INSTRUCTION MANUAL

TC-215
TC-225
INCH

SPECIAL NOTES

(1) Copying of this document either wholly or in part is prohibited.

(2) The contents of this document are subject to change without notice.

(3) This document has been complied very carefully. If any questions arises, please contact the shop where you bought the product.

Thank you for purchasing Brother CNC tapping centers model TC-215 and TC-225.
The compactly designed model provides excellent performance, reliability, operation ease and economy.
Please read the manuals carefully so that you can operate and maintain the machine in the correct manner, guaranteeing prolonged machine service life at its top performance level.

Contents

8-3-1-5-1. Center hole work
" 1-5-2. Hole boring work
" 1-5-3. Tap work
" 1-5-4. Reamer
" 1-5-5. Counter bore hole work
" 1-5-6. Counter bore tap work
" 1-5-7. The counter bore reamer work
" 1-5-8. Top face cutting
" 1-5-9. Pattern, Z axis position, XY position, peripheral feed
" 1-5-10. Coolant
" 1-5-11. XY Movement
" 1-5-12. Signal output
" 1-5-13. Subprogram
" 1-5-14. Program stop
8-3-2. Setting of Machining Order
8-3-3. Setting of Tool Pattern
8-3-4. Setting of Cutting Tool/Condition
8-3-5. Tool Assign
" 5-1. Information necessary for tool assignment
" 5-2. Assign operation
" 5-3. Forced tool assign
" 5-4. Sampled display of assign
" 5-5. Tool check
" 5-6. Tool Reassign
8-3-6. Program and Memory Display: Order
8-3-7. External Program Input and Output
" 7-1. External memory display
" 7-2. Program output to the external unit
" 7-3. Program input from the external unit
" 7-4. External program delete
" 7-5. External data output
" 7-6. Data input from the external unit
8-3-8. Setting of Working Zero Position
8-3-9. Setting in the Magazine

I Cautions for handling the machine

Cautions for handling the machine

o Installation and service location (See Chapter III Installation 1. "Service conditions".)

1. Install the machine at the flat and stable place.

2. Do not install the machine at the place near fire or heating element, or where temperature steeply changes.

3. Do not install the machine at the place under sunlight, near a vent hole, or with a high temperature.

4. Use dry clean air for the air blast equipment.

o Operation

1. Confirm the following before turning on the power:
   * No obstacle is left near the machine.
   * The CNC controller door is close.

2. Wear the uniform suitable for the operation.
   (Safety shoes, protectors, goggles, etc.)

3. Confirm the following before starting the operation:
   * No dirt or dust is found on the taper portion at the end of the spindle.
   * No dirt or dust is found on the taper portion of the tool held by the magazine, or no chipping or abrasion is found on each section of the machine.
   * The tool is securely held and no pull-stud is loosened.

4. Do not touch the rotational or moving element during operation.

5. Use components specified by us for the electrical equipment in the CNC controller.

6. Turn off the power before leaving the machine.

7. Be sure to warm up the machine before restarting it after shutdown for 3 days or more. (For spindle rotation, table movement, and ATC)

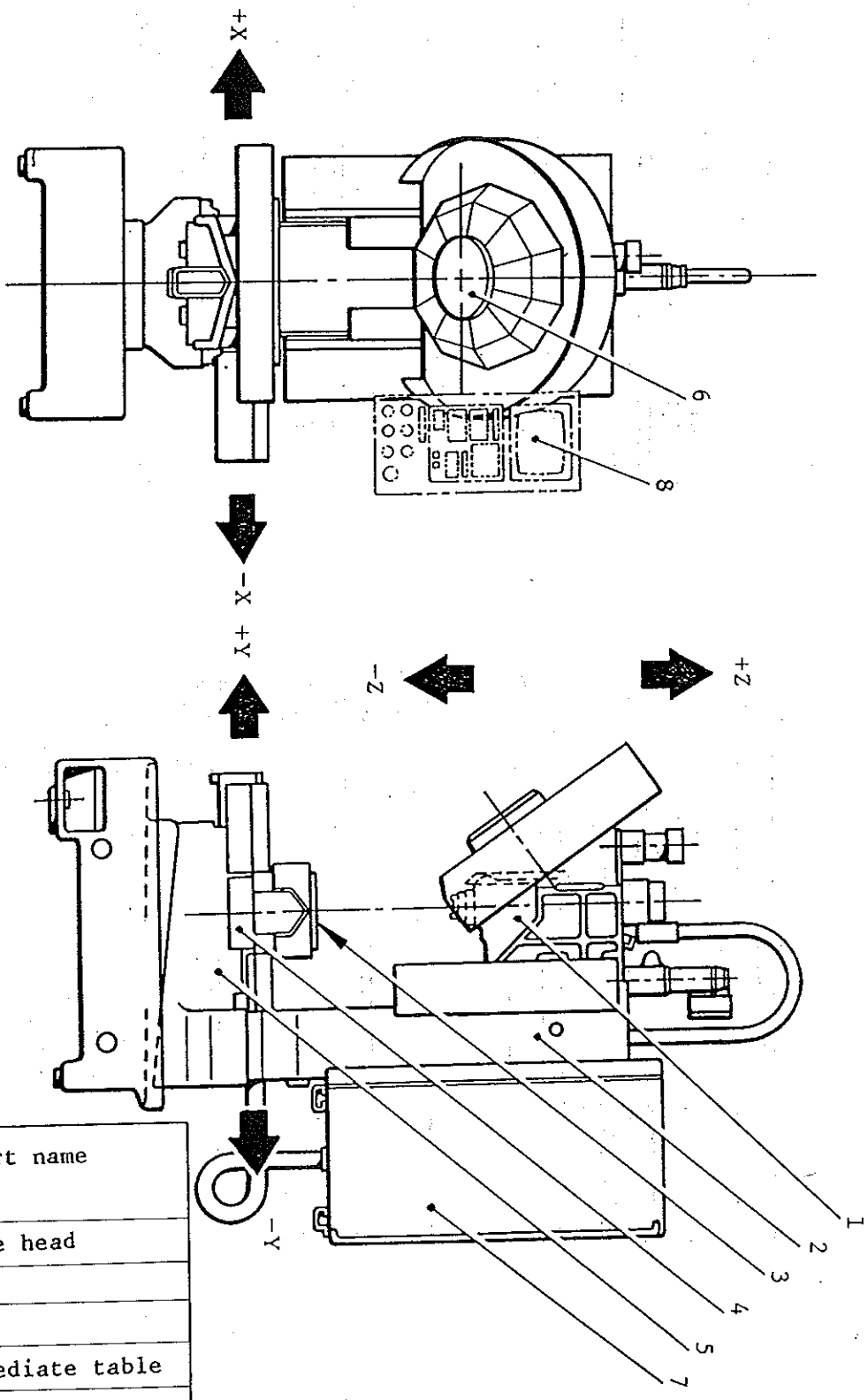
o Adjustment and component installation

1. Turn off the power before starting adjustment and checking.

2. Do not use compressed air to clean the machine body, CNC controller, and operation panel, or to eliminate chips. Pay special attention to protecting the machine from water.

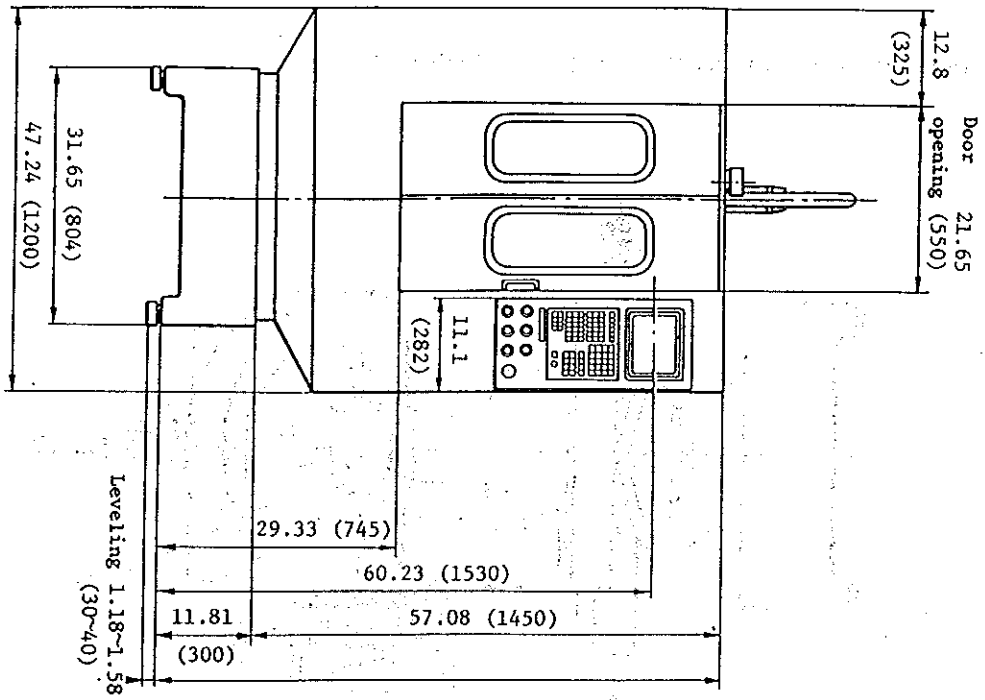
II Outline

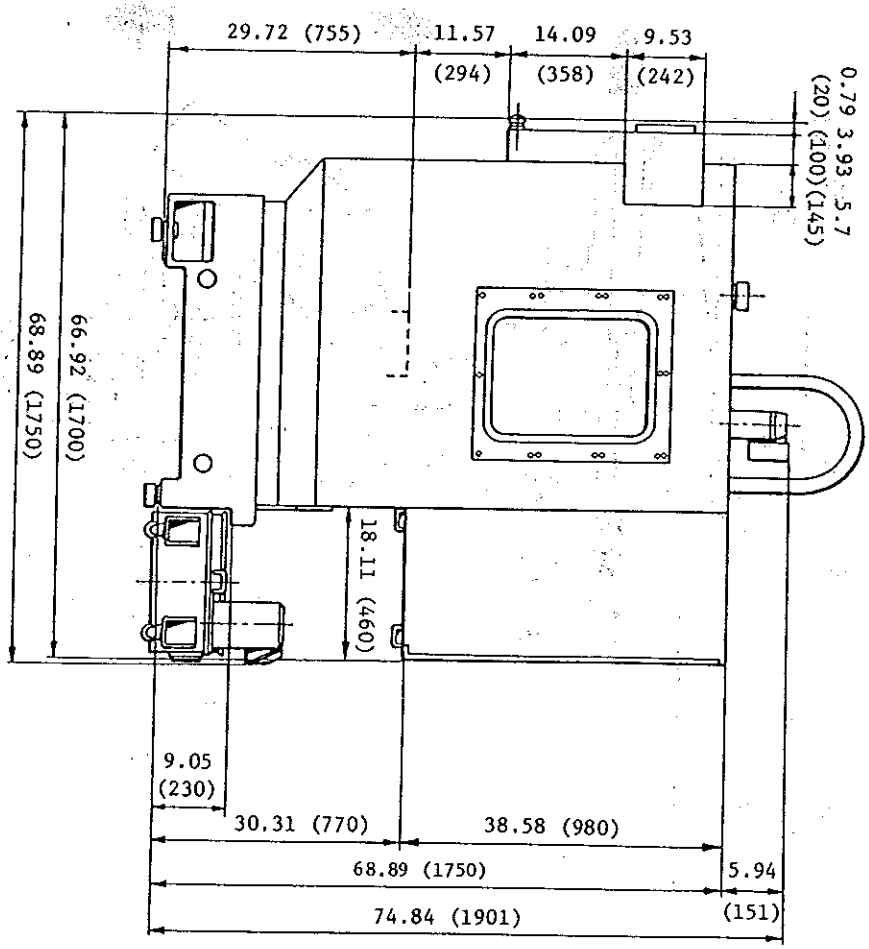
1. Each part name of machine and axis direction



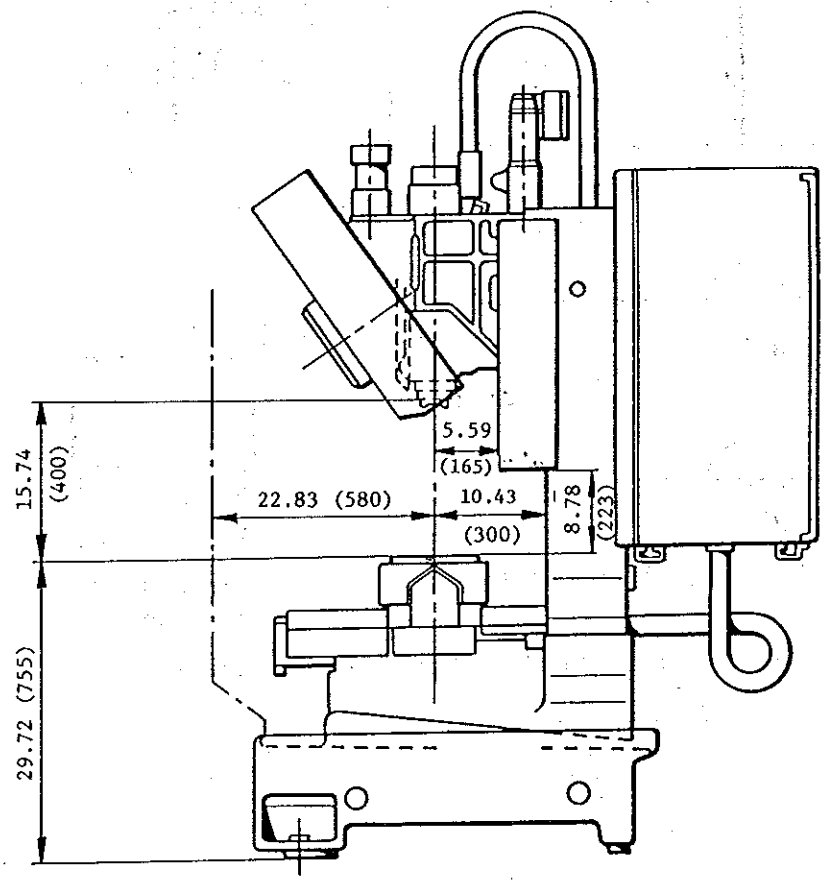
Part No.	Part name
1	Spindle head
2	Column
3	Table
4	Intermediate table
5	Base
6	Magazine
7	Control case
8	Operation panel

2. Outline drawing

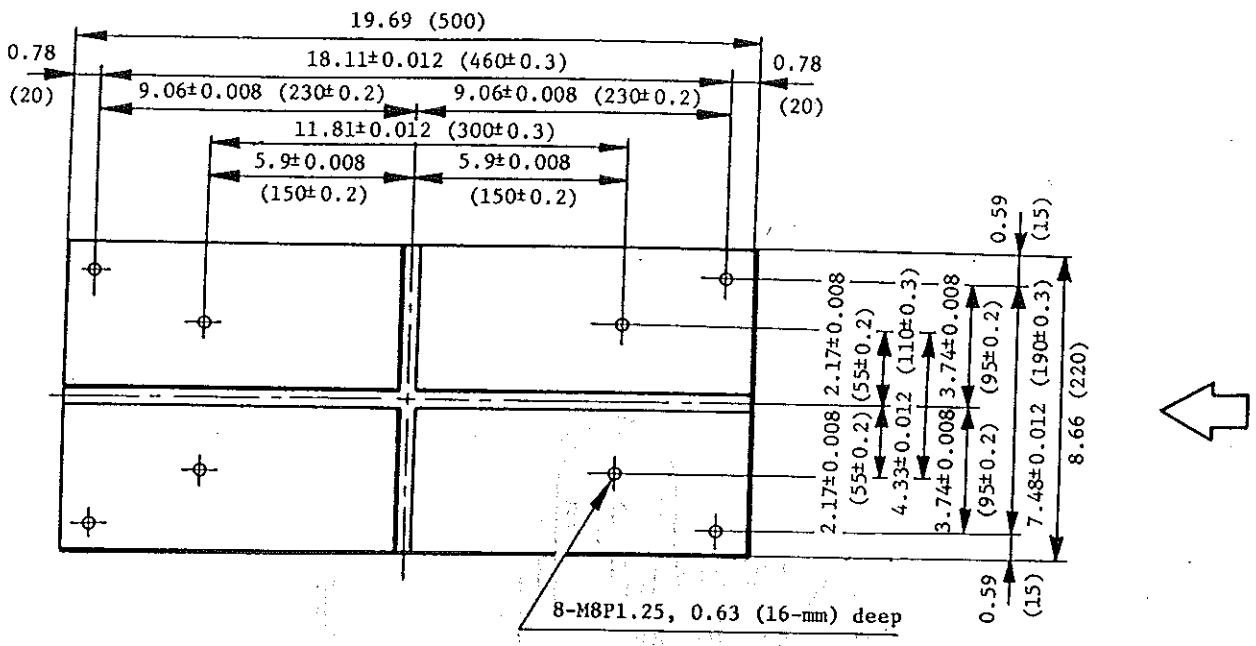


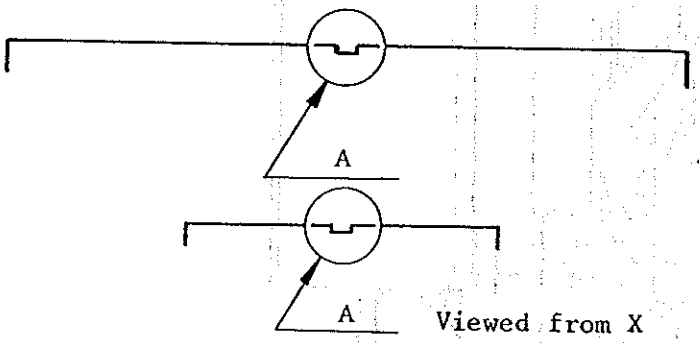


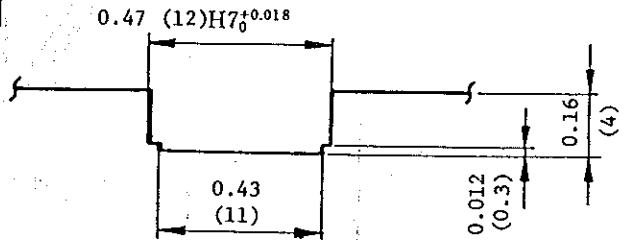
TC-215


15.74
(400)
29.72 (755)
22.83 (580)
5.59
(165)
10.43
(300)
8.78
(223)

Dimensions of table top

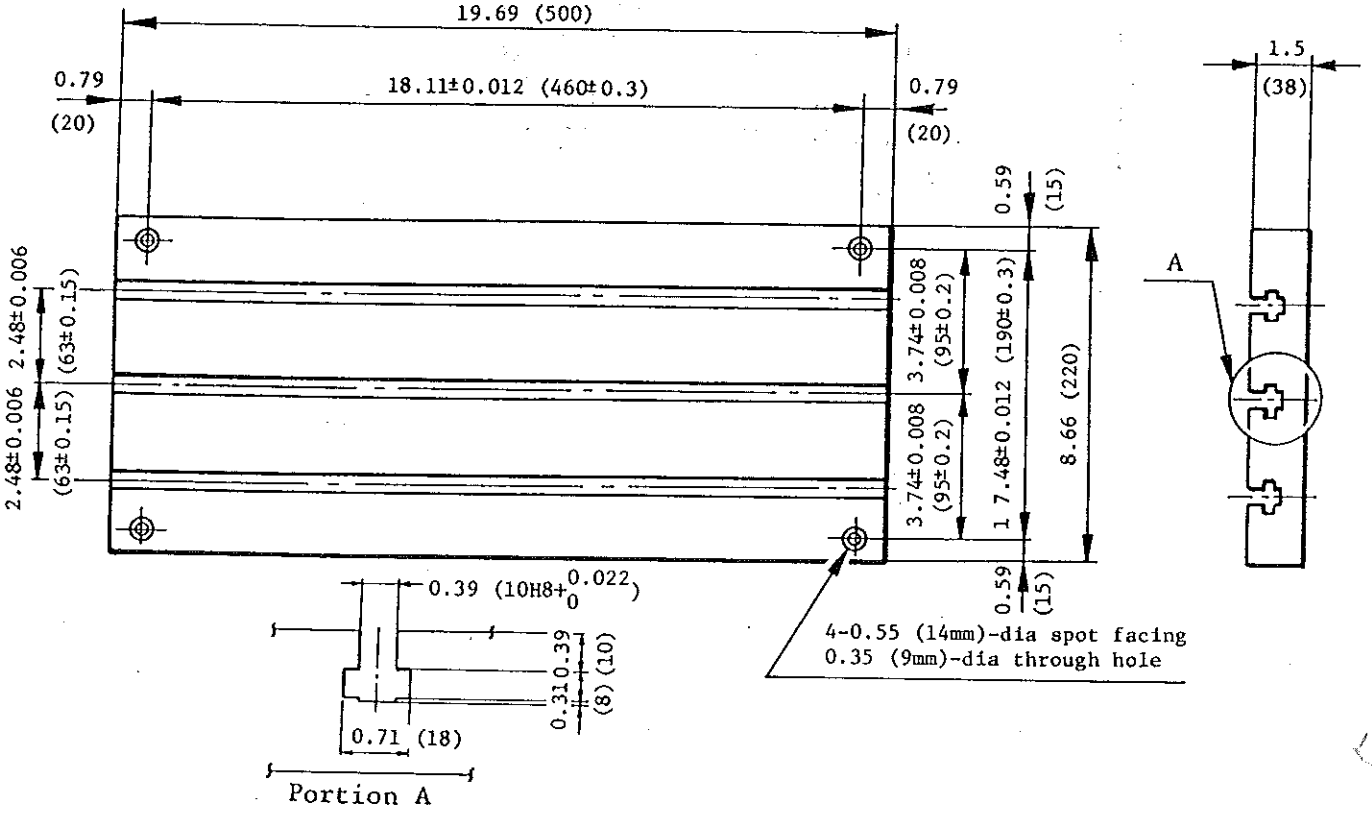


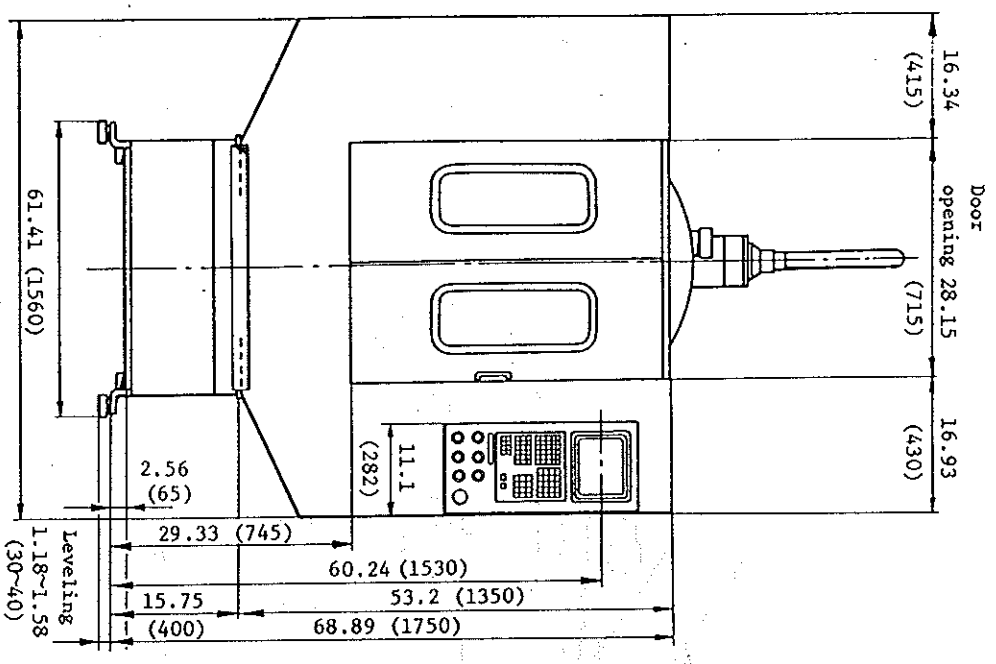


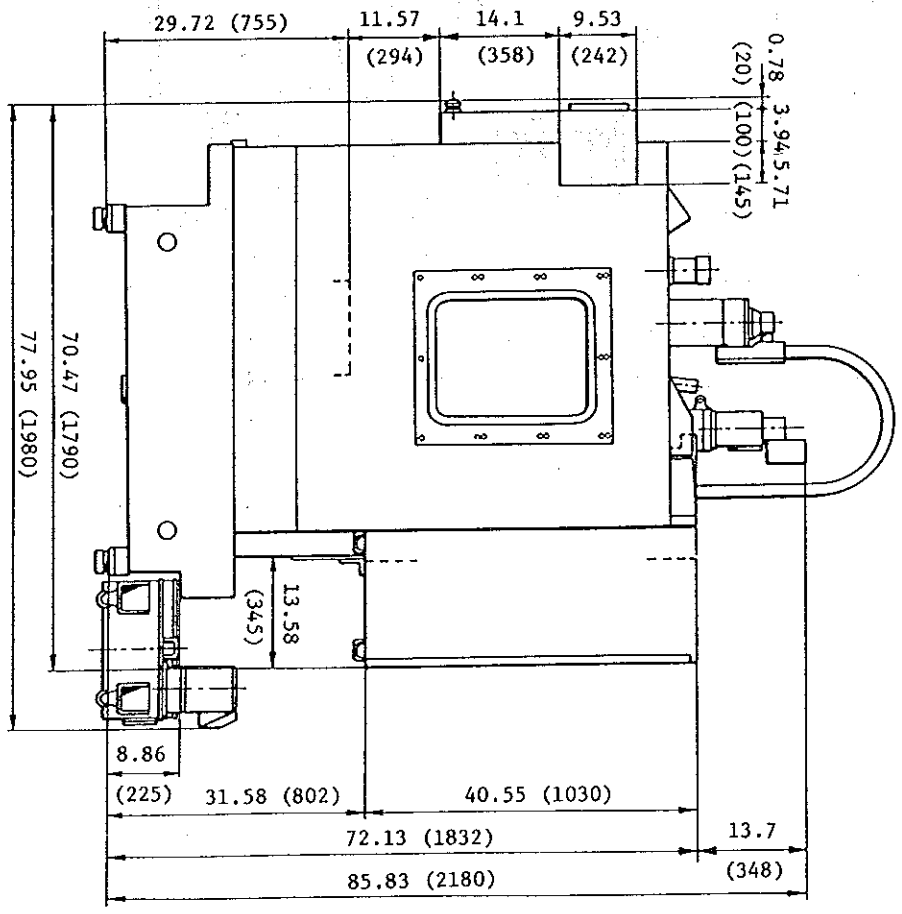


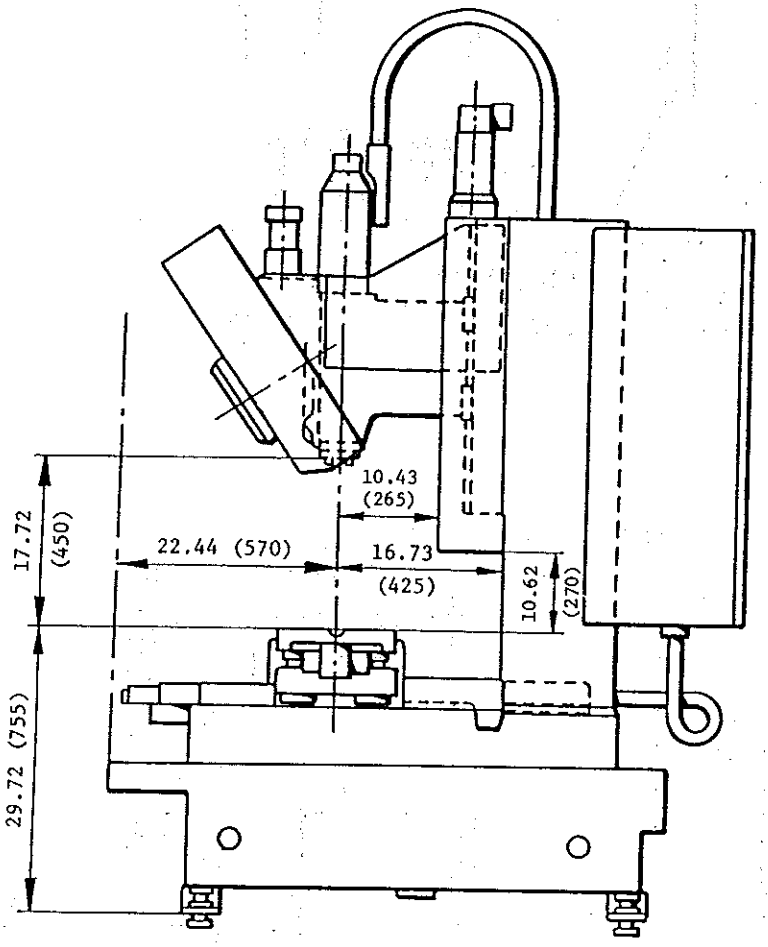
Details of portion A

Dimensions of subtable top (TC-215: Optional accessory)

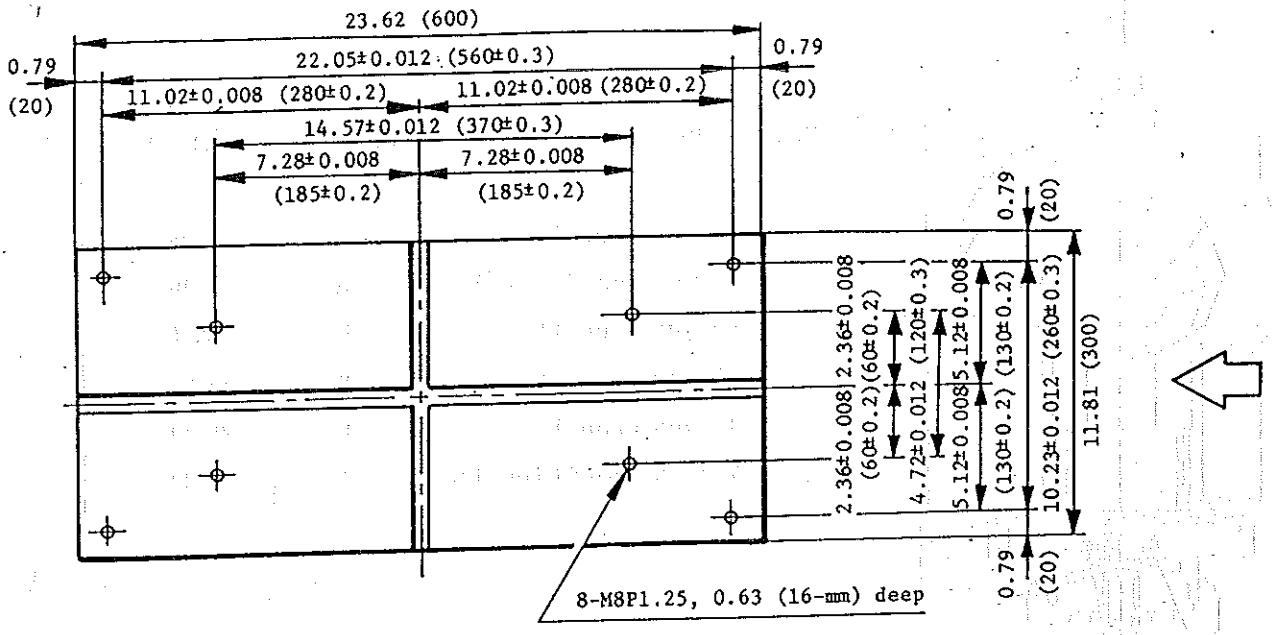


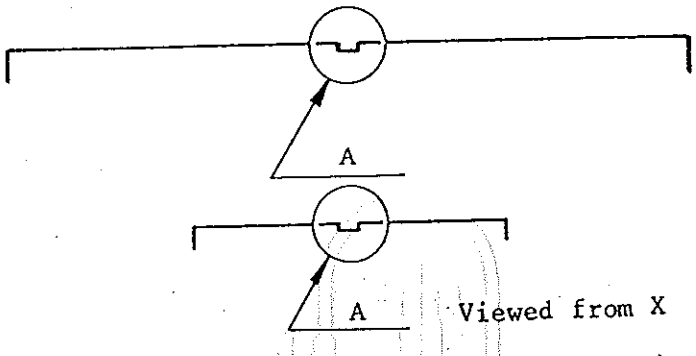

16.34
(415)
Door
opening 28.15
(715)
16.93
(430)
61.41 (1560)
11.1
(282)
2.56
(65)
Leveling
1.18~1.58
(30~40)
29.33 (745)
60.24 (1530)
15.75
(400)
53.2 (1350)
68.89 (1750)

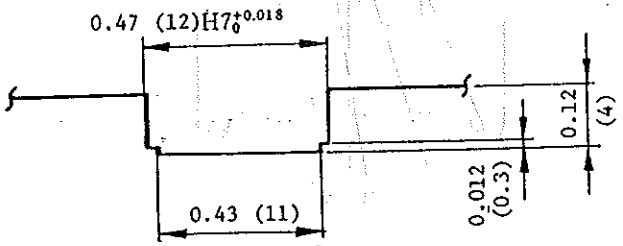

29.72 (755)
11.57
(294)
14.1
(358)
9.53
(242)
0.78
(20)
3.94
(100)
5.71
(145)
77.95 (1980)
70.47 (1790)
13.58
(345)
8.86
(225)
31.58 (802)
40.55 (1030)
72.13 (1832)
13.7
(348)
85.83 (2180)


10.43
(265)
22.44 (570)
16.73
(425)
10.62
(270)
17.72
(450)
29.72 (755)

Dimensions of table top

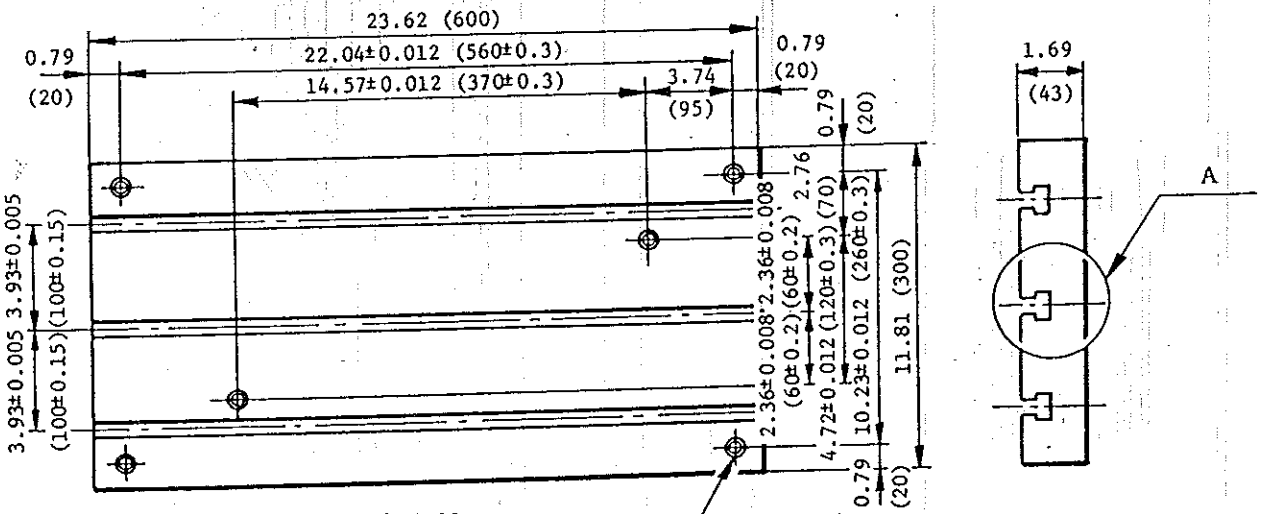


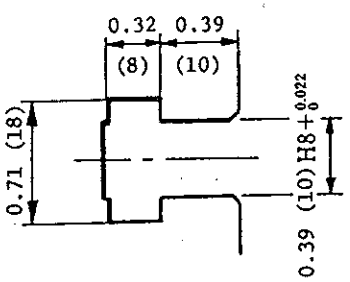




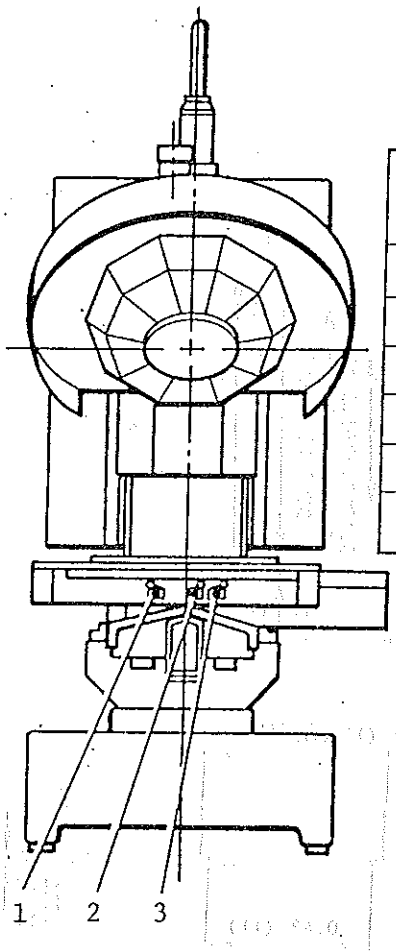
Details of portion A

Dimensions of subtable top (TC-225: Optional accessory)



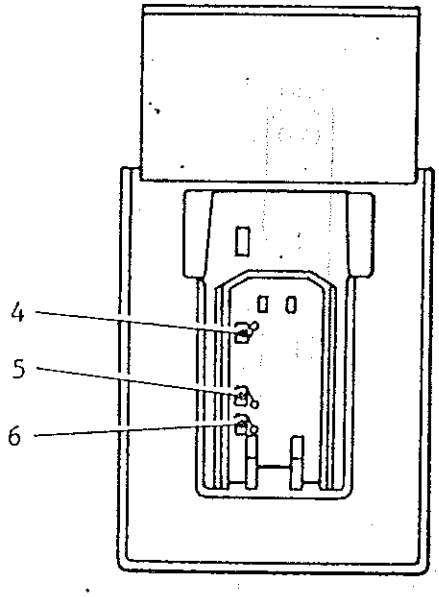


Each part name of LS

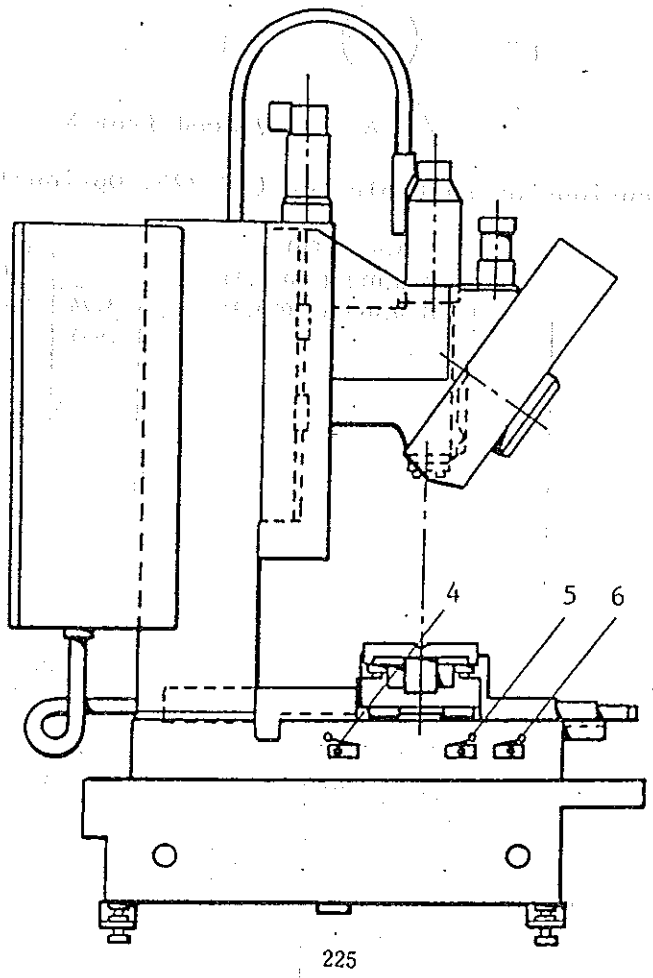


Part No.	Part name	Contact	Cable No.
1	+X overrun LS	B	25.26
2	X zero-position LS	A	39.40
3	-X overrun LS	B	27.28
4	-Y overrun LS	B	31.32
5	+Y overrun LS	B	29 30
6	Y zero-position LS	A	41.42

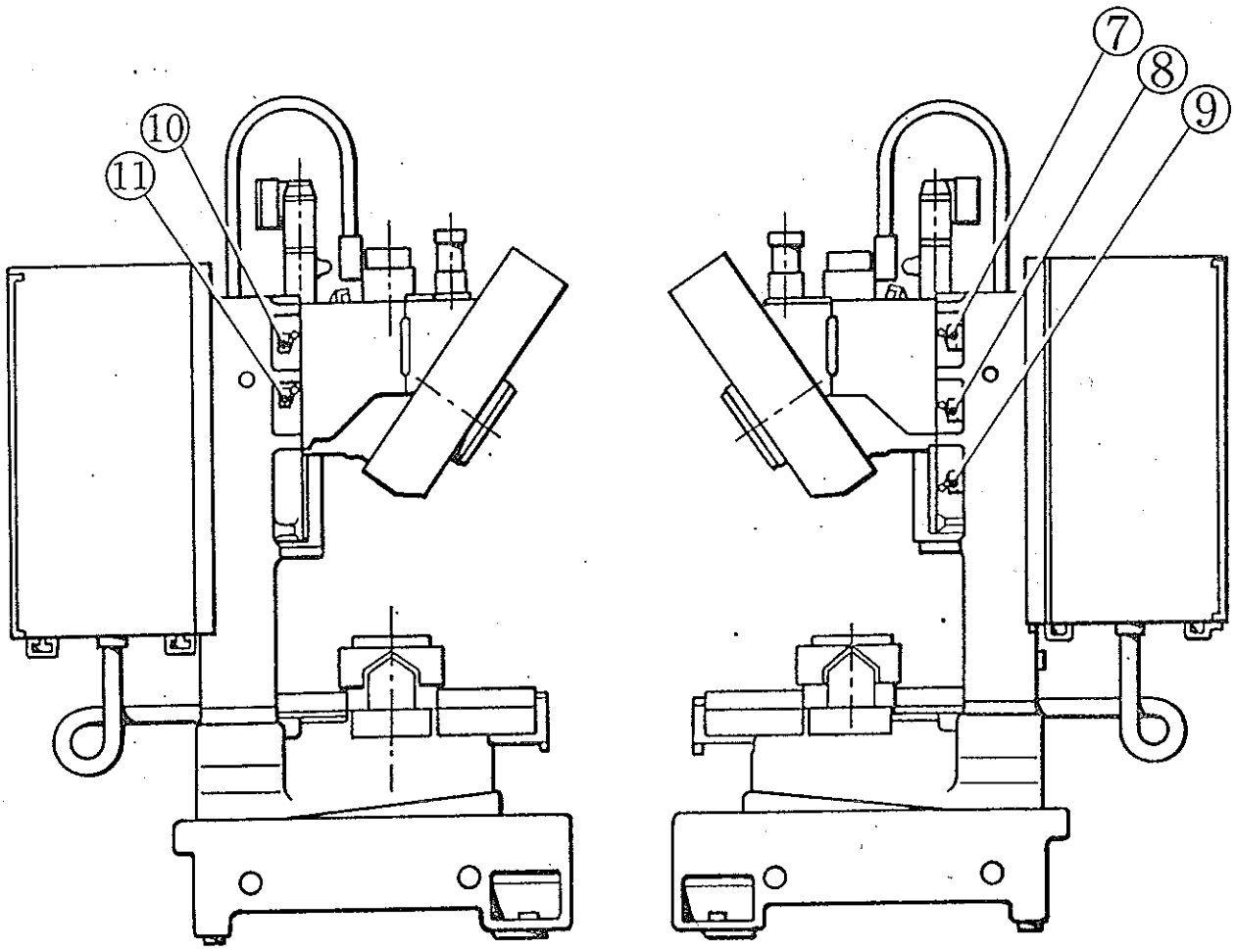
X-axis (common to 215 and 225)



215



Y 225



Left side drawing　　Right side drawing

Z-axis ARC (common to 215 and 225)

Part No.	Part name	Contact	Cable No.
7	+Z overrun LS	B	33.34
8	Z zero-position LS	A	43.44
9	-Z overrun LS	B	35.36
10	Z ATC zero-position LS	A	45.46
11	Z ATC area LS	A	47.48

3. Features and configuration of machine

Features

(1) ATC (Automatic Tool Changer)

* Non-cutting time can be saved because the machine uses a unique armless ATC mechanism to change tools simply and quickly.

* Since the magazine rotates clockwise and counterclockwise because of using the random shorter circuit system, it approaches the tool to be used through the closest way in a short time.

* Non-cutting time can be saved because X/Y positioning is made during rotation of the magazine.

(2) Tapping

* No tapper is needed and wasteful air cut amount can be the same with that of drilling because of the completely synchronous feed based on one pitch per turn of spindle.

* Maximum speed of spindle during tapping
  215 --- 4000min^{-1}(rpm)
  225 --- 3000min^{-1}(rpm)

(3) Spindle head

* Because the spindle uses precision bearings, a high accuracy is maintained for a long time.

* It is hardly needed to supply grease because of the grease-sealed system.

* The speed can be controlled between 6 and 6000 min^{-1}(rpm) by using an AC servo motor and assigning an optional peripheral speed of the tool (cutting speed) (however, the speed of up to 10 ~ 10000 min^{-1}(rpm) can be controlled for an optional specifications).

(4) Feed mechanism (X, Y, and Z)

* It is unnecessary to replace and maintain the brush because of using the AC servomotor.

* The machine is designed to have no stick slip or backlash because the AC servomotor is directly connected with the precision ball screw, the guide uses an LM guide, and smooth feed is possible.

(5) Interactive CNC controller developed by us

* It is possible to easily input data and quickly make programming because the equipment uses the automatic programming system and stores different data and menu. Though programming is backed up, the bubble-memory cassette (optional accessory) can be also used when an external memory is required.

