CNC SYSTEMS OSP-U100M OSP-U10M

SPECIAL FUNCTION MANUAL (No. 2)

(2nd Edition)

Pub. No. 4188-E-R1 (ME51-193-R2) Nov. 2000



SAFETY PRECAUTIONS

The machine is equipped with safety devices which serve to protect personnel and the machine itself from hazards arising from unforeseen accidents. However, operators must not rely exclusively on these safety devices: they must also become fully familiar with the safety guidelines presented below to ensure accident-free operation.

This instruction manual and the warning signs attached to the machine cover only those hazards which Okuma can predict. Be aware that they do not cover all possible hazards.

1. Precautions Relating to Machine Installation

- (1) Install the machine at a site where the following conditions (the conditions for achievement of the guaranteed accuracy) apply.
 - Ambient temperature: 17 to 25°C (62.6 to 77°F)
 - Ambient humidity: 40% to 75% at 20°C (68°F) (no condensation)
 - Site not subject to direct sunlight or excessive vibration; environment as free of dust, acid, corrosive gases, and salt spray as possible.
- (2) Prepare a primary power supply that complies with the following requirements.
 - Voltage: 200 V
 - Voltage fluctuation: \pm 10% max.
 - Power supply frequency: 50/60 Hz
 - Do not draw the primary power supply from a distribution panel that also supplies a major noise source (for example an electric welder or electric discharge machine) since this could cause malfunction of the CNC unit.
 - If possible connect the machine to a ground not used by any other equipment. If there is no choice but to use a common ground, the other equipment must not generate a large amount of noise (such as an electric welder or electric discharge machine).
- (3) Installation Environment

Observe the following points when installing the control enclosure.

- Make sure that the CNC unit will not be subject to direct sunlight.
- Make sure that the control enclosure will not be splashed with chips, water, or oil.
- Make sure that the control enclosure and operation panel are not subject to excessive vibrations or shock.
- The permissible ambient temperature range for the control enclosure is 0 to 40°C (32 to 104°F).
- The permissible ambient humidity range for the control enclosure is 30 to 95% (no condensation).
- The maximum altitude at which the control enclosure can be used is 1000 m (3281 ft.).

2. Points to Check before Turning on the Power

- (1) Close all the doors of the control enclosure and operation panel to prevent the entry of water, chips, and dust.
- (2) Make absolutely sure that there is nobody near the moving parts of the machine, and that there are no obstacles around the machine, before starting machine operation.
- (3) When turning on the power, turn on the main power disconnect switch first, then the CONTROL ON switch on the operation panel.

3. Precautions Relating to Operation

- (1) After turning on the power, carry out inspection and adjustment in accordance with the daily inspection procedure described in this instruction manual.
- (2) Use tools whose dimensions and type are appropriate for the work undertaken and the machine specifications. Do not use badly worn tools since they can cause accidents.
- (3) Do not for any reason touch the spindle or tool while spindle indexing is in progress since the spindle could rotate: this is dangerous.
- (4) Check that the workpiece and tool are properly secured.
- (5) Never touch a workpiece or tool while it is rotating: this is extremely dangerous.
- (6) Do not remove chips by hand while machining is in progress since this is dangerous. Always stop the machine first, then remove the chips with a brush or broom.
- (7) Do not operate the machine with any of the safety devices removed. Do not operate the machine with any of the covers removed unless it is necessary to do so.
- (8) Always stop the machine before mounting or removing a tool.
- (9) Do not approach or touch any moving part of the machine while it is operating.
- (10) Do not touch any switch or button with wet hands. This is extremely dangerous.
- (11) Before using any switch or button on the operation panel, check that it is the one intended.

4. Precautions Relating to the ATC

- (1) The tool clamps of the magazine, spindle, etc., are designed for reliability, but it is possible that a tool could be released and fall in the event of an unforeseen accident, exposing you to danger: do not touch or approach the ATC mechanism during ATC operation.
- (2) Always inspect and change tools in the magazine in the manual magazine interrupt mode.
- (3) Remove chips adhering to the magazine at appropriate intervals since they can cause misoperation.

Do not use compressed air to remove these chips since it will only push the chips further in.

(4) If the ATC stops during operation for some reason and it has to be inspected without turning the power off, do not touch the ATC since it may start moving suddenly.

5. On Finishing Work

- (1) On finishing work, clean the vicinity of the machine.
- (2) Return the ATC, APC and other equipment to the predetermined retraction position.
- (3) Always turn off the power to the machine before leaving it.
- (4) To turn off the power, turn off the CONTROL ON switch on the operation panel first, then the main power disconnect switch.

6. Precautions during Maintenance Inspection and When Trouble Occurs

In order to prevent unforeseen accidents, damage to the machine, etc., it is essential to observe the following points when performing maintenance inspections or during checking when trouble has occurred.

- (1) When trouble occurs, press the emergency stop button on the operation panel to stop the machine.
- (2) Consult the person responsible for maintenance to determine what corrective measures need to be taken.
- (3) If two or more persons must work together, establish signals so that they can communicate to confirm safety before proceeding to each new step.
- (4) Use only the specified replacement parts and fuses.
- (5) Always turn the power off before starting inspection or changing parts.
- (6) When parts are removed during inspection or repair work, always replace them as they were and secure them properly with their screws, etc.
- (7) When carrying out inspections in which measuring instruments are used for example voltage checks make sure the instrument is properly calibrated.
- (8) Do not keep combustible materials or metals inside the control enclosure or terminal box.
- (9) Check that cables and wires are free of damage: damaged cables and wires will cause current leakage and electric shocks.
- (10) Maintenance inside the Control Enclosure
 - a) Switch the main power disconnect switch OFF before opening the control enclosure door.
 - b) Even when the main power disconnect switch is OFF, there may some residual charge in the servo amplifier and spindle drive unit, and for this reason only service personnel are permitted to perform any work on these units. Even then, they must observe the following precautions.
 - Servo amplifier

Discharge the residual voltage one minute after turning off the breaker inside the unit.

- Spindle drive unit

Discharge the residual voltage one minute after turning off the main power disconnect switch.

c) The control enclosure contains the NC unit, and the NC unit has a printed circuit board whose memory stores the machining programs, parameters, etc. In order to ensure that the contents of this memory will be retained even when the power is switched off, the memory is supplied with power by a battery. Depending on how the printed circuit boards are handled, the contents of the memory may be destroyed and for this reason only service personnel should handle these boards.

- (11) Periodic Inspection of the Control Enclosure
 - a) Cleaning the cooling unit

The cooling unit in the door of the control enclosure serves to prevent excessive temperature rise inside the control enclosure and increase the reliability of the NC unit. Inspect the following points every three months.

- Is the fan motor inside the cooling unit working?

The motor is normal if there is a strong draft from the unit.

- Is the external air inlet blocked?

If it is blocked, clean it with compressed air.

7. General Precautions

- (1) Keep the vicinity of the machine clean and tidy.
- (2) Wear appropriate clothing while working, and follow the instructions of someone with sufficient training.
- (3) Make sure that your clothes and hair cannot become entangled in the machine. Machine operators must wear safety equipment such as safety shoes and safety goggles.
- (4) Machine operators must read the instruction manual carefully and make sure of the correct procedure before operating the machine.
- (5) Memorize the position of the emergency stop button so that you can press it immediately at any time and from any position.
- (6) Do not access the inside of the control panel, transformer, motor, etc., since they contain high-voltage terminals and other components which are extremely dangerous.
- (7) If two or more persons must work together, establish signals so that they can communicate to confirm safety before proceeding to each new step.

8. Symbols Used in This Manual

The following warning indications are used in this manual to draw attention to information of particular importance. Read the instructions marked with these symbols carefully and follow them.



INTRODUCTION

This special function manual contains instructions for the OSP. A careful reading of the manual will be of great assistance in obtaining the full benefit of all the superior functions the machine has to offer. For the most complete understanding, this manual should be read in conjunction with the "Operation Manual for OSP", as the two manuals are very closely related.

CLASSIFICATION

- A. MID-BLOCK RESTART FUNCTION
- B. DNC-A
- C. DNC-B HIGH-SPEED RM BUFFER METHOD
- D. SEMI-SYNCHRONIZED OPERATION OF MACHINING CENTER WITH DIGITIZER (TD)
- E. WARM-UP FUNCTION
- F. SPINDLE OVERLOAD DETECTION FUNCTION
- G. SIMPLIFIED LOAD MONITORING FUNCTION
- H. EXTERNAL PROGRAM SELECTION FUNCTION
- I. EXTERNAL M SIGNAL OUTPUT FUNCTION
- J. CYCLE TIME REDUCTION FUNCTION

K. F1-DIGIT FEED COMMAND FUNCTION (BY PARAMETERS)

- L. ARBITRARY-ANGLE CHAMFERING FUNCTION
- M. 3-D CIRCULAR INTERPOLATION FUNCTION
- N. AUTOMATIC ATTACHMENT INDEXING FUNCTION
- O. AXIS NAME DESIGNATION FUNCTION
- P. SLOPE MACHINING FUNCTION
- Q. MULTIPLE-POINT SPINDLE ORIENTATION FUNCTION
- R. INDEX TABLE EXTERNAL MANUAL OPERATION FUNCTION
- S. AUTOMATIC SCHEDULE PROGRAM UPDATE FUNCTION
- T. ADDITIONAL AXIS (ROTARY AXIS) FUNCTION

U. CYLINDER SIDE-SURFACE MACHINING FUNCTION

V. 2-PALLET APC PALLET IDENTIFICATION FUNCTION

W. TOOL LIST FILE FUNCTION

X. SERVO FILE SELECTION FUNCTION (SELECTION BY WORKPIECE WEIGHT SETTING)

Since this operation manual describes the special specifications (optional specifications) of OSP, some portions may not apply to your machine. Refer to your machine's Final Specifications to check the specifications before reading this manual.

TABLE OF CONTENTS

<u>PAGE</u>

A. MID-BLOCK RESTART FUNCTION

SECTION 1 FUNCTION OVERVIEW	A-1
SECTION 2 COMMAND FORMAT FOR SEQUENCE RESTART OPERATION	A-2
1. Restart from Designated Sequence	A-2
2. Restarting from Designated Block	A-3
3. Restart from the Designated Block Preceding the Restart Block	A-4
SECTION 3 OPTION DESIGNATION	A-5
1. Axis Positioning Order Designation	A-5
2. Designation of SequenCe Restart Point - ;M Option	A-6
3. Designation of Sequence Restart Point - ;TS Option	A-7
4. Axis Movements without Option Designation	A-7
SECTION 4 OPERATIONS	A-8
1. Sequence Restart from the Designated Sequence	A-8
2. Sequence Restart from the Position where Cutting Tool is Actually Located	A-9
3. Sequence Restart from a Required Position in the Program	A-11
SECTION 5 SUPPLEMENTS	A-12
B. DNC-A	

SECTION 1	FUNCTIONS	B-1
------------------	-----------	-----

SECTION 2 OPERATING PROCEDURES	B-2
1. Operating Procedure for DNC	B-2
2. Operation Function in the DNC Mode	B-4
SECTION 3 SETTING UP NC OPTIONAL PARAMETERS	B-6
SECTION 4 ERROR MESSAGES	B-8
SECTION 5 SPECIFICATIONS	B-9
1. Data Formats	B-9
1-1. Output Data Format Examples	B-10
1-2. Input Data Format Examples	B-11
2. Example RS232C Interface Connections and Timing Charts	B-12
2-1. Start/Stop Synchronization Mode	B-12
2-2. Control DC Code	B-14
2-3. Control DC Code on Type 2	B-19
C. DNC-B HIGH-SPEED RM BUFFER METHOD	
SECTION 1 OVERVIEW	C-1
SECTION 2 DNC OPERATION PANEL	C-2
SECTION 3 BUFFER OPERATION	C-4
1. Buffer Operation of NC Programs	C-5
1-1. Operation Methods	C-5
1-2. Command Format	C-6
2. Scheduled Operation by Schedule Program	C-7
2-1. Operation Methods	C-7
2-2. Command Format for Schedule Program Selection	C-8

3.	Memory Mode Operation Using a Program Stored in Memory	C-9
4.	Supplements	C-9
SECT	ION 4 BATCH TRANSFER OF NC PROGRAMS	C-11
1.	Operation Procedure	C-11
2.	Command Format	C-13
3.	Supplements	C-16
SECT	ION 5 PARAMETERS	C-17
1.	Parameters which cannot be Set at OSP	C-17
2.	Parameters which can be Set at OSP	C-18
SECT	ION 6 PROTOCOL A	C-22
1.	Communication Format	C-22
2.	Message Format	C-24
3.	Command List	C-25
4.	Data Reception	C-26
5.	Data Transmission	C-27
SECT	ION 7 PROTOCOL B	C-28
1.	Communication Format	C-28
2.	DC (Device Control) Codes	C-28
3.	Data Reception	C-29
4.	Data Transmission	C-31
SECT		C-33
		0-00
1.	Input Format	C-33
2.	Output Format	C-34

SECTION 9 SPECIFICATIONS	C-35
1. RS232C Interface	C-35
SECTION 10 SCREEN DISPLAY	C-39
1. Check Screens	C-39
2. Run Guide Screens	C-45
SECTION 11 ERROR MESSAGES	C-46
SECTION 12 ALARM MESSAGE CHART	C-48
1. Communication Alarm	C-48
2. NC Alarm	C-51
SECTION 13 APPENDIX	C-52
1. DNC HISTORY Contents Code Tables	C-52
D. SEMI-SYNCHRONIZED OPERATION OF MACHINING CENTER WITH DIGITIZED (TD)	R
SECTION 1 OVERVIEW	D-1
SECTION 2 DNC OPERATION PANEL	D-2
SECTION 3 SEMI-SYNCHRONIZED OPERATION	D-4
1. Operation Procedure for Semi-synchronized Operation	D-5
2. Memory Mode Operation Using A Program Stored in Memory	D-5

3.	Supplei	ments	D-6
SECTI	ON 4	PARAMETERS	D-7
1.	Parame	eters which cannot be Set at OSP	D-7
2.	Parame	eters which can be Set at OSP	D-8
SECTI	ON 5	SCREEN DISPLAY	D-12
1.	Check	Screens	D-12
2.	Run Gu	uide Screens	D-17
SECTI	ON 6	ERROR MESSAGES	D-18
SECTI	ON 7	ALARM MESSAGE CHART	D-19
1.	Commu	unication Alarm	D-19
2.	NC Ala	rm	D-22
SECTI	ON 8	APPENDIX	D-23
1.	DNC H	ISTORY Contents Code Tables	D-23
E. WA	RM-UP	PFUNCTION	
SECTI	ON 1	OVERVIEW	E-1
SECTI	ON 2	REGISTRATION OF WARM-UP PROGRAM	E-2
1.	Registe	ering Warm-up Program	E-2
2.	Format		E-2
SECTI	ON 3	OPERATIONS	E-3

SECTION 4	TIMING CHART	E-4
SECTION 5	SUPPLEMENTS	E-5
SECTION 6	ALARM	E-6
SECTION 7	PARAMETER SETTING	E-7
F. SPINDLE	OVERLOAD DETECTION FUNCTION	
SECTION 1	OVERVIEW	F-1
SECTION 2	OPERATIONS	F-2
SECTION 3	SPINDLE DRIVE MOTOR STARTING CURRENT	F-3
SECTION 4	SPINDLE OVERLOAD DETECTION	F-5
SECTION 5	ALARM	F-6
G. SIMPLIFI	ED LOAD MONITORING FUNCTION	
SECTION 1	OUTLINE	G-1
SECTION 2	DETAILS OF FUNCTION	G-2

H. EXTERNAL PROGRAM SELECTION FUNCTION

SECTI	ON 1	EXTERNAL PROGRAM SELECTION A (PUSHBUTTON TYPE)	H -1		
1.	. Main and Schedule Program Selection				
2.	Inhibiting the Selection of the Same Program				
З.	Program	n Selection	H -1		
4.	Supplei	ments	H-1		
SECTI	ON 2	EXTERNAL PROGRAM SELECTION B (ROTARY SWITCH TYPE)	H-2		
1.	Main ar	nd Schedule Program Selection	H-2		
2.	Inhibitir	ig the Selection of the Same Program	H-2		
3.	Program	m Selection	H-2		
4.	Precau	tions	H-2		
SECTI	ON 3	EXTERNAL PROGRAM SELECTION C (BCD TYPE)	H-3		
1.	Overvie	€₩	Н-3		
2.	Program Selection Range				
З.	Main and Schedule Program Selection				
4.	Inhibiting the Selection of the Same Program				
5.	Supple	ments	H-4		
6.	Timing	Chart	H-4		
7.	Parity C	Check	H-5		
7-1	. Pa	arameters	H-5		
8.	Alarms		H-5		
I. EXT	ERNA	L M SIGNAL OUTPUT FUNCTION			

SECTION 1	EXTERNAL M SIGNAL OUTPUT FUNCTION	I-1
1. Over	view	I-1

2.	External M Codes	-1
3.	Timing Chart	1-1

J. CYCLE TIME REDUCTION FUNCTION

SECT	ION 1 CYCLE TIME REDUCTION FUNCTION	J-1
1.	Ignoring Spindle Rotation Answer: M300 (1 Block)	J-1
2.	Ignoring/Confirming Answer Signal Other Than Spindle Rotation Answer: M301 M302 (1 Block)	J-2

K. F1-DIGIT FEED COMMAND FUNCTION (BY PARAMETERS)

SECTIO	N 1 OUTLINE	1
SECTIO	N 2 F1-DIGIT FEED COMMANDS K-	2
SECTIO	N 3 PARAMETER METHOD K-	3
1. C	perating Procedure	3
2. N	linimum and Maximum Values of Setting Data K-	5
3. C	ther Items K-	6
SECTIO	N 4 FEED SWITCH METHOD K-	7
1. C	peration Methods	7
2. S	elected State	7
3. F	1-digit Feed Rotary SwitchesK-	7
SECTIO	N 5 SUPPLEMENTS ĸ-	8

L. ARBITRARY-ANGLE CHAMFERING FUNCTION

SECTION 1	OUTLINE	L-1
SECTION 2	PROGRAMMING FORMAT	L-2
SECTION 3	FUNCTION	L-3
SECTION 4	SUPPLEMENTS	L-5
M. 3-D CIRC	ULAR INTERPOLATION FUNCTION	
SECTION 1	OVERVIEW	M-1
SECTION 2	COMMAND FORMAT	M-2
SECTION 3	3-D ARC ROTATING DIRECTION	M-5
SECTION 4	SHORTER ARC AND LONGER ARC	M-11
SECTION 5	SUPPLEMENTS	M-12
SECTION 6	ALARM CODES	M-13
SECTION 7	PARAMETERS	M-16

N. AUTOMATIC ATTACHMENT INDEXING FUNCTION

SECTION 1 AUTOMATIC ATTACHMENT INDEXING FUNCTION	N -1
1. Overview	N-1
2. Index Commands	N -1
3. Angle Commands	N -1
4. Command Format	N-2
5. Alarm	N-2
O. AXIS NAME DESIGNATION FUNCTION	
SECTION 1 OUTLINE	0-1
SECTION 2 AXIS NAME DESIGNATION	0-2
1. Renaming a Machine Axis	O-2
SECTION 3 APPLICATION RANGE	O-6
SECTION 4 DISPLAY	0-7
SECTION 5 PROGRAM EXAMPLE (FOR MCM MODELS)	O-8
SECTION 6 SUPPLEMENTS	O-10

P. SLOPE MACHINING FUNCTION

SECT	ION 1	OUTLINE	P-1
SECT	ION 2	COORDINATE CONVERSION FUNCTION IN THE AUTOMATIC AND MDI OPERATION MODES	P-2
1.	Setting	of Slope Coordinate System	P-2
2.	G Code	es Usable During Coordinate Conversion	P-4
3.	Mnemo	oniC Codes Usable During Coordinate Conversion	P-8
4.	Upper	Limit Return (M52)	P-8
5.	Axis Co	ommand Cancel	P-8
6.	Pulse H	Handle Overlap	P-8
SECT	ION 3	COORDINATE CONVERSIONS IN THE MANUAL OPERATION MODE	P-11
1.	Setting	Coordinate Conversion Parameters	P-11
2.	Coordii when S	nate Conversion Parameter Settings Switching to the Manual Operation Mode	P-13
3.	Execut	ing Coordinate Conversion	P-14
SECT	ION 4	ACTUAL POSITION DATA DISPLAY	P-15
SECT	ION 5	PARAMETER SETTING	P-17
Q. M	ULTIPL	E-POINT SPINDLE ORIENTATION FUNCTION	
SECT	ION 1	FUNCTION OUTLINE	Q-1
SECT	ION 2	MULTIPLE-POINT SPINDLE ORIENTATION FUNCTION	Q-2

SECTION 3	TOOL BREAKAGE DETECTION AND AUTOMATIC TOOL LENGTH OFFSET FUNCTIONS	Q-3
R. INDEX TA	ABLE EXTERNAL MANUAL OPERATION FUNCTION	
SECTION 1	OVERVIEW	R-1
SECTION 2	OPERATION	R-2
SECTION 3	SUPPLEMENTS	R-3
S. AUTOMA	TIC SCHEDULE PROGRAM UPDATE FUNCTION	
SECTION 1	OVERVIEW	S-1
SECTION 2	RESTRICTIONS TO SCHEDULE PROGRAMS	S-2
SECTION 3	OPERATION	S-3
1. Design	ating Automatic Schedule Program Update	S-3
2. Editing	Schedule Program	S -4
3. Registe	ering Schedule Program to be Updated	S-5
SECTION 4	SCREEN DISPLAY	S-6
SECTION 5	ERROR MESSAGES	S-7

T. ADDITIONAL AXIS (ROTARY AXIS) FUNCTION

SECTION 1 NORMAL OPERATION SPECIFICATION	T-1
1. Outline	T-1
2. Axis Nomenclature	T -1
3. Program Format	T-1
3-1. Least Setting Unit	T -1
3-2. Programmable Range	T -1
3-3. Rotation Direction Command	T -1
3-4. Program EXamples	T-5
3-5. Feedrate in Simultaneous Two-axis Control	T -7
3-6. Calculating Feedrate Command F	T-8
4. Alarm	T-10
SECTION 2 MULTI-TURN SPECIFICATION	T-12
1. Outline	T-12
2. Terminology	T-12
3. Program Format	T-13
3-1. Least Setting Unit	T-13
3-2. Programmable Range	T-13
3-3. Rotation DireCtion Command	T-13
3-4. Feedrate Commands	T-13
4. Operations	T-14
4-1. Automatic and MDI Modes	T-14
4-2. Manual Mode	T-15
4-3. Manual Interruption and Pulse Handle Overlap	T-15
5. Parameter Setting	T-16
5-1. Parameters	T-16
5-2. Rotary Axis with Limit	T-16
6. Actual Position Data Display	T-17
7. Power On/Off and NC Reset	T-18
8. Sequence Restart and Home Position	T-19
8-1. Sequence Restart	T-19

8-2	2. Hor	ne Position	T-20
	8-2-1.	Positioning at Home Position	T-20
	8-2-2.	Incremental Command After Positioning at Home Position	T-20
9.	Work Zei	o Offset	T-2 1
10.	Limit Che	eck	T-22
11.	Installatio	on of Multi-turn NC Rotary Table	T-23
11	-1. Sett	ing System Parameters	T-24
	11-1-1.	Machine Zero Offset	T-24
	11-1-2.	Home Position Data	T-24
12.	Alarm Lis	st	T- 2 4

U. CYLINDER SIDE-SURFACE MACHINING FUNCTION

SECTION 1	OUTLINE	U-1
SECTION 2	PROGRAMMING FORMAT	U-2
1. Cylinde	er Side-Surface Machining Mode	U-2
2. Corres	ponding Basic Axis	U-4
3. Machin	ing Commands	U-5
SECTION 3	ANIMATION FUNCTION	U-14
SECTION 4	ALARM CODES	U-16
V. 2-PALLET	APC PALLET IDENTIFICATION FUNCTION	
SECTION 1	OUTLINE	V-1
SECTION 2	SYSTEM VARIABLES FOR PALLET IDENTIFICATION	V-2
SECTION 3	PALLET IDENTIFICATION COMMAND	V-3

1.	Programming Format	V-3
2.	Application Example (Program Example)	V-3
3.	Alarms	V-3

W. TOOL LIST FILE FUNCTION

SECTION 1 AUTOMATIC SELECTION OF ANIMATION DATA FILE AND TOOL LIST FILE	W-1
1. Overview of Animation Data File and Tool List File	W-2
1-1. Animation Data File	W-2
1-2. Tool List File	W-3
2. Automatic Selection Operation	W-5
3. Relationship between Program Selection Method and Operation Method	W-8
SECTION 2 TOOL LIST DISPLAY FUNCTION	W-9
1. Selecting the Tool List Display Function	W-10
2. Selecting the Tool List File	W-13
2-1. Automatic Tool List File Selection	W-13
2-2. Manual Tool List File Selection	W-13
3. Selection of Tool List Display Screens and Display Contents	W-17
3-1. Tool List Display	W-17
3-2. Setup Required Tool List	W-22
3-3. Unnecessary Tool List	W-27
SECTION 3 PARAMETERS	W-31
SECTION 4 ALARM AND ERROR	W-32

X. SERVO FILE SELECTION FUNCTION (SELECTION BY WORKPIECE WEIGHT SETTING)

SECT	ION 1	OVERVIEW	X-1
SECT	ION 2	SETTING THE WORKPIECE WEIGHT (TABLE LOAD)	X-2
1.	Operati	on	X-2
2.	Parame	eter Setting Display	X-3
3.	Parame	eter Setting Range	X-3

A. MID-BLOCK RESTART FUNCTION

SECTION 1 FUNCTION OVERVIEW

The mid-block restart function covered by this section is closely related with the sequence restart function provided as a standard function of the OSP. Therefore, basic knowledge of the sequence restart function is necessary to use this mid-block function. Before reading this section, it is recommended to read "Return Search and Sequence Restart" in Operation Manual (Publication No. 4106-E)-Operation-thoroughly.

If programmed cutting operation is interrupted halfway, it is possible to restart the operation from the beginning of the interrupted block with the standard sequence restart function. The mid-block sequence restart function allows the operation to restart from the point where the operation has been interrupted. In addition, the operator can designate the axis to return to the sequence restart point first using the option designation function.



The mid-block restart function allows the interrupted operation to restart from the interrupted point in the following sequence.

- (1) Operation is interrupted halfway during the execution of a block of commands.
- (2) Retract the cutting tool to a desired point.
- (3) Manually bring the cutting tool to a point close to the restart point.
- (4) After searching the block from which the sequence is to restart, press the CYCLE START button. The commands are internally processed* up to the restart point.
- (5) Press the SEQ. RESTART button. The cutting tool is positioned at the restart point in a manual cutting feedrate.
- (6) Press the CYCLE START button, and the interrupted operation is resumed from the restart point.
 - * In this processing, the commands are processed without actual machine operations like in the machine lock state operations.

SECTION 2 COMMAND FORMAT FOR SEQUENCE RESTART OPERATION

This section explains the command formats used to designate sequence restart operation.

1. Restart from Designated Sequence

= RS --- sequence-name [,number-of-repetitions] [;option-designation] [WRITE]

- [Supplement] 1. Designation in [] can be omitted.
 - 2. The following option designation is possible. Details of option designation is explained in SECTION 2.
 - ; ± * Axis movement priority
 - ;M* Restart point designation option
 - ;TS Restart point designation option

Example 1:

= RS - N100 [WRITE]

Sequence restart from sequence name N100

Example 2:

= RS - N100,3 [WRITE]

When the same sequence name appears in the program more than one time, it is necessary to designate which of the sequence name should be searched for restart, first, second, etc. with a comma (,) placed preceding it.

If such data is not set, the sequence restart occurs from the designated sequence name appearing first.

The allowable maximum number for this setting is 9999.

With the example program above, sequence restarts from the third N100.

Example 3:

= RS - N100;-Z [WRITE]

Sequence restarts from the N100 sequence. Positioning to the restart point is carried out by axes other than Z-axis first when the SEQ. RESTART button is pressed. Z-axis returns to the sequence restart point when the SEQ. RESTART button is pressed next.

2. Restarting from Designated Block

= RS - block-count-data [;option-designation] [WRITE]

[Supplement] 1. Designation in [] can be omitted.

- 2. Option designation as explained in 2-1 also applies in this case.
- 3. Block count data must be smaller than 65535.

Example 4:

= RS - 127 [WRITE]

Sequence restarts from the 127th block.

Example 5:

= RS - 127; + Z [WRITE]

Sequence restarts from the 127th block; Positioning to the restart point is carried out by Z-axis first when the SEQ. RESTART button is pressed. The other axes return to the sequence restart point when the SEQ. RESTART button is pressed next.

3. Restart from the Designated Block Preceding the Restart Block

```
= RS - * [number-of-blocks] [;option-designation] [WRITE]
```

[Supplement] 1. Designation in [] can be omitted.

2. The block count data corresponding to the block at which the control has been reset is registered in the NC and it is called out by the designation of an asterisk (*).

Example 6:

= RS - * [WRITE]

Sequence restarts from the sequence at which the cursor (\uparrow) was located just before the control has been reset.

Example 7:

= RS ---- *--5 [WRITE]

Sequence restarts from the sequence five blocks ahead the sequence where the cursor (\uparrow) was located just before the control has been reset.

SECTION 3 OPTION DESIGNATION

Using the option designation, it is possible to specify the order in which axis positioning is carried out at the sequence restart point. Designation of the restart point is also possible by the option designation.

1. Axis Positioning Order Designation

;±*

- (1) With this option code, designation of the axis movement order for carrying out positioning at the sequence restart point is possible.
- (2) Designate the following items.

± Either "±", "+", or "–" sign * X, Y, or Z

- (3) The option code determines the axis positioning order in the following manner:
 - + * The axis designated is first position at the sequence restart point when the SEQ.RESTART button is pressed. Positioning of the other axes occurs when the SEQ.RESTART button is pressed in the second time.
 - -* The axis designated is positioned at the sequence restart point when the SEQ.RESTART button is pressed in the second time. With the first pressing of the SEQ.RESTART button, the other two axes are positioned at the sequence restart point.



2. Designation of SequenCe Restart Point - ;M Option

;M∗

- (1) Program is processed internally from the block one block preceding the designated sequence restart sequence or block and the point where the actual position data of the designated axis matches the data of the position where the axis is actually located is taken as the restart point.
- (2) Designate X, Y or Z at the symbol " * ".

The order of axis movement can be designated by an option code.



3. Designation of Sequence Restart Point - ;TS Option

- ;TS
 - (1) Program is processed internally from the block one block preceding the designated sequence restart sequence or block and the point at which axis movements stop due to the pressing of the SLIDE HOLD button is taken as the restart point.
 - (2) The order of axis movement can be designated by an option code " $\pm *$ ".



4. Axis Movements without Option Designation

When the SEQ. RESTART button is pressed while no option code is designated, the three axes are positioned at the start point of the designated sequence or block at a manual cutting feedrate.



SECTION 4 OPERATIONS

1. Sequence Restart from the Designated Sequence

(1) Select the AUTO OPERATION mode by pressing the AUTO key.



(2) Press function key [F8] (EXTEND) two times.



(3) Press function key [F2] (RESTART).



- (4) Input a sequence name.
- (5) Press the [WRITE] key.



(6) Press the SEQ. RESTART button.

Positioning is carried out at the start point of the designated sequence in a manual cutting feedrate.

(7) Press the CYCLE START button.

Programmed operations are resumed.
2. Sequence Restart from the Position where Cutting Tool is Actually Located

- (1) Position any of X, Y and Z axes at a point where programmed operations are to resume.
- (2) Select the AUTO OPERATION mode by pressing the AUTO key.



(3) Press function key [F8] (EXTEND) two times.



(4) Press function key [F2] (RESTART).



- (5) Input the name of the sequence preceding the one in which the restart point is included and also the restart point designation option ";M*". Input the name of axis which has been positioned in step (1).
- (6) Press the WRITE key.



(7) Press the CYCLE START button.

Program is processed internally from the sequence one block preceding the designated sequence and when the actual position data of the designated axis matches the data of the axis position where the axis designated is actually located, the SLIDE HOLD lamp lights up and internal processing of the data completes.

This actual position is taken as the restart point.

(8) Press the SEQ. RESTART button.

Positioning is carried out at the restart point in a manual cutting feedrate. After the completion of positioning, the control is placed in the slide hold state.

(9) Press the CYCLE START button.

Programmed operations are resumed.

Note: The restart point will be shifted a little in the tool advancing direction from the required restart point.

Example 1:



The procedure *to* restart the sequence from X130. Y310. Z-30. is explained below (The actually returned position is slightly ahead of the designated position.):

(a) Position X-axis at X130. point.

In this positioning operation, Y and Z axes may be located at any point.

(b) Input the command as indicated below.

RS - N04;MX + Z WRITE

(c) Press the CYCLE START button.

The program is processed internally from the commanded position (X90. Y100. Z–30.) in sequence N03 and at a position a little shifted from the required restart point (X130. Y310. Z–30.) in the tool advancing direction, the internal processing ends and the SLIDE HOLD lamp lights up.

The actual position reached in this process is taken as the restart point.

(d) Press the SEQ. RESTART button.

Positioning of Z-axis occurs at the restart point in a manual cutting feedrate. After the completion of positioning, the RUN lamp begins flickering.

- (e) Press the SEQ. RESTART button again. X and Y axes are positioned at the restart point and after the completion of the positioning, the SLIDE HOLD lamp lights up.
- (f) Press the CYCLE START button.

Programmed operations are resumed.

3. Sequence Restart from a Required Position in the Program

(1) Select the AUTO OPERATION mode by pressing the AUTO key.



(2) Press function key [F8] (EXTEND) two times.



(3) Press function key [F2] (RESTART).



- (4) Input the name of a sequence before the one which contains the required restart point and the restart point designation option code ";TS".
- (5) Press the WRITE key.



(6) Press the CYCLE START button.

Program is processed internally from the sequence one block preceding the designated sequence.

- (7) When internal processing reaches the required restart point, press the SLIDE HOLD button. This point is taken as the sequence restart point.
- (8) To change the sequence restart point press the CYCLE START button, and internal processing begins again. Press the SLIDE HOLD button when required point reached.
- (9) Repeat pressing the CYCLE START and SLIDE HOLD buttons until the required restart point is reached.
- (10) Press the SEQ. RESTART button.

The axes are positioned at the required sequence restart point in a manual cutting feedrate. After the completion of positioning, the control is placed in the slide hold state.

(11) Press the CYCLE START button.

Programmed operations are resumed.

SECTION 5 SUPPLEMENTS

- (1) Axis movement order option "+*" and "-*" cannot be specified at the same time.
- (2) If shift amount by manual intervention operation differs from the previous cutting, sequence restarts from a point offset by this difference.

B. DNC-A

SECTION 1 FUNCTIONS

DNC-A for the OKUMA OSP controls the transmission of NC programs and data between a CNC machine and the host processor through an RS232C interface. It provides the following two major functions:

- (1) Downloading Receives NC programs (or files) from the host computer.
 - [INPUT] The CNC machine can also verify a received NC program (file) by matching the received NC program (file) stored in its memory with the NC program (file) that is sent from the host computer for verification.
- (2) Uploading Uploads NC programs (or files) to the host computer.

[OUTPUT] The CNC machine can retransmit the NC programs (files) so that they can be verified at the host computer.

SECTION 2 OPERATING PROCEDURES

1. Operating Procedure for DNC

(1) Press the EDIT AUX key.



(2) The display shown below will appear on the screen.

PROG (PERATION				7/07/16 1	4:10:00
				5	1/0//15	4-10-00
		1	 1			

(3) Press function key [F8] (EXTEND) twice to cause "DNC" to be displayed in function key guide area corresponding to the [F4] key.

PROG OPERATION		 107/	07/16 14.	10:00
		97/	07/15 14.	10.00
=EX				
=EX =				
ROTECT	DNC	1	[EXTEND

(4) Press function key [F4] (DNC), and the display shown below will appear.

FRUG U		1110		 97	//07/15	14:10:00
=DNC						
>					DHO	1
INFUT	VULEVI				uurr	1

(5) Pressing function keys [F1] and [F2] at this moment will enable input and output processing, respectively. Press function key [F7] (DNC QUIT) to quit the DNC mode.

2. Operation Function in the DNC Mode

You can perform two types of functions in the DNC mode as explained below.

- (1) Downloading an NC Program (File)
 - Press function key [F1] (Input).
 - Enter the name of the NC program (file) to be downloaded.
 - Press the [WRITE] key.

The general format of the IN command is shown below.



- If main file name is omitted and extension name is specified, 'A' is assumed as main file name.
- If main file name is specified and extension name is omitted, 'MIN' is assumed as extension name.
- If both main file name and extension name are omitted, nothing is assumed by the CNC machine. The NC program to be transmitted is determined by the host computer.

If file name output mode is specified (bit 7 of NC optional parameter bit Nos. 8, 13, 14, 21 and 22 are set to setting 1), 'A.MIN' is transmitted to the host computer.

- (2) Uploading an NC Program (File)
 - Press function key [F2] (Output).
 - Enter the name of the NC program (file) to be uploaded.
 - Press the [WRITE] key.

The general format of the OUT command is shown below.



as extension name.

SECTION 3 SETTING UP NC OPTIONAL PARAMETERS

(1) NC Optional Parameter (Bit) No. 10

The NC optional parameter bit No. 10 specifies whether the verify data is to be transmitted in the upload mode and whether the NC program (file) name is to be transmitted.

· · · · ·		D# 0	Mada
	BIT 4	вкз	Wiode
А	1	1	Verify data transmitted on output, no NC program (file) name transmitted
в	1	0	Verify data transmitted on output, NC program (file) name transmitted
С	0	1	No verify data transmitted on output, no NC program (file) name transmitted
D	0	0	No verify data transmitted on output, NC program (file) name transmitted

Table 1

(2) NC Optional Parameter (Word) No. 43

The NC optional parameter word No. 43 selects the DNC-A inpuinoutput channel.

Parameter value	Channel
0	CN0: (TT:)
1	CN1:
2	CN2:
3	CN3:
4	CN4:

(3) NC Optional Parameter (Bit) No. 1

The NC optional parameter bit No. 1 defines the tape delimiter code.

Table 3

	Bit 3	Mode
А	1	% (A5 in hex)
В	0	NUL (00 in hex)

NC optional parameters described in (1) through (3) are enabled immediately whenever they are reset; the CNC machine need not be switched off and on.

Table 2

(4) NC Optional Parameter (Bit) Nos. 8, 13, 14, 21, 22

The NC optional parameter bit data Nos. 8, 13, 14, 21, 22 specify whether the request file name is to be transmitted from the host computer in the download mode.

Table 4

	Bit 7	Mode
A	1	Request file name transmitted on input.
В	0	No request file name transmitted on input.

(5) RS232C Interface Parameters for CN0: (TT:) through CN4:

Refer to SECTION 3, "TAPE PUNCHER INTERFACE" in Operation Manual (Publication No. 3754-E) for the parameter settings of the RS232C interface parameters for channels CN0: (TT:) through CN4:.

SECTION 4 ERROR MESSAGES

Two types of errors may occur in the DNC mode.

5244 DNC device

A device other than CN0:, CN1:, CN2:, CN3:, and CN4: is designated by parameter (bit) data. [Index] None

[Code] None

[Measures to Take] Designate the correct data.

