GE Fanuc Automation

Computer Numerical Control Products

Series 16/18/160/180-PC

Connection Manual

GFZ-62773EN/01 *July 2002*

Warnings, Cautions, and Notes as Used in this Publication

Warning

Warning notices are used in this publication to emphasize that hazardous voltages, currents, temperatures, or other conditions that could cause personal injury exist in this equipment or may be associated with its use.

In situations where inattention could cause either personal injury or damage to equipment, a Warning notice is used.

Caution

Caution notices are used where equipment might be damaged if care is not taken.

Note

Notes merely call attention to information that is especially significant to understanding and operating the equipment.

This document is based on information available at the time of its publication. While efforts have been made to be accurate, the information contained herein does not purport to cover all details or variations in hardware or software, nor to provide for every possible contingency in connection with installation, operation, or maintenance. Features may be described herein which are not present in all hardware and software systems. GE Fanuc Automation assumes no obligation of notice to holders of this document with respect to changes subsequently made.

GE Fanuc Automation makes no representation or warranty, expressed, implied, or statutory with respect to, and assumes no responsibility for the accuracy, completeness, sufficiency, or usefulness of the information contained herein. No warranties of merchantability or fitness for purpose shall apply.

Export Control Information

This document and the information contained in this document are classified with an Export Control Classification Number of 2E201. It is restricted and controlled under US export regulations. If you intend to export (or reexport), the document, directly or indirectly, or technical information relating thereto supplied hereunder or any portion thereof, it is your responsibility to ensure compliance with U.S. export control regulations and, if appropriate, to secure any required export licenses in your own name.

Table of Contents

PREFACE

1. LIST OF FUNCTIONS 1

2. FUNCTION SPECIFICATIONS THAT DIFFER FROM THE M series 9

2.1 AXIS CONTROL 10
2.1.1 Name of Axes 10
2.1.2 Increment System 11
2.1.3 Specifying the Rotation Axis 11
2.1.4 Mirror Image 11
2.1.5 Follow–up 11
2.1.6 Rotary Axis Roll Over 12
2.2 PREPARATIONS FOR OPERATION 13
2.2.1 Stored Stroke Limit 1 13
2.3 REFERENCE POSITION ESTABLISHMENT 14
2.3.1 Manual Reference Position Return 14
2.3.2 Setting the Reference Position without Dogs 14
2.3.3 Reference Position Return 14
2.3.4 2nd to 4th Reference Position Return 14
2.4 AUTOMATIC OPERATION 16
2.4.1 Feed Hold 16
2.4.2 Machine Lock 16
2.4.3 Single Block 17
2.5 INTERPOLATION FUNCTION 18
2.5.1 Positioning 18
2.5.2 Linear Interpolation/Circular Interpolation 18
2.6 FEEDRATE CONTROL/ACCELERATION AND DECELERATION CONTROL 19
2.6.1 Rapid Traverse Rate 19
2.6.2 Changing the Rapid Traverse Rate, Time Constant, and Servo Loop Gain According to the Positioning DistanceConstant Positioning Time Control 19
2.6.3 Rapid Traverse Override 28
2.6.4 T–axis , C–axis Jog Override Signal 29
2.6.5 Look–Ahead Control 30
2.7 AUXILIARY FUNCTION 31
2.7.1 Distribution End Signal 31
2.7.2 2nd Auxiliary Function 31
2.7.3 Auxiliary Function Lock 31
2.8 SPINDLE SPEED FUNCTION 32
2.9 TOOL FUNCTION 33
2.9.1 Tool Offset Value/Tool Offset Number/Tool Offset Memory 33
2.9.2 Tool Life Management 33
2.10 DISPLAY/SET/EDIT 35
2.10.1 Waveform Diagnosis Display 35
2.10.2 Graphic Display 35
2.10.3 Multi–language Display 35
2.11 MEASUREMENT 36
2.11.1 Skip Function 36

2.12 PMC CONTROL FUNCTION 37
2.12.1 PMC Axis Control 37
2.13 PREPARATORY FUNCTION (G FUNCTION) 38

3. PRESSING FUNCTION 41

3.1 PUNCH FUNCTION (1-CYCLE PRESSING) 42
3.1.1 Block in which Punching is Made 42
3.2 POSITIONING & PRESSING OFF (G70) 44
3.3 NIBBLING FUNCTION 45
3.4 NIBBLING BY M FUNCTION 48
3.5 PRESS FUNCTION 50
3.5.1 1–Cycle Press 50
3.5.2 Continuous Press (Nibbling) 52
3.5.3 Manual Press 56
3.5.4 Press Start Lock Signal (Input) PFL <G230#0> 57
3.5.5 Press Start Assistance Signal (Output) DPF <F230#6> 58
3.5.6 Press Start Waiting Signal (Input) PFW <G230#1> and
Press Start Waiting Signal B (Input) PFWB <X1004#4> 58
3.5.7 Press Start Signal B (Output) PFB <Y1004#3> 59
3.5.8 Press Stop Signal Neglect (Input) EPE <G230#5> 59
3.5.9 Two–step Selection (Input) SNP for Nibbling <G230#6> 60
3.5.10 Press Start Auxiliary Signal B DSPF<F230#5> 60
3.6 HIGH SPEED PRESS CONTROL FUNCTION 61
3.7 EXTERNAL OPERATION FUNCTION EF, EFS, FIN 64

4. FUNCTIONS TO SIMPLIFY PROGRAMMING 65

4.1 AUTOMATIC REPOSITIONING (G75) 66
4.2 MULTI-PIECE MACHINING FUNCTION 68
4.2.1 Base Point Command of Multi-Piece Machining (G98) 68
4.2.2 Multi-Piece Machining Commands (G73, G74) 69
4.2.3 Setting of Machining Method for Multi-Piece Machining 69
4.3 Y–AXIS CRACK CANCEL 73

5. TOOL FUNCTION (T FUNCTION) 74

5.1 TOOL SELECTION FUNCTION 75
5.2 TOOL OFFSET 77
5.3 TURRET AXIS CONTROL (T AXIS CONTROL) 78
5.3.1 T Command Neglect Signal (Input) TNG <G233#5> 82
5.3.2 Tool Change Signal (Input) TCNG <G233#6> 82
5.3.3 Turret Indexing Completion Signal (Output) TIE <F236#6> 83
5.3.4 T Code Display Signal (Input) TI00 – TI32 <G234 – G237> 84
5.3.5 Number of Punches Signal (Output) PN00 – PN31 <F234 – F237> 85
5.3.6 T–axis Machine Zero Point Position Signals RP1T – RP16T <F244, F245> 85
5.4 MULTIPLE–TOOL CONTROL (OUTPUT) CMOK <F232#5>, MIE <F232#7> 87
5.5 T–CODE PRE–ISSUE FUNCTION 95
5.6 TOOL DATA SETTING FUNCTION 99

6. C–AXIS CONTROL 105

6.1 C AXIS CONTROL (DIE ANGLE INDEXING) 106

6.2 C–AXIS SYNCHRONIZATION CONTROL 107
6.3 C–AXIS OFFSET FUNCTION 112
6.3.1 C–axis Offset Type A 112
6.3.2 C–axis Offset Type B 114

7. SAFETY ZONE CHECK 121

7.1 TYPE A 122
7.2 TYPE B 123
7.3 SETTING THE SAFETY ZONE 124
7.4 SETTING THE TOOL SHAPE AREA 125
7.5 AUTOMATIC SAFETY–ZONE SETTING 126
7.5.1 Workpiece Holder Detection Command 126
7.5.2 Detecting Workpiece Holder Position Using an External Signal 128
7.5.3 Displaying the Safety Zones and Tool Zone 132
7.6 SIGNAL 133
7.7 PARAMETER 136
7.8 ALARM AND MESSAGE 142

8. WORKPIECE HOLDER INTERFERENCE AVOIDANCE FUNCTION 143

8.1 TYPE A 144
8.2 TYPE B 146
8.3 TOOL/WORKPIECE HOLDER AREAS 149
8.4 SIGNAL 150
8.5 PARAMETER 152
8.6 NOTE 153

9. CONTROL FUNCTION 154

9.1 FEED HOLD SIGNAL B/FEED HOLD LAMP SIGNAL B 155
9.2 RESET KEY SIGNAL 156
9.3 SWITCHING THE PUNCHING AND LASER MODES 157
9.4 2–ms INTERVAL ACCELERATION/DECELERATION FOR RAPID TRAVERSE 159
9.5 MULTIPLE–WORKPIECE MACHINING RETRACE FUNCTION 160

APPENDIX

A. CNC AND PMC INTERFACE 167

A.1 ADDRESS LIST 168
A.2 LIST OF SIGNALS 171
A.2.1 List of Signals in the Order of Functions 171
A.2.2 List of Signals in the Order of Symbols 173
A.2.3 List of Signals in the Order of Addresses 175

B. ALARM LIST 177

PREFACE

When designing a punch press, refer to the following connection manuals: “FANUC Series 16/18/160/180–MODEL C CONNECTION MANUAL (HARDWARE)” (B–62753EN), “FANUC Series 16/18/160/180–MODEL C CONNECTION MANUAL (FUNCTION)” (B–62753EN–1), and “FANUC Series 16/18/160/180–PC CONNECTION MANUAL (FUNCTION)” (B–62773EN, this manual). The “FANUC Series 16/18/160/180–MODEL C CONNECTION MANUAL (HARDWARE)” and “FANUC Series 16/18/160/180–MODEL C CONNECTION MANUAL (FUNCTION)” explain the general connections required to configure a machine tool. This manual provides supplementary information on connection related to the punch press function, that is not covered by the other two manuals.

Name of Manuals	Specification	
FANUC Series 16/18/160/180–MODEL C (HARDWARE) CONNECTION MANUAL	B–62753EN	
FANUC Series 16/18/160/180–MODEL C (FUNCTION) CONNECTION MANUAL	B–62753EN–1	
FANUC Series 16/18/160/180–PC (FUNCTION) CONNECTION MANUAL	B–62773EN	*

The following items are explained for each function.

1. **General**
   Describes feature of the function. Refer to Operator’s manual as required.
2. **Signals**
   Describes names, functions, output conditions and addresses of the signals required to realize a function.
3. **Parameters**
   Describes parameters related with a function.
4. **Alarms and messages**
   Lists the alarms and messages related with a function in a table.
5. **Reference item**
   List the related items of the related manuals in a table.

A list of addresses of all signals, a list of signals and a list of alarms are described in the appendix of this manual. Refer to it as required.

Applicable models

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

Product Name	Abbreviations	
FANUC Series 16–PC	16–PC	M series
FANUC Series 160–PC	160–PC	
FANUC Series 18–PC	18–PC	
FANUC Series 180–PC	180–PC	

Notation of bit type and bit axis type parameters

Data No.	Data (#0 to #7 indicates bit position)							
	#7	#6	#5	#4	#3	#2	#1	#0
0000			SEQ			INI	ISO	TVC

Notation of parameters other than bit type and bit axis type

Data No.	Data
1023	

Related Manuals

The table below lists manuals related to Series 16/18/160/180–PC. In the table, this manual is marked with an asterisk (*).

Table 1 Related Manuals

Manual name	Specification number	
FANUC Series 16/18/160/180–PC DESCRIPTIONS	B–62772EN	
FANUC Series 16/18/160/180–MODEL C CONNECTION MANUAL (HARDWARE)	B–62753EN	
FANUC Series 16/18/160/180–MODEL C CONNECTION MANUAL (FUNCTION)	B–62753EN–1	
FANUC Series 16/18/160/180–PC (FUNCTION) CONNECTION MANUAL	B–62773EN	*
FANUC Series 16/18/160/180–PC OPERATOR'S MANUAL	B–62774EN	
FANUC Series 16/18/160/180–MODEL C MAINTENANCE MANUAL	B–62755EN	
FANUC Series 16/18/160/180–MODEL C PARAMETER MANUAL	B–62760EN	
FANUC Series 16/18/160/180–PC PARAMETER MANUAL	B–62780EN	
FANUC Series 16/18/160/180 PROGRAMMING MANUAL (Macro Compiler/Macro Executer)	B–61803E–1	
FAPT MACRO COMPILER (For Personal Computer) PROGRAMMING MANUAL	B–66102E	

1 LIST OF FUNCTIONS

General

For details of connections that are not related to the punch press function, refer to the “FANUC Series 16/18/160/180–MODEL C CONNECTION MANUAL (FUNCTION)” (B–62753EN–1). Most of the functions described in B–62753EN–1 can be used with the FANUC Series 16/18/160/180–PC. The remainder either cannot be used with the FANUC Series 16/18/160/180–PC or have different specifications when used with the FANUC Series 16/18/160/180–PC. The table below indicates whether the functions described in B–62753EN–1 can be used with the FANUC Series 16/18/160/180–PC. For an explanation of the differences in the specifications, see Chapter 2.

List of Functions

The table below lists the functions.
The following symbols are used in the table:

- □ : The function can be used with the FANUC Series 16/18/160/180–PC.
- × : The function cannot be used with the FANUC Series 16/18/160/180–PC.
- Δ : The function has different specifications when used with the FANUC Series 16/18/160/180–PC.

Item			Using the function with the FANUC Series 16/18/160/180–PC
1	Controlled axes		□
	Setting each axis		
		Name of axes	Δ : See chapter 2
		Increment system	Δ : See chapter 2
		Specifying the rotation axis	Δ : See chapter 2
		Controlled axes detach	□
		Outputting the movement state of an axis	□
		Mirror image	Δ : See chapter 2
		Follow–up	Δ : See chapter 2
		Servo off (mechanical handle)	□
		Position switch	□
	Error compensation		
		Stored pitch error compensation	□
		Backlash compensation	□

	Item		Using the function with the FANUC Series 16/18/160/180–PC
1	Settings related to servo–controlled axes		
		Parameters related to servo	□
		Absolute position detection	□
	Settings related with coordinate systems		
		Machine coordinate system	□
		Workpiece coordinate system/ addition of workpiece coordinate system pair	Δ : “Addition of Workpiece coordinate System Pair” is unavailable.
		Rotary axis roll over	Δ : See chapter 2
	Simple synchronous control		□
	Tandem control		□
2	Emergency stop		□
	CNC ready signal		□
	Overtravel check		
		Overtravel signal	□
		Stored stroke limit 1	Δ : See chapter 2
		Stored stroke limit 2,3	Δ : Stored stroke limit 3 is unavailable.
		Chuck/tailstock barrier (T series)	×
		Tool post interference check (T series, two–path control)	×
	Alarm signal		□
	Start lock/interlock		□
	Mode selection		□
	Path selection/display of optional path names		×
	Status output signal		□
	VRDY off alarm ignore signal		□
3	Jog feed/incremental feed		□
	Manual handle feed		□
	Manual handle interruption		□
	Tool axis direction handle feed function (M system)		×

Item			Using the function with the FANUC Series 16/18/160/180–PC
4	Manual reference position return		Δ : See chapter 2
	Setting the reference position without dogs		Δ : See chapter 2
	Reference position shift		□
	Reference position return		Δ : See chapter 2
	2nd reference position return/ 3rd,4th reference position return		Δ : See chapter 2
	Floating reference position return		□
5	Cycle start/feed hold		Δ : See chapter 2
	Reset and rewind		□
	Testing a program		
		Machine lock	Δ : See chapter 2
		Dry run	□
		Single block	Δ : See chapter 2
	Manual absolute on/off		Δ : “Manual absolute off” is unavailable
	Optional block skip/ addition of optional block skip		□
	Sequence number comparison and stop		□
	Program restart		×
	Tool retraction and return (M series)		×
	Exact stop/exact stop mode/ tapping mode/cutting mode (M series)		Δ : Tapping mode is unavailable
	Balance cut (2–path control for T system)		×
	DNC operation		□
	Manual intervention and return		□

Item			Using the function with the FANUC Series 16/18/160/180–PC
6	Positioning		Δ : See chapter 2
	Linear interpolation		Δ : See chapter 2
	Circular interpolation		Δ : See chapter 2
	Thread cutting		×
	Single direction positioning (M series)		×
	Helical interpolation		□
	Involute interpolation (M series)		×
	Polar coordinate interpolation		×
	Cylindrical interpolation		×
	Polygonal turning (T series)		×
	Normal direction control (M series)		×
7	Feedrate control		
		Rapid traverse rate	Δ : See chapter 2
		Cutting feedrate clamp	□
		Feed per minute	□
		Feed per revolution/ manual feed per revolution	×
		F1–digit feed (M series)	×
		Feedrate inverse time specification (M series)	×
		Override	
		Rapid traverse override	Δ : See chapter 2
		Feedrate override	□
		Second feedrate override	□
		Override cancel	□
		Automatic corner override (M series)	□
		External deceleration	□
		Feed stop function	□
		Feedrate clamping by arc radius	□
		Automatic corner deceleration (M series)	□

Item				Using the function with the FANUC Series 16/18/160/180–PC
7			Look–ahead control	Δ : See chapter 2
			High–precision contour control by RISC (M series)	×
		Acceleration/deceleration control		
			Automatic acceleration/ deceleration	□
			Bell–shaped acceleration/ deceleration for rapid traverse	□
			Linear acceleration/deceleration after interpolation for cutting feed	□
			Bell shaped acceleration/ deceleration after interpolation for cutting feed	×
			Linear acceleration/deceleration before interpolation for cutting feed	□
			Corner control	
			In–position check	□
			In–position check independently of feed/rapid traverse	□
			Error detect (T series)	×
			Feed forward in rapid traverse	□
8	Miscellaneous function/ 2nd auxiliary function			Δ : See chapter 2
	Auxiliary function lock			Δ : See chapter 2
	Multiple M commands in a single block			□
	High–speed M/S/T/B interface			□
	Waiting M code (two–path control)			×
	M code loop check function			×
9	Spindle speed function			Δ : See chapter 2
10	Tool function			□
	Tool compensation value/ tool compensation number/ tool compensation memory			Δ : See chapter 2
	Tool life management			Δ : See chapter 2

Item			Using the function with the FANUC Series 16/18/160/180–PC
10	Cutter compensation		
		Cutter Compensation B, C (M series)	Δ : Compensation B is unavailable
		Tool nose radius compensation (T series)	×
11	Decimal point programming/ pocket calculator type decimal point programming		□
	G code system (T series)		Δ : See chapter 2
	Program configuration		□
	Inch/metric conversion		□
	High speed cycle cutting		×
	Custom macro		
		Custom macro	□
		Interruption type custom macro	□
		Custom macro variables common to two–path control (T series (two–path control))	×
	Canned cycle (M series)/canned cycle for hole machining (T series)		×
	External motion function (M series)		Δ : See chapter 3
	canned cycle (T series)/multiple repetitive canned cycle (T series)		×
	Mirror image for double turrets (T series)		×
	Index table indexing function (M series)		×
	Scaling (M series)		□
	Coordinate system rotation		□
	Three–dimensional coordinate conversion (M series)		×
	Retrace (M series)		□
	Macro compiler/executer		□

Item			Using the function with the FANUC Series 16/18/160/180–PC
12	Display/set		
		Clock function	□
		Displaying operation history	□
		Help function	□
		Displaying alarm history	□
		Servo tuning screen	□
		Spindle tuning screen	×
		Waveform diagnosis display	Δ : See chapter 2
		Self–diagnosis	□
		Display of hardware and software configuration	□
		Position display neglect	□
		Run hour and parts count display	□
		Graphic display/ dynamic graphic display	Δ : See chapter 2
		Displaying operating monitor	□
		Stamping the machining time	□
		Software operator’s panel	□
		Multi–language display	Δ : See chapter 2
		Remote diagnosis	□
	Edit		
		Part program storage length	□
		No. of registered programs	□
		Memory protection key	□
		Password function	□
		Background editing	□
		Playback	×
		Conversational programming with graphic function	□

Item			Using the function with the FANUC Series 16/18/160/180–PC
13	Reader/puncher interface		□
	Remote buffer		□
	DNC1 interface		□
	DNC2 interface		□
	External I/O device control		□
	Simultaneous input and output operations (M series)		□
	External program input		□
14	Tool length measurement (M series)		×
	Automatic tool length measurement (M series)/automatic tool offset (T series)		×
	Skip function		
		Skip function	Δ : See chapter 2
		High–speed skip signal	×
		Multi–step skip (T series)	×
		Torque limit skip (T series)	×
	Entering compensation values		×
15	PMC axis control/PMC axis speed control function		Δ : See chapter 2
	External data input		□
	External workpiece number search		□
	Spindle output control by the PMC		□
	External key input		□
	Direct operation by PMC/MMC		□

2 FUNCTION SPECIFICATIONS THAT DIFFER FROM THE M series

Some of the functions described in the “FANUC Series 16/18/160/180–MODEL C CONNECTION MANUAL (FUNCTION)” (B–62753EN–1) have different specifications when used with the FANUC Series 16/18/160/180–PC. This chapter describes these differences.

2.1 AXIS CONTROL

2.1.1 Name of Axes

Difference

Axis names can be selected from X, Y, Z, A, B, C, U, V, W, and T. X and Y, however, are automatically selected and always assigned to the basic axes.

Parameter

1020	Name of the axis used for programming for each axis

[Data type] Byte axis

Set the name of the program axis for each control axis, with one of the values listed in the following table:

Axis name	Set value	Axis name	Set value	Axis name	Set value	Axis name	Set value
X	88	U	85	A	65	T	84
Y	89	V	86	B	66		
Z	90	W	87	C	67		

Note 1 If the system supports the function for machining multiple workpieces, addresses U, V, and W cannot be used for the names of the axes controlled by the CNC.

Note 2 If macro functions A, B, U, V, and W for a punch press are used, addresses A, B, U, V, and W cannot be used for the names of the axes controlled by the CNC.

Note 3 When the secondary auxiliary function is provided, address B cannot be used as an axis name.

Note 4 If the C–axis synchronous control function is supported, the C_2–axis is automatically assigned the number immediately subsequent to the axis number of the C_1–axis. Subscripts are not added. (The two axes have the same axis name, C.)

Example: When the fourth axis is C_1, the fifth axis automatically becomes C_2.

	#7	#6	#5	#4	#3	#2	#1	#0
16200	UVW	ABM						

[Data type] Bit

ABM To store and call a pattern, addresses A and B:

0 : Are used.
1 : Are not used. (The A and B axes can be used.)

UVW To execute a macro function, addresses U, V, and W:

0 : Are used.
1 : Are not used. (The U, V, and W axes can be used.)

2.1.2 Increment System

Difference

Two increment systems, IS–A and IS–B, are supported.

2.1.3 Specifying the Rotation Axis

Difference

When the T–axis or C–axis control function is used, the T–axis or C–axis automatically becomes the rotation axis.

2.1.4 Mirror Image

Difference

The following are not inverted:

- Direction of manual operation
- Motion toward the reference position in an automatic reference position return
- Repositioning
- Motion of tool position compensation and C–axis position compensation

CAUTION
If the value of the T–axis mirror image is set to 1, the amount of travel is inverted, resulting in incorrect turret indexing. (Never specify this setting.)

2.1.5 Follow–up

CAUTION
When the T–axis control function is used, the servo–off signal is generally used for a shot pin after T–axis positioning. If the *FLWU signal is set to 0, the follow–up function operates and sets the T–axis machine position signal and turret indexing completion signal to 0. If this occurs, turret indexing by a subsequent T command will result in incorrect positioning.
When turning the servo–off signal on or off during T–axis control or automatic operation, set the *FLWU signal to 1 in advance.

2.1.6
Rotary Axis Roll Over

Difference

The rotary axis roll over function cannot be used together with T–axis or C–axis control.

2.2 PREPARATIONS FOR OPERATION

2.2.1 Stored Stroke Limit 1

Difference

If the end point specified for rapid traverse positioning that constitutes part of an automatic operation falls outside a predetermined range, no axial movements are made. Instead, an alarm is output. (Stroke check before travel)

Alarm and message

Number	Message	Contents
4700	PROGRAM ERROR (OT+)	The value specified in the X–axis move command exceeded the positive value of stored stroke limit 1. (Advance check)
4701	PROGRAM ERROR (OT–)	The value specified in the X–axis move command exceeded the negative value of stored stroke limit 1. (Advance check)
4702	PROGRAM ERROR (OT+)	The value specified in the Y–axis move command exceeded the positive value of stored stroke limit 1. (Advance check)
4703	PROGRAM ERROR (OT–)	The value specified in the Y–axis move command exceeded the negative value of stored stroke limit 1. (Advance check)

2.3 REFERENCE POSITION ESTABLISHMENT

2.3.1 Manual Reference Position Return

Difference Parameter No.1240 cannot be used.

2.3.2 Setting the Reference Position without Dogs

Difference Parameter No.1240 cannot be used.

2.3.3 Reference Position Return

Difference Executing the G28 command causes reference position returns for all axes.
Parameter No.1240 cannot be used.

2.3.4 2nd to 4th Reference Position Return

Difference The signal posts notification that the tool is at the second, third, or fourth reference position.

(output condition) The signal is set to 1 when:

- The tool will be at the position specified in parameters 1241 to 1243 after the reference position has been established by a reference position return.

Parameter

16600	Width for the second reference position on each axis
16601	Width for the third reference position on each axis
16602	Width for the fourth reference position on each axis

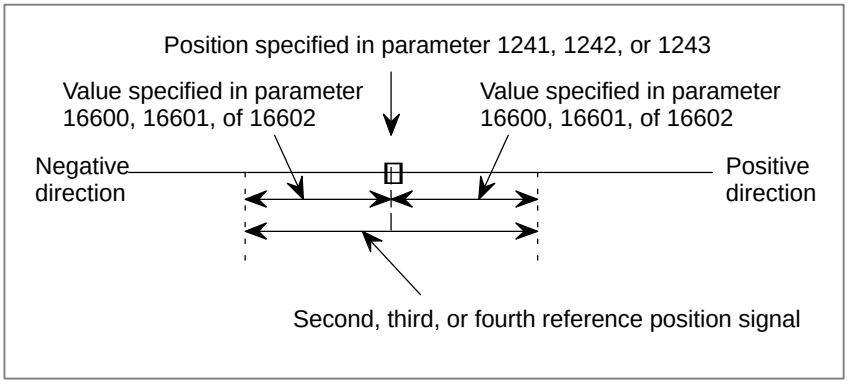
[Data type] Word

[Unit of data]

Increment system	IS–A	IS–B	Unit
Millimeter machine	0.01	0.001	mm
Inch machine	0.001	0.0001	inch

[Valid data range] 0 to 65535

The parameters specify the width for the second, third, or fourth reference position of the machine coordinate system. The second, third, or fourth reference position signal is output within the range shown below:



Note 1 For a rotation axis, the specified reference position output range must not include 0 of the machine coordinate system.

2.4 AUTOMATIC OPERATION

2.4.1 Feed Hold

Difference

(a) Nibbling mode

When the *SP signal is set to 0 during positioning to the first punch point in nibbling mode, positioning stops. This sets the STL signal to 0 and the SPL signal to 1, such that the system enters the feed hold state.

Also, when the *SP signal is set to 0 during pitch movement from the first punch point to the last punch point, the system enters the feed hold state. The system can enter the feed hold state after a pitch movement which sets the *SP signal to 0 provided the NSP bit (bit 2 of parameter 16181) is set accordingly. If this setting is made, press start signal PF and nibbling signal NBL are set to 0 when press stop signal *PE is set to 0.

(b) Workpiece holder escape mode

If the *SP signal is set to 0 in escape mode, the system stops after the escape is completed.

2.4.2 Machine Lock

Difference

Even in the machine lock state, the press function and external operation function can be executed. So, miscellaneous function lock signal AFL, T–command ignore signal TNG, and press start lock signal PFL should all be set to 1 and external operation function selection signal EFS to 0 in the machine lock state.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
16001						PRC		

[Data type] Bit

PRC When the machine lock signal, MLK, is set to 1, a program check is:

0 : Not executed.
1 : Executed.
The machine position data is updated although the actual position is not changed. This setting is invalid for the machine lock signal of each axis.

2.4.3 Single Block

Difference

If the SBK signal is set to 1 while a pattern such as a bolt hole circle (G26) is being specified, the operation does not stop at the end of each program block. It stops only after the tool has been positioned to each punch point and punching completed. In this case, feed hold signal SPL is set to 1 when the cycle start lamp signal STL is set to 0, posting notification that it is not the end of a single block of the program. Once the operation of one block has been completed, both the STL and SPL signals are set to 0 and the operation stops.
If the SBK signal is set to 1 during nibbling or automatic repositioning, a stop is made upon the completion of a series of actions.

2.5 INTERPOLATION FUNCTION

2.5.1 Positioning

Difference

In G00 mode, punching is executed after the completion of axial movements.
Generally, the F command cannot be specified in G00 mode. The F command can be specified only when the G0F bit (bit 0 of parameter 16050) is set accordingly.
For T– or C–axis command blocks, nonlinear interpolation positioning is performed, even if linear interpolation positioning is specified.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
16050								G0F

[Data type] Bit

G0F For a rapid traverse command (G00), the X–axis or Y–axis rapid traverse feedrate is set to the value:

0 : Specified in the parameter.
1 : Specified by the F code.

Note 1 If no F command is detected, alarm 011 occurs.

Note 2 If the speed specified with the F command exceeds that set in parameter 1420, the specified speed is reduced to that specified in the parameter.

Note 3 In nibbling mode, pitch movement is performed at the speed specified in parameter 1420.

Note 4 The programmable rapid traverse override function cannot be used.

2.5.2 Linear Interpolation/ Circular Interpolation

Difference

The T or C command cannot be specified in G01, G02, or G03 mode.

Alarm and message

Number	Message	Contents
4600	T, C COMMAND IN INTERPOLATION	In the linear interpolation (G01) mode or circular interpolation (G02, G03) mode, a T command or C–axis command was specified.

2.6 FEEDRATE CONTROL/ ACCELERATION AND DECELERATION CONTROL

2.6.1 Rapid Traverse Rate

Difference

Punching starts once positioning has been completed.
A rapid traverse rate is overridden by the following values when a switch is pressed on the machine operator's panel:
25%, 50%, 75%, 100%
The LRP bit (bit 1 of parameter 1401) cannot be used.

2.6.2 Changing the Rapid Traverse Rate, Time Constant, and Servo Loop Gain According to the Positioning Distance Constant Positioning Time Control

General

In automatic rapid traverse, the rapid traverse rate, time constant, and servo loop gain can be varied according to the positioning distances for individual axes, as specified in the parameters. By using this function, positioning accuracy can be improved.

X–axis, Y–axis:
- The rapid traverse rate, time constant, and servo loop gain can be varied according to the positioning distance. (Up to seven levels)
- The servo loop gain for rapid traverse and cutting feed can be changed.

T–axis:
- The rapid traverse time constant and servo loop gain can be varied according to the positioning angle. (Up to three levels)

C–axis:
- The rapid traverse rate and time constant can be varied according to the positioning angle. (Up to three levels)

In automatic rapid traverse, positioning for the X– and Y–axes can be executed in a specified period, independently of the positioning distance. If this function is used in nibbling mode, positioning can always be completed within a specified period, irrespective of the length of the nibbling pitch. This enables smooth punching. (Two levels)

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
16050	KLV	PCT	CT2	KLC	NCT			

[Data type] Bit

NCT Constant control of positioning time is:

0 : Always enabled.
1 : Enabled only when the nibbling command is executed.
This parameter is valid when the PCT bit (bit 6 of parameter 16050) is set to 1.

KLC When rapid traverse is executed in automatic operation, the function to change the time constant and C–axis rapid traverse feedrate among three levels according to the positioning angles:

0 : Invalidated.
1 : Validated. See the descriptions of parameters 16040 to 16147.

CT2 In constant control of the positioning time, the times specified in parameters 16095 to 16102 are:

0 : Not changed.
1 : Doubled.

PCT Constant control of positioning time is:

0 : Invalidated.
1 : Validated. The parameter is validated when parameter KLV (No.16050, #7) is set to 1.

KLV When rapid traverse is executed in automatic operation, the function to change the time constant and X–axis and Y–axis rapid traverse feedrates among seven levels according to the positioning distance is:

0 : Invalidated.
1 : Validated. See the descriptions of data 16055 to 16094.

	#7	#6	#5	#4	#3	#2	#1	#0
16051	PGC			LPG	KLT			

[Data type] Bit

KLT When rapid traverse is executed in automatic operation, the function to change the servo loop gain of position control and time constant of T–axis rapid traverse among three levels according to the indexed angle is:

0 : Invalidated.
1 : Validated. See the descriptions of parameters 16177 to 16124.

LPG When rapid traverse is executed in automatic operation, the function to change the servo loop gain of X–axis and Y–axis position control among seven levels according to the positioning distance is:

0 : Invalidated.
1 : Validated. The parameter is validated when parameter KLV (No.16050, #7) is set to 1.

PGC Servo loop gains of X–axis and Y–axis position control to be used in rapid traverse and cutting feed:

0 : Are the same.
1 : Can be set separately.

16055	Distance D1 to level 1 (in mm)
16056	Distance D2 to level 2 (in mm)
16057	Distance D3 to level 3 (in mm)
16058	Distance D4 to level 4 (in mm)
16059	Distance D5 to level 5 (in mm)
16060	Distance D6 to level 6 (in mm)

16061	Distance D1 to level 1 (in inches)
16062	Distance D2 to level 2 (in inches)
16063	Distance D3 to level 3 (in inches)
16064	Distance D4 to level 4 (in inches)
16065	Distance D5 to level 5 (in inches)
16066	Distance D6 to level 6 (in inches)

[Data type] Two–word

[Unit of data]

Increment system	IS–A	IS–B	Unit
input in millimeters	0.01	0.001	mm
input in inches	0.001	0.0001	inch

[Valid data range] 0 to 99999999

Each of the parameters set the positioning distance to use the function to change the time constant and X–axis and Y–axis rapid traverse feedrate among seven levels according to the positioning distance. (Identical values are set for the X and Y axes.)
The data is validated when parameter KLV (No.16050, #7) is set to 1.

Note 1 The values set here must satisfy the following relationship:
D1 < D2 < D3 < D4 < D5 < D6.

Note 2 The values can be changed among seven levels or less. When the values are to be changed among four levels, set D4 to 99999999.

16067	X–axis rapid traverse feedrate of level 1
16068	X–axis rapid traverse feedrate of level 2
16069	X–axis rapid traverse feedrate of level 3
16070	X–axis rapid traverse feedrate of level 4
16071	X–axis rapid traverse feedrate of level 5
16072	X–axis rapid traverse feedrate of level 6
16073	X–axis rapid traverse feedrate of level 7

[Data type] Two–word

[Unit of data]
[Valid data range]

Increment system	Unit of data	Valid data range
Millimeter machine	1 mm/min	30 to 240000
Inch machine	0.1 inch/min	30 to 96000

Each of the parameters set the X–axis rapid traverse rate for the corresponding distance.

16074	X–axis rapid traverse time constant of level 1
16075	X–axis rapid traverse time constant of level 2
16076	X–axis rapid traverse time constant of level 3
16077	X–axis rapid traverse time constant of level 4
16078	X–axis rapid traverse time constant of level 5
16079	X–axis rapid traverse time constant of level 6
16080	X–axis rapid traverse time constant of level 7

[Data type] Word

[Unit of data] msec

[Valid data range] 8 to 4000

Each of the parameters set the X–axis rapid traverse time constant for the corresponding positioning distance.

16081	Y–axis rapid traverse rate of level 1
16082	Y–axis rapid traverse rate of level 2
16083	Y–axis rapid traverse rate of level 3
16084	Y–axis rapid traverse rate of level 4
16085	Y–axis rapid traverse rate of level 5
16086	Y–axis rapid traverse rate of level 6
16087	Y–axis rapid traverse rate of level 7

[Data type] Two–word

[Unit of data]
[Valid data range]

Increment system	Unit of data	Valid data range
Millimeter machine	1 mm/min	30 to 240000
Inch machine	0.1 inch/min	30 to 96000

Each of the parameters set the Y–axis rapid traverse rate for the corresponding distance.

16088	Y–axis rapid traverse time constant of level 1
16089	Y–axis rapid traverse time constant of level 2
16090	Y–axis rapid traverse time constant of level 3
16091	Y–axis rapid traverse time constant of level 4
16092	Y–axis rapid traverse time constant of level 5
16093	Y–axis rapid traverse time constant of level 6
16094	Y–axis rapid traverse time constant of level 7

[Data type] Word

[Unit of data] msec

[Valid data range] 8 to 4000

Each of the parameters set the Y–axis rapid traverse time constant for the corresponding positioning distance.

Relationship between positioning distances and data numbers

Level	Positioning distance d	X–axis		Y–axis	
		Rapid traverse feedrate	Rapid traverse time constant	Rapid traverse feedrate	Rapid traverse time constant
1	0□d□D1	16067	16074	16081	16088
2	D1□d□D2	16068	16075	16082	16089
3	D2□d□D3	16069	16076	16083	16090
4	D3□d□D4	16070	16077	16084	16091
5	D4□d□D5	16071	16078	16085	16092
6	D5□d□D6	16072	16079	16086	16093
7	D6□d	16073	16080	16087	16094

16095	X–axis positioning time of level 1 (Rapid traverse override of 100% or 75%)
16096	X–axis positioning time of level 1 (Rapid traverse override of 50% or 25%)
16097	X–axis positioning time of level 2 (Rapid traverse override of 100% or 75%)
16098	X–axis positioning time of level 2 (Rapid traverse override of 50% or 25%)

16099	Y–axis positioning time of level 1 (Rapid traverse override of 100% or 75%)
16100	Y–axis positioning time of level 1 (Rapid traverse override of 50% or 25%)
16101	Y–axis positioning time of level 2 (Rapid traverse override of 100% or 75%)
16102	Y–axis positioning time of level 2 (Rapid traverse override of 50% or 25%)

[Data type] Byte

[Unit of data] msec

[Valid data range] 32 to 248

When constant control of the positioning time is applied, each of the parameters set the X–axis or Y–axis positioning time for the positioning distance of level one or two.
The parameters are validated when parameter KLV (No.16050, #7) and PCT (No.16050, #6) are set to 1.

Note 1 When this function is used, parameters 16067, 16068, 16074, 16075, 16081, 16082, 16088, and 16089 are invalidated. Constant control of the positioning time is applied, irrespective of the positioning distance.

16103	X–axis servo loop gain of level 1
16104	X–axis servo loop gain of level 2
16105	X–axis servo loop gain of level 3
16106	X–axis servo loop gain of level 4
16107	X–axis servo loop gain of level 5
16108	X–axis servo loop gain of level 6
16109	X–axis servo loop gain of level 7

[Data type] Word

[Unit of data] 0.01 sec^{-1}

[Valid data range] 1 to 9999

Each of the parameters set the servo loop gain of X–axis position control for the corresponding positioning distance.

16110	Y–axis servo loop gain of level 1
16111	Y–axis servo loop gain of level 2
16112	Y–axis servo loop gain of level 3
16113	Y–axis servo loop gain of level 4
16114	Y–axis servo loop gain of level 5
16115	Y–axis servo loop gain of level 6
16116	Y–axis servo loop gain of level 7

[Data type] Word

[Unit of data] 0.01 sec^{-1}

[Valid data range] 1 to 9999

Each of the parameters set the servo loop gain of Y–axis position control for the corresponding positioning distance.

16117	T–axis angle to level 1
16118	T–axis angle to level 2

[Data type] Word

[Unit of data] 0.1 deg

[Valid data range] 0 to 3600

Each of the parameters set the indexed angle to use the function for changing the T–axis rapid traverse time constant and servo loop gain of position control among three levels according to the indexed angle.
The parameters are validated when parameter KLT (No.16051, #3) is set to 1

Note 1 The value of level one must be smaller than the value of level 2.

16119	T–axis rapid traverse time constant of level 1
16120	T–axis rapid traverse time constant of level 2
16121	T–axis rapid traverse time constant of level 3

[Data type] Word

[Unit of data] msec

[Valid data range] 8 to 4000

Each of the parameters set the T–axis rapid traverse time constant to use the function for changing the T–axis rapid traverse time constant and servo loop gain of position control among three levels according to the indexed angle.

16122	T–axis servo loop gain of level 1
16123	T–axis servo loop gain of level 2
16124	T–axis servo loop gain of level 3

[Data type] Word

[Unit of data] 0.01 sec^{-1}

[Valid data range] 1 to 9999

Each of the parameters set the servo loop gain of T–axis position control to use the function for changing the T–axis rapid traverse time constant and servo loop gain of position control among three levels according to the indexed angle.

16140	C–axis angle to level 1
16141	C–axis angle to level 2

[Data type] Two–word

[Unit of data] 0.01 deg (IS–A)/0.001 deg (IS–B)

[Valid data range] 0 to 99999999

Each of the parameters set the positioning angle to use the function for changing the C–axis rapid traverse feedrate and time constant among three levels according to the positioning angle.
The data is validated when parameter KLC (No.16050, #4) is set to 1.

Note 1 The value of level 1 must be smaller than the value of level 2.

16142	C–axis rapid traverse rate of level 1
16143	C–axis rapid traverse rate of level 2
16144	C–axis rapid traverse rate of level 3

[Data type] Two–word

[Unit of data] 1 deg/min

[Valid data range] 30 to 240000

Each of the parameters set the C–axis rapid traverse rate to use the function for changing the C–axis rapid traverse rate and rapid traverse time constant among three levels according to the positioning angle.

16145	C–axis rapid traverse time constant of level 1
16146	C–axis rapid traverse time constant of level 2
16147	C–axis rapid traverse time constant of level 3

[Data type] Word

[Unit of data] msec

[Valid data range] 8 to 4000

Each of the parameters set the C–axis rapid traverse to use the function for changing the C–axis rapid traverse rate and rapid traverse time constant among three levels according to the positioning angle.

16160	Servo loop gain in cutting feed

[Data type] Word axis

[Unit of data] 0.01 sec^{-1}

[Valid data range] 1 to 9999

For each axis, the parameter sets the servo loop gain of position control in cutting feed.
The parameter is validated when parameter PGC (No.16051, #7) is set to 1.

Note 1 The parameter can only be set for the X and Y axes.

2.6.3 Rapid Traverse Override

Difference

In automatic rapid traverse, the rapid traverse rate can be overridden by the value determined by the ROV1 and ROV2 signals.
If the function for varying the speed and time constant according to the positioning distance is used (the KLV bit, bit 7 of parameter 16050, is set to 1), linear acceleration/deceleration is executed according to the overridden rapid traverse rate of the corresponding level and the specified time constant.
For positioning under constant positioning time control (the PCT bit, bit 6 of parameter 16050, is set to 1), rapid traverse override is disabled and is always set to 100%.