Specifications for machine body

TC-215

Movement	X-axis movement (Table, longitudinal)	11.81inch (300mm)
	Y-axis movement (Table, cross)	8.66inch (220mm)
	Z-axis movement (Spindle head, vertical)	7.87inch (200mm)
	Distance between table top and spindle end	7.87 ~ 15.75inch (200 ~ 400mm) High column based on optional specifications [5.91inch (150mm)] 13.78 ~ 21.6inch (350 ~ 550mm) High column based on optional specifications [9.84inch (250mm) 17.72 ~ 25.6inch (450 ~ 650mm)
	Distance between column front and spindle center line	11.81inch (300mm)
Table	Size of table work area	19.69inch x 8.66inch (500mm x 220mm)
	Maximum load of table	165lbs (75kg)(Uniform)
Spindle	Spindle speed	6 - 6000min^{-1}(rpm) Optional specifications 10 ∽ 10000min^{-1}(rpm)
	Maximum speed for tapping	4000min^{-1}(rpm)
	Spindle taper hole	NT30
	AC spindle servomotor	1.5kW (Continuous)/ 2.1kW (30 min)
Feed and related matters	High travel rate (max)	65.6ft/min (20m/min)
	Cutting travel rate	0.19 - 196.85inch/min (5 - 5000mm/min)
	Minimum movement unit	0.0001inch (0.001mm)
	Feed AC servomotor	X, Y :0.15kW Z :0.3kW
ATC and related matters	Tool shank system	BT30
	Pull-stud type	P30T-2

ATC and related matters	Number of tools to be housed		10
	Maximum tool diameter		1.97inch (50mm)
	Maximum tool length		7.87inch (200mm) (From the gauge line)
	Maximum tool weight/total tool weight		3lbs/53lbs (1.5kg/12kg)
	Tool selection system		Random shorter circuit
	Tool change time (Tool to tool)		1.8s (MAS011)
	Tool change time (Chip to chip)		4.0s
Machine size and weight	Machine height		74.84inch (1901mm)
	Floor area size required		47.24inch X 95.67inch (1200mm X 2430mm)
	Machine weight		2500lbs (1140kg)
Accuracy	Positioning accuracy		±0.0039/11.81inch }6334 (±0.01mm/300mm) JIS B6334
	Repeated positioning accuracy		±0.0002inch (±0.005mm)
Power supply	3 Phase AC V ± 10%, 50/60 ± 1 Hz		
Electric capacity		6000 min^{-1}(rpm)	1000 min^{-1}(rpm)
	Continuous rating	4.8KVA	5.4KVA
	30-min rating	5.5KVA	6.0KVA
	Instantaneous rating	10KVA	12KVA

Specifications for machine body

TC-225

Movement	X-axis movement (Table, longitudinal)	16.54inch (420mm)
	Y-axis movement (Table, cross)	11.81inch (300mm)
	Z-axis movement (Spindle head, vertical)	9.84inch (250mm)
	Distance between table top and spindle end	7.87∽17.71inch (200∽450mm) High column based on optional specifications [5.9inch (150mm)] 13.78∽23.62inch (350∽600mm)
	Distance between column front and spindle center line	16.73inch (425mm)
Table	Size of table work area	23.62 x 11.81inch (600mm x 300mm)
	Maximum load of table	330lbs (150kg) (Uniform)
Spindle	Spindle speed	6 - 6000min^{-1}(rpm)
	Maximum speed for tapping	3000min^{-1}(rpm)
	Spindle taper hole	NT30
	AC spindle servomotor	3.1kW(Continuous)/ 4.1kW(30 min)
Feed and related matters	High travel rate (max)	X, Y 65.6ft/min (20m/min) Z 49.2ft/min (15m/min)
	Cutting travel rate	0.19 - 196.85inch/min (50 - 5000mm/min)
	Minimum movement unit	0.0001inch (0.001mm)
	Fee AC servomotor	X, Y :0.41kW Z :0.72kW
ATC and related matters	Tool shank system	BT30
	Pull-stud type	P30T-2

ATC and related matters	Number of tools to be housed	10
	Maximum tool diameter	3.15inch (80mm)
	Maximum tool length	7.87inch (200mm) (From the gauge line)
	Maximum tool weight/total tool weight	6 lbs/40 lbs. (3.0kg/20kg)
	Tool selection system	Random shorter circuit
	Tool change time (Tool to tool)	2.2s (MAS011)
	Tool change time (Chip to chip)	5.2s
Machine size and weight	Machine height	84.45inch (2145mm)
	Floor area size required	61.42inch X 70.47inch (1560mm X 1790mm)
	Machine weight	3760 lbs (1880kg)
Accuracy	Positioning accuracy	±0.00039/11.81inch (±0.01mm/300mm) JIS B6334
	Repeated positioning accuracy	±0.0002inch (±0.005mm)
Power supply	AC 3-phase V ± 10%, 50/60 ± 1 Hz	
Electric capacity	Continuous rating	7.0 KVA
	30 min rating	9.0 KVA
	Instantaneous rating	16.5 KVA

Specifications for CNC system
For both TC-215 and -225

Function	Details
Number of control axes	Three axes of X, Y, and Z
Number of simultaneous control axes	Simultaneous biaxial control is possible for X- and Y-axes. Simultaneous uniaxial control is possible for Z-axis.
Minimum setting unit (inch)	0.0001 inch
Minimum movement unit (inch)	0.0001 inch
Command system	Absolute
Maximum command value (inch)	±99.9999 inch
Display	9-inch CRT
Decimal-point entry	Decimal point can be used for data entry.
MDI	Different NC data is assigned through key-in operation.
Home-position return	Automatic home-position return is made by key-in operation.
Machining home-position return	Automatic home-position return is made by key-in operation.
Present position display	It is possible to display the present coordinate position of each axis and magazine status.
Coordinate system setting	It is possible to set the coordinate system at the position commanded for X-, Y-, and Z-axes.
Over travel	As soon as the overrun limit switch of each axis operates, the operation stops.
Stroke limit	It is possible to set a work area for each axis with parameters.
Emergency stop	Every command and the machine are immediately stopped by emergency stop.
High travel rate	Axis-direction travel rate of 65.6 ft/min for the X- and Y-axes (5-step travel rate for manual operation), Axis-direction travel rate of 49.2 ft/min only for the Z-axis of 225 (22-step travel rate for manual operation)
Cutting travel rate	Axis-direction travel rate of 0.19-196.85 inch/min for each axis (22-step travel rate for manual operation)
Step movement	Four-step movement between 0.0001 and 0.1 inch is possible for each axis.
Temporary stop	It is possible to temporarily stop the movement of each axis. The operation can be continued by restarting each axis.
Automatic acceleration/ deceleration	Automatic acceleration/deceleration is made by the movement command.
Pitch feed	Completely synchronous feed is made between screw pitch and one turn of spindle.

Function	Details
Interpolation function	Linear interpolation
Fixed cycle	It is possible to perform drilling and tapping according to a series of fixed operation sequence.
Machining start point setting	It is possible to restart the operation from halfway of the program.
Pitch error correction	It is possible to correct mechanical error due to each screw feed of X-, Y-, and Z-axes. The correction interval is constant and can be set up to 15 points.
Dry run	The program can be quickly checked through the high travel rate in the manual operation mode.
Dwell	It is possible to make the Z-axis dwell for the time assigned by the parameter only when chamfering or spot facing is complete.
Tool function	A necessary tool is automatically assigned among the entered tools according to the edited program.
Spindle function	A spindle speed is automatically assigned by setting a tool diameter along with the set material and cutting condition.
Backlash correction	It is possible to correct in pulses the lost motion produced in the mechanism for each axis within the range of 0 to 311 pulse.
Tool length setting	It is possible to automatically store the tool length measured by installing it on the machine in the tool data memory.
Automatic programming	Automatic programming is made by selecting the input data and interactive menu.
Background programming	A program can be also edited during memory operation. (However, there is partial limitation.)
Program-number search	It is possible to call the stored program with a 4-digit program number.
Self-checking alarm	The CNC controller performs different checks by itself and displays the error on the CRT if any.
External program and data calling	It is possible to transfer or delete the external program and data using the bubble-memory unit (optional accessory).
Restart	This specifies, in memory run mode, machine stop or consutive operation at the of program.
Schedule program	Continuous operation by combining 8 programs in the memory run mode is possible.

Standard accessory

	215	225
Splash guard (for 215)	o	
Splash guard (for 225)		o
Instruction manual: 1	o	o
Anchor bolt: 4	o	o
Leveling plate: 4	o	o
Lamp SLL-28H	o	o
Fuse S (0.5 A): 2	o	
Fuse S (2.0 A): 1		o
Fuse S (3.0 A): 1	o	
Fuse S (5.0 A): 1		o
Fuse S (0.5 A): 1		o
Fuse S (5 A) : 1		o
Spindle alarm fuse: 1		o

Optional specifications

High column (5.91inch)	o	o
High column (9.84inch)	o	
1,000-rpm specifications	o	

Optional accessory

Item	215	225
Coolant equipment (3000 in^3)	o	o
Coolant solenoid valve	o	o
Subtable (for 215)	o	
Subtable (for 225)		o
Bubble-memory unit (with one bubble-memory cassette)	o	o
Bubble memory cassette (128KB)	o	o
Spindle-taper airblast equipment 0.4-0.5MPa (4-5fkg/cm^2	o	o
Tool breakdown detector	o	o
RS-232C interface	o	o

2-1. Machining capacity

Though the upper limit of the capacity depends on cutting conditions, consider the following values as the approximate criteria:

TC-215

Type of machining	Condition	S45C	FC20	ADC12
Drilling	Diameter, inch (mm) Feed rate inch/rev (mm/rev)	ϕ0.43 (11) 0.004 (0.1)	ϕ0.47 (12) 0.004 (0.1)	0.59 (15) 0.009 (0.25)
Tapping	Diameter, inch (mm) Pitch Thread (mm)	5/16 (M5) 18 (1.25)	3/8 (M10) 16 (1.5)	7/16 (M12) 1.75 (14)
End milling	Diameter, 0.78 inch Cutting rate inch^3/min (cm^3/min)			2.44 (40)

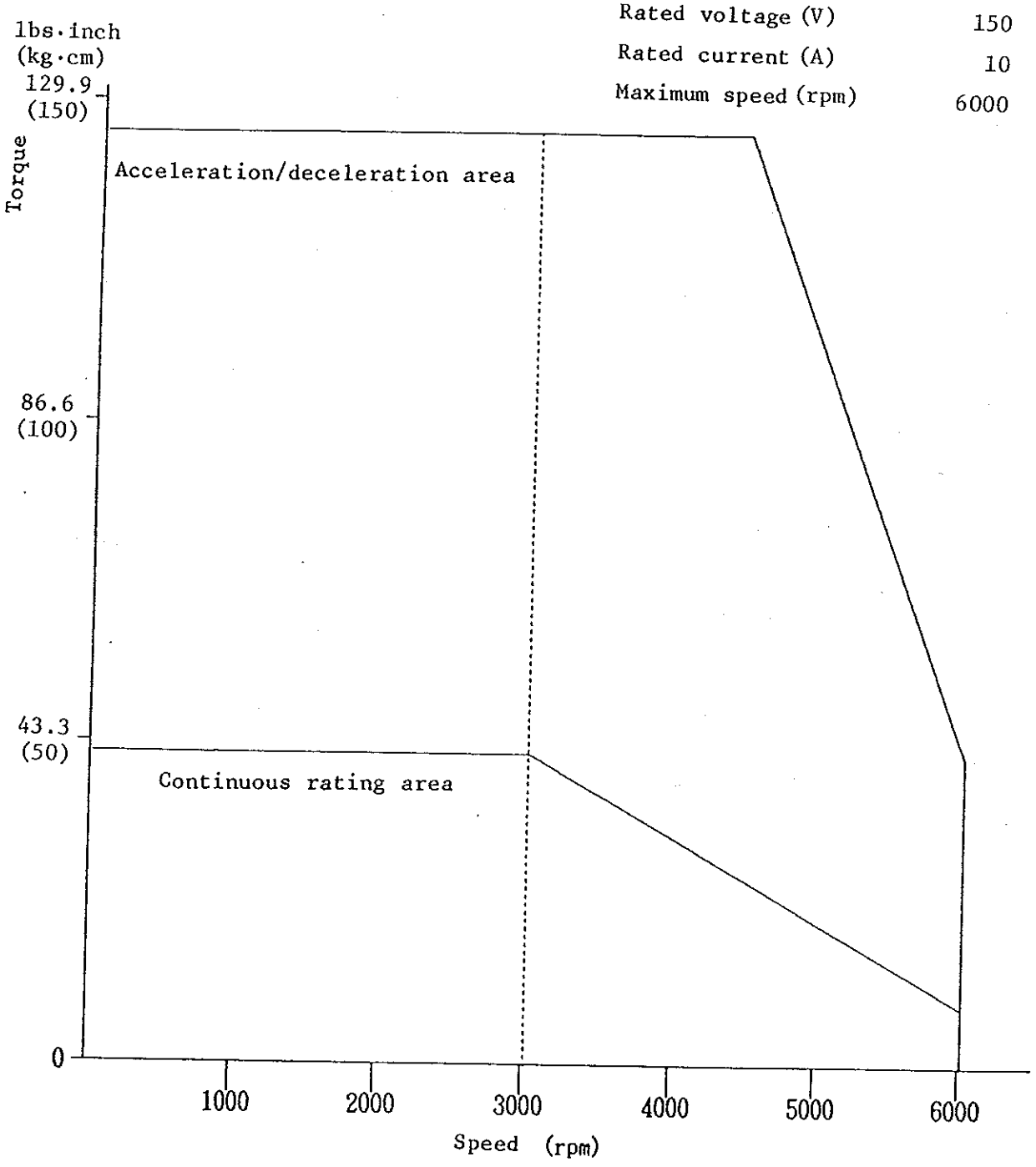
TC-225

Type of machining	Condition	S45C	FC20	ADC12
Drilling	Diameter, inch (mm) Feed rate inch/rev (mm/rev)	0.67 (17) 0.006 (0.15)	0.79 (20) 0.006 (0.15)	0.98 (25) 0.008 (0.20)
Tapping	Diameter, inch (mm) Pitch Thread (mm)	1/2 (M14) 13 (2.0)	5/8 (M16) 11 (2.0)	3/4 (M18) 10 (2.5)
End milling	Diameter, 0.78 inch Cutting rate inch^3/min (cm^3/min)	0.40 (6.6)	0.73 (12)	3.66 (60)
Face milling	Diameter, 3.1 inch Cutting rate inch3/min (cm3/min)	0.85 (14)	3.26 (37)	9.15 (150)

2-2. Spindle motor characteristics (TC-215)

RA10M-1500W-3000RPM (Max6000RPM)

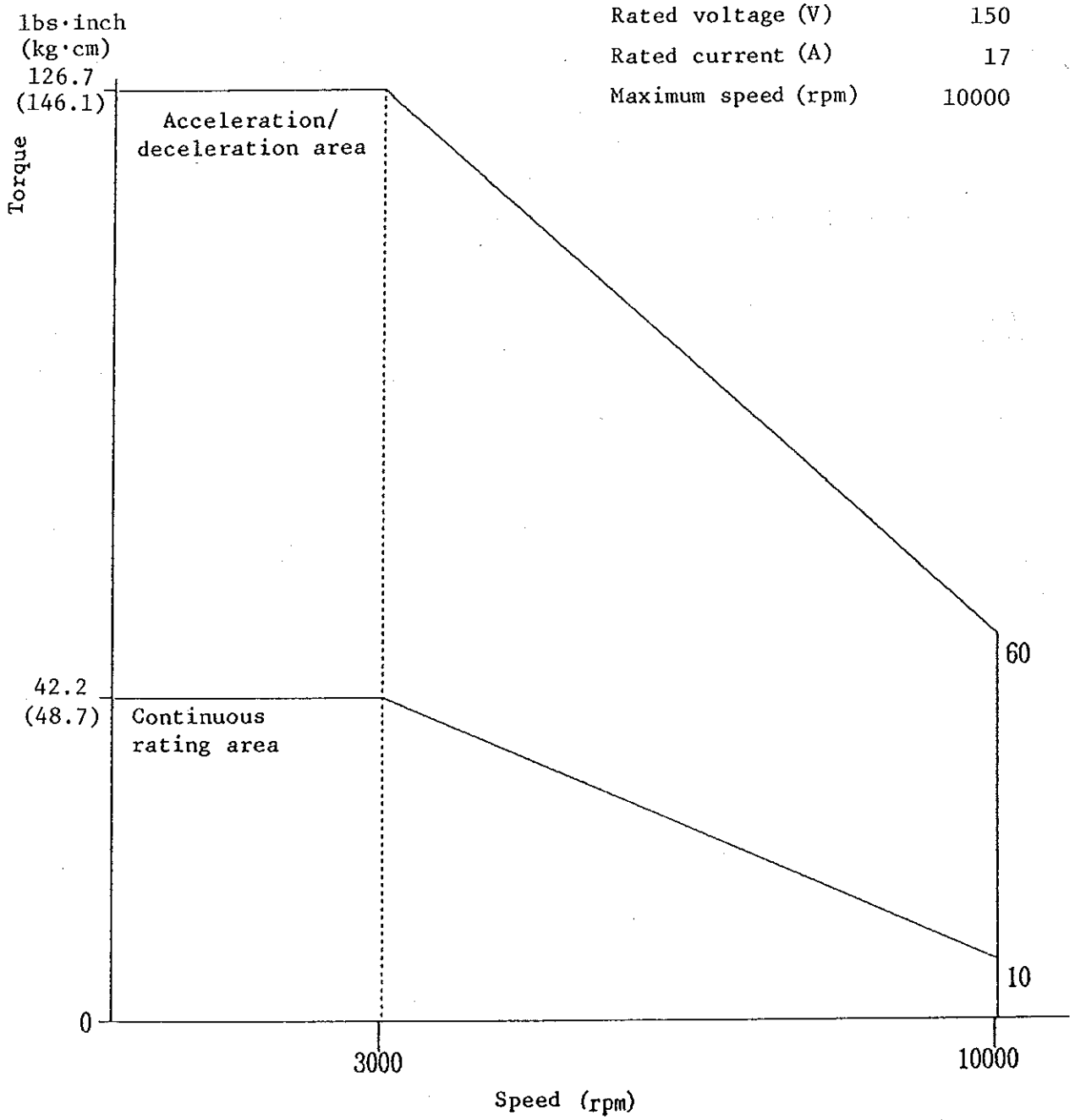
Rated output	(W)	1500
Rated torque	(lbs·inch)	42.2
Rated speed	(rpm)	3000
Rated voltage	(V)	150
Rated current	(A)	10
Maximum speed	(rpm)	6000



Spindle motor characteristics (TC-215 -- Optional specifications (10,000 rpm))

RA12M-1500W-3000RPM (Max10000RPM)

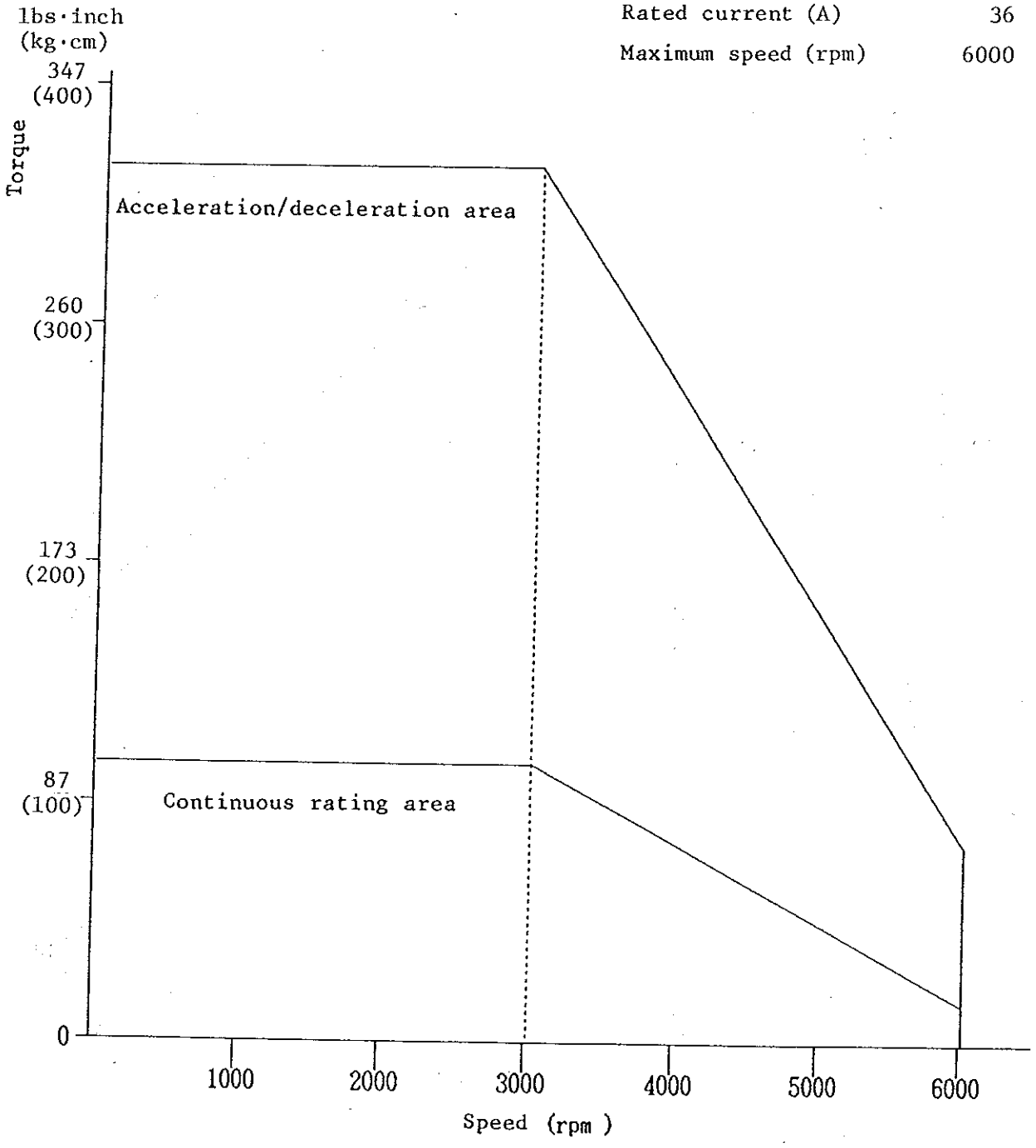
Rated output	(W)	1500
Rated torque	(lbs·inch)	42.2
Rated speed	(rpm)	3000
Rated voltage	(V)	150
Rated current	(A)	17
Maximum speed	(rpm)	10000



Spindle motor characteristics (TC-225)

BL820-20BM121FXP

Rated output	(W)	3100
Rated torque	(lbs·inch)	104
Rated speed	(rpm)	3000
Rated voltage	(V)	140
Rated current	(A)	36
Maximum speed	(rpm)	6000



III Installation

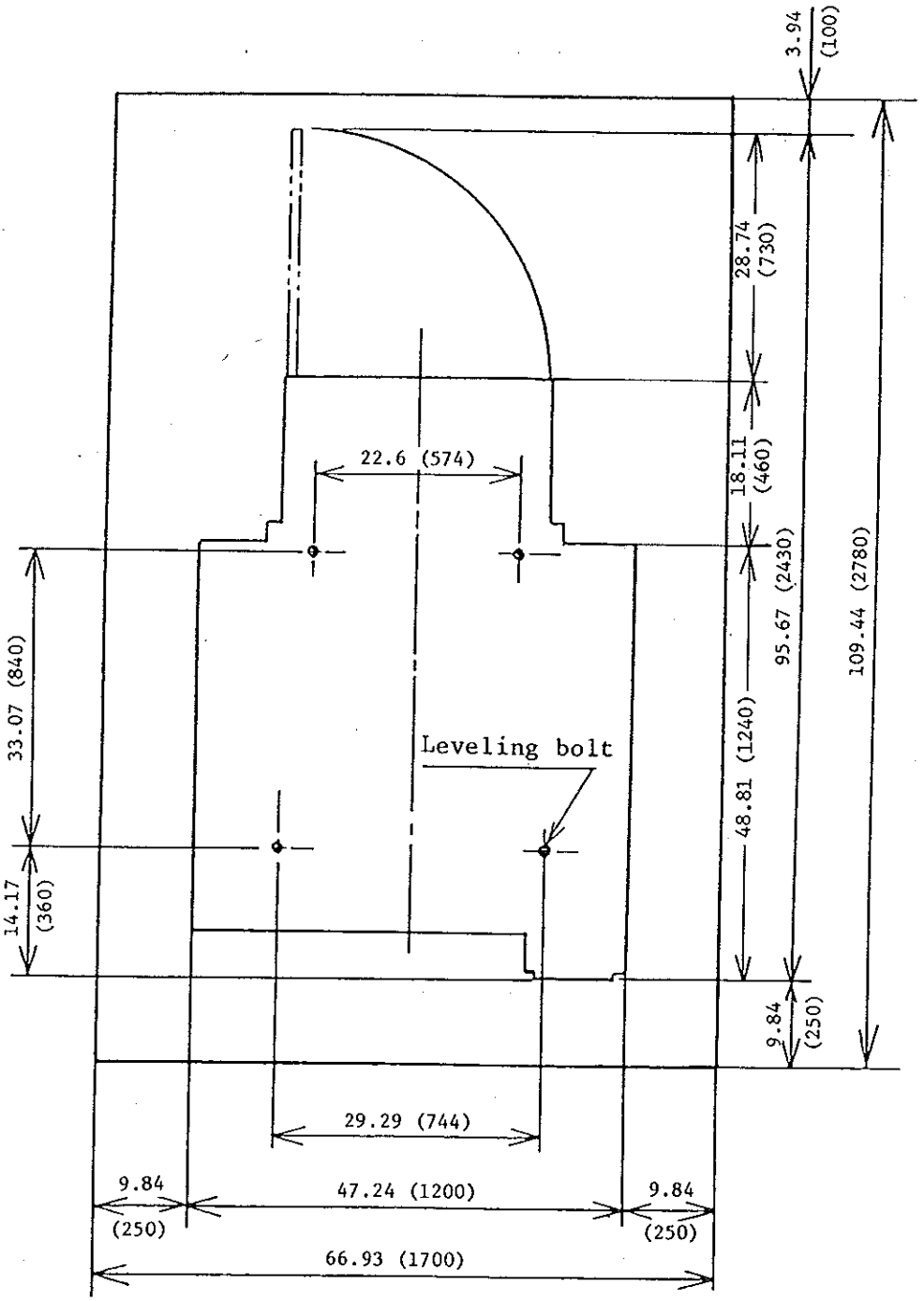
1. Service conditions

Install the CNC controller and the machine at the place with little dust to protect the moving elements of them from dust. Do not install them at the place where they are exposed to the sunlight, near a heating element, or where temperature steeply changes.

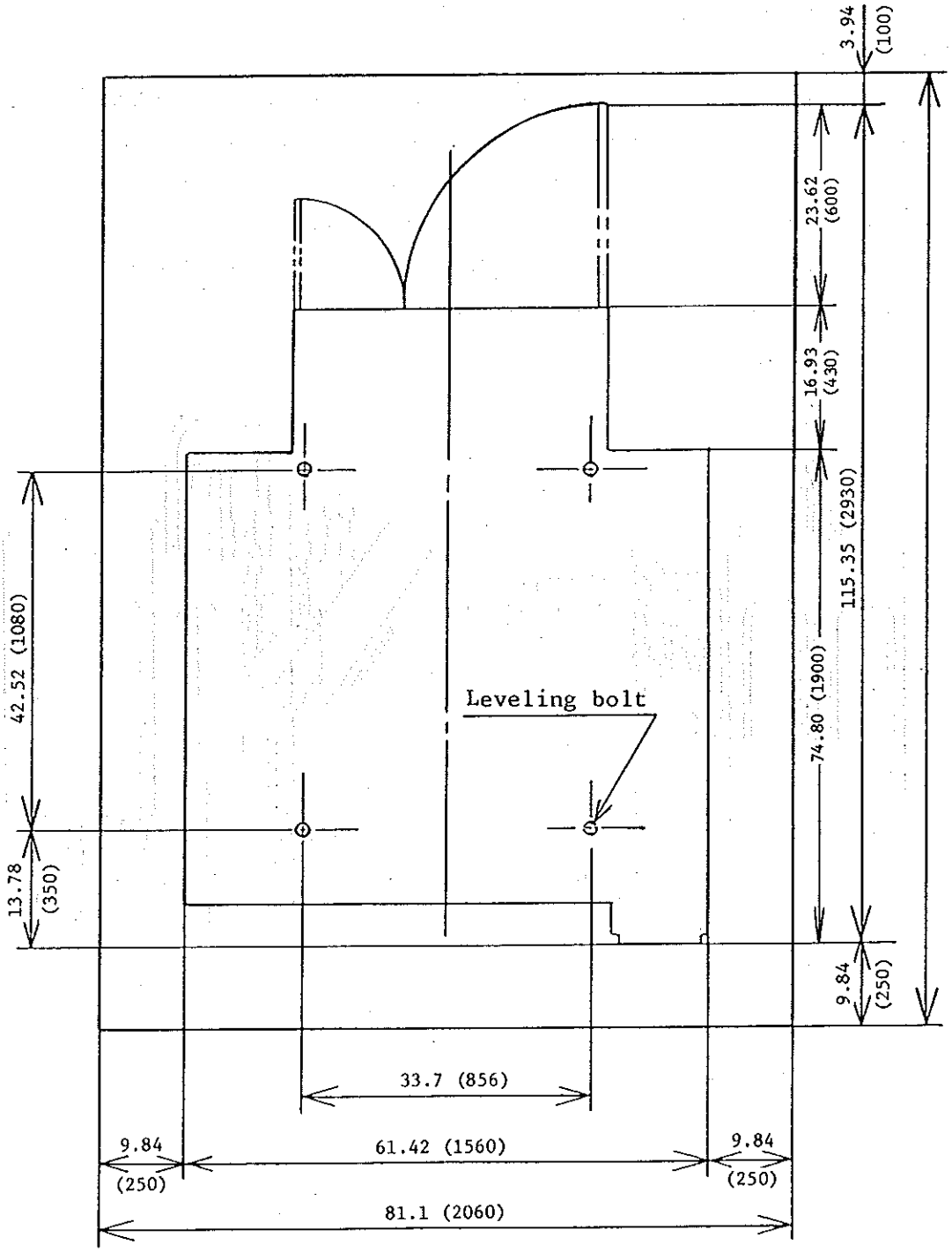
Ambient temperature	During operation: 0-45°C
	During storage or transportation: -20°-60°C
Temperature change	Up to 1.1°C/min
Humidity	75% or less (Relative humidity)
Vibration	0.5G or less

2. Installation area

TC-215



TC-225


3.94
(100)
23.62
(600)
16.93
(430)
115.35 (2930)
74.80 (1900)
42.52 (1080)
13.78
(350)
Leveling bolt
9.84
(250)
33.7 (856)
9.84
(250)
61.42 (1560)
81.1 (2060)

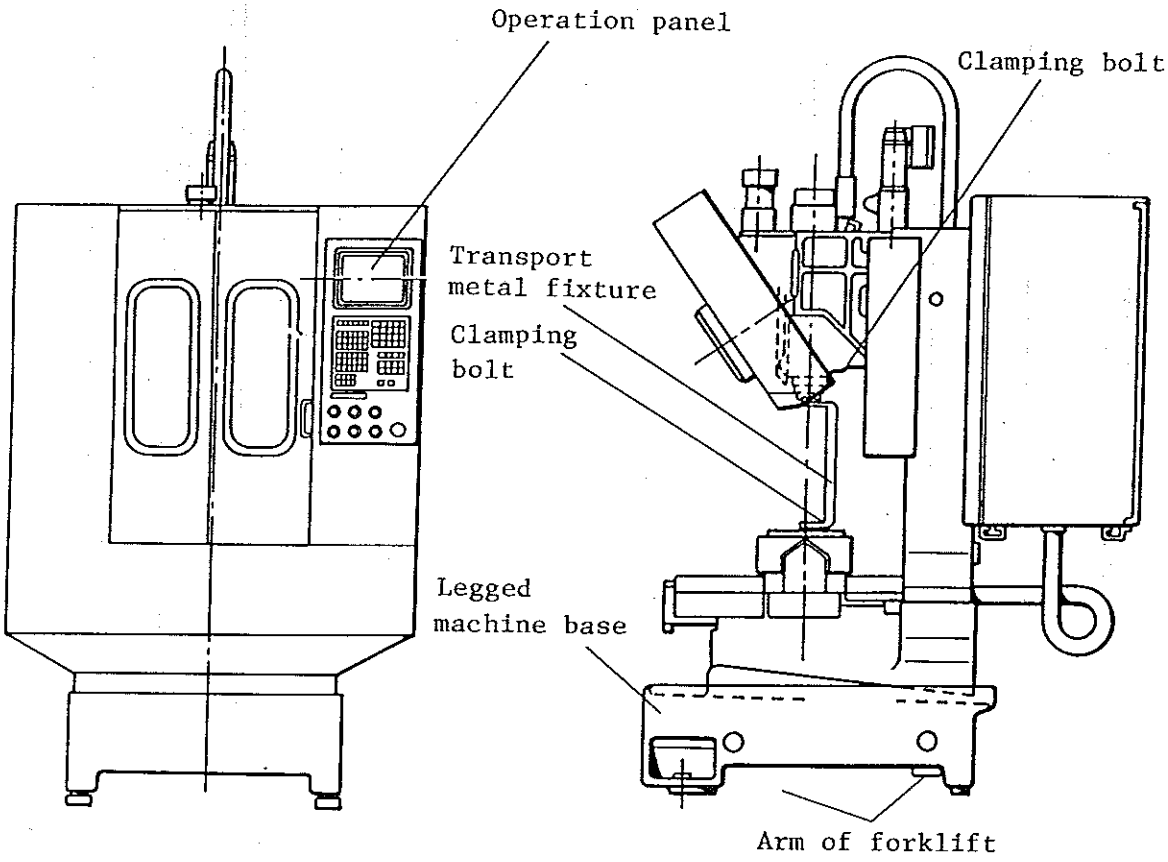
3. Transportation of Machine

The electrical and mechanical blocks of this machine are integrated. It is possible, therefore, to transport the machine by simply disconnecting the power cord.

Before transporting the machine, fix the spindle head and the table using the transport metal fixture and clamping bolts.

In consideration of the weight (TC-215: 1140 kg, TC-225: 1880kg) the machine should be transported by forklift by inserting the fork arms under the legged machine base. Be careful not to vibrate the machine or subject it to shocks during transportation.

After completion of the transportation, loosen the clamping bolts and remove the transport metal fixture.



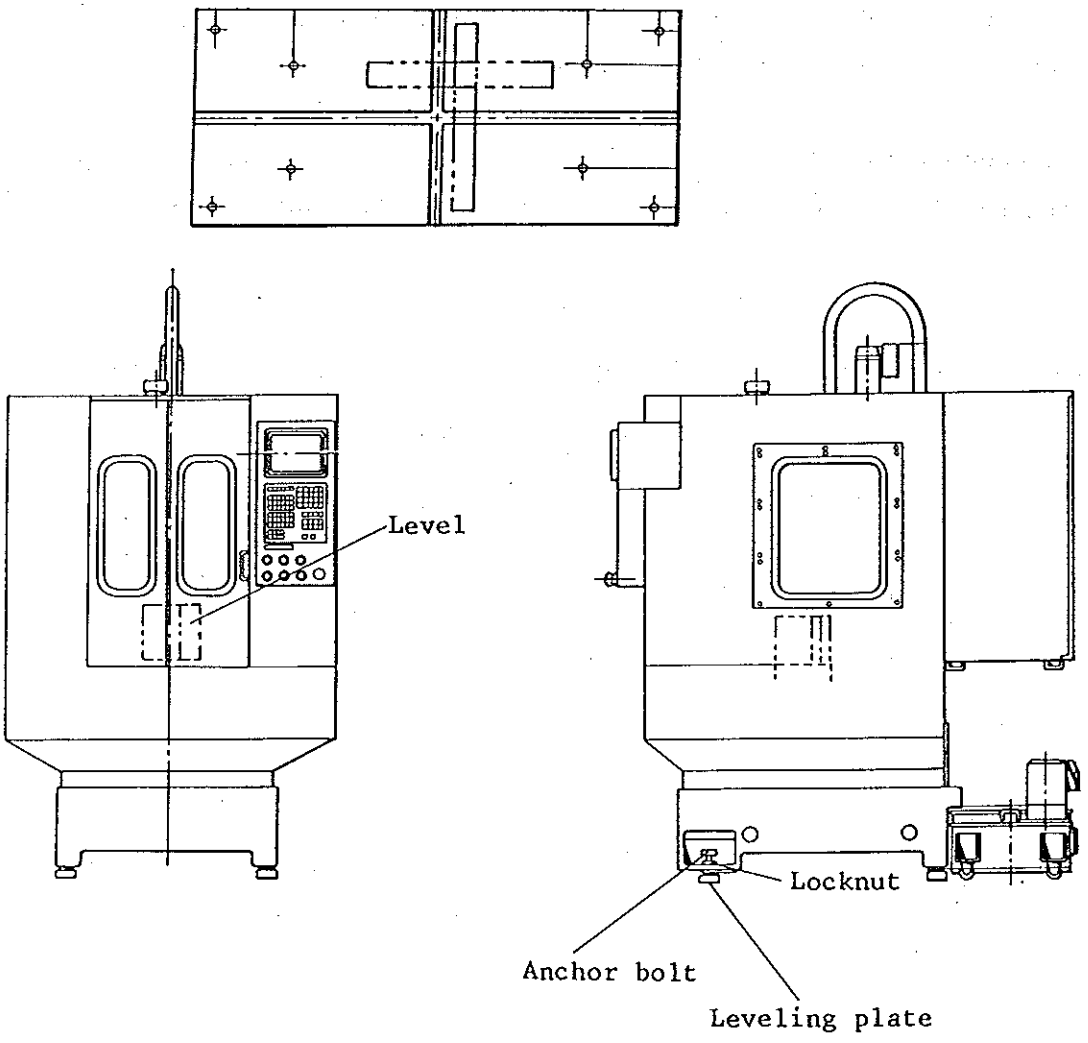
4. Machine installation

Leveling the machine correctly is the important condition to improve the performance and accuracy of the machine.

Place a level at the center of the table top as shown in the figure below and adjust the anchor bolt at the bottom of the base so that the difference of reading between longitudinal (X-axis) and cross (Y-axis) directions will be 0.0004 inch or less and then fix the bolt with a locknut.

Check the level for several days. After that, regularly check it according to necessity. Be sure to use the leveling plate, the standard accessory, under the anchor bolt.

Level



5. Power supply connection

Connection between the power supply and the CNC controller should be made by customers.

When opening or closing the control case door, conform to the following:

Before opening the door -- Be sure to set the breaker handle to "reset".

Before closing the door -- Be sure to set the breaker handle to "off".

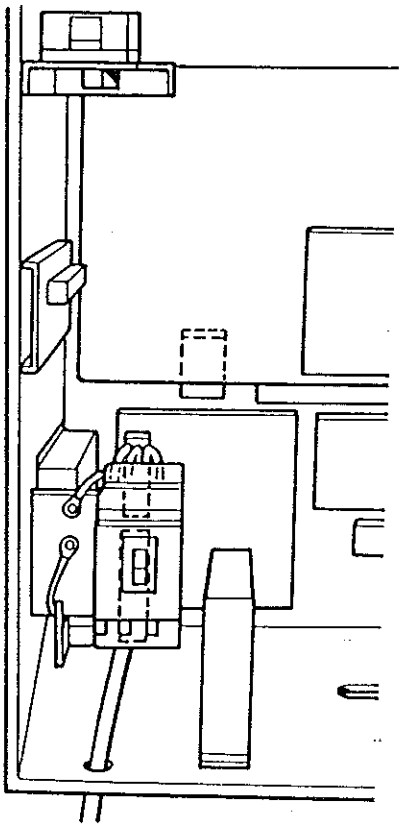
Use a 4-core vinyl cabtyre cable and be sure to make earthing. (Class 3 level earthing)

Be sure to make exclusive connection to prevent the machine from being affected by other large equipment.

To connect the CNC system, use a non-fuse breaker conforming to the specifications and procedure shown in the table below.

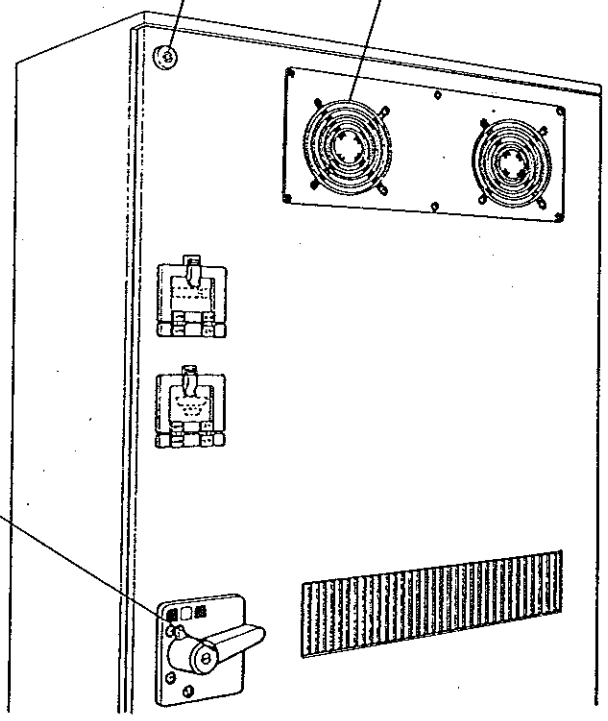
		215		225	
Input power supply		AC 3-phase V ± 10%, 50/60 ± 1 Hz			
		6000min^{-1} (rpm)	10000min^{-1} (rpm)		
Electric capacity	Continuous rating	4.8KVA	5.4KVA	7.0KVA	
	30-min rating	5.5KVA	6.0KVA	9.0KVA	
	Instantaneous rating	10KVA	12KVA	16.5KVA	
Power supply connection		Nominal cross section	Cable length	Nominal cross section	Cable length
		0.005inch^2	787inch or less	0.009inch^2	787inch or less
		0.009inch^2	1181inch or less	0.012inch^2	1181inch or less
		0.012inch^2	1969inch or less	0.022inch^2	1969inch or less

1. TC-215



Red cable: Terminal R
White cable: Terminal S
Black cable: Terminal T
Green cable: Terminal E

Power-supply connection terminal:
4-core cabtyre cable

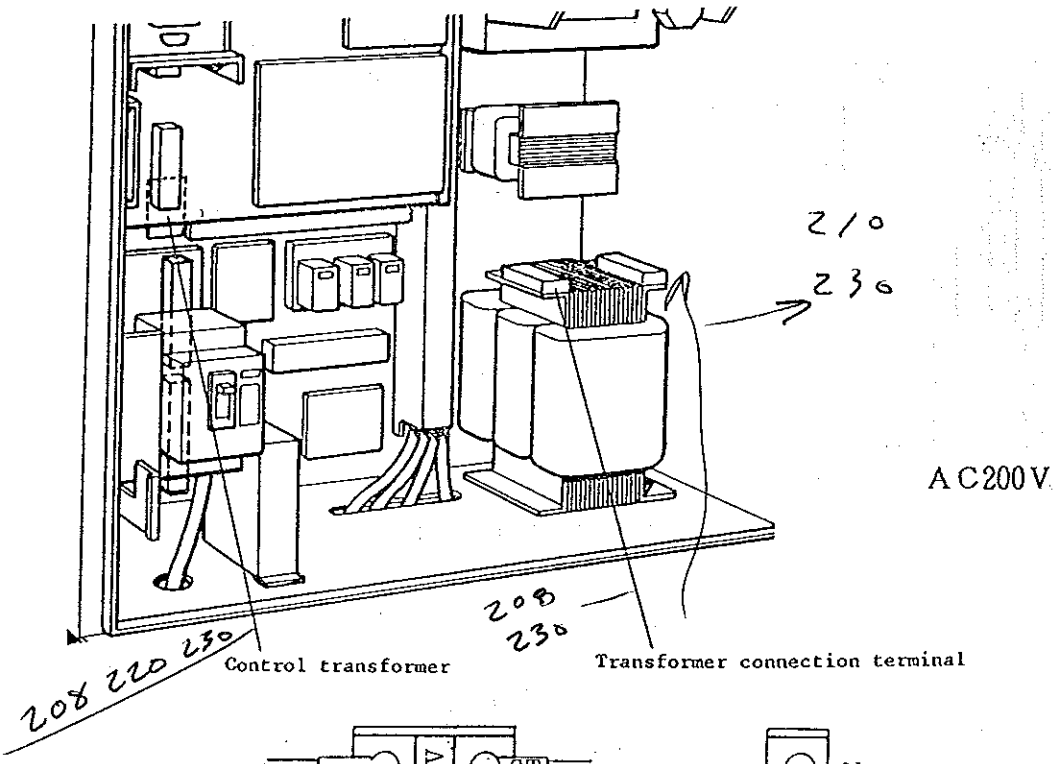


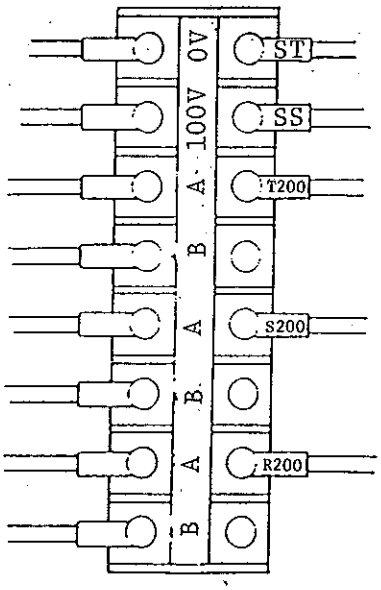
If the supply voltage is too high at the customer side:

The input power-supply voltage of this machine uses standard VAC. If your supply voltage ranges are high VAC, change the transformer connection terminal as shown in the figure below. In this case, be sure to replace the following 2 places:

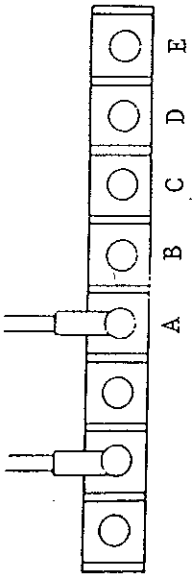
(1) Transformer

(2) Control transformer



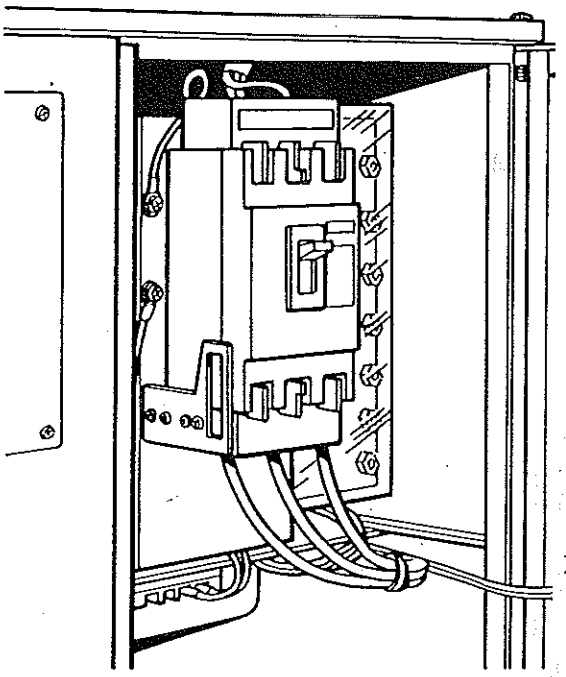


Transformer connection terminal



Control transformer connection terminal

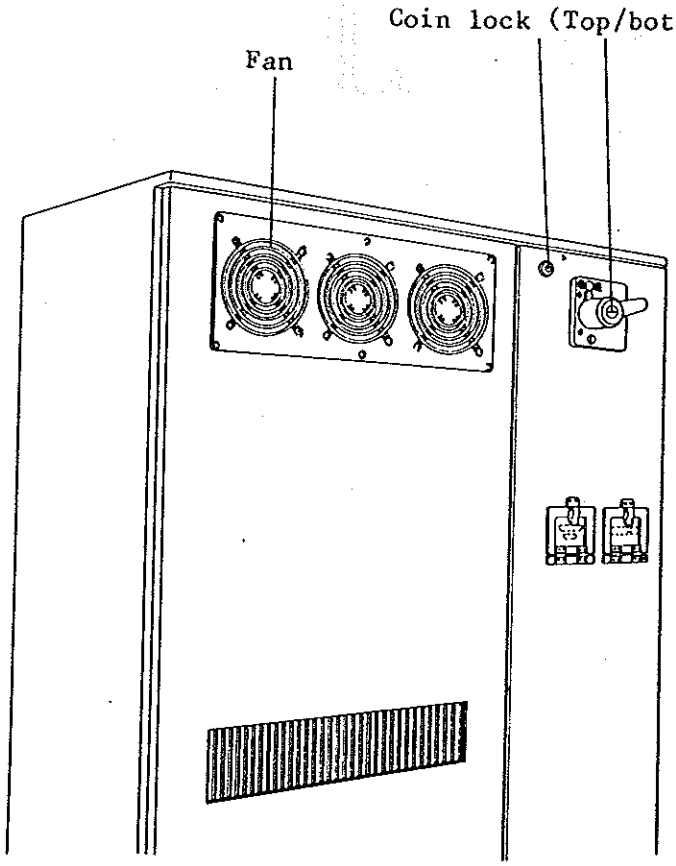
2. TC-225



4-core vinyl cabtyre cable
Power-supply connection terminal
Earth terminal
Connect the cable from the top of the CNC controller to the power-supply connection terminal shown in the figure at left.

