



PROGRAMMING MANUAL

FOR

MAZAK

MAZATROL CAM M-2

EIA/ISO



INTRODUCTION

This manual will describe the EIA/ISO program functions of MAZATROL CAM M-2.

Refer to the respective manuals mentioned below for MAZATROL language functions and the general handling procedure for the machine.

1. Operating Manual for MAZATROL CAM M-2
2. Programming Manual for MAZATROL CAM M-2

Please note that no other function than mentioned in the manual is provided.

The specifications of the machine are subject to change for improvement without advance notice.



Programming Manual for EIA/ISO

CONTENTS

1. OPERATION HANDLING

1-1	Paper Tape Input and Output	1-2
1-2	EIA/ISO Programs	1-11
1-3	Tool Offset Data	1-15
1-4	Work Offset Parameters	1-17
1-5	Operation	1-20

2. PROGRAM

2-1	Cautions as to MAZATROL CAM M-2	2-2
2-2	Feed Function (F-function)	2-7
2-3	Preparatory Function (G-function)	2-8
2-4	Correction Function	2-19
2-5	Canned Cycle Function	2-46
2-6	Spindle Function (S-function), Tool Functions (T- and E-functions), Auxiliary Function (M-function), Second Auxiliary Function (B-function)	2-68
2-7	Mirror Image	2-70
2-8	Sub-programs	2-71
2-9	User Macro (option)	2-72



1. OPERATION HANDLING

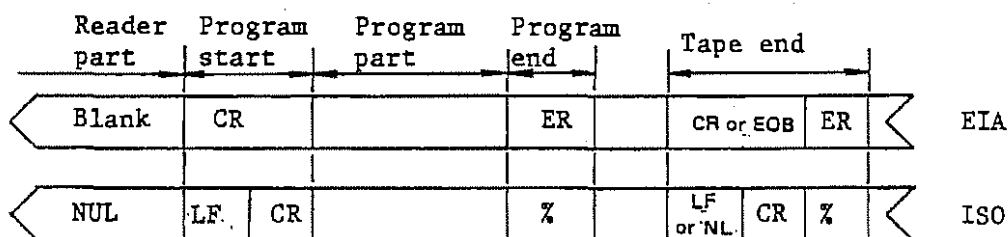


1-1 Paper Tape Input and Output

1-1-1 EIA/ISO program tape formats

The following paper tape punch programs are provided.

	EIA	ISO
Leader part	Blank	NUL
Program start	CR or EOB	LR or NL, CR
Program part	-	-
Program end	ER	%
Tape end	CR or EOB, ER	LF or NL, CR, %



(1) Leader part

The information punch before CR (EIA) or LF (ISO), the first instruction on instruction tape, is called leader part. Whatever other combinations of code punch holes than LF and CR is selectable for the leader part since their parity check is not carried out.

(2) Program start

The CR (EIA) or LF (ISO) punching immediately after the leader part signifies the start of the programs.

(3) Program part

The punch part holes between the program start and end are named program part. They represent transfer, ON/OFF and other machining instruction informations.

Work number holes are punched at the head of the program part in accordance with address "0". Work numbers are registered but not displayed on programs when paper tape is read. The programs without work number punch holes can be read, provided their work number is registered in advance.



(4) Program end

ER (EIA) or % (ISO) punch holes at the end of programs signifies their end. Address "0" signifies tape end if the code is not given to the end of programs.

(5) Tape end

In tape I/O all-load, all-compare or all-punch mode, tape end signifies the completion of tape read or punching.



1-1-2 Call of pictures

When the program control picture is on the screen, call pictures by depressing the **TAPE I/O** menu key.

CMT I/O	RENUMBER	BUBBLE DIRECT.	PROGRAM ERASE	ALL ERASE		DNC I/O	TAPE I/O	PROGRAM
------------	----------	-------------------	------------------	--------------	--	------------	-------------	---------

↑

WORK NO.	SIZE	TAPE I/O						
1	1	14	PUNCH					
2	10	2	PUNCH					
3	9999	15	PUNCH					
4								
5								
6								
7								
8								
9								
10								
11								
12								
13								
14								
15								
16								
RAM								
USED 31 / 580								
*** PROGRAM FILE ***								
()								
LOAD	ALL LOAD	PUNCH	ALL PUNCH	COMPARE	ALL COMP			START
TAPE-NC	TAPE-NC	NC-TAPE	NC-TAPE	NC-TAPE	NC-TAPE			

Tape I/O picture

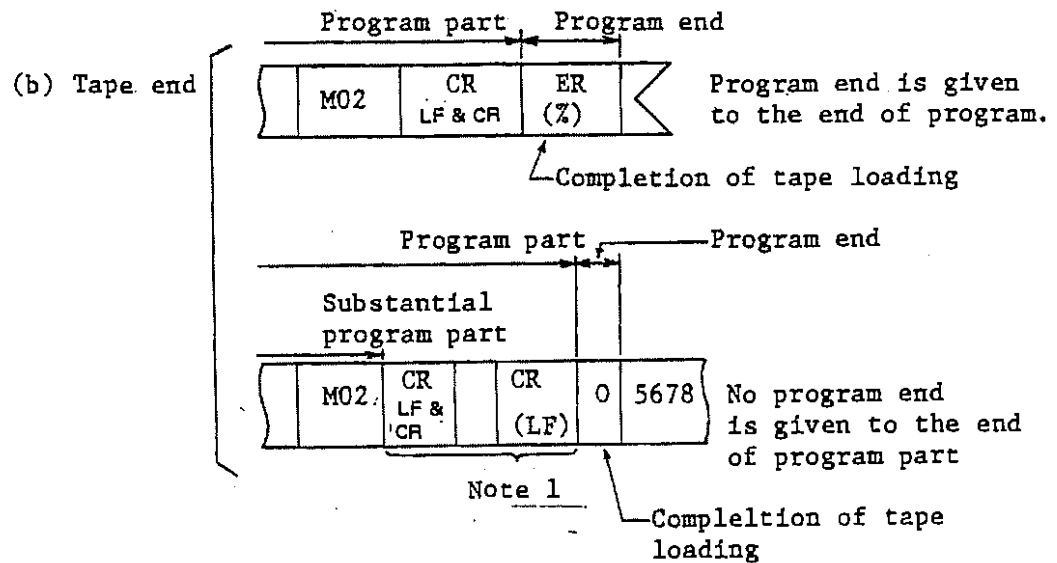
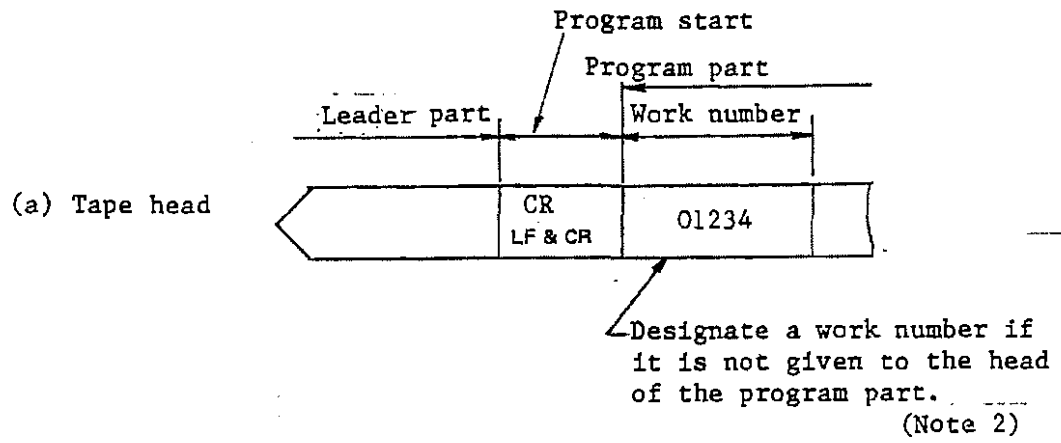


1-1-3 Tape I/O mode

(1) Load

Register only one paper tape punch hold program on the numerical controller. Even the programs without work number are registrable. When loading paper tape, judge whether one block consists of more than 64 characters including those of the comment part. If this is the case, carry out alarm setting.

(Tape formats)





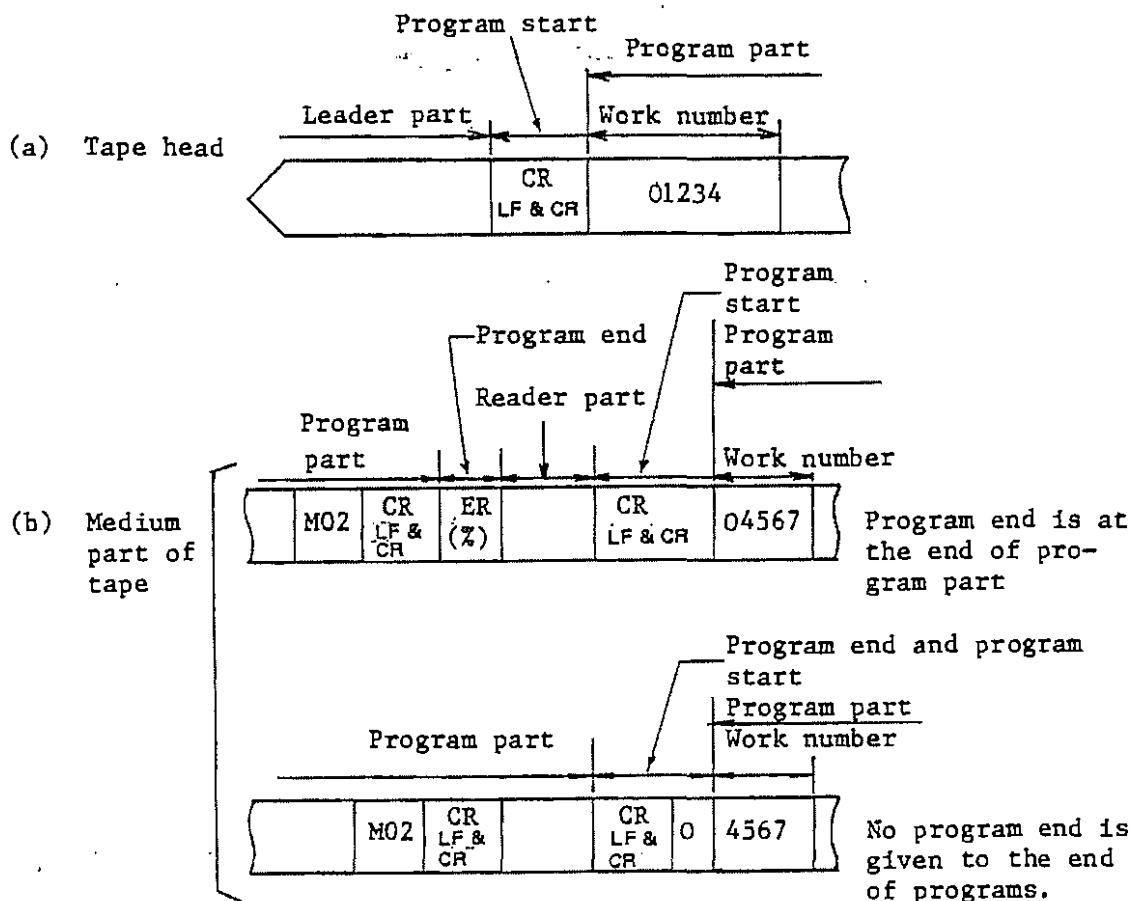
Note 1) If neither "ER" (EIA) nor "%" (ISO) is given to the end of the substantial program part, the address "0" at the next program signifies program end. The nonsignificant information such as a long space between program is therefore dealt with as program part. Therein, alarm position is taken up if one block consists of more than 64 characters. In this case shorten the space.

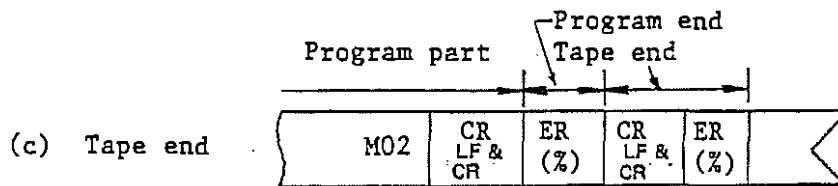
Note 2) Even if the work No. has been punched, the tape will be judged as a tape without work No., if the reading of it is started at the portion following the work No..

(ii) All load

Register all the paper tape punching programs up to tape end on the numerical controller. Alarm position is taken up if no work number is given to the head of each program part.

(Tape formats)

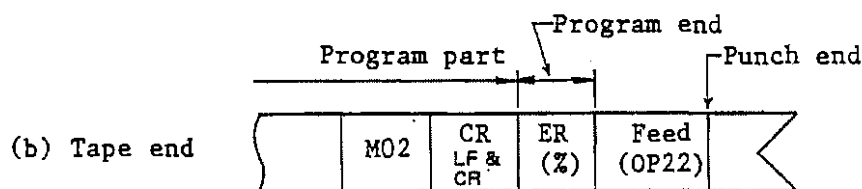
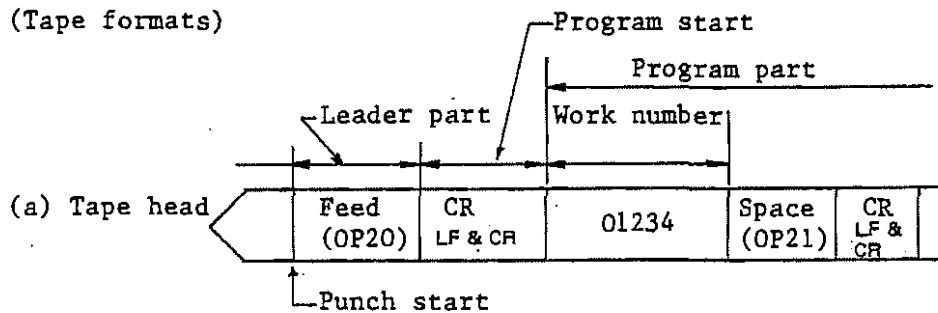




(iii) Punching

Paper tape is punched for only one EIA/ISO program with a signated work number

(Tape formats)

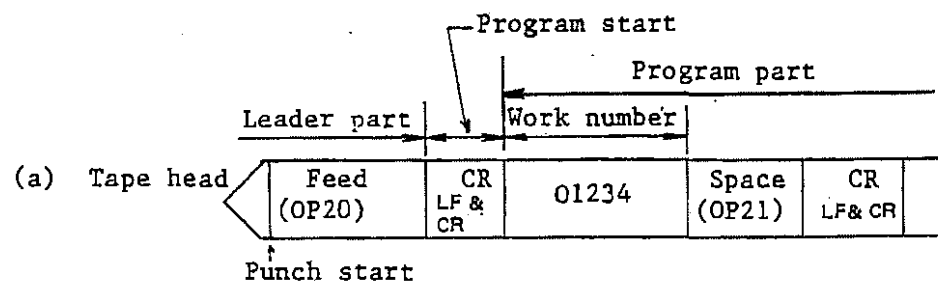


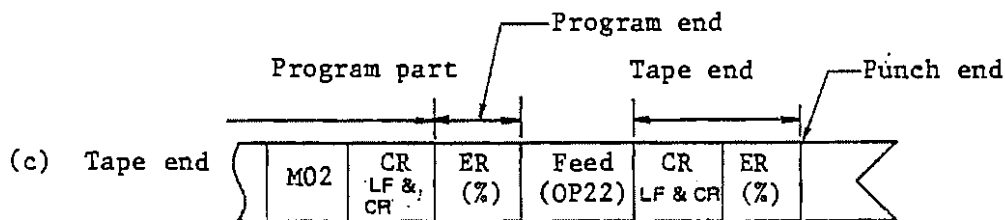
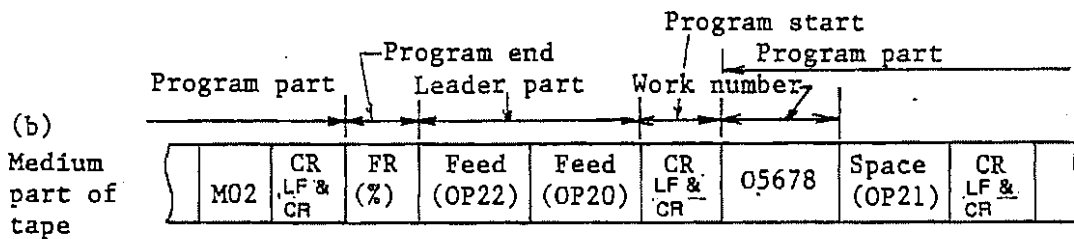
(OP20), (OP21) and (OP22) are the numbers of punch holes set using EIA/ISO parameters.

(iv) All punching

Paper tape is punched for all the EIA/ISO programs registered on the numerical controller.

(Tape formats)





(v) Compare

Only one paper tape punching program is identified with the ones with the same work number that are registered on the numerical controller.

(Tape formats)

The same as "Load"

(vi) All compare

All the paper tape punching programs up to tape end are identified with the ones with same work number that are registered on the numerical controller.

(Tape formats)

The same as "All load"



1-1-4 Handling

1) Load

- ① Set paper tape on the tape reader.
- ② Depress the **[LOAD]** menu key.
- ③ Set a work number if it is not given to paper tape.
- ④ Press the **[START]** menu key.

→ The information on paper tape will be read and registered on the numerical controller.

ii) All Load

- ① Set paper tape on the tape reader.
- ② Press the **[ALL LOAD]** menu key.
- ③ Depress the **[START]** menu key.

→ The information on paper tape will be read and registered on the numerical controller.

iii) Punching

- ① Set paper tape on the tape puncher.
- ② Press **[PUNCH]** menu key.
- ③ Set work numbers for punching programs.
- ④ Depress **[START]** menu key.

→ Information will be transferred to paper tape from the numerical controller.

iv) All punching

- ① Set paper tape on the tape puncher.
- ② Press the **[ALL PUNG]** menu key.
- ③ Press the **[START]** menu key.

→ Paper tape will be punched for all the EIA/ISO programs registered on the numerical controller.



v) Compare

- ① Set paper tape on the tape reader.
- ② Depress the COMPARE menu key.
- ③ Set a work number if it is not given to paper tape.
- ④ Press the START menu key.

→ The programs on paper tape will be compared with those registered on the numerical controller and the ones that do not agree with the latter will be displayed.

vi) All compare

- ① Set paper tape on the tape reader.
- ② Depress ALL COMP menu key.
- ③ Press the START menu key.

→ All the programs on paper tape will be compared with those registered on the numerical control and those that do not agree with the latter will be displayed.



1-2 EIA/ISO Programs

1-2-1 Call of pictures

Depress the **PROGRAM** menu key in picture selection mode for call.

POSITION	COMMAND	GRAPHIC	PROGRAM	TOOL FILE	TOOL DATA	TOOL LAYOUT	PAR	DIAGNOS.
----------	---------	---------	---------	--------------	--------------	----------------	-----	----------

↑

Program list range

N001 G91 G80 S800 T01 M03; N002 G92 X0 Y0 Z0; N003 M06 T00; N004 G00 X-50.0 Y-25.0; N005 G45 Z-48.0 M01; N006 G01 Z-4.0 F60; N007 Y-130.0 F85; N008 G02 X-20.0 Y-20.0 I-20.0 J0; N009 G01 X-150.0; N010 G03 X-30.0 Y30.0 I-30.0 J0; N011 G01 Y50.0; N012 X80.0 Y60.0; N013 G02 X30.0 Y10.0 I30.0 J-40.0; N014 G01 X90.0 T02; N015 G28 Z0;	N016 M06 T01; N017 G00 X-25.0 Y-25.0; N018 S1000 M03; N019 G00 G45 Z-48.0 M02; N020 G99 X0 Y0 Z-31.0 R0 Q3000 F100; N021 X-25.0 Y-50.0;							
*** WORK PROGRAM NO.9999 ***								
()								
SEARCH DATA	INSERT	ERASE	EXCHANGE					

← Message
← Menu

Program edit range

EIA/ISO program picture



1-2-2 Making and edit of EIA/ISO programs

(i) Making

Search for new work numbers and press the PROGRAM menu key.

WORK NO. SEARCH	UNIT NO. SEARCH	PROGRAM				DETAILED INFORM	WPC MSR	PROGRAM FILE
--------------------	--------------------	---------	--	--	--	--------------------	------------	-----------------

Next depress the displayed EIA/ISO PROGRAM menu key.
EIA/ISO program edit mode will then be obtained.

	EIA/ISO PROGRAM	MAZATROL PROGRAM						
--	--------------------	---------------------	--	--	--	--	--	--

(ii) Edit

EIA/ISO program pictures are automatically obtained when the work numbers of EIA/ISO programs are searched for.

WORK NO. SEARCH	UNIT NO. SEARCH	PROGRAM				DETAILED INFORM	WPC MSR	PROGRAM FILE
--------------------	--------------------	---------	--	--	--	--------------------	------------	-----------------

Depress the PROGRAM menu key. Edit mode will then be obtained.

1-2-3 Edit mode - handling

(Edit mode menu)

SEARCH DATA	INSERT	ERASE	EXCHANGE					
----------------	--------	-------	----------	--	--	--	--	--

(Address menu) (Change over among 4 modes at SHIFT)

G N / O	X I A U	Y J B V	Z K C W	F R P Q	M S L #	SP T D (EOB E H)	SHIFT
------------	------------	------------	------------	------------	------------	-------------	--------------	-------

(i) Data search (the designated character strings after the cursor at the program list range are searched for.)

- ① Transfer the cursor to search start position and press SEARCH DATA menu key.
- ② Press the ten-key and address menu key to enter the character strings to be searched for in the edit range.



③ Depress the INPUT key.

→ The cursor will transfer to the character string searched for.

(ii) Insertion (a desired character string is inserted at the position of the cursor at the list range.)

① Transfer the cursor to insertion place and depress the INSERT menu key.

② Press the ten-key and address menu key to enter in the edit range the character strings to be inserted.

③ Depress the INPUT Key.

→ The character strings at the edit range will be entered at the position of the cursor in the list range.

(iii) Erasure (the character strings at a desired range are erased from the position of the cursor at the list range.)

① Transfer a cursor to erasure start position and depress the ERASE menu key.

② Transfer a cursor to erasure completing position.

③ Press the INPUT key.

→ The character strings between the cursors will be erased.

(iv) Exchange (the character strings at a block where the cursor positions are exchanged with other characters.)

① Transfer the cursor to a block for exchange and depress the EXCHANGE menu key.

→ The block to be subjected to exchange is transferred to the edit range.

② Exchange the characters at the edit range using the ten-key or address menu key.

③ Depress the INPUT key.

→ The character strings at the list range will be exchanged with those at the edit range.



Note 1) No. "0" number is not displayed in EIA/ISO program pictures because it is registered as work number on the numerical controller. It is therefore not necessary to give any work number to the head of programs using the address "0".

Note 2) Machining is not possible unless all the T-code tools to be used in EIA/ISO programs are registered on tool data pictures. They are registrable in tool data pictures using the tool registration function.



1-3 Tool Offset Data

Tool offset data, the values for the tool diameter correction and tool position offset in EIA/ISO programs, are registered on tool offset data pictures.