5245 DNC verify

In reading operation with verify designation, data following "R" and that following "V" do not match. [Index] None

[Code] The number of records in which the verify error has occurred.

0306 Command character

A command that is not in the command table has been designated.

[Character-string] Command characters which have been input

[Code] None

[Probable Faulty Locations]

1) Spelling error of a command

2) Input of a command not allowed in the mode currently selected

[Measures to Take] Input the command using function keys.

SECTION 5 SPECIFICATIONS

1. Data Formats

Data is transferred all in ISO code. The end of data is identified by two consecutive NUL (00) or % (A5 in hex) codes.



- (1) When transmission of verify data is not specified, the control command V and the subsequent data are not transmitted; instead, two consecutive % or NUL codes are transmitted and transmission terminates (see Example 3 and 4 in 1-1 "Output Data Format Examples").
- (2) When transmission of the NC program (file) name is not specified, the NC program (file name following the control command is not transmitted (see Example 1 and 3 in 1-1 "Output Data Format Examples").
- (3) The NC program (file) name may be omitted in the download mode. In this case, the program (file) name 'A.MIN' is assumed if it is not specified on the CNC machine side (see Example 2 in 1-2 "Input Data Format Examples").
- (4) Either % or NUL code can be defined in a parameter as the tape delimiter code. The host computer must use the defined tape delimiter code when sending a NC program (file) to the CNC machine.
- (5) The host computer need not transmit the V control command and subsequent data in the download mode if verify processing is not required. In this case, the host computer must send two consecutive % (or NUL) codes instead of the V control command (see Example 2 in 1-2 "Input Data Format Examples").
- (6) Only the data between the LF code and the tape delimiter code (i.e., DATA 1) is stored in memory.

1-1. Output Data Format Examples

Example 1: When A is selected in parameter setting in (1) and (3) in Section 3



Example 2: When B is selected in parameter setting in (1) and A in (3) in Section 3



Example 3: When C is selected in parameter setting in (1) and B in (3) in Section 3



Example 4: When D is selected in parameter setting in (1) and B in (3) in Section 3



NC program (file) name

1-2. Input Data Format Examples

Example 1: When A is selected in parameter setting in (3) in Section 3 and verify data is present



Example 2: When A is selected in parameter setting in (3) in Section 3 and no verify data is present

				-		
Р	%	CR	LF	N	 NUL	NUL

When the NC program (file) name is not present

- (a) If the store file name is specified, the received data is stored in bubble memory under that name.
- (b) If the request file name is specified and the store file name is omitted, the received data is stored under the request file name.
- (c) If both the request and store file names are omitted, 'A.MIN' is assumed as the name of the received file.

2. Example RS232C Interface Connections and Timing Charts

Since the CNC machine and the host computer are likely to use different RS232C interface lines, it is necessary to check the RS232C interface lines used in the host computer system and use an appropriate, dedicated RS232C cable.

2-1. Start/Stop Synchronization Mode

This mode of communications is possible using optional interfaces CN1 - CN4. Note that this mode cannot be used with standard CN0 interface.

(1) Sample Connection Diagram

r					
NC machine			Host o	computer	
Signal name	Pin No.			Signal name	
FG	(1))(\diamond	FG	
TXD	(2))($\stackrel{\frown}{\rightarrow}$	RD	
RXD	(3))∢	5	SD	
RTS	(4)) (5		
стѕ	. (5)	، لـــر	$\stackrel{-}{\rightarrow}$		
DSR	(6))	Ç	ER	
SG	(7))(Ϋ́,	SG	
RG1	(9)	>►	¢	CS	
DTR	(20))•	\diamond	DR	
EX-INT	(23))	¢	RS	
L			L		





- 3) The CNC machine sends the Data Request signal RG1.
- 4) The host computer, on receiving RG1, starts transmitting a brie of serial data.
- 5) The Data Request signal is reset on the start bit of the serial data byte.
- (b) Upload mode timing chart



- 1) The CNC machine sends a byte of transmitted data when the Data Busy signal EX-INT is high.
- 2) The host computer resets the Data Busy signal when it reads the stop bit of the received serial data byte. The Data Busy signal must be reset each time a serial data brie is received.
- 3) The host computer sets the Data Busy signal when it completes processing of the received data byte.
- [Supplement] Since this example configuration uses the EX-INT signal, NC optional parameter bit for the "no EX-INT signal" mode (bit 1 of NC optional parameter bit Nos. 8, 13, 14, 21, 22) must be set to "0".

2-2. Control DC Code

The DC (Device Control) codes are defined as follows irrespective of the code of the NC program data to be transmitted (ISO or EIA):

	Character	8	7	6	5	4		3	2	1
DC1	Tape Reader Start				0		0			0
DC2	Tape Punch Start				0		0		0	
DC3	Tape Reader Stop	0			0		0		0	0
DC4	Tape Punch Stop				0		0	0		

[Supplement] 1. The RS232C interface parameters must be set to use DC codes (refer to SECTION 3, "Tape Puncher Interface" in Operation Manual (Publication No. 4106-E) for the parameter settings of the RS232C interface parameters).

2. DC codes are automatically generated by the NC machine. The program need not consider the DC codes.

Sample Connection Diagrams:

(1) Example 1



(a) Download mode timing charts

1) When request-file name output mode is not specified (B is set in (4) in Section 3 "SETTING UP NC OPTIONAL PARAMETERS")



2) When request-file name output mode is not specified (A is set in (4) in Section 3 "SETTING UP NC OPTIONAL PARAMETERS")



(b) Upload mode timing charts

1) When request-file name output mode is not specified (Either A or C is set in (1) in Section 3 "SETTING UP NC OPTIONAL PARAMETERS")



Received Data RXD

- [Supplement] 1. When CTS is set off during data transfer, the data transfer is suspended within two characters.
 - 2. Since this example configuration does not use the EX-INT signal, NC optional parameter bit for the "no EX-INT signal" mode (bit 1 of NC optional parameter bit Nos. 8, 13, 14, 21, 22) must be set to "1".

(2) Example 2



(a) Download mode timing charts

The timing chart in this mode is identical to that given in Example 1.

(b) Upload mode timing charts



[Supplement] Since this example configuration uses the EX-INT signal, NC optional parameter bit for the "no EX-INT signal" mode (bit 1 of NC optional parameter bit Nos. 8, 13, 14, 21,22) must be set to 0.

2-3. Control DC Code on Type 2

The Control DC code on type 2 configuration uses only DC codes to control data transfer between the host computer and a CNC machine through an RS232C interface.

- (1) While the standard Control DC code configuration allows only the CNC machine to generate DC codes, this configuration also allows the host computer to generate DC codes.
- (2) The CNC machine uses four device control codes (DC1, DC2, DC3, and DC4) and the host computer two codes (DC1 and DC3).

DC Code	NC Machine	Host Computer
DC1	Enables data read. (a) Initiates a read. (b) Resumes a read.	Enables data read. (a) Responds to DC2. (b) Resumes a read.
DC2	Placed before the transmitted data to request the host computer to receive the transmitted data.	
DC3	Requests suspension of data transmission.	Requests suspension of data transmission.
DC4	Identifies the end of data transmission.	

Table 5

(3) Sample Connection Diagrams



[Supplement] Since this example configuration does not use the EX-INT signal, NC optional parameter bit for the "no EX-INT signal" mode (bit 1 of NC optional parameter bit Nos. 8, 13, 14, 21 and 22) must be set to 1.

- (a) Download mode timing charts
- 1) When request-file name output mode is not specified (B is set in (4) in Section 3 "SETTING UP NC OPTIONAL PARAMETERS") The timing chart in this mode is identical to that given in 2-2, "Control DC Code".
- 2) When request-file name output mode is specified (A is set in (4) in Section 3 "SETTING UP NC OPTIONAL PARAMETERS")



- (b) Upload mode timing charts
- 1) When file name output mode is not specified (Either A or C is set in (1) in Section 3 "SETTING UP NC OPTIONAL PARAMETERS")



2) When file name output mode is not specified (Either B or D is set in (1) in Section 3 "SETTING UP NC OPTIONAL PARAMETERS")



C. DNC-B HIGH-SPEED RM BUFFER METHOD

SECTION 1 OVERVIEW

This section explains the communication link between the OSP and the host computer to transfer part programs between these two devices.

(1) There are two types of program transfer methods. One is the batch transfer method in which part programs are transferred independently of operation status of the OSP and the other is on-line mode program transfer in which part programs are transferred from the host computer to the buffer memory of the OSP and the machine is controlled by the program being sent. The latter mode of program transfer is called "buffer operation" in this manual.

Note that these two methods of program transfer cannot be selected at the same time.

- (2) The schedule operation is possible in the buffer operation mode by setting the parameters.
- (3) Two types of protocol used for communication between the host computer and the OSP are selectable by the setting of parameter. These are protocol A and protocol B.

Protocol A Shake hand method in which transmission and reception of data are repeated each other.

Protocol B Communication is controlled by the DC (device control) codes which are output from the OSP.

SECTION 2 DNC OPERATION PANEL



- (1) COMMUNICATION ON/OFF switch
 - (a) COMMUNICATION ON

Turning on power supply after placing the switch in the ON position establishes the communication mode between the OSP and the host computer. The COMMUNICATION lamp lights up.

(b) COMMUNICATION OFF

Turning on power supply after placing the switch in the OFF position disconnects communication line between the OSP and the host computer.

- 1) Once power supply has been turned off, changing the switch setting position has no effect and the established state cannot be changed.
- 2) With protocol A, turn the switch in the OFF position once and then back to the ON position to recover the communication alarm state* from the NC. Even in this case, the previously established communication ON state is not changed.
 - * The state in which either of the following alarm messages (see 12-1 "Communication Alarm") is displayed.
 - 4218 DNC-B communication error
 - 4219 DNC-B message format error

(2) BUFFER OPERATION ON/OFF switch

(a) BUFFER OPERATION ON

By setting the switch in the ON position with the DNC COMMUNICATION ON state, buffer operation is enabled and the BUFFER OPERATION ON lamp lights up.

(b) BUFFER OPERATION OFF

Buffer operation is disabled.

- 1) If the switch is set at the BUFFER OPERATION OFF position during buffer operation, the buffer operation enabled state is continued until the buffer operation has completed.
- 2) With protocol A, ON/OFF state of the buffer operation can be changed by the command from the host computer. If the command from the host computer does not match the switch setting, the command from the host computer is given priority.

However, if communication alarm occurs, ON/OFF state of the buffer operation is controlled by the setting of the BUFFER OPERATION ON/OFF switch setting at the OSP operation panel.

SECTION 3 BUFFER OPERATION

Buffer operation is possible in the automatic mode while the BUFFER OPERATION ON lamp at the DNC operation panel lights up.

Buffer operation function varies depending on the NC program transmission method (with or without file name designation).

Operation Type	Program Type	Operation Method	Buffer C (DNC m Without Designa	Deration ode: ON, Bu File Name ttion	uffer opera With F Desigr	ation: ON) ile Name nation	Norma Operat	l Memory ion
			Memory	Host Computer	Memory	Host Computer	Memory	Host Computer
Manual	Main	A	×	×	0	×	0	×
Mode	Program	В	×	×	0	×	0	Х
Operation		S	×	0	0	0	0	×
Operation According	Schedule Program	_	×	×	0	0	0	×
to	Main	A	×	х	0	×	0	×
Schedule	Program	В	×	×	0	×	0	×
Program		S	×	×	0	0	0	×

Table 1

[Supplement] 1. "O"indicates "operation possible" and " × " indicates "operation impossible".

- 2. Operation Methods:
 - A Normal operation
 - B Large-volume operation

S Subprogram and branch function are not checked in operation method B.

The following operations are possible depending on "with" and "without" file name designation.

Without file name designation Buffer operation of NC program

With file name designation Buffer operation of NC program Scheduled operation based on schedule

> program Memory mode operation based on programs stored in memory

1. Buffer Operation of NC Programs

- 1-1. Operation Methods
 - (1) If "file name not used" state is selected,
 - (a) Place the DNC COMMUNICATION ON/OFF switch at ON.
 - (b) Turn on power supply.
 - (c) Select the AUTO OPERATION mode by pressing the AUTO key.
 - (d) Place the BUFFER OPERATION ON/OFF switch at ON.
 - (e) Press the CYCLE START button. (PROGRAM SELECT operation is not necessary.)
 - (2) If "file name used" state is selected,
 - (a) Place the DNC COMMUNICATION ON/OFF switch at ON.
 - (b) Turn on power supply.
 - (c) Select the AUTO OPERATION mode by pressing the AUTO key.
 - (d) Place the BUFFER OPERATION ON/OFF switch at ON.
 - (e) After pressing function key [F1] (PROGRAM SELECT), input the main program file name.
 - (f) Press the [WRITE] key
 - (g) The screen displays prompt "Request file name data?!", requesting the operator to input the file address character-string managed by the host computer.
 - (h) Press the [WRITE] key.
 - (i) Press the CYCLE START button.

1-2. Command Format

(Possible only when "file name used" state is selected.)

= PS [main-program-file-name] [WRITE]

Request file name data?! file-address-character-string requested [WRITE]

(1) Main Program File Name

[program file name].[extension]

- (a) Designate a character-string of up to 16 characters beginning with an alphabet. Alphabets, numbers and hyphen (-) can be used.
- (b) The extension is MIN.
- (c) The main program file name designated in this step is registered as the file name for buffer operation.
- (d) If a main program file name is omitted, A.MIN is used.
- (e) To designate a device name in the program file name, designate the host computer (DNC:).
- (2) File Address Character-string Requested

Character-string of up to 30 characters

Following characters can be used:

0-9, A-Z, !, ", #, \$, %, &, ', <, >, (,), @,

Transmission of the NC program of the request-file-name designated in this step is requested to the host computer.

If the request-file-name is omitted, the main program file name is designated.

2. Scheduled Operation by Schedule Program

(Possible only when "file name used" state is selected.)

2-1. Operation Methods

- (1) Place the DNC COMMUNICATION ON/OFF switch at ON.
- (2) Turn on power supply.
- (3) Select the AUTO OPERATION mode by pressing the AUTO key.
- (4) Place the BUFFER OPERATION ON/OFF switch at ON.
- (5) After pressing function keys [F8] (EXTEND) two times and then [F4] (SP SELECT), input the schedule program file name.
- (6) Press the [WRITE] key.
- (7) The screen displays prompt "Request file name data?!", requesting the operator to input the file address character-string managed by the host computer.
- (8) Press the [WRITE] key.
- (9) Confirm that the schedule program is selected on the program display screen.
- (10) Press the CYCLE START button.

For the details of schedule program, refer to OSP Programming Manual.

2-2. Command Format for Schedule Program Selection

= SS [Schedule-program-file-name] [WRITE]

Request file name data?! file-address-character-string requested [WRITE]

(1) Schedule Program File Name

[program file name].[extension]

- (a) Designate a character-string of up to 16 characters beginning with an alphabet. Alphabets, numbers and hyphen (-) can be used.
- (b) The extension is SDF.
- (c) The schedule program file name designated in this step is registered as the file name for schedule operation.
- (d) If a schedule program file name is omitted, S.SDF is used.
- (e) To designate a device name in the program file name, designate the host computer (DNC:).
- (2) File Address Character-string Requested

Character-string of up to 30 characters

following characters can be used:

0-9, A-Z, !, ", #, \$, %, &, ', <, >, (,), @,

?, ¥, , [,], , , , , , +, -, *, /, =, ., SP

Transmission of the schedule program of the request-file-name designated in this step is requested to the host computer.

If the request-file-name is omitted, the schedule program file name is designated.

3. Memory Mode Operation Using a Program Stored in Memory

The operation method is quite the same as used for normal memory mode operation.

Only the difference is the designation of the memory (MD1:) as the device name when designating the schedule program file name and the main program file name (including main programs in the schedule program).

4. Supplements

- (1) Subprograms only registered in the library program can be used.
- (2) Size of part program has no limit for program transfer.
- (3) Only schedule programs or main programs can be transmitted.

If subprograms and/or other main programs continue following the M02 code of the main program, do not forget to set the "M02 reset output used" state by the parameter.

- (4) At the instant the CYCLE START button is pressed, transfer of a part program begins (GTD sent for protocol A, DC1 or DC2 sent for protocol B) and then the program execution begins.
- (5) Change of parameter setting for file name used or not-used status becomes active when power supply is turned off and then on again.
- (6) Even in the buffer operation mode, intervention of manual operation is possible as usual.
- (7) Even in the buffer operation ON mode, the return search and sequence restart commands can be executed.
- (8) In the buffer operation mode, when "file name not used" is selected, the following commands cannot be executed:
 - Schedule program selection
 - Number search
 - Sequence search using the cursor
 - Schedule program search
- (9) When "file name not used" state is selected, program selection is always required when buffer operation mode is switched form ON to OFF. If program selection is not carried out, program execution does not begin.
- (10) When "file name used" state is selected, program selection is always required when buffer operation mode is switched between ON and OFF. If program selection is not carried out, program execution does not begin.
- (11) When selecting a program in the file name used state, designation of a main program name and subprogram name is not possible.
- (12) Irrespective of the setting of the BUFFER OPERATION ON/OFF switch, the selected buffer operation on or off mode is continued until the operation in progress has been completed even if the setting of the switch is changed.

This function is effective for program transfer operation.

If the buffer operation on/off mode is switched by the command using protocol A, an alarm occurs if such an attempt is made during operation irrespective of the setting of the buffer operation on/off mode.

- (13) Batch transfer of NC program is impossible during buffer operation (buffer operation mode on and process in between cycle start and M02 execution, or NC reset).
- (14) After the execution of MDI mode operation, buffer operation is not possible unless the NC is once reset.

Interruption by MDI mode operation during buffer mode operation may be executed in normal operation intervention manner.

- (15) Automatic program selection when power is turned on:
 - (a) File name not used

BUFFER OPERATION ON Program selection is not executed. BUFFER OPERATION OFF Program selection is executed.

(b) File name used

The buffer operation on/off mode and the file name being selected when power has been turned off are stored in memory. If the buffer operation on/off mode currently active and the mode active before power off match, automatic program selection is executed. When buffer operation mode is on, the main program of the file name stored is requested to the host computer when the CYCLE START button is pressed.

If the previous and current buffer operation on/off modes do not match, automatic program selection is not executed. When buffer operation mode is on, no operation occurs when the CYCLE START button is pressed.
SECTION 4 BATCH TRANSFER OF NC PROGRAMS

NC programs can be transferred from the NC to the host computer or from the host computer to the NC in batch.

1. Operation Procedure

Follow the steps below:

- (1) Place the DNC COMMUNICATION ON/OFF switch at ON.
- (2) Turn on power supply.
- (3) Place the BUFFER OPERATION ON/OFF switch at OFF.
- (4) Select the PROG OPERATION mode by pressing the EDIT AUX key.
- (5) Press function keys [F8] (EXTEND) and then [F4] (DNC). This displays the DNC screen.In this status, batch transfer of the program is possible.

1								-	
(PROG 0	PERATION	DNC-B] `
						97	7/07/15	14:10:00	
1									
l	=EX								
	=EX =DNC								
	>								
							DNC		_
	INPUT	OUTPUT	VERIFY				QUIT		
~						-	-	-	~
	(F1	D(F2)) (F 3) (F 4) (F 5)(F 6	(F7)	(F 8)]
					Fig 1				
					· .9. ·				

1	c	١.
1	О)
۰.	_	5

(a) For program request to the host computer

After pressing function key [F1] (INPUT), input the file name of the file to be stored in the memory.

(b) For sending NC program to the host computer

After pressing function key [F2] (OUTPUT), input the file name of the file stored in the memory.

(c) For program verification

To verify the NC program already stored in the memory, press function key [F3] (VERIFY) and input the file name of the file stored in the memory.

(7) Press the [WRITE] key.

If "file name used" state is selected by the parameter, follow the step (8) and (9) below.

If such state is not selected, batch transfer of the designated file begins.

(8)

(a) For program request to the host computer

Key in the request-file name designation data.

(b) For sending NC program to the host computer

Key in the transmission-file-name designation data.

(c) For program verification

Key in the request-file-name designation data.

(9) Press the [WRITE] key.

This begins transmission of the NC program in batch.

- (10) After the completion of program transmission, press function key [F7] (DNC QUIT).
- [Supplement] 1. As request-file-name designation data and transmission-file-name designation data, input the program file name data (character-string) which is managed by the host computer. This allows the input data to be sent to the host computer.
 - 2. For program verify operation, the following message is displayed after the completion of verification in accordance with the results of the verification.

data all same

File and received NC data match.

DNC verify unsame-2

Although the file has terminated, received NC data is left.

DNC verify unsame-1

Although the received NC data has terminated, file data is left.

DNC verify unsame-n

The number (n) of mismatch data sets in verify operation.

2. Command Format

(1) Input



[Supplement] The "file-address-character-string requested" column becomes effective when "file name used" state is selected by parameter. If "file name used" state is not selected, it should be omitted.



[Supplement] The "file-address-character-string transmitted" column becomes effective when "file name used" state is selected by parameter. If "file name used" state is not selected, it should be omitted.

(3) Verify >V stored-file-name extension 1 WRITE ſ Request file name data?! file-address-character-WRITE string requested (Note) Character-string of up to 30 characters; following characters can be used: 0 - 9, A - Z, !, ", #, \$, %, &, ', <, >, (,), @, ?, ¥, , [,], ,;,:,, +, -, *, /, =, , SP Transmission of the NC program of the requestfile-name designated in this step is requested to the host computer. If the request-file-name is omitted, the stored-filename is designated. Three characters of alphabets Designate a character-string of up to 16 characters beginning with an alphabet. Alphabets, numbers and hyphen (-) can be used. The transmitted NC program is stored in the memory with the file name and extension designated in third step. If the designation of stored-file-name is omitted with only extension designated, "A" is assigned. If the designation of extension is omitted with only the stored-file-name designated, "MIN" is assigned.

[Supplement] The "file-address-character-string requested" column becomes effective when "file name used" state is selected by parameter. If "file name used" state is not selected, it should be omitted.

3. Supplements

- (1) Transmission and Reception of NC Programs
 - (a) Protocol A

Method 1 File name is not added to RTD command and the first GTD command.

- Method 2 File name is added to RTD command and the first GTD command.
- (b) Protocol B

Method 1 File name is not transmitted.

Method 2 For program transmission, file name is first transmitted; for program reception, DC2 code, file name and DC4 code are transmitted.

In each protocol, switching between method 1 and method 2 is made by the parameter. After changing parameter setting, it is necessary to turn off power supply and then turn it back on. Note that the parameter used for switching the method is used in common with the one used for switching the operation method in the buffer mode operation.

(2) For program "input" operation, if the file assigned with the file name and extension same as those designated is already existing, prompt "file exist overwrite? (Y/N)!" appears on the screen.

If "Y" is input for this prompt, the existing file is deleted and new file is input from the host computer.

If "N" is input, no processing occurs.

SECTION 5 PARAMETERS

1. Parameters which cannot be Set at OSP

The parameters indicated below cannot *be* set at the OSP and setting is possible only by using the SET command from the host computer.

(Parameters in the table below, conversion codes, codes after conversion (45th - 48th bytes of SET command))

(For protocol A only)

Parameter	Function	Unit	Range	Initial Value when Power is Turned On
Nb	Minimum buffer area of CNC when transmitting [GTD]	bytes	1 - 4000	2000
No	Maximum overrun amount in reception	bytes	2 - 2000	50
Ne	Number of retries allowed when transmission error is detected	times	0 - 100	10
Тр	[SAT] transmission polling time for DNC, or response time-out duration by CNC for host computer	S	1 - 999	5
То	Response time-out duration by host computer for CNC	s	1 - 999	20
Ti	Minimum time interval between transmission bytes	ms	0 - 10	10
Тх	Minimum switching time from reception to transmission	ms	0 - 100	100
Tw*	Standby time for reception of [WAT] used by both of CNC and host computer; upper limit value is Tp or To, whichever smaller.	S	0 - Tp 0 - To (Whichever smaller)	5

Table 2

*Tw: Communication is executed in the following manner when "Tw" value exceeding the specified range is set by the host computer using the SET command.

(1) When Tw value > Tp value

In the remote buffer mode operation or batch transmission mode, the OSP sends the GTD command after waiting for "Tw" when it received the WAT command from the host computer. The SAT command is not sent out during this period.

(2) When Tw value > To value

If program data to be transmitted is not ready in the NC program for transmission from the OSP to the host computer operation, the OSP sends the WAT command after and elapse of "Tp". If the host computer sends the GTD command in response to this after waiting for "Tw", the OSP regards that, after an elapse of "To", there is no answer to the WAT command. In this case the communication error occurs.

2. Parameters which can be Set at OSP

Some parameters are required to be changed frequently during operation while some are not changed once they have been set. The OSP differentiates these two types of parameters to allow easy setting of parameters. The former is assigned to NC optional parameter (long word), (word) and (bit), and the latter to the communication parameter (dedicated screen).

- (1) NC Optional Parameter (Bit)
 - (a) Bit 0 of No. 1 /Bit 1 of No. 11

Coding system of part programs

<u>Bit 0 of No. 1</u>	<u>Bit 1 of No. 11</u>	
	1	ASCII
1	0	ISO
0	0	EIA
Initial setting: 1		

[Supplement] ASCII code cannot be used if protocol B is selected.

For protocol B, do not set "1" for bit 1 of No. 11. The following alarm occurs if "1" set for this bit.

4037 Alarm D CCP-CPU error 4

(b) Bit 2 of No. 1

TV check, creation designation

This parameter determines whether or not the TV check, creation is made for the NC program to be received or transmitted - a space is inserted before LF (or CR LF) so that a total number of characters in a record including the LF (or CR LF) will be even, or whether the record contains an even number of characters is checked.

- 1: TV check, creation is executed.
- 2: TV check, creation is not executed.
- Initial setting: 0

(c) Bit 3 of No. 11

End of record code

1: LF 0: CR LF Initial setting: 0

(d) Bit 0 - bit 7 of No. 43 and No. 44

Data at the 41st to 44th bytes of [SAT] or [SET] command

[SAT] command From OSP to the host computer [SET] command From the host computer to OSP

Bit 0 of No. 44

Buffer is cleared when NC is reset.
 Buffer is not cleared when NC is reset.
 Initial setting: 0

[Supplement] Other bits have not been defined.

- (2) NC Optional Parameter (Word)
 - (e) No. 16

Number of bytes in one block for transmission of NC program from OSP to the host computer in protocol A.

If setting is "0", it follows Nb sent from the host computer.

Initial setting: 0

Setting range: 0 - 4000

(f) No. 24

File read-out waiting time T2 used for protocol B (Time duration between transmission of DC4 code and transmission of DC1 code)

Initial setting: 0 (s)

Setting range: 0 - 9999 (s)

If setting is "0", time duration of T1 (Parameter No. 35) is automatically adopted.

(g) No. 35

Response timer used for protocol B (Time duration between transmission of DC1 code and input of NC program)

Initial setting: 10 (s)

Setting range: 1 - 9999 (s)

- (3) NC Optional Parameter (Long Word)
 - (h) No. 13

In the remote mode operation, program is started after the specified number of character of the NC program is written to the buffer.

Initial setting: 0

Setting range: 0 - 99999999

- [Supplement] If the setting is smaller than 256, it is handled as 256. If the setting is greater than the main program buffer capacity, program is started when the main program buffer became full.
- (4) Communication Parameters

Communication parameters are displayed and set in the parameter setting mode.

- (a) Select the PARAMETER SET mode by pressing the PARAMETER key.
- (b) Display the COMMUNICATION PARAMETER screen.
- (c) Set the required parameters.

This setting is possible by using function key [F2] (SELECT). Each time this key is pressed, contents are automatically changed within a setting range.

- (d) Press function key [F5] (BACKUP). This operation is always required when data is set or changed.
- (e) Turn off power supply and then turn it back on.

The communication parameters become effective when power reapplied.

(5) Description of Communication Parameters

	*			<u>ا</u> ا	97/07/15	4:10:00
	* (0)	WINDERATIO	N PAHAMETE	:H *		
NU. 1 COMUNNIC, 2 BAUD RATI 3 RS232C S 4 RS232C P, 5 TRANSMIT 6 MESSAGE I 7 PROTOCOL 8 NC PROGR 9 LAST 'DC: 10 SYN/NAK I 11 M02 RESE ACT POSIT(WO	ATION DEVIC E TOP BITS ARITY CODE END CODE A/B MM TRANSMIT 3'CODE ENAL ENABLE T OUTPUT EI XABLE T OUTPUT EI RK) – R	CE T METHOD BLE NABLE (100.000 Mtd	¥ -200. 00	00	Ci 24 EVI 1: 1: 2 YI YI YI 2 -300.000	4A 000 EN 60 TX A A ES ES ES
=CMPS =			1			

F1F2F3F4F5F6F7FB

No.	Name	Description
1	COMMUNICATION	Designation of device name used for communication
	DEVICE	Select from CHA: and CHB:.
		Initial setting: CHA:
2	BAUD RATE (BPS)	Designation of baud rate
		Select from 110, 150, 300, 600, 1200, 2400, 4800, 9600, 19200, 38400, 89300.
		Initial setting: 2400
3	RS232C STOP	Designation of number of stop bits
	BIIS	Select from 1 and 2.
		Initial setting: 1
4	RS232C PARITY	Designation of parity added at RS232C interface
		Select from NON, EVEN and ODD.
		Initial setting: EVEN
5	TRANSMIT CODE	Designation of transmission code (excluding NC programs)
		Select from ISO and ASCII.
		Initial setting: ISO
		[Supplement] The transmission code is not used for protocol B.
6	MESSAGE END	Designation of message end code
	CODE	Select from ETX and CR.
		Initial setting: ETX
		[Supplement] The end code is not used in protocol B.

No.	Name	Description						
7	PROTOCOL A/B	Selection of protocol						
		Select from A (protocol A) and B (protocol B).						
		Initial setting: A						
8	NC PROGRAM	Designation of "file name used" or "file name not used" state for						
	METHOD	program transmission						
		Select from A (not used) and B (used).						
		Initial setting: A						
9	LAST 'DC3' CODE	Designation whether or not DC3 code at the end of NC program						
	ENABLE	reception is transmitted						
		Select from YES (transmitted) and NO (not transmitted).						
		Initial setting: YES						
		[Supplement] The DC3 code is not used in protocol A.						
10	SYN/NAK ENABLE	Designation whether or not [SYN] and [NAK] are transmitted at						
		an occurrence of NC alarm or NC reset						
		Select from YES (transmitted) and NO (not transmitted).						
		Initial setting: YES						
		[Supplement] This is not used in protocol A.						
11	M02 RESET	Designation whether or not reset signal ([RST] command for						
	OUTPUT ENABLE	protocol A, [SYN] code for protocol B) is transmitted to the host						
		computer upon execution of M02 and M30.						
		Select from YES (transmitted) and NO (not transmitted).						
		Initial setting: YES						

SECTION 6 PROTOCOL A

1. Communication Format

When the power is turned ON, the NC and host computer are prepared for operation. When this preparation is completed, transmissions from the NC and reception from the host computer begins. From this point on, transmissions and receptions are repeated in an alternating manner.



Fig. 2 Example of RS232C Link



Fig. 3 Timing Chart at Communication START

[Supplement] The initial transmission request occurs 2 seconds after a READY status is established at both the NC and host computer.





Minimum time interval between transmission bytes	: Ti ≦ t
"Transmission to reception" switching time	: $0 \leq ts \leq To$
"Reception to transmission" switching time	: $Tx \leq t \leq Tp$
Overrun which follows CTS signal OFF	:n ≦ 2 bytes
Overrun amount which follows reception RTS signal OFF	: n ≦ No
 To: Timeout period Tp: Polling time interval Tx: Minimum "reception to transmission" switching time No: Maximum overrun amount during reception 	
	 Minimum time interval between transmission bytes "Transmission to reception" switching time "Reception to transmission" switching time Overrun which follows CTS signal OFF Overrun amount which follows reception RTS signal OFF To: Timeout period Tp: Polling time interval Tx: Minimum "reception to transmission" switching time No: Maximum overrun amount during reception

[Supplement] There is no specific designation for "Ti".

2. Message Format

Data exchanges (hereafter referred to as "messages") between the NC and host computer occurs using the data format shown below.



All data, including the Check Sum, is comprised of ISO or ASCII codes (parameter switching).

However, when the "DATA" portion consists of NC program or NC data, EIA codes can be used for that data only.

(1)	Check Sum	The COMMAND to END code data total is shown at the Check Sum area, where the final 1-byte of the total is expressed as a hexadecimal, 2-digit value. (Transmission occurs in order, from high to low.)
(2)	Command	The COMMAND is comprised of 3 characters, and indicates the message type.
(3)	Data	This is the text data of variable length. The format and content of this data is determined according to the COMMAND. (Numeric data is transmitted in order, from high to low.)
(4)	END Code	This code indicates the end of the message. Either "ETX" or "CR" may be used as this code (parameter switching).
		When ISO code is used: ETX (03H) or CR (8DH) When ASCII code is used: ETX (03H) or CR (0DH)
		The message must not contain any data which appears in the same pattern as

the END code.

3. Command List

Command	Description	Data Area	Reply From Host
SYN	Initialize command	None	SYN
RDY	Initialize completed	None	SYN, RDY
RST	DNC reset notification	None	ARS
ALM	DNC alarm notification	None	AAL
SAT	DNC status notification	Status	SET, CLB, SYN
GTD	NC Program data transmission request	NC Program file name	DAT, EOD, WAT
RTY	Repeat request (retry)	Error status	Previous message
RTD	NC Program data upload request	NC Program file name	GTD
DAT	NC Program data upload	NC Program data	GTD
WAT	Wait	None	GTD
EOT	NC Program data uploading completed	None	SET

(1) Transmission Commands (NC \rightarrow host computer)

(2) Reception Commands (host computer→NC)

Command	Description	Data Area	Reply From NC
SYN	Initialize command	None	RDY
RDY	Initialize completed	None	RST, ALM, SAT,
			GTD
ARS	RST reply	None	Same as above
AAL	ALM reply	None	RST, SAT
CLB	Buffer delete	None	RST, ALM, SAT,
			GTD, RTD
SET	NC status change	Status	Same as above
DAT	NC Program data transmission	NC Program data	Same as above
EOD	NC Program data transmission	None	Same as above
	completed		
WAT	Wait command	None	GTD
RTY	Repeat transmission (retry)	Error status	Previous message
GTD	NC Program data transmission request	None	DAT, WAT, EOD

4. Data Reception

(1) When File Name is not Designated:



5. Data Transmission

(1) When File Name is not Designated:



(2) When File Name is Designated:



SECTION 7 PROTOCOL B

1. Communication Format

Data transmission/reception requests are always sent from the NC to the host computer using DC codes (Device Control Code).



Fig. 5 Example of RS232C Link

2. DC (Device Control) Codes

The DC codes will be as shown in the Table below, regardless of the code system (ISO or EIA) of the NC program data which is being transmitted.

Character		8	7	6	5	4		3	2	1
DC1	Tape Reader START				0		0			0
DC2	Tape Puncher START				0		0		0	
DC3	Tape Reader STOP	0			0		0		0	0
DC4	Tape Puncher STOP				0		0	0		

[Supplement] DC codes are generated automatically by the NC, and need not be included in the program.

3. Data Reception

(1) When File Name is not Designated:





(2) When File Name is Designated:

1

4. Data Transmission

(1) When File Name is not Designated:



ī.

(2) When File Name is Designated:



4188-E C-32-R1 C. DNC-B HIGH-SPEED RM BUFFER METHOD

L

SECTION 8 DATA FORMAT

1. Input Format

<For ISO Code >



< For EIA Code >



- The transmission data must begin with a "%" or "ER" code, and must also end with a "%" or "ER" code. If the data does not begin with either a "%" or "ER" code, all the data (from the very beginning) will be processed as significant data, and the initial "%" or "ER" code which appears will be processed as the END code.
- When ISO code is used, an [LF] or [CR LF] code should be designated at the end of each record within the program. Switching between the "[LF] only" and the "[CR LF]" format is possible (parameter setting).
- When EIA code is used, a [CR] code should be designated at the end of each record within the program.
- [Supplement] 1. The continuing transfer of a subprogram following a main program is possible only during batch transfer operations. In such cases, a breakpoint code should not be used between the main program and the subprogram, or between two subprograms.
 - 2. The maximum record length is 156 bytes.

2. Output Format

<For ISO Code>



- When ISO code is used, the output will begin with a [CR LF] or [% LF] code. When EIA code is used, output will begin with a [ER CR] code.
- The machining program data will be output following the Program Name (No.).
- When ISO code is used, the output will end with a [CR LF] or [% LF] code. When EIA code is used, output will end with an [ER CR] code.

SECTION 9 SPECIFICATIONS

1. RS232C Interface

(1) Communication Format

Asynchronous Transmission

With this format, data is sent by preceding each character with a start bit and following it with a stop bit.

1 character consists of the ① START Bit (1 bit), ② Information Bit (8 bits), ③ Parity Bit (1 bit), and the ④ STOP Bit (2 bits), as shown in Fig. 6 below.



Fig. 6 Bit Configuration

- (2) Baud Rate (BPS)
 110, 300, 600, 1200, 2400, 4800, 9600, 19200, 38400, 89300
- (3) Data Configuration
 - START Bit (1 bit)

Data Bits (8 bits)

Parity Bit ("1 bit" or "none" \rightarrow selected by parameter setting)

STOP bits ("1 bit" or "2 bits" \rightarrow selected by parameter setting)

(4) Parity Check (Character Parity)

"Odd/Even parity" or "no parity" (selected by parameter setting)

- (5) Maximum Data Transmission Distance
 - RS232C : Maximum cable length of 15 meters
 - RS-422 : Maximum cable length of 1200 meters (an RS232C RS422 converter is used)

Fiber Optics : Maximum cable length of 2000 meters (a Fiber Optics - RS232C converter is used)







(7) Signal Descriptions

Pin No. (DB25)	Signal Name	Signal Direction	Description
1	FG	_	Safety ground
2	TXD	Output	Transmission Data
			Data line from OSP to external device
3	RXD	Input	Transmission Data
			Data line from external device to OSP
4	RTS	Output	Transmission Request
			Switches ON when transmission or reception begins.
			It then remains ON.
5	CTS	Input	Transmission Possible
			When this signal is OFF, no data will output from the OSP.
			This signal is used for BUSY/READY control.
			When this signal is not used, the RTS signal at the NC should be connected.

Pin No. (DB25)	Signal Name	Signal Direction	Description
6	DSR	Input	Data Set Ready This signal indicates that a "communication enabled" status exists at the external
			device.
			If this signal switches OFF during communications, an error will be activated at the OSP.
			BUSY/READY control is not possible at this signal. When this signal is not used, the DTR signal at the OSP should be connected.
7	SG	_	For signal connection
20	D⊺R	Output	Operation Preparation Completed
			This signal switches ON when the OSP is ready for operation. When this signal is OFF, any data which is transmitted to the OSP will not be received.

(8) Connector Layout



SECTION 10 SCREEN DISPLAY

1. Check Screens

(1) Display the communication status and parameters on the CHECK DATA screen. (This screen is available only for protocol A.)

