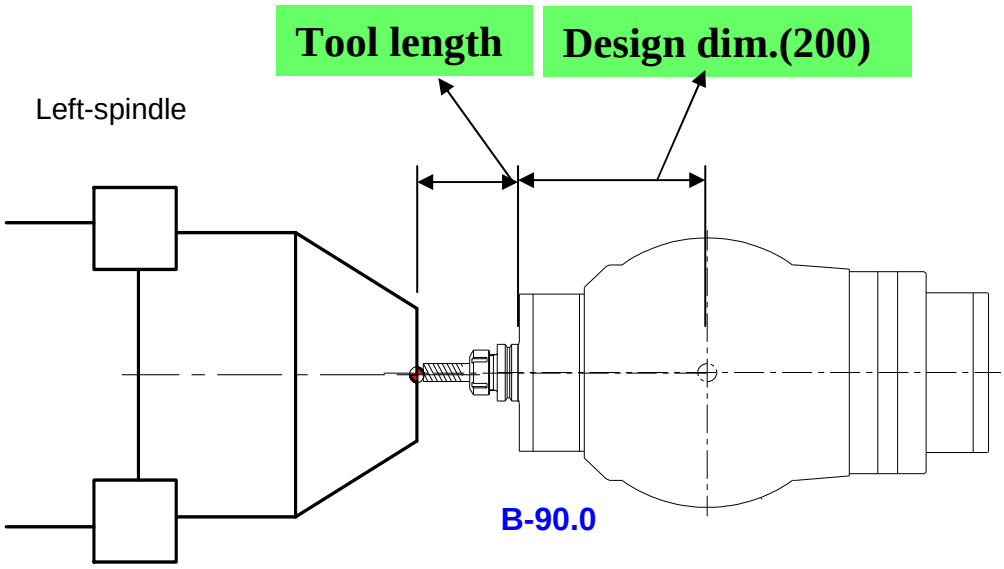
[TYPE 3 : Turning tool]

Measuring X/Z offset value
for G400 function (**B0, S0**)

[TYPE 4 : Milling tool]

Measuring X offset value
for G400 function (**B0**)
After M101 (spindle unclamp)

2) Comprehension tool offset at using TCP function in 16iT



Tool offset(Z) = Tool length + Design Dim.(200)
⇒ Tool offset(Z) is Plus value (+)

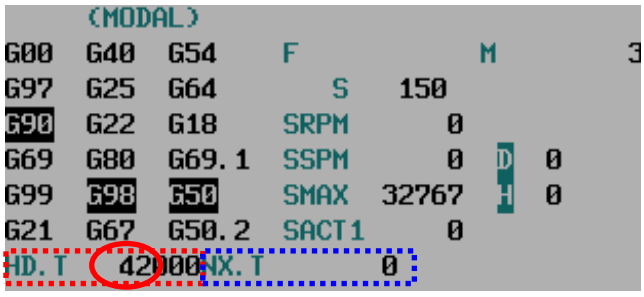
Using TCP function in 16 iT
→ *Par.19666 ≒ 200 (Design dim.)*
DNC run → *Par.138#7 = 1*

4.3 ATC recovery in MX series

[ATC OP PANEL]

1) Select [JOG] mode (Main OP)
2) Select [MANUAL] mode (ATC OP)
3) Select [BRAKE RELEASE] mode
4) Press [START] button.
5) Select [ATC STEP REV.] mode
6) Changer arm is returned step by step, when [START] button is pressed.
7) Return changer arm to home position
8) Check whether changer arm home position sensor is on
9) Check whether milling spindle tool is clamped
10) Select [ATC 1-CYCLE] mode.
11) Press [START] button
12) ATC changing is executed and completed.
13) Check D-data.
    D448 = Next tool No.
    D450 = Spindle Head tool No.
    Check whether D450 value is equal to Head tool value.
14) If D450 value is not equal to head tool value,
    command " M45T△△000 ;"
    △△ is real spindle tool No.

Note) K52.7=1 (ATC is used)

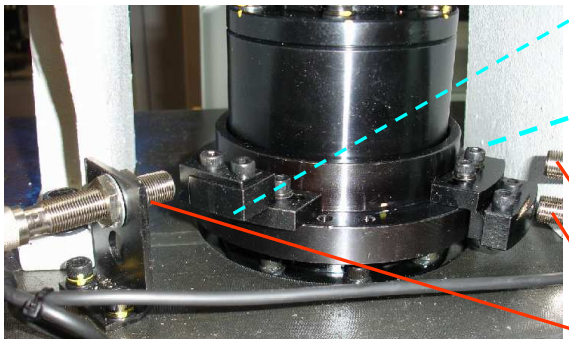


[TOOL DATA]

NO.	ADDRESS	DATA
0015	D0430	3413
0016	D0432	546
0017	D0434	0
0018	D0436	3276
0019	D0438	0
0020	D0440	137
0021	D0442	0
0022	D0444	34
0023	D0446	0
0024	D0448	0
0025	D0450	42
0026	D0452	41
0027	D0454	27256
0028	D0456	2389
0029	D0458	0

4. Tool changing and Tool offset

1) 24T,40T magazine



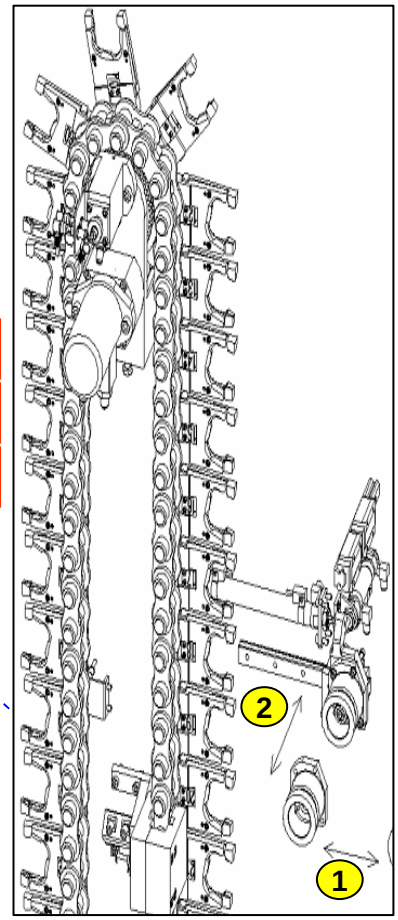
CLAMP
UNCLAMP
DOG

HOME DOG

CLAMP S/W

HOME S/W

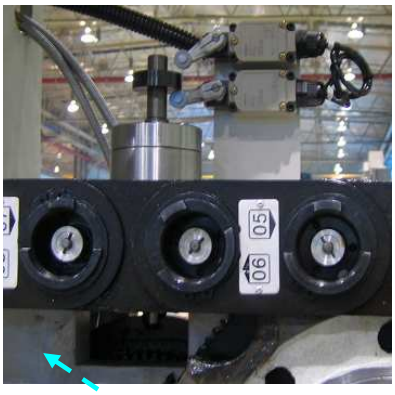
UNCLAMP S/W



Magazine Home

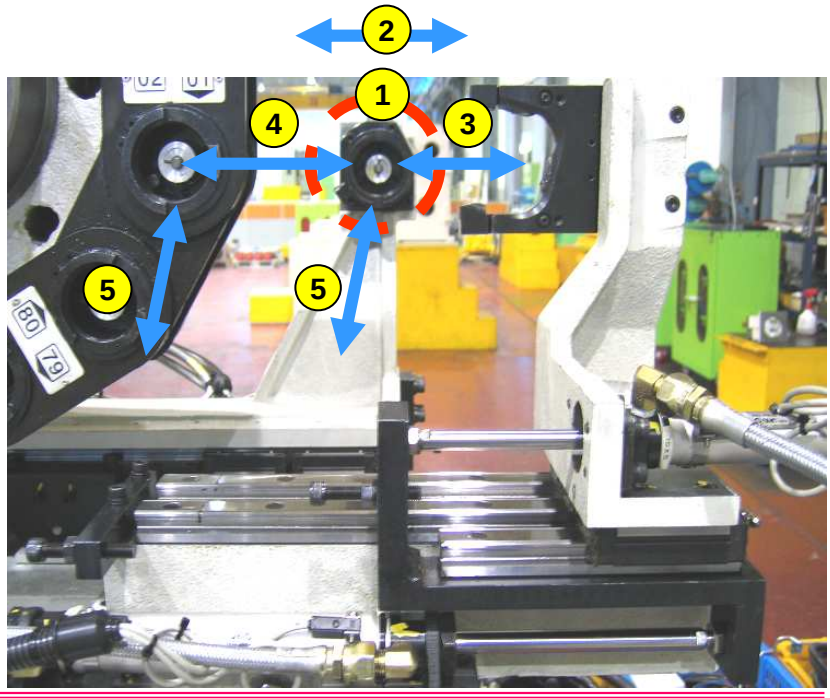
1. Shifter w-pos. / m-pos.
2. Shifter retract/advance

2) 80T magazine



ATC HOME DOG

1. Shifter retract/advance
2. Shifter w-pos./ m-pos.
3. Gripper w-pos./ h-pos.
4. Gripper w-pos./ m-pos.
5. Gripper In/ Out



5. Miscellaneous functions

5. Miscellaneous functions

5.1 Miscellaneous functions of MX series

5.1.1 M-code list (1/6) of MX series

☆ : option

No.	UPPER UNIT	LOWER UNIT	option
M00	PROGRAM STOP	PROGRAM STOP	
M01	OPTIONAL PROGRAM STOP	OPTIONAL PROGRAM STOP	
M02	PROGRAM END	PROGRAM END	
M03	LEFT SPINDLE FORWARD ROTATION	LEFT SPINDLE FORWARD ROTATION	
M04	LEFT SPINDLE REVERSE ROTATION	LEFT SPINDLE REVERSE ROTATION	
M05	LEFT SPINDLE STOP & SELECTION	LEFT SPINDLE STOP & SELECTION	
M06	ATC TOOL CHANGE COMMAND		
M07	BED SHOER COOLANT ON	BED SHOER COOLANT ON	
M08	COOLANT ON (UPPER SIDE)	COOLANT ON (UPPER SIDE)	
M09	COOLANT OFF (UPPER SIDE)	COOLANT OFF (UPPER SIDE)	
M10	PARTS UNLOADER DOWN TO CHUCK	PARTS UNLOADER DOWN TO CHUCK	☆
M11	PARTS UNLOADER RETURN TO HOME	PARTS UNLOADER RETURN TO HOME	☆
M12	COOLJECT ON (UPPER SIDE)	COOLJECT ON (UPPER SIDE)	☆
M13	LEFT SPINDLE COOLANT BLOWER OFF	LEFT SPINDLE COOLANT BLOWER OFF	☆
M14	LEFT SPINDLE AIR BLOW ON	LEFT SPINDLE AIR BLOW ON	☆
M15	LEFT SPINDLE AIR BLOW OFF	LEFT SPINDLE AIR BLOW OFF	☆
M16	BED SHOER COOLANT OFF	BED SHOER COOLANT OFF	☆
M17	MACHINE LOCK ACT	MACHINE LOCK ACT	
M18	MACHINE LOCK CANCEL	MACHINE LOCK CANCEL	
M19	LEFT SPINDLE ORIENTATION	LEFT SPINDLE ORIENTATION	
M20	GANTRY LOADER CALL	GANTRY LOADER CALL	☆
M21	OPTIONAL BLOCK SKIP ACT		
M22	OPTIONAL BLOCK SKIP CANCEL		
M23	PROGRAM START CHECK	PROGRAM START CHECK	
M24	CHIP CONVEYOR RUN	CHIP CONVEYOR RUN	
M25	CHIP CONVEYOR STOP	CHIP CONVEYOR STOP	
M26	T.S.C COOLANT ON (MILLING SPINDLE)		
M27	T.S.C COOLANT OFF (MILLING SPINDLE)		
M28	POLYGON MODE SELECT	POLYGON MODE SELECT	
M29	RIGID TAPPING MODE SELECT	RIGID TAPPING MODE SELECT	
M30	PROGRAM END & REWIND	PROGRAM END & REWIND	
M31	LEFT CHUCKING INTERLOCK BY-PASS	LEFT CHUCKING INTERLOCK BY-PASS	
M33	MILLING SPINDLE FORWARD ROTATION	REVOLVING SPINDLE FORWARD ROTATION	
M34	MILLING SPINDLE REVERSE ROTATION	REVOLVING SPINDLE REVERSE ROTATION	
M35	MILLING SPINDLE STOP & C1-AXIS SELECT	REVOLVING SPINDLE STOP & C1-AXIS SELECT	
M45	MILLING SPINDLE HEAD TOOL NO, EXTERNAL SET		
M46	P.T.S.BODY CLAMP & BAR ADVANCE		☆

5.1.1 M-code list (2/6) of MX series

☆ : option

No.	UPPER UNIT	LOWER UNIT	option
M47	P.T.S.BODY UNCLAMP & BAR RETRACT		☆
M48			
M49	MILLING SPINDLE ORIENTATION		
M50	BAR-FEEDER COMMAND 1	BAR-FEEDER COMMAND 1	☆
M51	BAR-FEEDER COMMAND 2	BAR-FEEDER COMMAND 2	☆
M52	FRONT SPLASH GUARD DOOR OPEN	FRONT SPLASH GUARD DOOR OPEN	☆
M53	FRONT SPLASH GUARD DOOR CLOSE	FRONT SPLASH GUARD DOOR CLOSE	☆
M54	PARTS COUNT	PARTS COUNT	
M55	REPEAT PROGRAM AFTER M02 / M30	REPEAT PROGRAM AFTER M02 / M30	
M56	TAPPING MODE SELECT	TAPPING MODE SELECT	
M57	TAPPING MODE CANCEL	TAPPING MODE CANCEL	
M58		STEADY REST CLAMP	
M59		STEADY REST UNCLAMP	
M60		TOOL CHANGE WITH MOVING AXES	
M61	LEFT SPINDLE WINDING CHANGE LOW SPEED	LEFT SPINDLE WINDING CHANGE LOW SPEED	
M62	LEFT SPINDLE WINDING CHANGE HIGH SPEED	LEFT SPINDLE WINDING CHANGE HIGH SPEED	
M63	LEFT SPINDLE FORWARD & UPPER COOLANT ON	LEFT SPINDLE FORWARD & UPPER COOLANT ON	
M64	LEFT SPINDLE REVERSE & UPPER COOLANT ON	LEFT SPINDLE REVERSE & UPPER COOLANT ON	
M65	LEFT SPINDLE STOP & UPPER COOLANT OFF	LEFT SPINDLE STOP & UPPER COOLANT OFF	
M66	LEFT CHUCKING LOW PRESSURE	RIGHT CHUCKING LOW PRESSURE	☆
M67	LEFT CHUCKING HIGH PRESSURE	RIGHT CHUCKING HIGH PRESSURE	☆
M68	LEFT CHUCK CLAMP	LEFT CHUCK CLAMP	
M69	LEFT CHUCK UNCLAMP	LEFT CHUCK UNCLAMP	
M70	TAIL STOCK QUILL LOW PRESSURE ADVANCE	TAIL STOCK QUILL LOW PRESSURE ADVANCE	
M72	SPINDLE CONVERT ROTATION DIRECTION	SPINDLE CONVERT ROTATION DIRECTION	
M73	SPINDLE NORMAL ROTATION DIRECTION	SPINDLE NORMAL ROTATION DIRECTION	
M74	ERROR DETECT ON	ERROR DETECT ON	
M75	ERROR DETECT OFF	ERROR DETECT OFF	
M76	CHAMFERING ON	CHAMFERING ON	
M77	CHAMFERING OFF	CHAMFERING OFF	
M78	TAIL STOCK QUILL ADVANCE	TAIL STOCK QUILL ADVANCE	
M79	TAIL STOCK QUILL RETRACT	TAIL STOCK QUILL RETRACT	
M80	Q-SETTER SWING ARM DOWN	Q-SETTER SWING ARM DOWN	☆
M81	Q-SETTER SWING ARM UP	Q-SETTER SWING ARM UP	☆
M82	TOUCH PROBE ON	TOUCH PROBE ON	☆
M83	TOUCH PROBE OFF	TOUCH PROBE OFF	☆
M84		TURRET C.W. ROTATION	
M85		TURRET C.C.W. ROTATION	

5.1.1 M-code list (3/6) of MX series

☆ : option

No.	UPPER UNIT	LOWER UNIT	option
M86		LOWER A-AXIS TORQUE SKIP	
M87		LOWER A-AXIS TORQUE SKIP CANCEL	
M88	C1-AXIS LOW PRESSURE CLAMP	C1-AXIS LOW PRESSURE CLAMP	
M89	C1-AXIS HIGH PRESSURE CLAMP	C1-AXIS HIGH PRESSURE CLAMP	
M90	C1-AXIS UNCLAMP	C1-AXIS UNCLAMP	
M91	EXTERNAL M-CODE M91	EXTERNAL M-CODE M91	☆
M92	EXTERNAL M-CODE M92	EXTERNAL M-CODE M92	☆
M93	EXTERNAL M-CODE M93	EXTERNAL M-CODE M93	☆
M94	EXTERNAL M-CODE M94	EXTERNAL M-CODE M94	☆
M98	SUB PROGRAM CALL	SUB PROGRAM CALL	
M99	END OF SUB PROGRAM	END OF SUB PROGRAM	
M100	MILLING SPINDLE CLAMP		
M101	MILLING SPINDLE UNCLAMP		
M103	RIGHT SPINDLE FORWARD ROTATION	RIGHT SPINDLE FORWARD ROTATION	
M104	RIGHT SPINDLE REVERSE ROTATION	RIGHT SPINDLE REVERSE ROTATION	
M105	RIGHT SPINDLE STOP & SELECTION	RIGHT SPINDLE STOP & SELECTION	
M106	B-AXIS CURVIC COUPLING CLAMP		
M107	B-AXIS CURVIC COUPLING UNCLAMP		
M108	T.S.C COOLANT ON (LEFT SPINDLE)	T.S.C COOLANT ON (LEFT SPINDLE)	☆
M109	T.S.C COOLANT OFF(LEFT SPINDLE)	T.S.C COOLANT OFF(LEFT SPINDLE)	☆
M110	B-AXIS DISC CLAMP		
M111	B-AXIS DISC UNCLAMP		
M112	B-AXIS CURVIC & DISC CLAMP		
M113	B-AXIS CURVIC & DISC UNCLAMP		
M114	RIGHT SPINDLE AIR BLOW ON	RIGHT SPINDLE AIR BLOW ON	☆
M115	RIGHT SPINDLE AIR BLOW OFF	RIGHT SPINDLE AIR BLOW OFF	☆
M116	WORK EJECTOR OPERATION COMMAND	WORK EJECTOR OPERATION COMMAND	☆
M117		LOWER A-AXIS TORQUE CONTROL ON	
M118		LOWER A-AXIS TORQUE CONTROL RESET	
M119	RIGHT SPINDLE ORIENTATION	RIGHT SPINDLE ORIENTATION	
M121		LOWER Z2-AXIS TORQUE CONTROL ON	
M124		LOWER Z2-AXIS TORQUE CONTROL RESET	
M125	SYNCHRO CONTROL WITH Z2-AXIS		
M126	SYNCHRO CONTROL CANCEL		
M131	RIGHT CHUCKING INTERLOCK BY-PASS	RIGHT CHUCKING INTERLOCK BY-PASS	
M135	MILLING SPINDLE STOP & C2 AXIS SELECTION	REVOLVING SPINDLE STOP & C2 AXIS SELECTION	

5.1.1 M-code list (4/6) of MX series

☆ : option

No.	UPPER UNIT	LOWER UNIT	option
M138	ROOF SHOWER COOLANT ON		☆
M139	ROOF SHOWER COOLANT OFF		☆
M144	ATC WAITING POT RETRACT		
M145	ATC WAITING POT ADVANCE		
M146	ATC WAITING POT CHANGE POSITION		
M147	ATC WAITING POT HOME POSITION		
M148	ATC WAITING POT HOME POSITION CHECK		
M149	ATC MAINTENANCE MODE ON		
M150	ATC MAINTENANCE MODE OFF		
M151	ATC CHANGER START FLAG		
M152	ATC 1 CYCLE		
M153	ATC DOOR OPEN		
M154	ATC DOOR CLOSE		
M155	AUTO MILLING SPINDLE ORIENTATION		
M156	V-GROOVE AIR ON		
M157	ATC CHANGER MOTOR BRAKE RELEASE		
M158	ATC CHANGER MOTOR BRAKE ENGAGE		
M159			
M160			
M161	RIGHT SPINDLE WINDING CHANGE LOW SPEED	RIGHT SPINDLE WINDING CHANGE LOW SPEED	
M162	RIGHT SPINDLE WINDING CHANGE HIGH SPEED	RIGHT SPINDLE WINDING CHANGE HIGH SPEED	
M163	RIGHT SPINDLE FORWARD & UPPER COOLANT ON	RIGHT SPINDLE FORWARD & LOWER COOLANT ON	
M164	RIGHT SPINDLE REVERSE & UPPER COOLANT ON	RIGHT SPINDLE REVERSE & LOWER COOLANT ON	
M165	RIGHT SPINDLE STOP & UPPER COOLANT OFF	RIGHT SPINDLE STOP & LOWER COOLANT OFF	
M168	RIGHT CHUCK CLAMP	RIGHT CHUCK CLAMP	
M169	RIGHT CHUCK UNCLAMP	RIGHT CHUCK UNCLAMP	
M170	INTERFERENCE CHECK RELEASE	INTERFERENCE CHECK RELEASE	
M171	INTERFERENCE CHECK RELEASE CANCEL	INTERFERENCE CHECK RELEASE CANCEL	
M172	Y-AXIS REFERENCE INTERLOCK RELEASE		
M173	Y-AXIS REFERENCE INTERLOCK RELEASE CANCEL		
M174	AUTO B-AXIS UN/CLAMP OFF & B-AXIS UNCLAMP		
M175	AUTO B-AXIS UN/CLAMP ON & B-AXIS CLAMP		
M188	C2-AXIS LOW PRESSURE CLAMP	C2-AXIS LOW PRESSURE CLAMP	
M189	C2-AXIS HIGH PRESSURE CLAMP	C2-AXIS HIGH PRESSURE CLAMP	
M190	C2-AXIS UNCLAMP	C2-AXIS UNCLAMP	
M192	SIMUL.FEEDRATE ON(BALANCE CUTTING)	SIMUL.FEEDRATE ON(BALANCE CUTTING)	
M193	SIMUL.FEEDRATE OFF(BALANCE CUTTING)	SIMUL.FEEDRATE OFF(BALANCE CUTTING)	

5.1.1 M-code list (5/6) of MX series

☆ : option

No.	UPPER UNIT	LOWER UNIT	option
M194			
M200	TOOL LOAD MONITOR OFF	TOOL LOAD MONITOR OFF	☆
M201	TOOL LOAD MONITOR ON	TOOL LOAD MONITOR ON	☆
M203	SPINDLE FORWARD SPEED SYNCHRO COMMAND	SPINDLE FORWARD SPEED SYNCHRO COMMAND	
M204	SPINDLE REVERSE SPEED SYNCHRO COMMAND	SPINDLE REVERSE SPEED SYNCHRO COMMAND	
M205	SPINDLE SYNCHRO STOP COMMAND	SPINDLE SYNCHRO STOP COMMAND	
M206	SPINDLE SYNCHRO RELEASE COMMAND	SPINDLE SYNCHRO RELEASE COMMAND	
M208	SPINDLE SPEED & PHASE CONFRIM COMMAND	SPINDLE SPEED & PHASE CONFRIM COMMAND	
M209	SPINDLE SPEED CONFIRM COMMAND	SPINDLE SPEED CONFIRM COMMAND	
M210	PARTS UNLOADER WAITING IN LEFT CHUCK	PARTS UNLOADER WAITING IN LEFT CHUCK	☆
M211	COMPOSITE CONTROL ACT	COMPOSITE CONTROL ACT	
M212	COMPOSITE CONTROL CANCEL	COMPOSITE CONTROL CANCEL	
M213	SPINDLE FORWARD PHASE SYNCHRO COMMAND	SPINDLE FORWARD PHASE SYNCHRO COMMAND	
M214	SPINDLE REVERSE PHASE SYNCHRO COMMAND	SPINDLE REVERSE PHASE SYNCHRO COMMAND	
M215		TURRET UNCLAMP BEFORE T-CODE COMMAND	
M217		Z2-AXIS INTERLOCK(A-AXIS REF) RELEASE	
M218		Z2-AXIS INTERLOCK(A-AXIS REF) RELEASE CANCEL	
M220		UPPER X-AXIS INTERLOCK RELEASE	
M221		UPPER X-AXIS INTERLOCK RELEASE CANCEL	
M226	WORK EJECTER ADVANCE	WORK EJECTER ADVANCE	☆
M227	WORK EJECTER RETRACT	WORK EJECTER RETRACT	☆
M230	PARTS UNLOADER STOPPER RETURN	PARTS UNLOADER STOPPER RETURN	☆
M231	PARTS UNLOADER STOPPER ADVANCE	PARTS UNLOADER STOPPER ADVANCE	☆
M232	PARTS UNLOADER BODY ADVANCE	PARTS UNLOADER BODY ADVANCE	☆
M233	PARTS UNLOADER BODY RETURN	PARTS UNLOADER BODY RETURN	☆
M234	PARTS UNLOADER BUCKET DOWN	PARTS UNLOADER BUCKET DOWN	☆
M235	PARTS UNLOADER BUCKET RETURN	PARTS UNLOADER BUCKET RETURN	☆
M236	PARTS UNLOADER ARM DOWN	PARTS UNLOADER ARM DOWN	☆
M237	PARTS CATCHER ARM RETURN	PARTS CATCHER ARM RETURN	☆
M250	MACHINE MAINTENANCE MODE ON	MACHINE MAINTENANCE MODE ON	
M251	MACHINE MAINTENANCE MODE OFF	MACHINE MAINTENANCE MODE OFF	
M252		TOOL CHANGE AREA INTERLOCK OFF	
M253		TOOL CHANGE AREA INTERLOCK OFF CANCEL	
M254		A-AXIS INTERLOCK IGNORE	
M255		A-AXIS INTERLOCK IGNORE CANCEL	
M256	PARTS UNLOADER CONTINUOS ACT W/RIGHT CHUCK	PARTS UNLOADER CONTINUOS ACT W/RIGHT CHUCK	☆

5.1.1 M-code list (6/6) of MX series

☆ : option

No.	UPPER UNIT	LOWER UNIT	option
M289	CANNED CYCLE ON WITH M89,M90 OF LEFT C1-AXIS	CANNED CYCLE ON WITH M89,M90 OF LEFT C1-AXIS	
M296	DYNAMIC RADIUS/DIAMETER CONVERT ON	DYNAMIC RADIUS/DIAMETER CONVERT ON	
M297	DYNAMIC RADIUS/DIAMETER CONVERT CANCEL	DYNAMIC RADIUS/DIAMETER CONVERT CANCEL	
M300	Z-AXIS MIRROR IMAGE ON	Z-AXIS MIRROR IMAGE ON	
M301	Z-AXIS MIRROR IMAGE OFF	Z-AXIS MIRROR IMAGE OFF	
M308	T.S.C COOLANT ON (RIGHT SPINDLE)	T.S.C COOLANT ON (RIGHT SPINDLE)	☆
M309	T.S.C COOLANT OFF(RIGHT SPINDLE)	T.S.C COOLANT OFF(RIGHT SPINDLE)	☆
M310	PARTS UNLOADER WAITING IN RIGHT CHUCK	PARTS UNLOADER WAITING IN RIGHT CHUCK	☆
M311	PARTS UNLOADER RETRACT FROM RIGHT CHUCK	PARTS UNLOADER RETRACT FROM RIGHT CHUCK	☆
M312	LEFT SPINDLE COOLANT BLOWER ON	LEFT SPINDLE COOLANT BLOWER ON	☆
M313	RIGHT SPINDLE COOLANT BLOWER OFF	RIGHT SPINDLE COOLANT BLOWER OFF	☆
M314	RIGHT SPINDLE COOLANT BLOWER ON	RIGHT SPINDLE COOLANT BLOWER ON	☆
M315	LEFT SPINDLE STOP (NOT CONFIRM)	LEFT SPINDLE STOP (NOT CONFIRM)	
M316		Z2-AXIS CURRENT LOAD READ (MINUS)	
M317		Z2-AXIS CURRENT LOAD READ (PLUS)	
M320	GANTRY LOADER CALL	GANTRY LOADER CALL	☆
M325	RIGHT SPINDLE STOP (NOT CONFIRM)	RIGHT SPINDLE STOP (NOT CONFIRM)	
M326		A-AXIS CURRENT LOAD READ	
M389	CANNED CYCLE ON WITH M189,M190 OF RIGHT C2-AXIS	CANNED CYCLE ON WITH M189,M190 OF RIGHT C2-AXIS	
M900	WAITING M-CODE	WAITING M-CODE	

5.2 Miscellaneous functions of TT series

5.2.1 M-code list (1/4) of TT series

☆ : option

No.	UPPER UNIT	LOWER UNIT	option	Remarks
M00	U-PROGRAM STOP	L-PROGRAM STOP		
M01	U-OPTIONAL STOP	L-OPTIONAL STOP		
M02	U-PROGRAM END	L-PROGRAM END		
M03	U-LEFT SPINDLE FORWARD	L-LEFT SPINDLE FORWARD		
M04	U-LEFT SPINDLE REVERSE	L-LEFT SPINDLE REVERSE		
M05	U-LEFT SPINDLE STOP	L-LEFT SPINDLE STOP		
M06	U-COOL JET ON	L-COOL JET ON	☆	
M07	U-COOLANT FLUSING ON	L-COOLANT FLUSING ON		
M08	U-COOLANT ON	L-COOLANT ON		
M09	U-COOLANT OFF	L-COOLANT OFF		
M10	U-LEFT PARTS CATCHER RECEIVE ADV.	L-LEFT PARTS CATCHER RECEIVE ADV.	☆	
M11	U-LEFT PARTS CATCHER RECEIVE RET.	L-LEFT PARTS CATCHER RECEIVE RET.	☆	
M12	U-LEFT COOLANT BLOWER ON	L-LEFT COOLANT BLOWER ON	☆	
M13	U-LEFT COOLANT BLOWER OFF	L-LEFT COOLANT BLOWER OFF	☆	
M14	U-LEFT SPINDLE AIR BLOW ON	L-LEFT SPINDLE AIR BLOW ON	☆	
M15	U-LEFT SPINDLE AIR BLOW OFF	L-LEFT SPINDLE AIR BLOW OFF	☆	
M16	U-COOLANT FLUSING OFF	L-COOLANT FLUSING OFF		
M17	U-M/C LOCK ACT	L-M/C LOCK ACT		
M18	U-M/C LOCK CANCEL	L-M/C LOCK CANCEL		
M19	U-LT SPINDLE ORIENTATION	L-LT SPINDLE ORIENTATION		
M21	U-OPTIONAL BLOCK SKIP ON	L-OPTIONAL BLOCK SKIP ON		
M22	U-OPTIONAL BLOCK SKIP CANCEL	L-OPTIONAL BLOCK SKIP CANCEL		
M24	U-CHIP CONVEYOR START	L-CHIP CONVEYOR START		
M25	U-CHIP CONVEYOR STOP	L-CHIP CONVEYOR STOP		
M28	U-POLYGON MODE ON	L-POLYGON MODE ON	☆	REV. SP
M29	U-RIGID TAPPING ON	L-RIGID TAPPING ON	☆	
M30	U-PROGRAM END & REWIND	L-PROGRAM END & REWIND		
M31	U-LT SPINTERLOCK BY PASS(SPDL.T/S)	L-LT SPINTERLOCK BY PASS(SPDL.T/S)		
M33	U-REV.SPINDLE FORWARD	L-REV.SPINDLE FORWARD		REV. SP
M34	U-REV.SPINDLE REVERSE	L-REV.SPINDLE REVERSE		REV. SP
M35	U-REV.SPINDLE STOP OR REV. SPINDLE(C1-AXIS) SELECTION	L-REV.SPINDLE STOP OR REV. SPINDLE(C1-AXIS) SELECTION		REV. SP
M46				M
M47				M

5.2.1 M-code list (2/4) of TT series

☆ : option

No.	UPPER UNIT	LOWER UNIT	option	Remarks
M50	U-BAR FEEDER COMMAND 1	L-BAR FEEDER COMMAND 1	☆	
M51	U-BAR FEEDER COMMAND 2	L-BAR FEEDER COMMAND 2	☆	
M52	U-SPLASH GUARD DOOR OPEN	L-SPLASH GUARD DOOR OPEN	☆	
M53	U-SPLASH GUARD DOOR CLOSE	L-SPLASH GUARD DOOR CLOSE	☆	
M54	U-PARTS COUNT	L-PARTS COUNT	☆	
M55	U-PROGRAM RESTART ON	L-PROGRAM RESTART ON		
M56	U-TAPPING MODE SELECT	L-TAPPING MODE SELECT		
M57	U-TAPPING MODE CANCEL	L-TAPPING MODE CANCEL		
M60	U-TOOL INDEX & AXIS MOVE	L-TOOL INDEX & AXIS MOVE		
M61	U-LEFT WINDING CHANGE M-CODE(LOW)	L-LEFT WINDING CHANGE M-CODE(LOW)		
M62	U-LEFT WINDING CHANGE M-CODE(HIGH)	L-LEFT WINDING CHANGE M-CODE(HIGH)		
M63	U-LT SP. FORWARD & COOLANT ON	L-LT SP. FORWARD & COOLANT ON		
M64	U-LT SP. REVERSE & COOLANT ON	L-LT SP. REVERSE & COOLANT ON		
M65	U-LT SP.STOP & COOLANT OFF	L-LT SP.STOP & COOLANT OFF		
M68	U-LEFT CHUCK CLAMP	L-LEFT CHUCK CLAMP		
M69	U-LEFT CHUCK UNCLAMP	L-LEFT CHUCK UNCLAMP		
M72	U-REV.SPINDLE DIR. CONVERT	L-REV.SPINDLE DIR. CONVERT		REV. SP
M73	U-REV.SPDL DIR. CONVERT CANCEL	L-REV.SPDL DIR. CONVERT CANCEL		REV. SP
M74	U-ERROR DETECT ON	L-ERROR DETECT ON		
M75	U-ERROR DETECT OFF	L-ERROR DETECT OFF		
M76	U-CHAMFERING ON	L-CHAMFERING ON		
M77	U-CHAMFERING OFF	L-CHAMFERING OFF		
M78				
M79				
M80	U-Q SETTER DOWN COMMAND	L-Q SETTER DOWN COMMAND		
M81	U-Q SETTER UP COMMAND	L-Q SETTER UP COMMAND		
M84	U-TURRET CW ROTATION	L-TURRET CW ROTATION		
M85	U-TURRET CCW ROTATION	L-TURRET CCW ROTATION		
M86	U-TORQUE LIMIT SKIP ON			
M87	U-TORQUE LIMIT SKIP CANCEL 1			
M88	U-LEFT SPINDLE LOW CLAMP	L-LEFT SPINDLE LOW CLAMP		REV. SP
M89	U-LEFT SPINDLE HIGH CLAMP	L-LEFT SPINDLE HIGH CLAMP		REV. SP
M90	U-LEFT SPINDLE UNCLAMP	L-LEFT SPINDLE UNCLAMP		REV. SP
M91	U-EXTERNAL M91	L-EXTERNAL M91	☆	
M92	U-EXTERNAL M92	L-EXTERNAL M92	☆	
M93	U-EXTERNAL M93	L-EXTERNAL M93	☆	
M94	U-EXTERNAL M94	L-EXTERNAL M94	☆	

5.2.1 M-code list (3/4) of TT series

☆ : option

No.	UPPER UNIT	LOWER UNIT	option	Remarks
M98	U-SUB PROGRAM CALL	L-SUB PROGRAM CALL		
M99	U-PROGRAM END or RE-START	L-PROGRAM END or RE-START		
M103	U-RIGHT SPINDLE FORWARD	L-RIGHT SPINDLE FORWARD		S,MS,SY
M104	U-RIGHT SPINDLE REVERSE	L-RIGHT SPINDLE REVERSE		S,MS,SY
M105	U-RIGHT SPINDLE STOP	L-RIGHT SPINDLE STOP		S,MS,SY
M108	U-THROUGH THE SPINDLE COOALNT ON	L-THROUGH THE SPINDLE COOALNT ON	☆	S,MS,SY
M109	U-THROUGH THE SPINDLE COOALNT OFF	L-THROUGH THE SPINDLE COOALNT OFF	☆	S,MS,SY
M112	U-RIGHT COOLANT BLOWER ON	L-RIGHT COOLANT BLOWER ON	☆	S,MS,SY
M113	U-RIGHT COOLANT BLOWER OFF	L-RIGHT COOLANT BLOWER OFF	☆	S,MS,SY
M114	U-RIGHT SPINDLE AIR BLOW ON	L-RIGHT SPINDLE AIR BLOW ON	☆	S,MS,SY
M115	U-RIGHT SPINDLE AIR BLOW OFF	L-RIGHT SPINDLE AIR BLOW OFF	☆	S,MS,SY
M116	U-WORK EJECTOR ADVANCE	L-WORK EJECTOR ADVANCE	☆	S,MS,SY
M117	B-AXIS TROQUE CONROL ON	L-Z2 AXIS TROQUE CONROL ON		S,MS,SY
M118	B-AXIS TROQUE CONROL CANCEL	L-Z2 AXIS TROQUE CONROL CANCEL		S,MS,SY
M119	U-RT SPINDLE ORIENTATION	L-RT SPINDLE ORIENTATION		S,MS,SY
M131	U-RT SPINTERLOCK BY PASS(SPDL.T/S)	L-RT SPINTERLOCK BY PASS(SPDL.T/S)		S,MS,SY
M135	U-REV.SPINDLE STOP OR REV. SPINDLE(C2-AXIS) SELECTION	L-REV.SPINDLE STOP OR REV. SPINDLE(C2-AXIS) SELECTION		REV. SP
M161	U-RIGHT WINDING CHANGE M-CODE(LOW)	L-RIGHT WINDING CHANGE M-CODE(LOW)		S,MS,SY
M162	U-RIGHT WINDING CHANGE M-CODE(HIGH)	L-RIGHT WINDING CHANGE M-CODE(HIGH)		S,MS,SY
M163	U-RIGHT SP. FORWARD & COOLANT ON	L-RIGHT SP. FORWARD & COOLANT ON		S,MS,SY
M164	U-RIGHT SP. REVERSE & COOLANT ON	L-RIGHT SP. REVERSE & COOLANT ON		S,MS,SY
M165	U-RT SP.STOP & COOLANT OFF	L-RT SP.STOP & COOLANT OFF		S,MS,SY
M168	U-RIGHT CHUCK CLAMP	L-RIGHT CHUCK CLAMP		S,MS,SY
M169	U-RIGHT CHUCK UNCLAMP	L-RIGHT CHUCK UNCLAMP		S,MS,SY
M170	U-UPPER TURRET INTERFEARNCE CHECK OFF			S,MS,SY
M171	U-UPPER TURRET INTERFEARNCE CHECK ON			S,MS,SY
M186	PART UNLOADER INTERLOCK CANCEL (ONLY TT15/18)			S,MS,SY
M187	PART UNLOADER INTERLOCK CANCEL OFF (ONLY TT15/18)			S,MS,SY
M188	U-RIGHT SPINDLE LOW CLAMP	L-RIGHT SPINDLE LOW CLAMP		REV. SP
M189	U-RIGHT SPINDLE HIGH CLAMP	L-RIGHT SPINDLE HIGH CLAMP		REV. SP
M190	U-RIGHT SPINDLE UNCLAMP	L-RIGHT SPINDLE UNCLAMP		REV. SP
M192	U-SYN. FEEDRATE CONTROL ON	L-SYN. FEEDRATE CONTROL ON		
M193	U-SYN. FEEDRATE CONTROL OFF	L-SYN. FEEDRATE CONTROL OFF		
M195	U-RIGHT SPINDLE P.C USE	L-LEFT SPINDLE P.C USE		S,MS,SY
M196	U-RIGHT SPINDLE P.C RETURN	L-LEFT SPINDLE P.C RETURN		S,MS,SY

5.2.1 M-code list (4/4) of TT series

☆ : option

No.	UPPER UNIT	LOWER UNIT	option	Remarks
M200	U-TOOL MONITOR OFF	L-TOOL MONITOR OFF		
M201	U-TOOL MONITOR ON	L-TOOL MONITOR ON		
M203	U-SPEED SYN. CONTROL FORWARD	L-SPEED SYN. CONTROL FORWARD		S,MS,SY
M204	U-SPEED SYN. CONTROL REVERSE	L-SPEED SYN. CONTROL REVERSE		S,MS,SY
M205	U-SYN. CONTROL STOP	L-SYN. CONTROL STOP		S,MS,SY
M206	U-SYN. CONTROL RELEASE	L-SYN. CONTROL RELEASE		S,MS,SY
M208	U-SYN. PHASE OK	L-SYN. PHASE OK		S,MS,SY
M209	U-SYN. SPEED & ARRIVAL OK	L-SYN. SPEED & ARRIVAL OK		S,MS,SY
M211	U-C AXIS COMPOSITION ON	L-C AXIS COMPOSTION ON		MS,SY
M212	U-C AXIS COMPOSITION OFF	L-C AXIS COMPOSTION OFF		MS,SY
M213	U-SPEED&PHASE SYN. CONTROL FWD.	L-SPEED&PHASE SYN. CONTROL FWD.		S,MS,SY
M214	U-SPEED&PHASE SYN. CONTROL RVS.	L-SPEED&PHASE SYN. CONTROL RVS.		S,MS,SY
M215	U-UPPER TURRET UNCLAMP	L-LOWER TURRET UNCLAMP		
M226	U-WORK EJECTER ADVANCE	U-WORK EJECTER ADVANCE		S,MS,SY
M227	U-WORK EJECTER RETRACT	U-WORK EJECTER RETRACT		S,MS,SY
M230	U-PTC STOPPER RET.	L-PTC STOPPER RET.	☆	S,MS,SY
M231	U-PTC STOPPER ADV.	L-PTC STOPPER ADV.	☆	S,MS,SY
M232	U-PTC BODY ADV.	L-PTC BODY ADV.	☆	S,MS,SY
M233	U-PTC BODY RET.	L-PTC BODY RET.	☆	S,MS,SY
M234	U-PTC BUCKET ADV.	L-PTC BLCKET ADV.	☆	S,MS,SY
M235	U-PTC BUCKET RET.	L-PTC BLCKET RET.	☆	S,MS,SY
M236	U-PTC ARM ADV.	L-PTC ARM ADV.	☆	S,MS,SY
M237	U-PTC ARM RET.	L-PTC ARM RET.	☆	S,MS,SY
M252	U-TOOL CHANGE INTERLOCK OFF	L-TOOL CHANGE INTERLOCK OFF		
M253	U-TOOL CHANGE INTERLOCK ON	L-TOOL CHANGE INTERLOCK ON		
M254	U- CHUK CLAMP MOVE INTERLOCK RELEASE	L- CHUK CLAMP MOVE INTERLOCK RELEASE		S,MS,SY
M255	U- CHUK CLAMP MOVE INTERLOCK ON	L- CHUK CLAMP MOVE INTERLOCK ON		S,MS,SY
M256	U- PARTS UNLOADER CONTINUOUS OPERATION	L- PARTS UNLOADER CONTINUOUS OPERATION		S,MS,SY
M289	U- LEFT SPINDLE HIGH CLAMP CONTROL	L- LEFT SPINDLE HIGH CLAMP CONTROL		M,MS,SY
M389	U-RIGHT SPINDLE HIGH CLAMP CONTROL	L-RIGHT SPINDLE HIGH CLAMP CONTROL		M,MS,SY
M310	U-RIGHT PARTS CATCHER RECEIVE ADV.	L-RIGHT PARTS CATCHER RECEIVE ADV.	☆	S,MS,SY
M311	U-RIGHT PARTS CATCHER RECEIVE RET.	L-RIGHT PARTS CATCHER RECEIVE RET.	☆	S,MS,SY
M315	U-LEFT SPINDLE STOP(NOT CONFIRM)	L-LEFT SPINDLE STOP(NOT CONFIRM)		
M318	U-CHUCK UNCLAMP & WORK EJECTOR ON (ONLY TT15/18)	L-CHUCK UNCLAMP & WORK EJECTOR ON (ONLY TT15/18)		M,MS,SY
M325	U-RIGHT SPINDLE STOP(NOT CONFIRM)	L-RIGHT SPINDLE STOP(NOT CONFIRM)		S,MS,SY
M900	**U-WAITING M-CODE (M900~M999)**	**L-WAITING M-CODE (M900~M999)**		

6. Fixed cycle functions

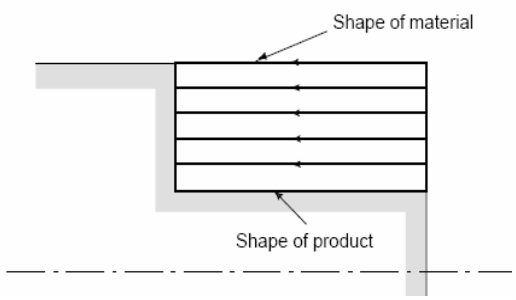
6.1 Simple Fixed Cycle (G90,G92,G94)

Machining operation that requires a simple stock removal pattern cycle to be repeated is usually programmed using several blocks of commands. However, if a fixed cycle function is used, such an operation can be programmed in a single block.

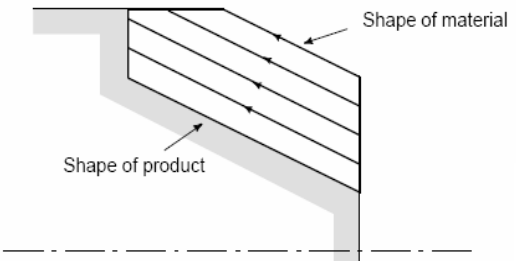
Three types of simple fixed cycle patterns are provided.

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

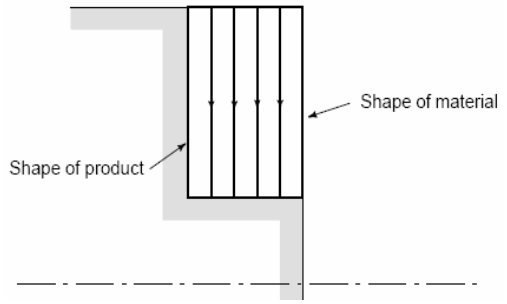
(1) Straight cutting cycle (G90)



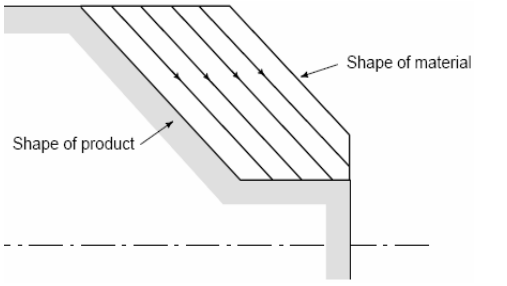
(2) Taper cutting cycle (G90)



(3) Face cutting cycle (G94)



(3) Face taper cutting cycle (G94)



NOTE

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

2 The following two applications can be performed.

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

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

(Example)

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

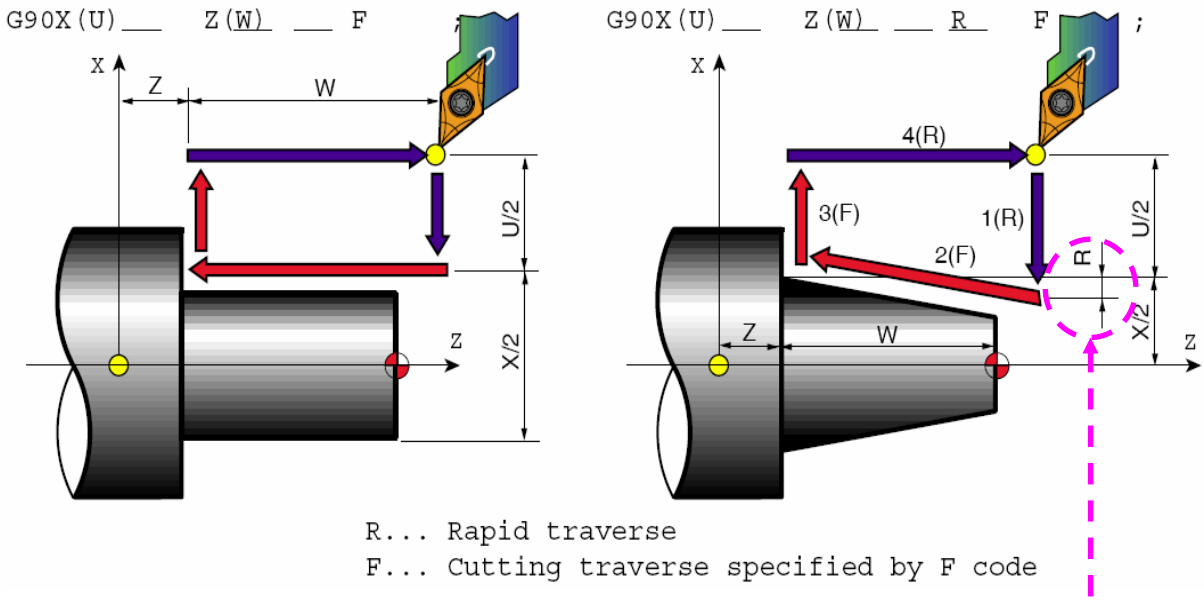
6.1.1 O.D. and I.D. turning cycle (G90)

Straight or taper cutting cycle to be executed in the Z axis direction can be programmed using a single block of commands.

(1) Command format

G90 X(U)_ Z(W)_ R_ F_

X(U) : X coordinate at the end point of Z
Z(W) : The end point of Z
R - : The taper size in radius when cutting from the start point to **X+** direction
R + : The taper size in radius when cutting from the start point to **X-** direction
F : Feedrate

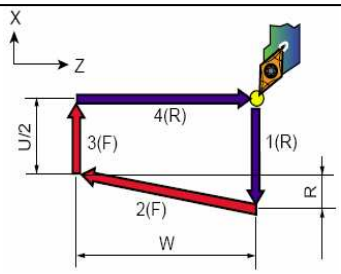


Note 1) G90 command, F (feedrate) and Dimension words (X,Z,R) are modal.
Note 2) Designate the taper size in radius
Note 3) When cutting of ID and in lower unit, above format can be used with same one.

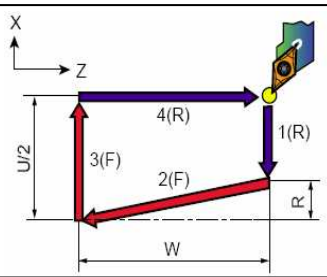
(2) Taper direction

Taper is generated as illustrated below according to the sign of an R word.