They are quite ineffective in MAZATROL programs.

1-3-1 Call of pictures

Depress the **TOOL OFFSET** menu key in tool data picture mode for call.

DRUM NO.	TEACH	TOOL CHANGE	INCR.	TOOL ASSIGN	TOOL OFFSET	PREVIOUS PAGE	NEXT PAGE
----------	-------	-------------	-------	-------------	--------------------	---------------	-----------

↑

NO.	OFFSET	NO.	OFFSET	NO.	OFFSET	NO.	OFFSET	
1:	-12345.678	17:	-12345.678	33:	-12345.678	49:	-12345.678	POSITION
2:	0.	18:	0.	34:	0.	50:	0.	X-12345.678
3:	0.	19:	0.	35:	0.	51:	0.	Y-12345.678
4:	0.	20:	0.	36:	0.	52:	0.	Z-12345.678
5:	0.	21:	0.	37:	0.	53:	0.	A-12345.678
6:	0.	22:	0.	38:	0.	54:	0.	
7:	0.	23:	0.	39:	0.	55:	0.	
8:	0.	24:	0.	40:	0.	56:	0.	
9:	0.	25:	0.	41:	0.	57:	0.	
10:	0.	26:	0.	42:	0.	58:	0.	MACHINE
11:	0.	27:	0.	43:	0.	59:	0.	X-12345.678
12:	0.	28:	0.	44:	0.	60:	0.	Y-12345.678
13:	0.	29:	0.	45:	0.	61:	0.	Z-12345.678
14:	0.	30:	0.	46:	0.	62:	0.	A-12345.678
15:	0.	31:	0.	47:	0.	63:	0.	
16:	0.	32:	0.	48:	0.	64:	0.	

	TEACH		INCR.	POSITION SET	ALL ERASE	TOOL DATA		PAGE
--	-------	--	-------	--------------	-----------	-----------	--	------

Tool offset data picture



1-3-2 Registration of offset distance

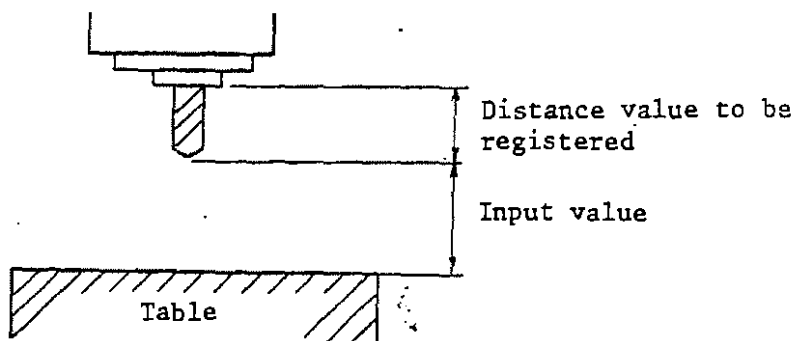
- (i) Enter data with the cursor transferred to the offset number to be registered.

One hundred twenty eight offset values are registrable on the maximum.

(1st page: offset Nos. 1 - 64
2nd page: offset Nos. 65 - 128

- (ii) Input of tool length as per TEACH memory

Enter the distance between the table and cutter tip with the menu inverted by depressing the TEACH menu key. Tool length will then be registered.



- (iii) Incremental input

Enter data after inverting the INCR. menu by pressing its key. An input value will then be added to an already-registered value.

- (iv) Current value setting

Depress the POSITION SET menu key when the same value as the current value in position/command picture. The cursor will then transfer so that the current value is changeable. Press the menu key again or lower the cursor. The cursor will then return to the original position.

- (v) Depress the TOOL DATA menu key. Tool data picture will then be reset.



1-4 Work Offset Parameters

Work offset data, the values for the setting of work coordinate systems using G54 to G59 in EIA/ISO programs, are registered on work offset parameter picture.

They are quite ineffective in MAZATROL programs.

1-4-1 Call of pictures

Use the **PREVIOUS PAGE** or **NEXT PAGE** menu key in the following of parameter cutting condition pictures.

Work offset parameter picture												
G54				G56				G58				SHIFT
X-12345.678				X-12345.678				X-12345.678				X -1.234
Y-12345.678				Y-12345.678				Y-12345.678				Y -1.234
Z-12345.678				Z-12345.678				Z-12345.678				Z -1.234
A-12345.678				A-12345.678				A-12345.678				A -1.234
G55				G57				G59				MACHINE
X-12345.678				X-12345.678				X-12345.678				X-12345.678
Y-12345.678				Y-12345.678				Y-12345.678				Y-12345.678
Z-12345.678				Z-12345.678				Z-12345.678				Z-12345.678
A-12345.678				A-12345.678				A-12345.678				A-12345.678
()												
TEACH	INCR.									PREVIOUS PAGE	NEXT PAGE	

Work offset parameter picture



1-4-2 Registration of offset values

- (i) Enter data with the cursor transferred to an offset to be registered.

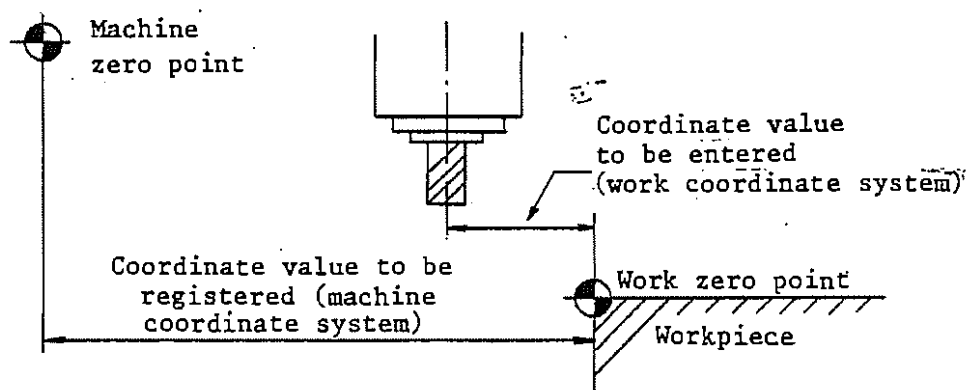
Six G54 - 59 work coordinates are registrable.

The six registered work coordinates can be shifted using the coordinate shift.

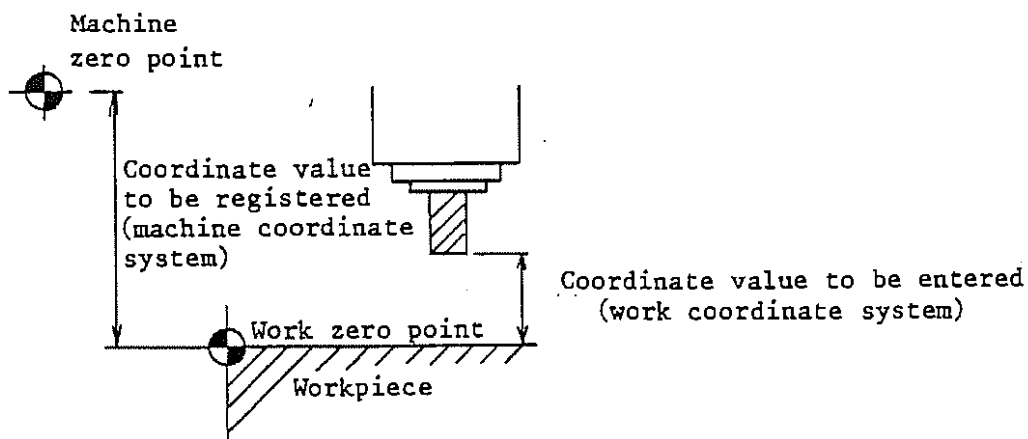
- (ii) Coordinate system input as per **TEACH**

Enter the coordinate value of the cutter tip in the work coordinate system to be established, with the **TEACH** menu inverted by pressing its key. The coordinate value of the 0-point viewed from that of the machine of work will then be registered.

(X- and Y-axis)



(Z-axis)





(iii) Incremental input

Enter data after inverting INCR. menu by pressing its key.
The input value will then be added to a value already
registered.

(iv) Depress the PREVIOUS PAGE or NEXT PAGE menu key.

Another parameter picture will then be obtained.



1-5 Operation

1-5-1 Restart

EIA/ISO programs are the same as MAZATROL ones in that restart is possible during programs. Take the following steps therefor after checking that the work numbers selected correspond to EIA/ISO programs.

1. Mode: automatic operation
2. Depress the menu key.
3. The question message "SEQUENCE NO. TO RESTART?" appears. Then, enter a desired sequence number (note: no picture is displayed).

Example:

4. The question message "BLOCK NO. TO RESTART?" appears. Then, enter a desired block number.

Example:

Note i) Give block No. 1 to the blocks including a corresponding sequence number. Addition will then be made until another sequence number appears.

N001	___;	Sequence No.1	Block No. 1
___;	"	1	" 2
___;	"	1	" 3
N002	___;	" 2	" 1
___;	"	2	" 2

Note ii) The blocks for which no input is made or only setting is entered are interpreted as block No. 1.

5. A message asking "SUB PROGRAM REPEAT NUMBER" will appear. Specify at which sequence No. and block No. (set earlier) re-start should occur.



Note i) This function is used when re-start is required half way through a subprogram which is frequently called by the main program.

Note ii) When no data on the number of subprogram calls are entered or only the INPUT key is depressed, the first sequence will be selected automatically because it is interpreted that re-start should occur during the first sequence.

6. Press the CYCLE START key. Programs will then be run starting with their set sequence or block number.

Note 1) On restart, the S- and M-codes in the programs before set block are ineffective. Prior to run, therefore, set a necessary value in MDI mode.

Note 2) Commanding to restart at a block including G02 or G03 code will result in alarm.

1-5-2 Optional block skip

Enter a slash and numeral /n (n = 1 ~ 9) in the order of mention at the head of a desired block in EIA/ISO programs, invert the automatic operation menu by pressing its BLOCK SKIP n (n = 1 ~ 9) key. The information at a block with a number corresponding to skip number (n) to which /n instruction is given is neglected.

WORK NO. SEARCH	WORK LIGHT	MACHINE LOCK	SINGLE BLOCK	OPTIONAL STOP	DRYRUN	BLOCK SKIP	SCREEN OFF	RESTART
--------------------	---------------	-----------------	-----------------	------------------	--------	---------------	---------------	---------

(Automatic operation menu)



Depress the **BLOCK SKIP** menu key. The following menu will then be obtained.

BLOCK SKIP1	BLOCK SKIP2	BLOCK SKIP3	BLOCK SKIP4	BLOCK SKIP5	BLOCK SKIP6	BLOCK SKIP7	BLOCK SKIP8	BLOCK SKIP9
----------------	----------------	----------------	----------------	----------------	----------------	----------------	----------------	----------------

Depress a menu key. A menu will then invert. Press an inverted menu key. Inversion erasure will then occur.

Depress the **MENU SELECT** key. Resetting will then occur to the automatic operation menu.

All other operation and handling steps than above require the same handling as the operation as per MAZATROL programs.



2. PROGRAMS

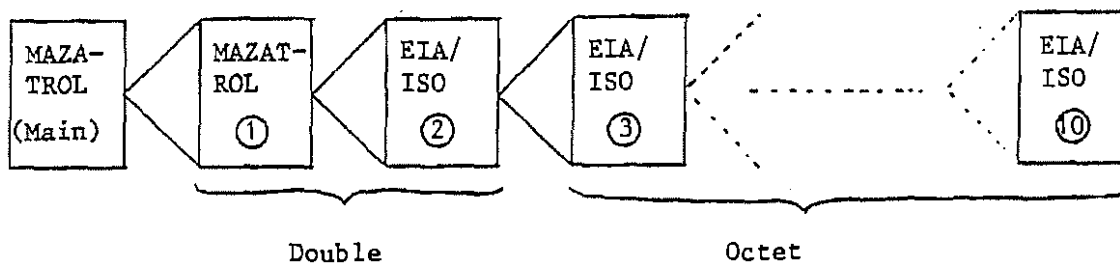


2-1 Cautions as to MAZATROL CAM M-2

2-1-1. Sub-programs

- (1) EIA/ISO programs can be called using the sub-program unit in MAZATROL programs. Alarm position is however taken up if MAZATROL programs are called from EIA/ISO programs.
- (2) Sub-program nesting in MAZATROL programs: up to double
- (3) Sub-program nesting in EIA/ISO programs: up to octet nesting is possible.

The maximum tenfold nesting is possible.



2-1-2. Basic coordinate systems

- (1) Note the following when calling the EIA/ISO ones among MAZATROL programs using the sub-program unit.
 - (i) The basic coordinate systems set in MAZATROL programs are effective except for 0.
 - (ii) The auxiliary coordinate systems set in MAZATROL programs are absolutely neglected.
 - (iii) When return is performed from EIA/ISO programs (sub-programs) to MAZATROL ones, the basic coordinate systems set in the former are cancelled and those previously set in the latter become effective (0 takes 0).



2-1-3. Difference between manual program mode unit and EIA/ISO programs

(1) Helical cutting

	Specification of pitch	Arc	Diameter correction	Input format
Manual program mode	As per address P (Pitch differs: alarm)	Larger arcs than circumferential are permitted also.	Possible for only the arcs whose quadrant does not change.	(G17) G02 X_Y_Z_ { I_J_ } P_ G03 X_Y_Z_ { R_ } P_
EIA/ISO	Automatic	Only smaller arcs than circumferential are permitted	Possible	(G17) G02 X_Y_Z_ { I_J_ } G03 X_Y_Z_ { R_ } G02 X_Y_ { Z_ } { I_J_ } G03 X_Y_ { A_ } { R_ }

(2) Initial setting for synchronous feed (G95) and non-synchronous feed (G94)

Manual program mode synchronous feed (G95)

EIA/ISO nonsynchronous feed (G94)

(3) Multi-piece machining

Manual program multi-piece machining is possible. Only mode the first piece is fed if G91

instruction is given to SNO 1.

EIA/ISO multi-piece machining is not possible.

2-1-4. Tool path

(1) Tool paths are drawn in EIA/ISO program on the supposition that all the X-, Y- and Z-axes locate at the first origin regardless of the current position of the machine.

(2) The path for ATC is drawn so that the X- and Y-axes move toward the first origin.

(3) The exchange of system variables in G10 (offset program input) or user macro is not performed on tool transfer.



2-1-5. Correction of tool length and radius

- (1) In MAZATROL, MAZATROL brings tool tip to a coordinate position automatically specified through referring to the input tool length of tool data. The input of tool length therefore ensures the integration of programs without using G43 and G44.
- (2) It is necessary to set the tool length to "0" for the run of the programs on paper tape (the programs employing G43 and G44).

If tool length is set to "0" when the programs are used as MAZATROL sub-programs, no machining is possible therein. Therefore carry out edit for the correction of tool length.

- (3) In correcting tool radius (G41, 42), such parameter setting is possible that reference is made to that of tool data instead of using D-code for setting correction value. In this case, correction value is obtained by adding offset value to the radius of tool data, provided D-code instruction is given.



2-1-6. Mirror image

- (1) Note the following when calling EIA/ISO programs in MAZATROL programs using the sub-program unit.
 - (i) Cancell (M90) the mirror image in MAZATROL programs at block preceding sub-program unit and give instruction at the head of EIA/ISO programs.
 - (ii) The mirror image specified using EIA/ISO programs is cancelled on the return to MAZATROL ones.
- (2) The mirror image of MAZATROL CAM M-2 is symmetrical with respect to the origin of work coordinate system, which should be noticed particularly in incremental instruction (G91) mode.
- (3) Don't give any of G28, G30, G92, G54 - G59 instructions in principle when the mirror image is applied.
- (4) The mirror image is not applied to the directions of canned cycle drilling and of the one-directional positioning in accordance with G60.
- (5) M codes (M90 - M93) for mirror image are effective in the blocks following the blocks which includes the mirror image instruction.

2-1-7. ATC

Since some machines with MAZATROL CAM M2 carry out ATC operation accompanying axis movement or ATC operation which does not end at the first reference point of Z axis, care should be taken especially when programming with incremental instruction and when restarting.



2-1-8. Others

- (1) MAZATROL performs the read several blocks ahead of the block being run. Note that the data exchange for a memory such as G10 (offset program input) may be underway several blocks ahead on the resetting with the resetting button.
- (2) MAZATROL may carry out internal code conversion and make blocks to ensure the passage of EIA/ISO program tape.

e.g.) ATC (automatic tool change)

	Tape	Numerical control instruction
Next time tool selection	T <input type="text"/> <input type="text"/> <input type="text"/>	M (600 + <input type="text"/> <input type="text"/> <input type="text"/>)
ATC	M06	M06: stop for the orientation of spindle and preparation for ATC by shifting tool toward spindle side + T <input type="text"/> <input type="text"/> <input type="text"/> : ATC operation

```
N001 G00 X-100. Y-100. Z-100. T1;  
N002 G91 G28 XO YO ZO M06;  
:
```

Output from numerical controller on above programming

- 1. M601 (Specification of tool No. 1 for next time) N001
- 2. G00 X-100. Y-100. (Positioning) Z-100.
- 3. G00 X-100. Y-100. (Transfer "0" positioning) Z-100.
- 4. G00 XO YO ZO M06 (Return to zero point and ATC preparation) N002
- 5. T1 (Automatic fitting of No. 1 tool on spindle)



2-2 Feed function (F-function)

	Feed per minute (G94)	Synchronous feed (G95)
mm	1 mm/min - 9999 mm/min (F1 - F9999)	0.01 mm/rev - 500.00 mm/rev (F1 - F50000)
inch	0.01 inch/min - 600.00 inch/min (F1 - F60000)	0.0001 inch/rev 50.0000 inch/rev (F1 - F500000)

Note: Axis of rotation

° Nonsynchronous feed

Metric system: Rotates in feedrate of $1^\circ/\text{min.}$ on feed instruction of F1 (1 mm/min.).

Inch system: Rotates in feedrate of $0.1^\circ/\text{min.}$ on feed instruction of F1 (0.01 mm/rev.).

° Synchronous feed

Metric system: Rotates in feedrate of $0.01^\circ/\text{rev.}$ on feed instruction of F1 ($0.001^\circ/\text{rev.}$ on feed instruction of F1 (0.0001 inch/rev.).



2-3 Preparatory Function (G-function)

G-codes

G-code	Group	Function
G00	A	Positioning (Rapid Traverse)
G01		Linear Interpolation (Straight Cut)
G02		Arc Interpolation CW
G03		Arc Interpolation CCW
G04	00	Dwell
G07	00	Cancellation of Instruction Axis (SIN Interpolation)
G09	00	Exact Stop Check
G10	00	Offset Program Input
G17	B	XY Plane Selection
G18		ZX Plane Selection
G19		YZ Plane Selection
G22	00	Call of Sub-programs
G23		Sub-program Return
G27	00	Return to 1st Zero Point, Check
G28		Return to 1st Zero Point
G29		Automatic Return from 1st Zero Point
G30		Return to 2nd Zero Point
G31		Skip Function
G40	C	Tool Nose Radius Correction, Cancel
G41		Tool Nose Radius Correction, Left
G42		Tool Nose Radius Correction, Right
G43	D	Tool Length Correction, Plus (+)
G44		Tool Length Correction, Minus (-)
G45	00	Tool Position Offset, Increase
G46		Tool Position Offset, Decrease
G47		Tool Position Offset, Double Increase
G48		Tool Position Offset, Double Decrease
G49	D	Tool Length Correction, Cancel
G54	E	Selection of Work Coordinate System 1
G55		Selection of Work Coordinate System 2
G56		Selection of Work Coordinate System 3
G57		Selection of Work Coordinate System 4
G58		Selection of Work Coordinate System 5



G-code	Group	Function
G59	E	Selection of Work Coordinate System 6
G60	00	One-directional Positioning
G61	F	Exact Stop Check Mode
G64	F	Continuous Cutting Mode
G65	00	User Macro Unmodal Call
G66	G	User Macro Modal Call
G67		Cancellation of User Macro Call
G71	H	True-circle Cycle (CW)
G72		True-circle Cycle (CCW)
G73		High-speed Peck Drilling Canned Cycle
G74		Reverse Tapping Canned Cycle
G75		Fine Boring Canned Cycle
G76		Fine Boring Canned Cycle
G77		Back Facing Canned Cycle
G80	H	Canned Cycle Cancel
G81		Drilling Canned Cycle, Spot Drilling
G82		Drilling Canned Cycle, Counterboring
G83		Peck Drilling Canned Cycle
G84		Tapping Canned Cycle
G85		Boring Canned Cycle
G86		Boring Canned Cycle
G87		Back Boring Canned Cycle
G89	Boring Canned Cycle	
G90	I	Absolute Instruction
G91		Incremental Instruction
G92	00	Coordinate System Designation
G94	J	Nonsynchronous feed
G95		Synchronous feed
G98	K	Return to Initial Point in Canned Cycle
G99		Return to R Point in Canned Cycle



- Note 1. ▀ mark indicates that the marked G code will be executed when an EIA/ISO program is started or called as a subprogram by a MAZATROL program.
- Note 2. G codes in 00 group are non-modal instructions which are effective only in the block to which it belongs.
- Note 3. Any number of G codes belonging to different groups can be set in a block. If a block includes two or more G codes belonging to a same group, only the last of them is effective.
- Note 4. When a G code of A group or anyone of G27, G28, G29, G30 and G31 is effected during canned cycle, G80 is executed with the canned cycle cancelled automatically.



2-3-1. Selection of planes (G17, G18, G19)

Select planes using G-code for arc interpolation and tool nose-radius correction.

G17	Plane X-Y
G18	Plane X-Z
G19	Plane Y-Z

Transfer instruction is not concerned in the selection of planes.

2-3-2. Positioning (G00)

Positioning is made at a specified point through rapid traverse.

Instruction procedure

G00 X ___ Y ___ Z ___ A ___ ;

2-3-3. One-directional positioning (G60)

Finally one-directional precise positioning is possible except for lost motion.



Starting point

End point

(In case final positioning transfer
is made right to left.)

Use G60 instead of G00 as follows for instruction.

G60 X ___ Y ___ Z ___ A ___ ;

Use parameters to set overrun distance and positioning direction. Even if the positioning transfer direction specified agrees with that set using a parameter, the tool returns over a distance set using a parameter after the positioning at the end point.

Note 1) G60 - one-shot G-code

Note 2) In fixed cycle, one-directional positioning is performed only for the first X-Y plane.



Note 3) One-directional positioning is performed also when transfer distance "0" instruction is given.

Note 4) The mirror image is not applied to the direction set using a parameter.

Note 5) Alarm position is taken up if G60 instruction is given in radius correction mode (G41, 42).

2-3-4. Linear interpolation (G01)

Linear interpolation is possible using instruction:

G01 X ___ Y ___ Z ___ A ___ F ___;

The feed speed at this time is specified by F. It remains effective until a new instruction is given. Therefore no instruction is necessary each time.