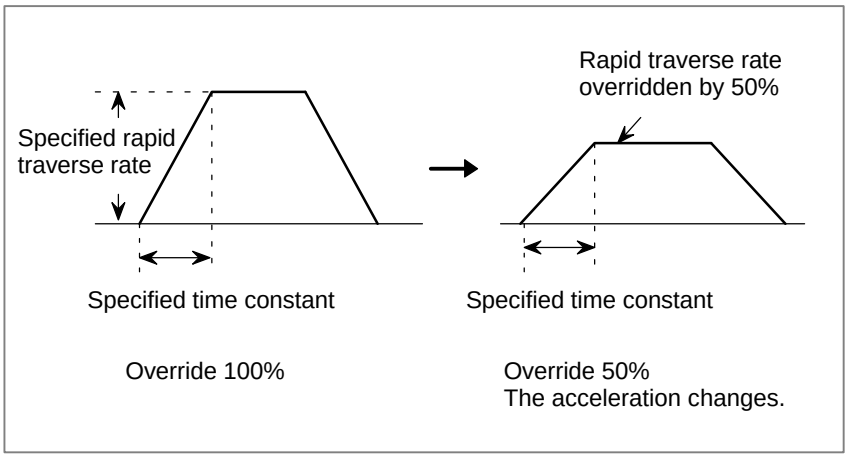


Fig. 2.6.3 (a) Rapid Traverse Override for the X– and Y–Axes

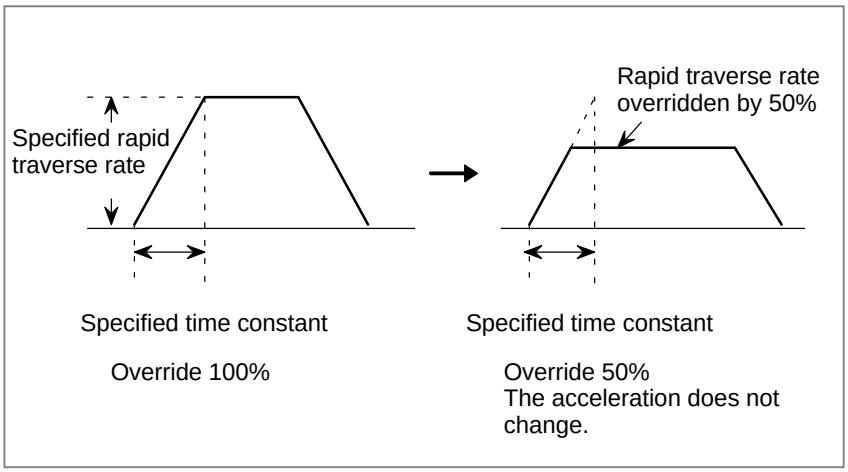


Fig. 2.6.3 (b) Rapid Traverse Override for an Axis Other than the X– and Y–Axes

The rapid traverse override depends on the states of the ROV1 and ROV2 signals when information relating to a block is read and stored into the buffer. Any change in the state of the ROV1 or ROV2 signal does not affect a block, if the change is made while the block is being executed. Also, the change does not affect the subsequent block if the block is stored into the buffer prior to the change.
If a specification is made to set press start signal PF to 1 before the completion of positioning, the PF signal is set to 1 before the end of positioning only when the rapid traverse override is 100%. For other than a non–100% rapid traverse override, the PF signal is set to 1 upon the completion of positioning.
In nibbling mode, rapid traverse override is valid for positioning to the first punch point. For positioning to subsequent nibbling pitches, the rapid traverse override becomes invalid and is always set to 100%.

ROV1	ROV2	X, Y axes	T, C axes
0	0	100%	100%
0	1	75%	100%
1	0	50%	50%
1	1	25%	50%

2.6.4 T–axis , C–axis Jog Override Signal

General

The T–axis and C–axis jog override can be set by input signals that differ from the conventional input signals, G010 and G011 (*JV0 to *JV15).

*JVT1, *JVT2 <G233#0, #1>

[Classification] Input signal

[Function] Selects the T–axis and C–axis jog feedrate.

*JVT1	*JVT2	Override value of T and C axis
1	1	25%
1	0	50%
0	1	75%
0	0	100%

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G233							*JVT2	*JVT1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
16052			TJG					

[Data type] Bit

TJG The jog override signals for the T–axis and C–axis (G233, #0 and #1) are:

0 : Not used.
1 : Used.

2.6.5 Look–Ahead Control

WARNING

This function cannot be used when high–speed press control (HSP bit, bit 0 of parameter 16000) is applied.

2.7 AUXILIARY FUNCTION

2.7.1 Distribution End Signal

Difference

The DEN signal is set to 1 also in the following states:

- Punch completion wait state (*PFIN or *NFIN) for the press function once all axial movements have been completed
- Completion wait state (FIN) for the external operation function once all axial movements have been completed

The DEN signal can also be used for the external operation function. If a specification is made to set the press start signal (PF) to 1 before the completion of positioning, the external operation function signal (EF) is also set to 1 before the completion of positioning. The DEN signal should be used as a gate signal for starting operation after movement when the external operation function is used.

2.7.2 2nd Auxiliary Function

NOTE
When the 2nd auxiliary function is provided, the A/B macro function cannot be used to store and call a specified pattern.

2.7.3 Auxiliary Function Lock

Difference

The internal processing for the following M codes is executed, independently of the AFL signal:

- Forming mode, forming mode cancel
- Nibbling mode, nibbling mode cancel
- Workpiece clamp, workpiece unclamp
- Switching between punch mode and laser mode

These M codes can be output, even when the AFL signal is set to 1, by setting the PMA bit accordingly (bit 5 of parameter 16001).

2.8
SPINDLE SPEED FUNCTION

Difference

S–code output and analog voltage control by the PMC are possible. The other spindle control functions cannot be used.

2.9 TOOL FUNCTION

2.9.1 Tool Offset Value/ Tool Offset Number/ Tool Offset Memory

Difference

- Setting Range of Tool Offset Value

Increment system	Metric input	Inch input
IS - A	□9999. 99 mm	□999. 999 inch
IS - B	□999. 999 mm	□99. 9999 inch

- Tool Compensation Number
  32, 64, 99, 200 or 400
- Tool Offset Memory
  Cutter compensation memory only is available.

2.9.2 Tool Life Management

General

The function sets the maximum punch count for each tool on the CRT screen. If the actual punch count for a selected tool exceeds the predetermined maximum punch count, tool expired signal PTLCH is output. Upon detecting this signal, the PMC outputs an alarm or instructs the operator to change the tool.
The actual punch count and maximum punch count can be displayed and set on the <OFFSET/SETTING> screen by pressing the [TOOL] and [TOOLLIFE] soft keys.

Signal

Tool expired signal PTLCH<F232#0>

[Classification] Output signal

[Function] Posts notification that the tool has reached the end of its service life.

[Output condition] The signal is set to 1 in the following cases:

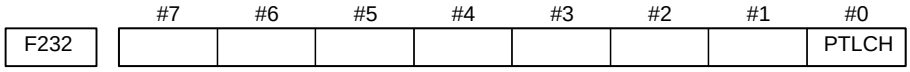
- When the T command is specified when the actual punch count of the corresponding tool has exceeded the predetermined maximum punch count, that is, when the tool has reached the end of its service life.
- When the actual punch count for the corresponding tool exceeds the predetermined maximum punch count, that is, when the tool reaches the end of its service life, during punching after the T command has been specified.

The signal is set to 0 in the following cases:

- When the CNC enters the reset state.
- When the actual punch count is preset, or a value less than the maximum punch count is entered.
- When a tool which has not yet reached the end of its service life is selected.

Note 1 This signal is not output if the maximum punch count is set to 0.

Signal address



2.10 DISPLAY/SET/EDIT

2.10.1 Waveform Diagnosis Display

NOTE
The display will act abnormally when the high–speed press control function is enabled (HSP bit, bit 0 of parameter 16000).

2.10.2 Graphic Display

Difference

There are no system parameters related to the graphic display.
No dynamic graphic display is provided.

2.10.3 Multi–language Display

Difference

The display can be set to English, Japanese, German, French, Italian, or Spanish output.

2.11
MEASUREMENT

2.11.1
Skip Function

Difference

G33 is used to specify the function. The address of the input signal is SKIP <X1004, #0>.

2.12 PMC CONTROL FUNCTION

2.12.1 PMC Axis Control

CAUTION
This function cannot be used when the high–speed press control function is enabled (HSP bit, bit 0 of parameter 16000).

2.13 PREPARATORY FUNCTION (G FUNCTION)

A number following address G determines the meaning of the command for the concerned block.
G codes are divided into the following two types.

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

(Example)
G01 and G00 are modal G codes in group 01.

```
G01X–▯
   Y▯   } G01 is effective in this range.
   X▯
G00Y–▯
```

Explanations

1. When the clear state (bit 6 (CLR) of parameter No. 3402) is set at power–up or reset, the modal G codes are placed in the states described below.

(1) The modal G codes are placed in the states marked with ◤ as indicated in Table 3.
(2) G20 and G21 remain unchanged when the clear state is set at power–up or reset.
(3) For G22 and G23, G22 is set at power–up. However, G22 and G23 remain unchanged when the clear state is set at reset.
(4) The user can select G00 or G01 by setting bit 0 (G01) of parameter No. 3402.
(5) The user can select G90 or G91 by setting bit 3 (G91) of parameter No. 3402.
(6) The user can select G17, G18, or G19 by setting bit 1 (G18) and bit 1 (G19) of parameter No. 3402.

2. G codes of group 00 other than G10 and G11 are one–shot G codes.
3. When a G code not listed in the G code list is specified, or a G code that has no corresponding option is specified, alarm No. 010 is output.
4. Multiple G codes can be specified in the same block if each G code belongs to a different group. If multiple G codes that belong to the same group are specified in the same block, only the last G code specified is valid.
5. G codes are indicated by group.
6. The G code system is set by parameter GSB (No.3401#6).

Table 2.13 G code list (1/2)

System A	System B	Group	Meaning
G00	G00	01	Positioning (Rapid traverse)
G01	G01		Linear interpolation (Cutting feed)
G02	G02		Circular interpolation (CW)
G03	G03		Circular interpolation (CCW)
G04	G04	00	Dwell
G08	G08		Look–ahead control
G09	G09		Exact stop
G10	G10		Data setting
G11	G11		Data setting mode cancel
G20	G20	06	Input in inch
G21	G21		Input in mm
G22	G22	04	Stored stroke limit function on
G23	G23		Stored stroke limit function off
G26	G26	00	Bolt hole circle
G28	G50		Automatic reference point return
G30.1	G30.1		Floating reference point return
G32	G32		Automatic safety zone setting
G33	G33		Skip function
G38	G38		Bending compensation X
G39	G39		Bending compensation Y
G40	G40	07	Cutter compensation cancel
G41	G41		Cutter compensation left
G42	G42		Cutter compensation right
G45	G45	00	Linear punching
G46	G46		Circular punching (CW)
G47	G47		Circular punching (CCW)
G50	G34	11	Scaling on
G51	G35		Scaling off
G52	G93	00	Local coordinate system setting
G53	G53		Machine coordinate system selection
G54	G54	14	Work coordinates system 1 selection
G55	G55		Work coordinates system 2 selection
G56	G56		Work coordinates system 3 selection
G57	G57		Work coordinates system 4 selection
G58	G58		Work coordinates system 5 selection
G59	G59		Work coordinates system 6 selection
G61	G61	15	Exact stop mode
G62	G62		Automatic corner override
G64	G64		Continuous cutting mode
G65	G95	00	Custom macro simple call
G66	G96	12	Custom macro modal call
G67	G97		Custom macro modal call cancel

Table 2.13 G code list (2/2)

System A	System B	Group	Meaning
G68	G68	00	Circular nibbling
G69	G69		Linear nibbling
G70	G70		Positioning & press off
G72	G72		Standard point command
G73	G75		Multi–piece machining command X
G74	G76		Multi–piece machining command Y
G75	G27		Automatic repositioning
G76	G28		Line at angle
G77	G29		Arc
G78	G36		Grid I
G79	G37		Grid II
G84	G84	16	Coordinate rotating on
◤ G85	◤ G85		Coordinate rotating off
G86	G66	00	Share proof
G87	G67		Square
G88	G78		Radius
G89	G79		Cut at angle
◤ G90	◤ G90	03	Absolute command
◤ G91	◤ G91		Incremental command
G92	G92	00	Coordinate system setting
G98	G98		Coordinate system setting (Multi–piece machining)

3 PRESSING FUNCTION

3.1 PUNCH FUNCTION (1-CYCLE PRESSING)

This control sends a signal “Start press and punch” to the machine after moving a tool to the position commanded in a predetermined block. When the machine receives this signal, it starts pressing. As a result, punching is made on a workpiece by the selected tool. After punching, the press motion stops, and a signal returns to the NC to indicate that “punch has finished”.
Thus, NC proceeds to the execution of the next block. In this manner, punching on a workpiece by press motion is executed by data transfer between the NC and the machine, and it is necessary to know the blocks to be punched, in advance.
This description is made from the viewpoints of the NC side. Since details may differ depending upon the machine tool builders, refer to the machine tool builder’s manual without fail.
For details of the pressing function, refer to section 3.5.

3.1.1 Block in which Punching is Made

Punching is made in a block where the X-axis or Y-axis is positioned at rapid traverse, in principle.
In other words, punching is not done in a block where the X-axis or Y-axis is not positioned at rapid traverse. Blocks where punching is done are as follows:

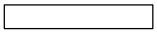
(1) Block where X-axis or Y-axis is positioned in the positioning mode (G00)

CAUTION

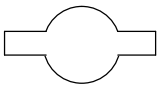
If the same position as the present tool position is commanded by address X or Y, positioning is not done, but punching is executed. (This is regarded as the positioning command with movement amount 0)
G00G91X0; . . . Punching is made.
This applies to such a case that punching is done at the same position using a different tool.

Examples

Tool 01 profile 

Tool 02 profile 

N711G00G90X50.0Y30.0T02; **. . .** Punching is made using tool 02
N712X50.0Y30.0T01; **. . .** Punching is made using tool 01
The punch profile at (50, 30) position is as shown below.



No punching is made in case of N712T01;, N712T01C50.01;

CAUTION
Punching is not done in T single block where the X-axis or Y-axis moves for tool offset.

(2) Block where pattern function G26, G76, G77, G78, G79, G86, G87 or G89 was commanded

Punching is made after positioning to respective points on a pattern.

Punching is not done in the following cases, even if the block corresponds to (1) or (2).

(a) MDI mode is selected.
(b) M code is commanded.
(c) Blocks inserted between M code of workpiece clamp and M code of workpiece unclamp which are employed for repositioning of workpiece.
(d) Block where positioning & punch off (G70) was commanded.

CAUTION
Punching is not done even in G00 mode if the block is irrespective of positioning such as coordinate system setting (G92), local coordinate system setting (G52), standard point command (G72), dwell (G04), etc.

3.2 POSITIONING & PRESSING OFF (G70)

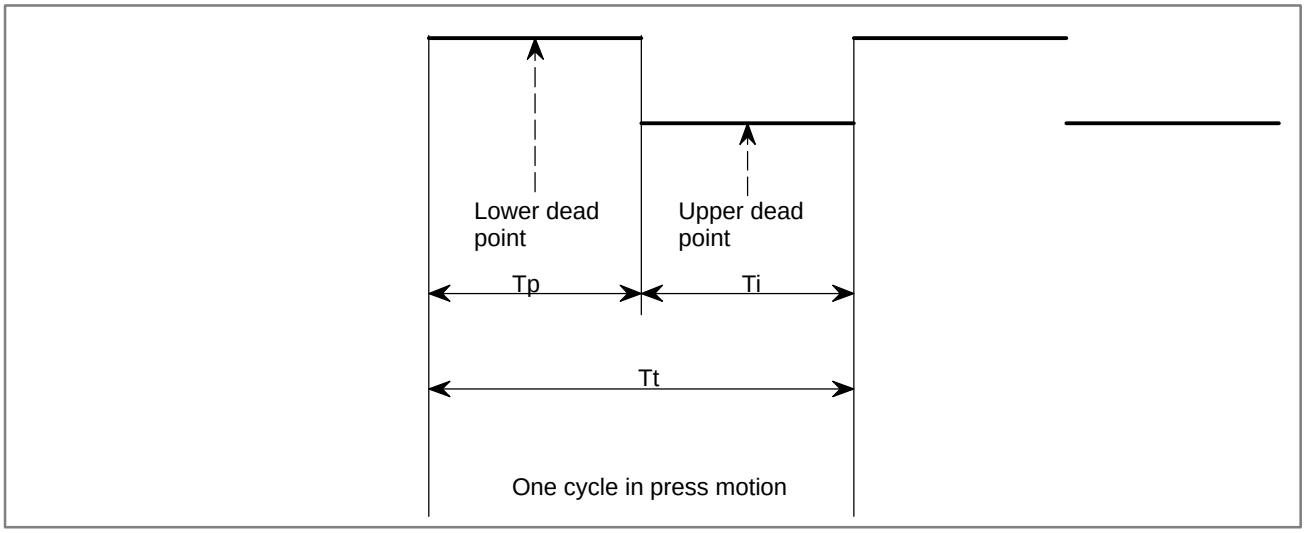
Punching is made in a block where the X-axis or Y-axis if positioned at rapid traverse, in principle.
Command the following code, if it is not desired to punch a workpiece after positioning a tool to the commanded position at rapid traverse.

G70X__Y__;

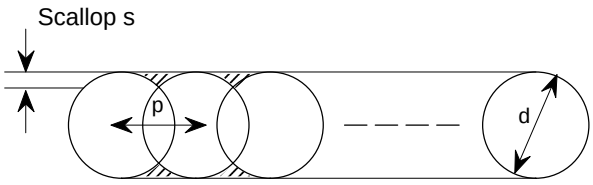
CAUTION
1 G70 is an one–shot G code.
2 Rapid traverse is made in a G70 block even if in G01, G02 or G03 mode.

3.3 NIBBLING FUNCTION

Nibbling means sequential repeated punching without stopping press motion.
Assume Tt be the time required for one-cycle press motion. The remaining time obtained by subtracting punching time Tp from Tt (or, Ti = Tt – Tp) is the time allowable for positioning.



The maximum distance (maximum pitch) which can be positioned in time Ti is limited by various conditions, such as machine, servo motor, and others as well as time Ti.
In this NC, the maximum nibbling pitch determined by these conditions is preset as a parameter.
On the other hand, the nibbling pitch is commanded by a program. If the commanded pitch exceeds the maximum pitch preset by the parameter, an alarm is produced.
Since this pitch can be specified directly, programming can be done, while taking the scallop into consideration.

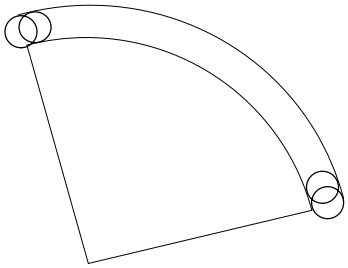
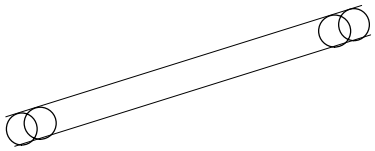


The relation between pitch p and scallop s is as shown below in case of linear nibbling

$$p = 2\sqrt{ds - s^2}$$

where d: Tool diameter

The following functions are prepared for nibbling.

Functions	Description
Circular nibbling (G68)	
Linear nibbling (G69)	
Nibbling by M function	M12; . . . . . } . . . . . } Nibbling is performed in these blocks. . . . . . } . . . . . } M13; (Note) Other M codes may be used instead of M12 and M13 depending upon machine tool builders.

CAUTION

(1) The maximum pitches in G68 and G69 are set by parameters No. 16186 (for mm input) and No. 16187 (for inch input).

(2) If T code is commanded in G68 or G69 block, nibbling is started after the X and Y axes have moved to the first punch point and also a tool has been selected.

(3) M code is not commandable in G68 and G69 blocks.

(4) For the rapid traverse to the first punch point, the rapid traverse override is effective when it is specified by the rapid traverse override switch on the machine operator's panel or by F1-digit specification. For the pitch movement up to the final point, the rapid traverse override is ineffective and fixed to 100%.

(5) If G68 or G69 is commanded using the single block operation, nibbling is made up to the last punch point, and then, stopped.

(6) If feed hold is applied halfway during the movement to the first punch point, the X and Y axes stop at once.
These axes also stop immediately when the feed hold is applied halfway during the pitch movement from the first point to the last point.
However, this can be changed by parameter NSP (No. 16181#2) in such a way that the X and Y axes stop after pitch movement.

(7) In a block just after G68 or G69, the tool does not move by the incremental amount from the tool position when nibbling ends, but moves from the programmed end point of the arc or straight line by the incremental amount.

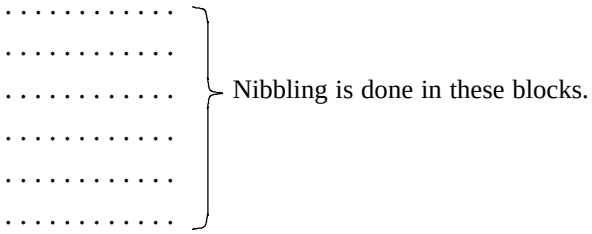
Parameter

Refer to parameters No.16181 ~ No. 16194 in the parameter manual (B–62780EN).

3.4 NIBBLING BY M FUNCTION

In addition to the circular or linear nibbling according to the G68 or G69 command, this control can perform nibbling by M function. In other words, it can execute nibbling in the blocks from a block with the M code of nibbling mode to a block with the M code of nibbling mode cancel as shown below.

M12; (M code of nibbling mode)



M13; (M code of nibbling mode cancel)

In this manual, the M code of nibbling mode is described as M12, while the M code of the nibbling mode cancel is described as M13. However, since these M codes may be different from those specified above in certain machine tool builders, you are requested to read these M codes correspondingly according to the manual prepared by these machine tool builders.
Don’t use this nibbling by M function in a different way other than specified in this manual, since there are certain restriction about its use.

WARNING
Each of the M codes for nibbling mode and nibbling mode cancel must be commanded in a single block.

NOTE

(1) The following commands only are executable in nibbling mode.
   (i) X, Y positioning command by G00
   Provided that the T code and F1-digit command can be included in the same block where the X, Y positioning is made by G00 to the first punch point of nibbling.
   (ii) G26 (bolt hole circle), G76 (line at angle), G77 (arc), G78, G79 (grid), G86 (share proofs), G87 (square), G88 (radius), G89 (cut at angle)
   The movement amounts along the X-axis and Y-axis to respective positioning points should not exceed the parameter set value (Nos. 16188, 16189), except when the first positioning point is equivalent to the first punch point of nibbling.
   (iii) G01, G02, G03, G41 and G42 commands.

(2) The positioning distance commandable by the X-Y positioning command by G00 is not composite distance $\sqrt{\Delta x^2 + \Delta y^2}$ obtained by the movement amounts along X-axis and Y-axis.
   If the absolute value of the movement amount along either X-axis or Y-axis exceeds the parameter set value (Nos. 16188, 16189), alarm (No. 4521) is issued.
   This provision also applies to G26, G76, G77, G78, G79, G86, G87, G88 and G89, correspondingly.

(3) When offset was made by the tool diameter by G41 or G42 to G01, G02 and G03 as described, the offset straight line or circular arc is divided by the pitch commanded by address Q.
   Be careful since the above division differs form such a case that a commanded circular arc is divided by a commanded pitch, like in G68 (circular nibbling).

Parameter

Refer to parameter No.16181 ~ No. 16194 in the parameter manual (B–62780EN).

Reference

Series 16/18/160/180–PC OPERATOR'S MANUAL I–9.4 "Nibbling by M Function".

3.5 PRESS FUNCTION

3.5.1 1–Cycle Press

1) Press start signal (output) PF <Y1004#2>
2) Press stop signal (input) *PE <X1004#7>
3) Punch finish signal for 1–cycle press (input) *PFIN <X1004#5>

In the punching block, the PF signal goes to 1 after positioning if tape or memory command input is already selected. In the machine tool, this signal makes the press start for punch operation. When the time set in parameter 16030 elapses after the *PE signal goes to 0, the PF signal goes to 0. Use the *PE signal to stop the press.
When the time set in parameter 16040 elapses after the *PFIN signal goes to 0, processing goes to the next block.

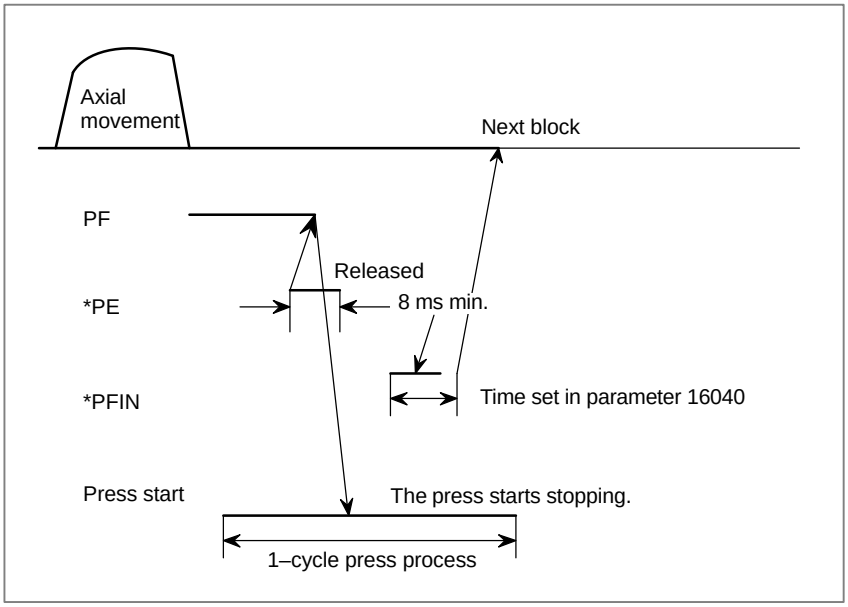


Fig. 3.5.1(a) 1–Cycle Press Process

Several parameters for the PF signal can be used to adjust the timing of 1–cycle press.
The PF signal can be set to 1 before the end of positioning depending on setting in parameter 16012. If the set time is longer than the time required for deceleration during axial movement, the PF signal goes to 1 at the same time deceleration starts. If the condition for setting the PF signal to 1 is satisfied in all X–, Y–, and C–axes, the PF signal goes to 1 during simultaneous positioning for each axis. (See Fig. 3.5.1(b).)

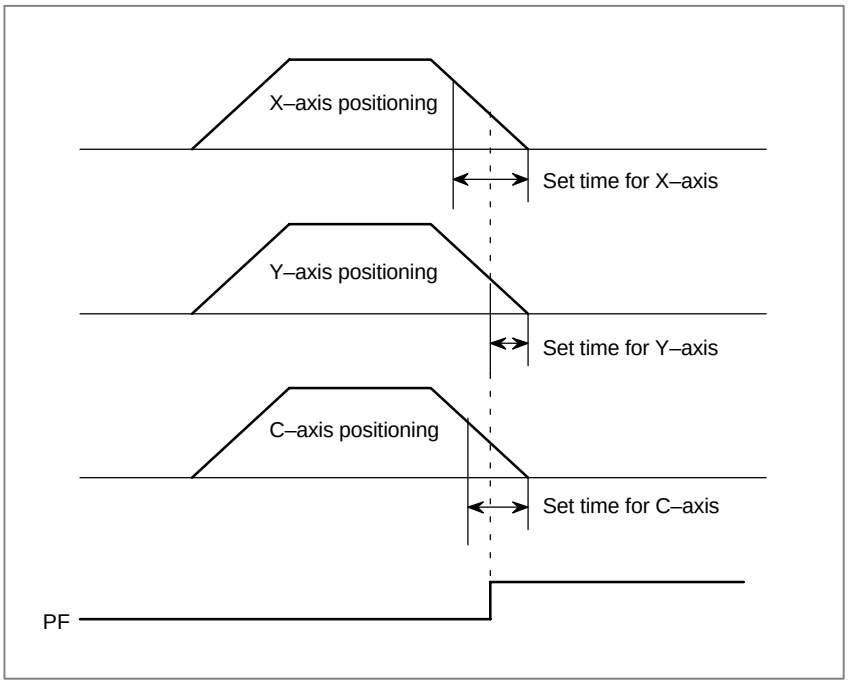


Fig.3.5.1(b) Quick Generation of the PF Signal

NOTE
The timer for quick generation of the PF signal can be set for up to seven steps for the X– and Y–axes or up to three steps for the C–axis using parameters KLV (No.16050#7) and KLC (No.16050#4) according to the positioning distance.

With the X– and Y–axes, the PF signal can be set to 1 with parameter PFE (No. 16001#1) and PE2 (No. 16001#3) when the absolute value of the positional deviation becomes equal to or less than the value set in parameter 16010. The quick timer described above is enabled when the PF signal is set to 1.
To clamp the hit rate, the shortest time required for the steps from setting the *PFIN signal to 0 to setting the PF signal to 1 can be specified with parameter 16036. The PF signal is not set to 1 until the specified time elapses even when the tool completes positioning for the next block and all conditions for setting the PF signal to 1 are already satisfied after the *PFIN signal goes to 0.

In the block between the M code for entering the forming mode (setting in parameter 16008) and the M code for canceling the forming mode (setting in parameter 16009), the PF signal goes to 1 after the time set in parameter 16032 elapses. When the time set in parameter 16033 elapses after the *PFIN signal goes to 0, processing goes to the next block.
(See Fig. 3.5.1(c).)

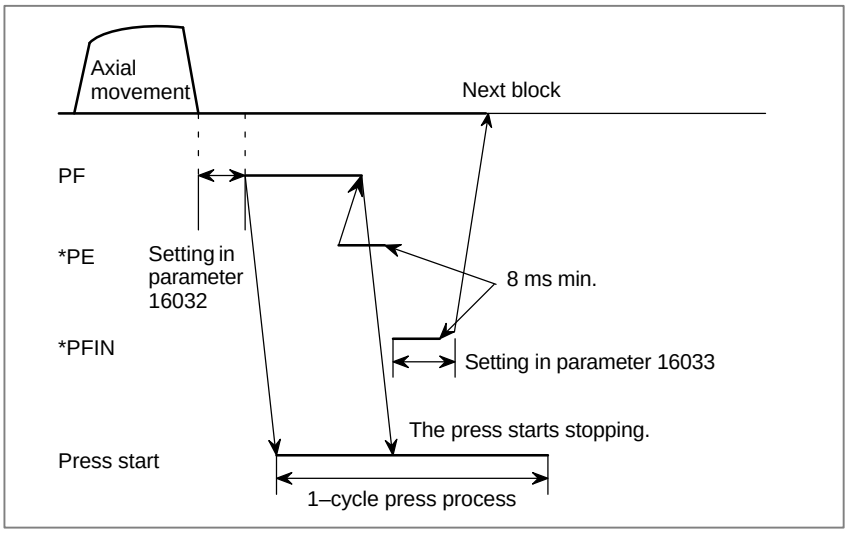


Fig.3.5.1(c) 1–Cycle Press Process in the Forming Mode

NOTE
1 Parameter TCF (No.16003#5) can be used to set the PF signal to 1.
2 The PF signal goes to 0 in case of emergency stop, external reset, or reset when the PF signal is 1. However, the PF signal can be set to 0 after the *PE signal is set to 0 according to setting in RPF (No.16000#2).

3.5.2 Continuous Press (Nibbling)

1) Nibbling signal (output) NBL <Y1004#1>
2) Nibbling finish signal (input) *NFIN <X1004#6>
3) 1–cycle press select signal (input) CPS <G230#2>
4) Nibbling completion signal (output) NBLE <F230#7>

When tape or memory command input is already selected, linear or circular nibbling can be performed in the block between the G68 or G69 code or the M code for entering the nibbling mode (setting of parameter 16183) and the M code for canceling the nibbling mode (setting of parameter 16184). When the time set in parameter 16034 elapses after the tool is positioned at the first punch point, the PF signal goes to 1. In the machine tool, this signal makes the press start for punch operation.

In nibbling, the PF signal goes to 1 at the same time the NBL signal goes to 1. Since the PF and NBL signals remain 1 till nibbling is completed, press operation can be repeated periodically without the press stopping. When the *NFIN signal goes to 0 after completion of punching, the tool starts moving to the next punch point.

The *NFIN signal goes to 0, and the tool starts moving to the last punch point at the same time the NBLE signal goes to 1. If the *PE signal goes to 0 after the end of positioning to the last punch point, the PF signal goes to 0. Accordingly, the press is stopped. In this case, the NBL and NBLE signals go off at the same time the PF signal goes to 0. After the tool is positioned at the last punch point and the *NFIN signal goes to 0, the time set in parameter 16035 elapses, then processing goes to the next block. (See item (a) in Subsection 3.5.2)

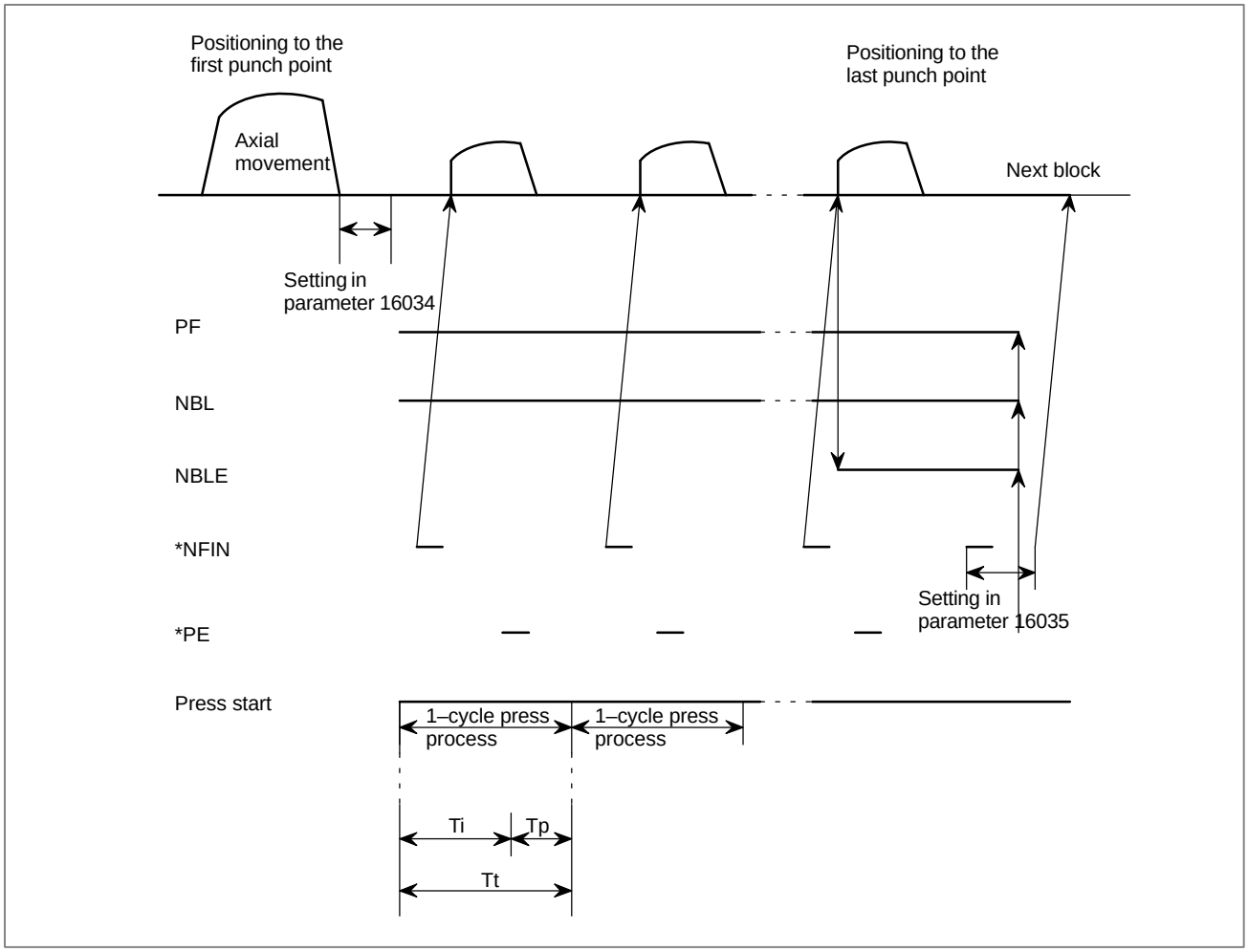


Fig.3.5.2(a) Nibbling (Continuous Press) Process

CAUTION

1 Let the time required for 1–cycle press operation be Tt and the time required for punch operation be Tp. The time permissible for positioning Ti is found by the following formula: Ti = Tt – Tp. The maximum distance allowable for positioning requiring the time Ti, namely the maximum nibbling pitch, is determined when the conditions such as selection of the time Ti, a machine, and a servo motor, or other conditions are satisfied.

2 Since the NBLE signal goes to 1 when the M code for canceling the nibbling mode is read, nibbling by the M code is performed a little later than the time when the tool starts moving to the last punch point.

With the machine that sets the *PE signal to 0 after the *NFIN signal goes to 0 in a press cycle, no punch operation may be performed at the last punch point if the distance for positioning to the last punch point is shorter than the specified one. This is due to the following reason. Setting the *NFIN signal to 0 starts positioning to the last punch point. However the tool completes positioning before the *PE signal goes to 0. Then the *PE signal going to 0 sets the PF signal to 0, thus causing the press to stop. With the machine under this condition, therefore, use parameter NED (No. 16003#7) to specify the following operation. After the end of the last positioning in the nibbling block, the *NFIN signal goes to 0. Then, the *PE signal going to 0 can set the PF signal to 0. (See item (b) in Subsection 3.5.2)

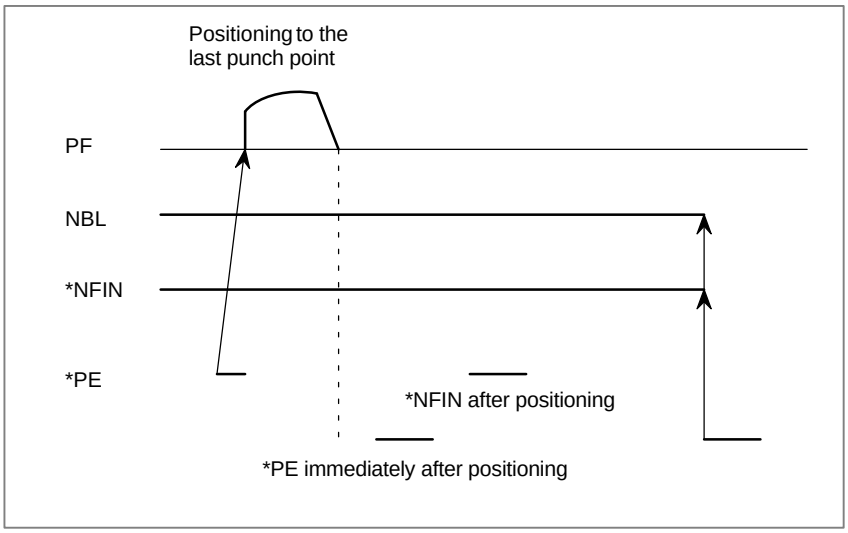


Fig.3.5.2(b)

In the nibbling block when the CPS signal is 1, when the time set in parameter 16034 elapses after positioning to the first punch point ends, the PF signal goes to 1, but the NBL signal does not go to 1. When the *PFIN signal goes to 0, the tool starts positioning to the next punch point. When the *PE signal goes to 0, the time set in parameter 16030 elapses, then the PF signal goes to 0. When the tool complete positioning to the next punch point, the time set in parameter 16034 elapses, then the PF signal goes to 1. Thereafter, the operation is repeated until the tool moves to the last punch point. (See Fig. 3.5.2(c).)

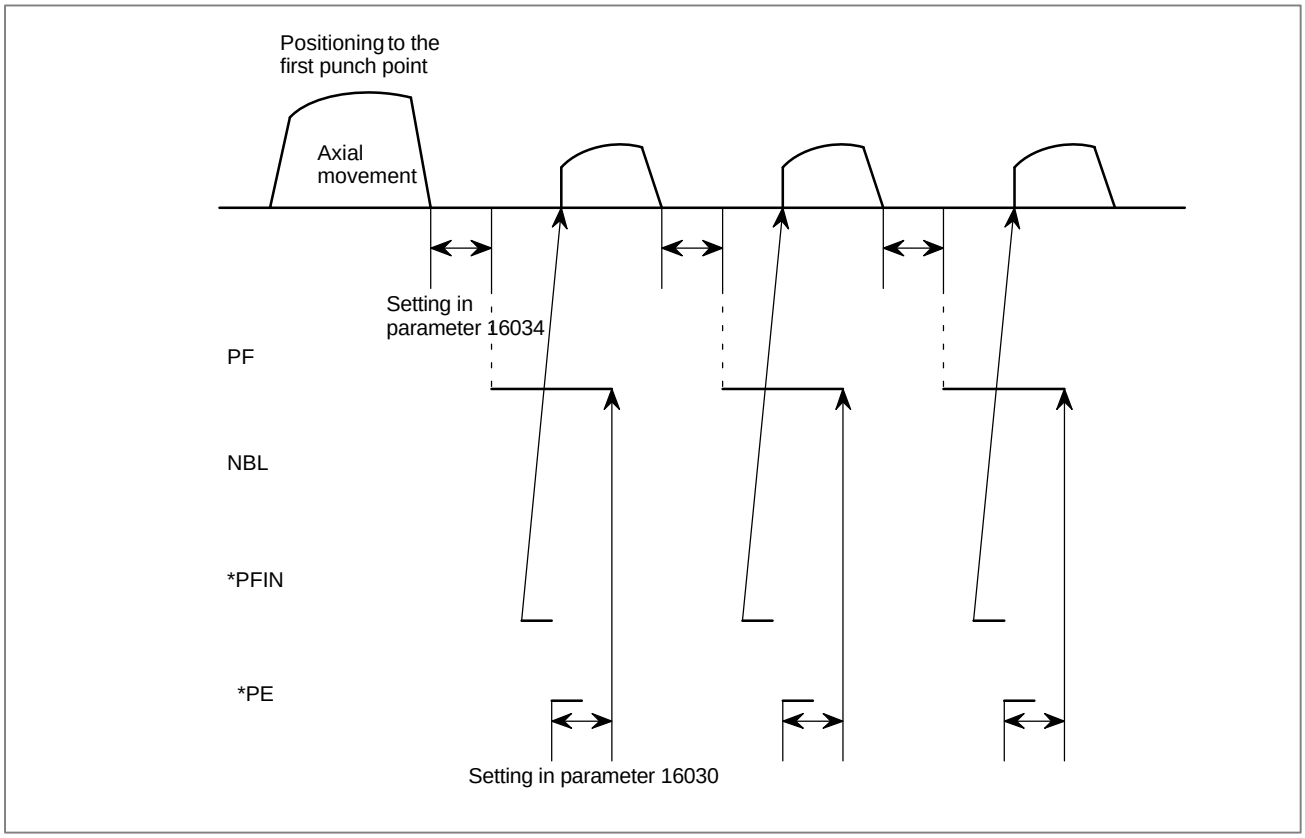


Fig.3.5.2(c) Nibbling when the CPS Signal is 1

When the CPS signal is set to 0 during nibbling in the state that the CPS is 1, the PF signal goes to 1 at the same time the NBL signal goes to 1. Then the above nibbling continues. When the CPS signal is set to 1 during nibbling in the state that the CPS signal is 0, the *PE signal going to 0 sets the PF and NBL signals to 0. Then the above nibbling when the CPS signal is 1 continues.