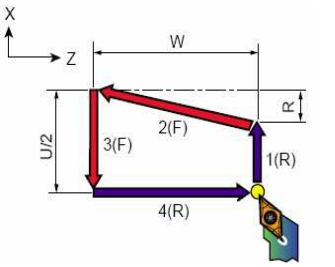
R<0 : OD turning (U<0, W<0)



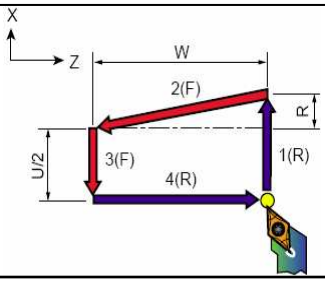
R>0 : OD turning (U<0, W<0)



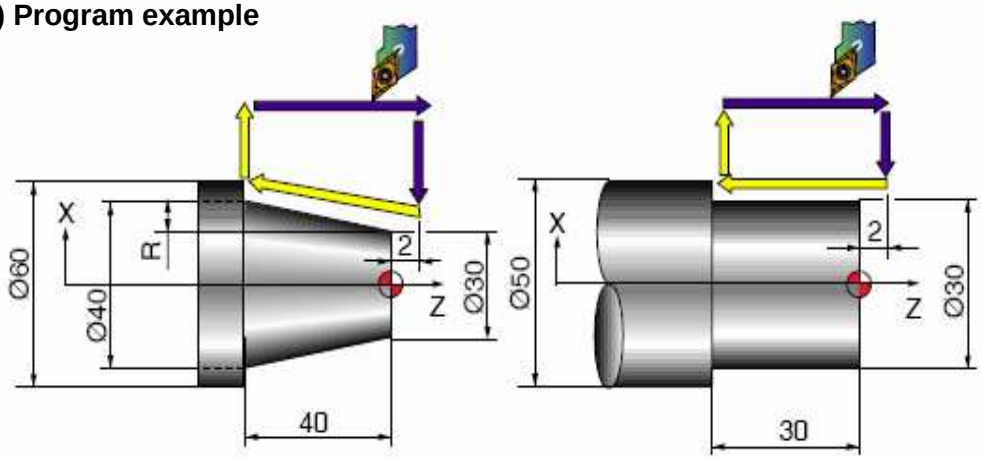
R<0 : ID turning (U>0, W<0)



R>0 : ID turning (U>0, W<0)



(3) Program example




```
G50 S2000 T0100 ;
G96 S200 M03 ;
G00 X62.0 Z2.0 T0101 M8 ;
G90 X50.0 Z-40.0 F0.25 ;
      X40.0 ;
      Z-20.0 R-2.75 ; (from X29.5 Z2.)
      Z-40.0 R-5.25 ; (from X29.5 Z2.)
G30 U0 W0 ;
M01;
```

```
G50 S2000 T0100 ;
G96 S200 M03 ;
G00 X52.0 Z2.0 T0101 M8 ;
G90 X40.0 W-32.0 F0.25 ;
      X30.0 ;
G30 U0 W0
M01
```

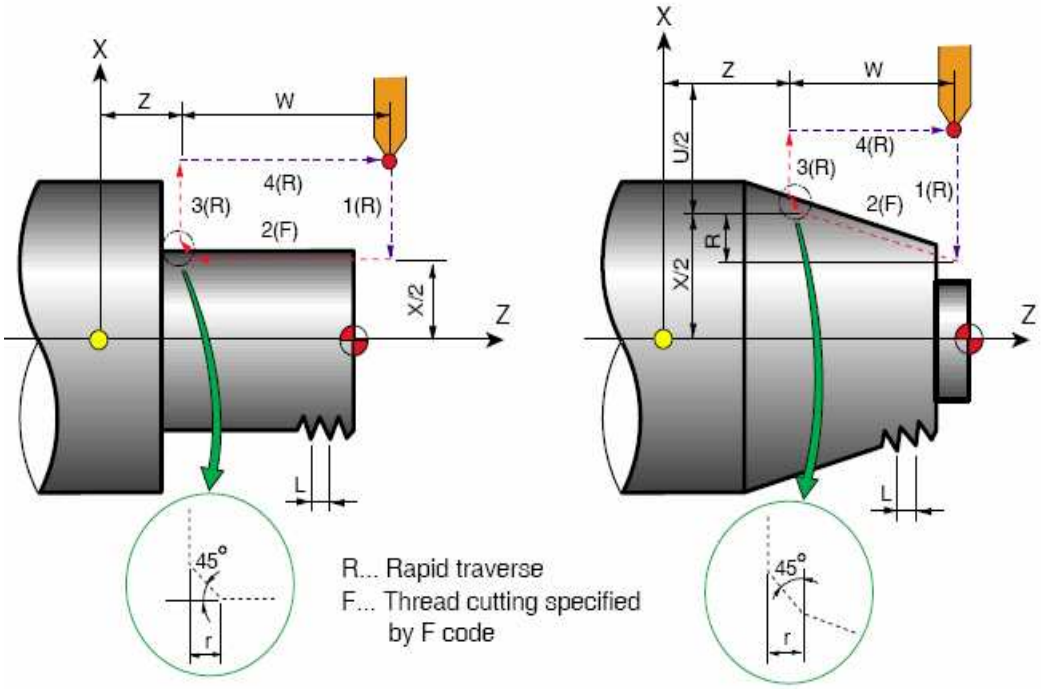
6.1.2 Thread Cycle (G92)

Straight thread and taper thread cutting cycle can be programmed using a single block of commands.

(1) Command format

G92 X(U)_ Z(W)_ R_ F_

X(U) : X axis coordinate of thread process position of each time
Z(W) : The end point of Z
R - : The taper size in radius when cutting from the start point to **X+** direction
R + : The taper size in radius when cutting from the start point to **X-** direction
F : Thread lead (pitch X n)



Note 1) To cut the thread by repeating the thread cutting pattern, never change the spindle speed during thread cutting.

Note 2) Do not use constant surface control mode.

Note 3) The feedrate override function is invalid during thread cutting cycle.

Note 4) At the portions near the start and end point of thread, incomplete threads are cut due to automatic deceleration/acceleration. Therefore, allow margin in these portions so that required length of thread can be cut correctly.

Note 5) when cutting of ID and in lower unit, above format can be used.

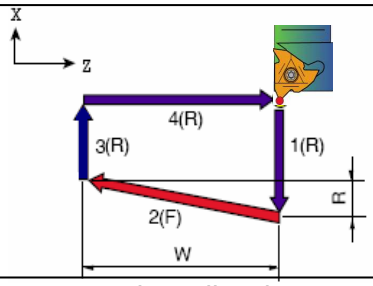
Note 6) Chamfering width ”r” is set for parameter. NO.5130 (0.1L to 12.7L)

Note 7) For the thread lead and the spindle speed, F x S ≤ 4800

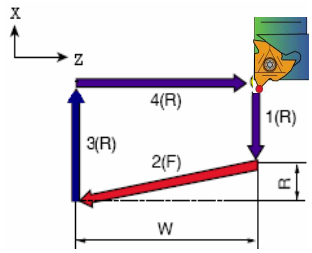
(2) Taper direction

Taper is generated as illustrated below according to the sign of an R word.

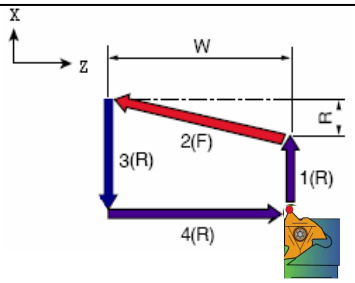
R< 0 : OD threading (U<0, W<0)



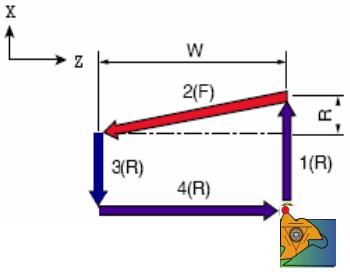
R>0 : OD threading (U<0, W<0)



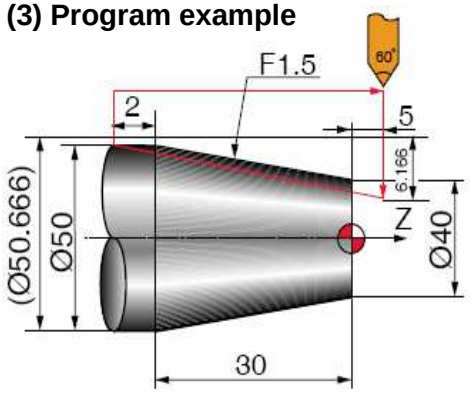
R<0 : ID threading (U>0, W<0)

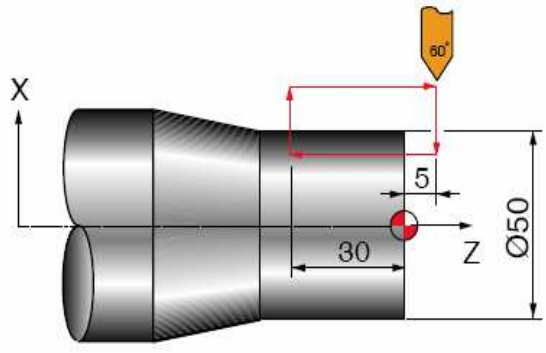


R>0 : ID threading (U>0, W<0)



(3) Program example






```
G50 S1000 T0100 ;
G97 S1000 M03 ;
G00 X70.0 Z5.0 T0101 M8 ;
G92 X49.4 Z-32.0 R-6.166 F1.5 ;
     X49.0  ;
     X48.72 ;
     X48.48 ;
     X48.32 ;
     X48.2  ;
G30 U0 W0 ;
```

```
G50 S1000 T0100 ;
G97 S1000 M03 ;
G00 X60.0 Z5.0 T0101 M8 ;
G92 X49.4 Z-30.0 F0.25 ;
     X49.0  ;
     X48.72 ;
     X48.48 ;
     X48.32 ;
     X48.2  ;
G30 U0 W0 ;
```

6.1.3 End Face Turning Cycle (G94)

Straight and taper cutting cycle to be executed along the face (X-axis direction) can be programmed using a single block of commands.

If the same pattern is repeated to generate multiple tool paths, simply specify the coordinate values that change according to the progress of the cutting.

(1) Command format

G94 X(U)_ Z(W)_ R_ F_

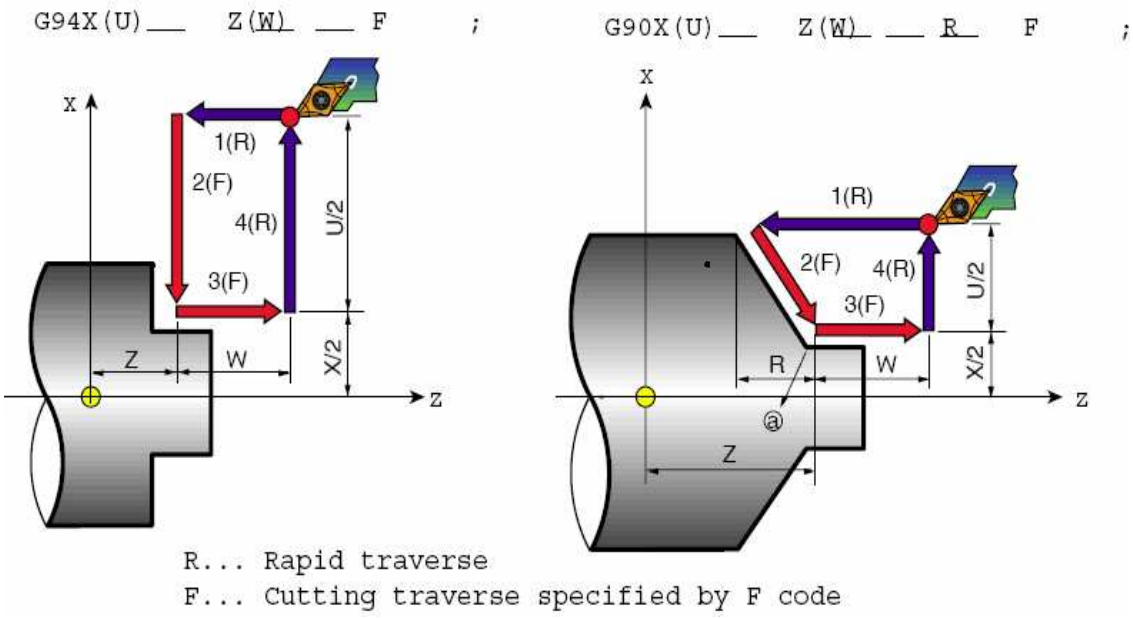
X(U) : End point of X

Z(W) : End point of inclination in Z = ⓐ point of cycle distance

R - : The inclined radius value when cutting from the start point to **Z+** direction

R + : The inclined radius value when cutting from the start point to **Z-** direction.

F : Cutting feedrate



Note 1) G90 command, F (feedrate) and Dimension words (X,Z,R) are modal.

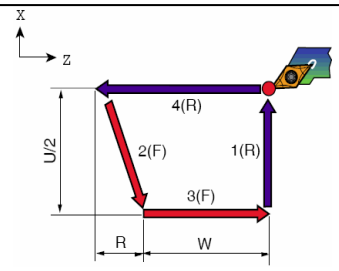
Note 2) Designate the taper size in radius

Note 3) When cutting of ID and in lower unit, above format can be used

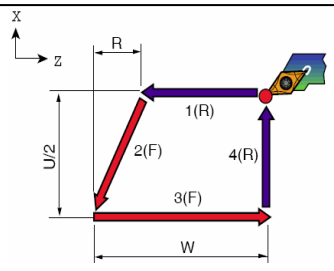
(2) Taper direction

Taper is generated as illustrated below according to the sign of an R word.

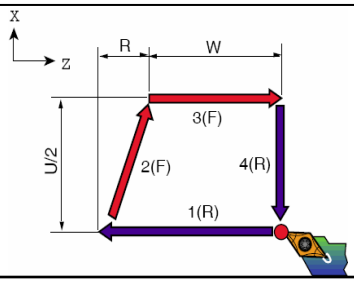
R<0 : OD turning (U<0, W<0)



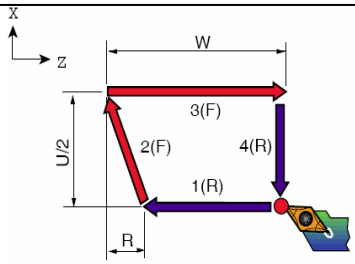
R>0 : OD turning (U<0, W<0)

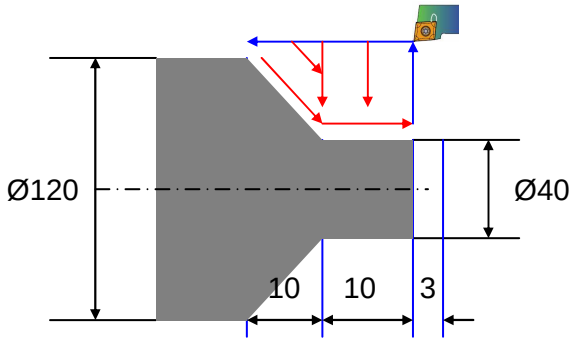


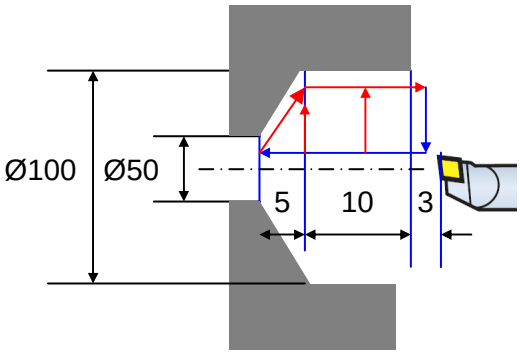
R<0 : ID turning (U>0, W<0)



*** R>0** : ID turning (U>0, W<0)








```
G50 S2000 T0100 ;
G96 S180 M03 ;
G00 X126.0 Z3.0 T0101 M8 ;
G94 X40.0 Z-5.0 F0.3 ;
      Z-10.0   ;
G00 Z-10.0  ;
G94 X82.8 R-5.4     ; (from X126.0 Z-15.4)
      X40.0 R-10.75 ; (from X126.0 Z-20.75)
G30 U0 W0 ;
M01 ;
```

```
G50 S2000 T0100 ;
G96 S180 M03 ;
G00 X45.0 Z2.0 T0101 M8 ;
G94 X100.0 Z-5.0 F0.25 ;
      Z-10.0   ;
      Z-10.0 R-5.0 ; (from X45.0 Z-15.0)
G00 X45.0 Z10.0    ;
G30 U0 W0 ;
M01 ;
```

6.2 Simple Fixed Cycle (G70-G76)

The multiple repetitive cycle option makes programming simple by making the most of the fixed cycle capability. For example, by defining the finish shape, necessary tool paths to carry out roughing cycle are automatically generated.

The function also provides fixed threading cycles.

(1) Notes on Multiple Repetitive cycle (G70-G76)

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

9. G74, G75, and G76 also do not support the input of a decimal point for P or Q. The least input increments are used as the units in which the amount of travel and depth of cut are specified.
10. When #1 = 2500 is executed using a custom macro, 2500.000 is assigned to #1. In such a case, P#1 is equivalent to P2500.
11. Tool nose radius compensation cannot be applied to G71, G72, G73, G74, G75, G76, or G78.
12. The multiple repetitive cycle cannot be executing during DNC operation.
13. Interruption type custom macro cannot be executed during executing the multiple repetitive cycle.
14. The multiple repetitive cycle cannot be executing during Advanced Preview Control mode.

(2) Comparison with each canned cycle

< Rough turning cycle > Rough OD & ID turning cycle (G71) Rough Face cycle (G72) Pattern repeating Cycle (G73)	⇔	**< Finish turning cycle >** Finish cycle of G71,G72,G73 (G70)
		<Single turning cycle > OD & ID turning cycle (G90) End face turning cycle (G94)
< Drilling + Grooving > Z-axis peck drilling cycle (G74) X-axis peck drilling cycle (G75) * G75 :Drilling, grooving, Cut-off	⇔	**< High speed Drilling >** Z-axis drilling cycle (G83) Z-axis tapping cycle (G84) Z-axis boring cycle (G85) X-axis drilling cycle (G87) X-axis tapping cycle (G88) X-axis boring cycle (G89)
< Multiple thread > Multiple thread cycle (G76)	⇔	**< the other thread cycle >** Constant lead thread (G32) Variable lead thread (G34) Single thread cycle (G92)

6.2.1 Rough O.D. turning cycle (G71)

When the finish shape is defined (A to A' to B), rough turning paths to remove stock with depth of cut of Δd are automatically generated. In the rough turning cycle, finishing stock of Δu/2 and Δw/2 are left on the defined finish shape.

(1) Command format

```
G71 U (Δd)  R( e) ;
G71 P(ns) Q(nf) U(Δu) W(Δw) F(f) S(s) T(t) ;
N (ns) ….. ;
        ….. F_ ;
        ….. S_ ;
        ….. T_  ;
N (nf) …..  ;
```

Δd : cut depth (radius) of one time
The direction of in-feed is determined according to A-A' direction
e : escape volume and direction (always 45˚ escape)
ns : The first sequence no. used to program the finishing shape.
nf : The last sequence no. used to program the finishing shape
Δu : distance (diameter) and direction of finishing stock in the X axis direction
Δw : distance (radius) and direction of finishing stock in the Z axis direction
f, s, t : the feedrate, spindle speed and tool number
Ant F,S and T function contained in blocks N (ns) to N (nf) in the cycle is ignored in G71 and the F,S or T in the G71 block is effective.

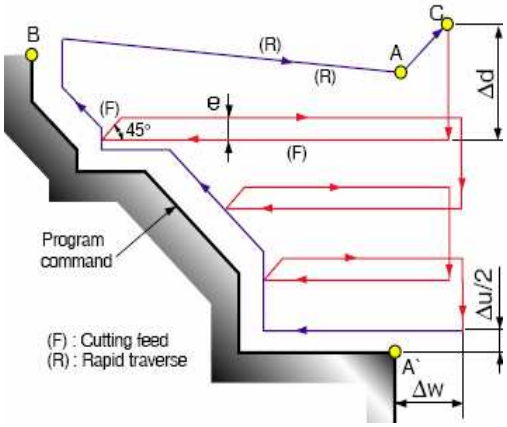
Note 1) F,S and T commands, if designated in the blocks defining the finishing shape, are all ignore and those designated in the G71 block or preceding one are valid. If the optional constant surface speed control function is selected, the G96 or the G97 command, if designated in the blocks defining the finishing shape, is ignored and the G-code designated in the G71 blocks or preceding one is valid.

Note 2) The tool path between A and A' is specified in the block with sequence number "ns" including G00 or G01.

Note 3) The subprogram cannot be called from the block between sequence number "ns" and "nf".

(2) Type- Ⅰ

The shape A' to B must be monotonously (steadily) increasing or decreasing pattern in both X and Z axis. In the block of tool path between A and A' (sequence number "ns"), a move command in the z axis cannot be specified.



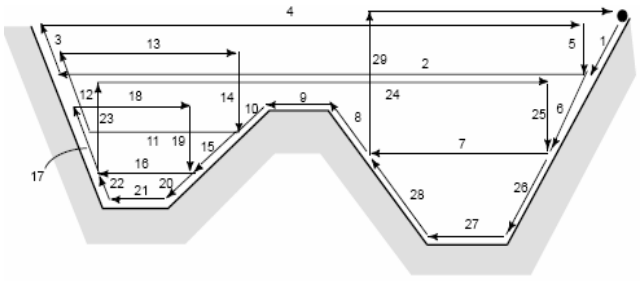
< TYPE- Ⅰ >

G71 U10.0 R5.0 ;
G71 P100 Q200 ….. ;
N100 G01 X____ ;

….. ;
N200 X___ Z___ ;

(3) Type- Ⅱ

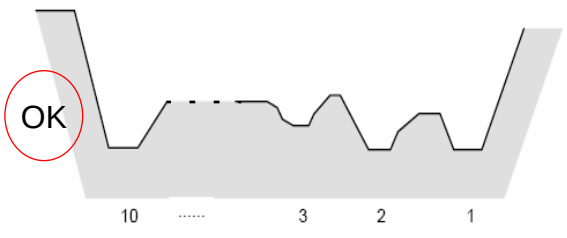
The shape A' to B must be monotonously (steadily) increasing or decreasing pattern in Z axis. The X-axis does not need to be monotonous increasing or decreasing pattern. It may have up to 10 concaves (pockets). The first cut portion need not be vertical. Any profile is permitted if monotonous change is shown along the Z axis. When the first block does not include Z motion and type Ⅱ is to be used, W0 must be specified.

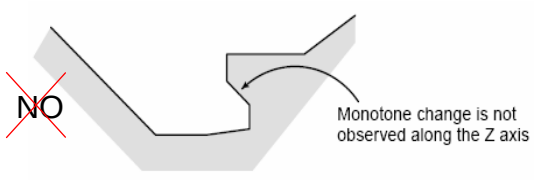


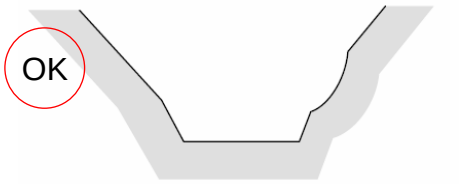
< TYPE- Ⅱ >

G71 U10.0 R5.0 ;
G71 P100 Q200 ….. ;
N100 G01 X____ Z___ ;

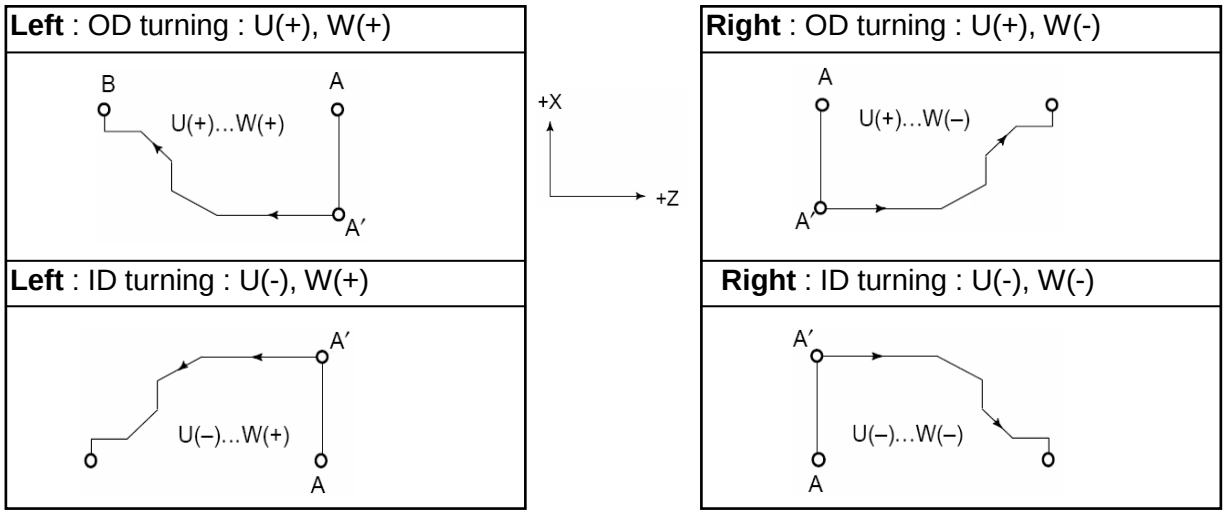
….. ;
N200 X___ Z___ ;



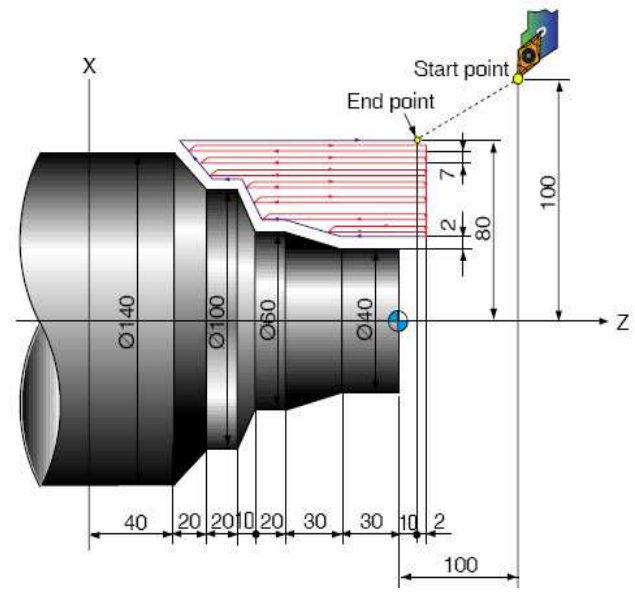




(4) Cutting direction of Type Ⅰ(A-A'-B)



(5) Program example (Type Ⅰ)




```
G00 X200.0 Z100.0 :
G00 X160.0 Z10.0 :
G71 U7.0 R1.0 :
G71 P014 Q021 U4.0 W2.0 F0.3 S550 :
N014 G00 G42 X40.0 S700 :
G01 W-40.0 F0.15 :
    X60.0 W-30.0 :
    W-20.0 :
    X100.0 W-10.0 :
    W-20.0 :
    X140.0 W-20.0 :
N021    G40 U2.0 :
G70 P014 Q021 :
G00 X200.0 Z100.0 :
M30 :
```

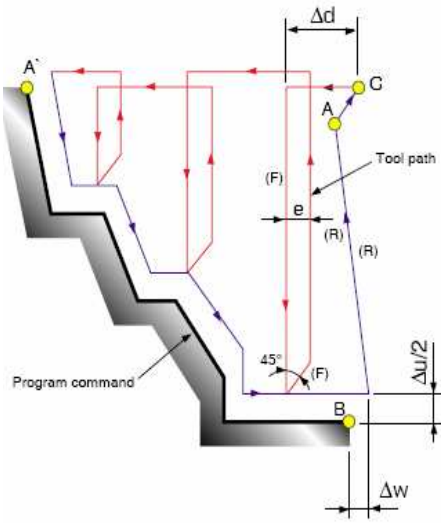
6.2.2 Rough facing cycle (G72)

The cycle is programmed and executed in the same manner as for G71 cycle.
In this cycle, however, cutting is carried out by the tool motion along the X-axis.
The program to carry out the finish cycle for shape (A to A' to B)

(1) Command format

```
G72 U (Δd)  R( e) ;
G72 P(ns) Q(nf) U(Δu) W(Δw) F(f) S(s) T(t) ;
N (ns) ..... ;
        ..... F_ ;
        ..... S_ ;
        ..... T_  ;
N (nf) .....  ;
```

Δd : cut depth (radius) of one time
The direction of in-feed is determined according to A-A' direction
e : escape volume and direction (always 45˚ escape)
ns : The first sequence no. used to program the finishing shape.
nf : The last sequence no. used to program the finishing shape
Δu : distance (diameter) and direction of finishing stock in the X axis direction
Δw : distance (radius) and direction of finishing stock in the Z axis direction
f, s, t : the feedrate, spindle speed and tool number
Ant F,S and T function contained in blocks N (ns) to N (nf) in the cycle is ignored in G72 and the F,S or T in the G72 block is effective.



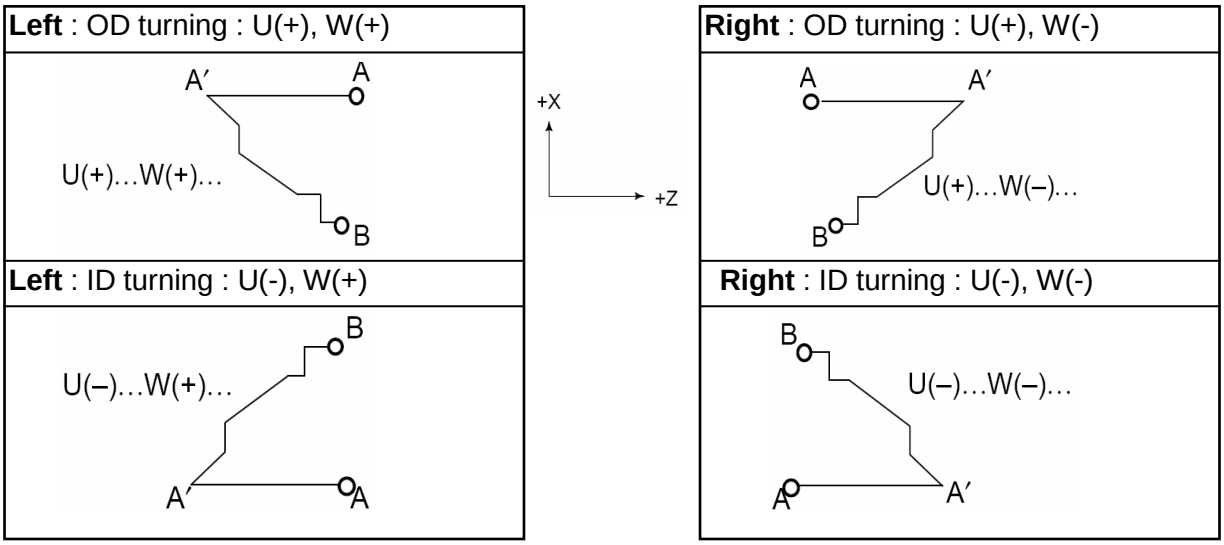
Note 1) the shape between A and A' is defined in the block assigned with sequence number "ns". In this area, tool motion is controlled in either G00 or G01 mode and no Z-axis command is allowed.

Note 2) The shape A'-B must be monotonously increasing or decreasing both in the X-axis and A-axis direction.

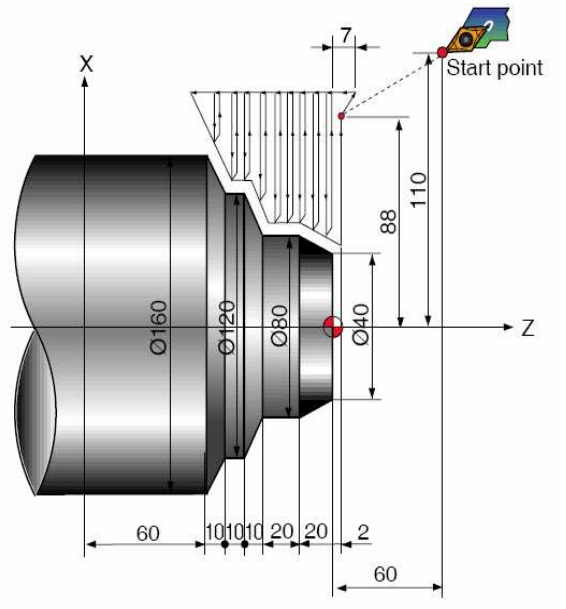
Note 3) the in-feed mode along A-A' conforms to the mode used to designate the tool motion A-A' as in the G71 cycle.

Note 4) All of these cutting cycles are made parallel to X-axis.

(2) Cutting direction (A-A'-B)



(3) Program example




```
G00 X220.0 Z60.0 :
G00 X176.0 Z2.0 :
G72 W7.0 R1.0 :
G72 P014 Q021 U4.0 W2.0 F0.3 S550 :
N014 G00 G41 Z-70.0 S700 :
X160.0 :
G01 X120.0 Z-60.0 F0.15 :
     W10.0 :
     X80.0 W10.0 :
     W20.0 :
     X36.0 W22.0 :
N021      G40 :
G70 P014 Q021 :
G00 X220.0 Z60.0 :
M30 :
```

6.2.3 Pattern repeat cycle (G73)

A designated cutting pattern can be executed repeatedly by shifting the position.

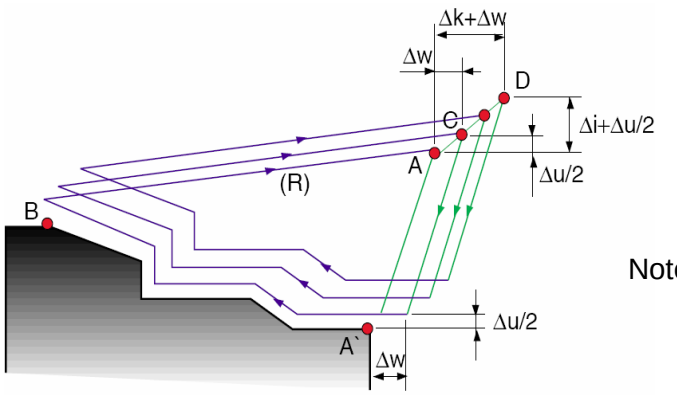
This cycle is used to effectively cut a forged or cast work-piece on which rough shape is already formed. (ex. Casting or forging method etc.)

The program to carry out the finish cycle for shape (A to A' to B)

(1) Command format

```
G73 U (Δi) W(Δk) R( d) ;
G73 P(ns) Q(nf) U(Δu) W(Δw) F(f) S(s) T(t) ;
N (ns) ….. ;
        ….. F_ ;
        ….. S_ ;
        ….. T_  ;
N (nf) …..  ;
```

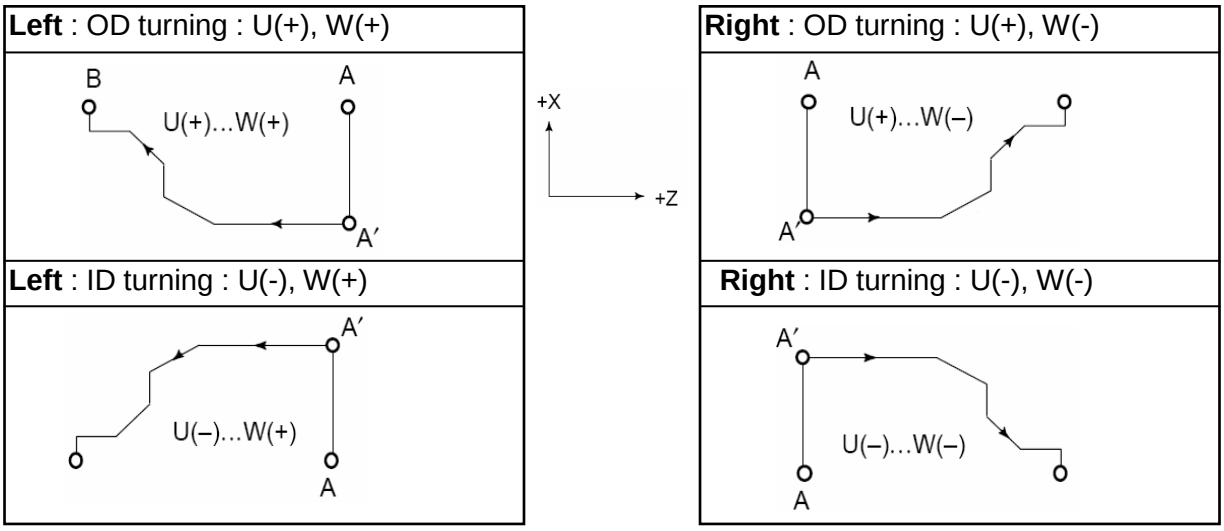
Δi : distance and direction of relief in the X-axis direction (radius)
Δk : distance and direction of relief in the Z-axis direction
d : the number of division
the roughing cycle is repeated by the number of times set for "d".
ns : The first sequence no. used to program the finishing shape.
nf : The last sequence no. used to program the finishing shape
Δu : distance (diameter) and direction of finishing stock in the X axis direction
Δw : distance (radius) and direction of finishing stock in the Z axis direction
f, s, t : the feedrate, spindle speed and tool number
Ant F,S and T function contained in blocks N (ns) to N (nf) in the cycle is ignored in G73 and the F,S or T in the G73 block is effective.



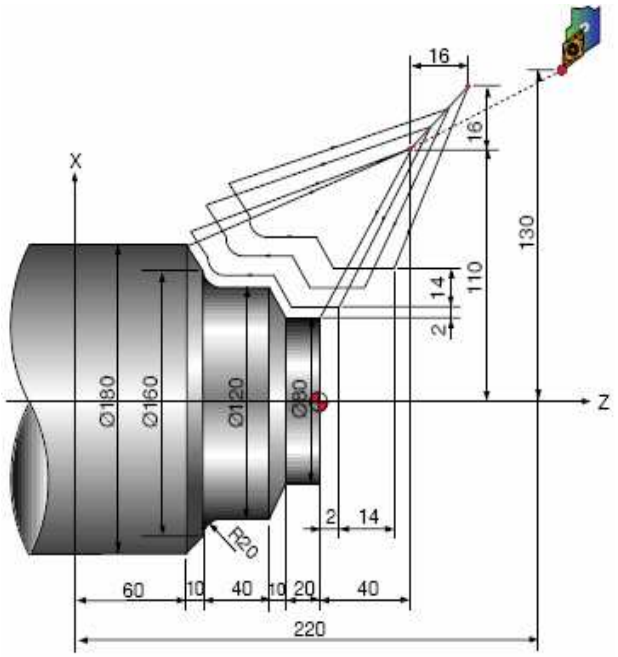
Note 1) while the values Δi and Δk, or Δw are specified by address U and W respectively, the meanings of them are determined by the presence of address P and Q in G73 block. When P and Q are not specified in a same block, address ΔU and ΔW indicates u and w respectively.

Note 2) In the G73 mode, there are four possible patterns and, therefore, the sign of Δu, Δw, ΔI and Δk must be determined carefully. After the completion of the cycle, the cutting tool returns to point A.

(2) Cutting direction (A-A'-B)



(3) Program example




```
G00 X260.0 Z80.0 :
G00 X220.0 Z40.0 :
G73 U14.0 W14.0 R3 :
G73 P014 Q020 U4.0 W2.0 F0.3 S0180 :
N014 G00 G42 X80.0 Z2.0 :
G01 W-20.0 F0.15 S0600 :
X120.0 W-10.0 :
W-20.0 S0400 :
G02 X160.0 W-20.0 R20.0 :
G01 X180.0 W-10.0 S0280 :
N020 G40 :
G70 P014 Q020 :
G00 X260.0 Z80.0 :
M30 :
```

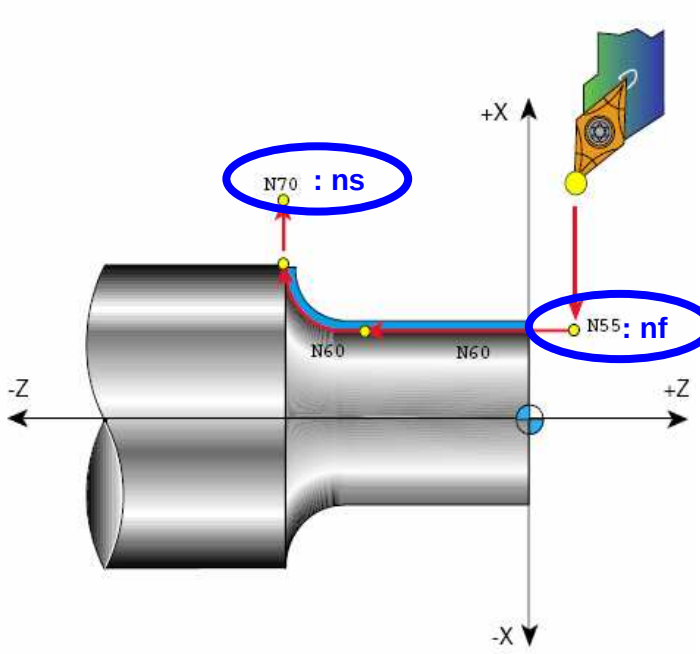
6.2.4 Finishing cycle (G70)

After carrying out a rough cycle, using G71,G72 or G73 cycle, the finishing cycle can be executed using the G70 cycle.

(1) Command format

G70 P(ns) Q(nf) ;

ns : The first sequence no. used to program the finishing shape.
nf : The last sequence no. used to program the finishing shape



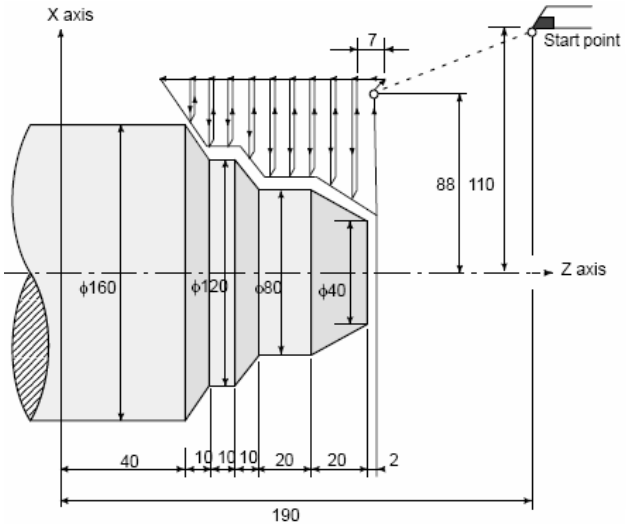
Note 1) In the G70 mode, F,S and T commands designated in the G71,G72or G73 block are all ignored and those designated in the blocks between “ns” and “nf” block are used.

Note 2) After the completion of G70 cycle, the cutting tool returns to the start point at a rapid traverse. Then , the block that follows the G70 block is continuously executed.

Note 3) Subprogram call from a block, within “ns” and “nf” blocks, is not available.

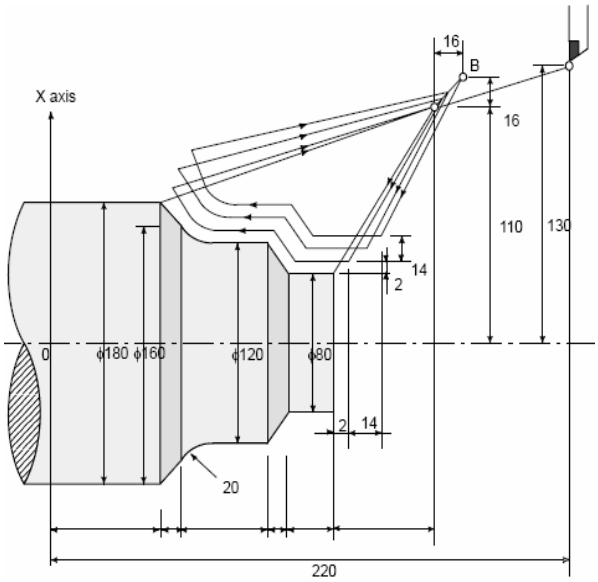
(2) Program example

< Stock removal in facing (G72) >




```
G50 X220.0 Z190.0 ;
G00 X176.0 Z132.0 ;
G72 W7.0 R1.0 ;
G72 P014 Q019 U4.0 W2.0 F0.3 S550 ;
N014  G00 Z58.0 S700 ;
G01 X120.0 W12.0 F0.15 ;
   W10.0 ;
   X80.0 W10.0 ;
   W20.0 ;
N019     X36.0 W22.0 ;
G70 P014 Q019 ;
```

< Pattern repeating (G73) >




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

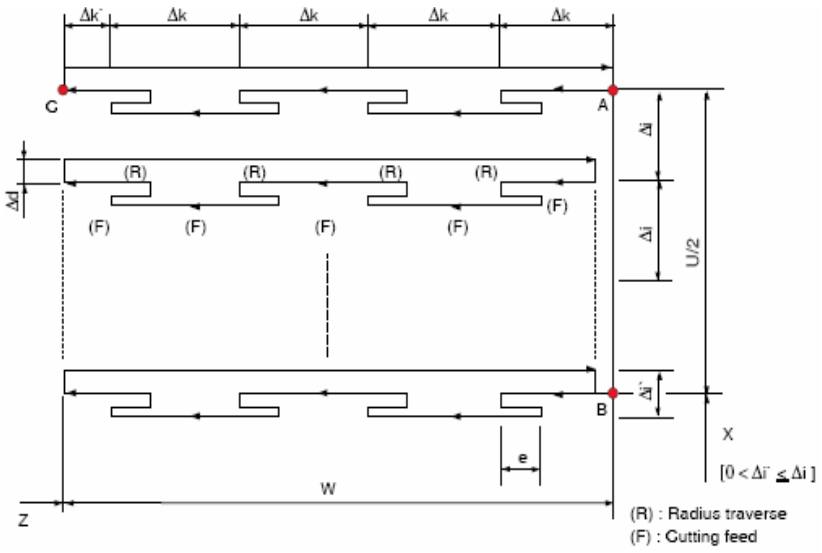
6.2.5 End face peck drilling cycle (G74)

The illustrated cycle is programmed using the command format indicated in below
This cycle permits chip disposal for O.D turning operation. If X(U) and P words are omitted, the program calls for only Z-axis motion, there designating peck-drilling cycle.

(1) Command format

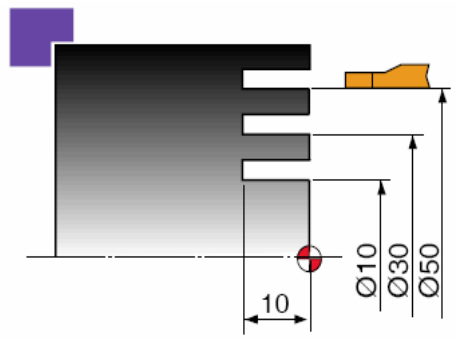
G74 R(e) ;
G74 X(U)_ Z(W)_ P(Δi) Q(Δk) R(Δd) F(f) ;

e : retraction amount
X : X-coordinate of point B
U : distance between point A and B (incremental)
Z : Z-coordinate of point C
W : distance between point A and pint C
Δi : distance in the X-axis direction (P1000 = 1mm)
Δk : distance in the Z-axis direction (Q1000 = 1mm)
Δd : relief amount of the tool at the cutting bottom. The Δd is always plus However, if address X(U) and Δi are omitted, the relief direction can be specified by the desired sign.
f : feedrate

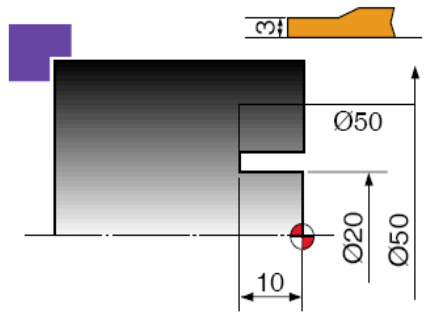


Note 1) While both e and Δd are specified by address R, the meanings of them are determined by the present address X(U). When X(U) is specified, Δi is use.
Note 2) The cycle machining is performed by G74 command with X(U) specification.

(2) Program example




```
G50 S2000 T0100 :
G96 S80 M03 :
G00 X50.0 Z1.0 T0101 :
G74 R1.0 :
G74 X10.0 Z-10.0 P10000 Q3000 F0.1 :
G00 X200.0 Z200.0 T0100 :
M30 :
```

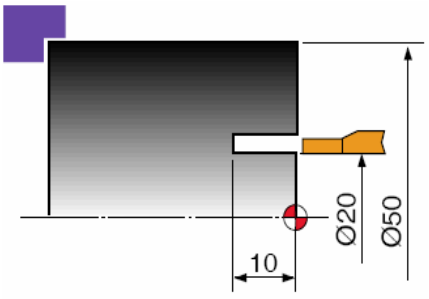



```
G50 S2000 T0100 :
 G96 S80 M3 :
 G0 X47.0 Z1.0 T0101M8 :
 G74 R1.0 :
 G74 Z-10.0 Q3000 F0.1 :
 G0 U-5.0 :
 G74 X20.0 Z-10.0 P2500 Q3000 F0.1 :
 G0 X200.0 Z200.0 T0100 :
 M30 :
```

< Drilling cutting cycle >

G74 R_ ;

G74 Z_ Q_ F_ ;




```
G00 X20.0 Z1.0 :
G74 R1.0 :
G74 Z-10.0 Q3000 F0.1 :
G00 X200.0 Z200.0 :
M30 :
```

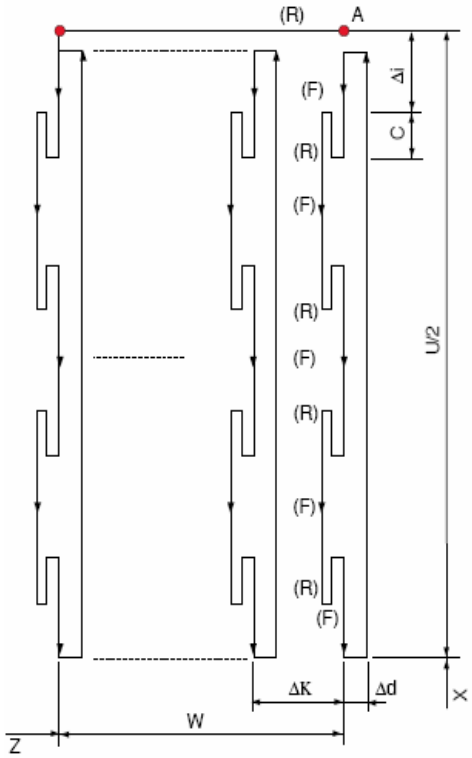
6.2.6 End face peck drilling cycle (G75)

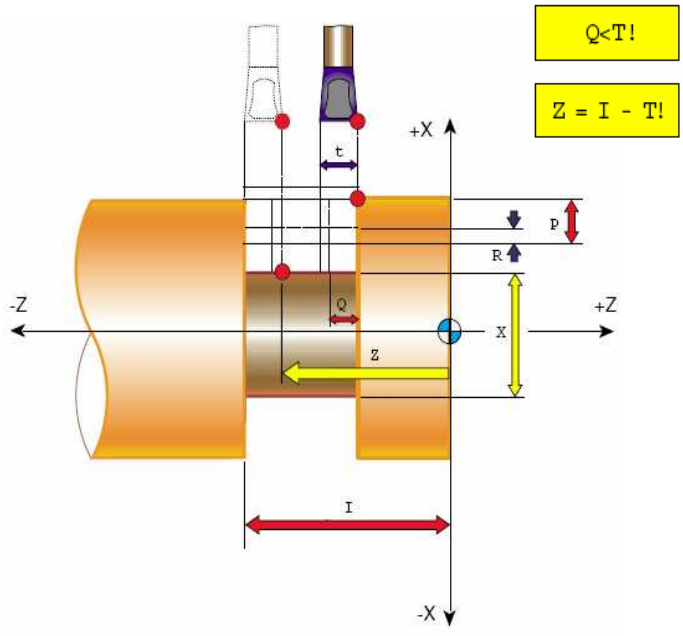
The illustrated cycle is programmed using the command format indicated in below
This cycle is called by the G75 command is the same pattern as that called by G74 command. The difference between these patterns is the pattern execution direction. In the G75 cycle, in-feed is made in the X-axis direction. while in the G74 cycle, in-feed is made in the Z-axis direction.

(1) Command format

G75 R(e) ;
G75 X(U)_ Z(W)_ P(Δi) Q(Δk) R(Δd) F(f) ;

e : retraction amount
X : X-coordinate of point B
U : distance between point A and B (incremental)
Z : Z-coordinate of point C
W : distance between point A and pint C
Δi : distance in the X-axis direction
Δk : distance in the Z-axis direction (Q1000=1mm)
Δd : relief amount of the tool at the cutting bottom. The Δd is always plus However, if address X(U) and Δi are omitted, the relief direction can be specified by the desired sign.
f : feedrate



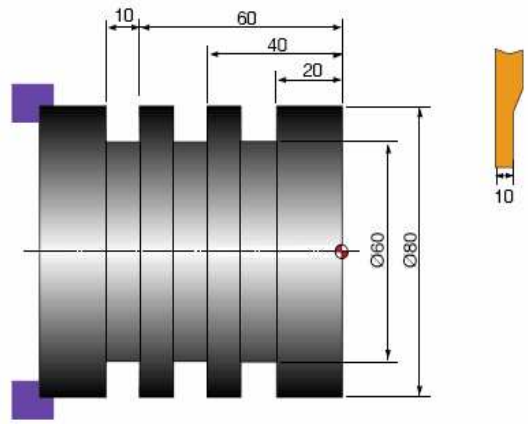


Note 1) Chip breaking is impossible in this cycle , and grooving in X axis and peck drilling in X axis (in this case Z, W, and Q are omitted) are possible.

Note 2) The format used to designate the cutting-off cycle is indicated above.

Note 3) Both G74 and G75 are used for grooving and drilling, and permit the tool to relief automatically. Four symmetrical patterns are considered, respectively.