	AUTO OPE	RATION					N	0
	CHECK DATA			DNC (B)			<u>/07/15</u> 52	<u>14:10:00</u> 1mm
	HOST CPU CNC STAT PARAMETE CONVERT	J STATU: TUS ER ND Ne Tp To Ti Tx Tw CODE ED CODE	5	1 2000 by 50 by 10 ti 5 se 20 se 0 ms 100 ms 5 se 00 00	te mes c c ec ec c			
= = = >	EX DNc							
PF SE	NOGRAM /	ACTUAL POSIT.	PART PROGRAM	BLOCK DATA	SEARCH		CHECK Data	[EXTEND]

F1F2F3F4F5F6F7F8

Table 3

No.	Name	Description
1	HOST CPU STATUS	Displays the second byte data in [SET] command. Select from CHA: and CHB:. Set "CHB:". Initial setting: CHA:
2	CNC STATUS	Displays the second byte data in [SAT] command. 0: Not ready status 1: CNC in normal status 2: Buffer operation status 3: Alarm status
3	PARAMETER Nb - Tw	Displays contents of [SET] and [SAT] commands from 9th to 40th bytes.
4	CONVERT CODE/CONVERTED CODE	Displays code conversion designation for NC programs.

AUTO OF	PERATION					<u>N</u> 97/07/15	<u>3</u> 14:10:00
CHECK (ATA		* DNC RE	CEIVE	DATA *	PAGE 50	1mm
\$0000	C644535	9 4E03B2	7B2 D24	45 9 03	444153C5	FDSYN. 72	2RDY. DASE
\$0010	D430B1A	O AOAOA	DAO AO3	0B744	30303033	T01	07D0003
\$0020	B230303	0 413030	030 353	030B1	B4303030	2000A000	050014000
\$0030	3030303	6 B43030	30 353	03030	30303030	00064000	1200000000
\$0040	3003050	6 53C5D4	103 C5C	65305	04030506	0.EFSET.	EFSELEF
\$0050	5305D40	13 C5C65	SC5 D40	30506	53C5D403	SET. EFSI	I.EFSEI.
\$0060	0506530	5 D403C	DCD 53C	50403	05065305	E-SET.E	-SET.EFSE
\$0070	D403050	5 53U5D4	103 C5C	00000	04030506	1.EPSEL	EFSEL EF
\$UU8U	5305040	13 UDUDD.	505 D40	36566	53050403	3E1.EF3	ILEFSEL.
20090	6566536	5 D403C	56 536	50403	65665365	EFSEI.E	-SELEFSE
REA	D POINTE	R \$C90C	TRANSFI	ER COL	DE ISO	DSR OFF	CTS OFF
WRI	TE POINT	ER \$001F	PROGRAM	I CODE	SO	DTR ON	RTS OFF
	RMto	t				·	
=EX							
=CH							
=F 50							
=							
ROGRAM	ACTUAL	PART	BLOCK			CHECK	
FLECT	POSIT	PROGRAM	ΩΔΤΔ	SEAR	∵H ATC/AF	C DATA	
						-1 2010	

(2) Display the RECEIVE DATA on the CHECK DATA screen.

The list of the data received from the host computer is displayed.

No.		Contents
1	1 The following four	(a) To position the display pointer at "0"
	methods can be used	CP [WRITE]
	to change the display pointer.	(b) To position the display pointer at the required position
		CP Pointer-value [WRITE]
		Designate the pointer value in hexadecimal number (\$ symbol is not necessary). When the designated value causes the pointer to move outside the receive buffer area, the pointer is replaced with the pointer of the last area of the receive buffer.
		(c) To return the display pointer from the current position CP ;B [WRITE]
		This replaces the currently displayed pointer with the "current display pointer - \$80 (hexadecimal)". If the above indicated command is input while the current display pointer value is 0 or smaller, it is replaced with "0".
		(d) To advance the display pointer from the current position
		CP ;N [WRITE]
		This replaces the currently displayed pointer with the "current display pointer + \$80 (hexadecimal)". If this replacement causes the pointer to move beyond the receive buffer, the pointer is replaced with the pointer of the last area in the receive buffer.
1	The display right	(a) ISO code
	character coding	CP ;I [WRITE]
	changed	(b) ASCII code
	as required.	CP ;S [WRITE]
		(c) EIA code
		CP ;E [WRITE]
		Note: The data of the code which cannot be expressed in characters are displayed in ".".
2	READ POINTER/WRITE POINTER	This displays the read-out pointer and write pointer of the received data.
3	TRANSFER CODE	The code used for transmission of data other than NC program is displayed.
4	PROGRAM CODE	The code used for transmission of NC program is displayed.

No.	Contents							
5	DSR signal (input)	Indicates the status of the data set ready (DSR) signal.						
		(a) Protocol A/B						
		If this signal is OFF, the following alarm/error is generated:						
		3218 DNC start condition 1*						
		2426 DNC-B Remote operation 305 '4218'						
		4218 DNC-B communication error 305						
		5290 DNC can not transmit 4*						
		5295 DNC transmitting 305 4218						
		* If the signal has not been turned on even once after						
		turning on power supply.						
6	DTR signal (output)	Data terminal ready						
		(a) Protocol A						
		Normally ON						
		(b) Protocol B						
		After power is turned on, the signal is turned on when the						
		buffer mode operation starts with the NC program						
		transferred in batch. The signal is turned off if an alarm						
		occurs or the NC is reset.						
7	CTS signal (input)	Clear to send						
		(a) Protocol A/B						
		If this signal is OFF, the following alarm/error is generated:						
		2426 DNC-B Remote operation 305 '4218'						
		4218 DNC-B communication error 305						
		5295 DNC transmitting 305 4218						
8	RIS signal (output)							
		Normally ON						
		(b) Protocol B						
		After power is turned on, the signal is turned on when the						
		buffer mode operation starts with the NC program						
		transferred in batch. The signal is turned off if an alarm						
		occurs or the NC is reset.						

AUTO OF	ERATION	1	DNC-PROGE	RAM	· · ·	N	2
CHECK D	ATA	÷	* DNC HIS	STORY *	[97 P#	/0//15 1 GE 51	1.10:00] 1mm
P-N0 C\$0000 C\$0002 C\$0003 C\$0004 C\$0005 C\$0006 C\$0007 C\$0008 C\$0009		TIME : 3:24, 11 : 3:24, 33 : 3:28, 80 : 3:28, 80 : 3:28, 90 : 3:33, 40 : 3:33, 40 : 3:33, 40 : 3:38, 10 : 3:38, 10	STA 5 5 5 5 5 5 5 5 5 5 5 5 5 5 5 5 5 5 5	TUS IT 2 5 2 4 2 5 2 4 2 5 2 4 2 5 2 4 2 5 2 4 2 5 2 4 2 5 3 C\$	EM CC 1 5 5 5 5 5 5 5 5 5 5 5 5 5 5 5 5 5 5	Solution 54 5 6 7 5 54 6 7 5 54 6 7 5 54 6 7 5 54 5 6 7 5 54 5 6 7 5 5 6 7 5 5 6 7 5 5 4 5 6 7 5 5 4 5 5 5 4 5 5 5 4 5 5 5 4 5 5 5 4 5 5 5 4 5 5 4 5 5 5 4 5 5 5 4 5 5 4 5 5 5 4 5 5 5 4 5 5 4 5 5 5 4 5 5 5 4 5 5 5 4 5 5 5 5 4	
=EX =:CH =F 50 =			H-M1;	d			
PROGRAM SELECT	ACTUAL POSIT.	PART PROGRAM	BLOCK DATA	SEARCH		CHECK DATA	[EXTEND]

(3) Display the DNC HISTORY page on the CHECK DATA screen.

F1F2F3F4F5F6F7F8

No.		Description
1	P-NO	The entry pointer of the DNC HISTORY saving area is
		displayed. Character C at the beginning of the pointer
		indicates that the information displayed is the contents in the
		information saving area for normal operation. Character E at
		the beginning of the pointer indicates that the information
		displayed is the contents in the information saving area for the
		occurrence of abnormality.
		The display area can be changed in the following method.
		(a) To display the information for the normal DNC HISTORY
		saving area.
		CP ;C [WRITE]
		When the power is turned on, this display area is selected.
		(b) To display the information for the abnormal DNC HISTORY saving area.
		CP ;E [WRITE]
		The display pointer can be changed in the same manner
		as explained in (1).
2	TIME	The time elapsed from the power on is displayed in
		hour:minute: second.

No.		Description						
3	STATUS	The communication operation status is displayed in a coded						
		number.						
		0: Initial state						
		1: Not ready for communication						
		2: Ready for communication						
		3: Communication in execution						
		4: Communication alarm						
		5: Communication system alarm						
4	ITEM	The DNC HISTORY kind is displayed in a coded number.						
		1: Command						
		2: Notification for correct ending of command						
		3: Notification for abnormal ending of command						
		4: Internal request						
		5: Transmission message						
		6: Line event						
5	CONTENTS	Details of DNC HISTORY kind is indicated in a coded number.						
		For details, refer to Appended Table.						
6	CURRENT POINTER	The latest information saving point for normal DNC HISTORY						
7	LAST ERROR	The latest information saving point for abnormal DNC						
	POINTER	HISTORY						

2. Run Guide Screens

(1) BUFFER MODE

PRO	G OPERATION		RUN GL	IIDE	[0]		
	RUNNING	FILE		SI	ELECT ERR	OR FILE	4.10.00
RM M E M E	BUFFER MODE AIN FILE NAM MPTY AIN PROGRAM MPTY	IE NAME	BELECTED	MAIN EMPTY MAIN EMPTY SUB F	FILE NAME PROGRAM N ILE NAME	IAME	
R R	UNNING METHO Mitcl	D		SELEC	ted Runni	NG METHO	D
=GD =GD =GD = T ME	INIT	DELETE	RENAME			RUN GUIDE	[EXTEND]
Ē	1)(F 2)(F3) (F 4)	(F 5) <u>F 6</u>)	(F 7)	F 8)

(2) SCHEDULE MODE

$\left(\right)$	PROG 0	PERAT10N		RUN GL	JIDE				
		RUNNING	FILE		SE	elect err	7/07/15 1 OR FILE	4:10:00	1
	SCHEDULE MODE SCHEDULE PROGRAM FILE OMF.SDF MAIN FILE NAME OMF-4VA-A.NIN RUNNING MAIN PROGRAM NAME EMPTY REQUEST FILE NAME OMF-4VA-A.MIN RUNNING METHOD R-Mtd			runn i Ng	MAIN FILE NAME EMPTY MAIN PROGRAM NAME EMPTY SUB FILE NAME EMPTY SELECTED RUNNING METHOD				
	=GD =GD =GD =								
	TIME	INIT	DELETE	RENAME			RUN GUIDE	[EXTEND]	
	(F 1)(F 2)(F 3)(F 4) <u>F5</u>) <u>F</u> 6)	(F7)	<u>F 8</u>	

SECTION 11 ERROR MESSAGES

5290 DNC can not transmit

An attempt is made to transfer an NC program in batch while DNC communication is impossible. [Character-string] None

[Code]

- 1-> Attempt during communication alarm
- 2-> DNC communication off
- 3-> Not in interruption mode (DNC-C2, C3)
- 4-> Not ready

5295 DNC transmitting

For data transfer by DNC-B

Communication error occurred during batch transfer of DNC data, resulting in abnormal end. [Character-string] Communication alarm number such as 4218, 4219, 4220, or 4037 [Code]

For the cause of the alarm, refer to the codes of the alarm indicated by the character-string. [Character-string] None

[Code] None

Data input/output or verification is impossible by setting the NC optional parameter bit No.1 bit0=0/No.11 bit1=0 (EIA).

This is because two or more same settings exist at the NC optional parameter (bit) No. 27, 28, 29, 30, 31, 49 (EIA code settings that represent '=','*','[',']','\$','#') or a code in the EIA code table is set.

Or, the parameter values from Nb to Tw set with the SET command from the host are abnormal. (For example, this error occurs if Tp=0 is set. For detail of each parameter, OSP-U100/U10M Special Functions Manual (No. 2) DNC-B high speed RM buffer system (SECTION 5 PARAMETER) and OSP -U100/U10M Operation Manual.

For data transfer by DNC-C

Attempt was made to execute a command that cannot be executed during communication with the host computer.

[Code]

- 1-> An attempt to execute the following commands while an NC program is being sent in the INDEPENDENT mode
 - Program selection
 - Schedule program selection
 - Number search
 - Restart
 - Number search using the cursor
 - Schedule program search
 - Sequence stop
 - Library
- [Measures to Take]

DNC-B

If a character-string appears indicating the alarm number, refer to the corresponding alarm.

If no character-string or code appears, check the communication parameter values.

DNC-C

Wait until the NC program transfer is completed.
5296 Remote operation mode

In the DNC-B mode operation, following operations are attempted while buffer operation is in progress with the buffer operation mode ON. This error occurs both with the protocol A and B.

[Character-string] None

[Code]

- 1-> Batch transfer of NC program (impossible only during buffer operation)
- 3-> Main program name or subprogram file name is designated when selecting a program in the NC program transfer method (2) during buffer operation mode.

5301 Check point setting

Illegal setting of check point

[Character-string] None

[Code]

1-> Input other than pointer value (hexadecimal) and option

SECTION 12 ALARM MESSAGE CHART

1. Communication Alarm

Communication alarm is the alarm which has occurred in communication link (protocol A) between the host computer and the OSP and does not have influence on NC operations. Once this alarm has occurred, it can be reset either by the reception of [SYN] command from the host computer or switching COMMUNICATION ON/OFF switch of the CNC from OFF to ON.

4218 DNC-B communication error

Communication error occurred during communication with the host computer using the protocol A/B.

[Character-string] None

[Code] YZZ (hexadecimal)

Y: OSP state where the error occurred

- 0-> Operation uncompleted state (DNC switch is OFF or CCP system is in abnormal state)
- 1-> OSP normal state
- 2-> Buffer operation state
- 3-> Alarm state or communication error state
- ZZ: Cause
- 01-> Check sum error occurred successively the number of times (Ne) set at the parameter. (Protocol A)
- 02-> Parity error occurred successively the number of times (Ne) set at the parameter. (Protocol A)
- 04-> No response *1
 - CTS signal OFF (Protocol A/B)
 - Response timer T1 time over (Protocol B)
- 05-> DSR signal OFF
- 06-> Invalid response or TV check error
 - NC program data codes specified at the NC optional parameter (bit) No.1 bit0/No.11 bit1 do not match the settings on the host.
 - The NC received a non-existing command. (Protocol A)
 - Transfer error was detected the number of times (Ne) set at the parameter. (Protocol A)
 - When TV check is executed, each record length is not composed of even number of characters.
 - Code conversion error
 - The NC received DC1, DC3, SYN, or NAK code while receiving the NC program. (Protocol B)
- 08-> Reception buffer overflow
- 09-> The NC received a signal from the host computer while not waiting for any response.
- 0A-> Overrun error (Protocol A/B)
- 0B-> Framing error (Protocol A/B)

[Probable Faulty Locations]

- 01-> Communication parameters or NC optional parameters are wrongly set.
- 02-> Communication cable or, in rare case, RS232C I/F is faulty.
- 03-> Hardware, software, or their parameters of the host computer are faulty.

[Measures to Take]

- 01-> Check the communication parameters related with the cable specifications (such as RS232C I/F channels used, parity, the number of stop bits, transfer code, end code, and baud rate) and the NC optional parameters. Correct improper parameters.
- 02-> Check the hardware, software, and their parameters of the host computer.
- 03-> Check if the NC is connected with the channel for DNC-B and the communication cable is in normal state.

[Related Specifications] DNC-B

4219 DNC-B message format error

Error has occurred in the reception message in the protocol A.

As for the cause of the alarm, "6. Reception of unexpected message" is reported to the host computer.

[Code]

None-> Message exceeds 72 bytes (4,000 bytes for NC program).

XXXXYY-> Mistake in the data designated by the [SET] command

XXXX-> Hexadecimal number of the wrong data (numerical data)

YY-> Byte position inside the data

[Probable Faulty Locations]

- 1) Defective software used at the host computer
- 2) Text format mismatch due to mismatch of the version of DNC-B specification at the CNC
- [Measures to Take]

Check the set data against the upper and lower limits of the text format specified in the DNC-B specification.

[Related Specifications] DNC-B

4220 DNC-B communication system abort

Unrestorable error has occurred in the protocol A/B. (This alarm is not cleared unless the power is turned on again.)

[Code]

- 10-> Overrun error has occurred.
- 11-> Framing error has occurred.

[Probable Faulty Locations]

- 1) Defective RS232C interface
- 2) Defective hardware or software of the host computer
- 3) Mismatch between the actual communication line specification (baud rate, code configuration, etc.) and the communication set by the communication parameters.

[Measures to Take]

- 1) Check the hardware and software of the host computer.
- 2) Check the following communication specifications between the actual communication line specification and the communication parameters:
 - RS232C interface channel being used
 - Parity
 - Number of stop bits
 - Transmission code
 - End code
 - Baud rate

[Related Specifications]

DNC-B

4037 CCP-CPU error

An error occurred with the CCP-CPU of the DNC-B specification.

This alarm is not cleared unless the power is turned off and then on again.

[Code]

- 1-> A memory parity error or write error occurred in the CCP memory test.
- 2-> The CCP control program cannot be loaded correctly.
- 3-> The CCP control program cannot be started correctly.
- 4-> The CCP communication channel cannot be opened correctly.

Or, the program data is written in ASCII codes though the protocol B is used (NC optional parameter bit No.11 bit1=1).

Or, data input/output or verification is impossible by setting the NC optional parameter bit No.1 bit0=0/No.11 bit1=0 (EIA). This is because two or more same settings exist at the NC optional parameter (bit) No. 27, 28, 29, 30, 31, 49 (EIA code settings that represent '=','*','[',']','\$','#') or a code in the EIA code table is set.

- 5-> A CCP self-diagnostics error occurred.
- 6--> An exception error occurred with the CCP CPU
- 7-> An error in the operation of the CCP control program occurred(task start control error).
- 8-> An error in the operation of the CCP control program occurred (real time task error).
- 9-> An error in the operation of the CCP control program occurred (time sharing task switching control error).

[Measures to Take]

- 1) NC optional parameter setting error
- 2) Defective CCP Board
- 3) CCP control software has not been stored in Memory.
- 4) Defective Memory.
- 5) Defective CCP control software.
- 6) CCP Board switch setting error.

2. NC Alarm

3218 ALARM C DNC start condition

In the DNC-B mode operation, communications cannot be done normally.

[Code]

- 1-> In the DNC-B buffer operation mode, the system is not in the ready state or the communication alarm state.
- 2-> In the DNC-B buffer operation mode, the schedule program is being executed.
- 5-> The main program buffer size is not greater than 12 m when the program runs using the buffer memory.

[Measures to Take]

Turn the DNC mode ON after pressing the RESET button.

[Related Specifications]

DNC-B

2426 ALARM B DNC-B remote operation

During DNC-B remote mode operations, an error has occurred.

[Character-string]

Communication alarm number (such as 4218, 4219, 4220 or 4037) or error number

[Code]

When a communication alarm occurs, the character-string shows the alarm number.

For the cause of the alarm, refer to the alarm D of the displayed number.

When an error occurs, the character-string shows the error number.

For the cause of the error, refer to the error of the displayed number.

[Probable Faulty Locations]

An error or a communication alarm occurred during remote operation through DNC-B. [Measures to Take]

Refer to the corresponding alarm or error.

[Related Specifications]

DNC-B

SECTION 13 APPENDIX

1. DNC HISTORY Contents Code Tables

(1) Kind = 1 (Command)

Upper Code	Lower Code
Command kind	Appended data length
1: Open	
2: Close	
3: Request for NC program	File name length
4: Call of NC program	
5: Transmission of NC program	File name length
6: Notification of NC reset	
7: Notification of NC alarm	

(2) Kind = 2 (Notification for correct ending of command)

Upper Code	Lower Code
Normal end code	Not used = 0
0: Processing completion	
1: Request for NC program	
2: Request for NC program	
transmission	
3: NC reset	
4: NC alarm	
5: NC reset/alarm	
6: Forced termination	

Upper Code	Lower Code
Abnormal end code	Error detail code
1: Temporary error	1: Check sum error
2: Permanent error	2: Parity error
3: NC reset	3:
4: NC alarm	4: No answer signal
5: Forced termination	5: DSR OFF
6: Buffer clear	6: Ineffective answer signal
7: Initialization	7: Record length overflow
8: NC reset/alarm	8: Reception buffer overflow
 –1: Command issuing sequence error 	9: Record reception in other than data reception standby state
–14: Data error	10: Overrun error
-15: Parameter error	11: Framing error
-16: Command error	-59: Appended data length range error
	-60: Command issuing error
	–62: Error in selecting with/without significant data
	-63: Command kind selection error
	-64: Command request selection error
	–91: Answer error for SAT
	–92: NC program end code selection error
	-93: EIA/ASCII conversion error
	-94: NC program length range error
	–95: NC program record end code selection error
	–96: NC program data code selection error.
	 –112: Answer signal monitoring time range error
	-128: SAT, SET command parameter selection error

(3) Kind = 3 (Notification for abnormal ending of command)

(4) Kind = 4 (Internal request)

Upper Code	Lower Code
Macro status	Macro event
0: Initial state	1: Open
1: Awaiting macro start	2: Close
2: Awaiting request for	3: Request for reception
reception	4: Request for transmission
 Awaiting request for transmission 	5: NC alarm
4: Awaiting reception of the	6: NC reset
first block	7: Time-out
*4 Only for protocol A	8: Request for forced termination
	9:
	10: Code conversion error

Upper Code	Lower Code
Message kind	Command data length
(a) Protocol A	
1: GTD	
2: DAT	DAT : Data length/16
3: RST	Others : Data length
4: ALM	
5: SAT	
6: RTD	
7: RTY	
8: SYN	
9: RDY	
10: WAT	
11: EOD	
12: ARS	
13: AAL	
14: CLB	
15: SET	
(b) Protocol B	Data length/16 (incl. DC code)
–127: DC1	
–128: DC3	
–129: Data	
–130: SYN	
–131: NAK	

(5) Kind = 5 (Transmission message)

(6) Kind = 6 (Line event)

	Upper Code	Lower Code		
Line sta	tus	Line event		
(a) Proto	Dool A			
0:	Awaiting answer signal for	1:	Reception of SYN	
	initialization	2:	Reception of RDY	
1:	Awaiting end of initialization	3:	Reception of ARS	
2:	Data being received	4:	Reception of AAL	
3:	Awaiting reception after	5:	Reception of CLB	
	transmitting DAT/RTD	6:	Reception of SET	
4:	Awaiting reception after	7:	Reception of DAT	
_	transmitting WAT	8:	Reception of EOD	
5:	Awaiting reception after	9:	Reception of WAT	
6.	Awaiting recention after	10:	Reception of GTD	
0.	transmitting ALM	11:	Reception of RTY	
7:	Awaiting reception after	12:	Reception of data other	
	transmitting RST signal		than answer	
8:	Awaiting reception after	13:	No answer back	
	transmitting EOD	14:	DSR signal off	
		15:	Transfer error 1	
		16:	Transfer error 2	
(b) Proto	pcol B			
0:	Reception being requested	1:	Normal character	
1:	Data being received	2:	Transfer error 1	
2:	Transmission being requested	3:	Transfer error 2	
3:	Data being transmitted	4:	No answer signal	
4:	Alarm signal being requested	5:	CTS signal on	
5:	Reset signal being requested	6:	CTS signal off	
		7:	DSR signal off	
		8:	Forced termination	

D. SEMI-SYNCHRONIZED OPERATION OF MACHINING CENTER WITH DIGITIZER (TD)

SECTION 1 OVERVIEW

- (1) Semi-synchronized operation is executed between the machining center and the TD (digitizer).
- (2) The semi-synchronized operation means that the digitized data obtained during scanning processing carried out by the TD digitizer is transferred to the NC machine so that machining a workpiece is machined in parallel with scanning while scanning is being conducted.
- (3) In this mode of operation, lag between the machining position and scanning position is less than one line of scanning.
- (4) In the semi-synchronization operation, delay is controlled to be less than 1K bytes in character length.
- (5) In addition, the DNC operation in the ordinary DNC-B high-speed RM buffer method is also possible. For details of the high-speed RM buffer method, refer to the instruction manual for "DNC-B high-speed RM buffer method". Operation mode can be selected between the "semi-synchronized" and the "DNC-B high-speed RM buffer" by setting proper parameter; the DNC operation panel is used in common to these two modes.
- (6) The information in this section covers only the NC related information. For the operations of the TD digitizer, refer to the instruction manual for the TD.

SECTION 2 DNC OPERATION PANEL



- (1) COMMUNICATION ON/OFF switch
 - (a) COMMUNICATION ON

Turning on power supply after placing the switch in the ON position establishes the communication mode between the TD and the host computer. The COMMUNICATION lamp lights up.

(b) COMMUNICATION OFF

Turning on power supply after placing the switch in the OFF position disconnects communication line between the TD and the host computer.

Once power supply has been turned off, changing the switch setting position has no effect and the established state cannot be changed.

Turn the switch in the OFF position once and then back to the ON position to recover the communication alarm state* from the NC. Even in this case, the previously established communication ON state is not changed.

* The state in which either of the following alarm messages (see 7-1 "Communication Alarm") is displayed.

4218 DNC-B communication error

4219 DNC-B message format error

(2) BUFFER OPERATION ON/OFF switch

(a) BUFFER OPERATION ON

By setting the switch in the ON position with the TD COMMUNICATION ON state, buffer (semi-synchronized) operation is enabled and the BUFFER OPERATION ON lamp lights up.

(b) BUFFER OPERATION OFF

Buffer (semi-synchronized) operation is disabled.

If the switch is set at the BUFFER OPERATION OFF position during semi-synchronized operation, the buffer operation enabled state is continued until the buffer operation has completed.

However, if communication alarm occurs, ON/OFF state of the buffer (semi-synchronized) operation is controlled by the setting of the BUFFER OPERATION ON/OFF switch setting at the OSP operation panel.

Note: The DNC operation panel is used in common to the DNC-B high-speed RM buffer operation and the semi-synchronized operation.

In the high-speed RM buffer operation mode (NC optional parameter (bit) No. 37, bit 2: 0), the operation panel is used for the high-speed RM buffer operation, and in the semi-synchronized operation mode (NC optional parameter (bit) No. 37, bit 2: 1), it is used for semi-synchronized operation.

SECTION 3 SEMI-SYNCHRONIZED OPERATION

Semi-synchronized operation is possible in the automatic mode while the BUFFER OPERATION ON lamp at the DNC operation panel lights up.

Semi-synchronized operation function varies depending on the NC program transmission method (with or without file name designation).

Table	1
-------	---

			Buffer Operation (DNC mode: ON, Buffer operation: ON)				Normal Memory	
Operation Type	Program Type	Operation Method	Without File Name Designation		With File Name Designation		Operation	
			Memory	TD	Memory	TD	Memory	TD
Manual	Main	A	×	×	0	×	0	×
Mode Operation	Program	В	×	×	0	×	0	×
operation		S	×	0	0	×	0	×

[Supplement] 1. " \bigcirc "indicates "operation possible" and " \times " indicates "operation impossible".

- 2. Operation Methods:
 - A Normal operation
 - B Large-volume operation
 - S Subprogram and branch function are not checked in operation method B.
- 3. In the semi-synchronized operation mode, operation using a schedule program is not possible.

The following operations are possible depending on "with" and "without" file name designation.

Without file name designation Buffer operation of NC program (digitized data)

With file name designation Buffer operation of NC program Memory mode operation based on programs stored in memory

1. Operation Procedure for Semi-synchronized Operation

(1) Selecting the Semi-synchronized Operation Mode

Both the DNC-B high-speed RM buffer operation and the semi-synchronized operation use the same DNC operation panel. Selection of the operation mode is possible by the setting for NC optional parameter (bit) data (No. 37, bit 2).

The procedure to select the semi-synchronized operation mode is indicated below:

- (a) Press the PARAMETER key.
- (b) Press function key [F7] (ITEM) to display the NC optional parameter (bit) data setting screen.
- (c) Press the page key to display the screen which contains No. 37.
- (d) Move the cursor to No. 37 bit 2 and set "1".

The setting should be "0" to select the high-speed RM buffer operation mode.

- (e) Press the backup function key to back up the data. After that turn off the power once and turn it back on again.
 - Note: Once the parameter is set, it is not necessary to repeat the same setting operation when the power is turned on or off. Setting is necessary only when the operation mode should be changed.
- (2) Starting the Semi-synchronized Operation

Operation Procedure:

- (a) Place the DNC COMMUNICATION ON/OFF switch at ON.
- (b) Turn on power supply.
- (c) Select the AUTO OPERATION mode by pressing the AUTO key.
- (d) Place the BUFFER OPERATION ON/OFF switch at ON.
- (e) Press the CYCLE START button. (PROGRAM SELECT operation is not necessary.)

2. Memory Mode Operation Using A Program Stored in Memory

The operation method is quite the same as used for normal memory mode operation.

Only the difference is the designation of the memory (MD1:) as the device name when designating the main program file name.

3. Supplements

- (1) Subprograms only registered in the library program can be used.
- (2) Size of part program has no limit for program transfer.
- (3) Only main programs can be transmitted.

If subprograms and/or other main programs continue following the M02 code of the main program, do not forget to set the "M02 reset output used" state by the parameter.

- (4) At the instant the CYCLE START button is pressed, transfer of the NC program (digitized data) begins (GTD sent for protocol A) and then the program execution begins.
- (5) Change of parameter setting for file name used or not-used status becomes active when power supply is turned off and then on again.
- (6) Even in the semi-synchronized operation mode, intervention of manual operation is possible as usual.
- (7) After the buffer operation mode is turned off, it is always necessary to select a program. Unless a program is selected, cycle start is impossible.
- (8) Regardless of the on or off state of the buffer operation mode, the state does not change even if the setting for the BUFFER OPERATION ON/OFF switch. This is also applied when a program is being transmitted for buffer operation.

If the buffer operation mode is changed using a command, in the "buffer operation mode on" always causes an alarm regardless of the current mode.

- (9) Batch transfer of NC programs using the DNC function is not possible. (Ignored, "No Command" message is displayed.)
- (10) The RS command (restart search) cannot be used. (Ignored, "No Command" message is displayed.)
- (11) Schedule program selection is not possible. (Ignored, "No Command" message is displayed.)
- (12) After the execution of MDI mode operation, semi-synchronized operation is not possible unless the NC is once reset.

Interruption by MDI mode operation during buffer mode operation may be executed in the same manner as normal operation intervention.

(13) Automatic program selection when power is turned on:

BUFFER OPERATION ONProgram selection is not executed.BUFFER OPERATION OFFProgram selection is executed.

(14) In the semi-synchronized operation mode, three blocks of commands are buffered.In the cutter radius compensation mode, eight blocks of commands are buffered.

SECTION 4 PARAMETERS

1. Parameters which cannot be Set at OSP

The parameters indicated below cannot be set at the NC and setting is possible only by using the SET command from the TD.

(Parameters in the table below, codes after conversion (45th - 48th bytes of SET command))

Parameter	Function	Unit	Range	Initial Value when Power is Turned On
Nb	Minimum buffer area of CNC when transmitting [GTD]	bytes	1 - 60	30
No	Maximum overrun amount in reception	bytes	2 - 2000	50
Ne	Number of retries allowed when transmission error is detected	times	0 - 100	10
Тр	[SAT] transmission polling time for DNC, or response time-out duration by CNC for host computer	S	1 - 999	5
То	Response time-out duration by host computer for CNC	S	1 - 999	20
Ti	Minimum time interval between transmission bytes	ms	0 - 10	10
Tx	Minimum switching time from reception to transmission	ms	0 - 100	100
Tw*	Standby time for reception of [WAT] used by both of CNC and host computer; upper limit value is Tp or To, whichever smaller.	S	0 - Tp 0 - To (Whichever smaller)	5

Table 2

*Tw: Communication is executed in the following manner when "Tw" value exceeding the specified range is set by the host computer (TD) using the SET command.

(1) When Tw value > Tp value

In the remote buffer mode operation or batch transmission mode, the OSP sends the GTD command to the host computer (TD) after waiting for "Tw" when it received the WAT command from the host computer (TD). The SAT command is not sent out during this period.

(2) When Tw value > To value

If program data to be transmitted is not ready in the NC program transmission from the OSP to the host computer operation, the OSP sends the WAT command after and elapse of "Tp". If the host computer sends the GTD command in response to this after waiting for "Tw", the OSP regards that, after an elapse of "To", there is no answer to the WAT command. In this case the communication error occurs.

2. Parameters which can be Set at OSP

The parameters indicated below can be set at the NC.

Some parameters are required to be changed frequently during operation while some are not changed once they have been set. The OSP differentiates these two types of parameters to allow easy setting of parameters. The former is assigned to NC optional parameter (long word), (word) and (bit), and the latter to the communication parameter (dedicated screen).

- (1) NC Optional Parameter (Bit)
 - (a) Bit 0 of No. 1 /Bit 1 of No. 11

Coding system of NC programs

Bit 0 of No. 1	Bit 1 of No. 11	
	1	ASCII
1	0	ISO
0	0	EIA

Initial setting: 1

(b) Bit 2 of No. 1

TV check, creation designation

This parameter determines whether or not the TV check, creation is made for the NC program (digitized data) to be received or transmitted - a space is inserted before LF (or CR LF) so that a total number of characters in a record including the LF (or CR LF) will be even, or whether the record contains an even number of characters is checked.

- 1: TV check, creation is executed.
- 0: TV check, creation is not executed.

Initial setting: 0

(c) Bit 3 of No. 11

End of record code of NC program data (digitized data)

1: LF 0: CR LF Initial setting: 0

(d) Bit 3 of No. 37

Selection of operation mode - DNC-B high-speed RM buffer method or semi-synchronized operation mode

- 1: Semi-synchronized operation
- 0: DNC-B high-speed buffer
- *Note:* After changing the setting, it is necessary to turn off the power once and then turn it back on again to make the new setting effective.

(e) Bit 0 - bit 7 of No. 43 and No. 44

Data at the 41st to 44th bytes of [SAT] (CNC \rightarrow TD) or [SET] (TD \rightarrow CNC) command.

Bit 0 of No. 44

- 1: Buffer is cleared when NC is reset.
- 0: Buffer is not cleared when NC is reset.

Initial setting: 0

[Supplement] Other bits have not been defined.

(2) Semi-synchronized Operation Communication Parameters

Communication parameters are displayed and set in the parameter setting mode.

Note that the communication parameter screen cannot be displayed in normal operation method. (screen selection using the ITEM function keys.) Follow the steps below to display the communication parameter screen.

- (a) Select the PARAMETER SET mode by pressing the PARAMETER key.
- (b) Display the COMMUNICATION PARAMETER screen.
- (c) Press the page key to display the COMMUNICATION PARAMETER screen.
- (d) Set the required parameters.

This setting is possible by using function key (SELECT). Each time this key is pressed, contents are automatically changed within a setting range.

- (e) Press function key (BACKUP). This operation is always required when data is set or changed.
- (f) Turn off power supply and then turn it back on.

The communication parameters become effective when power reapplied.

(3) Description of Communication Parameters

_	PARAMETER SET				
	* COMMUNECATION	PARAMETE	R *	7/07/15 1	4:10:00
	1. COMMUNICATION DEVICE 2. BAUD RATE 3. RS232C STOP BITS 4. RS232C PARITY 5. TRANSMIT CODE 6. MESSAGE END CODE 7. PROTOCOL A/B 8. NC PROGRAM TRANSMIT METHOD 9. LAST 'DC3' CODE ENABLE 10. SYN/NAK ENABLE 11. MO2 RESET OUTPUT ENABLE X ACT POSIT (WORK) 799. 999	Y 799. 999	-140	Z 20, 000	CHA: 2400 1 EVEN ISO ETX A A YES YES YES
	=CMPS =				
	SET SELECT	BACKUP	ITEM †	ITEM↓	

F1F2F3F4F5F6F7FB

No.	Name	Description
1	COMMUNICATION DEVICE	Designation of device name used for communication through RS232C interface
		Select from CHA: and CHB:. Set "CHB:".
		Initial setting: CHA:
2	BAUD RATE (BPS)	Designation of baud rate
		Select from 110, 150, 300, 600, 1200, 2400, 4800, 9600, 19200, 38400, 89300.
		Initial setting: 2400
3	RS232C STOP BITS	Designation of number of stop bits
		Select from 1 and 2.
		Initial setting: 1
4	RS232C PARITY	Designation of parity added at asynchronized communication
		Select from NON, EVEN and ODD.
		Initial setting: EVEN
5	TRANSMIT CODE	Designation of transmission code (excluding NC programs)
		Select from ISO and ASCII.
		Initial setting: ISO
		[Supplement] The transmission code is not used for protocol B.

No.	Name	Description
6	MESSAGE END	Designation of message end code
	CODE	Select from ETX and CR.
		Initial setting: ETX
		[Supplement] The end code is not used in protocol B.
7	PROTOCOL A/B	Protocol A only (setting is not possible.)
		Initial setting: A
8	NC PROGRAM	Designation of "file name used" or "file name not used" state
	TRANSMIT	for NC program (digitized data) transmission
METHOD		Select from A (not used) and B (used).
		Initial setting: A
9	LAST 'DC3' CODE	Setting is not possible.
	ENABLE	Initial setting: YES
		[Supplement] Not used
10	SYN/NAK ENABLE	Setting is not possible.
		[Supplement] Not used
11	M02 RESET OUTPUT	Designation whether or not reset signal ([RST] command for
	ENABLE	protocol A, [SYN] code for protocol B) is transmitted to the
		host computer (TD) upon execution of M02 and M30.
		Select from YES (transmitted) and NO (not transmitted).
		Initial setting: YES

SECTION 5 SCREEN DISPLAY

1. Check Screens

(1) Display the communication status and parameters page on the CHECK DATA screen.

AUTO O	PERATION					N	0
CHECK	DATA		DNC (B)	PAGE	67 67	14:10:00 1mm
HOST C CNC ST PARAME CONVER	PU STATU ATUS TER No Ne Tp To Ti Tx Tw RT CODE RTED CODE	s	0 0 2000 by 50 by 10 tir 5 sec 10 msc 5 sec A0 A0	te nes c c c c ec c			
=							
PROGRAM	ACTUAL	PART PROCEDAN	BLOCK	SEARCH		CHECK	

F1F2F3F4F5F6F7FB

No.	Name	Description
1	HOST CPU STATUS	Displays the second byte data in [SET] command for TD transmission.
2	CNC STATUS	Displays the second byte data in [SAT] command for NC transmission.
		 0: Not ready status 1: CNC in normal status 2: Buffer operation status 3: Alarm status
3	PARAMETER Nb - Tw	Displays contents of [SET] and [SAT] commands from 9th to 40th bytes.
4	CONVERT CODE/ CONVERTED CODE	Displays code conversion designation for NC programs (digitize data).

(2) Display the RECEIVE DATA on the CHECK DATA screen.