CAUTION

1 Override signals ROV1 and ROV2 for rapid traverse are effective during positioning to the first punch point in the nibbling block. After that, the ROV1 and ROV2 signals are ineffective during positioning for the pitch to the last punch point, and the override rate is fixed to 100%.

2 When the single block signal is set to 1 during nibbling, the machine stops after consecutive nibbling operations are completed.

3 While the tool moves to the first punch point during nibbling, setting the automatic operation stop (*SP) signal to 0 stops the automatic operation of the CNC. At the same time, positioning stops immediately, setting the automatic operation starting (STL) signal to 0 and the automatic operation stopping (SPL) signal to 1.
When the *SP signal is set to 0 during pitch movement from the first punch point to the last punch point, automatic operation of the CNC is stopped immediately. However, automatic operation of the CNC can also be stopped after the end of the pitch movement when the *SP signal is set to 0 by the setting of parameter NSP (No. 16181#2). In this case, the PF and NBL signals are turned off when the *PE signal goes to 0.

3.5.3 Manual Press

1) Manual press start signal (input) MPS <G230#3>

2) Continuous manual press signal (input) MNS <G230#4>

Setting the MPS signal to 1 can set the press start (PF) signal to 1. The signal is used when a 1–cycle press operation is performed with the push button on the machine operator’s panel.
During manual press operation, the control of the PF signal depends on the status of the MNS signal. When the MPS signal changes from 0 to 1 in the state that the MNS signal is 0, the PF signal goes to 1. When the time set in parameter 16030 elapses after the *PE signal goes to 0, the PF signal goes to 0. (See Fig. 3.5.3(a).)
When the MPS signal changes from 0 to 1 in the state that the MNS signal is 1, the PF signal goes to 1. The state of the PF signal is held until the MPS signal goes to 0. When the MPS signal changes from 1 to 0, the PF signal goes to 0. (See Fig. 3.5.3(b).)

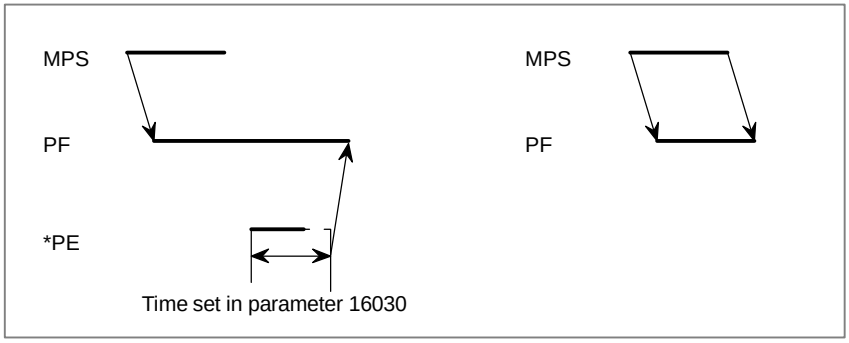


Fig.3.5.3(a) MNS Signal=0 **Fig.3.5.3(b) MNS Signal=1**

3.5.4 Press Start Lock Signal (Input) PFL <G230#0>

When the PFL signal is 1, the PF signal does not go to 1, but can be locked in the block for 1–cycle press. The tool completes positioning in the state that the PFL signal is 1 and the time set in parameter 16031 elapses, then the next block is executed. (See Fig. 3.5.4(a).)

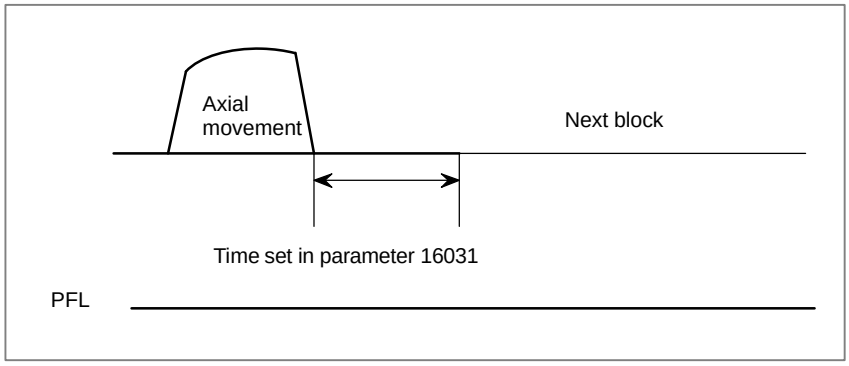


Fig.3.5.4(a) 1–Cycle Press Operation when the PFL Signal is 1

In the same way, the PF signal and the nibbling (NBL) signal do not go to 1 in the block for nibbling when the PFL signal is 1. When the tool completes positioning to the first punch point, the timer set in parameter 16034 starts. If the PFL signal is 1 in this case, the time set in parameter 16031 elapses, then the tool starts positioning to the next punch point. If the PFL signal remains 1, the tool completes positioning for each pitch, then the previously described processing is repeated. (See Fig. 3.5.4(b).)

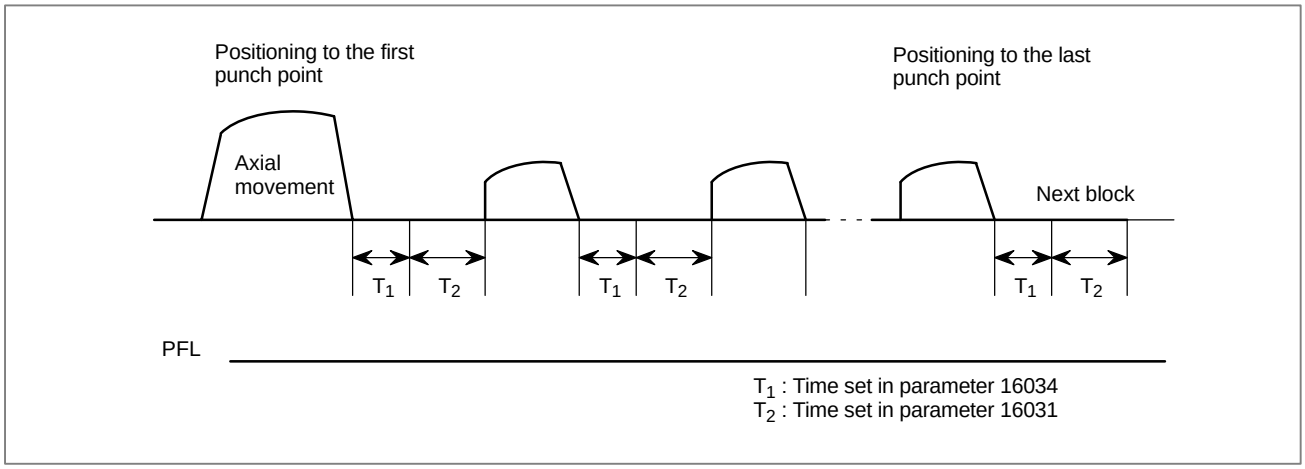


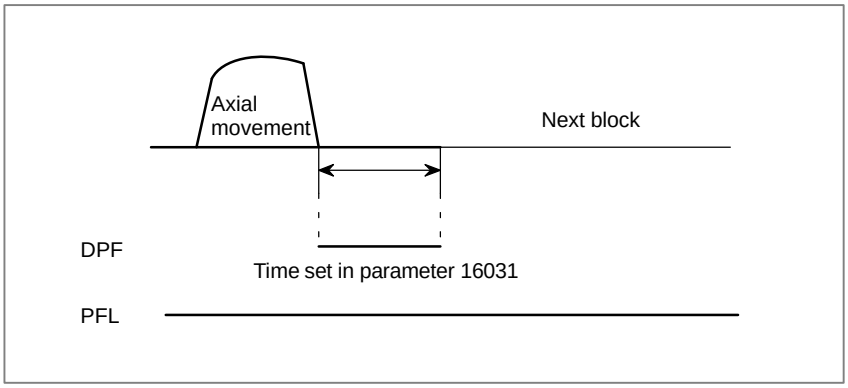
Fig.3.5.4(b) Nibbling when the PFL Signal is 1

When the PFL changes from 1 to 0, the tool completes positioning for the pitch, the time set in parameter 16034 elapses, then the PF and NBL signals go to 1. When the PFL changes from 0 to 1, the tool completes positioning for the pitch, the press stop (*PE) signal goes to 0, then the PF and NBL signals go to 0. These operations are the same as those when nibbling starts and ends. (See Subsection 3.5.2)

NOTE
The PFL signal is effective for manual press operation.

3.5.5 Press Start Assistance Signal (Output) DPF <F230#6>

When the press start lock (PFL) signal is 1 during tape or memory operation, DPF signal is set to 1, not the press start (PF) signal. To lock the press start operation due to some reason related to the machine, the automatic operation of the CNC can be stopped according to the supervision of the DPF signal if necessary, or the CNC can be made to enter the alarm state with the external data input function.
The DPF signal goes to 0 after the time set in parameter 16031 elapses, then the next block is executed.



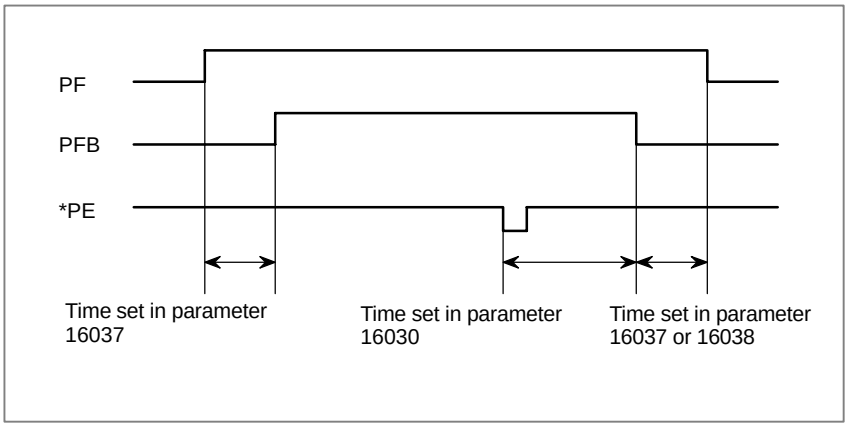
3.5.6 Press Start Waiting Signal (Input) PFW <G230#1> and Press Start Waiting Signal B (Input) PFWB <X1004#4>

When the press start lock (PFL) signal is 0 in the block for punching, setting the PFW signal to 1 inhibits the press start (PF) signal from going to 1. The PFW signal can be used when an attempt is made to apply an interlock to the press start operation according to the mechanical conditions. However, the PFW signal is ineffective if the PF signal already goes to 1.
Both the PFWB signal and the PFW are provided. The PFWB signal is read directly from the machine by the CNC, not via the PMC. The use of the PFWB is the same as that of the PFW signal. The PFWB signal is made effective by parameter PWB (No. 16002#5).

CAUTION
These signals are ineffective for continuous manual pressing.

3.5.7 Press Start Signal B (Output) PFB <Y1004#3>

The press start (PF) signal goes to 1 and the time set in parameter 16037 elapses, then the PFB signal goes to 1. The press stop (*PE) signal goes to 0 and the time set in parameter 16030 elapses, then the PFB signal goes to 0. After that, the time set in parameter 16037 or 16038 elapses, then the PF signal goes to 0.
The PFB signal is used, for example, for the following. The PF signal controls on and off of the brake for the press while the PFB signal controls on and off of the clutch for the press.



CAUTION
1 When the PFB signal is not used, always set parameters 16037 and 16038 to 0.
2 When the setting in parameter PF9 (16002#6) is 1, the time set in parameter 16038 is used as the time required for the process from setting the PFB to 0 to setting the PF signal to 0.

3.5.8 Press Stop Signal Neglect (Input) EPE <G230#5>

When the setting in parameter DPE (No. 16003#6) is 1, the EPE signal enables switching of the press stop (*PE) signal between effective and ineffective states. When the EPE signal is 0, the *PE signal is ignored. When the EPE signal is 1, the *PE signal is made effective.

CAUTION
The EPE signal is ineffective for manual press operation.

3.5.9 Two–step Selection (Input) SNP for Nibbling <G230#6>

If the machine can change the rotation of the flywheel, it can perform nibbling at high and low speeds. The maximum pitch movable with the nibbling command is limited and set in parameter. The SNP signal is used to switch between the maximum moving pitches for high–speed nibbling and low–speed nibbling. The signal can also be used to change the maximum moving pitches according to the thickness of a board blank to be machined.
When the SNP signal is 0, the machine uses the maximum movement pitch set in parameters 16186 to 16189. When the SNP signal is 1, the machine uses the maximum movement pitch set in parameter 16190 to 16193.

NOTE
1 Even when the SNP signal is not used, the maximum movement pitch can be selected by nibbling with the M code set in parameter 16185.
2 Nibbling with the M code set in parameter 16185 and nibbling with the SNP are effective when setting in parameter NPC (No. 16181#1) is 1.

3.5.10 Press Start Auxiliary Signal B DSPF<F230#5>

[Classification] Output signal

[Function] Notifies the PMC of a punching block.

[Output condition] Once positioning has been completed as part of automatic operation, this signal is output in a punching block in which press start signal PF is output. This signal is also output when press start wait signals PFW and PFWB are set to 1. The signal is not output in manual press mode.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F230			DSPF					

Parameter

Refer to the parameter manual B–62780EN/01 for details of parameters No.16000 to 16040, and No.16181 to No. 16194.

3.6 HIGH SPEED PRESS CONTROL FUNCTION

As the speed of the press mechanism increases, the time between the end of pressing and the beginning of the next positioning has a greater effect on the hit count of the punch press.
If the time can be estimated, the hit count of a conventional punch press can be prevented from decreasing by issuing the punch completion signal early. Many recent press mechanisms cannot output the punch completion signal early, however.
This function has been developed to improve the hit rate by enabling the controller to detect the punch completion signal at high speed and to start the next positioning with the minimum of delay.

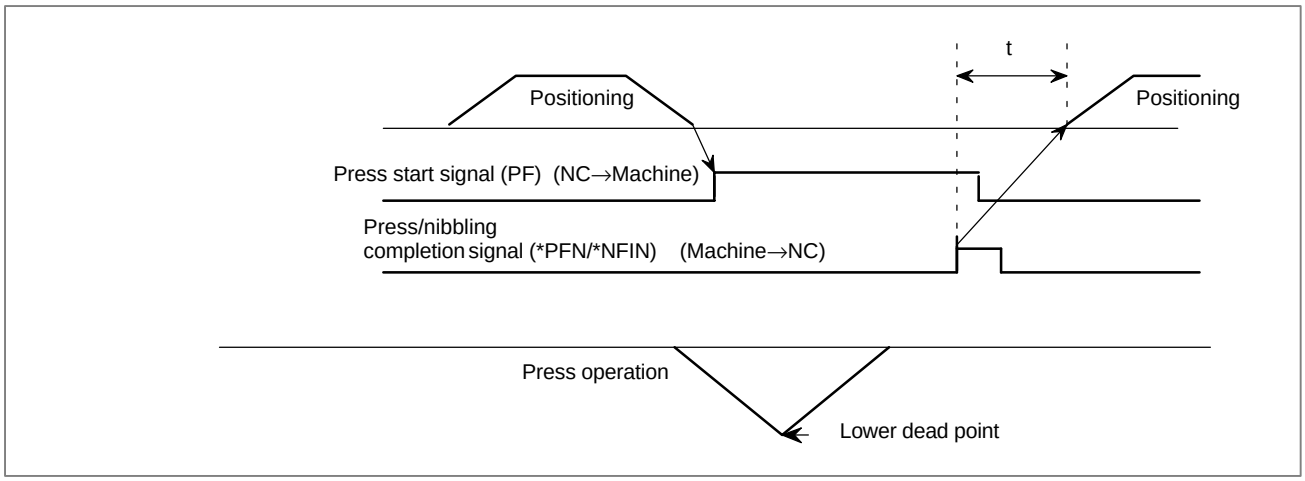


Fig. 3.6 Processor High Speed Press Control Function

In a conventional system having no high–speed press control function, the time t indicated above is set to 17 ms (fixed) plus a variation of up to 2 ms. The time t can be reduced to 2 ms if the *PFIN or *NFIN signal is output 17 ms earlier than the estimated press end time.

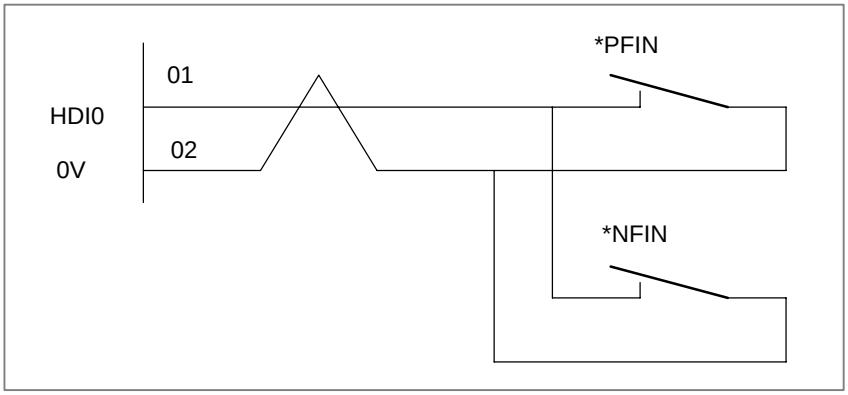
If the press operation cycle is constant, the time interval between the end of a press operation and the beginning of positioning can be specified by issuing *PFIN or *NFIN early. Recent systems vary the press operation cycle, depending on the thickness of the workpiece and other conditions. Such systems cannot output *PFIN or *NFIN early, however.
To compensate for this disadvantage and improve the hit rate, recent systems must detect the *PFIN or *NFIN signal at high speed and start the next positioning with the minimum of delay. When this function is used, the time t indicated in Fig. 3.6 can be reduced as follows:

1 When high–speed DI is used: Fixed time of 1 ms + variation of up to 2 ms
2 When an I/O card is used: Fixed time of 3 ms + variation of up to 2 ms

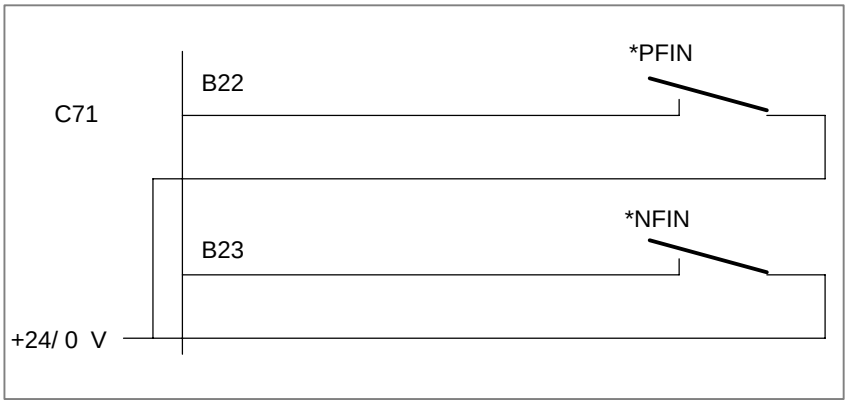
Signal

Connection of signal *PFIN and *NFIN is as follows:

1 When high speed DI is used



2 When I/O card is used



Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
16000							HCI	HSP

[Data type] Bit

HSP High–speed press control is:

0 : Disabled.
1 : Enabled.

HCI Under high–speed press control, the *PFIN signal to complete punching for single–cycle pressing, and the *NFIN signal to complete punching for continuous pressing are valid for:

0 : Standard address (X1004).

When this is selected, the maximum stop time, from when the punching complete signal is input until movement along an axis starts, is 5 msec.

1 : High –speed DI address HDI0 (both *PFIN and *NFIN).

When this is selected, the maximum stop time, from when the punching complete signal is input until movement along an axis starts, is 3 msec.

Limitations

Using the high–speed press control function imposes the following limitations:

1 The PMC cannot execute axis control.

2 The look–ahead control function cannot be used.

3 High–speed press control does not function if followed by a non–punching block. That is, the non–punching block starts at the conventional timing.

4 If punching is executed after positioning, the servo waveform display will act abnormally during the time period from the end of positioning to the beginning of the next positioning.

3.7 EXTERNAL OPERATION FUNCTION EF, EFS, FIN

External operation function signal (output) EF<F008#0> External operation function select signal (output) EFS<G230#7> External operation function finish signal (input) FIN<G004#3>

When a tape or memory command input is selected, press start signal PF is turned to 1 after positioning in a block to be punched. However, if signal EFS is 1, signal PF is not turned to 1, but signal EF is turned to 1. Perform tapping and other operation by this signal on the machine side. Turn signal FIN to 1 to turn signal EF to 0, and resultantly turn signal FIN to 0 when the operation has finished completely. The control proceeds to the next block after signal FIN has been turned to 0.
If signal PF is preset to be turned to 1 (parameter (No. 16012)) before completion of positioning, signal EF is also turned to 1 before completion of positioning. Accordingly, signal EF should be gated with distribution end signal DEN. Since signal EF is turned to 1 unconditionally, if signal EFS is 1 in a block to be punched, if no motion is desirable by the receipt of this signal on the machine side, treat signal EF as required, and turn FIJN to 1 at the above timing. If signal EFS is 0 and press lock signal PFL is 1 in the block to be punched neither signal EF nor PF is turned to 1, and the control proceeds to the next block after time No. 16031 has passed. Signal EFS is ineffective in blocks with nibbling (Nibbling by G68, G69 and M code).

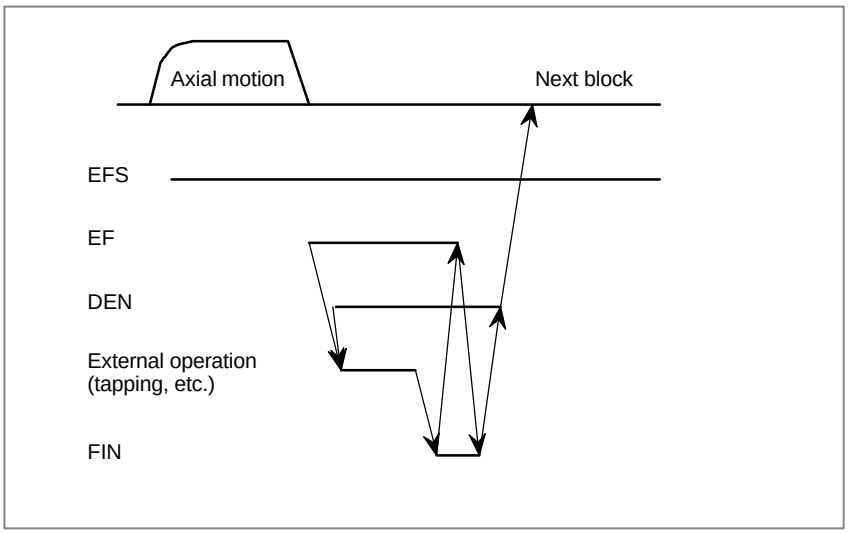
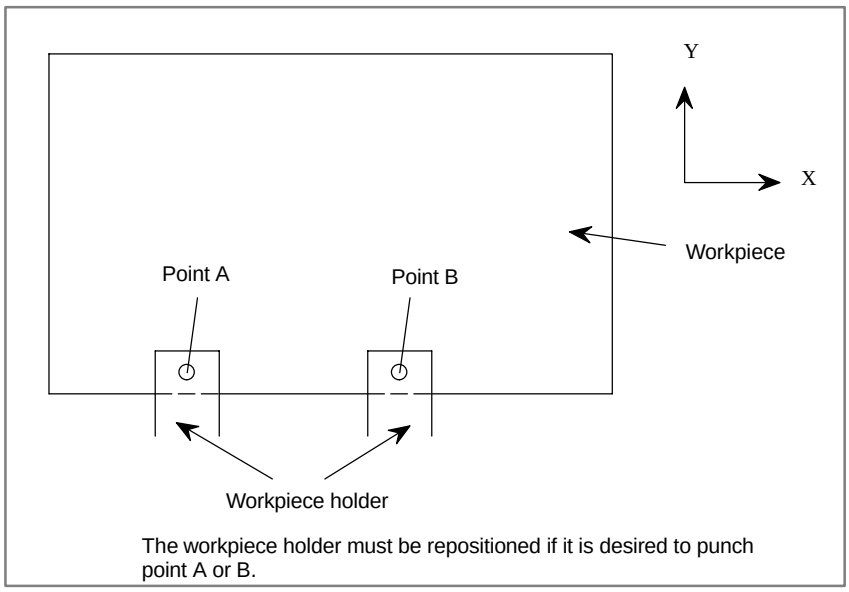


Fig.3.7 Timing Chart for External Operation Signal

4 FUNCTIONS TO SIMPLIFY PROGRAMMING

4.1 AUTOMATIC REPOSITIONING (G75)

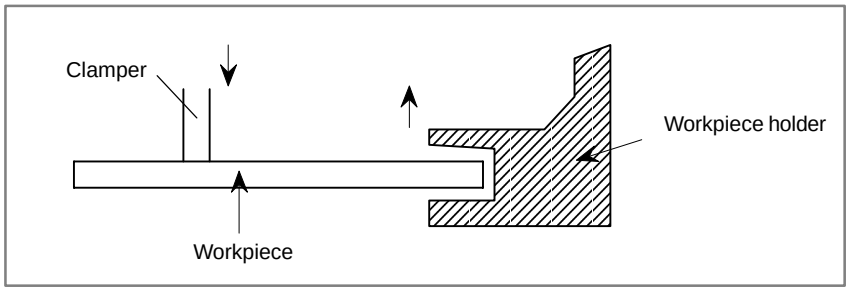
By changing the hold position of a workpiece by the workpiece holders, a workpiece having a size larger than the stroke in X-axis direction of the machine can be machined.
If it is desired to punch a workpiece at the workpiece holder position when the workpiece was set to the machine, the hold position of the workpiece must be changed.



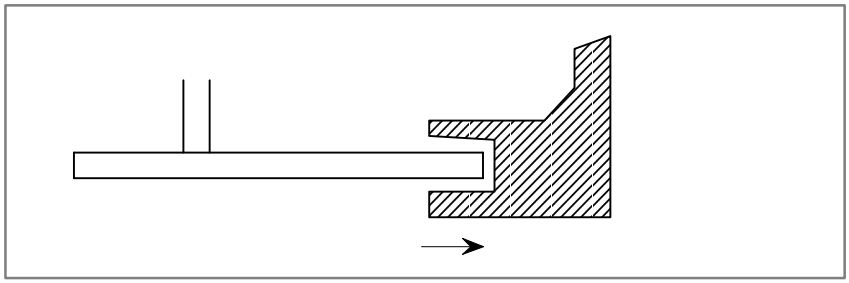
The workpiece holder must be repositioned if it is desired to punch point A or B.

Repositioning of a workpiece is generally done according to the following procedure, assuming that the workpiece is positioned at a location where the repositioning of the workpiece is executable.

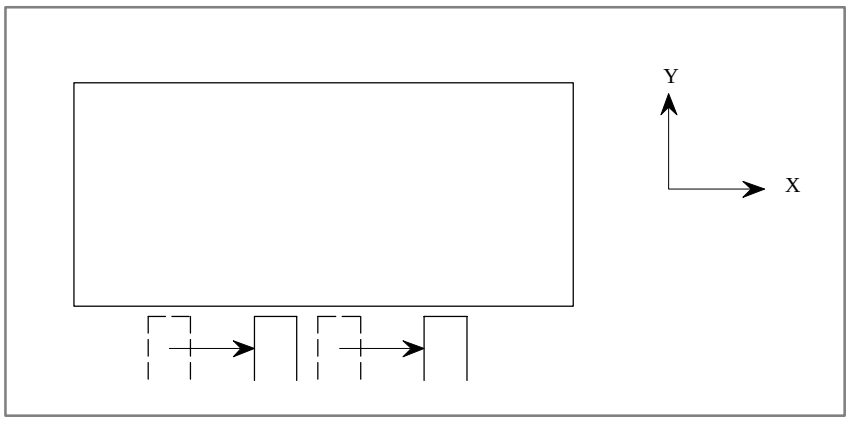
1) The claw of the workpiece holder is opened, and also the clamper depresses the workpiece concurrently to fix the workpiece as a general procedure, so that the workpiece is not deviated from the table.



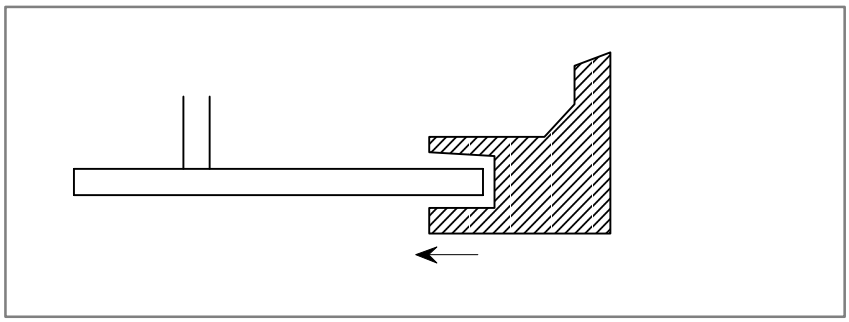
2) The workpiece holder moves in the Y-axis direction and separates from the workpiece.



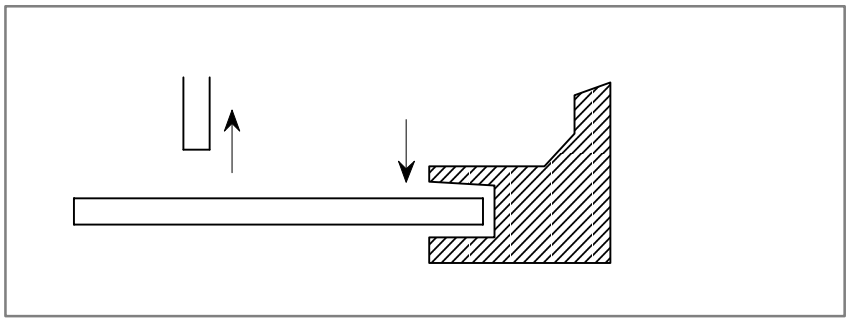
3) The workpiece holder moves in the X-axis direction to relocate the hold position.



4) The workpiece holder moves in the Y-axis direction to return to the position where it can hold the workpiece.



5) The claw of the workpiece holder is closed to hold the workpiece, and the clamper lifts and separates from the workpiece concurrently.



A series of the above operation can be done by one-block command including G function.

G75X x ;

The above command is executed by being divided into the following 5 blocks.

1 M10;
2 G70G91 y_R ;
3 G70G91X –x ;
4 G70G91Y –y_R ;
5 M11;

Parameter

Refer to parameters No.16209 and 16210 in the parameter manual (B–62780EN).

4.2 MULTI-PIECE MACHINING FUNCTION

The multi-piece machining function enables several sheets of product with the same punching shape to be produced from a single sheet of material at a time by simple commands.
This function allows so called “trial machining” that performs punching only on a sheet of product from the machining command tape for “multi-piece machining” by a simple setup method, therefore the machining command tape can be easily checked before full machining.

4.2.1 Base Point Command of Multi-Piece Machining (G98)

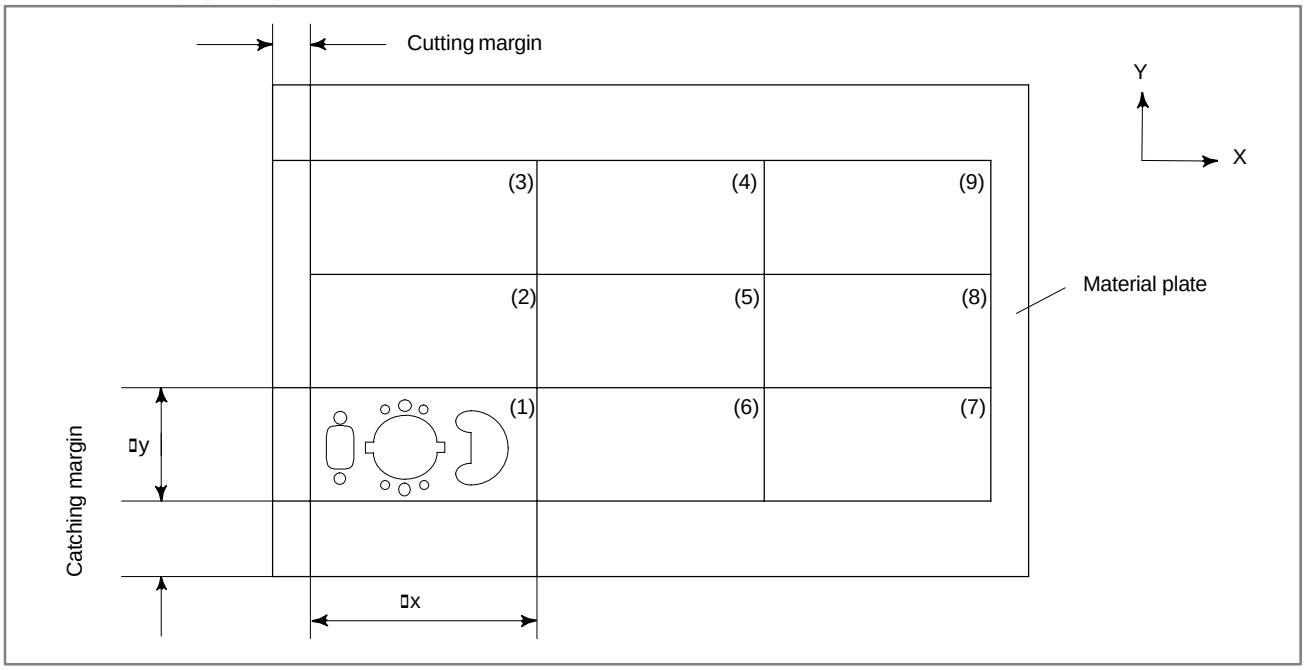


Fig.4.2.1

Parts (2) to (9) as shown above have the same punching shape as part (1). Machining commands to punch on a sheet of material must be specified on the product part at the lower left ((1)).
The point at the lower left of the set of multi-products (point B shown in Fig. 4.2.1; called as “Base point of multi-piece machining” hereafter) must be specified prior to the machining commands to punch on a product part by G98 as absolute co-ordinates under the system of co-ordinates specified using G92 command. In the G98 command, the X-axial and Y-axial lengths of one product and the numbers of products in the directions of X-axis and Y-axis must be specified.

G98X xb Y yb I Δx J Δy P nx K ny ;

xb: X-axis coordinate value of the base point of multi-piece machining
yb: Y-axis coordinate value of the base point of multi-piece machining
Δx: X axial length of one product part (a positive number)
Δy: Y axial length of one product part (a positive number)
nx: The number of products in the X axial direction (Note)
ny: The number of products in the Y axial direction (Note)

4.2.2 Multi-Piece Machining Commands (G73, G74)

Specify any of the following commands, and then multi-piece machining is performed by calling machining commands stored using the macro function.

G73 W ωn Q q ; or
G74 W ωn Q q ;

where

ωn: A macro number

q: Machining start area specification

- q=1 Machining starts from the lower left area ((1) in Fig. 4.2.1)
- q=2 Machining starts from the lower right area ((7) in Fig. 4.2.1)
- q=3 Machining starts from the upper left area ((3) in Fig. 4.2.1)
- q=4 Machining starts from the upper right area ((9) in Fig. 4.2.1)

G73 goes on punching in the X axial direction, whereas G74 goes on punching in the Y axial direction in grid parts-line.

4.2.3 Setting of Machining Method for Multi-Piece Machining

When products are machined using a NC tape for multi-piece machining, any desired machining method can be selected according to a set-up from MDI.
Input a setting value into the setting data number 16206 in MDI mode according to the desired machining method.

Setting value for No. 16206

0: The NC tape for multi-piece machining is not used

1: Trial punching for multi-piece machining

2: Machining on the reset of material punched for trial

3: Full machining on a material for multi-piece machining

If trial punching is selected, only the lower-left product part of material ((1) in Fig. 4.2.1). As a result, macros except 60 to 89 are executed while storing, and blocks specified in G73/G74 are all ignored.
Machining on the reset of material punched for trial signifies that after punching on only the lower-left product part of material, the remaining product parts of material are machined entirely. For this purpose, no machining is performed during the storage of macros, and the machining on the lower-left product part is skipped when commands of G73/G74 are executed.
For full machining on a material, no machining is performed during the storage of macros, but machining over the entire product parts is performed by G73/G74.
Set to “0” if NC tape for the multi-piece machining is not used.

Signal

Multi–piece machining setting signal MLP1, MLP2<G231#0,#1>

[Classification] Input signal

[Function] The signals can set the method used for machining multiple workpieces. These signals are effective when the MLP bit (bit 3 of parameter 16201) is set to 1. Generally, the signals must be changed while the CNC is in the reset state.

MLP1	MLP2	Method used to machine multiple workpieces
0	0	The command for machining multiple workpieces is not used.
1	0	Trial machining when machining multiple workpieces
0	1	Remainder machining after trial machining when machining multiple workpieces
1	1	Complete machining of multiple workpieces

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G231							MLP2	MLP1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
16201					MLP	MPC		

[Data type] Bit

MPC When the number of machined workpieces is counted in multiple–workpiece machining:

0 : The number of actually machined workpieces is counted.
1 : The number is incremented by one when complete machining or remainder machining is executed (but not when trial machining is executed).

MLP Setting for taking multiple workpieces depends on:

0 : The set parameter (No.16206).
1 : A signal (MLP1 or MLP2) input from the PMC machine.

16206	Machining pattern when multiple workpieces are taken

[Data type] Byte

[Valid data range] 0 to 3

Parameter 16206 sets a machining pattern when multiple workpieces are taken.

0 : A program without the G73 or G74 command for machining when multiple workpieces are taken is used.

Note 1 Alarm 4539 is issued if the G73 or G74 command is found with this setting.

1 : A program containing the G73 or G74 command is used and test machining is executed.

2 : A program containing the G73 or G74 command is used and the remaining processing is executed after test machining.

3 : A program containing the G73 or G74 command is used and the entire machining is executed.

16228	Number of character that can be stored for a U or V macro function

[Data type] Byte

[Valid data range]

Setting Value	Number of macro storage characters
0	3200
1	11008
2	22272
3	27072

Alarm and message

Number	Message	Contents
4531	U/V MACRO FORMAT ERROR	An attempt was made to store a macro while storing another macro using a U or V macro. A V macro was specified although the processing to store a macro was not in progress. A U macro number and V macro number do not correspond with each other.
4532	IMPROPER U/V MACRO NUMBER	The number of an inhibited macro (number beyond the range from 01 to 99) was specified in a U or V macro command.
4533	U/V MACRO MEMORY OVERFLOW	An attempt was made to store too many macros with a U or V macro command.
4534	W MACRO NUMBER NOT FOUND	Macro number W specified in a U or V macro command is not stored.
4535	U/V MACRO NESTING ERROR	An attempt was made to call a macro which is defined three times or more using a U or V macro command. An attempt was made to store 15 or more macros in the storage area for macros of number 90 to 99.
4536	NO W, Q COMMAND IN MULTI-PIECE	W or Q was not specified in the command for taking multiple workpieces (G73, G74).
4537	ILLEGAL Q VALUE IN MULTI-PIECE	In the command for taking multiple workpieces (G73, G74), Q is set to a value beyond the range from 1 to 4.
4538	W NO. NOT FOUND IN MULTI-PIECE	Macro number W specified in the command for taking multiple workpieces (G73, G74) is not stored.
4539	MULTI-PIECE SETTING IS ZERO	The command for taking multiple workpieces (G73, G74) was specified although zero is specified for the function to take multiple workpieces (No. 16206 or signals MLP1 and MLP2 (PMC address G231, #0 and #1)).
4540	MULTI-PIECE COMMAND WITHIN MACRO	The command for taking multiple workpieces (G73, G74) was specified when a U or V macro was being stored.
4542	MULTI-PIECE COMMAND ERROR	Although G98P0 was specified, the G73 command was issued. Although G98K0 was specified, the G74 command was issued.
4543	MULTI-PIECE Q COMMAND ERROR	Although G98P0 was specified, the Q value for the G74 command was not 1 or 3. Although G98K0 was specified, the Q value for the G73 command was not 1 or 2.
4544	MULTI-PIECE RESTART ERROR	In the command for resuming taking multiple workpieces, the resume position (P) is set to a value beyond the range from 1 to total number of workpieces to be machined.

Reference item

Section I–14.5 “Multi–piece Machining Function” in Series 16/18/160/180–PC OPERATOR’S MANUAL.

4.3 Y–AXIS CRACK CANCEL

The M–codes which is set by parameters (No. 16610 to 16614) are commanded, the crack between work coordinate system and machine coordinate system of Y–axis repositioning motion is canceled.

Example1

```
Y 1 5 2 5 M 3 0 ;
```

Y–axis moves including the crack of repositioning.

Example2

```
Y 1 5 2 5 ;
M 3 0 ;
```

The work coordinate system is preset to the machine coordinate system to cancel the crack of repositioning by M30. (Y–axis does not move.)

Limitations

The cancel M–code must be different from another special meaning M–code.
But M02 and M30 is possible.

Parameter

Refer to parameters No.16610 to 16614 in the parameter manual (B–62780EN).

5 TOOL FUNCTION (T FUNCTION)

5.1 TOOL SELECTION FUNCTION

By specifying an up to 8–digit numerical value following address T, tools can be selected on the machine.
One T code can be commanded in a block. Refer to the machine tool builder's manual for the number of digits commandable with address T and the correspondence between the T codes and machine operations.
When a move command and a T code are specified in the same block, the commands are executed in one of the following two ways:

(i) Simultaneous execution of the move command and T function commands.
(ii) Executing T function commands upon completion of move command execution.
The selection of either (i) or (ii) depends on the machine tool builder's specifications. Refer to the manual issued by the machine tool builder for details.

The selection of either (i) or (ii) depends on the machine tool builder's specifications. Refer to the manual issued by the machine tool builder for details.

The T command must be given without fail to the block before a block where punching is first made by press motion in one program or to the same block where punching is first made by press motion.
If the T command is not given to these blocks, the press start signal which instructs "Punch by press motion", is not sent to the machine, and machining does not proceed to the next block.

Examples

```
O1000G92 . . . . . ;
N1G00G90X__Y__;
. . . . . .
. . . . . .
```

Punching should be made in N1 block.
However, since no T command is given to a block before N1 block or N1 block, no punching is made, and machining does not proceed to the next block.

CAUTION

1 The correspondence between commandable T codes and tools depends upon machine tool builders.
The commandable T codes are set in tool registering screen before shipment from factory (III-10.8). If a commanded T code was not registered, alarm (No. 4602) is produced.