2-3-5. Arc interpolation (G02, G03)

The tool in use is movable along an arc using the following instructions.

i) Arc at XY plane

G17 $\left\{ \begin{array}{l} G02 \\ G03 \end{array} \right\} X \text{ ___ } Y \text{ ___ } \left\{ \begin{array}{l} R \text{ ___} \\ I \text{ ___ } J \text{ ___} \end{array} \right\} \left. \begin{array}{l} F \text{ ___} \\ \text{YDU} \end{array} \right\};$

ii) Arc at XZ plane

G18 $\left\{ \begin{array}{l} G02 \\ G03 \end{array} \right\} X \text{ ___ } Z \text{ ___ } \left\{ \begin{array}{l} R \text{ ___} \\ I \text{ ___ } K \text{ ___} \end{array} \right\} F \text{ ___};$

iii) Arc at YZ plane

G19 $\left\{ \begin{array}{l} G02 \\ G03 \end{array} \right\} Y \text{ ___ } Z \text{ ___ } \left\{ \begin{array}{l} R \text{ ___} \\ J \text{ ___ } K \text{ ___} \end{array} \right\} F \text{ ___};$

Note 1) . IOJOKO is omissible.

Note 2) Priority is given to I.J.K. when it is issued at the same time as R or one-turn arc instruction is issued and to R in other cases.



2-3-6. Helical cutting (G02, G03)

Helical cutting is possible by using another straight axis instruction simultaneously with the arc interpolation in accordance with arc instruction. Therein the tool is spirally movable.

$$\begin{array}{l} \text{G17} \left\{ \begin{array}{l} \text{G02} \\ \text{G03} \end{array} \right\} \begin{array}{l} \text{X} \text{---} \text{Y} \\ \text{---} \end{array} \left\{ \begin{array}{l} \text{R} \text{---} \\ \text{I} \text{---} \text{J} \end{array} \right\} \left\{ \begin{array}{l} \text{Z} \text{---} \\ \text{A} \text{---} \end{array} \right\} \text{F} \text{---}; \\ \text{G18} \left\{ \begin{array}{l} \text{G02} \\ \text{G03} \end{array} \right\} \begin{array}{l} \text{X} \text{---} \text{Y} \\ \text{---} \end{array} \left\{ \begin{array}{l} \text{R} \text{---} \\ \text{I} \text{---} \text{K} \end{array} \right\} \left\{ \begin{array}{l} \text{Y} \text{---} \\ \text{A} \text{---} \end{array} \right\} \text{F} \text{---}; \\ \text{G19} \left\{ \begin{array}{l} \text{G02} \\ \text{G03} \end{array} \right\} \begin{array}{l} \text{Y} \text{---} \text{Z} \\ \text{---} \end{array} \left\{ \begin{array}{l} \text{R} \text{---} \\ \text{J} \text{---} \text{K} \end{array} \right\} \left\{ \begin{array}{l} \text{X} \text{---} \\ \text{A} \text{---} \end{array} \right\} \text{F} \text{---}; \end{array}$$

Instruction procedure: fundamentally, another 1-axis transfer instruction not included in arc planes is simply added to the above instructions for arc interpolation.

Note 1) This neglect G07 instruction in tool path.

Note 2) On restart, computation is carried out on the assumption that the tool has completed transfer in accordance with the actual instruction even if G07 instruction is given.

2-3-7. Cancellation of instruction axis (G07)

An axis (virtual axis) for the interpolation without transferring 1-axis can be given using the following instructions.

```
G07 0; ( : virtual axis)
G07 1; ( : actual axis)
( = X,Y,Z,A)
```

SIN interpolation is possible using the 1-axis of an arc in helical cutting instruction as virtual axis using the above instructions.



Note:1 G07 instruction is neglected midst a tool path.

Note:2 When restarting, calculation is carried out assuming that movement has been done for an axis instruction during sequence No. searching, even if G07 instruction has been effected.

2-3-8. Return to zero point (G27-G30)

8.1 G27___ Return to zero point, check

G27 X___ Y___ Z___ A___;

This function checks whether programs return accurately to the position corresponding to the machine zero point.

The tool in use is positioned at a point specified in rapid traverse using instruction.

If it positions at the zero point of the machine, a lamp corresponding thereto turns on.

Thereafter the run of the next block continues. To stop the tool there, therefore, put auxiliary functions such as M00 and M01 therein or use single block mode.

Note 1: During correction, the point to be reached in accordance with G27 is specified with offset distance taken into account.

8.2 G28___ Return to 1st zero point, check

G28 X___ Y___ Z___ A___;

The tool is positioned at the intermediate points (specified points in one block) of all designated axes through rapid traverse first and at the zero point of all the axes through the rapid traverse from the intermediate points.



G28 instruction is generally used for ATC. Thereon cancel tool nose radius correction, tool position offset and tool length correction in principle.

Note 1) The tool in use does not move at all in any axis not designated at G28 block.

Note 2) Don't run G28 while the machine is locked. Otherwise the current position counter will shift.

8.3 G29 ___ Automatic return from 1st zero point

The tool is positioned at the intermediate points designated using G28 as stated above through rapid feed and further at specified points through the feed from therefrom using instruction:

G29 X ___ Y ___ Z ___ A ___ ;

Note 1) The intermediate points do not transfer even if the work coordinate system is changed after the return to zero point in accordance with G28. When G29 instruction is given, the tool is positioned at a designated point in a new coordinate system after passing the intermediate points.

Note 2) G30 intermediate point is neglected.

8.4 G30 ___ Return to 2nd zero point

A designated axis can be returned to the 2nd zero point using instruction:

G30 X ___ Y ___ Z ___ A ___ ;



2-3-9. Dwell (G04)

Dwell is performed using instruction:

G04 X (t) or G04 P (t)

Namely the run of a block starts t(m.sec.)(max. 9,999,999 sec.) after that of the block preceding it.

(e.g.) 2.5 sec. dwell

G04 X2.5; (G04 X2500;)

G04 P2.5; (G04 P2500;)

2-3-10. Exact stop check (G09)

At the end of a block including G09, in-position is checked with feed speed lowered to "0" before proceeding to the next block. This function is used to develop sharp edge at a corner. G09 is effective at only designated blocks.

Note 1) Positioning mode (G00, G60), in-position is automatically checked even if G09 instruction is not given.

2-3-11. Exact stop check mode (G61) and cutting mode (G64)

(1) Exact stop check mode (G61)

In-position is checked using G61 instruction with feed speed lowered to "0" at the end of blocks using an instruction for the transfer after G61 instruction before G64 instruction before proceeding to the next block.

(2) Cutting mode (G64)

Transfer starts to the next block using an instruction for the transfer after G64 instruction before G61 instruction immediately so that deceleration is minimized at the end of blocks.



Even in G64 mode, however, deceleration is performed until feed speed takes to "0" and in-position is checked at the blocks to which positioning mode or exact stop check (G09) instruction is given.

2-3-12. Coordinate system setting (G92)

Before transferring the tool using absolute instruction, specify a coordinate system using instruction:

```
G92 X x Y y Z z A a ;
```

Accordingly such a coordinate system is established that the tip of the tool installed on the spindle locate at (x, y, z, a). It is called work coordinate system.

Note 1) If G92 is used in offset, such coordinate system is established that a position designated in accordance with G92 is taken up before hand.

Note 2) Alarm position is taken up if G92 instruction is given in tool nose radius correction mode.

2-3-13. Work coordinate system (G54 - G59)

G92 is not used to establish coordinate systems. Any of the six coordinate systems specific to the machine that have been established beforehand is selectable using G54 - G59.

G54	----	Work coordinate system	1
G55	----	"	2
G56	----	"	3
G57	----	"	4
G58	----	"	5
G59	----	"	6



2-3-14. Change of work coordinate systems as per program instruction

Six work coordinate systems G54 - G59 are prepared. If they are not enough or they are transferred for each program, their position is changeable using program instruction:

```
G10 L2 P p X x Y y Z z A a ;
```

p = 1 - 6: designated correspondingly to work coordinate systems 1 - 6

Note 1) Coordinate-system shift distance is changeable if p = 0.

Note 2) On the check of tool path, the exchange of the data of G54 - G59 is hazardous. G10 is therefore neglected.

2-3-15. Skip function

The same as G01 in that the issuance of linear interpolation instruction is possible through that of axial transfer instruction after G31. If skip signals are entered from outside on its way, it is interrupted for the run of the next block. One-shot G31 is effective for the block being run.

Note 1) G31 is effective only when a touch-sensor is provided on the spindle tool.

Operation stops at the block when another tool is stuck.

Note 2) Alarm position is taken up if G31 instruction is given with tool nose radius correction kept underway.

Note 3) The speed of the feed in accordance with G31 is designated using a parameter.

The speed of dry run is also designated using a parameter.



2-4 Correction Function

2-4-1. Correction of tool length (G43, G44, G49)

The position of the end point of the Z-axis can be shifted in + or - direction over a distance set in the offset memory using instruction:

$$\left. \begin{matrix} G43 \\ G44 \end{matrix} \right\} Z \text{ ______ } H \text{ ______ }; \text{ or } \left. \begin{matrix} G43 \\ G44 \end{matrix} \right\} H \text{ ______ };$$

(1) Offset direction

G43 + direction offset

G44 - direction offset

(2) Specification of offset distance

H-code is used to designate offset numbers.

Offset numbers H00 - H128 can be designated (128 numbers including D-code used for tool nose radius correction).

The offset distance corresponding to offset number 00 (H00) is always 0.

(3) Cancellation of tool length offset

When cancelling offset, use G49 instruction or designate H00.

Cancelling operation starts immediately thereon.

Note 1) Alarm position is taken up when offset distance changes at the same block as arc instruction (G02, G03).

Note 2) The change of offset distance through that of offset number does not mean the addition of new offset distance value to old one.

Note 3) D-code is not usable for tool length offset.



(4) Offset axis

Tool length correction is performed for the Z-axis as well as another axis through parameter setting. Offset axes can be designated using a designated axial address α in the same block as G43 and G44.

Tool length correction is possible for the α -axis using instruction:

G43 α H ; (α : a desired axis)

G44

Note 4) Tool length correction is possible for only one desired axis each time. Alarm position is taken up when the following instruction is used. Cancel the tool length correction underway for the change-over of axes.

G43 Z H ; Alarm position is taken up
G43 X H ; at this block.

Note 5) Cancel tool length correction in principle before using G28 or G30 return to zero point instruction.

Note 6) The tool length correction in accordance with G43 and G44 can be replaced with the registration of tool length using the tool data picture.

Note 7) Offset distance increases 10 times when tool length correction is carried out for the A-axis or the A-axis in an inch system is rotary.



2-4-2. Tool position offset (G45 - G48)

The tool transfer distance in as designated axis can be increased, decreased, increased two times or decreased two times by a value set in the offset distance memory using G45 - G48 instructions.

G-code	Functions
G45	Increase by the value set in offset memory
G46	Decrease by the value set in offset memory
G47	2 times increase by the value set in offset memory
G48	2 times decrease by the value set in offset memory

Both H- and D-codes are usable. Programs become easy to understand if D- or H-code is used to select the address for tool radius or length correction respectively.

Which of H- and D-codes is used for tool position offset is decided using a parameter.

Set transfer distance to "0" when changing only offset distance in incremental instruction (G91) mode. In absolute (G90) mode, no operation is made even if "0" is designated.

Offset distance +123.45 (offset number 01)

NC instruction	G91G45XOD01	G91G45X-OD01
Equivalent instruction	G91X123.45	G91X-123.45

- Note 1) The arc interpolation (G02, G03) correction for only 1/4 and 3/4 circle instructions is possible using G45 - G48 instructions.
- Note 2) If H-code is used in G43 or G44 mode, the Z-axis transfers over offset distance. Therefore use D-code when using G45 - G48 instructions in G43 and G44 modes.
- Note 3) In canned cycle, G45 - G48 are neglected.
- Note 4) Alarm position is taken up if G45 - G48 are used in G41 or G42 mode.



2-4-3. Tool nose radius correction (G40, G41, G42)

The path of a tool with radius R for cutting the work with a certain shape must be set apart by R therefrom. The function for preparing it is called tool radius correction function.

(1) Offset distance

Designate offset distance using D-code.

Offset numbers D00 -D128 can be designated (128 numbers including H-code used for tool length correction).

Offset distance 0 always corresponds to offset number 00 (D00).

(2) Selection of planes (G17, G18, G19)

Offset computation is performed in a plane selected using plane selecting G-code (G17, G18, G19).

Alarm position is taken up if the planes selected in G41 and G42 modes is subjected to change-over.

(3) Tool nose radius correction

G-code	Function
G40	Tool nose radius correction - cancel
G41	Offset on the left hand with respect to tool advance direction
G42	Offset on the right hand with respect to tool advance direction

Note 1) Alarm position is taken up to the change-over from G41 to G42 or from G42 to G41.

Alarm position is taken up when coordinate system setting instructions (G92, G54 - G59) are used in radius correction mode.



(4) Details of tool radius correction

① Cancellation mode

Cancellation mode position is taken up immediately after power supply or resetting and after the completion of the run of EIA/ISO programs. In cancellation mode, vector size is always "0" and agrees with the central path of the tool in use.

② Start-up

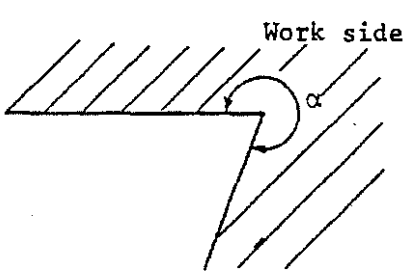
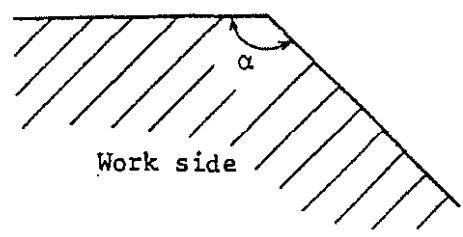
The operation in offset mode after the blocks satisfying all the following requirements are run in cancellation mode is called start-up.

- (a) G41 or G42 instruction is given or G41 or G42 mode position is taken up using them.
- (b) Tool radius correction offset distance is not "0".
- (c) Instruction is given and transfer distance is not "0" even if only one axis (except I, J and K) exists in offset planes. Type A and type B of start-up are available which are to be designated with parameter OP2.

Alarm position is taken up if arc instruction (G02, G03) is used for start-up.

Note 2) Definition of "inside" and "outside"

More than 180° or $1 - 180^\circ$ intersection angle measured on work side of the 2-block transfer instructions in programs is called "inside" or "outside" respectively.

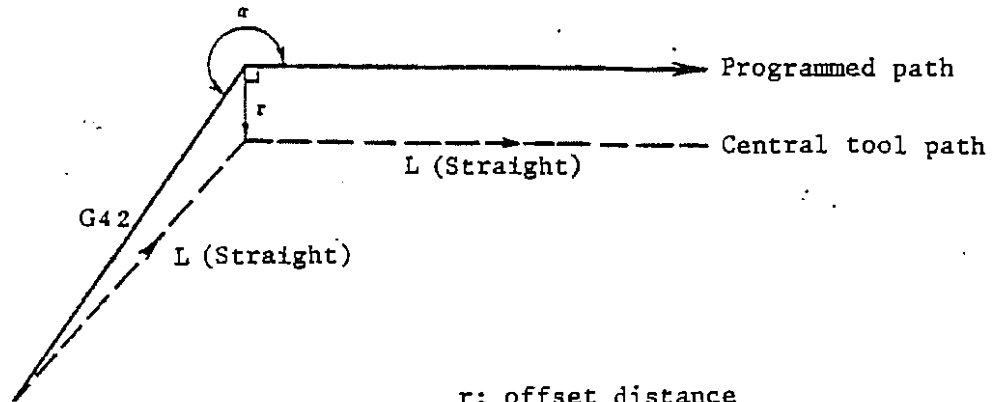
Inside	Outside
$180^\circ \leq \alpha$	$0^\circ \leq \alpha < 180^\circ$
	



(Type A)

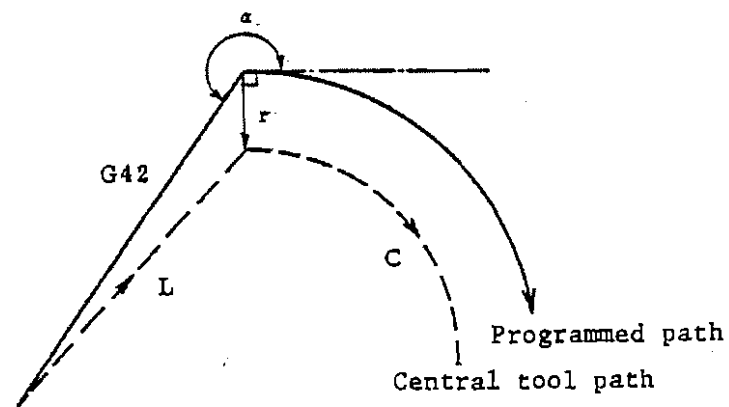
(i) Running inside ($180^\circ \leq \alpha$)

Straight → Straight



Characters L and C in the drawings inserted hereunder indicate straight and arc running respectively.

Straight → Arc

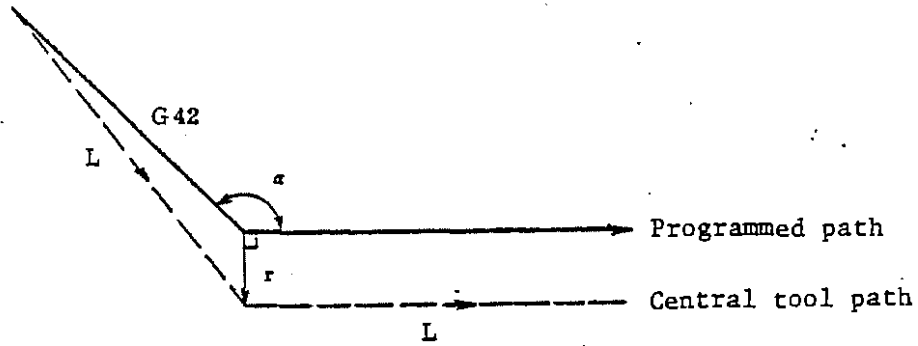


Note 3) "Intersection" is obtained through the offset of two blocks over (r) from a programmed path.

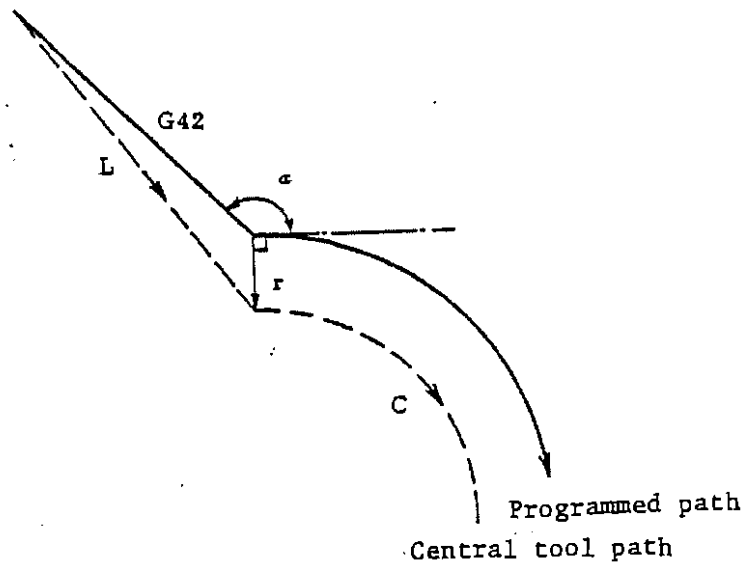


(ii) Outside obtuse-angle running ($90^\circ \leq \alpha < 180^\circ$)

Straight \rightarrow Straight



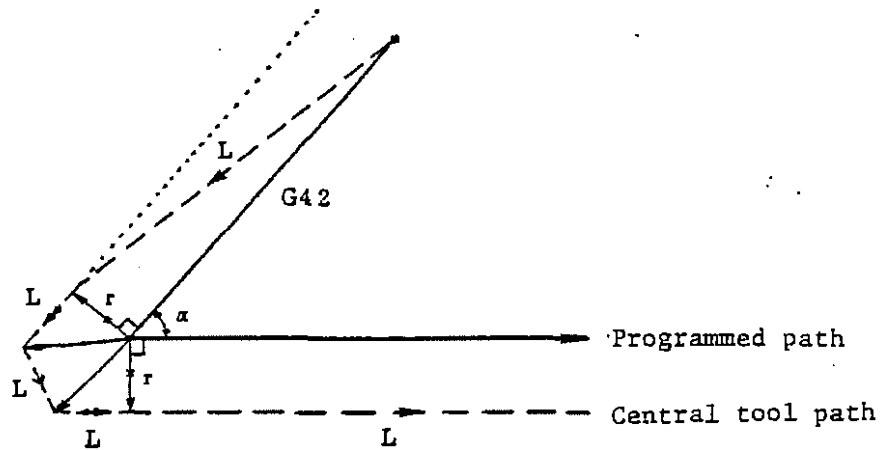
Straight \rightarrow Arc



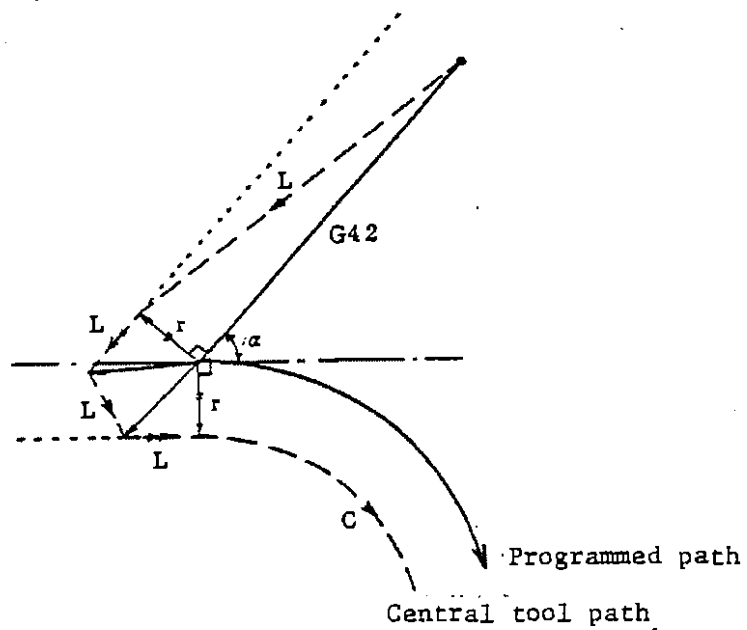


(iii) Outside acute-angle running ($\alpha < 90^\circ$)