AUTO OPI	ERATION					N	3
CHECK D	ATA	*	DNC REC	EIVE DA	ATA * 🛄	PAGE 50	4-10-00] 1mm
\$0000 \$0010 \$0020 \$0030 \$0040 \$0050 \$0060 \$0070 \$0080 \$0090	C6445359 D430B1A0 B2303030 30303C5C6 53C5D403 C5C653C5 D403C5C6 53C5D403 C5C653C5	4E03B7B A0A0A0A 4130303 B430303 53C5D40 C5C6530 D403C50 C5C6530 D403C50 D403C50	12 D244 10 A030 10 3530 10 3530 13 C5C6 15 D403 16 53C5 17 C5C6 18 C5C6 19 C5C6 10 C5C6 10 C5C6 10 C5C6 10 C5C6 10 C5C6 10 C5C5 10 C5C5	5903 4 B744 3 30B1 E 3030 3 53C5 D C5C6 5 D403 C 53C5 D C5C6 5 D403 C	144153C5 30303033 34303030 30303030 30303030 30303030	FDSYN. 72 T01 2000A000 0.064000 0.EFSET. SET.EFSE EFSET.EF T.EFSET. SET.EFSE EFSET.EF	RDY. DASE 07D0003 50014000 50000000 EFSET. EF T. EFSET. SET. EFSE EFSET. EF T. EFSET. SET. EFSE
REAL WRI [*] =EX =CH =F 50	d pointer Te pointer R-Mito	\$C90C \$001F	TRANSFE PROGRAM	r Code Code	E 1S0 1S0	DSR OFF DTR ON	CTS OFF RTS OFF
= PROGRAM SELECT	ACTUAL PA POSIT. PI	ART Rogram	BLOCK DATA	SEARCH		CHECK DATA	[EXTEND]

The list of the data received from the TD is displayed.

No.		Contents
1	1 The following four methods can be used to change the display	(a) To position the display pointer at "0"
		CP [WRITE]
	pointer.	(b) To position the display pointer at the required position
		CP Pointer-value [WRITE]
		Designate the pointer value in hexadecimal number (\$ symbol is not necessary). When the designated value causes the pointer to move outside the receive buffer area, the pointer is replaced with the pointer of the last area of the receive buffer.
		(c) To return the display pointer from the current position
		CP ;B [WRITE]
		This replaces the currently displayed pointer with the "current display pointer - \$80 (hexadecimal)". If the above indicated command is input while the current display pointer value is 0 or smaller, it is replaced with "0".
		(d) To advance the display pointer from the current position
		CP ;N [WRITE]
		This replaces the currently displayed pointer with the "current display pointer + \$80 (hexadecimal)". If this replacement causes the pointer to move beyond the receive buffer, the pointer is replaced with the pointer of the last area in the receive buffer.
1	The display right	(a) ISO code
	character coding system can be changed as required.	CP ;I [WRITE] When the power is turned on, this coding system is selected.
		(b) ASCII code
		CP ;S [WRITE]
		(c) EIA code CP ;E [WRITE]
		Note: The data of the code which cannot be expressed in characters are displayed in ".".
2	READ POINTER/WRITE POINTER	This displays the read-out pointer and write pointer of the received data.
3	TRANSFER CODE	The code used for transmission of data other than NC program is displayed.
4	PROGRAM CODE	The code used for transmission of NC program is displayed.

No.	Contents		
5	DSR signal (input)	Indicates the status of the data set ready (DSR) signal.	
		(a) Protocol A/B	
		If this signal is OFF, the following alarm/error is generated:	
		3218 DNC start condition 1* 2426 DNC-B Bemote operation 305 '4218'	
		4218 DNC-B communication error 305	
		5290 DNC can not transmit 4*	
		* If the signal has not been turned on even once after	
		turning on power supply.	
6	DTR signal (output)	Data terminal ready	
	U U U U U U U U U U U U U U U U U U U	(a) Protocol A	
		Normally ON	
		(b) Protocol B	
		After power is turned on, the signal is turned on when the	
		buffer mode operation starts with the NC program	
		transferred in batch. The signal is turned off if an alarm	
		occurs or the NC is reset.	
7	CTS signal (input)	Clear to send	
		(a) Protocol A/B	
		If this signal is OFF, the following alarm/error is generated:	
		4218 DNC-B communication error 305	
		5295 DNC transmitting 305 '4218'	
8	RTS signal (output)	Request to send	
		(a) Protocol A	
		Normally ON	
		(b) Protocol B	
		After power is turned on, the signal is turned on when the	
		butter mode operation starts with the NC program	
		transferred in batch. The signal is turned off if an alarm	
		occurs of the INU IS reset.	

(3) Display the DNC HISTORY on the CHECK DATA screen.

No.	Description		
1	P-NO	The entry pointer of the DNC HISTORY saving area is	
		displayed. Character C at the beginning of the pointer	
		indicates that the information displayed is the contents in the	
		information saving area for normal operation. Character E at	
		the beginning of the pointer indicates that the information	
		displayed is the contents in the information saving area for the	
		occurrence of abnormality.	
		The display area can be changed in the following method.	
		 (a) To display the information for the normal DNC HISTORY saving area. 	
		CP ;C [WRITE]	
		When the power is turned on, this display area is selected.	
		(b) To display the information for the abnormal DNC HISTORY	
		saving area.	
		CP ;E [WRITE]	
		The display pointer can be changed in the same manner	
		as explained in 5-1, (2).	
2		The time elapsed from the power on is displayed in	
		hour:minute: second.	
3	STATUS	The communication operation status is displayed in a coded	
		number.	
		0: Initial state	
		1: Not ready for communication	
		2: Ready for communication	
		3: Communication in execution	
		4. Communication alarm	
		The DNC HISTORY kind is displayed in a coded number	
1.		1: Command	
		2: Notification for correct ending of command	
		3: Notification for abnormal ending of command	
		4: Internal request	
		5: Transmission message	
		6: Line event	
5	CONTENTS	Details of DNC HISTORY kind is indicated in a coded number.	
		For details, refer to Appended Table.	
6	CURRENT POINTER	The latest information saving point for normal DNC HISTORY	
7	LAST ERROR	The latest information saving point for abnormal DNC	
1	POINTER	HISTORY	

2. Run Guide Screens



SECTION 6 ERROR MESSAGES

5295 DNC transmitting

For data transfer by DNC-B

Communication error occurred during batch transfer of DNC data, resulting in abnormal end. [Character-string] Communication alarm number such as 4218, 4219, 4220, or 4037

[Code]

For the cause of the alarm, refer to the codes of the alarm indicated by the character-string. [Character-string] None

[Code] None

Data input/output or verification is impossible by setting the NC optional parameter bit No.1 bit0=0/No.11 bit1=0 (EIA).

This is because two or more same settings exist at the NC optional parameter (bit) No. 27, 28, 29, 30, 31, 49 (EIA code settings that represent '=','*','[',']','\$','#') or a code in the EIA code table is set.

Or, the parameter values from Nb to Tw set with the SET command from the host are abnormal. (For example, this error occurs if Tp=0 is set. For detail of each parameter, OSP-U100/U10M Special Functions Manual (No. 2) DNC-B high speed RM buffer system (SECTION 5 PARAMETER) and OSP -U100/U10M Operation Manual.

For data transfer by DNC-C

Attempt was made to execute a command that cannot be executed during communication with the host computer.

[Code]

- 1-> An attempt to execute the following commands while an NC program is being sent in the INDEPENDENT mode
 - Program selection
 - Schedule program selection
 - Number search
 - Restart
 - Number search using the cursor
 - Schedule program search
 - Sequence stop
 - Library

[Measures to Take]

DNC-B

If a character-string appears indicating the alarm number, refer to the corresponding alarm. If no character-string or code appears, check the communication parameter values.

DNC-C

Wait until the NC program transfer is completed.

5301 Check point setting

Illegal setting of check point

[Character-string] None

[Code]

1-> Input other than pointer value (hexadecimal) and option

SECTION 7 ALARM MESSAGE CHART

1. Communication Alarm

Communication alarm is the alarm which has occurred in communication link (protocol A) between the host computer (TD) and the OSP and does not have influence on NC operations. Once this alarm has occurred, it can be reset either by the reception of [SYN] command from the host computer (TD) or switching COMMUNICATION ON/OFF switch of the CNC from OFF to ON.

4218 DNC-B communication error

Communication error occurred during communication with the host computer using the protocol A/B.

[Character-string] None

[Code] YZZ (hexadecimal)

Y: OSP state where the error occurred

- 0-> Operation uncompleted state (DNC switch is OFF or CCP system is in abnormal state)
- 1-> OSP normal state
- 2-> Buffer operation state
- 3-> Alarm state or communication error state
- ZZ: Cause
- 01-> Check sum error occurred successively the number of times (Ne) set at the parameter. (Protocol A)
- 02-> Parity error occurred successively the number of times (Ne) set at the parameter. (Protocol A)
- 04-> No response *1
 - CTS signal OFF (Protocol A/B)
 - Response timer T1 time over (Protocol B)
- 05-> DSR signal OFF
- 06-> Invalid response or TV check error
 - NC program data codes specified at the NC optional parameter (bit) No.1 bit0/No.11 bit1 do not match the settings on the host.
 - The NC received a non-existing command. (Protocol A)
 - Transfer error was detected the number of times (Ne) set at the parameter. (Protocol A)
 - When TV check is executed, each record length is not composed of even number of characters.
 - Code conversion error
 - The NC received DC1, DC3, SYN, or NAK code while receiving the NC program. (Protocol B)
- 08-> Reception buffer overflow
- 09-> The NC received a signal from the host computer while not waiting for any response.
- 0A-> Overrun error (Protocol A/B)
- 0B-> Framing error (Protocol A/B)

[Probable Faulty Locations]

- 01-> Communication parameters or NC optional parameters are wrongly set.
- 02-> Communication cable or, in rare case, RS232C I/F is faulty.
- 03-> Hardware, software, or their parameters of the host computer are faulty.

[Measures to Take]

- 01-> Check the communication parameters related with the cable specifications (such as RS232C I/F channels used, parity, the number of stop bits, transfer code, end code, and baud rate) and the NC optional parameters. Correct improper parameters.
- 02-> Check the hardware, software, and their parameters of the host computer.
- 03-> Check if the NC is connected with the channel for DNC-B and the communication cable is in normal state.

[Related Specifications] DNC-B

4219 DNC-B message format error

Error has occurred in the reception message in the protocol A.

As for the cause of the alarm, "6. Reception of unexpected message" is reported to the host computer.

[Code]

None-> Message exceeds 72 bytes (4,000 bytes for NC program).

XXXXYY-> Mistake in the data designated by the [SET] command

XXXX-> Hexadecimal number of the wrong data (numerical data)

YY-> Byte position inside the data

[Probable Faulty Locations]

- 1) Defective software used at the host computer
- 2) Text format mismatch due to mismatch of the version of DNC-B specification at the CNC
- [Measures to Take]

Check the set data against the upper and lower limits of the text format specified in the DNC-B specification.

[Related Specifications] DNC-B

4220 DNC-B communication system abort

Unrestorable error has occurred in the protocol A/B. (This alarm is not cleared unless the power is turned on again.)

[Code]

- 10-> Overrun error has occurred.
- 11-> Framing error has occurred.

[Probable Faulty Locations]

- 1) Defective RS232C interface
- 2) Defective hardware or software of the host computer
- 3) Mismatch between the actual communication line specification (baud rate, code configuration, etc.) and the communication set by the communication parameters.

[Measures to Take]

- 1) Check the hardware and software of the host computer.
- 2) Check the following communication specifications between the actual communication line specification and the communication parameters:
 - RS232C interface channel being used
 - Parity
 - Number of stop bits
 - Transmission code
 - End code
 - Baud rate

[Related Specifications]

DNC-B

4037 CCP-CPU error

An error occurred with the CCP-CPU of the DNC-B specification.

This alarm is not cleared unless the power is turned off and then on again.

[Code]

- 1-> A memory parity error or write error occurred in the CCP memory test.
- 2-> The CCP control program cannot be loaded correctly.
- 3-> The CCP control program cannot be started correctly.
- 4-> The CCP communication channel cannot be opened correctly.

Or, the program data is written in ASCII codes though the protocol B is used (NC optional parameter bit No.11 bit1=1).

Or, data input/output or verification is impossible by setting the NC optional parameter bit No.1 bit0=0/No.11 bit1=0 (EIA). This is because two or more same settings exist at the NC optional parameter (bit) No. 27, 28, 29, 30, 31, 49 (EIA code settings that represent '=','*','[',']','\$','#') or a code in the EIA code table is set.

- 5-> A CCP self-diagnostics error occurred.
- 6--> An exception error occurred with the CCP CPU
- 7-> An error in the operation of the CCP control program occurred(task start control error).
- 8-> An error in the operation of the CCP control program occurred (real time task error).
- 9-> An error in the operation of the CCP control program occurred (time sharing task switching control error).

[Measures to Take]

- 1) NC optional parameter setting error
- 2) Defective CCP Board
- 3) CCP control software has not been stored in Memory.
- 4) Defective Memory.
- 5) Defective CCP control software.
- 6) CCP Board switch setting error.

2. NC Alarm

3218 ALARM C DNC start condition

In the DNC-B mode operation, communications cannot be done normally.

[Code]

- 1-> In the DNC-B buffer operation mode, the system is not in the ready state or the communication alarm state.
- 2-> In the DNC-B buffer operation mode, the schedule program is being executed.
- 5-> The main program buffer size is not greater than 12 m when the program runs using the buffer memory.

[Measures to Take]

Turn the DNC mode ON after pressing the RESET button.

[Related Specifications]

DNC-B

2426 ALARM B DNC-B remote operation

During DNC-B remote mode operations, an error has occurred.

[Character-string]

Communication alarm number (such as 4218, 4219, 4220 or 4037) or error number

[Code]

When a communication alarm occurs, the character-string shows the alarm number.

For the cause of the alarm, refer to the alarm D of the displayed number.

When an error occurs, the character-string shows the error number.

For the cause of the error, refer to the error of the displayed number.

[Probable Faulty Locations]

An error or a communication alarm occurred during remote operation through DNC-B. [Measures to Take]

Refer to the corresponding alarm or error.

[Related Specifications]

DNC-B

SECTION 8 APPENDIX

1. DNC HISTORY Contents Code Tables

(1) Kind=1 (Command)

Upper Code	Lower Code
Command kind	Apponded data longth
	Appended data length
1: Open	
2: Close	
3: Request for NC program	File name length
4: Call of NC program	
5: Transmission of NC program	File name length
6: Notification of NC reset	
7: Notification of NC alarm	

(2) Kind = 2 (Notification for correct ending of command)

Upper Code	Lower Code
Normal end code	Not used = 0
0: Processing completion	
1: Request for NC program	
2: Request for NC program	
transmission	
3: NC reset	
4: NC alarm	
5: NC reset/alarm	
6: Forced termination	

Upper Code	Lower Code
Abnormal end code	Error detail code
1: Temporary error	1: Check sum error
2: Permanent error	2: Parity error
3: NC reset	3:
4: NC alarm	4: No answer signal
5: Forced termination	5: DSR OFF
6: Buffer clear	6: Ineffective answer signal
7: Initialization	7: Record length overflow
8: NC reset/alarm	8: Reception buffer overflow
 -1: Command issuing sequence error 	9: Record reception in other than data reception standby state
-14: Data error	10: Overrun error
-15: Parameter error	11: Framing error
-16: Command error	-59: Appended data length range error
	-60: Command issuing error
	–62: Error in selecting with/without significant data
	-63: Command kind selection error
	-64: Command request selection error
	–91: Answer error for SAT
	–92: NC program end code selection error
	-93: EIA/ASCII conversion error
	-94: NC program length range error
	–95: NC program record end code selection error
	–96: NC program data code selection error.
	 –112: Answer signal monitoring time range error
	–128: SAT, SET command parameter selection error

(3) Kind = 3 (Notification for abnormal ending of command)

(4) Kind = 4 (Internal request)

Upper Code	Lower Code
Macro status	Macro event
0: Initial state	1: Open
1: Awaiting macro start	2: Close
2: Awaiting request for	3: Request for reception
reception	4: Request for transmission
 Awaiting request for transmission 	5: NC alarm
4: Awaiting reception of the	6: NC reset
first block	7: Time-out
*4 Only for protocol A	8: Request for forced termination
	9:
	10: Code conversion error
(5) Kind = 5 (Transmission message)

Upper Code	Lower Code
Message kind	Command data length
(a) Protocol A	
9: GTD	
10: DAT	DAT : Data length/16
11: RST	Others : Data length
12: ALM	
13: SAT	
14: RTD	
15: RTY	
16: SYN	
17: RDY	
18: WAT	
19: EOD	
20: ARS	
21: AAL	
22: CLB	
23: SET	
(b) Protocol B	Data length/16 (incl. DC code)
–127: DC1	
–128: DC3	
–129: Data	
–130: SYN	
– 13 1: NAK	

(6) Kind = 6 (Line event)

Upper Code		Lower Code		
Line status		Line event		
(a) Proto	pcol A			
0:	Awaiting answer signal for	1:	Reception of SYN	
	initialization	2:	Reception of RDY	
1:	Awaiting end of initialization	3:	Reception of ARS	
2:	Data being received	4:	Reception of AAL	
3:	Awaiting reception after	5:	Reception of CLB	
	transmitting DAT/RTD	6:	Reception of SET	
4:	Awaiting reception after	7:	Reception of DAT	
-		8:	Reception of EOD	
5:	Awaiting reception aπer	9:	Reception of WAT	
6.		10:	Reception of GTD	
0.	transmitting ALM	11:	Reception of RTY	
7:	Awaiting reception after	12:	Reception of data other	
	transmitting RST signal		than answer	
8:	Awaiting reception after	13:	No answer back	
	transmitting EOD	14:	DSR signal off	
		15:	Transfer error 1	
		16:	Transfer error 2	
(b) Proto	pcol B			
0:	Reception being requested	1:	Normal character	
1:	Data being received	2:	Transfer error 1	
2:	Transmission being requested	3:	Transfer error 2	
3:	Data being transmitted	4:	No answer signal	
4:	Alarm signal being requested	5:	CTS signal on	
5:	Reset signal being requested	6:	CTS signal off	
		7:	DSR signal off	
		8:	Forced termination	

E. WARM-UP FUNCTION

SECTION 1 OVERVIEW

The warm-up function described in this section allows to warm up the machine through the execution of the predetermined program (warm-up program) by automatically turning on power supply to the machine in accordance with the setting of the calendar timer.

That is, when power supply to the machine is turned on in accordance with the setting of the calendar timer, the OSP automatically selects and executes the warm-up program prepared by the user.

NOTICE

: This section deals with the information necessary to effectively use the warm-up function incorporated in the OSP. Since the operation procedures for the calendar timer are not discussed in this section, refer to the instruction manual for the calendar timer, provided separately.

Please note that the operation procedure for a calendar timer will vary depending on the manufacturer and the type. Therefore, it must be confirmed that the manual provided with the machine is the one for the calendar timer actually installed.

SECTION 2 REGISTRATION OF WARM-UP PROGRAM

Registration of a warm-up program in the user area of memory (MD1:) by the file name of "W.WAF" is necessary to activate the warm-up function.

1. Registering Warm-up Program

The procedure to register the warm-up program is identical to the procedure to register part programs in the memory, which is explained in SECTION 2, "PROGRAM OPERATION" of III. DATA OPERATION in the Operation Manual.

2. Format

The warm-up program format is identical to the program format for main programs. The program should begin with the program name, beginning with character O, and end with code M02 or M30.

Example:

O9999 S200 M3 VC128=50 G0 Z500 NA1 X-200 Y-200 X200 Y200 G4 F5 VC128=VC128-1 IF [VC128 GT 0] NA1 M5 M2

SECTION 3 OPERATIONS

Machine warm-up is carried out in the order described below:

(1) Set the cycle on and cycle off time on the calendar timer in accordance with the operation schedule.

Duration between cycle on to cycle off should be more than three minutes. For cycle power off, refer to items (2) and (3) in SECTION 5. "SUPPLEMENTS".

- (2) Power supply to the NC is automatically turned on in accordance with the setting of the calendar timer.
- (3) After an elapse of 90 seconds after power supply to the control has been turned on, operation end alarming buzzer will sound for the duration set by parameter. After the cycle end alarming buzzer has stopped, the control is automatically placed in the automatic mode and the warm-up program* is automatically selected and executed.

After the completion of the warm-up program (the execution of M02 or M30 in the warm-up program), or when the RESET button on the operation panel is pressed, machine stops and the program having been selected before the power has been turned off is automatically selected**.

* warm-up program:

Warm-up program should have the file name of W.WAF and program name may be designated as required.

** automatic program selection:

Whether the program is to be selected or not after the completion of the warm-up program is determined by the setting of parameter.

SECTION 4 TIMING CHART

Calendar timer output	
Control power	
Operation end alarming buzzer	
Automatic selection of warm-up program	
Automatic execution of warm-up program	
Reset by execution of M or M30, or manual press of RESET button	02 sing
Automatic selection of program having been se before power shut off	lected

Fig. 1 Timing Chart for Warm-up Program Execution

SECTION 5 SUPPLEMENTS

- (1) After the completion of the warm-up program, the program selected through the operation panel is automatically selected. Therefore, if the warm-up program (file name W.WAF) is selected through the operation panel before turning off power supply, W.WAF is selected again after the completion of warm-up program.
- (2) If the calendar timer output is turned off while the warm-up program is being executed, the warmup program is not interrupted and executed up to the end of the program.
- (3) If the file W.WAF is not existent in the memory when power is automatically turned on by the calendar timer function, an alarm will occur. See 6. "ALARM".
- (4) For the period from automatic power on to the cycle start of the warm-up program, controls (mode selection keys, function keys, alphanumeric keys) on the NC operation panel, CYCLE START button and the external cycle start button are all disable. However, if the selection of the warm-up program is impossible, all controls are operative.
- (5) If the operator presses the RESET button while the warm-up program is being executed, the program having been selected before power shutoff is automatically selected. Note that when "0" is designated at the optional parameter (bit) No. 4, the automatic program selection is ineffective.
- (6) The warm-up function is ignored even if the calendar timer is actuated, if its actuation is after the display of the ACTIVE POSITION data display with power supply to the OSP turned on by the operator.
- (7) The end of program command in the warm-up program is ignored by the automatic power shutoff function. That is, power supply will not be turned off even when the M02 or M30 in the warm-up program is executed.



: If the operator turns off control power while the calendar timer output is on and the control power is turned on again, the warm-up program is executed again.

SECTION 6 ALARM

The following alarm is added when the warm-up function is selected.

3227 ALARM C Warming up

Program selection for the warmup program cannot be carried out normally.

[Code] XX

- XX=1-> The warmup program (W.WAF) is not registered in the bubble memory (BB1:).
- XX=2-> Although the warmup program is registered in the bubble memory, selection of it is impossible.
- XX=20-> The operation door is open.

XX=21-> The loader is not at its retract end.

[Probable Faulty Locations]

Contents of the warmup program

Program Example:

OXXX

:

RTS WRITE (must be M02 or M30.)

[Measures to Take]

Check the warmup program.

SECTION 7 PARAMETER SETTING

For the activation of the warm-up program, setting of the following machine user parameter is necessary.

5. Buzzer						
Parameter No.	Parameter Name	Function	Effective Commands	Initial Value	Setting Range	Setting Unit
7	Operation end buzzer timer	Time duration in which the operation end alarming buzzer sounds	SET ADD	10	0 to 600	(s)

Operation Procedure:

- (1) Select the PARAMETER SET mode by pressing the PARAMETER key.
- (2) Press function key [F6] (ITEM ↑) or [F7] (ITEM ↓) until the CRT displays the *MC USER PARAMETER (5. Buzzer) * page.
- (3) Locate the cursor on parameter No. 7 using the cursor control keys.



- (4) After pressing function key [F1] (SET), key in the required duration through the keyboard.
- (5) Press the [WRITE] key.

Check the data of parameter No. 7.

F. SPINDLE OVERLOAD DETECTION FUNCTION

SECTION 1 OVERVIEW

- (1) The meter relay is used for the spindle motor load meter to trigger an alarm signal if the spindle load exceeds a preset level. This overload detection function detects excessively heavy cutting condition or collision of the cutting with the workpiece due to operation error immediately. Upon detection, the control triggers level A alarm and stops machine operation.
- (2) The load meter with the meter relay asillustrated at the right is connected to the motordrive circuit.
- (3) Set the maximum load setting pointer at the required load value position by rotating the knob. When the motor starts, the meter pointer moves according to the motor load; if the pointer swings beyond the setting pointer, the relay contact closes. This is detected as level A alarm and stops all axes rotation.



SECTION 2 OPERATIONS

(1) This function is made effective and ineffective by designating the following M codes in the automatic or MDI mode.

M142 Spindle overload detection function ineffective

- M143 Spindle overload detection function effective.
- (2) When power supply to the control is turned on or when the control is reset, M142 is effective.

Therefore, it is necessary to designate M143 to make the spindle overload detection function effective.

(3) In the manual operation mode, the spindle overload detection function is always ineffective and thus the related alarm does not occur.

SECTION 3 SPINDLE DRIVE MOTOR STARTING CURRENT

- (1) When the spindle motor starts, the starting current flows to cause the load meter pointer to swing beyond the max. load setting pointer. The control ignores this overcurrent status so that alarm will not occur. For this purpose, the control establishes the immune period as Indicated below.
 - (a) The control determines that the spindle is rotating at a fixed speed when an actual spindle speed is 75 125 percent of the programmed speed.
 - (b) Spindle overload condition is not checked until the spindle speed reaches the fixed speed, i.e., within duration T1 from the spindle start.
 - (c) However, there are cases where the spindle load meter pointer still lies at the right side of the preset point even after the spindle speed reaches the fixed speed because the response of the spindle load meter pointer is slow. This also might occur if the setting value is relatively small. Thus the control does not check the spindle overload condition for a period T2 until the spindle rotation reaches the stationary state after the fixed speed condition has been reached. This period T2 is set with the parameter (Machine user parameter "4. Spindle overload time" [6. spindle].



Fig. 1

(2) The CRT screen display for the spindle overload timer is shown below. Time setting range is 0
 - 60 seconds in increments of 1 second. Setting is made using the [F1] (SET) or [F2] (ADD).
 The Initial value is one second (1).

	PARAMETER SET			
	* MC USER PAI	RAMETER *	97/07/15	14:10:00
	Spindle 1.Spindle stops when an alarm of 2.The minimum of "S" is the mir 3."No tool"interlock is made in 4.Spindle overload allowable t 5.	of level Boccu nimum speed of neffective time	urs motor	0) 0 0 0 [10 ms]
	6. 7. 8. 9. 10. 11.			
-				
	SET ADD CAL SEARCH	ITEM	† ITEM↓	
	F1(F2)(F3)(F4)	FSF	6) (F 7	(<u>F 8</u>)

- (3) The following states are assumed that the spindle has not reached the fixed speed:
 - (a) The spindle is not rotating, or the spindle start (forward or reverse) command is given.
 - (b) The spindle stop command is given while the spindle is rotating.
 - (c) A new S command is given while the spindle is rotating.
 - (d) When the spindle speed override switch setting is changed while the spindle is rotating.

SECTION 4 SPINDLE OVERLOAD DETECTION

When the spindle overload detection is effective, the spindle overload alarm occurs immediately if the spindle load meter pointer moves beyond the setting pointer.

If the spindle load meter pointer fluctuates as illustrated below when the setting is at "B", the spindle overload alarm occurs. Therefore, the setting should be at "A".



SECTION 5 ALARM

1726 ALARM A Spindle overload

The spindle load detection meter relay has been turned on due to overload.

[Index] None

[Character-string] None

[Code] None

[Probable Faulty Locations]

- 1) Cutting under overload condition
- 2) Setting error of the spindle load detection mater relay
- Error in setting the startup current immune period which is set at the "4.Spindle overload time" [6.Spindle] machine user parameter.

G. SIMPLIFIED LOAD MONITORING FUNCTION

SECTION 1 OUTLINE

This function monitors the spindle load and activates an alarm if the parameter-designated load value is exceeded.

A total of five parameter sets is used to control this function. The desired parameter set is selected using system variable VSLNO, with monitoring ON/OFF switching executed by the M142/M143 commands.

SECTION 2 DETAILS OF FUNCTION

(1) Monitoring ON/OFF Switching

M142 Simplified load monitoring function OFF

M143 Simplified load monitoring function ON

(2) Selecting the Spindle Load Monitoring Parameter No.

There are 5 sets of setting values for the "LOAD MAX", "LOAD TIMER", and "MONIT. MOD (Monitoring Mode)" items. The desired parameter set is selected from these 5 sets.

VSLNO Monitoring parameter No. selection system variable

A VSLNO backup is executed. Therefore, if no designation is made, the value designated for the previous operation will be adopted. If no previous designation exists, "1" will be adopted.

(3) Ignoring the Spindle Motor's START Current

When spindle rotation is executed from a stopped condition, there will be cases in which the "maximum load value" will be exceeded due to the START current at the spindle motor. This START overload condition is handled by the following 2 actions:

- (a) The overload is ignored until the actual spindle speed is within 75% to 125% of the speed command value.
- (b) When the actual spindle speed enters this 75% to 125% range, the system then waits for the spindle to reach a constant speed. During this waiting period (designated by machine user parameter "4. Spindle overload time" [6. spindle].), the overload condition is ignored.



Fig. 1

- (4) Overload Monitoring Format
 - (a) When the monitoring parameter is selected at VSLNO, and the M143 command is executed to switch the monitoring function ON, overload monitoring will then begin from the point when the spindle reaches the constant speed.
 - (b) If the spindle load exceeds the parameter-designated maximum load value, and that overload is maintained continuously until the "continuous overload time" is exceeded, the spindle overload alarm will be activated.

The alarm level can be changed by changing the monitoring mode.

0 : Alarm [A]

30 : Alarm [D]

(5) Display

On the 2nd page of the ACTUAL POSITION display, on the CURRENT MAIN PROGRAM screen, and on the READ MAIN PROGRAM screen, a " ∇ " scale mark is displayed at the 1st line of the "spindle load" bar-graph. This " ∇ " scale mark indicates the maximum load value.



AUTO OPERATION	A. MIN	0	MA3	3 15
PROGRAM	* CURENT MAIN PROGRAM *	2	1/07/15	mm
Y100 Z100 X0 Y0 Z0		DIS	X Y Z B	0.000 0.000 0.000 0.000
>X100 Y100 Z100		Co O N H=	0 F S	0.0 0 0.000
LOAD MAX SPINDLE LOAD ACT POSIT(WORK)	LOAD MONTOR(110%) TITIT 165% X Y Z 0.000 0.000 A-Mtd	0.00	0 B	0.000
=₽\$ =				
PROGRAM ACTUAL PA SELECT POSIT. PR	RT BLOCK IOGRAM DATA SEARCH		CHECK DATA	[EXTEND]
F1F2	F3F4F5F	6	F7	(FB)
READ MAII	N PROGRAM Screen			

(Display is identical for CURRENT MAIN PROGRAM)

When the monitoring function is ON (M143), LOAD MONITOR (*** %) (***: Maximum Load Value) items are displayed to indicate that the monitoring function is operating.

If the spindle load exceeds the maximum load value, the LOAD MONITOR (*** %), item will change from white to red. (In monochrome systems, it will become brighter.)

(6) Parameters

(a) Constant spindle speed waiting period setting

This parameter designates the waiting period during which the system waits for the spindle to reach a constant speed. (For details, refer to item (3) above.)

Machine user parameter "4. Spindle overload time" [6. spindle]

PARAMETER SET		
<pre>/ PARAMETER SET NO.6 Spindle 1.Spindle sto 2.The minimum 3."No tool"in 4.Spindle ove 5. 6. 7. 8. 9.</pre>	* MC USER PARA os when an alarm of of "S" is the minir terlock is made inet rload allowable tim	97/07/15 14:10:00 METER * Num speed of motor 0 ffective 0 me 0 [10 ms]
	CAL SEARCH	ITEM 1 ITEM 1
F1)F2 Setting range) <u>F3</u> F4(;	F5F6F7FB 0-60s.
Initial value	:	Varies depending on machine models.

(b) Spindle overload monitoring parameter setting

Values are set at LOAD MAX, LOAD TIMER, and MONIT. MOD in order for spindle overload monitoring to occur.



NO. One of the possible five parameter sets is selected at VLSNO, and an : asterisk is displayed to the left of the selected parameter No. LOAD MAX The maximum allowable load (in percentage) is designated. ÷ Setting range 0 - 280 (%) Command [SET], [ADD] Initial value 0 (%) LOAD TIMER : The maximum permissible period during which the spindle load can continuously exceed the maximum load value is designated (s). Setting range 0 - 120 (s) Command [SET], [ADD] Initial value 0 (s) MONIT. MOD : The overload alarm level is designated. Setting value = 0 Alarm [A] 30 . . . Alarm [D] Initial value0

- (7) System Variables
 - (a) Spindle overload monitoring parameter No.

[VSLNO]

- Reading/writing possible
- Data value range: 1 to 5
- VSLNO is stored in memory (backup). Therefore, if no designation is made, the value designated for the previous operation will be adopted. If no value has been designated previously, "1" will be adopted.
- (b) Spindle overload monitoring parameter

[VSLDT (i, j)]

- i : Parameter No. (1 - 5)

- j	:	1Ioad value
		2 Continuous overload time
		3 Monitoring mode

- Only reading possible
- (8) Alarms
 - (a) Alarm A 1242 Spindle overload
 - (b) Alarm D 4229 Spindle overload

Codes (for both 1) and 2))

XXXXYYYYY

- XXXX : Overload monitoring parameter No.
- YYYY: Overload (%) when alarm was activated

H. EXTERNAL PROGRAM SELECTION FUNCTION

SECTION 1 EXTERNAL PROGRAM SELECTION A (PUSHBUTTON TYPE)

This function selects the main and schedule programs by pressing the pushbutton switch corresponding to the desired program.

1. Main and Schedule Program Selection

Whether the program selection is to be made for the main programs or the schedule program is selected by the setting of the parameter (bit) data.

Parameter (Bit) No. 17	Main/Schedule Program
Bit 5 = 1	Schedule programs
Bit 5 = 0	Main programs

2. Inhibiting the Selection of the Same Program

By setting proper parameter (bit) data, it is possible to inhibit the loading of the selected program if the same program resident in the memory is selected again. This can save the program loading time.

Parameter (Bit) No. 17	Same Program Loading
Bit 6 = 1	Inhibited
Bit 6 = 0	Allowed

3. Program Selection

Selection of a desired program is possible by pressing the pushbutton switch corresponding to that program.

Selection of a program from P1.MIN (or SDF) to P16.MIN (or SDF) is possible.

Note that the number of selectable programs differs depending on the specification.

After the completion of program selection, the lamp corresponding to the program number lights up. This lamp, while lit, indicates that the corresponding program is selected.

4. Supplements

- (1) Program selection is possible only in the automatic mode.
- (2) While a program is being selected, pressing another program selection button is ignored.
- (3) While the control is in the alarm state, program selection is not carried out.
- (4) An alarm occurs if the program designated for selection is not found.
- (5) An attempt to select another program during the selected program execution causes an alarm.
- (6) Program selection during the editing or copying execution is suspended until the completion of the editing on copying.

SECTION 2 EXTERNAL PROGRAM SELECTION B (ROTARY SWITCH TYPE)

This function selects the main and schedule programs using the rotary switch and the program selection pushbutton switch.

1. Main and Schedule Program Selection

Whether the program selection is to be made for the main programs or the schedule program is selected by the setting of the parameter (bit) data.

Parameter (bit) No. 17	Main/Schedule Program
Bit 5 = 1	Schedule programs
Bit 5 = 0	Main programs

2. Inhibiting the Selection of the Same Program

By setting proper parameter (bit) data, it is possible to inhibit the loading of the selected program if the same program resident in the memory is selected again. This can save the program loading time.

Parameter (bit) No. 17	Same Program Loading
Bit 6 = 1	Inhibited
Bit 6 = 0	Allowed

3. Program Selection

Selection of a program is possible by specifying the desired program using the rotary switch. Selection of a program from P1.MIN (or SDF) to P16.MIN (or SDF) is possible.

Note that the number of selectable programs differs depending on the specification.

After the completion of program selection, the program selection indicating lamp lights up. This lamp lights up when the program number selected by the rotary switch and the program number actually selected coincide each other.

4. Precautions

- (1) Program selection is possible only in the automatic mode.
- (2) While a program is being selected, pressing another program selection button is ignored.
- (3) While the control is in the alarm state, program selection is not carried out.
- (4) An alarm occurs if the program designated for selection is not found.
- (5) An attempt to select another program during the selected program execution causes an alarm.
- (6) Program selection during the editing or copying execution is suspended until the completion of the editing or copying.

SECTION 3 EXTERNAL PROGRAM SELECTION C (BCD TYPE)

1. Overview

This function selects the main and schedule programs using the BCD (binary coded decimal) number input from an external device.

2. Program Selection Range

(1) 4-digit BCD Numbers

P * * * *.MIN (main program)





BCD 1st digit : 0 through 9, A through F
 BCD 2nd digit : 0 through 9, A through F

[Supplement] Designation of "P0000.MIN (or P00.MIN)" is not allowed.

Selectable range is from P0001.MIN to PFFFF.MIN or from P01.MIN to PFF.MIN.

3. Main and Schedule Program Selection

Whether the program selection is to be made for the main programs or the schedule program is selected by the setting of the parameter (bit) data.

Parameter (bit) No. 17	Main/Schedule Program
Bit 5 = 1	Schedule programs
Bit 5 = 0	Main programs

4. Inhibiting the Selection of the Same Program

By setting proper parameter (bit) data, it is possible to inhibit the loading of the selected program if the same program resident in the memory is selected again. This can save the program loading time.

Same Program Loading
Inhibited
Allowed

5. Supplements

- (1) Program selection is possible only in the automatic mode.
- (2) While a program is being selected, pressing another program selection button is ignored.
- (3) While the control is in the alarm state, program selection is not carried out.
- (4) An alarm occurs if the program designated for selection is not found.
- (5) An attempt to select another program during the selected program execution causes an alarm.
- (6) Program selection during the editing or copying is being executed is suspended until the completion of the job being in execution.

6. Timing Chart



- (1) The strobe signal is maintained the program selection completion answer is output from the NC. The strobe signal is turned off when the program selection completion answer is received.
- (2) The program number output signal is turned on after the completion of program selection until the next program selection begins. This signal is not output while a program is being selected.

7. Parity Check

The parity check function can be added to external program selection C (BDC type). By adding this function, the reliability of signals can be improved.

This function is a standard feature for NC versions from A709K (NG main: MAA07.09AK.POL).

7-1. Parameters

Machine user parameter No. 14 External program select

- 1. Check parity (even) is made effective
 - 1: Executes parity check in the even-parity scheme.
 - 0: Does not execute parity check in the even-parity scheme.
- 2. Check parity (odd) is made effective
 - 1: Executes parity check in the odd-parity scheme.
 - 0: Does not execute parity check in the odd-parity scheme.

If both bits are "0", parity check is not executed. If both bits are "1", parity check is executed in the even-parity scheme.

8. Alarms

3223 Program select

In the external program selection operation, an error occurred during the selection of a file name which is input from the FCP board.

[Code]

X :

- 1-> Automatic mode is not selected.
- 2-> The command buffer is not empty.
- 3-> An error when reading a main program in the PDO task.

XYZZAABB:

X=1 Automatic select alarm

BB:

- Bit0--> No specified schedule program file
 - 1-> No specified main program
 - 2-> No specified subprogram file
 - 3-> No specified main subprogram
 - 4-> No specified subprogram
 - 5-> No M02 or M03 up to the end of main program
 - 6-> No RTS up to the end of subprogram
 - 7-> Program buffer cannot register the entire program.

```
AA:
```

- Bit0-> The main program has more than 31 sequence names.
 - 1-> A sequence name consists of more than five characters including N.
 - 2-> No subprogram name is specified following CALL statement.
 - 3-> A subprogram name consists of more than five characters including O.
 - 4-> G and M code macro program names have not been defined.
 - 5-> More than 63 subprograms are called.
 - 6-> A block contains more than 156 characters.
 - 7-> Main file name and extension have more than 20 characters in total.