2 No T code is commandable in the following blocks.
(i) G10 (Offset value setting)
(ii) G22 (Stored stroke limit function on)
(iii) G23 (Stored stroke limit function off)
(iv) G92 (Coordinate system setting)
(v) G52 (Local coordinate system setting)
(vi) G72 (Standard point command)
(vii) G75 (Automatic repositioning)
(viii) G98 (Base point command for multi-piece machining)
(ix) G73, G74 (Multi-piece machining command)

3 If tape or memory operation is made in the T command neglect status, the T command is ignored, and the operation is made as if the T command were not given.
The press start signal is not sent to the machine side in a block to be punched, and processing does not proceed to the next block. If a program is checked by marking to a workpiece by using a marking tool, for example, select the marking punch tool by a T command in the MDI mode in advance, and perform the tape or memory operation without reset operation in the T command neglect status. Now, punching is made using the tool selected in the MDI mode.

4 If automatic operation is applied to the machine side by the cycle start when the cycle start lamp signal is not sent, i.e., in the reset status, the cycle start lamp signal is sent.
If a block to be punched appears before a T command is given after this cycle start lamp signal has been sent, the press start signal is not sent to the machine side, and also machining does not proceed to the next block.
The cycle start lamp signal is stopped by reset operation.

5 The press start signal can be sent by setting a parameter TCF (No. 16003#5) even if no T command is given to a block before the block where the punching is made or the block in which punching is made. In this case, the integrated value of the number of punch times may differ from actual number of punch times about respective tools.

5.2 TOOL OFFSET

Tool offset is applicable to respective T codes in the X-axis and Y-axis directions.

Since use of this tool offset function depends upon machine tool builders, refer to the machine tool builder's manual.

NOTE
1 Tool offset compensation applies to tools numbered from 1 to 9999.
2 Tool offset values are set in the tool registering screen in the unit of the least command increment for each X axis and Y axis before shipment of the machine from the machine tool builder's factory.
3 If data are inputted by inch in a millimeter system machine, or if data are inputted by millimeter in an inch system machine, a tool offset error is produced within the sum of a half of the least input increment and half of the least command increment. This error is not accumulated.

Parameter

Refer to parameter No.16263 in the parameter manual (B–62780EN).

5.3 TURRET AXIS CONTROL (T AXIS CONTROL)

The tool positions corresponding to respective tool numbers on the turret axis are set in the tool registered screen in the unit of least command increment. By commanding one of T0 to T9999 using tape, MDI, or memory command, the control unit calculates the moving quantity of the turret axis from the position of a tool corresponding to this T code on the turret axis and the present turret axis position. The turret axis is positioned in such a direction that the absolute value of the moving quantity is less than 180 degrees. If the absolute value of the moving quantity is 180 degrees, the turret axis is positioned in the plus (+) direction. (Whether the plus direction means the clockwise direction or not depends upon machine tools).

Simultaneously when the positioning motion of the turret axis is started, the commanded decimal 4–digit numeral is sent by the binary code relay contact. The T code read command signal TF to give the code readout timing on the machine side is turned to 1 when the time (TMF) preset by a parameter (No. 3010) has passed after T code signal was sent. Read the T code signal and perform corresponding motion after reading signal TF on the machine side.

After completion of corresponding motion, turn signal FIN to 1. Turning signal FIN to 1 causes signal TF to be turned to 0. Turn signal FIN to 0 after signal TF has been turned to 0. Since signal transfer is just the same as in miscellaneous function (M function), the T code signal remains held until a new T code is commanded next.

If the turret axis has already been positioned when signal FIN goes to 0, the control proceeds to the next block. If the turret axis has not being positioned yet, the control proceeds to the next block after the turret axis has been positioned.

If an M code is commanded concurrently in a block to which a T code is previously commanded, signal FIN should be turned to 1 after all motions have been completed. When a workpiece is punched by press motion using a selected tool, a considerable load is applied to the turret. Accordingly, the turret is mechanically clamped by using shot pins or the like, in general. Since the mechanical clamping force of shot pins, etc. is stronger than the clamping force of the servo motor, the turret axis is usually set to the servo–off condition.

In such a case, the servo–off condition should be released to the servo–on condition to remove mechanical clamp of shot pins, etc. before starting the motion of the turret axis. The following description shows an example of processing to be made on the machine side in such a case.

Assume that interlock signal *ITT of the turret axis is 0 (interlock condition) and serve–off signal of the turret axis SVFT is 1 (servo–off condition) when the turret is being mechanically clamped by using shot pins or the like.

When T0 to T9999 are commanded by tape, MDI, or memory command, the control unit calculates the moving quantity of the turret axis as described previously.

The moving quantity of the turret axis is 0, if T01 is commanded when tool number 01 is being selected, for example. Let’s consider such a case that the moving quantity of the turret is not 0, i.e., the turret axis is positioned. The positioning of the turret is going to start soon after the T code was sent. However, the axial motion is not commanded because signal *ITT is 0, and the turret axis remains stopped in practice.

On the other hand, axis moving signal MVT is set to 1, even if signal *ITT is 0. Accordingly, whether the positioning of the turret axis is executed or not can be known by signal MVT on the machine side.
The above description may be summarized as follows. Signal MVT goes to 1 simultaneously when the T code is sent, and signal TF goes to 1 after the time TMF has passed. Read the T signal, and perform corresponding motion on the machine side when signal TF is 1. If signal MVT is 1 when signal TF is 1, release the turret axis from being mechanically clamped by shot pins or the like, and turn signal SVFT to 0 after confirming that the turret axis has been unclamped mechanically (Mount a timer on the machine side, if required). A current flows to the servo motor to excite it.
If the servo motor shaft was rotated by punch motion, etc., while signal SVFT is 1, the servo motor is going to be reset to the position where signal SVFT is 1 when signal SVFT is set to 0. After turning signal SVFT to 0, wait for a while on the machine side, and then, turn signal *ITT to 1 to start the motion of the turret axis. When the motion ends, signal MVT is turned to 0. Mechanically clamp the turret by means of shot pins or the like. However, since the turret axis is not always stopping mechanically due to the delay of servo motor, etc., when signal MVT goes to 0, clamp the turret mechanically by means of shot pins after signal MVT has been turned to 0, and turret axis in–position signal INPT has been turned to 1. In other words, turn signal *ITT and SVFT to 1 under the above condition. When signal SVFT is turned to 1, a current cannot be fed to the servo motor, causing the servo motor to be deenergized. Mechanically clamp the turret by using shot pins or the like on the machine side after the timer has reached the time–out point, if required. Turn signal FIN to 1 after the mechanical clamp and the motion for the T code have been completed on the machine side. When signal FIN is turned to 1, signal TF goes to 0. Then, when signal FIN is turned to 0, the control proceeds to the next process for example, press start signal PF is turned to 1 if a punch operation is to be performed in a block with a T–code command.
The above description covers an example only. If the mechanical clamp fails due to a certain cause, for example, its decision and processing should be made on the machine side. If the moving quantity of the turret axis is 0, the turret axis is not positioned, and it is not necessary to release its mechanical clamp, unlike in turret axis positioning.
In other words, if signal MVT is 0 when signal TF is 1, perform corresponding motion only according to the T code signal without releasing mechanical clamp.

CAUTION
1 The number of commandable T codes is limited to 136.
2 If the T–axis is absent, the T–code signal transfer only is done. Whether the T–axis is provided or not can be set by a parameter TCL (No. 16260#4)
3 Up to four digits of the T command are effective for control of the T–axis.

An example of a time chart for above description is shown below.

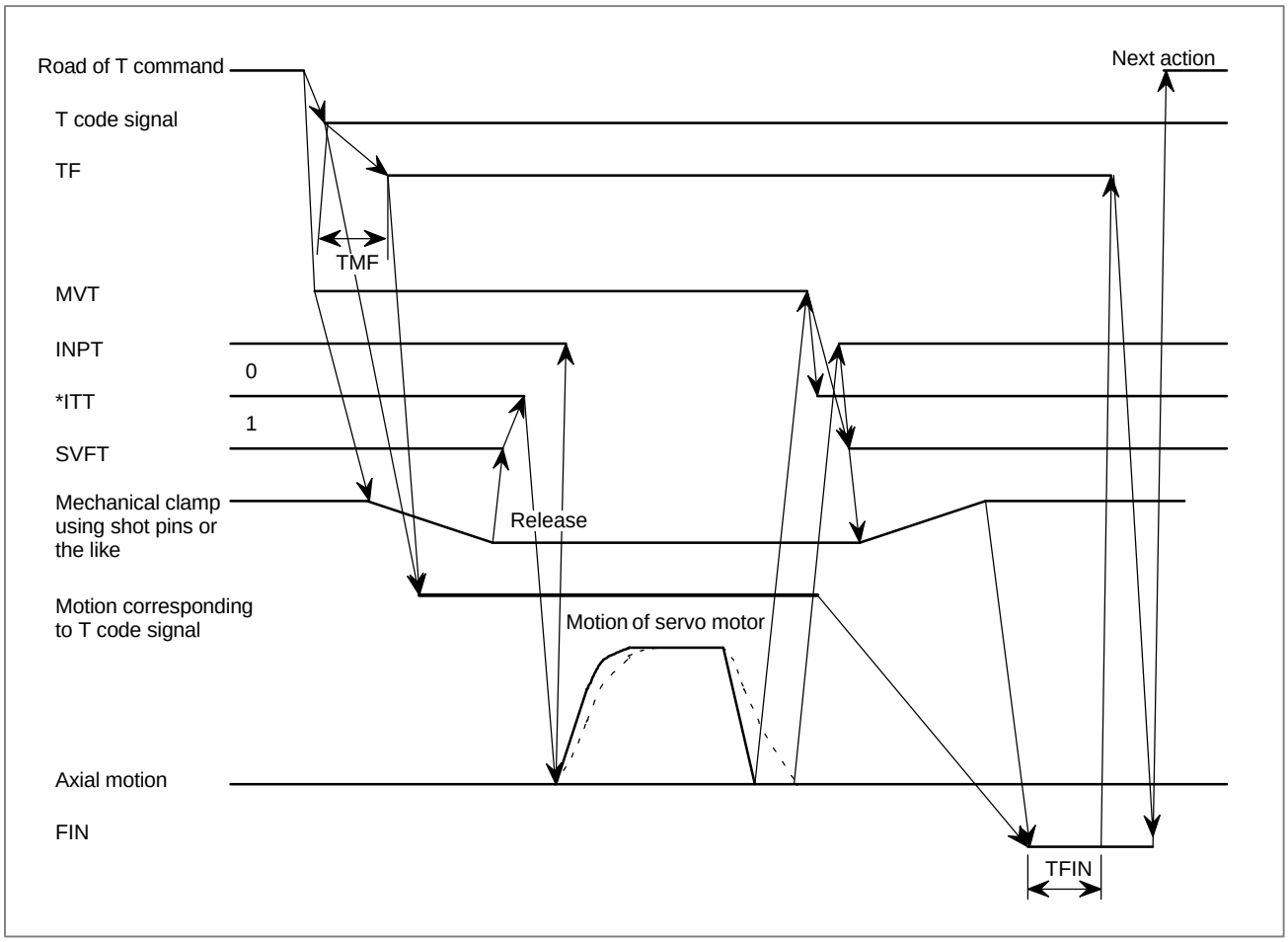


Fig.5.3(a)

If a feed axis & direction select signal + T of the turret axis is turned to 1 with jog feed selected, the turret axis moves in the + direction, and if – T is turned to 1, the turret axis moves in the –direction to select a tool, respectively.

In other words, when the feed axis & direction select signal is turned to 0, the turret axis moves to select the nearest tool in the decelerable/stoppable direction, and then stops. However, this motion is performed only after completion of manual reference position return of the turret axis after power on. In the period after the power is turned on or after the servo ready signal is set to 0 in case of emergency stop and before manual reference position return of the turret axis is completed and the machine position is determined, setting the feed axis direction select signal to 0 decelerates the turret axis movement and stops it immediately. It is requested, as described above, to move the turret axis after confirming that the turret has been released from being mechanically clamped with shot pins or the like in the jog feed mode as well as in the T–code command mode, if the turret is mechanically clamped by using shot pins, etc.

An example of the manual reference point return of the turret axis will be shown as a time chart below. When the manual reference point return of the turret axis is completed, the tool number preset in the tool registering screen is automatically selected. If it is desired to perform mechanical motion in correspondence with the selected tool when the manual reference point return has been completed, make sure that the feed axis & direction select signal has been turned to 1, or zero point return signal ZPT of the turret axis has been turned to 1 before starting such a work on the machine side.

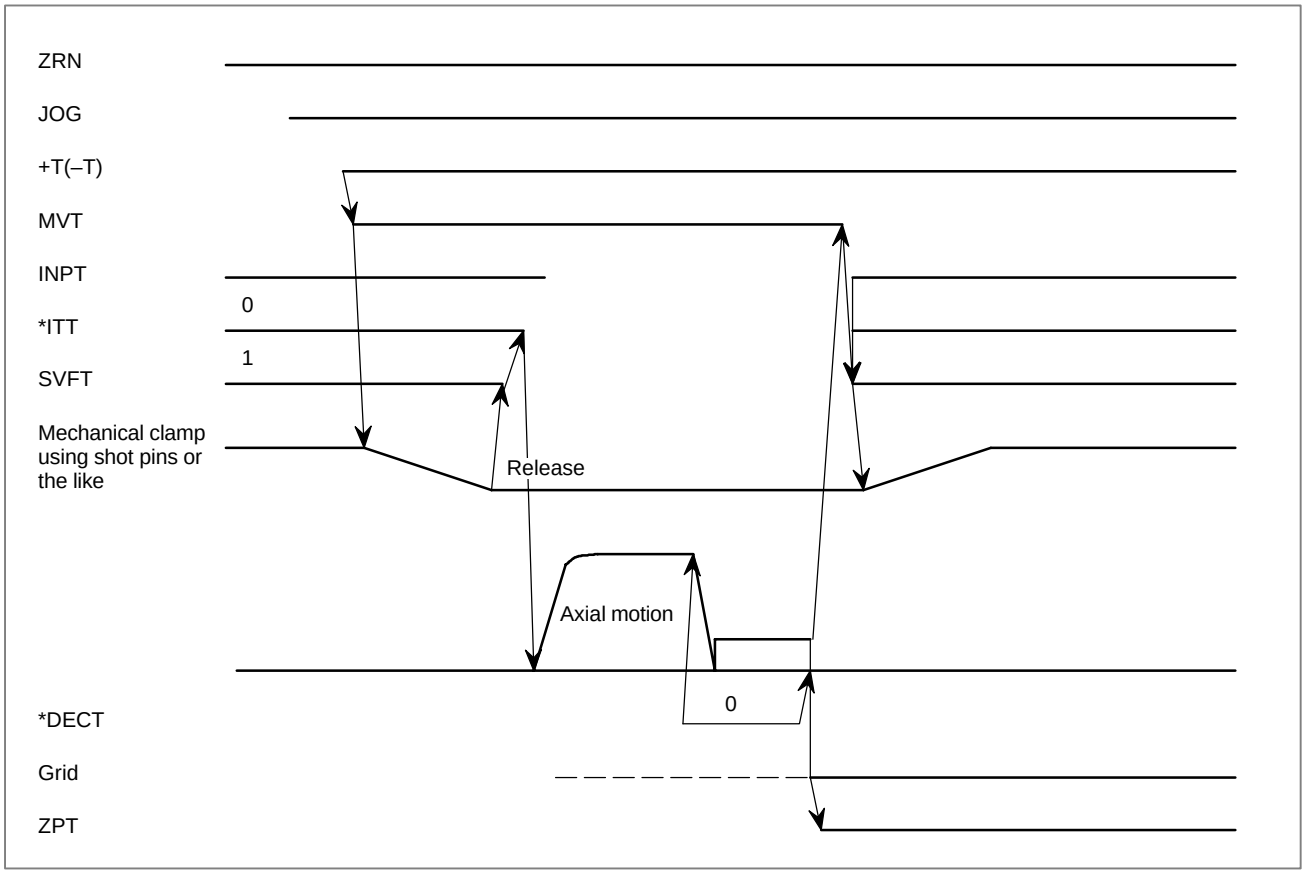


Fig.5.3(b) Manual Reference Point Return for Turret Axis

Parameter

Refer to parameters No.16260 to 16270 in the parameter manual (B–62780EN).

Reference item

III–11.4.3 Displaying and Setting Items on the Tool Registration Screens in Series 16/18/160/180–PC Operator's Manual B–62774EN/01.

5.3.1 T Command Neglect Signal (Input) TNG <G233#5>

If T command neglect signal (input) TNG is 1 when a T code is commanded by tape, MDI, or memory command, the commanded T code is ignored. When a block information is read and a T code is included in the block, the control unit calculates the moving quantity of the turret axis from the position of a tool corresponding to the commanded T code on the turret axis and the present turret axis position.
The signal TNG condition at this time is effective. When signal TNG is 1 at this time, the commanded T code is ignored, the moving quantity of the turret axis is not calculated signal TF is not turned to 1 after the executing block motion has been completed, and also the turret axis is not positioned. Be careful with the difference from auxiliary function lock signal AFL and press lock signal PFL.

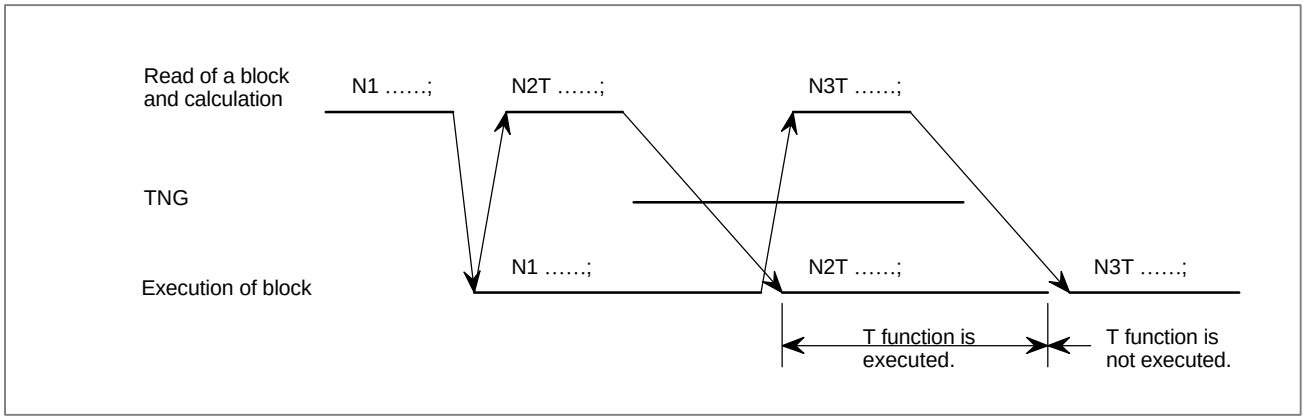


Fig.5.3.1

5.3.2 Tool Change Signal (Input) TCNG <G233#6>

When the T code is instructed by the tape, MDI, or Memory command, the turret axis positioning operation can be made to allow the tool corresponding to the the instructed T code to be selected. The position where the selected tool is punched by press operation and that where tools are replaced may differ depending on the machine. In this case, the tool to be replaced can be shifted to the replacement position in the manual mode. In addition, the tool to be replaced can be readily positioned to the replacement position by the following approach. Namely, the tool is positioned to the tool replacement position corresponding to the instructed T code, the T code of the tool to be selected is set to the punching position in the tool change screen of the tool registration screen beforehand. If the T code is instructed when the tool change signal (input) TCNG is 1 by the MDI command, the control unit carries out processing as if the T code set in the tool change screen was instructed for the instructed T code. In other words, the tool corresponding to the converted T code is positioned to the punching position and the converted T code is output as the T code signal.
The signal TCNG is valid only for the MDI command. Also, the signal TNG is valid even if the signal TCNG is 1.

5.3.3 Turret Indexing Completion Signal (Output) TIE <F236#6>

Although the turret axis control has been described in detail up to this point, the following is included in the description:
The above explanation shows only one example. For example, in the case of mechanical clamping, the machine side should judge that no clamping can be carried out for some reason and should also carry out some countermeasures for it.
The machine side should perform confirmation, judge to see if it is proper to carry out clamping, or perform some countermeasures, assuming that the position cannot be mechanically clamped for some reason (in general, shot pins cannot be inserted) when the turret axis was positioned at the location set in the tool registration screen for example, the commanded T code beforehand.
There is no problem even if the following conditions should occur as far as the machine side can perform the above confirmation.
Namely, when the T05 is commanded and turret indexing is performed while the T01 tools are being selected, the turret axis reduces its speed and stops and the axis moving signal MVT of the turret axis is turned to 0 at the same time the deceleration is terminated if the automatic operation stop signal *SP is turned to 0 during turret axis positioning.
Afterwards, in–position signal INPT of the turret axis is turned to 1 and becomes the automatic operation stop state.
Even if signal MVT is 0 and INPT is 1, the following results if the machine side judges whether shot pins can be inserted or not:
Namely, clamping by shot pins can be done if shot pins can be inserted and no clamping can be done if shot pins cannot be inserted.
When signal *SP is turned to 1 and automatic operation is restarted by cycle start, the turret axis can be positioned at the commanded T05 location and the T05 tools can be selected.
The following turret indexing completion signal TIE was added for reducing processing at the machine side.
In other words, signal TIE is 0 when the turret axis movement is started (including when interlock is applied to the axis movement with interlock signal *ITT of the turret axis set to 0), and it is 1 only when the positioning of the turret axis to the instructed T code is completed. Consequently, if it is confirmed that signal MVT is 0, signal INPT is 1 and signal TIE is 1, and the turret is clamped by a shot pin, it is not necessary to check whether it is a position in which to clamp at the machine side.

CAUTION

1 When turret indexing for an instructed T code is over, signal TIE is turned to 1; however the transmission timing is at the moment when deceleration begins. The turret must be neither clamped nor released from clamping, therefore, using a shot pin, etc. merely on receiving the signal TIE. The conditions of signals MVT and INPT must be applied.
2 After emergency stop or the servo alarm is released after power on, the TIE signal remains 0 until manual reference position return of the turret axis is completed, and the machine position of the turret axis is established.

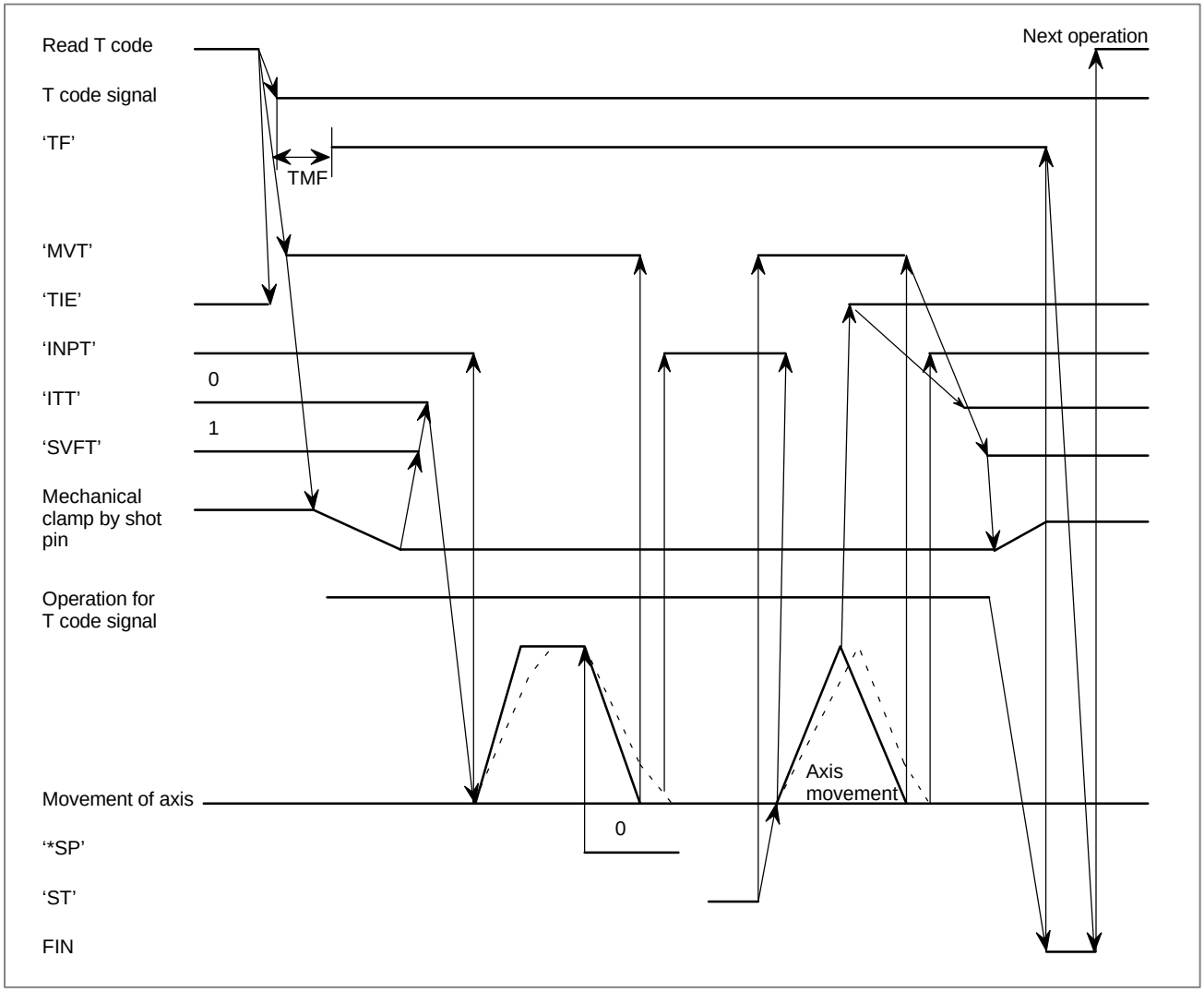


Fig.5.3.3

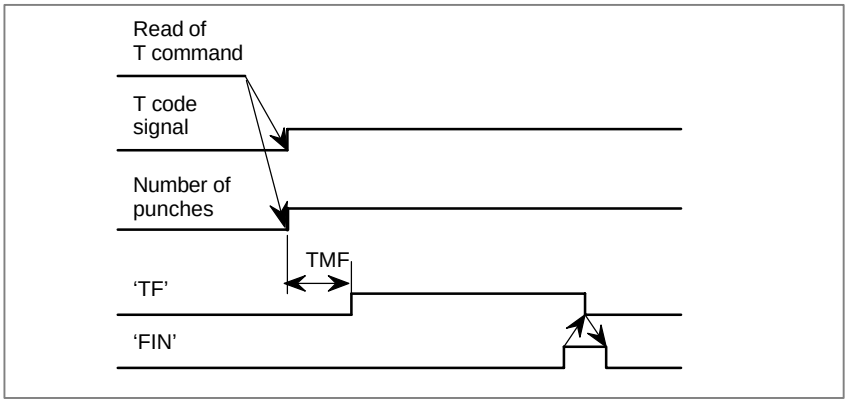
5.3.4 T Code Display Signal (Input) TI00 – TI32 <G234 – G237>

The station number (tool number) being set when the power is turned on is displayed on the position display screen. The number to be displayed should be set to a PMC address <G234 – G237>.
The station number (Tool number) selected present after the power supply is turned on can be displayed in the position display in case of the machine which does not do the turret indexing (No T axis control).
Input in binary code of 32 points.

CAUTION
1 Set the data to PMC address <G234 – G237> within 2 sec after power is turned on.
2 This signal becomes effective when parameter PWT (No. 16262#2) is set to 1.
When parameter JGT (No.16262#1)=1, T codes can be displayed on the position display screen in the Jog mode by the signals TI00 to TI31.

5.3.5 Number of Punches Signal (Output) PN00 – PN31 <F234 – F237>

The content of the number of punches of the tool until at that time which corresponds to T code instructed when T command is done is outputted to PMC from CNC.
It is a binary code signal of 32 points.
Correspond to the number of punches and the signal as follows.
Number of punches = 2□Pi (0 to 99999999)
However Pi = 0 when PNi is 0.
Pi = 1 when PNi is 1.



When the T command is done, T code signal is turned to 1 and the number of punches up to that point corresponding to the T code is outputted. The content is maintained until a new T command is done.

CAUTION
Signals PN00 to PN31 are 0 until T command is done for the first time after the power supply is turned on.

5.3.6 T–axis Machine Zero Point Position Signals RP1T – RP16T <F244, F245>

[Classification] Output signal

[Function] These signals notify the PMC that the machine coordinate of the T–axis agrees with the machine zero point. The PMC is thus aware of the special tool indexing point.

[Output condition] When the machine coordinate of the T–axis becomes the position set in parameters 16680 to 16695 under T–axis control, the bit signal corresponding to the parameter is output.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F244	RP8T	RP7T	RP6T	RP5T	RP4T	RP3T	RP2T	RP1T
F245	RP16T	RP15T	RP14T	RP13T	RP12T	RP11T	RP10T	RP9T

Parameter

16680	Position of machine zero point 1 on T–axis
16681	Position of machine zero point 2 on T–axis
16682	Position of machine zero point 3 on T–axis
16683	Position of machine zero point 4 on T–axis
16684	Position of machine zero point 5 on T–axis
16685	Position of machine zero point 6 on T–axis
16686	Position of machine zero point 7 on T–axis
16687	Position of machine zero point 8 on T–axis
16688	Position of machine zero point 9 on T–axis
16689	Position of machine zero point 10 on T–axis
16690	Position of machine zero point 11 on T–axis
16691	Position of machine zero point 12 on T–axis
16692	Position of machine zero point 13 on T–axis
16693	Position of machine zero point 14 on T–axis
16694	Position of machine zero point 15 on T–axis
16695	Position of machine zero point 16 on T–axis

[Data type] Two–word

[Unit of data] Least command increment for T–axis

[Valid data range] 0 to 99999999

When the machine coordinate of the T–axis matches a position specified in parameters 16680 to 16695, the corresponding signal RP1T to RP16T (F244, F245) is output.

WARNING

If the servo–off signal is used under T–axis control, setting the *FLWU signal (G007, #7) to 0 causes the follow–up function to operate at servo–off. This sets the TIE and RP1T to RP16T signals to 0. T–axis indexing may not be executed correctly. To prevent such problems from occurring, set the *FLWU (G007, #7) signal to 1.

5.4 MULTIPLE–TOOL CONTROL (OUTPUT) CMOK <F232#5>, MIE <F232#7>

General

A common way of handling the tools, i.e., the metal dies, for punch presses is to store each die in the tool holder. To select a tool, it is necessary to index the magazine containing the required tool holder to the position at which the tool is changed with a T command and to set the tool holder on the punch holder at that position.
In the multiple–tool system, however, one tool holder contains different types of metal dies. This makes it possible to change tools by simply moving the relevant tool holder. The multiple–tool system, therefore, reduces the time taken to change tools and enables a larger variety of tools to be used, enhancing the range of machining that can be performed.

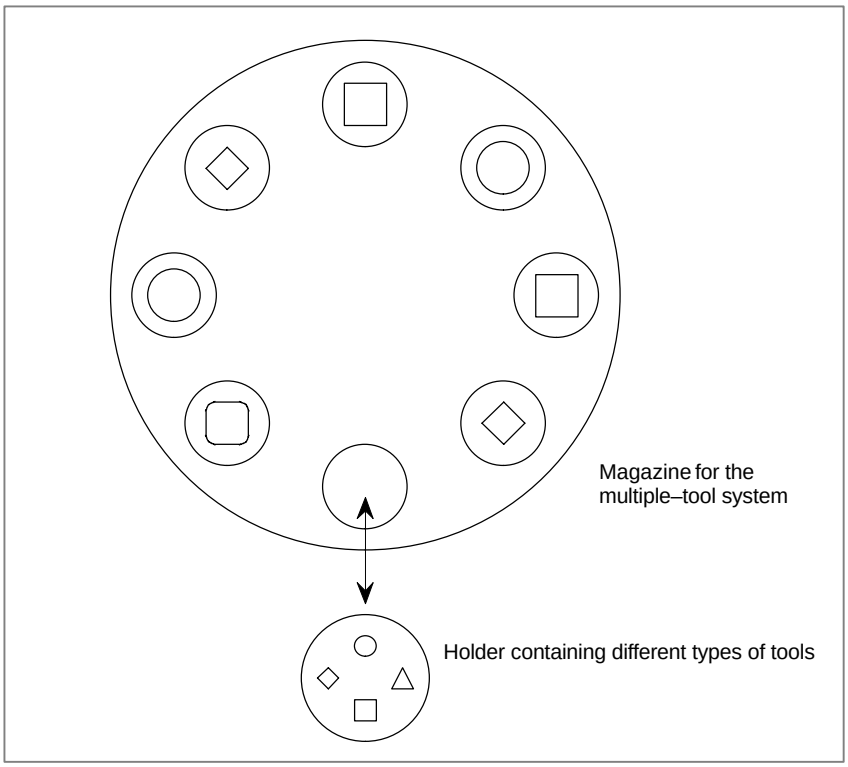
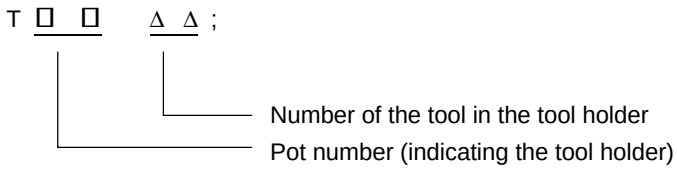


Fig.5.4(a)

Function

1) Pot number

In the multiple–tool system, each tool holder is assigned a pot number specified as part of a three–or four–digit T code as follows:



When the CNC controls the indexing of the tool holder, the T code shall contain a two–digit pot number from 00 to 99 as shown above. Specify a three–or four–digit T code to use the multiple–tool system. The first two digits are the pot number and the last two digits are the number of the tool. When the T code is specified, a binary code signal is sent to the machine.

CAUTION

1 The control numbers that the CNC uses to control the indexing of the tool holders must be registered in the tool registering screen. The number of parameters registered must not exceed the number of tools controlled by indexing (136 maximum).
2 The control numbers that the CNC uses to control the indexing of the tool holders that are not part of the multiple–tool system must be specified by a T code consisting of two digits or less.
3 The tool numbers in the multiple–tool system must be set in the tool registering screen for multiple–tool of tool registering screen.

2) Relationships between the multiple–tool system and the C axis

The tools in the multiple–tool system are selected as the C axis rotates. When a tool is selected, it is positioned at the tool reference position. The line passing through the tool reference position and the center point of the tool holder is parallel to the Y axis.

Specify the angular displacement of each tool from the tool reference position in the tool registering screen for multiple–tool. When T code for the tool is specified, the CNC reports the angular displacement to the C axis and tool is positioned at the reference position.

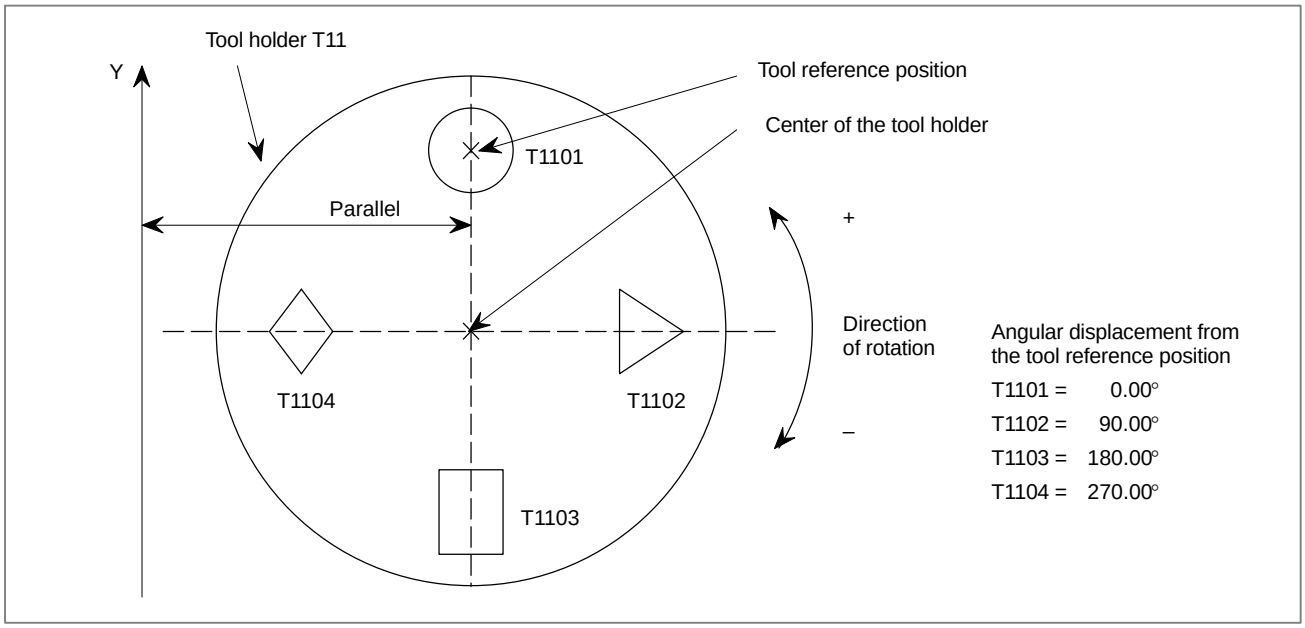


Fig. 5.4(b)

CAUTION
When the angular displacement from the tool reference position is set to 0, the tool that is selected when the tool holder has been indexed is used.

3) Tool position compensation

The centers of the tools move from the center of the tool holder when using the multiple–tool system. Therefore, the tool positions need to be compensated. However, the compensation function in the multiple–tool system only operates in the direction of the Y axis. Therefore, specify the distance between the tool reference position and the center of the tool holder for each tool in the tool registering screen for multiple–tool as the Y–axis compensation.

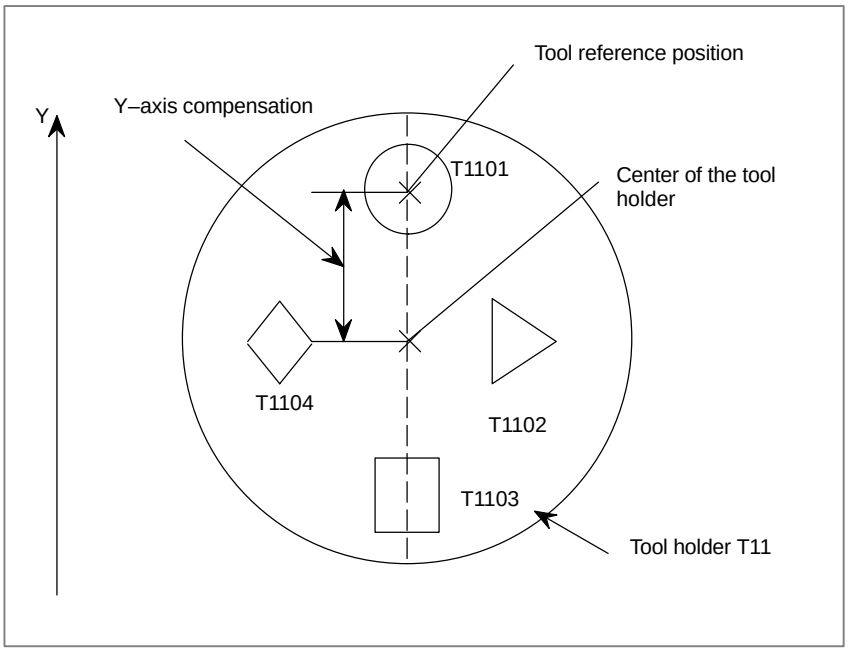


Fig 5.4(c)

CAUTION
To use the compensation function, set parameter TDF (No. 16263#2) to 1.

Tool change

The following tool changes are possible using the multiple–tool system.

(1) A regular tool (not in a multiple–tool holder) to another regular tool
(2) A regular tool to a tool in a multiple tool holder
(3) A tool in a multiple–tool holder to a regular tool
(4) A tool in a multiple–tool holder to a tool in another multiple–tool holder
(5) A tool in a multiple tool holder to another tool in the same multiple tool holder

Tool changes (2) to (5) are described below.

(2) A regular tool to a tool in a multiple tool holder

If the C axis is not positioned at the reference position when the T command to select a tool in the multiple–tool system is issued, the C axis automatically returns to the reference position.

Then, the necessary T code and the corresponding TF signal are output. The machine returns the FIN signal as soon as the tool has been indexed, as specified by the first two digits of the T code.

When the FIN signal is returned, the CNC moves the C axis as specified by the last two digits of the T code and starts indexing the required tool. When indexing the tool starts, the tool indexing signal, CMOK, is switched to 1. When the tool has been indexed, the MIE signal (the completion signal) is switched to 1 and the CMOK signal is switched to 0. This completes indexing the tool.