Red cable: R／L 1
White cable: S／L 2
Black cable: T／L 3
Green cable: ⏚

Breaker handle (also used for the power switch)

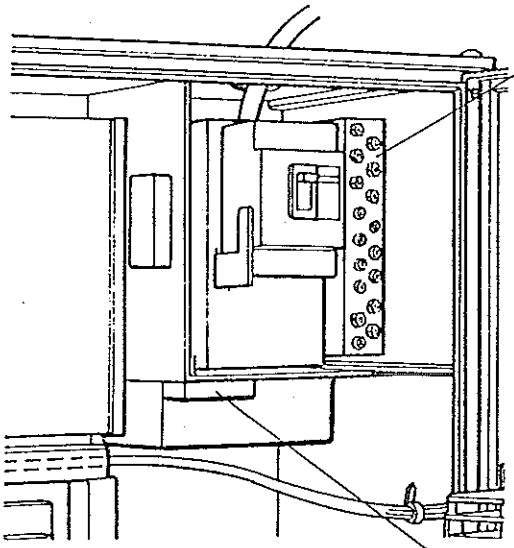


If the supply voltage is too high at the customer side:

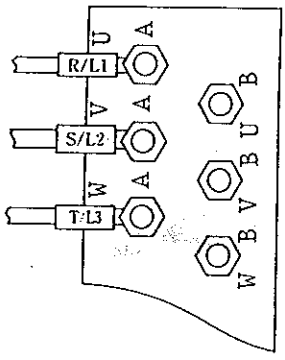
The input power-supply voltage of this machine uses standard. If your supply voltage ranges are high VAC, change the transformer connection terminal as shown in the figure below. In this case, be sure to replace the following 2 places:

(1) Transformer

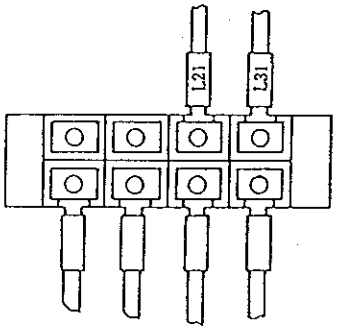
(2) 4-pole terminal block



In case of voltage "A"

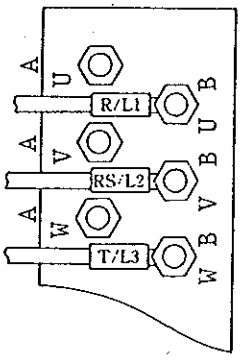


Transformer connection terminal

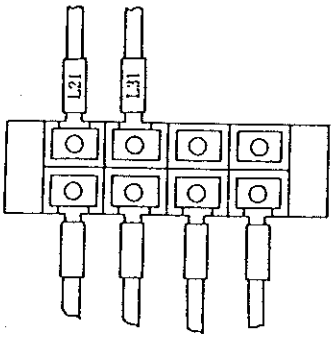


4-pole terminal block

In case of voltage "B"



Transformer connection terminal



4-pole terminal block

Please check the right list.
Connect your supply voltage
based on your voltage.

In case of voltage "A"	In case of voltage "B"
200	220
208	230
220	240
346	380
380	415
400	440

IV Maintenance

1. Routine inspection

Be sure to check the following every two months:

1. The machine level is correctly maintained

2. The power supply and the supply voltage are normal

3. The tank is kept clean.

2. Lubrication

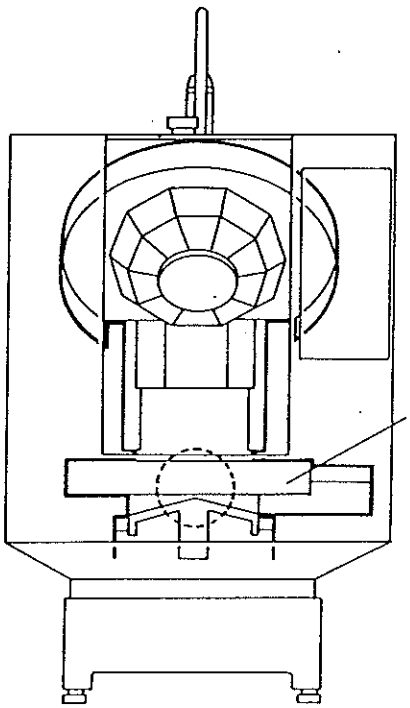
2-1. Lubrication for ball screw and LM guide

Consider the lubrication cycle of approx. one year as a criterion and supply oil according to necessity. Use the lubricating oil equivalent to those in the table below and feed oil with a grease gun according to the figures shown on pages 50 to 54.

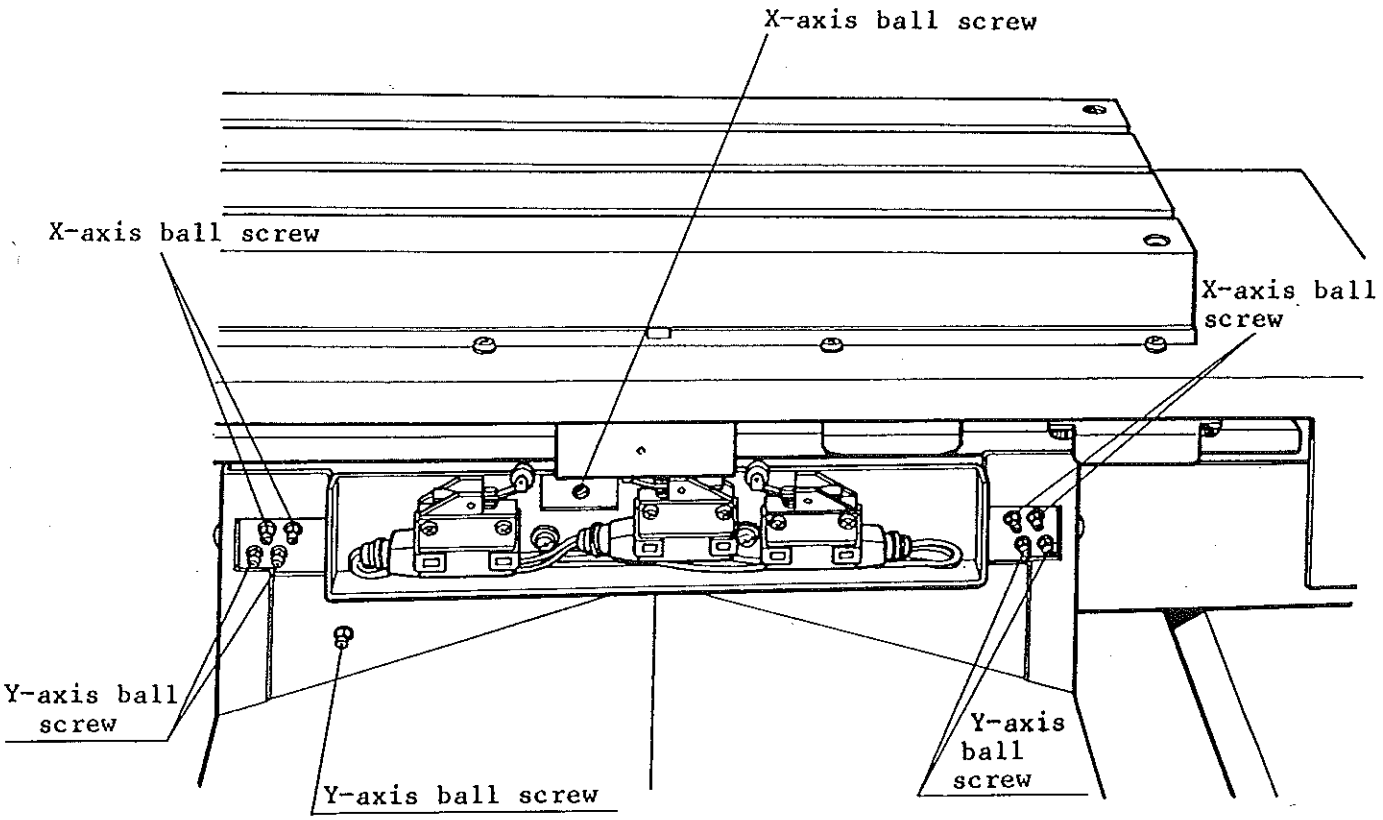
Product name	Maker
ALBANIA GREASE No. 2	SHOWA SHELL SEKIYU K.K.
MOBILUX 2	MOBIL OIL CO., LTD.
BEACON 2	ESSO Oil Co., Ltd.
MULTINOX GREASE 2	NIPPON OIL CO., LTD.

TC-215

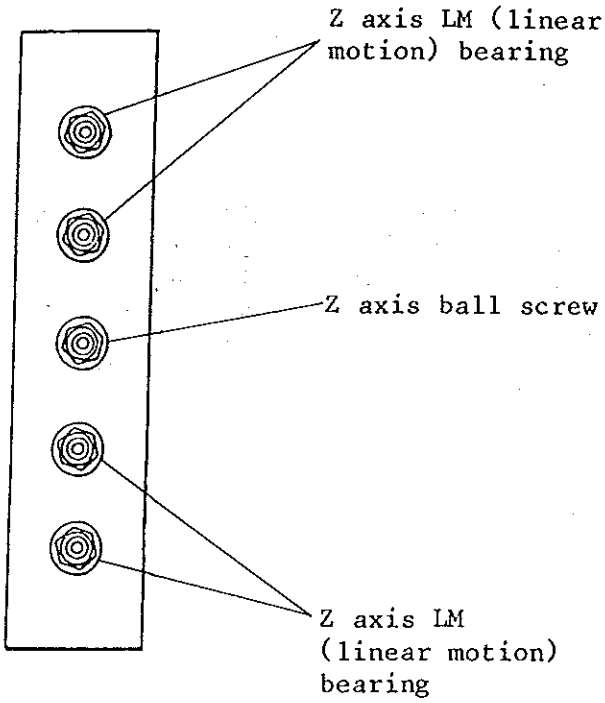
X- and Y-axes



Remove the table front cover and feed oil by approx. 1-2 gr for the LM guide and by approx. 5-6 gr for the ball screw by paying attention to excessive lubrication.



(Z axis)



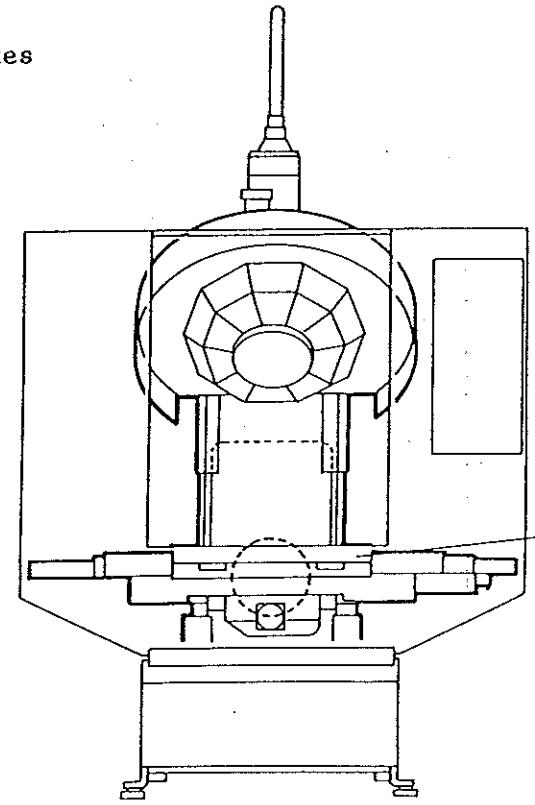
Supply 1 to 2g of oil to each LM guide and 5 to 6g of oil to the ball screw. Be careful not to cause an overload.

Top view

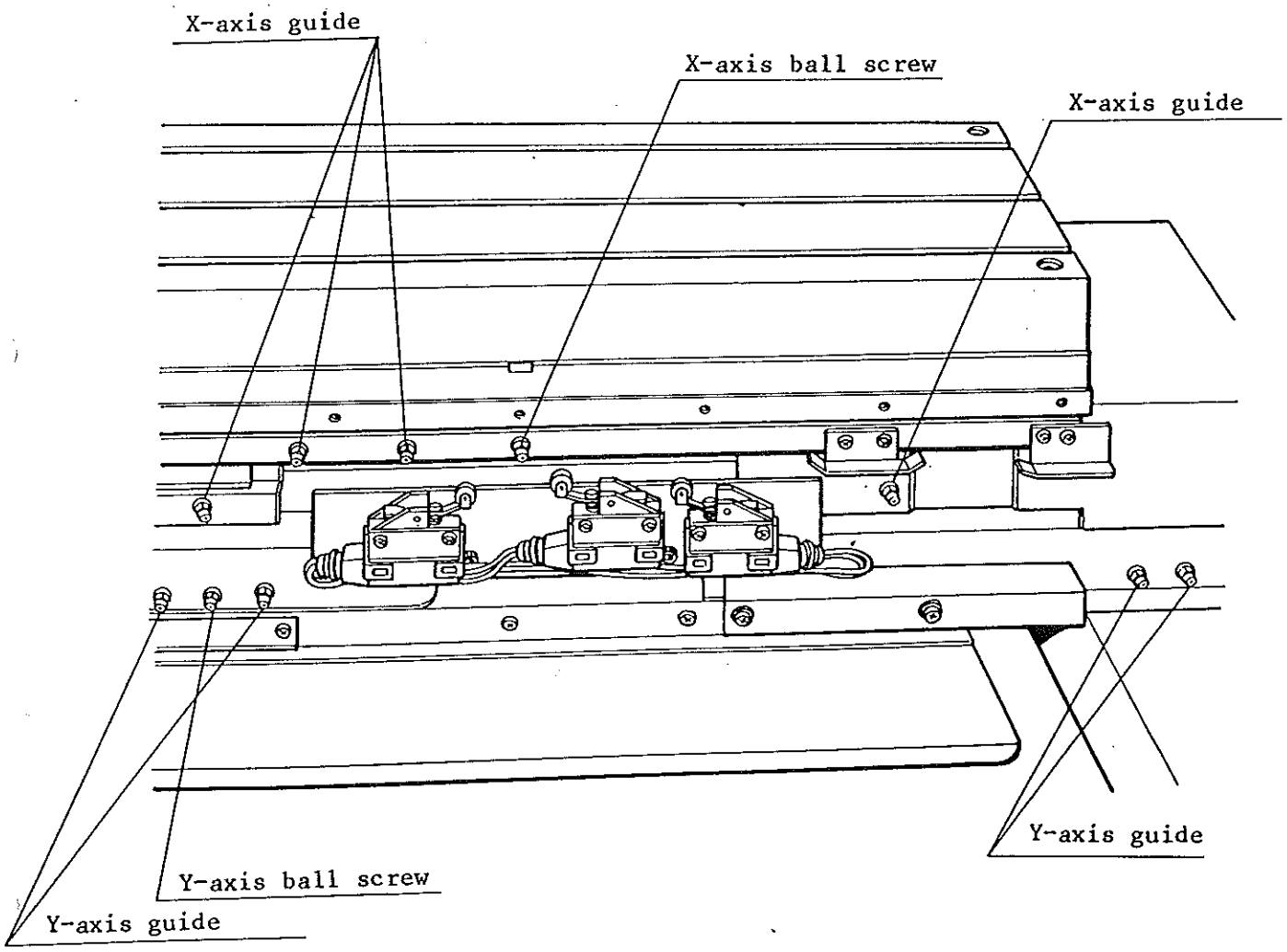
Lower section of spindle head

TC-225

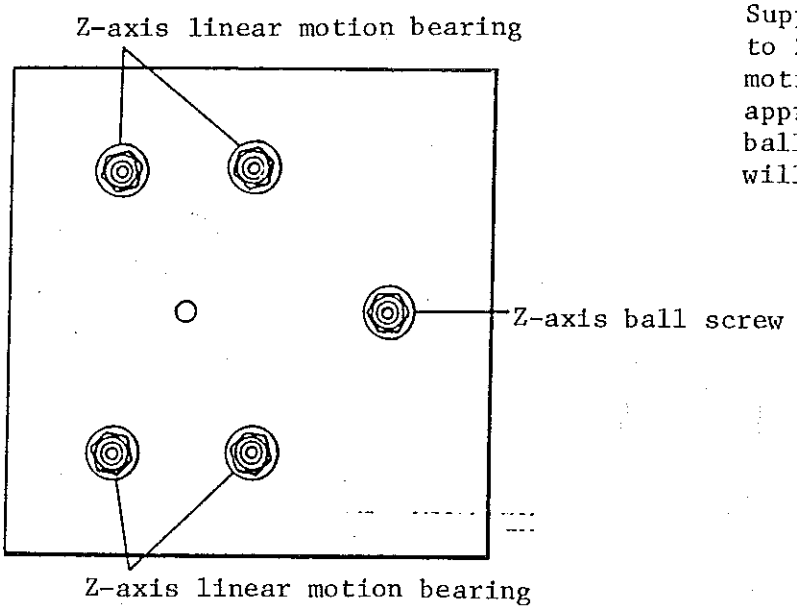
X- and Y-axes



Remove the table front cover and feed oil by approx. 1-2 gr for the LM guide and by approx. 5-6 gr for the ball screw by paying attention to excessive lubrication.



(Z-axis)

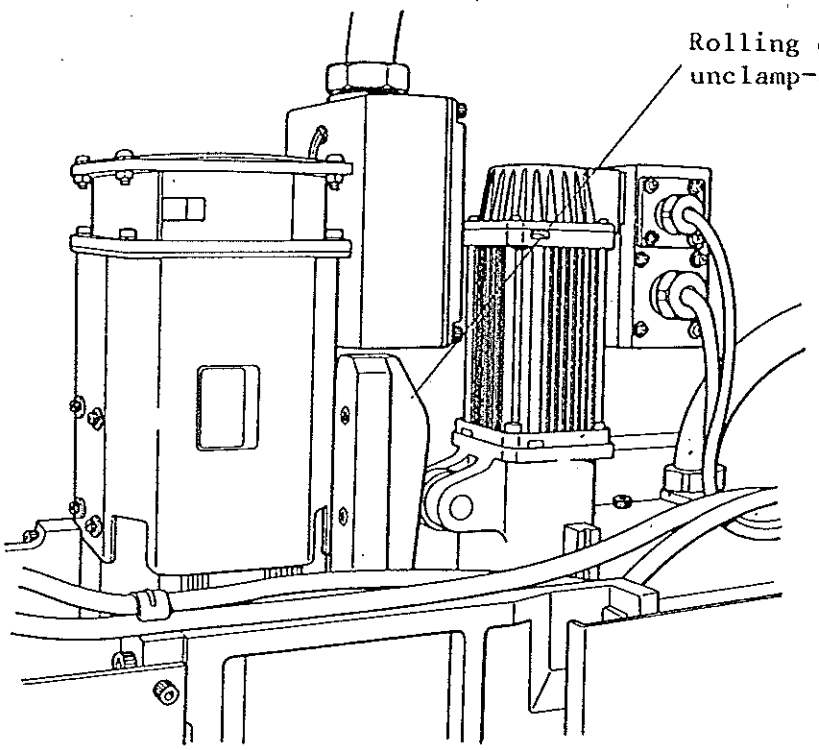


Supply oil of approx. 1 to 2 g to the linear motion bearing and of approx. 5 to 6 g to the ball screw so that they will not be overloaded.

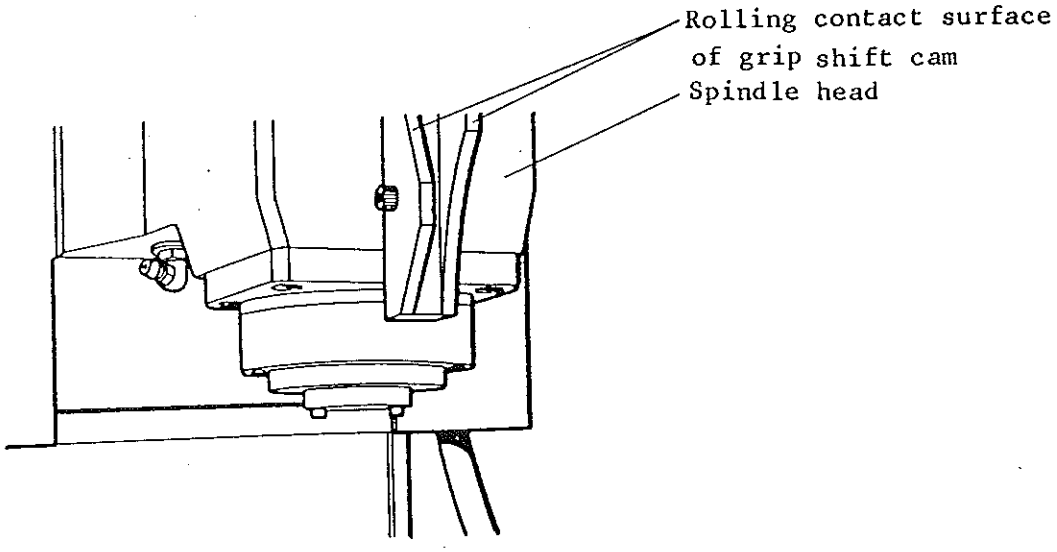
2-2. Other lubrication

(TC-215 and -225)

Apply EPINOX GREASE or equivalent to the parts below according to service condition. Apply grease to the rolling contact surface of the plate cam rollers (unclamp cam and arm cam).



Apply grease to the rolling contact surface of the cam (spindle head).



V OPERATION

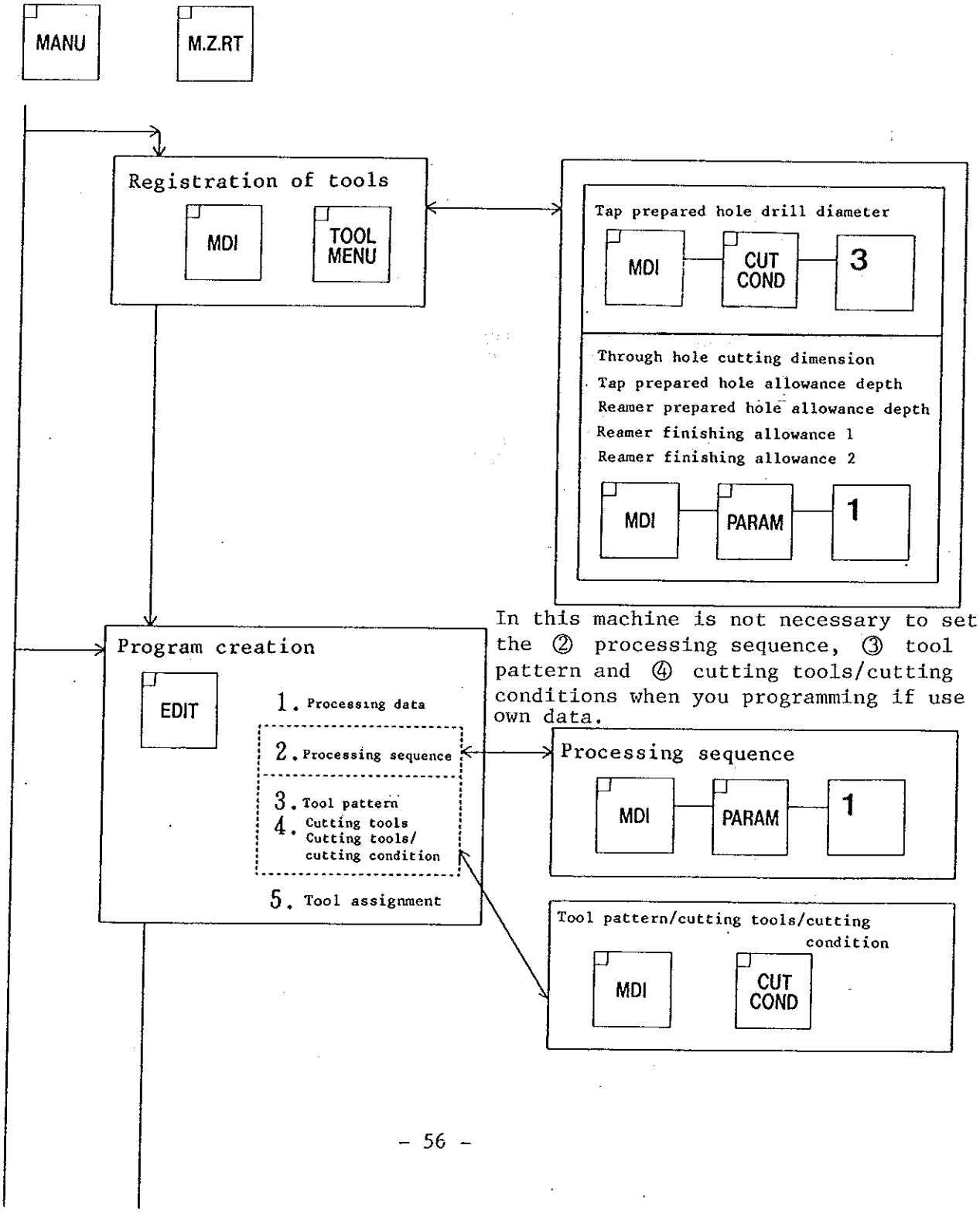
1. OPERATION PROCEDURE

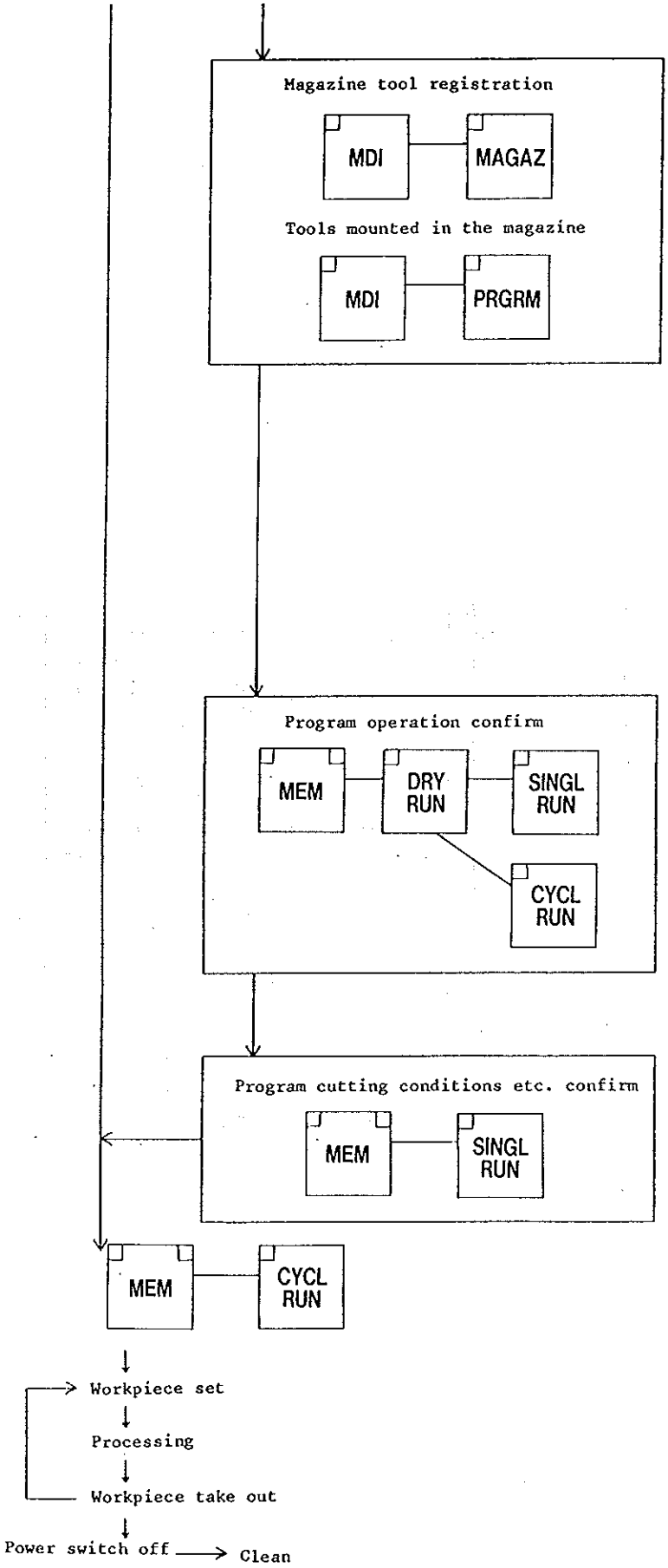
(Installation of tools)

Test

Power supply switch ON

Power ON




Magazine tool registration
MDI
MAGAZ
Tools mounted in the magazine
MDI
PRGRM
Program operation confirm
MEM
DRY RUN
SINGL RUN
CYCL RUN
Program cutting conditions etc. confirm
MEM
SINGL RUN
MEM
CYCL RUN
Workpiece set
Processing
Workpiece take out
Power switch off
Clean

2. POWER ON AND OFF

2-1. Power On

Before the power supply is switched on, make sure to see:

(1) If no obstacle exist around the machine.

(2) If the door on the CNC control equipment does not open.

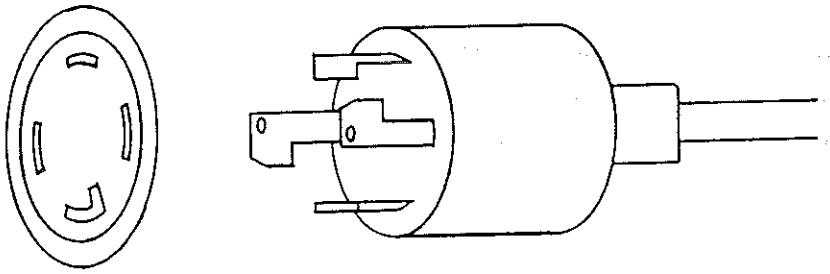
(3) If the taper at the edge of main shaft is clean.

(4) If the taper of the tool held on the magazine is clean or if neither tipping nor abrasion exists.

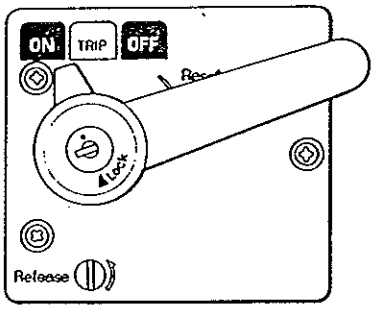
(5) If the tools are maintained well, or if the * is not loosened.

(6) If quantity of the cutting oil in the tank is proper.

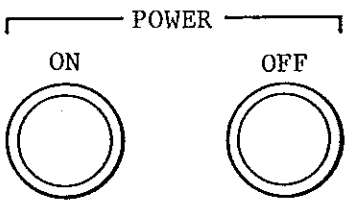
1. Plug socket in



2. CNC control box



3.



* The fans in the CNC control box and the main shaft motor rotate.
* The motors of the X, Y, and Z axes are servo-locked.

* Do not switch on the power supply again within three seconds after the power is switched off.
* When the power supply to the operation panel on the CNC control equipment is switched on, check to see if the fan on the rear panel rotates.

2-2. Power Off

When the power supply is switched off, make sure to see:

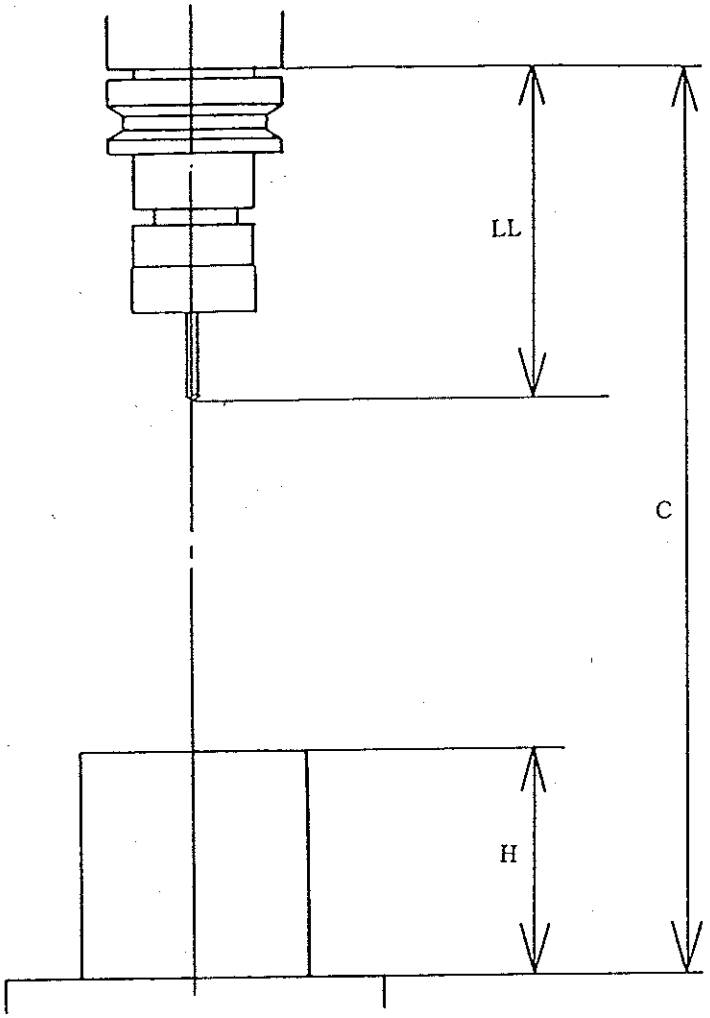
(1) If all movable portions at the machine side stop.

(2) If program edit, change of various data, and input and output operation related to external warehouses are completed.

1. Operation panel power supply: OFF
2. CNC control box power supply: OFF
3. Plug socket: OFF

3. CAUTIONS FOR SETTING WORKPIECES

Restrictions of Height and Length of Workpieces, Jigs and Tools
To shorten the non-cutting time, refer to the figure given below which is concurrently used for rotating the magazine at the time of ATC and for determining the table position, then check to see if no interference exists between the tools, workpieces, jigs and others.



LL: Maximum length of the tool set in the magazine

C: Z axis zero position length of parameter Z

H: Maximum height of the workpieces and jig tools

(1) When H C-1-0.39,
Unnecessary to try to avoid the interference

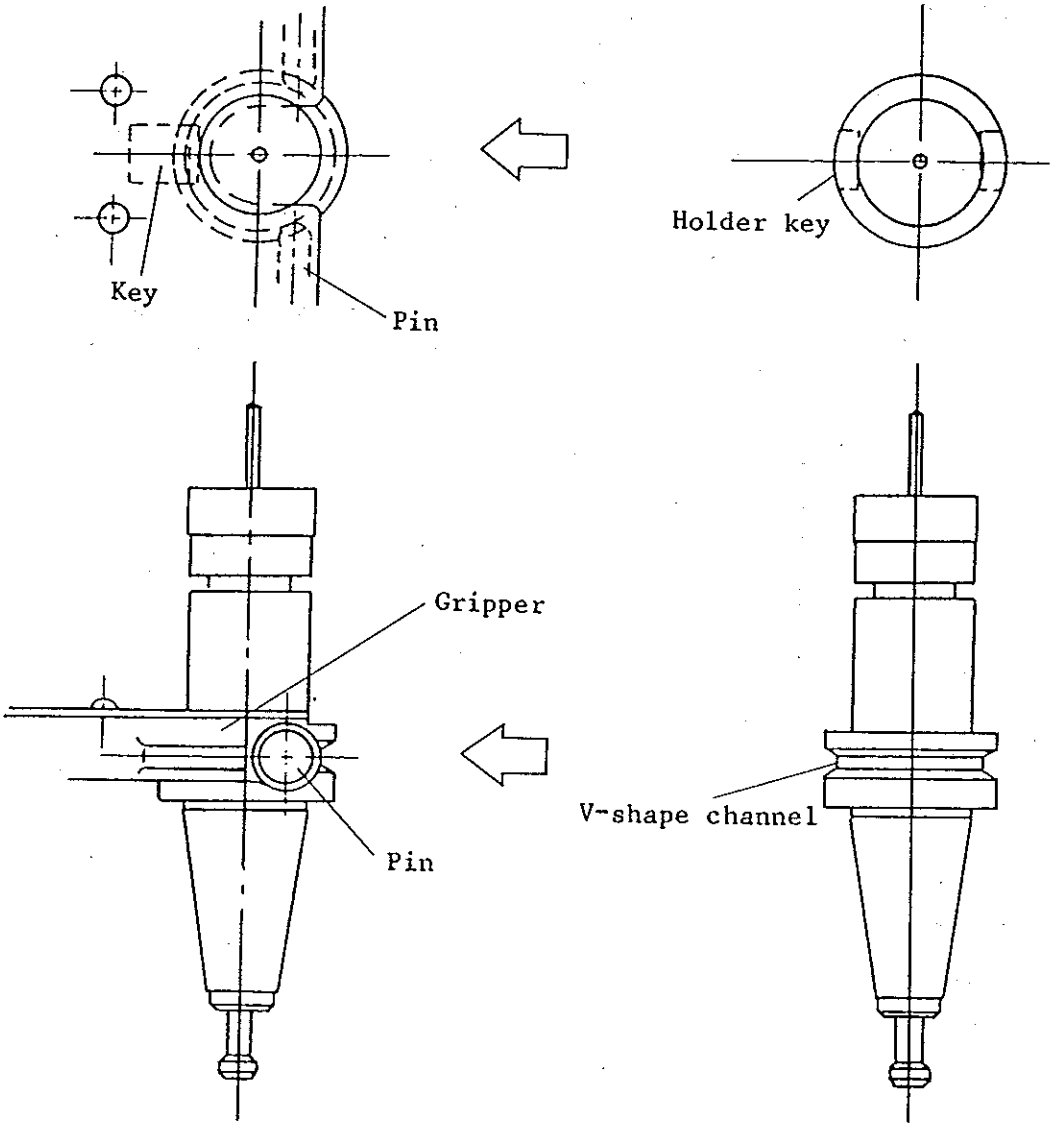
(2) When H | C-1-0.39,
Since the interference may exist due to the positions of workpieces and jig tools, installation position of tools, or processing procedure, check for the interference, then start operation.

4. SETTING AND REMOVAL OF TOOLS

4-1. Setting of Tools

(1) Select the magazine number by using the [MDI] mode and [PRGRM] (Block run).
Set the magazine number opposite to the number in which the tool is set, and activate the magazine. (The magazine number in which the tool is set will appear on the top.)

(2) Place accurately the magazine key to the holder key of the tool, set the magazine pin in the holder V-shape channel of tool, then push them into the gripper. (Fix them correctly.)

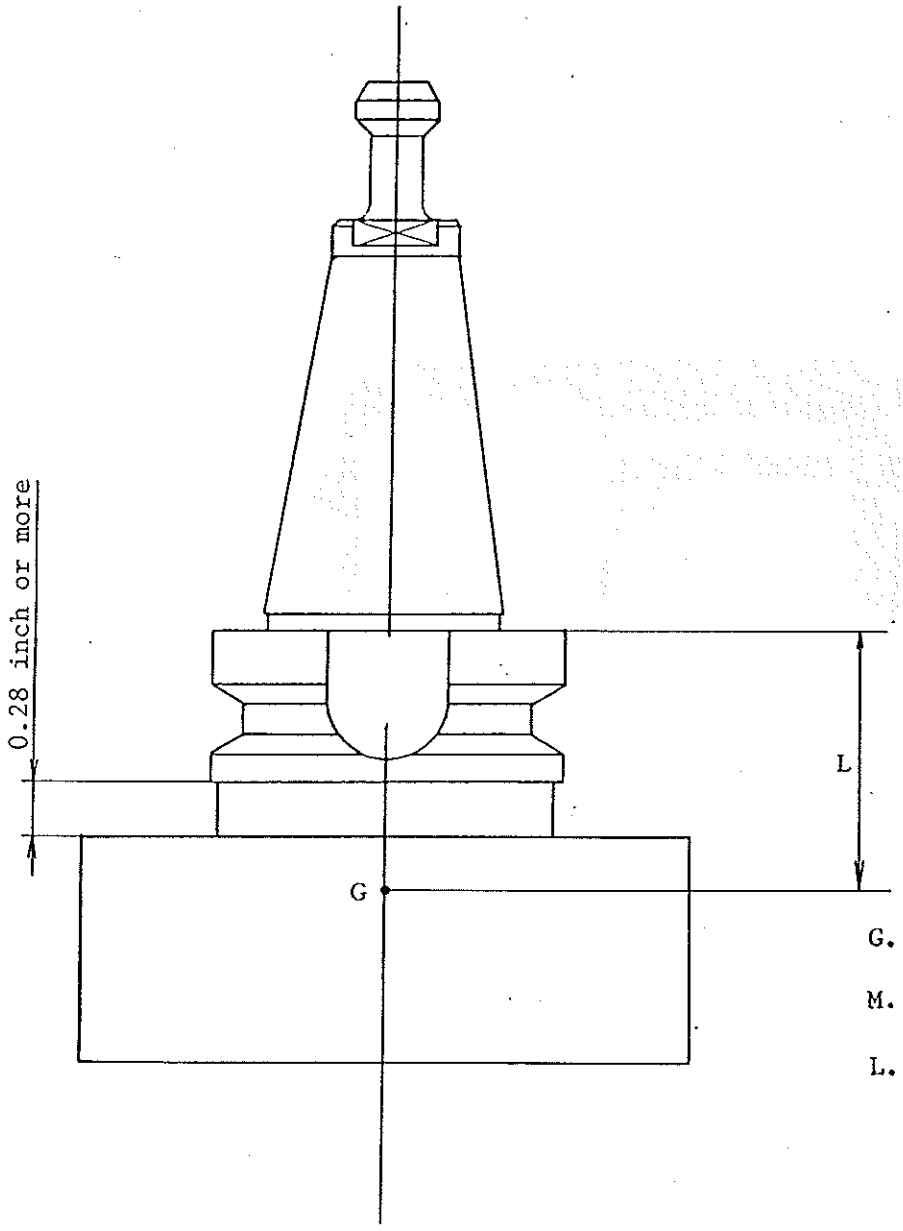


4-2. Removal of Tools

Move the pin from the holder V-shape channel, and push the pin out.

(Note) Restriction of tool holder

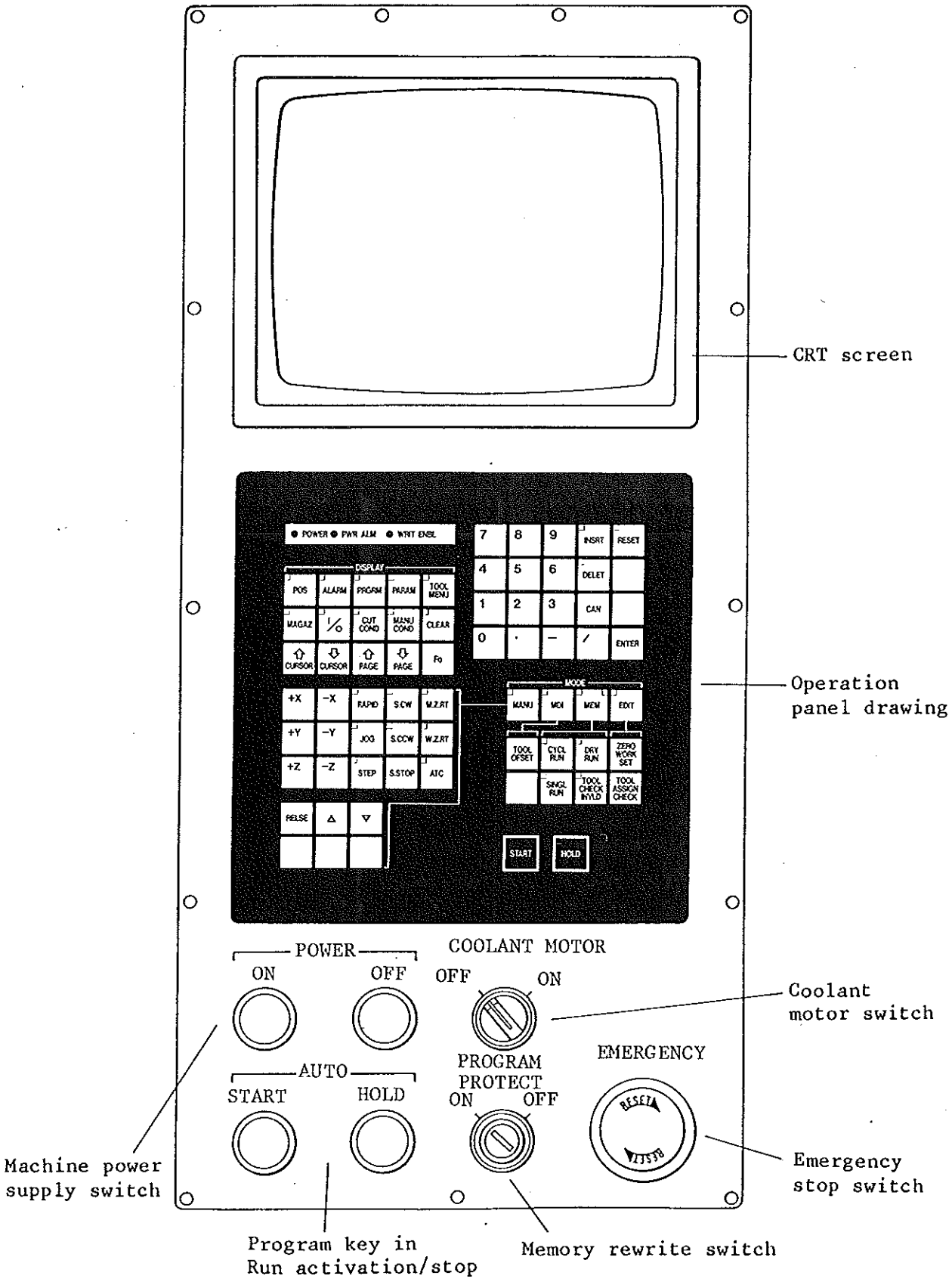
Use a tool as shown in the figure below.