(2) Program example




```
N10 G50 S500 T0100 :
    G97 S_   M03 :
    G00 X90.0 Z1.0 T0101 :
        X82.0 Z-60.0 :
    G75 R1.0 :
    G75 X60.0 Z-20.0 P3000 Q20000 F0.1 :
    G00 X90.0
        X200.0 Z200.0 T0100 :
    M30 :
```

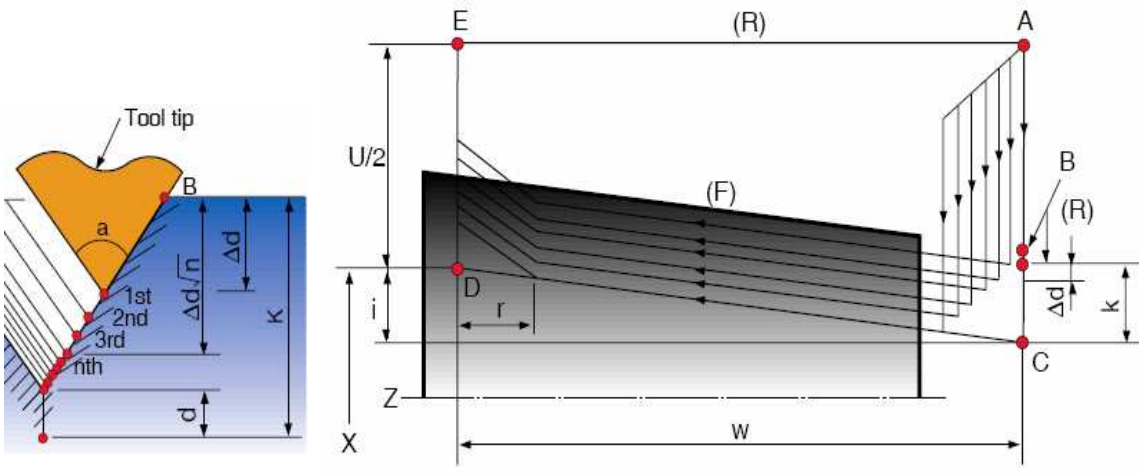
6.2.7 Multiple thread cutting cycle (G76)

Using the G76 command, the thread cutting cycle as illustrated below is executed.

(1) Command format

G76 P(m)(r)(a) Q(Δd min) R(d) ;
G76 X(U)_ Z(W)_ R(i) P(k) Q(Δd) F(L) ;

m : repetitive count in finishing (1 to 99)
r : chamfering amount (2-digit number)
(0.0L to 9.9L in 0.1L increment **:** L means the thread lead.)
a : angle of tool tip (thread) (2-digit number)
(one of six kinds of angle 80˚, 60 ˚, 55 ˚, 30 ˚, 29 ˚ and 00 ˚)
* P(m)(r)(a) : m, r and a are specified by address P at the same time.
→ *P021260 (m=2, r=1.2L, a=60˚)*
Δd min : minimum cutting depth (radius, omit the decimal point → *Q100=0.1mm*)
(If the depth of cut, calculated for each thread cutting pass, becomes smaller than this value, the depth of cut is clamped at this value.)
R(d) : finishing allowance (+,radius, omit the decimal point → *R200=0.2mm*)
R(i) : difference of thread radius
(If i=0, ordinary straight thread cutting can be made.
R- : OD taper thread to X+ , R+: ID taper thread to X-)
P(k) : height of thread (+,radius, omit the decimal point → *P900=0.9mm*)
Q(Δd) : depth of cut in 1st cut (+,radius, omit the decimal point → Q5*00=0.5mm*)
L : lead of thread (same as G32)
U,W : minus (determined by the direction of the tool path AC and CD)
R : minus (determined by the direction of the tool path AC)



Note 1) Cautions to be taken into consideration are the same as those when programming thread cutting in the G32 or G92 mode.

Note 2) Chamfering distance designation is also valid for the G92 cycle.

Note 3) By using this cycle, one edge cutting is performed and the load on the tool tip is reduced. Cutting amount per one cycle is held constant.

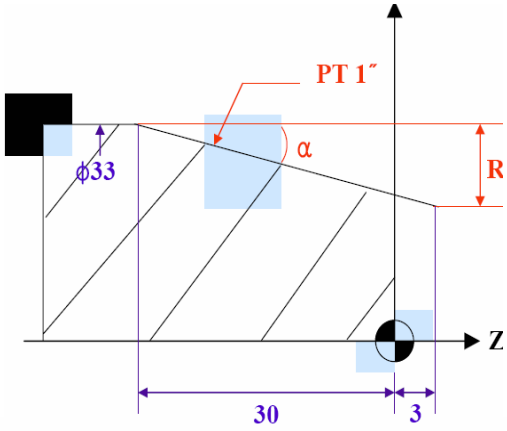
Note 4) Four symmetrical patterns are considered corresponding to the sign of each address.

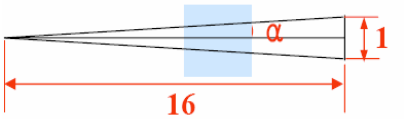
Note 5) ***The table of P(k)*** *(height of thread)*

- meter thread : P(k) =0.6495 x Pitch
- whitworth thread : P(k) =0.6403 x Pitch
- unified thread : P(k) =0.6134 x Pitch
- gas thread : P(k) =0.6403 x Pitch

ex) meter thread : Pitch 6.0 → 0.6495 x 6.0 = 3.897→ P3897

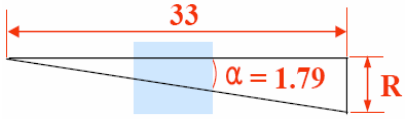
Note 6) R value in taper thread : In PT thread the taper is 1/ 16





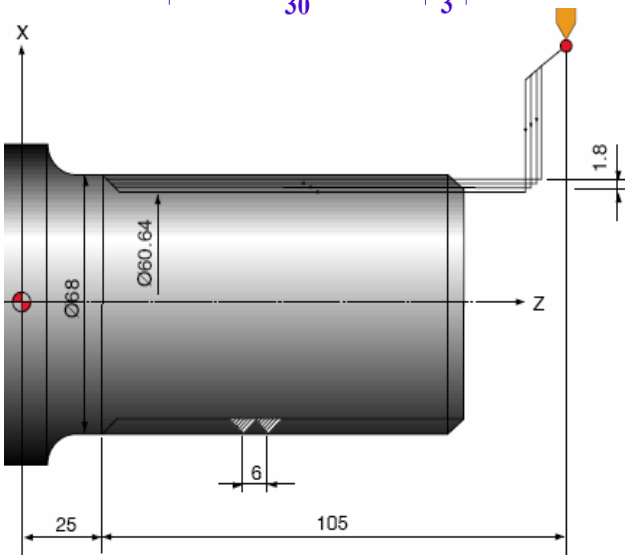
$\tan \alpha = 0.5 / 16$

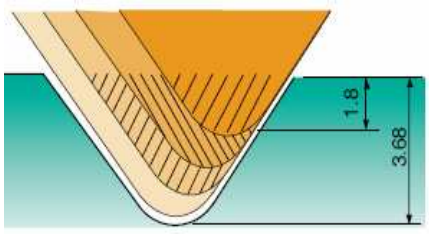
$\alpha = \tan^{-1}(0.5 / 16) = 1.79$



$\tan 1.79 = R / 33$

$R = \tan 1.79 \times 33 = 1.031$



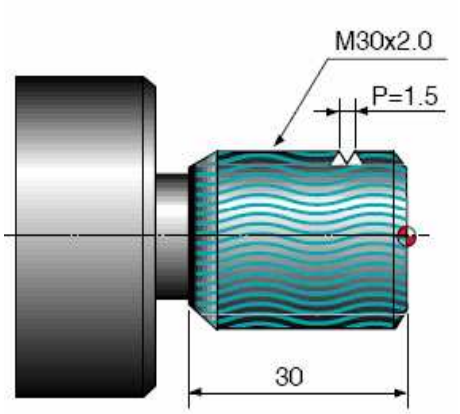


G00 X80.0 Z130.0 :

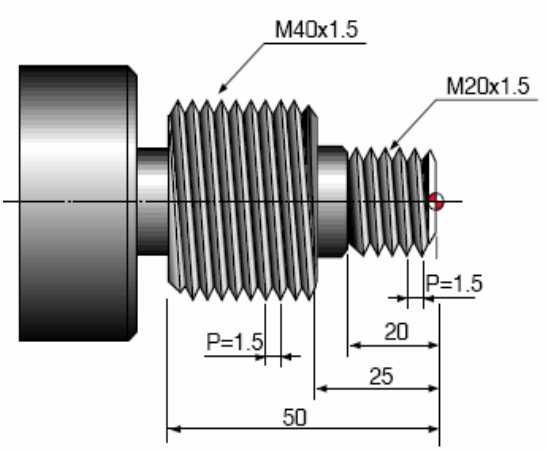
G76 P011060 Q100 R200 :

G76 X60.64 Z25.0 P3680 Q1800 F6.0 ;

(2) Program example




```
N10 G97 S1000 M03
    T0100
    G00 X50.0 Z5.0 T0101
    G76 P021060 Q100 R100
    G76 X28.2 Z-32.0 P900 Q500 F1.5
    G00 X200.0 Z200.0 T0100
    M30
```




```
G97 S800 M03
T0300
G00 X30.0 Z5.0 T0303
G76 P021060 Q100 R100
G76 X18.2 Z-20.0 P900 Q500 F1.5
G00 X50.0 Z-20.0
G76 P021060 Q100 R100
                    Omissible
G76 X38.2 Z-52.0 P900 Q500 F1.5
G00 X200.0 Z200.0 T0300
M30
```

6.3 Single block during a canned cycle

In a canned cycle, the single block stop points are as follows.

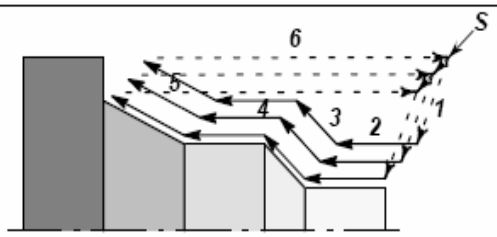
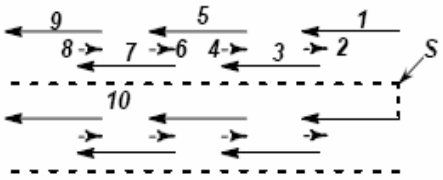
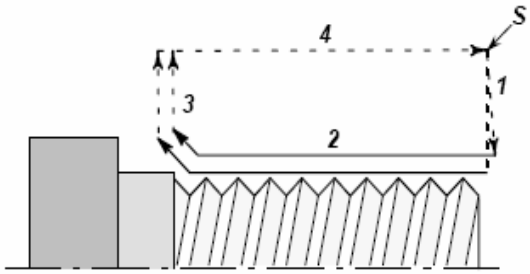
S : Single block — Rapid traverse — Cutting feed

	Tool path		Explanation
☆G90 (Outer/inner turning cycle)	Straight cutting cycle	Taper cutting cycle	Tool path 1 to 4 is assumed as one cycle. After 4 is finished, a stop is made.
☆G92 (Threading cycle)	Straight threading cycle	Taper threading cycle	Tool path 1 to 4 is assumed as one cycle. After 4 is finished, a stop is made.
☆G94 (End surface turning cycle)	Straight end surface cutting cycle	Taper end surface cutting cycle	Tool path 1 to 4 is assumed as one cycle. After 4 is finished, a stop is made.
☆G70 (Finishing cycle)			Tool path 1 to 7 is assumed as one cycle. After 7 is finished, a stop is made.
☆G71 (Outer surface rough machining cycle) G72 (End surface rough machining cycle)	This figure shows the case for G71. G72 is the same.		Each tool path 1 to 4,5 to 8,9 to 12, 13 to 16 and 17 to 20 is assumed as one cycle. After each cycle is finished, a stop is made.

S : Single–block stop

Rapid traverse

Cutting feed

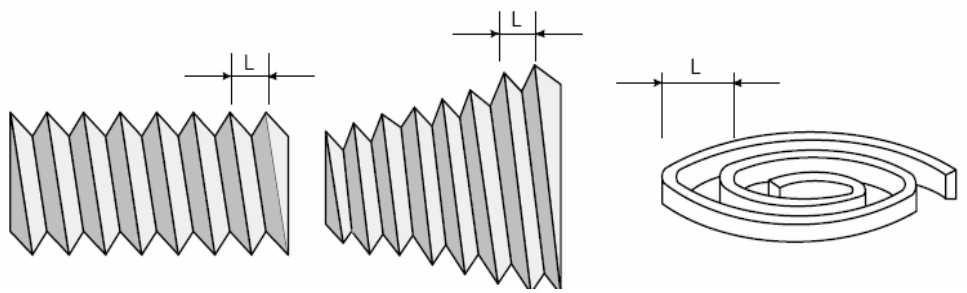
	Tool path	Explanation
☆G73 (Closed–loop cutting cycle)		Tool path 1 to 6 is assumed as one cycle. After 10 is finished, a stop is made.
☆G74 (End surface cutting–off cycle) G75 (Outer/inner surface cutting–off cycle)	 This figure shows the case for G74. G75 is the same.	Tool path 1 to 10 is assumed as one cycle. After 10 is finished, a stop is made.
☆G76 (Multiple repetitive threading cycle		Tool path 1 to 4 is assume as one cycle. After 4 is finished, a stop is made.

7. Thread cycles

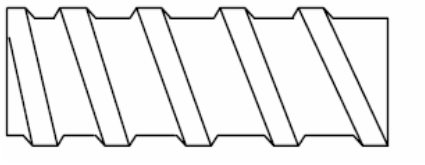
7. Thread cycles

7.1 Thread cycles

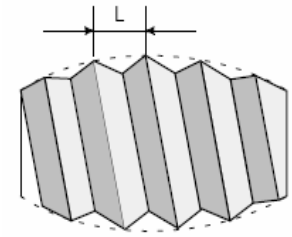
(1) Constant lead thread (G32,G76,G92)



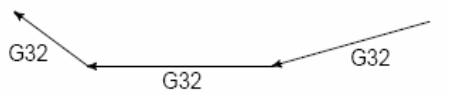
(2) Variable-lead cycle (G34)



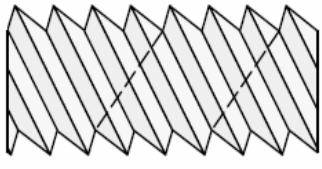
(3) Circular thread (G35,G36)



(4) Continuous thread (G32)



(5) Multiple thread (G32,G34,G76,G92)



7.1.1 Constant lead thread (G32)

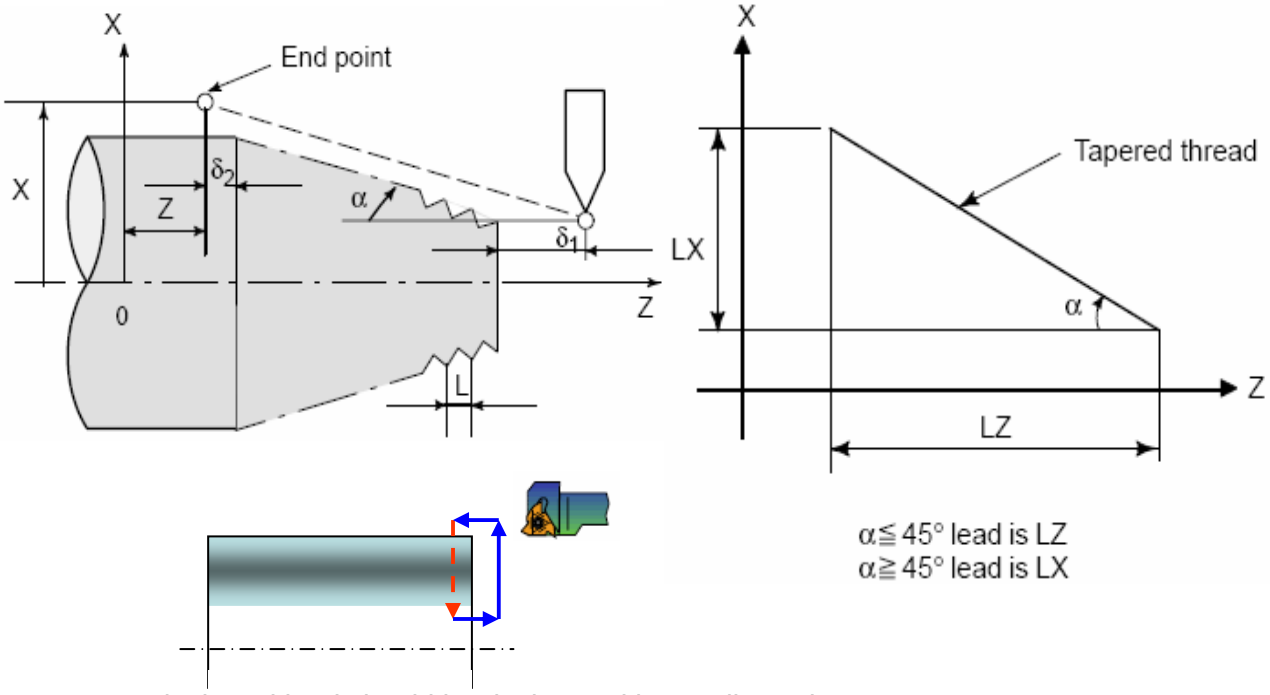
Tapered screws and scroll threads in addition to equal lead straight threads can be cut by using a G32 command.

The spindle speed is read from the position coder on the spindle in real time and converted to a cutting feedrate for-feed-per minute mode, which is used to move the tool.

(1) Command format

G32 X(U)_ Z(W)_ F_ ;

X(U) : The end point of X of threading operation
Z(W) : The end point of Z of threading operation
F : Thread lead of long axis
* Straight thread : G32 Z_ F_ ;
Face thread : G32 X_ F_ ;



Note 1) Thread lead should be designated in a radius value.

Note 2) In the G32 mode, thread cutting pattern is not repeated.

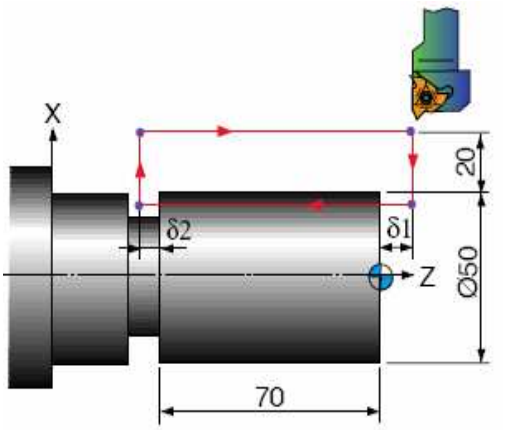
Note 3) For the thread lead, designate the lead along the Z axis if the thread tapered angle is less than 45˚. If the tapered angle is 45˚ to 90˚, designate the lead along the X-axis.

Note 4) For cutting TPI cutting, calculate the pitch between the threads in “mm” in units of 1/10000 mm and designate the result of calculation following address F.

To cut 14 TPI thread, designate as F1.8143 .

(2) example

< straight thread >



Lead of screw : 3mm

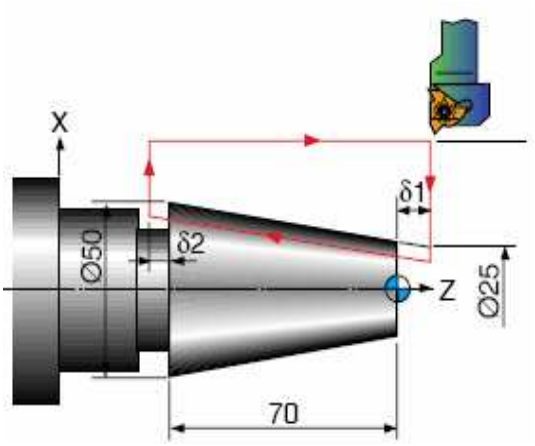
δ1 : 5mm

δ2 : 1.5mm

Depth of cut : 1mm(2cut two times)

```
G50 T0100 :
G97 S800 M03 :
G00 X90.0 Z5.0 T0101 M8 :
    X48.0 :
G32 Z-71.5 F3.0 :
G00 X90.0 :
    Z5.0 :
    X46.0 :
G32 Z-71.5 :
G00 X90.0 :
    Z5.0
    X150.0 Z150.0 T0100 :
M30 :
```

< Taper thread >



Lead of screw : 3mm

δ1 : 5mm

δ2 : 1.5mm

Depth of cut : 1mm(2cut two times)

```
G50 S800 T0100 :
G97 S800 M03 :
G00 X90.0 Z5.0 T0101 :
    X22.026 :
G32 X49.562 Z-71.5 F3.0 :
G00 X90.0 :
    Z5.0 :
    X21.052 :
G32 X48.588 Z-71.5 :
G00 X90.0 :
    Z5.0 :
    X150.0 Z150.0 T0100 :
M30 :
```

7.1.2 Continuous thread (G32)

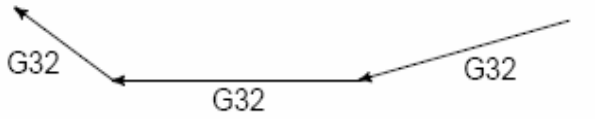
This function for continuous thread cutting is such that fractional pulse output a joint between move blocks are overlapped with the next move for pulse processing and output. (block overlap)

Therefore, discontinuous machining sections caused by the interruption of move during continuous block machining are eliminated, thus making it possible to continuously direct the block for thread cutting instructions.

(1) Command format

```
G32 X1(U1)_  Z1(W1)_  F_   ;
G32 ----   ;
G32 Xn(Un)_ Zn(Wn)_ F_     ;
```

X1 ~ Xn (U1 ~ Un) : The end point of X of each threading operation
Z1 ~ Zn (W1 ~Wn) : The end point of Z of each threading operation
F : Thread lead of long axis



Note 1) Block overlap is effective even for G01 command, producing a more excellent finishing surface.

Note 2) When extreme micro blocks continue, no block overlap may function.

(2) example

```
G40G30U0W0
T0404M42
G97S130M3
G0X6.2Z1.01M8
# 1=0
N52G0U-.0082W0.0005
G32U-.813W-1.5F.2
G32U.2748W-4.398F.2
G0U.5382
W5.898
# 1=[ # 1+1 ]
IF [ # 1LT15 ] GOTO52
# 1=0
N47G0U0
G32U-.813W-1.5F.2
G32U.2748W-4.398F.2
G0U.5382
W5.898
# 1=[ # 1+1 ]
IF [ # 1LT2 ] GOTO47
G30U0W0M9
M30
```

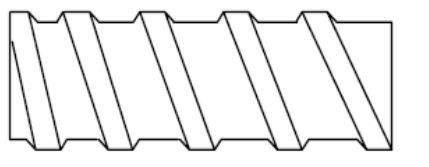
7.1.3 Variable-lead thread (G34)

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

(1) Command format

G34 X(U)_ Z(W)_ F_ K_ ;

X(U) : The end point of X of threading operation
Z(W) : The end point of Z of threading operation
F : lead in longitudinal axis direction at the start point
K : increment and decrement of lead per spindle revolution



Note 1) Address other than K are the same as in straight / taper thread cutting with G32.
Note 2) the “Thread cutting cycle Retract” is not effective for G34.

(2) example

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

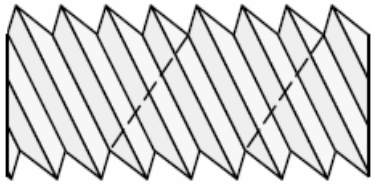
G34 Z-72.0 F8.0 K0.3 ;

7.1.4 Multiple thread cutting (G32,G34,G76,G92)

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

(1) Command format

Type -1 Using the start angle (Q address)	**Type -2** Shift the start point of every angle except putting start angle
G32 X(U)_ Z(W)_ F_ *Q_*	**G00 *X1_ Z1_*** **G32 X(U)_ Z(W)_ F_**
G34 X(U)_ Z(W)_ F_ K_ *Q_*	**G00 *X1_ Z1_*** **G34 X(U)_ Z(W)_ F_ K_**
G76 *X(U)_ Z(W)_ K_ D_ F_ A_ P2* **→ *use FS15 tape format***	**G00 *X1_ Z1_*** **G76 P_ Q_ R_** **G76 X(U)_ Z(W)_ R_ P_ Q_ F_**
G92 X(U)_ Z(W)_ R_ F_ *Q_*	**G00 *X1_ Z1_*** **G92 X(U)_ Z(W)_ R_ F_**
Q : threading start angle	X1 _ Z1_ : start point of thread



Note 1) The start angle is not a modal value. It must be specified each time it is used. If a value is not specified, 0 is assumed.

Note 2) The start angle (Q) increment is 0.001 degrees. Note that no decimal point can be specified. 180˚ → Q180000

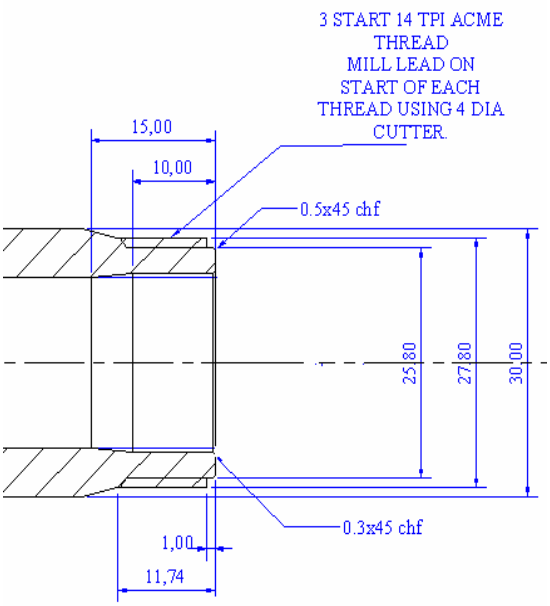
Note 3) A start angle (Q) of between 0 and 360000 can be specified. If a value greater than 360000 (360˚) is specified, it is rounded down to 360000.

Note 4) For the G76 multiple-thread cutting command, always use the FS15 tape format.

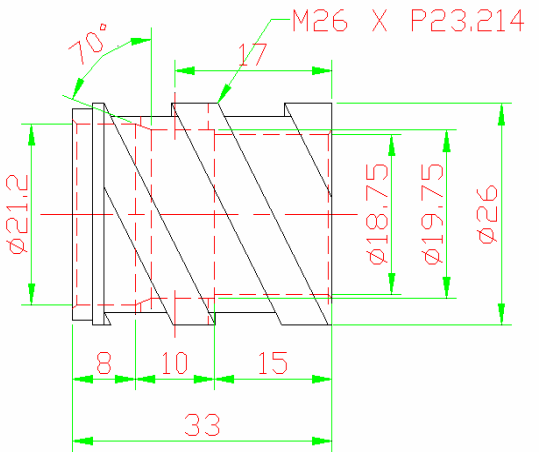
(2) Example-1 (G32)

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

(3) Example-2 (G76)

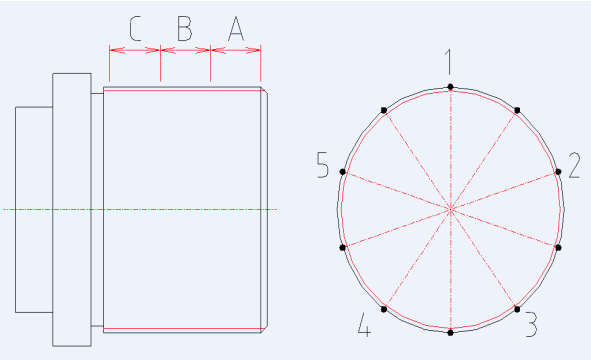



```
G28 U0 W0 V0
G54G40G80G99
G50S3000
G97S1000M3T0909
G0Z17.442M8
X29.
G76P010200Q100R.05
G76X25.8Z-11.5P1000Q400R0F6.35
G0Z19.558
G76P010200Q100R.05
G76X25.8Z-11.5P1000Q400R0F6.35
G0Z21.674
G76P010200Q100R.05
G76X25.8Z-11.5P1000Q400R0F6.35
G0X200.Z10.
G28U0 W0 V0 M09
M30
```




```
G28 U0 W0
G54G40G80G99
M85
G97S100M3
G00Z10.0T1111M8
G0X35.0
G76P020000Q50R30
G76X29.4Z-34.0P1300Q50F23.214
G00Z17.738
G76P020000Q50R30
G76X29.4Z-34.0P1300Q50F23.214
G00Z25.476
G76P020000Q50R30
G76X29.4Z-34.0P1300Q50F23.214
G00X40.0Z25.476
G28U0 W0 M9
M30
```

(4) Example-3 (G92)




```
G28U0W0
T0303
G97S80M3
G0X188.0Z33.0M8
G92X185.0Z-140.0F25.0
G0Z35.5
G92X185.0Z-140.0F25.0
G0Z38.0
G92X185.0Z-140.0F25.0
G0Z40.5
G92X185.0Z-140.0F25.0
G0Z43.0
G92X185.0Z-140.0F25.0
G0Z45.5
G92X185.0Z-140.0F25.0
G0Z48.0
G92X185.0Z-140.0F25.0
G0Z50.5
G92X185.0Z-140.0F25.0
G0Z53.0
G92X185.0Z-140.0F25.0
G0Z55.5
G92X185.0Z-140.0F25.0
M1
```

```
N20G0Z33.0
G92X184.95Z-140.0F25.0
G0Z35.5
G92X184.95Z-140.0F25.0
G0Z38.0
G92X184.95Z-140.0F25.0
G0Z40.5
G92X184.95Z-140.0F25.0
G0Z43.0
G92X184.95Z-140.0F25.0
G0Z45.5
G92X184.95Z-140.0F25.0
G0Z48.0
G92X184.95Z-140.0F25.0
G0Z50.5
G92X184.95Z-140.0F25.0
G0Z53.0
G92X184.95Z-140.0F25.0
G0Z55.5
G92X184.95Z-140.0F25.0
M1
N30G0Z33.0
G92X184.90Z-140.0F25.0
G0Z35.5
G92X184.90Z-140.0F25.0
G0Z38.0
G92X184.90Z-140.0F25.0
G0Z40.5
G92X184.90Z-140.0F25.0
G0Z43.0
G92X184.90Z-140.0F25.0
G0Z45.5
G92X184.90Z-140.0F25.0
G0Z48.0
G92X184.90Z-140.0F25.0
G0Z50.5
G92X184.90Z-140.0F25.0
G0Z53.0
G92X184.90Z-140.0F25.0
G0Z55.5
G92X184.90Z-140.0F25.0
G28 U0 W0
M30
```

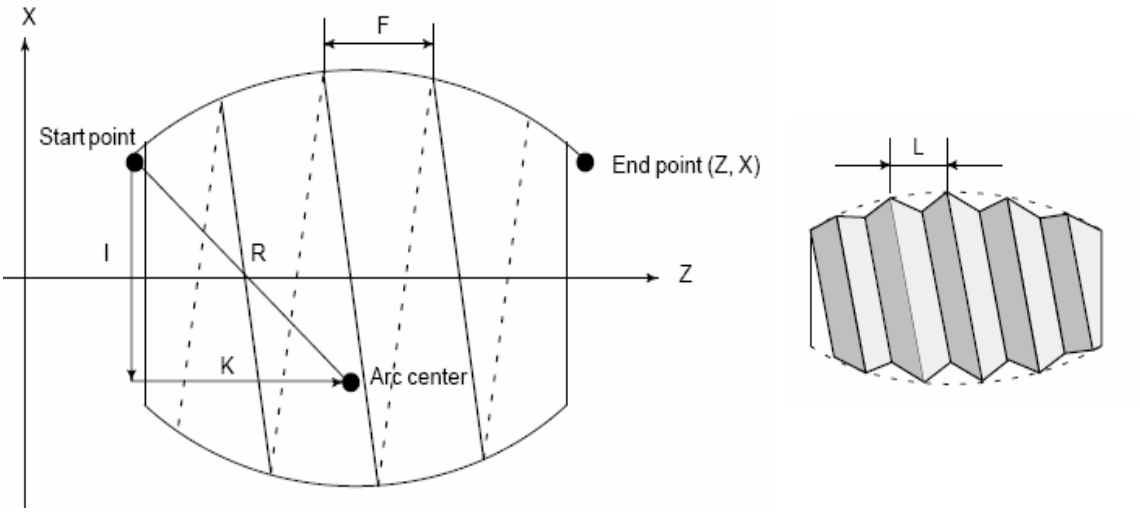
7.1.5 Circular threading (G35,G36)

Using the G35 and G36 commands, a circular thread, having the specified lead in the direction of the major axis, can be machined.

(1) Command format

G35 X(U)_ Z(W)_ I_ K_ F_ Q_ ;
or
G35 X(U)_ Z(W)_ R___ F_ Q_ ;

G35 : clockwise circular threading command
G36 : counterclockwise circular threading command
X (U) : specify the arc end point of X (in the same way as for G02,G03)
Z (W) : specify the arc end point of Z (in the same way as for G02,G03)
I,K : specify the arc center relative to the start point, using relative coordinates (in the same way as for G02,G03)
R : specify the arc radius
F : specify the lead in the direction of major axis
Q : specify the shift of the threading start angle (0˚~360˚ , Q180000=180˚)



Note 1) If R is specified with I and K, only R is effective.

Note 2) If an additional axis other than the X- and Z –axes is provided, circular threading can be specified for a plane other than the ZX plane. The method of specification is the same as that for G02 and G03.

Note 3) Parameter 3405#3 = 1

G35/G36 → CW / CCW circular threading

G37 → automatic tool compensation Z

G37.1/G37.2 → automatic tool compensation X / Z

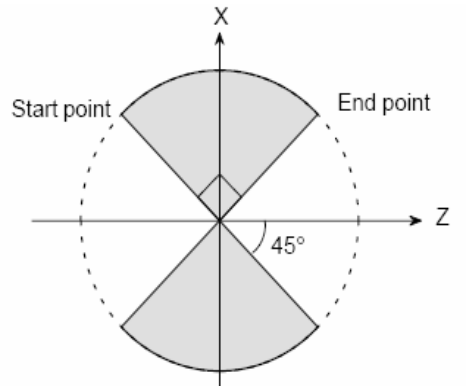
Note 4) Parameter 3405#3=0

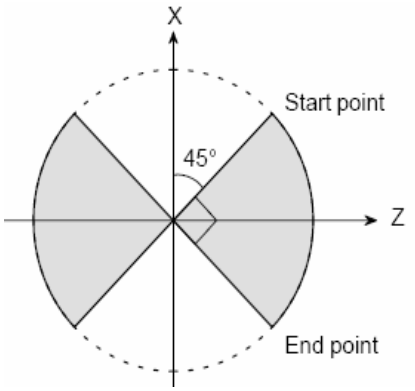
G36 → automatic tool compensation X

Note 5) Lead angle = Lead / (2πR) = Lead / (πd)

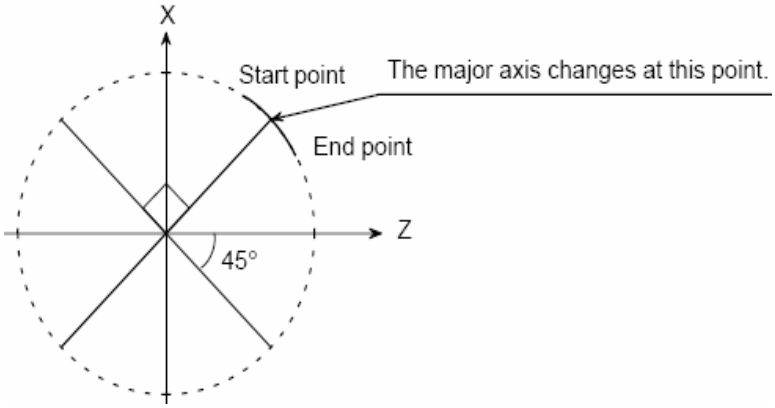
(2) Limitations

An arc must be specified such that it falls within a range in which the major axis of the arc is always the Z–axis or always the X–axis, as shown in Fig. (a) and (b). If the arc includes a point at which the major axis changes from the X–axis to Z–axis, or vice versa, as shown in Fig (c), P/S alarm 5058 is issued.





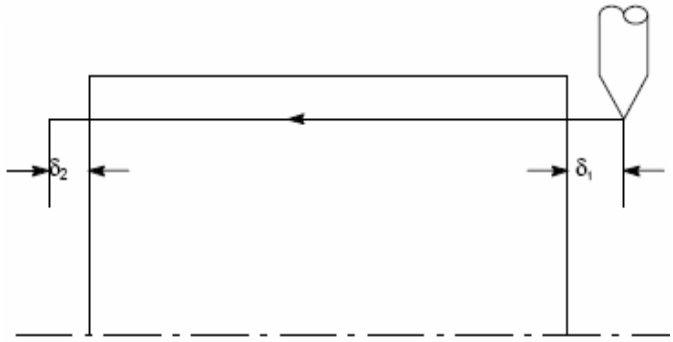
(a) Range in which the Z–axis is the major axis **(b) Range in which the X–axis is the major axis**



(c) Example of arc specification which causes an alarm

7.2 Incorrect threaded length

The leads of a thread are generally incorrect in δ_1 and δ_2, as shown in Fig. D.1 (a), due to automatic acceleration and deceleration.
Thus distance allowances must be made to the extent of δ_1 and δ_2 in the program.



(a) Incorrect thread position

(1) How to determine δ_2

$$\delta_2 = \frac{LR}{1800 *} \text{ (mm)}$$

R : Spindle speed (min^{-1})
L : Thread lead (mm)

When time constant T of the servo system is 0.033 s.

(2) How to determine δ_1

$$\delta_1 = \frac{LR}{1800 *}(-1-lna) \text{ (mm)}$$
$$= \delta_2(-1-lna) \text{ (mm)}$$

R : Spindle speed (min^{-1})
L : Thread lead (mm)

When time constant T of the servo system is 0.033 s.

(3) Example

$$\delta 1 = \frac{3.6 \times L \times n}{1800}$$

$$\delta 2 = \frac{L \times n}{1800}$$

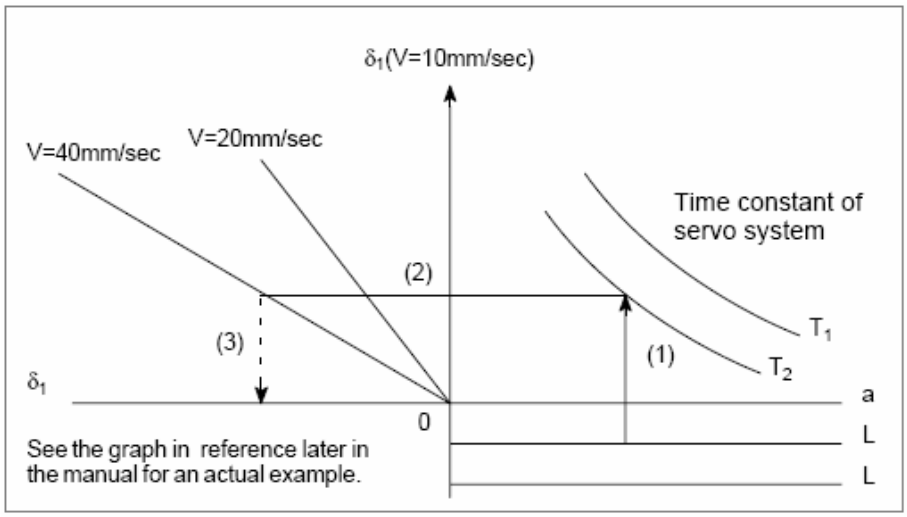
(a) Tolerance of thread	(c) Constant
0.005	4.298
0.010	3.605
0.015	3.200
0.020	2.912

L = Lead of thread
n = Rotating time of main spindle

V =100m/min
L = 1.5
D = 50mm
a = 0.01
n = (V x 1000) / (D x π)
= (100X1000) / (50 x π)
= 640rpm
δ1 = a x δ2
= 3.605 x 1.5 x 640/1800
δ2 = 1.5 x 640/1800

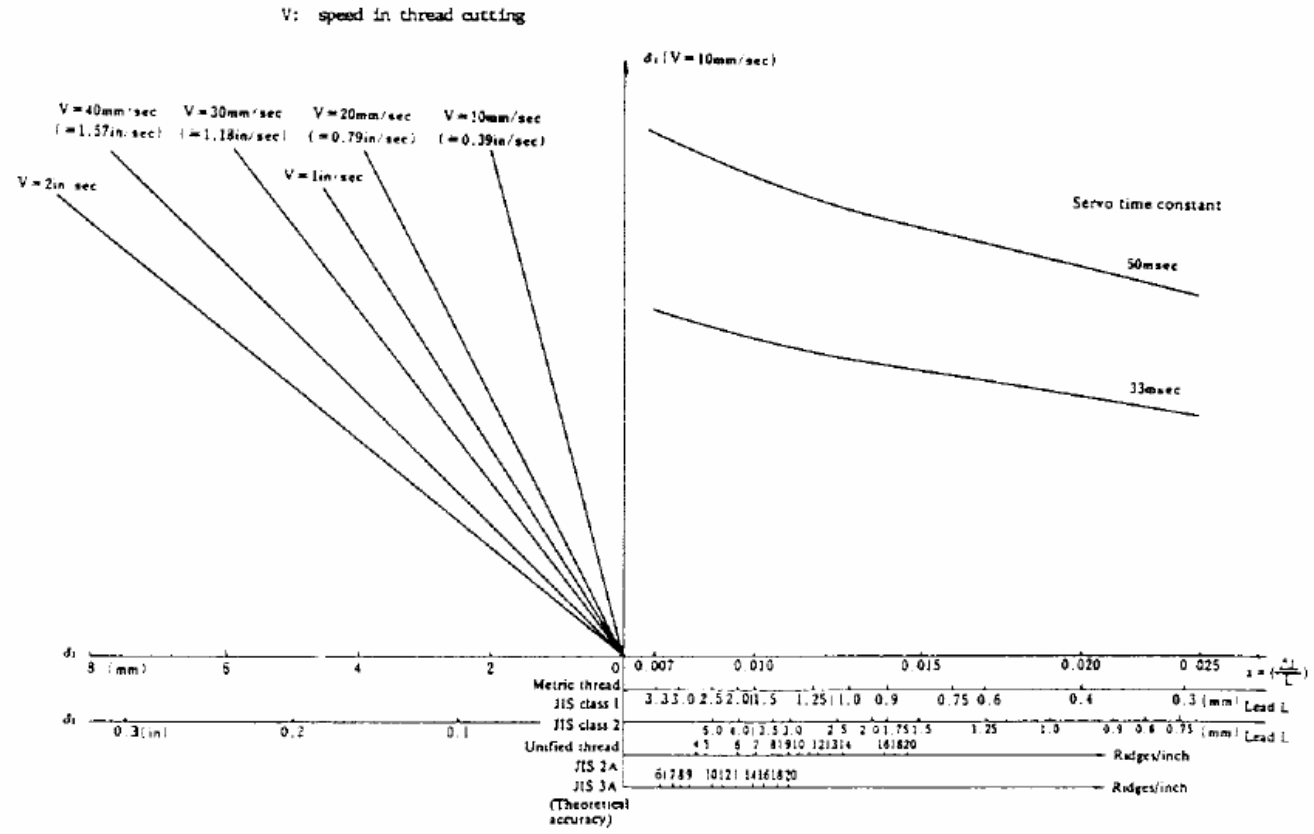
(4) How to use monograph

First specify the class and the lead of a thread. The thread accuracy, α, will be obtained at (1), and depending on the time constant of cutting feed acceleration/ deceleration, the δ_1 value when V = 10mm / s will be obtained at (2). Then, depending on the speed of thread cutting, δ_1 for speed other than 10mm/ s can be obtained at (3).



Nomograph

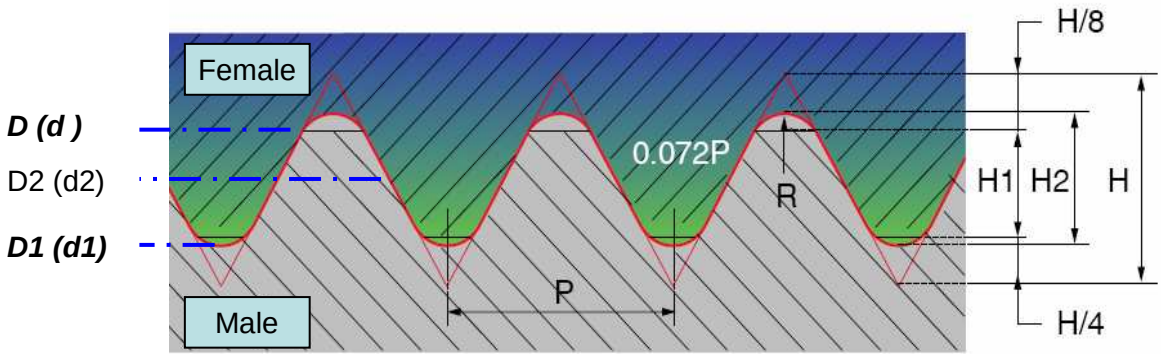
Notes)
The equations for δ1, and δ2 are for when the acceleration / de-celeration time constant for cutting feed is 0.



Nomograph for obtaining approach distance δ_1

7. Thread cycles

7.3 Cutting depth of thread process (Metric 60˚)



< D=d , D2=d2, D1=d1 >

H = 0.86603P

H1 = 0.54127P = 5/8H

D2 = D-0.64952P

D1 = D-1.08253P

▶ Example M35xP1.0

→ D(d) =35

→ ap = R + H2

= 0.07 + (0.25+0.2+0.1+0.05)

= 0.67

PITCH	P1.0	1.0	1.25	1.5	1.75	2.0	2.5	3.0	3.5	4.0	4.5	5.0
Total infeed		***0.67***	***0.8***	***0.94***	***1.14***	***1.28***	***1.58***	***1.89***	***2.20***	***2.50***	***2.80***	***3.12***
CUTTING DEPT	H2	0.6	0.74	0.89	1.05	1.19	1.49	1.79	2.08	2.38	2.68	2.98
CORNER ROUND	R	0.07	0.09	0.11	0.13	0.14	0.18	0.22	0.25	0.29	0.32	0.36
SCREW CUTTING NUMBER OF TIMES	1	0.25	0.30	0.30	0.30	0.30	0.30	0.35	0.35	0.35	0.40	0.45
	2	0.20	0.20	0.20	0.25	0.25	0.28	0.30	0.35	0.35	0.35	0.35
	3	0.10	0.11	0.14	0.16	0.20	0.24	0.26	0.30	0.30	0.30	0.32
	4	0.05	0.08	0.12	0.12	0.14	0.20	0.22	0.25	0.26	0.28	0.30
	5		0.05	0.08	0.10	0.11	0.15	0.18	0.20	0.23	0.25	0.25
	6			0.05	0.07	0.08	0.11	0.13	0.15	0.20	0.22	0.25
	7				0.05	0.06	0.09	0.10	0.12	0.17	0.20	0.20
	8					0.05	0.07	0.08	0.10	0.14	0.15	0.17
	9						0.05	0.07	0.08	0.10	0.12	0.15
	10							0.05	0.05	0.10	0.10	0.15
	11							0.05	0.05	0.08	0.08	0.10
	12								0.05	0.05	0.08	0.10
	13									0.05	0.05	0.08
	14										0.05	0.06
	15										0.05	0.06

8. Nose radius compensation functions

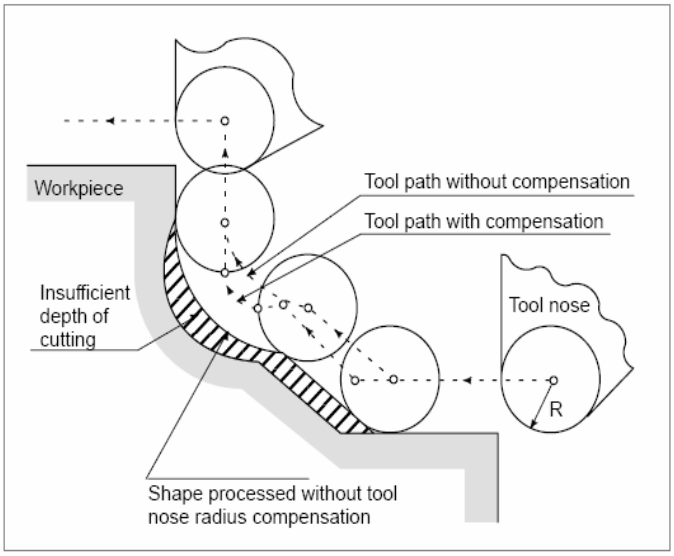
8.1 Tool nose radius offset

8.1.1 Overview of tool nose radius compensation

Although a cutting tool has rounded nose, a program is written assuming an imaginary tool nose. This causes an error between the programmed tool paths and the actual tool paths when the programmed shape includes a circular arc or tapered portions.

The nose R compensation function automatically calculates the error and correct the tool paths so that they are generated as programmed.

(1) Tool path of tool nose radius compensation

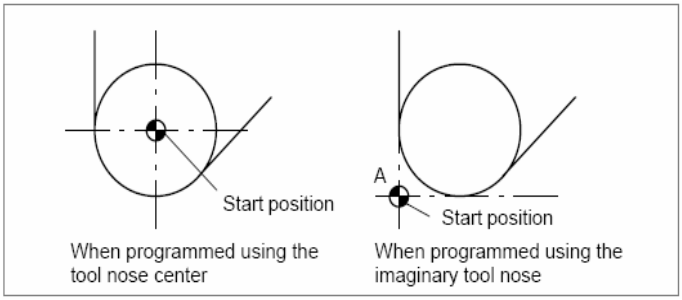


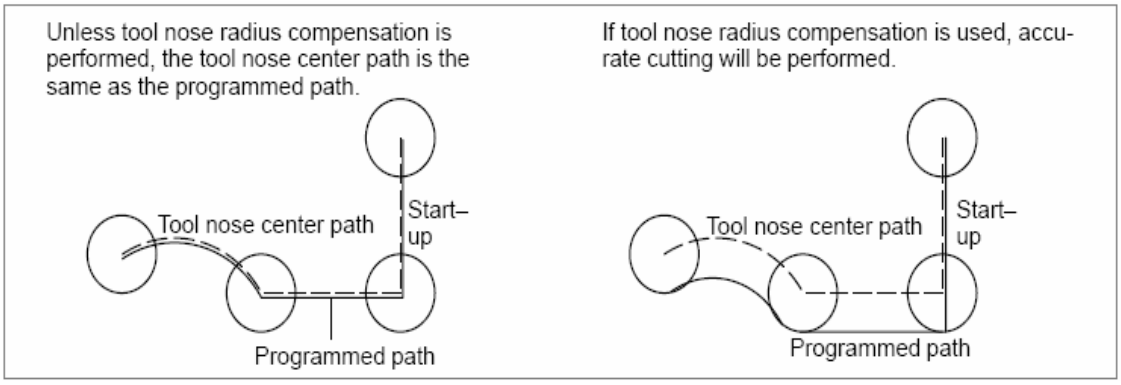
(2) Imaginary tool nose

The tool nose at position A in following figure does not actually exist. The imaginary tool nose is required because it is usually more difficult to set the actual tool nose radius center to the start position than the imaginary tool nose.

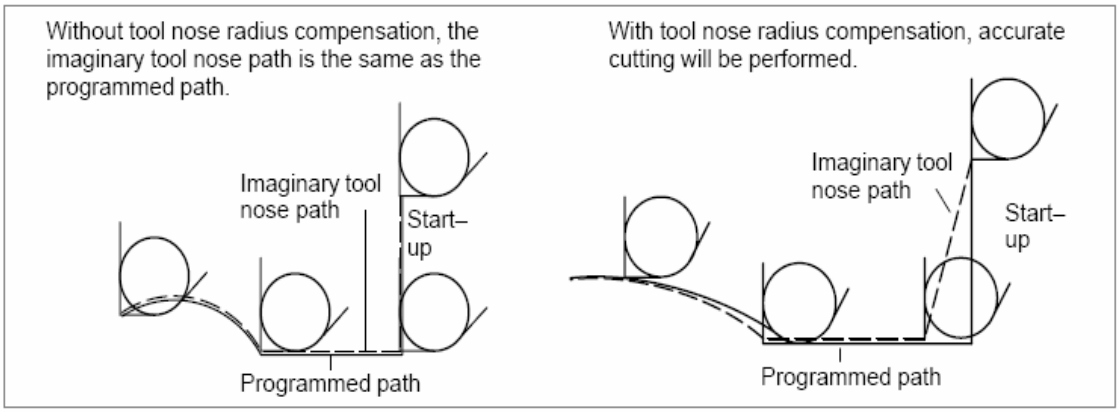
Also when imaginary tool nose is used, the tool nose radius need not to be considered in programming.

The position relationship when the tool is set to the start position is shown in the following figure.