Tool selection command T1102

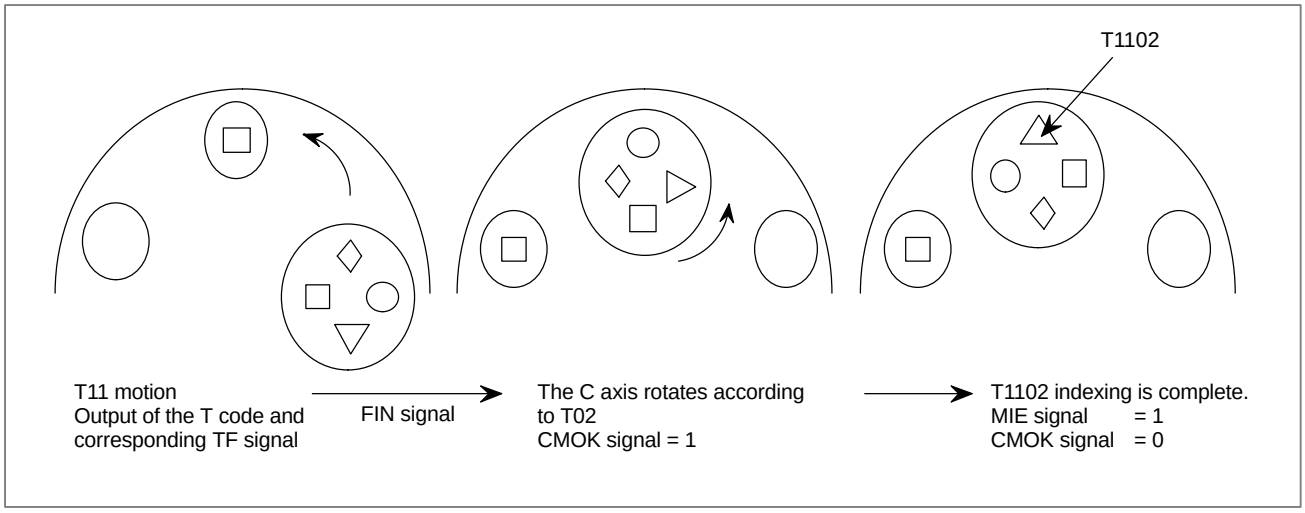
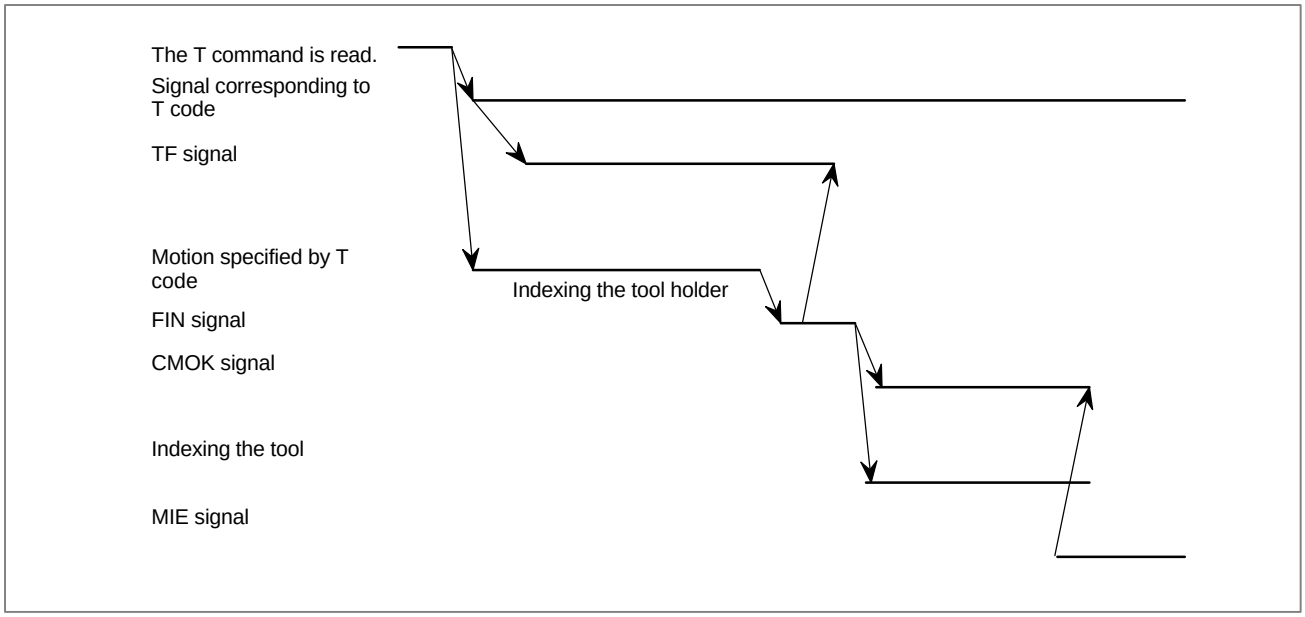


Fig.5.4(d)

Timing diagram



(3) A tool in a multiple–tool holder to a regular tool

If the C axis is not positioned at the reference position when the T command for selecting a regular tool is issued after a tool in a multiple tool holder is selected, the C axis automatically returns to the reference position. Because the tool in a multiple tool holder is moved with this command, the CMOK signal is switched to 1 and the MIE signal is switched to 0. When the C axis has returned to the reference position, the CMOK signal is switched to 0.

Then, the necessary T code and the corresponding TF signal are output. The machine returns the FIN signal as soon as the tool has been indexed, as specified by the T code. This completes indexing the tool.

Regular tool selection command T10

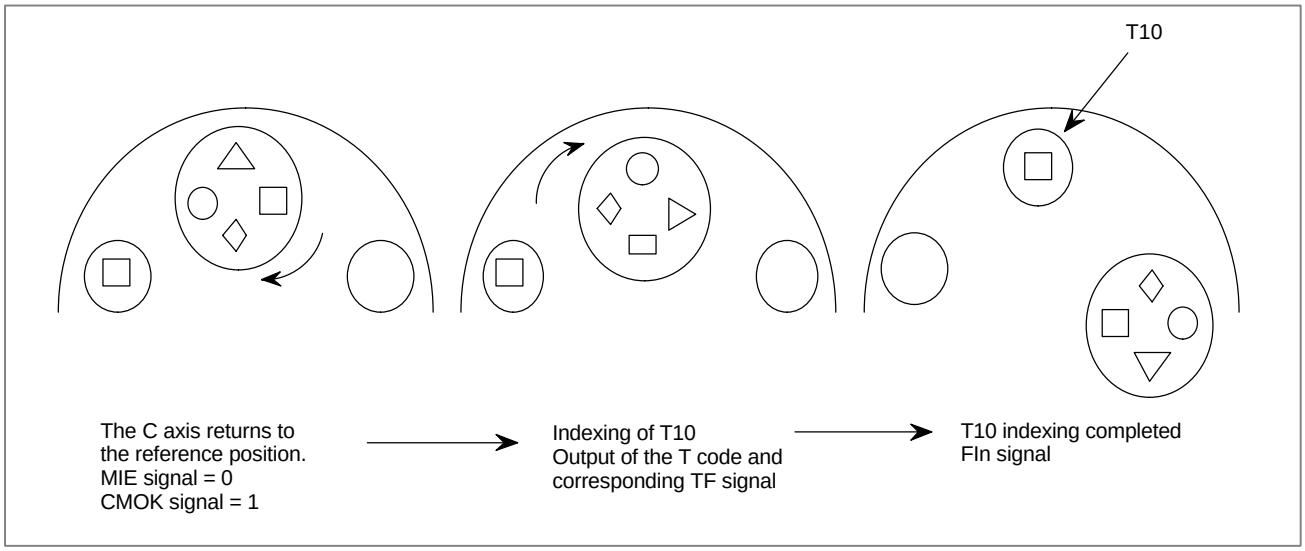
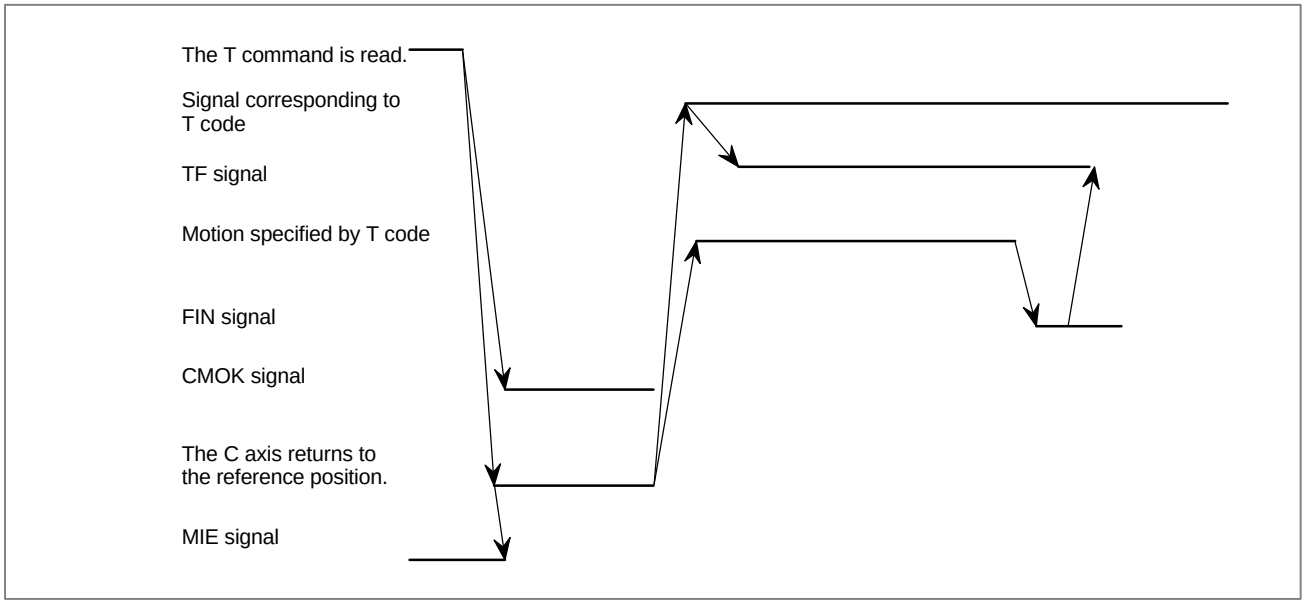


Fig.5.4(e)

Timing diagram



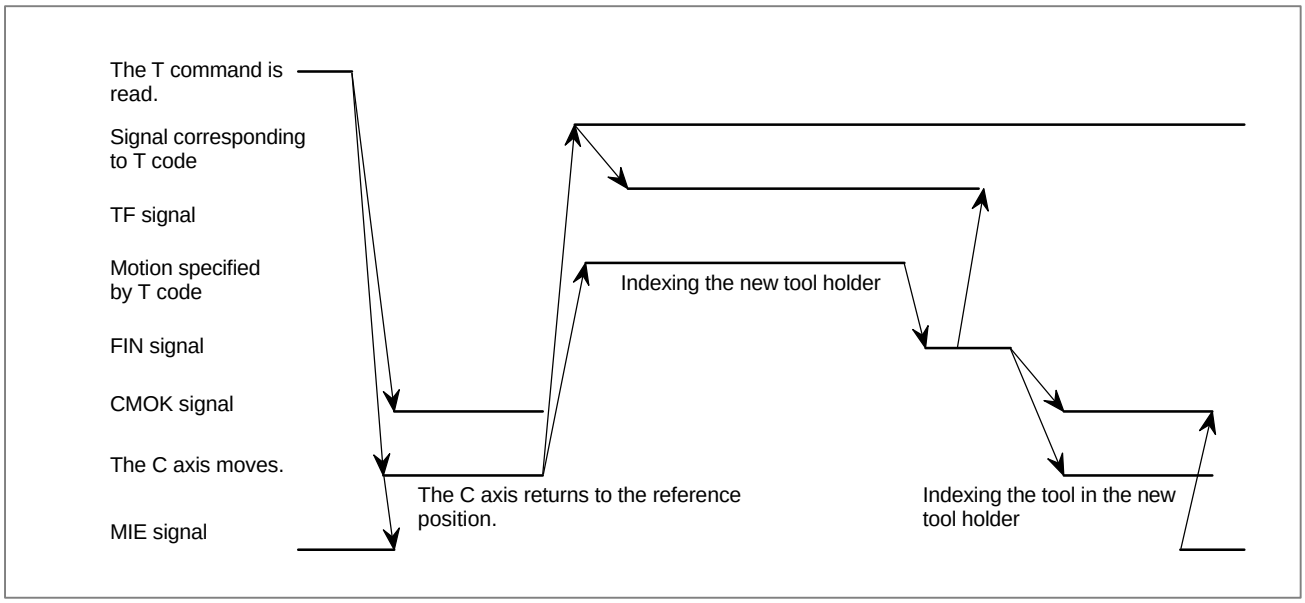
(4) A tool in a multiple–tool holder to a tool in another multiple–tool holder

If the C axis is not positioned at the reference position when the T command for selecting a tool in a multiple–tool holder is issued after a tool in another multiple tool holder is selected, the C axis automatically returns to the reference position. Because the tool in the first multiple tool holder is moved with this command, the CMOK signal is switched to 1 and the MIE signal is switched to 0. When the C axis has returned to the reference position, the CMOK signal is switched to 0.

Then, the necessary T code and the corresponding TF signal are output. The machine returns the FIN signal as soon as the tool has been indexed, as specified by the first two digits of the T code.

When the FIN signal is returned, the CNC moves the C axis as specified by the last two digits of the T code and starts indexing the required tool. When indexing the tool starts, the tool indexing signal, CMOK, is switched to 1. When the tool has been indexed, the MIE signal is switched to 1 and the CMOK signal is switched to 0. This completes indexing the tool.

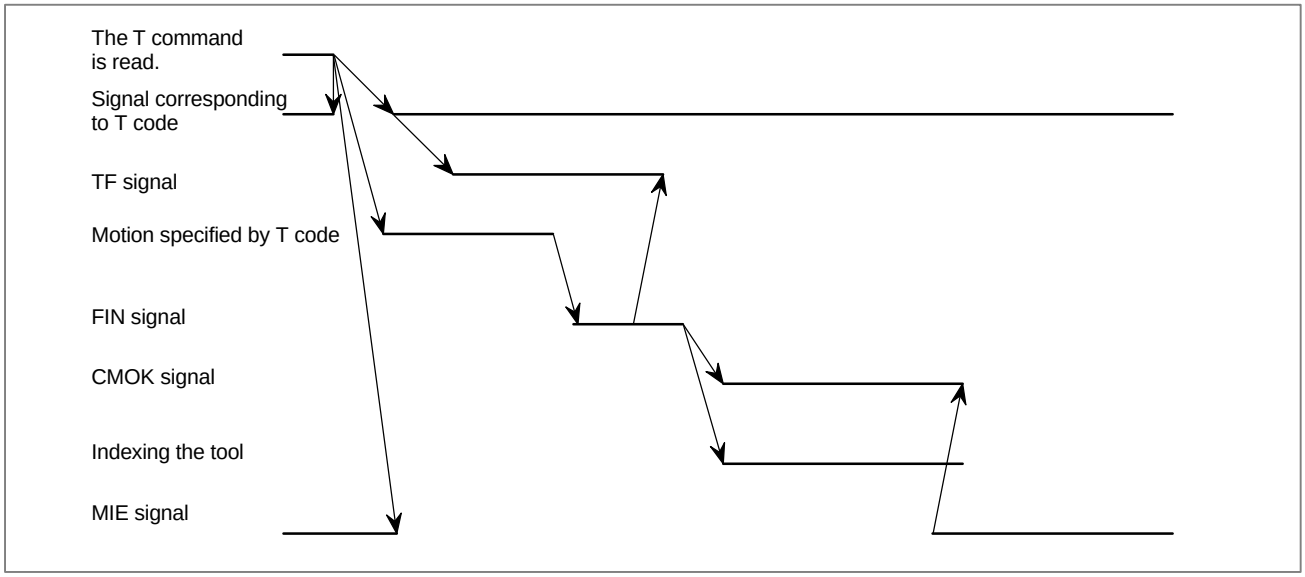
Timing diagram



(5) A tool in a multiple tool holder to another tool in the same multiple tool holder

When the T command for a tool in a multiple–tool holder is issued after another tool in the same tool holder has been selected, the necessary T code and the corresponding TF signal are output. The machine then returns the FIN signal. (If necessary, move the tool holder according to the T command, and return the FIN signal after completion.)
When the FIN signal is returned, the CNC moves the C axis as specified by the last two digits of the T code and starts indexing the required tool. When indexing the tool starts, the tool indexing signal, CMOK, is switched to 1. When the tool has been indexed, the MIE signal is switched to 1 and the CMOK signal is switched to 0. This completes indexing the tool.

Timing diagram



Caution on using the multiple–tool system

- It is necessary to combine the C–axis control option to use multiple–tool control
- When using multiple–tool control, do not issue a C–axis command.
- When using multiple–tool control, set parameter No. 3032 to 3 or 4. To index a tool in a multiple–tool holder, specify a T code consisting of at least three digits. To index a regular tool, specify a T code consisting of two digits or less.
- The MIE signal, which indicates that indexing the tool in a multiple–tool holder is complete, is switched to 1 when a tool has been indexed. However, switching the MIE signal to 1 is synchronized with the start of C–axis deceleration.
- When using multiple–tool control, set parameter CRM (No. 16362#5) to 0.

✍ It is possible to create an image of a tool in the multiple–tool system on the graphic screen. Specify the tool contour on the tool contour setting screen. Tools in the multiple system are assigned numbers 101 to 164, as specified by the T code. The procedure for inputting and outputting the contours of tool in the multiple–tool system is the same as for regular tools.

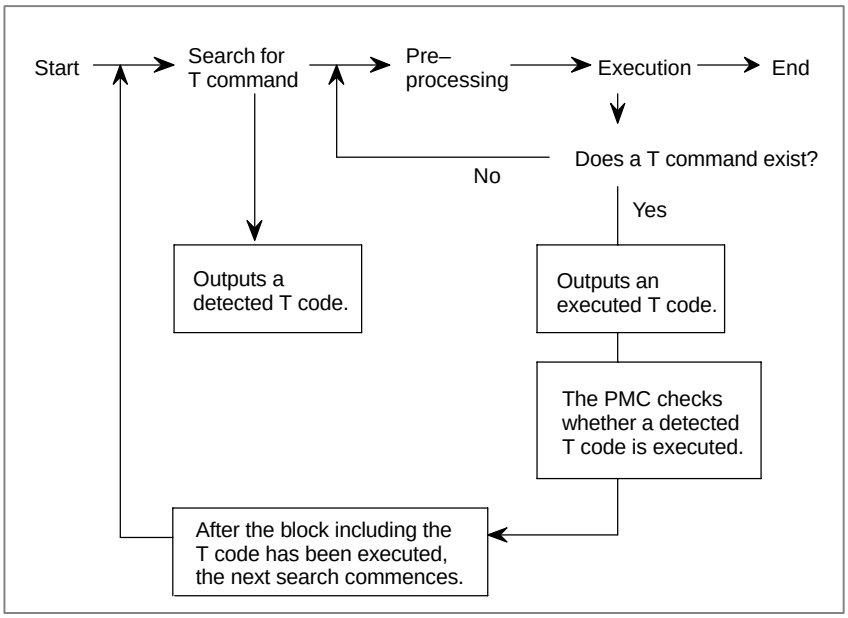
Reference item

Series 16–PC Operator's manual
II–11.5 Multiple Tool Control
III–11.4.3 Displaying and Setting Items on the Tool Registration screen.

5.5 T–CODE PRE–ISSUE FUNCTION

General

In automatic operation, this function searches the program to be executed for T commands and outputs the T codes in advance. The machine can prepare for tool change before execution of the tool change command.



Basic procedure

When automatic operation starts, the CNC reads up to 30 blocks of the program to be executed and stores them into the ring buffer. While executing a program, the CNC always stores up to 30 blocks to be executed in its buffer. If T commands are detected in the buffer, corresponding binary codes are output to PMC addresses F246 to F249 as pre–read T code signals in the order in which they are executed. At the same time, T–code read command signal BTF is inverted. Upon detecting this inversion, the PMC should read the pre–read T–code signals and execute the necessary processing. After completing the operation indicated by the pre–read T–code signal, the PMC should invert the BTFIN signal.

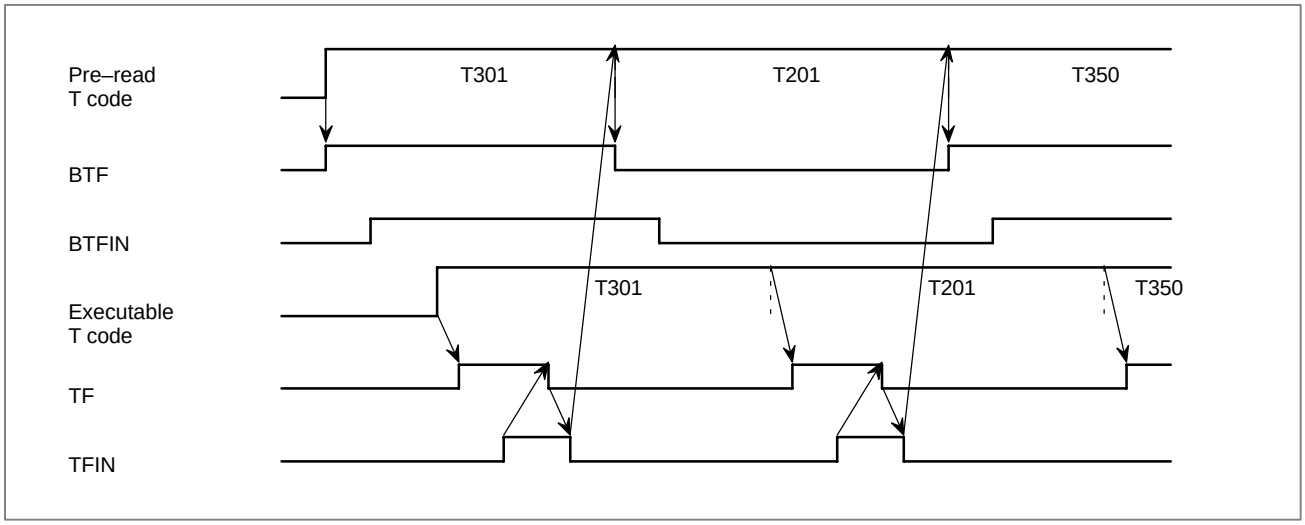
The BTF and BTFIN signals have the following functions: When the BTF and BTFIN signals have different bit states, the CNC outputs the pre–read T–code signal. After checking that the pre–read T–code operation has been completed, the PMC inverts the bit of the BTFIN signal. This ends the pre–read T–code operation. In other words, the opposite logical states of the BTF and BTFIN signals indicate that the CNC is issuing a pre–read T–code signal. When the signals have identical states, the next pre–read T–code can be output.

Once the pre–read T–code operation has been completed, the actual T command is executed. The command is executed under usual T function control. Once this T command has been processed, a subsequent pre–read T–code is output. That is, a pre–read T code is output after the current T code has been executed while the BTF and BTFIN signals have identical states.

The pre–read T code processing is separate from the usual T code processing. The machining program does not stop, instead continuing even while the CNC is waiting for BTFIN. If the CNC is waiting for BTFIN when the actual T command is executed, however, the program enters the wait state.

Examples

```
G90G92X1800. Y1500. ;
G00X500. Y500. T301 ;
G26I50. J0. K8 ;
G72X800. Y500. ;
G26I50. J0. K8T201 ;
G00X1000. Y1000. T350 ;
%
```



Reset processing

When all blocks have been pre–read, the CNC sets the pre–read T code signal to 0 and inverts the BTF signal to indicate that it has reached the end of the program. If necessary, the PMC should invert the BTFIN signal.
If the RESET key is pressed, an emergency stop is instigated, or other reset processing is executed during the output of a pre–read T code signal, the CNC executes the following reset processing: Sets the bit of the BTF signal to the bit of the BTFIN signal; Sets the pre–read T code signal to 0. The CNC clears the thirty blocks stored in the buffer, assuming that they have already been executed.

Executing a special program

To enable T–code pre–reading, a buffer that can store up to 30 blocks is provided. T codes detected in the buffer are output as pre–read T code signals. Operations and variable rewriting of custom macro statements are not regarded as being CNC statements. They are processed before being stored in the buffer. Some macro programs cannot be executed.

Examples

```
O0100 ;                                  O9500 ;
N1 G92 X1800. Y1500. ;                   N10 #100 = 1.
N2 G90 G00 X1000. Y1000. T400 ;          N20 IF [#5001 EQ 0. ] GOTO 50 ;
N3 G65 P9500 X500. Y500. T500 ;          N30 G90 G00 X#24 Y#25 T#20 ;
N4 G72 X100. Y100. ;                     N40 GOTO 60 ;
N5 G26 I10. J0 K4 T600 ;                 N50 T#100 ;
    ⋮                                    N60 M99 ;
```

The sample macro program is executed in the following sequence: N10, N20, N50, then N60. Pre–read T codes for T400, T1, and T600 are output in this order.

The G10 command is also processed before being stored.

Signal

Pre–issued T code signal BT00 – BT31 <F246 – F249>

Pre–issued T code strobe signal BTF<F232#1>

[Classification] Output signal

[Function] Informs a pre–issued T code has been specified.

[Output condition] Refer to the item of “output condition” and “basic procedure”

Pre–issued T code complete signal BTFIN<G248#1>

[Classification] Input signal

[Function] Informs that processing a pre–issued T code has been completed.

[Operation] Refer to the item of “Operation” and “Basic procedure”.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G248							BTFIN	
F232							BTF	
F246	BT07	BT06	BT05	BT04	BT03	BT02	BT01	BT00
F247	BT15	BT14	BT13	BT12	BT11	BT10	BT09	BT08
F248	BT23	BT22	BT21	BT20	BT19	BT18	BT17	BT16
F249	BT31	BT30	BT29	BT28	BT27	BT26	BT25	BT24

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
16260							BST	

[Data type] Bit

BST The function used to output a T code beforehand is:

0 : Disabled.
1 : Enabled.

Limitations

1. In MDI mode, a single block is issued in advance.
2. Manual operation intervention is not possible.
3. The operation can be resumed after feed hold, but not after the following:
   - Program number search, sequence number search
   - Finding the beginning of a program
   - Editing
4. If an M code that must not be buffered is specified, subsequent T codes are not pre–issued before the M code has been executed.

   G90 G00 X1000. Y1000. ; □ Current block
   M00 ;
   G72 X100. Y100. ;
   G26 I10. J0 K4 T600 ; □ Output only after M00 is executed

5.6 TOOL DATA SETTING FUNCTION

General

This function makes the area used for registering tool numbers, tool position compensation, punch count, and other tool data available to the user. This enables the user to customize the registered tool data. This allows the conventional tool data setting method to be changed and the number of tools that can be registered to be increased considerably.

Registration area

A memory area consisting of 16KB (16□1024 bytes) is provided for the tool data setting function.

Type and size of tool data

Tool data can be customized, as listed below, by specifying parameters. Individual tools cannot have more than one setting. All registered tools will have the same setting.

Size (byte) / Data	0	2	4	Description
Tool number	×	□	□	2 bytes: T command having up to 4 digits 4 bytes: T command having up to 8 digits
Tool position compensation X, Y	□	□	□	2 bytes: –32768 to +32767 4 bytes: 0 to □99999999
T–axis machine position	□	×	□	4 bytes: 0 to 99999999
Punch count	□	□	□	2 bytes: 0 to 65535 4 bytes: 0 to 99999999
Tool number for tool change	□	×	×	When used, 2 bytes: T command having up to 4 digits 4 bytes: T command having up to 8 digits
Tool figure for graphics(*1)	□	×	×	Always 13 bytes
Tool life management(*2)	□	×	×	When used, 2 bytes: 0 to 65535 4 bytes: 0 to 99999999
Multitool subtool number(*3)	□	□	×	2 bytes: Up to 4 digits Magazine number plus subtool number can be registered.
Multitool angle(*3)	□	×	□	4 bytes: 0 to 360000 deg
Multitool tool position compensation X, Y(*3)	□	□	□	2 bytes: –32768 to +32767 4 bytes: 0 to □99999999
Tool figure for multitool graphics(*3)	□	×	×	Always 13 bytes

*1 The graphic function is necessary.
*2 The tool life management function is necessary.
*3 The multitool control function is necessary.

□ : Can be selected by setting a parameter
× : Cannot be selected by setting a parameter

Tool number

When specifying up to four digits with a T command, select 2 bytes. When specifying five or more digits with a T command, select 4 bytes. If the 4–byte option is selected, T–axis control is possible with a T command having five or more digits.

X–axis and Y–axis tool position compensation

Enable or disable tool position compensation. A range of either two or four bytes can be selected thus enabling compensation values to be set in the corresponding range, in output increments.

T–axis machine position

Set this when T–axis control is used.

Punch count

Select whether to use individual punch counts. A range of either two or four bytes can be selected. This item must be set when the tool life management function is used.

Tool change tool number

Set this when using the tool change function. The data agrees with that of the tool number.

Tool figure

Select this when drawing a tool using the graphic function. Each tool requires 13 bytes of data.

Figure data	: 1 byte
Vertical dimension data	: 4 bytes
Horizontal dimension data	: 4 bytes
Angle data	: 4 bytes

Tool life management

Select this when using the tool life management function. The data agrees with that of the punch count.

Multitool subtool number

Select this when using multitool control. Up to four digits can be specified for a subtool number when the corresponding number of digits is registered.
This item can also be used to specify whether a magazine number and subtool number are registered together or separately. When separately registering the magazine number and subtool number, the number of subtools to be registered must be set as a parameter.

Multitool angle

Set a subtool indexing angle when using multitool control. If the magazine number and subtool number are registered together, the angle for tools other than the multitool must be set to 0.

Multitool tool position compensation

Set this when executing multitool tool position compensation in the X and Y directions.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
16280	UTL	UTS	UCT	UPC	UTC	UOY	UOX	UT8

[Data type] Bit

This parameter is valid when the function used to set tool data is specified. In the following description, n represents the number of tools to be stored.

UT8 As a tool number:

0 : Up to eight digits can be input. (n 4–byte numbers)
1 : Up to four digits can be input. (n 2–byte numbers)

UOX A tool position compensation value along the X–axis is:

0 : Not stored.
1 : Stored. See the description of the OX4 bit (bit 1 of parameter 16281).

UOY A tool position compensation value along the Y–axis is:

0 : Not stored.
1 : Stored. See the description of the OY4 bit (bit 2 of parameter 16281).

UTC Under T–axis control, the machine position on the T–axis is:

0 : Not stored.
1 : Stored. The valid data range is 0 to 99999999. (n 4–byte values)

Note 1 When T–axis control is used (TCL bit (bit 4 of parameter 16260) is 1), this bit should be set to 1.

UPC The punching count of an individual tool is:

0 : Not stored.
1 : Stored. See the description of the PC4 bit (bit 4 of parameter 16281).

Note 1 When the tool life management function is used, this bit should be set to 1.

UCT Tool numbers for changing tools are:

0 : Not stored.
1 : Stored. The number of digits is the same as that for the UT8 bit (bit 0 of parameter 16280).

UTS A graphic tool figure is:

0 : Not stored.
1 : Stored. (n 13–byte values)

UTL The tool life management data is:

0 : Not stored.
1 : Stored. The data is stored in the same way as for the PC4 bit (bit 4 of parameter 16281).

	#7	#6	#5	#4	#3	#2	#1	#0
16281				PC4		OY4	OX4	

[Data type] Bit

This parameter is valid when the function to set tool data is specified. In the following description, n represents the number of tools to be stored.

OX4 For the tool position compensation value along the X–axis:

0 : n four–byte values can be stored. The valid data range is 0 to □99999999.

1 : n two–byte values can be stored. The valid data range is –32768 to +32767.

OY4 For the tool position compensation value along the Y–axis:

0 : n four–byte values can be stored. The valid data range is 0 to □99999999.

1 : n two–byte values can be stored. The valid data range is –32768 to +32767.

PC4 For the punching count of individual tools:

0 : n four–byte values can be stored. The valid data range is 0 to 99999999.

1 : n two–byte values can be stored. The valid data range is 0 to 65536.

	#7	#6	#5	#4	#3	#2	#1	#0
16282		MTS	CMT			MOY	MOX	

[Data type] Bit

This parameter is valid when the function used to set tool data is specified. In the following description, m represents the number of subtools stored for a multi–tool.

MOX The tool position compensation value along the X–axis for a multi–tool is:

0 : Not stored.

1 : Stored. See the description of the MX4 bit (bit 1 of parameter 16283).

MOY The tool position compensation value along the Y–axis for a multi–tool is:

0 : Not stored.

1 : Stored. See the description of the MY4 bit (bit 2 of parameter 16283).

CMT The tool numbers for a multi–tool:

0 : Are the magazine number plus subtool number. When this is selected, m equals n (number of tools stored).

1 : Consist of the Magazine number and subtool number, which are separately stored. (m 2–byte numbers)

MTS The graphic tool figure for a multi–tool is:

0 : Not stored.

1 : Stored. (n 13–byte data items)

Note 1 This bit is valid when the CMT bit (bit 5 of parameter 16282) is set to 1.

	#7	#6	#5	#4	#3	#2	#1	#0
16283						MY4	MX4	

[Data type] Bit

This parameter is valid when the function used to set tool data is specified. In the following description, m represents the number of subtools stored for a multi–tool.

MX4 For the tool position compensation value along the X–axis for a multi–tool:

0 : m four–byte values can be specified. The valid data range is 0 to □99999999.

1 : m two–byte values can be specified. The valid data range is –32768 to +32767.

MY4 For the tool position compensation value along the Y–axis for a multi–tool:

0 : m four–byte values can be specified. The valid data range is 0 to □99999999.

1 : m two–byte values can be specified. The valid data range is –32768 to +32767.

16284	Number of tools to be stored

[Data type] Word

[Valid data range] 0 to

This parameter specifies the number of tools to be stored for the function used to set tool data.

16285	Number of digits in the tool number of an subtool for a multi–tool

[Data type] Word

[Valid data range] 0 to 4

This parameter specifies the number of digits that can be specified for the tool number of an subtool for which multiple tool control is executed by the function used to set tool data.

16286	Number of subtools for a multi–tool

[Data type] Word

[Valid data range] 0 to

This parameter specifies the number of subtools for which multiple tool control is executed by the function used to set tool data.

Sample registration data

Conditions: T–axis machine position of 6 digits; 300 tools to be registered; tool number consisting of 8 digits; X, Y position compensation consisting of 6 digits; punch count of 8 digits; The graphic function and tool life management function are provided.

Data to be used for a single tool

T–axis machine position	4 bytes
Tool number	4 bytes
Position compensation	8 bytes
Punch count	4 bytes
Graphic	13 bytes
Tool life management	4 bytes
Subtotal	37 bytes

□Total 300□37 = 11100 bytes
(about 11KB)

The values of the parameters follow:

	#7	#6	#5	#4	#3	#2	#1	#0
16280	1	1	0	1	1	1	1	0
16281	–	–	–	0	–	0	0	–
16282	–	0	0	–	–	0	0	–
16283	–	–	–	–	–	0	0	–
16284	300							
16285	0							
16286	0							

Alarm

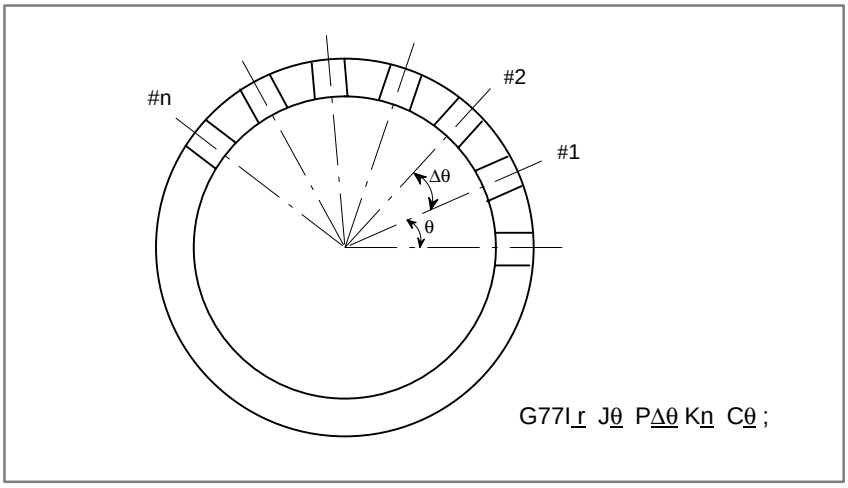
No.	Message	Contents
4549	ILLEGAL TOOL DATA FORMAT	The size of the registered tool data patterns has exceeded the upper limit of 16KB.

6 C–AXIS CONTROL

6.1 C AXIS CONTROL (DIE ANGLE INDEXING)

For predetermined dies (tools) on a turret, the angular position of the die can be changed with a command from a tape, a memory or MDI.
In the past, it was necessary to use many dies when the die shape is the same but the die arrangement is different. Even in such a case, this new function makes it possible to conduct the operations only with one die since the function can freely change the angular position of the die. Further, since chamfering of four corners of a workpiece can be performed only with one die, the time required for die change is reduced, resulting in shorter machining time.
Further, in pattern function for a circular geometry bolt hole circle, arc pattern, and arc nibbling function, the C axis is automatically controlled so that one side of the die always faces the center of arc at each punching position.

CAUTION
The punch section and the die section of the die can be controlled by the same motors or C axis synchronously controlled by the respective servo motors.



Parameter

Refer to parameters No. 16360 to 16469 in the parameter manual (B–62780EN).

Reference item

II–19.3 C–axis Control in Series 16/18/160/180–PC OPERATOR'S MANUAL.

6.2 C–AXIS SYNCHRONIZATION CONTROL

General

The C–axis synchronization control function enables operation of the punch section of the die in sync with the die section of the die by using a separate servo motor for each section. C–axis synchronization control simultaneously issues the same command to both servo motors. So, synchronization error correction, normally required to detect and reduce deviation between the two servo motors during synchronous operation, is not applied.
In C–axis synchronization control, the master axis is referred to as the C_1–axis, and the slave axis as the C_2–axis.

Axis configuration

When C–axis synchronization control is applied, the C_2–axis immediately follows the C_1–axis in any case. So, be particularly careful when making a connection to the machine.

Example 1: When the C_1–axis is the third axis $\rightarrow$ C_2–axis: Fourth axis
Example 2: When the C_1–axis is the fifth axis $\rightarrow$ C_2–axis: Sixth axis

Command

When C–axis synchronization control is applied, the same increment system, maximum values, and so forth, as used for ordinary C–axis control, are used. The C_1–axis and C_2–axis are controlled by programmed C commands.

Manual reference position return

In manual reference position return, C–axis synchronization control is not applied, but a reference position return operation is independently performed for each of the C_1–axis and C_2–axis. A deceleration signal for reference position return and a reference position return completion signal are provided separately for the C_1–axis and C_2–axis, so that reference position return is performed in the ordinary way. Each signal is provided for each axis number.

Create a sequence on the PMC side so that synchronous operation is enabled when the reference position return completion signals for both the C_1–axis and C_2–axis are set to 1.

CAUTION
Upon power–on, emergency stop, or servo alarm release, the machine positions for the C_1–axis and C_2–axis will shift. In such cases, therefore, always perform manual reference position return to correct the machine positions.

Manual operation

When C–axis synchronization control is applied, jog feed, incremental feed, and handle feed cannot be performed. Synchronous operation, however, is enabled by setting synchronization control signal SYNCJn <G140> (n: Axis number of the C_2–axis) to 1. In this case, synchronous operation is performed by issuing commands to the C_1–axis only; no commands can be issued to the C_2–axis.

Synchronization error alarm

During C–axis synchronous operation, position deviation of the C_1–axis and C_2–axis is monitored. The maximum allowable position deviation is set in parameter Nos. 16364 and 16365. If the position deviation error for the C_1–axis or C_2–axis exceeds the value set in the corresponding parameter during synchronous operation, alarm 4603 is issued.

Two parameters are provided for setting the maximum allowable position deviation errors. One parameter is effective when nibbling is performed, and the other when nibbling is not performed.

Synchronization

The positions on the C_1–axis and C_2–axis may be displaced relative to each other during power–off or in the event of an emergency stop. This function enables the correction of such displacement. After the follow–up performed immediately after power–on, a compensation pulse signal is output to the C_2–axis to match the C_2–axis machine position to that of the C_1–axis machine position. Note that this function is effective only when the absolute position detection function is applied to the C–axis.

CAUTION

1 To use C–axis synchronization, set bit 4 (ACS) of parameter No. 16360 to 1.
2 The synchronization function is enabled after a reference position has been established.
3 A synchronization error is represented by the position deviation on the C_2–axis when a stop is performed. The position deviation is indicated by the detection unit as diagnosis No. 300. When this value exceeds the value set in parameter No. 16368, servo alarm No. 410 is issued. This alarm can be released by using the reset key, but the position on the C_2–axis will remain displaced. So, correct the position by performing manual reference position return.
4 Synchronization is not performed when a servo alarm is released. Perform position correction by means of manual reference position return.
5 When the C–axis offset function is used, synchronization may not be performed normally if an emergency stop or reset operation is performed during C–axis movement. So, alarm No. 4605 is issued if a C command is specified. In this case, perform position correction by means of manual reference position return.
6 If the position of the C_1–axis is displaced relative to that of the C_2–axis, with 0° between the positions, synchronization cannot be performed. Perform position correction by means of manual reference position return.

Signal

C–axis control status signal SC1 <F233#0> SC2 <F233#1>

[Classification] Output signal

[Function] These signals notify the PMC of whether C–axis independent control or C–axis synchronization control is applied.

[Output condition] These signals are set to 1 upon power–on in the following cases:

SC1: When C–axis control is applied

SC2: When C–axis synchronization control is applied (when using the C_1–axis and C_2–axis)

These signals are set to 0 in the following cases:

SC1: When C–axis control is not applied

SC2: When C–axis synchronization control is not applied (when C–axis independent control is applied)

Synchronous manual feed select signal SYNCJ4 ~ SYNCJ8 <G140>

[Classification] Input signal

[Function] C–axis synchronization control is applied when jog feed, handle feed, or incremental feed is performed.

SYNCJn
- 4 –– The fourth axis is the C_2–axis.
- 5 –– The fifth axis is the C_2–axis.
- 6 –– The sixth axis is the C_2–axis.
- 7 –– The seventh axis is the C_2–axis.
- 8 –– The eighth axis is the C_2–axis.

[Operation] When this signal is set to 1, a move command for the C_1–axis is also issued to the C_2–axis.
Never change the setting of this signal while movement is being performed.
Set this signal to 0 in other than manual operation mode.

CAUTION
1 During synchronous operation, an input signal provided for each axis must be entered separately and simultaneously for the C_1–axis and C_2–axis.
2 An output signal provided for each axis is output separately for the C_1–axis and C_2–axis.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G140	SYNC8	SYNC7	SYNC6	SYNC5	SYNC4			
	#7	#6	#5	#4	#3	#2	#1	#0
F233							SC2	SC1

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
16360				ACS				SYN

[Data type] Bit

SYN C–axis synchronous control is:

0 : Disabled.
1 : Enabled.

ACS Under C–axis synchronous control, synchronization is:

0 : Disabled.
1 : Enabled.

16364	Upper limit of error under C–axis synchronous control

[Data type] Word

[Unit of data] Units of detection

[Valid data range] 0 to 32767

When the absolute value of the position error between the C_1–axis and C_2–axis exceeds the value set in this parameter under C–axis synchronous control, alarm 4603 occurs.

16365	Upper limit of error under C–axis synchronous control (during continuous pressing)

[Data type] Word

[Unit of data] Units of detection

[Valid data range] 0 to 32767

When the absolute value of the position error between the C_1–axis and C_2–axis exceeds the value set in this parameter while continuous pressing signal NBL is set to 1, alarm 4603 occurs.

16368	Maximum compensation in C–axis synchronization

[Data type] Word

[Unit of data] Units of detection

[Valid data range] 0 to 65535

This parameter specifies the maximum compensation in C–axis synchronization under C–axis synchronous control. If an actual compensation value exceeds this value, an excessive error alarm occurs in the stop state or during travel.

CAUTION

For C–axis synchronization control, two servo motors are used: one for the C_1–axis and the other for the C_2–axis. This means that for the C_1–axis and C_2–axis, the same values must be set in the parameters (for setting a feedrate, time constant, and so forth) provided for each axis. If different values are set, normal synchronous operation cannot be expected.

For the following parameters, however, different values can be set for the C_1–axis and C_2–axis:

Grid shift	(No. 1850)
Backlash compensation	(Nos. 1851, 1852, 16390 to 16429)
C–axis offset	(No. 16430 to 16469)

Alarm and message

Number	Message	Contents
4603	C AXIS SYNCHRONOUS ERROR	The difference between the position deviation value of C_1 axis and C_2 axis exceeds the parameter value (No. 16364, 16365) with the C–axis synchronous control function.
4605	NEED ZRN	The C axis synchronization is not done normally.