ZZ:

- Bit0-> Main file name and extension begin with a numeric character.
 - 1-> A device name consists of more than three characters or no colon (:) is designated following a device name.
 - 2-> A program name does not begin with "O".
 - 3-> A program name contains characters other than alphanumeric characters.
 - 4-> A program name has more than five characters including "O".
 - 5-> Option designation is made with more than 16 characters.
 - 7-> Request for program selection is found during execution of a schedule program.

Y:

- Bit0-> Request for program selection is found during execution of a main program.
 - 1-> An attempt to select a program in a mode which is not supported.
 - 3-> A floppy disk drive is designated as a device name.
- X=2 Defective memory

Symbols "*" and "?" are specified in a file name.

[Measures to Take]

Display the "RUN GUIDE" screen in the EDIT. AUX mode to check the file name for which external program selection was impossible. Then, select the program manually to check the error again.

3713 External program select parity check

When selecting a program according to the external device command, the NC conducted a parity check as specified by the parameter (machine user parameter "external program select"). The result is detection of a parity error.

[Index] None

[Character-string] None [Code] None

I. EXTERNAL M SIGNAL OUTPUT FUNCTION

SECTION 1 EXTERNAL M SIGNAL OUTPUT FUNCTION

1. Overview

The function outputs contact point signals to an external device, corresponding to the external M code written in a program. The external device conducts processing in response to the reception of the signal and after the completion of the processing, it returns the answer back signal to the NC. It then proceeds the program to the next step assuming that the commanded processing is complete upon the reception of the answer back signal from the external device.

For external M codes, two different numbers of output points, four or eight, are prepared. In either case, only one point is provided for an answer back signal.

2. External M Codes

For external M code outputs, M181 - M188 are provided:

Four points M181 - M184 Eight points M181 - M188

[Supplement] In a program, one block can contain only one external M code.

3. Timing Chart



3. The control is placed in the S.T.M. function execution status while an external M signal is being output and program execution stops.

J. CYCLE TIME REDUCTION FUNCTION
SECTION 1 CYCLE TIME REDUCTION FUNCTION

This function reduces non-cutting time by executing spindle rotation command and axis motion command or other command simultaneously to shorten total machining cycle time.

1. Ignoring Spindle Rotation Answer: M300 (1 Block)

When the M300 code is designated in a program, answer signals for spindle start/stop command is ignored:

(1) M03/M04

The program advances to the next program block without confirming the answer to the M03 or M04 code (spindle rotation).

(2) M05

Spindle stop operation begins simultaneously with the execution of axis motion commands. The program advances to the next program block without confirming the answer to the M05 code (spindle stop).

(3) M19

Spindle orientation operation begins simultaneously with the execution of axis motion commands. The program advances to the next program block without confirming the answer to the M19 code (spindle stop position).

(4) Example Program

N100 G00 X100 M3 M300 N101 Z10(M3)

[Supplement] 1. Designate the M300 code with a spindle rotation command in the same block.

- 2. To confirm the answer signal for the M03/M04 in the next block, designate the spindle start (M03/M04) command in that block again.
- 3. If the M300 code is designated in the fixed cycle mode, an alarm occurs.

2. Ignoring/Confirming Answer Signal Other Than Spindle Rotation Answer: M301 M302 (1 Block)

When the M301 code is designated in a program, answer signals other than the answer signal for the M03/M04 code are ignored. The M302 code cancels the M301 mode and the answer signals for the programmed M codes are confirmed.

These M codes (M301/M302) are effective to the following M codes:

- M32 (door close)
- M33 (door open)
- M181 M188 (external M signal)
- (1) Example Program

N200 N201 N202	M33M301 M181 M302	M301	(Ignoring M33 answer signal) (Ignoring M181 answer signal) (Confirming M33/M181 answer signal)
	:		

[Supplement] The M301 code is a one-shot M code. There fore, designate it with the M code for which an answer signal should be ignored in the same block.

K. F1-DIGIT FEED COMMAND FUNCTION (BY PARAMETERS)

SECTION 1 OUTLINE

Conventionally, as an alternative method, the cutting feedrate has been able to be designated by the F address character followed by a 1-digit integer. The feedrate can then be controlled by the feed switch setting value which corresponds to the designated 1-digit number.

This method is referred to as the "F1-digit feed function".

This function provides two methods for setting a feedrate – using the feed switches on the operation panel and using the parameter.

SECTION 2 F1-DIGIT FEED COMMANDS

- F1-digit feed command methods
 - F1 to F4 (feed switch method, 4-set specification)
 - F1 to F8 (feed switch method, 8-set specification)
 - F1 to F9 (parameter method)

Any other designation will be processed as a regular F command which directly designates a cutting feedrate.

- Real numbers (including variables) such as F1. etc., will be processed as regular F commands.

Therefore, variables may not be used in the F1-digit command format.

Example:

F1	F1-digit Operation will be according to the F1 feed switch setting	ng.
F200	200 mm/min. feed	
F5.	5 mm/min. feed	

 $\begin{bmatrix} LA1 = 8 \\ F = LA \end{bmatrix}$ 8 mm/min. feed

SECTION 3 PARAMETER METHOD

1. Operating Procedure

The data setting procedure for F1-digit feed by parameters feed format is described below.

(1) On-Screen Display

When in the automatic or MDI operation mode, press function key [F2] shown in Fig. 1 to proceed to the Parameter F1-Digit Setting screen as shown in Fig. 2 below.

=EX							
LIBRARY P.SET	PARAM. F1 SET	TOOL DISPLAY	PER- SONAL	DIAG		MESSAGE	[EXTEND]
(F 1)(F 2)(F3) <u>(F4</u>) (F 5)[F6]	(F7)	F 8]

Fig. 1 Function Keys in the Automatic (MDI) Operation Mode



(2) Unit System Setting

In the same manner as for other data setting modes (zero setting mode, tool data setting mode, and parameter setting mode) the unit system is determined by the setting at <u>NC optional parameter (bit)</u> No. 9.

The unit system which is currently selected will be displayed at the 3rd line of the on-screen display.

Parameter No.	Bit No.	Description	When "1" is set:	When "0" is set:	Initial Factory Setting
	0	In the data setting mode, the length is designated as "INCH" or "MM".	INCH	MM	0
	1	In the data setting mode, data is set in units of "1 mm", "1 inch", "1°", and "1 s", or in the minimum setting unit.	1 mm, 1 inch, 1°, 1 s	Minimum setting unit	1
	2				0
9	3				0
	4				0
	5				0
	6	For settings involving decimal points, the unit system for decimal point position is designated as "1 mm", "1 inch", "1°", "1 s".	Yes	According to bit 1	0
	7				

(3) Function Key Commands

Function keys may be used to execute the following commands:

For data setting purposes, the [SET] and [ADD] keys are used.

If the number of controlled axes exceeds 4, the AXIS CHANGE function can be displayed. When this function key is pressed, the ACT POSIT (WORK) at the 17th line can be switched.

To end the F1 PARAMETER SET screen and return to the original display page, press function key [QUIT].

(4) Updating Data By Page Forward/Back Operation

In addition to the normal setting procedure in which function keys [SET] and [ADD] are used, the page keys [P \downarrow] and [P \uparrow] can be used to add or subtract a given value (designated at the optional parameter (long word) No. 22) in a continuous manner over a number of pages. (Details follow.)

A timer is provided for this continuous data updating operation to prevent abrupt data changes.

Moreover, the addition and subtraction of the variable speed data designated at the optional parameter (long word) No. 22 is always executed using the "minimum setting unit" value ("1 μ m" for MM, "1/10,000 inch" for INCH).

2. Minimum and Maximum Values of Setting Data

(1) Maximum Feed Value (right-side data in Fig. 2)

∫ Minimum value	:	0 (For both "inch" and "mm", unit systems)
C Maximum value	:	Maximum F command value
		(4,000 mm/min or 157.4803 ipm)

(2) Feedrate (left-side data in Fig. 2)

ſ	Minimum value	:	0	(For both "inch" and "mm" unit systems)
≺				

Maximum value : The maximum value in (1) above (maximum feedrate)

If the maximum value is exceeded during data addition operations executed using the keys, processing will occur as follows:

-	Feedrate :	:	The calculated feedrate will be changed to the "maximum feed value", and set accordingly.
-	Maximum Feed Value :	:	If the "maximum feed value" is exceeded, the following error will be activated:
			5209 Input data overflow

3. Other Items

(1) System Variables

All setting data are held in system variables and can be referred to and set in programs.

System Variable Name	Description	Subscript Command	Remarks
VPF?F[*]	F1-digit parameter Feedrate	With 1 to 9	Unit conversion data
VPF1M[*]	F1-digit parameter Maximum feed value	With 1 to 9	Unit conversion data
VPF1C	F1-digit parameter Variable speed data	without	Optional parameter (long word) No. 22

- * Subscript from 1 to 9
- (2) Backup

The F1-digit parameter setting data is automatically backed up in memory at fixed intervals. Therefore, this setting data will be saved even if the power is switched OFF.

(3) Variable Speed Data Setting

Acceleration/deceleration data of F1-digit parameters "feedrate" and "maximum feed value" is set at optional parameter (long word) No. 22 within the following setting range:

∫ Minimum value : 0 ∫ mum value : 4,000,000

(4) F Displays on Actual Position Display Screen and Other Screens

On the ACTUAL POSITION and BLOCK DATA screens in the automatic operation mode, the F1-digit command is displayed in the same manner as the normal F4-digit command.

Metric system	Displayed to one digit following the decimal point (the numeral at the 2nd
	decimal place is rounded off)

^L Inch system Displayed to two digits following the decimal point (the numeral at the 3rd decimal place is rounded off)

SECTION 4 FEED SWITCH METHOD

1. Operation Methods

Feedrate setting rotary switches are provided on the F1-digit feed operation panel and required feedrates should be set using these rotary switches. When an F command (F1 to F4, or F1 to F8) is read, the rotary switch is selected meeting the read F command and the axis will move at the feedrate set by the selected rotary switch.

2. Selected State

Which of the feedrate setting rotary switches is selected can be visually confirmed by the indicating lamp provided for each of the rotary switches.

3. F1-digit Feed Rotary Switches



SECTION 5 SUPPLEMENTS

- (1) The F1-digit feed status (F1 F9) is not cleared when the NC is reset. It can be cleared by designating a direct F command or by switching the power OFF and back ON again.
- (2) The F1-digit feedrate setting may be changed while cutting is in progress.
- (3) The feedrate override function is inoperative while the F1-digit feed command is being executed.
- (4) An alarm will be activated if the F1-digit feed command is executed while in the "feed per revolution (G95)" mode.

L. ARBITRARY-ANGLE CHAMFERING FUNCTION

SECTION 1 OUTLINE

For some workpieces, chamfering (C-chamfer, R-chamfer) is desirable at the corners.

If the corners have arbitrary angles, however, this presents difficulties when chamfering is executed using usual programs (G01, G02, G03).

The "Arbitrary-angle chamfering function" enables the start and end points of chamfering to be determined automatically simply by designating the corner's apex and the desired chamfer amount.





C-chamfer (chamfer amount: 5 mm)

R-chamfer (chamfer amount: 5 mm)

SECTION 2 PROGRAMMING FORMAT

CHFC (CHFR) Xp_Yp_L_Q_I_J_ On the G17 plane CHFC (<u>Chamfer C</u>) C-chamfer command, one shot CHFR R-chamfer command, one shot Xp, Yp Corner apex, target value L Chamfer amount, one shot (positive number) L = 0 - 99999999 Q Travel amount after chamfer (travel amount toward next block's target value after completion of corner chamfer), one shot (positive number) Q = 0 - 99999999 I, J Virtual advancing direction (vector), one shot Direction is indicated, but command value is according to the unit system.)

SECTION 3 FUNCTION

The chamfering command (C-chamfer, R-chamfer) is executed at the corner area between the block in which the chamfer command has been designated and the block following it.



When a Q command is present:

In order to prevent uncut and cutter marks following the chamfering of the final corner (N7 & N2 corner) in multi-cornered shapes such as that shown in Fig. 1 above, a slight amount of overlapping cutting is required.

In such cases, use a Q command to designate the amount of overlapping cutting. After corner chamfering is completed, axes will amount toward the point specified in the next block only by this Q amount.



Fig. 2

When the I or J command is present (virtual advancing direction):

When chamfering the final corner of a multi-cornered workpiece as shown in Fig. 1, the I or J command can be used to execute the chamfering operation by designating the virtual advancing direction.

When the I or J command is designated in the chamfering command block, operation will occur based on the designated this virtual advancing direction.

[Supplement] The above explanation is made assuming that the G17 plane (X-Y plane) is selected. The following applies when other planes are selected:

> G18 plane Zp, Xp, K, I G19 plane Yp, Zp, J, K

SECTION 4 SUPPLEMENTS

- (1) The chamfering command is effective only in the G01 mode. If attempted in another mode, an alarm will be activated.
- (2) Chamfering will be executed on the plane which has been designated in the plane selection operation.
- (3) In order to ensure that chamfering is executed at the corner area between two straight lines, the mode for the next block must be either G00 or G01. If any other mode is designated, an alarm will be activated.
- (4) An axis travel command must be present in the next block following a chamfering command.
- (5) A chamfering command is also effective during an AG command, or during cutter radius compensation.
- (6) To execute a chamfering command in the incremental mode, designate the travel amount in the next sequence in terms of the travel amount from the end point of chamfering.

NOTICE

: When a chamfering command is designated in the cutter radius compensation mode, virtual advancing direction (I and J commands) of the cutter in regard to cutter radius compensation cannot be designated.

Regardless of the mode selected, any Q, L, I, or J commands which have been designated in a chamfering block will be taken as chamfering Q, L, I, or J commands.

M. 3-D CIRCULAR INTERPOLATION FUNCTION

SECTION 1 OVERVIEW

(1) The 3-D circular interpolation function executes circular interpolation in a required plane in addition to XY, YZ, and ZX planes in which circular interpolation is possible with conventional function.

3-D circular interpolation function:



Fig. 1

(2) Designation of the center and end point of the required arc makes axis motion possible along an arc in a spatial plane.

SECTION 2 COMMAND FORMAT

G02/G03 Xp_Yp_Zp_I_J_K_(P_Q_R_)

G02, G03 : Rotating direction (see 3. "3-D Arc Rotating Direction")

Xp, Yp, Zp : Coordinate values of end point

- Xp
 X or U axis

 Yp
 Yor V axis

 Zp
 Z or W axis
- I, J, K : Coordinate values of arc center
- (P, Q, R) : Coordinate values of a point in the arc in reference to the arc start point

These are necessary if a plane cannot be determined by Xp, Yp, Zp, I, J, and K.

- (1) When all of I, J, and K are designated, the arc is assumed to be 3-D arc. In other cases, the arc is assumed to be 2-D arc in XY, YZ, or ZX plane.
- (2) An arc is defined in the plane where the following three points lie:
 - Start point (actual position)
 - Arc center
 - End point (target position)



Fig. 2

- (3) The end point can be designated in either absolute value or incremental value according to the G code (G90 or G91).
- (4) The coordinate values of the arc center should be designated in incremental value regardless of G90 or G91.
- (5) If the start point, arc center, and end point are on a straight line (arc to be defined has 180° or 360° central angle), the plane in which the arc should be defined cannot be determined. In this case, either the following items by designating (P, Q, R) is necessary so that the plane can be determined.
 - Coordinate values outside the straight line
 - Normal vector on the plane including the arc set "0" or "1" at NC optional parameter (bit) No. 37 bit 5 for selecting either the above items
- (6) When P, Q, and R commands are designated with setting "0" at the optional parameter (bit) No. 37, but 5, an arc is defined in the plane where the following three points lie:
 - Start point (actual position)
 - Arc center
 - Point defined by (P, Q, R)

Example:



When P, Q, and R commands are designated with setting "1" at theoptional parameter (bit) No. 37, bit 5, an arc is defined in the plane where the following two points and a vector lie:

- Start point (actual position)
- Arc center
- Vector defined by (P, Q, R)

Example:



(7) The coordinate values of (P, Q, R) used to define the plane should be designated in incremental value regardless of G90 or G91.

[Supplement]

- 1. If any of Xp, Yp, and Zp is not designated, the coordinate value designated in a preceding block becomes effective for the omitted axis.
- 2. If (P, Q, R) is designated, the plane is defined using the start point, arc center, and (P, Q, R) even when the plane can be defined using the start point, arc center, and end point.

SECTION 3 3-D ARC ROTATING DIRECTION

The arc rotating direction should be designated based on the arc rotating direction when the 3-D arc is projected on the 2-D plane.

Command format:

G256 I_ (J_, K_) (designating the plane of projection)

I+ : The arc projected on the G19 plane is viewed from the X-axis positive direction.

I- : The arc projected on the G19 plane is viewed from the X-axis negative direction.

J+ : The arc projected on the G18 plane is viewed from the Y-axis positive direction.

J- : The arc projected on the G18 plane is viewed from the Y-axis negative direction.

 $K+ \hspace{0.1 cm}:\hspace{0.1 cm} \text{The arc projected on the G17 plane is viewed from the Z-axis positive direction.}$

K-: The arc projected on the G17 plane is viewed from the Z-axis negative direction.

Example:



G256 K–1

G02 X_Y_Z_I_J_K_

In the illustration above, when the arc projected on the G17 plane is viewed from the Z-axis negative direction, the rotating direction is G02 (CW).

To cancel the plane of projection, designate as follows:

G256 or G256 I0 (J0, K0)

[Supplement]

- 1. Designate an arbitrary numeral following I, J, and K. Any of the I, J, and K words must be designated.
- 2. When another plane of projection has been designated while a plane of projection is selected, the plane designated later becomes effective.

2 G256 I-100 G02 I-100 is canceled and the J100 mode is established. G256 J100 ÷

[Supplement] 3. If P, Q, and R words are designated simultaneously of G256, G256 is ignored.

- 4. When an arc rotating direction cannot be determined on the plane designated by G256, an alarm occurs. An arc rotating direction cannot be determined when the projected path of the 3-D arc is taken as a straight line.
- 5. Whether or not the projected path of a 3-D arc is a straight line is judged while referencing the tolerance (allowable amount) set at the NC optional parameter (word) No. 59. Here, the tolerance refers to the allowable width within which the projected path is taken as a straight line. This allows the arc rotating direction to be determined correctly.





Define segment L1 by connecting the start point (S') and end point (E') of the projected path. Also define segment L2 by connecting S' and the center point (0'). Draw vertical line at point E' to L2; the point of intersection between L2 and the vertical line is defined as E". Length E'E" is defined as L which is compared to the value set for optional parameter (word) No. 59.

 $|L| \leq$ Optional parameter (word) No. 59



The projected path is assumed to be straight line.

|L| > Optional parameter (word) No. 59

The projected path is not assumed to be straight line.

- (1) Arc Rotating Direction Definition without G252 Command
 - (a) The arc rotating direction (G02, G03) is defined based on the rotating direction of the arc projected on the XY plane and viewed from the +Z direction. See the illustration below.



(b) If the projected profile in the XY plane is assumed to be straight line (See Note), the arc rotating direction is defined based on the rotation direction of the arc projected on the YZ plane and viewed from the +X direction.



Fig. 7

(c) If the projected profile in the XY plane and that in the YZ plane are both assumed to be straight line (See Note), the arc rotating direction is defined based on the rotation direction of the arc projected on the ZX plane and viewed from the +Y direction.





Note: Optional parameter (word) No. 59 is used as the "tolerance to determine whether the projected line of 3-D arc on a plane is straight line or not" (Refer to SECTION 7, "PARAMETERS".).

- (2) The projection plane where the 3-D arc is projected can be fixed to a required plane by designating G17, G18, or G19. Whether this function is effective or not can be changed by the setting for optional parameter (bit).
 - (a) The G17, G18, and G19 codes are used to designate the plane on which the 3-D arc is projected to determine the rotating direction of the 3-D arc. rn this case, the following alarm occurs if the rotating direction cannot be determined by the projected path because the projected path is determined to be straight line.

ALARM B 2255 Data word: arc calculation 9

(b) The following parameter is used to set this function effective or ineffective.

Optional parameter (bit) No. 37, bit 3



Example:	N1 N2	G0 G17	X100	Y0	Z0				
	N3	G02	X0	Y0	Z100	I–33.333	J66.667	K66.667	F1000
	N4	G18							
	N5	G02	X0	Y100	Z0	166.667	J66.667	K–33.333	
	N6	G19							
	N7	G02	X100	Y0	Z0	166.667	J_33.333	K66.667	
	N8	G0	X0	Y0	Z0				
	N9	M02							

The 3-D arc rotating direction defined by the N3 block is determined by the G02 code using the path projected on the XY plane. Similarly, the 3-D arc rotating direction defined by the N5 block and that by the N7 block are determined by the G02 code using the path projected on the YZ and ZX plane, respectively.

SECTION 4 SHORTER ARC AND LONGER ARC

The circular interpolation will be performed from the are starting point to the are end point for shorter or longer are wite the designated center.

Command format:

G172/G173 X_Y_Z_I_J_K_(P_Q_R_)

- G172 : Shorter arc
- G173 : Longer arc
- X, Y, Z : Coordinate values of end point
- I, J, K : Coordinate values of arc center
- P, Q, R : Normal vector of the plane which includes the arc (When "0" is set at the optional parameter (bit) No. 37, bit 5)

Arbitrary vector from the start point on the arc plane (When "1" is set at the optional parameter (bit) No. 37, bit 5)



G172 : $\theta < 180^\circ$

G173 : *θ* > 180°

- [Supplement] When angle θ = 180°, the rotating direction may be reverse depending on the command error. In such cases, designate G256 (designation of projection plane) or any of P, Q, and R. When this is done, circular interpolation will be executed as follows.
 - G172→G02 G173→G03

SECTION 5 SUPPLEMENTS

- (1) The cutter radius compensation function (G41, G42) cannot be used.
- (2) High speed drawing of line drawing and animated simulation is not possible.
- (3) Envelope of animated simulation is not possible.
- (4) The tool length offset function must be set effective in a block before the 3-D arc is defined.
 In other words, the G53 G59 codes must not be designated in the same block as the 3-D arc definition block.

SECTION 6 ALARM CODES

2246 Data word: 'I', 'J' or 'K'

Improper I, J, or K command

[Code]

None-> The G256 command has two or more I, J, or K values.

Others-> Specified I, J, or K command in hexadecimal

[Probable Faulty Locations]

The G02 or G03 (circular cutting) command block has an I, J, or K command (coordinate of arc center) of which value is out of the setting range. Or, the G187 command (tolerance control mode ON) block has an I, J, K command (selection of control axis) of which value is out of the setting range.

[Measures to Take]

Data setting range :

Coordinate of arc center : \pm 99999.999 mm (\pm 99999.999 inch) Selection of control axis : 0 to 1

2255 Data word: arc calculation

Arc center cannot be calculated with given arc radius command, or arc radius error is greater than the permissible value which is set at optional parameter (long word) No. 3.

[Code]

- 1-> An 'R' command value is smaller than one-half of the distance between the start and end points. An error does not occur when the amount smaller than this value is within the permissible error range set at NC optional parameter (long word) No. 3.
- 2-> An error occurred during coordinate calculation of the other axis.
- 3-> Radius error exceeds permissible value which is set at NC optional parameter (long word) No. 3. (local coordinate system)
- 4-> Commanded radius is zero (0), or the radius command is not specified.
- 5-> Another axis parallel to the plane currently selected, or the corresponding rotary axis is specified.
- 6 or 10-> Radius error exceeds permissible value which is set at NC optional parameter(long word) No.3. (work coordinate system)
- 7-> Calculation error during calculation of a radius
- 8-> In 3-D circular interpolation, the plane on which the arc is defined cannot be determined. (No P, Q or R command)
- 9-> When the projection plane is designated, the arc rotating direction in the designated plane cannot be determined.
- 11-> In 3-D circular interpolation. The designated axis has been canceled.
- 12-> In 3-D circular interpolation, the 3-D coordinate conversion command or an axis other than the rotary axis copied in the G11 mode has been designated.

[Probable Faulty Locations]

- 1) Calculation error of arc radius R, or I, J and K commands in the circular interpolation mode
- 2) Setting error of optional parameter (long word) No. 3
- Program Example:

G0X0Y0 Optional parameter (long word) No. 3: 10

G2X100I49.99 F100

[Measures to Take]

- 1) Check the program for arc radius calculation error.
- 2) Check the setting of optional parameter (long word) No. 3.

2256 Data word: plane selection

With a plane designation G code(G17 (XY plane), G18 (ZX plane), G19 (YZ plane)), an axis parallel to the horizontal or vertical axis of the plane selected, or the corresponding rotary axis is specified.

[Code]

- None-> Two or more of the following has been designated simultaneously:
 - X-axis, U-axis, rotary axis corresponding to the X-axis
- 1-> Two or more of the following has been designated simultaneously: Y-axis, V-axis, rotary axis corresponding to the Y-axis
- 2-> Two or more of the following has been designated simultaneously: Z-axis, W-axis, rotary axis corresponding to the Z-axis
- 3-> With the Z-axis oscillation specification machine, a circular interpolation command has been designated in other than the Xp-Yp plane (designated by G17) in the M291 (oscillation start) mode.
- 4-> With the Z-axis oscillation specification machine, M291 (oscillation start) has been designated in a plane other than the Xp-Xp plane (designated by G17).

[Probable Faulty Locations]

Error in designating G17, G18 or G19 command

Program Example:

G17X0W0

[Measures to Take]

Correct the program.

2259 Data word: coordinate conversion

Operation error occurred during coordinate system conversion between the work and the local coordinate system.

[Code]

- 1-> Operation error occurred during conversion from local coordinate system to work coordinate system. Operation error occurred while rotation copy of the normal vector of 3-D circular interpolation is being executed in the G11 mode.
- 2-> Operation error occurred during conversion from work coordinate system to local coordinate system.

[Probable Faulty Locations]

Operation error during conversion

2359 Data word: rotating axis command

Wrong rotating axis command

[Code]

- 1-> Rotary table or index table command programmed during the following modes or programmed simultaneously with a G code establishing them.
 - RS232C interface channel being used
 - Circular interpolation mode (G02, G03) [index table only]
 - Tool length offset mode (G54 G59)
 - Cutter radius compensation mode (G41, G42)
 - Three dimensional compensation mode (G44)
 - Figure enlargement or reduction mode (G51)
 - Coordinate computation mode
 - Area machining mode
 - Note: In the cylindrical side surface machining mode (G175), this alarm occurs only when the tool length offset command or 3-D tool compensation command is specified.

- 2-> Index table command programmed during the following modes or programmed simultaneously with a G code establishing them.
 - Skip function (G31)
 - Local coordinate system specification mode (G11)
 - During copy function
 - Work coordinate system specification command (G92)
- 3-> Index axis command programmed simultaneously with another axis during G01 mode.
- 4-> Command for a rotary axis of the circular table which has been removed.
- [Probable Faulty Locations]

Wrong G code active when the rotary axis command is specified.

[Measures to Take]

Check the error condition, and then specify the correct command.

SECTION 7 PARAMETERS

(1) Optional Parameter (Word) No. 59

Tolerance to determine whether the projected path of 3-D arc on a plane is straight line or not

Initial value	:	0
Minimum value	:	0
Maximum value	:	32767
Setting unit	:	μm
Effective command	:	S (setting) only

(2) Optional Parameter (Bit) No. 37, bit 3

1 : 3-D arc (G17, G18, G19) projection plane fixing function effective 0 : 3-D arc (G17, G18, G19) projection plane fixing function ineffective Initial value : 0

- (3) Optional Parameter (Bit) No. 37, bit 5
 - 1 : Coordinate value (P, Q, R) on the plane including arc
 - 0 : Normal vector (P, Q, R) on the plane including arc

N. AUTOMATIC ATTACHMENT INDEXING FUNCTION

SECTION 1 AUTOMATIC ATTACHMENT INDEXING FUNCTION

1. Overview

This function indexes an attachment in intervals of 5° , 15° , or 45° .

The index angle intervals (5, 15 or 45°) are determined by the machine specification. Refer to the specification for approval.

2. Index Commands

The attachment index command includes the direction through which the attachment is rotated.

M94 Attachment indexing in the forward direction M95 Attachment indexing in the reverse direction

3. Angle Commands

Attachment index angle is defined as absolute angle position in the forward rotation direction with the front position taken as 0° .


4. Command Format





(1) The M code and an angle command must be designated in the same block.(2) The angle command should be given after the spindle rotation has stopped.

5. Alarm

2394 Data word: 'RH'

Command value of attachment angle command RH is outside 0° to 359° , or an angle not supported by the specification is specified (usually, attachment is indexed in units of 5° , 15° or 45°).

[Code] RH command value in hexadecimal

[Probable Faulty Locations]

Program error

Program Example: N000 RH=360 M94 WRITE

2732 Data word: RH

Command value of attachment angle command RH is outside 0° to 359°, or an angle not supported by the specification is specified (usually, attachment is indexed in units of 5°,15° or 45°). [Index] None

[Character-string] None [Code] RP command value in hexadecimal [Probable Faulty Locations] Program error Program Example: N000 RH=360 M94

O. AXIS NAME DESIGNATION FUNCTION

SECTION 1 OUTLINE

As a rule, coordinate axis names used in the NC program are specific to each machine model. For example, table motion is defined as the X-axis.

However, the axis name designation function enables the axis names which are used in a program to be selected by the programmer.

When this function is used for machining with an angular attachment, or for MCM five-face machining models, the tool travel axis (cycle axis) can be handled as the Z-axis at all the machining surfaces.



Fig. 1 Program Coordinate System

By designating the axis names as shown above, the Z-axis and the Y-axis in the program coordinate system will correspond to the Y-axis negative direction and the Z-axis positive direction in the machine coordinate system, respectively.

SECTION 2 AXIS NAME DESIGNATION

The names of the standard axes (X, Y, Z) are designated by G-code commands.

1. Renaming a Machine Axis

Programming format:

G14 Xx Yy Zz

X, Y, Z Program axes (new axis names) x, y, z Machine axis No.

x, y, z Values	Corresponding Machine-specific Axis Names	Remarks
±1	Х	A minus (–)
±2	Y	symbol indicates the opposite axis
± 3	Z	direction.

Example: For a "G14 X–3 Y–2 Z–1" designation, the new coordinate system will be as shown below. \square Machine axis No.

Program axis



The following command is used to cancel the axis switching function (return to the machine coordinate system):

Programming format: [G14] or [G14 X1 Y2 Z3]

[Supplement] The rotating direction of 3-D circular interpolation is judged taking the selected plane as the projection plane. (same as when the optional parameter (bit) No. 37 bit 3 is "1".)

- The following alarm will be activated if the program axis names are not designated for all of the X, Y, and Z axes (except for the cancel command).

Alarm B 2253 Data word: axis command

Codes	:	01 X-axis command only
		02 Y-axis command only
		03 X- and Y-axis command only
		04 Z-axis command only
		05 Z- and X-axis command only
		06 Z- and Y-axis command only

- The following alarm will be activated if the machine axis No. designation is other than $\pm 1, \pm 2$ and ± 3 or if the same numbers exist.

Alarm B 2253 Data word: axis command

Code : Command value

- Machine axis No. combinations are shown in Table 1 and in Fig. 2. If other combinations are used, the following alarm will be activated:

Alarm B 2253 Data word: axis command



- The following alarm will be activated if designation is attempted during the tool length offset, cutter radius compensation, fixed cycle, 3-D tool offset, and/or arc mode.

Alarm B 2263 Data word: G code 18

- Once designated, a coordinate system will not be changed when the power is switched ON or OFF, or when the NC is reset.

Machining Surface	Coordinate System No	Axis	Desigr No.	nation	Remarks
	Gysten No.	X	Y	Z	
А	1 2 3 4	1 -1 2 -2	2 2 -1 1	3 3 3 3	
В	5 6 7 8	1 -1 -3 3	-3 3 -1 1	2 2 2 2	
С	9 10 11 12	1 -1 3 -3	3 3 -1 1	-2 -2 -2 -2	
D	13 14 15 16	-3 3 2 -2	2 2 -3 3	1 1 1 1	
E	17 18 19 20	3 -3 2 -2	2 2 3 3	-1 -1 -1 -1	

Table 1 Coordinate Systems And Axis Designation Nos.



Shaded areas indicate X-Y planes.

Fig. 2 Coordinate System Types

SECTION 3 APPLICATION RANGE

(1) Modes

In the automatic and MDI modes, the NC program is processed according to the program coordinate system. In other modes, processing will be performed according to the machine coordinate system.

(2) Command Axis

Standard axes (X,Y, Z) and axis-related commands (I, J, K, P, R) are also processed according to the program coordinate system.

(3) External Axis

The mirror image switch setting (X,Y, Z) is processed according to the program coordinate system.

(4) G Codes

Axis related G codes are processed according to the program coordinate system.

G17, G18, G19	(Plane designation)
G54, G55, G56	(Tool length offset)
G02, G03	(Circular interpolation)
G41, G42	(Cutter radius compensation)

(5) M Codes

All M codes except those shown below are processed according to the program coordinate system. M22, M23, M24, M25 (Clamp/unclamp)

SECTION 4 DISPLAY

The program axes are displayed at the lower right of the machine axes X, Y, and Z on the ACTUAL POSITION screen.

Mirror image indica	ation							
Machine axis								
Program	Program axis_							
(positive/negative is indicated by the sign)								
AUTO OPERA	TION		N	$\frac{1}{1000}$				
	1000 000	0	97707715 14.	10.00				
`-X +`	1600.000	U						
v	1100 000	N	00	1				
1 x	1100.000	IN	· · · ·	I				
7	1000.000	F	0.0 H	0				
		010 11	Ŭ					
W	0.000	S	0 D	0				
		г	CUECK					
	SIT. PROGRAM DATA	SEARCH	DATA	[extend]				
F1	F 2) (F 3) (F 4) <u>F</u> 5)	F6)F7)	= 8)				
	Fig. 3 ×4 Di	splay (or	×16 Display)					

SECTION 5 PROGRAM EXAMPLE (FOR MCM MODELS)

(1) Fixed Cycle (Drilling) Program Example

O100						
N01	G0	Z100				
N02				⊤1		
N03				M77		
N04				M63		
N05				M75		
N06	G15			H1	N06:	Workpiece coordinate system selection 1
N07	G14	X1	Y 3	Z–2	N07:	Axis name designation
N08	G56	Z100		H1	N08:	Tool length offset Z-axis
N09	CALL	01				
N10	G53				N10:	Tool length offset cancel
N11				M74		
N12	G15			H2	N12:	Workpiece coordinate system selection 2
N13	G14	X2	Y 3	Z1	N13:	Axis name designation
N14	G56	Z100		H1	N14:	Tool length offset Z-axis
N15	CALL	01				
N16	G53	G0	X1000	M77		
N17	G14				N17:	Axis name cancel
N18				M2		
O1						
N1	G71	Z100				
N2	G81	X20	Y20	Z–5	R40	F100
N3	X40					
N4	X60					
N5	G80					
RTS						



Fig. 4

As shown in the above program, the same machining operation can be executed for all machining surfaces using a single subprogram by switching axes.

SECTION 6 SUPPLEMENTS

(1) There are cases when G codes and actual position data in the BLOCK DATA screen differ from those in the NC program. This is because the BLOCK DATA screen shows the machine axis motion, and differences occur between the machine coordinate system and the program coordinate system due to axis switching.

G codes which may differ from those in the NC program:

G17, G18, G19	(plane designation)
G54, G55, G56	(tool length offset)
G02, G03	(circular interpolation)
G41, G42	(cutter radius compensation)

Axes which may differ from the program command:

Standard axes (X, Y, Z) and axis-related commands (I, J, K)

(2) Zero Point Offset (G92)

A command value designated in the G92 mode is processed as a command value designated in the new coordinate system. Therefore, care should be exercised to the machining surface and axis to which the command applies.



The value which is entered in zero offset mode is stored in memory as machine axis data. In the above example, the Z-axis (machine axis) zero point will be rewritten as shown above.

(3) Upper Limit Return in Fixed Cycle

The M52 command which is designated for fixed cycle, coordinate calculations, and others is effective for the Z-axis (machine axis) in the machine coordinate system. If the Z-axis in the program is designated as the X or Y machine axis, an alarm of level B will be activated when the M52 command is designated.

(4) Feedrate

Normally, the Z-axis feedrate differs from that of the other axes (X, Y). Therefore, the following precautions should be observed during the programming operation:

- To designate a feedrate command (F-command), always verify the machining surface and the axis (machine axis) which will actually move. Special care must be exercised when designating the X- or Y-axis as the Z machine axis in the program, so that the maximum Z-axis feedrate is not exceeded.
- (5) G Codes Effective for Machine Coordinate System

When a G code, for example G22 (programmable travel limit), G30 (home position return), and others which are effective for the machine coordinate system, has been designated, it causes axis motion in the machine coordinate system, and does not cause axis motion in the program coordinate system.

Example:

```
G22 X1000 Y1000 Z1000 I-1000 J-1000
K-1000
```

Positive travel end limits for X, Y, and Z = 1000

Negative travel end limits for X, Y, and Z = -1000

NOTICE

When the same part program is performed repeatedly by switching the coordinate system, if rapid feed commands are designated for multiple axes in the program, various positioning paths will be generated since the Z-axis feedrate differs from that of the other axes (X, Y), and, depending on the command value, there may be cases in which the tool axis (Z-axis) will interfere with the workpiece. Use extra care to avoid such an occurrence.

P. SLOPE MACHINING FUNCTION

SECTION 1 OUTLINE

This function enables a part program created in the X-Y-Z coordinate system to be executed in any arbitrary coordinate system. This is done by converting the arbitrary coordinate system commands to machine commands.



The arbitrary coordinate system shown above (created by executing a parallel or rotational shift of the basic coordinate system) is called a "slope coordinate system".



- (1) Parallel shift of a coordinate system parallel is executed by designating the desired X, Y, and Z shift amount from the zero point of the basic coordinate system to the zero point of the arbitrary coordinate system.
- (2) With either the X', Y', or Z' axis of the parallel shifted arbitrary coordinate system serving as the pivot point (center), the desired rotation angle is then designated to execute a rotational shift.
- (3) The X"-Y"-Z" coordinate system created by steps 1 and 2 above is referred to as the "slope coordinate system".

This function permits up to 3 coordinate shift conversions, with steps 1 and 2 above being counted as 1 conversion.

SECTION 2 COORDINATE CONVERSION FUNCTION IN THE AUTOMATIC AND MDI OPERATION MODES

To perform 3-D coordinate conversion, first create a slope coordinate system, and then designate machining commands in the normal manner in the slope coordinate system.

1. Setting of Slope Coordinate System

Either of the following two command formats may be used:

(1) [G69 [X_] [Y_] [Z_]
$$\begin{bmatrix} P_{-} \\ Q_{-} \\ R_{-} \end{bmatrix}$$
]

- X, Y, Z: Parallel shift amount for each coordinate axis
 - P: Rotation angle in the X-Y plane (Z-axis pivot)
 - Q: Rotation angle in the Z-X plane (Y-axis pivot)
 - R: Rotation angle in the Y-Z plane (X-axis pivot)
- P, Q, R: -9999.9999 to 9999.9999 (including cases of rotational axis with limit)
- * The rotation direction in the P, Q, and R commands is CCW as viewed from the positive direction of an axis not included in the plane.