< Tool path when programming using ***the tool nose center*** >



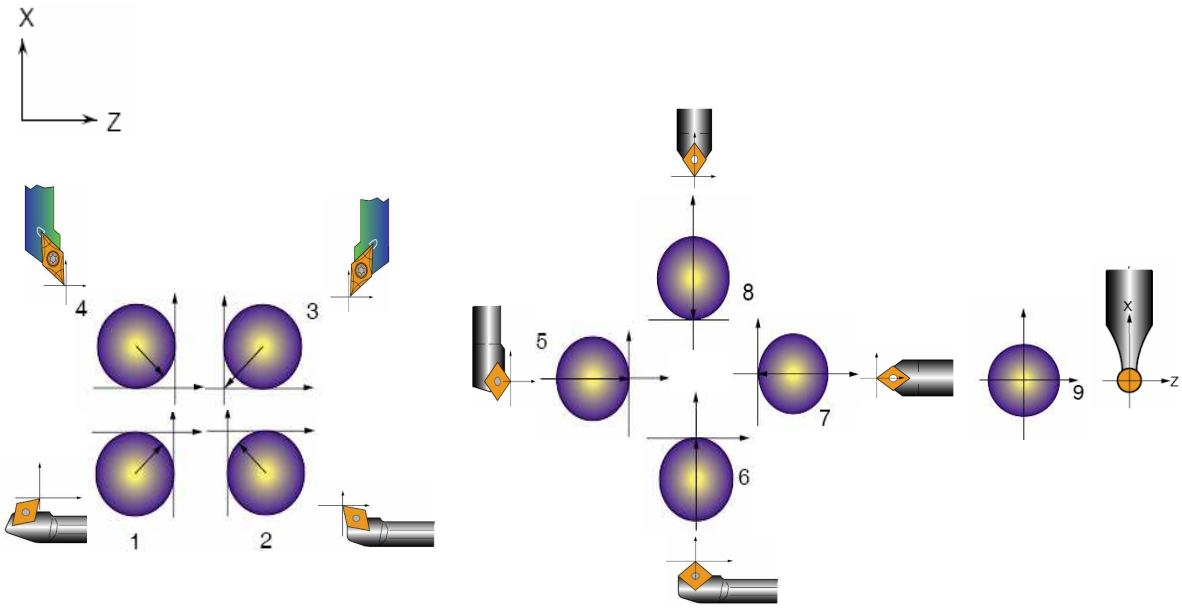
< Tool path when programming using ***the imaginary tool nose*** >

8.1.2 The direction of imaginary tool nose.

The direction of the imaginary tool nose viewed from the tool nose center is determined by the direction of the tool during cutting, so it must be set in advance as well as offset values.

The direction of the imaginary tool nose can be selected from the eight specifications together with their corresponding codes.

The following apply when the tool geometry offset and tool wear offset option are selected.



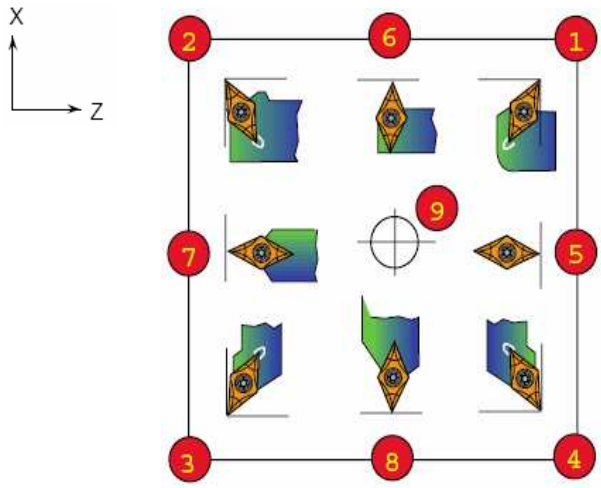
Imaginary tool nose numbers 0 and 9 are used when the tool nose center coincides with the start position.

Virtual tool nose directions 1 to 8 can be used only in the G18(Z-X) plane.

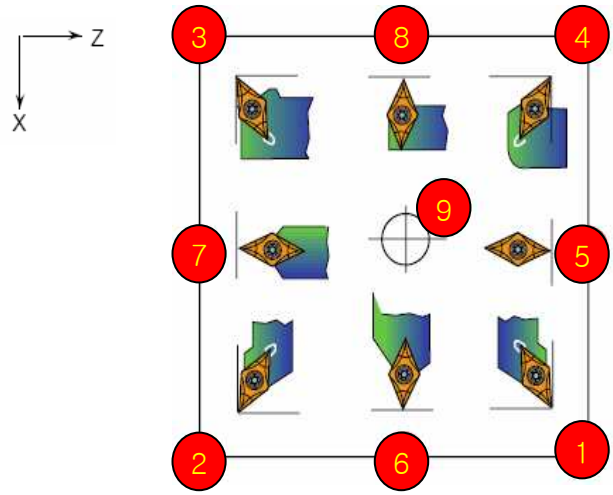
For virtual tool nose 0 to 9, compensation is applied in both the G17 and G19 planes.

OFFSET/GEOMETRY			01000	N0000
NO.	X	Z	R	T
01	0.000	0.000	0.8	3
02	0.000	0.000	0.000	0

1) The direction of imaginary tool nose In upper turret



2) The direction of imaginary tool nose In lower turret



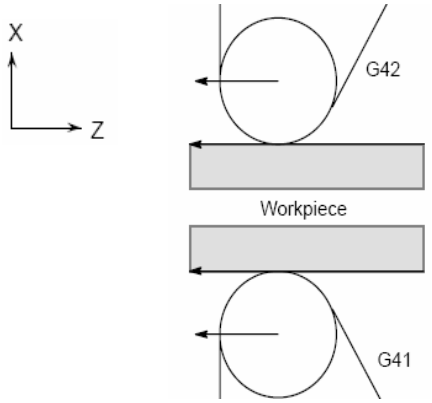
8.1.3 Work position and Move command

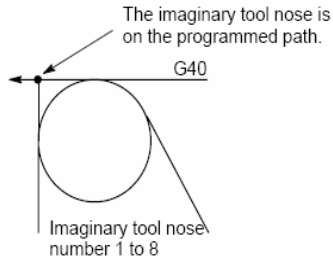
In tool nose radius compensation , the position of the work-piece with respect to the tool must be specified.

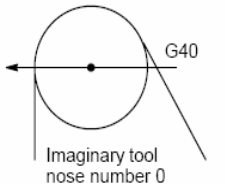
G code	Workpiece position	Tool path
G40	(Cancel)	Moving along the programmed path
G41	Right side	Moving on the left side the programmed path
G42	Left side	Moving on the right side the programmed path

1) The direction of imaginary tool nose In upper turret

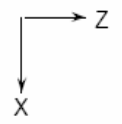
The tool offset to the opposite side of the work-piece.

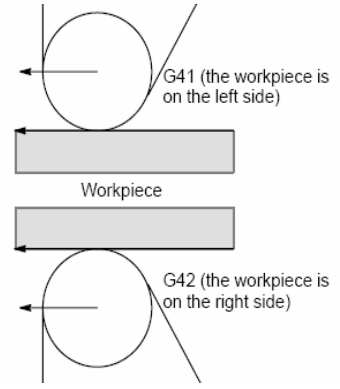






2) The direction of imaginary tool nose In lower turret





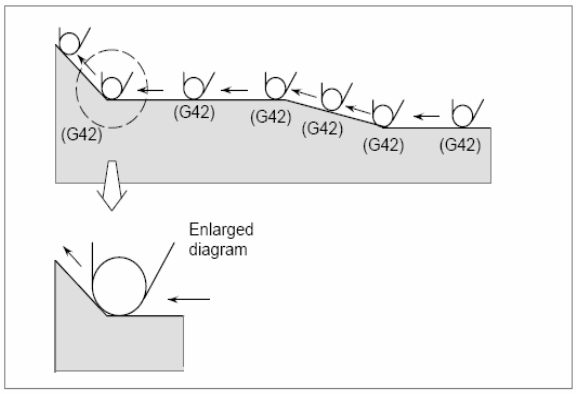
Notes) G40,G41 and G42 are modal.

Don't specify G41 while in the G41 mode. If you do, compensation will not work properly. (The G42 is same.)

G41 or G42 mode blocks in which G41 or G42 are not specified are expressed by (G41) or (G42) respectively.

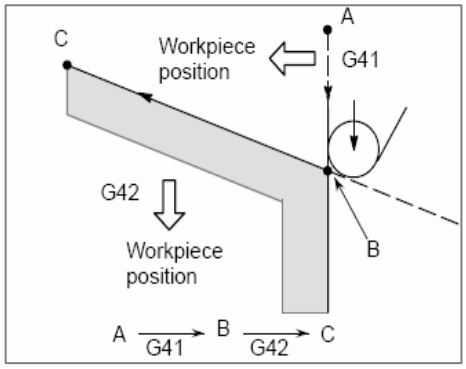
3) Tool movement when the work-piece position does not change

When the tool is moving, the tool nose maintains contact with the work-piece.



4) Tool movement when the work-piece position changes

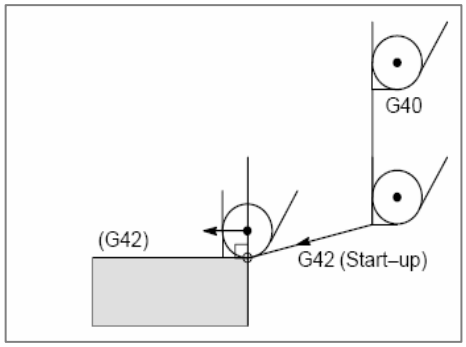
The work-piece position against the tool changes at the corner of the programmed path as shown in the following figure.



Although the work-piece does not exist on the right side of the programmed path in this case, the existence of the work-piece is assumed in the movement from A to B. the work-piece position must not be changed in the block nest to the start-up block. In this example, if the block specifying motion from A to B were the start-up block, the tool path would not be the same as the one shown.

5) Start-up

The block in which the mode changes to G41 or G42 from G40 is called the start-up block.



G40 ;

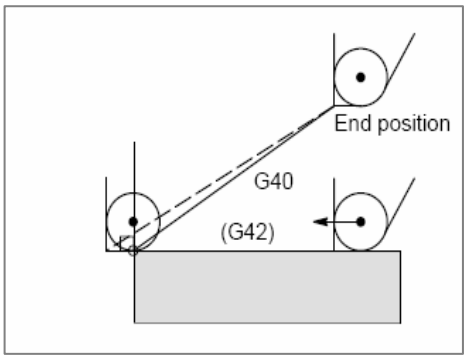
G41 ; (start-up block)

Transient tool movements for offset are performed in the start-up block.

In the block after the start-up block, the tool nose center is positioned vertically to the programmed path of that block at the start position.

6) Offset cancel

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



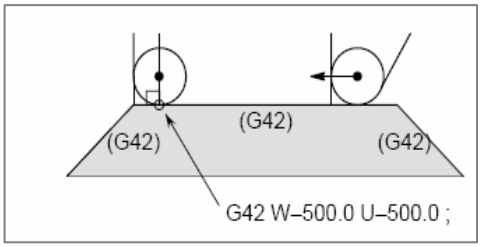
G41 ;

G42 ; (offset cancel block)

The tool nose center moves to a position vertical to the programmed path in the block before the cancel block. The tool is positioned at the end position in the offset cancel block (G40) as shown.

7) Specification of G41/G42 in G41/G42 mode

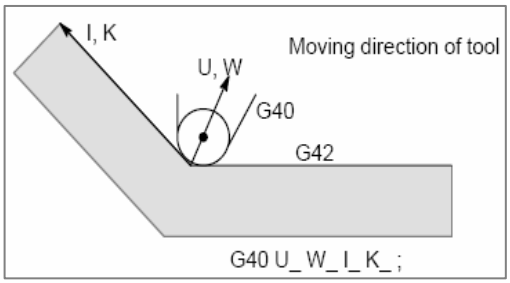
When is specified again in G41/G42 mode, the tool nose center is positioned vertical to the programmed path of the preceding block at the end of the preceding block.



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

8) Tool movement when the moving direction of the tool in a block which includes a G40 command is different from the direction of the work-piece.

When you wish to retract the tool in the direction specified by X(U) and Z(W) cancelling to the tool nose radius compensation at the end of machining the first block in the figure below, specify the following : **G40 X(U)_ Z(W)_ I_ K_ ;**



The work-piece position specified by address I and K is the same as that in the preceding block. If I and/or K is specified with G40 in the cancel mode, the I and/or K is ignored.

G40 X_Z_I_K_ : tool nose radius compensation

G40 G02 X_Z_I_K_: circular interpolation

G40 G01 X_ Z_ :

G40 G01 X_Z_I_K_; offset cancel mode
(I and K are ineffective.)

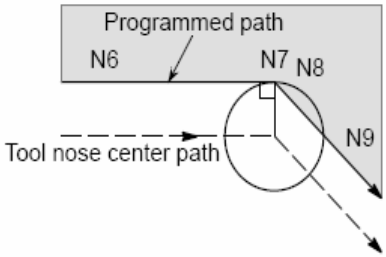
* I , K : radius value

8.1.4 Notes on tool nose radius compensation

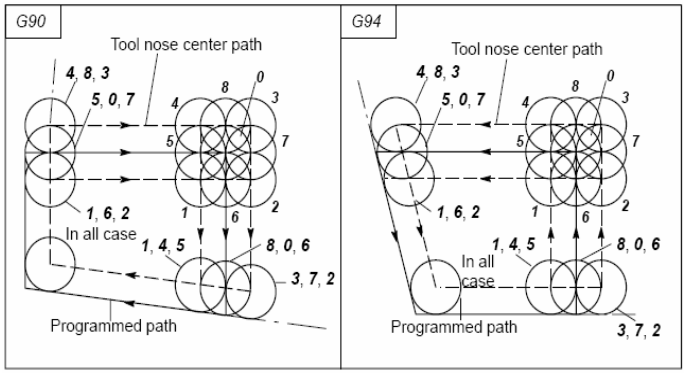
1) Tool movement when two or more blocks without a move command should not be programmed consecutively.

If two or more of the above blocks are specified consecutively, the tool nose center comes to a position vertical to the programmed path of the preceding block at the of the preceding block. However, if the no movement commands is, the above tool motion is attained only with one block.

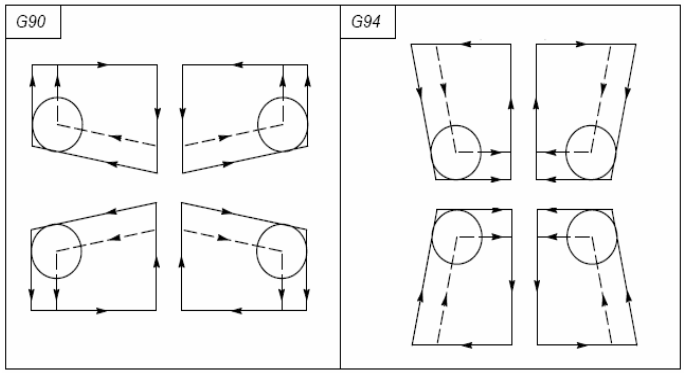
```
(G42 mode)
N6 W1000.0 ;
N7 S21 ;
N8 M04 ;
U9 U–1000.0 W1000.0 ;
```



2) Tool nose radius compensation with G90 or G94



< The tool nose center path is generally parallel to the program path. >

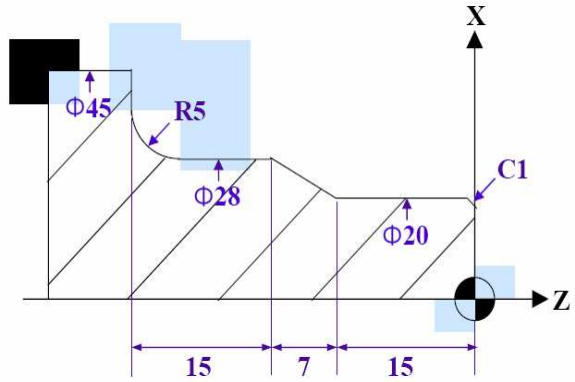


< The direction of the tool offset regardless of the G41/G42 mode >

3) Tool nose radius compensation with G71,G72,G73,G74,G75,G76

In the Multiple repetitive cycles, the cycle deviates by a tool nose radius compensation vector. During the cycle, no intersection calculation is performed.

4) Example before tool nose compensation and after one



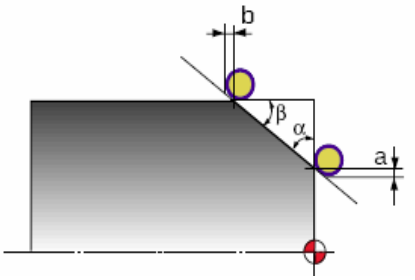
< Before >

```
O1001 ;
G50 S2000 ;
G96 S200 M03 ;
G00 X25. Z0 T0101 M08 ;
G01 X0 (Z0) F0.15 ;
G00 X16. Z1. ;
G01 X20. Z-1. F0.2 ;
    (X20.) Z-15. ;
     X28. Z-22. ;
    (X28.) Z-32. ;
G02 X38. Z-37. R5. ;
G01 X47. (Z-37.) ;
G00 X200. Z200. T0100 M09 ;
M30 ;
```

< After >

```
O1002 ;
G40 T0100 ;
G50 S2000 ;
G96 S200 M03 ;
G00 X25. Z0 T0101 M08 ;
G01 X-1.6 (Z0) F0.15 ;
G00 X16. Z3. ;
G42 (X16.) Z1. ;
G01 X20. Z-1. F0.2 ;
    (X20.) Z-15. ;
     X28. Z-22. ;
    (X28.) Z-32. ;
G02 X38. Z-37. R5. ;
G40 G01 X47. (Z-37.) ;
G00 X200. Z200. T0100 M09 ;
M30 ;
```

8.1.5 Calculation formula of bite nose



* Calculation formular of compensation volume

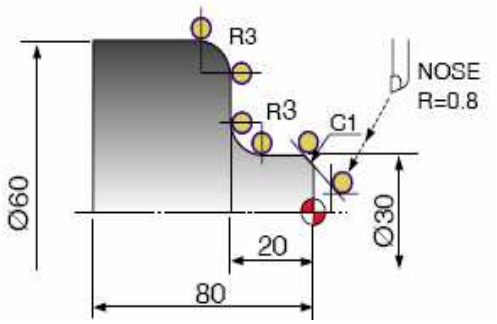
$a = r(1-\tan\frac{\alpha}{2})$

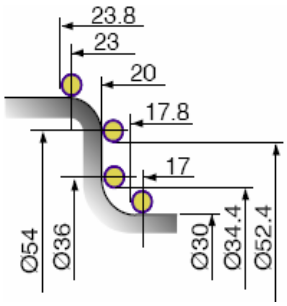
$b = r(1-\tan\frac{\beta}{2})$

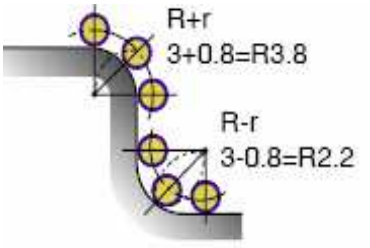
r = Rvalue of bite

Bite Nose	a	b
0.4	0.468	0.234
0.8	0.937	0.468

Example)







Concave R = R–r
Convex R = R+r
R : Circumference R
r : Bite r

```
N10 G50 S1500 T0100 :
    ≀
N20 G50 S2000 T0303 :
G96 S180 M03 :
G00 X35.0 Z5.0 M08 :
    Z0:
G01 X-1.6 F0.2 :
G00 X25.063 Z1.0 :
G01 X30.0 Z-1.468 F0.17 :
    Z-17.8 :
G02 X34.4 Z-20.0 R2.2 :
G01 X52.4 :
G03 X60.0 Z-23.8 R3.8 :
G01 Z-80.0 :
G00 X150.0 Z150.0 :
T0300 :
M30 :
```

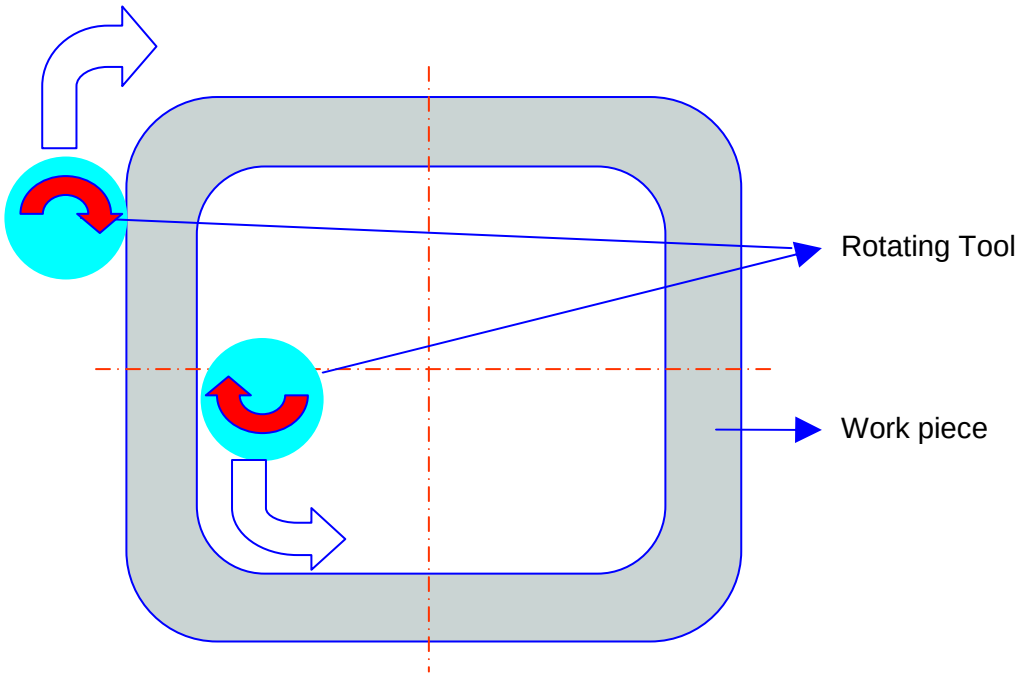
8.2 Cutter radius compensation

1) Down cutting direction in Milling

< Out-side milling in Down cutting >

G41 : Cutter radius compensation

G02 : Tool moving direction



< In-side milling in Down cutting >

G41 : Cutter radius compensation

G03 : Tool moving direction

9. C-axis Control Function

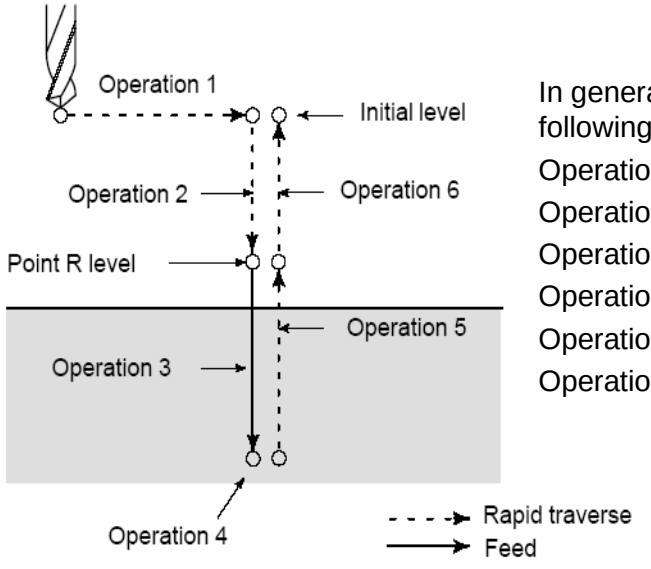
9.1 Canned cycle for hole machining (G80~G89)

The canned cycle for drilling simplifies the program normally by directing the machining operation commanded with a few blocks, using one block including G code.

This canned cycle conforms to JIS B 6314.

Following is the canned cycle table.

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



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

Operation 1 : Positioning of X(Z) and C axis

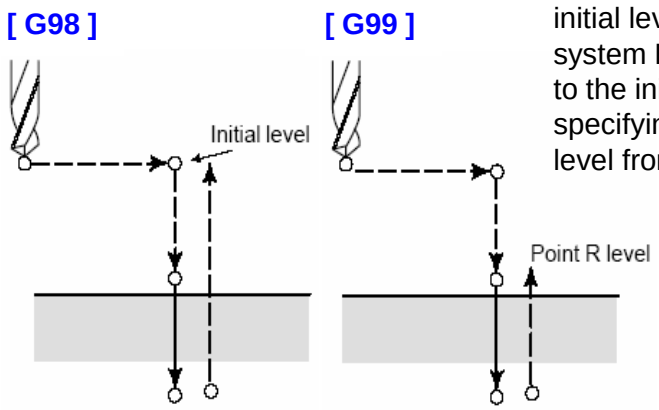
Operation 2 : Rapid traverse up to point R level

Operation 3 : Hole machining

Operation 4 : Operation to point R level

Operation 5 : Retraction to point R level

Operation 6 : Rapid traverse up to the initial point



In G code system A, the tool returns to the initial level from the bottom of a hole. In G code system B or C, specifying G98 returns the tool to the initial level from the bottom of a hole and specifying G99 returns the tool to the point-R level from the bottom of a hole.

9.1.1 Front Drilling Cycle(G83) / Side Drilling Cycle(G87)

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

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

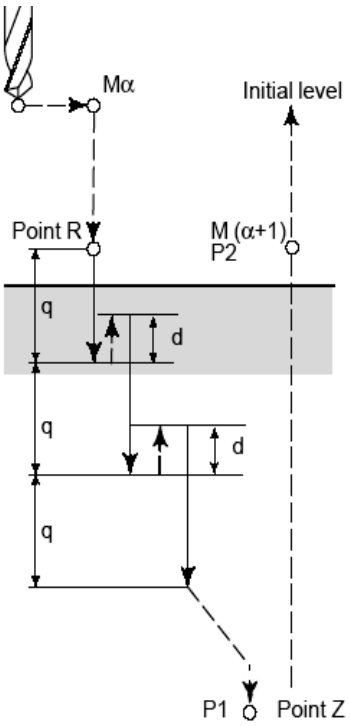
G83 X(U)_ C(H)_ Z(W)_ R_ Q_ P_ F_ K_ M_ ;

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

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

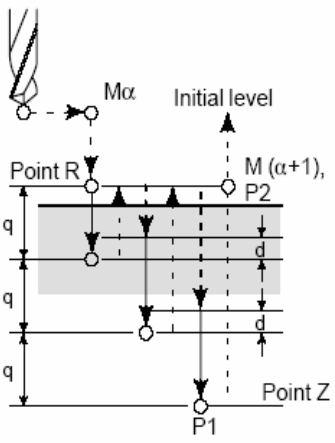
[High-speed peck drilling cycle]

Parameter No.5102#2 (RTR)=0



[Peck drilling cycle]

Parameter No.5102#2 (RTR)=1



Mα	:	M code for C–axis clamp
M(α+1)	:	M code for C–axis unclamp
P1	:	Dwell specified in the program
P2	:	Dwell specified in parameter No. 5111
d	:	Retraction distance specified in parameter No. 5114

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

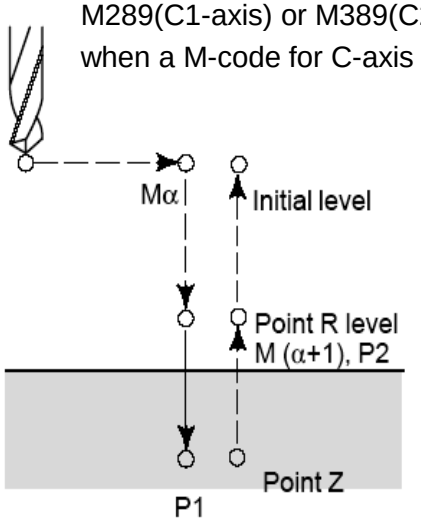
G83 X(U)_ C(H)_ Z(W)_ R_ P_ F_ K_ M_ ;

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

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

  M89 (C1-axis) or M189(C2-axis) is used for C-axis clamp mode.

  M289(C1-axis) or M389(C2-axis) should be commanded before G83 or G87 when a M-code for C-axis clamp is used in G-code block.



Mα : M code for C–axis clamp
M(α+1) : M code for C–axis unclamp
P1 : Dwell specified in the program
P2 : Dwell specified in parameter No. 5111

[Example]

```
M05
M61                                  Left spindle winding change low speed
M35                                  Select C-axis mode
M289
G28 H0.
M33 S2000                            Rotate the milling spindle
G00 X50.0 C0.0                       Positioning the drill along the X and C axes
G83 Z-40.0 R-5.0 Q5000 F500 M89      Drilling hole 1 (M89 : High clamp)
C90.0 Q5000                          Drilling hole 2
C180.0 Q5000                         Drilling hole 3
C270.0 Q5000                         Drilling hole 4
G80                                  Cancel the drilling cycle
M35                                  Stop Milling spindle
```

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

This cycle performs tapping.

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

Tapping is performed by rotating the spindle clockwise. When the bottom of the hole has been reached, the spindle is rotated in the reverse direction for retraction. This operation creates threads.

Feedrate overrides are ignored during tapping. A feed hold does not stop the machine until the return operation is completed.

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

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

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

  M89 (C1-axis) or M189(C2-axis) is used for C-axis clamp mode.

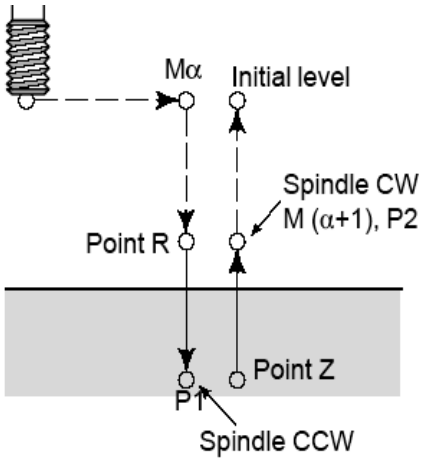
  M289(C1-axis) or M289(C2-axis) should be commanded before G84 or G88 when a M-code for C-axis clamp is used in G-code block.

M29 S_ ; (M29 : Rigid tap mode)

* M72/ M73 : Left/ Right hand tap select

→ This M-code must be commanded before M29.

If M73 is commanded, Tapping thread direction is return to normal direction.



[Example] M12x1.5 TAP

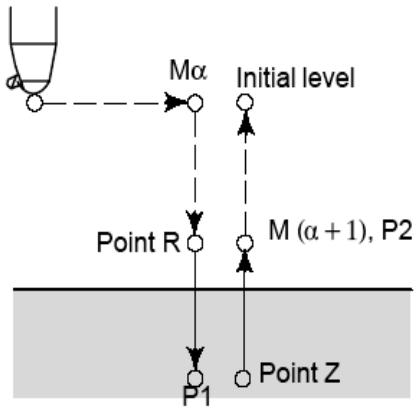
M05	
M61	
M35	Select C-axis
M289	
G28 H0.	
G00 X50.0 C0.0	Positioning the drill along the X and C axes
M29 S1000	Rigid tap mode
G84 Z-30.0 R-5.0 F1500 M89	Drilling hole 1
C90.0	Drilling hole 2
C180.0	Drilling hole 3
C270.0	Drilling hole 4
G80	Cancel the drilling cycle
M35	Stop Milling spindle

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

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

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

* X_ C_ or Z_ C_ : Hole position data
* Z_ or X_ : The distance from point R to the bottom of the hole.
* R_ : The distance from the initial level to point R level
* P_ : Dwell time at the bottom of a hole
* F_ : Cutting feedrate
* K_ : Number of repeats (When it is needed)
* M_ : M code for C-axis clamp (When it is needed)
  M89 (C1-axis) or M189(C2-axis) is used for C-axis clamp mode.
  M289(C1-axis) or M289(C2-axis) should be commanded before G85 or G89 when a M-code for C-axis clamp is used in G-code block.



[Example]

M05	
M61	
M35	Select C-axis mode
M289	
G28 H0.	
M33 S2000	Rotate the milling spindle
G00 X50.0 C0.0	Positioning the drill along the X and C axes
G85 Z-40.0 R-5.0 P500 F300 M89	Drilling hole 1
C90.0	Drilling hole 2
C180.0	Drilling hole 3
C270.0	Drilling hole 4
G80	Cancel the drilling cycle
M35	Stop Milling spindle

9.2 Rigid tapping in 2-axis mode and 3-axis mode (M29)

This M29 command is the rigid tapping M-code.

M29 allows tapping function to valid through synchronous control between spindle and axes.

When commanding M29, M29 finishes immediately. Rigid tapping mode, after executing G84, will be released by G80 or CNC reset.

The spindle must be selected before executing the rigid tapping.

This M29 must be executed in G97 mode (constant surface control cancel) and G98(feed per minute mode) and must be used before commanding G-code.

If M29 is commanded in G96(constant surface control mode), alarm will occur.

G84(G88) cycle that don't use the M29 code will be activated with normal tapping.

In case that a left-handed thread will be cut, command M72 before M29.

It can be executed the high-speed and high-precision tapping work through the synchronous control between the tapping axis and the spindle during the tapping cycle.

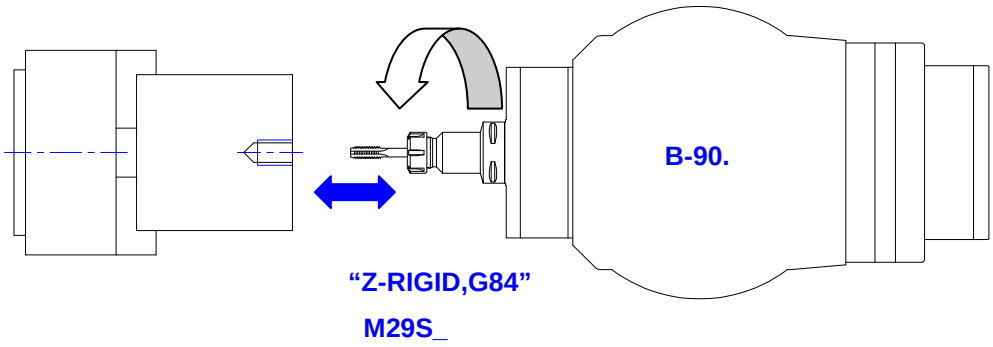
M29 S**** : spindle speed in rigid tapping is set by S-code. Single command in a block.

< notes >

1) Feed override of axis and spindle are fixed at 100% and cannot be changed during rigid tapping.
2) Single block and feed hold are not available during rigid tapping
3) The command of machine lock and dry-run is available during rigid tapping.
4) In case of resetting CNC during rigid tapping, the motion will be stopped immediately, and the rigid tapping mode will be released. But, the tapping mode (G84,G88) will not be released. Tapping mode can be released by "G80"
5) Command "G80" surely to be release the tapping mode after completing the rigid tapping.

In case that tapping mode is not released, it can be occurred the malfunction or machine because the spindle override is fixed at 100%.

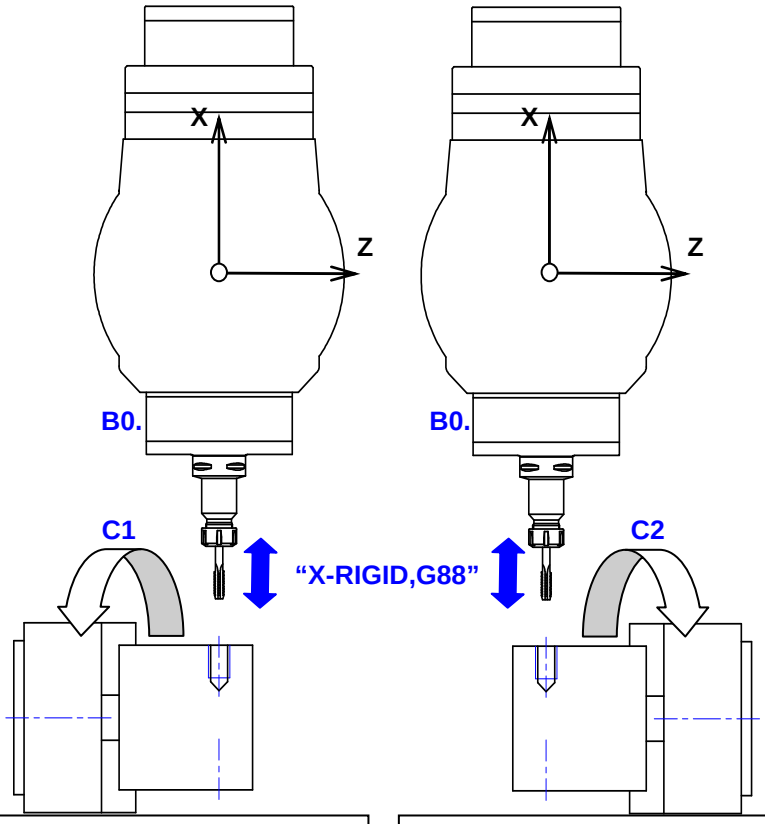
9.2.1 3-axis Z-direction Rigid tapping in Upper unit



< example >

G97 M35 : select the rev. spindle in the constant surface control cancel.
(M05 : left spindle , M105 : right spindle)
G28 U0 W0 : 1st reference point return of X and Z axis
G28 H0 : 1st reference point return of C-axis
T02002 : call the tool offset
M72 : use only when tapping the left-handed thread.
G98 G00 X135.0 Z20.0 B-90. M8 :
move to the start position in the feed mode per minute
Z position is an initial position of rigid tapping.
M29 S500 : command the spindle speed when rigid tapping and milling tool
G84 Z-25.0 R-5. F1000 M89 :
call the tapping cycle
Z position is a final position of rigid tapping.
R position is a distance from initial position to start position of actual rigid tapping.
Regarding spindle rpm and feed, refer to the value which is set according to the size of tap in rigid tap standard sheet.
G80 : complete the tapping mode
M73 : by commanding M73, restore the rotational direction of spindle which had been changed.
G28 U0 W0 :
M30 : end of program

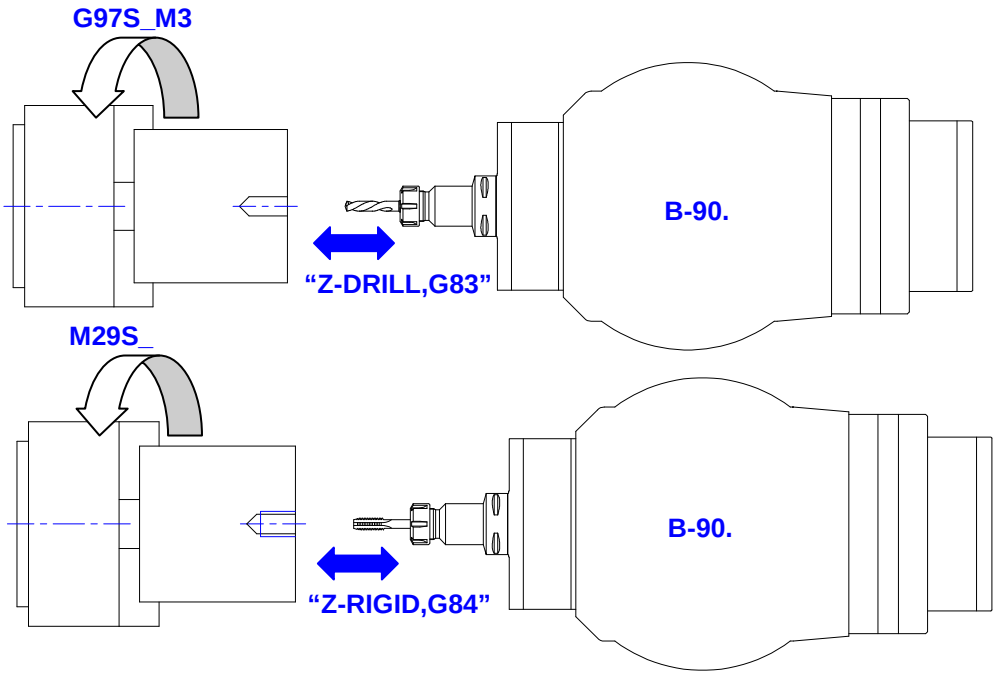
9.2.2 3-axis X-direction Rigid tapping in Upper unit




```
***UPPER TURRET ON LEFT SPINDLE***
N9(X-RIGID,G84@B0. ON LS)
T02000
G28U0.V0.W0.
G54G99G97G80G40M5
M6T02000
G28U0.V0.M61(LS-LOW WINDING)
M35(C1 SELECT)
M90
G28H0.
G50C0.
M101
G0B0.
T02002
G0X200.Z0.C0.
G0X100.
M289(SELECT C1 CLAMP CONTROL)
M29S500
M73(72=L/H, M73=R/H)
G88X70.R-5.F1.0M89
C90.
C180.
C270.
G80M90
M35
G28U0.V0.W0.
M1
```

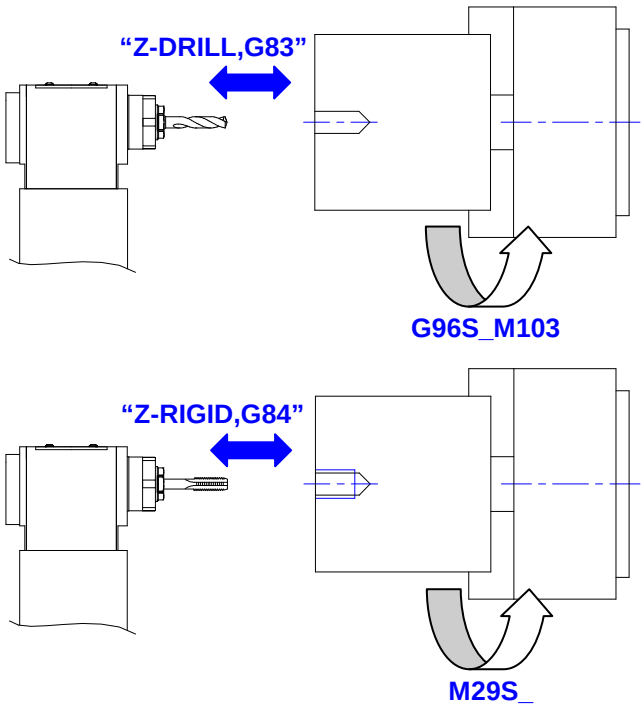
```
***UPPER TURRET ON RIGHT SPINDLE***
N10(X-RIGID,G84@B0. ON RS)
T02000
G28U0.V0.W0.
G55G99G97G80G40M105
M6T02000
G28U0.V0.M161(RS-LOW WINDING)
M135(C2 SELECT)
M211(COMPOSITE MODE ACT)
M190
G28H0.
G50C0.
M101
G0B0.
T02002
G0X200.Z0.C0.
G0X100.
M389(SELECT C2 CLAMP CONTROL)
M29S500
M73(72=L/H, M73=R/H)
G88X70.R-5.F1.0M189
C90.
C180.
C270.
G80M190
M135
M212(COMPOSITE MODE CANCEL)
G28U0.V0.W0.
M1
```

9.2.3 2-axis Drilling and Rigid Tapping in Upper unit (No C-axis control function)




```
***UPPER TURRET***
N1(Z-DRILL@B-90. ON LS)
T01000
G28U0.V0.W0.
G54G99G80G40M5
M6T08000
T02000
G28U0.V0.M62
T08008
G400B-90.J1.
G97S1000M3
G0Z10.
X0.
G83Z-25.R-5.Q20000F0.5
G80
G28U0.V0.W0.M5
M1

N2(Z-RIGID@B-90 ON LS)
T02000
G28U0.V0.W0.
G54G99G97G80G40M5
M6T08000
G28U0.V0.M62
T08008
G400B-90.J1.
G0Z10.
X0.
M73(M73=R/H,M72=L/H)
M29S400
G84Z-20.R-5.F1.5P500
G80
G28U0.V0.W0.M5
M30
```

9.2.4 2-axis Drilling and Rigid Tapping in Lower unit (No C-axis control function)




```
***LOWER TURRET***
N1(Z-DRILL ON RS)
G28U0.W0.
G55G99G80G40M105
T0505
M162
G97S1000M103
Z-10.
X0.
G83Z25.Q20000R5.F5.0
G80
G28U0.W0.M105
M1

N2(Z-RIGID ON RS)
G28U0.W0.
G55G99G97G80G40M105
T0606
M162
Z-10.
X0.
M73(M73=R/H,M72=L/H)
M29S400
G84Z20.R5.F1.5P500
G80
G28U0.W0.M105
M30
```

9.3 Cylindrical interpolation (G07.1)

The amount of travel of a rotary axis specified by an angle is once internally converted to a distance of a liner axis along the outer surface so that linear interpolation or circular interpolation can be performed with another axis. After interpolation, such a distance is converted back to the amount of travel of the rotary axis.

The cylindrical interpolation function allows the side of a cylinder to be developed for programming. So programs such as a program for cylindrical ca grooving can be created very easily.

G07.1 IP r ; Starts the cylindrical interpolation mode

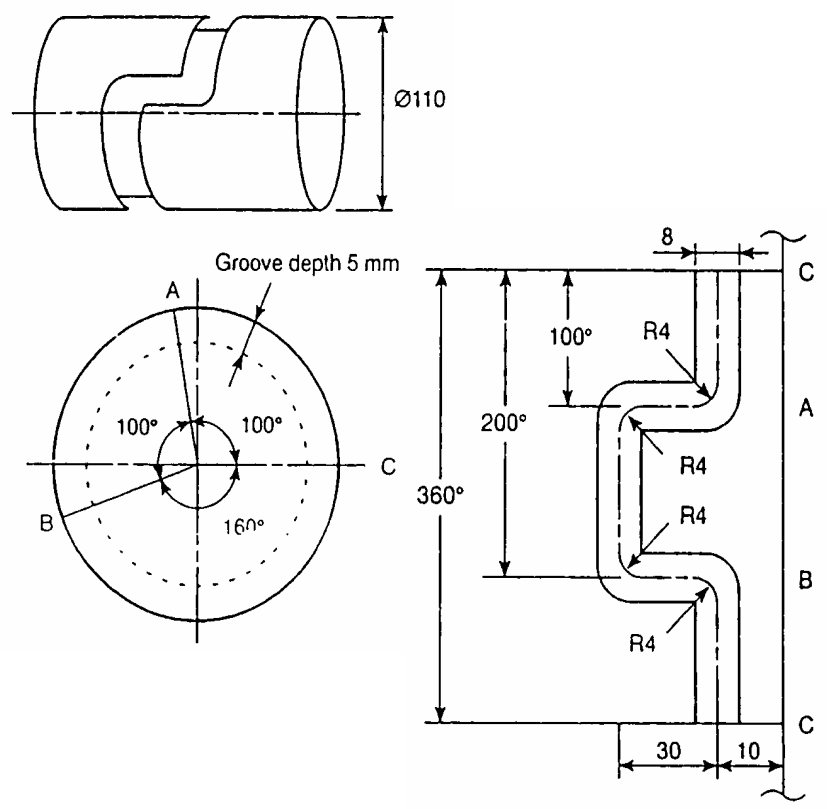
:

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

* IP : An address for the rotation axis
* r : Radius value of the cylinder

Specify G07.1 IP r ; and G07.1 IP 0; in separate blocks.

G117 can be used instead of G07.1.



[Example]

```
M35
G97 S500 M33
X115.0 Z-10.0
G98 G01 X100.0 F50
G19 W0 H0
G07.1 C50.0
C95.416
G02 Z-14.0 C100.0 R4.0
G01 Z-36.0
G03 Z-40.0 C104.583 R4.0
G01 C195.416
G03 Z036.0 C200.0 R4.0
G01 Z-14.0
G02 Z-10.0 C204.584 R4.0
G01 C360.0
G07.1 C0
X115.0
G00 X200.0 Z100.0 M35
```

9.4 Polar interpolation (G12.1/G13.1)

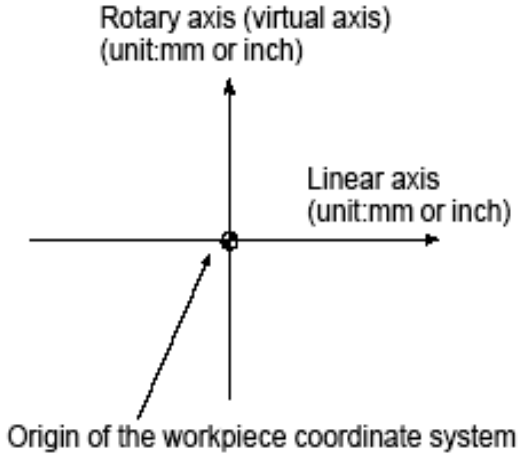
Polar coordinate interpolation is a function that exercises contour control in converting a command programmed in a Cartesian coordinate system to the movement of a linear axis (movement of a tool) and the movement of a rotary axis (rotation of a workpiece). This method is useful in cutting a front surface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

G12.1 ; Starts polar coordinate interpolation mode

:

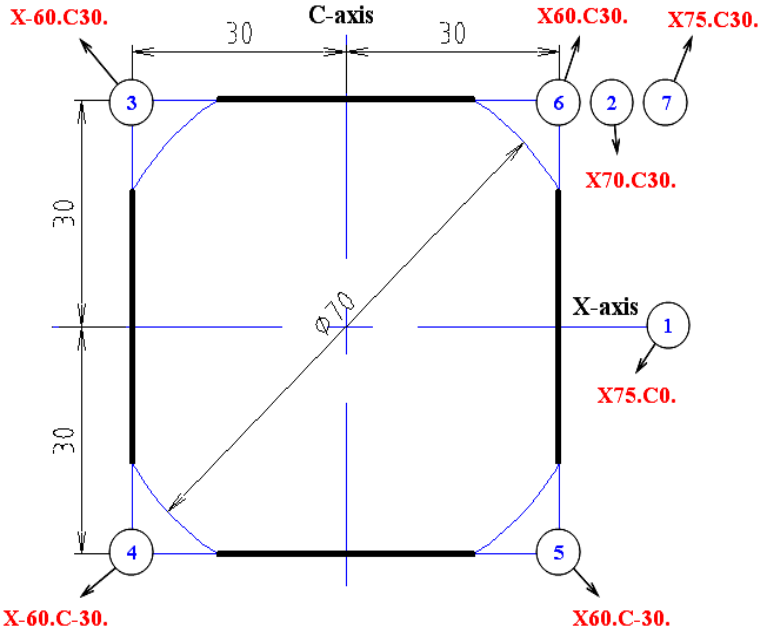
G13.1 ; Polar coordinate interpolation mode is cancelled.

* G112 and G113 can be used in place of G12.1 and G13.1 respectively.



G12.1 starts the polar coordinate interpolation mode and selects a polar coordinate interpolation plan. Polar coordinate interpolation is performed on this plane.

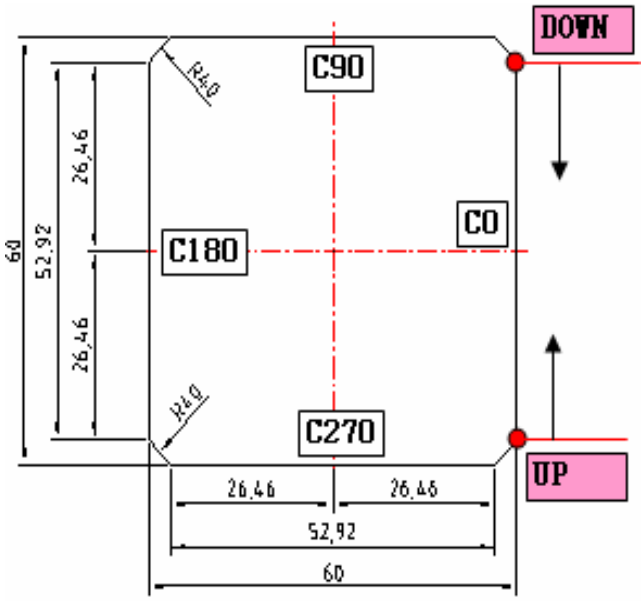
When the power is turned on or the system is reset, polar coordinate interpolation is canceled (G13.1).