6.3 C–AXIS OFFSET FUNCTION

General

For the punch press, a tool is used which allows the angle to be changed according to the machine. This type of rotary tool is subject to CNC C–axis control. A C command is used to change the angular position of a tool. When a machine controlling multiple tools of this type is used, mechanical adjustment of each tool is necessary to ensure correct punching.

This function simplifies mechanical adjustment by performing automatic C–axis offset when indexing up to 20 tools that allow angle changes. A machine that uses C–axis synchronization control enables offset to be performed separately for each of the C_1–axis and C_2–axis.

The C–axis offset function supports two types of offset operations.

6.3.1 C–axis Offset Type A

General

Upon the completion of tool indexing on the turret by issuing a T command, this function performs offset at the same time as an angle is specified with a C command.

Operation

Set the number of a tool that supports angle changes in a parameter (Nos. 16370 to 16389). Following the T command set in the parameter, the CNC performs C–axis offset according to the value set in the corresponding parameter (Nos. 16430 to 16469). An example program for performing this operation is shown below.

Parameter No. 16370 = 10 (Tool number 10)
Parameter No. 16430 = 10 (Offset 0.1 deg)

```
(Example program)
      :
N100  X__  Y__T10  ;
      :
N110  X__  Y__T20  ;
      :
N120  X__  Y__T10  C  90.  ;
      :
N130  G28  ;
```

The N100 block specifies rotary tool T10. So, C–axis movement by an offset of 0.1 deg is performed once positioning along the X–axis, Y–axis, and T–axis has been completed, and completion signal FIN for the T command is returned.

The N110 block specifies a tool change command. So, C–axis offset is canceled, and reference position return is performed.

The N120 block specifies rotary tool T10. So, C–axis movement by 90.00 deg plus an offset of 0.1 deg is performed once positioning along the X–axis, Y–axis, and T–axis has been completed, and completion signal FIN for the T command is returned.

The N130 block performs reference position return while cancelling C–axis offset.

Cancelling the offset

C–axis offset is canceled by the following programmed commands:

(a) T command
(b) G28 command

C–axis offset is not canceled by a reset, but can be canceled by setting bit 0 (RC0) of parameter No. 16362. When C–axis synchronization control is applied, and the C_1–axis offset differs from the C_2–axis offset, however, the ($C_1 - C_2$) offset is not canceled. Instead, this ($C_1 - C_2$) offset is canceled upon execution of the next T command or G28 command.

CAUTION

1 When using this function, set bit 2 (MAI) of parameter No. 16360 to 1 and bit 3 (MAB) of parameter No. 16360 to 0.
2 When using this function, enable movement along the T axis (by setting bit 5 (CRM) of parameter No. 16362 to 0) only when the tool is at the reference position on the C–axis.
3 When multiple–tool control is applied together with this function, this function is applied to the multiple–tool holder. Set the number of the multiple–tool holder as that of a tool that supports angle change.
4 If the C_1–axis offset differs from the C_2–axis offset when C–axis synchronization control is applied, no synchronization error check is made as part of an offset operation or C–axis positioning including offset cancellation.

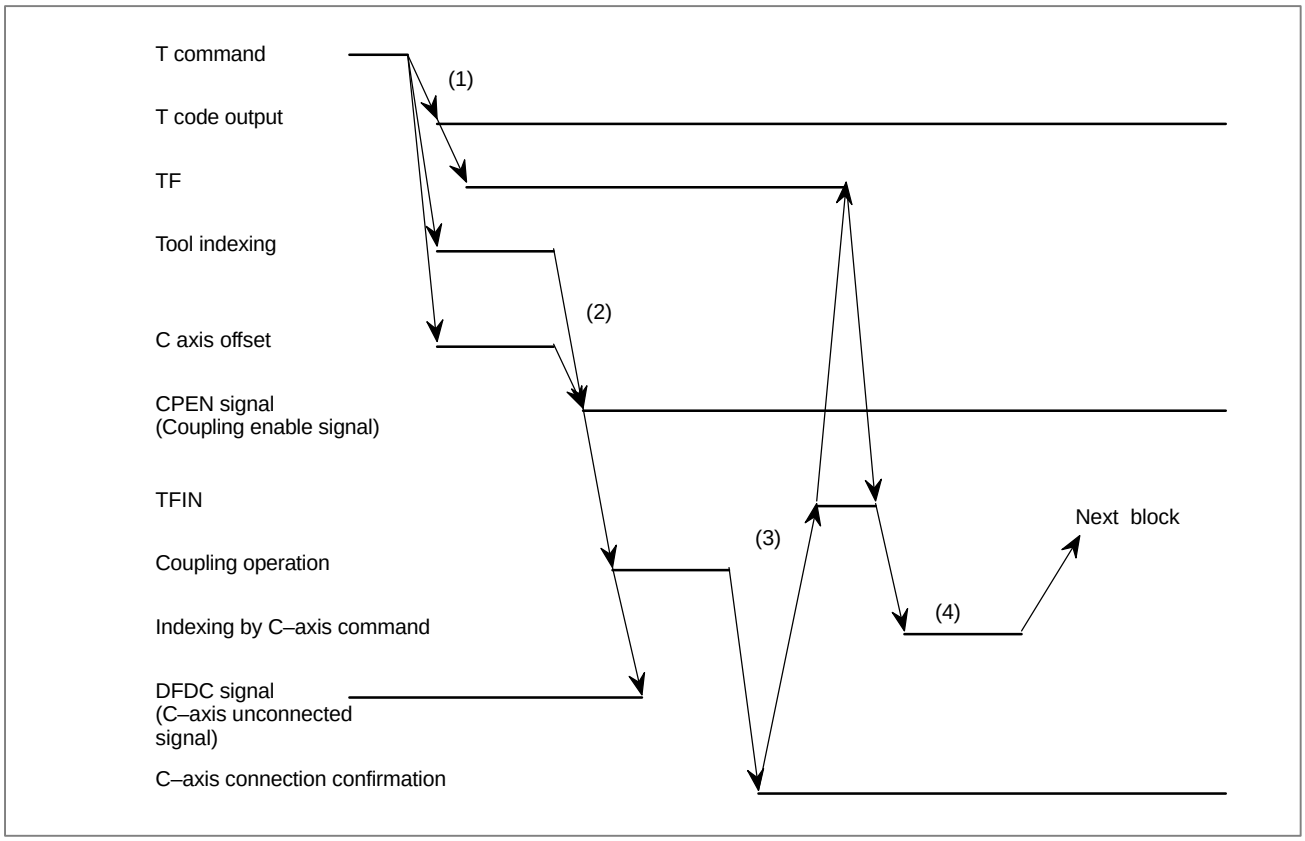
6.3.2 C–axis Offset Type B

General

This function performs C–axis offset as part of a tool change operation based on a T command, either before or after the mechanical attachment/detachment of rotary tools and tool holders.

Operation

(1) Exchanging a regular tool with a rotary tool or a tool in a multiple–tool holder



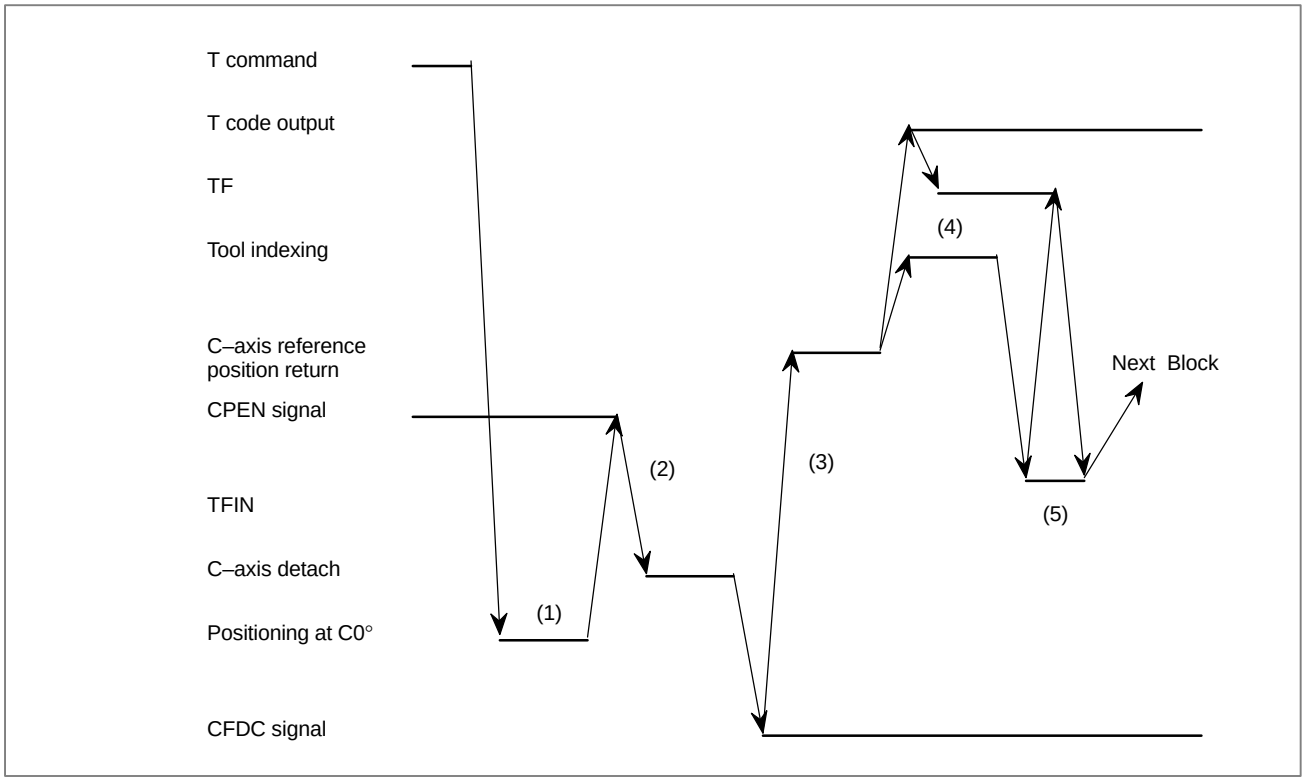
(1) When a T command supporting angle change or multiple–tool control is specified, a T code and the TF signal are output. At the same time, tool indexing and C–axis offset are performed.

(2) Upon the completion of tool indexing and C–axis offset, the CNC sets the coupling enable signal CPEN to 1. Using this signal, the machine couples the C–axis and tool holder. The PMC sets C–axis unconnected signal DFDC to 0 to notify the CNC that coupling has been completed.

(3) Then upon the completion of tool change, the PMC returns the TFIN signal to the CNC.

(4) Tool angle change is performed according to the C–axis command. When multiple–tool control is applied, tool indexing is performed within the multiple–tool holder.

(2) Exchanging a rotary tool or tool in a multiple–tool holder with a regular tool



(1) When a regular tool command is specified, the CNC performs positioning at an angle of 0° on the C–axis. When a multiple–tool holder is selected, the tool at 0° within the multiple–tool holder is indexed. Then, the CNC sets the CPEN signal to 0.

(2) When the CPEN signal is set to 0, the machine detaches the C–axis from tool holder.

(3) Once the C–axis is detached, the PMC sets the CFDC signal to 1. When the CFDC signal is set to 1, the CNC cancels the C–axis offset, and performs reference position return.

(4) Then, a T code and TF signal are output, after which tool indexing is performed.

(5) The PMC returns the TFIN signal to the CNC upon the completion of tool change.

CAUTION
1 When using this function, set bit 2 (MAI) of parameter No. 16360 to 1 and bit 3 (MAB) of parameter No. 16360 to 1.
2 When using this function, enable movement along the T axis (by setting bit 5 (CRM) of parameter No. 16362 to 0) only when the tool is at the reference position on the C–axis.
3 When multiple–tool control is applied together with this function, this function is applied to the multiple–tool holder. Set the number of the multiple–tool holder as that of a tool that supports angle change.
4 If the C_1–axis offset differs from the C_2–axis offset when C–axis synchronization control is applied, no synchronization error check is made as part of an offset operation or C–axis positioning including offset cancellation.

Signal

Coupling enable signal CPEN <F232#4>

[Classification] Output signal

[Function] When C–axis offset type B is used, this signal posts that C–axis offset is performed, and that the C–axis and tool holder are ready for coupling.

[Output condition] See “Operation” in Section 6.3.2.

C–axis unconnected signal CFDC <G233#7>

[Classification] Input signal

[Function] When C–axis offset type B is used, this signal posts that the C–axis and tool holder are not mechanically coupled, hence are inoperative.

[Operation] See “Operation” in Section 6.3.2.

CAUTION
When the C–axis and tool holder are not mechanically coupled, set this signal to 1.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G233	CFDC							

	#7	#6	#5	#4	#3	#2	#1	#0
F232				CPEN				

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
16360					MAB	MAI		

[Data type] Bit

MAI The function for compensating the C–axis position is:

0 : Invalidated.
1 : Validated.

MAB The function B for compensating the C–axis position is:

0 : Invalidated. (Type A)
1 : Validated. (Type B)

	#7	#6	#5	#4	#3	#2	#1	#0
16362			CRM					RCO

[Data type] Bit

RCO At reset, compensation of C–axis position is:

0 : Not canceled.
1 : Canceled.

CRM According to a T command, the machine is:

0 : Moved along the C–axis to the reference point.
1 : Not moved along the C–axis to the reference point.

16370	Number of tool 1 for which C–axis control can be executed
16371	Number of tool 2 for which C–axis control can be executed
16372	Number of tool 3 for which C–axis control can be executed
16373	Number of tool 4 for which C–axis control can be executed
16374	Number of tool 5 for which C–axis control can be executed
16375	Number of tool 6 for which C–axis control can be executed
16376	Number of tool 7 for which C–axis control can be executed
16377	Number of tool 8 for which C–axis control can be executed
16378	Number of tool 9 for which C–axis control can be executed
16379	Number of tool 10 for which C–axis control can be executed
16380	Number of tool 11 for which C–axis control can be executed
16381	Number of tool 12 for which C–axis control can be executed
16382	Number of tool 13 for which C–axis control can be executed
16383	Number of tool 14 for which C–axis control can be executed
16384	Number of tool 15 for which C–axis control can be executed
16385	Number of tool 16 for which C–axis control can be executed
16386	Number of tool 17 for which C–axis control can be executed
16387	Number of tool 18 for which C–axis control can be executed
16388	Number of tool 19 for which C–axis control can be executed
16389	Number of tool 20 for which C–axis control can be executed

[Data type] Word

[Valid data range] 0 to 9999

Each of the parameters set the number of a tool for which C–axis control can be executed.

16430	C–axis position compensation 1 to use function for compensating the C–axis position
16431	C–axis position compensation 2 to use function for compensating the C–axis position
16432	C–axis position compensation 3 to use function for compensating the C–axis position
16433	C–axis position compensation 4 to use function for compensating the C–axis position
16434	C–axis position compensation 5 to use function for compensating the C–axis position
16435	C–axis position compensation 6 to use function for compensating the C–axis position
16436	C–axis position compensation 7 to use function for compensating the C–axis position
16437	C–axis position compensation 8 to use function for compensating the C–axis position
16438	C–axis position compensation 9 to use function for compensating the C–axis position
16439	C–axis position compensation 10 to use function for compensating the C–axis position
16440	C–axis position compensation 11 to use function for compensating the C–axis position
16441	C–axis position compensation 12 to use function for compensating the C–axis position
16442	C–axis position compensation 13 to use function for compensating the C–axis position
16443	C–axis position compensation 14 to use function for compensating the C–axis position
16444	C–axis position compensation 15 to use function for compensating the C–axis position
16445	C–axis position compensation 16 to use function for compensating the C–axis position
16446	C–axis position compensation 17 to use function for compensating the C–axis position
16447	C–axis position compensation 18 to use function for compensating the C–axis position
16448	C–axis position compensation 19 to use function for compensating the C–axis position
16449	C–axis position compensation 20 to use function for compensating the C–axis position

[Data type] Two–word

[Unit of data] 0.01 deg (IS–A) /0.001 deg (IS–B)

[Valid data range] 0 to □99999999

Each of the parameters set the C–axis position compensation (C_1–axis position compensation in C–axis synchronous control) to use the function for compensating the C–axis position.
These compensated values correspond to the tool numbers set in parameters 16370 to 16389.
The values validated when parameter MAI (No. 16360, #2) is set to 1.

16450	C_2–axis position compensation1 to use function for compensating the C–axis position
16451	C_2–axis position compensation2 to use function for compensating the C–axis position
16452	C_2–axis position compensation3 to use function for compensating the C–axis position
16453	C_2–axis position compensation4 to use function for compensating the C–axis position
16454	C_2–axis position compensation5 to use function for compensating the C–axis position
16455	C_2–axis position compensation6 to use function for compensating the C–axis position
16456	C_2–axis position compensation7 to use function for compensating the C–axis position
16457	C_2–axis position compensation8 to use function for compensating the C–axis position
16458	C_2–axis position compensation9 to use function for compensating the C–axis position
16459	C_2–axis position compensation10 to use function for compensating the C–axis position
16460	C_2–axis position compensation11 to use function for compensating the C–axis position
16461	C_2–axis position compensation12 to use function for compensating the C–axis position
16462	C_2–axis position compensation13 to use function for compensating the C–axis position
16463	C_2–axis position compensation14 to use function for compensating the C–axis position
16464	C_2–axis position compensation15 to use function for compensating the C–axis position
16465	C_2–axis position compensation16 to use function for compensating the C–axis position
16466	C_2–axis position compensation17 to use function for compensating the C–axis position
16467	C_2–axis position compensation18 to use function for compensating the C–axis position
16468	C_2–axis position compensation19 to use function for compensating the C–axis position
16469	C_2–axis position compensation20 to use function for compensating the C–axis position

[Data type] Two–word

[Unit of data] 0.01 deg (IS–A) /0.001 deg (IS–B)

[Valid data range] 0 to □99999999

Each these parameters specifies C_2–axis position compensation value for the C–axis position compensation function. The parameter values correspond to the tool numbers specified in parameters 16370 to 16389. These parameters are valid when both the SYN and MAI bits (bits 0 and 2 of parameter 16360) are set to 1.

7 SAFETY ZONE CHECK

This is the safety function to set the safety zone for protecting the workpiece holder that holds the workpiece set on the carriage, and disable punching in that area or forbid the tool to approach thereinto.

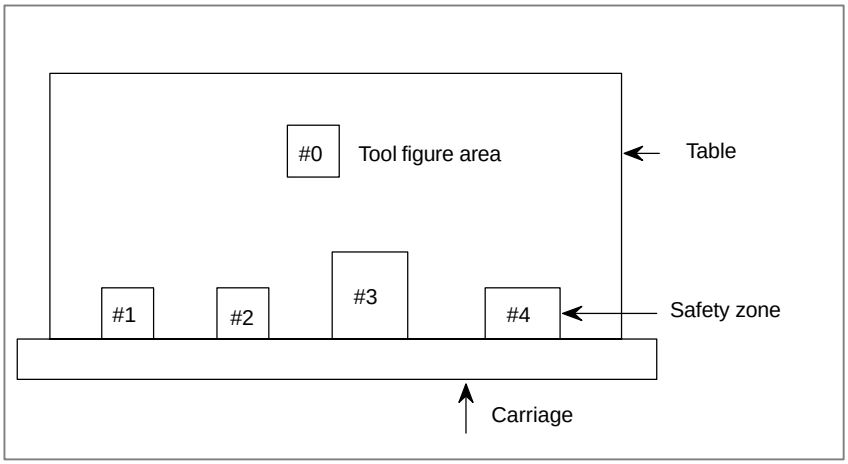


Fig. 7

This function permits to set tool figure area (#0) and up to four safety zones (#1 - #4), as shown above.

Two types of safety zone check methods are prepared.

7.1 TYPE A

Punch forbidden area and approach forbidden area

The safety zone is settable in two types, punch forbidden area and approach forbidden area, that are set by the parameter SZ1 to SZ4 (No. 16501#0 - #3) shown below.

1) Punch forbidden area
   When the tool figure area goes into the safety zone and the punching is commanded, an alarm (Nos. 4800 to 4803) is given to disable punching. In the case of positioning & punching command in the automatic operation mode, when the end point of positioning is in the punch forbidden area, an alarm is given without moving the axis. (Previous check)
   In the case of move command without punching, the tool figure area can go into the punch forbidden area, but manual punching is impossible after going into this area.

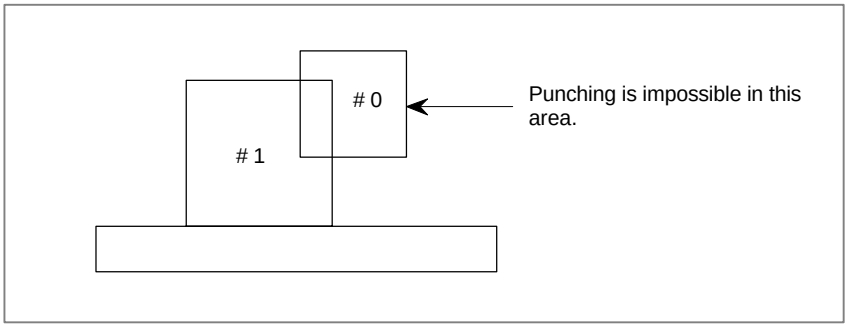


Fig. 7.1 (a)

2) Approach forbidden area
   The tool figure area can not go into the safety zone. When the tool figure area approaches into the safety zone by the move command, the axis is immediately stopped and an alarm (Nos. 4810 - 4837) is given. This is valid in either manual or automatic operation mode.

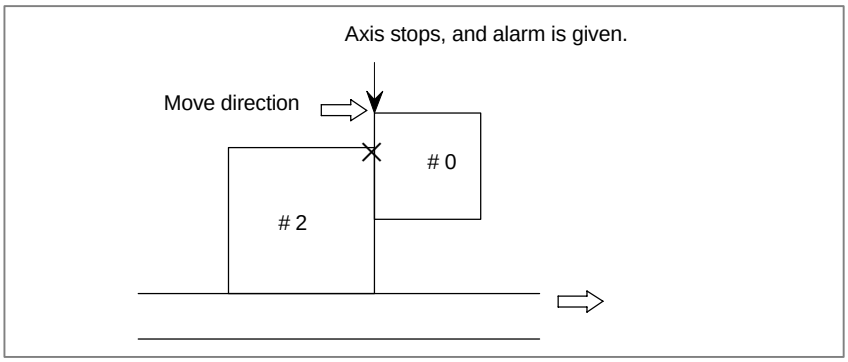


Fig. 7.1 (b)

7.2 TYPE B

General

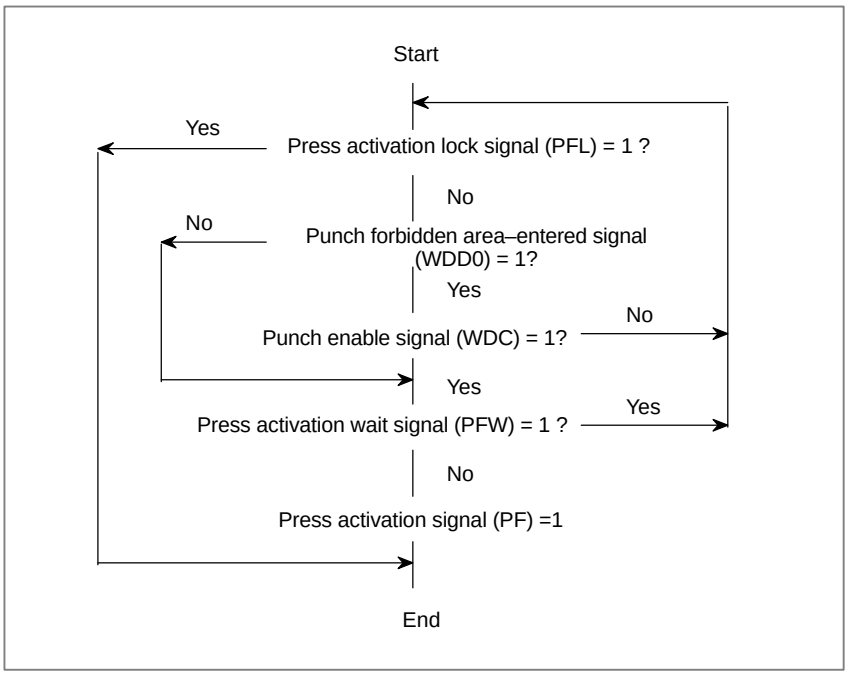
By setting bit 0 (SF0) of parameter No. 16500, the type B safety zone check can be selected. With type B, no alarm is issued even if a tool enters a safety zone; after confirming the safety of the situation, the operator can perform a punch operation, or can position the tool to the next punching position without performing punching.

With type B, all safety zones are handled as punch forbidden areas.

Punch forbidden area

If the tool enters a safety zone, the punch–forbidden area–entered signal WDD0 (F231#7), used to provide notification that the tool has entered a punch forbidden area, is set to 1 in the punch block. When punching is to be performed in a punch–forbidden area, punch–enable signal WDC (G232#4) is set to 1 after the operator confirms that punching can be performed safely. After detecting the WDC signal, the CNC sets the PF signal to 1.

When punching is not to be performed in a punch forbidden area, the press activation lock signal PFL (G230#0) is set to 1. Upon detecting the PFL signal, the CNC moves on to execute the next block.



CAUTION

1 If the tool enters a punch forbidden area during nibbling, the WDD0 signal is set to 1 one punching position before the tool enters punch forbidden area.
2 With type B, the setting of SZ1 to SZ4 (bits 0 to 3 of parameter No. 16501) is ignored.

7.3 SETTING THE SAFETY ZONE

Set the machine coordinate value when the workpiece holder is positioned at the tool center (punching position), in the parameters 16505 - 16516 in output units.

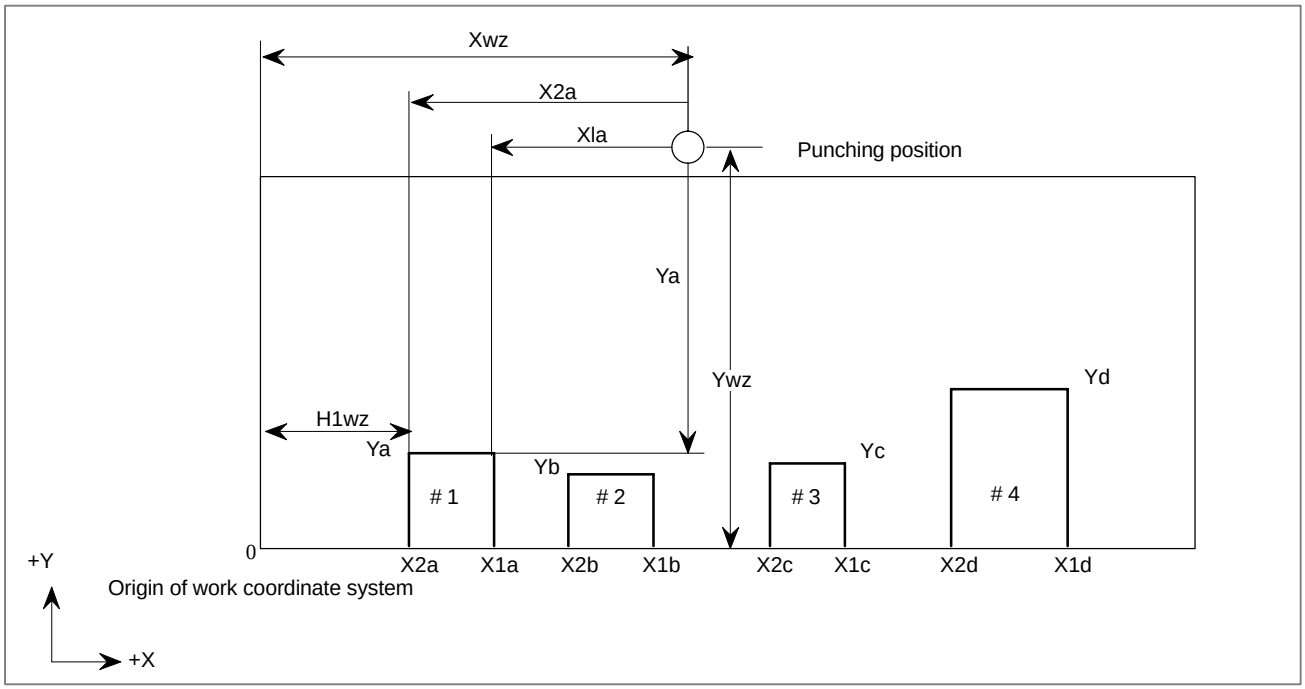


Fig. 7.3

Regarding #1 in Fig. 7.3, the safety zone is specified at both ends (X1a, X2a) for the X-axis direction, and at the forward end (Ya) of the workpiece holder for the Y-axis direction. The specifying method is the same as for #2, #3 and #4.
Considering the setting value in the work coordinate system, it is the value obtained by subtracting the set value of automatic coordinate system from the workpiece holder position in the work coordinate system.
For example, set value of X2a is as follows in Fig. 7.3.

Set value (X2a) = (H1wz) – (Xwz)

Set four safety zones to be arranged sequentially in the order of #1, #2, #3 and #4 from the origin to positive of the X-axis.

7.4 SETTING THE TOOL SHAPE AREA

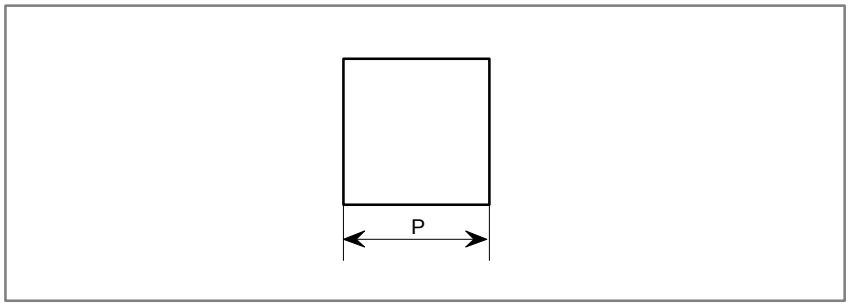


Fig. 7.4 (a)

The specification of the area of tool figure sets the size in the X direction and Y direction of the tool by the parameter (No. 16517 to 16532, 16551 to 16558).
The setting unit is output unit.
Twelve kinds of or less tool figure can be set.

The tool shape area can be changed by using the programmable parameter input function (G10). Therefore, when multiple tools are used, it is possible to specify the tool shape area meeting the tool No. (Txx).

When there are an area of the punched tool and an area with the laser oscillator for special, first set two safety zones for the workpiece holder. Reserve the remaining two safety zones for the imaginary workpiece holder.
When the workpiece holder (a) approaches to the laser oscillator in Fig. 7.4 (b) below, it is judged as the approaching of the tool area to the imaginary safety zone.

CAUTION
DI signals of the PMC select a tool figure area.

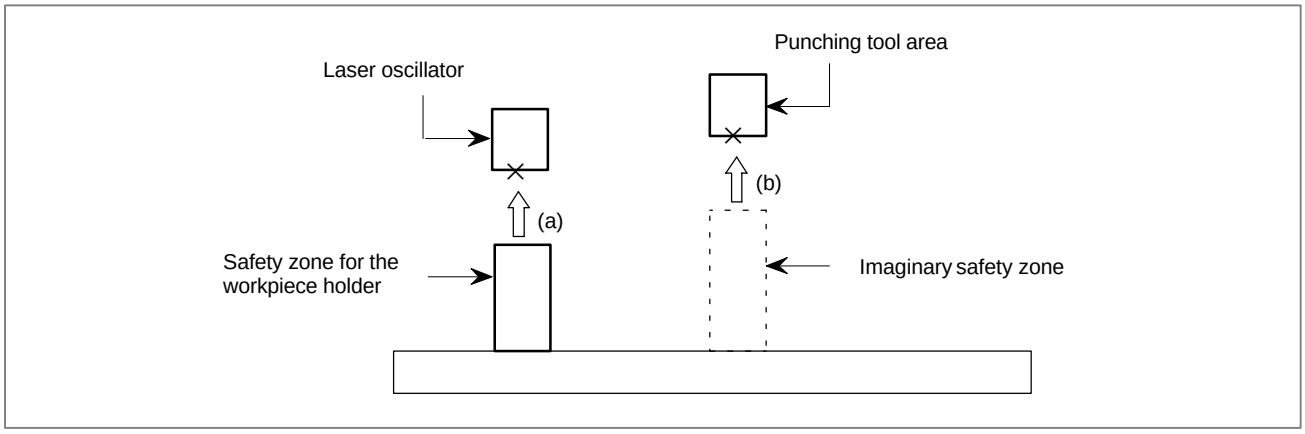


Fig. 7.4 (b)

7.5 AUTOMATIC SAFETY–ZONE SETTING

Workpiece holder position detector signal (input) SAFZ <X1004#1>

The detector installed in the machine automatically detects the positions of the workpiece holders mounted on the carriage. The detected values are set in the safety zone setting parameter.

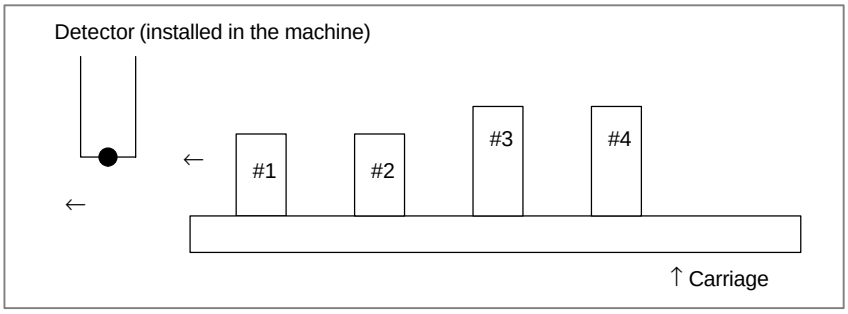


Fig. 7.5

As shown in Fig. 7.5, the SAFZ signal is turned on and off when workpiece holders 1 to 4 pass by the detector. The safety zone is set according to the status of the SAFZ signal. The position along the X–axis when the edge of the SAFZ signal rises is specified as one end of the workpiece holder. The position when the edge of the SAFZ signal falls is specified as the other end of the workpiece holder.

7.5.1 Workpiece Holder Detection Command

G32X x F f P p Q q ;

G32 is used to specify detection of the positions of workpiece holders. Before this command is issued, the workpiece holder detector must be made ready.

Specify the command for movement along the X–axis during detection after address X. The move direction can be specified as either the positive or negative direction. When the direction is specified as positive, the detection start position must be at a more negative position than that of workpiece holder 1. In addition, the distance between the detection start position and workpiece holder 1 must be sufficient for the velocity to be constant. Workpiece holders are detected in ascending order of the holder numbers, such as 1, 2, 3, and 4. When the move direction is specified as negative, workpiece holders are detected in descending order, such as 4, 3, 2, 1. Other conditions are the same as for positive movement. Specify the feedrate during detection in F (mm/min, 0.01 inch/min) in the same way as that during usual interpolation.

The positions of the workpiece holders are obtained from the specified machine position when the edge of the workpiece holder position detector signal (SAFZ) rises of falls. However, the actual machine position contains an error such as servo delay unlike the specified machine position. Compensate the error using individual numeric valuers after addresses P and Q. Namely, specify the compensation when the edge of the SAFZ signal rises with the numeric value after address P, and the compensation when the edge of the SAFZ signal falls with the numeric value after address Q.

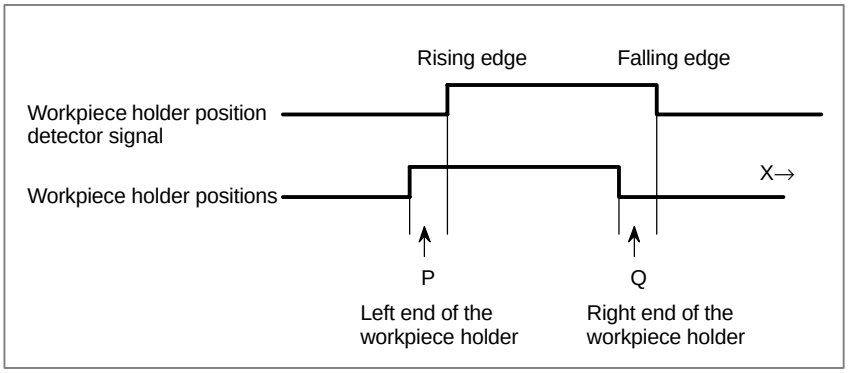
The servo delay can be calculated by the following formulas:

$\Delta E = T1 \square F + T2 \square F$ (exponential acceleration/deceleration)

$\Delta E = 1/2T1 \square F + T2 \square F$ (linear acceleration/deceleration)

- ΔE: Servo delay
- T1: Automatic acceleration/deceleration time constant
- T2: Servo time constant
- F: Feedrate

The sign for compensation is defined as positive when compensation is made in the opposite direction to the move direction specified by the detection command.



CAUTION

1 G32 is a single–shot G code.
2 This function is ineffective if reference position return along the X–axis is not completed.
3 The positions of the workpiece holders along the Y–axis cannot be detected automatically.
4 When the execution of the workpiece holder detection command is started, the specified safety zone is cleared.
5 Before the workpiece holder detection command is issued, the workpiece holder detector must be made ready with the M function . After the end of detection, the ready state of the detector must be released.

```
Mxx    ;        (Making the workpiece holder
                detector ready)
G32X ____F ____P ____Q ____  ;
MΔΔ    ;        (Releasing the ready state of the
                workpiece holder detector)
```

6 The error caused by fluctuation in the workpiece holder position detector (SAFZ) signal, namely, the following detection error may develop:

Detection error = f (mm/min) □ 8/60 mm

7.5.2 Detecting Workpiece Holder Position Using an External Signal

Detection method 1

Automatic safety zone setting is enabled when reference position return along the X–axis is completed immediately after CNC power–on. After selecting manual reference position return mode, set automatic safety zone setting request signal SAFRQ to 1. At this time, the workpiece holder position detector must be ready for operation.

When reference position return has not yet been completed, perform manual reference position return. For automatic safety zone setting, set the SAFRQ signal to 1 after reference position return along the X–axis. Next, enter feed axis direction select signal +X or –X. Then, the tool moves along the X–axis to position A, set in parameter No. 16535, either at the manual rapid traverse rate or at the feedrate set in parameter No. 16536.

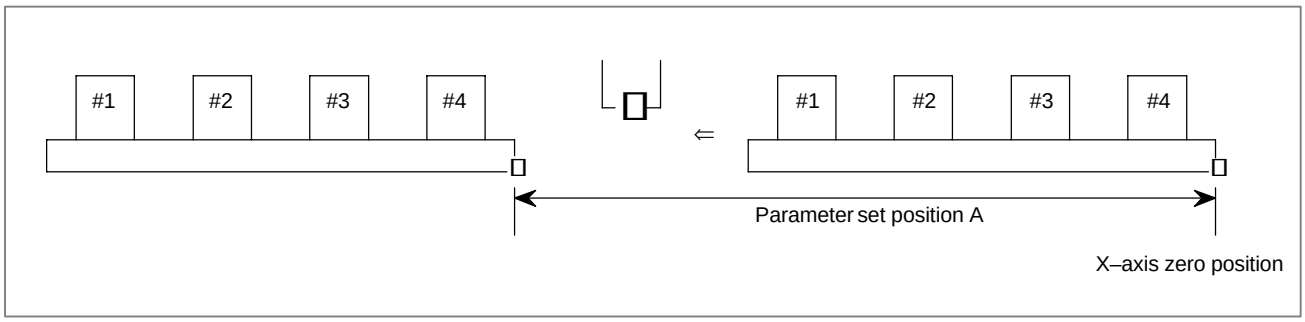


Fig. 7.5.2

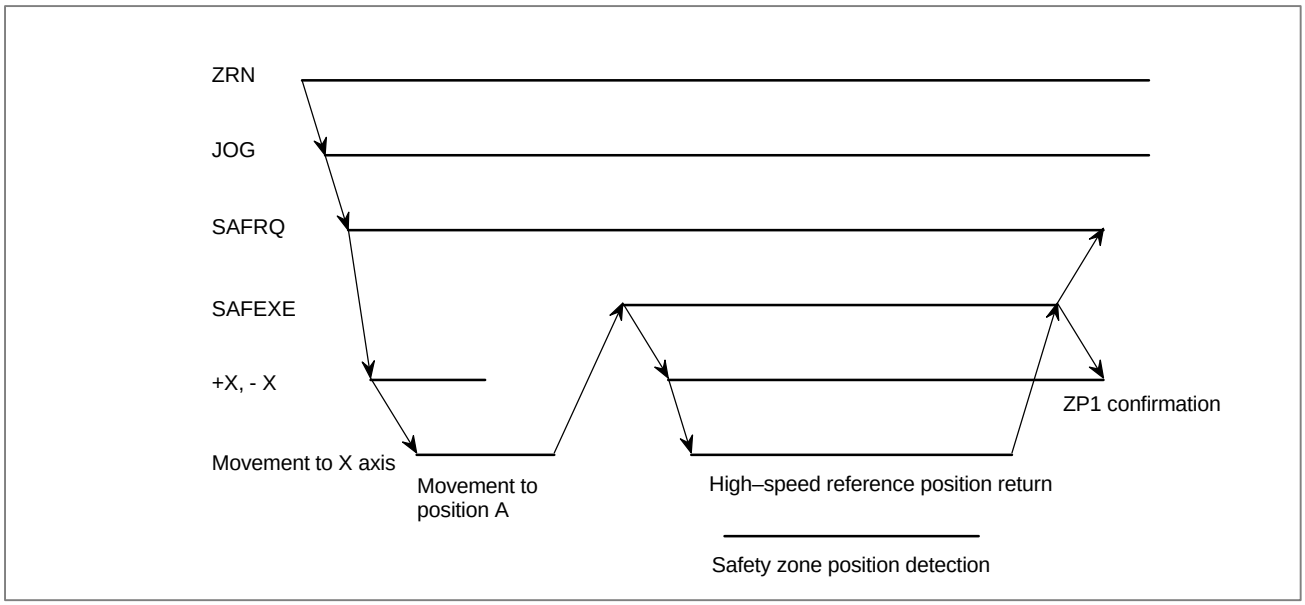
Once movement along the X–axis has been completed, the CNC clears the currently set safety zones, then sets safety zone setting ready signal SAFEXE to 1. Automatic safety zone setting is enabled when the SAFEXE signal is output.

If feed axis direction select signal +X or –X is entered again after the SAFEXE signal has been set to 1, the tool performs high–speed reference position return along the X–axis. During high–speed reference position return, the CNC sets each safety zone at the rising or falling edge of the workpiece holder position detector signal.

Safety zones are set in order from #1 to #2 to #3 to #4 if movement is made for detection in the positive direction along the X axis. Safety zones are set in order from #4 to #3 to #2 to #1 if movement is made for detection in the negative direction along the X axis.