G. Center of gravity in the entire tool
M. Mass of the entire tool (Kg)
L. Distance from the gauge line to the center of gravity in the entire tool

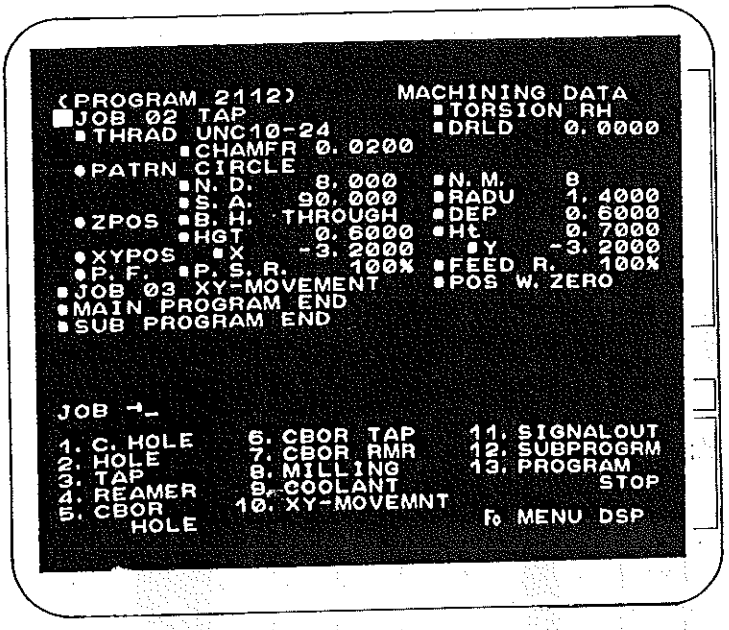
Use the tool the condition
of which is M x L = 15.6 (lbs·inch)
In addition, use the tool with the following conditions:
M x L $\leqq$ 7.8 (lbs·inch) TC-215
$\leqq$ 15.6 (lbs·inch) TC-225

5. OPERATION PANEL



5-1. CRT Screen

ex) Program Edit Screen



Program display area (data display area)

Key in request area

Teaching data area

Program display area

This area display the data and program necessary for the operation of the machine.

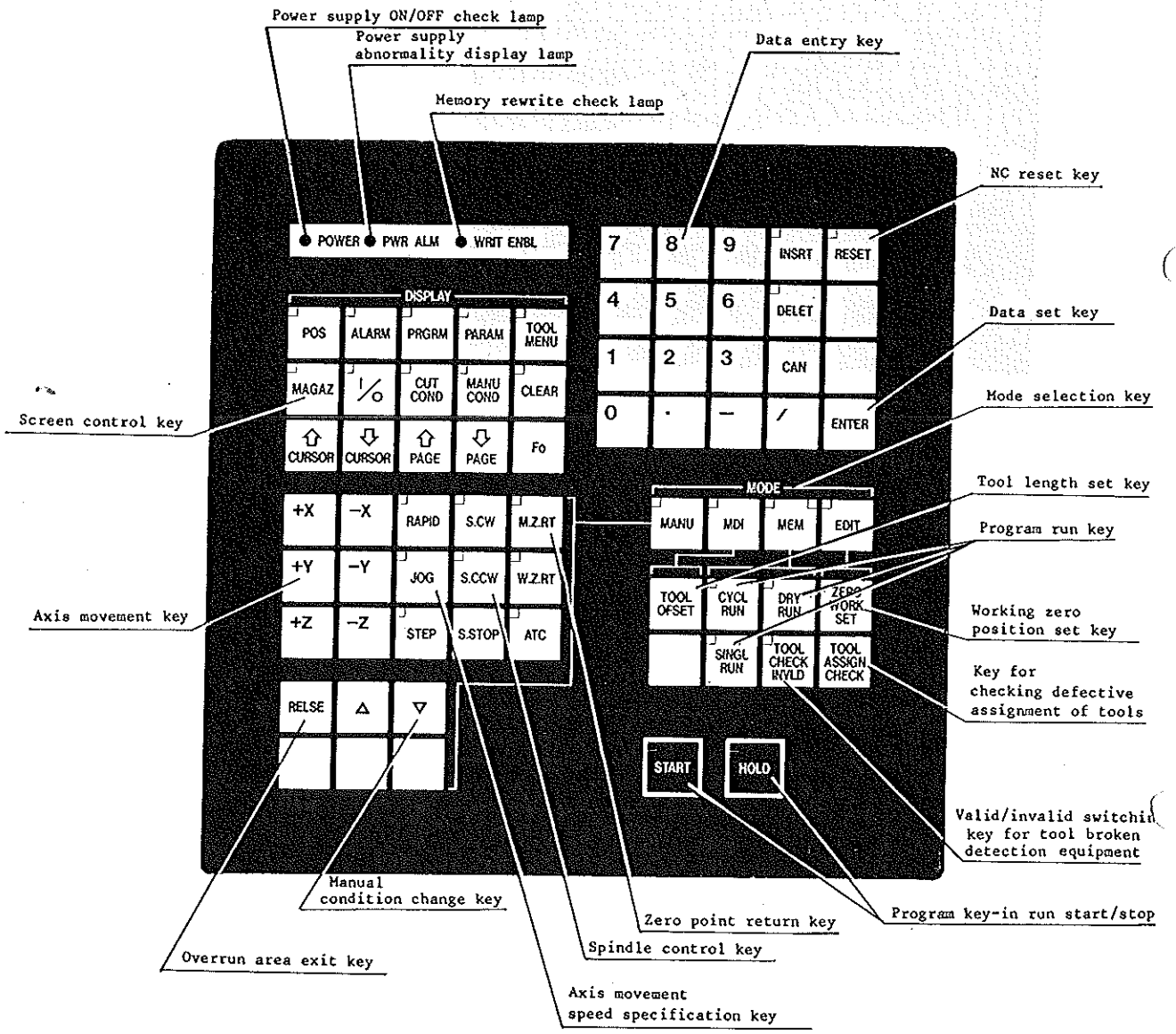
Key-in request area

Excepting manual run mode, the machine is operated in interactive mode, requesting necessary key-in items. To query from NC, key in the required data or select the right one among the teaching data menu and key in item number.

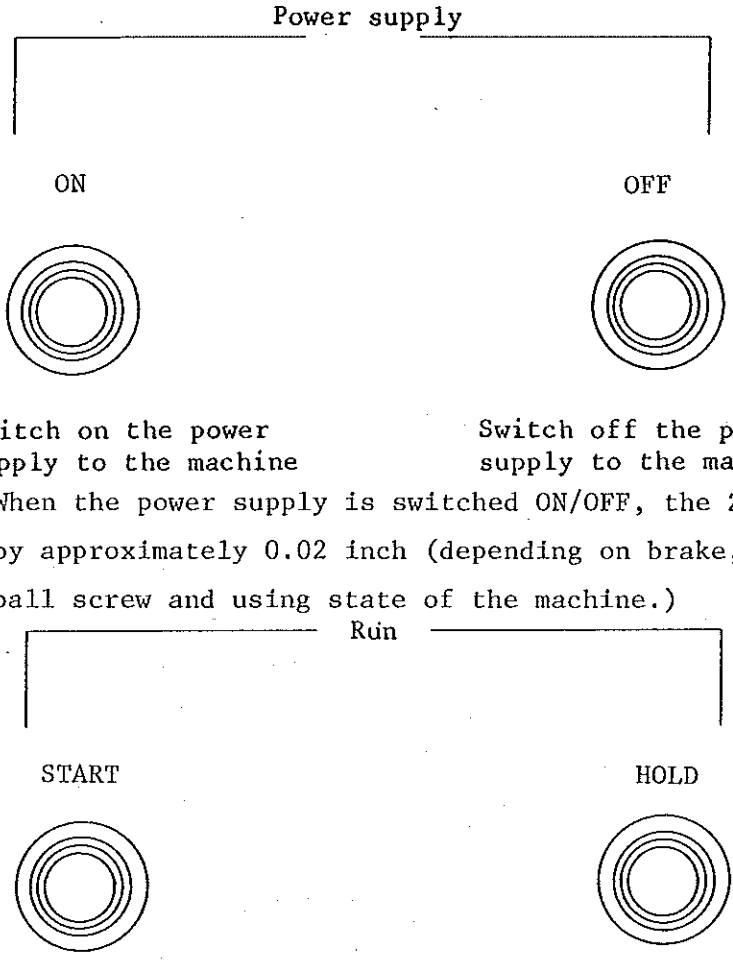
Teaching data area

This area teaches the data in menu among the built-in NC data that are required by the machine.

5-2. Operation panel



5-2-1. Function of the switch



(Caution) When the power supply is switched ON/OFF, the Z axis drops by approximately 0.02 inch (depending on brake, guides, ball screw and using state of the machine.)

START Switch to start key-in/memory run, tool assignment at the time of program edit, program alignment and input/output of external program.

(The START key has the same function. Either key can start the operation.)

Use this switch take precedence when machining.

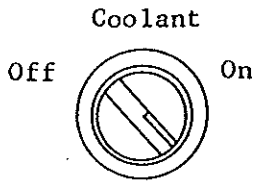
HOLD Switch to stop the movement of the X, Y and Z axes, and other operations when the program selected by the key-in and memory run is executed. (This switch does not stop rotating the spindle.)

When this switch is set to on during tool replacement or tap working, the operation stops after it is finished.

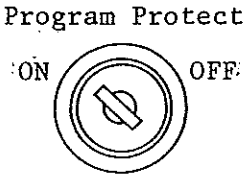
(The S.STOP key has the same function. Either key can stop the operation.)

Use this switch take precedence when machining.

Coolant Motor (Special accessory)



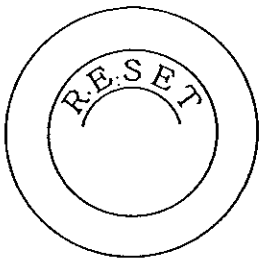
This switch switches on and off the coolant motor. The coolant lubrication is switched on and off by the open and close instruction of the coolant valve or by the coolant motor switch.



ON
Programs can be edited, and various data can be entered and changed.

OFF
This switch protects programs and data in memory from being cleared.
* Except when programs and data are to be changed or added, set the switch to the impossible side.
* Even if this switch has been set to possible, the state is set to impossible immediately after power on. Set this switch to impossible once and then set it to possible.

Emergency

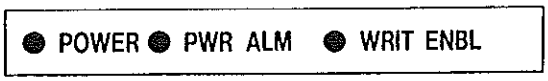


In case of the emergency during the machine operation (malfunction of the machine caused by entry error, program creation error, and others), this switch is used. Upon pressing this button, the machine stops on the spot.

Method of release Turn the button in the arrowed direction to release the emergency stop, and the initial condition of power ON will be resumed.

(Caution) When the power supply is switched ON/OFF, the Z axis drop by approximately 0.02 inch (depending on brake, guide, ball screw, and using state of the machine.)

5-2-2. Function of lamp



o Power supply

When the power supply is switched on, the red lamp comes on.

o Power supply abnormality

When the input voltage is + 15% or more of the rated voltage, the power supply is regarded to be abnormal, then the red lamp comes on. (The power supply is automatically disconnected.)

Rated voltage - When the voltage is 15% or more of the rated voltage, alarm (error) is displayed.

o Memory rewrite possible

When the memory switch is set to on, the green lamp comes on.

(Caution) When the power supply is switched ON/OFF, the Z axis drops by approximately 0.02 inch (depending on brake, guide, ball screw, and using state of the machine.)

5-2-2. Function of Keys

<table>
<tr><th>Key</th><th>Function</th></tr>
<tr><td>RESET</td><td>Key to return the machine to the initial condition.
* After the alarm (error) is displayed, the alarm display is cleared before the release.
* When this key is pressed after programs are entered, all entry data are made invalid.
* When this key is pressed in the course of memory run, the machine, together with the X, Y, and Z axes, and spindle, stop on the spot. (When this key is pressed during tapping or tool replacement, the machine stops after the operation is completed.)</td></tr>
<tr><td>RELSE</td><td>Key to exit from the overrun area or stoke over state, and to move the Z axis from the ATC area manually.</td></tr>
<tr><td>MANU</td><td>Key to select the manual run mode (spindle revolve,
move, cutting move, step move, ATC)</td></tr>
<tr><td>+X -X
+Y -Y
+Z -Z</td><td>Axis move key
Keys to move the X, Y, Z axes toward positive or negative direction in the manual run mode.</td></tr>
<tr><td>RAPID</td><td>Key to rapidly move the X, Y and Z axes toward
negative direction in the manual run mode.
<table>
<tr><th>MAX / Model</th><th>X and Y axes</th><th>Z axis</th></tr>
<tr><td>TC-215</td><td>65.6 ft/min</td><td>65.6 ft/min</td></tr>
<tr><td>TC-225</td><td>65.6 ft/min</td><td>49.2 ft/min</td></tr>
</table>
The maximum move speeds of the X, Y, and Z axes for parameter 2 are provided by 5-step speed change (10, 25, 50, 75, 100 %).</td></tr>
</table>

Key	Function
JOG	Key to move the X, Y, Z axes toward positive or direction in the manual operation mode. * Up to 22-speed changes from 2.0 to 196.9 inch/min are provided for the X, Y and Z axes. When the power is switched on, the X, Y, and Z axes are set to 2.0 inch/min.
STEP	Key to move, step by step, the X, Y, and Z axes toward positive or negative direction in the manual mode. * 4-step change (0.0001, 0.001, 0.01, and 0.1 inch) is provided for one-step distance in the X, Y, and Z axes.
S.CW	Key to forward the spindle (clockwise vied from the motor top) in the manual mode. * 21-step change from 100 to 6000 min-1 (rpm) is provided. * 22-step change from 100 to 6000 min-1 (rpm) is provided (Only when 10000min-1) When the power is switched on, the speed is set to 100 min-1 (rpm).
S.CCW	Key to forward the spindle (counterclockwise vied from the motor top) in the manual mode. * 21-step change from 100 to 6000 min-1 (rpm) is provided. * 22-step change from 100 to 6000 min-1 (rpm) is provided (Only when 10000min-1) When the power is switched on, the speed is set to 100 min-1 (rpm).
S.STOP	Key to stop the spindle.
△	Key to change speed (ascend) (rapid move, cutting move, step move, and spindle rotation).
▽	Key to change speed (descend) (rapid move, cutting move, step move, and spindle rotation).

Key	Function
M.Z.RT	Key to return the X, Y and Z axes to the machine zero position. * Simultaneous movement is provided: X and Y axes together, Z axis alone. In the machine zero coordinate, the X and Y axes are set to 0, and the Z axis is set to the value of the Z axis zero position for parameter 2. This key is valid in the manual un mode.
W.Z.RT	Key to return the table to any working zero position set (both X and Y axes). However, the Z axis is returned (the Z axis rises). * This key is valid in the manual run mode. * This key operation assumes that the working zero position is set in the key entry run mode or that the memory run is performed in the key input run mode.
ATC	Key to replace the magazine tool in the manual mode. Pressing the key four times completes the tool replacement.
0 ~ 9 / . −	Keys to key in data (values).
ENTER	Key to key in data of input request area.
CAN	Key to cancel data of input request area (valid for data prior to pressing the ENTER key.)
INSRT	Key to add data to stored data.
DELET	key to delete data from store programs.

Key	Function
POS	Key to indicate distance from the machine zero position and form the working zero position for each axis, and information on tools set in the magazine.
ALARM	Key to confirm all the displays in the event of error (alarm). (Displays only one major error in other screens.)
PRGRM	Key to display and set programs in program edit, memory run, and key in run.
PARAM	Key to key in data necessary for machine run.
TOOL MENU	Key to key in the data of 60 tools (type, length, diameter, overall length, life, etc. of drill, tap, etc.). Data can be entered in the key in run mode. In other modes, only data can be displayed.
MAGAZ	Key to assign in tool No. of 60 tools stored to each magazine No. Data can be set in the key entry run mode. In other modes, current state of the magazine tools can be displayed.
I/O	Key to check the cutting condition of I/O at the time of failure or repair.
CUT COND	Key to set the cutting condition of tools (feedrate and feed amount). * Sets the tool pattern and the combination of the tools necessary for working. * Sets the cutting tools, cutting conditions, and working conditions for 10 types of materials (feed rate and feed amount for each tool). * Sets hole size before cutting for tapping. Data can be set in the key in run mode. In other mode, data can only be displayed.

Key	Function
MANU COND	Key to set the various types of move rates and spindle rotation speed. * Rapid feed rate: 5-step change of 10, 25, 50, 75, 100% * Cutting move rate: 22-step change of 2.0, 2.5, 3.1, 3.7, 4.5, 5.5, 6.7, 8.1, 9.6, 11.4, 13.4, 15.7, 19.7, 24.6, 27.6, 31.5, 39.4, 49.2, 62.0, 78.7, 124, 96.9, inch/min * Spindle r.p.m. 6000 min-1 (rpm) specification (Standard): 21-step change of 100, 125, 155, 190, 230, 280, 340, 410, 490, 580, 800, 1000, 1250, 14000, 1600, 2000, 2500, 3150, 4000, 6000 10000min-1 (rpm) specification (special): 22-step change (adding 10000 to the standard)
CLEAR	Key to turn off the CRT.
⇧ CURSOR	Key to back setting item.
⇩ CURSOR	Key to advance setting item.
⇧ PAGE	Key to back setting item (for 1 screen).
⇩ PAGE	Key to advance setting item (for 1 screen).
F0	Key to be used at the time of changing input item when menu screen is changed or when the working data is completely entered.

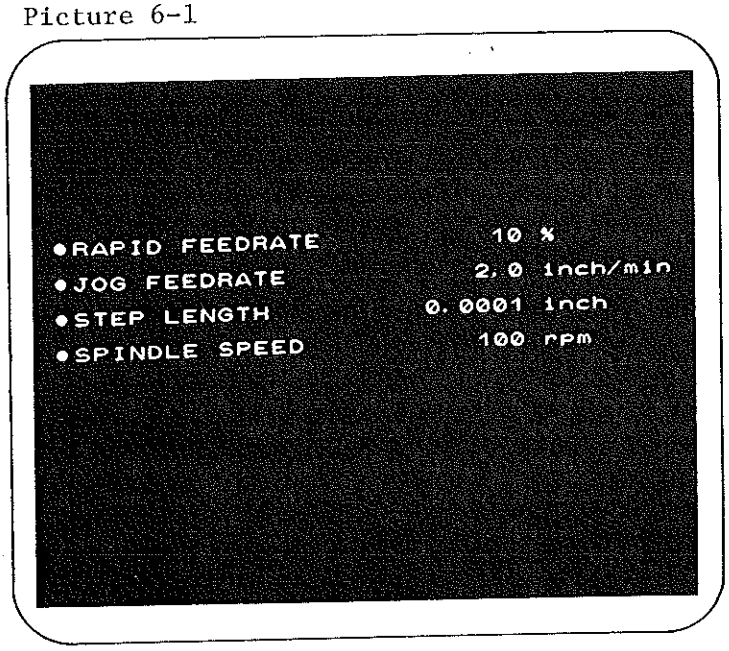
Key	Function
Key In Run MDI	Key to perform the key-in run (keying in necessary data, or piling blocks). This key is used to key in or change data of parameter, tool menu, cutting tool conditions and tool length set.
TOOL OFSET	Key to key in the tool menu automatically by calculating the tool length when the tool is mounted in the spindle in the key in run mode.
MEM	Key to execute memory run by calling stored program.
CYCL RUN	Key to executed called program consecutively from start to end.
SINGL RUN	Key to executed called program individually for each single action.
DRY RUN	Key to execute memory run with rapid feedrate and ATC, and check the program in a short time with rapid feedrate of table movement. By setting dry run offset amount in parameter 2, the Z axis only can be set off.
EDIT	Key to edit programs and to provide communication with external equipment.

Key	Function
ZERO WORK SET	Key to change the current X and Y axes position to zero working position in the program edit mode and insert it into program automatically, when the working zero position of edited program to machine zero position is unknown.
TOOL ASSIGN CHECK	Key to check the cause for no assignment of tool specified in program edit. Further press the [ALARM] key, and check causes for all of trouble.
TOOL CHECK INVLD	Key to set whether function of the tool broken detector is valid or invalid. The initial mode can be set in parameter (whether the tool broken detection function is valid or invalid when the power supply is switched on).
START	Key to start the key-in and memory run, to assign tools in program edit, and save/load external program. (The start switch as the same function. Either of the start key or the start switch can start operation.)
HOLD	Key to stop moving the X, Y, and Z axes during the operation of program selected in the key-in and memory run. * This key is invalid during tool replacement or tapping. (The Stop switch has the same function. Either of the hold key or the stop key can stop operation.)

6. MANUAL RUN

When the power supply is switched on, the screen shown below is displayed. (At the time of parameter initial mode selection)

Picture 6-1



Manual run mode display

By [MANU] mode,

The axes can be returned to zero position by pressing the [M.Z.RT] key and [W.Z.RT] key.

The speed, step width and rotating speed can be set in the above screen, and the [RAPID], [JOG], and [STEP] keys can be pressed. Data in the rapid movement or in the cutting movement is moved only when the [RAPID] or [JOG] key is being pressed.

In addition, [S.CW], [S.CCW] and [S.STOP] keys are available.

By pressing the [ATC], the tools can be replaced in the single action.

6-1. Zero Position Return

Return X, Y, and Z axes to the machine zero position and the spindle to the orientation position. In this stage, the coordinate value are as follows: both X and Y are positioned at 0, and the Z axis is set to the value of the Z axis zero position dimension of parameter 2.

* After switching ON the power supply, be sure to execute the zero position return.

For the X and Y axes, move the table to the center of the machine, for the Z axis, move the spindle tip face approximately 11.81 inch higher than the table top. Press the [M.ZRT] key in the manual run mode.

The Z axis ascends to return to zero position and the LED comes on. Then, the X and Y axes return to zero position. After completion of zero position return of the three axes, the LED goes off.

The positions of the three axes are checked by the [POS] key.

* After zero position movement, the axis cannot be moved out of the parameter stroke.

Refer to 7-2-2, parameter (system) stroke.)

When the axis are requested to move out the stroke, move them while the [RELSE] key in being pressed.

6-2. Working Zero Position Return (W.Z.RT)

The axes return to working zero position of the program set by the [MDI] or to working zero position of the program executed by the [MEM] (the position of the program which was executed immediately before the selection of manual mode).

The axes return to working zero position. The Z axis returns to machine zero position, then the X and Y axes return to working zero position of the program which was executed immediately before the selection of manual mode.

The three axes are checked by the [POS] key.

6-3. Rapid Feed

The X, Y, and Z axes are move at a high speed.
The axes can be moved while the key is being pressed.

The cursor is set in RAPID FEEDRATE on the CRT display (manual run mode display).

The speed is increased or decrease by the [△] or [▽] keys.

The rapid feedrate is selected from among the five-step change of 10%, 25%, 50%, 75% and 100% set by parameter 2.

* The above operation is omitted when the initial value (display) is selected.

ex) The axes are moved at a high speed by the [RAPID] key. The cursor is set in the RAPID FEEDRATE. The cursor is set at step of 75% by the [△] key. (The maximum feedrate of parameter 2 is set to 65.6 ft/min.)

[+X] [-X]

The X axis moves toward positive or negative direction at 49.2 ft/min while this key is being pressed.

[+Y] [-Y]

The Y axis moves toward positive or negative direction at 49.2 ft/min while this key is being pressed.

[+Z] [-Z]

The Z axis moves toward positive or negative direction at 49.2 ft/min while this key is being pressed.

6-4. Jog Feedrate (JOG)

The X, Y, and Z axes are moved at the jog feedrate. The axes are moved while the axis movement key is being pressed.

The cursor is set in JOG FEEDRATE in the [MANU] mode.

The jog feedrate is increased or decrease by the [△] and [▽] keys. The feedrate is selected among from 22 steps from 1.97 to 196.8 in. inch/min.

* When the axes are set to the initial value (display), the above operation is omitted.

ex) The axes are moved by [JOG] key.

Cursors [△] and [▽] are set to the jog feed rate. Set the feedrate to 196.8 inch/min by the .

[+X] [-X]

The X axis moves toward positive or negative direction at 196.8 inch/min while this key is being pressed.

[+Y] [-Y]

The Y axis moves toward positive or negative direction at 196.8 inch/min while this key is being pressed.

[+Z] [-Z]

The Z axis moves toward positive or negative direction at 196.8 inch/min while this key is being pressed.

6-5. Step Length

The X, Y, and Z axes are moved step by step. Each axis moves one step at every pressing of this key. One step length is selected from among the four steps.

The cursor is set in STEP LENGTH in the manual mode by using [⇧ CURSOR] and [⇩ CURSOR] keys.

Step length is increased or decreased by the [△] or [▽] key. The length is selected from among four steps of 0.0001, 0.001, 0.01 and 0.1 inch.

* When the axes are set to the initial value (display), the above operation is omitted.

ex) The [STEP] key moves the axes by step.

The [⇧ CURSOR] and [⇩ CURSOR] are set to 0.1inch/step by [△] key.

The X axis moves by 0.1inch/step toward positive or negative direction by every pressing of the [+X] or [-X] key.

The Y axis moves by 0.1inch/step toward positive or negative direction by every pressing of the [+Y] or [-Y] key.

The Y axis moves by 0.1inch/step toward positive or negative direction by every pressing of the [+Y] or [-Y] key.

6-6 Spindle Revolve

The spindle is revolved.
The revolving direction is specified as follows: clockwise direction is forward revolution, and counterclockwise direction is backward revolution.
The [STEP] key stops the revolution.

The manual run mode and the cursor are set to SPINDLE SPEED.

The speed is increased or decreased by the [△] or [▽] key. The speed is selected from among 21 steps of 100 to 6000 min^{-1} (rpm).

* When set to the initial value (screen display), the above operation is omitted.

ex) The cursor is set in SPINDLE SPEED.

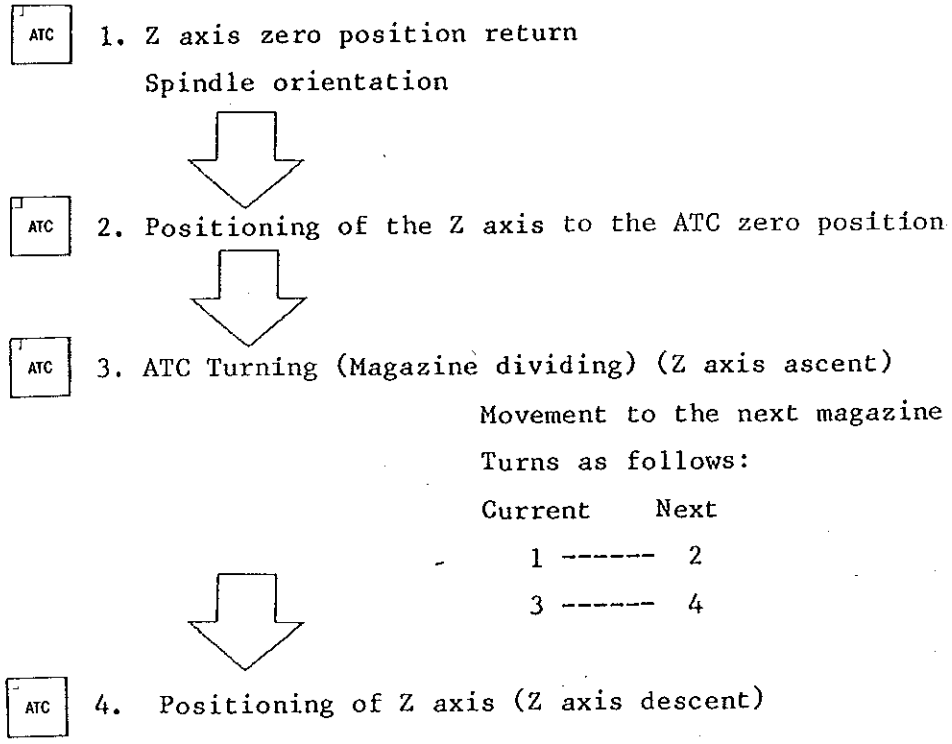
The spindle is set to 6000 min^{-1} (rpm) by the [△] key.

The spindle forwards at 6000 min^{-1} (rpm) by the [S.CW] key, and the spindle reverses at 6000 min^{-1} (rpm) by the [S.CCW] key.

The spindle stops by the [S.STOP] key.

6-7. Tool Replacement Single Action

The magazine tools are replaced manually.



The operation is completed by pressing the key four times.

* Press the [ATC] key while pressing the [RESET] key at the ATC zero position of the Z axis (under the above condition 2), and the ATC magazine indexes the positions one by one clockwise.

7. MDI

In the key [MDI] run mode, one block run or block piling run can be performed.

The key-in items necessary for block run is displayed by the [PRGRM] key.

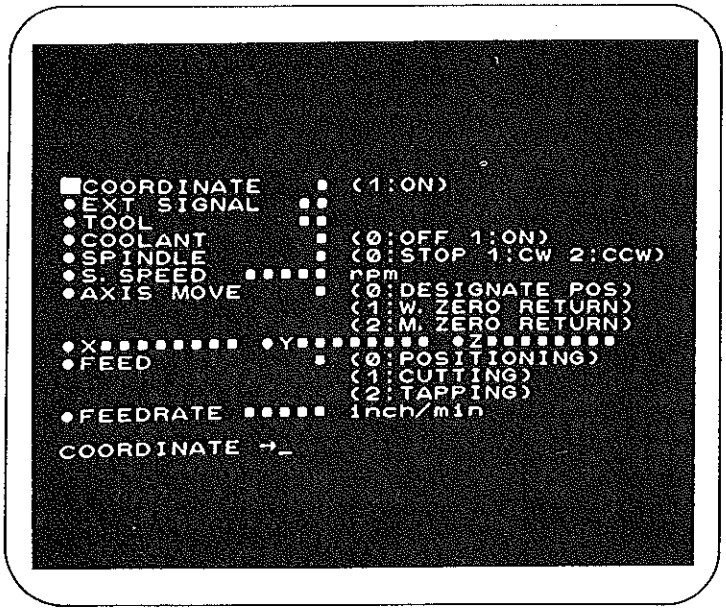
In addition, [PARAM] , [TOOL MENU] , [MAGAZ] and [CUT COND] conditions can be changed.

(When other modes are selected, only these modes can only be displayed.)

The machine control state can be checked by the input and output display.

7-1. Block Run

[MDI] mode, [PRGRM]



Disregard to MEMORY PROTECT block run.

By only the above items (one block only) or by combination of the above items (piling of blocks), a sequence of the flow is completed, and the block run is performed.
The operation flows from the top to the bottom.

Item	Description
Coordinate system setting	The X and Y axes are returned to the zero position of the current machine position. The working coordinate system of the Z axis is set to the value offset from the machine coordinate position of 400.000 by the tool length (displayed by the [ ] key).
External signal	The signal is output to operate other equipment. * When an operation cannot be performed in the machine, when the index rotation is performed, two-digit signal is output so that the external equipment can be operated as ordered.6. * Two-digit signal setting The following digits cannot be used because they have been used for the internal signals: 00, 01, 04, 05, 08, 09, 30, and 90 to 99. (For details, refer to 7-6, Input/Output Display.)
Tools	Sets the magazine number where the necessary tool is mounted. * Random approach (Refer to Section II. Outline 2. Features and Configuration of the Machine.)
Coolant	Sets the open and close of the coolant valve. (0: OFF 1: ON) Before the instruction, set the coolant motor switch on the operation panel to ON. * The machine is lubricated after the block run is carried out. To stop the lubrication, set the the valve to OFF.
Spindle Rotation speed	Stops rotating the spindle. The speed can be selected as follows: TC-215 6 to 6000 min^{-1} (rpm) Special specification 10 to 10000 min^{-1} (rpm) TC-225 6 to 6000 min^{-1} (rpm) * The spindle is continuously rotated after the block run is carried out. To stop the rotation, set the rotation speed to 0. (0: stop 1: Clockwise rotation 2: Counterclockwise rotation) Once the rotation speed is set, the set value does not change after the operation ends. Set the speed as requested.

Item	Description
Axis movement X ■■■■■■ Y ■■■■■■ Z ■■■■■■	Moves the X, Y, and Z axes to the free position, machine zero position, and working zero position. Sets the free working coordinate value. Both X and Y axes can move together, and the Z axis can move alone.
Feed	Each axis can be fed as follows: (0: Positioning 1: Cutting 2: Tap) 0: Positioning / Rapid movement — Maximum feedrate of X, Y, and Z of parameter 1 is made valid. 1: Cutting / Jog movement — The feedrate is determined according to the feedrate of the spindle. 2: Tap Tapping — The speed is determined by the spindle rpm setting.
Feedrate	The jog feedrate is set. The feedrate can be set freely from 0.2 to 196.9 inch/min.

o Modal state

Even after the operation ends, neither the spindle rotation nor the lubrication stops.

In addition, since the display of the speed and the feedrate is not cleared, move the cursor for stop.

Examples of block run

(1) Sets the new coordinate system.

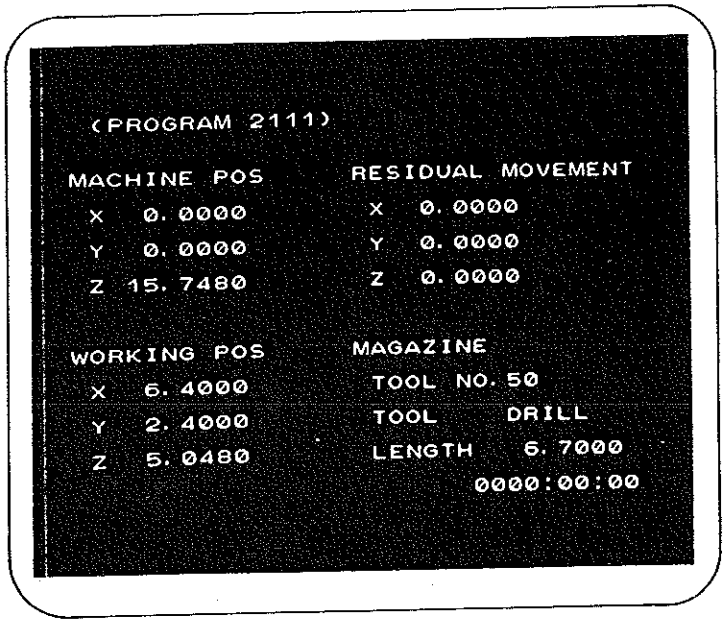
In the [MDI] mode and [PRGRM].

Set the [⇧ CURSOR] to the [COORDINATE].

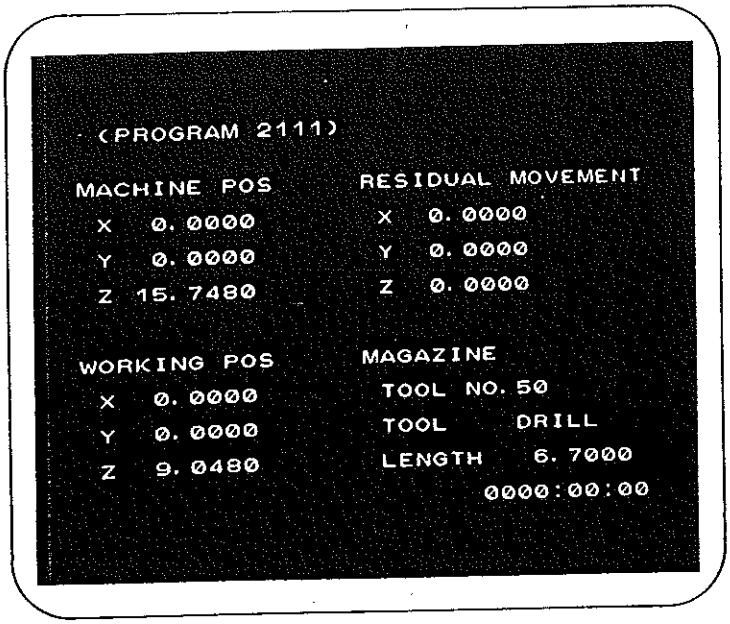
Set as follows: [/] (1:ON) [ENTER].

Then [START].

Check by the [POS] key.



⇩



(2) Outputs external signals.

Signal 25

In the [MDI] and [PRGRM] .

Set the [⇧ CURSOR] and [⇩ CURSOR] to the external signal.

Set signal 25, [START] .

The signal is sent to the output terminal, given in binary decimal two digits.

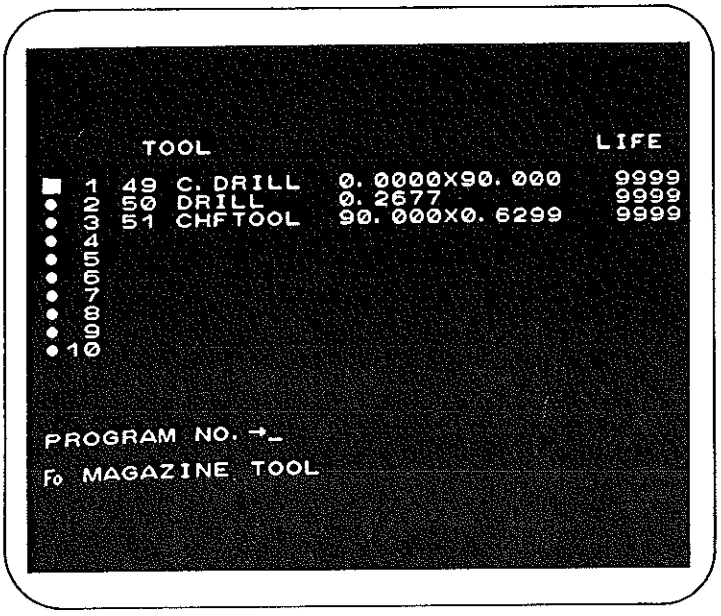
CODE	2				5 (4+1)			
BINARY DECIMAL	$(2^3=8)$	$(2^2=4)$	$(2^1=2)$	$(2^0=1)$	$(2^3=8)$	$(2^2=4)$	$(2^1=2)$	$(2^0=1)$
	0	0	1	0	0	1	0	1
TERMINAL	M28	M24	M22	M21	M18	M14	M12	M11
OUTPUT SIGNAL			•			•		•

The control box outputs signals as shown in the above figure.

(3) Selects the tools.

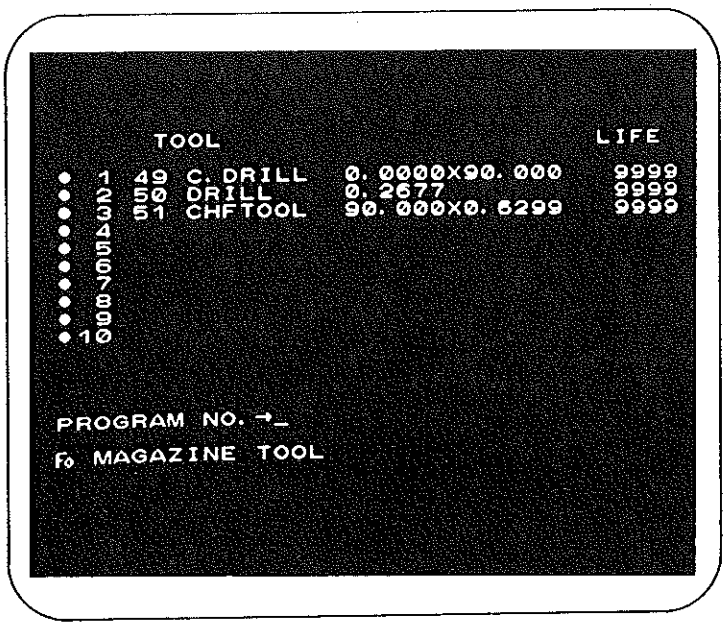
Set the [⇧ CURSOR] and [⇩ CURSOR] to the tool.
Select the chamfering tool mounted in magazine No. 3.
Set '3', push the [ENTER] and [START] .
Before the operation: Magazine tool No. 1 is attached to spindle.

[MAGAZ]



The current magazine tool selects No.1.

After the operation: Magazine tool No. 3 is attached the spindle.



(4) Opens the coolant valve.

In the [MDI] and [PRGRM].

Set the [↑ CURSOR] and [↓ CURSOR] in the coolant.

Set [1] to ON, [START].

Set the coolant motor on the operation panel to ON.

⟶ To close the coolant valve, set the [↑ CURSOR] and [↓ CURSOR] in the coolant, set [0] to OFF, then activate.
Switch off the coolant motor switch on the operation board.

(5) Rotate the spindle at 400 min^{-1} (rpm) in the clockwise direction.

In the [MDI] and [PRGRM].

Set the [↑ CURSOR] and [↓ CURSOR] in the spindle.

Set [1] (1:rotation), then set the rotation speed to [4000] 4000min-1 (rpm), and activate.

⟶ To stop rotating the spindle, set the [↑ CURSOR] and [↓ CURSOR] to the spindle.
Set [0] (0:stop), then activate.

(6) Move the X 5.9inch in the same direction from working coordinate value (X: 0.000, Y: 0.000, Z: 150.000).

Set the [⇧ CURSOR] and [⇩ CURSOR] in the axis movement.

Set [0] (0:position specification).

Set the cursor in the X axis, then set [-5, 9].

Then, set the feeding method.
Set [0] (0:positioning), [ENTER] and [START].

(7) Cutting rate (ϕ0.1535 drilling)

The Z axis cutter moves to position 130 at a rotating speed of 2,000 min^{-1} (rpm) and a feed rate of 7.8740 inch/min.

The coordinate values of the cutting position indicated on the position display are (X:0.000, Y:0.000, Z:5.9000), and the drill mounted in the spindle is tool No. 52.

Set the [MDI] and [PRGRM] keys to the spindle by pushing the [⇧ CURSOR] key in the [⇩ CURSOR] mode.
Push the [1] key (1: Clockwise Rotation), then push the [START] key after setting the number of revolutions to 2,000.
Adjust the [⇩ CURSOR] key according to the axis movement and push the [0] key (0: Position Setting).
Adjust the [⇩ CURSOR] key according to the axis movement and key-in [5.1000].
Push the [1] key (1: Cutting), then push the [START] key after setting the cutting speed to [7.8740].
* After completion of the cutting operation, reset the machine by shifting the axes.

Tool List Data	
Tool No.	52
Tool Name	Drill
Point angle	118.
Smaller diameter	*
Length of smaller diameter section	*
Center angle	*
Length of invalid section	*
Nominal outside diameter	0.1535
Pitch (threads per inch)	*
Length of cut	0.75
Spiral direction	*
Tool length	5.5
Life	9000

(8) Tapping (M5 nominal outside diameter and 0.8 pitch)

Tapping at Z axis position 5.1000 at a rotating speed of 2,000 min^{-1} (rpm).

The coordinate values of the tapping position indicated on the position display are (X:0.000, Y:0.000, Z:5.9000), and the P tap mounted in the spindle is tool No. 55.

Set the [CURSOR ⇩] key to the rotating speed by pushing the [PRGRM] key in the [CURSOR ⇩] mode, adjust the [CURSOR ⇧] or [MDI] to the spindle position. Set the [1] key (1: Clockwise Turn), and key-in [2000]. Adjust the [CURSOR ⇩] according to the axis movement, and push the [0] key (0: Position Setting). Then set the [CURSOR ⇩] to the Z axis, and key-in [5.1000].

Set the feed mode by pushing the [2] key (2: Tapping), and then push the [START] key.

* The tool is synchro-fed according to the pitch of the tap mounted in the spindle.

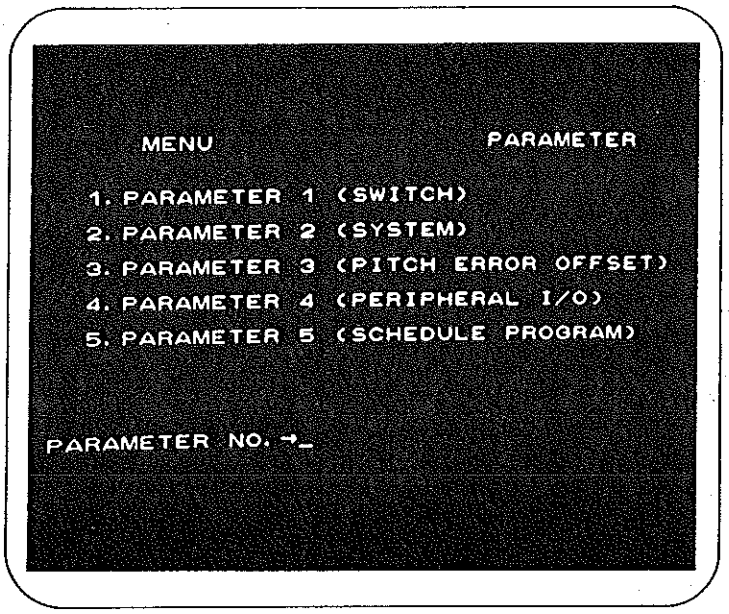
Upon completion of the tapping, the machine is automatically reset to the tapping coordinate position (X:0.000, Y:0.000, Z:5.9000).

Tool List Data	
Tool No.	55
Tool Name	P tap
Point angle	*
Smaller diameter	*
Length of smaller diameter section	*
Center angle	*
Length of invalid section	
Nominal outside diameter	UNF10
Pitch (threads per inch)	32
Length of cut	0.9
Spiral direction	Right
Tool length	5.5
Life	9000

7-2. Parameter

When this machine is delivered to you, the data necessary for the operation is set divided into the following five items. The data is very important, file them with care.

Set to [MDI] mode and [PARAM].



* When the machine is delivered to you, the test result table and parameter table are provided.

When menu numbers 1 to 5 are set, the details are displayed.
The displayed data can be altered according to the necessity.
Set [MDI] mode, parameter, and memory rewrite switch to on.
Set the menu number of the data to be altered, and position the cursor by using the [↑ CURSOR] and [↓ CURSOR] keys.
Set the necessary data. The data is divided into items to be altered by the [RESET] key and items to be altered by turning off the power supply (the power supply can be turned off on the operation panel). When an item is set, the message is displayed. (The screen is returned to the menu screen by the [Fo] key.
After the data is altered, set the memory rewrite switch to off.

Parameter TC-215

List of initialization values

1. Parameter (Switch)

ITEM	SET VALUE
MODE SELECT (0:MANUAL 1:MEMORY)	0
MANUAL FEED (0:RAPID 1:JOG 2:STEP)	0
TOOL BROKEN INTL MODE (0:VLD 1:INVLD)	/
RESTART (0:STOP 1:RESTART)	0
OPERATION TIME	0
TOOL LENGTH OFFSET VALUE	0
LENGTH AFTER THROUGH CUT	2. (0.1000)
STOP POSITION FORWARD WORK	2. (0.1000)
TAP PRIMARY HOLE DEPTH	6. (6.000)
REAMER PRIMARY HOLE DEPTH	1. (1.000)
REAMER FINISH ALLOW 1	0
REAMER FINISH ALLOW 2	0
MAXIMUM TIME OF EXT SIGNAL	0
DRY RUN OFFSET VALUE	0
DWELL (CHF)	0
DWELL (CBO)	0
REHR RETN SPD (0:RAPID 1:CUTNG)	0
SOFTWARE STROKE LIMIT X	-300. (-11.8110)
SOFTWARE STROKE LIMIT Y	-220. (-8.6614)
SOFTWARE STROKE LIMIT Z	0
MACHINING ORDER (1:INDV JOB-1PC 2:INDV TOOL-1PC) (3:INDV JOB 4:INDV TOOL)	2

2. Parameter (System)

	SET VALUE
BACKLASH VALUE X	0
BACKLASH VALUE Y	0
BACKLASH VALUE Z	0
STROKE X AXIS	300. (11.8110)
STROKE Y AXIS	220. (8.6614)
STROKE Z AXIS	200. (7.8740)
Z ORIGIN POSITION	400. (15.7480)
IN-POSITION WIDTH X	15
IN-POSITION WIDTH Y	15
IN-POSITION WIDTH Z	15
GRID SHIFT VALUE X	
GRID SHIFT VALUE Y	
GRID SHIFT VALUE Z	
GRID SHIFT VALUE SPINDLE	0
MAXIMUM DEVIATION X	20000
MAXIMUM DEVIATION Y	20000
MAXIMUM DEVIATION Z	20000
MAXIMUM DEVIATION SPINDLE	
XY TIME CONSTANT	100
Z TIME CONSTANT	150
SPINDLE TIME CONSTANT	200
XY MAXIMUM SPEED	20000 (787.4)
Z MAXIMUM SPEED	20000 (787.4)
SPINDLE SPEED (0:4000 1:6000 2:10000)	
X ORIGIN OFFSET VALUE	0
Y ORIGIN OFFSET VALUE	0
Z ORIGIN OFFSET VALUE	0
DATA CLEAR	

* The Z axis zero point dimension is different in the high column (special specification).
* The values of glid shift of the X, Y and Z axes depend on machine models.
* The position deviation limit value of the spindle depends on the rotation speed of the spindle.