(2) [G69H1]

[Supplement]

- ment] 1. When settings of X, Y, and Z are omitted, "0" is set.
 - 2. Ether P, Q, or R must be designated.
 - 3. This command executes the coordinate conversion by using the coordinate conversion parameter designated in the manual operation mode (refer to SECTION 3, "COORDINATE CONVERSIONS IN THE MANUAL OPERATION MODE" which follows).

As stated earlier, up to 3 coordinate conversion operations are possible with this function.

At the (1) command shown above, the number of conversions is counted at each G69 code. If a 4th conversion is attempted, this will be processed as a new 3rd conversion, and the old 3rd conversion will be canceled.

When the (2) command shown above occurs, all the current settings will be deleted (even if a coordinate conversion is currently in progress), with the setting operation then beginning from the 1st conversion. The number of conversions will be the same as the number set by the parameter.

If the (1) command is executed following the (2) command, the current "number of conversions" count will be updated accordingly, or, if it is a 4th conversion, the 3rd conversion setting will be changed.

The G68 command is used to cancel 1 of the coordinate conversion settings. The "number of conversions" count will be reduced accordingly. This command can be used in both the (1) and (2) formats.

- All G69 commands will be canceled if an NC reset or the M02 command is executed.
- 5. With the graphic display specification, there are cases in which the tool path immediately following a coordinate conversion differs from the path immediately preceding the conversion.

Example:



After the execution of G68, the tool path is displayed from (b).

2. G Codes Usable During Coordinate Conversion

G Code	Operation During Coordinate Conversion
G00 Positioning G01 Cutting feed G31 Skip	Linear interpolation in the slope coordinate system
G02 Circular interpolation G03	Circular interpolation in the slope coordinate system Helical cutting is impossible.
G10 Local conversion G11	Local conversion is executed with respect to the slope coordinate system.
G15 Work coordinate system selection G16	The work coordinate system which is to be the object of 3-D coordinate conversion is selected.
G17 to G19 Plane designation	A plane in the slope coordinate system is designated. Planes having an additional axis cannot be designated.
G22, G23 Programmable stroke limit	Stroke limit is designated in the work coordinate system (coordinate system before 3-D coordinate conversion occurs).
G40 to G42 Cutter radius compensation	Cutter radius compensation is executed in the slope coordinate system.
G43, G44 3-D offset	3-D offset is executed in the slope coordinate system.
G50, G51 Geometry enlarge/reduce	Geometry is enlarged or reduced in the slope coordinate system.
G53 to G59 Tool length offset	Refer to following page.
G60 One-direction positioning	One-direction positioning is executed in the machine coordinate system. (Verify movement before executing this command.)
G62 Programmable mirror image	Mirror image is executed according to parameter bit setting.
	Identical operation can also be executed using switches. (Refer to following page.)
Fixed cycle	Refer to following page.

G14 (axis name designation) and G92 (work coordinate system setting) cannot be designated while coordinate conversion is in progress.

G codes other than those shown above can be executed regardless of the coordinate system. (G04, G09, G20, G21, G30, G61, G64, G90, G91, G94, G95)

- Tool length offset (G53 to G59)

When tool offset the X- Y- or Z-axis direction is designated while a coordinate conversion is in progress, the coordinate system in which tool length offset is executed can be selected by the NC optional parameter bit No. 48, bit 1 (refer to SECTION 5, "PARAMETER SETTING").



If tool length offset along an additional axis (G57 to G59) is designated while a coordinate conversion is in progress, tool length offset will be executed in the coordinate system which includes the additional axis, irrespective of the slope coordinate system.

- Programmable mirror image (G62)

The desired mirror image coordinate system can be selected according to the bit status when the power is switched ON. The bit settings and the corresponding coordinate systems are shown in the Table below. (This applies to the mirror image function using switches.)

Optional Parameter B	it Status at Power ON	Corresponding Coordinate System	
No. 34, bit 2	No. 33, bit 2		
1	1	Work coordinate system	
1	0		
0	1	Slope coordinate system	
0	0	Local coordinate system	



- Fixed cycles (G71, G73, G74, G76, G80 to G87, G89, G274 and G284)

During coordinate conversions, rapid axis feed in the usual fixed cycle is converted to cutting feed. This cutting feedrate is designated by the NC optional parameter (long word) No. 14 (refer to SECTION 5, "PARAMETER SETTING").

Example: Drilling cycle (G81, G82)



Dwell for "P" seconds

parameter (long word) No. 14)



In fine-boring (G76) and back-boring (G87) cycles, a shift is executed to retract the tool tip. During coordinate conversions, this shift is executed in either the machine coordinate system or the slope coordinate system, according to the setting at NC optional parameter (bit) No. 11, bit 4 (refer to SECTION 5, "PARAMETER SETTING").

Example: Fine boring cycle (G76)



Designated by shift amount I, J

Fig. 2

[Supplement] When an attachment is used, the tool indexing position will vary according to the direction in which the tool is facing. Therefore, always verily the tool tip direction before designating the shift amount using I and J commands.

3. MnemoniC Codes Usable During Coordinate Conversion

Mnemonic Code	Description
OMIT LAA ARC GRDX GRDY SQRX SQRY BHC	Can be used as a coordinate calculation function in the slope coordinate system.
FMILR FMILF PMIL RMILR RMILO RMILI	Can be used as an area machining function in the slope coordinate system.
CHFC CHFR	Can execute arbitrary angle chamfering in the slope coordinate system.
COPY COPYE	Can be used in the slope coordinate system.

Mnemonic codes other than those listed above can be used regardless of the coordinate system. (NOEX, CALL, RTS, MODIN, MODOUT, GOTO, IF, MSG, NMSG, and NCYL)

[Supplement] Additional patterns of I-MAP may not be used.

4. Upper Limit Return (M52)

During coordinate conversions, the "upper limit return (M52)" command used in the fixed cycle, milling cycle, and other cycles is converted to the "R-point return (M54)" command. Moreover, the "upper limit return" command which is generated at the final cycle of the coordinate calculation function is also ineffective.

The "designated point return (M53)" command can be used.

5. Axis Command Cancel

The "axis command cancel" command is not effective during coordinate conversions.

6. Pulse Handle Overlap

Regarding pulse handle overlap during coordinate conversions, select the coordinate system according to the ON/OFF status of the slope selection switch.

Slope Selection Switch	Coordinate System
ON	Slope coordinate system
OFF	Machine coordinate system

- Slope Machining Example:



Fig. 3

_	:							
(A)	G333	PAT=1 X100	PAC=45	H6 7 100	D45	OSUB SZ00		
	G69	R90	1-100	2-100	1 40	G0 X0 Y300	Z100	
	CALL	OSUB				G81 G17 X-50	Y0 R5 Z-20	F84
	G68					X50		
	G88 G334					RTS		
В	G333	PAT=1	PAC=135	H6				
	G69	X–100	Y–100	Z–100	P45			
	G69	R90 OSUR						
	G68	0300						
	G68							
~	G334							
\odot	G333	PAT=1	PAC=225	H6				
	G69	X–100 R00	Y100	Z–100	P–135			
	CALL	N90 OSUB						
	G68							
	G68							
\sim	G334							
(\bigcirc)	G333	PAT=1	PAC=315	H6	D105			
	G69 G69	R90	1100	2-100	F139			
	CALL	OSUB						
	G68							
	G68							
	G334							
	:							

SECTION 3 COORDINATE CONVERSIONS IN THE MANUAL OPERATION MODE

By designating coordinate conversion parameters in the manual operation mode and by turning the "slope selection switch" ON, manual operation can be executed in the slope coordinate system.

Moreover, in the automatic and MDI operation modes, the designated coordinate conversion parameters can be transferred by executing the G69 H1 command.

1. Setting Coordinate Conversion Parameters

Call the 3-D coordinate setting screen using the following procedure:

- (1) Press the MANUAL key to select the manual operation mode.
- (2) Press function key [F8] (EXTEND) twice. (The function key [F1] will be assigned to the function key [F1].)
- (3) Press function key [F1] (3-D CD. CONVERT).

The 3-D coordinate setting screen consists of 2 pages, with either the 1st or 2nd page being displayed, depending on the setting at NC optional parameter (bit) No. 11, bit 5. (Refer to SECTION 5, "PARAMETER SETTING").

1st Display Page

MAN. OPEF	ATION			4:10:00
	* 3-D CC	DORDINATE SET *	<u>97/07/15 1</u> 1mm	4:10:00
1.ST CONV	SHIFT AMOUNT 7. X -999999.999 Y -999999.999 7 -99999.999	ROTARY AXIS P	REV. ANGLE -999999,999	
2.ND CONV	/. X -999999.999 Y -99999.999	Р	- 9999 9.999	
, 3. RD CONV	/. X -99999.999 Y -99999.999 Y -99999.999 Z -99999.999	Р	-99999.999	
ACT POSIT	X (WORK) –999999.9 A-Mtd	ү 999 –999999,999	Z 999999.999	A 999999.99
> > > >				
SET	ADD		QUIT	

F1F2F3F4F5F6F7F8

The parallel shift amount, rotational axis name (P, Q, R), and the rotation angle are designated in the same manner as when the G69 command is used.

2nd Display Page

MAN. OPERATION				
	* 3-D COORD	INATE SET *	<u>97/07/15</u> 1mr	14:10:00 n
3-D COOR. POS.		3-D COORD	INATE VALUE	
x -999999.999	1.ST REV	ROTARY AXIS P	REV. ANGLE -99999.999	
Y -99999.999	2.ST REV	Р	-99999.999	I
z -99999,999	3.ST REV	Р	-99999.999	1
ACT POSIT(WORK) =EX ≕EX =CS >	X _99999.999 A-Mtd	Y 99999.999	Z -99999.999	A -99999.999
SET ADD	D I SPL RESET	AY	QUIT	
F1)F2		4) (F 5) (F 6 F 7	IF B

The parallel shift amount is designated as a coordinate point position* in the slope coordinate system, and the rotational axis name and rotation angle are designated in the same manner as on the 1st display page.

- * "Coordinate point position in the slope coordinate system" refers to the spindle position in coordi- nate system which follows the 3-D coordinate conversion operation. In other words, it is the current position in the slope coordinate system.
- [Supplement] 1. Although the coordinate conversion parameter displays on the 1st and 2nd display pages are different, the content is the same. Differing parameter settings on the two pages are not permitted.
 - 2. Coordinate conversion parameters designated in the manual operation mode can be transferred to the automatic and MDI operation modes by designating the G69 H1 command. The parameters on the 1st display page (shift amount, rotational axis name, rotation angle) are transferred as they are. However, as for the parameters on the 2nd display page, the "total shift amount" is transferred as the "1st conversion" shift amount, and the shift amounts for the 2nd and 3rd conversions are transferred as "0". This should be noted when using the G68 command on the 2nd display page.
 - 3. If an asterisk (*) is entered as the rotational axis name for a given coordinate conversion, all subsequent coordinate conversion settings will be ignored. For example, if the 1st conversion axis name is designated as "*", the 2nd and 3rd conversion settings will be ignored, with only the 1st conversion being effective.
 - 4. An alarm will be activated if the G69 H1 command is attempted when the "coordinate conversion parameter inoperative" status (1st conversion axis name is " * ") is established.

2. Coordinate Conversion Parameter Settings when Switching to the Manual Operation Mode

There are occasions during automatic or MDI operation when it is necessary to switch to the manual operation mode to make a correction, etc. When switching to the mode at such a time, the data which exists immediately before switching occurs is automatically transferred and is displayed as coordinate conversion parameter data. The setting at NC optional parameter (bit) No. 11, bit 6 determines whether or not this automatic data transfer occurs when switching to the manual operation mode.

G69	X10	Y10	Z10	P45 💿
G69	X20	Y20	Z20	Q45 🕑
G69	X30	Y30	Z30	R45
:				Ŭ
1				
:				
G68				
:				
2				
:				
G68				
:				
(3)				
:				
G68				
:				
4				
:				
M02				

If a switch to the manual operation mode occurs at steps (1) to (4) during execution of the above program, data will be transferred as coordinate conversion parameter data as shown in the table below. (When NC optional parameter (bit) No. 11, bit 6 is "1".)

Conversion data not canceled by the G68 command will be transferred even if the M02 or RESET command is executed.

Tabl	e '	1
Tap	e	

Modo Switabing Points	Data to be Transferred			
Mode Switching Folins	1st Conversion	2nd Conversion	3rd Conversion	
1	a	b	॰	
2	a	ь	-	
3	a	-	-	
4	-	-	-	

When a data transfer occurs, all the current coordinate conversion parameter settings will be deleted. Therefore, the "no setting" (indicated by "--" marks in the above table) status exists at steps where no data transfer occurs.

3. Executing Coordinate Conversion

In the manual operation mode, turn the "slope selection switch" ON in order to make the designated coordinate conversion parameter settings effective.

The coordinate conversion operation will then be executed.

During coordinate conversion operation, rapid feed commands, cutting feed command, and pulse handle feed commands for X-, Y-, and Z-axes are processed in the slope coordinate system.

The feedrates and feed amounts are as shown below:

- Rapid feedrate The slowest machine axis speed is adopted for all machine axes.
- Cutting feedrate The designated speed is adopted.
- Pulse handle feed amount The designated distance is adopted.
- [Supplement] 1. Never switch the axis or change the coordinate conversion parameter setting while axis travel is in progress.
 - 2. All machine axes will be stopped if any axis reaches the limit position during a coordinate conversion operation.

SECTION 4 ACTUAL POSITION DATA DISPLAY

Actual position data in the slope coordinate system can be displayed. Switching between the slope coordinate system and the work coordinate system can be executed at the NC optional parameter (bit) No. 6, bit 6 setting.

However, to display actual position data in the slope coordinate system in the manual operation mode, the "slope selection switch" must be turned ON together with the bit switching operation.

- [Supplement] 1. This function applies only to the ACTUAL POSITION display screen called by pressing function key [F2]. On the program display screen, actual position data is displayed only in the work coordinate system.
 - 2. There are cases when a discrepancy may exist (never exceeding the "minimum setting unit" amount) between the designated coordinate value and the displayed value, due to limitations in the OSP's calculating accuracy.
 - 3. If a local coordinate conversion (G11) is designated during 3-D coordinate conversion operation, the local coordinate's current position will be displayed at the local-converted position relative to the work coordinate system. In other words, actual position data in the local coordinate system represents the [actually designated value] [3-D coordinate conversion].

Example:	G69 X10	Y10	Z10	P0	
	G11 X–20	Y–20	Z–20		
	G00 X49.5	Y195.8	85	Z–1090	Command at coordinates where local conversion is
					being executed following
					3-D coordinate
					conversion.
	O	- 141 - 14 D			D

	Х	Y	Z
Command value	49.5	195.85	-1090
imes4 ($ imes$ 2) enlarged display (slope coordinate system)	29.5	175.85	-1110
Local coordinates	59.5	205.85	-1080
Work coordinates	39.5	185.85	-1100

Current Position Display At Each Display Page (NC optional parameter (bit) No. 6, bit 6 is 1)



×4 (×2) Enlarged Display (Slope Coordinate System)

Local Coordinate Current Position Display

1	-					
/	AUTO OPERATION	TST	9.MIN	OTST9 N13	16 \	
1	ACTUAL POSITION			97/07/15 14:10 1mm	:00	
	LOCAL COORDINATES WORK COORDS (APA) WORK COORDS MACHINE COORDS FEEDBACK COORDS TARGET VALUE DISTANCE REMAINING MANUAL SHIFT ACTUL MANUAL SHIFT TOTAL PITCH ERROR COMP	X 39.500 39.500 39.500 2539.500 39.500 0.000 0.000 0.000 0.000	Y 185, 850 185, 850 185, 850 2685, 650 185, 850 0, 000 0, 000 0, 000 0, 000	Z -1100.000 -1100.000 -1100.000 1100.000 1400.000 -1100.000 0.000 0.000 0.000 0.000		 Local coordinates in reference to the work coordinate system The value obtained by subtracting 3-D coordinate conversion value from the command value is displayed.
	A-Mto =P0 =P0 = PROGRAM ACTUAL PART SELECT POSIT. PROG	j BLOCK RAM DATA	SEARCH	CHECK DATA [E>	TEND]	
	F1F2F	3)F4)F5F	6 F 7 F	8)	

SECTION 5 **PARAMETER SETTING**

NC optional parameter (long word)

No. 14 Cutting feedrate to which the rapid feedrate in the fixed cycle is converted during coordinate conversion operations.

Setting units	:	mm/min.
Setting range	:	1 - 20000
Initial value	:	20000

NC optional parameter (bit) (factory setting: 0)

No. 6, bit 6	Actual position d	ata display in the slope	coordinate system
--------------	-------------------	--------------------------	-------------------

	=0 : Actual position data in the work coordinate system=1 : Actual position data in the slope coordinate system
No. 11, bit 4	Tool tip retract motion during fixed cycle operation
	 =0 : Axis retract command in the machine coordinate system =1 : Axis retract command in the slope coordinate system
No. 11, bit 5	Coordinate conversion parameter setting screen in the manual operation mode
	=0 : 1st page =1 : 2nd page
No. 11, bit 6	Coordinate conversion parameter transfer when switched to the manual operation mode
	=0 : Not transferred =1 : Transferred
No. 33, bit 2	Mirror image in the slope coordinate system
	 =0 : According to the setting at NC optional parameter (bit) No. 34, bit 2 =1 : Slope coordinate system
No. 48, bit 1	Tool length offset in slope coordinate system
	=0 : Workpiece coordinate system =1 : Slope coordinate system

Q. MULTIPLE-POINT SPINDLE ORIENTATION FUNCTION

SECTION 1 FUNCTION OUTLINE

When executing the tool breakage detection and automatic tool length compensation, the tool tip must be brought into contact with the touch sensor. However, with some cutting tools, it becomes necessary to orient the spindle in a position other than the one supported as the standard spindle orientation position.

The function covered by this specification orients the spindle at such positions so that the tool breakage detection and automatic tool length offset function can be used for special tools.

The illustration below shows the spindle orientation positions.



SECTION 2 MULTIPLE-POINT SPINDLE ORIENTATION FUNCTION

(1) Spindle Orientation Direction

The direction in which the spindle is rotated to orient the spindle is determined by the M code commands.

- M19 Spindle orientation is executed in the spindle forward rotation direction.
- M118 Spindle orientation is executed in the spindle reverse rotation direction.
- M119 Spindle orientation direction is not definite; to shorten the spindle orientation time, spindle orientation is executed in the direction the spindle orientation position lies further from the present spindle position.
- (2) Multiple-point Spindle Orientation

The multiple-point spindle orientation function is called by M118 and M119 in addition to M19. The explanation is given assuming M19.⁻

(a) The command format is:

M19 RS = θ

- (b) θ represents an angle in units of degrees with decimal fractions rounded off.
- (c) Programmable range is from 0° to 360°

The required orientation angle θ is defined in reference to the spindle position when the tool change is carried out and the angle is measured in the spindle forwardfor rotation direction.



- [Supplement]
- When RS command is omitted after M19 (or M118 or M119), the spindle position selected for the tool change cycle execution is used as the spindle orientation position.

In other words, omission of an RS command is equivalent to M19 RS = 0.

2. Note that the RS command must always be programmed in the same block with the spindle orientation M code command (M19, M118, M119).

If an RS command is designated independently, it does not generate an alarm and the answer signal is turned on immediately. This command is cleared by the M19 command designated next.

RS commands are handled as a modal command.

SECTION 3 TOOL BREAKAGE DETECTION AND AUTOMATIC TOOL LENGTH OFFSET FUNCTIONS

The spindle can be oriented in the direction so that the special tool tip (boring bar, etc.) may be brought into contact with the touch sensor to execute the tool breakage detection and automatic tool length offset functions.

(1) Command Format

To execute the tool breakage detection or automatic tool length offset program using the multiple-point spindle orientation function, following format is used:

CALL OO30 PRS = θ

Designate the required spindle orientation angle in reference to the spindle position selected for tool change cycle and measured in the spindle forward rotation direction.

[Supplement] Difference between CALL OO30 and CALL OO30 PRS = θ

CALL OO30	The spindle is orient stopped at the position where the spindle has been oriented for the execution of the tool change cycle. If the spindle orientation function is elected, the spindle orientation pin is inserted at the spindle orientation position.
CALL OO30 PRS = θ	The spindle is orient stopped at the designated angle (θ). The spindle orientation pin is not inserted after the orientation even when the spindle orientation pin is available. Note that the pin is inserted when the orientation angle is 0° ($\theta = 0$).

R. INDEX TABLE EXTERNAL MANUAL OPERATION FUNCTION
SECTION 1 OVERVIEW



Fig. 1

The external manual operation panel is mounted at the front of the machine to turn the index table manually at the setup position to facilitate setup for the machine equipped with shield cover, etc.

- (1) Turn the B-AXIS MANUAL switch in the ON position and the control is placed in the manual B-axis operation mode.
- (2) Note that the control is reset in this case.
- In the manual B-axis operation mode, the control is in the following alarm state.
 ALARM C 3225 External manual operation
- (4) Turn the B-AXIS + / switch in the required position and press it.
- (5) The B-axis rotates in the selected direction while the switch is held down.
- (6) Positioning of B-axis is completed after command pulse is turned off.
- (7) After the completion of setup, return the switch to the OFF position.

SECTION 2 OPERATION



(1) B-AXIS MANUAL Switch

The manual B-axis operation mode is turned on and off according to the setting of the B-AXIS MANUAL switch.

Note that changing this switch setting resets the control.

In the manual B-axis operation mode, the control is in the following alarm state.

ALARM C 3225 External manual operation

In this alarm state, index table operation using the operation panel has the top priority to any other axis motion commands. In the manual mode, for example, normal manual rapid feed, manual cutting feed, or pulse handle feed is not possible. This is to ensure safety of operation.

Place the switch to the OFF position only after confirming that the index table has been clamped.

(2) B-AXIS + / - Switch

This switch determines the direction of index table rotation and rotates the index table.

(a) Rotating direction

Determine the index table rotating direction by placing the switch in the "+" or "-" position.

- (b) Index table rotation
- 1) When the switch is pressed, the index table rotation signal is turned on.
- 2) The index table keeps rotating in the selected direction while the switch is held down.
- 3) When the switch is released, the rotation signal is turned off; the table is positioned and clamped at the first 1° or 5° interval position.
- 4) The indexing angle interval conforms to the machine specification.

(3) CYCLE START Button

This button is the same as the CYCLE START button on the machine operation panel.

(4) SLIDE HOLD Button

This button is the same as the SLIDE HOLD button on the machine operation panel.

SECTION 3 SUPPLEMENTS

(1) When the B-AXIS MANUAL switch is turned from ON to OFF or from OFF to ON, the control is reset.

Therefore, place this- switch in the OFF position after confirming that the index table has been clamped. If it is placed to the OFF position while the index table is rotating, the index table stops rotating in the unclamped state.

To release this table unclamped state, refer to (5).

(2) When the manual B-axis mode is selected, axis feed (manual rapid feed, cutting feed, pulse handle feed, etc.) is not possible even in the manual mode.

Set the B-axis manual mode setting switch OFF to carry out normal manual operation.

This interlock is provided to ensure the safety during operation; when the manual B-axis mode is selected, an operator will be carrying out setup in the machine.

(3) Index table positioning is carried out in intervals of 1° or 5° according to the 1° index or 5° index specification.

Therefore, there may be slight delay until the B-axis starts decelerating after the B-axis motion button is released.



- (4) When the manual B-axis operation mode is selected, the axes being unclamped for pulse handle operation, etc.are clamped.
- (5) If the control is reset while the index table is rotating, the index table stops and remains unclamped.

In this state, although manual B-axis operation is possible, B-axis is not clamped.

(6) To clamp the B-axis, turn the B-AXIS MANUAL switch off, select the MDI or automatic mode and designate an index table operation command. The B-axis is clamped after the completion of positioning.

S. AUTOMATIC SCHEDULE PROGRAM UPDATE FUNCTION

SECTION 1 OVERVIEW

- (1) The OSP can use a schedule program so that different types of workpieces can be machined continuously using a pallet changer, etc. without operator's intervention.
- (2) Recently, flexibility of a schedule program, i.e., addition and/or change of a schedule program, is required for schedule operation using a schedule program to meet diversified needs in production.
- (3) The automatic schedule program update function allows an operator to add or change the schedule program safely without stopping schedule operation by designating the autofflatic update and registering the schedule program.

SECTION 2 RESTRICTIONS TO SCHEDULE PROGRAMS

Because the automatic schedule program update function, updates the schedule program while the machine is operating according to the schedule, the following restrictions must be observed to ensure safe operation.

The blocks which can be used in the schedule program are:

- PSELECT block Selects and executes the main program.
- VSET block Sets variables.
- END block Ends the schedule program.

Other blocks such as GOTO and IF blocks cannot be used.

SECTION 3 OPERATION

1. Designating Automatic Schedule Program Update

Designation of automatic schedule program update is made using the option designation when selecting the schedule program.

Procedure:

- (1) Select the automatic mode.
- (2) Press function key [F8] (EXTEND) two times.

The functions assigned to function keys are changed; the SEARCH NUMBER function is assigned to function key [F1].

(3) Press function key [F4] (SP-SELECT).

The "SS" prompt is displayed on the screen.

(4) Key in [*] and [;].

SS_*;

(5) Press the [WRITE] key.

The screen transfers to program selection directory screen, and the schedule program files in the memory are displayed.

- (6) Move the cursor to the required file name.
- (7) Press the [WRITE] key.
- (8) The following message is displayed on the screen.

What Is the option?

Key in "U".

The required schedule program is selected and at the same time the automatic update function is called.

- [Supplement] 1. Option designation
 - U: Automatic update function effective
 - N: Automatic update function ineffective

The option designation can be omitted. Default for option designation is set with the NC optional parameter (word) No. 13.

- No. 13: 0 N (automatic update function ineffective) 1 U (automatic update function effective)
- 2. It is possible to designate the option using the keyboard directly.

SS_ (schedule program file name) ;U [WRITE]

2. Editing Schedule Program

A schedule program can be edited in the same manner as editing machining programs, etc. However, the schedule program which is currently run or the schedule program registered for automatic editing cannot be edited directly. Copy the schedule program to other file to edit it.



[Supplement] The automatic update function compares the schedule program being run when it is registered and with the schedule program to be registered. If the contents in the blocks preceding the currently executed block do not agree with each other, registration is not possible. Therefore, addition and change for blocks to be executed later are allowed in program editing operation.

3. Registering Schedule Program to be Updated

Procedure:

- (1) Select the automatic mode by pressing AUTO key.
- (2) Press function key [F8] (EXTEND) two times.

The functions assigned to function keys are changed; the NEW SP ENTRY function is assigned to function key [F6].

(3) Press function key [F6] (NEW SP ENTRY).

The "SE" prompt is displayed on the screen.

(4) Key in [*].

SE *

(5) Press the [WRITE] key.

The screen transfers to program selection directory screen, and the schedule program files in the memory are displayed.

- (6) Move the cursor to the required file name.
- (7) Press the [WRITE] key.

After the completion of registration, the following message is displayed.

***.SDF entry finished

This registers the schedule program used to update the current schedule program. After the completion of execution of the current block, and before the execution of the next block, the registered update schedule program is selected automatically. Machining is continued with the new schedule program.

[Supplement] 1. The schedule program file name can be directly keyed in to be registered.

SE_ (schedule program file name) [WRITE]

2. To cancel the registered schedule program file, append option ;C to the registration command in the same manner as registering a schedule program.

Example: SE_*;C SE_S1.SDF ;C

When the registered schedule program file is canceled, the following message is displayed.

***.SDF entry cancel finished

3. The registered schedule program file is canceled when program selection or schedule program selection is carried out newly.

SECTION 4 SCREEN DISPLAY

The registered schedule program file names, etc. are displayed on the RUN GUIDE screen in the program operation mode.

									<u> </u>
(PROG 0	PERATION		RUN (SUIDE				
	R	UNNING FI	LE		SEL	ect erro	87/07/15 R FILE	14:10:00	
	SCHEDUL SCHEDU OMF.SD MAIN F OMF-4V MAIN P EMPTY RUNNIN S-Mtd SCHEDU UPDATE OMF1.S	E MODE ILE PROGRA IF ILE NAME (AA. MIN (ROGRAM N/ IG METHOD ILE FILE I D NEXT IDF	AM FILE AME NAME TO	RUNN ING BE	MAIN FIL EMPTY MAIN PRO EMPTY SUB FILE EMPTY SELECTED	E NAME GRAM NAN NAME RUNN I NG	NE METHOD		
	=EX =GD =				l 				_
	TIME	 INIT	DERETE	RENAME			RUN GUIDE	EXTEND	
	(F 1	D(F 2	DE	3) F 4) (F 5) <u>F 6</u>) (F 7) <u>F</u> 8)	

SECTION 5 ERROR MESSAGES

5356 Schedule program update mode

[Character-string] None

[Code]

- 1-> Registration of automatic update is made for the schedule program which has been selected without the "automatic update" designation.
- 2-> SP number search, using the function key or cursor key, was made for the program which has been selected with the "automatic update" designation.
- 3-> The program selected with the "automatic update" designation contains the GOTO or IF block.

5357 Schedule program update entry

[Character-string] None

[Code]

- 1-> The designated schedule program is not found.
- 2-> A schedule program has already been registered.
- 3-> The GOTO or IF block is used.
- 4-> The program (machining processes) is not identical in the range from the head of the program to the registration point.

5225 File change inhibit

An attempt to edit the protected file

[Character-string] File name attempted to edit

[Code] None

[Measures to Take]

Release the protect, or cease the file change operation. How to release the file protection (for A.MIN):

PROT A.MIN;C

T. ADDITIONAL AXIS (ROTARY AXIS) FUNCTION

SECTION 1 NORMAL OPERATION SPECIFICATION

1. Outline

Rotary tables have infinite movement range, and, differing from linear axes (X, Y, Z), they return to their original position in a rotation of 360°. From this special feature, rotary axes are programmed in the different manner from linear axes. This section provides programming rule for rotary axes so that readers can program rotary axis movement correctly and easily.

2. Axis Nomenclature

A-axis:	Axis rotating around X-axis
B-axis:	Axis rotating around Y-axis
C-axis:	Axis rotating around Z-axis

3. Program Format

3-1. Least Setting Unit

The least setting unit for the NC rotary table is 0.001° or 0.0001° . (Varies depending on the machine specification.)

3-2. Programmable Range

Absolute value :	0 - 360.000 (0.001 ° specification)
	0 - 360.0000 (0.0001° specification)
Incremental value:	\pm 360.000 (0.001 $^{\circ}$ specification)
	±360.0000 (0.0001 $^\circ$ specification)

3-3. Rotation Direction Command

Rotation direction commands should be programmed in conformity with ISO-841, ISO-1056 and JISB6310.

Command Method	Tape Command	Rotation Direction
Absolute command	M15	Forward direction
(4th axis)	M16	Reverse direction
Absolute command	M115	Forward direction
(5th axis)	M116	Reverse direction
Incremental command	+	Forward direction
	-	Reverse direction

Note: M codes used for determining the rotation direction of a rotary axis are all modal. M15, M16, M115, and M116 Installation Position and Rotation Direction of NC Rotary Table:

(1) Vertical Machining Center (MC-V)





: When controlling the NC rotary table as A-axis, it may not be installed to the left side of the spindle viewing the spindle from front due to the interference with the ATC.

C-axis

(2) Horizontal Machining Center (MC-H)



(3) Double-column Machining Center





X



Parameters for Rotary Axes

Rotary axes (the B-axis of the MC-H series machining centers not included) can be defined as A-, B-, or C-axis depending on the installation direction.

Axis name, rotation direction, and correspondence to the basic axis can be set using parameters.

Procedure:

- (1) Press the PARAMETER key to select the parameter setting mode.
- (2) Press function key [F8] (EXTEND) repeatedly until the "ROTARY AXIS PARAMETER" screen is displayed.

(3)

PARAMETER SET				7/07/15 1	4-10-00
	* ROTARY AXIS	PARAMETER	ع (*	//0//15 1	4.10.00
1. AXIS SELECT					
	AXIS NAME	REV. D	IR. (REVE	rse)	
4 AX1\$:	A	1			
5 AXIS:	*	0			
6 AXIS:	*	0			
AXIS NAM A B C	E CORRESPO 	NU BASE AX	.12		
ACT POSIT(WORK)	× 100.0	00 200	. 000	Z 300.000	
=		-1	1	1	
SELECT			ITEM	ITEML	EXTEND

F1F2F3F4F5F6F7FB

Move the cursor to the data column where data needs to be set using the cursor control keys and press function key [F1] (SELECT).

Set data at the following parameters.

AXIS NAME	:	A, B, C, * (no setting)
REV. DIR.	:	0 (forward rotation), 1 (reverse rotation)
CORRESPOND BASE AXIS	:	\pm Xp, \pm Yp, \pm Zp, $***$ (no setting)

(a) AXIS NAME

Select a name for the rotary axis to be used, axis names for rotary axes can only be used.

- [Supplement] 1. When more than one rotary axis is used, do not apply the same axis name to these axes.
 - 2. With the animated drawing specification, the rotary axis designated as drawing data is canceled when the axis name in parameter data has been changed. Reset the rotary axis in drawing data.

(b) REV. DIR.

Arrow directions as indicated below are defined as forward rotation directions of rotary tables with respect to the basic axes by ISO. Refer to these directions when selecting a rotation direction.



- [Supplement] When the rotation direction is changed, actual position data is also changed. In such cares, reset machine zero point data and work zero point data after the completion of step (4) below.
 - (c) CORRESPOND BASE AXIS

Select a corresponding basic axis when machining the periphery of a cylinder. For details, refer to U.

- (4) After necessary data has been set, press function key [F7] (BACKUP), turn off the power, and then turn on the power again.
 - Note 1 Data set on the screen is not effective unless the power is once turned off and then turned back on again.
 - Note 2 When the power is turned off without pressing, function key [F7] (BACKUP), data set on the screen will be lost.

NOTICE

 If new data settings are not registered by the step (4) procedure above, operation will occur according to the previous rotary axis parameter setting, resulting in an unexpected operation.
 When the rotary axis parameter settings are changed, be sure to execute the step (4) procedure.

3-4. Program EXamples

Example 1:

N1	G90	G0	C0	M15
N2			C0	

In N2 block, rotary table does not rotate since the same point is specified for C-axis.

Example 2:

N3 G90 G0 C0 M15 N4 G91 G1 C360 F36

In N4 block, the table rotates in the forward direction since positive value is specified for C-axis command in incremental mode. It makes one full turn in 10 minutes.

Example 3:

N5	G90	G0	C0	M15
N6			C200	

Example 4:

N7	G90	G0	C0	M15
N8	G91		C–160	

The rotary table carries out the same operations in the programs indicated in examples 3 and 4.

In example 3, the rotary table rotates in the reverse direction (M16) up to 200° position. That is, the rotary table rotates in the reverse direction by 160° .

In example 4, the commands are given in incremental values. The minus sign of the C command designates the reverse rotation direction, and thus the rotary table rotates in the reverse direction by 160° .

Example 5:

(A)	N9	G0		C0	Y200	M16
(B)	N10	G91	G1	C360	Y–50	F36
(C)	N11			C360	Y–50	



With the commands in blocks N10 and N11, Y-axis moves 100 mm (3.94 in.) while the rotary table makes two full turns (720°) and the resultant tool paths are as indicated above.



Example:



The commands in block N2 generate tool paths as indicated above.

Y-axis moves from point A to point B while the rotary table makes one turn.

The time T required for the execution of the commands is calculated as indicated below:

In the simultaneous interpolation mode of the C-axis with other linear axis, feedrate unit may be considered as:

1 mm/min = 1°/min

For the program example, time T may be calculated as below:

```
C-axis: 360 mm (= 360°)
Y-axis: 100 mm
```



Distance between point A and point B:

 $D = \sqrt{(360)^2 + (100)^2} = 373.631$

The control assumes the axis travel amount as D = 373.631 mm.

Since feedrate is programmed as F36 (mm/min), required time T is

T= D/F = 373.631/36 = 10.379 min

Therefore, the time required for the execution of the programmed commands is 10 min 23 s.

3-6. Calculating Feedrate Command F

(1) When Cutting is Made with Feeding Only A-axis



Assume the workpiece diameter is D = 200 mm, to obtain a feedrate of 150 mm/min on the workpiece OD, the feedrate to be programmed should be calculated in the following manner.

(a) One turn of a workpiece is equal to the cutter travel amount (L) as calculated below.

 $L = \pi \times D = 3.14 \times 200 = 628 \text{ mm}$

 (b) Time required for this cutting is calculated as indicated below if feedrate at the cutting point is f = 150 mm/min.

 $T = L/f = 628 \div 150 = 4.19 min$

(c) The travel amount L_{NC} calculated inside the control is

 $L_{NC}=360^\circ=360~mm$

(d) The feedrate to be programmed as an F word is calculated as below:

 $F = L_{NC}/T = 360 \div 4.19 = 86$

That is, programming F86 will provide the expected actual feedrate, 150 mm/min.

To simplify the calculation of a feedrate F, use the following formula:

 $F = \{360/(\pi \times D)\} \times f \dots (1)$

where,

- f = feedrate required at the cutting point
- D = workpiece diameter
- n = ratio of circumference of circle to its diameter

With this formula, feedrate in the example above is calculated as

 $F = \{360/(3.14 \times 200)\} \times 150 = 86$

For B- and C-axis, feedrate can be calculated using the same formula, For C-axis however, use two times the distance "r" between the table rotation center and the cutter position as "D".



(2) When Cutting is Made by Simultaneous A- and X-axis Movement

The example below gives the procedure how to determine the feedrate when feeding A-axis by 120° and X-axis by 150 mm in the simultaneous 2-axis control mode at a feedrate of 150 mm/min. In this example, workpiece diameter is D = 200 mm.

(a) The actual cutter travel distance L on the workpiece is

$$L = \sqrt{(\pi \times D \times a/360)^2 + x^2} = \sqrt{(3.14 \times 200 \times 120/360)^2 + 150^2} = 258 \text{ mm}$$

(b) Time required for this cutting is calculated as indicated below if feedrate at the cutting point is f = 150mm/min.

$$T = L/f = 258 \div 150 = 1.72 \text{ min}$$

(c) The travel amount L_{NC} calculated inside the control is

$$L_{\rm NC} = \sqrt{a^2 \times x^2} = \sqrt{120^2 \times 150^2} = 192$$
 mm

(d) The feedrate to be programmed as an F word is calculated as below:

$$F = L_{NC}/T = 192 \div 1.72 = 112$$

That is, programming F112 will provide the expected actual feedrate, 150 mm/min.

To simplify the calculation of a feedrate F, use the following formula:

$$\mathsf{F} = \{\sqrt{a^2 \times x^2} / \sqrt{(\pi \times \mathsf{D} \times a/360)^2 + x^2} \} \times \mathfrak{f} \dots (2)$$

where,

f = feedrate required at the cutting point

- D = workpiece diameter
- n = ratio of circumference of circle to its diameter
- a = incremental value of A-axis command (°)
- x = incremental value of B-axis command (mm)

Entering $a = 360^{\circ}$ and x = 0 mm, will yield formula (1).

If the simultaneous 2-axis control with B- and Y-axis is required, replace "a" and "x" with "b" and "y", respectively. Here, "b" represents the incremental value of B-axis and "y" the incremental value of Y-axis.