Upon the completion of high–speed reference position return, the SAFEXE signal is set to 0 to end automatic safety zone setting. After the SAFEXE signal has been confirmed as having been set to 0, the SAFRQ signal is set to 0.

Timing chart



Detection method 2

After automatic or manual reference position return along the X–axis, automatic safety zone setting is enabled when X–axis reference position return completion signal ZP1 is set to 1. In this case, safety zones can be set automatically if manual reference position return mode is selected, and the SAFRQ signal is set to 1. At this time, the workpiece holder position detector must also be ready for operation.

Then, when feed axis direction select signal +X or –X is entered, the tool moves along the X–axis to position A, set in parameter No. 16535, either at the manual rapid traverse rate or at the feedrate set in parameter No. 16536. At this time, the CNC clears the currently set safety zones.

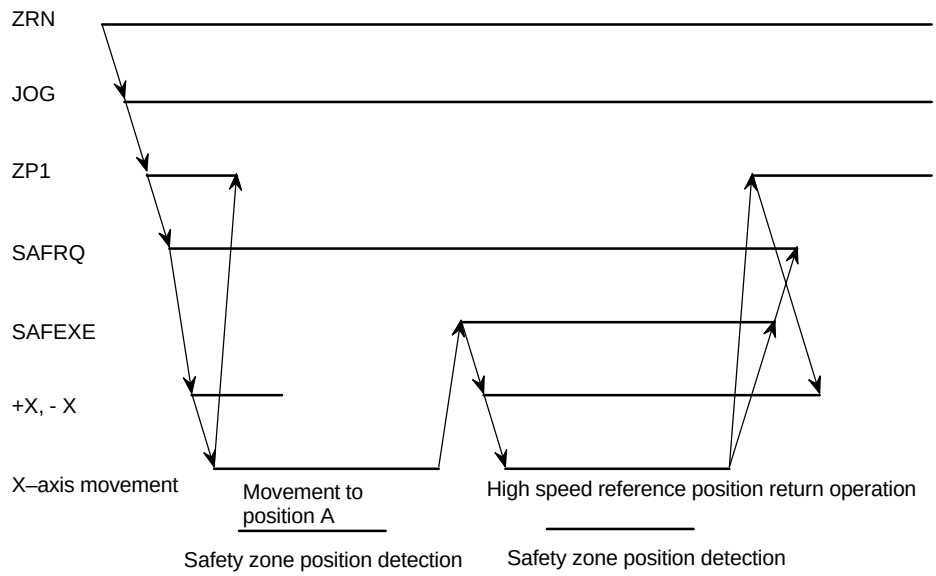
During movement to position A, the CNC memorizes the position of one end of each workpiece holder at the rising edge of the workpiece holder position detector signal.

Upon the completion of movement to position A, the SAFEXE signal is set to 1. If the feed axis direction select signal +X or –X is entered again after the SAFEXE signal has been set to 1, the tool performs high–speed reference position return along the X–axis. During high–speed reference position return, the CNC sets each safety zone by memorizing the position of the other end of each workpiece holder at the rising edge of the workpiece holder position detector signal.

Safety zones are set in order from #1 to #2 to #3 to #4 if movement is made for detection in the positive direction along the X axis during high–speed reference position return. Safety zones are set in order from #4 to #3 to #2 to #1 if movement is made for detection in the negative direction along the X axis during high–speed reference position return.

Upon the completion of high–speed reference position return, the SAFEXE signal is set to 0 to end automatic safety zone setting. After the SAFEXE signal has been confirmed as having been set to 0, the SAFRQ signal is set to 0.

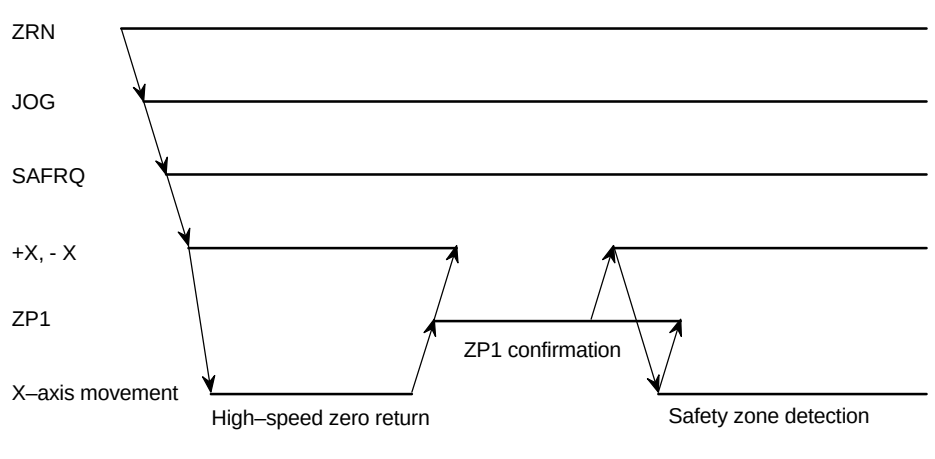
Timing chart



Once reference position return along the X–axis has been completed, the tool can be automatically moved to the X–axis reference position to perform subsequent workpiece holder detection operations.

When automatic safety zone setting request signal SAFRQ and feed axis direction select signal +X or –X are entered, the tool performs high–speed reference position return along the X–axis, starting from the current position. After checking the ZP1 signal upon the completion of this operation, perform actual detection.

Timing chart



CAUTION

1 Use bit 2 (ZI0) of parameter No. 16502 to switch between the detection methods.
2 The position of a workpiece holder on the Y–axis cannot be detected automatically.
3 To enable the execution of safety zone position detection, set a detection start position such that the distance between the detection start position and the first end of the workpiece holder is sufficient for the feedrate to stabilize.
4 If reset is performed during safety zone position detection, repeat the operation from reference position return along the X–axis.
5 The purpose of automatic detection is to automatically calculate the central position of a workpiece holder from the signal transitions. Accordingly, set workpiece holder width data in parameter Nos. 16540 to 16543 so that safety zones can be set correctly.
6 If the number of safety zones to be set does not match the detector signal transition count, during automatic safety zone setting, alarm 4871 is issued.
7 When the detector signal makes a transition when position deviation along the X–axis during movement for automatic safety zone setting exceeds the allowable range of values set in the parameters (Nos. 16538 and 16539), alarm 4870 is issued.

7.5.3 Displaying the Safety Zones and Tool Zone

After safety zone values are set automatically, they can be displayed on the safety zone screen as shown below. With this screen, the user can check whether the set values are valid.

Screen

Type A

```
SAFETY ZONE (ABSOLUTE)              O0017 N01234
 AREA #1              AREA #3
  X2=   100.000        X2=  1000.000
  X1=   200.000        X1=  1150.000
  Y =   100.000        Y =   110.000
 AREA #2              AREA #4
  X2=   500.000        X2=  1400.000
  X1=   600.000        X1=  1550.000
  Y =   100.000        Y =   110.000
 TOOL ZONE
  X =     5.000
  Y =    10.000

)_
MEM **** *** ***         11:32:41
[ TOOL ][      ][ SAFETY ][       ][ (OPRT) ]
```

Screen

Type B

```
SAFETY ZONE (ABSOLUTE)              O0017 N01234
 AREA #1              AREA #3
  W =   100.000        W =   100.000
  X =   200.000        X =  1150.000
  Y =   100.000        Y =   110.000
 AREA #2              AREA #4
  W =   100.000        W =   100.000
  X =   600.000        X =  1550.000
  Y =   100.000        Y =   110.000
 TOOL ZONE            ZONE NUMBER
  X =     5.000        N = 2
  Y =    10.000

)_
MEM **** *** ***         11:32:41
[ TOOL ][      ][ SAFETY ][       ][ (OPRT) ]
```

NOTE

The display items of type B are as follows:

W : Workpiece holder width
X : Workpiece holder central position relative to the tool center
Y : Workpiece holder tip position relative to the tool center

7.6 SIGNAL

Selection signal for tool area of safety zone SZTS0 ~ SZTS3 <G232#0 ~ #3>

[Classification] Input signal

[Function] These signals are used to select a tool area for safety zone checking. These signals must be changed while a T code is being read or upon reset of CNC.

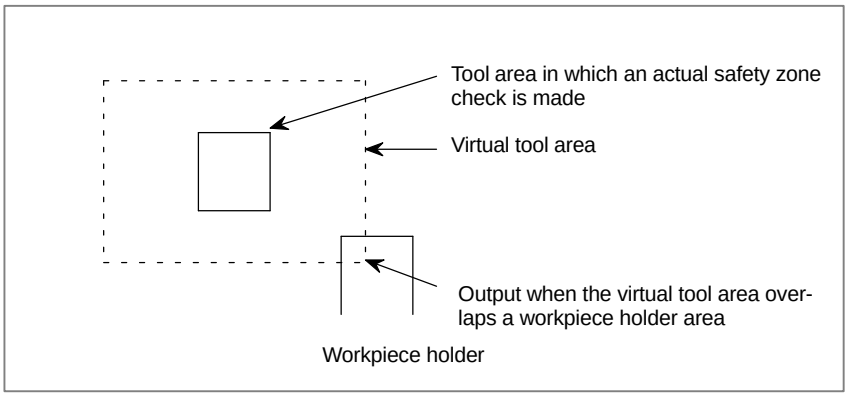
[Operation] The table below indicates the correspondence between the SZTS0 to SZTS3 signals and parameter settings.

SZTS0	SZTS1	SZTS2	SZTS3	Parameter No. for tool area	
				X–axis tool size	Y–axis tool size
0	0	0	0	16517	16518
1	0	0	0	16519	16520
0	1	0	0	16521	16522
1	1	0	0	16523	16524
0	0	1	0	16525	16526
1	0	1	0	16527	16528
0	1	1	0	16529	16530
1	1	1	0	16531	16532
0	0	0	1	16551	16552
1	0	0	1	16553	16554
0	1	0	1	16555	16556
1	1	0	1	16557	16558

In safety zone signal SZT1 ~ SZT12 <F240#0 ~ #7> <F241#0 ~ #3>

[Classification] Output signal

[Function] Each signal notifies the PMC that the corresponding tool area is in a safety zone. For example, a virtual area which is larger than the selected tool area can be set using the safety zone tool area select signals (SZTS0 to SZTS3). Thus, the command for positioning the tool near a workpiece holder is known beforehand, allowing interference to be avoided by changing the positioning path with the interlock function (for type B).



Signals SZT1 to SZT12 correspond to tool areas 1 to 12, respectively.

Punch forbidden area–entered signal WDD0 <F231#7>

[Classification] Output signal

[Function] When the type B safety zone check is used, this signal notifies the PMC that the tool has entered a punch forbidden area.

[Output condition] See “7.2 Type B”.

Punch enable signal WDC <G232#4>

[Classification] Input signal

[Function] When the type B safety zone check is used, this signal instructs the CNC to perform punching, even if the tool has entered a punch forbidden area, and the WDD0 signal is set to 1.

[Operation] See “7.2 Type B”.

Workpiece holder detection command SAFZ <X1004#1>

[Classification] Input signal

[Function] This signal notifies the CNC that the workpiece holder position has been detected with the automatic safety zone setting function.

[Operation] See “7.5 Automatic safety–zone setting”.

Automatic safety zone setting request signal SAFRQ <G232#5>

[Classification] Input signal

[Function] This signal notifies the CNC that automatic safety zone setting, based on an external signal, has been performed.

[Operation] Refer to 7.5.2 “Detecting workpiece holder position using an external signal”.

Safety zone setting ready signal SAFEXE <F231#6>

[Classification] Output signal

[Function] This signal notifies the PMC that the position set in parameter No. 16535 has been reached during automatic safety zone setting, based on an external signal.

[Output condition] Refer to 7.5.2 “Detecting workpiece holder position using an external signal”.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
X1004							SAFZ	
G232			SAFRQ	WDC	SZTS3	SZTS2	SZTS1	SZTS0
F231	WDD0	SAFEXE						
F240	SZT8	SZT7	SZT6	SZT5	SZT4	SZT3	SZT2	SZT1
F241					SZT12	SZT11	SZT10	SZT9

7.7 PARAMETER

	#7	#6	#5	#4	#3	#2	#1	#0
16500	YSF		SAT					SF0

[Data type] Bit

SF0 The safety zone of type:
0 : A is used.
1 : B is used.

SAT When punching is inhibited in the safety zone, the block in which a T command is specified is checked:
0 : In advance.
1 : After the FIN signal to complete the T command has been received.

YSF When a safety zone check is executed, the inhibited area along the Y axis extends from the values set in parameters 16507, 16510, 16513, and 16516:
0 : In the negative direction.
1 : In the positive direction.

	#7	#6	#5	#4	#3	#2	#1	#0
16501					SZ4	SZ3	SZ2	SZ1

[Data type] Bit

SZj When a safety zone check is executed, in the #j (j = 1 to 4) area,
0 : An entry is inhibited.
1 : Punching is inhibited.

	#7	#6	#5	#4	#3	#2	#1	#0
16502	SOF			SZI	GSZ	ZIO	SZC	MDP

[Data type] Bit

MDP On the safety zone setting display,
0 : The workpiece coordinate system is indicated.
1 : The machine coordinate system is indicated.

SZC On the safety zone setting display, the data for:
0 : Any zone can be changed.
1 : Those zones to be set automatically (parameter 16534) can be changed.

ZIO When the safety zone is automatically set by an external signal, the position of a workpiece holder is detected according to:
0 : The on and off states of the SAFZ signal used to detect the position of a workpiece holder.
1 : The on state of the SAFZ signal used to detect the position of a workpiece holder.

GSZ On the graphic screen, the safety zone is checked according to the position of a workpiece holder:
0 : Specified on the safety zone screen.
1 : Specified by graphic parameters.
(On the graphic screen, this check is executed in an area that is not related to the actual machining check.)

SZI Data set on the safety zone setting display is:

0 : Invalidated.
1 : Validated.

SOF In the safety zone check, tool position compensation is:

0 : Not considered.
1 : Considered.

16505	Positive X coordinate for safety zone 1
16506	Negative X coordinate for safety zone 1
16507	Y coordinate for safety zone 1
16508	Positive X coordinate for safety zone 2
16509	Negative X coordinate for safety zone 2
16510	Y coordinate for safety zone 2
16511	Positive X coordinate for safety zone 3
16512	Negative X coordinate for safety zone 3
16513	Y coordinate for safety zone 3
16514	Positive X coordinate for safety zone 4
16515	Negative X coordinate for safety zone 4
16516	Y coordinate for safety zone 4

[Data type] Two–word

[Unit of data]

Increment system	IS–A	IS–B	Unit
Millimeter machine	0.01	0.001	mm
Inch machine	0.001	0.0001	inch

[Valid data range] 0 to □99999999

Each set of the parameters specify safety zone 1, 2, 3, or 4.

Note 1 The values set for zone #n must be smaller than those set for zone #(n + 1). (n: 1 to 3)

Note 2 Zeros must be specified for zones which need not be specified.

16517	Size of tool area 1 in the X direction for the safety zone function
16518	Size of tool area 1 in the Y direction for the safety zone function
16519	Size of tool area 2 in the X direction for the safety zone function
16520	Size of tool area 2 in the Y direction for the safety zone function
16521	Size of tool area 3 in the X direction for the safety zone function
16522	Size of tool area 3 in the Y direction for the safety zone function
16523	Size of tool area 4 in the X direction for the safety zone function
16524	Size of tool area 4 in the Y direction for the safety zone function
16525	Size of tool area 5 in the X direction for the safety zone function
16526	Size of tool area 5 in the Y direction for the safety zone function
16527	Size of tool area 6 in the X direction for the safety zone function
16528	Size of tool area 6 in the Y direction for the safety zone function
16529	Size of tool area 7 in the X direction for the safety zone function
16530	Size of tool area 7 in the Y direction for the safety zone function
16531	Size of tool area 8 in the X direction for the safety zone function
16532	Size of tool area 8 in the Y direction for the safety zone function

[Data type] Two–word

[Unit of data]

Increment system	IS–A	IS–B	Unit
Millimeter machine	0.01	0.001	mm
Inch machine	0.001	0.0001	inch

[Valid data range] 0 to 99999999

The parameters set 12 tool areas for the safety zone function.
Refer to parameters No. 16551 to No. 16558

Note 1 A tool area is selected by signals SZTS0 to SZTS3 input from a PMC machine.

16533	Distance between the position detector of the workpiece holder and the punch

[Data type] Two–word

[Unit of data]

Increment system	IS–A	IS–B	Unit
Millimeter machine	0.01	0.001	mm
Inch machine	0.001	0.0001	inch

[Valid data range] 0 to □99999999

The parameter sets the distance between the position detector of the workpiece holder and punch.
The sign of the value set in the parameter corresponds to the direction the machine travels along the X–axis, assuming the punch position as zero.

16534 Number of zones to be detected for automatic setting

[Data type] Byte

[Unit of data] Piece

[Valid data range] 0 to 4

This parameter specifies the number of zones to be detected for automatic setting of a safety zone by an external signal.

Note 1 This parameter must be specified when automatic setting is executed.

16535 Retraction position from the X–axis reference position for automatic setting

[Data type] Two–word

[Unit of data]

Increment system	IS–A	IS–B	Unit
Millimeter machine	0.01	0.001	mm
Inch machine	0.001	0.0001	inch

[Valid data range] 0 to □99999999

This parameter specifies a clearance from the X–axis reference position for automatic setting of a safety zone by means of an external signal.

Note 1 Specify a position that is sufficiently distant to allow the speed at which the position of a workpiece holder is detected to become stable.

16536 X–axis rapid traverse rate for automatic setting

[Data type] Two–word

[Unit of data]

Increment system	Unit of data	Valid data range
Millimeter machine	1 mm/min	30 to 24000
Inch machine	0.1 inch/min	30 to 9600

This parameter specifies an X–axis rapid traverse rate for automatic setting of a safety zone by an external signal.

Note 1 When this parameter is set to 0, the feedrate along the X–axis for automatic detection equals the manual rapid traverse rate.

16537 X–axis rapid traverse time constant for automatic setting

[Data type] Word

[Unit of data] msec

[Valid data range] 8 to 4000

This parameter specifies an X–axis rapid traverse time constant for the automatic setting of a safety zone by an external signal.

Note 1 When this parameter is set to 0, the X–axis time constant for automatic detection equals the time constant for manual rapid traverse.

16538	Lower limit of position error for movement along the X–axis for automatic setting
16539	Upper limit of position error for movement along the X–axis for automatic setting

[Data type] Two–word

[Valid data range] 0 to 99999999

[Unit of data] Units of detection

These parameters specify the lower and upper limits, for the position error for movement along the X–axis , for the automatic setting of a safety zone by an external signal. These parameters must be specified for automatic setting.

Note 1 The values of these parameters must satisfy the following condition: Parameter 16538 <Parameter 16539

16540	Width of workpiece holder 1 along the X–axis for automatic setting
16541	Width of workpiece holder 2 along the X–axis for automatic setting
16542	Width of workpiece holder 3 along the X–axis for automatic setting
16543	Width of workpiece holder 4 along the X–axis for automatic setting

[Data type] Two–word

[Unit of data]

Increment system	IS–A	IS–B	Unit
Millimeter machine	0.01	0.001	mm
Inch machine	0.001	0.0001	inch

[Valid data range] 0 to 99999999

Each of the parameters specifies the width of a workpiece holder along the X–axis for the automatic setting of a safety zone by an external signal. The parameter values correspond to safety zones 1 to 4, specified in parameters 16505 to 16516, respectively.
When automatic setting is executed, these parameters must be set.

16551	X dimension of tool area 9 for the safety zone function
16552	Y dimension of tool area 9 for the safety zone function
16553	X dimension of tool area 10 for the safety zone function
16554	Y dimension of tool area 10 for the safety zone function
16555	X dimension of tool area 11 for the safety zone function
16556	Y dimension of tool area 11 for the safety zone function
16557	X dimension of tool area 12 for the safety zone function
16558	Y dimension of tool area 12 for the safety zone function

[Data type] Two–word

[Unit of data]

Increment system	IS–A	IS–B	Unit
Millimeter machine	0.01	0.001	mm
Inch machine	0.001	0.0001	inch

[Valid data range] 0 to 99999999

The parameters set 12 tool areas for the safety zone function.

Refer to parameters No. 16517 to No. 16532

Note 1 A tool area is selected by signals SZTS0 to SZTS3 input from a PMC machine.

7.8 ALARM AND MESSAGE

Number	Message	Contents
4800	ZONE : PUNCHING INHIBITED 1	When a safety zone check was executed, a punch command was specified in area 1 where punching is inhibited.
4801	ZONE : PUNCHING INHIBITED 2	When a safety zone check was executed, a punch command was specified in area 2 where punching is inhibited.
4802	ZONE : PUNCHING INHIBITED 3	When a safety zone check was executed, a punch command was specified in area 3 where punching is inhibited.
4803	ZONE : PUNCHING INHIBITED 4	When a safety zone check was executed, a punch command was specified in area 4 where punching is inhibited.
4810	ZONE : ENTERING INHIBITED 1 +X	When a safety zone check was executed, the machine moving in the positive X direction entered area 1 into which entry is inhibited.
4811	ZONE : ENTERING INHIBITED 1 –X	When a safety zone check was executed, the machine moving in the negative X direction entered area 1 into which entry is inhibited.
4812	ZONE : ENTERING INHIBITED 2 +X	When a safety zone check was executed, the machine moving in the positive X direction entered area 2 into which entry is inhibited.
4813	ZONE : ENTERING INHIBITED 2 –X	When a safety zone check was executed, the machine moving in the negative X direction entered area 2 into which entry is inhibited.
4814	ZONE : ENTERING INHIBITED 3 +X	When a safety zone check was executed, the machine moving in the positive X direction entered area 3 into which entry is inhibited.
4815	ZONE : ENTERING INHIBITED 3 –X	When a safety zone check was executed, the machine moving in the negative X direction entered area 3 into which entry is inhibited.
4816	ZONE : ENTERING INHIBITED 4 +X	When a safety zone check was executed, the machine moving in the positive X direction entered area 4 into which entry is inhibited.
4817	ZONE : ENTERING INHIBITED 4 –X	When a safety zone check was executed, the machine moving in the negative X direction entered area 4 into which entry is inhibited.
4830	ZONE : ENTERING INHIBITED 1 +Y	When a safety zone check was executed, the machine moving in the positive Y direction entered area 1 into which entry is inhibited.
4831	ZONE : ENTERING INHIBITED 1 –Y	When a safety zone check was executed, the machine moving in the negative Y direction entered area 1 into which entry is inhibited.
4832	ZONE : ENTERING INHIBITED 2 +Y	When a safety zone check was executed, the machine moving in the positive Y direction entered area 2 into which entry is inhibited.
4833	ZONE : ENTERING INHIBITED 2 –Y	When a safety zone check was executed, the machine moving in the negative Y direction entered area 2 into which entry is inhibited.
4834	ZONE : ENTERING INHIBITED 3 +Y	When a safety zone check was executed, the machine moving in the positive Y direction entered area 3 into which entry is inhibited.
4835	ZONE : ENTERING INHIBITED 3 –Y	When a safety zone check was executed, the machine moving in the negative Y direction entered area 3 into which entry is inhibited.
4836	ZONE : ENTERING INHIBITED 4 +Y	When a safety zone check was executed, the machine moving in the positive Y direction entered area 4 into which entry is inhibited.
4837	ZONE : ENTERING INHIBITED 4 –Y	When a safety zone check was executed, the machine moving in the negative Y direction entered area 4 into which entry is inhibited.
4870	AUTO SETTING FEED ERROR	The feed rate of safety zone auto setting is other than the parameter value (No. 16538, No. 16539).
4871	AUTO SETTING PIECES ERROR	In safety zone auto setting, the safety zone pieces are not correct. Or the position detector has gone wrong, please tell your machine tool builder.
4872	AUTO SETTING COMMAND ERROR	M code, S code or T code is specified with safety zone auto setting command (G32). G32 is specified in the nibbling mode, in the cutter compensation, in the rotation mode or the scaling mode.

8 WORKPIECE HOLDER INTERFERENCE AVOIDANCE FUNCTION

General

If the tool is positioned to the normal height (for punching), as shown below, the tool will interfere with the workpiece holder when the workpiece holder moves into the turret.

By means of this function, the CNC monitors the positions of the tool and workpiece holder to avoid interference between the workpiece holder and tool.

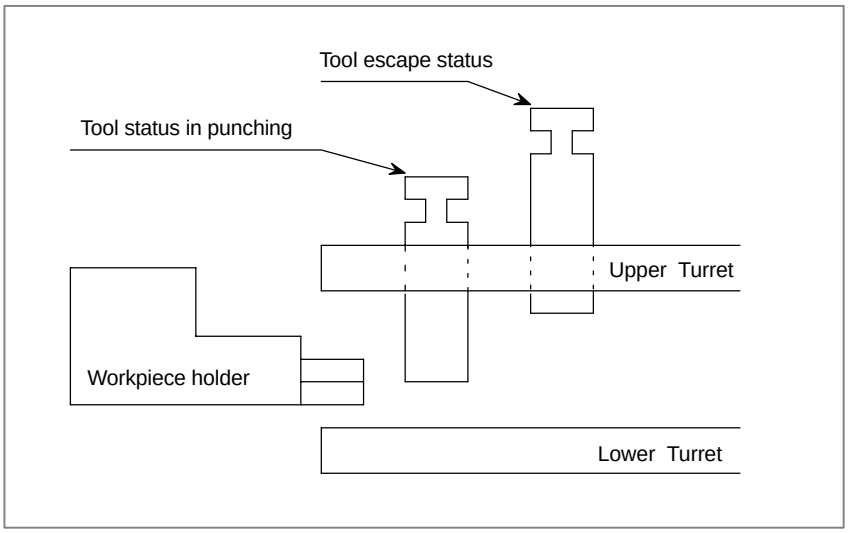


Fig. 8

By setting bit 6 (ACZ) of parameter No. 16502, the user can select either of two interference avoidance methods:

1 Type A: Interference is avoided by the PMC, by using the DI/DO signals.
2 Type B: Interference is automatically avoided by the CNC.

8.1 TYPE A

Specification

If the CNC makes a check to find the positioning path before the start of positioning, and the check reveals interference between the tool area and workpiece holder area, the CNC sets tool escape signal WHAL (F231#5), sent to the PMC, to 1, and simultaneously starts positioning.

When the WHAL signal is received, the PMC uses the axis interlock signal to stop tool movement along the axes. Then, the PMC retracts the tool to a position (tool escape position shown in Fig. 8) where it does not interfere with the workpiece holder. After the completion of an escape operation, the PMC sets tool escape completion signal WHALC (G232#7) to 1, and also releases the axis interlock signal to restart positioning.

When the WHALC signal is received, the CNC sets the WHAL signal to 0.

In nibbling operation

If, during nibbling, the tool enters an interference area during the next positioning operation, press activation signal PF (Y1004#2) and continuous press activation in–progress signal NBL (Y1004#1) are set to 0 when press activation stop signal *PE (X1004#7) for the current punching operation is set to 0. Moreover, the WHAL signal is set to 1 when punching completion signal *NFIN (X1004#6) for the continuous press is set to 0. Then, as in the case where 1–cycle press select signal CPS (G230#2) is set to 1, 1–cycle punch operation is performed until the tool moves out of the interference area.

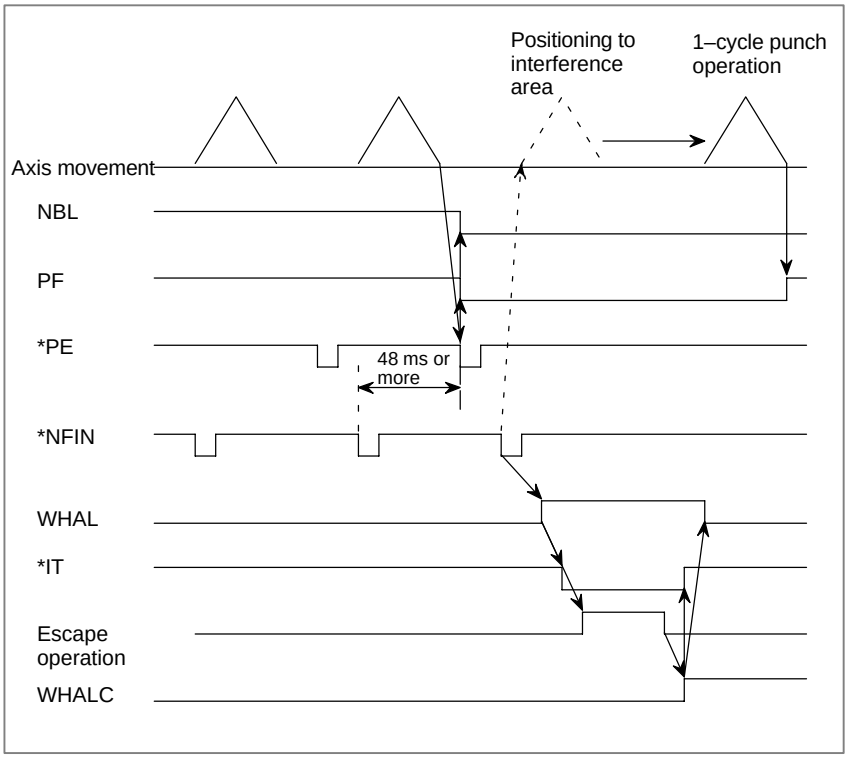


Fig. 8.1 (a)

CAUTION
After the *NFIN signal has been set to 0, at least 48 ms is required for the *PE signal to be set to 0. If this period is not provided, the *PE signal may not be used to set the PF and NBL signals to 0 and set the WHAL signal to 1.

Check of interference area

During automatic operation, the WHAL signal is set to 1 for any positioning operation that may cause the tool area to interfere with the workpiece holder area. (Fig. 8.1(b).)

When manual operation is performed, the WHAL signal is set to 1 only after the tool enters an interference area.

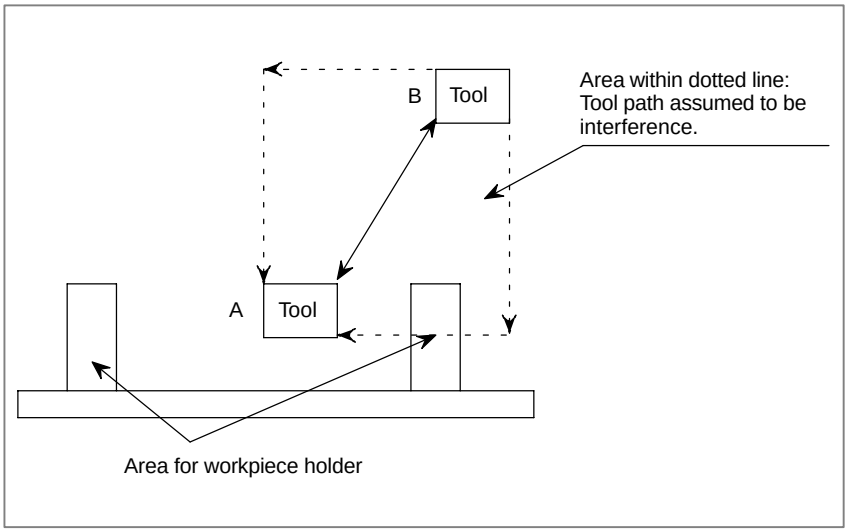


Fig. 8.1 (b) Positioning from A to B

NOTE
See also Section 8.2.

8.2 TYPE B

Specification

If, during automatic operation, a positioning operation may cause the tool area to interfere with the workpiece holder area, this function first moves the tool along a non–interfering axis, which may be either the X–axis or Y–axis, then moves the tool along the other axis, thus preventing the occurrence of interference.

This avoidance operation is performed only when the avoid operation signal ACZEXE (G232#6) is set to 1. So, if the tool does not interfere with the workpiece holder, normal positioning can be used for machining by setting the ACZEXE signal to 0.

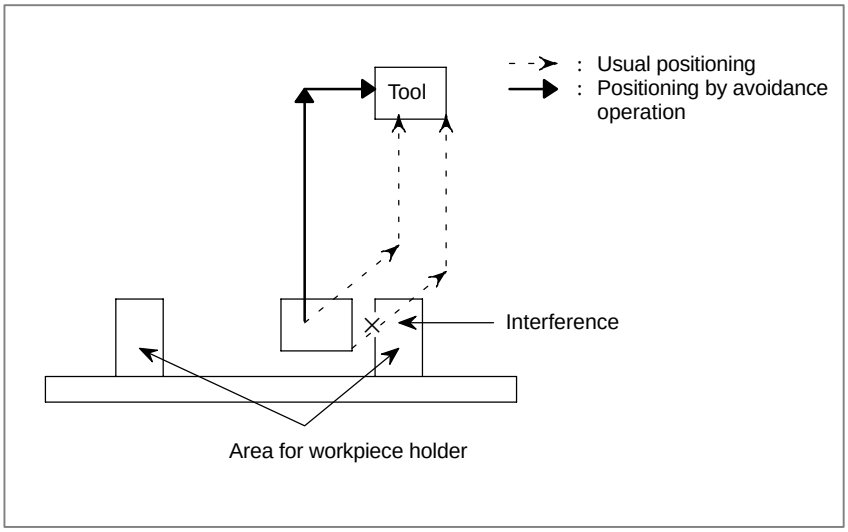
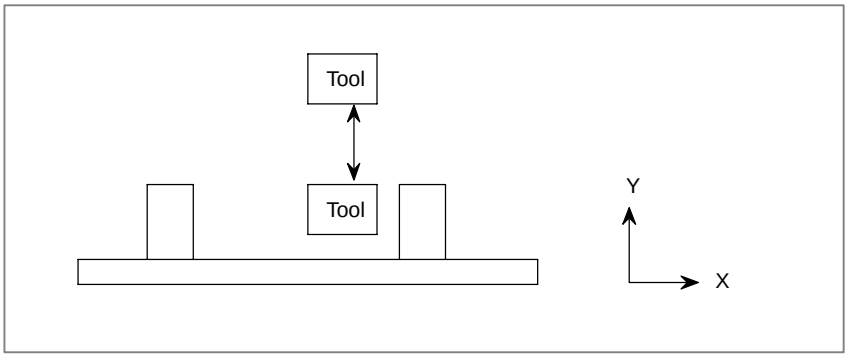


Fig. 8.2 (a)

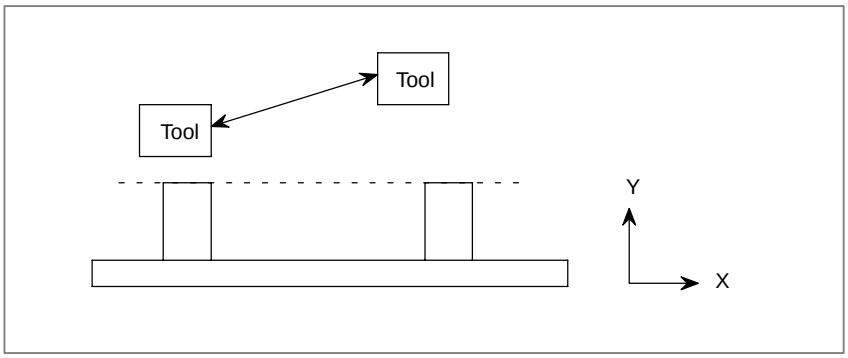
Determination of whether to perform avoidance operation

Avoidance operation is performed in all cases except those described below.

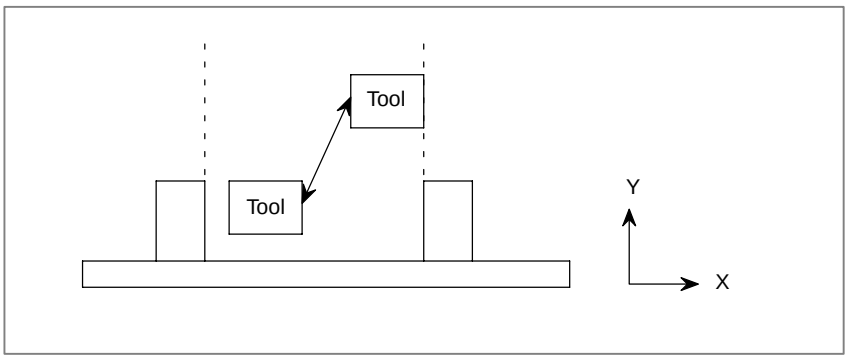
(1) When the tool does not move along the X–axis



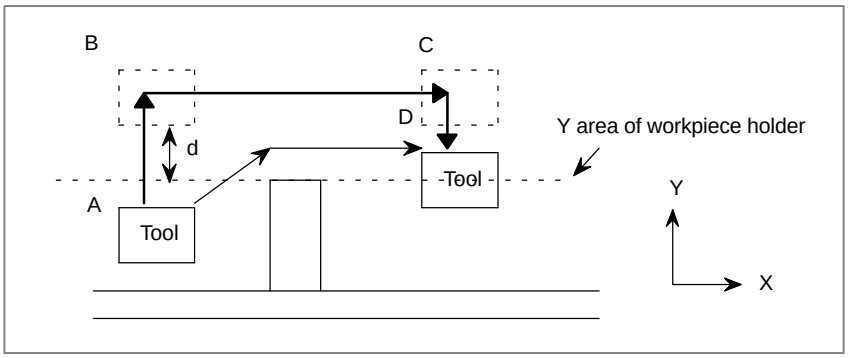
(2) When the start and end points of movement along the Y–axis are above the Y area of the workpiece holders



(3) When the tool does not cross the X area of a workpiece holder for movement along the X–axis



When both the start and end points on the Y–axis are in the Y area of the workpiece holders, the following avoidance operation is performed:



The avoidance operation modifies the positioning path from A to D to that from A to B to C to D.

1 Amount of travel between A and B on the Y–axis
= (workpiece holder position) + (tool area/2) + d
– (current position A)
d: Avoidance operation offset (parameter No. 16561)

2 Amount of travel between B and C along the X–axis =
(X command value)

3 Amount of travel between C and D along the Y–axis
= (Y command position D) – (current position C)

CAUTION

If feed hold signal *SP (G008#5) is set to 0 during an avoidance operation, the tool stops upon the completion of the avoidance operation.

8.3 TOOL/WORKPIECE HOLDER AREAS

(1) The workpiece holder area (parameter Nos. 16505 to 16516) of the safety zone function is used.

(2) The tool area (parameter Nos. 16517 to 16532, 16551 to 16558) of the safety zone function is used.

(3) When the WHAL signal (F231#5) is output, the tool area can be offset (parameter Nos. 16559, 16560).

8.4 SIGNAL

Tool escape completion signal WHALC <G232#7>

[Classification] Input signal

[Function] This signal posts the completion of tool escape.

[Operation] When this signal is set to 1, the WHAL signal is set to 0.

Tool escape signal WHAL <F231#5>

[Classification] Output signal

[Function] This signal posts notification that the tool may interfere with the workpiece holder during tool positioning.

[Output condition] This signal is set to 1 in the following cases:

(1) When, during automatic operation, a positioning operation that may cause the tool to interfere with the workpiece holder is started

(2) When, during manual operation, the tool enters the interference area of a workpiece holder

This signal is set to 0 in the following cases:

(1) When the WHALC signal, set to 1, is received

(2) In the reset state

CAUTION

If, during manual operation, the tool enters an interference area, for example, this signal is set to 0 only after the tool is moved out of the interference area.

Avoidance operation signal ACZEXE <G232#6>

[Classification] Input signal

[Function] This signal specifies whether to perform avoidance operation according to the type B specification.

[Operation] When this signal is set to 1, avoidance operation is performed as part of a positioning operation that may cause the tool to interfere with the workpiece holder.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G232	WHALC	ACZEXE						

	#7	#6	#5	#4	#3	#2	#1	#0
F231			WHAL					

8.5 PARAMETER

	#7	#6	#5	#4	#3	#2	#1	#0
16502		ACZ						

[Data type] Bit

ACZ The function used to prevent interference between workpiece holders of:

0 : Type A is used.

1 : Type B is used.

16559	Width of tool area along the X–axis for the function used to prevent interference between workpiece holders
16560	Width of tool area along the Y–axis for the function used to prevent interference between workpiece holders

[Data type] Two–word

[Unit of data]

Increment system	IS–A	IS–B	Unit
Millimeter machine	0.01	0.001	mm
Inch machine	0.001	0.0001	inch

[Valid data range] 0 to 99999999

Each of the parameters is specified to add a margin to the tool area (parameters 16517 to 16532, 16551 to 16558) and safety zone area (parameters 16505 to 16516) is checked, to output the WHAL signal (F231, #5) by the function used to prevent interference between workpiece holders. The parameter value is added to the tool area width when interference is checked.

16561	Compensation value used by the function used to prevent interference between workpiece holders

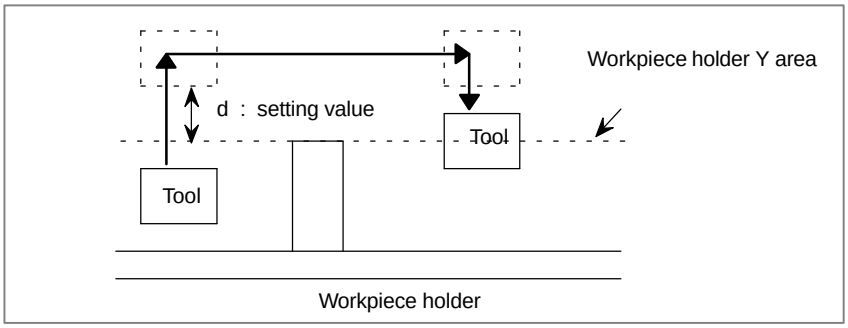
[Data type] Two–word

[Unit of data]

Increment system	IS–A	IS–B	Unit
Millimeter machine	0.01	0.001	mm
Inch machine	0.001	0.0001	inch

[Valid data range] 0 to 99999999

This parameter specifies a compensation value for retraction along the Y–axis when the function used to prevent interference between workpiece holders of type B is used.



8.6 NOTE

1. This function is optional.
2. The optional safety zone function must be specified at the same time.
3. The operation of this function is based on the safety zone function. So, see the specification of the safety zone function.
4. If the workpiece holder area is specified as a punch forbidden area (parameter No. 16501, SZj) according to the type B specification, a check is made before axis movement, even when the positioning and press–off command (G70) is specified.
5. Signals of type A can also be used when the type B specification is selected.

9 CONTROL FUNCTION

9.1 FEED HOLD SIGNAL B/FEED HOLD LAMP SIGNAL B

Feed hold signal B *SPR <X1008#5>

[Classification] Input signal

[Function] This signal suspends automatic operation.

[Operation] This signal has the same effect as feed hold signal *SP (G008#5). The CNC reads the input data directly, however, not via the PMC. This signal is valid when bit 4 (SPR) of parameter No. 16002 is set to 1.

Feed hold lamp signal B SPRL <F230#4>

[Classification] Output signal

[Function] This signal notifies the PMC that automatic operation has been suspended.

[Output condition] This signal has the same effect as feed hold lamp signal SPL (F000#4). This signal, however, is set to 1 when automatic operation is suspended by feed hold signal B (*SPR).