[Example]

```
G97 S1500 M33
G00 X75.0 C0.0 Z-5.0          (1)
G12.1 (Polar mode on)
G42 G01 X70.0 C30.0 F120      (2)
X-60.0                        (3)
C-30.0                        (4)
X60.0                         (5)
C30.0                         (6)
X75.0                         (7)
G40
G13.1 (Polar mode cancel)
G00 Z5.0
G28 U0 W0
```

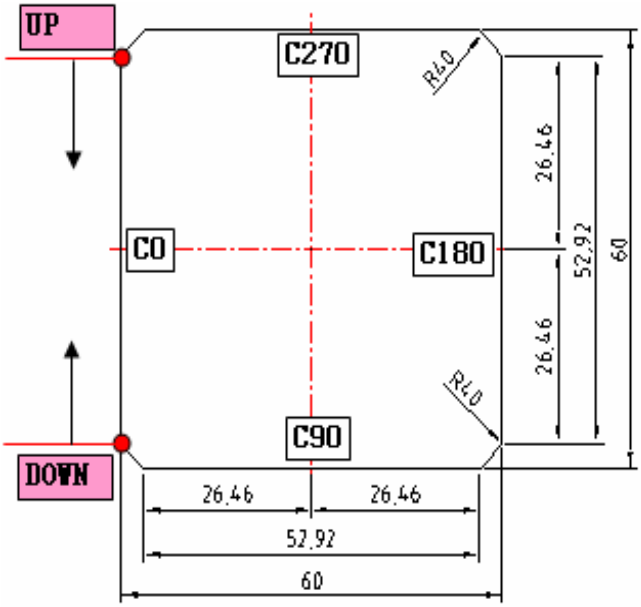
1) Upper unit on Left spindle



< **Down cutting** >	< **Up cutting** >
G40G80G98G17G54	G40G80G98G17G54
M05	M05
M61	M61
M35	**M35**
G28H0	G28H0
M101	M101
G97S2000M33	G97S2000M33
G0X100.Z5.0C0.	G0X100.Z5.0C0.
G0Z-35.0M08	G0Z-35.0M08
G12.1	G12.1
G41G01X60.C26.46F100	**G42**G01X60.C-26.46F100
G01C-26.46	G01C26.46
G02X52.92C-30.R40.	G03X52.92C30.R40.
G01X-52.92	G01X-52.92
G02X-60.0C-26.46R40.	G03X-60.0C26.46R40.
G01C26.46	G01C-26.46
G02X-52.92C30.R40.	G03X-52.92C-30.R40.
G01X52.92	G01X52.92
G02X60.0C26.46R40.	G03X60.0C-26.46R40.
G01X75.	G01X75.
G40	**G40**
G13.1	G13.1

Note) When programming polar coordinate interpolation in Lower unit of MX,
C-axis must be calculated as much as 15˚
(15˚ means a slant angle between Upper unit and Lower unit)

2) Upper unit on Right spindle



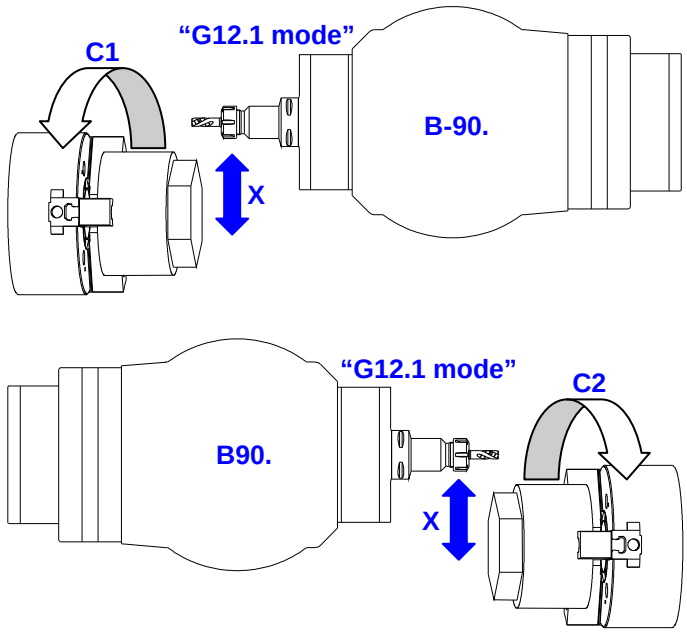
< Down cutting >

G40G80G98G17G55
M105
M161
M135
M35
M211
G28H0
M101
G97S2000M33
G0X100.Z-5.0C0
G0Z45.0M08
G12.1
G41G01X60.C26.46F100
G01C-26.46
G03X52.92C-30.R40.
G01X-52.92
G03X-60.0C-26.46R40.
G01C26.46
G03X-52.92C30.R40.
G01X52.92
G03X60.0C26.46R40.
G01X75.
G40
G13.1

< Up cutting >

G40G80G98G17G55
M105
M161
M135
M35
M211
G28H0
M101
G97S2000M33
G0X100.Z-5.0C0
G0Z45.0M08
G12.1
G42G01X60.0C-26.46F100
G01C26.46
G02X52.92C30.R40.
G01X-52.92
G02X-60.0C26.46R40.
G01C-26.46
G02X-52.92C-30.R40.
G01X52.92
G02X60.0C-26.46R40.
G01X75.
G40
G13.1

3) Upper, Hexagon-Mill at Polar coordinate mode on Left & Right-Spindle




```
***UPPER TURRET***
N1(END MILL@B-90. ON LS)
(HEXAGON IN G12.1)
T08000
G28U0.V0.W0.
G54G99G80G40M5
M6T08000
G28U0.V0.M61
M35
M90
G28H0.
G50C0.
T08008
G400B-90.J0
M101
G97S1000M33
G0X120.Z10.C0.(POS 1)
G1Z-12.F1.0(M88)
G12.1
G1G42X87.988C0.F0.5(POS 2)
X43.994C38.1(POS 3)
X-43.994(POS 4)
X-87.988C0.(POS 5)
X-43.994C-38.1(POS 6)
X43.994(POS 7)
X87.899C0.(POS 2B)
G40X120.C0.F2.0(POS 1B)
G13.1M90
G28U0.V0.W0.M35
M30
```

```
***UPPER TURRET***
N2(END MILL@B90. ON RS)
(HEXAGON IN G12.1)
T08000
G28U0.V0.W0.
G55G99G80G40M105
M6T08000
G28U0.V0.M161
M135(C2 SELECT)
M211
M190
G28H0.
G50C0.
T08008
G400B90.J0
M101
G97S1000M33
G0X120.Z-10.C0.(POS 1)
G1Z12.F1.0(M188)
G12.1
G1G42X87.988C0.F0.5(POS 2)
X43.994C38.1(POS 3)
X-43.994(POS 4)
X-87.988C0.(POS 5)
X-43.994C-38.1(POS 6)
X43.994(POS 7)
X87.899C0.(POS 2B)
G40X120.C0.F2.0(POS 1B)
G13.1M190
G28U0.V0.W0.M135
M212
M30
```

10. Two-path Control Function

10.1 Waiting M-code (M400-)

Control based on M codes is used to cause on tool post to wait for the other during machining. By specifying an M code in a machining program for each tool post, the two tool posts can wait for each other at a specified block. When an M code for waiting is specified in a block for on tool post during automatic operation, the other tool post waits for the same M code to be specified before staring the execution of the next block. This function is called the tool post waiting function.

A range of M codes used as M codes for waiting is to be set in the parameters (No.8110 and 8111) beforehand.

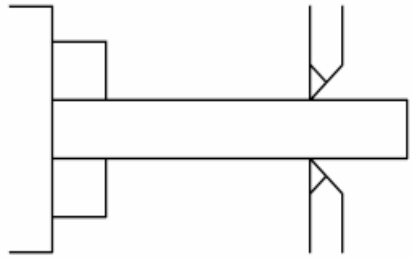
[Example] M400 to M499 are used as M codes for waiting
Parameter setting : No 8110=400 (Minimum M code for waiting)
No 8111=499 (Maximum M code for waiting)

[Example]

UPPER	***LOWER***
T25000	G28 U0 W0
G28 U0 W0 V0 M5	G54 G99 G80 G40
G54 G99 G80 G40	T02002
M6 T25000	***M410(Waiting M-code)***
G28U0 V0 M62	G00 X190.0 Z4.0
T25025	M195(SELECT LS ENCODER)
G400 B-45. J1.R2.	***M411(Waiting M-code)***
M410(Waiting M-code)	G01Z-50.0 F0.5
G50 S1500	X200.0
G96 S300 M3	U2.0 W1.0
G00 X195.0 Z2.0	G00 U2.0 Z4.0
M411(Waiting M-code)	X180.0
G01Z-50.0 F0.5	***M412(Waiting M-code)***
X200.0	G01 Z-50.0
U2.0 W1.0	X190.0
G00 U2.0 Z2.0	U2.0 W1.0
X185.0	G00 U2.0 Z4.0
M412(Waiting M-code)	***M413(Waiting M-code)***
G01 Z-50.0	G28U0 W0
X190.0	M30
U2.0 W1.0	
G00 U2.0 Z2.0	
M413(Waiting M-code)	
G28U0 W0 V0 M5	
M30	

10.2 Balance cut (G68/G69)

When a thin workpiece is to be machined as shown below, a precision machining can be achieved by machining each side of the workpiece with a tool simultaneously, this function can prevent the workpiece from warpage that can result when only one side is machined at a time. When both sides are machined at the same time, the movement of one tool must be in phase with that of the other tool. Otherwise, the workpiece can vibrate, resulting in poor machining. With this function, the movement of one tool post can be easily synchronized with that of the other tool post.



When G68 is specified in the programs for both tool post 1 and tool post 2, the pulse distribution of tool post 1 is synchronized with that of tool post 2 to start balance cutting.

Thus the two tool posts can move exactly at the same time to allow balance cutting.

G code	Meaning
G68	Balance cut mode
G69	Balance cut mode cancel

In the balance cut mode, balance cutting is performed only when a move command is specified for both tool posts. Balance cutting is performed even when different axes are specified for each tool post or an offset move command is specified. G68 or G69 must be specified in a single block. (Otherwise, a P/S alarm No.163 is raised.) When G68 or G69 is specified with on tool post, the tool post does not move until the execution for the other tool post proceeds to G68 or G69. And if cutting is specified with one tool post in the balance cut mode, the tool post does not move until the execution of or the other tool post proceeds to a cutting command.

[Example]

```
***UPPER ***
M410(Waiting M-code)
G50 S1500
G96 S300 M3
G00 X195.0 Z2.0
M411(Waiting M-code)
M192
G68
G01 Z-50.0 F0.3
X200.0
U2.0 W1.0
G69
M193
```

```
***LOWER***
M410(Waiting M-code)
G00 X195.0 Z2.0
M195(Select LS Encoder)
M411(Waiting M-code)
G68
G01 Z-50.0 F0.3
X200.0
U2.0 W1.0
G69
```

*M192/M193: Simultaneous Feedrate On/Off (Balance Cutting)

If M192 is commanded in Upper program, lower side axes are controlled by upper side feedrate override switch.

10.3 Composite control (M211/M212)

1) The composition control is the function to control a certain axis (C1,C2) of another path on 2 path. If M211 is commanded after all of spindles stop and 3-axis of Rev.spindle is selected, C-axis of another path can be controlled. And if M212 is commanded, the composition control will be cancelled.

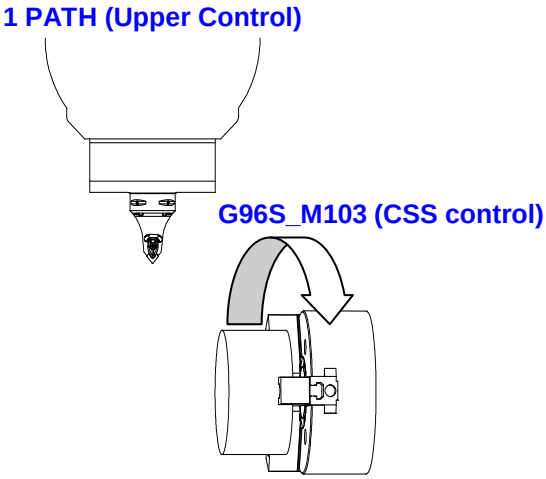
2) M-code related to the composition control
   * M211 : Execute the composition control
   * M212 : Cancel the composition control
   * M35 : In Lower turret, Rev.spindle stop and select C1-axis.
   * M135 : In Upper turret, Milling spindle stop and select C2-axis.

3) Conditions and methods of composition control
   * After turn on machine power, once Ref. position return of C1, C2 in manual mode must be operated. (Otherwise, alarm No 2132 occur.)
   * All of spindles must be stopped.
   * Rev.spindle (3-axis) of unit related to the composition control must be selected.

4) Clamping and Unclamping the C-axis of other path
   * Clamping and Unclamping the C-axis of other path is only valid in composition control mode
   * Low/ High clamp of C1-axis : M88 / M89
   * Low/ High clamp of C2-axis : M188/M189
   * M289 : Left spindle high clamp control
   * M389 : Right spindle high clamp control

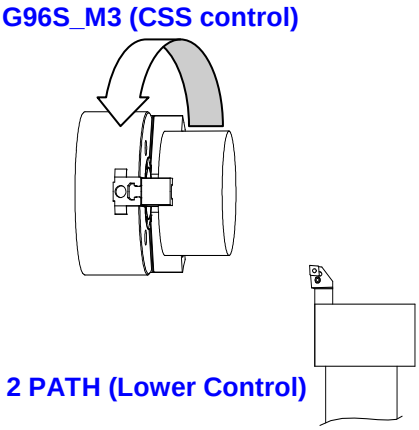
5) The method of using constant surface speed control (G96) in composition control mode
   * Constant surface speed control (G96) must be commanded after commanding Rev.spindle stop or rotation. (Otherwise, alarm No 2130 occur.)

10.3.1 Upper : 2-axis machining in CSS mode, Using the lower path's spindle (on Right-spindle)



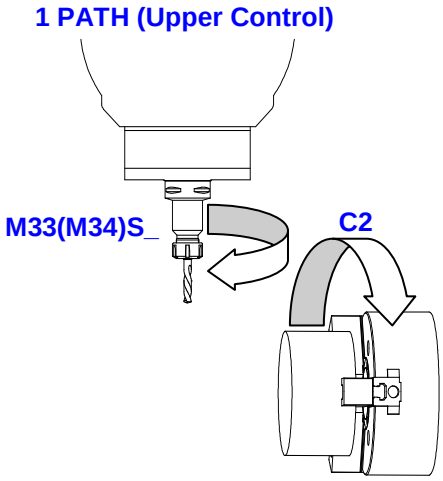

```
***UPPER***
G28 U0 V0 W0
M105 (Right-spindle stop)
      :
(Cutting program)
      :
M105 (Right-spindle stop)
G28 U0 V0 W0
```

10.3.2 Lower : 2-axis machining in CSS mode, Using the upper path's spindle (on Left-spindle)



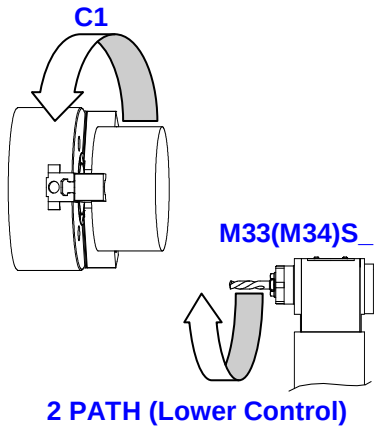

```
***LOWER***
G28 U0 W0
M05 (Left-spindle stop)
      :
(Cutting program)
      :
M05 (Left-spindle stop)
G28 U0 W0
```

10.3.3 Upper : 3-axis machining, Using the lower path's C-axis (C2 composition control mode)



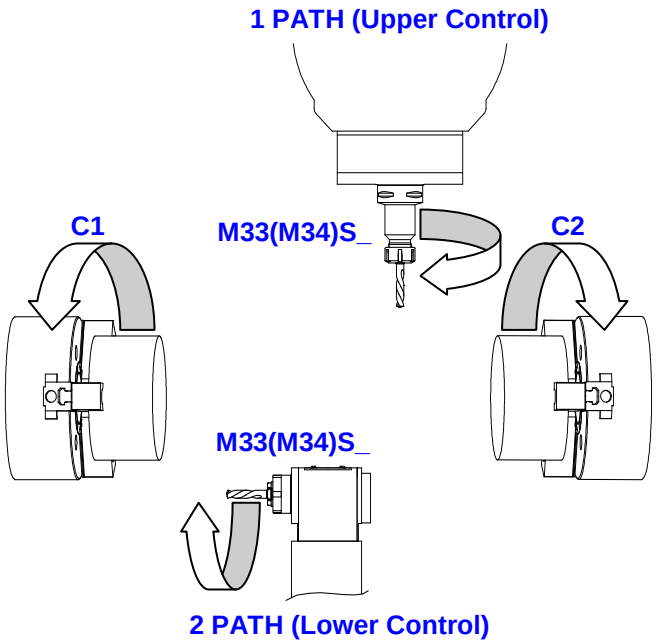

```
***UPRER***
G28 U0 V0 W0
M105 (Right-spindle stop)
M135 (C2-axis select)
M211 (Composite act)
      :
(Cutting program)
      :
M135 (Milling spindle stop)
G28 U0 V0 W0
M212 (Composite cancel)
```

10.3.4 Lower : 3-axis machining, Using the Upper path's C-axis (C1 composition control mode)




```
***LOWER***
G28 U0 W0
M05 (Left-spindle stop)
M35 (C1-axis select)
M211 (Composite act)
      :
(Cutting program)
      :
M35 (Rev.spindle stop)
G28 U0 W0
M212 (Composite cancel)
```

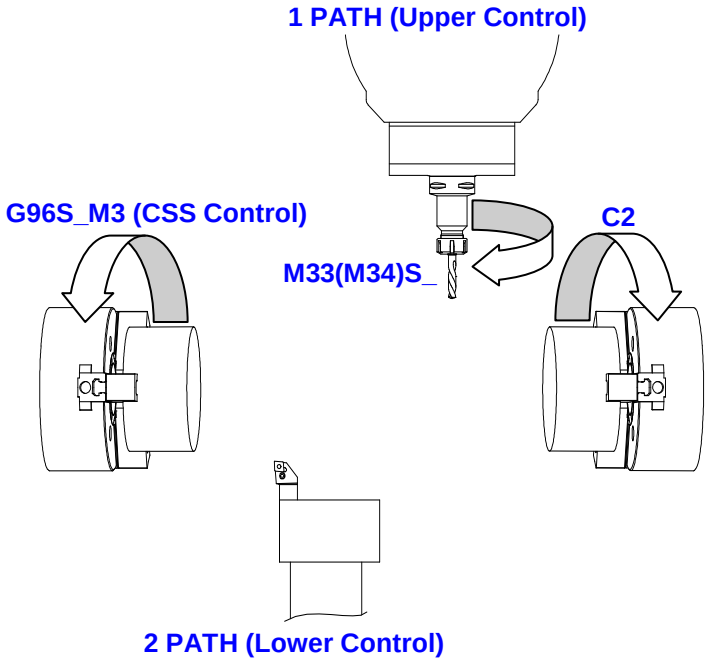
10.3.5 Upper : 3-axis machining, Using the lower path's C-axis (C2 composition control mode)
Lower : 3-axis machining, Using the upper path's C-axis (C1 composition control mode)




```
***UPPER***
G28 U0 V0 W0
M105 (Right-spindle stop)
M135 (C2-axis select)
M211 (Composite act)
      :
(Cutting program)
      :
M135 (Milling spindle stop)
G28 U0 V0 W0
M212 (Composite cancel)
```

```
***LOWER***
G28 U0 W0
M05 (Left-spindle stop)
M35 (C1-axis select)
M211 (Composite act)
      :
(Cutting program)
      :
M35 (Rev.spindle stop)
G28 U0 W0
M212 (Composite cancel)
```

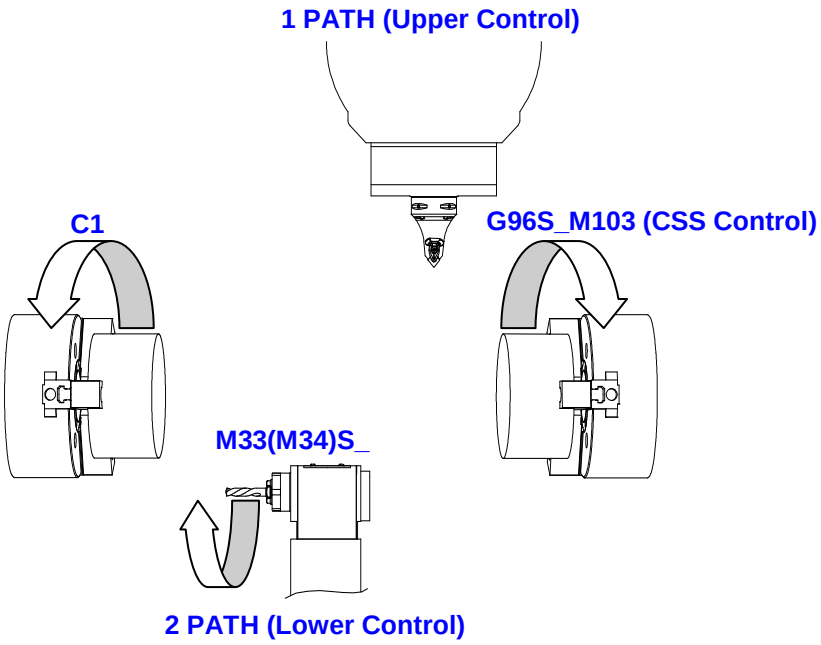
10.3.6 Upper : 3-axis machining, Using the lower path's C-axis (C2 composition control mode)
Lower : 2-axis machining in CSS mode, Using the upper path's spindle (on Left-spindle)




```
***UPPER***
G28 U0 V0 W0
M105 (Right-spindle stop)
M135 (C2-axis select)
M211 (Composite act)
G97 S2000 M33
M400 (Waiting M-code)
        :
(Cutting program)
        :
M135 (Milling spindle stop)
M410 (Waiting M-code)
G28 U0 V0 W0
M212 (Composite cancel)
```

```
***LOWER***
G28 U0 W0
M05 (Left-spindle stop)
M400 (Waiting M-code)
G96 S200 M03
        :
(Cutting program)
        :
        :
        :
M05 (Left-spindle stop)
M410 (Waiting M-code)
G28 U0 W0
```

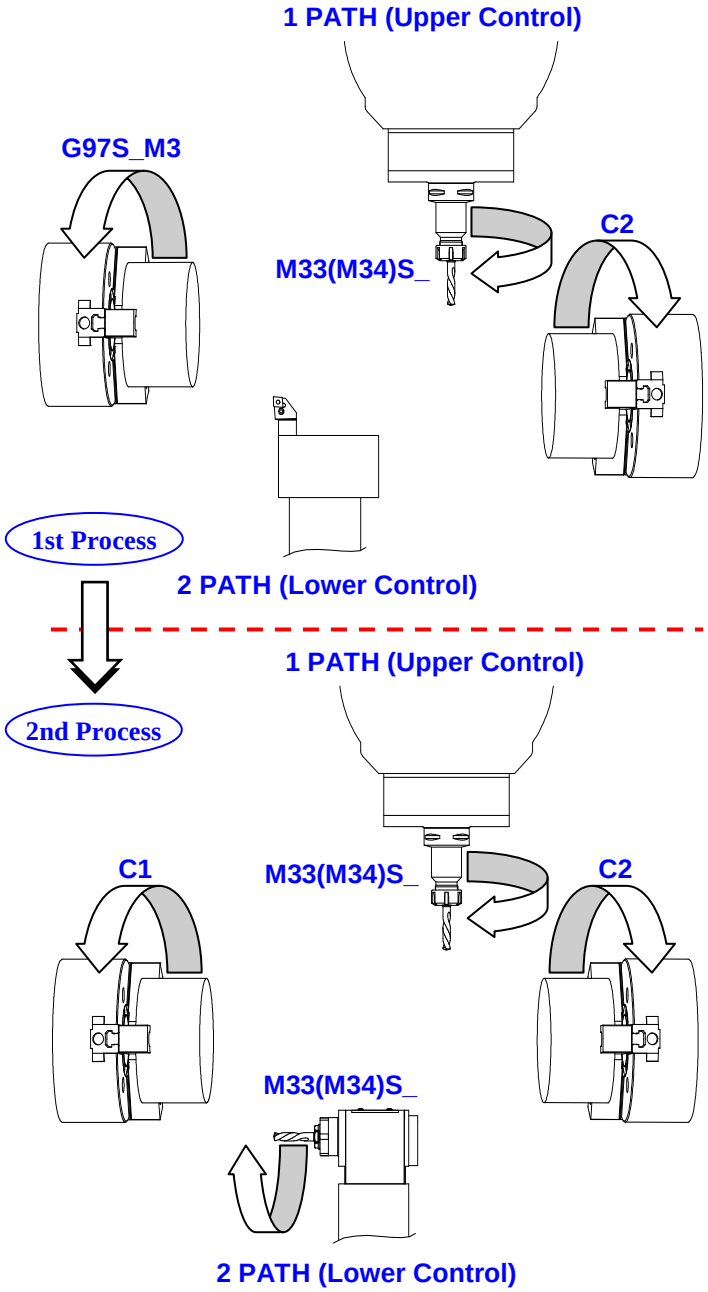
10.3.7 Upper : 2-axis machining in CSS mode, Using the lower path's spindle (on Right-spindle)
Lower : 3-axis machining, Using the upper path's C-axis (C1 composition control mode)




```
***UPPER***
G28 U0 V0 W0
M105 (Right-spindle stop)
M400 (Waiting M-code)
G96 S200 M103
        :
(Cutting program)
        :
        :
        :
M105 (Right-spindle stop)
M410 (Waiting M-code)
G28 U0 V0 W0
```

```
***LOWER***
G28 U0 W0
M05 (Left-spindle stop)
M35 (C1-axis select)
M211 (Composite act)
G97 S2000 M33
M400 (Waiting M-code)
        :
(Cutting program)
        :
M35 (Rev.spindle stop)
M410 (Waiting M-code)
G28 U0 W0
M212 (Composite cancel)
```

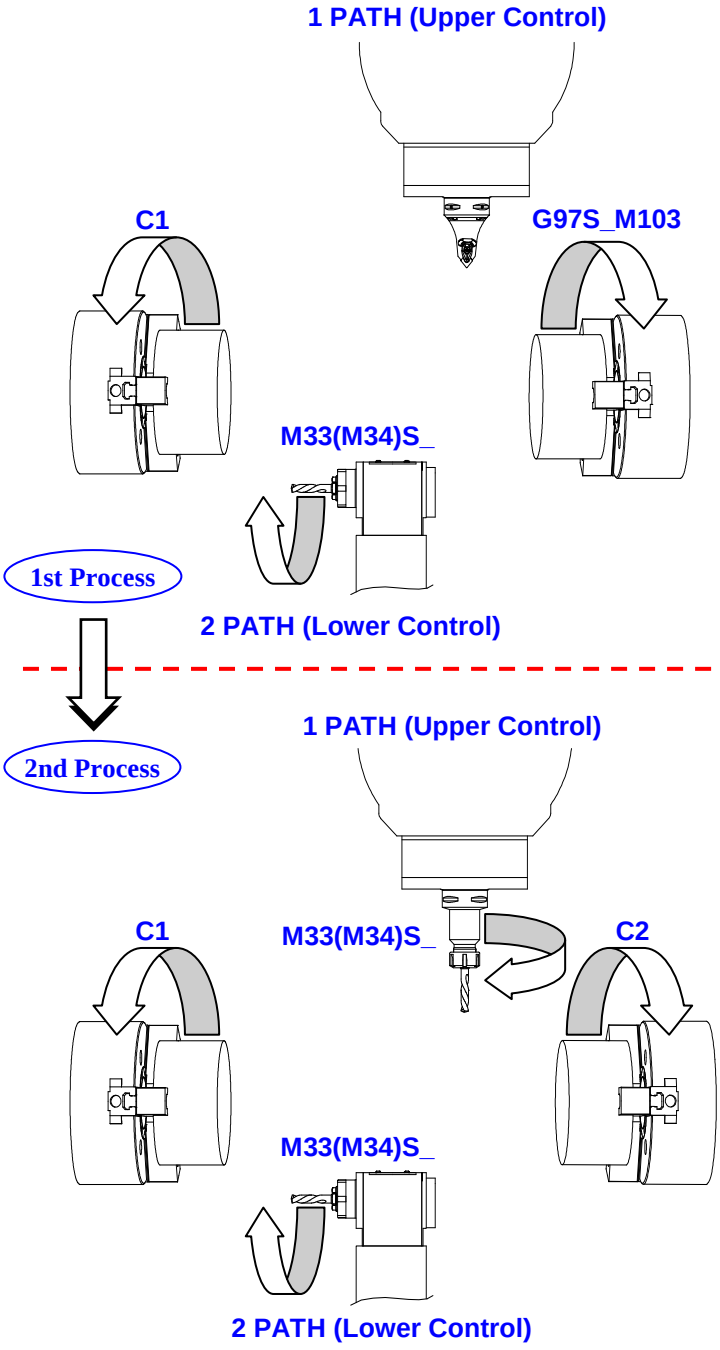
10.3.8 Upper : 3-axis machining, Using the lower path's C-axis
(C2 composition control mode)
Lower : 2-axis machining, Using the upper path's spindle (on Left-spindle)
followed by 3-axis machining in composite control




```
***UPPER***
G28 U0 V0 W0
M105 (Right-spindle stop)
M135 (C2-axis select)
M211 (Composite act)
      :
G97 S2000 M33
      :
(Cutting program)
      :
M135 (Milling spindle stop)
G28 U0 V0 W0
M212 (Composite cancel)
```

```
***LOWER***
G28 U0 W0
M05 (Left-spindle stop)
G97 S1000 M03
      :
(Cutting program)
      :
M05 (Left-spindle stop)
G28 U0 W0
M35 (C1-axis select)
      :
G97 S2000 M33
      :
(Cutting program)
      :
M35 (Rev.spindle stop)
G28 U0 W0
```

10.3.9 Upper : 2-axis machining, Using the lower path's spindle (on Right-spindle) followed by 3-axis machining in composite control
Lower : 3-axis machining, Using the upper path's C-axis (C1 composition control mode)

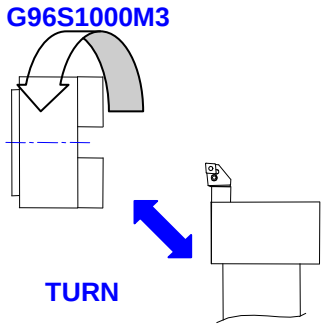


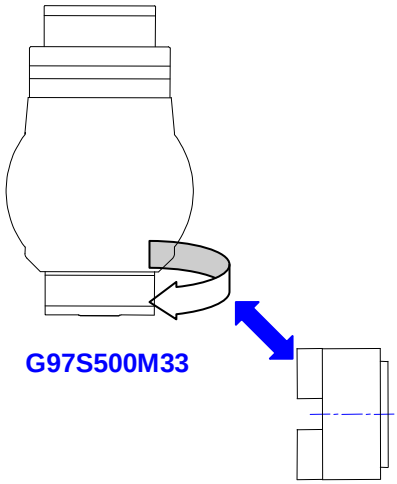
UPPER
G28 U0 V0 W0
M105 (Right-spindle stop)
G97 S1000 M103
:
(Cutting program)
:
M105 (Right-spindle stop)
M135 (C2-axis select)
:
G97 S2000 M33
:
(Cutting program)
:
M135 (Milling spindle stop)
G28 U0 V0 W0

LOWER
G28 U0 W0
M05 (Left-spindle stop)
M35 (C1-axis select)
M211 (Composite act)
:
G97 S1000 M03
:
(Cutting program)
:
M35 (Rev.spindle stop)
G28 U0 W0
M212 (Composite cancel)

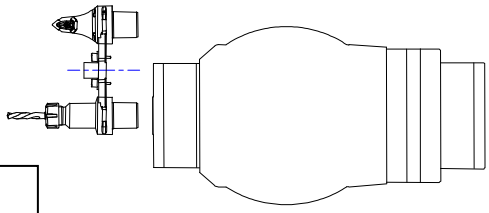
10.3.10 Upper : ATC between 3-axis machining, Using the lower path's spindle followed by 3-axis machining in composite control
Lower : 2-axis machining on Left spindle

```
***UPPER TURRET***
N1(Mill on Right spindle)
M212
G28U0W0V0
T11000
M06T11000
G28U0W0V0
T13000
T11011
M135
M101
G98
M211
G97S500M33
M401
G28 H0.
G0C90.
G04X20.0 (cutting)
M135
M212
M01

N2(TOOL CHANGE)
T13000
M06T13000
G28U0W0V0

N3(Mill on Right spindle)
T11000
M105
M135
M211
G98
M402
M101
G97S500M33
M403
G28 H0.
G04X20.0 (cutting)
M135
M212
M01
M30
```

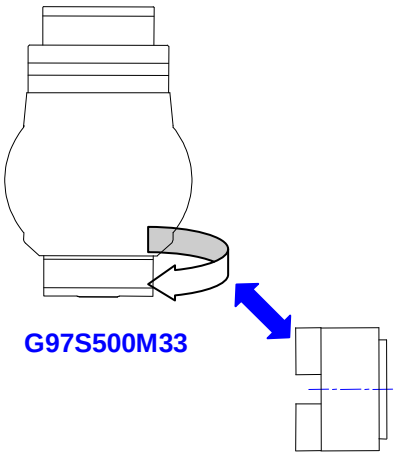




TOOL CHANGE



```
***LOWER TURRET***
N1(TURN on Left Spindle)
M212
G28U0W0A0
M05
T0600
M401
G96S1000M04
G04X35.0 (cutting)
M402
G97S **** M04
M403
G96S1000M04
G04X20.0 (cutting)
M05
G28U0W0
M01
M30
```



11. Torque Control Function

11.1 A –axis torque control : G300/G301 (B-axis torque control in TT series)

Z2-axis torque control : G302/G303

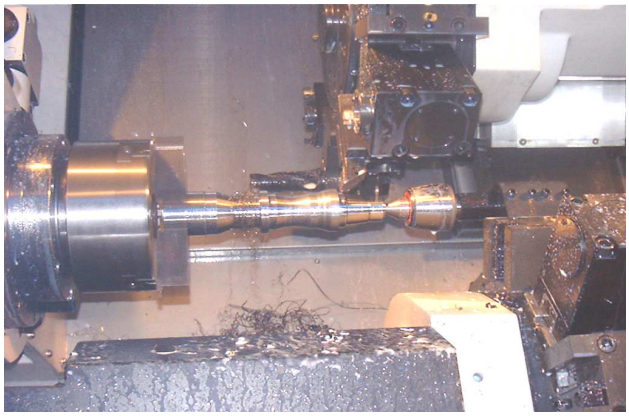
11.2 Cut-off confirmation : G350 (Applied to the machine having a right spindle)

11. Torque Control Function

11.1 A-axis torque control : G300/G301 (B-axis torque control in TT series)

Z2-axis torque control : G302/G303

In MX/TT machine of opposable twin spindle, upper and lower turret turning center, this function is executed by commanding G-code on program in order to machine the shaft with supporting the center by installing the tail-center at right spindle or lower tool holder.



[Example] Z2-axis Torque Control

[Format]

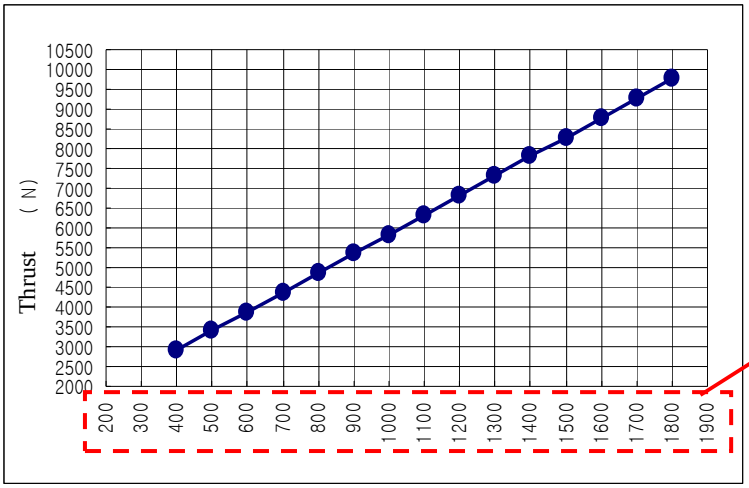
G300(G302) B +/-____ ;

+ : + direction (only Z2 axis)

- : - direction

G301(G303) ; Cancel

; Variable B = Torque Data



The commanded torque value (α22/3000i Fanuc Servo Motor)

[Example]

G0 A_

G300(G302) B-800.0

G301(G303)

(Note 1) For avoiding the no load alarm range(100mm), approach to the around of 100mm. Because A-axis position control isn't executed, the advance will keep on going until the tool confronts the load. Therefore, in order to avoid this problem, compare the position where G300 is commanded to the position where the tool stops by confronting the load. If the value is over 100mm, alarm 2072(Torque control command alarm) occurs. The no load range can be set in PMC data (D2000).

(Note 2) G300 is the code to call macro program as below and B variable is macro variable to transfer the torque data. (Be sure to mark the point on torque command data) (-)value means that the tool moves to (-) direction in machine coordinate system, and (+)value means that the tool moves to (+) direction in machine coordinate system. In controlling torque speed is set in PMC-60RPM. When the speed will be below 30mm/min, NC program run next block.

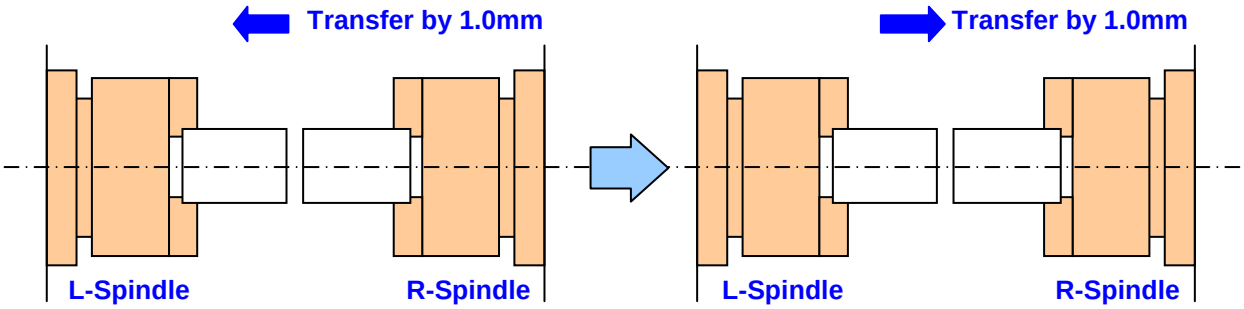
11.2 Cut-Off confirmation : G350 (Applied to the machine having a right spindle)

In the machine having the right-spindle, this function confirms whether the cutting is finished or no after cutting the workpiece. The hardware part is eliminated from the machine and the cut off confirmation function is executed using the torque control function of NC itself.

[Example]

```
G0 X_ Z_   ; Tool Escape after Cut-off
G350       ; Cut off Confirmation Function Enable
G4 U0.5    ; Dwell time 0.5sec
G28 A_     ; A-axis return Home Position
```

(Note) If G350 is commanded, A-axis will move in (+) direction by 1 second . At this time, the deviation is greater than 1.0mm, then executes the next block and if not, cut off confirmation error NC alarm No.3003 occurs.



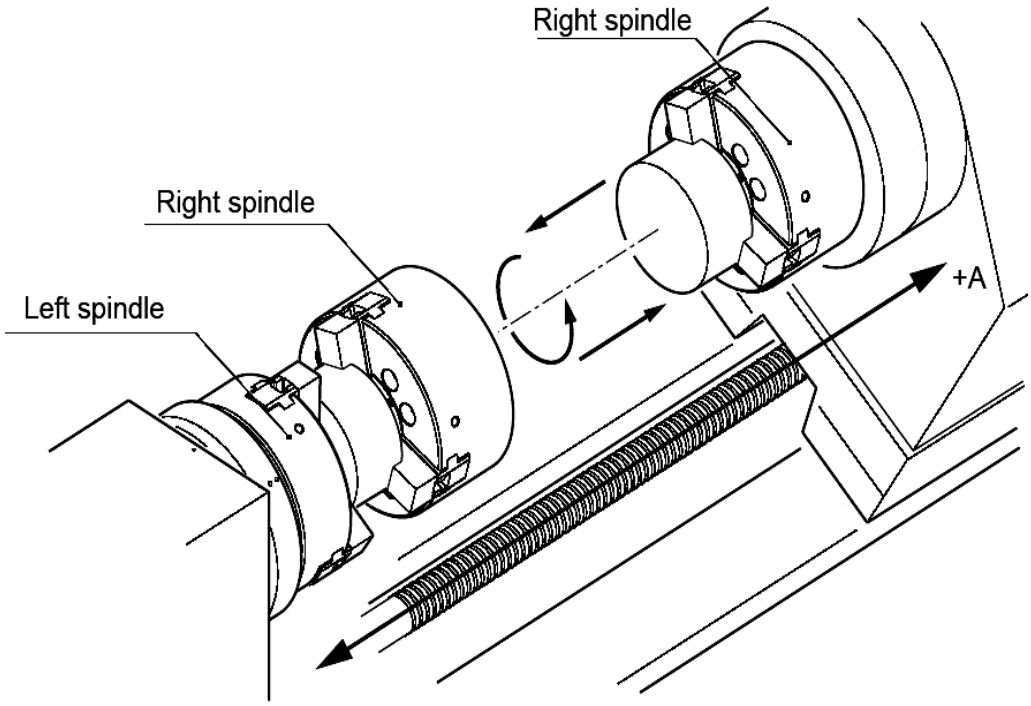
12. Work transfer & synchronous control

12.1 Work-piece transfer introduction

As figure shown below, if a machine is equipped with left spindle and right spindle, it needs to transfer the work-piece which is clamped at the chuck of left spindle to right spindle and needs to clamp the work piece.

According to the state for work-piece and machining condition, the methods to transfer the work-piece are various. Among these, the synchronous control and torque skip function are used to transfer the work-piece while the speed and phase of left spindle and right spindle make same.

Normally right spindle is moved to left spindle by ***A-axis*** and the work-piece in left spindle is transferred to right spindle.



Right spindle moves to left spindle by ***A-axis*** and the work-piece in left spindle is transferred to right spindle.

Note 1) In ***MX*** series, the advance and return axis of right spindle is ***A-axis*** and the ***A-axis*** is controlled by the ***lower unit's program***.

But, in ***TT*** series, the advance and return axis of right spindle is ***B-axis*** and the ***B-axis*** is controlled by the ***upper unit's program***.

12.2 Synchronous control & torque skip function

Synchronous control & torque skip function are used to transfer the work-piece between both spindles while they are rotating.

(1) Synchronous control

Synchronous control makes both spindles rotate at the same speed or phase, that is, synchronizing both spindles and it is used to transfer the workpiece from left spindle to right spindle.

Synchronous control of both spindles can be commanded and executed only by M-code in automatic mode, and spindle override command is available. Both spindles are stopped during synchronizing and synchronous control mode is released as follows.

-M205 command makes both spindles stop.

-M206 command releases the status of rotating in synchronous control. If left side rotates in 1000rpm and right side 500rpm before commanding synchronous control, and then command the left side in M203 with 2000rpm. After that, command M206, then left side will rotate in 2000rpm and right side in 500rpm.

- It is changed into manual mode (Handle, Jog, Reference point return mode)

- M02 or M30 is commanded.

- Pressing CNC reset switch during synchronous control

- Front door is opened. (Only when the door interlock is available)

(2) Torque skip function

To transfer the workpiece of left spindle to right spindle, right spindle will be moved and adhered closely to left spindle. This is difficult to determine the adhere point because the exact adhere point may differ according to the state of workpiece and each workpiece.

Torque skip function checks the torque of ***A-axis*** that moves right spindle when right spindle are adhere to left spindle. If the detected torque is greater than the torque to be set, ***A-axis*** feed is stopped, executes the next block that moves to the reverse direction. That is, torque skip function will skip the block which is working and executes next block when the detected torque is greater than the set torque value. At the time, torque that generates the skip is set to D472.

(Caution) Be sure not to change the value of D472. It may cause the malfunction of machine

Note) In TT series, torque skip function checks ***B-axis*** torque instead of A-axis

12.3 M-code for Synchronous control & Torque skip function

(1) M-code related to synchronous control

*** M203 S_ ; Spindle forward Speed Synchronous Command**

*** M204 S_ ; Spindle reverse Speed Synchronous Command**

*** M213 S_ ; Spindle forward Phase Synchronous Command**

*** M214 S_ ; Spindle reverse Phase Synchronous Command**

*** M205 ; Spindle Synchronous Stop Command**

*** M31 ; Left Spindle by-pass**

Left chuck can be clamped/unclamped during rotating. In the state of unclamp, left spindle can be rotated. However, it is only available in the automatic operation state.

*** M131 ; Right Spindle by-pass**

Right chuck can be clamped/unclamped during rotating. In the state of unclamp, right spindle can be rotated. However, it is only available in the automatic operation state.

(2) M-code & G-code related to torque skip function

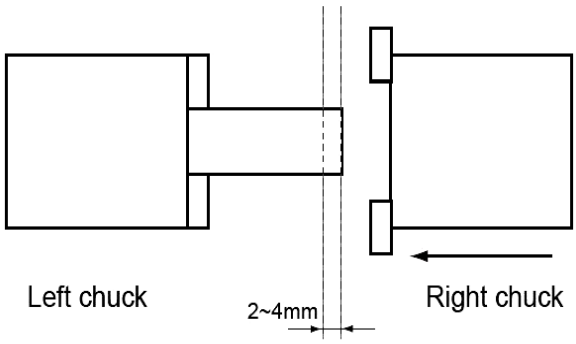
*** M86 ; Lower A-axis Torque skip function Act**

Make the torque skip function of A-axis available. Mainly it is used just before running G-code “G31” of the torque skip function

*** M87 ; Lower A-axis Torque skip function Cancel**

Make the torque skip function of A-axis unavailable. Mainly it is used after completing the workpiece transfer between left spindle and right spindle, that is, just after running G-code “G31” of the torque skip function.

*** G31 P98 A_ F_ ; Torque limit skip**



If A-axis is carried the load (torque) more than the setting value during moving to the commanded point (A_) at the commanded feed (F_), stops the movement and executes the next block.

Normally “A_” is set more than 2~4mm inside comparing to the actual adhesion point between left chuck and right spindle

G31 P98 : torque check

G31 P99 : skip signal check in measuring probe

[Caution]

- The torque skip function is available only in lower program and never move the axis by G00(Rapid traverse) after commanding “M86”(torque skip function Act). The torque value in rapid acceleration/deceleration may be greater than the torque value which is occurred by torque skip, this may cause the malfunction of machine. Be sure to operate the rapid traverse only after inactivating the torque skip function(M87).

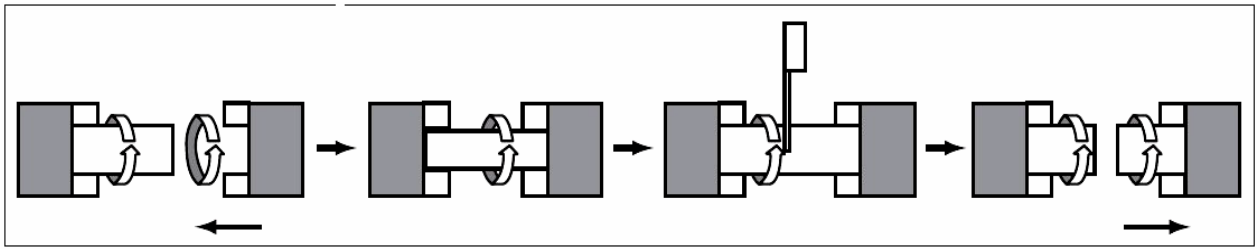
- Move the A-axis using M86 and G31 after commanding M203(M204) or M213(M214), and if the spindle override is too low when using the torque skip function, A-axis may not be moved because the spindle speed reached signal is not confirmed. At the time, check the state of spindle override.

12.4 Program methods using Synchronous control & Torque skip function

There are 4 kinds of example for the machining program that transfer the workpiece from spindle to right spindle as follows.

(1) Method-1

Synchronizing left spindle and right spindle, clamp the workpiece to both chuck while both spindles are rotating, Cut the center of workpiece, transfer the workpiece to right spindle. This can be used that the workpiece continuously supplied to left chuck in the bar type

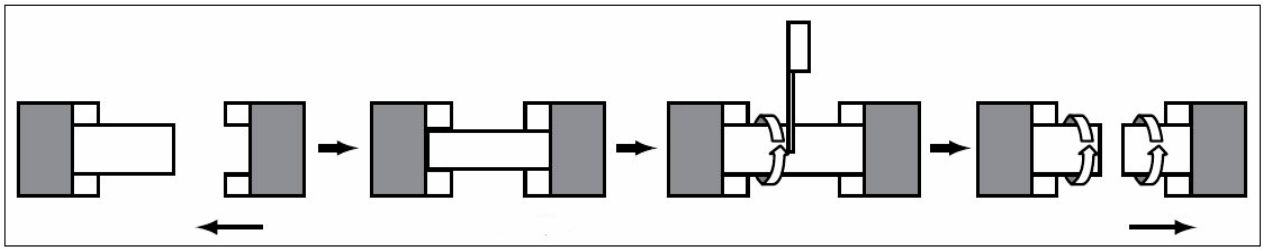


< LOWER PROGRAM >

```
M131          ; Rotate right spindle under the unclamp state of right chuck.
M169          ; Unclamp right chuck.
M203S_        ; Synchronizing right spindle and left spindle rotate to CW direction.
G00 A_.       ; Move right spindle to the front of left spindle using A-axis.
M86           ; Activate the torque skip function.
G31 P98 A_. F_  ; A-axis is moved by the commanded feed, left spindle and right
                  spindle is adhered, and if A-axis is carried the load (torque) more
                  than the setting value, stops the movement of A-axis and executes
                  the next block
M87           ; Make the torque skip function unavailable.
M168          ; Clamp right chuck.
M31           ; Left chuck can be unclamped during the rotation of left spindle.
M69           ; Unclamp left chuck.
G00 A_.       ; Move A-axis for cutting the workpiece.
M68           ; Clamp left chuck.
G01 X_. Z_.   ; Cut the workpiece.
```

(2) Method-2

Clamp the workpiece to both chuck while both spindles are stopped. And then synchronizing left spindle and right spindle, cut the center of workpiece, and transfer the workpiece to right spindle. This can be used that the workpiece continuously supplied to left chuck in the bar type

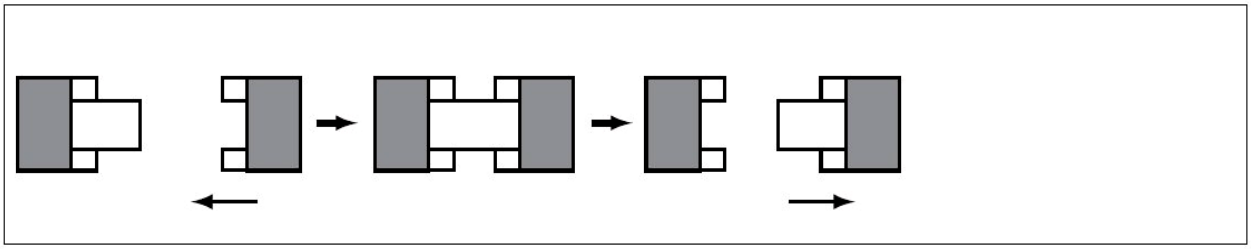


< LOWER PROGRAM >

```
M131          ; Rotate right spindle under the unclamp state of right chuck.
M169          ; Unclamp right chuck.
G00 A_.       ; Move right spindle to the front of left spindle using A-axis.
M86           ; Activate the torque skip function.
G31 P98 A_. F_  ; A-axis is moved by the commanded feed, left spindle and right
                  spindle is adhered, and if A-axis is carried the load (torque) more
                  than the setting value, stops the movement of A-axis and executes
                  the next block
M87           ; Make the torque skip function unavailable.
M168          ; Clamp right chuck.
M31           ; Left chuck can be unclamped during the rotation of left spindle.
M69           ; Unclamp left chuck.
G00 A_.       ; Move A-axis for cutting the workpiece.
M68           ; Clamp left chuck.
M203S_        ; Synchronizing left spindle and right spindle, rotate to CW direction.
G01 X_. Z_.    ; Cut the workpiece.
```

(3) Method-3

Right spindle is moved to left spindle when both spindles are stopped and the workpiece is transferred to right spindle. Spindle synchronous function is not used.

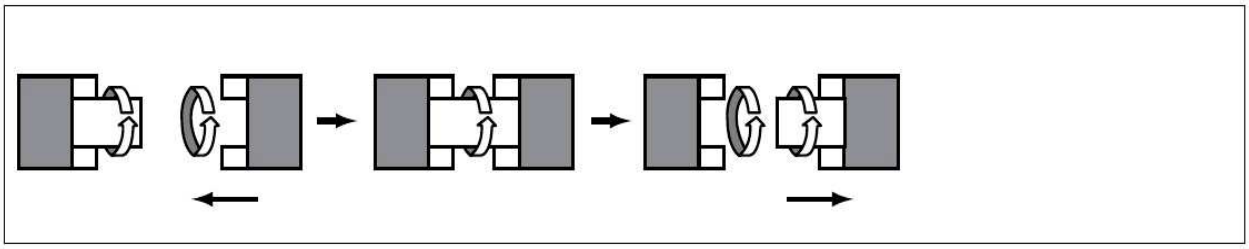


< LOWER PROGRAM >

```
M131            ; Rotate right spindle under the unclamp state of right chuck.
M169            ; Unclamp right chuck.
G00 A_.         ; Move right spindle to the front of left spindle using A-axis.
M86             ; Activate the torque skip function.
G31 P98 A_. F_  ; A-axis is moved by the commanded feed, left spindle and right
                  spindle is adhered, and if A-axis is carried the load (torque) more
                  than the setting value, stops the movement of A-axis and executes
                  the next block
M87             ; Make the torque skip function unavailable.
M168            ; Clamp right chuck.
M31             ; Left chuck can be unclamped during the rotation of left spindle.
M69             ; Unclamp left chuck.
G00 A_.         ; Move A-axis for next cutting
```

(4) Method-4

Synchronizing left spindle and right spindle, right spindle is moved to left spindle while both spindles are rotating, and the workpiece is transferred to right spindle.