Rotation speed of the spindle	6000	10000
Position deviation limit value of the spindle	36000	60000

Parameter TC-225

List of initialization values

1. Parameter (Switch)

ITEM	SET VALUE
MODE SELECT (0:MANUAL 1:MEMORY)	0
MANUAL FEED (0:RAPID 1:JOG 2:STEP)	0
TOOL BROKEN INTL MODE (0:VLD 1:INVLD)	1
RESTART (0:STOP 1:RESTART)	0
OPERATION TIME	0
TOOL LENGTH OFFSET VALUE	0
LENGTH AFTER THROUGH CUT	2.(0.1000)
STOP POSITION FORWARD WORK	2.(0.1000)
TAP PRIMARY HOLE DEPTH	6.(6.000)
REAMER PRIMARY HOLE DEPTH	1.(1.000)
REAMER FINISH ALLOW 1	0
REAMER FINISH ALLOW 2	0
MAXIMUM TIME OF EXT SIGNAL	0
DRY RUN OFFSET VALUE	0
DWELL (CHF)	0
DWELL (CBO)	0
REAR RETN SPD (0:RAPID 1:CUTNG)	0
SOFTWARE STROKE LIMIT X	-420.(-16.5354)
SOFTWARE STROKE LIMIT Y	-300.(-11.8110)
SOFTWARE STROKE LIMIT Z	0
MACHINING ORDER (1:INDV JOB-1PC 2:INDV TOOL-1PC) (3:INDV JOB 4:INDV TOOL)	2

2. Parameter (System)

ITEM	SET VALUE
BACKLASH VALUE X	0
BACKLASH VALUE Y	0
BACKLASH VALUE Z	0
STROKE X AXIS	16.5354(420)
STROKE Y AXIS	11.8110(300)
STROKE Z AXIS	9.8425(250)
Z ORIGIN POSITION	17.7165(450)
IN-POSITION WIDTH X	15
IN-POSITION WIDTH Y	15
IN-POSITION WIDTH Z	15
GRID SHIFT VALUE X	
GRID SHIFT VALUE Y	
GRID SHIFT VALUE Z	
GRID SHIFT VALUE SPINDLE	
MAXIMUM DEVIATION X	20000
MAXIMUM DEVIATION Y	20000
MAXIMUM DEVIATION Z	20000
MAXIMUM DEVIATION SPINDLE	
XY TIME CONSTANT	150
Z TIME CONSTANT	150
SPINDLE TIME CONSTANT	200
XY MAXIMUM SPEED	787.4(20000)
Z MAXIMUM SPEED	590.6(15000)
SPINDLE SPEED (0:4000 1:6000 2:10000)	
X ORIGIN OFFSET VALUE	0
Y ORIGIN OFFSET VALUE	0
Z ORIGIN OFFSET VALUE	0
DATA CLEAR	

* The values of the grid shift value of the X, Y, and Z axes depend on machine models.

3. Parameter (Pitch Error Offset)

* Values depend on machine models.

Item	Setting value
Pitch error 1	■X 0 ■Y 0 ■Z 0
Pitch error 2	■X 0 ■Y 0 ■Z 0
Pitch error 3	■X 0 ■Y 0 ■Z 0
Pitch error 4	■X 0 ■Y 0 ■Z 0
Pitch error 5	■X 0 ■Y 0 ■Z 0
Pitch error 6	■X 0 ■Y 0 ■Z 0
Pitch error 7	■X 0 ■Y 0 ■Z 0
Pitch error 8	■X 0 ■Y 0 ■Z 0
Pitch error 9	■X 0 ■Y 0 ■Z 0
Pitch error 10	■X 0 ■Y 0 ■Z 0
Pitch error 11	■X 0 ■Y 0 ■Z 0
Pitch error 12	■X 0 ■Y 0 ■Z 0
Pitch error 13	■X 0 ■Y 0 ■Z 0
Pitch error 14	■X 0 ■Y 0 ■Z 0
Pitch error 15	■X 0 ■Y 0 ■Z 0

4. Parameter (RS-232C)

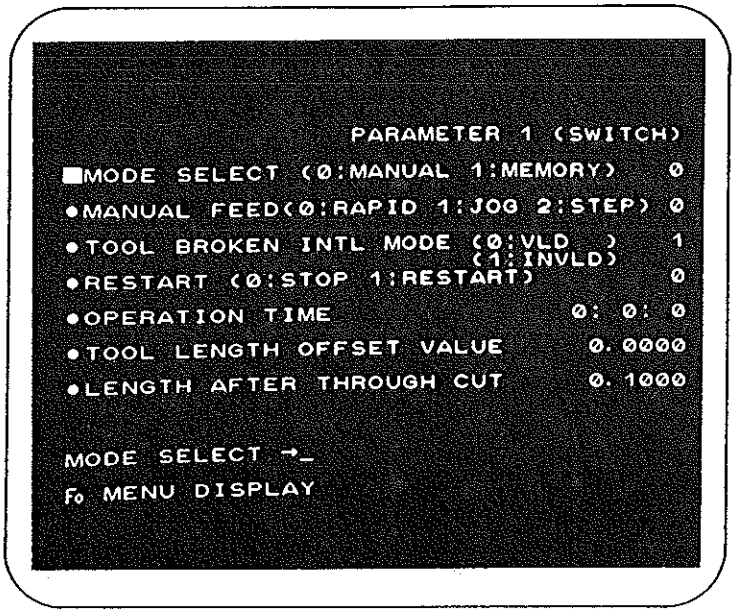
Item	Setting value
Baud rate (0: 300 1: 600 2: 1200 3: 2400 4: 4800)	4
Parity (0: Without 1: Odd number 2: Even number)	0
Stop bit (0: Bit 1: 2 Bit)	0
Connect (0: PTP/PTR 1: Computer 2: Printer 3: Valve)	3
How many lines in a page	0

5. Parameter 5 (Schedule program)

Item	Setting value
No. 1	
No. 2	
No. 3	
No. 4	
No. 5	
No. 6	
No. 7	
No. 8	

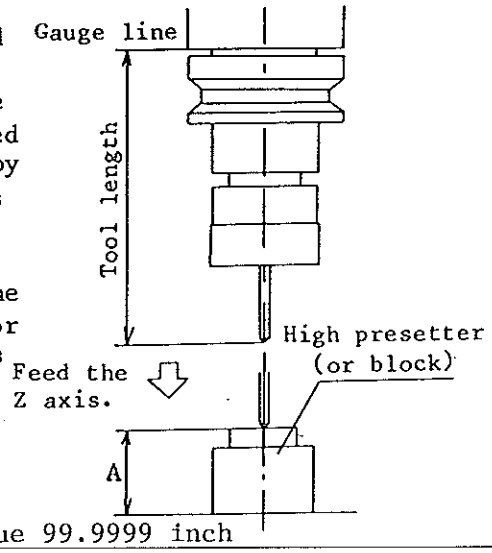
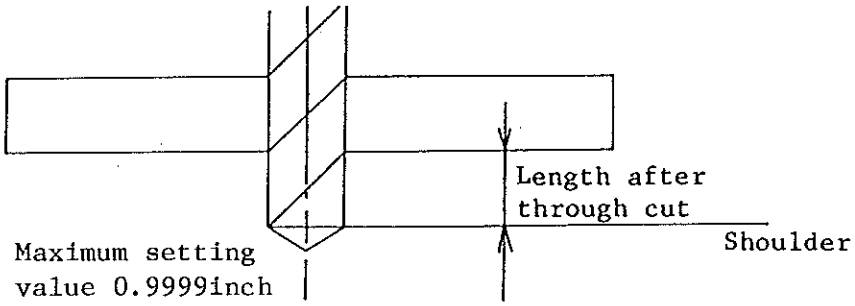
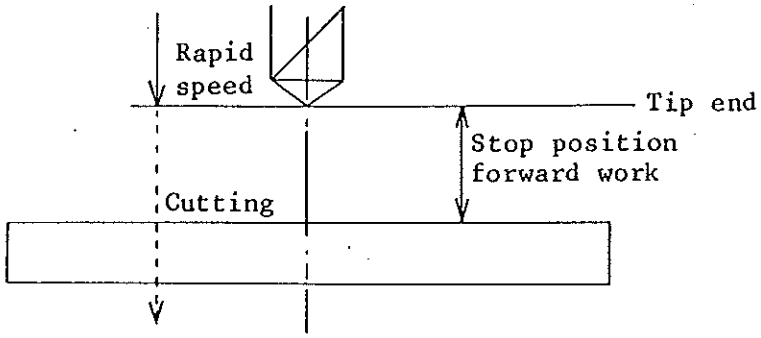
7-2-1. Parameter (Switch)

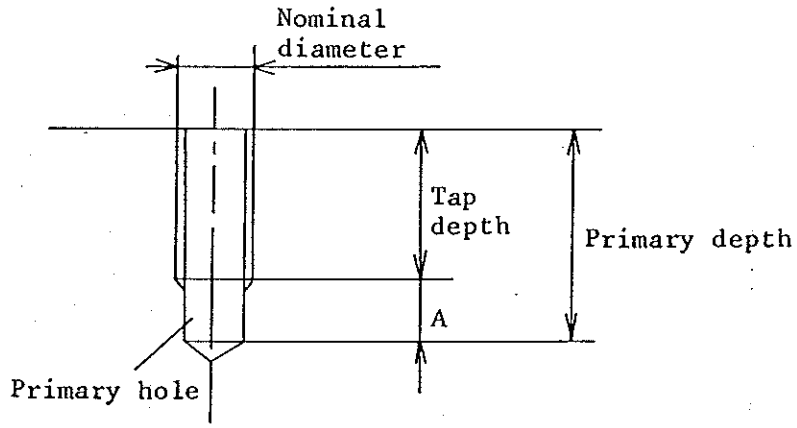
When the menu No. [1] is set, the following items are displayed.

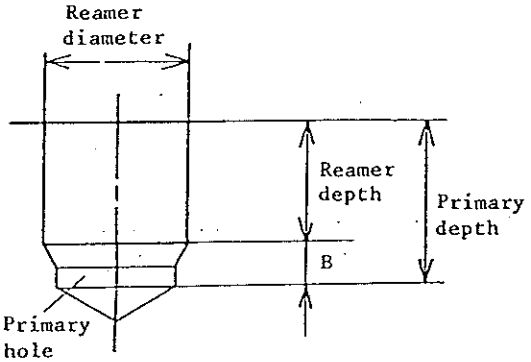
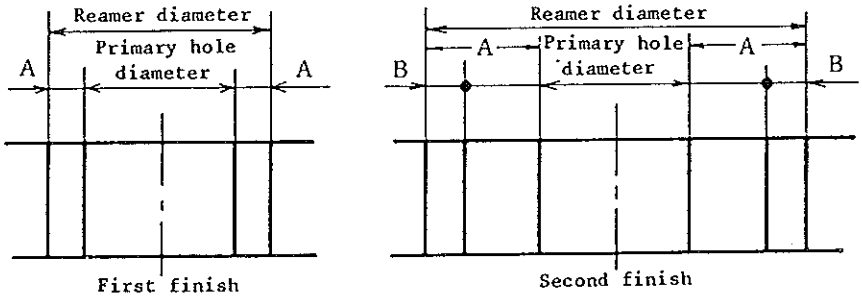


For the screens after the next page, refer to the parameter list for individual machine and initialization values.

Item	Contents
Mode selection (0: Manual 1: Memory)	Specifies the manual run or memory run mode when the power is switched on.
Tool broken INTL mode (0: Valid 1: Invalid)	Specifies whether the tool broken detector (special accessory) is valid or invalid in the memory run mode when the power is switched on.
Restart (0: Stop 1: Restart)	Specifies whether the operation stops or continues in the memory run when the program work is finished. The restart is usually unnecessary in the memory run. However, it is necessary to restart and to operate the machine for a certain time when the running-in is performed.
Operation time	Totalizes the time taken in the memory run and displays the time. The time is indicated for the automatic operation time in the position display. The initialization (start time) can be changed, and the setting time has effect on the automatic run time for the position display. Maximum totalized time: 9999 hours 59 minutes 59 seconds

Item	Contents
Tool length offset value	If tool length cannot be measured in advance, the tool length can be measured and stored in the tool menu by feeding the Z axis as shown in the right-hand figure. Set height A of the height presetter or block used in this operation.  Maximum setting value 99.9999 inch
Length after through cut	This sets feed amount after through cut. Maximum setting value 0.9999inch 
Stop position forward work	This sets the distance (air cut) from the changeover position from rapid feed rate to jog feedrate.  If thickness fluctuates heavily due to skin, adjust the position accordingly. The position before the cutting work is determine by the stop position forward work. If the stop position forward work is larger than the return position, the position is determined by the return position. Maximum setting value 0.999 inch

Item	Contents
Tap primary hole depth	Sets tap primary hole depth as the difference between primary hole depth and tap depth. When the primary depth is less than or equal to the depth in the blind hold processing, the parameter is used. Tap primary hole depth = (Primary depth) - (Tap depth) / Pitch = A/Pitch * Same as seat mounted tap.  * If the primary depth is less than or equal to the depth in the process data, the process is performed with the condition of prepared hole depth = (depth) + (tap prepared hole additional depth) × (Pitch). Maximum setting value: 99.999 threads

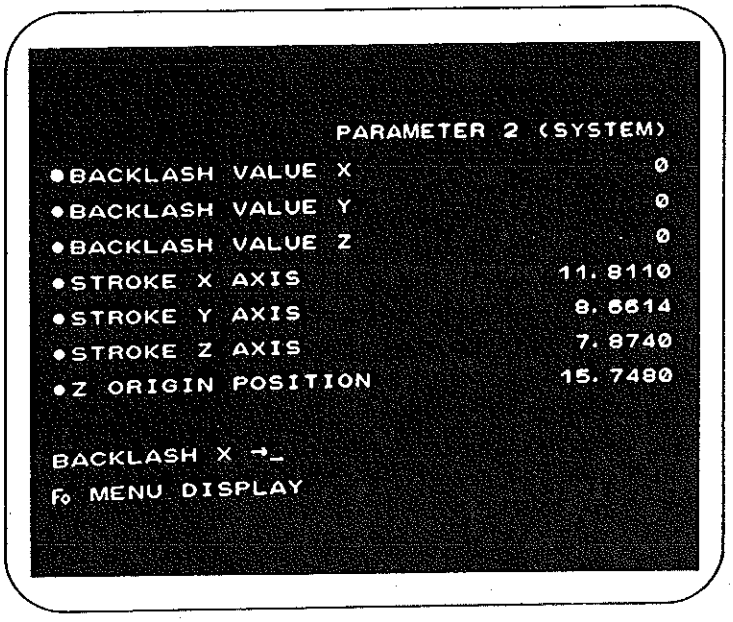
Item	Contents
Reamer primary hole depth	Sets the rate of difference between primary hole and reamer depth to the diameter of the hole in percentage. When the primary depth is less than or equal to the depth in the blind hole processing, the parameter is used. Reamer primary hole depth = (Primary hole depth) - (Reamer depth)/Reamer diameter = B/Reamer diameter * Same as seat mounted reamer.  * When the primary depth is less than or equal to the depth in the processing data, the work is performed with the condition of primary hole depth = (Depth) + (Reamer primary depth) x (Hole diameter). Maximum setting value: 99.999
Reamer finis allow 1	Sets the finish allow 1 or 2 in response to the reamer working. Set A + A.  Maximum setting value: 0.9999 inch
Reamer finish allow 2	Sets the finish allow 2 in response to the second reamer working. Set B + B shown in the above figure. Maximum setting value: 0.9999 inch

Item	Contents
Maximum time EXT signal	Sets an allowable response time of external signal from the time when the external signal is output to the time when the instruction indicating that the external operation is finished is received. Maximum setting value: 9999 sec.
Dry run offset value	Sets the offset value of the Z axis at the time of the test run (dry run) before the memory run. Maximum setting value: 99.9999 inch
Dwell (Chamfer)	Stops temporarily the Z axis to increase the accuracy after the completion of the chamfering. Effective when the following tools are assigned: drill, drill tap, drill reamer, and countering boring drill for chamfering. Setting value 0 to 99 Units of 0.1 sec.
Dwell (Counter boring)	Stops temporarily the Z axis to increase the accuracy after the completion of spotfacing. Setting value 0 to 99 Units of 0.1 sec.
Reamer return speed (0: Rapid feed 1: Cutting feed)	Specifies whether the return speed is set in the cutting feed or in the rapid speed.
Stroke limit X axis Stroke limit Y axis Stroke limit Z axis	Sets the movable range of the X and Y axis by the machine coordinate value, and sets the movable range of the Z axis from the top of the table to the lowest position of the tool end. Maximum setting value XY -99.9999 inch Maximum setting value Z 99.9999 inch
Machining order	(1: Indv job-1pc 2: Indv tool-1pc) (3: Indv job 4: Indv tool)

7-2-2. Parameter 2 (System)

When menu number [2] is set, the following items are displayed.

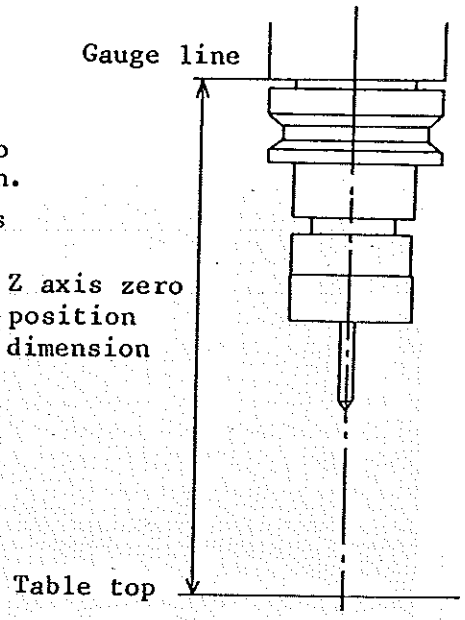
o Care must be taken to change the items, because the change has effects on the machine.



For the screens after the next page, refer to the parameter list for individual machine and initialization values.

Item	Contents
Backlash offset value of the X axis Backlash offset value of the Y axis Backlash offset value of the Z axis	The backlash offset value must be periodically measured according to the machine using state. In addition, the number of the pulses for each of X, Y, and Z axes. Offset pulse = Offset value (measured value u) −: 1.22 Convert the compensation value into pulse number by using the above formula. Maximum setting value: 255 pulses
Stroke of the X axis Stroke of the Y axis Stroke of the Z axis	The stroke limits of the X, Y, and Z axes have been set. The stroke limits from the axes are set.

	X	Y	Z
215	11.8	8.7	7.8
225	16.5	11.8	9.8

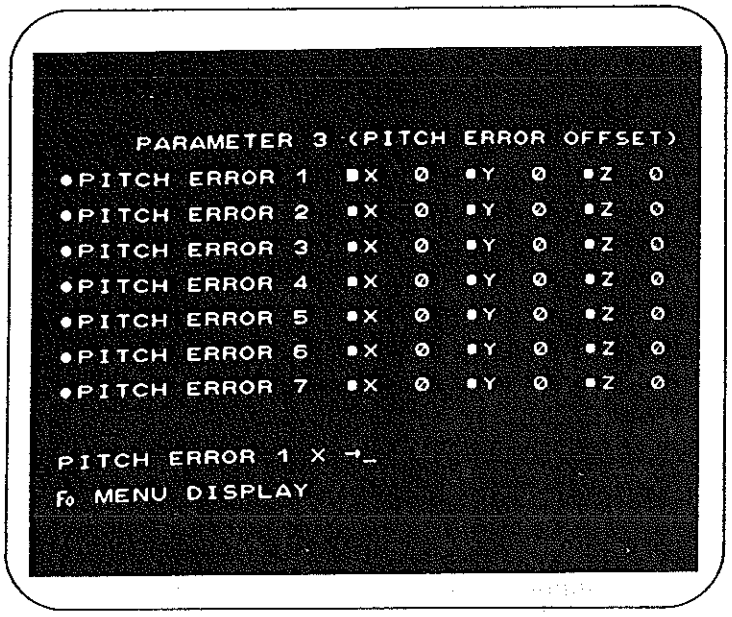
Item	Contents
Z origin position	The distance from the table top to the spindle end gauge line is set as the Z axis zero position dimension. *The table top does not include the subtable (Special accessory). 
In-position width of the X axis In-position width of the Y axis In-position width of the Z axis	Sets the allowable range of reaching position of the instruction value for each of the X, Y, and Z axes. When the value is set to 0, it takes time for each axis to reach to position 0. Therefore, it is necessary to set the valuewithin the allowable range. When the machine is delivered, the pulse is set to 15. The allowable range is specified as 15 pulses = 0.00072 inch in both positive and negative directions against the instruction value. * In the cutting mode, the in-position check is not performed. Maximum setting value: 255 pulses
Grid shift value of the X axis Grid shift value of the Y axis Grid shift value of the Z axis Grid shift value of the spindle	Sets the shift value of the motor zero position for each of the X, Y, and Z axes and spindle. In the zero position return, the zero position signal is detected, and the axes are moved to positive or negative direction for the grid shift value, so that the position is specified as the zero position. Maximum setting value: $\pm$ 9999 pulses
Maximum Deviation X Maximum Deviation Y Maximum Deviation Z Maximum Deviation Spindle	Sets the allowable deviation between the set value for each axis of X, Y, and Z and spindle and the actual machine position. Maximum setting value: 65535 pulses (Only the spindle 99999 pulses)

<table>
<tr><th>Item</th><th>Contents</th></tr>
<tr><td>X, Y and Z time constant, and spindle time constant</td><td>The time constant of acceration or deceleration at the time of activation or stopping of the X, Y, and Z and spindle.
Set the time of reaching to the desired speed.
Minimum setting value
<table><tr><th></th><th>XY axes</th><th>Z axis</th><th>Spindle</th></tr><tr><td>215</td><td>100</td><td>150</td><td>200</td></tr><tr><td>225</td><td>150</td><td>150</td><td>200</td></tr></table>Maximum setting value: 999 mec</td></tr>
<tr><td>XY Maximum speed
Z Maximum speed</td><td>Sets the maximum movement speed of the X, Y and Z axes.
Maximum setting value
<table><tr><th></th><th>XY axes</th><th>Z axis</th></tr><tr><td>215</td><td>65.6</td><td>65.6</td></tr><tr><td>225</td><td>65.6</td><td>49.2 ft/min</td></tr></table></td></tr>
<tr><td>Rotating speed of the spindle</td><td>Sets the maximum rotating speed of the spindle from among the following specifications:
1:6000min-1(rpm) - Standard specification
2:10000min-1(rpm) - Special specification</td></tr>
<tr><td>X ORIGIN OFFSET VALUE
Y ORIGIN OFFSET VALUE
Z ORIGIN OFFSET VALUE</td><td>Sets the delay amount in detecting motor zero position signal in return to the zero position operation.
This value is set to eliminate the delay amount in detecting motor zero position signal caused by the difference of the position generated when the dog kicks off the zero position limit.
Maximum setting value 9999 pulses</td></tr>
<tr><td>Data clear</td><td>This is set when memory data is deleted.
(Refer to 7-5-5. Batch Deletion in Key-in Run Mode, Cutting Tool Condition.)</td></tr>
</table>

7-2-3. Parameter 3 (Pitch Error Offset)

When the menu number is set to [3], the following items are displayed.

- o Since the parameter has effects on the accuracy, great care must be taken to alter the parameter.

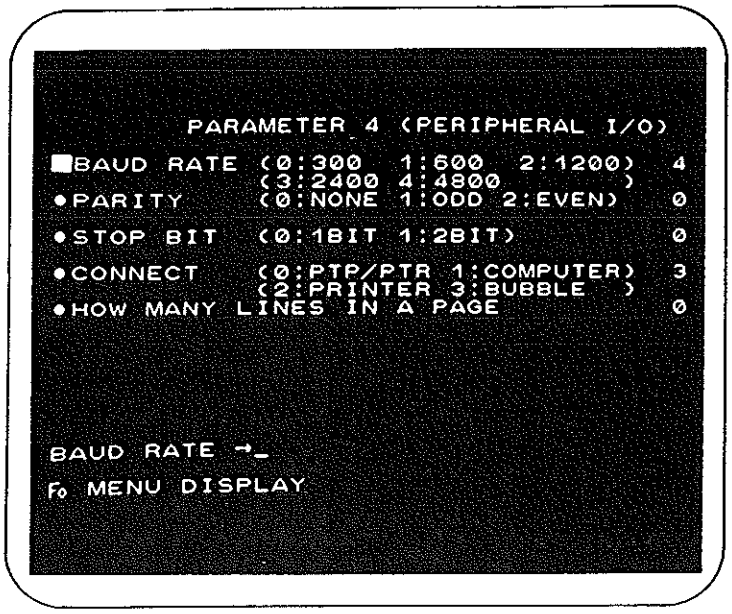


For the screens after the next page, refer to the parameter list for individual machine and initialization values.

Item	Contents
Pitch error 1 Pitch error 2 Pitch error 3 Pitch error 4 Pitch error 5 Pitch error 6 Pitch error 7 Pitch error 8 Pitch error 9 Pitch error 10 Pitch error 11 Pitch error 12 Pitch error 13 Pitch error 14 Pitch error 15	Compensate error by checking the pitch accuracy of the ball screw which moves the X, Y and Z axes. Pitch error of the X, Y and Z axes can be compensated. Up to 15 positions can be compensated with pitch of TC-215 to 0.8 inch or TC-225 to 1.6 inch from the zero position side. The compensation value must be given in pulse number (integer). Up to + 99 pulses can be compensated. Formula for converting pitch error into pulse: Compensation pulse number = compensation amount (measured value inch)/0.000048

7-2-4. Parameter 4 (RS-232C)

When the menu number is set to [4], the following items is displayed:

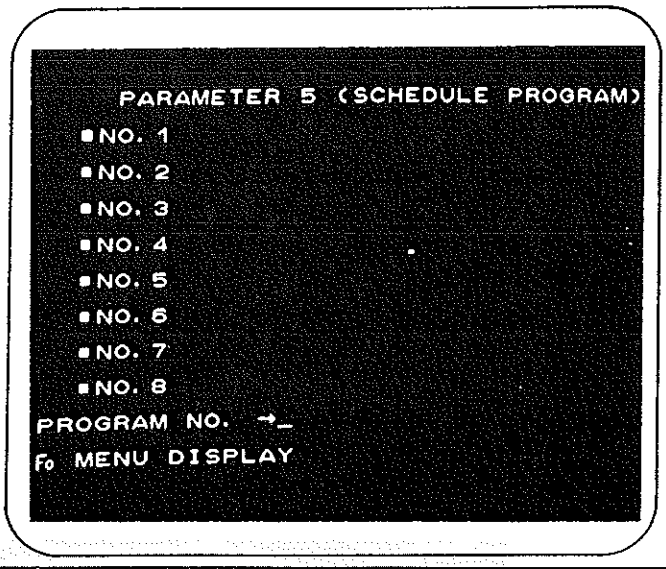


Item	Contents
Baud rate (0: 300 1: 600 2: 1200 3: 2400 4: 4800)	Sets the transfer rate of data. Provides compatible function necessary for connecting the CNC and various types of equipment.
Parity (0: 1 1: Odd 2: Even)	Check function for data transfer. Provides compatible function necessary for connecting the CNC and various types of equipment (via RS-232C). By adding parity bits (1) odd number or (2) even number and sending data to the connected equipment, checks whether the correct transfer has been performed. When the bits are set to 0, data is transferred without checking function.
Stop bit (0: 1 bit 1: 2 bits)	Sets whether the identification bit of data end is given in 1 bit or 2 bits. Provides the compatible function for connecting the CNC and the various types of equipment (via RS 232C).
Connection 0: PTP/PTR 1: Computer 2: Printer 3: Bubble	Selects the other equipment from among the tape puncher, tape reader, computer, printer, and bubble memory unit.

Item	Contents
How many lines in a page	Sets the number of lines per page when the machine is connected to the printer. Maximum setting value: 99 (When the number is set to 0, page feed is not set.)

7-2-5. Parameter 5 (Schedule Program)

When the menu number is set to [5], the following item is displayed.



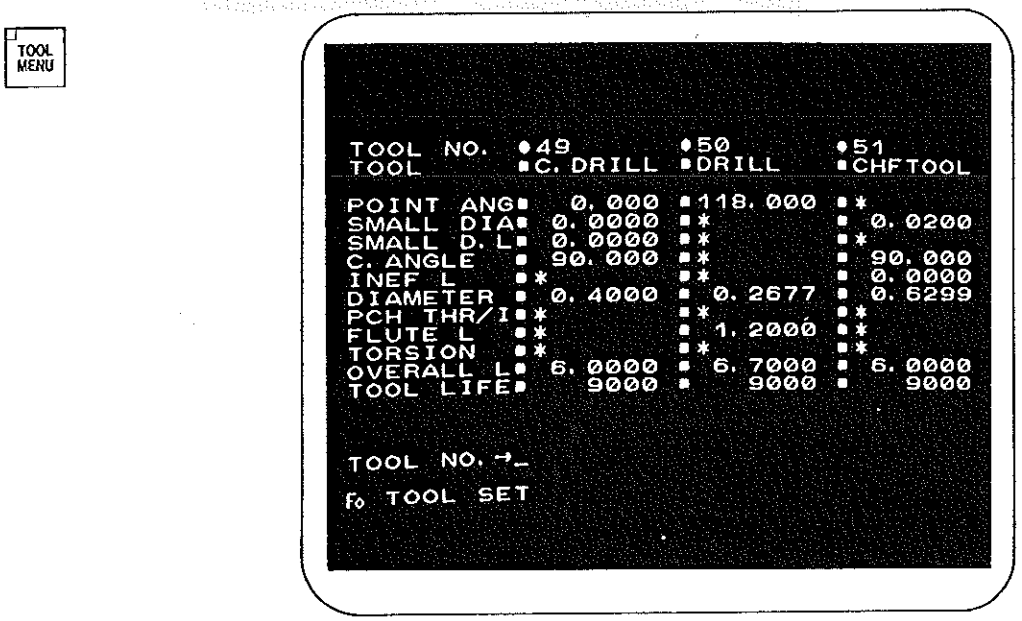
Item	Contents
No. 1 No. 2 No. 3 No. 4 No. 5 No. 6 No. 7 No. 8	This parameter is set when the programs are combined in the memory run. Up to eight program can be consecutively executed. When the programs are set, they are executed in the setting order from program No.1. The schedule program has merits under the following conditions: 1. When the working program has long jobs and the number of the jobs exceeds 69. 2. When more than one tool pattern cutting condition is contained in the same type of job according to the processed object. 3. When more than one working zero position is necessary to avoid complicating working programs in the case when an index table is used or different types of work is processed at the same time. ex) No. 1 1117 No. 2 1118 Program numbers 1117 and 1118 are consecutively executed. (For how to execute the programs, refer to Memory Run.)

7-3. Tool Menu

Register the number and data for the tool needed to operate the machine. There are 13 types of tools and up to 60 tools can be registered.

Anytime change is made in tool size or shape, be sure to alter the registered data.

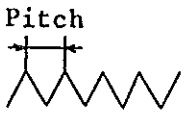
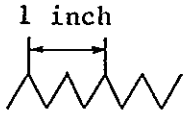
In addition, in this case, tools must be assigned to the related programs from all over again. Discrepancy in data of shape or size between registered tools and magazine tools, if any, may lead to instrumental breakage or accident. Therefore, be careful not to mix up the registered tools in tool storage.



Contents of necessary registration items

(Use this list by referring to the shape and dimensions of the tools.)

Item	Contents	Tool to be input
Tool No.	Registration number. Up to 60 numbers, from 1 to 60, can be registered.	
Tool name	Selected from among 13 types of teaching data areas.	All tools
Point angle: a	Angle at the point	Center drill, drill, chamfer drill, seat cutting drill, drill tap, drill reamer
Small diameter : d	Outer diameter	Center drill, drill, chamfer drill, cuttingtools, drill tap, drillreamer, stepped drill,chamfer tools
Small diameter length: l	Length of the outer diameter at the point	Center drill, chamfer drill, drill tap, drill reamer, stepped drill, counterboring tools
Center angle: θ	Large diameter angle of center drill Point angle of chamfer tool	Center drill, chamfer tool, chamfer drill
Invalid section length: LO	Chamfer length * Tap is given in threads and pitch.	Reamer, hand tap, spiral tap, drill tap, drill reamer, point tap, chamfer tools
Outer diameter, nominal size: D	Outer diameters of tools * Tap means nominal diameter. Registration method Select typs of screws according to the teaching data areas and set the size (outer diameter).	All tools

Item	Contents	Tool to be input
Pitch, threads per pitch: P	Tap: Meter screw  Inch screw  Threads per inch	
Effective cutting length : L	Effective cutting length	Drill, reamer, hand tap, spiral tap, end mill, counterboring tools, drill tap, drill reamer, counterboring drill, point tap, point tap, chamfer drill
Torsion	Torsion direction of tap Select (right or left) according to the instruction data areas.	Point tap, hard tap, spiral tap, drill tap
Tool length: LL	Length from the gauge line of the tool holder to the point of the tool	All tools
Life	Sets life in the registered tool. Time is given in minutes. The tool life is counted from the setting time. When the life expires, an error is displayed.	All tools

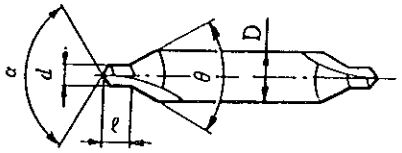
List of necessary input items of the registered tools

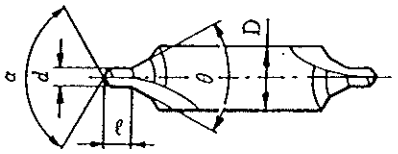
Item \ Tools	Center drill	Drill	Spiral tap	Point tap	Hand tap	chamfer tool	counter-boring tool
Point angle α	○	○	*	*	*	*	*
Small diameter d	○	*	*	*	*	○	○
Small diameter length l	○	*	*	*	*	*	○
Center angle θ	○	*	*	*	*	○	*
Invalid section length l	*	*	○	○	○	○	*
Outer diameter, nominal size D	○	○	○	○	○	○	○
Pitch, Threads P	*	*	○	○	○	*	*
Effective cutting length Lo	*	○	○	○	○	*	○
Torsion Direction	*	*	○	○	○	*	*
Tool length LL	○	○	○	○	○	○	○
Life Minute	○	○	○	○	○	○	○

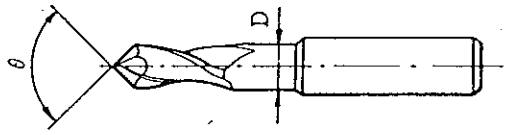
Item \ Tool	Chamfer drill	Stepped drill	Drill tap	End mill	Reamer	Drill reamer	
Point angle α	○	○	○	*	*	○	
Small diameter d	○	○	○	*	*	○	
Small diameter length l	○	○	○	*	*	○	
Center angle θ	○	*	*	*	*	*	
Invalid section length l	*	*	○	*	○	○	
Outer diameter, nominal size D	○	○	○	○	○	○	
Pitch, Threads P	*	*	○	*	*	*	
Effective cutting length Lo	○	○	○	○	○	○	
Torsion Direction	*	*	○	*	*	*	
Tool length LL	○	○	○	○	○	○	
Life Minute	○	○	○	○	○	○	

Shape and dimension of tools

1. Center drill

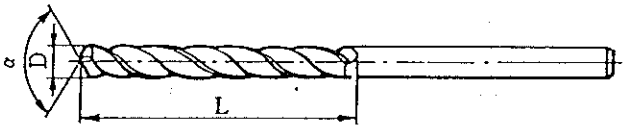




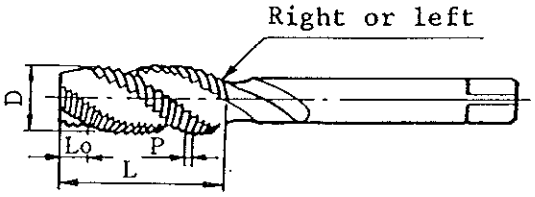


d = Ø
ℓ = Ø
α = Ø

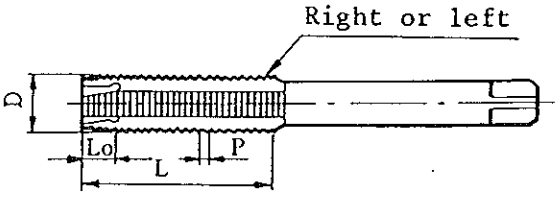
2. Drill



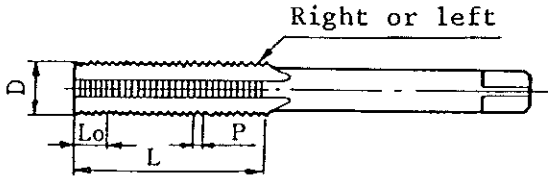
3. Spiral tap



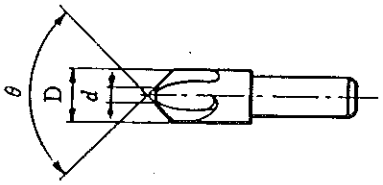
4. Point tap



5. Hand tap

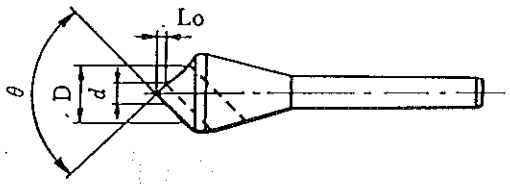


6. Chamfer tool

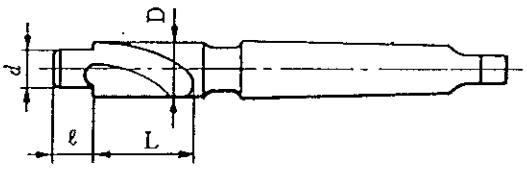


Lo = ∅

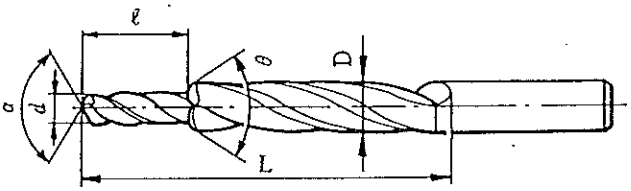
Chamfer tool



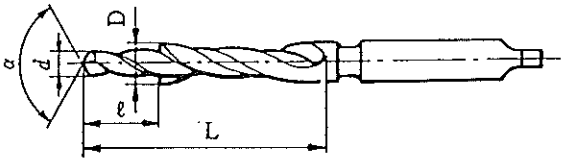
7. Counter boring tool



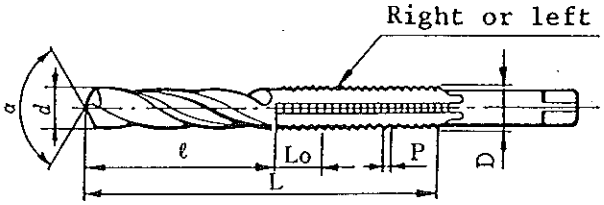
8. Chamfer drill



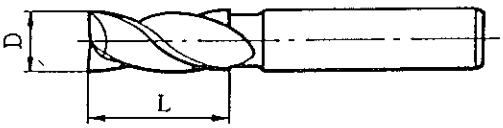
9. Stepped drill



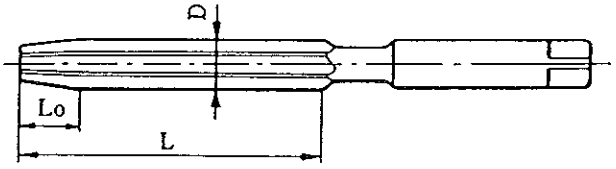
10. Drill tap



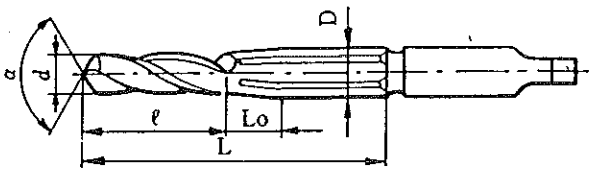
11. End mill



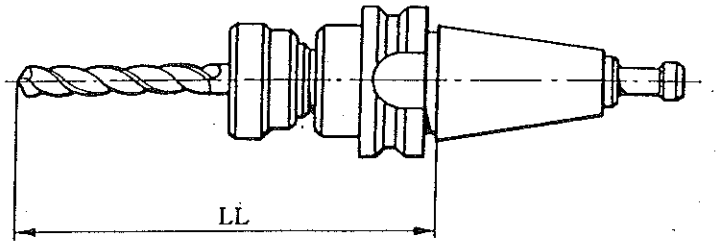
12. Reamer



13. Drill reamer

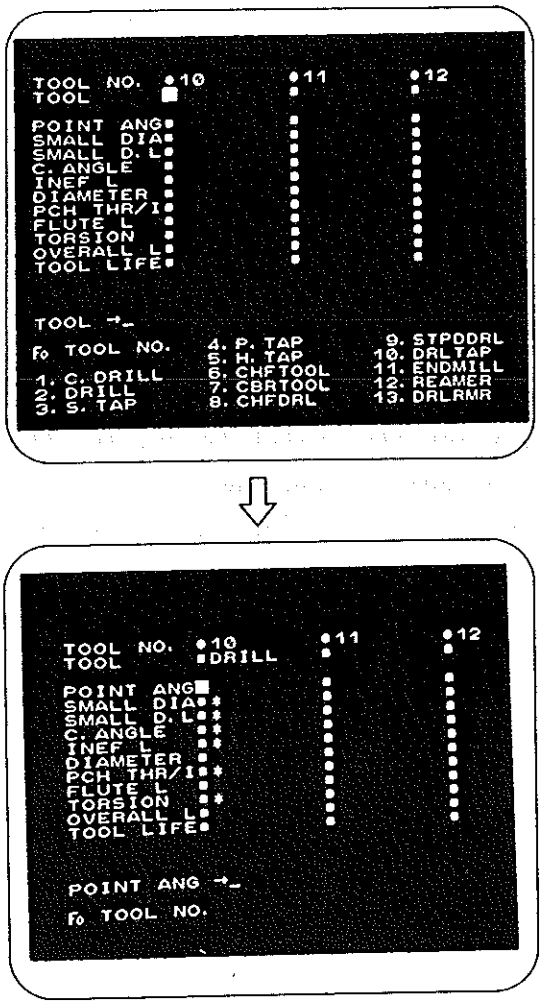


Length



Registration Method

Push [MDI] key and memory protect switch to on. Push [TOOL MENU] key and set cursors [⇧ CURSOR] and [⇩ CURSOR] in the tool number to be registered. Press the [Fo] key for setting data. Select the tool from among 13 types of tools in the instruction data area. When the tool is set, cursor [ENTER] will move over the input items of the selection tools, then set the data by using cursor [ENTER].



When the data is completely set, pressing the [Fo] key moves the cursor on the screen by [⇧ CURSOR] and [⇩ CURSOR] keys.
When the tool number to be displayed in the input request area [Tool →] is set, the applicable display screen appears. After the data is completely entered, set the memory switch to off.

o Auxiliary code

When tool names are registered, auxiliary codes 1 to 9 can be registered together. (If an auxiliary code is to be registered as 0, tool name without auxiliary code is registered.)

With an auxiliary code registered, similar tools (such as tools with different tool length, work materials, or tools different due to working) can be easily distinguished.
In general, tools in frequent use are registered with auxiliary code, and tools in rare use are registered without auxiliary code.

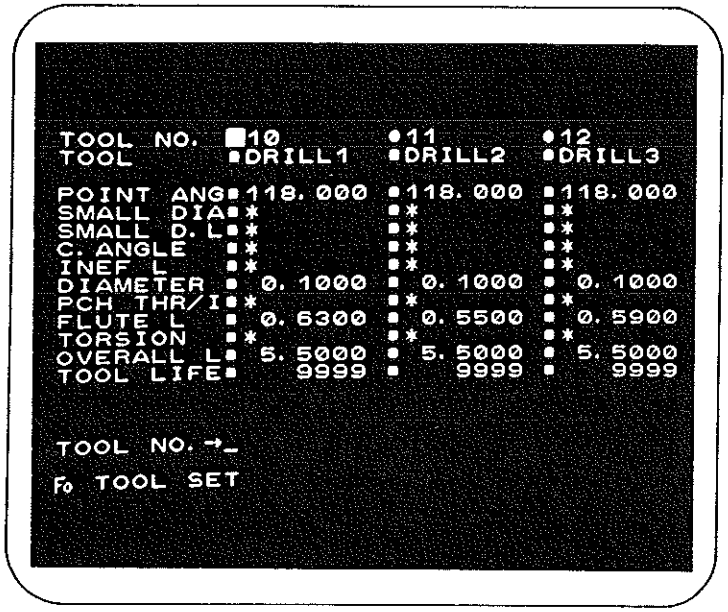
(ex) Aluminum -------- Without auxiliary code
Stainless ------- Auxiliary code 1
Soft steel ------ Auxiliary code 2 etc)

To assign tools with auxiliary code when the program edit tools are assigned, set the memory rewrite switch to on, set the tools with auxiliary code in program edit 4, cutting tool and cutting condition.

* The assignment of the auxiliary codes makes the desired assignment smooth.

ex) When three drills with different tool length are registered:
Set the key-in run mode and memory rewrite switch to on.

Set the [TOOL MENU] , [⇧ CURSOR] , and [⇩ CURSOR] to the number to be registered, and set the screen in the setting state by using the [F0] key.
Select 2. drill from the teaching area, then set auxiliary code 1, as follows: [2] [−] [1] .
Then, as mentioned in the previous page, enter the desired data into the necessary items.
Thus, the other two drills can be registered with auxiliary codes 2 and 3.


TOOL NO. ■10 ●11 ●12
TOOL ●DRILL1 ●DRILL2 ●DRILL3
POINT ANG ●118.000 ●118.000 ●118.000
SMALL DIA ●* ●* ●*
SMALL D.L ●* ●* ●*
C. ANGLE ●* ●* ●*
INEF L ●* ●* ●*
DIAMETER ● 0.1000 ● 0.1000 ● 0.1000
PCH THR/I ●* ●* ●*
FLUTE L ● 0.6300 ● 0.5500 ● 0.5900
TORSION ●* ●* ●*
OVERALL L ● 5.5000 ● 5.5000 ● 5.5000
TOOL LIFE ● 9999 ● 9999 ● 9999
TOOL NO. →_
F0 TOOL SET

Tool Length Setting

The tool length is calculated when the tool is mounted in the spindle, then it is automatically set in the tool menu.