- Straight \rightarrow Straight



Straight \rightarrow Arc

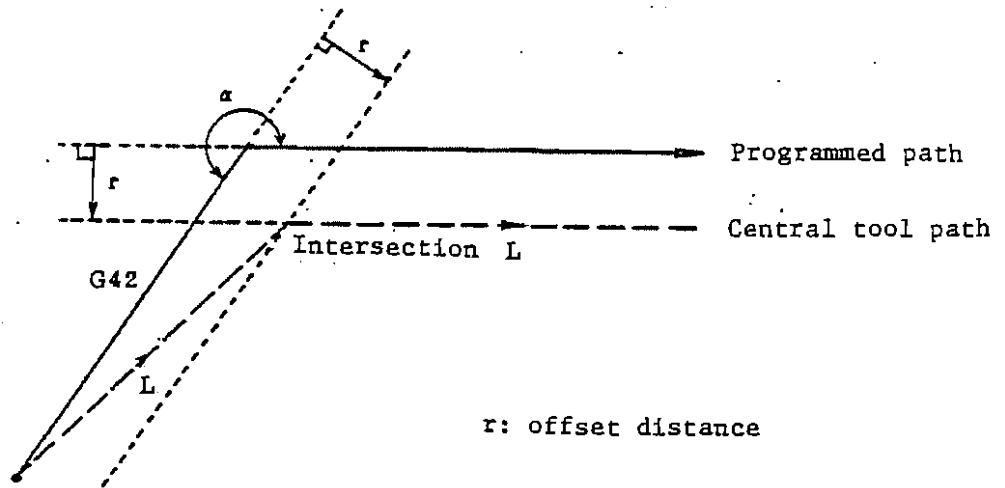




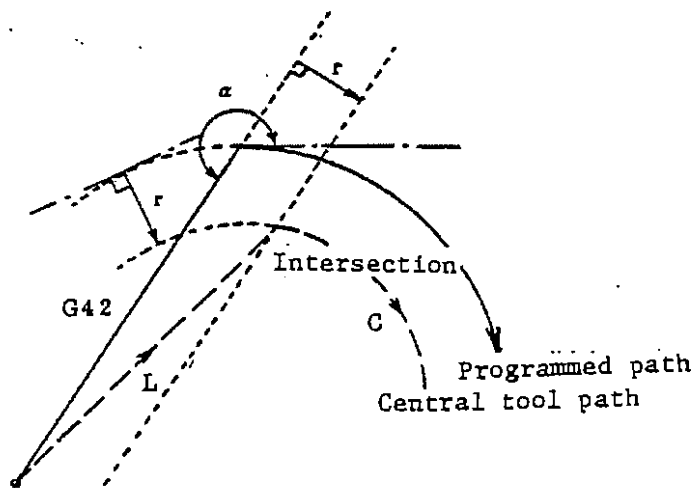
(Type B)

(i) Running inside ($180^\circ \cong \alpha$)

Straight \rightarrow Straight



Straight \rightarrow Arc

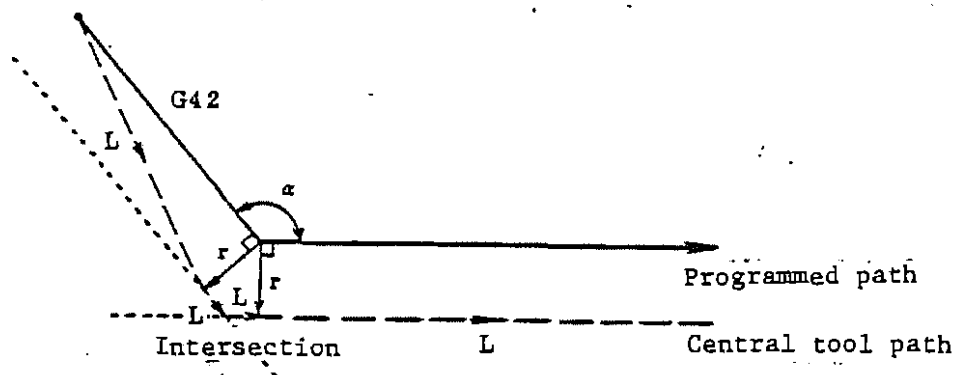


Note 3) "Intersection" is obtained through the offset of two blocks over (r) from a programmed path.

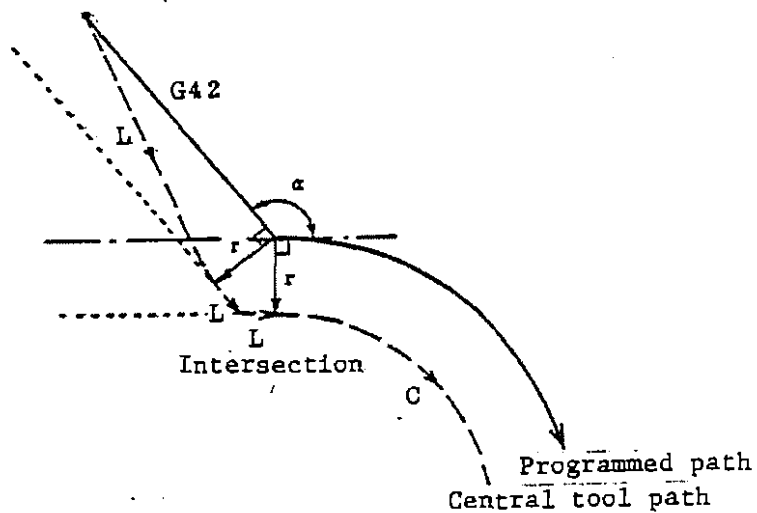


(ii) Outside obtuse-angle running ($90^\circ \leq \alpha < 180^\circ$)

Straight \rightarrow Straight



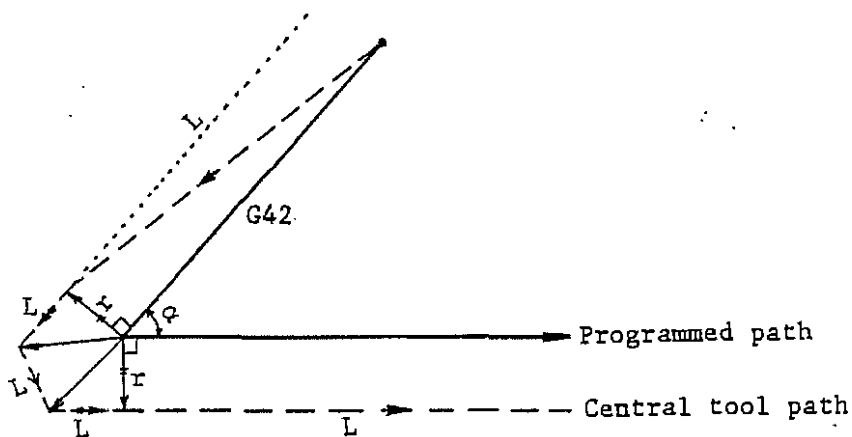
Straight \rightarrow Arc



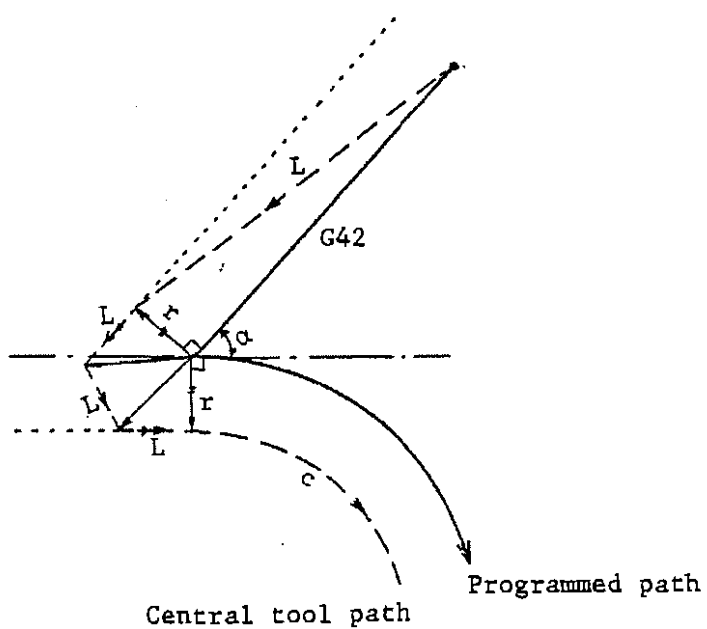


(iii) Outside acute-angle running ($\alpha < 90^\circ$)

Straight \rightarrow Straight



Straight \rightarrow Arc





③ Offset mode

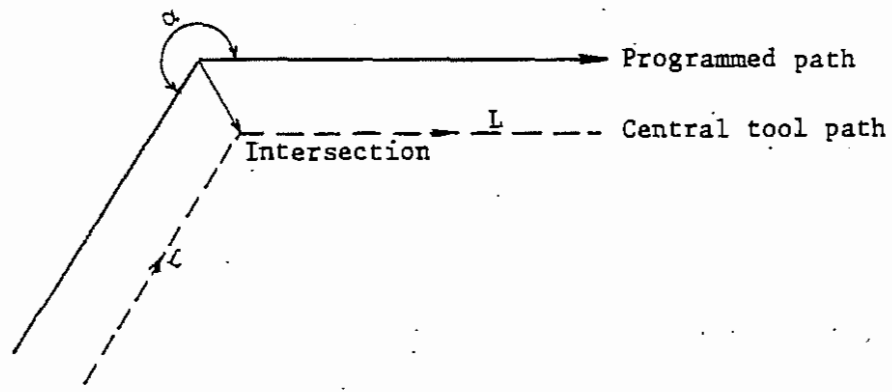
In offset mode, offset is carried out even if linear-interpolation, arc-interpolation or positioning instruction is given.

A shortage or excess occurs to the depth of cuts if more than six blocks not required to transfer are continuously for auxiliary functions and dwell in offset mode.

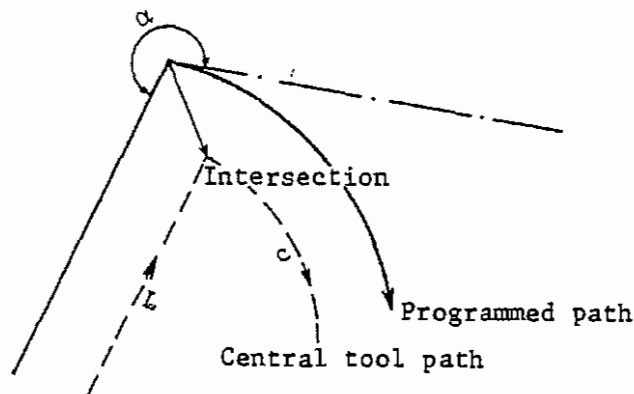
Alarm position is taken up if offset plane change-over is carried out in offset mode.

(i) Inside travel ($180^\circ \leq \alpha$)

Straight → Straight

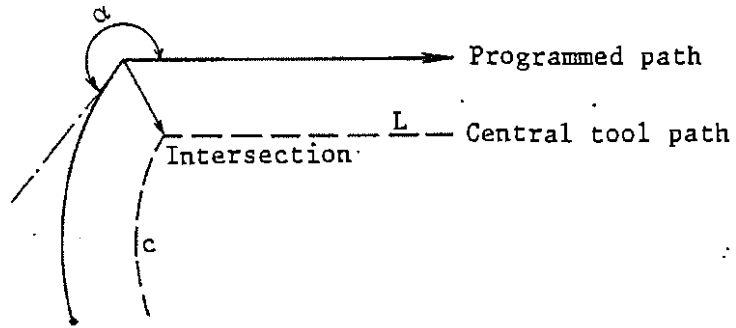


Straight → Arc

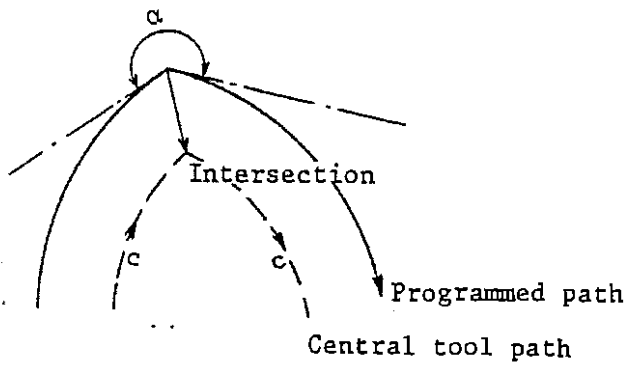




Arc → Straight

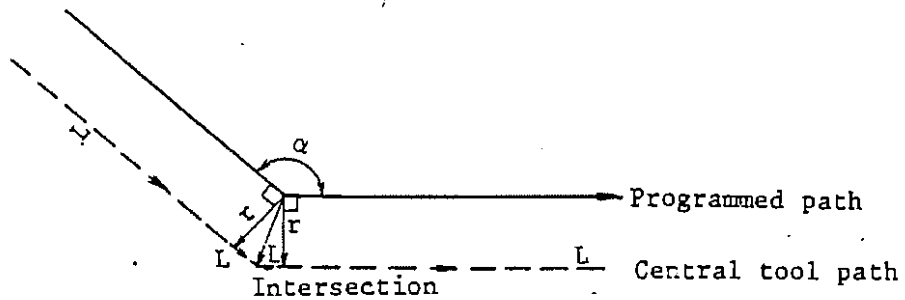


Arc → Arc



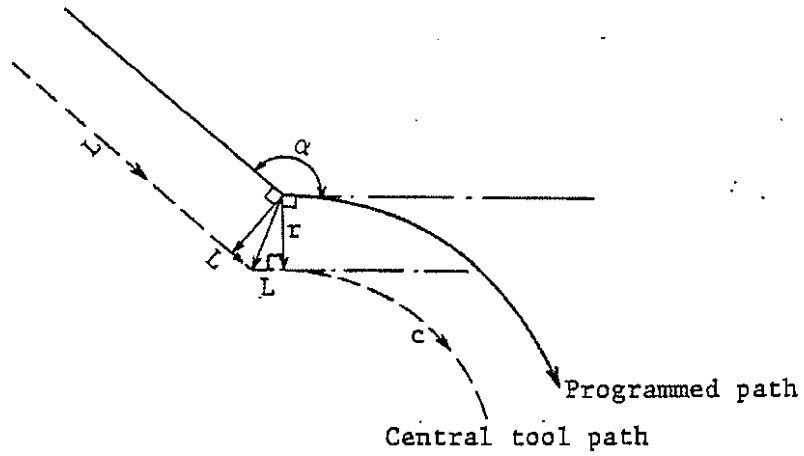
(ii) Outside obtuse-angle travel ($90^\circ \leq \alpha \leq 180^\circ$)

Straight → Straight

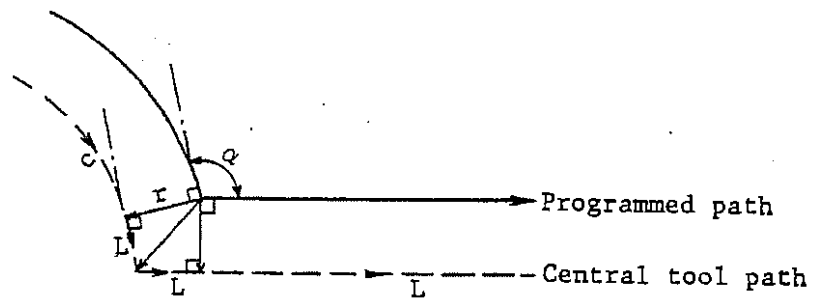




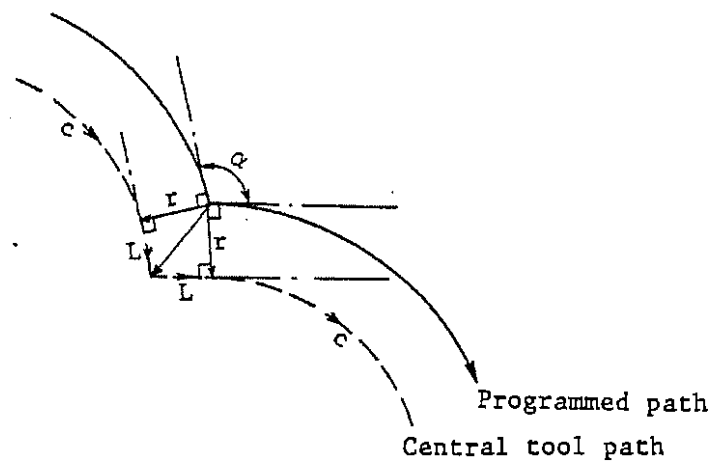
Straight → Arc



Arc → Straight



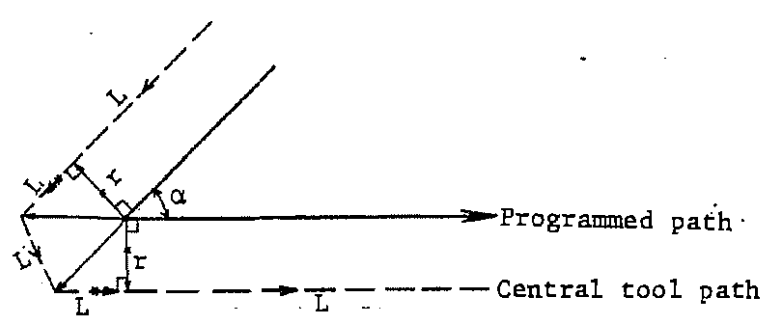
Arc → Arc



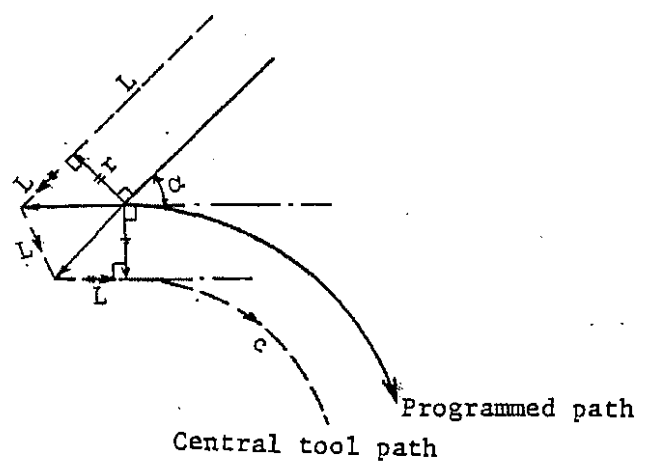


(iii) Outside acute-angle travel ($\alpha < 90^\circ$)

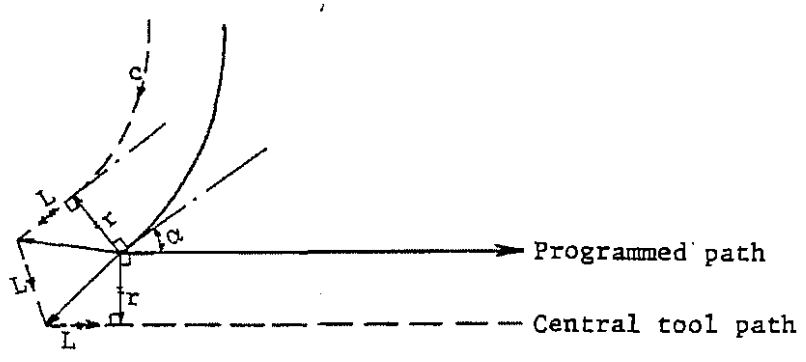
Straight \rightarrow Straight



Straight \rightarrow Arc

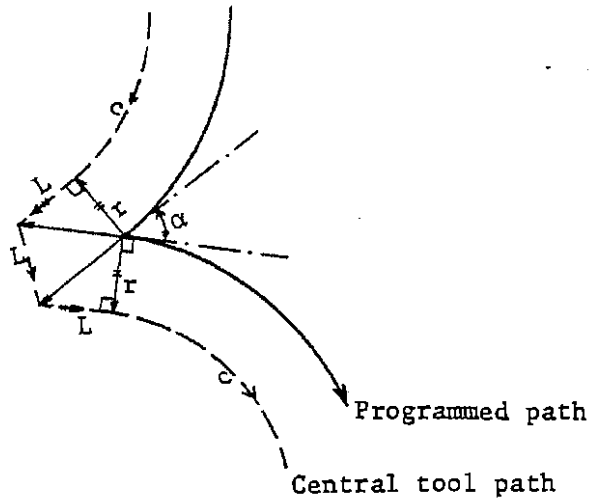


Arc \rightarrow Straight

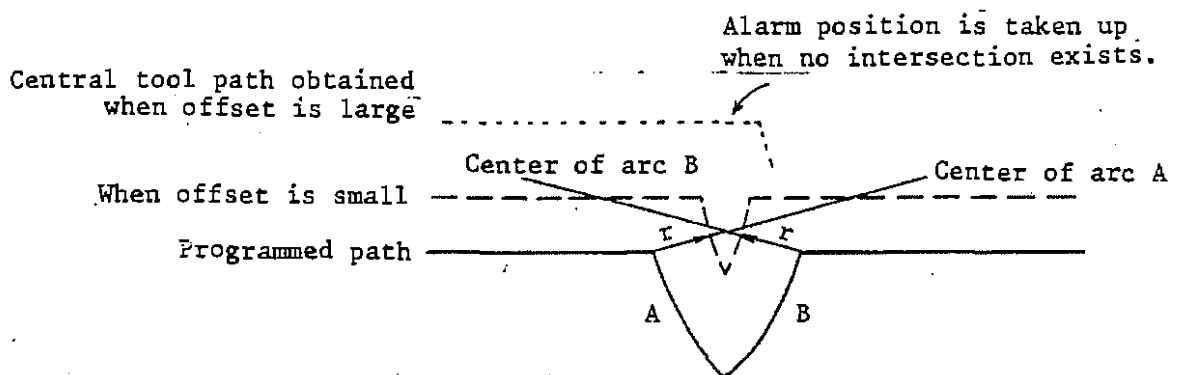




Arc → Arc



Note 4) An intersection existing at an offset route of the arcs when offset is small as illustrated below disappears in some cases when offset becomes large.





④ Offset cancellation

Cancellation mode position is taken up when a block that satisfies any of the following conditions is run in offset mode. The motion performed then is called offset cancellation.

(a) G40 instruction is given.

(b) Offset value: "0"

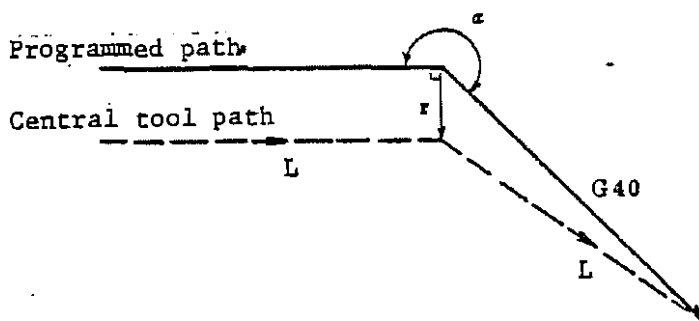
Alarm position is taken up if an arc instruction (G02, G03) is given on offset cancellation.

Offset cancellation of type A and type B are, similarly as in start-up, also available.