< LOWER PROGRAM >

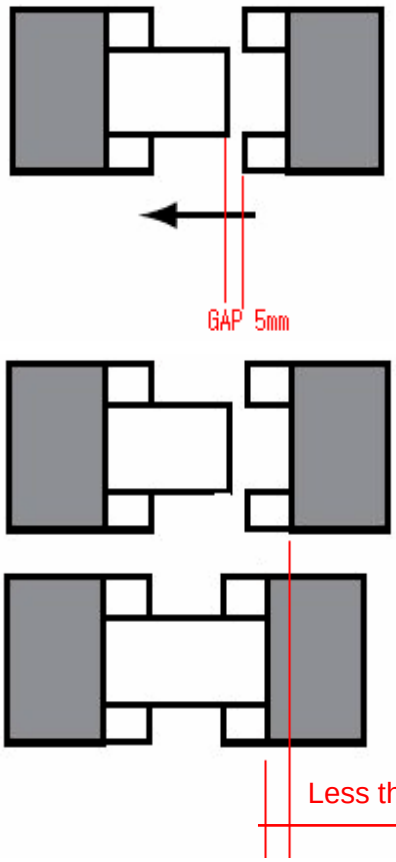
```
M131          ; Rotate right spindle under the unclamp state of right chuck.
M169          ; Unclamp right chuck.
M203S_        ; Synchronizing left spindle and right spindle, rotate to CW direction.
G00 A_.       ; Move right spindle to the front of left spindle using A-axis.
M86           ; Activate the torque skip function.
G31 P98 A_. F_ ; A-axis is moved by the commanded feed, left spindle and right
                spindle is adhered, and if A-axis is carried the load (torque) more
                than the setting value, stops the movement of A-axis and executes
                the next block
M87           ; Make the torque skip function unavailable.
M168          ; Clamp right chuck.
M31           ; Left chuck can be unclamped during the rotation of left spindle.
M69           ; Unclamp left chuck.
G00 A_.       ; Move A-axis for next cutting.
```

12.5 Program examples using Synchronous control & Torque skip function

12.5.1 Program examples of MX series

(1) MX example (G28H0)

< UPPER PROGRAM >
N500 (TRANSFER)
M170 (INTER. CHECK OFF)
G98G80G40
G28U0W0V0
G0B-90.
M?(WAITING 1)
M05
M14(AIR BLOWER)
M35
M?(WAITING 2)
M90
G28H0
M89
M?(WAITING 3)
M68
M01



< LOWER PROGRAM >
N500 (TRANSFER)
G28U0
G28W0
T?(VACANT POT)
G0W?(G30W0)
M?(WAITING 1)
M105
M114(AIR BLOWER)
M135
M?(WAITNG 2)
M190
G28H0
M189
M170 (INTERFERENCE CHECK RELEASE)
M87(TORQUE SKIP CANCEL)
G28A0
M131(R-SPD. INTER.BYPASS)
G98G0A?M169 (APPROACH)
G04U1.0
M86(TORQUE SKIP ON)
M254 (A-AXIS INTER.IGNORE)
G01G31P98A?F500 (ADV.)
M168 (R-CHUCK CLAMP)
G04U1.5
M69 (L-CHUCK UNCLAMP)
G04U1.0
G98G01A?F8000.(RET.)
M87 (TORQUE SKIP CNACEL)
G28A0
M255
M? (WAITING 3)
G28U0W0
M01

(2) MX example (M214)

```
< UPPER PROGRAM >
N500 (TRANSFER)
G98G80G40
G28U0W0V0
G0B-90.
M?(WAITING 1)
M05
M?(WAITING 2)
M05
M?(WAITING 3)
M01
```

```
< LOWER PROGRAM >
N500 (TRANSFER)
G98G80G40
G28U0W0
T?(VACANT POT)
G0W?(G30W0)
M?(WAITING 1)
M105
M170
M87  (TORQUE SKIP CANCEL)
G28A0
M131 (R-SPD.INTER.BYPASS)
M214S600 (SYNCH. REV.)
G98G0A?M169 (APPROACH)
G04U1.0
M86   (TORQUE SKIP ACT)
M254 (A-AXIS INTER.IGNORE)
G01G31P98A?F500 (ADV.)
M168 (R-CHUCK CLAMP)
G04U1.5
M31  (LT SPD.INTER.BYPASS)
M69  (L-CHUCK UNCLAMP)
G04U1.0
G98G01A?F8000.(RET)
G0A? (RET.)
M68
M215 (SYN. CONTROL STOP)
G28A0
M87 (TORQUE SKIP CANCEL)
G28U0
G28W0
M? (WAITING 2)
M105
M? (WAITING 3)
M01
```

(3) MX example (M214 & Parting)

```
< UPPER PROGRAM >
N500 (TRANSFER)
G98G80G40
G28U0W0V0
G0B-90.
M?(WAITING 1)
M05
M?(WAITING 2)
M05
M?(WAITING 3)
M05
G99
M?(WAITING 4)
M01
```

```
< LOWER PROGRAM 1/2  >
N500 (TRANSFER)
G98G80G40
G28U0W0
T?(VACANT POT)
G0W?(G30W0)
M?(WAITING 1)
M105
M170
M87  (TORQUE SKIP CANCEL)
G28A0
M131 (R-SPD.INTER.BYPASS)
M214S600 (SYNCH. REV.)
```

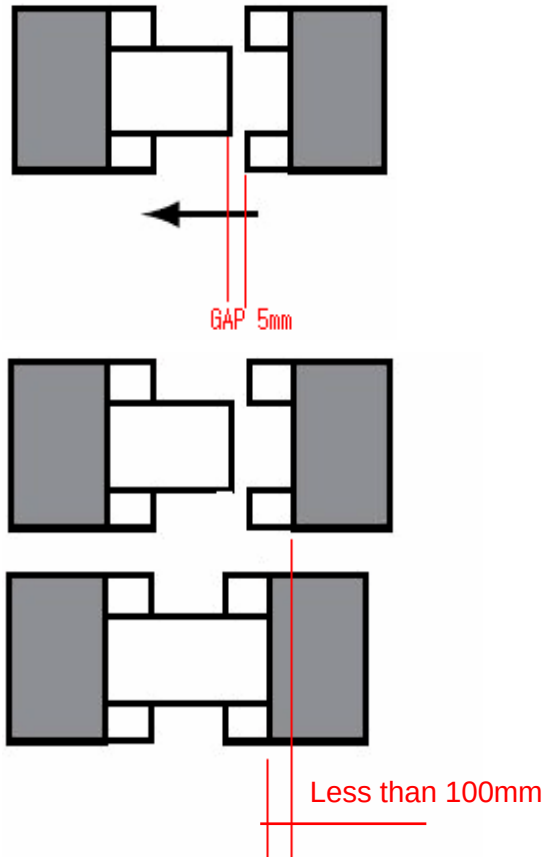
```
< LOWER PROGRAM  2/2 >
G98G0A?M169 (APPROACH)
G04U1.0
M86   (TORQUE SKIP ACT)
M254 (A-AXIS INTER.IGNORE)
G01G31P98A?F500 (ADV.)
M168 (R-CHUCK CLAMP)
G04U1.5
M31  (LT SPD.INTER.BYPASS)
M69  (L-CHUCK UNCLAMP)
G04U1.0
G98G01A?F8000.(RET)
M68  (L-CHUCK CLAMP)
M215 (SYN.CONTROL STOP)
M? (WAITING 2)
M105
M? (WAITING 3)
M217(Z2 A-ZXIS INTER. OFF)
G04U1.0
G54
G99
G0Z0.T?M08(PARTING TOOL)
G0X100.0  (APPROACH)
M204S? (SYNCH. REV.)
...
( PARTING PROGRAM )
...
G0X130.0M09
G28U0.
M205 (SYN.CONTROL START)
G28A0 (RET.)
M87 (TORQUE SKIP CANCEL)
G28U0
G28W0
M171 (INTER. CEHCK ON)
M105
M? (WAITING 4)
M01
```

12.5.2 Program examples of TT series

(1) TT example (G28H0)

< UPPER PROGRAM >
N500 (TRANSFER)
M?(WAITING 1)
G28U0W0V0
M35
T0100
M?(WAITING 2)
G28H0
G30U0W0V0
M?(WAITING 3)
M14
M170(INTER. CHECK OFF)
M131(U-RT SPDL INTERLOCK BYPASS SPDL T/S)
M169(U-RT CHUCK UNCLAMP)
G98G0B-750.0(DIA 40.)
M86(U-TORQUE LIMIT SKIP ON)
G01G31P98B-800.0F800 (TORQUE LIMIT SKIP)
G04U1.5
M168(U-RT CHUCK CLAMP)
M87(U-TORQUE SKIP CANCEL)
G4U1.0
M31(U-LT SPDL INTERLOCK BYPASS SPDL T/S)
M69(U-LT CHUCK UNCLAMP)
G28B0.0
M68(U-LT CHUCK CLAMP)
M?(WAITING 4)
G28U0W0V0
M171(INTER.CHECK ON)
M01

< LOWER PROGRAM >
N500 (TRANSFER)
M?(WAITING 1)
G28U0W0
M35
T1200
M?(WAITING 2)
G28H0
G30U0W0
M?(WAITING 3)
M114
M?(WAITING 4)
G28U0W0
M01



(2) TT example (M214 & parting)

< UPPER PROGRAM >
N500 (TRANSFER)
G28U0.0W0.0
G54
M87(U-TORQUE LIMIT SKIP CANCEL1)
M131(U-RT SPINTERLOCK BY PASS SPDL T/S)
M169(U-RT CHUCK UNCLAMP)
M214S100(U-SYN. CONTROL REVERSE)
G98G0B**-681.0**(DIA 30)
M86(U-TORQUE LIMIT SKIP ON)
G01G31P98B**-691.0**F800
M168(U-RT CHUCK CLAMP)
M87(U-TORQUE LIMIT SKIP OFF)
G4U1.
M31(U-LT SPINTERLOCK BY PASS SPDL T/S)
M69(U-LT CHUCK UNCLAMP)
G00**B-575.0**(MOVE+106.0)
M68(U-LT CHUCK CLAMP)
M215(U-SYN. CONTROL STOP)
M05
M**?**(WAITING 1)
M**?**(WAITING 2)
G28U0.0W0.0
G28B0.0
M**?**(WAITING 3)
G28U0W0
M01

< LOWER PROGRAM >
N500 (TRANSFER)
T1000
M**?**(WAITING 1)
M217(L-Z2 INTERLOCK OFF)
G04U1.0
G54
G99
G00Z0.0T1010M8
X130.0
X50.0
M214S600(L-SYN. CONTROL REVERSE)
G1X25.0F0.12
X50.0F3.0
X30.0F0.3
X15.0F0.12
X30.0F3.0
X20.0F0.3
X3.0F0.1
X5.0F3.0
X-2.0F0.05
X50.0F3.0
G0X130.M9
G28U0.0
M215(L-SYN. CONTROL STOP)
M**?**(WAITING 2)

M**?**(WAITING 3)
T1000
G28U0W0M215
M01

13. Tilting B-axis Function (MX series)

13.1 Automatic milling spindle orientation (G490)

Upper milling head spindle consists of curvic-coupling on front face.

The indexing of milling spindle is possible every 30˚ .

MX series has two functions in commanding for milling spindle orientation.

The one function(G490) is automatically milling spindle unclamped and milling spindle is oriented to designated degree . After orientation, milling spindle is automatically clamped.

The other function (M49) is needed to command unclamping (M101) for milling spindle before commanding orientation and to command clamping(M100) after commanding orientation . But this is only for maintenance by the engineer of machine builder.

(1) Command format

G490 S__

G490 : automatic milling spindle orientation command

S__ : orientation degree of milling spindle

S0 → 0˚

S300 → 30˚

S1800 → 180˚

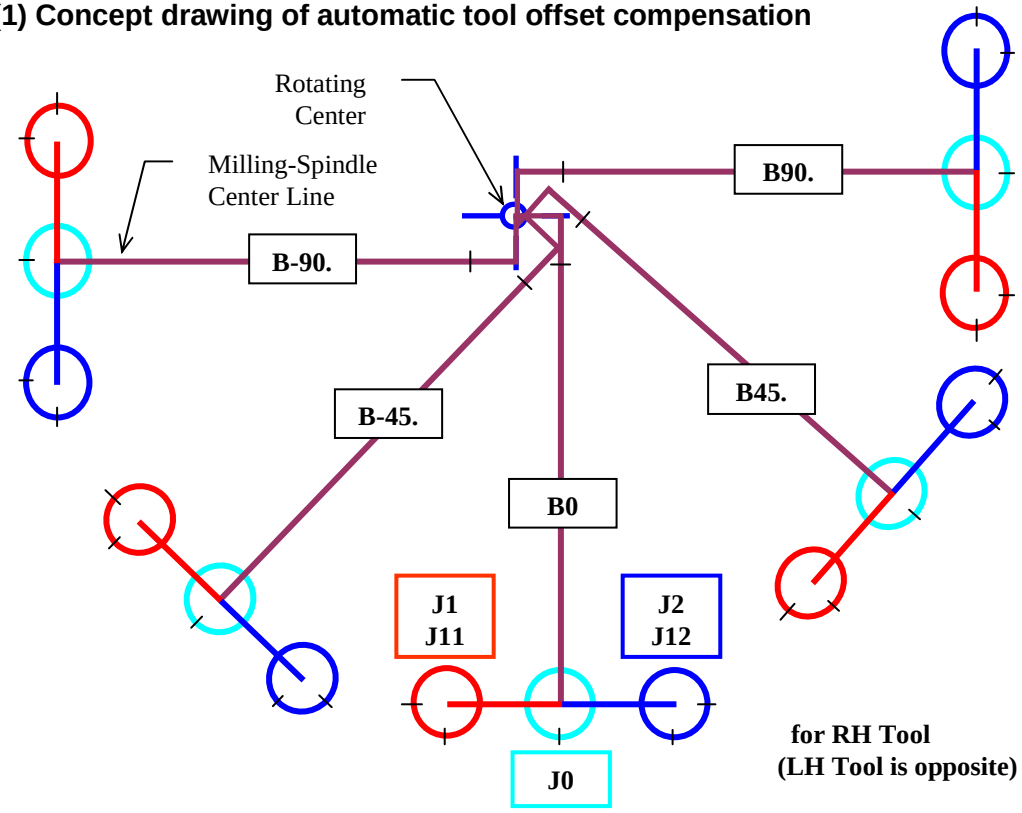
S3600 → 360˚

13.2 Automatic tool offset compensation (G400)

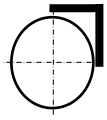
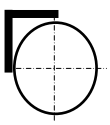
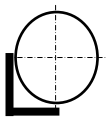
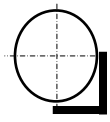
MX series has a automatic tool offset compensation function.

If we measure a tool offset at B axis 0˚, after command G400, tool offset no. is changed automatically and is saved in geometry offset no.399 and wear offset no. is saved in wear offset no.399.

(1) Concept drawing of automatic tool offset compensation

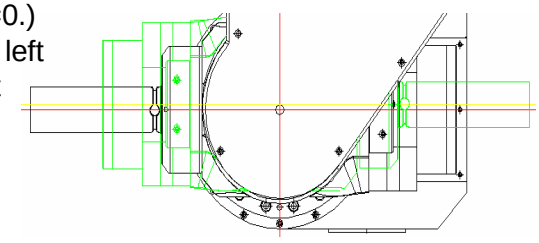


Orientation degree	J0.=0˚+M101	J1.=0˚	J2.=180˚	J11.= ω˚	J12.= ω˚+180˚

Tool type	Milling tool	Turning Tool			
Cutting quadrant		R1.	R2.	R3.	R4.
The direction of cutting Edge	-				

(2) B axis center offset value

When B-axis is moved to the spindle center line (X=0.) it has to be along the center line of the right and the left spindle. D428 is the center offset value between left and right spindle.



Ex) IF measured value by indicator is 0.1

→ New type : D428 = 50 , Macro #600 = 50

→ Old type : D428 = 0.05, Macro #600 = 50

(3) Command format

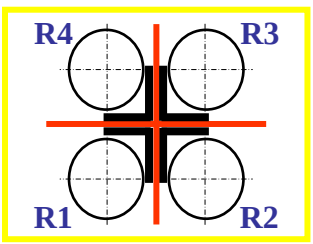
T□□ ●●●
G400 B_. J_. R_. K_. W_.

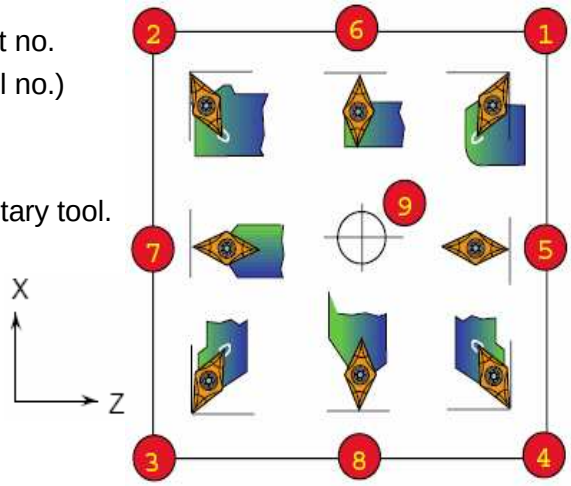
□□ : current tool no. in milling spindle.
●●● : geometry tool offset no. measured at B-axis 0˚
B : B-axis degree
J : clamped status of Milling spindle and orientation degree
J0 → Milling spindle becomes unclamped.
J1 → Milling spindle becomes oriented to 0˚and clamped.
J2 → Milling spindle becomes oriented to 180˚and clamped.
J11 → current degree (ω˚) of milling spindle for Mini-turret
J12 → current degree (ω˚)+180˚of Milling spindle for Mini-turret
ω˚means current degree of Milling spindle
R : the no. of cutting quadrant (1~4)
K : imaginary tool nose no. (0~9)
W : wear offset no.

Note 1) J11 and J12 are for mini-turret tool and before commanding J11 (J12), milling spindle orientation (G490S_) must be commanded. In case of not commanding G490S_ , milling spindle will be oriented S0.

G490S1200
G400 B__. J11. R__. W__.
……
G400 B__. J12. R__. W__.
……
G490S1200
G400 B__. J11. R__. W__.

Note 2) After command G400, new tool offset no. is T□□399. (□□ marks means Tool no.)
Note 3) after command G400 In milling tool, command M101 (spindle unclamp)
Note 4) R,K and W value are omissible for rotary tool.

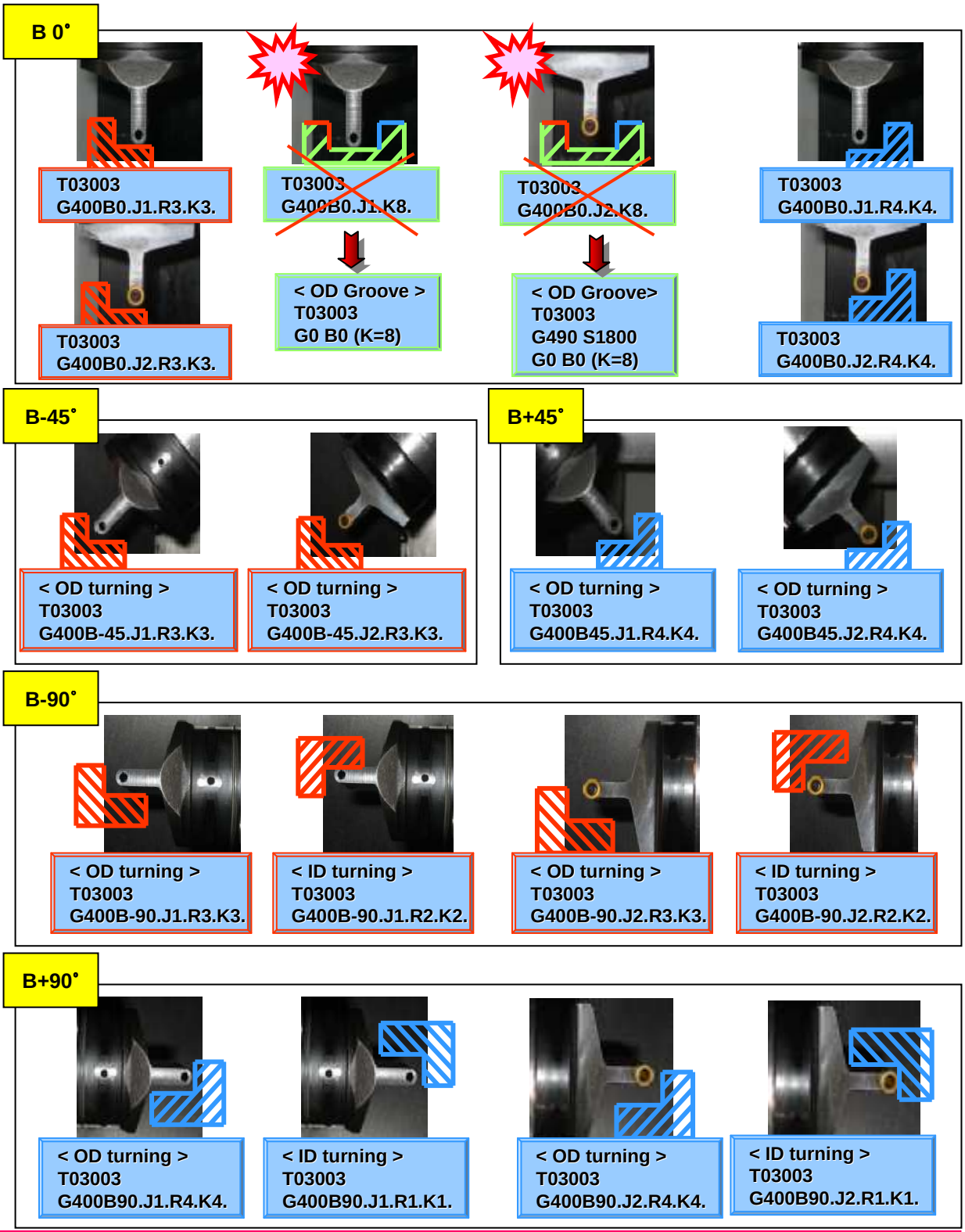




13.3 program examples of G400

13.3.1 Example for R5 groove tool

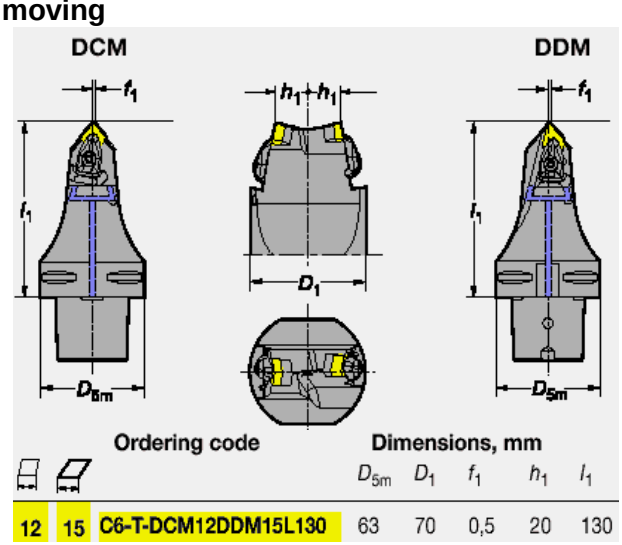
Current Head tool no. : 3 / Geometry Offset at B-axis 0˚ : 8 (G10003 X410.0 Z0. R5.0 Q8.)

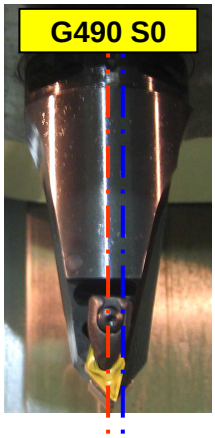


13.3.2 Tool offset for twin tool with Y axis moving

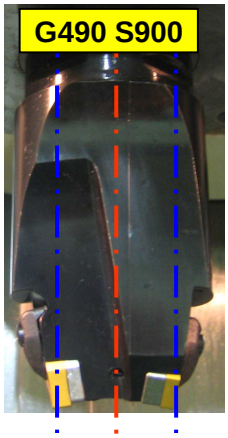
(1) Tool specification :

C6-T-DCM12DDM15L130 (SANDVIK)
Rough insert : CNMG 120408 PM4025
Finish insert : DNMG 150604 PF4045

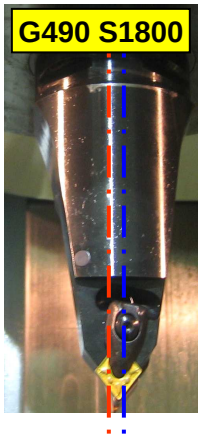




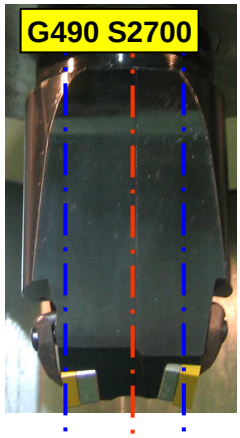
f1=0.5
(bottom-CNMG)
(top – DNMG)



h1=20. (DNMG)
h1=20. (CNMG)



f1=0.5
(bottom-DNMG)
(top –CNMG)



h1=20. (CNMG)
h1=20. (DNMG)

(2) Measuring the DNMG insert (T03004)

① G490 S900 (Y=0)

② Q-setter down

③ measuring X

OFFSET / GEOMETRY

NO.	X	Z	R	T
G 003	659.446	0.000	0.000	0
G 004	0.000	0.000	0.000	0

④ input Z value(f1=0.5) and nose R on offset screen

OFFSET / GEOMETRY

NO.	X	Z	R	T
G 003	659.446	-0.500	0.000	0
G 004	0.000	0.000	0.000	0

⑤ move the data of finish insert to G004

OFFSET / GEOMETRY

NO.	X	Z	R	T
G 003	0.000	0.000	0.000	0
G 004	659.446	-0.500	0.400	0

(3) Measuring the CNMG insert (T03003)

① G490 S2700 (Y=0)

② Q-setter down

③ measuring X

OFFSET / GEOMETRY

NO.	X	Z	R	T
G 003	659.406	0.000	0.000	0
G 004	659.446	-0.500	0.400	0

④ input Z value(f1=0.5) and nose R on offset screen

OFFSET / GEOMETRY

NO.	X	Z	R	T
G 003	659.406	-0.500	0.800	0
G 004	659.446	-0.500	0.400	0

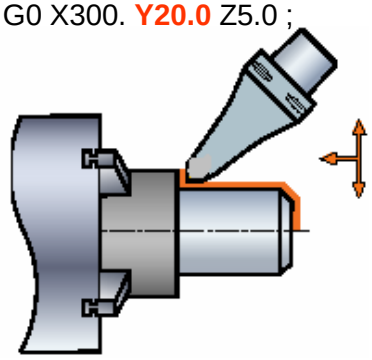
(4) example

< CNMG tool (M03) >

T03003 ;

G400 B-45. **J1**. R3. ;

G0 X300. **Y20.0** Z5.0 ;

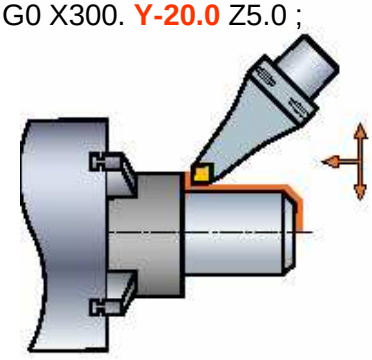


< CNMG tool (M04) >

T03003 ;

G400 B-45. **J2**. R3. ;

G0 X300. **Y-20.0** Z5.0 ;

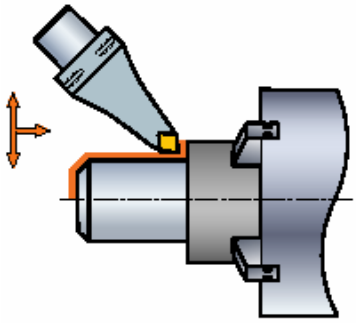


< CNMG tool (M103) >

T03003 ;

G400 B45. **J2**. R4. ;

G0 X300. **Y-20.0** Z-5.0 ;

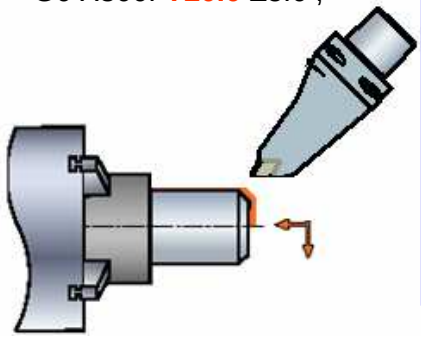


< DNMG tool (M03) >

T03004 ;

G400 B-45. **J2**. R3. ;

G0 X300. **Y20.0** Z5.0 ;

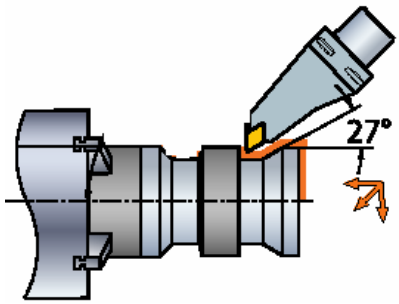


< DNMG tool (M04) >

T03004 ;

G400 B-45. **J1**. R3. ;

G0 X300. **Y-20.0** Z5.0 ;

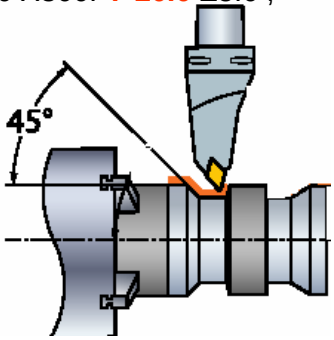


<DNMG tool (M04) >

T03004 ;

G400 B0. **J1**. R4. ;

G0 X300. **Y-20.0** Z5.0 ;



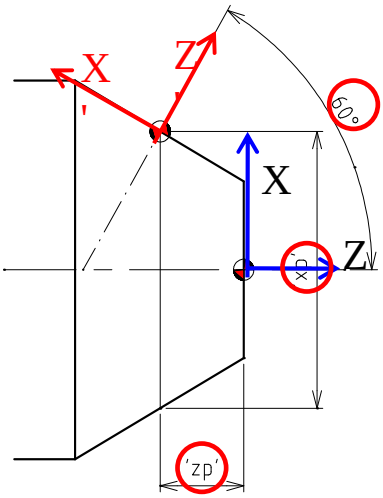
13.4 Three-dimensional coordinate conversion

13.4.1 three-dimensional coordinate conversion without B-axis rotating (G68.1/G69.1)

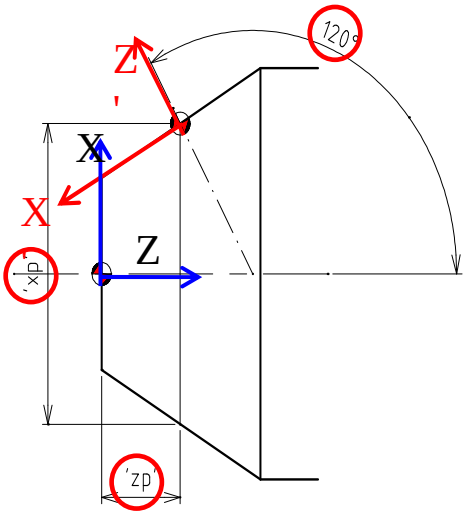
(1) Command format

```
G68.1  X_ Y_ Z_ I0 J1 K0 R_
.....
G69.1
```

X_ Y_ Z_ : center of rotation(absolute coordinates) on the X,Y and Z axis or parallel axes
I_ J_ K_ : direction of the axis of rotation
1 → the axis of rotation
R_ : Angular displacement
G69.1 : 3D coordinate conversion cancel



G68.1XxpY0ZzpI0J1K0R60.0



G68.1XxpY0ZzpI0J1K0R120.0

(2) Format example

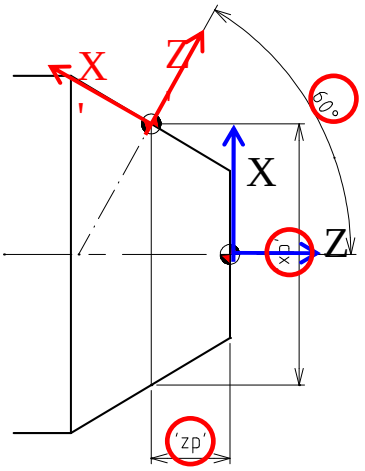
```
T01000          (Tool offset Cancel)
G00 B30.0       (B-axis Rotation)
M101
G97S3000M33
G68.1 X95.0Y0.0Z29.689I0J1K0R120.0 (3D-mode Act)
T01001          (Tool offset Call)
G00X0.0Y0.0Z50.0 (Approach Processing)
  .....
T01000          (Tool offset Cancel)
G69.1           (3D-mode Cancel)
```

13.4.2 Three-dimensional coordinate conversion with B-axis rotating (G368/G369)

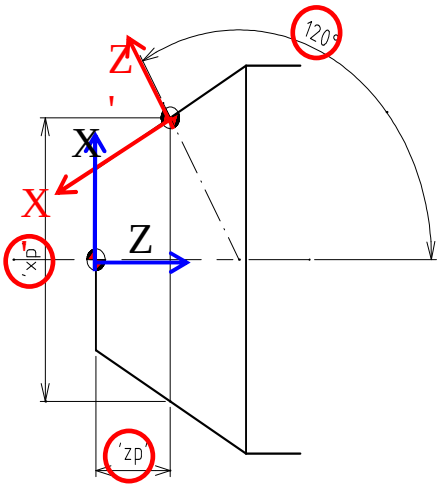
(1) Command format

```
G368 X_ Z_ D_ B_
.....
G369
```

X_ Z_	: center of rotation(absolute coordinates) on the X and Z axis or parallel axes
D_	: direction of feed axis. D0 → Z-axis direction / D1 → X-axis direction
R_	: rotation angle of actual B-axis
G369	: 3D coordinate conversion cancel



G368XxpZzpD0.B-30.0



G368XxpZzpD0B30.0

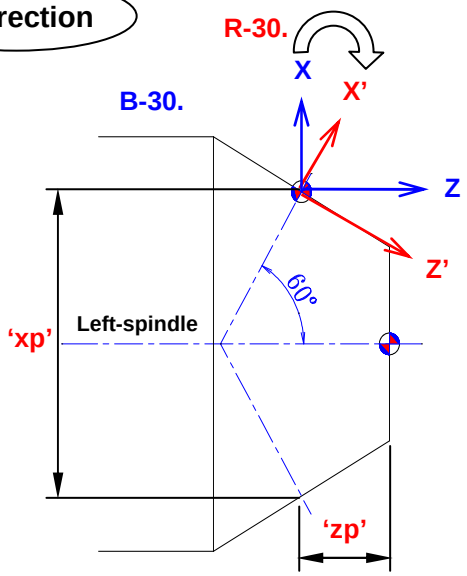
(2) Format example

T01001 (Tool offset Call)
M101
G97S3000M33
G368 X95.0Z29.689 D0.B30.0 (3D-mode Act)
G00X0.0Y0.0Z50.0 (Approach Processing)
....
G369 (3D-mode Cancel)

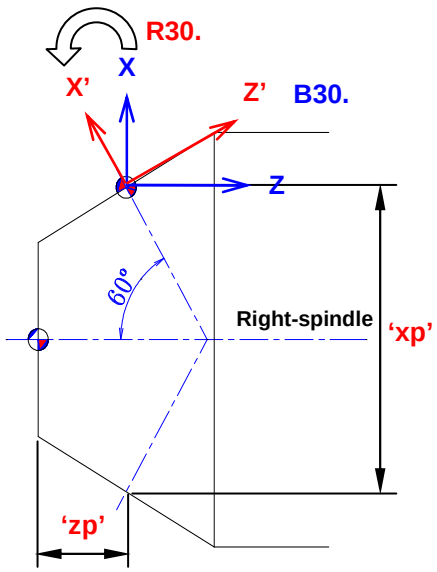
Note 1) G368 includes G400 function according to the D0/D1 with G68.1
G369 includes G28 and Tool offset cancel function with G69.1
ex) G369 => G69.1; G28 U0 W0 V0 M101;
G369 includes Live tool rotation stop and G490S0
Note 2) Tool offset conversion
G400 ⇒ Geometry offset 399 / G368 ⇒ Geometry offset 400

13.4.3 3D coordinate conversion along the direction of cutting feed axis

X-direction

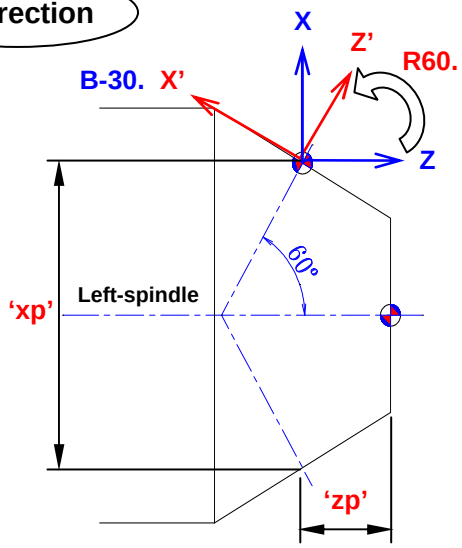


[Left-spindle]
G68.1 X**xp**Y0Z**zp**I0J1K0**R-30.0**
G368 X**xp**Z**zp**D**1.**B**-30.**

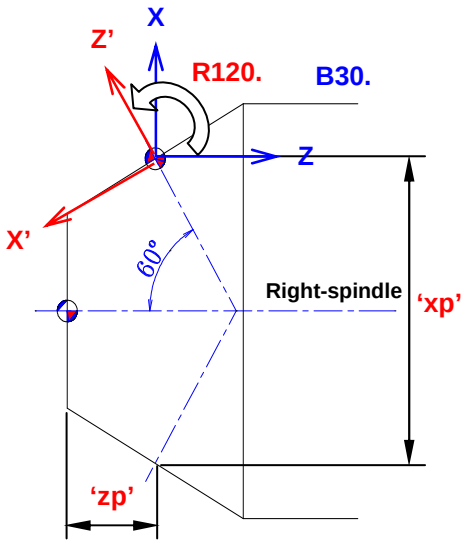


[Right-spindle]
G68.1 X**xp**Y0Z**zp**I0J1K0**R30.0**
G368 X**xp**Z**zp**D**1.**B**30.**

Z-direction



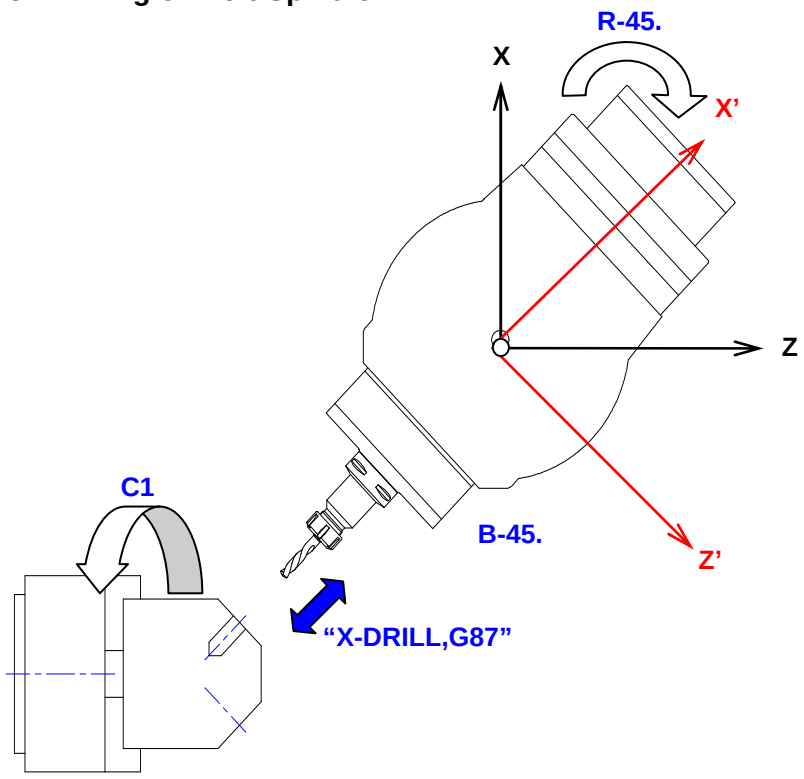
[Left-spindle]
G68.1 X**xp**Y0Z**zp**I0J1K0**R60.0**
G368 X**xp**Z**zp**D**0.**B**-30.**



[Right-spindle]
G68.1 X**xp**Y0Z**zp**I0J1K0**R120.0**
G368 X**xp**Z**zp**D**0.**B**30.**

13.5 Program examples for 3D Coordinate conversion

13.5.1 X-direction Drilling on Left-Spindle




```
***G68.1/G69.1***
N1(X-DRILL,G87@B-45. ON LS)
T01000
G28U0.V0.W0.
G54G99G80G40M5
M6T01000
G28U0.V0.M61(LS-LOW WINDING)
M35(C1 SELECT)
M90
G28H0.
G50C0.
M101(TOOL SP. UNCLAMP)
G97S1000M33
G0B-45.
(3D COORDINATE ROT.MODE)
G68.1X200.Z0.I0.J1.K0.R-45.
T01001
G0X20.Z0.C0.Y-10.
M289(SELECT C1 CLAMP CONTROL)
G87X-50.R-5.Q15000F0.5M89
C90.Q15000
C180.Q15000
C270.Q15000
G80M90
T01000
G69.1
M35
G28U0.V0.W0.
G28B0
M1
```

```
***G368/G369***
N1(X-DRILL,G87@B-45. ON LS)
T01000
G28U0.V0.W0.
G54G99G80G40M5
M6T01000
G28U0.V0.M61(LS-LOW WINDING)
M35(C1 SELECT)
M90
G28H0.
G50C0.
M101(TOOL SP. UNCLAMP)
G97S1000M33
T01001
(G0 B-45.)
(3D COORDINATE ROT.MODE)
G368X200.Z0.D1.B-45.
G0X20.Z0.C0.Y-10.
M289(SELECT C1 CLAMP CONTROL)
G87X-50.R-5.Q15000F0.5M89
C90.Q15000
C180.Q15000
C270.Q15000
G80M90
M35
G369
G28U0.V0.W0.
G28B0
M1
```

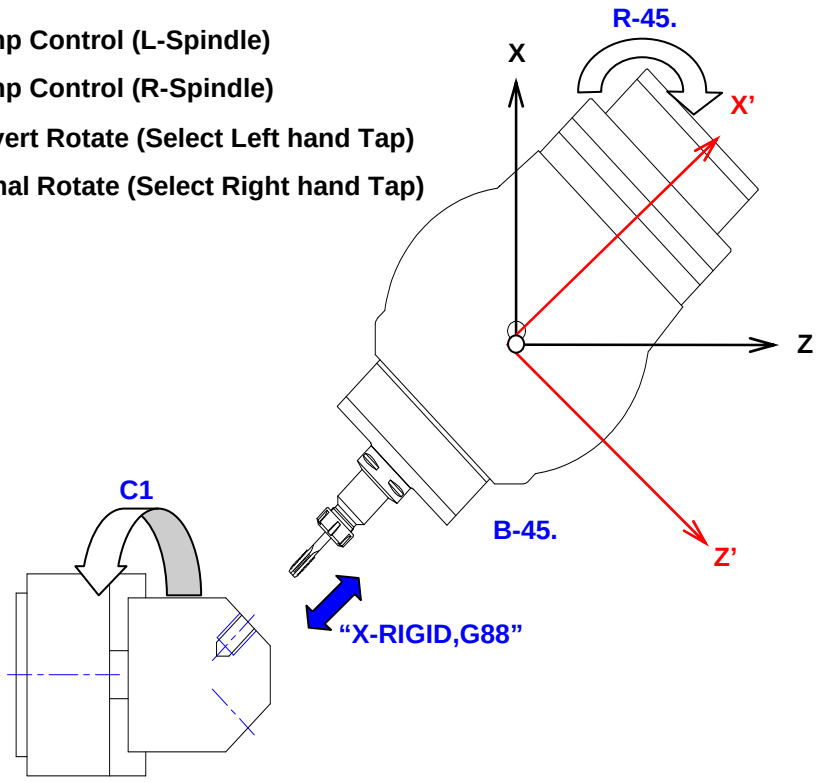
13.5.2 X-direction Tapping on Left-Spindle

***M289: Select C1 Clamp Control (L-Spindle)**

***M389: Select C2 Clamp Control (R-Spindle)**

***M72 : Live Tool Convert Rotate (Select Left hand Tap)**

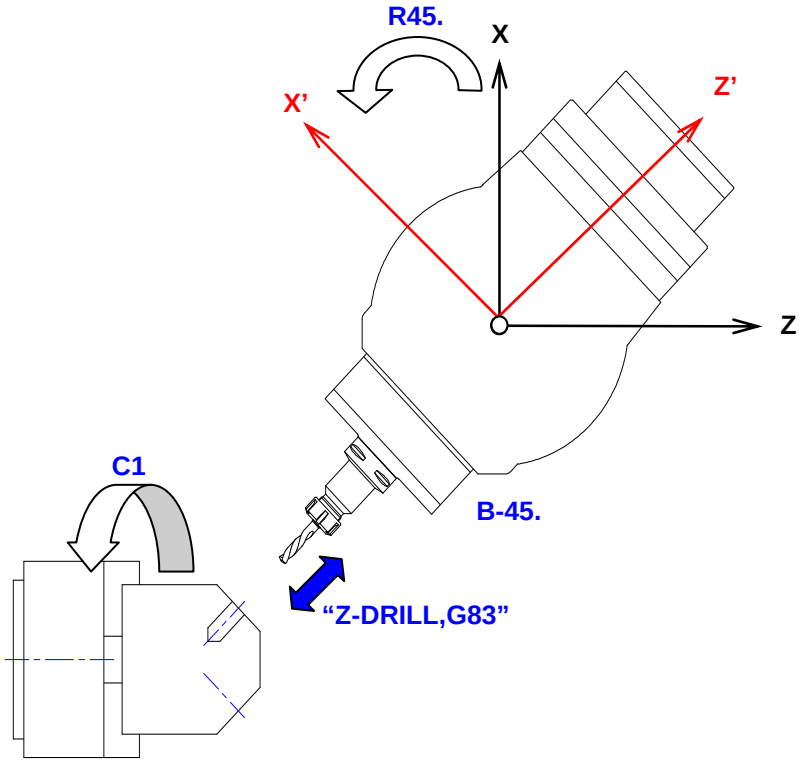
***M73 : Live Tool Normal Rotate (Select Right hand Tap)**




```
***G68.1/G69.1***
N2(X-RIGID,G88@B-45. ON LS)
T02000
G28U0.V0.W0.
G54G99G97G80G40M5
M6T02000
G28U0.V0.M61(LS-LOW WINDING)
M35(C1 SELECT)
M90
G28H0.
G50C0.
M101(TOOL SP. UNCLAMP)
G0B-45.
(3D COORDINATE ROT.MODE)
G68.1X200.Z0.I0.J1.K0.R-45.
T02002
G0X20.Z0.C0.Y-10.
M289(SELECT C1 CLAMP CONTROL)
M29S500
M73(72=L/H, M73=R/H)
G88X-40.R-5.F1.0M89
C90.
C180.
C270.
G80M90
T02000
G69.1
M35
G28U0.V0.W0.
G28B0
M1
```

```
***G368/G369***
N2(X-RIGID,G88@B-45. ON LS)
T02000
G28U0.V0.W0.
G54G99G97G80G40M5
M6T02000
G28U0.V0.M61(LS-LOW WINDING)
M35(C1 SELECT)
M90
G28H0.
G50C0.
M101(TOOL SP. UNCLAMP)
(G0B-45.)
T02002
(3D CORDINATE ROT.MODE)
G368X200.Z0.D1.B-45.
G0X20.Z0.C0.Y-10.
M289(SELECT C1 CLAMP CONTROL)
M29S500
M73(72=L/H, M73=R/H)
G88X-40.R-5.F1.0M89
C90.
C180.
C270.
G80M90
M35
G369
G28U0.V0.W0.
G28B0
M1
```

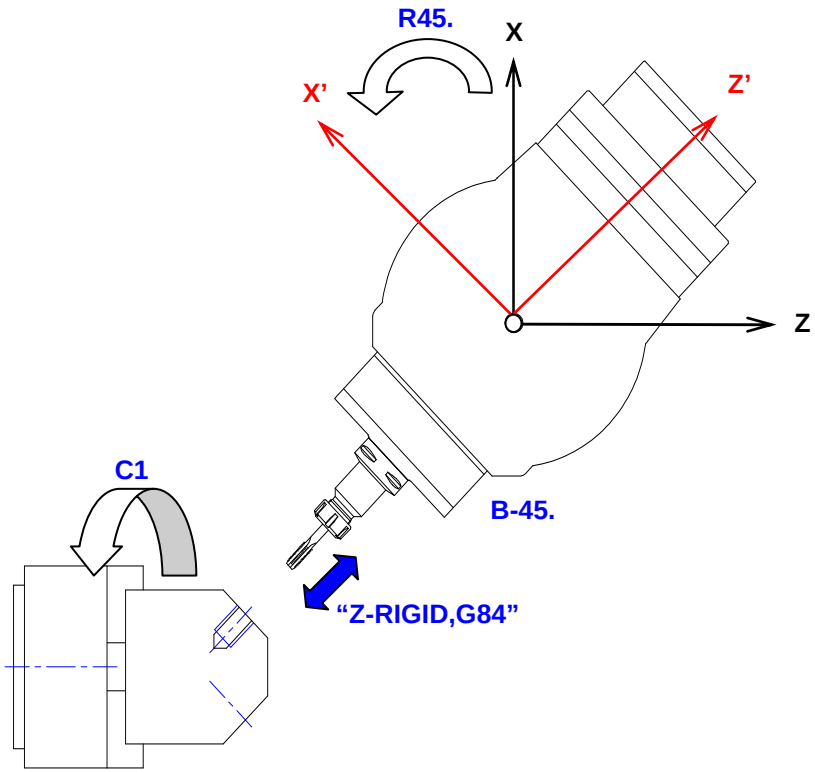
13.5.3 Z-direction Drilling on Left-Spindle




```
***G68.1/G69.1***
N3(Z-DRILL,G83@B-45. ON LS)
T01000
G28U0.V0.W0.
G54G99G80G40M5
M6T01000
G28U0.V0.M61(LS-LOW WINDING)
M35(C1 SELECT)
M90
G28H0.
G50C0.
M101(TOOL SP. UNCLAMP)
G97S1000M33
G0B-45.
(3D COORDINATE ROT.MODE)
G68.1X200.Z0.I0.J1.K0.R45.
T01001
G0X0.Z10.C0.Y-10.
M289(SELECT C1 CLAMP CONTROL)
G83Z-25.R-5.Q15000F0.5M89
C90.Q15000
C180.Q15000
C270.Q15000
G80M90
T01000
G69.1
M35
G28U0.V0.W0.
G28B0
M1
```

```
***G368/G369***
N3(Z-DRILL,G83@B-45. ON LS)
T01000
G28U0.V0.W0.
G54G99G80G40M5
M6T01000
G28U0.V0.M61(LS-LOW WINDING)
M35(C1 SELECT)
M90
G28H0.
G50C0.
M101(TOOL SP. UNCLAMP)
G97S1000M33
T01001
(G0B-45.)
(3D COORDINATE ROT.MODE)
G368X200.Z0.D0.B-45.
G0X0.Z10.C0.Y-10.
M289(SELECT C1 CLAMP CONTROL)
G83Z-25.R-5.Q15000F0.5M89
C90.Q15000
C180.Q15000
C270.Q15000
G80M90
M35
G369
G28U0.V0.W0.
M105
M1
```