Operation Procedure

(1) Set the tool to be measured automatically in the spindle. (Refer to Section 4, Tool Mounting and Removing.)

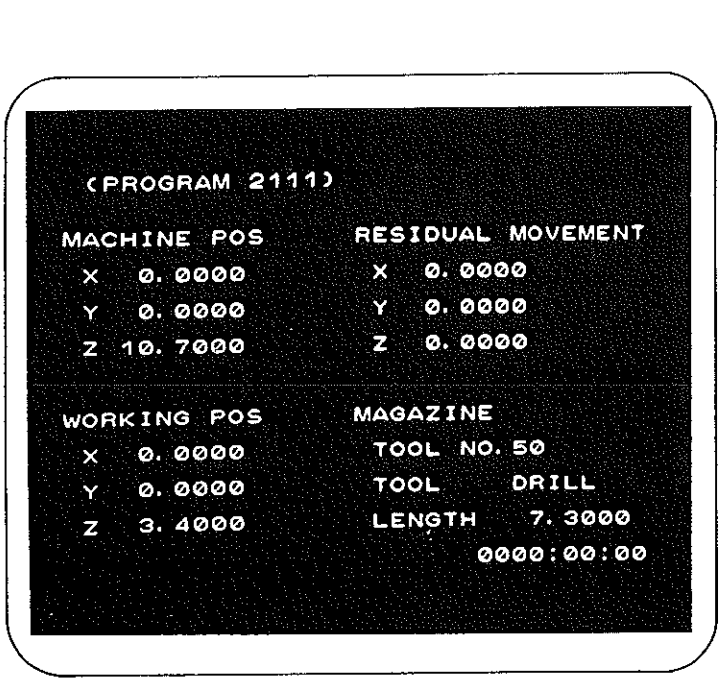
Check that the measurement tool is mounted in the magazine correctly on the [MAGAZ] screen.

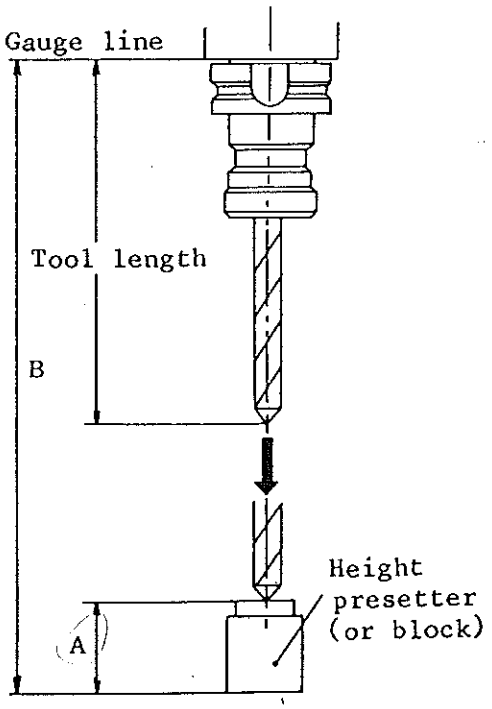
(2) Set the memory rewrite switch to on.

(3) Set the key-in run mode, parameter, and menu number [1] (1. switch).

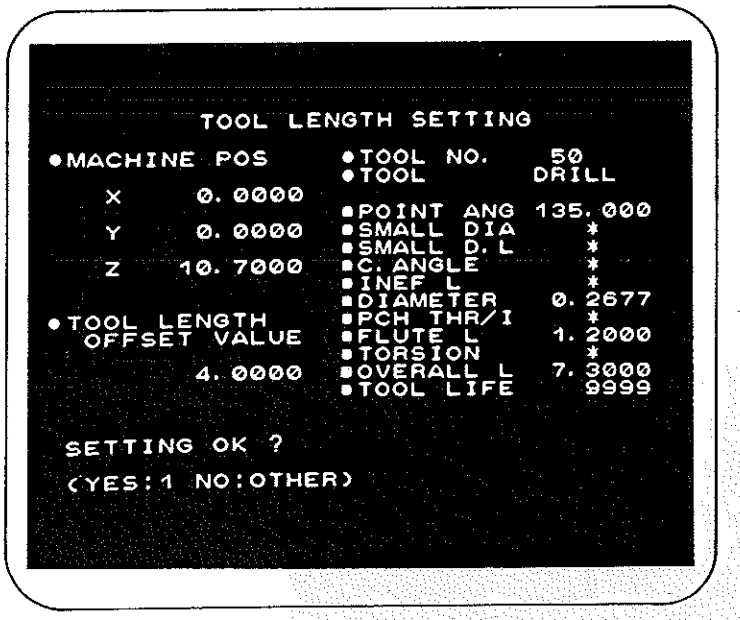
Set [⇧ CURSOR] and [⇩ CURSOR] in the tool length offset value, and set dimension A shown in the figure given below.

(4) In the manual mode, move the X axis to position 0 in the height presetter or the block.





(5) Set the [MDI] and [TOOL OFSET] to on.



(1: Set Other than 1: Not set)

As shown in the above figure, the tool length is set to 6.7.

Refer to Fig. 4-1. B : 10.67

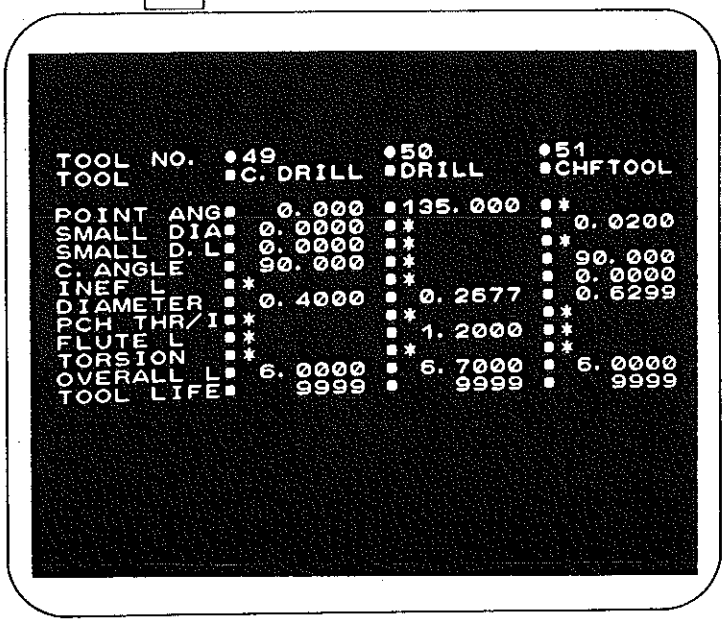
B - A = 6.7

A : 4.0

If the measured value is registered in the tool menu as tool length, set [1]. If not, set number other than 1.

(6) Set the memory rewrite switch to off.

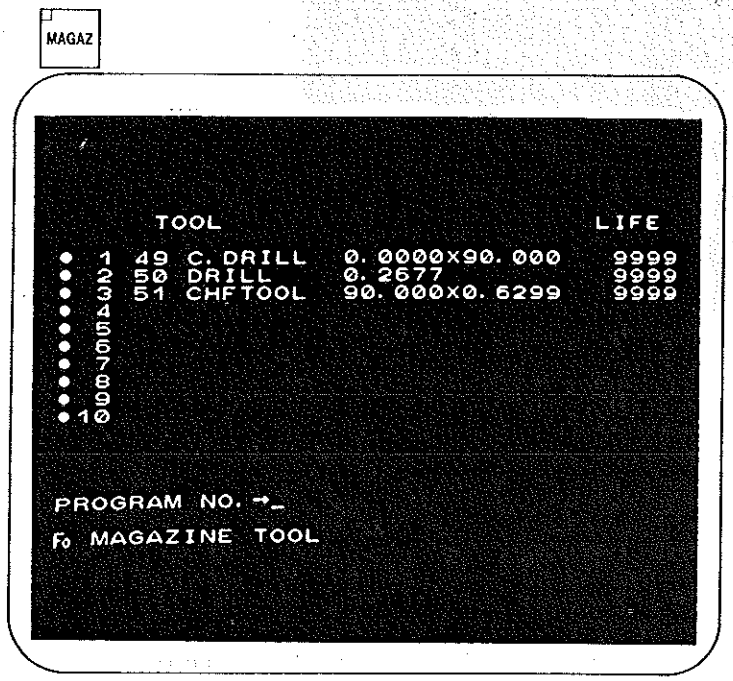
(7) By using the [TOOL MENU], check that the tool length is set.



7-4. Magazine Tools

This screen determines the number of the magazine (ATC) to install the tool registered in the tool menu.

* Before operation, check the number where the necessary tool is installed.



The tool condition set in the magazine used in the previous program edit is displayed.

Set the tool for work in the manual run or key-in run.
Set the [MDI] mode and memory rewrite switch to on.
Press the [MAGAZ] key, and the cursor flashes the magazine number on the spindle stock. Press the [F0] key to set the screen in the setting state. (The cursor can be moved.)
Set [⇧ CURSOR] and [⇩ CURSOR] the magazine number to be changed.
Set the tool number registered in the tool menu.
Install the tool having the same tool number as the magazine number on the screen.

* Set the tool to be used in the program edited for the memory run.

Set the [MDI] mode and memory rewrite mode, and press the [MAGAZ] key.

When 4-digit program number ex)1111, the tool assigned to the teaching data area is displayed. (The [Fo] key need not be pressed.)

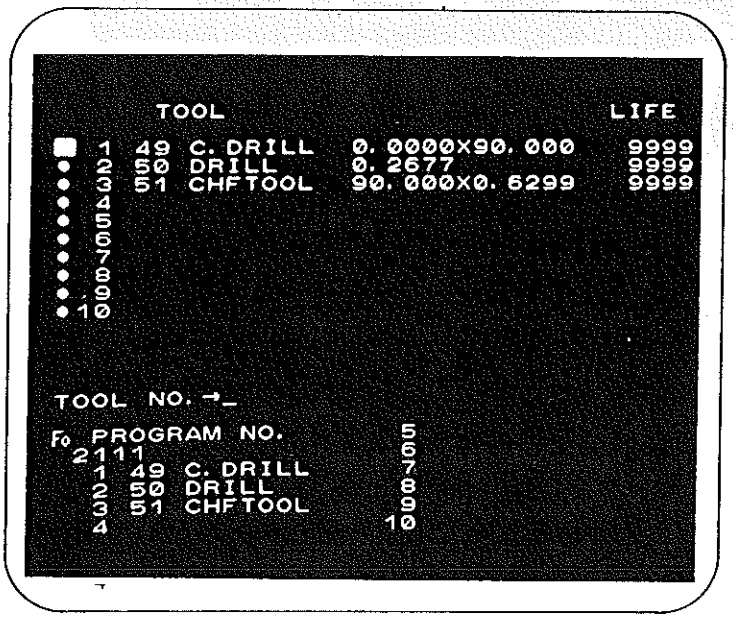
Set the cursor in the desired magazine number. (Any numberis allowed.)

Select tools in the teaching area, then set the tool number registered in the tool menu.

(When one-digity number is set, zero in the first position can be omitted.
ex) 01 -- 1)

When the number is set, set the memory rewrite switch to off.

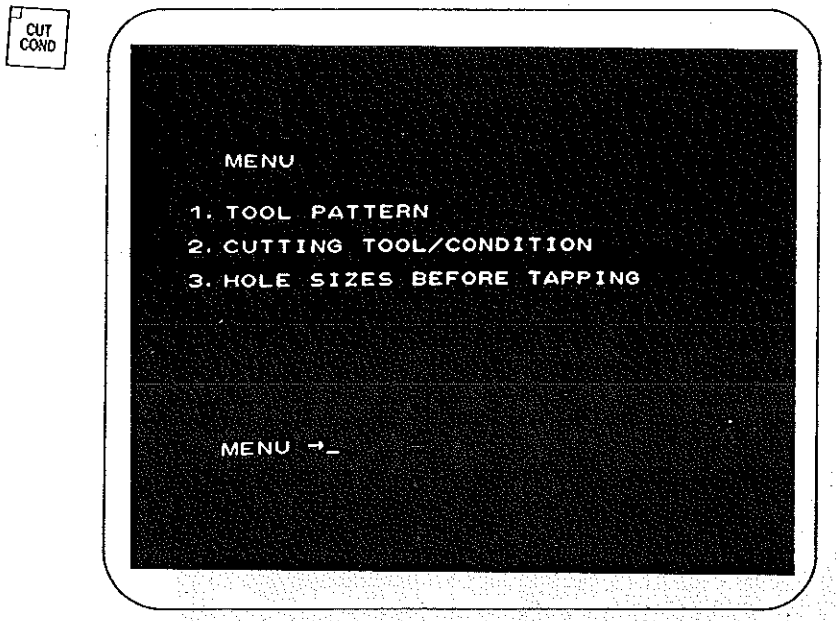
Install the tool having the same tool number as the magazine switch number on the screen.



(For details on how to install tools, refer to Section 4. Tool Mounting and Removing.)

7-5. Cutting Tool Condition

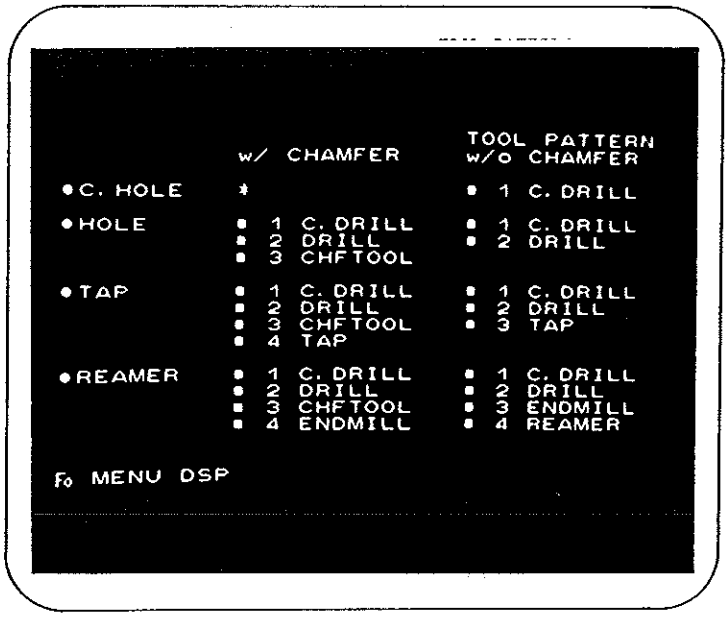
A combination of tools required for machining, peripheralvelocity (rotation speed on the peripheral), feed amount, andprimary hole diameter necessary for tap machining are set.
These conditions have effects on the memory run at the time of program edit.



When this machine is delivered to you, each type of cutting conditions has been set. These conditions can be displayed for check or the set values can be changed according to the necessity.

7-5-1. Tool Pattern

Tool pattern determines a combination of tools required for the completion of each matching.
Set the tool pattern [1].



A combination of tools is displayed on the screen, divided into a pattern with chamfer and a pattern without chamfer.

* Tap machining

Combination with chamfer at the time of machining

1. Center drill
2. Drill
3. Chamfer tool
4. Tap

After center hole machining, a hole before tapping is prepared. The hole is then chamfered, and is finally tapped.

Combination without chamfer at the time of machining

1. Center drill
2. Drill
3. Tap

After the center hole machining is completed, the hole before tapping is prepared. The hole is finally tapped.

When chamfer is specified in the program edit, necessary patterns are automatically selected from among the patterns with chamfer (see the table given below).

Machine type	With chamfer	Without chamfer
Center hole	*	1. Center drill
Hole	1. Center drill 2. Drill 3. Chamfer tool	1. Center drill 2. Drill
Tap	1. Center drill 2. Drill 3. Chamfer tool 4. Tap	1. Center drill 2. Drill 3. Tap
Reamer	1. Center drill 2. Drill 3. Chamfer tool 4. End mill 5. Reamer	1. Center drill 2. Drill 3. Counterboring tool 4. Reamer
Seat hole	1. Center drill 2. Drill 3. Counterboring tool 4. Chamfer tool	1. Center drill 2. Drill 3. Counterboring tool
Seat tap	1. Center drill 2. Drill 3. Counterboring tool 4. Chamfer drill 5. Tap	1. Center drill 2. Drill 3. Counterboring tool 4. Tap
Seat reamer	1. Center drill 2. Drill 3. Counterboring tool 4. Chamfering tool 5. End mill 6. Reamer	1. Center drill 2. Drill 3. Counterboring tool 4. End mill 5. Reamer
Top face cutting	*	1. End mill

Tool pattern initial setting

In general, tools are selected by the initial pattern. However, patterns are changed or deleted according to machining methods. For example, the center drill is deleted from the tool pattern for the holing process where the center process is unnecessary.

When a pattern is changed, operate according to the following procedure.

Set the memory switch to on in the [MDI] run mode.

Press the [CUT COND] key, and set the tool pattern.

Set the cursor in the portion to be changed by using the [PAGE↓], [PAGE↑], [CURSOR↑] and [CURSOR↓] keys.

Deletion --- The tool where the cursor flashes is deleted. In this case, data goes up by one.

ex)

	With chamfer				With chamfer
Hole	1. Center drill	[DELET]	Hole	1. Drill
	2. Drill			2. Chamfer tool
	3. Chamfer tool			3. *

Change of combination --- Select a tool to be changed from the teaching area, then set it.

ex)

	With chamfer				With chamfer
Tap	1. Center drill	[6] [START]	Tap	1. Center drill
	2. Drill			2. Chamfer drill
	3. Tap			3. Tap
	4. *			4. *

Insertion --- Adds a new tool before the tool where the cursor flashes. By pressing the key, data goes down by one.

ex)

	With chamfer				With chamfer
Hole	1. Drill		Hole	1. *
	2. Chamfer tool	[INSRT] [1] [ENTER]		2. Drill
	3. *			3. Chamfer tool

The tool is selected from the teaching area, then the insertion is completed.

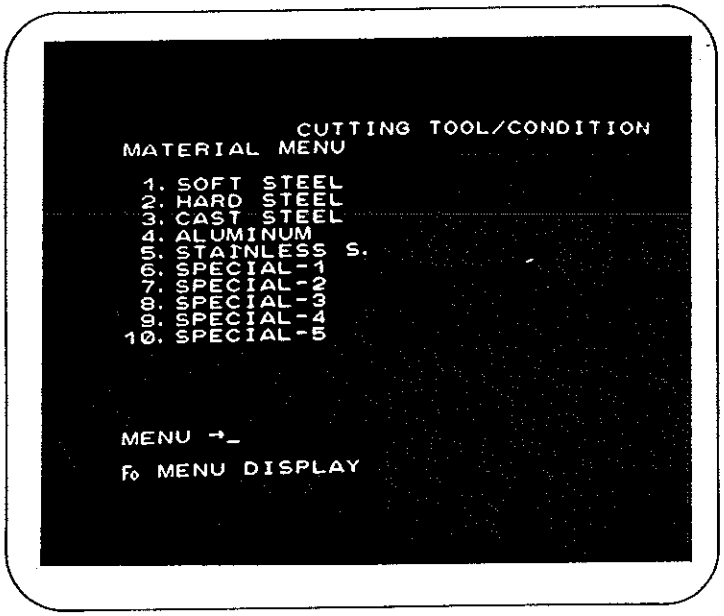
After the tool is set, set the memory rewrite switch to off.

* Unless the individual tool pattern for a program is set in program edit 3, the tool pattern is valid for the program.

7-5-2. Cutting Tool/Condition

The following 10 types of cutting conditions (peripheral velocity, feed amount) related to work materials are set. The setting values are specified in percentage when programs are edited.

Set the menu No. 2 of the cutting too/condition by the CUT COND key.



* The material menu cannot be changed.

The cutting tool/condition matched with the displayed material is set in work material numbers 1 to 5, which can be changed.
The user can set conditions in work material numbers 6 to 10.

Cutting condition can be changed according to the necessity.

In the [MDI] mode, set the memory switch to on.

In the cutting tool condition, set the cutting tool/condition [2], and set the material menu [4] (4.ADC12).

Display the change screen by using the [⇧PAGE], [⇩PAGE], [⇧CURSOR], and [⇩CURSOR] keys, then flash the cursor in the position to be changed.

Change of tool --- Select the new setting tool from the teaching data area, then set the tool.

Change of cutting condition --- Enter data, then set it.

Deletion --- Pressing the [DELET] key deletes the portion where the cursor flashes.

o Auxiliary code Auxiliary codes 1 to 9 can be set.

To assign smoothly the tools for the edited program, register the tools with auxiliary code.

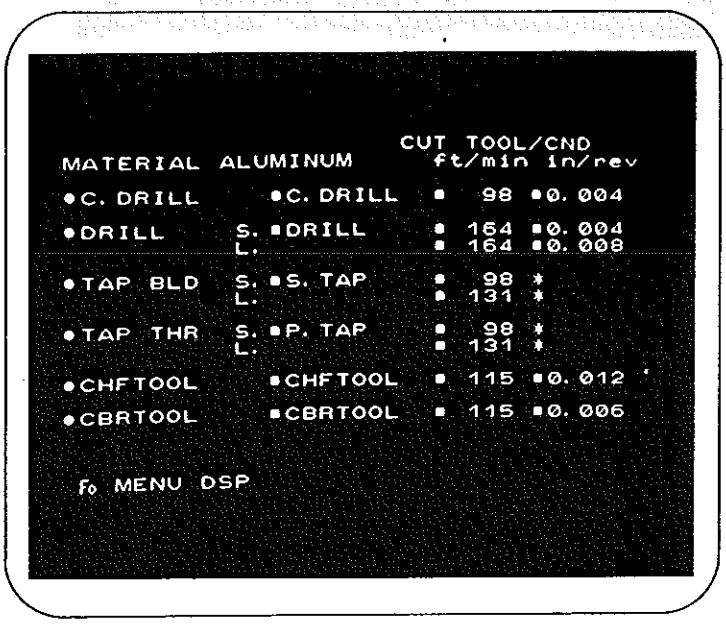
When the cutting condition of the used tool contains auxiliary code, the tool menu also selects tools having auxiliary code. Thereby, even the tools with similar diameter can be smoothly assigned.

When tools are selected from the teaching area in the above operation, the procedure of registering auxiliary codes is set as follows:

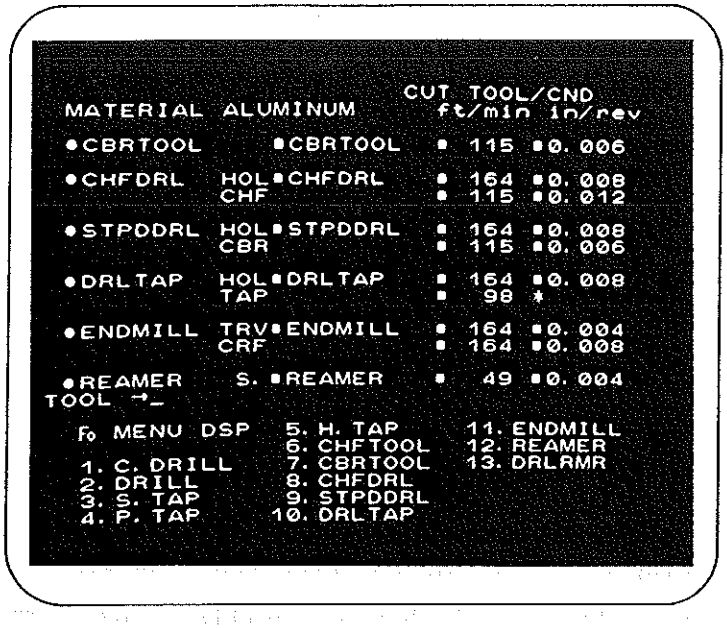
ex) When drill 2 is registered together with auxiliary code:

[2] [−] [1] [ENTER]

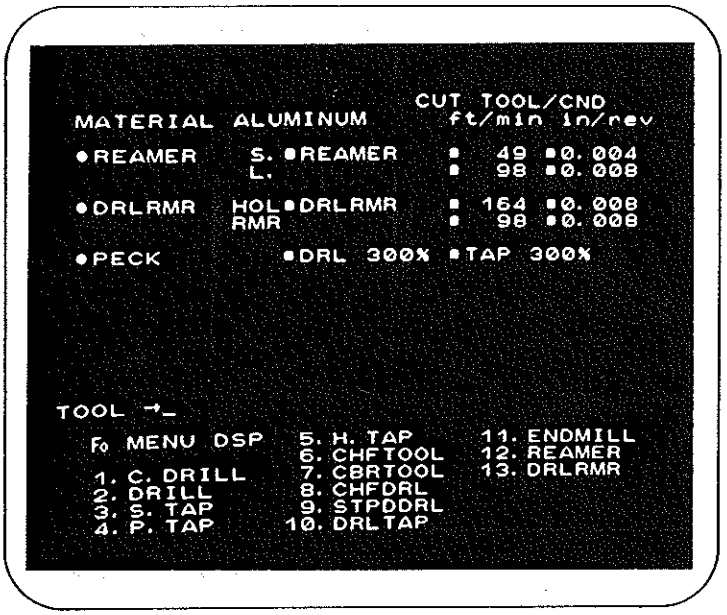
After the operation, set the memory rewrite switch to off.



Cutting condition display related to material menu 4. ADC12.
Set menu [4].

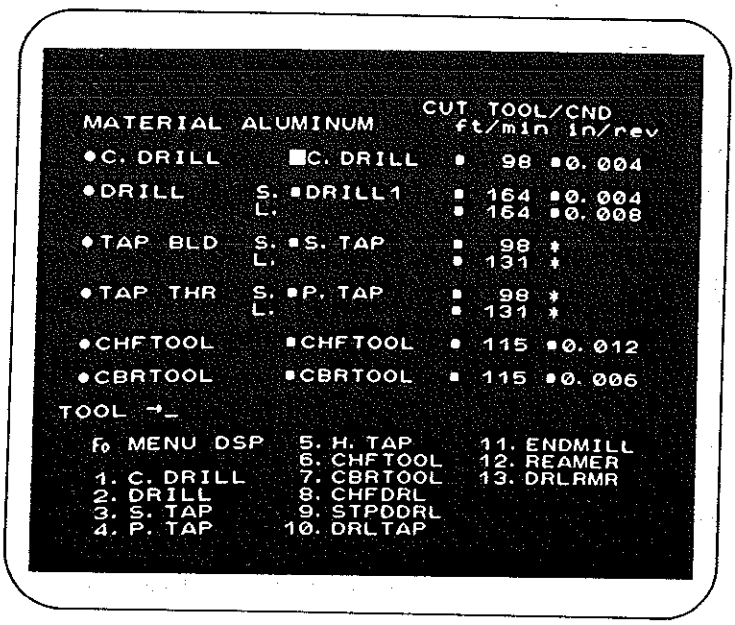


PAGE



Screen display	Contents
* Center drill * Drill * Tap Stop * Tap Through ¦ ¦ ¦ * End mill Vertical/ Horizontal ¦ ¦	Displays the working tool according to the working methods. Tap is displayed according to the stop hole working through hole work, and end mill is displayed according to vertical horizontal and vertical directions. In addition, cutting and other works are displayed.
0 - 3	Displays tool diameters. Displayed diameters are divided into two types: size from 0 to less than 0.12 inch, and size more than or equal to 0.12 inch.
Center drill Drill S tap ¦ ¦ End mill ¦ ¦	Displays the classification of tools. The taps to be displayed include S tap (Spiral tap) for stop hole working and P tap (Point tap) for through-hole working.
Peripheral velocity m/min Feed amount mm/revolution	Displays the cutting speed (peripheral velocity). Displays the feed amount for one revolution of the tool. Peripheral velocity and feed amount can be obtained by the following formulae. Peripheral velocity (peripheral rotating speed = diameter of tool X revolution rate X Peripheral velocity (ft/minute) = X (diameter of tool (inch) X (revolution rate min (rpm))/12

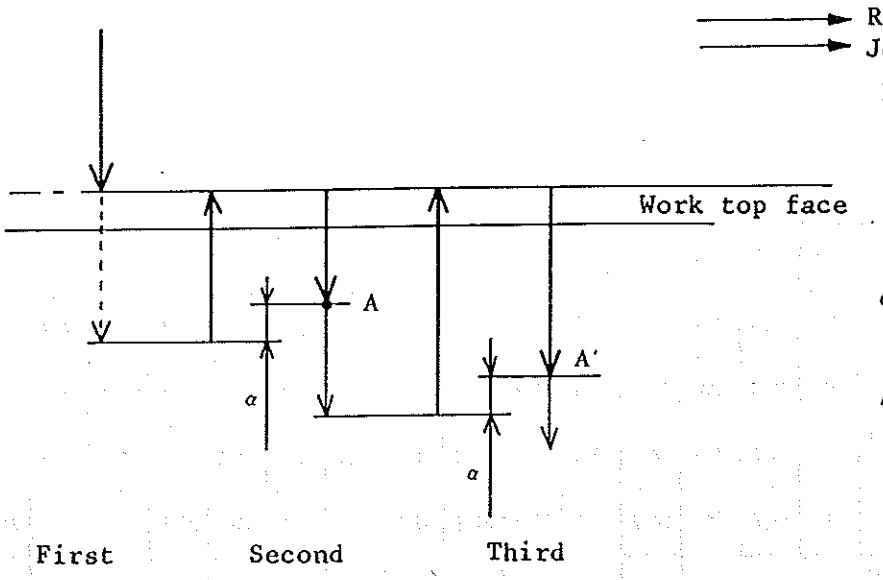
PAGE



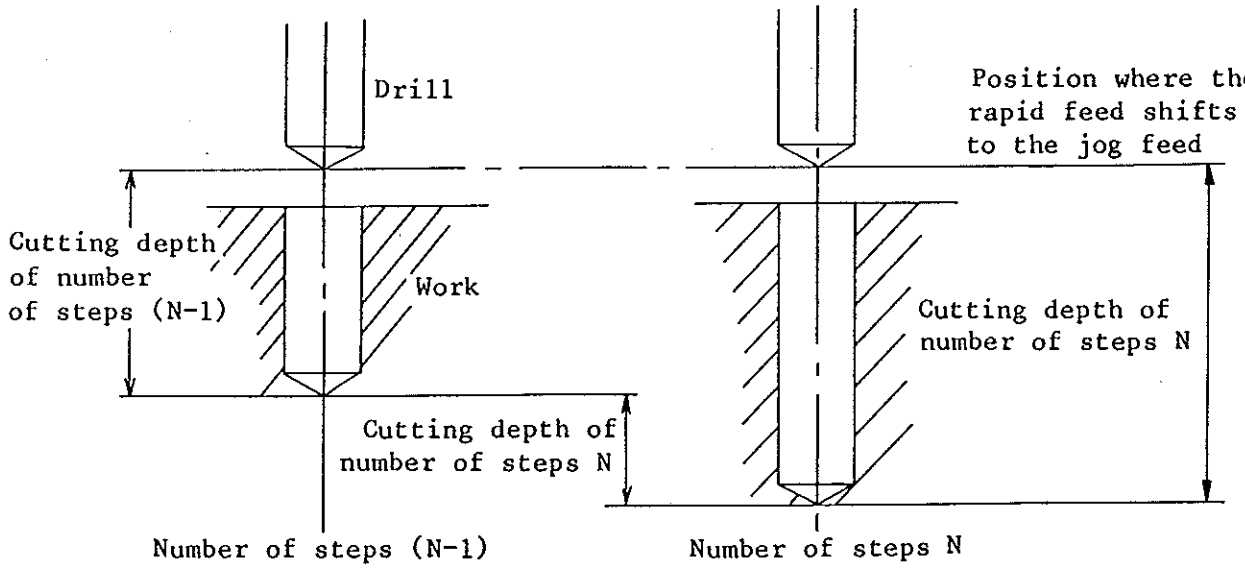
* Step

When a hole is deepened/tapped or material is difficult to hole/tap, the operation can be completed by the stepped tool. The number of steps can be set according to the necessity of the user. By standardizing the 100 (%) value of the ((number of step times) - (depth coefficient)) drawing built in the NC, set the rate to the value. Though the number of steps is unlimited, the cut depth of step at the 15th time or after uses one at the 14-th time.

1. Drill work step



After cutting, the drill step returns to the drill step return position at a high speed, then it goes to the stop position before cutting in the drill step at a high speed. The drill step cuts the material for cutting depth in jog feed, and it repeats these operations.



The cutting depth of the drill step is calculated by the following formula:

$$[\text{N-th cutting depth (inch)}] = \frac{[\text{cutting coefficient when the drill step is 100\%}] \times [\text{Drill outer diameter (inch)}] \times [\text{Rate of drill step (\%)}]}{100}$$

	Step	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15
300% (Aluminum)	Cutting depth coefficient	12	9.9	8.1	6.3	5.1	4.2	3.6	2.7	2.1	1.8	1.2	1.2	0.9	0.9	0.9
	Depth coefficient	12	21.9	30	36.3	41.4	45.6	49.2	51.9	54	55.8	57	58.2	59.1	60	60.9
200% (Cast steel)	Cutting depth coefficient	8	6.6	5.4	4.2	3.4	2.8	2.4	1.8	1.4	1.2	0.8	0.8	0.6	0.6	0.6
	Depth coefficient	8	14.6	20	24.2	27.6	30.4	32.8	34.6	36	37.2	38	38.8	39.4	40	40.6
100% (Soft steel)	Cutting depth coefficient	4.0	3.3	2.7	2.1	1.7	1.4	1.2	0.9	0.7	0.6	0.4	0.4	0.3	0.3	0.3
	Depth coefficient	4.0	7.3	10.0	12.1	13.8	15.2	16.4	17.3	18	18.6	19	19.4	19.7	20.0	20.3
80% (Hard steel)	Cutting depth coefficient	3.2	2.6	2.2	1.7	1.4	1.1	1.0	0.7	0.6	0.5	0.3	0.3	0.2	0.2	0.2
	Depth coefficient	3.2	5.8	8	9.7	11.1	12.2	13.2	13.9	14.5	15.0	15.3	15.6	15.8	16	16.2
70% (Stainless)	Cutting depth coefficient	2.8	2.3	1.9	1.5	1.2	1.0	0.8	0.6	0.5	0.4	0.3	0.3	0.2	0.2	0.2
	Depth coefficient	3.2	5.1	7	8.5	9.7	10.7	11.5	12.1	12.6	13.0	13.3	13.6	13.8	14	14.4

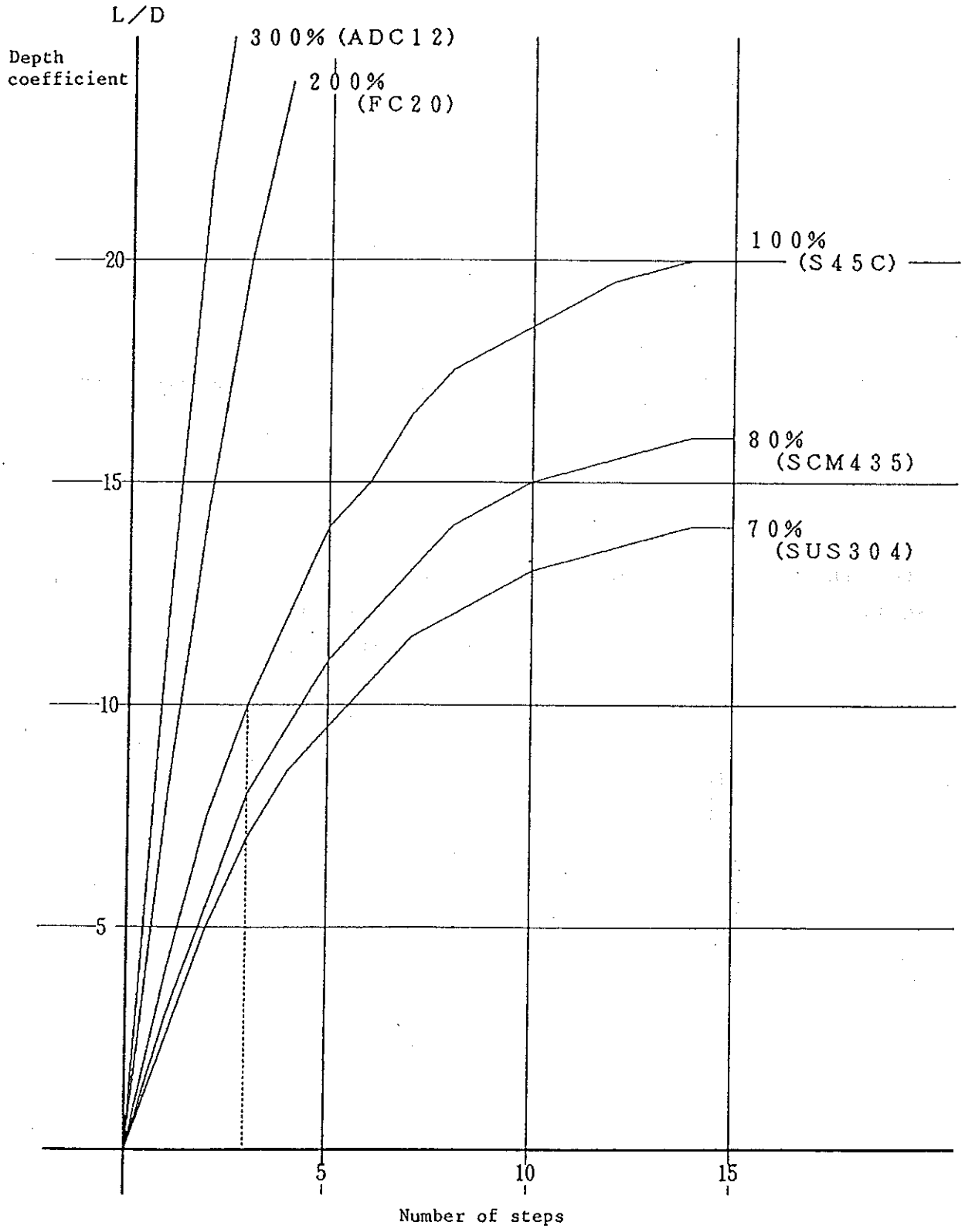
Note 1) (Cutting coefficient) = (Cutting depth)/(Hole diameter)

Note 2) (Depth coefficient) = (Cutting depth)/(Hole diameter)

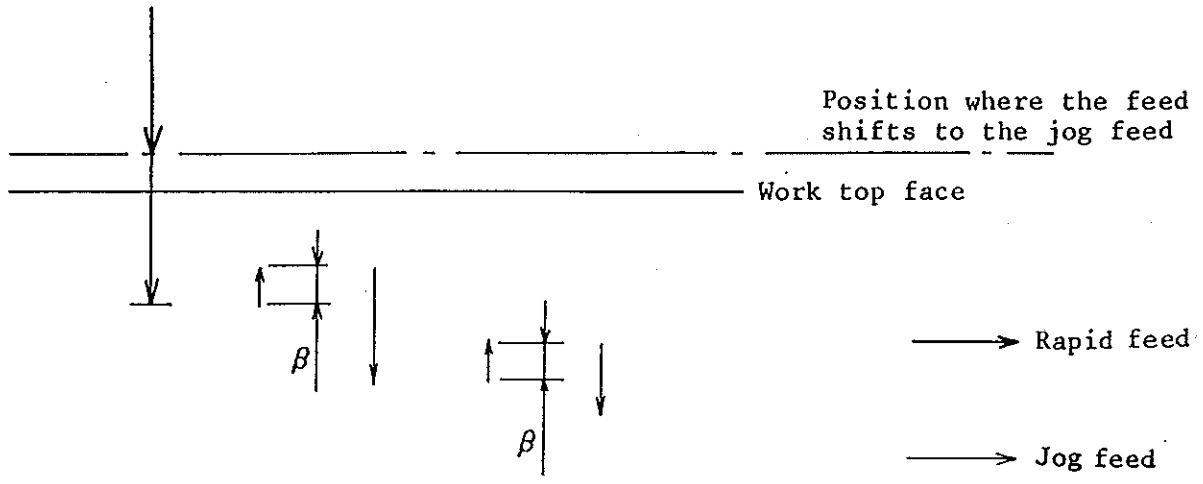
Note 3) Cutting depth at the 15th or after is the same as that at the 14th.

Note 4) When the cutting depth is less than minimum cutting depth (0.01inch), the minimum cutting depth is specified as the cutting depth.

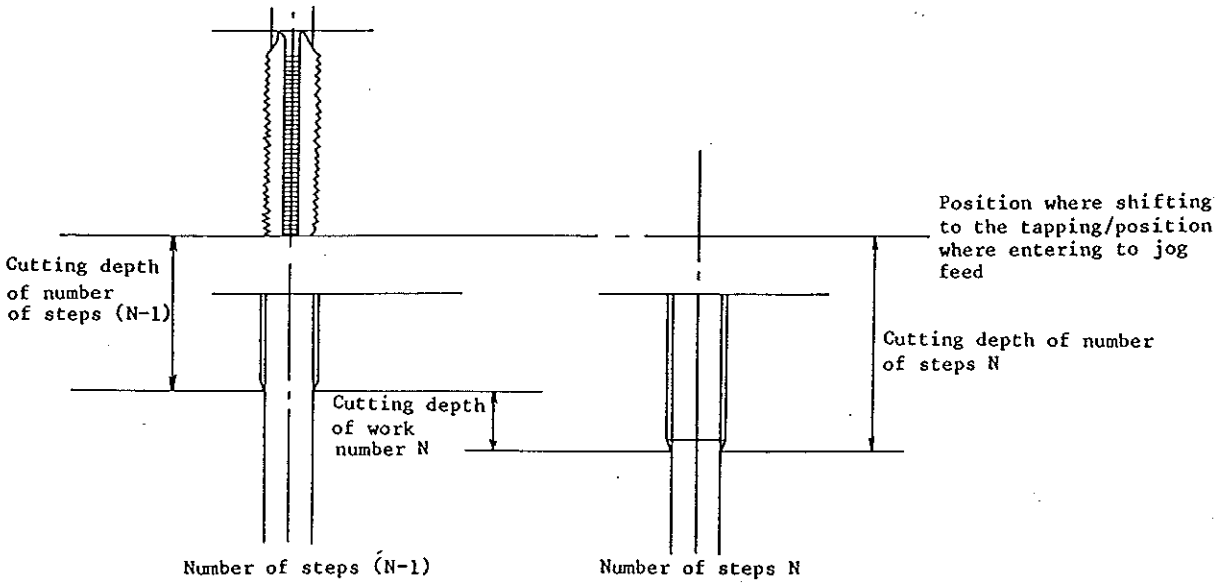
1-1. Drill work (Number of steps) - (Depth coefficient) graph



2. Step of tap work



After the tap step is tapped once, it returns to the tap step feedback in jog feed, then cuts the materials for the cutting depth from the position in jog feed. The tap step repeats these operations.



Cutting depth of the tap step can be calculated by the following formula:

$$[\text{N-th cutting depth (inch)}] = \frac{[\text{cutting depth when the tap step is 100\%}] \times [\text{Nominal diameter (inch)}] \times [\text{Tap step rate (\%)}]}{100}$$

	Step	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15
300% (Aluminum)	Cutting depth coefficient	6	5.1	3.9	3.3	2.4	2.1	1.8	1.2	1.2	0.9	0.6	0.6	0.3	0.3	0.3
	Depth coefficient	6	11.1	15	18.3	20.7	22.8	24.6	25.8	27	27.9	28.5	29.1	29.4	29.7	30
200% (Cast steel)	Cutting depth coefficient	4	3.4	2.6	2.2	1.6	1.4	1.2	0.8	0.8	0.6	0.4	0.4	0.2	0.2	0.2
	Depth coefficient	4	7.4	10	12.2	13.8	15.2	16.4	17.2	18	18.6	19	19.4	19.6	19.8	20
100% (Soft steel)	Cutting depth coefficient	2.0	1.7	1.3	1.1	0.8	0.7	0.6	0.4	0.4	0.3	0.2	0.2	0.1	0.1	0.1
	Depth coefficient	2.0	3.7	5.0	6.1	6.9	7.6	8.2	8.6	9	9.3	9.5	9.7	9.8	9.9	10
80% (Hard steel)	Cutting depth coefficient	1.6	1.4	1.0	0.9	0.6	0.6	0.5	0.3	0.3	0.2	0.2	0.2	0.1	0.1	0.1
	Depth coefficient	1.6	3	4	4.9	5.5	6.1	6.6	6.9	7.2	7.4	7.6	7.8	7.9	8	8.1
70% (Stainless)	Cutting depth coefficient	1.4	1.2	0.9	0.8	0.6	0.5	0.4	0.3	0.3	0.2	0.1	0.1	0.1	0.1	0.1
	Depth coefficient	1.4	2.6	3.5	4.3	4.9	5.4	5.8	6.1	6.4	6.6	6.7	6.8	6.9	7	7.1

Note 1) Value when the tap step rate is 100%.

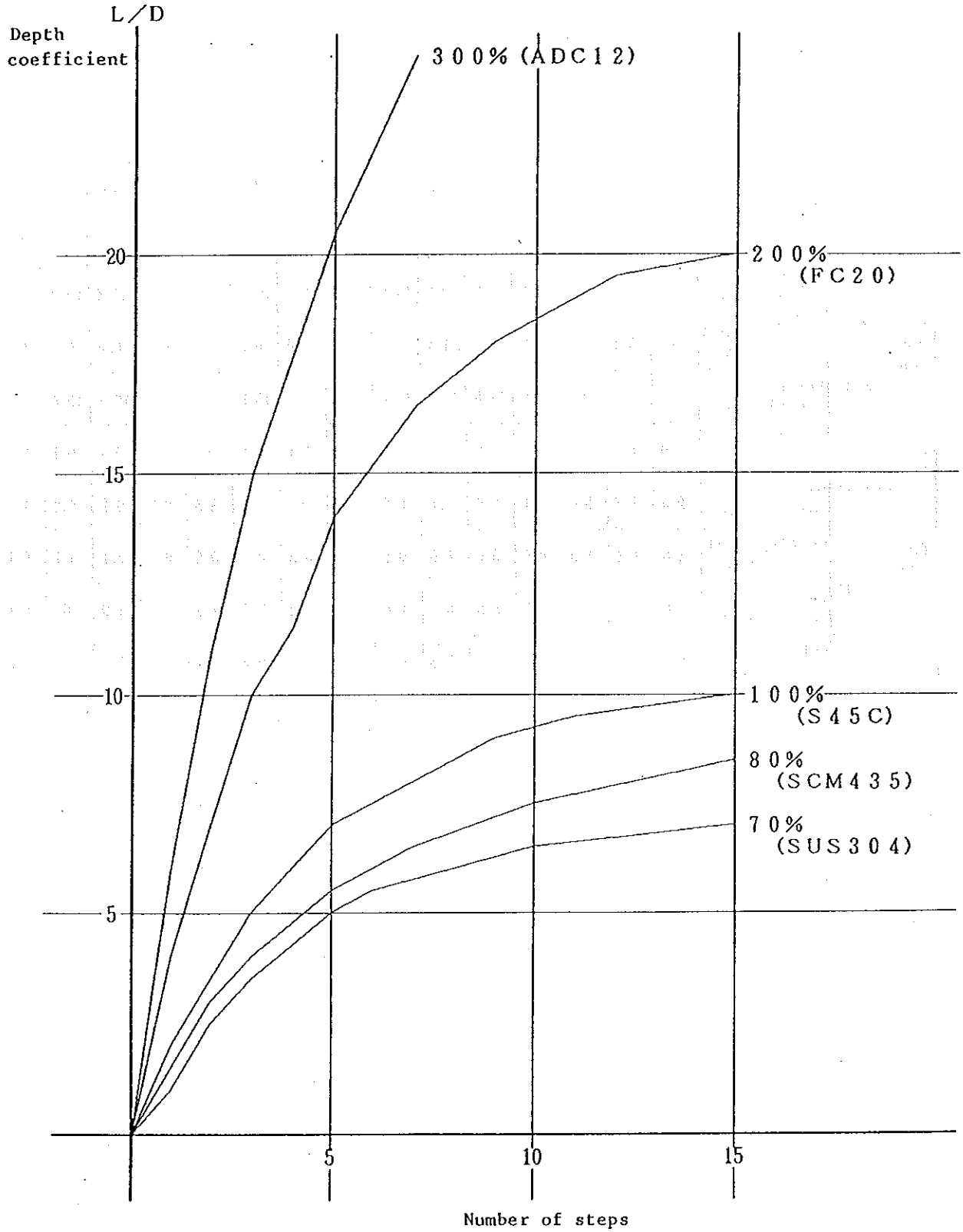
Note 2) (Cutting depth coefficient) = (Cutting depth)/(Nominal diameter)

Note 3) (Depth coefficient) = (Cutting depth)/(Nominal diameter)

Note 4) The cutting depth at the 15th or after is same as that at the 14th.