Simultaneous control including C-axis cannot provide a constant cutting speed at the cutting point because distance between the rotation center of C-axis and cutter changes as cutting progresses. In this case, divide the total cutting area into several segments in which change of distance is relatively small and calculate approximate feedrate F for each of these segments.

$$\mathsf{F}' = \{\sqrt{\mathsf{c}^2 \times \mathsf{x}^2} / (2 \times \pi \times \mathsf{R}' \times \mathsf{C}/360)\} \times \texttt{f} \cdots \cdots \odot \texttt{3}$$

where,

C = incremental value of C-axis command (°)

R' = average value of distances between the C-axis rotation center and the cutter within a segmented area

Note that formula ③ is applicable only to the cases where simultaneous 2-axis control between C-axis and X-axis, and that X-axis passes the center of the C-axis rotation, and R' is relatively large compared with x.



4. Alarm

If an alarm occurs with a rotary axis, the ALARM lamp lights up and the corresponding alarm number and alarm message are displayed on the CRT.

1008 SVP DIFF over

When the axis is in the target position or at a standstill, the target value, operation value, and actual position data are the same. With designation of an axis moving command, the NC starts calculating the path to the target position, thus creating operation values (RCON).

This time, the actual position data follows the operation values with a certain delay.

This delay amount obtained by subtracting [actual position] from [operation value] is termed following error. If the following error exceeds the allowable limit, this alarm occurs.

[Index] AXIS

[Code] Absolute DIFF value in hexadecimal [encoder unit]

[Probable Faulty Locations]

- 1) Mechanical trouble of ball screw, bearing or any other parts, which makes the axis movement heavy
- 2) Encoder, motor or drive unit of the problem axis
- 3) Servo parameters of the problem axis
- 4) SVP hardware of the problem axis

1002 SVP CON APA deviation

Deviation between CON and APA has exceeded the preset limit.

The deviation can be obtained by the formula:

[Calculated value written in SVP] – [Actual position read from SVP].

This value is monitored by the control software such as that for NC.

The SVP usually monitors DIFF. However, when the SVP becomes unable to monitor DIFF due to fatal abnormality, this monitoring function will prevent the system overrun.

[Index] AXIS

[Code] Absolute deviation value in hexadecimal [reference unit]

[Probable Faulty Locations]

SVP of the problem axis

1006 SVP APA speed

The position encoder value (RAPA) changes according to the motor speed. The NC checks the variation of position encoder values at regular intervals. If the feedrate of each axis is abnormally high compared with the normal rapid traverse rate, or the value changes in abnormal manner due to malfunction of the position encoder, the variation per unit time exceeds a fixed allowable limit, causing this alarm.

[Index] AXIS

[Code] Variation of the position encoder values in hexadecimal [encoder unit /3.2ms]

However, the following codes represents special meanings.

FFFFFFF-> The variation has exceeded the liner axis stroke.

FFFFFFE-> The variation has exceeded the rotary axis stroke (360°), or the positional data of the infinite axis has changed to a negative value.

FFFFFFD-> The variation has exceeded the infinite axis stroke.

[Probable Faulty Locations]

- 1) Position encoder of the problem axis or the cables connected with the encoder.
- 2) Motor for driving the problem axis
- 3) Drive unit of the problem axis
- 4) SVP hardware of the problem axis

2359 Data word: rotating axis command

Wrong rotating axis command

[Code]

- 1-> Rotary table or index table command programmed during the following modes or programmed simultaneously with a G code establishing them.
 - Circular interpolation mode (G02, G03) [index table only]
 - Tool length offset mode (G54 G59)
 - Cutter radius compensation mode (G41, G42)
 - Three dimensional compensation mode (G44)
 - Figure enlargement or reduction mode (G51)
 - Coordinate computation mode
 - Area machining mode
 - Note: Note: In the cylindrical side surface machining mode (G175), this alarm occurs only when the tool length offset command or 3-D tool compensation command is specified.
- 2-> Index table command programmed during the following modes or programmed simultaneously with a G code establishing them.
 - Skip function (G31)
 - Local coordinate system specification mode (G11)
 - During copy function
 - Work coordinate system specification command (G92)
- 3-> Index axis command programmed simultaneously with another axis during G01 mode.
- 4-> Command for a rotary axis of the circular table which has been removed.
- [Probable Faulty Locations]

Wrong G code active when the rotary axis command is specified.

[Measures to Take]

Check the error condition, and then specify the correct command.

SECTION 2 MULTI-TURN SPECIFICATION

1. Outline

Operations of the multi-turn NC rotary table can be commanded in the same manner as linear axes. The rotary table rotation direction commands (M15 and M16) are no longer effective, and the rotation direction is determined based on the relationship between the actual position coordinates and the target position coordinates.

Mirror image function is also usable for the multi-turn NC rotary table.

2. Terminology

In this section, the terms "position" and "coordinate" are defined as indicated below.

Position:

This indicates the physical position of the NC rotary table. That is, position data is expressed by any value between 0° and 360° .

 $0^\circ,\,360^\circ,\,and\,720^\circ$ indicate the same position.

Coordinate:

This indicates the theoretical coordinate position of the NC rotary table.

 $0^\circ,\,360^\circ,\,and\,720^\circ$ indicate different coordinate position.



These indicate physically the same position.

3. Program Format

3-1. Least Setting Unit

The least setting unit for the NC rotary table is 0.001° or 0.0001°.

(Varies depending on the machine specification.)

3-2. Programmable Range

-9999.999° - +9999.999° (0.001° specification) -9999.9999° - +9999.9999° (0.0001° specification)

3-3. Rotation DireCtion Command

If the programmed target position coordinate is larger than the actual position coordinate, the rotary table rotates in the forward direction. Conversely, if the target position coordinate is smaller than the actual position coordinate, the rotary table rotates in the reverse direction.



3-4. Feedrate Commands

Refer to SECTION 1, 3-5 and 3-6.

4. **Operations**

4-1. Automatic and MDI Modes

Commands, operations, and actual position display greater than 360 $^\circ$ (one turn) is possible regardless of the absolute or incremental mode.

Example:

N001	G90	G0	C0.0
N002	C540	0.0	
N003	G91	C540	.0

N001 Positioning of C-axis is made at 0° coordinate.

- N002 Positioning of C-axis is made at 540° coordinate. That is, the rotary table rotates by one and a half turns to make positioning at 180° position.
- N003 Positioning of C-axis is made at 1080° coordinate. That is, the rotary table rotates by one and a half turns from the current position to carry out positioning at 0° position.

Mirror image:

(1) In Absolute Mode (G90)

Dimension commands are reversed with the work zero point taken as the center of mirror image function.

Example: Mirror image C-axis ON

N001	G90	C–20
N002		C540



With the program above, C-axis is first positioned at + 20° coordinate and then rotates by 560° in the reverse direction to carry out positioning at -540° coordinate.

(2) In Incremental Mode (G91)

Dimension commands are reversed with the current NC rotary table position taken as the center of mirror image function.

Example: Mirror image C-axis ON



4-2. Manual Mode

Positioning is carried out based on the coordinate values in the same manner as with a linear axis.

4-3. Manual Interruption and Pulse Handle Overlap

Same as in the manual mode

5. Parameter Setting

5-1. Parameters

NC rotary table multiple turn effective/ineffective is set at the NC optional parameter (bit) No. 39.

Bit	bit 7	bit 6	bit 5	bit 4	bit 3	bit 2	bit 1	bit 0
No. 39			Multi-turn C-axis	Multi-turn B-axis	Multi-turn A-axis			

[Supplement] After the completion of setting, turn off the power supply once and then back it on again.

5-2. Rotary Axis with Limit

The operation of the NC rotary table with limit can be commanded in the same manner as linear axes, as in the case of the multi-turn NC rotary table. The relationship between the rotary table with limit and the multi-turn NC rotary table is as indicated below.

- (1) When both the rotary table with limit and the multi-turn NC rotary table are selected, the rotary table with limit is given priority.
- (2) When more than one multi-turn NC rotary table is used and the one is installed on the other, the lower table is recognized as a rotary table with limit.

6. Actual Position Data Display

The actual position data of NC rotary tables is displayed in coordinate values (-9999.999 - +9999.999° or -9999.9999 - +9999.9999°).

However, on the following screens, the data is displayed within 360°.

(1) Actual Position Display (2nd page)

/					
(AUTO OPERATION	TST	9.MIN	OTST9 N13	16
	ACTUAL POSITION			97/0//15_14: 1mm	10:00
	LOCAL COORDINATES WORK COORDS (APA) WORK COORDS MACHINE COORDS FEEDBACK COORDS TARGET VALUE DISTANCE REMAINING	X 39.500 39.500 39.500 39.500 2539.500 39.500 0.000	Y 185. 850 185. 850 185. 850 185. 850 2685. 650 185. 850 0. 000	Z -1100.000 -1100.000 -1100.000 -1100.000 1400.000 -1100.000 0.000	
	MANUAL SHIFT ACTUL MANUAL SHIFT TOTAL PITCH ERROR COMP	0.000 0.000 0.000	0.000 0.000 0.000	0.000 0.000 0.000	
	A-M =P0 =P0 =P0	td			
	PROGRAM ACTUAL PAR SELECT POSIT. PRO	T BLOCK Gram Data	SEARCH	CHECK DATA	[EXTEND]
	F1 F2	F3)(F4		-6(F7)	F 8)
			Fig. 1		

WORK COORDS (APA) is the position data display within 360°.

(2) NC Axis Data



RCON and RAPA are the position data display within 360°.

7. Power On/Off and NC Reset

Coordinate value of the NC rotary table on the work coordinate system is changed to any value within 360° corresponding to the actual coordinate value by the following operations.

- (1) Power ON/OFF
- (2) Machine lock ON to OFF
- (3) NC reset to clear alarm A

It is possible to select whether or not the coordinate value (on the work coordinate system) of the NC rotary table is changed to any value within 360° by the NC reset operation for other than clearing alarm A, using the parameter (NC optional parameter (bit) No. 2).

Bit 1 of NC optional parameter (bit) No. 2:

1 Reset to a value within 360°

0 Not reset

Note that the "NC reset" includes the following operations:

- Pressing the NC RESET button
- Mode reset operation (Resetting the mode by selecting the manual mode on the operation panel)
- Machine lock ON/OFF operation
- Reset by M code (M02, M30)

8. Sequence Restart and Home Position

8-1. Sequence Restart

With the multi-turn NC rotary table, return to the restart coordinate is accomplished within 360° movement by the sequence restart operation. In other words, positioning at the sequence restart point is carried out within one turn from the actual position and that position is taken as the restart coordinate.

Direction of the table rotation is determined based on the relationship between the actual coordinate and the sequence restart coordinate.

Example:



Assume that the actual coordinate is "1080".

N198 X50 N199 G90 C540

If restart is to be carried out from N199, the positioning at the sequence restart position occurs in the following manner.

Since the restart coordinate is 540°, the table rotates in the reverse direction by 180° from 1080° coordinate to carry out positioning at 900°. Then, the actual coordinate is changed from 900° to 540°.

8-2. Home Position

8-2-1. Positioning at Home Position

The multi-turn NC rotary table carries out positioning at the home position in the shortest path (within 360° turn).

Example:



Assume that 0° position is the home position.

(1) When the actual coordinate is 960°:

Positioning is carried out at the home position coordinate 1080° (0° position) in the forward rotation.

(2) When the actual coordinate is 840°:

Positioning is carried out at the home position coordinate 720° (0° position) in the reverse rotation.

(3) When the actual coordinate is 900° :

The case where the actual coordinate is equidistant from the two home position coordinates:

Positioning is carried out at the home position coordinate 1080° (0° position) in the forward rotation. That is, if the rotary table is equidistant from the two home positions, positioning at the home position is carried out in the forward direction.

8-2-2. Incremental Command After Positioning at Home Position

When an incremental command is specified in the block immediately following the home position command (M60, G30), the incremental command is executed in reference to the coordinate which the rotary table took just before the execution of the home position return command.

Example:



Assume that 0° position is taken as the home position.

N101	G90	C960
N102	M60	
N103	G91	C100

N101 Positioning at 960° position

N102 Positioning at the home position (1080°)

N103 \ldots . Positioning at 1060° (960 + 100) from 1080°

9. Work Zero Offset

The setting range of the work zero offset is indicated below:

-9999.999 - +9999.999° or -9999.9999 - +9999.9999°

When the G92 command is specified, the work zero offset value is calculated so that the actual coordinate will take the coordinate value specified.

10. Limit Check

The setting is made so that rotary table movable range is as indicated below on the machine coordinate system (H = 0).

 $-9720 - +9720^{\circ}$ (actual coordinate display, for \pm 27 turns)

Therefore, on work coordinate systems, the limit value is increased/decreased by the value equivalent to the work zero offset amount.



11. Installation of Multi-turn NC Rotary Table

Use of a multi-turn NC rotary table requires the setting of system parameters as explained in SECTION 2, 11-1.

If the NC rotary table is *to* be rotated while the machine zero offset has not been set, rotate it only in the forward direction.

If it is rotated in the reverse direction while the machine zero offset has not been set, an alarm occurs (alarm A 1243 CON 0 pass). When the alarm lamp is illuminated, turn off the power supply to the NC once and then turn it back on.

After the installation of a multi-turn NC rotary table, set the system parameters. (Refer to 11-1.) After the system parameters have been set, always backup the set data by pressing function key [BACKUP]. Note that the set system parameter data will be cleared if power to the NC is turned off before the data backup operation.

The data backup operation is explained below:

		•				
					97/07/15 1	4:10:00
⇒BA						
				T		
READ	PUMCH				BACKUP	[EXTEND]
F	1 F 2 F	3 F	4)(F	- 5)(F	6)(F 7)	(F B)

- (1) Press any one of the mode selection keys; ZERO SET, TOOL DATA, and PARAMETER.
- (2) Press function key [F8] (EXTEND).

This displays the screen as indicated above.

(3) Press function key [F7] (BACKUP).

A prompt "= BA" is displayed on the console line and data backup is executed continuously. After the completion of backup of data, a prompt "=" appears on the console line.

This backs up the data.

NOTICE

After setting system parameters, carry out data backup operation, turn off the power supply once and then turn it back on again.

11-1. Setting System Parameters

11-1-1. Machine Zero Offset

Calculation, setting, and addition of the machine zero offset data for a multi-turn NC rotary table are carried out as explained below.

(1) Calculation

Machine zero offset (within 360°) is calculated from the actual position (CON within 360°) of the multi-turn NC rotary table.

Calculation is made so that the machine zero offset data always falls within within 360°.

(2) Setting

Set the value which will fall within within 360°.

(3) Addition

The value set is added to the present machine zero offset data.

Carry out this operation in such a way that the value always falls within the range of 0 and 359.999°.

11-1-2. Home Position Data

The home position data for the rotary table should be within the range below:

within 360°

Note that the setting of home position is not necessary if not required.

12. Alarm List

1243 CON 0 pass

The SRSCON (final command value) value becomes negative. This alarm occurs if the SRSCON goes to the minus side from point A.

[Index] AXIS

[Code] SRSCON value (hexadecimal) at an occurrence of alarm

- [Probable Faulty Locations]
 - 1) Setting error of system parameter (machine zero)
 - 2) Wrong command value (point left to the machine zero is specified)

[Operation Example] An alarm occurs when X-axis moves beyond X0 under the following setting:

Machine zero : 2000.000

-travel limit : -2500.000

Programmed command : G0 X-2001.000

[Measures to Take]

- 1) Reset the machine zero at system parameter.
- 2) Change the command value (must be to the plus side from the machine zero).
U. CYLINDER SIDE-SURFACE MACHINING FUNCTION

SECTION 1 OUTLINE

This function processes rotational axes in the same manner as linear axes, enabling the planes including the rotational axis to be selected. Cylinder side-surface machining can be executed by simply designating the usual plane machining image on the plane which is obtained by developing the cylinder side-face.



 a) Rotational axis is used to execute plane development of the cylinder side-surface.



 b) The plane which is obtained by developing the cylinder side-surface is designated using a G code (G17, G18, or G19).



c) The same image as that used for normal plane machining is designated, and machining begins.



 d) Cylinder side-surface machining proceeds.

SECTION 2 PROGRAMMING FORMAT

Follow the procedure below to execute cylinder side-surface machining:

- (1) Select the plane which includes the rotational axis (G17, G18, or G19).
- (2) Turn on the cylinder side-surface machining mode (G175R_).
- (3) Designate machining commands. (For details regarding machining commands, refer to SECTION 2, 3. "MA- CHINING COMMANDS".)
 - Designate rotational axis commands in units of "degrees".
 - Designate F commands in terms of the feedrate on the plane which is obtained by developing the cylinder side-surface.
- (4) Turn off the cylinder side-surface machining mode (G174).

1. Cylinder Side-Surface Machining Mode

- Cylinder Side-Surface Machining Mode ON

```
G17 Xp_Yp_
G18 Zp_Zp_
G19 Yp_Zp_ 
G19 Yp_Zp_
```

- G17 : XpYp Plane selection
- G18 : ZpXp Plane selection
- G19 : YpZp Plane selection
- Xp : X-axis, U-axis, or the rotational axis whose corresponding basic axis is X-axis
- Yp : Y-axis, V-axis, or the rotational axis whose corresponding basic axis is Y-axis
- Zp : Z-axis, W-axis, or the rotational axis whose corresponding basic axis is Z-axis (For details regarding the "corresponding basic axis", refer to SECTION 2, 2 which

follows.)

- G175 : Cylinder side-surface machining mode ON
- R : Cylinder radius (1 5000 mm)

The axes included in the selected plane are determined by the axis address in the block where plane selection is designated.

Example:

G17 A_X_ (A = -Y)	AX plane
G17 B_Y_ (B = X)	BY plane
G18 C_Z_ (C = -X)	CZ plane
G18 A_X_ (A = Z)	AX plane
G19 B_Y_ (B = –Z)	BY plane
$G19 C_Z (C = Y)$	CZ plane

- Cylinder Side-Surface Machining Mode OFF
 - G174
- [Supplement]
- In the cylinder side-surface machining mode, be sure to designate the plane which includes the rotational axis.
 - 2. A plane cannot be selected using the basic axes, corresponding rotational axes, and parallel axes.
 - 3. The rotational axis name, rotation direction, and the corresponding basic axis must be designated in advance at the "rotation axis" parameter. (For details regarding the "rotational axis" parameter, refer to T of this manual.)
 - 4. Select the multi-turn specification for the additional axis. Set "1" for appropriate bit of NC optional parameter (bit) No. 39.

Bit	bit 7	bit 6	bit 5	bit 4	bit 3	bit 2	bit 1	bit 0
No. 39			Multi-turn C-axis	Multi-turn B-axis	Multi-turn A-axis			

- 5. Rotational axis addresses "A", "B", and "C" are designated as angle commands.
- 6. Do not designate both the plane selection command (G17, G18, or G19) and the cylinder side-surface machining mode ON command (G175) in the same block.
- 7. The following functions can be designated only in the cylinder side-surface machining mode.
 - a) Arbitrary angle chamfering function
 - b) Coordinate calculation function
 - c) Cutter diameter compensation function
 - d) Graphics enlargement/reduction function
 - e) Area machining function
- 8. The cylinder side-surface machining mode cannot be called or canceled in the following modes.
 - a) Fixed cycle function
 - b) Cutter diameter compensation function
 - c) Area machining function
 - d) Coordinate calculation function
 - e) Arbitrary angle chamfering function
 - f) 3-D coordinate conversion function
- 9. The following commands cannot be designated in the cylinder side-surface machining mode.
 - a) 3-D coordinate conversion function
 - b) 3-D circular interpolation

2. Corresponding Basic Axis

In cylinder side-surface machining, the rotational axis is replaced by a basic axis. The basic axis which is used to replace the rotational axis is referred to as the "corresponding basic axis". This corresponding basic axis can be either of the two basic axes which are not used as the center (pivot) axis of the rotational axis.

The plane which is formed by the corresponding basic axis and the center (pivot) axis is identical to the plane which is obtained by developing the cylinder side-surface.



The directions of the corresponding basic axes shown above conform to the ISO Standard.

3. Machining Commands

The machining command is executed after the cylinder side-surface machining mode has been switched ON. The cylinder side-surface length is then converted to degrees in order to designate the rotational axis.

The conversion formula is given below.



The F command is designated by the speed on the plane which is obtained by developing the cylinder side-surface. For details regarding the method for obtaining the cutting feedrate, refer to T, 3-5. 3-6 of this manual.

- [Supplement] 1. When referring to the Programming Manual regarding the command procedures, it should be noted that, unless otherwise stated, the X-axis is the horizontal axis, the Y-axis is the vertical axis, and the Z-axis is the axis which is vertical to the selected plane. The I, J, and K commands should be considered in the same manner.
- (1) Positioning (G00) and Linear Interpolation (G01)

Except for the feedrate command, program commands can be designated in the usual manner in the positioning and linear interpolation modes. Angle commands (AG) can also be used in the usual manner. For further details, refer to the Programming Manual.

(2) Circular Interpolation (G02, G03)

Arc on the G17 (Xp-Yp) plane	G02	Xn Vn I I F
	G03	<u>, , , , , , , , , , , , , , , , , , , </u>
Arc on the G18 (Zp-Xp) plane	G02	
	G03	2p_xp_K_ I_ F_
Arc on the G19 (Yp-Zp) plane	G02	
	G03	Yp_Zp_J_K_F_

I : Plus or minus distance between the start point and the center of the Xp-axis

J : Plus or minus distance between the start point and the center of the Yp-axis

K : Plus or minus distance between the start point and the center of the Zp-axis

F : Cutting feedrate (speed on the plane which is obtained by developing the cylinder side-surface)

Example:



The rotational axis designation cannot be made until the cylinder side-surface length has been converted to an angle value (°).

For the above example:

- Rotational axis is the A axis.
- Corresponding basic axis is the -Y axis.

Therefore, the A and J address characters are designated in units of degrees.

Rotational axis parameters

Axis Name	:	A-axis
Rotation direction	:	Plus
Corresponding basic axis	:	–Y axis

(3) Cutter Radius Compensation (G40, G41, G42)

Cutter radius compensation is executed on the plane which is obtained by developing the cylinder side-surface. Function and commands are the same as usual. Refer to the Programming Manual for detail.

(4) Arbitrary Angle Chamfering

When chamfering (C-chamfer, R-chamfer) a corner having an arbitrary angle, simply designate the corner apex and the chamfer amount. The start and end points for chamfering will then automatically be determined, and chamfering will be executed accordingly.

G17 (Xp-Xp) plan	e CHFC	
	CHFR 5	
G18 (Xp-Xp) plan	e CHFC	
		Xp_ Yp_ [Q_][K_][I_]
G17 (Xp-Xp) plan	e CHFC	
		Xp_ Yp_ [Q_][J_][K_]
CHEC · C	Chamfer commar	hc

	•	
CHFR	:	R-chamfer command
Xp, Yp, Zp	:	Corner apex coordinate value
L	:	Chamfer amount (0 - 2099999999 mm)
Q	:	Travel amount after chamfering (0 - 2099999999 mm)
I, J, K	:	Virtual advancing direction (direction vector of Q)
		I Xp-axis direction
		J Yp-axis direction
		Z Zp-axis direction

[Supplement] 2. The "L" chamfer amount and the "Q" travel amount after chamfering are always designated as length commands.



Address characters B and I are designated as angle values (degrees).

Rotational axis parameters

Axis name	:	B-axis
Rotation direction	:	Plus
Corresponding basic axis	:	X-axis

(5) Fixed Cycle

Fixed cycle machining is executed in the cylinder side-surface coordinates. A conventional linear axis is used as the cycle axis.

Shift amounts I, J, and K in the fine-boring (G76) and back-boring (G87) cycles are designated as angle values when corresponding rotational axes are present. However, "I" (depth of cut per pass) and "J" (tool tip retract amount) in the deep-boring cycle (G83) are designated as length values, regardless of the plane which has been selected.

For details, refer to the Programming Manual.

Example: Fine-boring cycle



When cylinder radius R = 50 mm

Address character "J" represents the shift amount in the A-axis (-Y axis) direction, and is therefore designated as an angle value (degrees).

Rotational axis parameters

Axis name	:	A-axis
Rotation direction	:	Plus
Corresponding basic axis	:	-Y



Address characters I and J in the deep-boring cycle must be designated as length values.

Rotational axis parameters

Axis name	:	A-axis
Rotation direction	:	Plus
Corresponding basic axis	:	-Y

[Supplement] 3.

Shift amounts I, J, and K in the fine-boring (G76) and back-boring (G87) are designated for the Xp-, Yp-, and Zp-axis directions, respectively.

(6) Coordinate Calculation Function

A coordinate calculation operation is executed in the cylinder side-surface coordinates.

The following commands are designated as angle values (degrees):

Reference point coordinate, I and J of the grid, staggered grid, and square (interval in the horizontal and vertical directions), Q and R of the staggered grid (stagger interval in the horizontal and vertical directions)

For details, refer to the Programming Manual.



Address character "I" represents the shift amount in the A-axis (-Y axis) direction, and is therefore designated as an angle value (degrees).

Rotational axis parameters

Axis Name	:	A-axis
Rotation direction	:	Plus
Corresponding basic axis	:	–Y

(7) Area Machining

Area machining is executed in the cylinder side-face coordinates. A conventional linear axis is used as the cycle axis.

The coordinate values of the reference point, I (length in the horizontal-axis direction), and J (length in the vertical-axis direction) are designated as angle values (degrees) when corresponding rotational axes are present.

For details, refer to the Programming Manual.

Example: Surface machining



Address character "I" represents the length in the A-axis (Z axis) direction, and is therefore designated as an angle value.

Rotational axis parameters

Axis name	:	A-axis
Rotation direction	:	Plus
Corresponding basic axis	:	-Y

[Supplement] 4. In area machining, the I command is for the horizontal axis direction and the J command is for the vertical axis direction.

Table 1	1
---------	---

	I (Horizontal axis)	J (Vertical axis)			
G17	Angle command in the A-axis (-Y-axis) direction	Length command in the X-axis direction			
	Angle command in the B-axis (X-axis) direction	Length command in the Y-axis direction			
G18	Angle command in the C-axis (-X-axis) direction	Length command in the Z-axis direction			
	Angle command in the A-axis (Z-axis) direction	Length command in the X-axis direction			
G19	Angle command in the B-axis (–Z-axis) direction	Length command in the Y-axis direction			
	Angle command in the C-axis (Y-axis) direction	Length command in the Z-axis direction			

The above table is effective only when the rotation directions of rotational axes and the corresponding basic axes conform to the ISO Standard. The corresponding basic axis is shown in parentheses in the above table.

SECTION 3 ANIMATION FUNCTION

A graphic display of the cylinder side-surface machining operation can be executed. Note however only tool movement on the square pole which is obtained by developing the cylinder can be displayed.

(1) Blank Definition

The blank is defined as a square pole, as shown below.



- h : Cylinder height
- ℓ : Circumferential length (range of rotational axis movement)
- t : Length equivalent to the radius (an arbitrary value can be designated because there is no display in the radial direction.)
- (2) Screen Data

Screen data for the cylinder side-surface machining is designated as shown below.

(a) 1st Page

Rotational axis

Designate a "*" setting.

(b) 3rd Page

Drawing center and drawing range

In addition to conventional length (mm) settings and adding operations, angle ($^{\circ}$) settings and adding operations are also possible.

[F1 (SET) "Setting value (°)"; D [WRITE]

By designating ";D", the angle value is converted to a length value based on the conversion coefficient.

Calculation Coefficient (mm/°)

Set the cylinder radius value followed by ";R".

The following calculation will be performed:

 $2\pi \approx$ setting value"/360

This value serves as the coefficient for converting the angle value (°) to a length value (mm).

If the coefficient determined here is different from that in the program (value for address "R" designated simultaneously with the G175 command), the animation function will be inaccurate.

(c) 4th Page

Master axis designation

Designate the name of the rotational axis. The cursor direction will be fixed in the corresponding basic axis direction (designated by rotational axis parameters).

- [Supplement] 1. All settings other than those described above are executed in the usual manner. For details, refer to SECTION 4 "ANIMATION FUNCTION" of the Special Function No. 1 Manual (Publication No. 4118-E).
 - 2. Even if 360° is exceeded during machining operation, coordinates will be displayed consecutively. However, the blank definition "L" basically must be within the following range $0 \le \ell \le 360^\circ$. Designate the movable range of the rotational axis taking this into consideration.

SECTION 4 ALARM CODES

2249 Data word: 'R'

'R' command value for arc radius command is outside the allowable range:(-99999.999) - (+99999.999) or zero (0).

The R command value in the G175 mode (cylindrical side surface cutting ON) is outside 1 to 5,000 mm.

Or, the G131 command (Super Hi-NC control mode ON) block has a R command (Minimum Block Length on Shape Reconstruction) of which value is out of the setting range.

[Code]

- 1-> With G178 or G179, the taper angle command R and the gap between taper start and end points (K, I) were specified at the same time.
- 2--> 'R' command is 0.
- 3-> A calculation error occurred during analysis of the taper angle command R specified with G178 or G179.

Others->Command value in hexadecimal

[Probable Faulty Locations]

'R' command value in a program

Program Example:

G0X0Y0

G2X100R0F100 <- Alarm code 2

[Measures to Take]

Specify the correct command value.

Or, refer to the instruction manual, and order the value of Minimum Block Length on Shape Reconstruction which is appropriate for the G131 command block.

2263 Data word: G code

Wrong G code command

[Code]

- 1-> G11/G69 (parallel/rotation shift of coordinate) and G10/G68 (cancel of parallel/rotation shift of coordinate) are specified during the G236 (copy) mode.
- 2-> G11/G69 (parallel/rotation shift of coordinate), G10/G68 (cancel of parallel/rotation shift of coordinate) and G237 (copy mode off) are specified in the G51 (shape enlargement/reduction) mode.
- 3-> G11/G69 (parallel/rotation shift of coordinate), G10/G68 (cancel of parallel/rotation shift of coordinate) and G237 (copy mode off) are specified during coordinate calculation.
- 4-> G30 (positioning at home position) is specified in other than a normal mode.
- 5-> G92 (work coordinate system setting) is specified in other than a normal mode.
- 6-> G92 (work coordinate system setting) is specified in the G51 (shape enlargement/reduction) mode.
- 7-> G92 (work coordinate system setting) is specified in the G10/G68 (cancel of parallel/rotation shift of coordinate) or G236 (copy) mode.
- 8-> G41 (cutter offset to left) or G42 (cutter offset to right) is specified in other than a normal mode.
- 9-> G31 (skip) command is specified in other than a normal mode.
- 14-> G41 (cutter offset to left) or G42 (cutter offset to right) is specified during the execution of the graphic function.
- 15-> The fixed thread cutting cycle is specified during parallel or rotation shift of the coordinate in the G11 mode. G178, G179, TRDL, or TRDT command was executed during the following operations; Enlargement/reduction(G51), coordinate parallel or rotational shift(G10), copy(G236).

- 16-> The cutter radius compensation command is specified in the 3-D circular interpolation mode.
- 17-> The 3-D circular interpolation command is specified in the cutter radius compensation mode.
- 18-> The axis name designation (G14) is specified while any of the following operations is in progress.
 - 1) Local coordinate system setting (G11)
 - 2) Scaling (G51)
 - 3) Fixed cycle
 - 4) Cutter radius compensation (G41 or G42)
 - 5) 3-D tool compensation (G44)
 - 6) Tool length offset (G54, G55, G56)
 - 7) Circular interpolation mode including 3-D circular interpolation (G02, G03, G172, or G173)
 - 8) 3-D coordinate conversion (G69)
 - 9) Cylindrical surface side machining (G175)
 - 10) An address character other than O and N is not designated after the designation of cutter radius compensation cancel (G40).
- 20-> Work coordinate system setting (G92) is specified during 3-D coordinate conversion (G69).
- 21-> The 3-D coordinate conversion ON or OFF command (G68 or G69) is specified in the G14 mode.
- 22-> The 3-D coordinate conversion ON or OFF command (G68 or G69) is specified in the G11 mode.
- 23-> The cylindrical side surface machining ON or OFF command (G175 or G174) is specified while any of the following operations is in progress.
 - 1) Fixed cycle
 - 2) Area machining
 - 3) Cutter radius compensation
- 24-> The cylindrical side surface machining ON or OFF command (G175 or G174) is specified while coordinate calculation or arbitrary angle chamfering is in progress.
- 25-> The cylindrical side surface machining ON or OFF command (G175 or G174) is specified during 3-D coordinate conversion (G69).
- 26-> The cylindrical side surface machining ON command (G175) is specified when the plane not including a rotary axis is selected.
 - Note: This alarm occurs if the cylindrical side surface machining is attempted on the machine not provided with a rotary axis having a limited or unlimited rotation range.
- 27-> The 3-D coordinate conversion command (G69) is specified in the cylindrical side surface machining mode (G175).
- 28-> A 3-D circular interpolation command is specified in the 3-D coordinate conversion mode (G69).
- [Probable Faulty Locations]

G code in a program

Program Example:

G52X0 WRITE

[Measures to Take]

Correct the program error which is specified by the alarm code displayed.

V. 2-PALLET APC PALLET IDENTIFICATION FUNCTION

SECTION 1 OUTLINE

The 2-pallet APC pallet identification function is used with parallel 2-pallet APCs and rotary 2-pallet APCs to determine whether the pallet on the machine table is an "A" (or L) pallet or a "B" (or R) pallet. This identification function can be executed in the part program using system variables and the pallet identification command.



< Gate type vertical loading APC>

<Gate type horizontal loading APC>

SECTION 2 SYSTEM VARIABLES FOR PALLET IDENTIFICATION

Name	VPLTK			
Description	Type of pallet on the machine table READ only			
Attribute				
Data	 A (L) pallet B (R) pallet No pallet, or pallet ID not possible) 			

Program example:

N001	IF [VPLTK EQ1] N100 IF [VPLTK EQ2] N200 MSG [PALLET NG] Used only when the message function is provided GOTO NEND
N100	(part program for A-pallet)
	GOTO NEND
N200	(part program for B-pallet)
	: M02
NEND	M02

Program proceeds to "N100" for A-pallet, or to "N200" for B-pallet. Program changes according to the identified pallet.

SECTION 3 PALLET IDENTIFICATION COMMAND

This command enables the program to determine whether the pallet on the machine table is an "A" (L) or "B" (R) pallet.

1. Programming Format

[RP = * M289]

M289	:	Pallet identification command				
RP = *	:	Data of the pallet on the machine table				
		☆: 1 A (L) pallet				
		☆: 2 B (R) pallet				

2. Application Example (Program Example)

N100	M60
N200	RP = 1 M289
	:

After a pallet change occurs, the pallet on the machine table is identified. If the pallet is an "A" pallet, program operation continues. If the pallet is not an "A" pallet, an alarm of level B will be activated and the program will be stopped at the N200 block.

3. Alarms

2730 Pallet discrimination NG

When the pallet is checked according to the "M289 RP=*" command, the pallet being checked is not the designed one.

[Index] None

[Character-string] None

[Code] None

[Measures to Take]

Press the RESET button, and execute the command again. At this time, check the VPLTN system variable to determine the pallet type.

2453 Data word: 'RP'

RP command value is improper.

[Code] Hexadecimal of RP command value caused an alarm

[Measures to Take]

Press the RESET button and designate again.

Designate 1 or 2 for an RP command value.

2731 Data word: RP

RP command value is improper. [Index] None [Character-string] None [Code] Hexadecimal of RP command value caused an alarm [Measures to Take] Press the RESET button and designate again. Designate 1 or 2 for an RP command value. W. TOOL LIST FILE FUNCTION

SECTION 1 AUTOMATIC SELECTION OF ANIMATION DATA FILE AND TOOL LIST FILE

By the "NC SAVE" operation, the IGF function transfers the animation data file and the tool list file at the same time the machining program is transferred to the program storage area in the NC. If the parameter is so set that these files are not created, the files are not transferred.

At the NC, when program selection is made, the animation data file (extension of ADT) and the tool list file (extension of UTT), both having the same file name as the machining program are automatically searched and selected. Based on the selected animation data file and the tool list file, the NC enters the animation data and displays the tool list.

The animation data file and the tool list file are briefly explained in what follows. Automatic selection of the animation data file and the tool data file by the program selection operation at the NC is also described.

1. Overview of Animation Data File and Tool List File

The animation data file and the tool list file are briefly explained below.

1-1. Animation Data File

The animation data file contains the definition of the workpiece blank shape which is expressed by the UGC (user graphic commands). The file name is the same as the program and assigned with the extension of ADT. The NC enteres (sets) the workpiece blank shape used for animation display based on the data in this file. The basic rules applied to the animation data file are:

(1) The file name of the animation data file is the same as the name of the machining program which uses the animation data file. Extension is fixed to "ADT".

If the machining program name is ABC.MIN, the animation data file to be used for this program is ABC.ADT.

- (2) The attribute of the animation data file is sequential file, and the number of characters in a record is within 158 characters.
- (3) The animation data file should not contain S, T, M, and G codes which are used in ordinary NC machining, the sequence names and program name. The M02 (program end code) is valid and the data up to M02 is interpreted as the UGC. The data which follows M02 is ignored.
- (4) The UGC is the command which can be interpreted as the NC's animation data.

Example: DEF WORK[n] ORGIN H1 INDEX A0 CYLNDR 0P,[250,0],266,0,81 CYLNDRI 0P,[250,0]10,0,81 END

These from DEF to END are called the UGC.

(5) In the animation data file, the symbol "\$" which is used in a program to indicate the program continues to the next line cannot be used.

1-2. Tool List File

The tool list file contains the tool data such as the tool number and tool type of the cutting tools to be used for the machining program. The file name is the same as the machining program and assigned with the extension of UTT. The NC enters the tool number, tool classification number, tool diameter, tool nose angle and tool nose radius as the data to define the tool shape to be used for animation drawing. It also uses the file for the display of the tool list, explained later. The basic rules applied to the tool list file are:

(1) The file name of the tool list file is the same as the name of the machining program which uses the tool list file. Extension is fixed to "UTT".

If the machining program name is ABC.MIN, the tool list file to be used for this program is ABC.UTT.

- (2) The attribute of the tool list file is sequential file, and the number of characters in a record is within 158 characters.
- (3) A line beginning with "*" in the tool list file is regarded as the comment file and no format check is made for such lines.

* TLN.,TI,MT,DIA, ,NT,PITCH ,ANGL,N-DIA,ONL,OND Comment line T1,1,2,120.00, ,2.54,120,100.00, 1,1 Data line

- (4) Up to 128 tools can be set in a tool list file.
- (5) The tool list file data is separated into 10 fields, each separated by ",". For the data having the unit of "length", the unit system conforms to the unit system used for creating the machining program. Since there is no information to indicate the unit system in the tool list file, the unit system presently adopted for NC operation is used.

Details of each field is explained in Table 1.