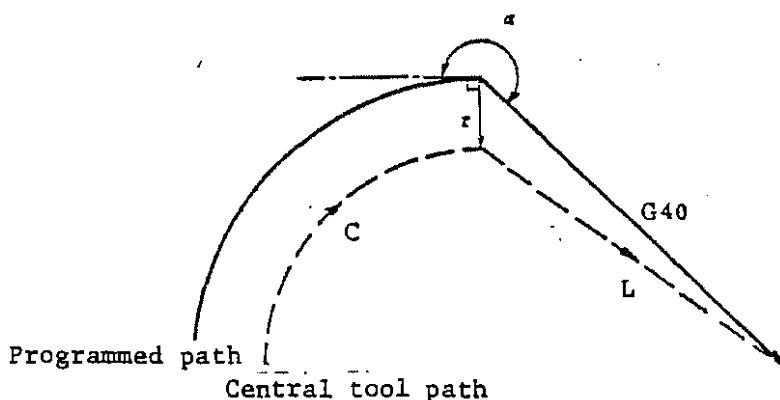
(Type A)

(i) Inside travel ($180^\circ \leq \alpha$)

Straight \rightarrow Straight

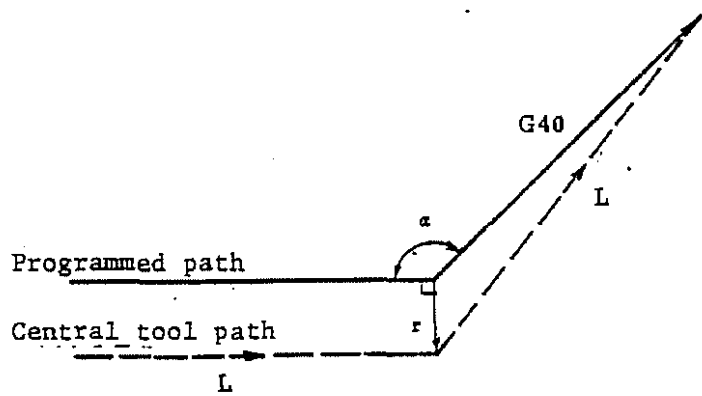


Arc \rightarrow Straight

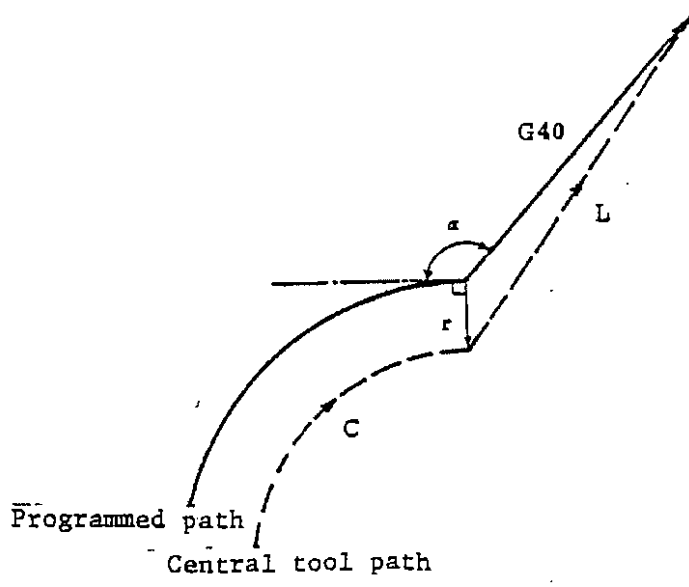




- (ii) Outside obtuse-angle travel ($90^\circ \leq \alpha < 180^\circ$)
Straight \rightarrow Straight

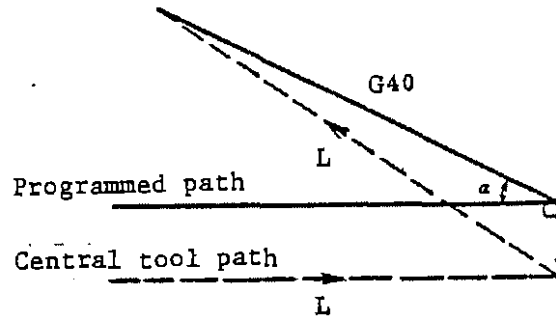


Arc \rightarrow Straight

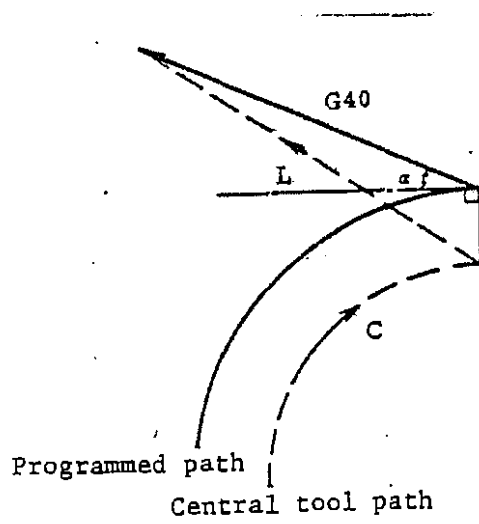




(iii) Outside acute-angle travel ($\alpha < 90^\circ$)
Straight \rightarrow Straight



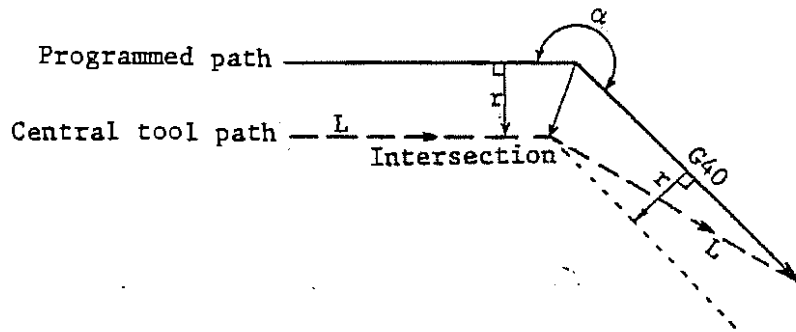
Arc \rightarrow Straight



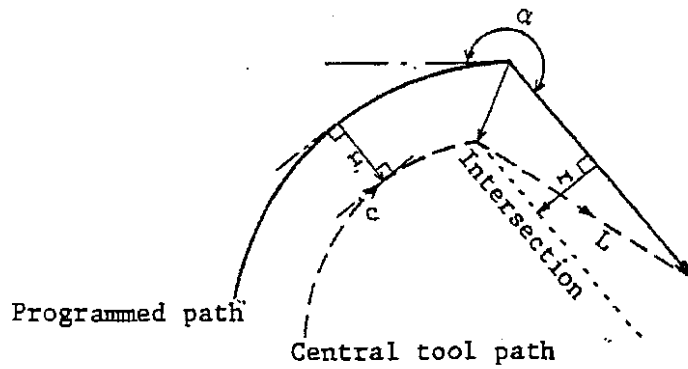


(Type B)

(i) Inside travel ($180^\circ \leq \alpha$)
Straight \rightarrow Straight



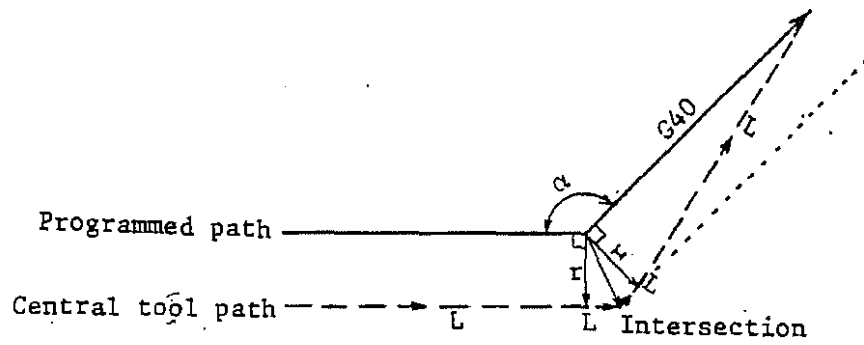
Arc \rightarrow Straight



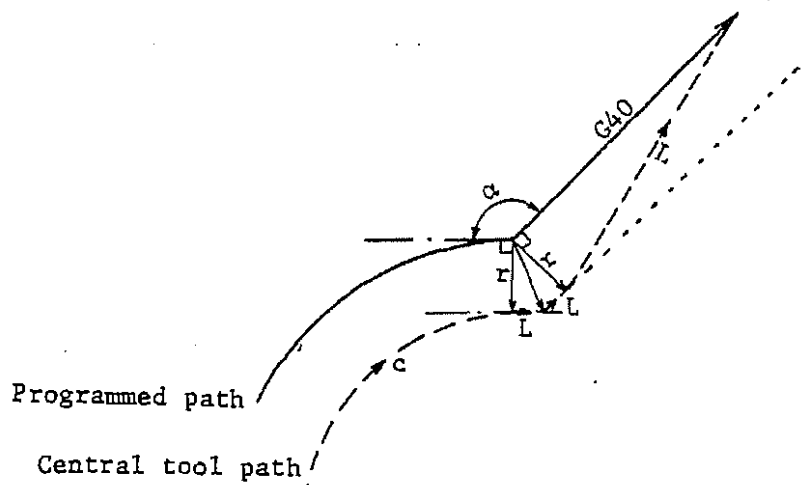


(ii) Outside obtuse-angle travel ($90^\circ \leq \alpha < 180^\circ$)

Straight → Straight

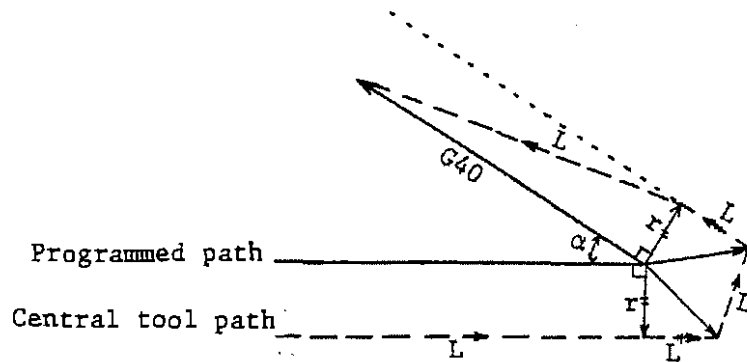


Arc → Straight

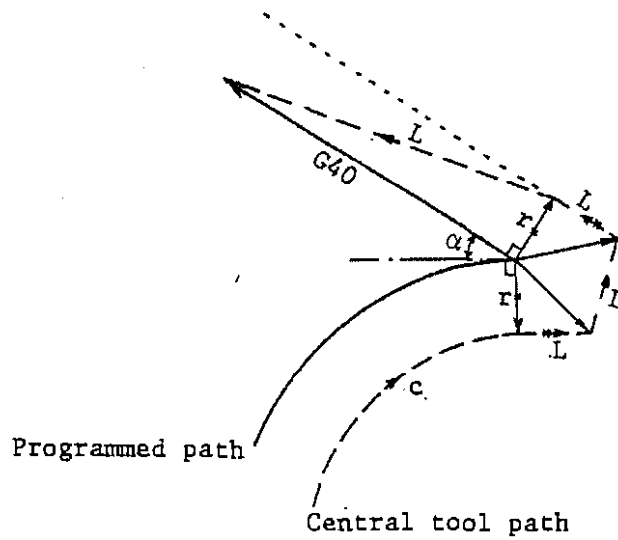




(iii) Outside acute-angle travel ($\alpha < 90^\circ$)
Straight \rightarrow Straight



Arc \rightarrow Straight





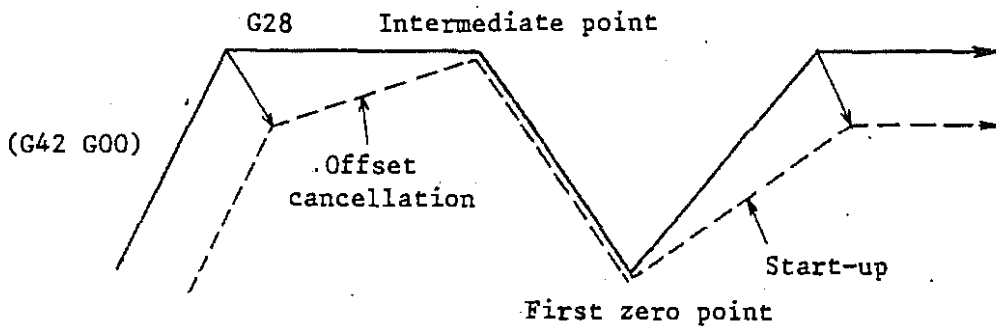
⑤ Temporary offset cancelling instruction

If any of the instructions mentioned below is given in offset mode, offset is temporarily cancelled and automatic resetting is made to offset mode.

Offset cancellation and start-up are performed in the same manner as " 4 offset cancellation" and " 2 start-up".

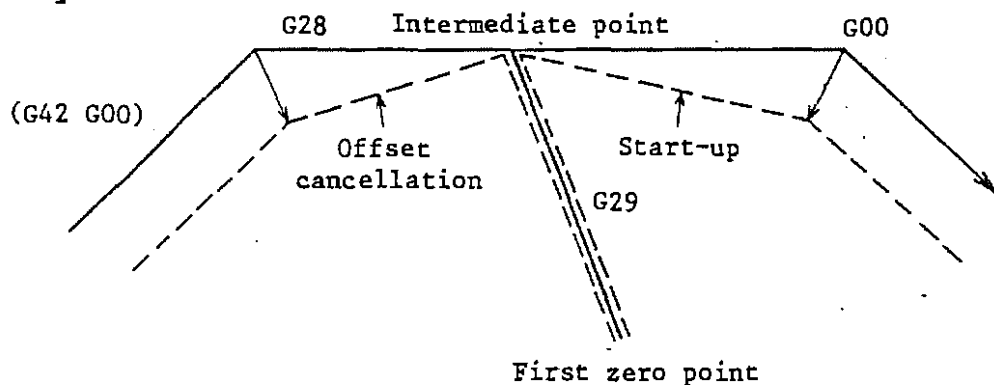
(i) G28, G30 (Return to zero point)

If G28 (G30) instruction is given in offset mode, offset is cancelled at the intermediate point and after origin resetting automatic resetting is performed to offset mode.



(ii) G29 (Return from first zero point)

If G29 instruction is given in offset mode, offset is cancelled at the intermediate point and automatic resetting is performed to offset mode.





⑥ Non-transfer blocks

The blocks mentioned below are called non-transfer block.
Offset causes no transfer there.

M03 S500 B180; .. M.S.B. code output	} No transfer is performed.
G04 X1000; Dwell	
G10 P01 R100; ... offset value setting	
(G17) Z 2000; ... transfer outside offset planes	
G90; only G-code	
; only EQB	
(G17) G91 X0 Y0;. transfer distance: 0	

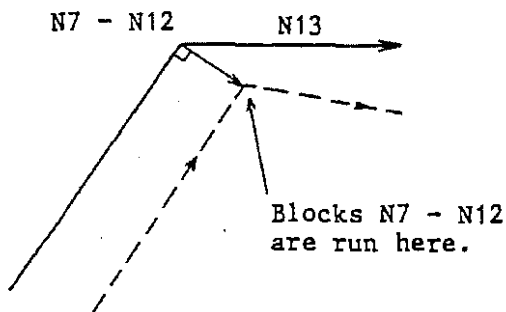
(i) Instruction is given on start-up.

No offset vector is made when an instruction is given on start-up.

(ii) Instruction is given in offset mode and at the same time as offset cancellation.

(a) The ordinary central tool route is used when more than six non-transfer blocks are not continuously designated.

(b) When more than six non-transfer blocks are continuously designated, a vector with a size equal to that of offset is made orthogonally with the transfer direction of the block preceding them. Therein, however, an excess occurs to the depth of cuts.



```

N6 G91 X100. Y100.;
N7 S500;
N8 M03;
N9 B180;
N10 G04 X1000;
N11 Z200.;
N12 G90;
N13 X100.;

```



⑦ Corner travel

On outside travel correction, corner-turn travel is performed. The tool performs single-block stop when the single block is turned on.

Its travel is performed at the same speed as the preceding block.

⑧ General cautions in offset

(1) Designation of offset value

The following methods are available for specifying offset value (parameter change-over).

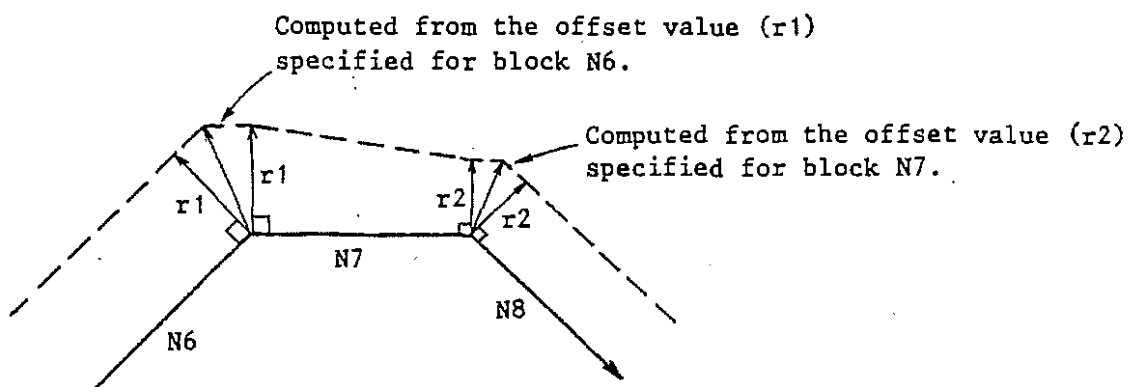
1. Designation of offset value number with D-code
2. Reference is made to the tool nose radius-registered on tool data. Tool nose radius + offset value is used when the latter is specified using D-code.

Chamfering tools are assumed to have radius "0".

D-code is used to specify tool position offset value as well as tool nose radius correction offset value. Alarm position is however taken up if tool position offset (G45 - G48) instructions are used in tool nose radius correction mode (G41/42).

(ii) Change of offset value

Offset value is generally changed when the tool is replaced in cancellation mode. When this change is made in offset mode, however, the vector at the end point of blocks is computed using the offset value specified therefor.





Note 5) Alarm position is taken up if D-code instructions are given to one block when arc instruction is given in offset mode.

Note 6) Alarm position is taken up if offset value is set negative (-).



2-4-4. D- and H-functions

D-function is used to correct tool nose radius (tool nose radius correction, tool position offset).

H-function is used to correct tool length (tool length correction, tool position offset).

Note 1) Correction value "0" always corresponds to D00 and H00.

Note 2) Tool correction numbers are totaled 128 (01 - 128).

Note 3) Be sure to use D-code for tool nose radius correction (G40, G41, G42).

Be sure to use H-code for tool length correction (G43, G44, G49).

Which of D- and H-codes is used for tool position offset (G45, G46, G47, G48) depends on set parameter.

2-4-5. Program input of offset value

The program input of offset value is possible using instruction:

```
G10 P ___ R ___;      P: offset number
                      R: offset value
```

Note 1) Specified offset value is always absolute.

Note 2) G10 is disregarded on tool path since the exchange of offset value is hazardous.



2-5 Canned Cycle Function

Canned cycle simplifies programs since the machining usually designated for several blocks is designated for one block.

Canned cycle G-code

G code	Uses	Operation at hole bottom		Return	Used address
		Dwell	Spindle		
G70					
G71	True circle cycle (CW)			Rapid traverse	X,Y,Z,R,P,Q, D,L
G72	" (CCW)			"	"
G73	High-speed peck drilling cycle			Rapid traverse	X,Y,Z,R,Q,L
G74	Back-tap cycle	(Provided)	Normal rotation	Cutting feed	X,Y,Z,R,(P), L
G75	Fine boring cycle	Provided	Bottom finish - orientation stop	Run-off rapid traverse	Z,Y,Z,R,P,Q, L
G76	Fine boring cycle	Provided	Orientation stop	"	X,Y,Z,R,P,Q, L
G77	Back facing cycle	Provided			X,Y,Z,R,P,L
G78					
G79					
G80	Canned-cycle cancellation				
G81	Drill and spot drill cycle	Not provided		Rapid traverse	X,Y,Z,R,L
G82	Drill and counter boring cycle	Provided		Rapid traverse	X,Y,Z,R,P,L
G83	Peck drilling cycle			Rapid traverse	X,Y,Z,R,Q,L
G84	Tap cycle	(Provided)	Reverse rotation	Cutting feed	X,Y,Z,R,(P) L



G code	Uses	Operation at hole bottom		Return	Used address
		Dwell	Spindle		
G85	Boring cycle	Not provided		Cutting feed	X,Y,Z,R,L
G86	Boring cycle	Provided	Stop	Rapid traverse	X,Y,Z,R,P,L
G87	Back-boring cycle	Not provided	Normal rotation	Cutting feed	X,Y,Z,R,P,Q, L
G89	Boring cycle	Provided		Cutting feed	X,Y,Z,R,P,L

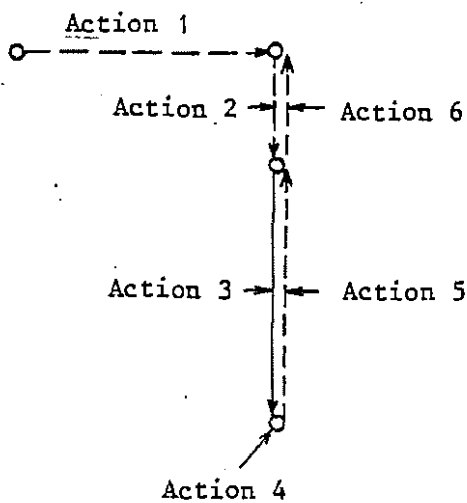
* (): parameter setting



(1) Actions

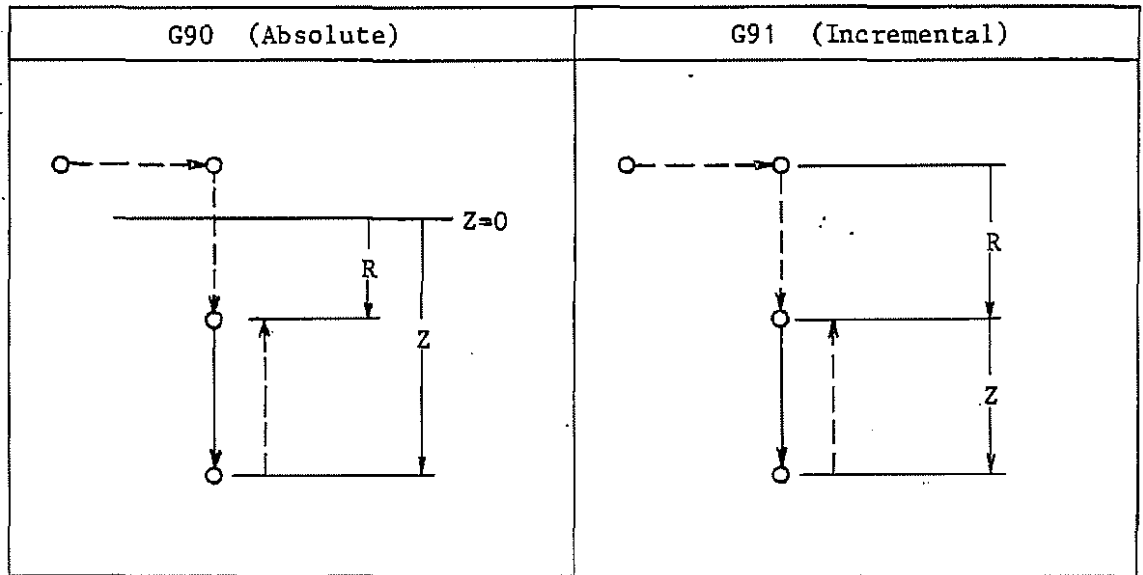
Canned cycle is generally a sequence of the following six actions.