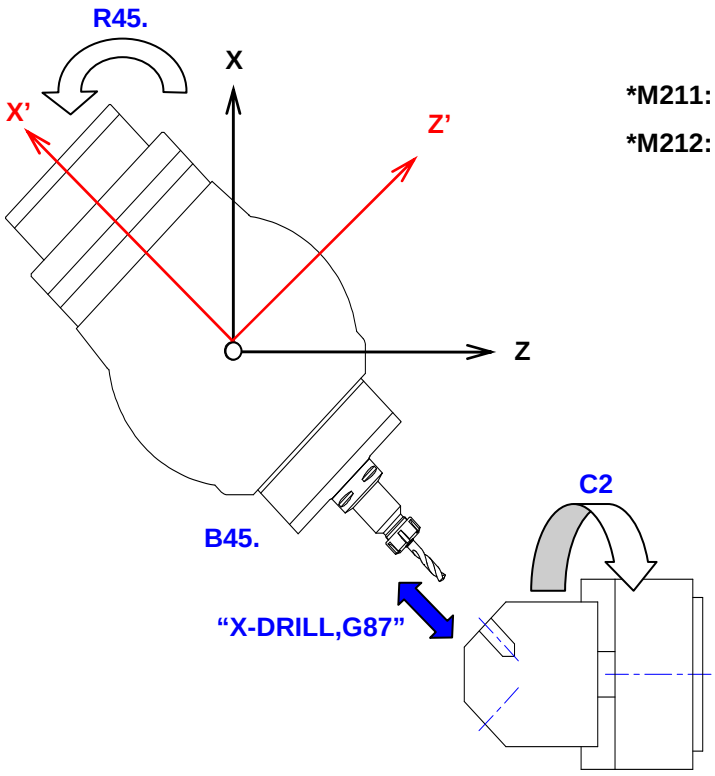
13.5.4 Z-direction Tapping on Left-Spindle




```
***G68.1/G69.1***
N4(Z-RIGID,G84@B-45. ON LS)
T02000
G28U0.V0.W0.
G54G99G97G80G40M5
M6T02000
G28U0.V0.M61(LS-LOW WINDING)
M35(C1 SELECT)
M90
G28H0.
G50C0.
M101(TOOL SP. UNCLAMP)
G0B-45.
(3D COORDINATE ROT.MODE)
G68.1X200.Z0.I0.J1.K0.R45.
T02002
G0X0.Z10.C0.Y-10.
M289(SELECT C1 CLAMP CONTROL)
M29S500
M73(72=L/H, M73=R/H)
G84Z-20.R-5.F1.0M89
C90.
C180.
C270.
G80M90
T02000
G69.1
M35
G28U0.V0.W0.
M1
```

```
***G368/G369***
N4(Z-RIGID,G84@B-45. ON LS)
T02000
G28U0.V0.W0.
G54G99G97G80G40M5
M6T02000
G28U0.V0.M61(LS-LOW WINDING)
M35(C1 SELECT)
M90
G28H0.
G50C0.
M101(TOOL SP. UNCLAMP)
(G0B-45.)
T02002
(3D CORDINATE ROT.MODE)
G368X200.Z0.D1.B-45.
G0X0.Z10.C0.Y-10.
M289(SELECT C1 CLAMP CONTROL)
M29S500
M73(72=L/H, M73=R/H)
G84Z-20.R-5.F1.0M89
C90.
C180.
C270.
G80M90
M35
G369
G28U0.V0.W0.
M1
```

13.5.5 X-direction Drilling on Right-Spindle



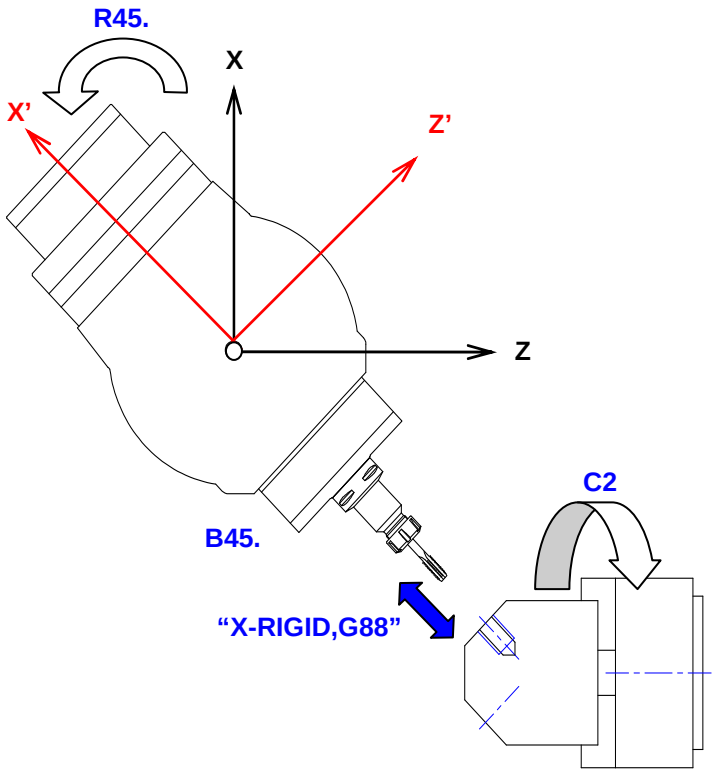
***M211: Composite Control mode Act**

***M212: Composite Control mode Cancel**

```
***G68.1/G69.1***
N5(X-DRILL,G87@B45. ON RS)
T01000
G28U0.V0.W0.
G55G99G80G40M105
M6T01000
G28U0.V0.M161(RS-LOW WINDING)
M135(C2 SELECT)
M211(COMPOSITE MODE ACT)
M190
G28H0.
G50C0.
M101(TOOL SP. UNCLAMP)
G97S1000M33
G0B45.
(3D COORDINATE ROT.MODE)
G68.1X200.Z0.I0.J1.K0.R45.
T01001
G0X20.Z0.C0.Y-10.
M389(SELECT C2 CLAMP CONTROL)
G87X-50.R-5.Q15000F0.5M89
C90.Q15000
C180.Q15000
C270.Q15000
G80M90
T01000
G69.1
M135
M212(COMPOSITE MODE CANCEL)
G28U0.V0.W0.
M1
```

```
***G368/G369***
N5(X-DRILL,G87@B45. ON RS)
T01000
G28U0.V0.W0.
G55G99G80G40M105
M6T01000
G28U0.V0.M161(RS-LOW WINDING)
M135(C2 SELECT)
M211(COMPOSITE MODE ACT)
M190
G28H0.
G50C0.
M101(TOOL SP. UNCLAMP)
G97S1000M33
T01001
(G0 B45.)
(3D COORDINATE ROT.MODE)
G368X200.Z0.D1.B45.
G0X20.Z0.C0.Y-10.
M389(SELECT C2 CLAMP CONTROL)
G87X-50.R-5.Q15000F0.5M89
C90.Q15000
C180.Q15000
C270.Q15000
G80M90
M135
M212(COMPOSITE MODE CANCEL)
G369
G28U0.V0.W0.
M1
```

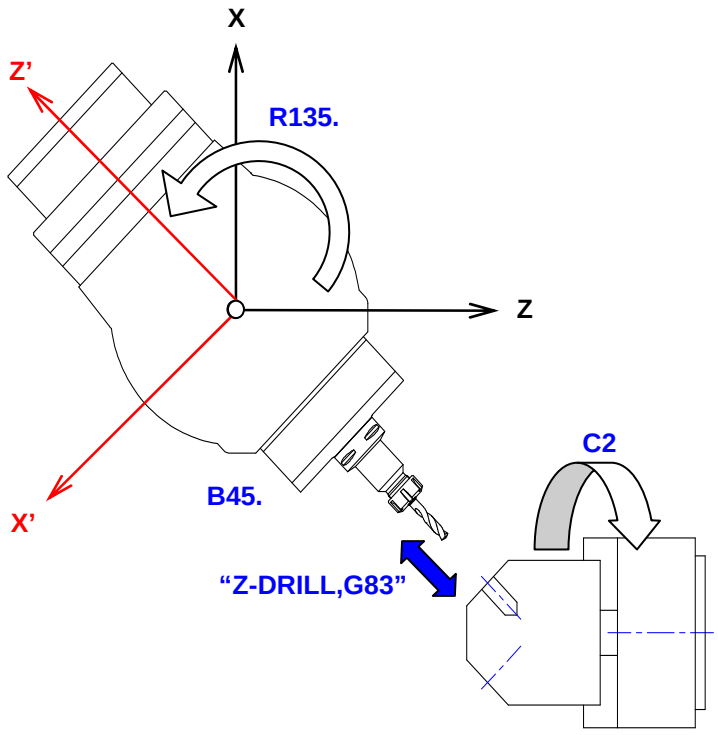
13.5.6 X-direction Tapping on Right-Spindle




```
***G68.1/G69.1***
N6(X-RIGID,G88@B45. ON RS)
T02000
G28U0.V0.W0.
G55G99G97G80G40M105
M6T02000
G28U0.V0.M161(RS-LOW WINDING)
M135(C2 SELECT)
M211(COMPOSITE MODE ACT)
M190
G28H0.
G50C0.
M101(TOOL SP. UNCLAMP)
G0B45.
(3D COORDINATE ROT.MODE)
G68.1X200.Z0.I0.J1.K0.R45.
T02002
G0X20.Z0.C0.Y-10.
M389(SELECT C2 CLAMP CONTROL)
M29S500
M73(72=L/H, M73=R/H)
G88X-40.R-5.F1.0M89
C90.
C180.
C270.
G80M90
T02000
G69.1
M135
M212(COMPOSITE MODE CANCEL)
G28U0.V0.W0.
M1
```

```
***G368/G369***
N6(X-RIGID,G88@B45. ON RS)
T02000
G28U0.V0.W0.
G55G99G97G80G40M105
M6T02000
G28U0.V0.M105
M161(RS-LOW WINDING)
M135(C1 SELECT)
M211(COMPOSITE MODE ACT)
M190
G28H0.
G50C0.
M101(TOOL SP. UNCLAMP)
(G0B45.)
T02002
(3D CORDINATE ROT.MODE)
G368X200.Z0.D1.B45.
G0X20.Z0.C0.Y-10.
M389(SELECT C2 CLAMP CONTROL)
M29S500
M73(72=L/H, M73=R/H)
G88X-40.R-5.F1.0M89
C90.
C180.
C270.
G80M90
M135
M212(COMPOSITE MODE CANCEL)
G369
G28U0.V0.W0.
M1
```

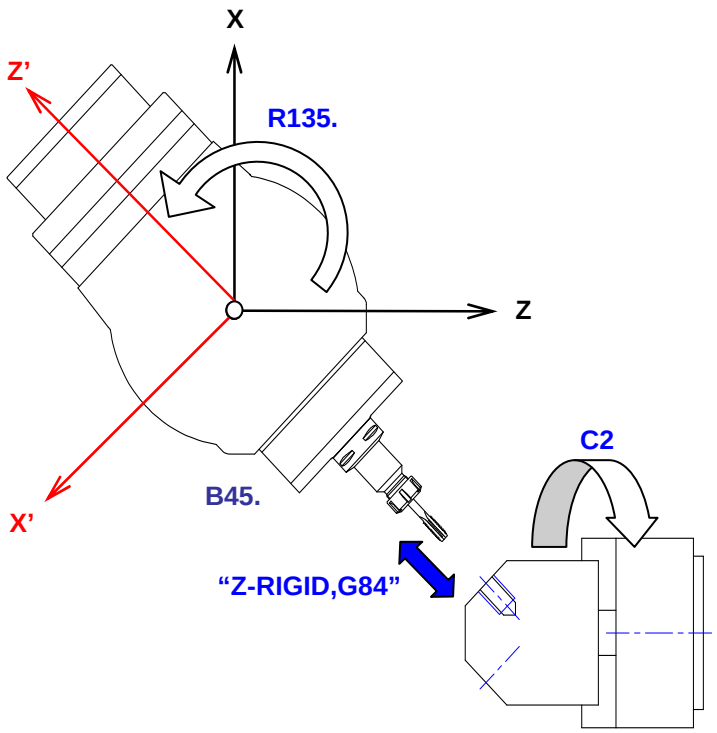
13.5.7 Z-direction Drilling on Right-Spindle




```
***G68.1/G69.1***
N7(Z-DRILL,G83@B45. ON RS)
T01000
G28U0.V0.W0.
G55G99G80G40M105
M6T01000
G28U0.V0.M161(RS-LOW WINDING)
M135(C2 SELECT)
M211(COMPOSITE MODE ACT)
M190
G28H0.
G50C0.
M101(TOOL SP. UNCLAMP)
G97S1000M33
G0B45.
(3D COORDINATE ROT.MODE)
G68.1X200.Z0.I0.J1.K0.R135.
T01001
G0X0.Z10.C0.Y-10.
M389(SELECT C2 CLAMP CONTROL)
G83Z-25.R-5.Q15000F0.5M89
C90.Q15000
C180.Q15000
C270.Q15000
G80M90
T01000
G69.1
M135
M212(COMPOSITE MODE CANCEL)
G28U0.V0.W0.
M1
```

```
***G368/G369***
N7(Z-DRILL,G83@B45. ON RS)
T01000
G28U0.V0.W0.
G55G99G80G40M105
M6T01000
G28U0.V0.M161(RS-LOW WINDING)
M135(C2 SELECT)
M211(COMPOSITE MODE ACT)
M190
G28H0.
G50C0.
M101(TOOL SP. UNCLAMP)
G97S1000M33
T01001
(G0 B45.)
(3D COORDINATE ROT.MODE)
G368X200.Z0.D0.B45.
G0X0.Z10.C0.Y-10.
M389(SELECT C2 CLAMP CONTROL)
G83Z-25.R-5.Q15000F0.5M89
C90.Q15000
C180.Q15000
C270.Q15000
G80M90
M135
M212(COMPOSITE MODE CANCEL)
G369
G28U0.V0.W0.
M1
```

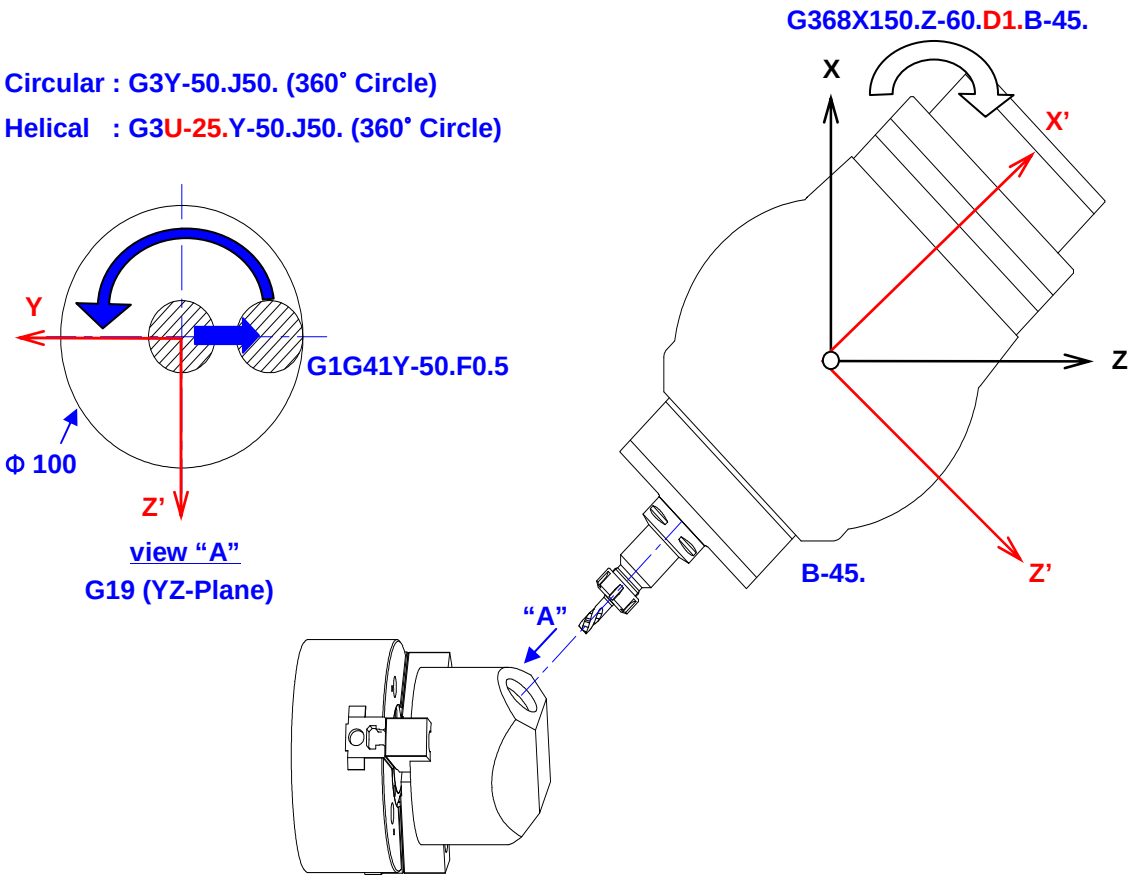
13.5.8 Z-direction Tapping on Right-Spindle




```
***G68.1/G69.1***
N8(Z-RIGID,G84@B45. ON RS)
T02000
G28U0.V0.W0.
G55G99G97G80G40M105
M6T02000
G28U0.V0. M161(RS-LOW WINDING)
M135(C2 SELECT)
M211(COMPOSITE MODE ACT)
M190
G28H0.
G50C0.
M101(TOOL SP. UNCLAMP)
G0B45.
(3D COORDINATE ROT.MODE)
G68.1X200.Z0.I0.J1.K0.R135.
T02002
G0X0.Z10.C0.Y-10.
M389(SELECT C2 CLAMP CONTROL)
M29S500
M73(72=L/H, M73=R/H)
G84Z-20.R-5.F1.0M89
C90.
C180.
C270.
G80M90
T02000
G69.1
M135
M212(COMPOSITE MODE CANCEL)
G28U0.V0.W0.
M1
```

```
***G368/G369***
N8(Z-RIGID,G84@B45. ON RS)
T02000
G28U0.V0.W0.
G55G99G97G80G40M105
M6T02000
G28U0.V0.M161(RS-LOW WINDING)
M135(C2 SELECT)
M211(COMPOSITE MODE ACT)
M190
G28H0.
G50C0.
M101(TOOL SP. UNCLAMP)
(G0B-45.)
T02002
(3D CORDINATE ROT.MODE)
G368X200.Z0.D0.B45.
G0X0.Z10.C0.Y-10.
M389(SELECT C2 CLAMP CONTROL)
M29S500
M73(72=L/H, M73=R/H)
G84Z-20.R-5.F1.0M89
C90.
C180.
C270.
G80M90
M135
M212(COMPOSITE MODE CANCEL)
G369
G28U0.V0.W0.
M1
```

13.5.9 Circular& Helical Interpolation on Left-Spindle (G368/G03)

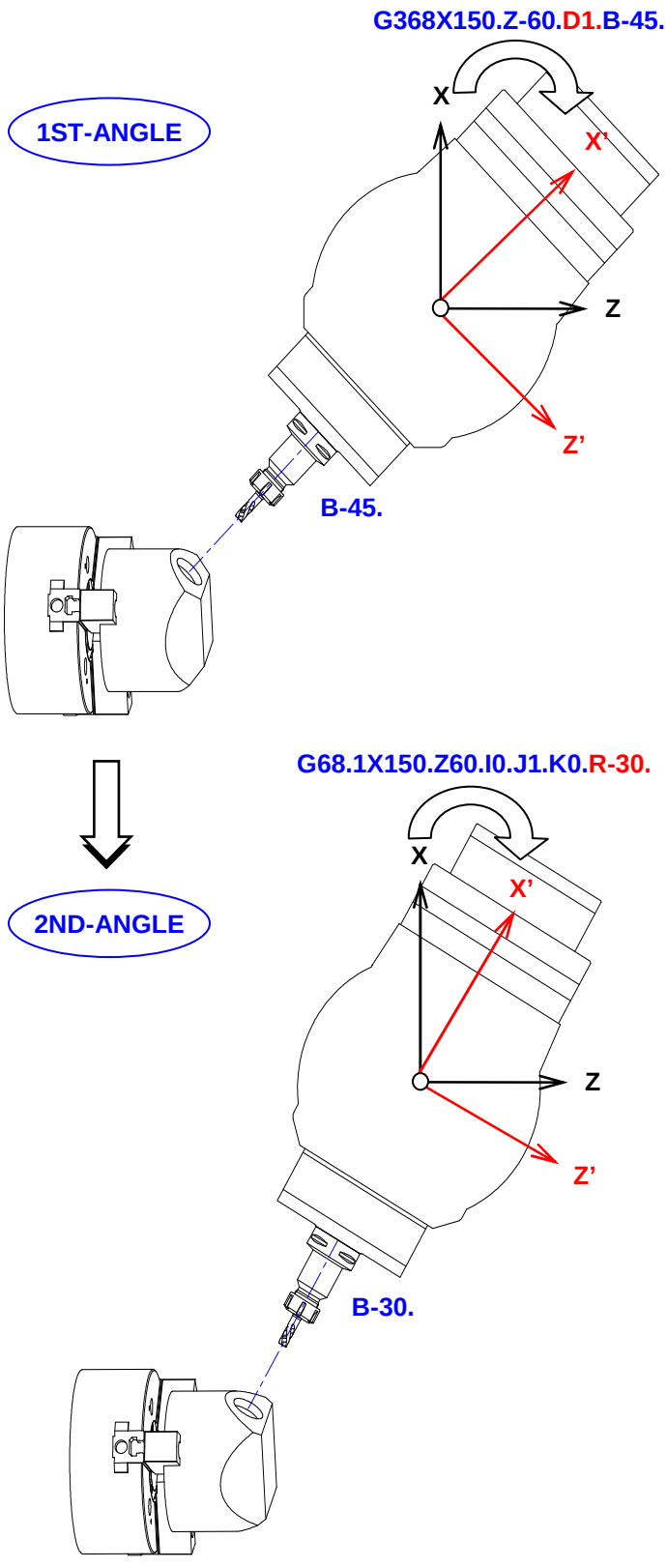



```
***G368/G369***
N1(SLOPE-POCKET@B-45. ON LS)
T08000
G28U0.V0.W0.
G54G99G80G40M5
M6T8000
G28U0.V0.M61
M35
M90
G28H0.
G50C0.
M101(TOOL SP. UNCLAMP)
G97S1000M33
T08008
(3 D CORDINATE ROT. MODE)
G368X150.Z-60.D1.B-45.(X-DIRECTION)
G0X10.Z0.C0.Y0.
G1X-10.F1.M89
G19(YZ-PLANE)
G1G41Y-50.F0.5
G3Y-50.J50. (CIRCULAR)
G1G40Y0.F1.
G0X10.M90
M35
G369
G28U0.V0.W0.
M30
```

```
***G368/G369***
N1(SLOPE-POCKET@B-45. ON LS)
T08000
G28U0.V0.W0.
G54G99G80G40M5
M6T8000
G28U0.V0.M61
M35
M90
G28H0.
G50C0.
M101(TOOL SP. UNCLAMP)
G97S1000M33
T08008
(3 D CORDINATE ROT. MODE)
G368X150.Z-60.D1.B-45.(X-DIRECTION)
G0X10.Z0.C0.Y0.
G1X-10.F1.M89
G19(YZ-PLANE)
G1G41Y-50.F0.5
G3U-25.Y-50.J50. (HELICAL)
G1G40Y0.F1.
G0X10.M90
M35
G369
G28U0.V0.W0.
M30
```

13.5.10 2-Angles Slope-Pocket on Left-Spindle (G368/G69.1/G68.1/G369)

```
***2-ANGLE G368/G369***
N1(SLOPE-POCKET@B-45. ON LS)
T08000
G28U0.V0.W0.
G54G99G80G40M5
M6T8000
G28U0.V0.M61
M35
M90
G28H0.
G50C0.
M101(TOOL SP. UNCLAMP)
G97S1000M33
T08008
(3 D CORDINATE ROT. MODE)
G368X150.Z-60.D1.B-45. (1ST)
G0X10.Z0.C0.Y0.
G1X-10.F1.M89
G19(YZ-PLANLE)
G1G41Y-50.F.5
G3Y-50.J50.
G1G40Y0.F1.
G0X250.M90
M35
G0C90.
T08000
G69.1
G28V0
G0B-30.
(3 D CORDINATE ROT. MODE)
G68.1X150.Z-60.I0.K0.J1.R-30. (2ND)
T08008
G97S1000M33
G0X10.Z0.Y0.
G1X-10.F1.M89
G19(YZ-PLANE)
G1G42Y-50.F0.5
G2Y-50.J50.
G1G40Y0.F1.
G0X150.M90
M35
G369
G28U0.V0.W0.
M30
```



Appendix

A. Keep relay list

A.1 Keep relay list (1/3) of MX 1-path

ADDRESS		MEANING
K0.0		
K0.1	KSREST	When the servo diriven steady rest(2 path) is used(→1), or not(→0).
K0.2	KEUCHS	When a door is closed, A door interlock switch enters ON state(→1), or OFF state(→0)
K0.3	KAZRN	In REF.mode, reference point is returned by one touch of jog switch(→1), or not(→0)
K0.4	KCPRS	Coolant pressure switch is used(→1), or not(→0)
K0.5	KPRSCH	Alarm of feed hold is generated in high-low chucking(→0), or not(→1)
K0.6		
K0.7	KAPRS	Air pressure switch is used(→0), or not(→1)
K1.0	KRD	Red PATROL LIGHT is used(→1), or not(→1)
K1.1	KYW	Yellow PATROL LIGHT is used(→1), or not(→0)
K1.2	KGN	Green PATROL LIGHT is used(→1), or not(→0)
K1.3	KCBSPL	Coolant blow of chuck runs with spindle rotating(→1), or not(→0)
K1.4	KSTREF	In REF.mode, reference point is returned by CYCLE START switch(→1), or not(→0)
K1.5	KM23	M23 FUNCTION is used(→1) or not(→0)
K1.6	KHINT	
K1.7	KTMCHP	Chip conveyor is stopped by timer NO.9(→1) or M25(→0) after M24
K2.0	KTMLK	Aux. machine lock function is used(→1), or not(→0)
K2.1	KNQHD	In Handle Mode, Tool offset compensation by Q-Setter is used(→0), or not(→1)
K2.2	KMABS	MANUAL ABSOLUTE function is used(→0), or not(→1)
K2.3	KSAR	Spindle speed signal(SAR) is used(→0), or not(→1)
K2.4	KOPSKP	Toggle switch in op panel is used for optional block skip function(→0), or input X7.6 in bar-feeder(→1)
K2.5	KSAPOF	Separate Auto Power Off function is used(→1), or not(→0)
K2.6	KLHUSE	When M03, M04 is ordered without M66,M77 command, the alarm occurs(→1), or not(→0)
K2.7	KTDRN	When DRY RUN issued during auto machining, dry run alarm soccurs(→1), or not(→0)
K3.0	KFHSPC	Spindle and coolant stop running on FEED HOLD or SINGLE BLOCK ALARM (→1), or not(→0)
K3.1	KMLKI	Reference point return is ignored after machine lock(→1), or not(→0)
K3.2	KF0%IT	Rapid override is available on feedrate override 0%(→1), or not(→0)
K3.3	KR25%	Rapid override is available with 25%(→1), or not(→0)
K3.4	KM00	Spindle & coolant stopped by M00, M01 run again with cycle start button in MEM(→1), or not(→0)
K3.5	KCOLTG	High pressure coolant runs when manual coolant switch is on(→1), or not(→0)
K3.6	KCLFIN	When m-code of coolant on is ordered, m-code is finished by motor(→1) or internal relay operation(→0)
K3.7	KTCL	Coolant is valid even splash guard is opened(→1), or not(→0)
K4.0		
K4.1	KLED	When EMG.STOP button is released in not ready, all LEDs on operation panel flicker(→1), or not(→0)
K4.2	KM7M8	When low and high coolant are used, M7 or M8 command is available without M9(→1), or not(→0)
K4.3	KSAFD	The safety auto door is used(→1), or not(→0)
K4.4	KIGPOW	Motor power on/off function that cut power of PSM in opened door is used(→0), or not(→1)
K4.5	KLBKEY	The locking operation runs in safety switch when the power is off(→1), or on(→0)
K4.6	KLUB	The low level check switch of lubrication unit is normal closed contact(→1)or normal open contact(→0)
K4.7	KQADWR	The Q-setter arm can be downed without checking the all axes in their reference position(→1), or on(→0)
K5.0	KHPOK	HYD. pressure switch is used(→0), or not(→1)
K5.1	KMQL	MQL is used(→1), or not(→0)
K5.2	KHYDON	Main hydraulic pressure is off on emergency(→1), or not(→0)
K5.3	KCYSTP	Temporary stop of ATC is valid(→1), or not(→0) in cycle stop
K5.4		

A.1 Keep relay list (2/3) of MX 1-path

ADDRESS		MEANING
K5.5	KSYMBO	The portable ATC OP specification is NON-CE symbol when ATC 24 tools or 40 tools(→1), or CE symbol(→0)
K5.6	KMCHA	Spindle speed change is used by M-Code (→1), or Not(→0)
K5.7	KACHA	Spindle speed range change is used by speed range (→1), or Not(→0)
K6.0	KM135	M135 function is used (→1), or Not(→0)
K6.1		
K6.2		
K6.3	KMQST	Manually operated Q-SETTER is used(→1), or not(→0)
K6.4	KCJET	The COOLJET coolant pump for DHIAC is used(→1), or not(→0)
K6.5	KM50	Bar feeder is advanced by M50(→1), or auto mode signal(→0)
K6.6	KMODE	Bar feeder is advanced by auto mode(→1), or stl(→0). It is available on setting K6.5(KM50) to 0
K6.7	KM50DU	Bar feeder advance is ON or OFF by M50(→1), or only ON(→0)
K7.0	KLNSSG	Single bar load type of LNS BAR-FEEDER is used(→1), or not(→0)
K7.1	KLNSSP	MINI/SPRINT type of BAR-FEEDER is used(→1), or not(→0)
K7.2	KHYDBF	Hydraulic type of BAR-FEEDER is used(→1), or not(→0)
K7.3	KSMW	SMW type of BAR-FEEDER is used(→1), or not(→0)
K7.4	KNEWM6	The new type of M06 tool change is used(→1), or not(→0)
K7.5	KQSET	Q-setter is used (→1), or not (→0)
K7.6	KCHPXL	PX. switch for examing the state of main-chuck is not used(→1), or used(→0)
K7.7	KCHPXR	PX. switch for examing the state of sub-chuck is not used(→1), or used(→0)
K8.0	KCHSPM	Sperate type of main-foot switch(clamp/unclamp) is used(→1) or not(→0)
K8.1	KORCB	Orientation function of angle index type is used(→1), or not(→0)
K8.2	KABSPL	Air blow of chuck runs with spindle rotating(→1), or not(→0)
K8.3	KCKNIT	Chuck un/clamped is valid by foot sw even while bar feeding of BAR-FEEDER(→1), or not(→0)
K8.4	KQPBUC	To clamp and unclamp a body are available on advanced tailstock quill(→1) or not(→0)
K8.5	KPARTS	Parts catcher is used (→1), or not (→0)
K8.6	KLCOUT	Left-spindle is outward chucking type(→1), or not(→0)
K8.7	KQULCF	Quill of tailstock advance and retract is confirmed by switch(→1), or not(→0)
K9.0	KCHIP	CHIP Conveyor alarm is displayed(→1), or not(→0)
K9.1		
K9.2	KTCHK	Spindle tool no. mismatch alarm(2115) is ignored(→1), or not(→0)
K9.3	KCUCPS	C-axis pressure switch is used(→0), or not(→1)
K9.4	KSTD	When standard door interlock function is used(→1), or not(→0)
K9.5	KPTS	Programmable tail stock(PTS) is used (→1), or not (→0)
K9.6	KATQC	Torque control function is used(→1), or not(→0)
K9.7	KGL	Gantry loader is used(→1), or not(→0)
K10.0	KSPJOG	In auto mode, spindle jogging is invalid(→1), or not(→0)
K10.1		
K10.2		
K10.3	KQCFST	When K8<7> was setting "1", quill advance is kept by switch(→0), or timer(→1)
K10.4	KSPARE	Input signal is used to finish M91 and M92(→0), or not(→1)
K10.5	KMGON	This bit can't use. (Only for Ez-guide T)
K10.6	KMILLS	When door interlock function of mills market is used(→1) or not(→0)
K10.7	KTEC	When door interlock function of italy tecnomach is used(→1) or not(→0)
K50.3	KYREF	When BT-axis moving, Y-axis referenece position interlock is released(→1) or not(→0)

A.1 Keep relay list (3/3) of MX 1-path

ADDRESS		MEANING
K50.7	KINTF	The parameter for interference area check function is valid(→1) or not(→0)
K51.6	KEDGE	Safety EDGE sensor is used(→1), or not(→0)
K52.0	K24TS	When 24-TOOLS magazine is used(→1) or not(→0)
K52.1	KCFILT	When coolant filter is used(→1) or not(→0)
K52.2	K80TS	When 80-TOOLS magazine is used(→1) or not(→0)
K52.3	K40TS	When 40-TOOLS magazine is used(→1) or not(→0)
K52.4	KDUSWU	The spindle warming up function is not used(→1) or used(→0)
K52.5	KMIST	Oil mist collector is used(→1), or not(→0)
K52.6	KOPDIG	When operator's door interlock is used(→1) or not(→0)
K52.7	KATC	When ATC is used(→1) or not(→0)
K53.0	KZRLS	The interlock between Z-axis and waiting pot in ATC 80 tools is ignored(→ 1), or not(→ 0)
K53.1	KLIGN	Lower unit alarm is ignored(→1) or not(→0)
K53.2	KUXIGN	The interlock function forX-AXIS in low is released(→1) or not(→0)
K53.3	KBACLP	When B-axis automatic clamp/unclamp function is used(→1) or not(→0)
K53.4	KBSVOF	When B-axis servo off function is used(→1) or not(→0)
K53.5	KATCOP	When portable ATC OP is used(→1) or not(→0)
K53.6	KNEWB	New B-axis(Roller Gear Cam type) is used(→1), or not(→0)
K53.7	KPBAST	Bucket and arm advance of parts catcher are operated separately(→0) or simultaneously(→1)
K54.0	K0TYPE	TAIL STOCK is used(→ 1), or not(→ 0)
K54.1	KAIGZ	When axes moving to ATC postion, the 2'nd reference position of Z-axis is ignored(→ 1), or not(→ 0)
K54.2	KINCH	When inch system is used(→ 1), or not(→ 0)
K54.3	KINVER	When the invereter for ATC arm is used(→ 1), or not(→ 0)
K54.5	K3DOOR	When auto 3 door system is used(→ 1), or not(→ 0)
K54.6	KKNOLL	When the KNOLL coolant system is used(→ 1), or not(→ 0)
K54.7	KURDOG	When the dog for X1,Z1 and Y-axis reference point return is used(→ 0), or not(→ 1)
K55.0	KPSUNC	When operation mode is changed, the C-axis is automatically unclamp(→ 0), or not(→ 1)
K55.1	KX2ITK	When the interlock function between lower turret and left chuck is used(→ 1), or not(→ 0)
K55.3	KPTFAS	PARTS UNLOADER M-CODE M210, M310 is finished without confirm (→1), with confirm(→0)
K55.5	KCUSEN	Auto start command is valid without chucking condition(→ 1), or not(→ 0)
K55.7	KSDTS	The Servo driven Tailstock is used(→ 1), or not(→ 0)
K60.0		
K56.0	KSOPU	When the soft operating panel is used(→ 1), or not(→ 0)
K56.4	KLATD	When the auto door is installed in long bed machine model .(→ 1), or not(→ 0)
K56.5		
K56.6	KSRXZ	X2 and Z2 -axis can not be moved with steady rest clamp.(→ 1), or be moved (→ 0)
K56.7		
K60.0		
K60.1	KPRGEQ	"Cycle start" is available when the program number is same in upper and lower(→ 1), or not(→ 0)
K60.2	KFAST8	M06 Tool cahnge can be executed and finished without waiting pot position (→ 1), or not(→ 0)
K60.3	KSKMO	If milling spindle is oriented to 0 degreee, orientation is executed in M06 (→ 1), or not (→ 0)
K60.4	KBSCO	When BED SHOWER COOLANT MOTOR is turned on in auto running automatically (→ 1), or not(→ 0)
K60.5	KPTCINT	Parts Unloader is returned to home position by pressing cycle start button (→ 1), or not(→ 0)
K65.0	KROBOT	ROBOT interface is used (→ 1), or not(→ 0)
K65.1	KREPT	Repeat switch is used (→ 1), or not(→ 0)

A.2 Keep relay list (1/4) of MX 2-path

ADDRESS		MEANING
K0.0	KNOTW	Waiting code is ignored(→1), or not(→0)
K0.2	KEUCHS	When a door is closed, A door interlock switch enters ON state(→1), or OFF state(→0)
K0.3	KAZRN	In REF.mode, reference point is returned by one touch of jog switch(→1), or not(→0)
K0.4	KCPRS	Coolant pressure switch is used(→1), or not(→0)
K0.5	KPRSCH	Alarm of feed hold is generated in high-low chucking(→0), or not(→1)
K0.6	KFORI	The first orientation of the 2nd spindle is performed(→1) to set shift value ,or not(→0)
K0.7	KAPRS	Air pressure switch is used(→0), or not(→1)
K1.0	KRD	Red PATROL LIGHT is used(→1), or not(→1)
K1.1	KYW	Yellow PATROL LIGHT is used(→1), or not(→0)
K1.2	KGN	Green PATROL LIGHT is used(→1), or not(→0)
K1.3	KALAPF	IN ALARM AUTO POWER OFF FUNCTION is used(→1) or not(→0)
K1.4	KSTREF	In REF.mode, reference point is returned by CYCLE START switch(→1), or not(→0)
K1.5	KM23	M23 FUNCTION is used(→1) or not(→0)
K1.6	KHINT	CYCLE START is available without reference return not completed(→1), or not(→0)
K1.7	KTMCHP	Chip conveyor is stopped by timer NO.9(→1) or M25(→0) after M24
K2.0	KTMLK	Aux. machine lock function is used(→1), or not(→0)
K2.1	KNQHD	In Handle Mode, Tool offset compensation by Q-Setter is used(→0), or not(→1)
K2.2	KMABS	MANUAL ABSOLUTE function is used(→0), or not(→1)
K2.3	KSAR	Spindle speed signal(SAR) is used(→0), or not(→1)
K2.4	KOPSKP	Toggle switch in op panel is used for optional block skip (→0), or input X7.6 in bar-feeder(→1)
K2.5	KSAPOF	Separate Auto Power Off function is used(→1), or not(→0)
K2.6	KLHUSE	When M03, M04 is ordered without M66,M77 command, the alarm occurs(→1), or not(→0)
K2.7	KTDRN	When DRY RUN issued during auto machining, dry run alarm soccurs(→1), or not(→0)
K3.0	KFHSPC	Spindle and coolant stop running on FEED HOLD or SINGLE BLOCK ALARM (→1), or not(→0)
K3.1	KMLKI	Reference point return is ignored after machine lock(→1), or not(→0)
K3.2	KF0%IT	Rapid override is available on feedrate override 0%(→1), or not(→0)
K3.3	KR25%	Rapid override is available with 25%(→1), or not(→0)
K3.4	KM00	Spindle & coolant stopped by M00, M01 run again with cycle start button in MEM(→1), or not(→0)
K3.5	KCOLTG	High pressure coolant runs when manual coolant switch is on(→1), or not(→0)
K3.6	KCLFIN	When coolant on M-code is ordered, M-code is finished by motor(→1) or internal relay operation(→0)
K3.7	KTCL	Coolant is valid even splash guard is opened(→1), or not(→0)
K4.0	KTORI	Proximity sensor for checking of tool ref. position is used(→1), or not(→0)
K4.1	KLED	When EMG.STOP button is released in not ready, all LEDs on operation panel flicker(→1), or not(→0)
K4.2	KM7M8	When low and high coolant are used, M7 or M8 command is available without M9(→1), or not(→0)
K4.3	KSAFD	The safety auto door is used(→1), or not(→0)
K4.4	KIGPOW	Motor power on/off function that cut power of PSM in opened door is used(→0), or not(→1)
K4.5	KLBKEY	The locking operation runs in safety switch when the power is off(→1), or on(→0)
K4.6	KLUB	The low level check switch of lubrication unit is normal closed (→1) or normal open contact(→0)
K4.7	KQADWR	The Q-setter arm can be downed without checking the all axes in their ref. position(→1), or on(→0)
K5.0	KHPOK	HYD. pressure switch is used(→0), or not(→1)
K5.1	KMQL	MQL is used(→1), or not(→0)
K5.2	KHYDON	Main hydraulic pressure is off on emergency(→1), or not(→0)
K5.3	KCYSTP	Temporary stop of ATC is valid(→1), or not(→0) in cycle stop
K5.4	KTALM	Turret is unclamped in manual mode(→1), or not(→0)

A.2 Keep relay list (2/4) of MX 2-path

ADDRESS		MEANING
K5.5	KSYMBO	The portable ATC OP specification is NON-CE symbol when ATC 24 tools or 40 tools(→1), or CE symbol(→0)
K5.6	KMCHA	Spindle speed change is used by M-Code (→1), or Not(→0)
K5.7	KACHA	Spindle speed range change is used by speed range (→1), or Not(→0)
K6.1	K10TS	The turret with 10 tools is used(→1), or not(→0)
K6.2	K12TS	The turret with 12 tools is used(→1), or not(→0)
K6.3	KMQST	Manually operated Q-SETTER is used(→1), or not(→0)
K6.4	KCJET	The COOLJET coolant pump for Upper side is used(→1), or not(→0)
K6.5	KM50	Bar feeder is advanced by M50(→1), or auto mode signal(→0)
K6.6	KMODE	Bar feeder is advanced by auto mode(→1), or stl(→0). It is available on setting K6.5(KM50) to 0
K6.7	KM50DU	Bar feeder advance is ON or OFF by M50(→1), or only ON(→0)
K7.0	KLNSSG	Single bar load type of LNS BAR-FEEDER is used(→1), or not(→0)
K7.1	KLNSSP	MINI/SPRINT type of BAR-FEEDER is used(→1), or not(→0)
K7.2	KHYDBF	Hydraulic type of BAR-FEEDER is used(→1), or not(→0)
K7.3	KSMW	SMW type of BAR-FEEDER is used(→1), or not(→0)
K7.4	KNEWM6	The new type of M06 tool change(M06 is commanded with T-code) is used(→1), or not(→0)
K7.5	KQSET	Q-setter is used (→1), or not (→0)
K7.6	KCHPXL	PX. switch for examing the state of main-chuck is not used(→1), or used(→0)
K7.7	KCHPXR	PX. switch for examing the state of sub-chuck is not used(→1), or used(→0)
K8.0	KCHSPM	Sperate type of main-foot switch(clamp/unclamp) is used(→1) or not(→0)
K8.1	KORCB	Orientation function of angle index type is used(→1), or not(→0)
K8.2	KABSPL	Air blow of chuck runs with spindle rotating(→1), or not(→0)
K8.3	KCKNIT	Chuck un/clamped is valid by foot sw even while bar feeding of BAR-FEEDER(→1), or not(→0)
K8.4	KQPBUC	To clamp and unclamp a body are available on advanced tailstock quill(→1) or not(→0)
K8.5	KPARTS	Parts catcher is used (→1), or not (→0)
K8.6	KLCOUT	Left-spindle is outward chucking type(→1), or not(→0)
K8.7	KQULCF	Quill of tailstock advance and retract is confirmed by switch(→1), or not(→0)
K9.0	KCHIP	CHIP Conveyor alarm is displayed(→1), or not(→0)
K9.1	KLREVS	Rev.Tool is used(→1), or not used(→0) in Low Turret.
K9.2	KTCHK	Spindle tool no. mismatch alarm(2115) is ignored(→1), or not(→0)
K9.3	KCUCPS	C-axis pressure switch is used(→0), or not(→1)
K9.4	KSTD	When standard door interlock function is used(→1), or not(→0)
K9.5	KPTS	Programmable tail stock(PTS) is used (→1), or not (→0)
K9.6	KTQCON	Torque control function is used(→1), or not(→0)
K9.7	KGL	Gantry loader for MX-Machine is used(→1), or not(→0)
K10.0	KSPJOG	In auto mode, spindle jogging is invalid(→1), or not(→0)
K10.1	KT00	Program runs without alarm when T00000 instruction is ordered at auto mode(→1), or not(→0)
K10.2	KTIC	If "+" traverse is command with T-code, they don't start simultaneously(→1), or not(→0)
K10.3	KQCFST	When K8<7> was setting "1", quill advance is kept by switch(→0), or timer(→1)
K10.4	KSPARE	Input signal is used to finish M91 and M92(→0), or not(→1)
K10.6	KMILLS	When door interlock function of mills market is used(→1) or not(→0)
K10.7	KTEC	When door interlock function of italy tecnomach is used(→1) or not(→0)
K50.0		This bit can't use. (Only for internal use)
K50.1		This bit can't use. (Only for internal use)
K50.2		This bit can't use. (Only for internal use)

A.2 Keep relay list (3/4) of MX 2-path

ADDRESS		MEANING
K50.3	KYREF	When B-axis moving, Y-axis referenece position interlock is released(→1) or not(→0)
K50.4		This bit can't use. (Only for internal use)
K50.5		This bit can't use. (Only for internal use)
K50.6	KTUNS	When the input address for milling spindle tool unclamp is X26.6(→1), or X9.5(→0).
K50.7	KINTF	The parameter for interference area check function is valid(→1) or not(→0)
K51.6	KEDGE	Safety EDGE sensor is used(→1), or not(→0)
K52.0	K24TS	When 24-TOOLS magazine is used(→1) or not(→0)
K52.1	KCFILT	When coolant filter is used(→1) or not(→0)
K52.2	K80TS	When 80-TOOLS magazine is used(→1) or not(→0)
K52.3	K40TS	When 40-TOOLS magazine is used(→1) or not(→0)
K52.4	KDUSWU	The spindle warming up function is not used(→1) or used(→0)
K52.5	KMIST	Oil mist collector is used(→1), or not(→0)
K52.6	KOPDIG	When operator's door interlock is used(→1) or not(→0)
K52.7	KATC	When ATC is used(→1) or not(→0)
K53.0	KZRLS	The interlock between Z-axis and waiting pot in ATC 80 tools is ignored(→ 1), or not(→ 0)
K53.1	KLIGN	Lower unit alarm is ignored(→1) or not(→0)
K53.2	KUXIGN	The interlock function forX-AXIS in low is released(→1) or not(→0)
K53.3	KBACLP	When B-axis automatic clamp/unclamp function is used(→1) or not(→0)
K53.4	KBSVOF	When B-axis servo off function is used(→1) or not(→0)
K53.5	KATCOP	When portable ATC OP is used(→1) or not(→0)
K53.6	KNEWB	New B-axis(Roller Gear Cam type) is used(→1), or not(→0)
K53.7	KPBAST	Bucket and arm advance of parts catcher are operated separately(→0) or simultaneously(→1)
K54.0	KTTYPE	When the machine is T-type(→ 1), or ST-type(→ 0)
K54.1	KAIGZ	When axes moving to ATC postion, the 2'nd reference position of Z-axis is ignored(→ 1), or not(→ 0)
K54.2	KINCH	When inch system is used(→ 1), or not(→ 0)
K54.3	KINVER	When the invereter for ATC arm is used(→ 1), or not(→ 0)
K54.4	KNPMPG	When new type portable MPG is used(→ 1), or not(→ 0)
K54.5		
K54.6	KKNOLL	When the KNOLL coolant system is used(→ 1), or not(→ 0)
K54.7	KURDOG	When the dog for X1, Z1 and Y-axis reference point return is used(→ 0), or not(→ 1)
K55.0	KPSUNC	When the C-axis is unclamped automatically in manule mode(→ 0), or not(→ 1)
K55.1	KX2ITK	When the interlock function between lower turret and left chuck is used(→ 1), or not(→ 0)
K55.2	KLCJET	The COOLJET coolant pump for Lower side is used(→1), or not(→0)
K55.3	KPTFAS	PARTS UNLOADER M-CODE M210, M310 is finished without confirm (→1), with confirm(→0)
K55.4	KC1C2I	When the C1 & C2-axis reference point return alarm is ignored(→ 1), or not(→ 0)
K55.5	KCUSEN	Auto start command is valid without chucking condition(→ 1), or not(→ 0)
K55.6		
K55.7	KSDTS	When the servo driven tail stock is used (→ 1), or not(→ 0)
K56.0	KSOPU	When the soft operating panel is used(→ 1), or not(→ 0)
K56.1		
K56.2		
K56.3		
K56.4	KLATD	When the auto door is installed in long bed machine model .(→ 1), or not(→ 0)
K56.5	KUSR	When the steady rest is installed at the lower turret unit.(→ 1), or not(→ 0)
K56.6	KSRXZ	X2 and Z2 -axis can not be moved with steady rest clamp.(→ 1), or be moved (→ 0)

A.2 Keep relay list (4/4) of MX 2-path

ADDRESS		MEANING
K60.0	KQDTIC	When the lower turret is enable to index even the Q-setter is downed(→ 1), or not(→ 0)
K60.1	KPRGEQ	"Cycle start" is available when the program number is same in upper and lower(→ 1), or not(→ 0)
K60.2	KFAST8	M06 Tool cahnge can be executed and finished without waiting pot position (→ 1), or not(→ 0)
K60.3	KSKMO	If milling spindle is oriented to 0 degreee, orientation is executed in M06 (→ 1), or not (→ 0)
K60.4	KBSCO	When BED SHOWER COOLANT MOTOR is turned on in auto running automatically (→ 1), or not(→ 0)
K60.5	KPTCINT	Parts Unloader is returned to home position by pressing cycle start button (→ 1), or not(→ 0)
K60.6		
K60.7		
K65.0	KROBOT	ROBOT interface is used (→ 1), or not(→ 0)
K65.1	KREPT	Repeat switch is used (→ 1), or not(→ 0)

A.3 Keep relay list (1/3) of TT15/18SY

ADDRESS		MEANING
K0.0	KNOTW	Waiting code is ignored(→1), or not(→0)
K0.2	KSAFD	The safety auto door is used(→1), or not(→0)
K0.3		
K0.4	KCPRS	Coolant pressure switch is used(→1), or not(→0)
K0.5	KPRSCH	Alarm of feed hold is generated in high-low chucking(→0), or not(→1)
K0.6	KFORI	The first orientation of the 2nd spindle is performed(→1) to set shift value ,or not(→0)
K0.7	KAPRS	Air pressure switch is used(→0), or not(→1)
K1.0	KRD	Red PATROL LIGHT is used(→1), or not(→1)
K1.1	KYW	Yellow PATROL LIGHT is used(→1), or not(→0)
K1.2	KGN	Green PATROL LIGHT is used(→1), or not(→0)
K1.3	KAZRN	In REF.mode, reference point is returned by one touch of jog switch(→1), or not(→0)
K1.4	KSTREF	In REF.mode, reference point is returned by CYCLE START switch(→1), or not(→0)
K1.6	KHINT	CYCLE START is available without reference return not completed(→1), or not(→0)
K1.7	KTMCHP	Chip conveyor is stopped by timer NO.9(→1) or M25(→0) after M24
K2.0	KTMLK	Aux. machine lock function is used(→1), or not(→0)
K2.1	KNQHD	In Handle Mode, Tool offset compensation by Q-Setter is used(→0), or not(→1)
K2.2	KMABS	MANUAL ABSOLUTE function is used(→0), or not(→1)
K2.3	KSAR	Spindle speed signal(SAR) is used(→0), or not(→1)
K2.4	KOPSKP	Toggle switch in op panel is used for optional block skip function(→0), or input X7.6 in bar-feeder(→1)
K2.5	KSAPOF	Auto power off function is not divided into upper and lower(→0), divided(→1),
K2.6	KLHUSE	When M03, M04 is ordered without M66,M77 command, the alarm occurs(→1), or not(→0)
K2.7	KTDRN	When DRY RUN issued during auto machining, dry run alarm soccurs(→1), or not(→0)
K3.0	KFHSPC	Spindle and coolant stop running on FEED HOLD or SINGLE BLOCK ALARM (→1), or not(→0)
K3.1	KMLKI	Reference point return is ignored after machine lock(→1), or not(→0)
K3.2	KF0%IT	Rapid override is available on feedrate override 0%(→1), or not(→0)
K3.3	KR25%	Rapid override 50% is available by F25 Switch on the Membrane OP(→0), or not(→1)
K3.4	KM00	Spindle & coolant stopped by M00, M01 run again with cycle start button in MEM(→1), or not(→0)
K3.5	KCOLTG	Coolant flusing motor runs when manual coolant switch is on(→1), or not(→0)
K3.6	KCLFIN	When m-code of coolant on is ordered, m-code is finished by motor(→1) or internal relay operation(→0)
K3.7	KTCL	Coolant is valid even splash guard is opened(→1), or not(→0)
K4.0	KTORI	Proximity sensor for checking of tool ref. position is used(→1), or not(→0)
K4.1	KLED	When EMG.STOP button is released in not ready, all LEDs on operation panel flicker(→1), or not(→0)
K4.2	KM6M8	When upper coolant and cooljet are used, M6 or M8 command is available wthout M9(→1), or not(→0)
K4.4	KIGPOW	Motor power on/off function that cut power of PSM in opened door is used(→0), or not(→1)
K4.5	KLBKEY	The locking operation runs in safety switch when the power is off(→1), or on(→0)
K4.6	KLUB	The level check switch of lubrication unit has normal closed contact(→1)or normal open contact(→0)
K5.0	KHPOK	HYD. pressure switch is used(→0), or not(→1)
K5.2	KHYDON	Main hydraulic pressure is off on emergency(→1), or not(→0)
K5.4	KTALM	Turret is unclamped in manual mode(→1), or not(→0)
K5.6	KMCHA	Spindle speed change is used by M-Code (→1), or Not(→0)
K5.7	KACHA	Spindle speed range change is used by speed range (→1), or Not(→0)
K6.1	K10TS	The turret with 10 tools is used(→1), or not(→0)
K6.2	K12TS	The turret with 12 tools is used(→1), or not(→0)
K6.3	K24TS	The turret with 24 tools is used(→1), or not(→0)