Field No.	Symbol	Contents	Data Attribute, Unit	Range	Comment	
1*	TLN	Tool number	Integer	1 to 300	The range varies according to the specification. (Standard: 99)	
2*	ΤI	Tool classification number	Integer	1 to 7	 Centering drill Drill Tap Reamer Boring End mill Face mill 	
3	МТ	Material code	Integer	1 to 5	1: Cemented carbide 2: HSS 3 to 5:Others	
4*	DIA	Diameter	Unit integer mm/inch	0.000 to 9999.999 mm	Space is assumed to be "0".	
5	NT	Number of teeth	Integer	0 to 99	A space and "0" are regarded to be different indication.	
6	PITCH	Pitch	Unit integer	0.000 to 9999.999 mm	A space and "0" are regarded to be different indication.	
7*	ANGL	Tool nose angle	Unit integer	0. to 180.	"0" and space are assumed to be "180".	
8*	N-DIA	Tool nose diameter	Unit integer mm/inch	0.000 to 9999.999 mm	Space is assumed to be "0".	
9	ONL	Tool length offset number	Integer	1 to 300	The range varies according to the specification. (Standard: 30) Entry of a space is allowed.	
10	OND	Tool diameter offset number	Integer	1 to 300	The range varies according to the specification. (Standard: 30) Entry of a space is allowed.	

Table 1 Details of Fields in Tool List File Record

The data identified by "*", given to the field number, is used as the tool shape data when program selection is made.

2. Automatic Selection Operation

After the completion of machining program selection by the ordinary program selection operation, the NC executes search in the NC's program storage area for the animation data file and the tool list file. If the corresponding files exist in the program storage area, these files are automatically selected.

The procedure how the animation data file and the tool list file are selected in the NC's program storage area, following the program selection operation, is explained below. For the purpose of explanation, the machining program name of "TEST.MIN" is assumed for the program to be selected by the program selection operation. For the name of animation data file and tool list file, refer to 1.

- (1) In the automatic operation mode, press function key [F1] (PROGRAM SELECT) and key in the machining program name "TEST.MIN" (".MIN" is omissible.). Press the [WRITE] key, and the selection of the specified program begins.
- (2) After the completion of program selection, the animation data file, having the same file name as the program name but with extension of "ADT", is searched.
 - * For the program "TEST.MIN",

Animation data file "TEST.ADT" is searched.

Extension, indicating that the file is

the animation data file

Animation data file name, the same as the program name

(3) If the animation data file corresponding to the selected program, "TEST.ADT" in this example, is found in the program storage area, the following message is displayed in the console line.

"Reading the animation data file"

At the same time, automatic selection starts and the file is entered as the animation data. The data is used as the data for animation drawing function.



If the corresponding animation data file is not found in the program storage area, the operation advances to the tool list file search.

- (4) After the completion of animation data file search and selection, the tool list file, having the same file name as the program name but with extension of "UTT", is searched.
 - * For the program "TEST.MIN",

Tool list file "TEST.UTT" is searched.

Extension, indicating that the file is

the animation data file

Tool list file name, the same as the program name

(5) If the tool list file corresponding to the selected program, "TEST.UTT" in this example, is found in the program storage area, the following message is displayed in the console line.

"Reading the tool list file"

At the same time, automatic selection starts and the file is entered as the tool data for animation drawing. The data is also used as the list of tools used for the tool list display function to be explained later.



If the corresponding tool list file is not found in the program storage area, the operation advances to the next.

- (6) This completes the automatic selection of the animation data file and the tool list file following the automatic program selection.
- [Supplement] 1. Automatic selection of the animation data file and the tool list file is executed after the selection of the machining program if there are corresponding files in the program storage area. Therefore, if selection of the animation data file and the tool list file is not successful due to an occurrence of an error, the machining program can be started since the machining program has been selected correctly.
 - 2. If a machining program has been selected in any operation procedure other than the "program selection by inputting the program name at the command line", and if an error occurred during automatic selection of the animation data file and the tool list file, it does not generate an alarm because the error in file selection does not pose a problem for executing the selected program.

- [Supplement] 3. Selection of the animation data file and the tool list file is independent from each other. Therefore, if an error occurs during the search of the animation data file, animation data file selection ends at this point and the tool list file selection operation is continuously executed. If an error occurs during tool list file selection, the selection operation stops when the error occurred. Note that if the animation data file could not be searched, the previous animation data (workpiece blank shape) is used for the succeeding animation drawing.
 - 4. For the elements which have "length units" in the animation data, the unit system selected for NC operation is used.
 - 5. If the setting for NC optional parameter (bit) No. 48, bit 5 is "1", the offset data set for the number which is set for "OND" in the automatically selected tool list file is automatically set for "DIA" in the tool list file. The offset data to be set is 1/2 of the diameter (rounded off to the unit of microns).
 - 6. For the elements which have "length units" in the tool list file, the unit system is determined to be "mm" or "inch" according to the unit system selected for NC operation.

If a decimal point is used, the position of the decimal point corresponds to "mm" or "inch". If no decimal point is used in the data, it is assumed that a decimal point is placed at the end of the data.

Example: If "mm" is selected for NC operation

"200" is treaded as "200 mm".

3. Relationship between Program Selection Method and Operation Method

There are a variety of program selection and operation methods. (For details of program selection and operation methods, refer to the operation manual of OSP.) In this item, the relationship between the program selection method, in which the automatic selection of the animation data file and the tool list file is valid, and the program operation method is explained.

Program selection and program operation methods are summarized below.

Program selection methods:

- Selection by inputting a program name in the command line
- Schedule program selection
- External program selection
- DNC-C mode program selection
- Selection by the operation preparation function for interactively prepared program

Program operation methods:

[Supplement]

- A-method operation
- B-method operation
- S-method operation
- Multi-volume operation
- Schedule program operation

Please note that some of the program selection methods and operation methods indicated above require addition of optional specification.

The combinations between the program selection method and program operation method in which the automatic selection of the animation data file and the tool list file is possible are summarized in Table 2 below.

Operation Method	A-method	B-method	S-method	Multi- volume	Remote buffer	Schedule
Command input	0	C	C	0	×	Δ
Schedule	0	0	0	Δ	×	Δ
External	0	0	0	Δ	Δ	×
DNC-C	×	×	×	Δ	Δ	×
Operation ready function	0	0	0	Δ	Δ	Δ

Table 2 Combinations of Program Selection and Operation Methods for Automatic File Selection

 \bigcirc : Automatic reading of animation data file and tool list file is executed.

- \times : Automatic reading of animation data file and tool list file is not executed.
- Δ : Impossible combination

The command input method for program selection includes the selection of a warm up program.

SECTION 2 TOOL LIST DISPLAY FUNCTION

For the setup assisting function, the tool list display function is supported. By selecting the tool list file which is created in the predetermined format, the function displays the contents of the tool list file on the screen as the tool list in which the following information is displayed.

(1) Tool List

The contents of the selected tool list file are displayed.

(2) Setup Required Tool List

For the cutting tools specified in the selected tool list, the function checks whether all of these tools are set in the tool magazine. If there are tools not found in the magazine or spindle, such tools are displayed as the setup required tools.

(3) Unnecessary Tool List

The function checks if, in the magazine, cutting tools not specified in the tool list file exist. If there are such tools in the magazine, they are displayed as unnecessary tools.

The tool list file is automatically created by the IGF. However, it is possible to create it using the NC's program editing function conforming to the tool list file format.

For the format of the tool list file, refer to SECTION 1, 1-2.

The three types of tool list display indicated above are explained below.

1. Selecting the Tool List Display Function

The tool list display is possible in the automatic, MDI, or manual operation mode, or in the tool data setting mode. In any of these modes, first call the tool list display function by pressing a proper function key and then select the tool list display screen and the tool list file, the base of the tool list display.

The procedure used for calling the tool list display function in the indicated operation mode or tool data setting mode is explained in this item. In the next item, the other operation such as selection of the tool list display screen is explained.

The procedure for calling the tool list display function is explained assuming the automatic operation mode. The same procedure may be used in other modes (MDI, manual, tool data setting).

(1) Press the AUTO key to select the automatic mode. The screen for the automatic operation mode is displayed.



(2) Press function key [F8] (EXTEND) several times until function name "TOOL FILE DISP" is displayed for function key [F1].



- [Supplement] 1. In the operation executed in the tool data setting mode, function "TOOL FILE DISP" is assigned to function key [F6]. In the MDI or manual mode, this function is assigned to function key [F1] as in the automatic mode.
 - 2. The number the function key [F8] should be pressed until the function "TOOL FILE DISP" is displayed varies depending on the selected mode.

(3) Press function key [F1] (TOOL FILE DISP).



This selects the tool list display function and the following functions are assigned to the function keys as indicated below. The screen changes to the tool list display and if a tool list file has been selected, the corresponding tool list is displayed.

- F1: FILE SELECT
- F2: USE TOOL
- F3: DISUSED TOOL
- F4: LACK TOOL
- F7: QUT

With the operation explained above, the tool list display function is selected. The operation to follow is explained below.
2. Selecting the Tool List File

To display the tool lists, the tool list file must be selected. The following explains the procedure used for selecting the tool list file.

Selection of the tool list file is possible in either automatic selection where the tool list file is automatically selected when a machining program is selected or manual selection where the required tool list file name is keyed in after pressing function key [F1] (FILE SELECT).

2-1. Automatic Tool List File Selection

In the automatic tool list file selection method, the tool list file is not selected independently, but it is selected when a machining program is selected.

In the machining program selection operation, in the automatic operation mode, the tool list file (extension UTT) assigned with the same file name as the machining program is searched. If the corresponding file is found, it is automatically selected. For details of automatic selection, refer to SECTION 1, 2 "Automatic Selection Operation".

2-2. Manual Tool List File Selection

Press function key [F1] (FILE SELECT) and key in the tool list file name. Pressing the [WRITE] key selects the specified tool list file. The operating procedure is explained below.

(1) Select the tool list function display function, and display function name "FILE SELECT" for function key [F1].

AUTO	OPERATI	ON	A.MIN		0 N	1
* TOOL	LIST 😽		FILE (VAME: A.	UTT TEETH NO	4.10.00
NÔ. EXI	ST IT	EM	MATERIAL	DIAM.	/PITCH	hno/dno
5 **	* CEN	TER DRILL	TIC	10.0000	4N	5/5
=EX =EX =TLLS >LT	1					
FILE		DISUSED	LACK TOOL		QUIT	

(2) Press function key [F1] (FILE SELECT). The following prompt appears on the console line: >LT

The function waits for the input of a file name.



(3) Key in the file name following the prompt and press the [WRITE] key.



The tool list file of the specified file name is selected and the contents of the file are displayed on the screen.

With the operation explained above, the required tool list file is selected and the tool data is displayed according to the selected tool list file.

	A,MIN		<u>0 N</u>	1
* TOOL LIST *	FILE N	AME: A.U	<u>ן 97/07/15 1</u> תד	4:10:00
TOOL TOOL	TOOL		TEETH NO.	
NO. EXIST TIEM	<u>MATERIAL</u> TIC			<u>HNO/DNO</u> 5/5
=EX =EX =TLLS >LT_ABC.UTT	LACK			
SELECT TOOL TOOL	TOOL		QUIT	
	ഭഹഭ			<u> </u>
				<u>- 8</u>)
	A MIN		0 N	<u>ر</u>
AUTO OPERATION	A, MIN		0 N 97/07/15 1	1 4:10:00
AUTO OPERATION * TOOL LIST * TOOL TOOL	A,MIN FILE N TOOL	AME: ABC	0 N <u>97/07/15 1</u> .UTT TEETH NO.	1 4:10:00
AUTO OPERATION * TOOL LIST * TOOL TOOL NO. EXIST ITEM	A.MIN FILE N TOOL MATERIAL	AME: ABC	0 N 97/07/15 1 UTT TEETH NO. /PITCH	4:10:00 HNO/DNO
AUTO OPERATION * TOOL LIST * TOOL TOOL NO. EXIST ITEM 1 *** BORING BAR 2 *** DRILL	A.MIN FILE N TOOL MATERIAL 5 HSS	AME: ABC DIAM. 5.0000 2.0000	0 N 97/07/15 1 UTT TEETH NO. /PITCH 5N 2N	1 4:10:00 HNC/DNO 5/ 5 2/ 2
AUTO OPERATION * TOOL LIST * TOOL TOOL NO. EXIST ITEM 1 **** BORING BAR 2 *** DRILL 3 *** CENTER DRILL 4 *** PRILL	A.MIN FILE N TOOL MATERIAL 5 HSS 3	AME : ABC <u>DIAM.</u> <u>5.0000</u> 2.0000 <u>3.0000</u> <u>4.0000</u>	0 N 97/07/15 1 UTT TEETH NO. /PITCH 5N 2N 3N 4N	4:10:00 HNC/DN0 5/ 5 2/ 2 3/ 3
AUTO OPERATION * TOOL LIST * TOOL TOOL NO. EXIST ITEM 1 *** BORING BAR 2 *** DRILL 3 *** CENTER DRILL 4 *** DRILL 10 *** TAP	A. MIN FILE N TOOL MATERIAL 5 HSS 3 4 HSS	AME : ABC <u>DIAM.</u> <u>5.0000</u> 2.0000 3.0000 4.0000 10.0000	0 N 97/07/15 1 UTT TEETH NO. /PITCH 5N 2N 3N 4N 20N	1 4:10:00 5/ 5 2/ 4 3/ 3 4/ 4 30/ 4
AUTO OPERATION * TOOL LIST * TOOL TOOL NO. EXIST ITEM 1 *** BORING BAR 2 *** DRILL 3 *** CENTER DRILL 4 *** DRILL 10 *** TAP	A.MIN FILE N TOOL MATERIAL 5 HSS 3 4 HSS	AME: ABC <u>5.0000</u> 2.0000 3.0000 4.0000 10.0000	0 N 97/07/15 1 UTT TEETH NO. /PITCH 5N 2N 3N 4N 20N	HNO/DNO 5/ 9 2/ 3/ 3/ 3/ 30/
AUTO OPERATION * TOOL LIST * TOOL TOOL NO. EXIST ITEM 1 *** BORING BAR 2 *** DRILL 3 *** CENTER DRILL 4 **** DRILL 10 **** TAP	A.MIN FILE N TOOL MATERIAL 5 HSS 3 4 HSS	AME: ABC <u>DIAM.</u> <u>5.0000</u> <u>2.0000</u> <u>3.0000</u> <u>4.0000</u> 10.0000	0 N 97/07/15 1 UTT TETH NO. /PITCH 5N 2N 3N 4N 20N	1 4:10:00 5/ 2 2/ 3 3/ 3 30/ 2
AUTO OPERATION * TOOL LIST * TOOL TOOL NO. EXIST ITEM 1 *** BORING BAR 2 *** DRILL 3 *** CENTER DRILL 4 *** DRILL 10 *** TAP	A.MIN FILE N TOOL MATERIAL 5 HSS 3 4 HSS	AME: ABC <u>5.0000</u> 2.0000 3.0000 4.0000 10.0000	0 N 97/07/15 1 UTT TEETH NO. /PITCH 5N 2N 3N 4N 20N	HNO/DNO 5/ 9 2/ 3/ 3/ 3/ 30/
AUTO OPERATION * TOOL LIST * TOOL TOOL NO. EXIST ITEM 1 *** BORING BAR 2 *** DRILL 3 *** CENTER DRILL 4 *** DRILL 10 *** TAP	A.MIN FILE N TOOL MATERIAL 5 HSS 3 4 HSS	AME: ABC <u>5.0000</u> 2.0000 3.0000 4.0000 10.0000	0 N 97/07/15 1 UTT TEETH NO. /PITCH 5N 2N 3N 4N 20N	HNC/DNO 5/ 5 2/ 2 3/ 3 4/ 4 30/ 2
AUTO OPERATION * TOOL LIST * TOOL TOOL NO. EXIST ITEM 1 *** BORING BAR 2 *** DRILL 3 *** CENTER DRILL 4 *** DRILL 10 *** TAP	A.MIN FILE N TOOL MATERIAL 5 HSS 3 4 HSS	AME : ABC <u>5.0000</u> 2.0000 3.0000 4.0000 10.0000	0 N 97/07/15 1 UTT TEETH NO. /PITCH 5N 2N 3N 4N 20N	1 4:10:00 5/ 5 2/ 2 3/ 3 4/ 4 30/ 2
AUTO OPERATION * TOOL LIST * TOOL TOOL NO. EXIST ITEM 1 *** BORING BAR 2 *** DRILL 3 *** CENTER DRILL 4 *** DRILL 10 *** TAP =EX =TLLS	A.MIN FILE N TOOL MATERIAL 5 HSS 3 4 HSS	AME : ABC <u>5.0000</u> 2.0000 3.0000 4.0000 10.0000	0 N 97/07/15 1 UTT TEETH NO. /PITCH 2N 3N 4N 20N	HNC/DNO 5/ 5 2/ 2 3/ 3 4/ 4 30/ 2
AUTO OPERATION * TOOL LIST * TOOL TOOL NO. EXIST ITEM 1 *** BORING BAR 2 *** DRILL 3 *** CENTER DRILL 4 **** DRILL 10 *** TAP =EX =TLLS >LT ABC.UTT J T	A.MIN FILE N TOOL MATERIAL 5 HSS 3 4 HSS	AME: ABC 5.0000 2.0000 3.0000 4.0000 10.0000	0 N 97/07/15 1 UTT TEETH NO. /PITCH 2N 3N 4N 20N	HNC/DNO 5/ 5 2/ 2 3/ 3 4/ 4 30/ 2
AUTO OPERATION * TOOL LIST * TOOL TOOL NO. EXIST ITEM 1 *** BORING BAR 2 *** DRILL 3 *** DRILL 3 *** DRILL 10 *** TAP =EX =TLLS >LT ABC.UTT >LT FILE USE DISUSED	A. MIN FILE N TOOL MATERIAL 5 HSS 3 4 HSS	AME : ABC 5.0000 2.0000 3.0000 4.0000 10.0000	0 N 97/07/15 1 UTT TEETH NO. /PITCH 2N 3N 4N 20N	HNO/DNO 5/ 5 2/ 2 3/ 3 4/ 2 30/ 2
AUTO OPERATION * TOOL LIST * TOOL TOOL NO. EXIST ITEM 1 **** BORING BAR 2 **** DRILL 3 **** CENTER DRILL 4 **** DRILL 10 **** TAP =EX =TLS >LT ABC.UTT >LT FILE USE DISUSED SELECT TOOL TOOL	A. MIN FILE N TOOL MATERIAL 5 HSS 3 4 HSS	AME : ABC <u>5.0000</u> 2.0000 3.0000 4.0000 10.0000	0 N 97/07/15 1 UTT TEETH NO. /PITCH 2N 3N 4N 20N	HNC/DNO 5/ 5 2/ 2 3/ 3 4/ 4 30/ 2

- [Supplement] 1. If the [WRITE] key is pressed without keying in a file name, in step 3, the tool list file (extension of UTT) of the machining program is selected if a machining program has been selected. If no machining program has been selected, A.UTT file is selected.
 - 2. In step 3, it is possible to display the directory of the tool list files entered to the NC memory. The procedure to be used to display the directory of the tool list files is the same as the one used for displaying the directory of the machining programs. The outline of the procedure is explained below.

In the operation in step 3, key in an asterisk "*" instead of a file name and press the [WRITE] key.

>LT * [WRITE]

(<u></u>
	AUTO OPERATION	A.MIN		0 N	1)
ί.	* TODI LIST *		AME: A	<u>_97/07/15_</u> UTT	14:10:00	
	TOOL TOOL	TOOL	- N	TEETH NO.		
	NO. EXIST ITEM	MATERIAL	DIAM.	<u>/PITCH</u>	HNO/DN	0
	5 *** CENTER DRILL	ΠĻ	10.0000	4N	5/	5
	=EX					
	=EX					
	=TLLS NIT *					
ļ		LACK				-
(SELECT TOOL TOOL	TOOL				
1		1002	1	1		/
	FAFZFA	FALE	SIFE	ארד שור	<u>E A</u>	
1						
(AUTO OPERATION	A.MIN		0 N	1)
				1 27707713		
					14.1V.00 J	
	TOOL LIST E	NTRY INDEX			4.10.00	
	TOOL LIST FILE	NTRY INDEX			PAGE 1	
Ī	TOOL LIST FILE ABC. UTT	NTRY INDEX			PAGE 1	
İ	TOOL LIST FILE ABC.UTT A.UTT E.UTT	NTRY INDEX			PAGE 1	
	TOOL LIST E	NTRY INDEX			PAGE 1	
I	TOOL LIST E	NTRY INDEX			PAGE 1	
I	TOOL LIST FILE ABC. UIT A. UIT E. UIT TEST-1. UIT	NTRY INDEX			PAGE 1	
I	TOOL LIST FILE ABC. UIT A. UIT E. UIT TEST-1. UIT	NTRY INDEX			PAGE 1	
ļ	TOOL LIST FILE ABC. UIT A. UIT E. UIT TEST-1. UIT	NTRY INDEX			PAGE 1	
I	TOOL LIST FILE ABC. UIT A. UIT E. UIT TEST-1. UIT	NTRY INDEX			PAGE 1	
I	TOOL LIST FILE ABC. UTT A. UTT E. UTT TEST-1. UTT	NTRY INDEX			PAGE 1	
I	TOOL LIST FILE ADDC. UIT A. UTT E. UTT TEST-1. UTT =FX	NTRY INDEX			PAGE 1	
	TOOL LIST FILE ACUTT A.UTT E.UTT TEST-1.UTT =EX =TLLS	NTRY INDEX			PAGE 1	
	TOOL LIST FILE ACUTT A.UTT E.UTT TEST-1.UTT =EX =TLS >LT *	NTRY INDEX			PAGE 1	
	TOOL LIST FILE ADC. UTT A. UTT E. UTT TEST-1. UTT =EX =TLS >LT * What is the file name for	NTRY INDEX	ile selec	t ?	PAGE 1	
	TOOL LIST FILE ADDIT FILE A.UTT A.UTT E.UTT TEST-1.UTT =EX =TLS >LT * What is the file name for FILE USE DISUSED	NTRY INDEX	ile selec	t ?	PAGE 1	

F1F2F3F4F5F6F7FB

The screen changes to the tool list entry index screen and the directory of the entered tool list files (extension UTT) is displayed.

Find the required file name in the directory and select it by keying in the file name and pressing the [WRITE] key. Pressing the [WRITE] key after moving the cursor to the required file name also selects the file.

3. Selection of Tool List Display Screens and Display Contents

There are three types of tool list display. They are tool list display, in which the contents of the tool list file are displayed, the setup required tool list display, in which the list of the tools required to be set up is displayed, and the unnecessary tool list, in which the list of tools not used for the selected program is displayed. These display modes are explained below.

3-1. Tool List Display

On the tool list display screen, variety of related data such as tool type, material, and tool diameter is displayed. Therefore, the list is displayed in two screens: in the first screen (screen 1), the data necessary for cutting is displayed and in the next screen (screen 2), the data related with the tool shape is displayed.

- (1) Selecting the tool list display screen
 - (a) Select the tool list display function as explained in 2-2-1 and display the function name "USE TOOL" for function key [F2].

AUTO OPERAT	ION	A.MIN		O N	1
-				97/07/15 14	4:10:00
* TOOL LIST 1	ŧ	FILE NA	ME: A.UT		
	тги		DIAN I	ELIH NU.	
	IEM NTED DDIII	TIC	10.0000		
		110	10.0000		
EV					
=TUS					
>LT					
		TOOL			
		FUUL (/
	<u>とにろ</u> し				<u>- 8</u>]

(b) Press function key [F2] (USE TOOL) and the following prompt appears on the console line.

>TL

The contents of the tool list file presently selected are displayed in the order of the tool numbers in the tool list file.

On this screen, a line cursor is displayed so that the data can be easily checked on the screen.



With the operation procedure indicated above, the contents of the selected tool list file are displayed. Since the variety of data is displayed for a single tool, use the page keys and the cursor keys to change the display page. The procedure to change the display page is explained below.

Screen change by cursor keys and page keys:

- Cursor key 🕜

The cursor in the screen moves up. When this key is pressed while the cursor is at the top line, the screen scrolls.

- Cursor key 🕀

The cursor in the screen moves down. When this key is pressed while the cursor is at the bottom line, the screen scrolls.

- Cursor keys (C) and (C)

The screen changes between screen 1 and screen 2. The cursor remains located at the same tool number.

- Page key 🖽

The screen returns to the previous page. The tool data display screen changes in units of pages.

- Page key (

The screen advances to the next page. The tool data display screen changes in units of pages.

(c) To quit the tool list display, press function key [F7] (QUIT). If a mode selection key is pressed instead of function key [F7], the tool list display mode is canceled.

(2) Contents on the Tool List Display Screen

On the tool list display screen, 10 kinds of data set in the tool list file and tool mount status (whether the tool is mounted in the magazine or not) are displayed. For details of the data set in the tool list file, refer to SECTION 1, 1-2 "Tool List File".



(a) Tool number

Displays the tool number to be used. They are displayed in the order the tools are set in the tool list file.

(b) Tool mounted status

Indicates whether the tool is mounted in the magazine or spindle, or not.

ACT	The tool is mounted in the spindle.
NXT	The tool is scheduled to be mounted to the spindle next.
Pot number	When the tool is mounted in a pot in the magazine.
***	The tool is not mounted in the magazine or spindle.

(c) Tool classification number

Displays the tool kind.

In the tool list file, the tool kind is set by the code number. On the tool list display screen, the tool kind is shown by the tool kind name.

- 1: Center drill 2: Drill 3: Tap 4: Reamer
- 5: Boring 6: End mill 7: Face mill

(d) Material

Displays the tool material.

In the tool list file, the tool material is set by the code number. On the tool list display screen, the tool material is shown by the tool material name. If the setting in the tool list file is 3 to 5, the set number is displayed.

1: Cemented carbide

2: HSS

3 to 5: The code number is displayed.

(e) Tool diameter

Displays the tool diameter.

The value displayed is the nominal value and differs from the tool diameter offset value.

(f) Number of teeth and pitch

Displays either the number of teeth or pitch. If both of the number of teeth and pitch are set in the tool list file, priority is given to the number of teeth.

- N: When the number of teeth is displayed, the data is followed by N.
- P: When the pitch is displayed, the data is followed by P.
- (g) Tool length/diameter offset number

Displays the tool length offset number and tool diameter offset number.



F1F2F3F4F5F6F7FB

1) Tool nose angle

Displays the tool nose angle.

2) Tool nose diameter

Displays the tool nose diameter.

- [Supplement] 1. On screen 2, tool number, tool mounted status, and tool classification number (tool kind number) are displayed as on Screen 1 so that the displayed data can be easily recognized.
 - 2. If the data exceeds the allowable range, it causes an error.
 - 3. When the unit system is changed between "mm" and "inch", the set values are not changed.

3-2. Setup Required Tool List

The list of tools which are not mounted in the magazine or spindle, in comparison with the tools specified in the tool list file is displayed. The same information as the tool list explained in 3-1 is displayed except the "tool mounted status". Therefore, the list is given in two screens (screen 1 and screen 2).

Since the unnecessary tool list is displayed on the same screen, the screen is split into two display areas; the setup required tool list is displayed in the upper area and the unnecessary tool list is displayed in the lower area.

- (1) Selecting the Setup Required Tool List Screen
 - (a) Select the tool list function display function, and display function name "LACK TOOL" for function key [F4].



(b) Press function key [F4] (LACK TOOL). The following prompt appears on the console line:

```
>WT
```

The contents of the setup required tools are displayed in the order of the tool numbers set in the tool list file.



On this screen, the list of setup required tools is displayed in the upper area while the list of unnecessary tools displayed in the lower area.

A line cursor is displayed in the setup required tool list. The cursor moves to the unnecessary tool list when function key [F3] (DISUSED TOOL) is pressed.

AUTO	OPERATION		A	MIN	0	N 7/07/1	1 5 14:10:00
H LACI	K TOOL LIST	· * -		FILE N		BC. UT	5 14.10.00 [
NOL.	TEM	i Ma'	OOL FERIAL	DIAM.	NOSE ANGLE		TOP DIAM.
1	BORING BAR	1	5	5.0000	5.00	0	5.0000
4	DRILL		4	4,0000	4.00	00	4,0000
10	TAP		HSS	10.0000	30.00	00 2	20.0000
7 13 16 20	7 13 16 20		11 14 17 22	11 14 17 22		12 15 18 24	12 15 18 24
≂ill: >TL >₩T >LT	> 		1			1	
FILE	USE CT TOOL	DISUSED TOOL	LACK			QUI	Ť
(F)		IF 3	(F 4	ា្រភា	FEI	F 7	

With the operation explained above, the data of the cutting tools which require to be set up is displayed. In this list, the same kind of tool information as given in the tool list is displayed. To confirm all of the tools displayed in this list, use the page keys and cursor keys.

For details of the functions of the page keys and the cursor keys, refer to the explanation given in 3-1.

(2) Contents on the Setup Required Tool List Display Screen

The list displays the tools which are not presently mounted in the magazine or spindle in comparison with the tools specified in the tool list file. In this list, all tool data except the tool mount status is displayed. For details of the tool data given in the list, refer to 3-1 "Tool List Display".

The screen for the setup required tool list display is split into two areas, and the setup required tool list is given in the upper area while the lower area displays the unnecessary tool list.

An example of the list display is shown below.

<Display data in Screen 1>

AUTO	OPERATION		ļ	A. MIN	0	N (07/15_1	4.10	1
* LACK	TOOL LIST	*		FILE N		707715-1 8C. UTT	4:10	100
NO	ITEM	MAT	FRIAL	DIAM.	/PITCH	ŀ	INO/D	NO
1	BORING BAR	٩	5	5.0000	5N	·······	5/	5
4	DRILL		4	4.0000	4N		4/	4
10	TAP		HSS	10.0000	20N	:	30/	2
* DISU TOOL 7 13 16 20	JSED TOOL L NO. 7 13 16 20	_IST *	T00L 11 14 17 22	NO. 11 14 17 22		TOOL NO 12 1: 15 1: 18 1: 24 2:	-2 5 8 4	
=TLLS >TL >WT >LT								
FILE	USE T TOOL	D I SUSED TOOL	LACK TO			QUIT		
F 1)(F2)	F 3	F 4		۱		F 8	5

<Display data in Screen 2>

/						
{	AUTO	OPERATION	A	MIN	0 1	N 1
					97/07	/15 14:10:00
	* LACH	<toollist td="" ₩<=""><td></td><td>FILE NAM</td><td>ME: ABC.L</td><td>ΠT</td></toollist>		FILE NAM	ME: ABC.L	ΠT
	TOOL		TOOL		NOSE	
	NQ.	ITEM	MATERIAL	DIAM.	ANGLE	<u>TOP DIAM.</u>
	1	BORING BAR	5	5.0000	5.000	<u>5.0000</u>
	4	DRILL	4	4.0000	4.000	4.0000
	10	TAP	HSS	10.0000	30.000	20.0000
	 ✤ DIS T00I 7 13 16 20 	SUSED TOOL LIST _ NO. 7 13 16 20	* T00L 11 14 17 22	NO. 11 14 17 22	T0 12 15 18 24	OL NO. 12 15 18 24
	=TLLS >TL >WT >LT	3				
	FILE SELE(USE DIS CT TOOL T	USED LACK OOL TOO	L	Q	UIT
	(F 1	F2F	3) F 4	F50	F 6) (F 7	7 <u>F 8</u>

(3) Determination of the Setup Required Tools

The tools to be displayed in the setup required tool list are determined in the following manner.

For the system not equipped with the tool life management specification and automatic spare tool selection specification

In the cutting tools set in the tool list file, those not mounted in the magazine pot or spindle.

For the system equipped with the tool life management specification but not equipped with the automatic spare tool selection specification

In the cutting tools set in the tool list file, those other than defective tool are not mounted in the magazine pot or spindle.

For the system equipped with the tool life management specification and the automatic tool change specification

In the cutting tools set in the tool list file, those other than defective tool in the same tool group are not mounted in the magazine pot or spindle.

[Supplement] Whether the tool is set in the magazine or spindle is judged only from the tool number.

Therefore, if the tool number is found in the magazine pot, it does not mean that the tool to be used by the program exists in the magazine.

3-3. Unnecessary Tool List

The list of tools which are presently mounted in the magazine or spindle but not used by the program is displayed. How the tools are classified as unnecessary tools is explained later.

In the list, the tool numbers of the unnecessary tools and the defective tools (for the system equipped with the tool life management specification) are displayed. The tools are displayed in the order of the pot numbers sent from the programmable controller. For the display of the list, only one screen is used differing from the tool list display or setup required tool list.

- (1) Selecting the Unnecessary Tool List Screen
 - (a) Select the tool list function display function, and display function name "DISUSED TOOL" for function key [F3].

AUTO OF	PERATION		A.MI	N	0 1	J 1]	
* T00L L	IST *		FILE	NAME :	97/07	/15 14:10:00	
NO. EXIS	T ITEM		TOOL MATERIA	DIAM.	TEETH N	O. I. HNO/DN	10
5 ***	ÇENTE	RDRILL	TIC	10.00	00 4N	l 5/ 5	5
=EX							
=EX =TLLS							
<u>>LT</u>							_
FILE	USE	DISUSED	LACK		a	uit	
	TOOL		IOUL	I		•••• I	~

F1F2F3F4F5F6F7F8

(b) Press function key [F3] (DISUSED TOOL). The following prompt appears on the console line:

>UT

The contents of the unnecessary tools are displayed in the order of the pot numbers sent from the external programmable controller.



On this screen, the upper area is used to display the list of the setup required tools and the unnecessary tool list is displayed in the lower area.

A line cursor is displayed in the list so ~that the data check is facilitated. This line cursor moves to the list of the setup required tools if function key [F4] (LACK TOOL) is pressed.

With the operation explained above, the contents of the unnecessary tools are displayed to be checked. If the list uses more than one page, it is necessary to use the page keys and cursor keys to check the entire contents of the list. For details of the functions of the page keys and the cursor keys, refer to 3-1 "Tool List Display".

Note that the cursor keys \bigcirc and \bigcirc are not used in the display of the unnecessary tool list. If these keys are used, the display area of the setup required tool list given on the same screen is changed.

(2) Contents on the Unnecessary Tool List Display Screen

The unnecessary tool list includes the tool numbers of the tools other than those set in the tool list file and the defective tools (only for the system equipped with the tool~ life management specification). The tools are displayed in the order of the tool pot numbers sent from the external programmable controller.

The screen for the unnecessary tool list display is split into two areas. The setup required tool list is given in the upper area while the lower area displays the unnecessary tool list.



<Unnecessary tool list screen>

(a) Tool number

Displays the tool number of~ the unnecessary tools.

(b) Defective tool

When the system is equipped with the tool life management specification, an asterisk "*" is displayed for defective tools. When the tool life management specification is not selected, no defective tool information is given.

(3) Determination of the Unnecessary Tools

The tools displayed in the unnecessary tool list are determined in the following manner.

For the system not equipped with the tool life management specification and automatic tool change specification

Tools for which corresponding tools are not found in the cutting tools set in the tool list file.

For the system equipped with the tool life management specification but not equipped with the automatic tool change specification

Defective tools, or tools for which corresponding tools are not found in the cutting tools set in the tool list file.

For the system equipped with the tool life management specification and the automatic tool change specification

Defective tools, tools for which corresponding tools are not found in the cutting tools set in the tool list file, or tools for which the corresponding group number is no~t set in the tool list file.

SECTION 3 PARAMETERS

NC optional parameter (bit)

No. 48, bit 5

Sets whether or not the tool diameter offset value set for DIA in the tool list file is used as the offset data for the tool diameter offset number set for OND.

"1": The tool diameter set in the tool list file is used for the offset value for the tool diameter offset number.

A value 1/2 the value set for DIA in the tool list file is set for the tool diameter offset value. A value smaller than microns is rounded off.

"0": The tool diameter set in the tool list file not used for the offset value for the tool diameter offset number.

SECTION 4 ALARM AND ERROR

5374 Tool list file read

In reading the tool list file for program selection, an error was detected in the file. [Character-string] None

[Code] XYZZZZZZ

X-> Cause

- 0-> Operation preparation is not completed.
- 1-> The required field (data) is not found.
- 2-> The tool list data has a wrong attribute. (The integer, for example, has a decimal point.)
- 3-> The data is set exceeding its setting range.
- 4-> More than 128 tool are registered in the tool list.
- 5--> The same tool number is assigned to plural tools.
- Y -> Field number
- 0--> Tool number
- 1-> Tool type
- 2-> Material code
- 3--> Tool diameter
- 4-> The number of cutting edges
- 5--> Pitch
- 6--> Nose angle
- 7-> Tool nose diameter
- 8-> Tool length offset number
- 9-> Cutter radius compensation number
- ZZZZZ-> Line number where the error was detected (in decimal).

The message lines are also counted.

[Measures to Take]

Correct the field number data in the line indicated by the

5367 Animation data file read

In reading the animated data file for program selection, an error was detected in the file. [Character-string]

- 632-> Program end code
- 633-> ELETE statement
- 634-> DEF statement
- 636--> Improper value
- 367-> Illegal character
- 638-> User variable
- 639-> System variable
- 640--> Illegal command
- 641-> Program element over
- 642-> Coordinate data
- 643--> END statement
- 644-> Command over
- 645--> Command format
- 646--> Additional parameter
- 647-> Expression
- 648--> Shape registration area over

[Code] Line number where the error was detected (in decimal)

Depending on the error content, the actual error line may not correspond to the line indicated by the code.

[Measures to Take]

Correct the line indicated by the code by removing the error indicated by the character-string code.

X. SERVO FILE SELECTION FUNCTION (SELECTION BY WORKPIECE WEIGHT SETTING)

SECTION 1 OVERVIEW

The servo data of the feed motors (servomotors) used to drive machine tool axes is set assuming the allowable maximum workpiece weight. In actual operation, however, workpieces weighing the assumed maximum weight are handled only rarely and in the operation where workpieces lighter than the set maximum weight are handled, using the preset servo data cannot always operate the machine at the maximum performance. The servo data selection function, stated below, automatically selects the servo file that has the most optimum servo data, in response to the set workpiece weight (weight of table load) to operate the machine at the maximum performance.

SECTION 2 SETTING THE WORKPIECE WEIGHT (TABLE LOAD)

1. Operation

- (1) Select the PARAMETER SET screen.
- (2) Press the [ITEM \uparrow] or [ITEM \downarrow] key to display the WORK WEIGHT SET screen.

If the display setting for this parameter screen is "not displayed", the screen is not displayed. In this case, change the setting to "displayed". For details of the setting for screen display/not-display, refer to 1. "Display Select Screen" in IV. PARAMETER, SECTION 3 DESCRIPTION OF PARAMETER AND SETTING PROCEDURE in the Operation Manual of OSP-U100M/U-10M.

- (3) Set the workpiece weight (in tons) and press the [WRITE] key.
- (4) Press the BACKUP key.
- (5) When the power is turned on next, the servo data best matching the set workpiece weight is automatically selected. The maximum workpiece weight of the automatically selected servo data is displayed for MAX WORK WEIGHT (tons).
- **NOTICE** : (1) When the setting for the WORK WEIGHT SET item is "0" (initial value), or if the set workpiece weight exceeds the allowable maximum load of the machine, the servo data for the allowable maximum load is selected when the power is turned on.
 - (2) If the value set for WORK WEIGHT SET is changed, the following alarm message is displayed.
 - 4217 Parameter setting

2. Parameter Setting Display



Fig. 1

3. Parameter Setting Range

ltem	Effective Command	Initial Value	Min. Value	Max. Value	Setting Unit
Work weight set	Setting	0	0	99	ton

LIST OF PUBLICATIONS

Publication No.	Date	Edition
4188-E	March 1998	1st
4188-E-R1	November 2000	2nd

This manual may be at variance with the actual product due to specification or design changes.

Please also note that specifications are subject to change without notice. If you require clarification or further explanation of any point in this manual, please contact your OKUMA representative.