- Action 1 ----- X- and Y-axis positioning
- Action 2 ----- Rapid traverse to R-point
- Action 3 ----- Drilling
- Action 4 ----- Action at hole location
- Action 5 ----- Run-off to point R
- Action 6 ----- Rapid traverse to initial point

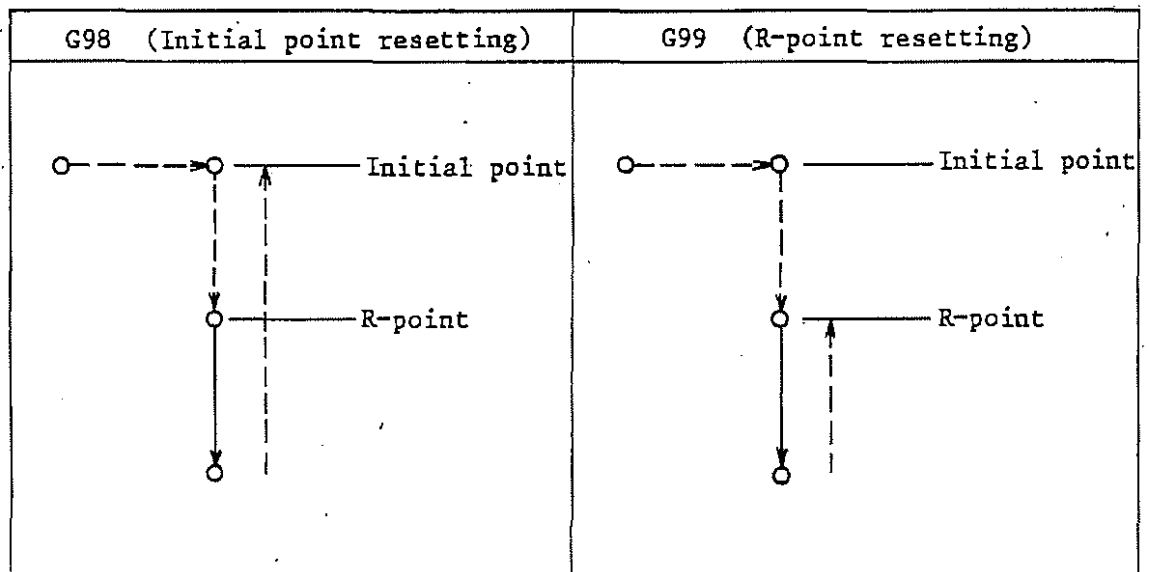




(2) How to give data to absolute instruction(G90) and incremental instruction (G91) -



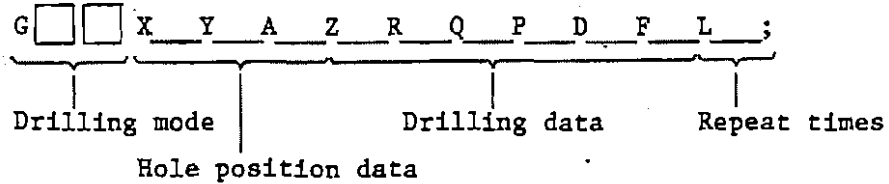
(3) Whether the tool is reset to R- or initial point is judged using G98/G99.





(4) Cycles

Designate canned-cycle machining data as illustrated below.



(i) Drilling mode (G)

Once designated, a drilling mode does not change until another drilling mode is designated or G-code instruction is given for cancelling fixed cycle. Therefore it is not necessary to designate the same drilling mode for each block when it is used continuously.

The G-code for cancelling canned cycle is included in G80 and A-group.

(ii) Drilling data

Z: Z-point

R: R-point

Q: lower hole diameter in G71 and G72, depth of each time's cut in G73 and G83, shift distance in G76 and G87

P: hole diameter in G71 and G72, duration of dwell at hole bottom

D: tool nose radius offset numbers in G71 and G72

F: cutting feed speed

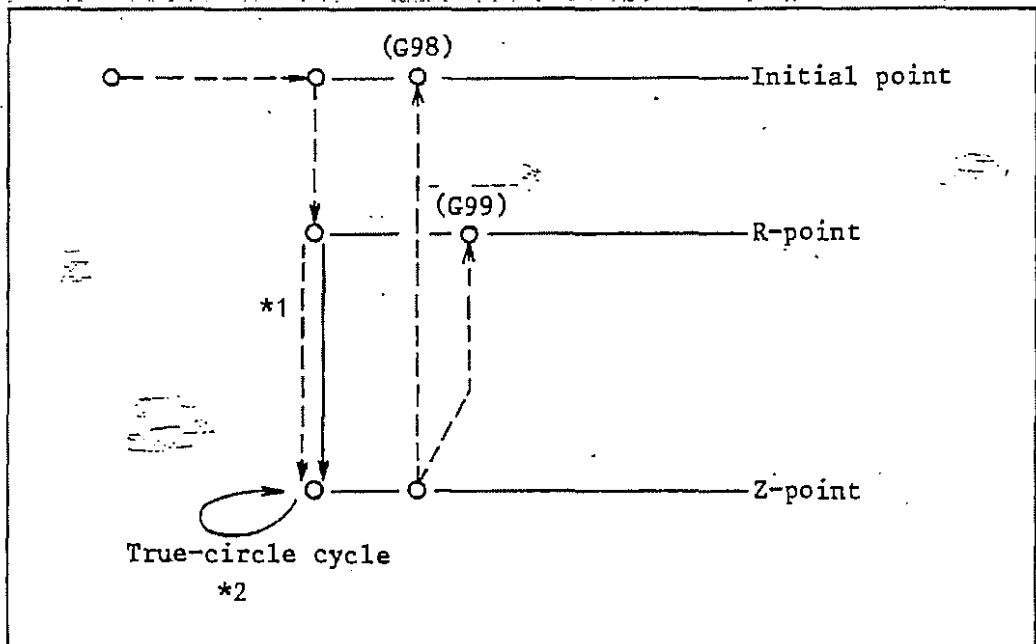
(iii) Repeat times "L"

"L" is regarded as "1" unless otherwise specified.

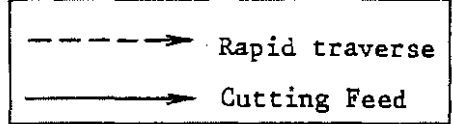
Only positioning is performed when L = 0.



G71 G72 (True-circle cycle)



- *1
- P: hole diameter
 - Q: lower hole diameter
 - D: offset number (tool radius)



EM6: allowance for lower hole diameter (parameter spot machining)

$\frac{(Dx2 + EM6)}{\text{Tool diameter}}$	$< Q$	Rapid traverse
$(Dx2 + EM6)$	$\geq Q$	Cutting feed (Feed $F = F \times 0.3$)

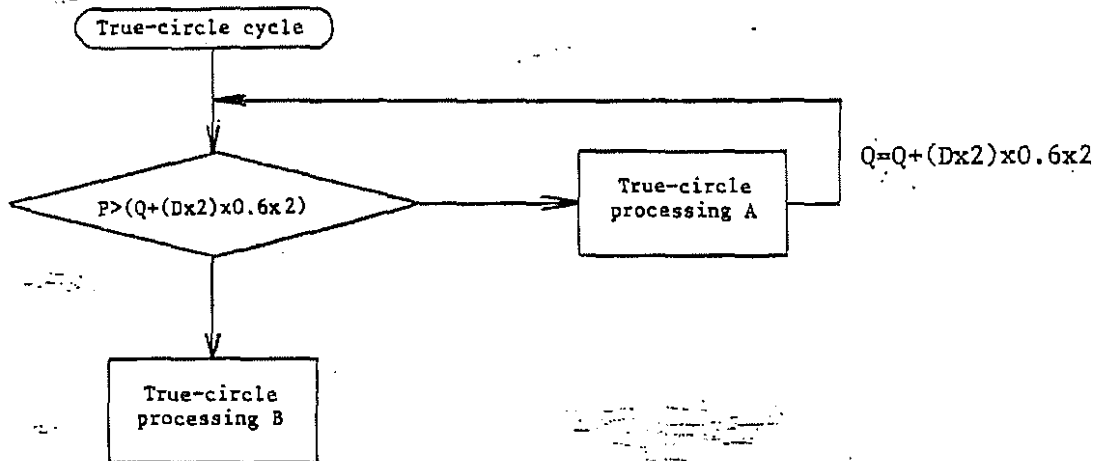
The feed from R-point to Z-point depends on programmed Q and D as well as parameter EM6.

Tool diameter is assumed to correspond to two times of the offset value specified by an offset number. Rapid traverse or cutting feed is carried out if tool diameter + EM6 is smaller or larger respectively than lower hole diameter Q.

Cutting feed however corresponds to 30% of programmed feed.



*2 Description of G71 (in G72, cutting direction should be assumed to be CCW.:

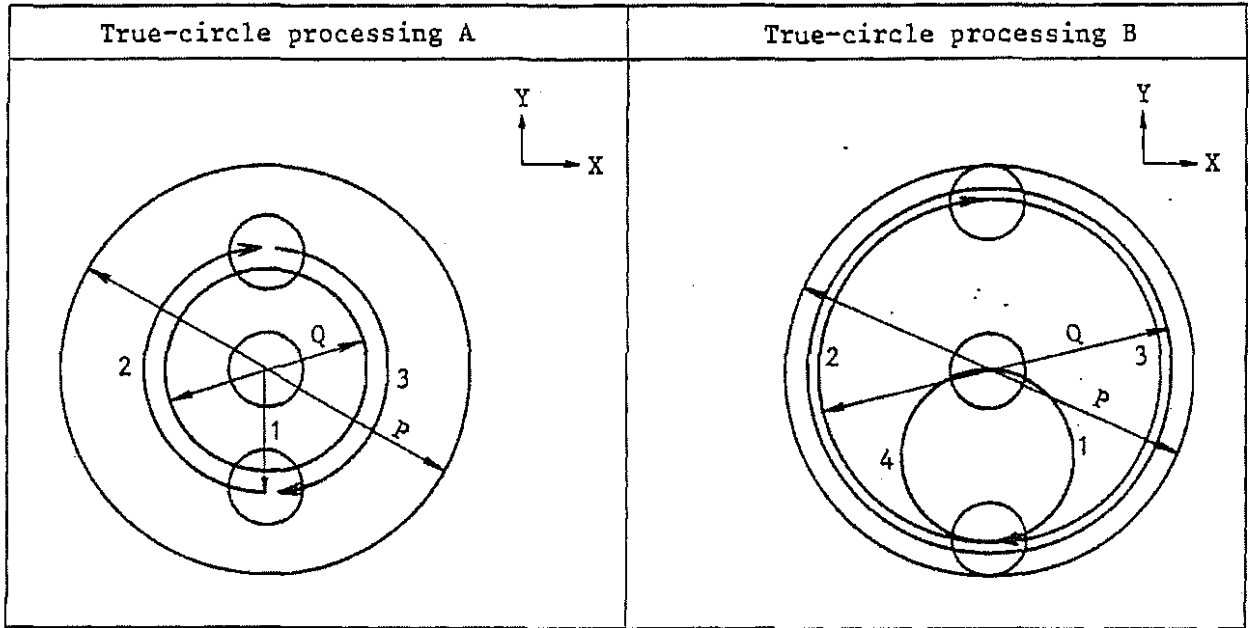


True-circle processing depends on programmed P, Q and D.

The depth of diametrical cuts corresponds to 60% of tool diameter (Dx2).

True-circle processing A is run if program hole diameter P is larger than the diameter of a one-time cut diametrical of lower hole Q. Thereafter the above action is repeated on the assumption that the hole diameter obtained then corresponds to lower hole diameter Q.

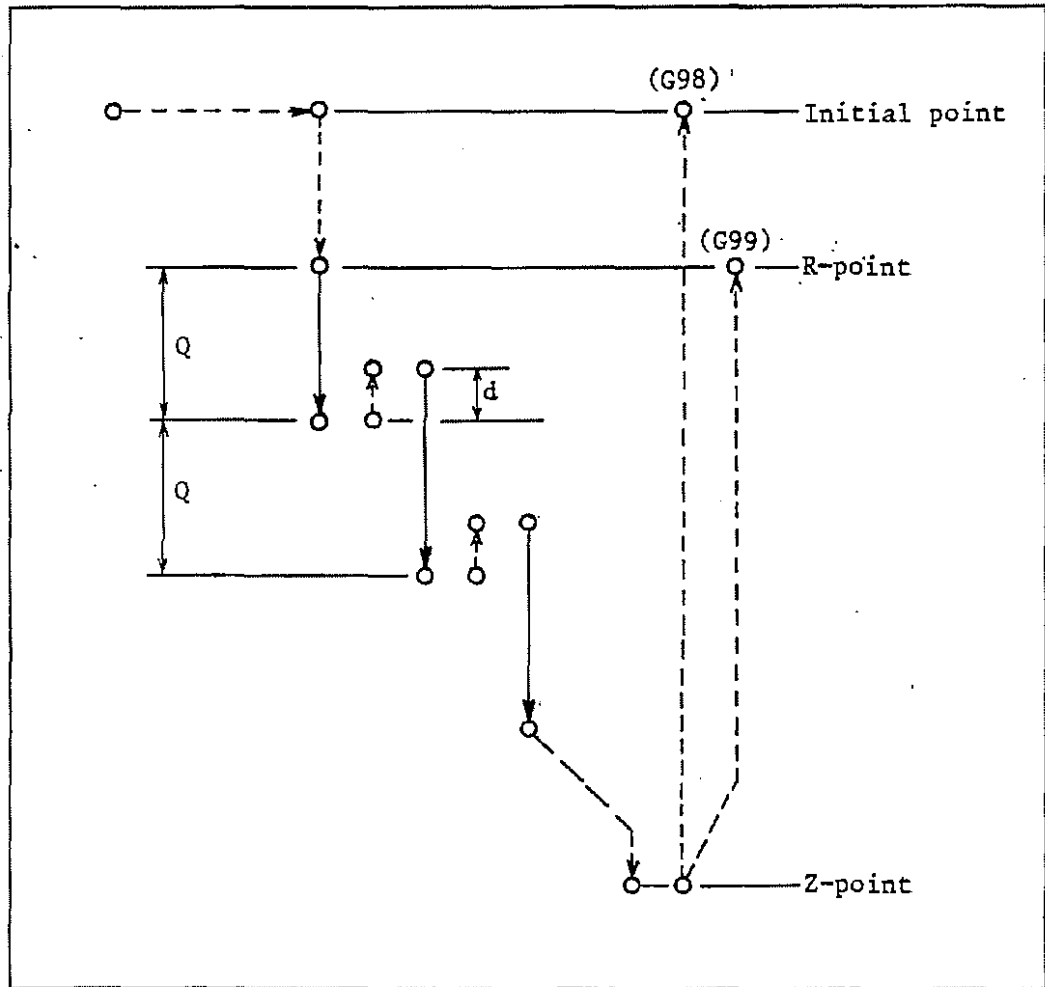
True-circle cycle is completed through true-circle processing B when diametrical cutting has made the hole diameter larger than program hole diameter P.





G73

(High-speed peck drilling canned cycle)



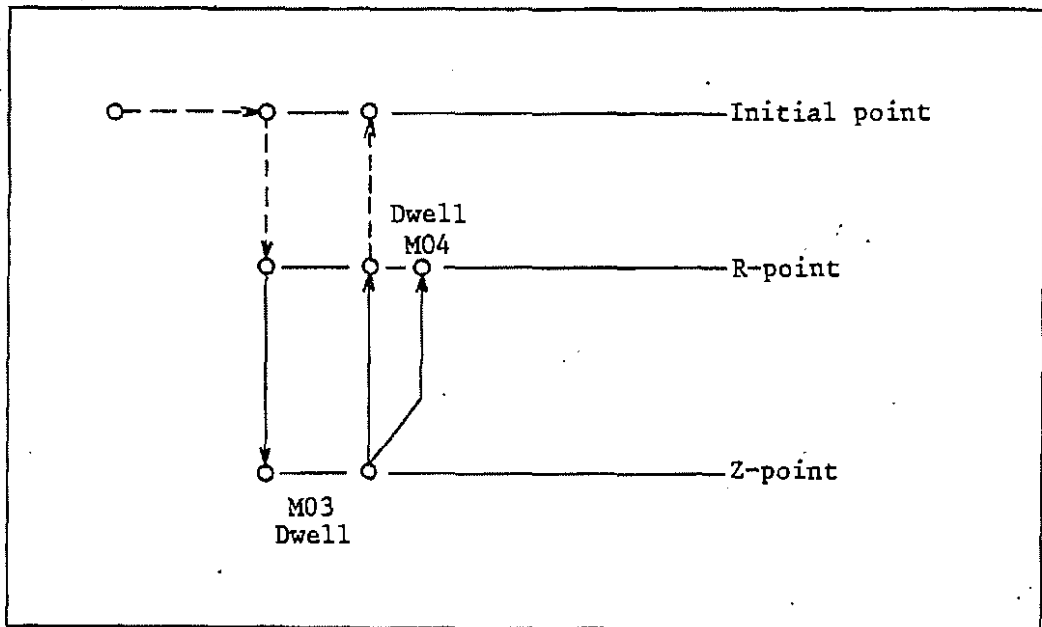
Run-off distance is set using parameter (PY3).

High-efficiency machining is possible since the intermittent feed in the Z-axis facilitates the discharge of cutting chips and run-off distance can be set fine.



G74

(Reverse tapping canned cycle)



Note 1) Override is disregarded during the tap with G74. Therefore it does not stop until the completion or resetting even if feed hold is performed.

In parameter spot machining BTD,

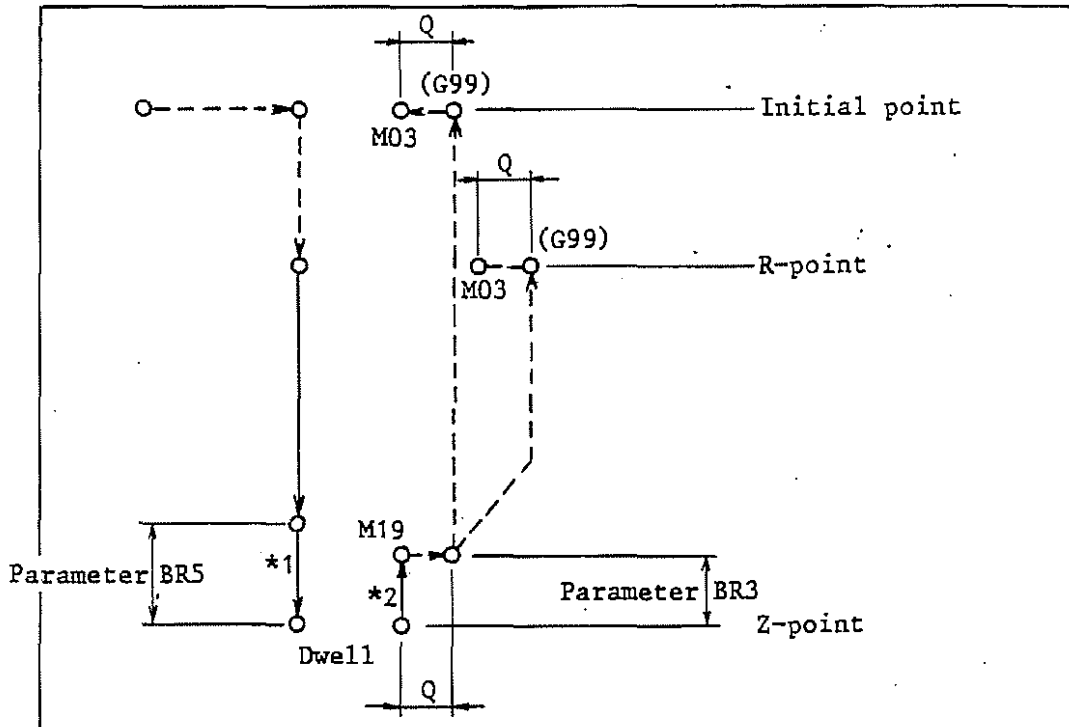
- Bit 0: dwell at hole bottom before M03
- Bit 1: dwell at hole bottom after M03
- Bit 2: dwell at R-point before M04

Dwelling time is to be specified by address P.



G75

(Fine boring canned cycle)



*1, *2 both feed and revolutions: 70%

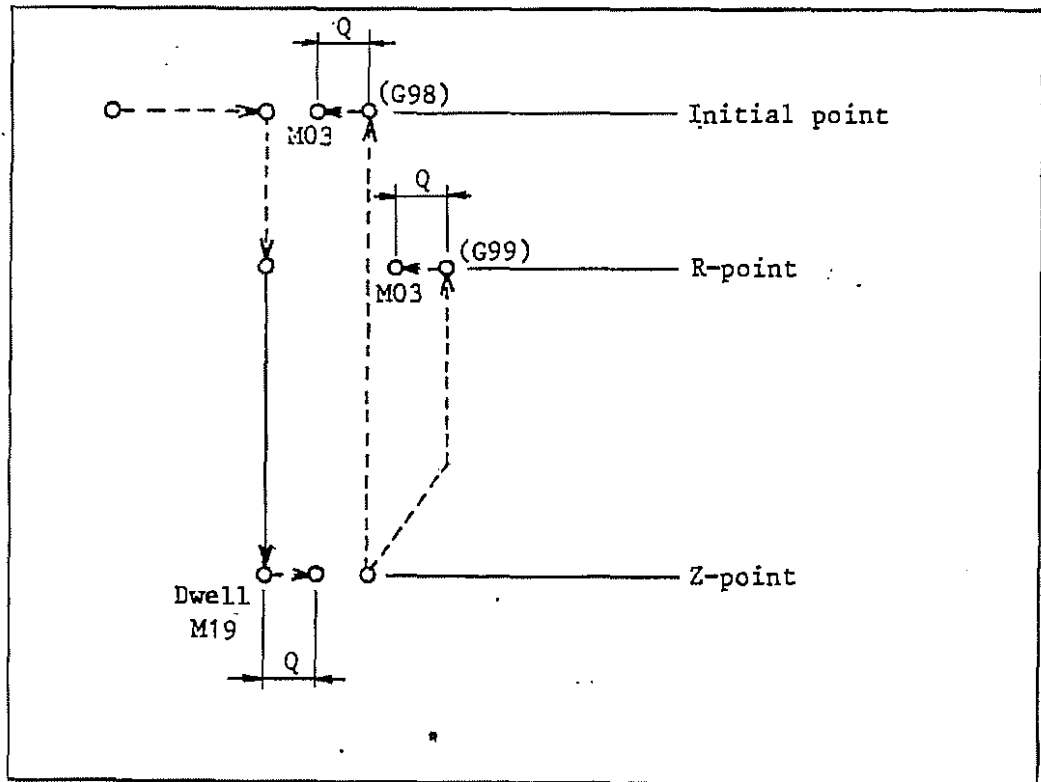
Shift direction: use parameter BR4 for setting

The spindle stops at constant revolutions position and shifts oppositely to tool tip before pulled off. Therefore, high-accuracy, high-efficiency boring is possible without damaging machining surface.