9.2 RESET KEY SIGNAL

RSTSW <F239#5>

[Classification] Output signal

[Function] This signal notifies the PMC that the reset key on the CRT/MDI panel has been pressed.

[Output condition] This signal is set to 1 while the reset key on the CRT/MDI panel is held down; the signal reverts to 0 when the reset key is released.

9.3 SWITCHING THE PUNCHING AND LASER MODES

The punching mode and laser mode can be switched by specifying M codes in parameters. An M code is specified in the first block for punching and for laser machining in a machining program. This will improve processing precision in the interpolation mode of the laser mode. In the laser mode, punching is not performed even in blocks where positioning is done by rapid traverse along the X and Y axes. Manual punching, however, is possible. Note that pattern commands such as G26 and nibbling commands such as G68 cannot be specified. (These commands cause alarm No. 4630.)

Examples

```
G92X1000.Y800.;
M40;                    ┐
G72X200.Y200.;          ├ Punching mode
G26I50.J0K10;T101;      ┘
M41;                    ┐
G90G00X100. Y100.;      │
G01X300.F300;           ├ Laser mode
Y300.;                  │
X100.;                  │
Y100.;                  ┘
M40;
G28M30;
```

WARNING

1 The punching mode and laser mode can be switched only when ALA in parameter No. 16240#0 is set to 1.
Set the M code to the parameter (No. 16244, 16245).

2 In the laser mode, the following parameters are invalid. The typical settings for the rapid traverse rate, rapid traverse time constant, and position control servo loop gain become valid.
Bit 4 of parameter LPG (No. 16051#4)
Bit 7 of parameter KLV (No. 16050#7)

3 The punching mode or the laser mode is entered immediately after the power is turned on or the system is reset. (Laser mode is entered when the parameter RLM (No. 16240#5) is set.)

4 The M codes for switching the punching and laser modes must be specified in a single block.

Signal

Laser mode set signal LASMOD <F239#6>

[Classification] Output signal

[Function] This signal notifies the PMC that laser mode is set.

[Output condition] When laser mode is set, this signal is set to 1. When punch mode is set, this signal is set to 0.

9.4 2–ms INTERVAL ACCELERATION/ DECELERATION FOR RAPID TRAVERSE

Overview

This function performs acceleration/deceleration for rapid traverse at 2 ms intervals in order to make servo motion smooth, thereby reducing the settling time required after positioning.

Specification

This function is valid for an axis for which the 2MPj parameter (bit 1 of parameter No. 1605) is set to 1. For any axis for which the 2MPj parameter is 1, acceleration/deceleration for rapid traverse (G00, G28, G70, manual rapid traverse, etc.) is performed at 2 ms intervals.

Time constant setting

- The time constant related to rapid traverse is specified in 2 ms units.
- When the rapid traverse bell–shaped acceleration/deceleration function is used, the valid data for parameter No. 1621 ranges from 0 to 128.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
16054							2MPj	

[Data type] Bit axis

2MPj Specify the acceleration/deceleration time for the rapid traverse command.

0 : 8ms
1 : 2ms

WARNING

1 This function can be specified for individual axes. However, it is recommended that the function be specified for no more than 4 axes. If this function is specified for more than 4 axes, other optional configurations (if any) may cause the operability and the processing speed of the CNC to greatly degrade, disabling normal control.
2 This function cannot be used together with the look–ahead control function.

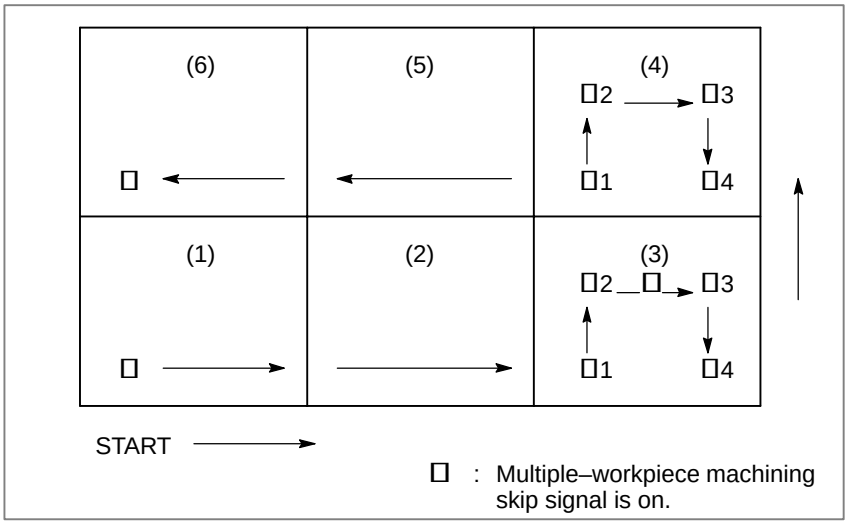
9.5 MULTIPLE–WORK-PIECE MACHINING RETRACE FUNCTION

Overview

If a multiple–workpiece machining skip signal is input for a retrace re–forward movement during multiple–workpiece machining, machining of the current workpiece is stopped and machining of another workpiece begins.

Examples

```
G98 X__ Y__ I__ J__ P2 K1
U1
X50. Y50. T505 (□1)
Y150.          (□2)
X200.          (□3)
Y50.           (□4)
V1
G75 W1 Q1
```



In a multiple–workpiece machining sequence for workpieces 1 to 6, if a multiple–workpiece machining skip signal is input at position □ during a retrace backward or re–forward movement in machining workpiece (3), the machining of workpiece (3) is stopped, and the machining of workpiece (4) begins.

Operation

1) If a multiple–workpiece machining skip signal (G248#3) is input:
   (1) Machining continues to 3 and stops there.
   (2) The multiple–workpiece machining skip preparation signal (F225#0) becomes 1.
2) If machining is restarted:
   (1) The multiple–workpiece machining skip preparation signal (F225#0) becomes 0.

(2) The multiple–workpiece machining skip completion signal (F225#2) becomes 1.

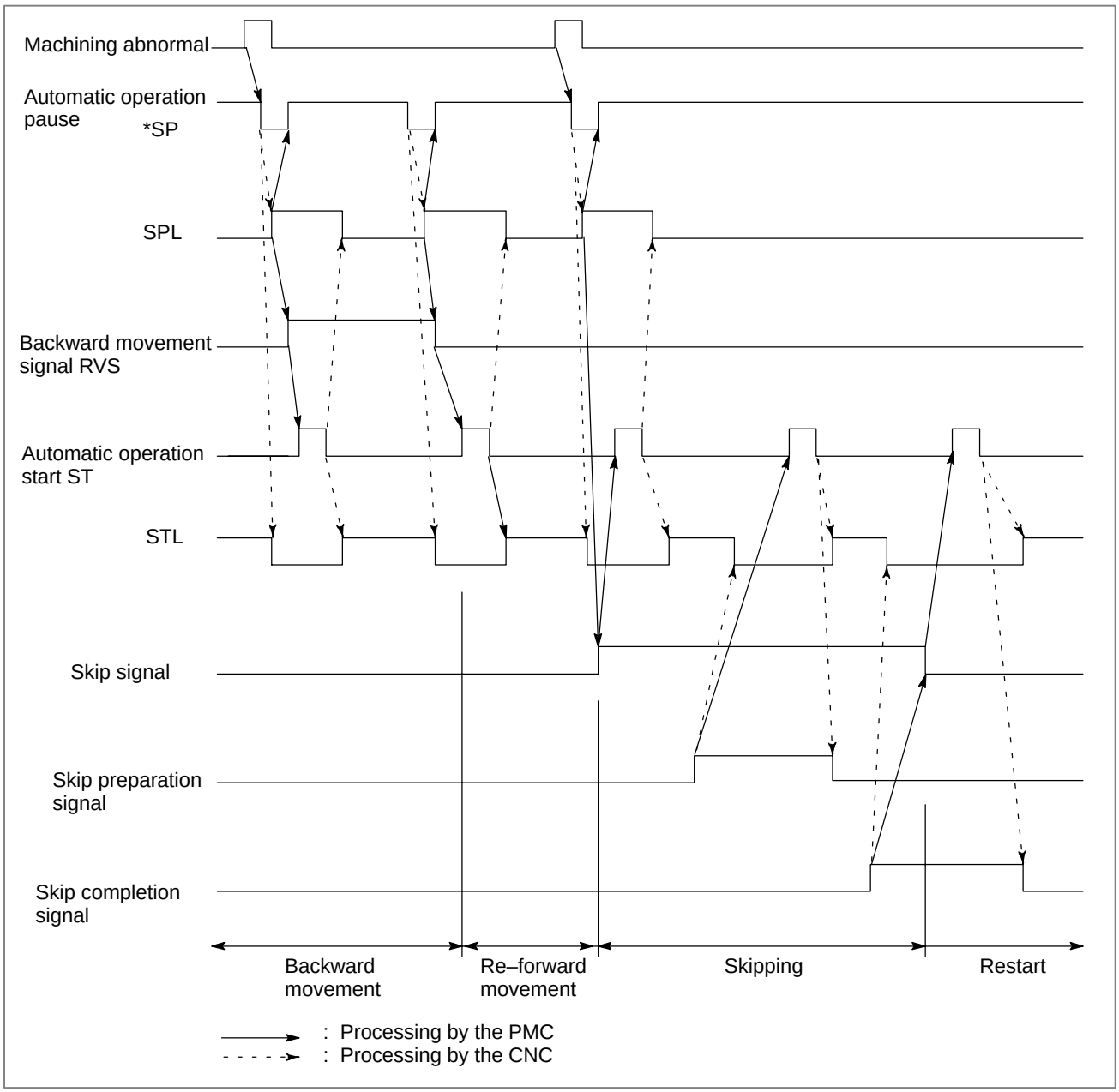
In this case, no axis movement occurs.

A skip should be performed on the machine side at the timing of the multiple–workpiece skip preparation or completion signal, as required.

3) If machining is restarted again.

(1) The multiple–workpiece machining skip completion signal (F225#2) becomes 0.

(2) The machining moves to the first block (□1) for the next workpiece, where machining is restarted.



NOTE

(1) The multiple–workpiece skip function is an option.

(2) It is necessary to specify the retrace function option simultaneously with the multiple–workpiece machining function. Refer to the applicable description of the retrace function.

CAUTION

(1) A skip does not occur during trial multiple–workpiece machining (when the setting of multiple–workpiece machining is 1).

(2) The multiple–workpiece skip signal can be detected only during a re–forward movement. The skip signal cannot be detected during a forward or backward movement.

(3) If the multiple–workpiece machining skip signal is input for a certain block, the machining moves to the end of that block and stops there. A skip occurs when machining is restarted.

(4) If a re–forward movement is stopped by feed hold, and the multiple–workpiece machining skip signal is input for a certain block, then machining is restarted, the machine moves to the end of that block and stops there. A skip occurs when machining is restarted.

(5) When a multiple–workpiece machining skip occurs, a backward movement for the skipped workpiece becomes impossible. If a backward movement is attempted for the next workpiece, the backward movement ends at the skipped position.

(6) When a multiple–workpiece machining skip occurs, the following information is re–set to the state in which it was when multiple–workpiece machining was started.

1) Absolute/incremental (G90/G91)
2) Cutter compensation mode (G40/G41/G42)
3) Plane selection (G17/G18/G19)
4) Programming data input (G10/G11)
5) Inch/metric input (G20/G21)
6) Stored stroke limit (G22/G23)
7) Punch/laser mode
8) Forming mode/cancel
9) Workpiece clamp/unclamp
10) Nibbling mode/cancel

Basically, the modes mentioned above must be canceled for a multiple–workpiece machining command.

(7) The multiple–workpiece machining function cannot be used together with:

1) Coordinate system rotation
2) Scaling, or
3) Advance T–code specification

Signal

Multiple–workpiece machining skip preparation signal RVSARV<F225#0>

[Classification] Output signal

[Function] Indicates to the PMC when the machine is ready for a multiple–workpiece machining skip.

[Output condition] This signal becomes 1 at the end of a block where the multiple–workpiece machining skip signal is input. It becomes 0 at the start of the next cycle.

Multiple–workpiece machining skip completion signal RVSSKE<F225#2>

[Classification] Output signal

[Function] Indicates to the PMC when a multiple–workpiece machining skip is completed.

[Output condition] If the RVSARV signal is 1, the RVSSKE signal becomes 1 at the completion of a skip after a cycle is started. It becomes 0 at the start of the next cycle.

Multiple–workpiece machining skip signal MGSKP<G248#3>

[Classification] Input signal

[Function] Specifies whether to perform a multiple–workpiece machining skip.

[Output condition] When this signal is 1, a multiple–workpiece machining skip is performed.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G248					MGSKP			

	#7	#6	#5	#4	#3	#2	#1	#0
F225						RVSSKE		RVSARV

APPENDIX

A CNC and PMC Interface

A.1 ADDRESS LIST

The addresses of the interface signals used to communicate between the CNC and PMC are as follows:

(1) Standard signals

Refer to the “FANUC Series 16/18/160/180–MODEL C Connection Manual (B–62753EN–1).”

(2) Signals dedicated to 16/18/160/180–PC

See the addresses below.

MT□PMC

Address	Bit number							
	#7	#6	#5	#4	#3	#2	#1	#0
X1000								
X1001								
X1002								
X1003								
X1004	*PE	*NFIN	*PFIN	PFWB			SAFZ	SKIP
X1005								
X1006								
X1007								
X1008			*SPR	*ESP				
X1009	*DEC8	*DEC7	*DEC6	*DEC5	*DEC4	*DEC3	*DEC2	*DEC1

PMC□MT

Address	Bit number							
	#7	#6	#5	#4	#3	#2	#1	#0
Y1000								
Y1001								
Y1002								
Y1003								
Y1004					PFB	PF	NBL	

PMC□CNC

Address	Bit number							
	#7	#6	#5	#4	#3	#2	#1	#0
G230	EFS	SNP	EPE	MNS	MPS	CPS	PFW	PFL
G231							MLP2	MLP1
G232	WHALC	ACZEXE	SAFRQ	WDC	SZTS3	SZTS2	SZTS1	SZTS0
G233	CFDC	TCNG	TNG				*JVT2	*JVT1
G234	TI07	TI06	TI05	TI04	TI03	TI02	TI01	TI00
G235	TI15	TI14	TI13	TI12	TI11	TI10	TI09	TI08
G236	TI23	TI22	TI21	TI20	TI19	TI18	TI17	TI16
G237	TI31	TI30	TI29	TI28	TI27	TI26	TI25	TI24
G238								
G239								
G240								
G241								
G242								
G243								
G244								
G245								
G246								
G247								
G248					MGSKP		BTFIN	
G249								

CNC□PMC

Address	Bit number							
	#7	#6	#5	#4	#3	#2	#1	#0
F225						RVSSKE		RVSARV
F230	NBLE	DPF	DSPF	SPRL				
F231	WDDO	SAFEXE	WHAL					
F232	MIE	TIE	CMOK	CPEN			BTF	PTLCH
F233							SC2	SC1
F234	PN07	PN06	PN05	PN04	PN03	PN02	PN01	PN00
F235	PN15	PN14	PN13	PN12	PN11	PN10	PN09	PN08
F236	PN23	PN22	PN21	PN20	PN19	PN18	PN17	PN16
F237	PN31	PN30	PN29	PN28	PN27	PN26	PN25	PN24
F238								
F239		LASMOD	RSTSW					
F240	SZT8	SZT7	SZT6	SZT5	SZT4	SZT3	SZT2	SZT1
F241					SZT12	SZT11	SZT10	SZT9
F242								
F243								
F244	RP8T	RP7T	RP6T	RP5T	RP4T	RP3T	RP2T	RP1T
F245	RP16T	RP15T	RP14T	RP13T	RP12T	RP11T	RP10T	RP9T
F246	BT07	BT06	BT05	BT04	BT03	BT02	BT01	BT00
F247	BT15	BT14	BT13	BT12	BT11	BT10	BT09	BT08
F248	BT23	BT22	BT21	BT20	BT19	BT18	BT17	BT16
F249	BT31	BT30	BT29	BT28	BT27	BT26	BT25	BT24

A.2
LIST OF SIGNALS

A.2.1
List of Signals in the Order of Functions

Function	Name	Symbol	Address	Item
External operation function	External operation function select signal	EFS	G230#7	3.7
Tool life management	Tool expired signal	PTLCH	F232#0	2.9.2
C–axis offset	Coupling enable signal	CPEN	F232#4	6.3.2
	C–axis unconnected signal	CFDC	G233#7	6.3.2
C–axis synchronization control	C–axis control status signal	SC1,SC2	F233#0,#1	6.2
Feed hold signal B	Feed hold signal B	*SPR	X1008#5	9.1
Feed hold lamp signal B	Feed hold lamp signal B	SPRL	F230#4	9.1
Skip function	Skip signal	SKIP	X1004#0	2.11.1
Safety zone check	Punch enable signal	WDC	G232#4	7.6
	Punch forbidden area entered signal	WDDO	F231#7	7.6
	Safety zone setting ready signal	SAFEXE	F231#6	7.6
	Workpiece holder position detector signal	SAFZ	X1004#1	7.5
	Selection signal for tool area of safety zone	SZTS0–SZTS3	G232#0–#3	7.6
	In safety zone signal	SZT1–SZT12	F240–F241#0–#3	7.6
	Automatic safety zone setting request signal	SAFRQ	G232#5	7.6
Multi–piece machining function	Multi–piece machining setting signal	MLP1,MLP2	G231#0,#1	4.2.3
Multi–piece machining retrace function	Multi–piece machining skip signal	MGSKP	G248	9.5
Turret axis control (T axis control)	T command neglect signal	TNG	G233#5	5.3.1
	Tool change signal	TCNG	G233#6	5.3.2
	Turret indexing completion signal	TIE	F236#6	5.3.3
	T code display signal	TI00–TI31	G234–G237	5.3.4
	Number of punches signal	PN00–PN31	F234–F237	5.3.5
	T–axis machine zero point position signals	RP1T–RP16T	F244–F245	5.3.6
T–code pre–issue function	Pre–issued T code complete signal	BTFIN	G248#1	5.5
	Pre–issued T code strobe signal	BTF	F232#1	5.5
	Pre–issued T code signal	BT00–BT31	F246–F249	5.5

Function	Name	Symbol	Address	Item
T–axis, C–axis jog override	T–axis, C–axis jog override signal	*JVT1*JVT2	G233#0,#1	2.6.4
Switching the punching and laser modes	Laser mode set signal	LASMOD	F239#6	9.3
Press function	Press stop signal	*PE	X1004#7	3.5.1
	Punch finish signal for 1–cycle press	*PFIN	X1004#5	3.5.1
	Press start signal	PF	Y1004#2	3.5.1
	Nibbling finish signal	*NFIN	X1004#6	3.5.2
	Nibbling signal	NBL	Y1004#1	3.5.2
	1–cycle press select signal	CPS	G230#2	3.5.2
	Nibbling completion signal	NBLE	F230#7	3.5.2
	Continuous manual press signal	MNS	G230#4	3.5.3
	Manual press start signal	MPS	G230#3	3.5.3
	Press start lock signal	PFL	G230#0	3.5.4
	Press start assistance signal	DPF	F230#6	3.5.5
	Press start waiting signal	PFW	G230#1	3.5.6
	Press start waiting signal B	PFWB	X1004#4	3.5.6
	Press start signal B	PFB	Y1004#3	3.5.7
	Press stop signal neglect	EPE	G230#5	3.5.8
	Two–step selection for nibbling signal	SNP	G230#6	3.5.9
	Press start auxiliary signal B	DSPF	F230#5	3.5.10
Multiple–tool control	Multiple tool indexing signal	MIE	F232#7	5.4
	Multiple tool indexing complete signal	CMOK	F232#5	5.4
Reset key signal	Reset key signal	RSTSW	F239#5	9.2
Workpiece holder interference avoidance function	Tool escape completion signal	WHALC	G232#7	8.1
	Tool escape signal	WHAL	F231#5	8.1
	Avoid operation signal	ACZEXE	G232#6	8.2

A.2.2 List of Signals in the Order of Symbols

Group	Symbol	Name	Address	Reference item
*	*JVT1*JVT2	T–axis, C–axis jog override signal	G233#0,#1	2.6.4
	*NFIN	Nibbling finish signal	X1004#6	3.5.2
	*PE	Press stop signal	X1004#7	3.5.1
	*PFIN	Punch finish signal for 1–cycle press	X1004#5	3.5.1
	*SPR	Feed hold signal B	X1008#5	9.1
A	ACZEXE	Avoid operation signal	G232#6	8.2
B	BT00–BT31	Pre–issued T code signal	F246–F249	5.5
	BTF	Pre–issued T code strobe signal	F232#1	5.5
	BTFIN	Pre–issued T code complete signal	G248#1	5.5
C	CFDC	C–axis unconnected signal	G233#7	6.3.2
	CMOK	Multiple tool indexing complete signal	F232#5	5.4
	CPEN	Coupling enable signal	F232#4	6.3.2
	CPS	1–cycle press select signal	G230#2	3.5.2
D	DPF	Press start assistance signal	F230#6	3.5.5
	DSPF	Press start auxiliary signal B	F230#5	3.5.10
E	EFS	External operation function select signal	G230#7	3.7
	EPE	Press stop signal neglect	G230#5	3.5.8
L	LASMOD	Laser mode set signal	F239#6	9.3
M	MIE	Multiple tool indexing signal	F232#7	5.4
	MLP1,MLP2	Multi–piece machining setting signal	G231#0,#1	4.2.3
	MNS	Continuous manual press signal	G230#4	3.5.3
	MPS	Manual press start signal	G230#3	3.5.3
	MGSKP	Multi–piece machining skip signal	G248#3	9.5
N	NBL	Nibbling signal	Y1004#1	3.5.2
	NBLE	Nibbling completion signal	F230#7	3.5.2

Group	Symbol	Name	Address	Reference item
P	PF	Press start signal	Y1004#2	3.5.1
	PFB	Press start signal B	Y1004#3	3.5.7
	PFL	Press start lock signal	G230#0	3.5.4
	PFW	Press start waiting signal	G230#1	3.5.6
	PFWB	Press start waiting signal B	X1004#4	3.5.6
	PN00–PN31	Number of punches signal	F234–F237	5.3.5
	PTLCH	Tool expired signal	F232#0	2.9.2
R	RP1T–RP16T	T–axis machine zero point position signals	F244–F245	5.3.6
	RSTSW	Reset key signal	F239#5	9.2
	RVSARV	Multi–piece machining skip ready signal	F225#0	9.5
	RVSSKE	Multi–piece machining skip completion signal	F225#2	9.5
S	SAFEXE	Safety zone setting ready signal	F231#6	7.6
	SAFRQ	Automatic safety zone setting request signal	G232#5	7.6
	SAFZ	Workpiece holder position detector signal	X1004#1	7.5
	SC1,SC2	C–axis control status signal	F233#0,#1	6.2
	SKIP	Skip signal	X1004#0	2.11.1
	SNP	Two–step selection for nibbling signal	G230#6	3.6.9
	SPRL	Feed hold lamp signal B	F230#4	9.1
	SZT1–SZT12	In safety zone signal	F240–F241#0–#3	7.6
	SZTS0–SZTS3	Selection signal for tool area of safety zone	G232#0–#3	7.6
T	TCNG	Tool change signal	G233#6	5.3.2
	TI00–TI31	T code display signal	G234–G237	5.3.4
	TIE	Turret indexing completion signal	F232#6	5.3.3
	TNG	T command neglect signal	G233#5	5.3.1
W	WDC	Punch enable signal	G232#4	7.6
	WDDO	Punch forbidden area entered signal	F231#7	7.6
	WHAL	Tool escape signal	F231#5	8.1
	WHALC	Tool escape completion signal	G232#7	8.1

A.2.3 List of Signals in the Order of Addresses

Address	Name	Symbol	Reference item
F225#0	Multi–piece machining skip ready signal	RVSARV	9.5
F225#2	Multi–piece machining skip completion signal	RVSSKE	9.5
F230#4	Feed hold lamp signal B	SPRL	9.1
F230#5	Press start auxiliary signal B	DSPF	3.5.10
F230#6	Press start assistance signal	DPF	3.5.5
F230#7	Nibbling completion signal	NBLE	3.5.2
F231#5	Tool escape signal	WHAL	8.1
F231#6	Safety zone setting ready signal	SAFEXE	7.6
F231#7	Punch forbidden area entered signal	WDDO	7.6
F232#0	Tool expired signal	PTLCH	2.9.2
F232#1	Pre–issued T code strobe signal	BTF	5.5
F232#4	Coupling enable signal	CPEN	6.3.2
F232#5	Multiple tool indexing complete signal	CMOK	5.4
F236#6	Turret indexing completion signal	TIE	5.3.3
F232#7	Multiple tool indexing signal	MIE	5.4
F233#0,#1	C–axis control status signal	SC1,SC2	6.2
F234–F237	Number of punches signal	PN00–PN31	5.3.5
F239#5	Reset key signal	RSTSW	9.2
F239#6	Laser mode set signal	LASMOD	9.3
F240–F241#0–#3	In safety zone signal	SZT1–SZT12	7.6
F244–F245	T–axis machine zero point position signals	RP1T–RP16T	5.3.6
F246–F249	Pre–issued T code signal	BT00–BT31	5.5
G230#0	Press start lock signal	PFL	3.5.4
G230#1	Press start waiting signal	PFW	3.5.6
G230#2	1–cycle press select signal	CPS	3.5.2
G230#3	Manual press start signal	MPS	3.5.3
G230#4	Continuous manual press signal	MNS	3.5.3
G230#5	Press stop signal neglect	EPE	3.5.8
G230#6	Two–step selection for nibbling signal	SNP	3.5.9
G230#7	External operation function select signal	EFS	3.7

Address	Name	Symbol	Reference item
G231#0,#1	Multi–piece machining setting signal	MLP1,MLP2	4.2.3
G232#0–#3	Selection signal for tool area of safety zone	SZTS0–SZTS3	7.6
G232#4	Punch enable signal	WDC	7.6
G232#5	Automatic safety zone setting request signal	SAFRQ	7.6
G232#6	Avoid operation signal	ACZEXE	8.2
G232#7	Tool escape completion signal	WHALC	8.1
G233#0,#1	T–axis, C–axis jog override signal	*JVT1*JVT2	2.6.4
G233#5	T command neglect signal	TNG	5.3.1
G233#6	Tool change signal	TCNG	5.3.2
G233#7	C–axis unconnected signal	CFDC	6.3.2
G234–G237	T code display signal	TI00–TI31	5.3.4
G248#1	Pre–issued T code complete signal	BTFIN	5.5
G248#3	Multi–piece machining skip signal	MGSKP	9.5
X1004#0	Skip signal	SKIP	2.11.1
X1004#1	Workpiece holder position detector signal	SAFZ	7.5
X1004#4	Press start waiting signal B	PFWB	3.5.6
X1004#5	Punch finish signal for 1–cycle press	*PFIN	3.5.1
X1004#6	Nibbling finish signal	*NFIN	3.5.2
X1004#7	Press stop signal	*PE	3.5.1
X1008#5	Feed hold signal B	*SPR	9.1
Y1004#1	Nibbling signal	NBL	3.5.2
Y1004#2	Press start signal	PF	3.5.1
Y1004#3	Press start signal B	PFB	3.5.7

B ALARM LIST

1) Program errors (P/S alarm)

Number	Message	Contents
4500	REPOSITIONING INHIBITED	A repositioning command was specified in the circular interpolation (G02, G03) mode.
4502	ILLEGAL COMMAND IN BOLT HOLE	In a bolt hole circle (G26) command, the radius (I) was set to zero or a negative value, or the number of holes (K) was set to zero. Alternatively, I, J, or K was not specified.
4503	ILLEGAL COMMAND IN LINE AT ANGLE	In a line-at-angle (G76) command, the number of holes (K) was set to zero or a negative value. Alternatively, I, J, or K was not specified.
4504	ILLEGAL COMMAND IN ARC	In an arc (G77) command, the radius (I) or the number of holes (K) was set to zero or a negative value. Alternatively, I, J, K, or P was not specified.
4505	ILLEGAL COMMAND IN GRID	In a grid (G78, G79) command, the number of holes (P, K) was set to zero or a negative value. Alternatively, I, J, K, or P was not specified.
4506	ILLEGAL COMMAND IN SHARE PROOFS	In a shear proof (G86) command, the tool size (P) was set to zero, or the blanking length (I) was 1.5 times larger than the tool size (P) or less. Alternatively, I, J, or P was not specified.
4507	ILLEGAL COMMAND IN SQUARE	In a square (G87) command, the tool size (P,Q) was set to zero or a negative value, or the blanking length (I, J) was three times larger than the tool size (P, Q) or less. Alternatively, I, J, P, or Q was not specified.
4508	ILLEGAL COMMAND IN RADIUS	In a radius (G88) command, the traveling pitch (Q) or radius (I) was set to zero or a negative value, or the traveling pitch (Q) was greater than or equal to the arc length. Alternatively, I, J, K, P, or Q was not specified.
4509	ILLEGAL COMMAND IN CUT AT ANGLE	In a cut-at-angle (G89) command, the traveling pitch (Q) was set to zero, negative value, or another value larger than or equal to the length (I). Alternatively, I, J, P, or Q was not specified.
4510	ILLEGAL COMMAND IN LINE-PUNCH	In a linear punching (G45) command, the traveling distance was set to zero or a value 1.5 times larger than the tool size (P) or less. Alternatively, P was not specified.
4511	ILLEGAL COMMAND IN CIRCLE-PUNCH	In a circular punching (G46, G47) command, the same position was specified for both start and end points of the arc, radius (R) of the arc was set to zero, or the pitch (Q) was set to a value exceeding the arc length. Alternatively, R or Q was not specified.
4520	T, M INHIBITED IN NIBBLING-MODE	T code, M code, G04, G70 or G75 was specified in the nibbling mode.
4521	EXCESS NIBBLING MOVEMENT (X, Y)	In the nibbling mode, the X-axis or Y-axis traveling distance was larger than or equal to the limit (No. 16188 to 16193).
4522	EXCESS NIBBLING MOVEMENT (C)	In the circular nibbling (G68) or usual nibbling mode, the C-axis traveling distance was larger than or equal to the limit (No. 16194).

Number	Message	Contents
4523	ILLEGAL COMMAND IN CIRCLE-NIBBL	In a circular nibbling (G68) command, the traveling pitch (Q) was set to zero, a negative value, or a value larger than or equal to the limit (No. 16186, 16187), or the radius (I) was set to zero or a negative value. Alternatively, I, J, K, P, or Q was not specified.
4524	ILLEGAL COMMAND IN LINE-NIBBL	In a linear nibbling (G69) command, the traveling pitch (Q) was set to zero, negative value, or a value larger than or equal to the limit (No. 16186, 16187). Alternatively, I, J, P, or Q was not specified.
4530	A/B MACRO NUMBER ERROR	The number for storing and calling by an A or B macro was set to a value beyond the range from 1 to 5.
4531	U/V MACRO FORMAT ERROR	An attempt was made to store a macro while storing another macro using a U or V macro. A V macro was specified although the processing to store a macro was not in progress. A U macro number and V macro number do not correspond with each other.
4532	IMPROPER U/V MACRO NUMBER	The number of an inhibited macro (number beyond the range from 01 to 99) was specified in a U or V macro command.
4533	U/V MACRO MEMORY OVERFLOW	An attempt was made to store too many macros with a U or V macro command.
4534	W MACRO NUMBER NOT FOUND	Macro number W specified in a U or V macro command is not stored.
4535	U/V MACRO NESTING ERROR	An attempt was made to call a macro which is defined three times or more using a U or V macro command. An attempt was made to store 15 or more macros in the storage area for macros of number 90 to 99.
4536	NO W, Q COMMAND IN MULTI-PIECE	W or Q was not specified in the command for taking multiple workpieces (G73, G74).
4537	ILLEGAL Q VALUE IN MULTI-PIECE	In the command for taking multiple workpieces (G73, G74), Q is set to a value beyond the range from 1 to 4.
4538	W NO. NOT FOUND IN MULTI-PIECE	Macro number W specified in the command for taking multiple workpieces (G73, G74) is not stored.
4539	MULTI-PIECE SETTING IS ZERO	The command for taking multiple workpieces (G73, G74) was specified although zero is specified for the function to take multiple workpieces (No. 16206 or signals MLP1 and MLP2 (PMC address G231, #0 and #1)).
4540	MULTI-PIECE COMMAND WITHIN MACRO	The command for taking multiple workpieces (G73, G74) was specified when a U or V macro was being stored.
4542	MULTI-PIECE COMMAND ERROR	Although G98P0 was specified, the G73 command was issued. Although G98K0 was specified, the G74 command was issued.
4543	MULTI-PIECE Q COMMAND ERROR	Although G98P0 was specified, the Q value for the G74 command was not 1 or 3. Although G98K0 was specified, the Q value for the G73 command was not 1 or 2.
4544	MULTI-PIECE RESTART ERROR	In the command for resuming taking multiple workpieces, the resume position (P) is set to a value beyond the range from 1 to total number of workpieces to be machined.

Number	Message	Contents
4549	ILLEGAL TOOL DATA FORMAT	The size of the registered tool data patterns has exceeded the upper limit of 16KB.
4600	T, C COMMAND IN INTERPOLATION	In the linear interpolation (G01) mode or circular interpolation (G02, G03) mode, a T command or C-axis command was specified.
4601	INHIBITED T, M COMMAND	In the block of G52, G72, G73, or G74, a T or M command was specified.
4602	ILLEGAL T-CODE	The specified T command is not cataloged on the tool register screen.
4603	C AXIS SYNCHRONOUS ERROR	The difference between the position deviation value of C1 axis and C2 axis exceeds the parameter value (No. 16364, 16365) with the C–axis synchronous control function.
4604	ILLEGAL AXIS OPERATION	A C-axis command was specified in the block containing a T command for multiple tools.
4605	NEED ZRN	The C–axis synchronization is not done normally.
4630	ILLEGAL COMMAND IN LASER MODE	In the laser mode, a nibbling command or pattern command was specified.
4650	IMPROPER G-CODE IN OFFSET MODE	In the cutter compensation mode, an inhibited G code (pattern command, G73, G74, G75, etc.) was specified.
4700	PROGRAM ERROR (OT +)	The value specified in the X-axis move command exceeded the positive value of stored stroke limit 1. (Advance check)
4701	PROGRAM ERROR (OT –)	The value specified in the X-axis move command exceeded the negative value of stored stroke limit 1. (Advance check)
4702	PROGRAM ERROR (OT +)	The value specified in the Y-axis move command exceeded the positive value of stored stroke limit 1. (Advance check)
4703	PROGRAM ERROR (OT –)	The value specified in the Y-axis move command exceeded the negative value of stored stroke limit 1. (Advance check)
4704	PROGRAM ERROR (OT +)	The value specified in the Z-axis move command exceeded the positive value of stored stroke limit 1. (Advance check)
4705	PROGRAM ERROR (OT –)	The value specified in the Z-axis move command exceeded the negative value of stored stroke limit 1. (Advance check)

2) Safety zone alarms

Number	Message	Contents
4800	ZONE : PUNCHING INHIBITED 1	When a safety zone check was executed, a punch command was specified in area 1 where punching is inhibited.
4801	ZONE : PUNCHING INHIBITED 2	When a safety zone check was executed, a punch command was specified in area 2 where punching is inhibited.
4802	ZONE : PUNCHING INHIBITED 3	When a safety zone check was executed, a punch command was specified in area 3 where punching is inhibited.
4803	ZONE : PUNCHING INHIBITED 4	When a safety zone check was executed, a punch command was specified in area 4 where punching is inhibited.
4810	ZONE : ENTERING INHIBITED 1 +X	When a safety zone check was executed, the machine moving in the positive X direction entered area 1 into which entry is inhibited.
4811	ZONE : ENTERING INHIBITED 1 –X	When a safety zone check was executed, the machine moving in the negative X direction entered area 1 into which entry is inhibited.
4812	ZONE : ENTERING INHIBITED 2 +X	When a safety zone check was executed, the machine moving in the positive X direction entered area 2 into which entry is inhibited.
4813	ZONE : ENTERING INHIBITED 2 –X	When a safety zone check was executed, the machine moving in the negative X direction entered area 2 into which entry is inhibited.
4814	ZONE : ENTERING INHIBITED 3 +X	When a safety zone check was executed, the machine moving in the positive X direction entered area 3 into which entry is inhibited.
4815	ZONE : ENTERING INHIBITED 3 –X	When a safety zone check was executed, the machine moving in the negative X direction entered area 3 into which entry is inhibited.
4816	ZONE : ENTERING INHIBITED 4 +X	When a safety zone check was executed, the machine moving in the positive X direction entered area 4 into which entry is inhibited.
4817	ZONE : ENTERING INHIBITED 4 –X	When a safety zone check was executed, the machine moving in the negative X direction entered area 4 into which entry is inhibited.
4830	ZONE : ENTERING INHIBITED 1 +Y	When a safety zone check was executed, the machine moving in the positive Y direction entered area 1 into which entry is inhibited.
4831	ZONE : ENTERING INHIBITED 1 –Y	When a safety zone check was executed, the machine moving in the negative Y direction entered area 1 into which entry is inhibited.
4832	ZONE : ENTERING INHIBITED 2 +Y	When a safety zone check was executed, the machine moving in the positive Y direction entered area 2 into which entry is inhibited.
4833	ZONE : ENTERING INHIBITED 2 –Y	When a safety zone check was executed, the machine moving in the negative Y direction entered area 2 into which entry is inhibited.
4834	ZONE : ENTERING INHIBITED 3 +Y	When a safety zone check was executed, the machine moving in the positive Y direction entered area 3 into which entry is inhibited.
4835	ZONE : ENTERING INHIBITED 3 –Y	When a safety zone check was executed, the machine moving in the negative Y direction entered area 3 into which entry is inhibited.
4836	ZONE : ENTERING INHIBITED 4 +Y	When a safety zone check was executed, the machine moving in the positive Y direction entered area 4 into which entry is inhibited.
4837	ZONE : ENTERING INHIBITED 4 –Y	When a safety zone check was executed, the machine moving in the negative Y direction entered area 4 into which entry is inhibited.

Number	Message	Contents
4870	AUTO SETTING FEED ERROR	The feed rate of safety zone auto setting is other than the parameter value (No. 16538, No. 16539).
4871	AUTO SETTING PIECES ERROR	In safety zone auto setting, the safety zone pieces are not correct. Or the position detector has gone wrong, please tell your machine tool builder.
4872	AUTO SETTING COMMAND ERROR	M code, S code or T code is specified with safety zone auto setting command (G32). G32 is specified in the nibbling mode, in the cutter compensation, in the rotation mode or the scaling mode.

Index

□Numbers□

1–cycle press, 50
2ms interval acceleration/deceleration for rapid traverse, 159
2nd auxiliary function, 31
2nd to 4th reference position return, 14

□A□

Address list, 168
Alarm list, 177
Automatic operation, 16
Automatic repositioning (G75), 66
Automatic safety–zone setting, 126
Auxiliary function, 31
Auxiliary function lock, 31
Axis control, 10

□B□

Base point command of multi–piece machining (G98), 68
Block in which punching is made, 42

□C□

C axis control (die angle indexing), 106
C–axis control, 105
C–axis offset function, 112
C–axis offset type A, 112
C–axis offset type B, 114
C–axis synchronization control, 107
Changing the rapid traverse rate, time constant and servo loop gain according to the positioning distance constant positioning time control, 19
CNC and PMC interface, 167
Continuous press (nibbling), 52
Control function, 154

□D□

Detecting workpiece holder position using an external signal, 128
Display/set/edit, 35
Displaying the safety zones and tool zone, 132
Distribution end signal, 31

□E□

External operation function EF, EFS, FIN, 64

□F□

Feed hold, 16
Feed hold signal B/feed hold lamp signal B, 155
Feedrate control/acceleration and deceleration control, 19
Follow–up, 11
Function specifications that differ from the M series, 9
Functions to simplify programming, 65

□G□

Graphic display, 35

□H□

High speed press control function, 61

□I□

Increment system, 11
Interpolation function, 18

□L□

Linear interpolation/circular interpolation, 18
List of functions, 1
List of signals, 171
List of signals in the order of addresses, 175
List of signals in the order of functions, 171
List of signals in the order of symbols, 173
Look–ahead control, 30

□M□

Machine lock, 16
Manual press, 56
Manual reference position return, 14
Measurement, 36
Mirror image, 11
Multi–language display, 35
Multi–piece machining commands (G73, G74), 69
Multi–piece machining function, 68
Multiple–tool control (output) CMOK <F232#5>, MIE <F232#7>, 87
Multiple–workpiece machining retrace function, 160

□N□

Name of axes, 10
Nibbling by M function, 48

Nibbling function, 45
Number of punches signal (output) PN00 – PN31 <F234 – F237>, 85

□P□

PMC axis control, 37
PMC control function, 37
Positioning, 18
Positioning & pressing OFF (G70), 44
Preparations for operation, 13
Preparatory function (G function), 38
Press function, 50
Press start assistance signal (output) DPF <F230#6>, 58
Press start auxiliary signal B DSPF<F230#5>, 60
Press start lock signal (input) PFL <G230#0>, 57
Press start signal B (output) PFB <Y1004#3>, 59
Press start waiting signal (input) PFW <G230#1>, 58
Press stop signal neglect (input) EPE <G230#5>, 59
Pressing function, 41
Punch forbidden area and approach forbidden area, 122
Punch function (1–cycle pressing), 42

□R□

Rapid traverse override, 28
Rapid traverse rate, 19
Reference position establishment, 14
Reference position return, 14
Reset key signal, 156
Rotary axis roll over, 12

□S□

Safety zone check, 121
Setting of machining method for multi–piece machining, 69
Setting the reference position without dogs, 14
Setting the safety zone, 124
Setting the tool shape area, 125
Single block, 17
Specifying the rotation axis, 11
Spindle speed function, 32
Stored stroke limit 1, 13
Switching the punching and laser modes, 157

□T□

T code display signal (input) TI00 – TI32 <G234 – G237>, 84
T command neglect signal (input) TNG <G233#5>, 82
T–axis , C–axis jog override signal, 29
T–axis machine zero point position signals RP1T – RP16T <F244, F245>, 85
T–code pre–issue function, 95
Tool change signal (input) TCNG <G233#6>, 82
Tool data setting function, 99
Tool function, 33
Tool function (T function), 74
Tool life management, 33
Tool offset, 77
Tool offset value/tool offset number/tool offset memory, 33
Tool selection function, 75
Tool/workpiece holder areas, 149
Turret axis control (T axis control), 78
Turret indexing completion signal (output) TIE <F236#6>, 83
Two–step selection (input) SNP for nibbling <G230#6>, 60
Type A, 122, 144
Type B, 123, 146

□W□

Waveform diagnosis display, 35
Workpiece holder detection command, 126
Workpiece holder interference avoidance function, 143

□Y□

Y–axis crack cancel, 73