A.3 Keep relay list (2/3) of TT15/18SY

ADDRESS		MEANING
K6.4	KCJET	COOL JET coolant pump is used(→1), or not(→0)
K6.5	KM50	Bar feeder is advanced by M50(→1), or auto mode signal(→0)
K6.6	KMODE	Bar feeder is advanced by auto mode(→1), or stl(→0). It is available on setting K6.5(KM50) to 0
K6.7	KM50DU	Bar feeder advance is ON or OFF by M50(→1), or only ON(→0)
K7.0	KLNSSG	Single bar load type of LNS BAR-FEEDER is used(→1), or not(→0)
K7.1	KLNSSP	MINI/SPRINT type of BAR-FEEDER is used(→1), or not(→0)
K7.2	KHYDBF	Hydraulic type of BAR-FEEDER is used(→1), or not(→0)
K7.3	KSMW	SMW type of BAR-FEEDER is used(→1), or not(→0)
K7.3	KSMW	SMW type of BAR-FEEDER is used(→1), or not(→0)
K7.4		
K7.5	KQSET	AUTO Q-setter is used (→1), or not (→0)
K7.6	KCHPXL	PX. switch for examing the state of main-chuck is not used(→1), or used(→0)
K7.7	KCHPXR	PX. switch for examing the state of sub-chuck is not used(→1), or used(→0)
K8.0	KCHSPM	Sperate type of main-foot switch(clamp/unclamp) is used(→1) or not(→0)
K8.1	KORCB	Orientation function of angle index type is used(→1), or not(→0)
K8.2	KABSPL	Air blow of chuck runs with spindle rotating(→1), or not(→0)
K8.3	KCKNIT	Chuck un/clamped is valid by foot sw even while bar feeding of BAR-FEEDER(→1), or not(→0)
K8.5	KPARTS	Parts catcher is used (→1), or not (→0)
K8.6	KPTUN	Parts unloader is used (→1), or not (→0)
K8.7		
K9.0	KTTS	The machine type of TT15/18S is used(→1), or not(→0)
K9.1	KTTMS	The machine type of TT15/18MS is used(→1), or not(→0)
K9.2	KTTSY	The machine type of TT15/18SY is used(→1), or not(→0)
K9.3	KCUCPS	C-axis pressure switch is used(→0), or not(→1)
K9.4	KSTD	When standard door interlock function is used(→1), or not(→0)
K9.5	KCHIP	The alarm detection function of CHIP Conyeyor is used(→1), or not(→0)
K9.6	KCLFSH	Coolnat flusing is used(→1), or not(→0)
K9.7	KGL	Gantry loader is used(→1), or not(→0)
K10.0	KSPJOG	In auto mode, spindle jogging is invalid(→1), or not(→0)
K10.1	KT00	Program runs without alarm when T0000 instruction is ordered at auto mode(→1),or not(→0)
K10.4	KSPARE	Input signal is used to finish M91 and M92(→0), or not(→1)
K10.5		This bit can't use.(Only for Ez-guide T)
K10.6	KMILLS	When door interlock function of mills market is used(→1) or not(→0)
K10.7	KTEC	When door interlock function of italy tecnomach is used(→1) or not(→0)
K50.0	KMQL	The Dry Coolant for MQL Dry Machining is used(→1) or not(→0)
K50.1	KMIST	OIL Mist Collector is used(→1) or not(→0)
K50.2	KALAPF	IN ALARM AUTO POWER OFF FUNCTION is used(→1) or not(→0)
K50.3	KM23	M23 FUNCTION is used(→1) or not(→0)
K50.5	KTQCO	Torque control is used(→1), or not(→0)
K50.6	KNORQ	Q-setter is available without reference return completed(→1), or not(→0)
K50.7	KCOLF	Coolnt filter change is used(→1), or not(→0)
K51.0	KSELY	This bit can't use. (Only for DHIAC)
K51.1	KMGI	MANUAL GUIDE I is used(→1) or not(→0)
K51.4	KRSHAW	MANUAL Q-SETTER is used(→1) or not(→0)

A.3 Keep relay list (3/3) of TT15/18SY

ADDRESS		MEANING
K51.6	KEDGE	Safety edge is used (→1) or not(→0)
K52.2	K12RET	Tool change is available at the 1st referance point(→1), at the 2nd referance point (→0)
K52.4	KDUSWU	Spindle warmming up function is used(→0), or not(→1)
K52.5	KCUSEN	Cycle start enable in the state of chuck unclamp(→1), or not(→0)
K52.6	KCHAIR	The air sensing for chuck-clamp confirmation is used(→1), or not(→0)
K52.7	KCLEV	The coolant level switch is used(→1), or not(→0)
K55.3	KALINT	Z, B, Y axis interlock function is used(→0), or not(→1)
K55.4	KC1C2I	C1& C2 ref. return is used(→0), or not(→1)
K55.6	KPSMAL	The running alarm of PSM module is detected(→0), or not(→1)
K55.7	KUIFIG	The interference check function for upper unit of SY machine is used(→1), or not(→0)
K60.4	KBSCO	When BED SHOWER COOLANT MOTOR is turned on in auto running automatically (→ 1), or not(→ 0)

A.4 Keep relay list (1/3) of TT20/25SY

ADDRESS		MEANING
K0.0	KNOTW	Waiting code is ignored(→1), or not(→0)
K0.2	KSAFD	The safety auto door is used(→1), or not(→0)
K0.4	KCPRS	Coolant pressure switch is used(→1), or not(→0)
K0.5	KPRSCH	Alarm of feed hold is generated in high-low chucking(→0), or not(→1)
K0.6	KFORI	The first orientation of the 2nd spindle is performed(→1) to set shift value ,or not(→0)
K0.7	KAPRS	Air pressure switch is used(→0), or not(→1)
K1.0	KRD	Red PATROL LIGHT is used(→1), or not(→1)
K1.1	KYW	Yellow PATROL LIGHT is used(→1), or not(→0)
K1.2	KGN	Green PATROL LIGHT is used(→1), or not(→0)
K1.3	KAZRN	In REF.mode, reference point is returned by one touch of jog switch(→1), or not(→0)
K1.4	KSTREF	In REF.mode, reference point is returned by CYCLE START switch(→1), or not(→0)
K1.5		
K1.6	KHINT	CYCLE START is available without reference return not completed(→1), or not(→0)
K1.7	KTMCHP	Chip conveyor is stopped by timer NO.9(→1) or M25(→0) after M24
K2.0	KTMLK	Aux. machine lock function is used(→1), or not(→0)
K2.1	KNQHD	In Handle Mode, Tool offset compensation by Q-Setter is used(→0), or not(→1)
K2.2	KMABS	MANUAL ABSOLUTE function is used(→0), or not(→1)
K2.3	KSAR	Spindle speed signal(SAR) is used(→0), or not(→1)
K2.4	KOPSKP	Toggle switch in op panel is used for optional block skip function(→0), or input X7.6 in bar-feeder(→1)
K2.6	KLHUSE	When M03, M04 is ordered without M66,M77 command, the alarm occurs(→1), or not(→0)
K2.7	KTDRN	When DRY RUN issued during auto machining, dry run alarm soccurs(→1), or not(→0)
K3.0	KFHSPC	Spindle and coolant stop running on FEED HOLD or SINGLE BLOCK ALARM (→1), or not(→0)
K3.1	KMLKI	Reference point return is ignored after machine lock(→1), or not(→0)
K3.2	KF0%IT	Rapid override is available on feedrate override 0%(→1), or not(→0)
K3.3	KR25%	Rapid override 50% is available by F25 Switch on the Membrane OP(→0), or not(→1)
K3.4	KM00	Spindle & coolant stopped by M00, M01 run again with cycle start button in MEM(→1), or not(→0)
K3.5	KCOLTG	Coolant flusing motor runs when manual coolant switch is on(→1), or not(→0)
K3.6	KCLFIN	When m-code of coolant on is ordered, m-code is finished by motor(→1) or internal relay operation(→0)
K3.7	KTCL	Coolant is valid even splash guard is opened(→1), or not(→0)
K4.0	KTORI	Proximity sensor for checking of tool ref. position is used(→1), or not(→0)
K4.1	KLED	When EMG.STOP button is released in not ready, all LEDs on operation panel flicker(→1), or not(→0)
K4.2	KM6M8	When upper coolant and cooljet are used, M6 or M8 command is available wthout M9(→1), or not(→0)
K4.4	KIGPOW	Motor power on/off function that cut power of PSM in opened door is used(→0), or not(→1)
K4.5	KLBKEY	The locking operation runs in safety switch when the power is off(→1), or on(→0)
K4.6	KLUB	The level check switch of lubrication unit has normal closed contact(→1)or normal open contact(→0)
K5.0	KHPOK	HYD. pressure switch is used(→0), or not(→1)
K5.2	KHYDON	Main hydraulic pressure is off on emergency(→1), or not(→0)
K5.4	KTALM	Turret is unclamped in manual mode(→1), or not(→0)
K5.6	KMCHA	Spindle speed change is used by M-Code (→1), or Not(→0)
K5.7	KACHA	Spindle speed range change is used by speed range (→1), or Not(→0)
K6.1	K10TS	The turret with 10 tools is used(→1), or not(→0)
K6.2	K12TS	The turret with 12 tools is used(→1), or not(→0)
K6.3		

A.4 Keep relay list (2/3) of TT20/25SY

ADDRESS		MEANING
K6.4	KCJET	COOL JET coolant pump is used(→1), or not(→0)
K6.5	KM50	Bar feeder is advanced by M50(→1), or auto mode signal(→0)
K6.6	KMODE	Bar feeder is advanced by auto mode(→1), or stl(→0). It is available on setting K6.5(KM50) to 0
K6.7	KM50DU	Bar feeder advance is ON or OFF by M50(→1), or only ON(→0)
K7.0	KLNSSG	Single bar load type of LNS BAR-FEEDER is used(→1), or not(→0)
K7.1	KLNSSP	MINI/SPRINT type of BAR-FEEDER is used(→1), or not(→0)
K7.2	KHYDBF	Hydraulic type of BAR-FEEDER is used(→1), or not(→0)
K7.3	KSMW	SMW type of BAR-FEEDER is used(→1), or not(→0)
K7.3	KSMW	SMW type of BAR-FEEDER is used(→1), or not(→0)
K7.4		
K7.5	KQSET	Q-setter is used (→1), or not (→0)
K7.6	KCHPXL	PX. switch for examing the state of main-chuck is not used(→1), or used(→0)
K7.7	KCHPXR	PX. switch for examing the state of sub-chuck is not used(→1), or used(→0)
K8.0	KCHSPM	Sperate type of main-foot switch(clamp/unclamp) is used(→1) or not(→0)
K8.1	KORCB	Orientation function of angle index type is used(→1), or not(→0)
K8.2	KABSPL	Air blow of chuck runs with spindle rotating(→1), or not(→0)
K8.3	KCKNIT	Chuck un/clamped is valid by foot sw even while bar feeding of BAR-FEEDER(→1), or not(→0)
K8.5	KPARTS	Parts catcher is used (→1), or not (→0)
K8.6	KLCOUT	Left-spindle is outward chucking type(→1), or not(→0)
K8.7	KRCOUT	Right-spindle is outward chucking type(→1), or not(→0)
K9.0	KTTS	The machine type of TT20S is used(→1), or not(→0)
K9.1	KTTMS	The machine type of TT20MS is used(→1), or not(→0)
K9.2	KTTSY	The machine type of TT20SY is used(→1), or not(→0)
K9.3	KCUCPS	C-axis pressure switch is used(→0), or not(→1)
K9.4	KSTD	When standard door interlock function is used(→1), or not(→0)
K9.5	KCHIP	The alarm detection function of CHIP Conyeyor is used(→1), or not(→0)
K9.6	KCLFSH	Coolnat flusing is used(→1), or not(→0)
K9.7	KGL	Gantry loader is used(→1), or not(→0)
K10.0	KSPJOG	In auto mode, spindle jogging is invalid(→1), or not(→0)
K10.1	KT00	Program runs without alarm when T0000 instruction is ordered at auto mode(→1),or not(→0)
K10.4	KSPARE	Input signal is used to finish M91 and M92(→0), or not(→1)
K10.5		This bit can't use.(Only for Ez-guide T)
K10.6	KMILLS	When door interlock function of mills market is used(→1) or not(→0)
K10.7	KTEC	When door interlock function of italy tecnomach is used(→1) or not(→0)
K50.0	KMQL	The Dry Coolant for MQL Dry Machining is used(→1) or not(→0)
K50.1	KMIST	OIL Mist Collector is used(→1) or not(→0)
K50.2	KALAPF	IN ALARM AUTO POWER OFF FUNCTION is used(→1) or not(→0)
K50.3	KM23	M23 FUNCTION is used(→1) or not(→0)
K50.5	KTQCO	Torque control is used(→1), or not(→0)
K50.6	KNORQ	Q-setter is available without reference return completed(→1), or not(→0)
K50.7	KCOLF	Coolnt filter change is used(→1), or not(→0)
K51.0	KSELY	This bit can't use. (Only for DHIAC)
K51.1	KMGI	MANUAL GUIDE I is used(→1) or not(→0)

A.4 Keep relay list (3/3) of TT20/25SY

ADDRESS		MEANING
K51.6	KEDGE	Safety edge is used (→1) or not(→0)
K52.2	K12RET	Tool change is available at the 1st referance point(→1), at the 2nd referance point (→0)
K52.4	KDUSWU	Spindle warmming up function is used(→0), or not(→1)
K52.5	KCUSEN	Cycle start enable in the state of chuck unclamp(→1), or not(→0)
K52.6	KCHAIR	The air sensing for chuck-clamp confirmation is used(→1), or not(→0)
K52.7	KPBAST	Bucket and arm advance of part cacher are operated separately(→0), simultaneously(→1)
K55.6	KPSMAL	The running alarm of PSM module is detected(→0), or not(→1)
K60.4	KBSCO	When BED SHOWER COOLANT MOTOR is turned on in auto running automatically (→ 1), or not(→ 0)
K55.3	KALINT	Z, B, Y axis interlock function is used(→0), or not(→1)
K55.4	KC1C2I	C1& C2 ref. return is used(→0), or not(→1)
K55.6	KPSMAL	The running alarm of PSM module is detected(→0), or not(→1)
K55.7	KUIFIG	The interference check function for upper unit of SY machine is used(→1), or not(→0)
K60.4	KBSCO	When BED SHOWER COOLANT MOTOR is turned on in auto running automatically (→ 1), or not(→ 0)

B. Alarm list

B.1 Alarm list (1/6) of MX series

NO.	Message
2001	WHETHER EMERGENCY BUTTON IS PRESSED OR OVERTRAVEL OF AXES IS DETECTED, RELEASE EMERGENCY BUTTON AND LIMIT SWITCH FOR OVERTRAVEL CHECK.
2002	THE ALARMS OF THE MAIN SPINDLE MOTOR AND SERVO UNIT DETECTED, CHECK ALARM DISPLAY ON THE SERVO UNIT THEN RETRY AFTER POWER OFF-ON.
2003	OVERCURRENT IN THE CONTROL POWER CIRCUIT, ABNORMALLY LARGE CURRENT FLOW IS DETECTED IN THE CONTROL POWER CIRCUIT, CHECK FOR CONTROL POWER SHORT CIRCUIT.
2004	HYD. PUMP MOTOR OVERLOAD IS DETECTED, CHECK OVERLOAD VALUE ON THE THERMAL RELAY AND MOTOR.
2005	HYD. PRESSURE IS DOWN, CHECK HYD. PRESSURE SWITCH, PRESSURE VALUE AND LEAKAGE OF HYDRAULIC.
2007	SPINDLE ROTATION IS ABNORMAL.
2009	REVOVING SPINDLE EMERGENCY STOP
2010	INVERTER FAULT ALARM
2011	KEEP RELAY SETTING ALARM. PLEASE CHECK THE KEEP RELAY(K5<6>,K5<7>)
2016	PSM POWER RUNNING ALARM. PLEASE CHECK AUXILLARY CONTACT ON THE MAGNETIC CONTACTOR (KM10 OR KM110).
2019	MILLING SPINDLE ORIENTATION OVERTIME
2020	COOLANT & LUB. PUMP MOTOR OVERLOAD OR Q11 FOR SERVO TURRET, CHIP CONVEYOR AND BAR FEEDER OVERLOAD IS DETECTED.
2021	TORQUE SKIP DATA ERROR
2022	SPINDLE ORIENTATION CONFIRM IS NOT DETECTED WITHIN 10SEC, CHECK ORIENTATION PARAMETER AND READJUST. ORIENTATION INDEX COMMAND ALARM.
2023	TOOL INDEX OVERTIME
2024	TOOL NUMBER COMMAND ALARM
2025	TOOL NUMBER SELECT KEEP RELAY SET ALARM
2027	SPINDLE SPEED ARRIVAL IS NOT DETECTED WITHIN 7SEC, CHECK SPINDLE SERVO UNIT.
2028	SPINDLE ROTATION CONDITION IS NOT READY, CHECK SPINDLE ROTATION CONDITION.
2029	SPLASH GUARD DOOR IS OPEN, CHECK SPLASH GUARD DOOR OPEN ANDLIMIT SWITCHES.
2030	M17 OR M18 IS COMMANDED IN ILLEGAL MODE, PLEASE CONFIRM MDI MODE.
2031	PLEASE MANUAL REF.POINT RETURN, OTHERWISE AUTO.OPERATION IS IMPOSSIBLE.
2032	FEEDHOLD PUSH BUTTON IS PRESSED.
2033	AIR OR COOLANT PRESSURE SWITCH IS DOWN.
2034	PROXIMITY SWITCH FOR CHUCK CLAMP CONFIRM POSITION ERROR, RESET SWITCH POSITION (PX1.M) (PX2.M)
2035	AXIS CHANGE OVERTIME.
2036	M68 OR M69, M168 OR M169 IS NOT FINISHED WITHIN 15SEC AFTER COMMAND, CHECK SOLENOID VALVE, POSITION SWITCH AND OPERATION CONDITION.

B.1 Alarm list (2/6) of MX series

NO.	Message
2037	M78 OR M79 IS NOT FINISHED, WITHIN 15SEC AFTER COMMAND, CHECK SOLENOID VALVE, POSITION SWITCH AND OPERATION CONDITION.
2038	M45,M46,M47 IS NOT FINISHED WITHIN 15SEC AFTER COMMAND, CHECK SOLENOID VALVE, POSITION SWITCH AND OPERATION CONDITION.
2039	M10 OR M11 COMMAND ALARM, CHECK SOLENOID VALVE, POSITION SWITCH AND OPERATION CONDITION.
2040	M50 OR M51 IS NOT FINISHED, WITHIN 15SEC AFTER COMMAND, CHECK SOLENOID VALVE, POSITION SWITCH AND OPERATION CONDITION.
2041	THE ALARMS OF THE BARFEEDER IS DETECTED, CHECK BAR-FEEDER.
2042	M52, M53, M152 OR M153 IS NOT FINISHED, WITHIN 15SEC AFTER COMMAND, CHECK SOLENOID VALVE, POSITION SWITCH AND OPERATION CONDITION.
2043	M58 OR M59 IS NOT FINISHED, WITHIN 15SEC AFTER COMMAND, CHECK SOLENOID VALVE, POSITION SWITCH, AND OPERATION CONDITION.
2045	Q-SETTER OPERATION ALARM. CHECK SENSOR AND SOLENOID. TO MEASURE X1-AXIS THAT Y-AXIS SHOULD BE LOCATED AT 0. (1ST.REF) POSITION
2046	CHUCK IS OPENED DURING THE SPINDLE ROTATION, CHECK HYD.PRESSURE.
2047	TURRET CLAMP SWITCH ERROR
2048	THE RESET COMMAND IS USED WHILE PROGRAM IS RUNNING.
2049	M88,M89,M90,M188,M189,M190 COMMAND ALARM. CHECK SOLENOID VALVE, POSITION SWITCH AND OPERATION CONDITION.
2050	LUB.PRESSURE IS DOWN OR LUBRICATION OIL LEVEL IS LOW,CHECK LUB.UNIT AND REPLENISH THE OIL.
2051	OIL COOLER ALARM
2052	M61 OR M62, M161 OR M162 IS NOT FINISHED WITHIN 15SEC AFTER COMMAND. PLEASE CHECK INITIAL CONDITION.
2053	WORK EJECTOR & DETECTOR ALARM
2054	MILLING SPINDLE IS UNCLAMPED.MILLING SPINDLE SHOULD BE CLMAPED IN TURNING MODE.
2055	ILLEGAL SYNCHRO INITIAL STATE IS DETECTING . PLEASE CHECK CHUCK STATE.
2056	THIS COMMAND CANNOT BE PERFORMED. PLEASE PERFORM THE REF. POSITION RETURN OF X & Y-AXIS BEFORE THIS COMMAND.
2057	TOOL LIFE COUNT END ALARM.
2058	PARTS COUNT END ALARM.
2059	M,T-CODE COMMAND ALARM WITH A-AXIS.
2060	CHUCK UNCLAMP STATUS.
2061	SAFETY LOCKING SWITCH ALARM. PLEASE CHECK CONDITION OF THE LOCKING KEY SWITCH OR WIRE CONDITION THAT IS CONNECTING BETWEEN NO.11 AND NO.12 IN SAFET LOCKING SWITCH.
2062	LIMIT SWITCH FOR PROGRAMABLE TAILSTOCK IS NOT OPERATED WITHN 5SEC AFTER COMMAND, CHECK SOLENOID VALVES AND LIMIT SWITCHES.
2063	PROXIMITY SWITCH FOR Q-SETTER IS NOT OPERATED WITHIN 5SEC AFTER COMMAND, CHECK SOLENOID VALVES AND PROXIMITY SWITCHES.
2064	M66 OR M67 COMMAND ON THE M/C, THE NEXT M03 OR M04 COMMAND.

B.1 Alarm list (3/6) of MX series

NO.	Message
2065	Y-AXIS IS NOT IN A REFERENCE POSITION. PLEASE RETURN THE Y-AXIS TO REFERENCE POSITION FOR BT-AXIS MOVING.
2066	CHECK POSITION DETECTING SWITCHES ARE OPERATED SIMULTANEOUSLY, CHECK SWITCH.
2067	TOOL SWITCH ALARM
2068	DRY RUN STATUS. IF YOU PRESS CYCLE START BUTTON, THIS ALARM WILL RESET.
2069	COOLANT FILTER CHANGE ALARM
2070	SPINDLE CAMMAND IS ABNORMAL. CHANGE SPINDLE COMMAND AFTER SPINDLE STOP(M05/M35/M65/M105/M165).
2071	CHIP CONVEYOR ALARM
2072	TORQUE ALARM
2073	ILLEGAL COMMAND ALARM
2074	COOLJET LOW PRESSURE OR FILTER BLOCK ALARM
2075	TOTAL CYCLE START ALARM
2076	HIGH PRESSURE FILTER OR COOLJET FILTER BLOCK
2077	MILLING SPINDLE CLAMP, UNCLAMP COMMAND ALARM. MILLING SPINDLE IS ROTATING OR SENSOR IS ABNORMAL.
2078	B-AXIS DISC CLAMP COMMAND IS NOT AVAILABLE WHEN NEW TYPE B-AXIS IS USED.
2079	B-AXIS SENSOR ALARM. PLEASE CHECK THE B-AXIS DISC CLAMP CHECK SENSOR AND CURVIC CLAMP CHECK SENSOR.
2080	B-AXIS ANGLE IS NOT A MULTIPLE OF 15 DEGREE
2081	B-AXIS POSITION S/W ALARM
2082	MILLING SPINDLE IS CLAMPED YET
2083	MILLING SPINDLE IS UNCLAMPED YET
2084	IN HANDLE MODE, CAN'T CHANGE TOOL OFFSET WITH Q-SETTER. PLEASE, CHANGE MODE TO JOG MODE AND TRY AGAIN.
2085	RETURN THE REFERENCE POINT OF X1 AND X2-AXIS FOR MOVING A-AXIS.
2086	TOOL MONITOR ALARM.DETECT ABNORMAL LOAD AND TOOL WARN ALARM
2087	TOOL MONITOR ALARM.DETECT ABNORMAL LOAD AND TOOL BROKEN ALARM
2088	THE SPINDLE SWITCHING IS INVALID IN RIGID TAPPING MODE. CANCEL RIGID TAPPING MODE TO SWITCH OTHER SPINDLE.
2089	B-AXIS IS NOT CLAMPED YET. PLEASE MAKE SURE THAT THE B-AXIS IS CLAMPED BY M106,M110 OR M112.
2090	PROGRAM RESTART TOGGLE SWITCH ON STATE.
2091	SERVO TURRET BATTERY LOW ALARM, PLEASE CHANGE BATTERY IN AMP. MODULE WITHIN 2 WEEKS.

B.1 Alarm list (4/6) of MX series

NO.	Message
2092	SPINDLE COMMAND ALARM. WHEN COMPOSITION CONTROL MODE, REV-SPINDLE RECOMMAND IS IMPOSSIBLE AFTER REV-SPINDLE STOP.
2093	REFERENCE COMMAND FROM REFERENCE POSITION KEEP STATE.
2094	HIGH LOW CHUCK SWITCHING STATE. OPERATE CHUCK AS UNCLAMPING AND CLAMPING BY FOOT SWTCH.
2095	RIGID TAPPING OR ORIENTATION COMMAND ALARM. M19 OR M119, M29 IS INVALID ON G96 MODE. PLEASE MODIFY PROGRAM.
2096	SPINDLE OVERRIDE OR FEEDRATE 0%.
2097	MACHINE LOCK ON STATUS.
2098	M00 OR M01 COMMAND.
2099	M02 OR M30 COMMAND.
2100	2100 MACHINE IS IN SERVICE MODE. 1)MACHINE WILL OPERATE WITH DOOR OPEN. 2)SPINDLE WILL NOT START UNLESS THE DOOR IS CLOSED.3)TAPPING OR THREADING MAY NOT WORK UNLESS THE DOOR IS
2101	2101 SPINDLE TOOL UNCLAMP ALARM
2102	2102 MAIN POWER PHASE ALARM
2103	2103 M06 OVERTIME ALARM
2104	2104 T-CODE COMMAND ERROR
2105	2105 ATC OPERATION COMMAND ERROR
2106	2106 TOOL POST SEARCH OVERTIME ALARM
2107	2107 ATC MAGAZINE ROTATION OVERTIME ALARM
2108	2108 ATC DOOR SWITCH ERROR
2109	2109 TOOL MAGAZINE SYNCHRO MAL-FUNCTION
2110	2110 ATC INTERLOCK ALARM
2111	2111 ATC MAGAZINE GUARD DOOR OPEN
2112	2112 MAINTENANCE MODE ON STATE
2113	2113 M149 COMMAND ERROR
2114	2114 SPINDLE TOOL INTERFERENCE ALARM
2115	2115 SPINDLE TOOL NO.IS MISMATCHED
2116	2116 MINI-TURRET 180 COMMAND ALARM. MILLING SPINDLE SHOULD BE ORIENTATED BEFORE COMMAND G400 J12.
2117	2117 MUST BE ATC 1CYCLE
2118	2118 ATC OP MANUAL MODE

B.1 Alarm list (5/6) of MX series

NO.	Message
2119	2119 SPINDLE AND WAITING TOOL NUMBER IS ZERO.
2120	2120 SPLASH GUARD DOOR MUST BE OPENED WHEN THE TOOL UNCLAMP OPERATION IS DOING IN MANUAL MODE
2121	2121 TOOL NUMBER SELECT KEEP RELAY IS NOT SET PROPERLY.
2122	2122 TOOL CHANGER MOTOR BRAKE RELEASE
2123	2123 TOOL SEARCH ILLEGAL ALARM
2124	2124 M06 COMMAND ILLEGAL POSITION
2125	2125 SPINDLE TOOL CLAMP/UNCLAMP ALARM
2126	2126 WAITING POT SWITCH ERROR
2127	2127 CHANGER ARM POSITION CHECK ALARM
2128	2128 MUST BE COMMAND NEXT TOOL NO.
2129	2129 ATC OPERATION COMMAND ERROR
2130	2130 CONSTANT SURFACE COMMAND ALARM.IN COMPOSITION CONTROL MODE, THE CONSTANT SURFACE CONTROL COMMAND FROM LOWER(UPPER) MUST BE COMMANDED AFTER MILLING ROTATING COMMAND
2131	2131 SPINDLE COMMAND ALARM. AFTER SPINDLE SELECTION, SPINDLE ROTATION COMMAND IS VALID(M05/M35/M105/M135). IF SPINDLE TO SELECT IS ROTATING, MUST BE STOP SPINDLE TO SELECT
2132	2132 ATFER MAIN POWER OFF/ON, C1 AND C2 AXES MUST BE RETURN REF.POINT BY MANUAL, OTHERWISE AUTO.OPERATION IS IMPOSSIBLE.
2133	2133 COMPOSITION CONTROL COMMAND ALARM. AFTER SELECT C2-AXIS(M135)FROM UPPER OR C1-AXIS(M35) FROM LOWER, M211 COMMAND IS POSSIBLE.
2134	2134 TURRET CONTROL UNIT ALARM. CHECK SERVO CONTORL UNIT IN CONTROL CABINET.
2135	2135 LEFT SPINDLE RIGID TAP IS VALID ONLY FOR THE UPPER PATH AND RIGHT SPINDLE RIGID TAP IS VALID ONLY FOR LOWER PATH
2150	2150 A-AXIS MOVEMONT IS PROHIBITED IN CHUCKING CLOSED CONDITION
2161	2161 SPINDLE WARM UP IS REQUESTED. PLEASE RUNNING ALL SPINDLES MORE THAN 15 MINUETES UNDER 1,000RPM
2162	2162 SPINDLE WARM UP IS COMPLETED
2165	2165 CHECK HOME POSITION OF EACH UNIT FOR PARTS CATCHER(STOPPER ADV:X64.5, BODY RET:X64.6, BUCKET RET:X65.2, ARM RET:X65.4)
2166	2166 A-AXIS IS REF.POINT AND Z1,Z2-AIXS IS IN RIGHT POSITION REGION FROM REF.POINT, M310/M311/M256 COMMAND CONDITION :A-AXIS IS REF. POINT AND Z1 AND Z2 AXIS IS IN LEFT POSITION REGION FROM REF.
2167	2167 CUTTING FEED COMMAND IS COMMANDED WHEN THE B-AXIS IS NOT CLAMPED YET.
2168	2168 RETURN A-AXIS TO THE REFERENCE POINT FOR MOVING Z2-AXIS OR COMMAND M217 TO RELEASE Z2 AXIS INTERLOCK.
2169	2169 EXIN DATA READ ALARM.
2170	2170 IT IS NOT ABLE TO MOVE Z-AXIS WHEN WAITING POT IS NOT IN THEIR HOME POSITION(WAITING POT UP AND HOME POSITION)
2173	2173 LUBRICATION OIL LEVEL IS LOW.

B.1 Alarm list (6/6) of MX series

NO.	Message
2174	2174 TOUCH PROBE UNIT ALARM. BATTERY IS LOW OR SIGNAL LEVEL IS LOW.
2175	2175 TRANSFERRING WITH FEED FROM THE TURRET UNCLAMP CONDITION WHICH IS IMPOSSIBLE.
2176	2176 THIS COMMAND CANNOT BE PERFORMED. PLEASE PERFORM THE REF. POSITION RETURN OF C1 & C2-AXIS BEFORE THIS COMMAND.
2177	2177 TOOL SPINDLE OVERLOAD ALARM, PLEASE CHECK CUTTING CONDITION IN PROGRAM.
2178	2178 C-AXIS HIGH/LOW CLAMP OR UNCLAMP M-COMMAND CONDITION ALARM. THE M88 OR M89,M90(M188 OR M189,M190) COMMAND OF THE OTHER PATH IS VALID IN COMPOSITION CONTROL STATE.
2179	2179 LOWER TURRET CAN NOT INDEX OR MOVE TO -Z2 DIRECTION IN INTERFERENCE AREA WITH LEFT SPINDLE. PLEASE MOVE THE Z2-AXIS TO THE RIGHT WHERE THE INTERFERENCE WITH LEFT SPINDLE IS NOT
2180	2180 X2-AXIS CAN NOT MOVE IN INTERFERENCE AREA WITH LEFT SPINDLE. PLEASE MOVE THE Z2-AXIS TO THE RIGHT WHERE THE INTERFERENCE WITH LEFT SPINDLE IS NOT EXIST.
2181	2181 KNOLL COOLANT SYSTEM ALARM@0A@
2182	2182 Y-AXIS & A-AXIS ARE IN INTERFERENCE AREA. PLEASE CHECK THE INTERFERENCE AREA
2185	2185 CHECK HOME POSITION OF EACH UNIT FOR PARTS CATCHER(STOPPER ADV:X64.5, BODY RET:XX64.6, BUCKET RET:X65.2, ARM RET:X65.4@0A@)
2186	2186 WHEN PARTS UNLOADER IS OPERATIING BY M256 COMMAND,THE Z1,Z2-AXIS CAN MOVE ONLY IN THE LEFT POSITION REGION FROM REF.POINT.
2187	2187 RIGHT SPINDLE ROTATION STATE. PLEASE SPINDLE STOP.
2188	2188 PROGRAM START M FUNCTION ALARM. PLEASE CHECK M-CODE (M23)
2189	2189 C-AXIS BRAKE NOT RELEASED, PLEASE CHECK C-AXIS UNCLAMP SOLENOIDE VALVE AND PRESSURE SWITCH.
2190	2190 PROGRAM RESTART ERROR BY OPTIONAL STOP. MUST BE IN STATE OF UPPER/LOWER UNIT AND MEM MODE TO RESTART PROGRAM STOPPED BY OPTIONAL STOP ON MEM MODE.
2192	2192 TAIL STOCK INTERLOCK ALARM, PLEASE CHECK THE POSITION OF Z-AXIS AND TAIL STOCK.
2193	2193 THE PROGRAM NUMBER OF UPPER UNIT AND ONE OF LOWER UNIT ARE NOT SAME. PLEASE CHECK PROGRAM NUMBER OR SET K60.1(CHECK PROGRAM NUMBER) TO 0

B.2 Alarm list (1/5) of TT series

NO.	Message
2001	2001 WHETHER EMERGENCY BUTTON IS PRESSED OR OVERTRAVEL OF AXES IS DETECTED, RELEASE EMERGENCY BUTTON AND LIMIT SWITCH FOR OVERTRAVEL CHECK.
2002	2002 THE ALARMS OF THE LEFT SPINDLE MOTOR AND SERVO UNIT DETECTED, CHECK ALARM DISPLAY ON THE SERVO UNIT THEN RETRY AFTER POWER OFF-ON.
2003	2003 OVERCURRENT IN THE CONTROL POWER CIRCUIT, ABNORMALLY LARGE CURRENT FLOW IS DETECTED IN THE CONTROL POWER CIRCUIT, CHECK FOR CONTROL POWER SHORT CIRCUIT.
2004	2004 HYD. PUMP MOTOR OVERLOAD IS DETECTED, CHECK OVERLOAD VALUE ON THE THERMAL RELAY AND MOTOR.
2005	2005 HYD. PRESSURE IS DOWN, CHECK HYD. PRESSURE SWITCH, PRESSURE VALUE AND LEAKAGE OF HYDRAULIC.
2006	2006 HYD.PUMP MOTOR IS NOT RUN, CHECK DGN.(Y) AND MOTOR STARTER (-KM31).
2007	2007 SPINDLE ROTATION IS ABNORMAL.
2008	2008 GEAR SHIFT LIMIT SWITCH IS ABNORMALLY OPERATING AS SWITCHS OPERATE SIMULTANEOUSLY.
2009	2009 THE ALARMS OF THE RIGHT SPINDLE MOTOR AND SERVO UNIT DETECTED, CHECK ALARM DISPLAY ON THE SERVO UNIT THEN RETRY AFTER POWER OFF-ON.
2010	2010 OIL LEVEL OF GEAR BOX IS LOW, CHECK THE OIL LEVEL OF GEAR BOX.
2011	2011 KEEP RELAY ALARM OF MACHINE SELECTION. PLEASE CHECK THE KEEP RELAY(K9<1>,K9<2> & K9<3>)
2016	2016 PSM POWER RUNNING ALARM. PLEASE CHECK AUXILLARY CONTACT ON THE MAGNETIC CONTACTOR (KM10, KM110).
2017	2017 SPINDLE OIL OVERFLOW, CHECK LEVEL S/W OR OIL DRAIN MOTOR.
2019	2019 MACHINE MODEL SELECTION DATA IS SETTED ABNORMALLY, CHECK.DGN(K9<1>,K9<2>,K9<3>).
2020	2020 COOLANT & LUB. PUMP MOTOR OVERLOAD OR Q11 FOR SERVO TURRET, CHIP CONVEYOR AND BAR FEEDER OVERLOAD IS DETECTED.
2021	2021 TORQUE LIMIT SKIP COMMAND(M86,M87) IS INDICATED INCORRECTLY.
2022	2022 SPINDLE ORIENTATION CONFIRM IS NOT DETECTED WITHIN 15SEC, CHECK ORIENTATION PARAMETER AND READJUST. ORIENTATION INDEX COMMAND ALARM.
2023	2023 EACH TOOL INDEXING IS NOT FINISHED WITHIN 7SEC AFTER TOOLINDEX COMMAND.(MANUAL OR AUTO).
2024	2024 TOOL CHANGE COMMAND ALARM. PLEASE CHECK TOOL COMMAND NUMBER OR PROGRAM STATE. IN ADDITION, TOOL COMMAND IS NOT AVAILABLE ON POLYGON MODE.
2025	2025 TOOL NUMBER SELECTION DATA IS SETTED ABNORNALLY, CHECK DGN(K6-B1,K6-B2).
2026	2026 GEAR SHIFT IS NOT FINISHED WITHIN 15SEC, REMOVE CAUSE AND PUSH ALARM RESET BUTTON.
2027	2027 SPINDLE SPEED ARRIVAL IS NOT DETECTED WITHIN 7SEC, CHECK SPINDLE SERVO UNIT.
2028	2028 SPINDLE ROTATION CONDITION IS NOT READY, CHECK SPINDLE ROTATION CONDITION.
2029	2029 SPLASH GUARD DOOR IS OPEN, CHECK SPLASH GUARD DOOR OPEN ANDLIMIT SWITCHES.
2030	2030 M17 OR M18 IS COMMANDED IN ILLEGAL MODE, PLEASE CONFIRM MDI MODE.
2031	2031 PLEASE MANUAL REF.POINT RETURN, OTHERWISE AUTO.OPERATION IS IMPOSSIBLE.
2032	2032 FEEDHOLD PUSH BUTTON IS PRESSED.

B.2 Alarm list (2/5) of TT series

NO.	Message
2033	2033 AIR OR COOLANT PRESSURE SWITCH IS DOWN.
2034	2034 PROXIMITY SWITCH FOR CHUCK CLAMP CONFIRM POSITION ERROR, RESET SWITCH POSITION (PX1.M) (PX2.M)
2035	2035 AXIS CHANGE OVERTIME.
2036	2036 M68/M168 OR M69/M169 IS NOT FINISHED WITHIN 15SEC AFTER COMMAND, CHECK SOLENOID VALVE, POSITION SWITCH AND OPERATION CONDITION.
2037	2037 M78 OR M79 IS NOT FINISHED, WITHIN 15SEC AFTER COMMAND, CHECK SOLENOID VALVE, POSITION SWITCH AND OPERATION CONDITION.
2038	2038 M10 OR M11 COMMAND ALARM, CHECK SOLENOID VALVE, POSITION SWITCH AND OPERATION CONDITION.
2039	2039 PARTS UNLOADER OVERTIME ALARM, CHECK SOLENOID VALVE, POSITION SWITCH AND OPERATION CONDITION.
2040	2040 M50 OR M51 IS NOT FINISHED, WITHIN 15SEC AFTER COMMAND, CHECK SOLENOID VALVE, POSITION SWITCH AND OPERATION CONDITION.
2041	2041 THE ALARMS OF THE BARFEEDER IS DETECTED, CHECK BAR-FEEDER.
2042	2042 M52, M53, M152 OR M153 IS NOT FINISHED, WITHIN 15SEC AFTER COMMAND, CHECK SOLENOID VALVE, POSITION SWITCH AND OPERATION CONDITION.
2044	2044 THE PARTS CATCHER IS ADVANCED, EVEN THOUGH M10 IS NOT COMMANDED.
2045	2045 M80 OR M81 COMMAND ALARM.
2046	2046 CHUCK IS OPENED DURING THE SPINDLE ROTATION, CHECK HYD.PRESSURE.
2047	2047 TURRET CLAMP SWITCH ALARM. IN THE MACHINE, CHECK KEEP RELAY 5.4 IS SET TO 0 OR CHECK TURRET CLAMP SWITCH.
2048	2048 THE RESET COMMAND IS USED WHILE PROGRAM IS RUNNING.
2049	2049 M88/M188,M89/M189,M90/M190 COMMAND ALARM. CHECK SOLENOID VALVE, POSITION SWITCH AND OPERATION CONDITION.
2050	2050 LUB.PRESSURE IS DOWN OR LUBRICATION OIL LEVEL IS LOW,CHECK LUB.UNIT AND REPLENISH THE OIL.
2051	2051 COOLANT LEVEL LOW IS CHECK, REPLENISH THE COOLANT.
2052	2052 M61 OR M62 IS NOT FINISHED WITHIN 15SEC AFTER COMMAND. PLEASE CHECK INITIAL CONDITION.
2053	2053 M116,M120 COMMAND ALARM. PLEASE CHECK THE SWITCHS, THE WORKPIECE AND OPERATION CONDITION. OR CUTTING POSITION BY M202 IS ERROR. PLEASE CKECK THE CUTTING STATE.
2054	2054 REV.-TOOL IS NOT ENGAGED WITHIN 3SEC, CHECK REV.-TOOL STATE.
2055	2055 ILLEGAL SYNCHRO INITIAL STATE IS DETECTING . PLEASE CHECK CHUCK STATE.
2056	2056 M206 IS NOT USED. PLEASE COMMAND M206.
2057	2057 TOOL LIFE COUNT END ALARM.
2058	2058 PARTS COUNT END ALARM.
2059	2059 M,T-CODE COMMAND ALARM IN B-AXIS CONTROL. CHECK WHICH PROPER M-CODE OR T-CODE IS USED IN B-AXIS MACHINING PROGRAM AND THEN CHECK SOLENOID VALVE OR SWITCH FOR M-CODE EXECUTION. CHECK THE SAME M-CODE IS EXECUTING IN THE BOTH MACHINING PROGRAMS.

B.2 Alarm list (3/5) of TT series

NO.	Message
2060	2060 CHUCK UNCLAMP STATUS.
2061	2061 SAFETY LOCKING SWITCH ALARM. PLEASE CHECK CONDITION OF THE LOCKING KEY SWITCH OR WIRE CONDITION THAT IS CONNECTING BETWEEN NO.11 AND NO.12 IN SAFET LOCKING SWITCH.
2062	2062 LIMIT SWITCH FOR PROGRAMABLE TAILSTOCK IS NOT OPERATED WITHN 5SEC AFTER COMMAND, CHECK SOLENOID VALVES AND LIMIT SWITCHES.
2063	2063 PROXIMITY SWITCH FOR Q-SETTER IS NOT OPERATED WITHIN 5SEC AFTER COMMAND, CHECK SOLENOID VALVES AND PROXIMITY SWITCHES.
2064	2064 M66 OR M67 COMMAND ON THE M/C, THE NEXT M03 OR M04 COMMAND.
2065	2065 TURRET TOOL NO.DETECTION ENCODER SIGNAL DISCONNECT.
2066	2066 CHECK POSITION DETECTING SWITCHES ARE OPERATED SIMULTANEOUSLY, CHECK SWITCH.
2067	2067 THE TOOL AND REV.SPINDLE INDEXING CONDITION ALARM. CHECK CONFIRMATION SWITCH IN TOOL POST UNIT.
2068	2068 RETURN B-AXIS TO THE REFERENCE POINT FOR MOVING Z2-AXIS OR COMMAND M217 TO RELEASE Z2 AXIS INTERLOCK.
2069	2069 COOLANT FILTER CHANGE ALARM
2070	2070 SPINDLE CAMMAND IS ABNORMAL. CHANGE SPINDLE COMMAND AFTER SPINDLE STOP(M05/M35/M135/M65/M105/M165).
2071	2071 CHIP CONVEYOR ALARM CHIP CONVEYOR ALARM
2072	2072 TORQUE CONTROL ALARM.
2073	2073 THIS COMMAND CANNOT BE PERFORMED. PLEASE PERFORM THE REF. POSITION RETURN OF X & Y-AXIS BEFORE THIS COMMAND.
2074	2074 IN HANDLE MODE, CAN'T CHANGE TOOL OFFSET WITH Q-SETTER. PLEASE, CHANGE MODE TO JOG MODE AND TRY AGAIN.
2075	2075 TRANSFERRING WITH FEED FROM THE TURRET UNCLAMP CONDITION WHICH IS IMPOSSIBLE.
2076	2076 TOOL SPINDLE OVERLOAD ALARM, PLEASE CHECK CUTTING CONDITION IN PROGRAM.
2077	2077 RETURN THE REFERENCE POINT OF X1 AND X2-AXIS AND RETURN THE ARM OF PART UNLOADER TO RETURN POSITION FOR MOVING B-AXIS.
2078	2078 OIL COOLER ALARM.
2079	2079 C-AXIS BRAKE NOT RELEASED, PLEASE CHECK C-AXIS UNCLAMP SOLENOIDE VALVE AND PRESSURE SWITCH.
2081	2081 GANTRY LOADER SERVO IS NOT READY
2082	2082 THERE ARE LOADER AXES INSIDE OF MACHINE. M80, M10, M03, M04, M63, M64, M78, M103, M104, M203, M204 AND T-CODE IS INVALID. MOVE Y-AXIS OF LOADER TO PLUS DIRECTION. THIS ALARM CAN BE RESET BY ONLY PRESSING NC RESET BUTTON ON MDI PANEL.
2083	2083 THERE ARE LOADER AXES INSIDE OF MACHINE. Q-SETTER DOWN, PARTS CATCHER ADVANCE, TAIL STOCK QUILL ADVANCE, TOOL INDEX, SPINDLE ROTATION, AND AXIS MOVING BESIDE OF HANDLE IS INVALID. MOVE Y-AXIS OF LOADER TO PLUS DIRECTION OVER TOP DOOR.
2084	2084 B-AXIS OR Z1 AXIS MOVING ALARM. WHEN THE DISTANCE BETWEEN B-AXIS AND Z1 AXIS IS SHORTER THAN D1112,B-AXIS OR Z1 AXIS MOVING IS IMPOSSIBLE.
2088	2088 LUBRICATION OIL LEVEL IS LOW.
2089	2089 SERVO TURRET BATTERY LOW ALARM, PLEASE CHANGE BATTERY IN AMP. MODULE WITHIN 2 WEEKS.

B.2 Alarm list (4/5) of TT series

NO.	Message
2090	2090 PROGRAM RESTART PUSH BUTTON SWITCH ON STATE.
2091	2091 THE SPINDLE SWITCHING IS INVALID IN RIGID TAPPING MODE. CANCEL RIGID TAPPING MODE TO SWITCH OTHER SPINDLE.
2093	2093 REFERENCE COMMAND FROM REFERENCE POSITION KEEP STATE.
2094	2094 HIGH LOW CHUCK SWITCHING STATE. OPERATE CHUCK AS UNCLAMPING AND CLAMPING BY FOOT SWTCH.
2095	2095 RIGID TAPPING OR ORIENTATION COMMAND ALARM. M19 OR M119, M29 IS INVALID ON G96 MODE. PLEASE MODIFY PROGRAM.
2096	2096 FEEDRATE OVERRIDE 0%.
2097	2097 MACHINE LOCK ON STATUS.
2100	2100 2100 MACHINE IS IN SERVICE MODE. 1)MACHINE WILL OPERATE WITH DOOR OPEN. 2)SPINDLE WILL NOT START UNLESS THE DOOR IS CLOSED.3)TAPPING OR THREADING MAY NOT WORK UNLESS THE DOOR IS CLOSED.
2101	2101 2101 PLEASE, CLOSE THE DOOR
2102	2102 2102 M310/M311 COMMAND CONDITION :B-AXIS AND X1,X2-AXIS ARE IN REFT. POINT, THE MACHINE COORDINATE OF Z1-AXIS MUST BE LESS THAN THE SETTING VALUE OF PARAMETER 6932(ONLY UPPER UNIT).
2104	2104 2104 CHECK HOME POSITION OF EACH UNIT FOR PARTS CATCHER(STOPPER ADV:X9.7, BODY RET:X11.2, BUCKET RET:X11.7, ARM RET:X22.7)
2106	2106 2106 RIGHT SPINDLE ROTATION STATE. PLEASE SPINDLE STOP.
2107	2107 2107 PROGRAM START M FUNCTION ALARM. PLEASE CHECK M-CODE (M23).
2108	2108 2108 SPINDLE WARM UP IS REQUESTED.
2109	2109 2109 SPINDLE WARM UP IS COMPLETED.
2110	2110 2110 CONSTANT SURFACE CONTROL COMMAND ALARM.IN COMPOSITION CONTROL MODE, THE CONSTANT SURFACE CONTROL COMMAND FROM LOWER(UPPER) MUST BE COMMANDED AFTER MILLING ROTATING COMMAND FROM UPPER(LOWER).
2111	2111 2111 SPINDLE COMMAND ALARM. AFTER SPINDLE SELECTION, SPINDLE ROTATION COMMAND IS VALID(M05/M35/M105/M135).IF SPINDLE TO SELECT IS ROTATING, MUST BE STOP SPINDLE TO SELECT BEFORE SPINDLE SELECTION COMMAND.
2112	2112 2112 C-AXIS HIGH/LOW CLAMP OR UNCLAMP M-COMMAND CONDITION ALARM. THE M88 OR M89,M90(M188 OR M189,M190) COMMAND OF THE OTHER PATH IS VALID IN COMPOSITION CONTROL STATE.
2113	2113 2113 RETURN THE REFERENCE POINT OF X AND Z-AXIS BEFORE TOOL CHANGE COMMAND.TOOL CHANGE INTERLOCK CAN BE OVERRIDDEN BY M-COMMAND(M252=INTERLOCK DISABLE & M253=INTERLOCK ENABLE).
2114	2114 2114 ATFER MAIN POWER OFF/ON, C1 AND C2 AXES MUST BE RETURN REF.POINT BY MANUAL, OTHERWISE AUTO.OPERATION IS IMPOSSIBLE.
2115	WHEN PARTS UNLOADER IS OPERATIING BY M310 COMMAND,THE Z1-AXIS CAN MOVE ONLY IF THE MACHINE COORDINATE OF Z1-AXIS IS LESS THAN THE SETTING VALUE OF PARAMETER 6932(ONLY UPPER UNIT)
2116	2116 2116 LEFT SPINDLE RIGID TAPPING IS VALID ONLY FOR THE UPPER TURRET AND RIGHT SPINDLE RIGID TAPPING IS VALID ONLY FOR THE LOWER TURRET.
2117	2117 2117 PROGRAM RESTART ERROR BY OPTIONAL STOP. MUST BE IN STATE OF UPPER/LOWER UNIT AND MEM MODE TO RESTART PROGRAM STOPPED BY OPSIONAL STOP ON MEM MODE.
2118	2118 2118 PLEASE, CHECK Q-SETTER CAP.
2119	2119 2119 NOT USE REV.(ROTARY) SPINDLE OPERATION AT THE HALF STATION(13-24).

B.2 Alarm list (5/5) of TT series

NO.	Message
2120	2120 2120 WHEN PARTS CATCHER IS OPERATIING BY M10 COMMAND, THE MACHINE COORDINATE OF Z2-AXIS MUST BE MORE THAN THE SETTING VALUE OF PARAMETER 6951(ONLY LOWER UNIT).
2121	2121 2121 THE PART UNLOADER IS NOT IN HOME POSITION. SET THE PART UNLOADER TO HOME POSITION.
2122	2122 2122 TURRET CONTROL UNIT ALARM. CHECK SERVO CONTORL UNIT IN CONTROL CABINET.
2123	2123 2123 COOLANT LEVEL IS LOW, REPLENISH COOLANT.
2124	2124 2124 Z,B,Y-AXIS INTERLOCK FUNCTION OFF. FOR Z,B,Y-AXIS INTERLOCK FUNCTION ON, SET K55.3 TO 0.
2125	2125 2125 MOVE X=30000 ON MACHINE COORDINATE.AND SET Y-AXIS REF.POINT.
2126	2126 2126 THIS COMMAND CANNOT BE PERFORMED. PLEASE PERFORM THE REF. POSITION RETURN OF Y-AXIS BEFORE THIS COMMAND.
2149	2149 2149 COMPOSITION CONTROL COMMAND ALARM. AFTER SELECT C2-AXIS(M135)FROM UPPER OR C1-AXIS(M35) FROM LOWER, M211 COMMAND IS POSSIBLE.
2150	2150 2150 B-AXIS MOVEMONT IS PROHIBITED IN CHUCKING CLOSED CONDITION.