Note 5) When the cutting depth is less than a minimum cutting depth [(Pitch)/10], the minimum cutting depth is specified as the cutting depth.

2-1. Tap work ((Number of steps) - (Depth coefficient)) graph



Cutting Tool/Condition List of Initial Values

TC-215

(When this machine is delivered to you, cutting tool/condition for each material is set as shown below.)
Considering tools and materials to be used, alter the set value accordingly.

* Set the cutting condition for end mill according to the necessity.

Material S45C SOFT STEEL

Machining Type	Registered Tool Name	Peripheral Velocity ft/min	Feed Amount inch/revolution
Center drill	■ Center drill	■ 39.	■ 0.002
Drill	S ■ Drill L	■ 33. ■ 59.	■ 0.002 ■ 0.004
Tap (BLD)	S ■ S tap L	■ 39. ■ 39.	* *
Tap (THR)	S ■ P tap L	■ 49. ■ 49.	* *
Chamfer tool	■ Chamfer tool	■ 49.	■ 0.004
Counterboring tool	■ Counterboring tool	■ 49.	■ 0.002
Chamfer drill	Hole chamfer ■ Chamfer drill	■ 59. ■ 49.	■ 0.004 ■ 0.004
Stepped drill	Hole counter-boring ■ Stepped drill	■ 59. ■ 49.	■ 0.004 ■ 0.002
Drill tap	Hole tap ■ Drill tap	■ 59. ■ 49.	■ 0.004 *
End mill	Vertical ■ End mill Horizontal	■ 0 ■ 0	■ 0.000 ■ 0.000
Reamer	■ Reamer	■ 20. ■ 39.	■ 0.004 ■ 0.008
Drill reamer	Hole reamer ■ Drill reamer	■ 59. ■ 39.	■ 0.004 ■ 0.008
Step	Drill 100% Tap 100%		

S: Tool diameter is within 0 to 0.1181 inch.
L: Tool diameter is more than 0.1181 inch.

Cutting Tool/Condition List of Initial Values

TC-215

(When this machine is delivered to you, cutting tool/condition for each material is set as shown below.)

Considering tools and materials to be used, alter the set value accordingly.

* Set the cutting condition for end mill according to the necessity.

Material SCM435 HARD STEEL

Machining Type	Registered Tool Name	Peripheral Velocity ft/min	Feed Amount inch/revolution
Center drill	■ Center drill	■ 33.	■ 0.002
Drill	S ■)rill L	■ 23. ■ 46.	■ 0.001 ■ 0.003
Tap (BLD)	S ■S tap L	■ 39. ■ 39.	* *
Tap (THR)	S ■ P tap L	■ 39. ■ 39.	* *
Chamfer tool	■ Chamfer tool	■ 39.	■ 0.003
Counterboring tool	■ Counterboring tool	■ 39.	■ 0.002
Chamfer drill	Hole chamfer ■ Chamfer drill	■ 46. ■ 39.	■ 0.003 ■ 0.003
Stepped drill	Hole counter-boring ■ Stepped drill	■ 46. ■ 39.	■ 0.003 ■ 0.002
Drill tap	Hole tap ■ Drill tap	■ 46. ■ 39.	■ 0.003 *
End mill	Vertical ■ End mill Horizontal	■ 0 ■ 0	■ 0.000 ■ 0.000
Reamer	■ Reamer	■ 16. ■ 33.	■ 0.004 ■ 0.008
Drill reamer	Hole reamer ■ Drill reamer	■ 46. ■ 33.	■ 0.003 ■ 0.008
Step	Drill 80% Tap 80%		

S: Tool diameter is within 0 to 0.1181 inch.
L: Tool diameter is more than 0.1181 inch.

Cutting Tool/Condition TC-215

List of Initial Values (When this machine is delivered to you, cutting tool/condition for each material is set as shown below.)

Considering tools and materials to be used, alter the set value accordingly.

* Set the cutting condition for end mill according to the necessity.

Material FC20 CAST STEEL

Machining Type	Registered Tool Name	Peripheral Velocity ft/min	Feed Amount inch/revolution
Center drill	Center drill	■ 49.	■ 0.003
Drill	S ■ Drill L	■ 43. ■ 82.	■ 0.002 ■ 0.006
Tap (BLD)	S ■ S tap L	■ 49. ■ 49.	■ * ■ *
Tap (THR)	S ■ P tap L	■ 49. ■ 49.	■ * ■ *
Chamfer tool	■ Chamfer tool	■ 49.	■ 0.006
Counterboring tool	■ Counterboring tool	■ 49.	■ 0.004
Chamfer drill	Hole chamfer ■ Chamfer drill	■ 82. ■ 49.	■ 0.006 ■ 0.006
Stepped drill	Hole counter-boring ■ Stepped drill	■ 82. ■ 49.	■ 0.006 ■ 0.004
Drill tap	Hole tap ■ Drill tap	■ 82. ■ 49.	■ 0.006 * *
End mill	Vertical ■ End mill Horizontal	■ 0 ■ 0	■ 0.000 ■ 0.000
Reamer	■ Reamer	■ 26. ■ 49.	■ 0.004 ■ 0.008
Drill reamer	Hole reamer ■ Drill reamer	■ 82. ■ 49.	■ 0.006 ■ 0.008
Step	Drill 200% Tap 200%		

S: Tool diameter is within 0 to 0.1181 inch.
L: Tool diameter is more than 0.1181 inch.

Cutting Tool/Condition TC-215

List of Initial Values (When this machine is delivered to you, cutting tool/condition for each material is set as shown below.)

Considering tools and materials to be used, alter the set value accordingly.

* Set the cutting condition for end mill according to the necessity.

Material ADC12 ALUM

Machining Type	Registered Tool Name	Peripheral Velocity ft/min	Feed Amount inch/revolution
Center drill	Center drill	■ 98.	■ 0.004
Drill	S ■ Drill L	■ 164. ■ 164.	■ 0.004 ■ 0.008
Tap (BLD)	S ■ S tap L	■ 98. ■ 131.	■ * ■ *
Tap (THR)	S ■ P tap L	■ 98. ■ 131.	■ * ■ *
Chamfer tool	■ Chamfer tool	■ 115.	■ 0.012
Counterboring tool	■ Counterboring tool	■ 115.	■ 0.006
Chamfer drill	Hole chamfer ■ Chamfer drill	■ 164. ■ 115.	■ 0.008 ■ 0.012
Stepped drill	Hole counter-boring ■ Stepped drill	■ 164. ■ 115.	■ 0.008 ■ 0.006
Drill tap	Hole tap ■ Drill tap	■ 164. ■ 98.	■ 0.008 *
End mill	Vertical ■ End mill Horizontal	■ 164. ■ 164.	■ 0.004 ■ 0.008
Reamer	■ Reamer	■ 49. ■ 98.	■ 0.004 ■ 0.008
Drill reamer	Hole reamer ■ Drill reamer	■ 164. ■ 98.	■ 0.008 ■ 0.008
Step	Drill 300% Tap 300%		

S: Tool diameter is within 0 to 0.1181 inch.
L: Tool diameter is more than 0.1181 inch.

Cutting Tool/Condition TC-215

List of Initial Values (When this machine is delivered to you, cutting tool/condition for each material is set as shown below.)

Considering tools and materials to be used, alter the set value accordingly.

* Set the cutting condition for end mill according to the necessity.

Material SUS304 STAINLESS

Machining Type	Registered Tool Name	Peripheral Velocity ft/min	Feed Amount inch/revolution
Center drill	■ Center drill	■ 26.	■ 0.002
Drill	S ■ Drill L	■ 20. ■ 20.	■ 0.001 ■ 0.002
Tap (BLD)	S ■ S tap L	■ 20. ■ 20.	* *
Tap (THR)	S ■ P tap L	■ 20. ■ 33.	* *
Chamfer tool	■ Chamfer tool	■ 33.	■ 0.002
Counterboring tool	■ Counterboring tool	■ 33.	■ 0.002
Chamfer drill	Hole chamfer ■ Chamfer drill	■ 33. ■ 33.	■ 0.002 ■ 0.002
Stepped drill	Hole counter-boring ■ Stepped drill	■ 33. ■ 33.	■ 0.002 ■ 0.002
Drill tap	Hole tap ■ Drill tap	■ 33. ■ 20.	■ 0.002 *
End mill	Vertical ■ End mill Horizontal	■ 0 ■ 0	■ 0.000 ■ 0.000
Reamer	■ Reamer	■ 13. ■ 26.	■ 0.004 ■ 0.008
Drill reamer	Hole reamer ■ Drill reamer	■ 33. ■ 26.	■ 0.002 ■ 0.008
Step	Drill 70% Tap 70%		

S: Tool diameter is within 0 to 0.1181 inch.
L: Tool diameter is more than 0.1181 inch.

Cutting Tool/Condition TC-225

List of Initial Values (When this machine is delivered to you, cutting tool/condition for each material is set as shown below.)

Considering tools and materials to be used, alter the set value accordingly.)

Material S45C Soft Steel

Machining Type	Registered Tool Name	Peripheral Velocity ft/min	Feed Amount inch/revolution
Center drill	■ Center drill	■ 39.	■ 0.002
Drill	S ■ Drill L	■ 33. ■ 59.	■ 0.002 ■ 0.004
Tap (BLD)	S ■ S tap L	■ 39. ■ 39.	* *
Tap (THR)	S ■ P tap L	■ 49. ■ 49.	* *
Chamfer tool	■ Chamfer tool	■ 49.	■ 0.004
Counterboring tool	■ Counterboring tool	■ 49.	■ 0.002 ■
Chamfer drill	Hole chamfer ■ Chamfer drill	■ 59. ■ 49.	■ 0.004 ■ 0.004
Stepped drill	Hole counter-boring ■ Stepped drill	■ 59. ■ 49.	■ 0.004 ■ 0.002
Drill tap	Hole tap ■ Drill tap	■ 59. ■ 49.	■ 0.004 *
End mill	Vertical ■ End mill Horizontal	■ 49. ■ 49.	■ 0.002 ■ 0.004
Reamer	■ Reamer	■ 20. ■ 39.	■ 0.004 ■ 0.008
Drill reamer	Hole reamer ■ Drill reamer	■ 59. ■ 39.	■ 0.004 ■ 0.008
Step	Drill 100% Tap 100%		

S: Tool diameter is within 0 to 0.1181 inch.
L: Tool diameter is more than 0.1181 inch.

Cutting Tool/Condition TC-225

(When this machine is delivered to you, cutting tool/condition for each material is set as shown below.)

Considering tools and materials to be used, alter the set value accordingly.

Material SCM435 HARD STEEL

Machining Type		Registered Tool Name	Peripheral Velocity ft/min	Feed Amount inch/revolution
Center drill		■ Center drill	■ 33.	■ 0.002
Drill	S L	■ Drill	■ 23. ■ 46.	■ 0.001 ■ 0.003
Tap (BLD)	S L	■ S tap	■ 39. ■ 39.	* *
Tap (THR)	S L	■ P tap	■ 39. ■ 39.	* *
Chamfer tool		■ Chamfer tool	■ 39.	■ 0.003
Counterboring tool		■ Counterboring tool	■ 39.	■ 0.002
Chamfer drill	Hole chamfer	■ Chamfer drill	■ 46. ■ 39.	■ 0.003 ■ 0.003
Stepped drill	Hole counter-boring	■ Stepped drill	■ 46. ■ 39.	■ 0.003 ■ 0.002
Drill tap	Hole tap	■ Drill tap	■ 46. ■ 39.	■ 0.003 *
End mill	Vertical Horizontal	■ End mill	■ 39. ■ 39.	■ 0.002 ■ 0.003
Reamer		■ Reamer	■ 16. ■ 33.	■ 0.004 ■ 0.008
Drill reamer	Hole reamer	■ Drill reamer	■ 46. ■ 33.	■ 0.003 ■ 0.008
Step		Drill 80% Tap 80%		

S: Tool diameter is within 0 to 0.1181 inch.
L: Tool diameter is more than 0.1181 inch.

Cutting Tool/Condition TC-225

(When this machine is delivered to you, cutting tool/condition for each material is set as shown below.)

Considering tools and materials to be used, alter the set value accordingly.

Material FC20 CAST STEEL

Machining Type	Registered Tool Name	Peripheral Velocity ft/min	Feed Amount inch/revolution
Center drill	■ Center drill	■ 49.	■ 0.003
Drill	S ■ Drill L	■ 43. ■ 82.	■ 0.002 ■ 0.006
Tap (BLD)	S ■ S tap L	■ 49. ■ 49.	* *
Tap (THR)	S ■ P tap L	■ 49. ■ 49.	* *
Chamfer tool	■ Chamfer tool	■ 49.	■ 0.006
Counterboring tool	■ Counterboring tool	■ 49.	■ 0.004
Chamfer drill	Hole chamfer ■ Chamfer drill	■ 82. ■ 49.	■ 0.006 ■ 0.006
Stepped drill	Hole counter-boring ■ Stepped drill	■ 82. ■ 49.	■ 0.006 ■ 0.004
Drill tap	Hole tap ■ Drill tap	■ 82. ■ 49.	■ 0.006 *
End mill	Vertical ■ End mill Horizontal	■ 49. ■ 49.	■ 0.004 ■ 0.006
Reamer	■ Reamer	■ 26. ■ 49.	■ 0.006 ■ 0.008
Drill reamer	Hole reamer ■ Drill reamer	■ 82. ■ 49.	■ 0.004 ■ 0.008
Step	Drill 200% Tap 200%		

S: Tool diameter is within 0 to 0.1181 inch.
L: Tool diameter is more than 0.1181 inch.

Cutting Tool/Condition TC-225

(When this machine is delivered to you, cutting tool/condition for each material is set as shown below.)

Considering tools and materials to be used, alter the set value accordingly.

Material ADC12 ALUMINUM

Machining Type	Registered Tool Name	Peripheral Velocity ft/min	Feed Amount inch/revolution
Center drill	■ Center drill	■ 98.	0.004
Drill	S ■ Drill L	■ 164. ■ 164.	0.004 0.008
Tap (BLD)	S ■ S tap L	■ 98. ■ 131.	* *
Tap (THR)	S ■ P tap L	■ 98. ■ 131.	* *
Chamfer tool	■ Chamfer tool	■ 115.	0.012
Counterboring tool	■ Counterboring tool	■ 115.	0.006
Chamfer drill	Hole chamfer ■ Chamfer drill	■ 164. ■ 115.	0.008 0.012
Stepped drill	Hole counter-boring ■ Stepped drill	■ 164. ■ 115.	0.008 0.006
Drill tap	Hole tap ■ Drill tap	■ 164. 98.	0.008 *
End mill	Vertical ■ End mill Horizontal	164. 164.	0.006 0.008
Reamer	■ Reamer	49. 98.	0.004 0.008
Drill reamer	Hole reamer ■ Drill reamer	164. 98.	0.008 0.008
Step	Drill 300% Tap 300%		

S: Tool diameter is within 0 to 0.1181 inch.
L: Tool diameter is more than 0.1181 inch.

Cutting Tool/Condition TC-225

(When this machine is delivered to you, cutting tool/condition for each material is set as shown below.)

Considering tools and materials to be used, alter the set value accordingly.

Material SUS304 STAINLESS

Machining Type	Registered Tool Name	Peripheral Velocity ft/min	Feed Amount inch/revolution
Center drill	■ Center drill	■ 26.	■ 0.002
Drill	S ■ Drill L	■ 20. ■ 33.	■ 0.001 ■ 0.002
Tap (BLD)	S ■ S tap L	■ 20. ■ 20.	* *
Tap (THR)	S ■ P tap L	■ 20. ■ 20.	* *
Chamfer tool	■ Chamfer tool	■ 33.	■ 0.002
Counterboring tool	■ Counterboring tool	■ 33.	■ 0.002
Chamfer drill	Hole chamfer ■ Chamfer drill	■ 33. ■ 33.	■ 0.002 ■ 0.002
Stepped drill	Hole counter-boring ■ Stepped drill	■ 33. ■ 33.	■ 0.002 ■ 0.002
Drill tap	Hole tap ■ Drill tap	■ 33. ■ 20.	■ 0.002 *
End mill	Vertical ■ End mill Horizontal	■ 33. ■ 26.	■ 0.002 ■ 0.002
Reamer	■ Reamer	■ 13. ■ 26.	■ 0.004 ■ 0.008
Drill reamer	Hole reamer ■ Drill reamer	■ 33. ■ 33.	■ 0.002 ■ 0.008
Step	Drill 70% Tap 70%		

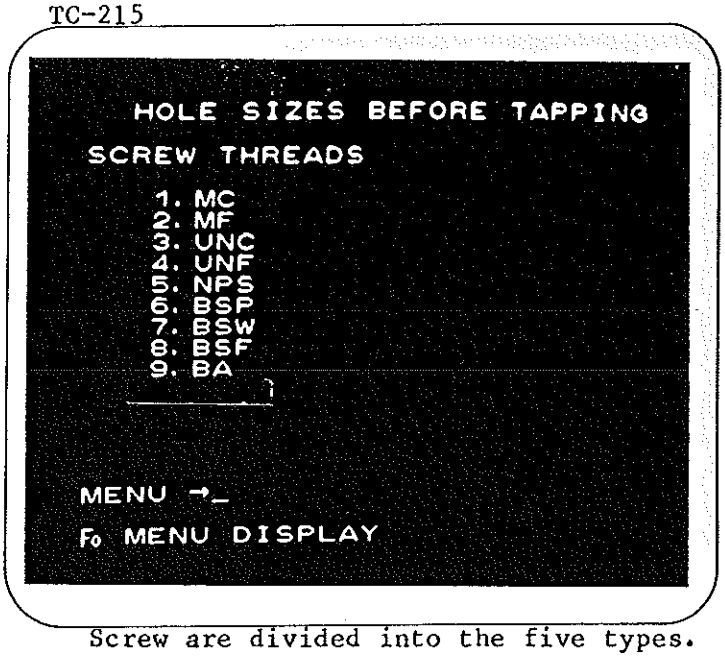
S: Tool diameter is within 0 to 0.1181 inch.
L: Tool diameter is more than 0.1181 inch.

7-5-3. Hole Sizes before Tapping

Hole sizes before tapping are displayed on the screen below. These hole sizes are set divided into screw types, nominal sizes, diameters, pitches, and threads.

Press the [CUT COND] key and press the [3] and [ENTER] Keys in the menu to set the hole sizes before tapping.

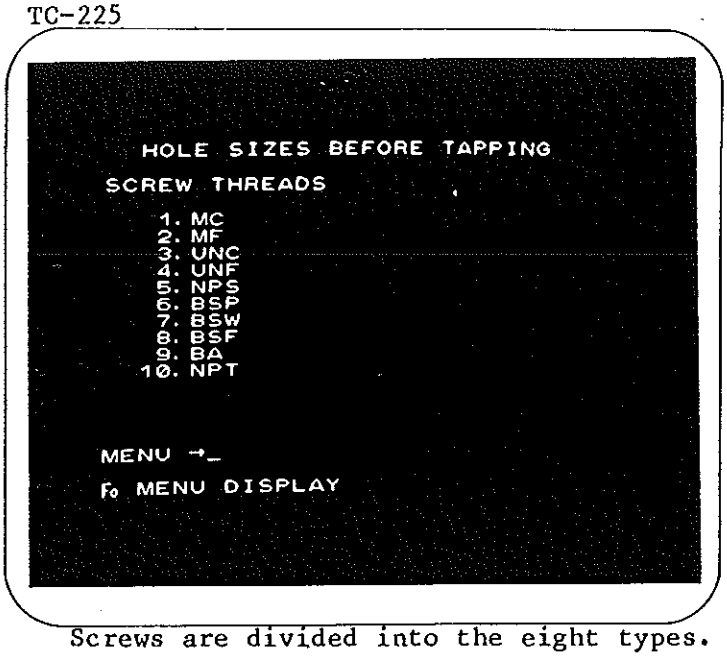
TC-215



* The screw menu cannot be changed.

Screw are divided into the five types.

TC-225



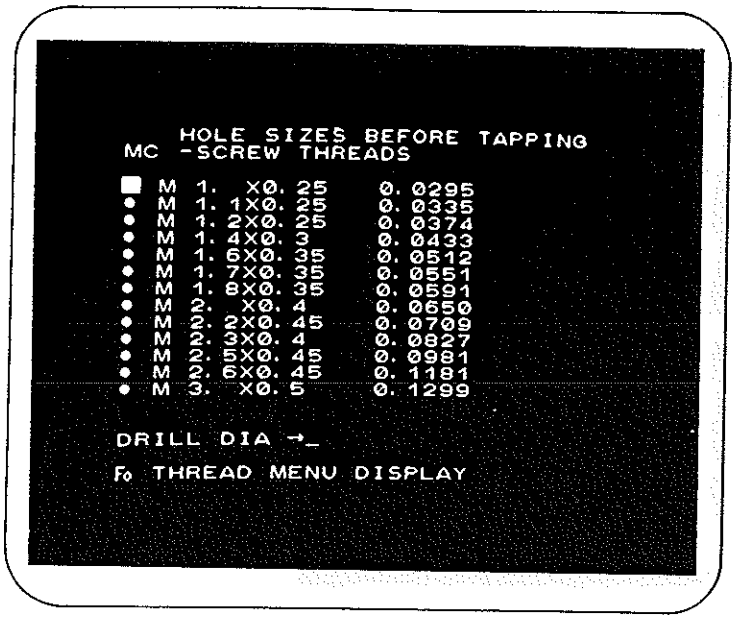
* The screw menu cannot be changed.

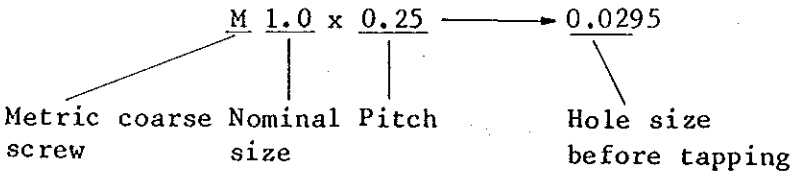
Screws are divided into the eight types.

Select and key in menu No. of required screw thread out of the menu screen, hole sizes before tapping are displayed.

Set the screw menu No.

Key in [1] , and set the hole sizes before tapping for the metric coarse screw.





When the above screws are processed, 0.0295 inch drill in the tool menu is used.

In actual operation, some diameter allowance is provided to holes for tool holder installation, tool vibration, and tool shape. The above display data may be used for controlling tool diameter hole diameter at your end.

Hole Sizes before Tapping TC215

List of initial values (When this machine is delivered to you, the sizes are divided into screw types and set as shown below.)

Considering tools, materials, etc. to be used, alter these values accordingly.

Metric screw threads

1. Metric coarse screw threads

Nominal size (Pitch)	Hole size before tapping
M 1.0×0.25	0.0295
M 1.1×0.25	0.0335
M 1.2×0.25	0.0374
M 1.4×0.3	0.0433
M 1.6×0.35	0.0512
M 1.7×0.35	0.0551
M 1.8×0.35	0.0591
M 2.0×0.4	0.0650
M 2.2×0.45	0.0709
M 2.5×0.45	0.0827
M 3.0×0.5	0.0981
M 3.5×0.6	0.1181
M 4.0×0.7	0.1299
M 4.5×0.75	0.1496
M 5.0×0.8	0.1693
M 6.0×1.0	0.2008
M 7.0×1.0	0.2402
M 8.0×1.25	0.2677
M 9.0×1.25	0.3071
M10.0×1.5	0.3386
M12.0×1.75	0.4130
M14.0×2.0	0.4844
M16.0×2.0	0.5625

2. Metric fine screw threads

Nominal size (Pitch)	Hole size before tapping
m 1.0×0.2	0.0315
m 1.1×0.2	0.0354
m 1.2×0.2	0.0394
m 1.4×0.2	0.0472
m 1.6×0.2	0.0551
m 1.8×0.2	0.0630
m 2.0×0.25	0.0709
m 2.2×0.25	0.0787
m 2.5×0.35	0.0866
m 3.0×0.35	0.1063
m 3.5×0.35	0.1260
m 4.0×0.5	0.1378
m 4.5×0.5	0.1575
m 5.0×0.5	0.1772
m 5.5×0.5	0.1969
m 6.0×0.75	0.2087
m 7.0×0.75	0.2480
m 8.0×1.0	0.2795
m 8.0×0.75	0.2874
m 9.0×1.0	0.3189
m 9.0×0.75	0.3268
m10.0×1.25	0.3465
m10.0×1.0	0.3583
m10.0×0.75	0.3661
m12.0×1.5	0.4219
m12.0×1.25	0.4331
m12.0×1.0	0.4375
m14.0×1.5	0.5000
m14.0×1.0	0.5156
m15.0×1.5	0.5469
m15.0×1.0	0.5625
m16.0×1.5	0.5781
m16.0×1.0	0.5938

3. Unified coarse screw threads

Nominal size	(No. of threads)	Hole size before tapping
UNC	1－64	0.0610
UNC	2－56	0.0728
UNC	3－48	0.0827
UNC	4－40	0.0906
UNC	5－40	0.1024
UNC	6－32	0.1102
UNC	8－32	0.1378
UNC	10－24	0.1535
UNC	12－24	0.1772
UNC	1／4－20	0.2047
UNC	5／16－18	0.2638
UNC	3／8－16	0.3189
UNC	7／16－14	0.3740
UNC	1／2－13	0.4331
UNC	9／16－12	0.4844
UNC	5／8－11	0.5469

4. Unified fine screw threads

Nominal size	(No. of threads)	Hole size before tapping
UNF	0－80	0.0512
UNF	1－72	0.0630
UNF	2－64	0.0748
UNF	3－56	0.0827
UNF	4－48	0.0945
UNF	5－44	0.1063
UNF	6－40	0.1181
UNF	8－36	0.1378
UNF	10－32	0.1614
UNF	12－28	0.1850
UNF	1／4－28	0.2165
UNF	5／16－24	0.2756
UNF	3／8－24	0.3386
UNF	7／16－20	0.3970
UNF	1／2－20	0.4688
UNF	9／16－18	0.5118
UNF	5／8－18	0.5781

5. National Pipe Thread Standard

Nominal size	Thread	Hole size before tapping
NPS	1/16-27	0.2610
NPS	1/8-27	0.3480
NPS	1/4-18	0.4531
NPS	3/8-18	0.5937

6. British Standard Pipe Thread

Nominal size	Thread	Hole size before tapping
BSP	1/16-28	0.2638
BSP	1/8-28	0.3465
BSP	1/4-19	0.4688
BSP	3/8-19	0.6094

7. British Standard Whitworth

Nominal size	Thread	Hole size before tapping
BSW	1/8-40	0.1024
BSW	3/16-24	0.1457
BSW	1/4-20	0.2008
BSW	5/16-18	0.2598
BSW	3/8-16	0.3150
BSW	7/16-14	0.3701
BSW	1/2-12	0.4219
BSW	9/16-12	0.4844
BSW	5/8-11	0.5469

8. British Standard Fine Thread

Nominal size	Thread	Hole size before tapping
BSF	3/16-32	0.1575
BSF	7/32-28	0.1850
BSF	1/4-26	0.2126
BSF	9/32-26	0.2441
BSF	5/16-22	0.2677
BSF	3/8-20	0.3268
BSF	7/16-18	0.3858
BSF	1/2-16	0.4375
BSF	9/16-16	0.5118
BSF	5/8-14	0.5625

9. British Association

Nominal size	Thread	Hole size before tapping
BA	0 x 1.000	0.2008
BA	1 x 0.900	0.1772
BA	2 x 0.810	0.1575
BA	3 x 0.730	0.1378
BA	4 x 0.660	0.1220
BA	5 x 0.590	0.1063
BA	6 x 0.530	0.0945
BA	7 x 0.480	0.0827
BA	8 x 0.430	0.0709
BA	9 x 0.390	0.0630
BA	10 x 0.350	0.0551
BA	11 x 0.310	0.0472
BA	12 x 0.280	0.0433
BA	13 x 0.250	0.0394
BA	14 x 0.230	0.0315

Hole sizes before tapping　TC-225

List of initial values　(when this machine is delivered to you, the hole sizes are divided into the screw types and are set as shown below.)

Considering tools, materials, etc. to be used, alter the set values accordingly.

1. Metric coarse screw threads

Nominal (Pitch) size	Hole size before tapping
M 1.0x0.25	0.0295
M 1.1x0.25	0.0335
M 1.2x0.25	0.0374
M 1.4x0.3	0.0433
M 1.6x0.35	0.0512
M 1.7x0.35	0.0551
M 1.8x0.35	0.0591
M 2.0x0.4	0.0650
M 2.2x0.45	0.0709
M 2.3x0.4	0.0787
M 2.5x0.45	0.0827
M 2.6x0.45	0.0866
M 3.0x0.5	0.0984
M 3.5x0.6	0.1181
M 4.0x0.7	0.1299
M 4.5x0.75	0.1496
M 5.0x0.8	0.1693
M 6.0x1.0	0.2008
M 7.0x1.0	0.2402
M 8.0x1.25	0.2677
M 9.0x1.25	0.3071
M10.0x1.5	0.3386
M12.0x1.75	0.4130
M14.0x2.0	0.4844
M16.0x2.0	0.5625
M18.0x2.5	0.6181

2. Metric fine screw threads

Nominal (Pitch) size	Hole size before tapping
m 1.0×0.2	0.0315
m 1.1×0.2	0.0354
m 1.2×0.2	0.0394
m 1.4×0.2	0.0472
m 1.6×0.2	0.0551
m 1.8×0.2	0.0630
m 2.0×0.25	0.0709
m 2.2×0.25	0.0787
m 2.5×0.35	0.0866
m 3.0×0.35	0.1063
m 3.5×0.35	0.1260
m 4.0×0.5	0.1378
m 4.5×0.5	0.1575
m 5.0×0.5	0.1772
m 5.5×0.5	0.1969
m 6.0×0.75	0.2087
m 7.0×0.75	0.2480
m 8.0×1.0	0.2795
m 8.0×0.75	0.2874
m 9.0×1.0	0.3189
m 9.0×0.75	0.3268
m10.0×1.25	0.3465
m10.0×1.0	0.3583
m10.0×0.75	0.3661
m12.0×1.5	0.4219
m12.0×1.25	0.4331
m12.0×1.0	0.4375
m14.0×1.5	0.5000
m14.0×1.0	0.5156
m15.0×1.5	0.5469
m15.0×1.0	0.5625
m16.0×1.5	0.5781
m16.0×1.0	0.5938
m17.0×1.5	
m17.0×1.0	
m18.0×2.0	
m18.0×1.5	
m18.0×1.0	

3. Unified coarse screw threads

Nominal size (No. of threads)	Hole size before tapping
UNC 1-64	0.0610
UNC 2-56	0.0728
UNC 3-48	0.0827
UNC 4-40	0.0906
UNC 5-40	0.1024
UNC 6-32	0.1102
UNC 8-32	0.1378
UNC 10-24	0.1535
UNC 12-24	0.1772
UNC 1/4-20	0.2047
UNC 5/16-18	0.2638
UNC 3/8-16	0.3189
UNC 7/16-14	0.3740
UNC 1/2-13	0.4331
UNC 9/16-12	0.4844
UNC 5/8-11	0.5469

4. Unified fine screw threads

Nominal size (No. of threads)	Hole size before tapping
UNF 0-80	0.0512
UNF 1-72	0.0630
UNF 2-64	0.0748
UNF 3-56	0.0827
UNF 4-48	0.0945
UNF 5-44	0.1063
UNF 6-40	0.1181
UNF 8-36	0.1378
UNF 10-32	0.1614
UNF 12-28	0.1850
UNF 1/4-28	0.2165
UNF 5/16-24	0.2756
UNF 3/8-24	0.3386
UNF 7/16-20	0.3970
UNF 1/2-20	0.4688
UNF 9/16-18	0.5118
UNF 5/8-18	0.5781

5. National Pipe Thread Standard

Nominal size	Thread	Hole size before tapping
NPS	1/16-27	0.2610
NPS	1/8-27	0.3480
NPS	1/4-18	0.4531
NPS	3/8-18	0.5937

6. British Standard Pipe Thread

Nominal size	Thread	Hole size before tapping
BSP	1/16-28	0.2638
BSP	1/8-28	0.3465
BSP	1/4-19	0.4688
BSP	3/8-19	0.6094

7. British Standard Whitworth

Nominal size	Thread	Hole size before tapping
BSW	1/8-40	0.1024
BSW	3/16-24	0.1457
BSW	1/4-20	0.2008
BSW	5/16-18	0.2598
BSW	3/8-16	0.3150
BSW	7/16-14	0.3701
BSW	1/2-12	0.4219
BSW	9/16-12	0.4844
BSW	5/8-11	0.5469
BSW	11/16-11	0.5945

8. British Standard Fine Thread

Nominal size	Thread	Hole size before tapping
BSF	3/16-32	0.1575
BSF	7/32-28	0.1850
BSF	1/4-26	0.2126
BSF	9/32-26	0.2441
BSF	5/16-22	0.2677
BSF	3/8-20	0.3268
BSF	7/16-18	0.3858
BSF	1/2-16	0.4375
BSF	9/16-16	0.5118
BSF	5/8-14	0.5625
BSF	11/16-14	0.6250

9. British Association

Nominal size	Thread	Hole size before tapping
BA	0 x 1.000	0.2008
BA	1 x 0.900	0.1772
BA	2 x 0.810	0.1575
BA	3 x 0.730	0.1378
BA	4 x 0.660	0.1220
BA	5 x 0.590	0.1063
BA	6 x 0.530	0.0945
BA	7 x 0.480	0.0827
BA	8 x 0.430	0.0709
BA	9 x 0.390	0.0630
BA	10 x 0.350	0.0551
BA	11 x 0.310	0.0472
BA	12 x 0.280	0.0433
BA	13 x 0.250	0.0394
BA	14 x 0.230	0.0315

10. American Pipe Taper Thread

Nominal size	Thread	Hole size before tapping
NPT	1/16-27	0.2500
NPT	1/8-27	0.3390
* NPT	1/4-18	0.4531
* NPT	3/8-18	0.5781

* Since the machining of taper threads marked with an asterisk is heavy-duty cutting, alter the cutting conditions including the cutting speed and step frequency as necessary.

Data can be changed in the following procedure.

* Nominal sizes of screws cannot be changed.

In the key-in run mode, set the memory rewrite switch to on.
Press the [CUT COND] key and [3] key to set hole sizes before trapping. Select and set the number from the screw menu.
Display the change screen and flash the cursor on the position to be changed by using the [⇧ PAGE], [⇩ PAGE], [⇧ CURSOR] and [⇩ CURSOR] key.
Set the change value of hole sizes before tapping.
Press the [DELET] key to delete the data.
After the change is completed, set the memory rewrite switch to off.

7-5-4. Change of Cutting Tool in Memory Run

When memory data for the cutting tool condition is change, the data, in general, must be changed before the memory run. (However, when the cutting condition during operation and exclusive cutting condition set in the program edit are changed, press the RESET key, then change them.)

Press the HOLD key to stop the memory run.
Check the current process, tools, work position, and work number.
Press the RESET key to change the mode to the key-in run mode.
Change the necessary data. For procedure to change data, refer to the change operation of each cutting condition.
Return the mode to the memory run, enter the program number, return the machine to the start point (refer to 9, Memory Run), then restart the machine by pressing the START key.

7-5-5. Collective Deletion of Cutting Tool Conditions

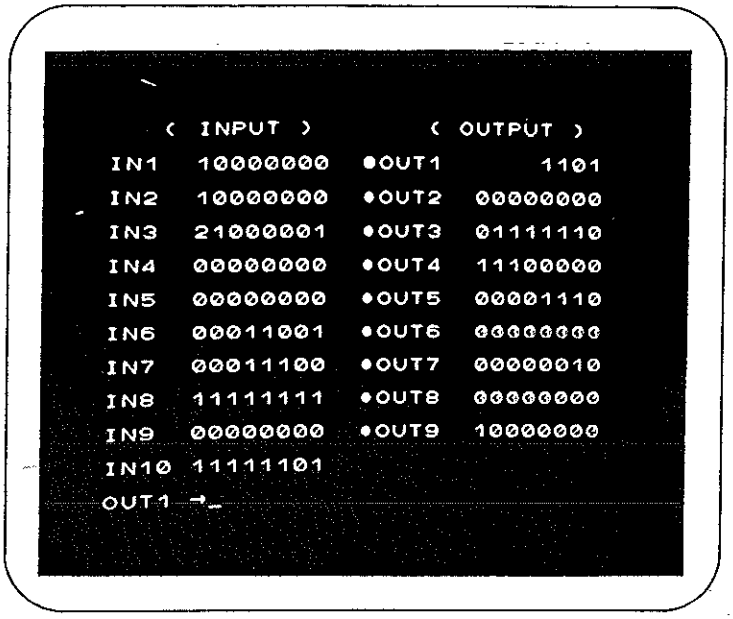
Various cutting conditions set in key-in run mode are usually deleted by the use of the DELET key. Collective deletion, however, can be achieved by the use of the following key operation:
Press the MDI key to set the memory switch to on.
Press the PARAM key, then press 2 key to set the system.
Display the last page by pressing the ⇩ PAGE key, and set the cursor in the data clear.

9293: All data deletion in tool menu, magazine tool, tool pattern, hole sizes before tapping, and cutting tool condition
9294: Deletion of tool menu data
9295: Deletion of tool pattern data
9296: Deletion of data on hole sizes before tapping
9297: Deletion of data on cutting tool/condition

Set the above 4-digit number to the parameter then push the ENTER key.

7-6. I/O Display

Press the [%] key, and the following screen will be displayed. (The input side on this screen is just displayed but cannot be altered.)



I/O can be displayed for each bit (8 to 1) of input and output sides, which allows you to grasp the machine condition for maintenance and repair.

On the above screen, bit with 1 included indicates signal entered, and bit with 0 included indicates no signal entered.
To those items marked with (*), this is reversely applied.
Besides, input bits 3 to 8 at the input side are displayed not in 1 but in 2. (Refer to the display contents.)

For every bit on the above screen, the following tables show the display contents.

I/O display contents

Input \ Bit	8	7	6	5	4	3	2	1
1	Memory protect enable	External signal end	Stop key	Start key	Supply voltage abnormal	Master CPU abnormal	Coolant thermal abnormal	Emergency stop
2	CRT vertical synchronize	Battery abnormal	Abnormal signal	External zero return start	Program select 8	Program select 4	Program select 2	Program select 1
3	Slave CPU RUN	Slave CPU READY	Slave CPU STOP	Slave CPU START	Reset key 1	External reset	Temperature abnormal	Spindle temperature abnormal
4	CPU abnormal	CPU abnormal abnormal	Z axis - overrun	Z axis + overrun	Y axis - overrun	Y axis + overrun	X axis - overrun	X axis + overrun
5	Spindle servo abnormal	Z axis servo abnormal	Y axis servo abnormal	X axis servo abnormal	Z axis zero LS	Y axis zero LS	X axis zero LS	Tool broken detection
6	Encoder signal 8	Encoder signal 4	Encoder signal 2	Encoder signal 1	Tool decelerate position LS	Z axis ATC area LS	Z axis ATC zero LS	
7			Indexing-error signal	Release key*	Spindle zero signal	Z axis zero signal	Y axis zero signal	X axis zero signal
8	CME	TDRA	RDA	SCE	MEMON	WPRT	ERROR	BUSY
9								
10							SW2	SW1

Display contents of SW1 and SW2 depend on the maximum rotation rate at the time of tapping.

1. Output display contents

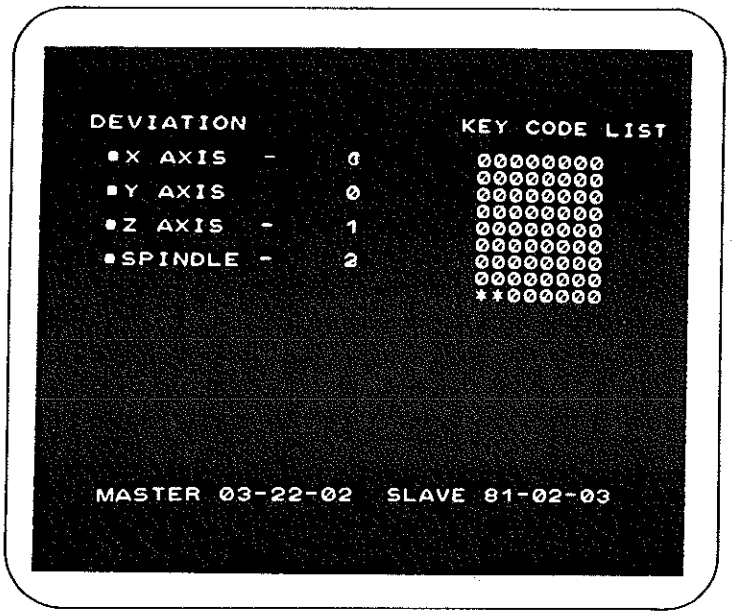
Bit / Input	8	7	6	5	4	3	2	1
1					Emergency stop F/F reset*	Disable*	Time aboard ON	CPU monitor timer ON
2	External signal output 28	External signal output 24	External signal output 22	External signal output 21	External signal output 18	External signal output 14	External signal output 12	External signal output 11
3	CRT extended output	CRT mask output*	Z axis zero complete monitor	Y axis zero complete monitor	X axis zero complete monitor	Servo ON monitor	Automatic operation	External signal output latch
4	Servo preparation end	Servo preparation end	Memory protect ON/OFF	Abnormal output	Cycle end	Coolant ON	Program stop	Automatic operation in progress
5				Spindle servo ON	XYZ servo ON	Disable*	Time aboard timer set	
6	Speed command 2 15	Speed command 2 14	Speed command 2 13	Speed command 2 12	Speed command 2 11	Speed command 2 10	Speed command 2 9	Speed command 2 8
7	Spindle zero detection enable	Z axis zero detection enable	Y axis zero detection enable	X axis zero detection enable	Air blast valve ON	ATC decelera-tion command	ATC CW/CCW	ATC start stop
8	Speed command 2 7	Speed command 2 6	Speed command 2 5	Speed command 2 4	Speed command 2 3	Speed command 2 2	Speed command 2 1	Speed command 2 0
9	Sampling		i REF sampling		Spindle sampling	Z axis sampling	Y axis sampling	X axis sampling

Press the [!/0] key for screen control, then press the [⇩ PAGE] key, and the below screen will be displayed.

This screen allows you to grasp the position deviation value of the X, Y and Z axes and spindle.

The right-hand display provides confirmation of the movement of each key on the operation panel.

The bits are usually set to 0, and when a key is moved, the corresponding bits are set to 1.



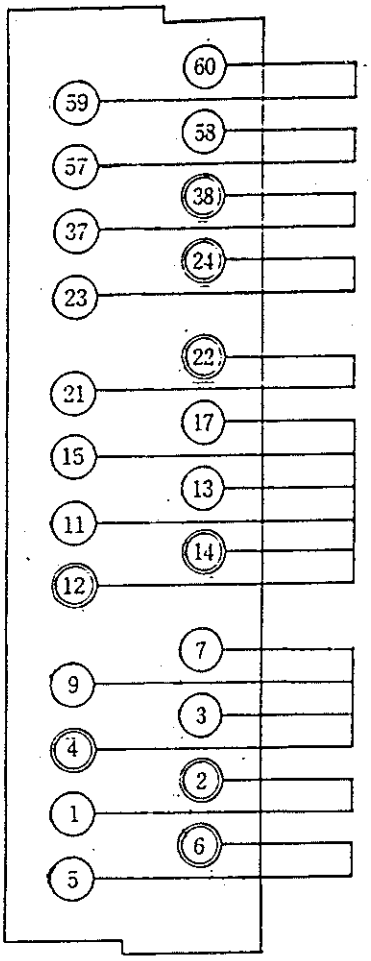
The following table shows the display contents of the key check.

	0	1	2	3	4	5	6	7
0	Figure key 0	Figure key 1	Figure key 2	Figure key 3	Figure key 4	Figure key 5	Figure key 6	Figure key 7
1	Figure key 8	Figure key 9		—	/	Insertion	Deletion	Can
2					Too offset	Tool Check Invid	Cycle run	Dry run
3		Zero work set	Rapid feed	Jog feed	Step	Tool check	Single RUN	ENTER
4	△	▽						
5	M.Z.RT	W.Z.RT	ATC	S.CW	S.CCW	S.STOP		
6	Fo		↓ CURSOR	↑ CURSOR		↓ PAGE		
7							I/O	
8			+ X	- X	+ Y	- Y	+ Z	-Z

7-6-1. Ddescription of External Input

1. TC-215

External input terminal

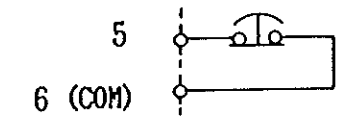
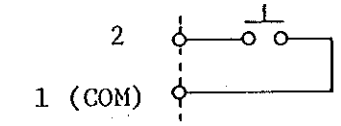
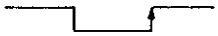
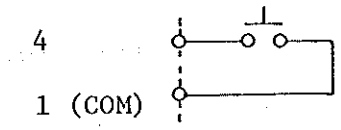
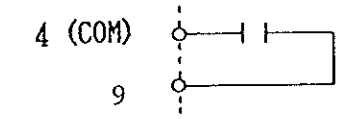
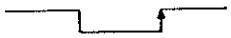
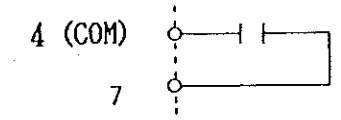
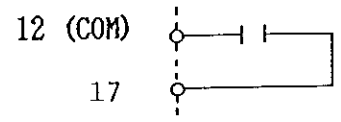
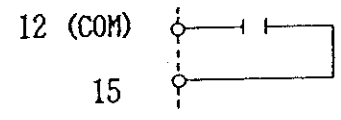
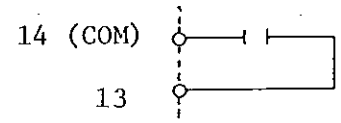


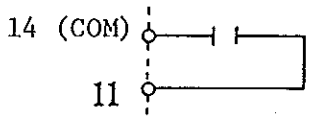
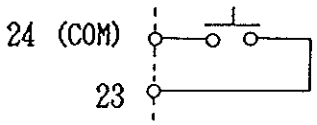
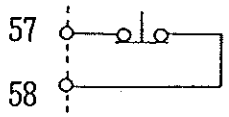
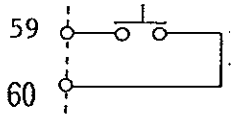
ON	External power on	
OFF	External power off△	
COM		
IN10	Tool broken detector△	
COM	External COM	
EXRESET	External use	
COM	Internal use	
EXTEMP		
PRO4	External program select	8
PRO3	"	4
PRO2	"	2
PRO1	"	1
COM	External program select	COM
COM	"	COM
MFIN	External signal finish	
EXORG	External machine zero return	
EXSTOP	External stop	
COM	External COM	
EXST	External start	
EMSA	External emergency stop△	

* Connect the no-voltage contact to each terminal. Note that the internal circuit is broken if 100V AC is applied to it.