G76

(Fine boring canned cycle)



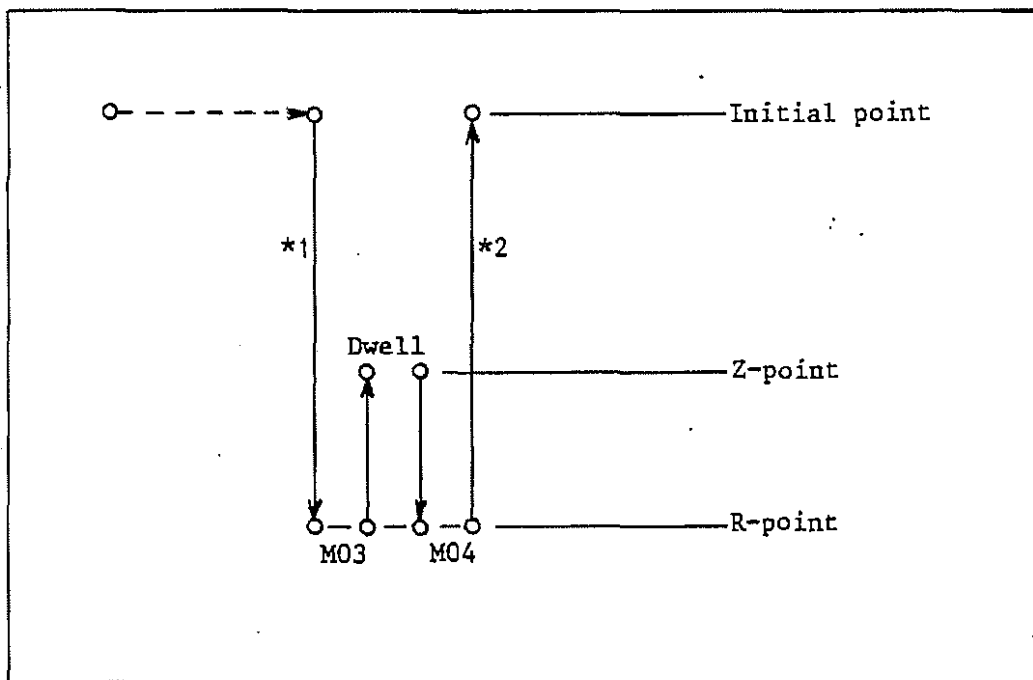
Shift direction: use parameter BR4 for setting

The same as G75 in that boring is possible without damaging machining surface.



G77

(Back facing canned cycle)



*1, *2 Use parameter DR4 to set feed speed.

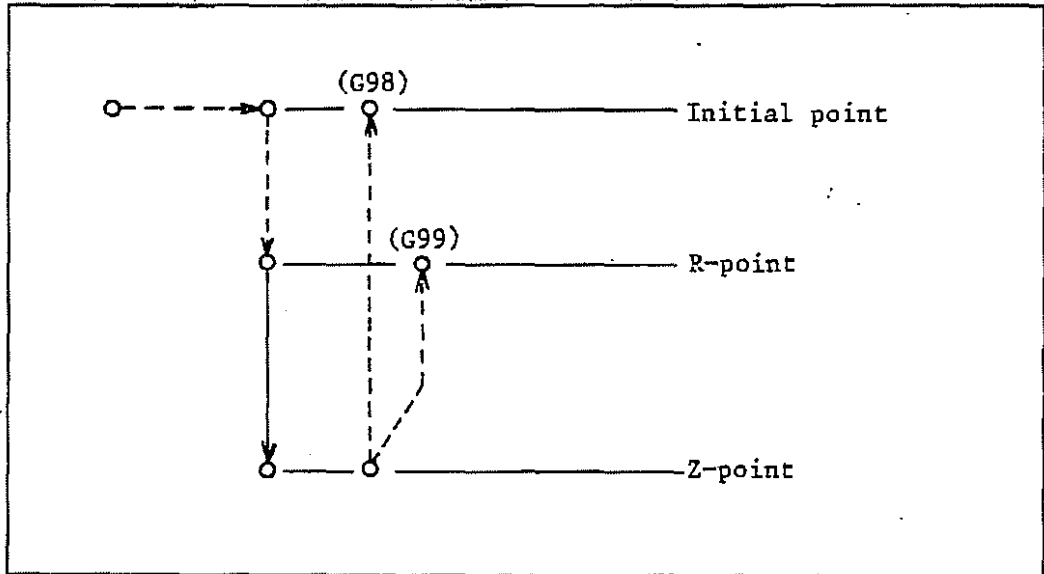
If feed speed is set to "0", programmed feed is carried out.

Note 1) Not concerned in G98 and G99.



G81

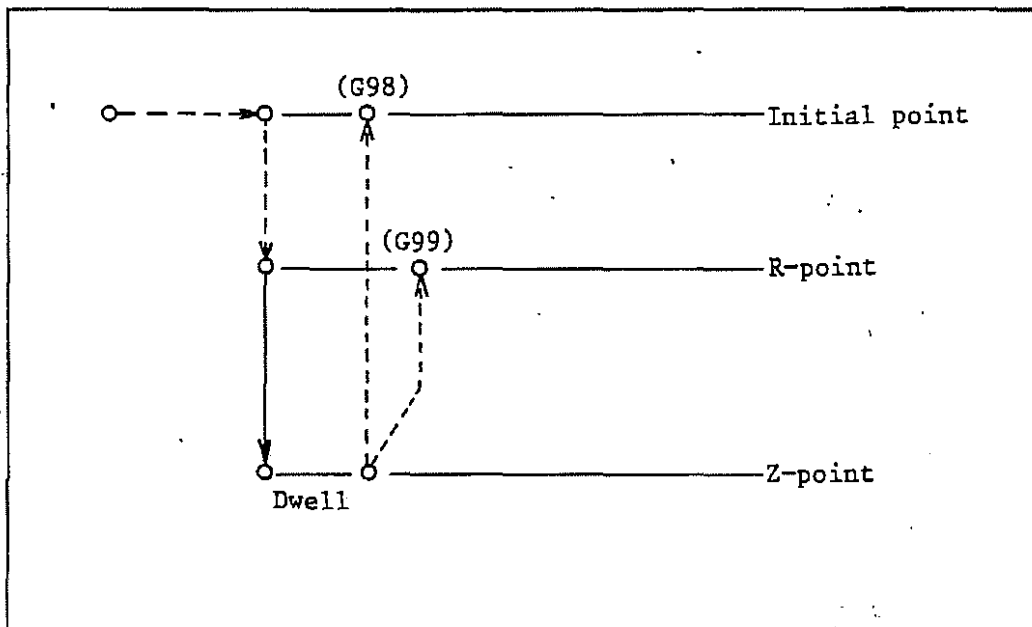
(Drilling canned cycle, spot drilling)





G82

(Drilling canned cycle, counterboring)

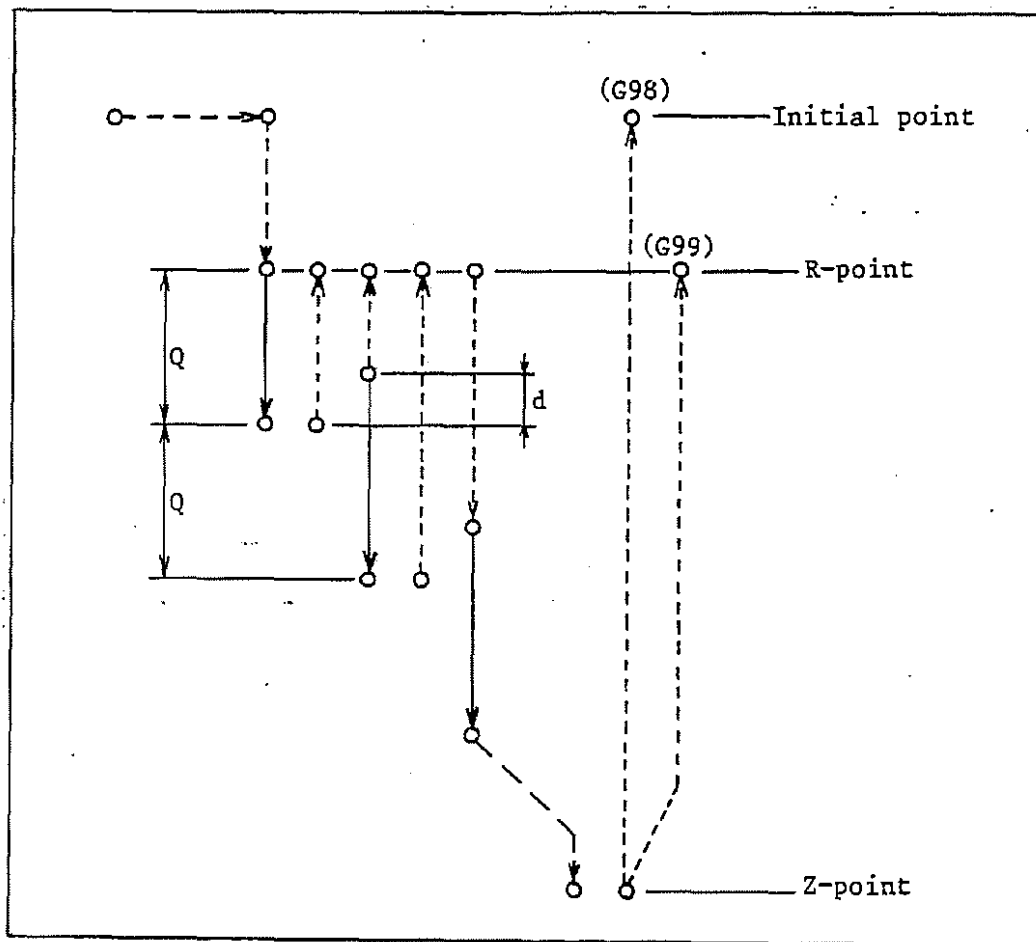


The depth of blind holes is improved in accuracy through the dwell at hole bottom.



G83

(Peck drilling canned cycle)

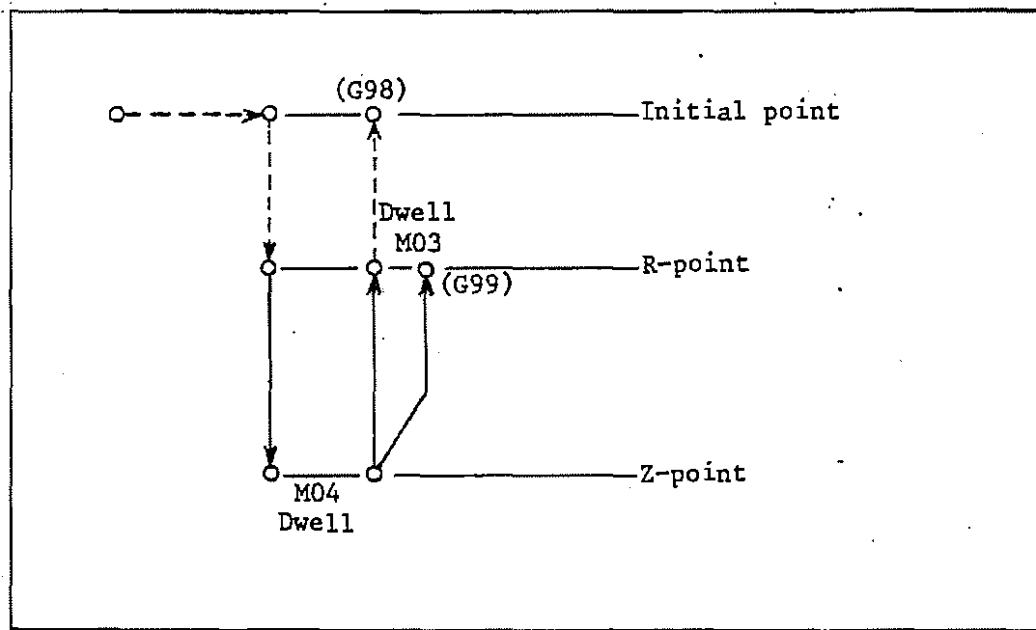


Use parameter (PY4) to set run-off distance α .



G84

(Tapping canned cycle)



Note 1) Override is disregarded during the tapping canned cycle with G84. Therefore, it does not stop until the completion of resetting even if feed hold is performed.

In parameter spot machining BTD:

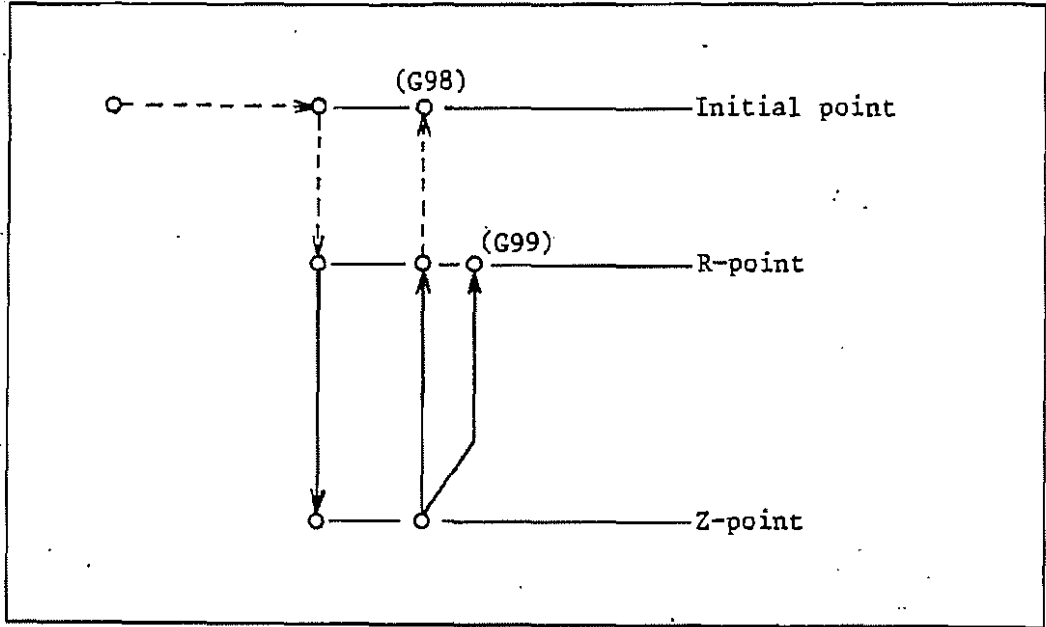
- Bit 0: dwell at hole bottom before M04
- Bit 1: dwell at hole bottom after M04
- Bit 2: dwell at R-point before M03

Dwelling time is to be specified by address P.



G85

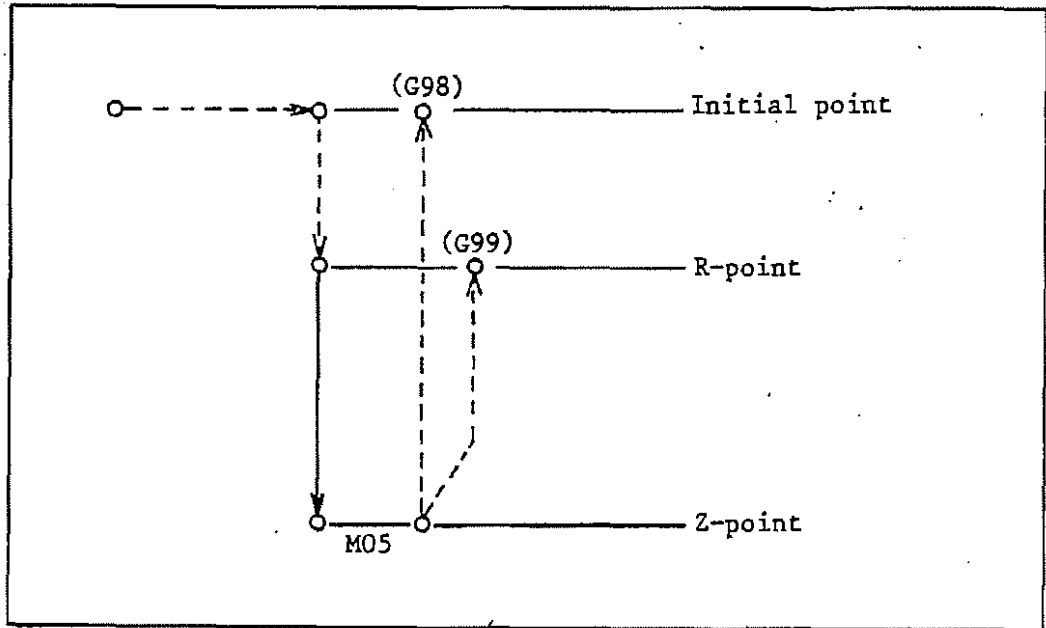
(Boring canned cycle)





G86

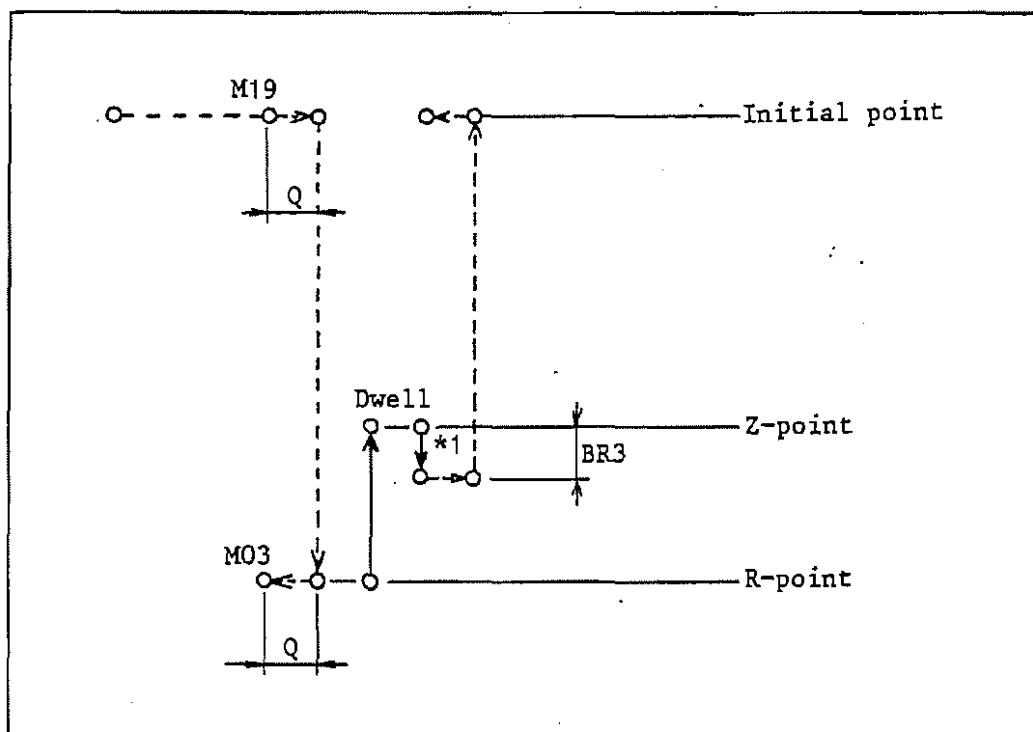
(Boring canned cycle)





G87

(Back boring canned cycle)



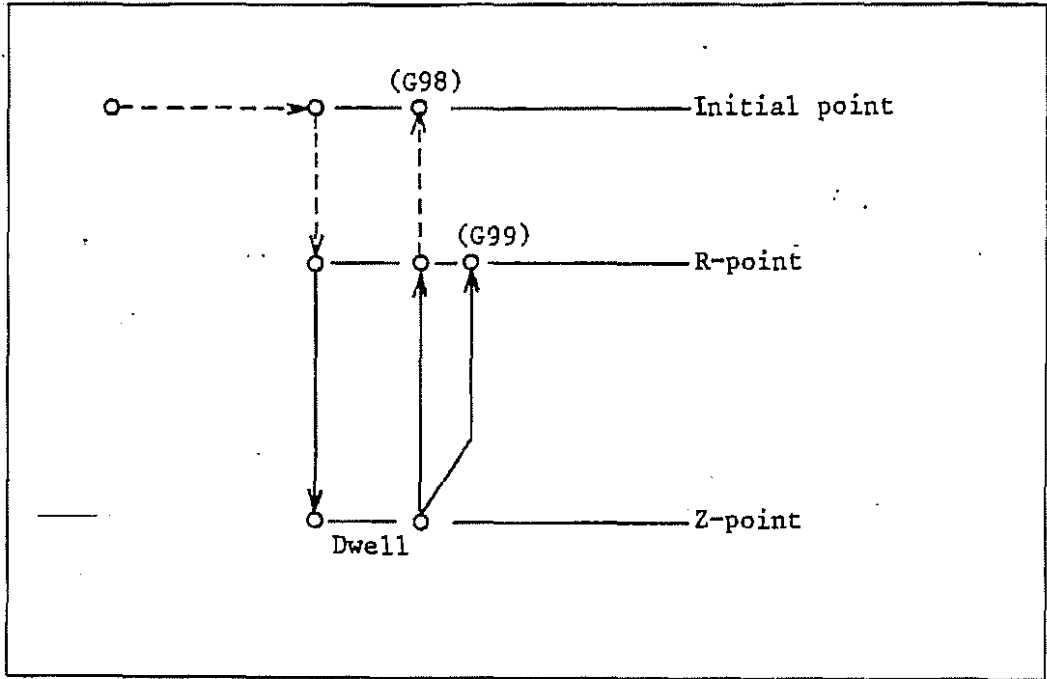
*1 Bit 3 of parameter BTB (0: no run-off is performed in Z-direction/1: run-off is performed in Z-direction). Shift direction: use parameter BB2 for setting

Note 1) Not concerned in G98 and G99.



G89

(Boring canned cycle)





- Note 1) Before giving fixed cycle instruction, set the spindle in operation using the auxiliary function (M-code).
Be sure to give M04 instructions (reverse rotation of spindle) to one block when using G75, G76, G86 and G87.
Be sure to give M03 instructions to one block for the sequence of normal rotation with G77 → reverse rotation cutting → normal rotation return.
- Note 2) In fixed cycle mode, drilling is carried out if even one of X, Y, A, Z and R data is present in the block being run. The cycle is run for each block after run once.
- Note 3) If an auxiliary function instruction is given to the same block as the fixed cycle, M-code is generated on the first positioning.
- Note 4) In fixed cycle mode, tool position offset instructions (G45 - G48) are disregarded.
- Note 5): If tool length offset instructions (G43, G44, G49) are given in fixed cycle mode, offset is carried out on the positioning (action 2) at the R-point.
- Note 6) G27 to G31 or a G code of group A causes the drilling cycle to be cancelled with G80 effected and the drilling data to be cleared.
-
- Note 7) Cautions for operator
- (a) Resetting
Drilling mode is changed over to cancellation mode (G80) and drilling data are cleared.
 - (b) Single block
The fixed cycle at a single block stops at the end point of actions 1, 2 and 6.
 - (c) Feed hold
Operation stops after continuing up to action 6 if feed hold is performed for actions 3 - 5 using G74 and G84.
 - (d) Override
Both the feed and spindle overrides in the actions 3 - 5 with G74 and G84 are 100%.



Note 8) When G88 instruction is given (G88 X_Y_Z_R_P;), the spindle stops after the dwell at hole bottom and if a single block is turned on the tool makes single block stop. Upon starting, the tool is reset to the R-point or initial point at rapid traverse rate.

2-6 Spindle function (S-function), tool functions (T- and E-functions), auxiliary function (M-function) and second auxiliary function (B-function)

2-6-1. Spindle function (S-function)

Capable of directly specifying spindle revolutions (rpm) using address S and the five-digit numeral (max. 65,000) following it.

2-6-1. Tool function (T and E functions)

Decides which of a parameter or T-code and M06 are used for ATC setting.

(1) T-code type ATC

Uses address T and the three-digit numeral following it to designate a tool to be fitted on the spindle this time. Uses address E and the three digit numeral following it to designate a tool to be fitted on the spindle next time.

(2) M06 type ATC

Uses address T and the three-digit numeral following it to designate a tool to be fitted on the spindle next time.

The T-code at the same block as M06 is disregarded.



2-6-3. Auxiliary function (M-function)

Generates signals for the ON/OFF control of the machine side in accordance with the three-digit numeral instruction following address M.

M-codes (1) - (3) are used for special meaning.

- (1) M00: Program Stop
- (2) M01: Optional Stop
- (3) M02, M30: NC Resetting

M-codes (4) - (12) are processed in the numerical controller.

- (4) M06: ATC Instruction (parameter is set to M06 type ATC)
 - (5) M48: Override effective
 - (6) M49: Override Cancellation
 - (7) M90: Mirror Image Cancellation
 - (8) M91: Mirror Image Symmetrical with respect to the X-axis of Work Coordinate System
 - (9) M92: Mirror Image Symmetrical with respect to the Y-axis of Work Coordinate System
 - (10) M93: Mirror Image Symmetrical with respect to the A-axis of Work Coordinate System
 - (11) M98: Sub-program Call
 - (12) M99: Sub-program Return
- } Refer to "Mirror image" for further details
- } Refer "Sub-programs" for further details.

Note 1) Don't use any instruction for M35 EIA/ISO program.

2-6-4. Second auxiliary function (B-function)

Uses the three-digit numeral following address B for index table positioning.



2-7 Mirror Image (M90 - M93)

M-code	Function
M90	Mirror image cancellation
M91	Mirror image symmetrical with respect to work coordinate system X = 0
M92	" Y = 0
M93	" A = 0

Mirror image becomes effective for the block next to that to which M91 - M93 instructions are given and on.

Note 1) Cancell the mirror image using M90 in principle when using G28 and G30 instructions as well as coordinate system setting instructions (G54 - 59, G92).



2-8 Sub-programs (G22, G23 or M98, M99)

(1) Making

The programs at whose end M99 or G23 is entered are sub-programs.

Note 1) It is permitted as illustrated below that no block is made for M99.

X ____ M99;

G23 should however constitute a block.

(2) Run

Use the following to call sub-programs.

M98P ____ L ____ ;

↑ ↑
 Call repeat times

↑
 Sub-program number

Call repeat time is 1 if L-instruction is omitted.

Note 2) M98P ____ L ____ can be given to the same block as transfer instruction.

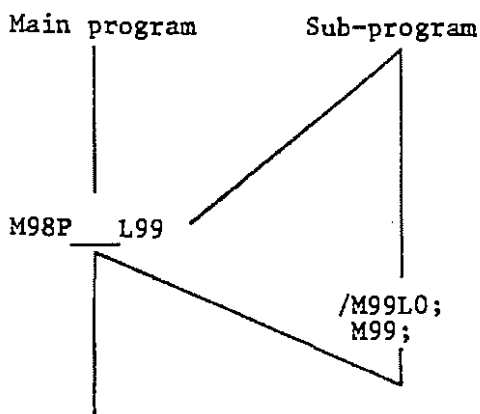
G23P ____ L ____ should however constitute a block.

(3) Special uses

i) If M99 is run in a main program, resetting is made to its head.

ii) M99L α;

This instruction compulsorily sets sub-program call repeat times L to α.



The example illustrated at the left: when optional block skip 1 is turned off, call times is set to 0 and resetting is made to the main program.



2-9 User Macro (option)

MAZATROL M-2 is provided with the instructions for the operation and branch in programs that are available for EIA/ISO programs. The sub-programs made from the fixed cycle with a parameter using the function can be utilized by other programs. Being useful for a special fixed cycle not fitted on M-2, it raises program efficiency and improves its functions.

The following are the main uses of the user macro.

- (1) Special machining cycle
 - a) Special drilling cycle ultra-peck drilling cycle
 - b) Simple die machining cycle ... semicylindrical machining

- (2) Machine control cycle
 - a) Special tool machining automatic change of
angle-head and long tools
 - b) Control of outer controller ... offset data supplied by
outer metering devices

- (3) Automation and adaptive control
Special measuring mode not provided on MMS



(1) Use of macro-programs

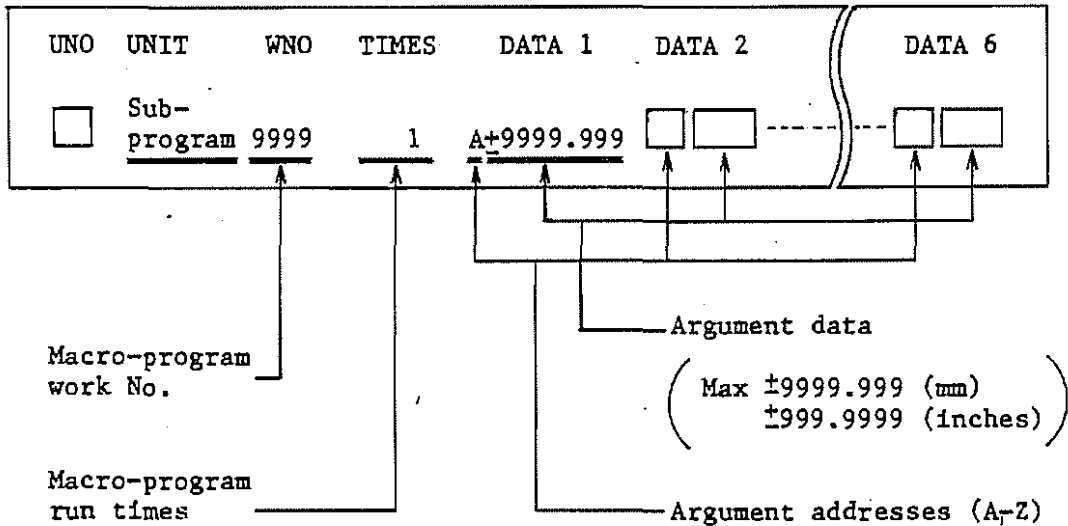
The macro-programs with macro G-code (refer to "Macro-instructions") and # variables (refer to "Variables") are made using EIA/ISO codes. They can be called using either of the MAZATROL language and EIA/ISO codes with the following formats.

(i) Macro-program call

MAZATROL language	Sub-program unit	Simple call	Run only on call
EIA/ISO	G65	Simple call	Run only on call
	G66	Modal call	Run is carried out for each axial transfer instruction until cancelled after call.
	G67	Modal call cancellation	

(ii) Input formats

o MAZATROL language: call by means of sub-program unit



Note) MAZATROL language does not ensure the G- and M-code call of sub-programs.



o EIA/ISO

G65P _____ { ~ } (Unmodal)



Program number



Argument address and data

G66P _____ { ~ } (Modal)

G67 (Modal cancellation)



(2) Variables

Variables, made of symbol # and a variable number following it, are expressed by (i= 1, 2 ...). Those available for M-2 are roughly divided in two - common variables and system variables.

(i) Common variables (#1 - #80)

Used to preserve computation values in macro-programs. Usable as axial transfer data as well as the data for S- and F-codes. The argument data for calling macro-programs are handed after substituted for the common variables corresponding to their respective argument addresses.

Address	Variable number	Address	Variable number	Address	Variable number	Address	Variable number
A	#1	H	#8	Q	/	V	#22
B	#2	I	#9	P	/	W	#23
C	#3	J	#10	Q	#17	X	#24
D	#4	K	#11	R	#18	Y	#25
E	#5	L	/	S	#19	Z	#26
F	#6	M	#13	T	#20	/	/
G	/	N	/	U	#21	/	/

Even if power supply turns off, the contents of common variables are preserved using a back-up power source. All the contents of #1 - #80 variables can be displayed and set on parameter pictures.

Note: Pay care to the place of decimal point of the common variable when it is to be used as the argument of a code. An instruction "G#1X-100, F2000" means G1000 (G10000 in inch system) causing alarm, if the value of #1 is 1. on the parameter common variable picture. If G01 is required, set the value of #1 on the common variable picture to 0.001 (0.0001).



(11) System variables

System variables are used for the program control of the numerical controller by reading and changing its current internal state. They as well as the cautions in their use are tabulated in the next page.

(System variables)

Variable number	Function	R	W *1*	Description
1000-1016	Macro-I/F input	o	—	Read of input I/F for user macro (1000 - 1015 - bit correspondence, 1016 - all bits)
1100-1116	Macro I/F output	—	o	Transmission to output I/F for user macro (1100 - 1115 - bit correspondence, 1116, all bits)
2001-2128	Too offset data	o	o	Tool offset data, those corresponding to D or H1 - 128
2500-2506	X-axis work coordinate system offset	o	o	X-axis work coordinate system offset data (2500 : Coordinate shift) (2501 : G54 - 2506 : G59)
2600-2606	Y-axis work coordinate system offset	o	o	Y-axis work coordinate system offset data (2600 : Coordinate shift) (2601 : G54 - 2606 : G59)
2700-2706	Z-axis work coordinate system offset	o	o	Z-axis work coordinate system offset data (2700 : Coordinate shift) (2701 : G54 - 2706 : G59)
2800-2806	A-axis work coordinate system offset	o	o	A-axis work coordinate system offset data (2800 : Coordinate shift) (2806 : G54 - 2806 : G59)
2510 2610 2710 2810 2910	MAZATROL FRM unit - X - Y - Z - A - θ	—	o	Rewrite of MAZATROL FRM unit FRM - X - Y - Z - A - θ

*2)

*2)



(System variables)

Variable number	Function	R	W *1*	Description
4001-4121	Modal informations	o	—	4001 - 4015 : G-code (separate table) 4102 : B-code 4107 : D-code 4109 : F-code 4111 : H-code 4113 : M-code 4114 : Sequence No. 4115 : Program numbers 4119 : S-code 4120 : T-code 4121 : E-code
4150	Spindle tool No.	o	—	Tool No. actually set on the spindle
4200	NC alarm	—	o	Alarm output through alarm number input #4200 ← 901 "User alarm No. 1" #4200 ← 910 "User alarm No. 10"
4203	NC control informations	—	o	bit0 0: feed hold effective, 1: ineffective bit1 0: feedoverride effective, 1: ineffective bit2 0: spindle override effective, 1: ineffective bit3 0: block stop effective, 1: ineffective
4500-4503	Macro interrupt information	o	—	4500 : Interrupted program No. 4501 : Interrupted unit No. (MAZATROL) 4502 : Interrupted sequence No. (MAZATROL) 4503 : Interrupted sequence No. (EIA/ISO)
5001-5004	Coordinate system of the work- piece in machining (Basic coord- inates)	o	—	5001 : Workpiece coordinate system X 5002 : Workpiece coordinate system Y 5003 : Workpiece coordinate system Z 5004 : Workpiece coordinate system A



(System variables)

Variable number	Function	R	W *1*	Description
5021-5024	Current position	o	—	5021 : Current position X 5022 : Current position Y 5023 : Current position Z 5024 : Current position A
5061-5064	Skip position	o	—	5061 : Skip position X 5062 : Skip position Y 5063 : Skip position Z 5064 : Skip position A
6001-6160 7001-7080 8001-8040 9001-9040	Drum No. 1 Drum No. 2 Tool length data Drum No. 3 (TOOL DATA screen) Drum No. 4	o	o	Tool length data for each tool pocket number
6201-6360 7201-7280 8201-8240 9201-9240	Drum No. 1 Drum No. 2 Tool breakage flag data Drum No. 3 (TOOL DATA screen) Drum No. 4	—	o	1 : Tool breakage indicating flag for each tool pocket number ON 0 : Tool breakage indicating flag for each tool pocket number OFF
6501-6660 7501-7580 8501-8540 9501-9540	Drum No. 1 Drum No. 2 Tool nose radius data Drum No. 3 (TOOL DATA screen) Drum No. 4	o	o	Tool nose radius data for each tool pocket number





- *1) Data setting for system variables is carried out only on the start of automatic operation. No data change is carried out on tool pass.
- *2) A printed board(option) is necessary for using macro- I/F input and output.
- *3) Rewriting of FRM unit in MAZATROL program can be done only in EIA/ISO program called by the subprogram immediately following the FRM unit.
- *4) Micro interrupt causes all modal information to be cancelled.

(G-code modal informations)

Variable number	Modal information
#4001	(Group 01) G00, G01, G02, G03, G33
#4002	(Group 02) G17, G18, G19
#4003	(Group 03) G90, G91
#4005	(Group 05) G94, G95
#4007	(Group 07) G40, G41, G42
#4008	(Group 08) G43, G44, G49
#4009	(Group 09) G73, G74, G76, G80, G81, G82, G83, G84, G85, G86, G87, G88, G89
#4010	(Group 10) G98, G99
#4012	(Group 12) G66, G67
#4014	(Group 14) G54, G55, G56, G57, G58, G59
#4015	(Group 15) G61, G62, G64



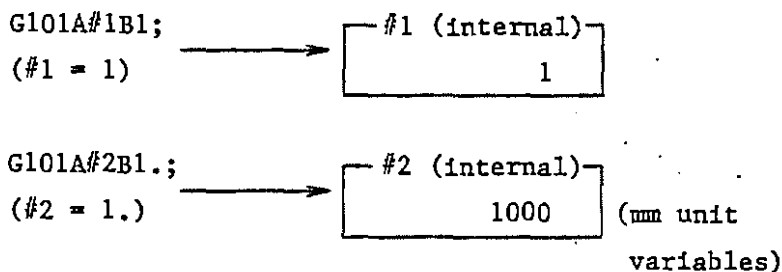
(iii) Use of variables

Take the following cautions in using common and system variables.

① Internal expression of variables

All variables are internally dealt with as integer with a code consisting of 32 bits and located (1/1000 and 1/10000 fixed decimal points for mm and inch respectively) only when referred to as the data with decimal point. The display without or with decimal point of program input is interpreted in view of the former or latter's format respectively.

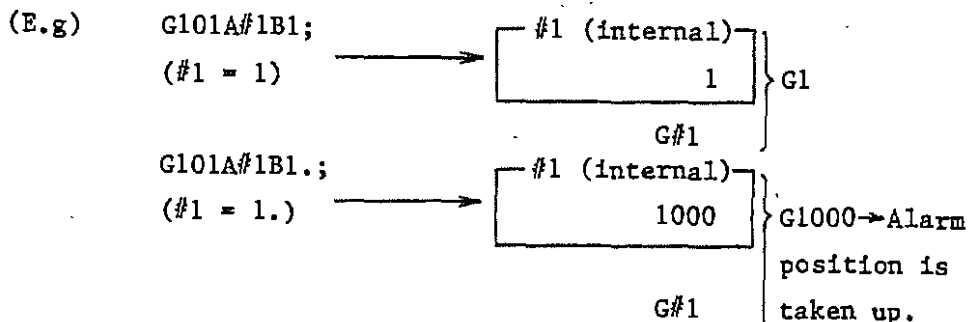
(E.g.)



Take care of the internal expression in using the variables.

o Use for codes

The use of # variables as code input value is possible for all addresses. Compare it with direct program input.



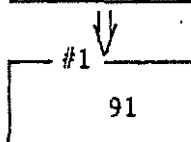
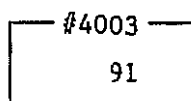


o Read of system variables

The contents of system variables are referred to in terms of their internal expression.

(E.g.) Modal read

G101A#1B#4003;
(#1 = #4003)



Note) The input of # variables to F-code is dealt with in the same manner as the direct input with decimal point.

(E.g) G101A#1B2.0; }
 (#1 = 2.0) } → F2.0;
 F#1; }



(3) Macro-instructions

The macro-instructions usable by M-2 can be divided in (1) definition and operation instructions and (2) branch instructions and all designated using three-digit G-code. The following are the G-code as well as the mode of their processing and their input format.

(i) Definition and operation instructions (G101 - G120)

A total of 20 instructions - 3 definition instructions and 17 operation instructions (four fundamental rule operation, trigonometric and logical operations) - are available.

G-code	Input format	Mode of operation	Function
101	G101 Aa Bb	$a = b$	Definition
102	G102 Aa Bb Cc	$a = b + c$	Sum
103	G103 Aa Bb Cc	$a = b - c$	Difference
104	G104 Aa Bb Cc	$a = b * c$	Product
105	G105 Aa Bb Cc	$a = b / c$	Quotient
106	G106 Aa Bb Cc	$a = b \sin C$	Sine (DEG)
107	G107 Aa Bb Cc	$a = b \cos C$	Cosine (DEG)
108	G108 Aa Bb Cc	$a = \tan^{-1} b/c$	Arc cotangent (DEG)
109	G109 Aa Bb	$a = \#b$	Variable number designation (I)
110	G110 Aa Bb	$a = \sqrt{b}$	Square root
111	G111 Aa Bb	$a = b $	Absolute values
112	G112 Aa Bb	$a(\text{BCD}) \leftarrow b(\text{BIN})$	BIN \rightarrow BCD conversion
113	G113 Aa Bb	$a(\text{BIN}) \leftarrow b(\text{BCD})$	BCD \rightarrow BIN conversion
114	G114 Aa Bb Cc	$a = b \text{ OR } C$	OR
115	G115 Aa Bb Cc	$a = b \text{ XOR } C$	Exclusive OR
116	G116 Aa Bb Cc	$a = b \text{ AND } C$	Logical product



G-code	Input format	Mode of operation	Function
117	G117 Aa Bb	a = ROUND b	Zero point five and over are counted as a unit and the rest is disregarded.
118	G118 Aa Bb	a = FLX b	Neglect of decimal places
119	G119 Aa Bb	a = FUP b	Raise of decimal places
120	G120 Aa Bb	#a = b	Variable number designation (II)

* Input value

(a: # variables only (constants are usable for G120)
b, c: # variables or constants

Up to 0.0005 mm (0.00005 inch) of error may be included in the calculation in one block.



(ii) Branch instructions (G200 - G204)

A total of four branch instructions - one unconditional branch instruction and three conditional branch instructions - are available. Branch address sequence numbers are indirectly definable using # variables.

G-code	Input format	Mode of processing
200	G200 Aa	GOTO N No.a
201	G201 Aa Bb Cc	IF b = c GOTO N No.a
202	G202 Aa Bb Cc	IF b < c GOTO N No.a
203	G203 Aa Bb Cc	IF b ≤ c GOTO N No.a
204	G204 Aa Bb Cc	IF b > c GOTO N No.a
205	G205 Aa Bb	GOTO W No.a N No.b

* Input values

a: # variables or constants (positive values only)

b, c: # variables or constants (as to b for G205, positive value only)

G200 through 204 are branching instructions for the same work No., while G205 is for branching across work Nos.

Note) Branching with G205 is not treated as subprogram nesting.



(4) DEBUG of macro-programs

- (i) Correct results are not always obtainable in graphic check since system ones of the # variables that can be handled in macro-programs are closely related to the actions of the machine. It is therefore necessary to carry out the check of macro-programs in dry run for example.

- (ii) No single block stop is performed in macro-instruction blocks not dealt with as independent block by the numerical controller. If it is desired to know the en-route results at an intermediate point in macro-programs for DEBUG and so forth, set M01 (optional stop) therefor or take another effective measure.

**YAMAZAKI MACHINERY WORKS, LTD.**

1, Norifune, Oguchi-cho, Niwa-gun,
Aichi-pref., Japan 480-01
Phone: Oguchi (05879) 5-1131
Telex: J59777
Cable: "MAZAK" NAGOYA JAPAN
Facsimile: 05879-5-2717

MAZAK CORPORATION

8025 Production Drive, Florence,
Kentucky 41042
Phone: 606-727-5700
Telex: 214674
Facsimile: 606-727-5833/5834

BUFFALO SERVICE OFFICE

7661 Seneca Avenue
East Aurora, NY 14052
Phone: 716-655-4388
Facsimile: 716-655-4915

CHICAGO REGIONAL OFFICE

600 N. Wheeling Road
Mt. Prospect, Illinois 60056
Phone: 312-298-0707 885-8311
Facsimile: 312-298-4748

HARTFORD REGIONAL OFFICE

10 Bidwell Road
South Windsor, Connecticut 06074
Phone: 203-528-9511
Facsimile: 203-528-2995

ATLANTA REGIONAL OFFICE

1964 Providence Court
College Park, Georgia 30337
Phone: 404-996-1030
Facsimile: 404-996-3533

LOS ANGELES MAZAK TECHNICAL CENTER

1333 West 190th Street
Gardend, California 90248
Phone: 213-327-7172
Facsimile: 213-538-4087

HOUSTON REGIONAL OFFICE

770 Belt Way, 8
Houston, Texas 77060
Phone: 713-931-7770
Facsimile: 713-931-6191

SEATTLE REGIONAL OFFICE

7857 South 180
Bldg. No. 1
Kent, Washington 98031
Phone: 206-251-6015
Facsimile: 206-251-5419

TULSA REGIONAL OFFICE

11424 East 58th Street
Tulsa, Oklahoma 74145
Phone: 918-252-9533
Facsimile: 918-250-8298

**YAMAZAKI MAQUINAS HERRAMIENTAS,
S.A. DE C.V.**

Boulevard Toluca 22-E
Planta Baja
Naucalpan De Juarez
Estado de Mexico
MEXICO
Phone: 905-358-5340
Facsimile: 0015253585341

YAMAZAKI MACHINERY EUROPE N.V.

Research Park, Grauwmeer 7
3030 Leuven
BELGIUM
Phone: 016-230902
Telex: 4662552
Facsimile: 016-224587

YAMAZAKI MACHINERY DEUTSCHLAND GmbH

Esslingerstrasse 4,
7320 Göppingen Deutschland
Phone: 7161-68057/68058
Telex: 41727653
Facsimile: 7161-73613

YAMAZAKI MACHINERY DEUTSCHLAND GmbH

Niederlassung Deutschland
Nordendstrasse 46
6082 Mörfelden-Walldorf
Deutschland
Phone: 06105-71025/26
Telex: 4185773
Facsimile: 6105-71272

YAMAZAKI MACHINERY UK LTD.

Buckholt Drive
Warndon Industrial Estate
Warndon, Worcester
UNITED KINGDOM
Phone: 0905-57024
Telex: 338889
Facsimile: 0905-57644