Notes:
1. The external input terminals marked with "△" are respectively shortcircuited as per the standard specification.
2. The terminals respectively surrounded by ◎ are terminals for the external COM (P24V).
3. Use the crimp-style terminal TB1.25-9-1 (NICHIFU) for the connection with the terminal block.

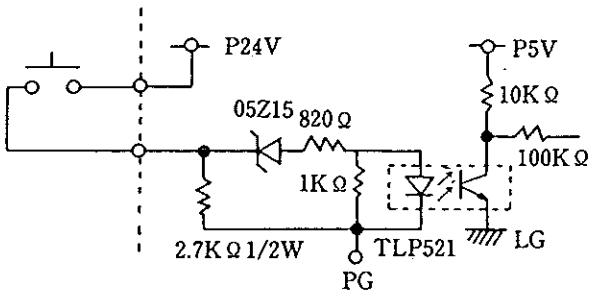
Description of External Input

Signal name	Circuit	Function
External emergency stop ((EMSA)		This button is used for emergency stop. Use external reset for resetting. Contact is of normal close type.
External start (EXST)		This button is used for starting memory run and key-in run, restarting them after stop, or restarting single run. Press the external start button and detach your finger, and the button will function. The timing is: 
External stop (EXSTOP)		This signal is used for decelerate and stop feeding during automatic operation. Use external start for restarting. This is not effective in ATC and tapping cycle. External stop functions upon pressing this button.
External zero return start (WXORG)		This signal is used for zero position return under external command. The machine is placed in the memory run mode. Timing is: 
External signal end		This is an end signal for external signal output. For timing, refer to the description of external output terminal.
External program select 8 (Pro8)		In memory run mode, this signal is used for selecting a program among established program numbers 1 to 15. Make this selection when external start
External program select 4 (Pro4)		becomes effective. In the event of no selection, or the selection of No.0, internally selected program will be executed. Use binary code for program selection.
External program select 2 (Pro2)		

Signal name	Circuit	Function
External program select		In memory run mode, this signal is used for selecting a program among established program numbers 1 to 15. Make this selection when external start becomes effective. In the event of no selection, or the selection of No.0, internally selected program will be executed. Use binary code for program selection.
External reset (EXRESET)		This signal provides the same function as NC reset on NC control panel, and used for reset when external emergency stop functions are executed. This signal acts when this terminal button is pressed.
External power OFF (OFF)		This signal allows switching off of NC power from outside. Contact is of normal close type.
External power ON (ON)		This signal allows switching ON of NC power from outside. Contact is of normal open type.

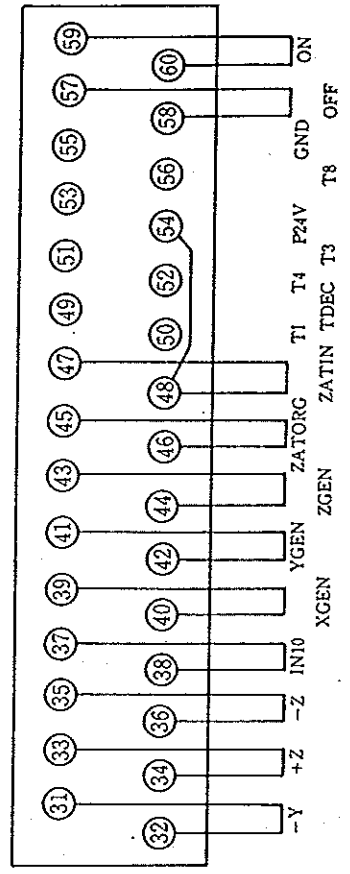
Note 1: Best use of external power ON/OFF, external machine zero return external start, external program select and external signal output allows factory automation or labor saving.

Note 2: Even number on input terminal is commonly used with P24V loaded. Their input circuit is as shown in the diagram.



2. TC-225

External input terminal



External power ON

External power OFF

Internal use

Internal use

Internal use

Internal use

Internal use

Internal use

Internal use

Internal use

Internal use

Internal use

Internal use

Tool broken detection△

Internal use

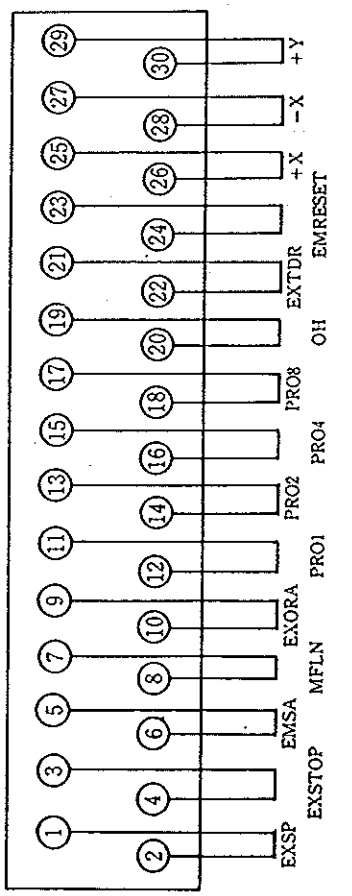
Internal use

Internal use

Apply no-voltage contact to each terminal.

Applying a high voltage such as 100V AC will cause internal breakdown.

The external input terminals marked with △ are respectively short-circuited as per the standard specification.



Internal use

Internal use

Internal use

External reset

Internal use

Internal use

External program select 8

External program select 4

External program select 2

External program select 1

External machine zero retrun start

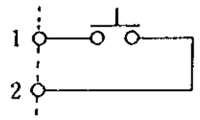
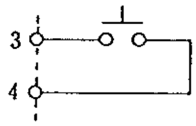
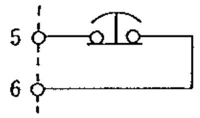
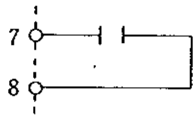
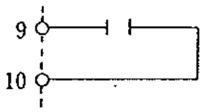
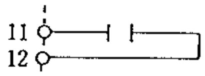
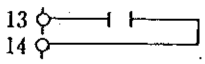
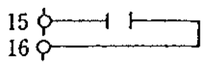
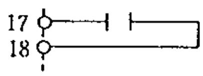
External signal finish

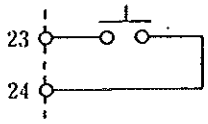
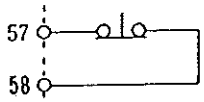
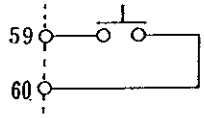
External emergency stop △

External stop

External start

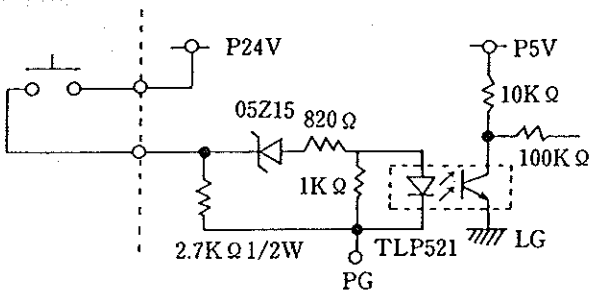
Description of External Input

Signal name	Circuit	Function
External start (EXST)		This button is used for starting memory run and , restarting them after stop, or restarting single run. Press the external start button and detach your finger, and the button will function. The timing is:
External emergency stop (WXSTOP)		This signal is used for decelerate and stop feeding during automatic operation. Use external start for restarting.
External emergency stop (EMSA)		This button is used for emergency stop. Use external reset for resetting. Contact is of normal close type.
External signal end (MFIN)		This is an end signal for external signal output. For timing, refer to the description of external output terminal.
External machine zero return start (EXORG)		This signal is used for machine zero position return under external command. Memory run mode is used. Timing is:
External program select 1 (Pro1)		In memory run mode, this signal is used for selecting a program among established program numbers 1 to 15. Make this selection when external start
External program select 2 (Pro2)		becomes effective. In the event of no selection, or the selection of No.0, internally selected program will be executed. Use binary code for program selection.
External program select 4 (Pro4)		
External program select 8 (Pro8)		

Signal name	Circuit	Function
External reset (EXRESET)		This signal provides the same function as NC reset on NC control panel, and used for reset when external emergency stop functions are executed. This signal acts when this terminal button is pressed.
External power OFF (OFF)		This signal allows switching off of NC power from outside. Contact is of normal close type.
External power ON (ON)		This signal allows switching ON of NC power from outside. Contact is of normal open type.

Note 1: Best use of external power ON/OFF, external machine zero return external start, external program select and external signal output allows factory automation or labor saving.

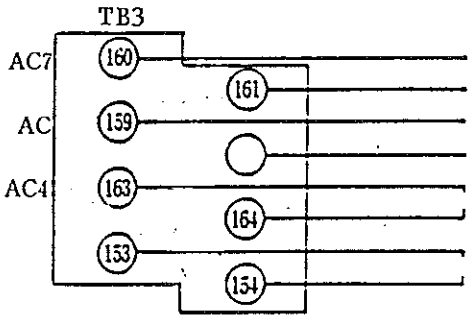
Note 2: Even number on input terminal is commonly used with P24V loaded. Their input circuit is as shown in the diagram.



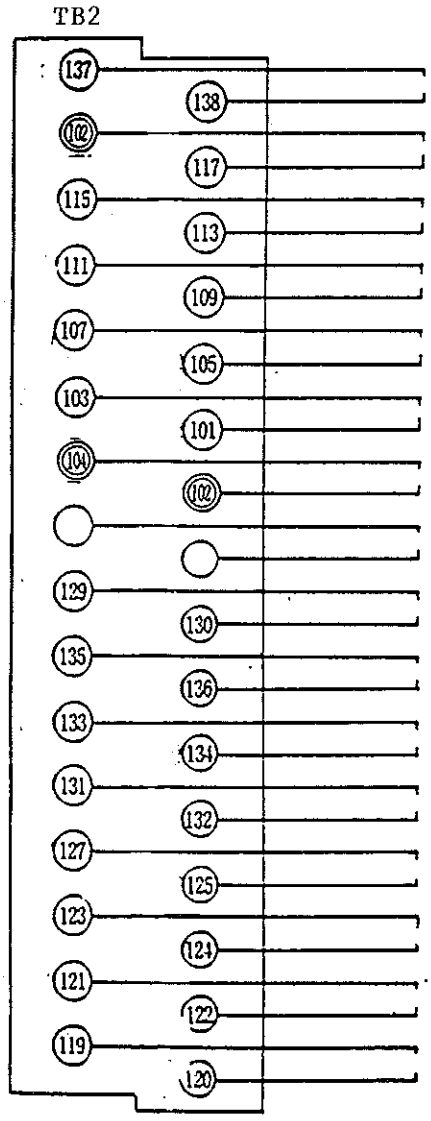
7-6-2. Description of External Output

1. TC-215

External output terminal



AC 0V
AC 24V 1.0A for spotlight
AC100V 1.0A
NC0 Unused
AC100V (AIR) for air valve (50VA)
AC100V for coolant valve (50VA)

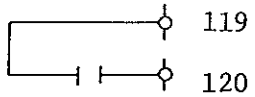
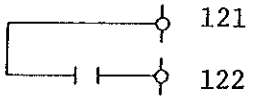
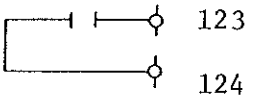
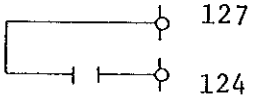
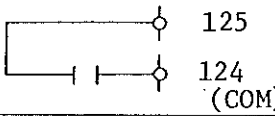
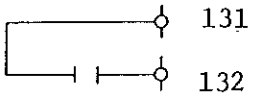
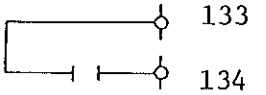
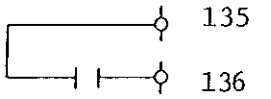
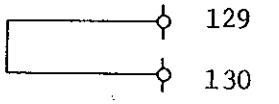


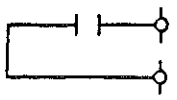
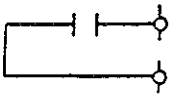
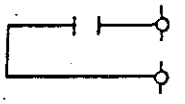
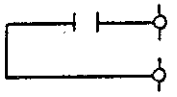
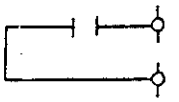
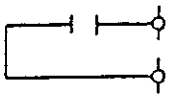
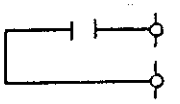
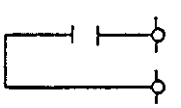
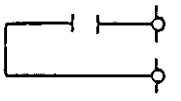
NC1 Unused
COM External signal output COM
MF Latch
M11 External signal output 11
M12 External signal output 12
M14 External signal output 14
M18 External signal output 18
M21 External signal output 21
M22 External signal output 22
M24 External signal output 24
M28 External signal output 28
COM External signal output COM
COM External signal output COM
Unused
STL Automatic run in progress
ALH Abnormal output
M30 Program end
M00 Program stop
ZGEN Z zero return end
YGEN Y zero return end
XGEN X zero return end
COM Zero return end
SV Servo ON
AUTO Automatic run

Notes: 1. The terminals surrounded by ◎ have been internally shortcircuited.

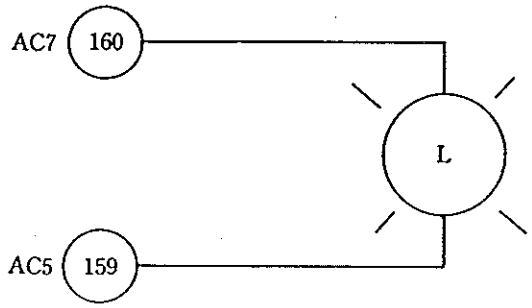
2. Use the crimp-style terminal TB1.25-9-1 (NICHIFU) for the connection with the terminal block.

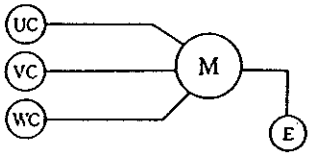
Description of external output

Signal name	Circuit	Function
Automatic run (AUTO)		When start button is pressed in memory and [w] , or automatic run is started, this output is continuously made until reset button is pressed, external reset input signal is entered, emergency stop input is entered, or other alarm is put out.
Servo ON (SV)		After NC power is switched ON and servo is turned ON, this output is continuously made until alarm is put out or NC power is switched OFF.
X zero return end (XGEO)		When the X, Y and Z axes return to zero position, this signal is put out.
Z zero return end (ZGEO)		
Y zero return end (YGEO)		
Program stop (YGEO)		When program stop is executed, this signal is put out. NC reset button or next start command turns off the output.
Program end (M30)		This signal is put out at the end of the program. NC reset button turns off the output under next start command.
Abnormal output (ALM)		If the alarm causing servo system turn off, alarm No. 0 through 33 is displayed on the XRT screen, this signal is put out.
Automatic run start		After automatic run start button is pressed, this signal is continuously put out until it halts or single stop.

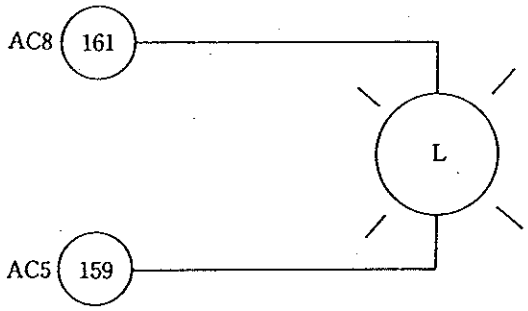
Signal name		Circuit	Function
External signal output	M28		When one of external signal output from 00 to 99 is specified in key-in and memory run, NC controller puts it out in decimal 2-digit BCD code. 1. External signal output is put out only one per job in memory run. 2. External signal output prohibited: 00 03 04 05 08 09 30 3. For external signal read timing, refer to the description of "external output terminal". 4. Program stop(M00) Program end (M30) Coolant ON: OFF Put out independently.
	M24		
	M22		
	M21		
	M18		
	M14		
	M12		
	M11		
	MF		

When 100 V AC spotlight is used

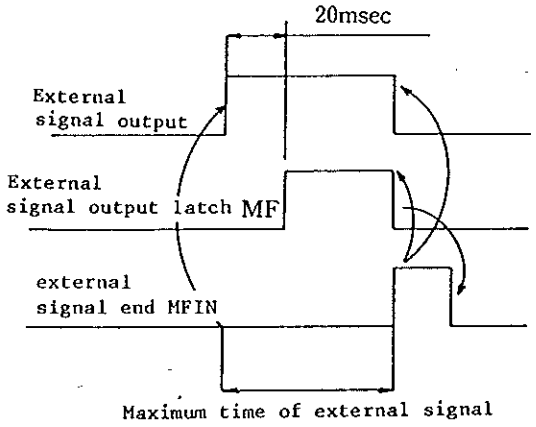




When 24 V AC spotlight is used



External signal output timing chart



4. (Supplementary comment)
External signal output latch is put out 20 msec after the external signal is put out. While the external signal output latch is being put out, external signal output is read. When the external signal output and external output signal end are put in, the external signal output latch is set to 0. A certain time after this, set the external signal end to 0.

* Before putting out external signal, set maximum time external signal for parameter. Maximum setting value is 9999 sec.
If set to 0, the maximum time becomes infinite.

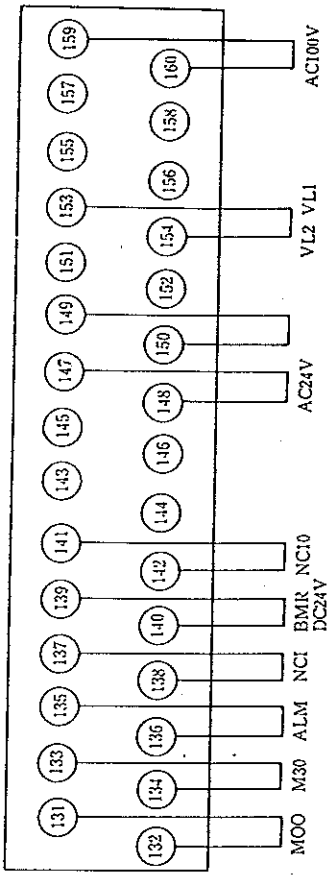
For the external signal output, separate a two-digit decimal number between the first and the second digits and convert each value into a four-digit binary number. The value "1" turns on the terminal concerned.

Example: When "external signal output 85" is input for operation in the machining program, the external signal output terminals M28, M14, and M11 are turned on.

External signal output	M28	M24	M22	M21	M18	M14	M12	M11
Digit position	Second digit				First digit			
Binary number	8 (2^3)	4 (2^2)	2 (2^1)	1 (2^0)	8 (2^3)	4 (2^2)	2 (2^1)	1 (2^0)

2. TC-225

Description of External Output Terminal Board



For spotlight (100VA)

ON/OFF by output 08
Single phase 100V
Solenoid valve
} Coolant output (usable up to 100VA)

Single phase 100V for air valve (50VA)

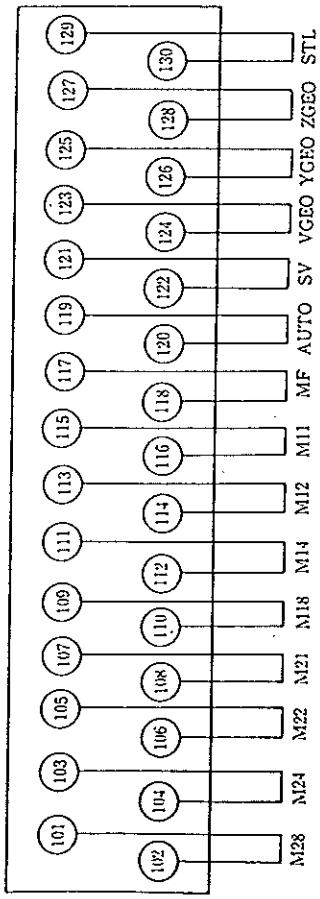
For spotlight (72VA) of European type

Internal use

Abnormal output

Program end M30

Program stop M00



Automatic run in progress

Z zero return finish

Y zero return finish

X zero return finish

Servo ON

Automatic run

External signal output latch

External signal output 11

External signal output 12

External signal output 14

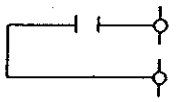
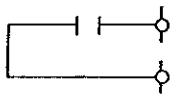
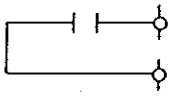
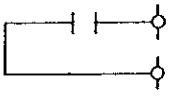
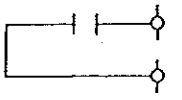
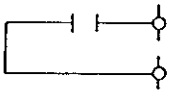
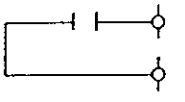
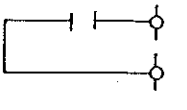
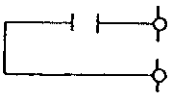
External signal output 18

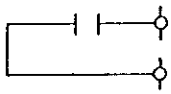
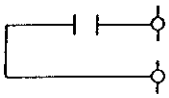
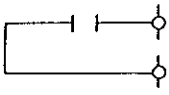
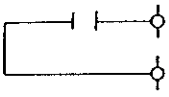
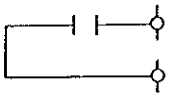
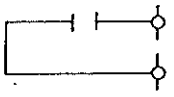
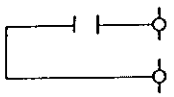
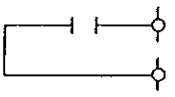
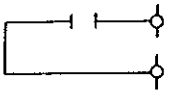
External signal output 21

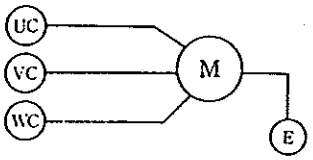
External signal output 22

External signal output 24

External signal output 28

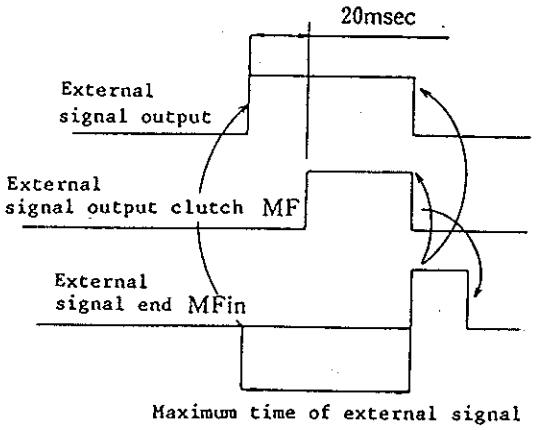
Signal name		Circuit	Function
	M28	 101 102	When one of external signal output from 00 to 99 is specified in key-in and memory run, NC controller puts it out in decimal 2-digit BCD code. 1. External signal output is put out only one per job in memory run. 2. External signal output prohibited: 04 05 08 09 30 3. For external signal read timing, refer to the description of "external output terminal". 4. Program stop (M30) Program end (M00) Coolant ON: OFF Put out independently.
External signal output	M24	 103 104	
	M22	 105 106	
	M21	 107 108	
	M18	 109 110	
	M14	 111 112	
	M12	 113 114	
	M11	 115 116	
	MF	 117 118	

Signal name	Circuit	Function
Automatic run (AUTO)	 119 120	When start button is pressed in memory and key-in run, or automatic run is started, this output is continuously made until reset button is pressed, external reset input signal is entered, emergency stop input is entered, or other alarm is put out. After NC power is switched ON and servo is turned ON, this output is continuously made until larm is put out or NC power is switched OFF.
Servo ON (SV)	 121 122	
X zero return end (XGEO)	 123 124	When the X, Y and Z axes return to zero position, this signal is put out.
Y zero return end (YGEO)	 125 126	
Z zero return end (ZGEO)	 127 128	
Automatic run start (STL)	 129 130	After automatic run start button is pressed, this signal is continuously put out until it halts or single stops.
Program stop (M00)	 131 132	When program stop is executed, this signal is put out. NC reset button or next start command turns off this output.
Program end (M30)	 133 134	This signal is put out at the end of program. NC reset button turns off the output under next start command.
Abnormal output (ALM)	 135 136	If the alarm causing servo system turning off, alarm No. 0 through 31, is displayed on the CRT screen, this signal is put out.



Coolant motor wiring diagram

External signal output timing chart



(Supplementary comment)
External signal output latch is put out 20 msec after the external signal is put out. While the external signal output latch is being put out, external signal output is read. When the external signal output and external output signal end are put in, the external signal output latch is set to 0. A certain time after this, set the external signal end to 0.

* Before putting out external signal, set maximum time of external signal for parameter. Maximum setting value is 9999 sec.

8. Program Edit

8-1. Before Program Edit

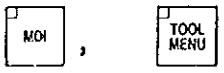
Before program edit, set the following items completely, otherwise, program will not be made correctly.

Program edit requires correct order. Follow the screen display carefully.

(1) Power switch ON

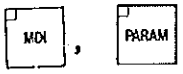
Power switch ON allows program edit (so far as the following items are set). For continuous operation of the machine, be sure to make machine zero position return.

(2) Tool registration



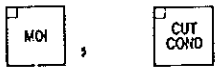
--- Check whether registered tools can be used for new program. (Can be checked by tool assignment.)

(3) Parameter confirmation



--- Check whether initial set value is acceptable. If the value is necessary to change, change it by using parameter 1 (switch).

(4) Cutting tool/condition confirmation



--- Check whether initial set value is acceptable.

1. Tool pattern
2. Cutting tool/condition
3. Hole sizes before tapping

If necessary, change the contents.

8-2. Insertion/Deletion/Cancellation of Data

Data may be required to be altered during the operation of this machine or editing program. If so, use the following procedure.

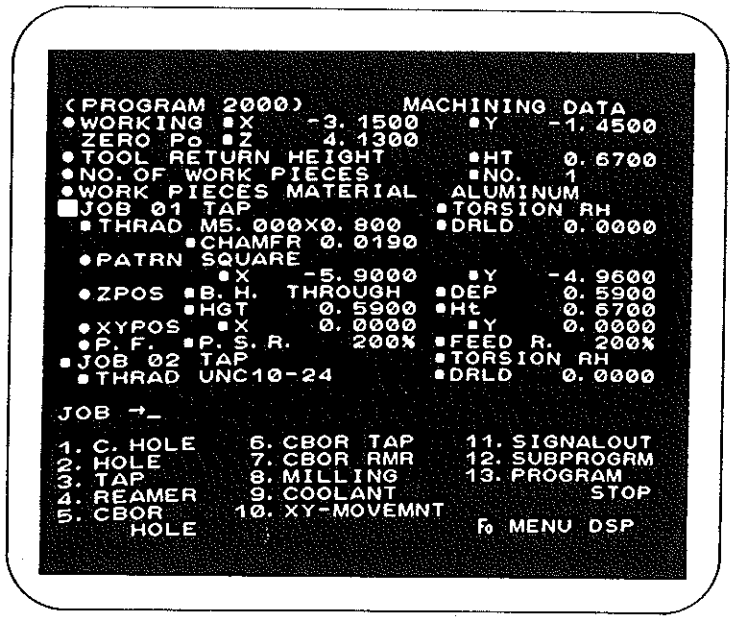
Make the PROGRAM PROTECT switch on.

(1) Insert --- Adds data of job or sub-program.

o Addition of job

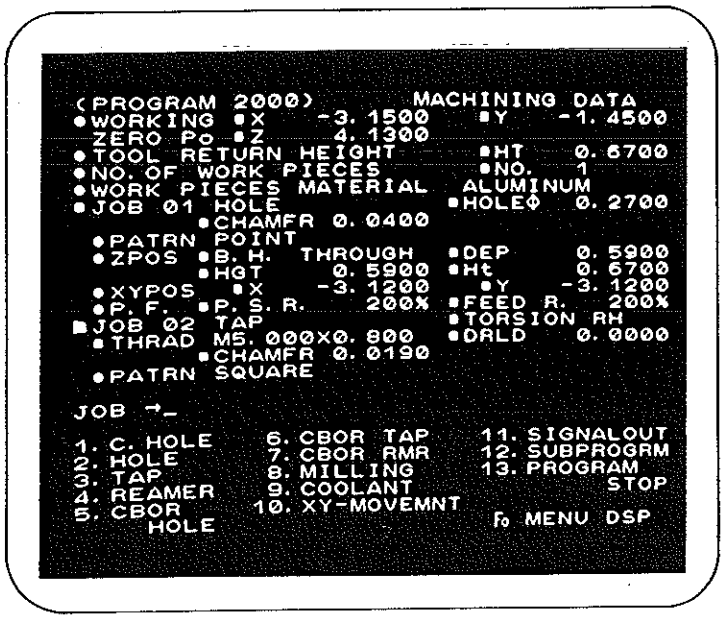
1. When new job is added to the position before the job

ex) When new job is inserted into the position before job 01 in the screen given below.
Move the cursor by using the ⇧CURSOR ⇩CURSOR keys to the position into which new job is inserted.



Press the INSRT key at job 01.
When job and last shape (except top face cutting) of job 01 is requested, enter the necessary data. Job 01 is changed to job 02.

Until data is completely input after the INSRT key is pressed, the contents of items after current job 01 disappears from the screen.



2. A job is added to the program in which machining data is completely input.

Move the cursor to the end of main program by using the ⇧ CURSOR and ⇩ CURSOR keys.

Press the INSRT key to set the last job or shape of next job and to input the machining data.

o Addition of sub-program

Move the cursor to the end of sub-program by using the ⇧ CURSOR and ⇩ CURSOR keys.

Press the INSRT key to set new sub-program number and to input the machining data.

After this program edit, press the Fo key. (The display returns to the menu screen.)

(2) Alter --- Alters the contents of data.

Move the cursor to the data to be altered by using the [⇧ CURSOR] and [⇩ CURSOR] keys, and set new data.

After this program edit, press the [Fo] key.

(The display returns to the menu screen.)

(3) Delete --- Deletes programs, program jobs, and sub-program.

o In deleting the data of job or sub-program, move the cursor to the data position by using the [⇧ CURSOR] and [⇩ CURSOR] key, and press the [DELET] key.

(When sub-program is deleted, the job using the sub-program must be also deleted.)

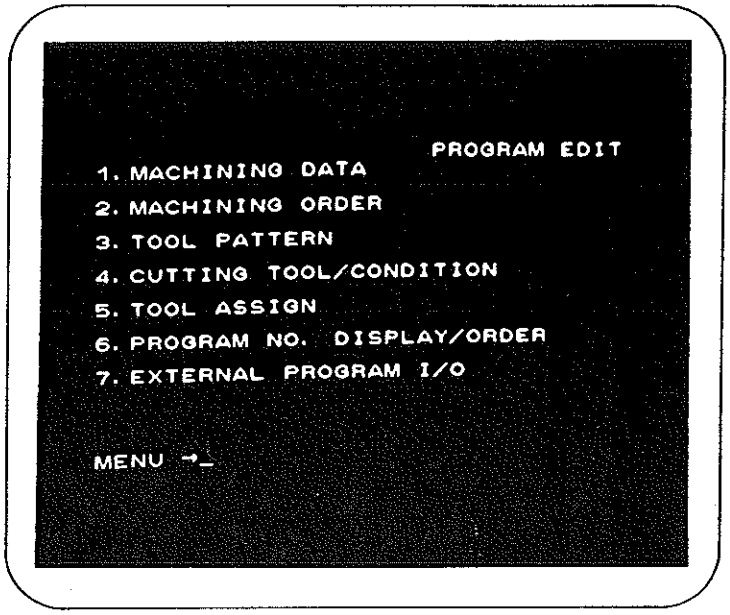
(4) Cancel

When input data is displayed on the input demand display area (before pressing the [ENTER] key), press the [CAN] key, and the input data will be deleted.

* After the completion of data correction, be sure to turn off the PROGRAM PROTECT switch.

8-3. Program Edit Contents

EDIT mode



Program edit menu screen

(1) MACHINING DATE — To complete program, necessary items for machining are sequentially input in the characteristic interactive mode provided by Brother Industries, Ltd.

(2) MACHINING ORDER — Sets the sequence order for machining the edited programs.

(3) TOOL PATTERN — Set the tool pattern for the edited programs.

(4) CUTTING TOOL/CONDITION — Sets the peripheral velocity and feedrate of the tools used for edited programs.

(5) TOOL ASSIGN — Assigns the registered tools to the tools used for edited programs and to the tool menu.

(6) PROGRAM NO. DISPLAY/ORDER

Displays the list of registered program numbers. (Performs program deletion and others.)

(7) EXTERNAL PROGRAM I/O

Inputs or outputs program data of the external memory unit (special accessory).

Press the [MDI] and [MAGAZ] key to register tools in the magazine.

↓

Execute the edited program.

* (2) MACHINING ORDER
  (3) TOOL PATTERN
  (4) CUTTING TOOL/ CONDITION

By pressing the [EDIT] key, the data is specified only for the program.

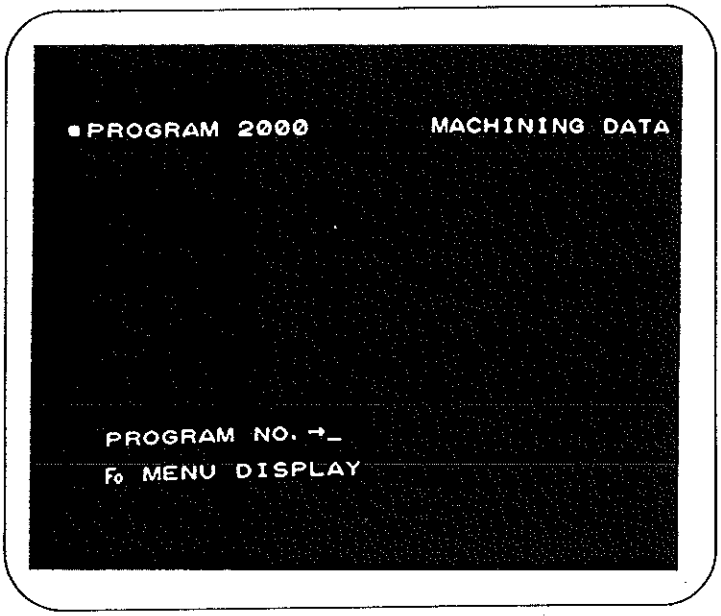
When machining order data of PARAMETER, cutting tool/condition data, and tool pattern data are accepted, check data, and omit the key operation.

8-3-1. Input of machining data

This screen allows the creation of machining data (program). (Data can be set in the item where the cursor flashes, and the cursor will descend.)

Set the [EDIT] key and PROGRAM PROTECT switch to on.

Machining data is set.



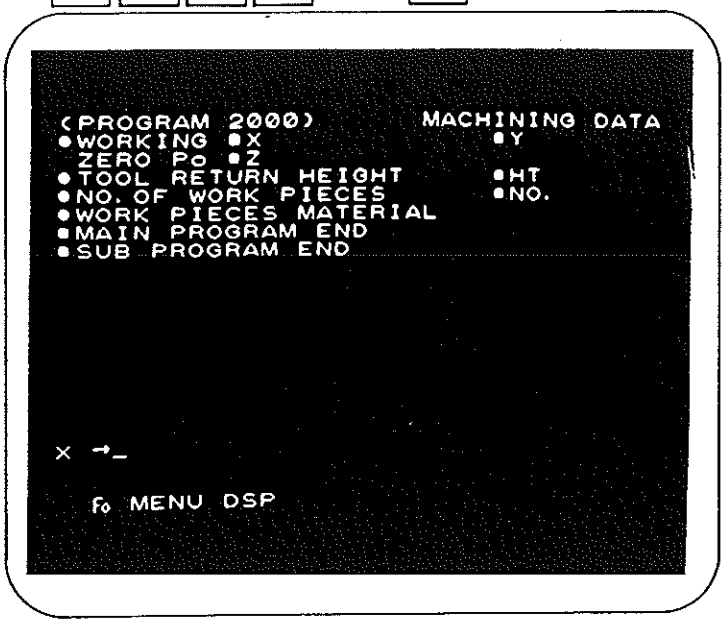
The program number on the left-hand screen was used previously.

When the program number on the left-hand screen is accepted in setting the program number, the number can be accessed only by pressing the [ENTER] key.

Set 4-digit program number.

Any program number can be set within four digits. (Program No. 0 cannot be used.)

ex) Press [2] [0] [0] [0] and [ENTER] keys for program number.



Major items of the program are displayed. Key in items by means of dialog on the screen display of reach machining.

1. Setting of machine zero position

o Set the machine zero position X, Y and Z.
Set the distance from machine zero position. (Refer to 6-1 Machine Zero Position Return.)
For the Z axis, the position from the table top (0) is set.

ex)

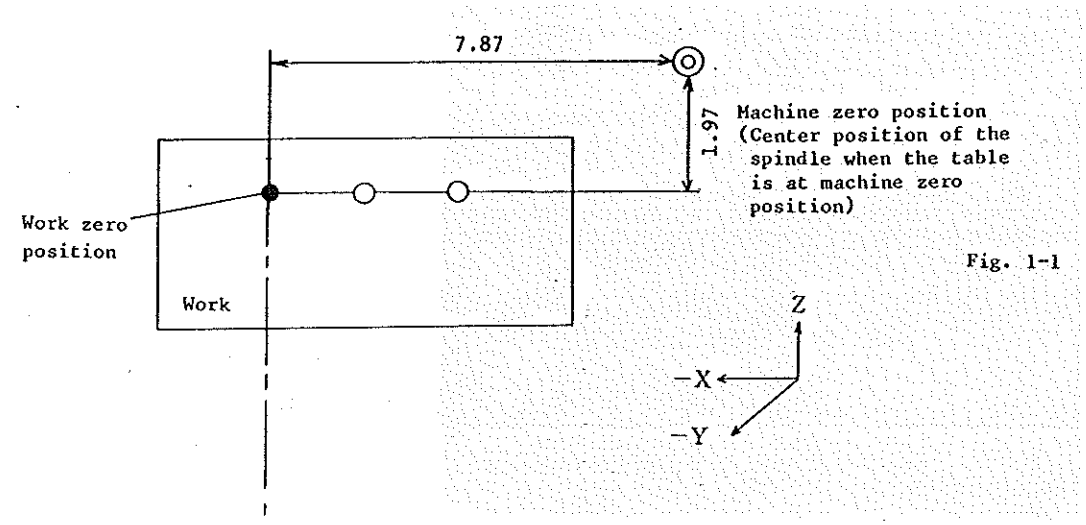


Fig. 1-1

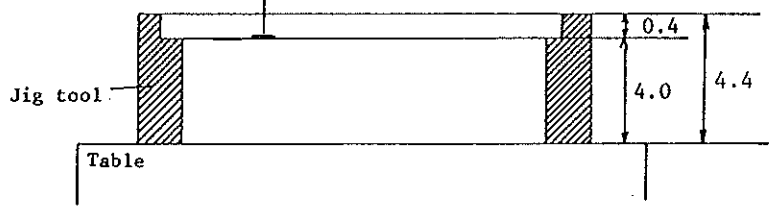


Fig. 1-2

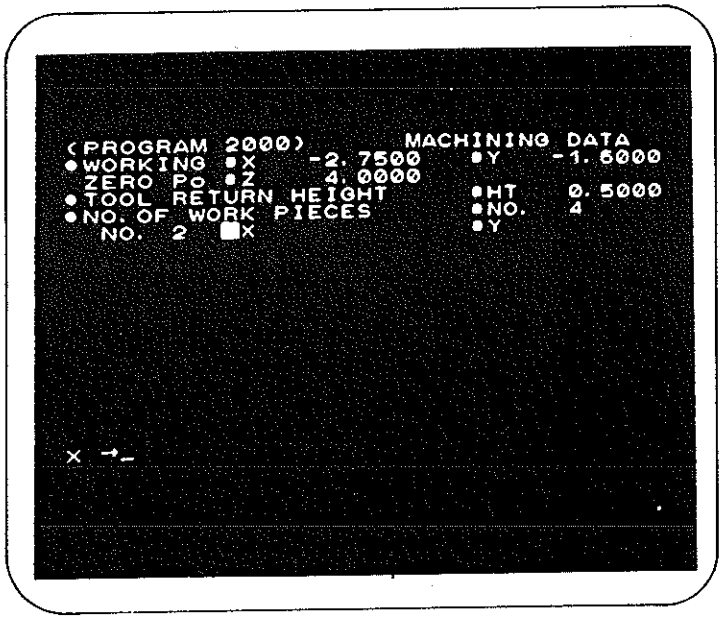
The above machine zero position is specified as follows:
(X, Y, Z) = (-7.87, -1.97, 4.0)

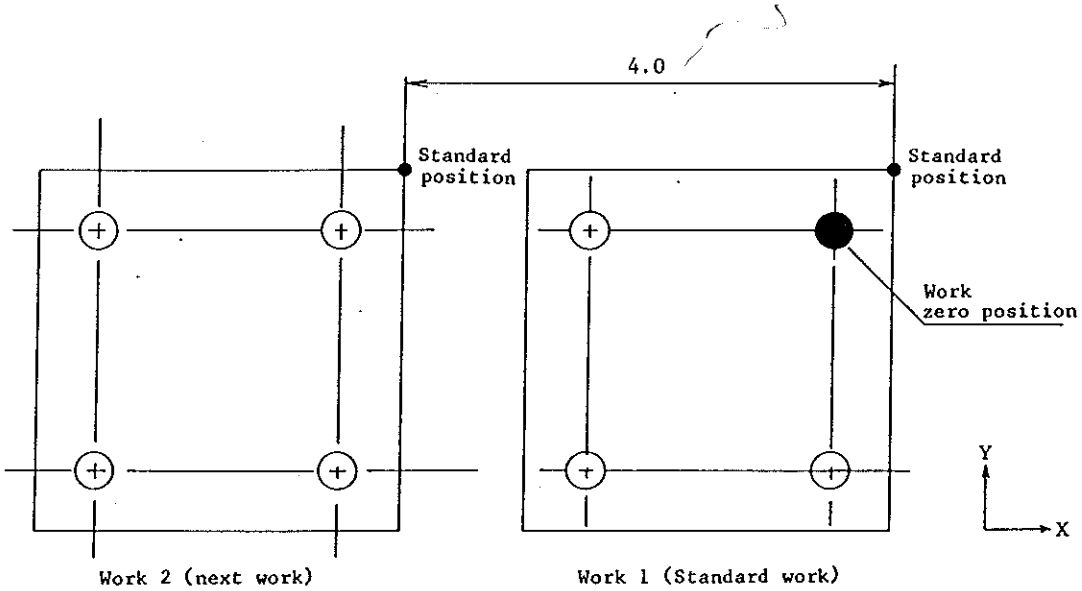
2. Setting of TOOL RETURN HEIGHT

When a job is ended and the machine shifts to next job, the rising height of the Z axis (top of the tool) must be set.
Set the position from the Z machine zero position.
When work zero position is set in the work bottom (refer to Figure 1-2), the TOOL RETURN HEIGHT is height added work thickness to height of workpiece upper surface and tool end.

3. Setting NO. OF WORKPIECES

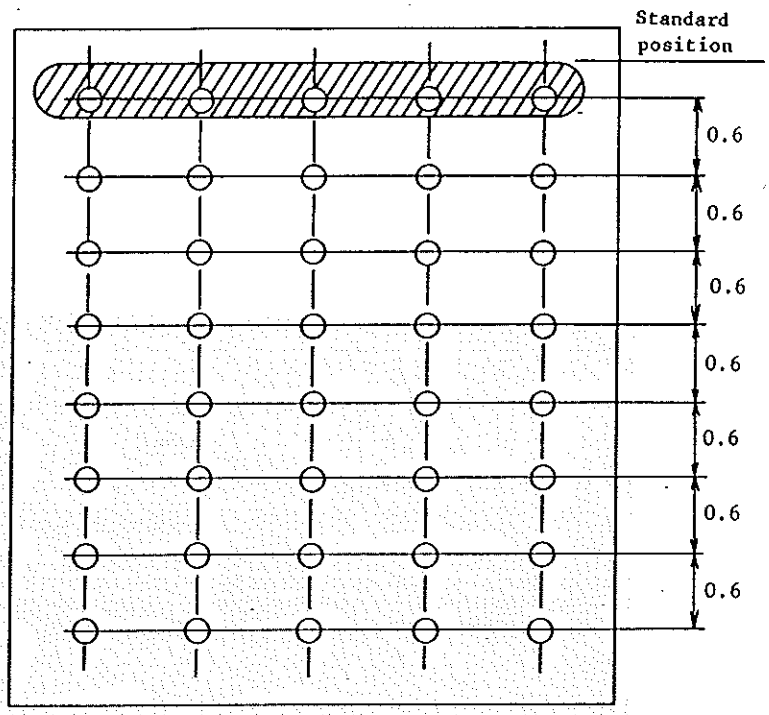
Set the number of workpiwces. Up to 12 workpieces can be set. When plural quantity is set, the distance form work start position for the standard work to work start position for each work is requested to be entered.





In the above figure, since the Y axis is in the same position, (X, Y) is specified as (-4.0, 0).

In addition, programs in the following cases can be edited as a program with plural workpieces.



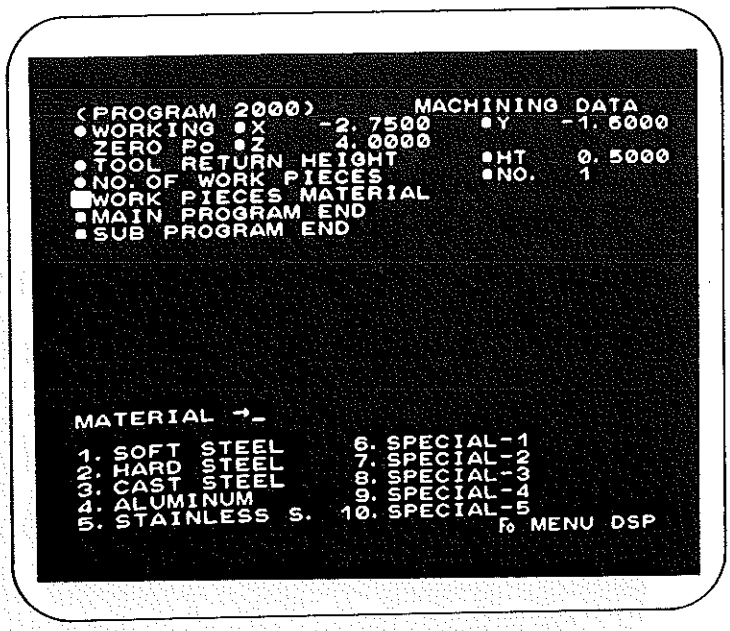
When many work holes are in order as shown in the above figure.

Regarding holes in one holizontal line with slant lines as one workpiece, total of eight workpiece can be set for working.

In this case, since the position of the X axis is the same from the standard position for standard work to standard position for each work, workpieces are as follows:

Work 2 (X, Y) = (0, -0.6)
Work 3 (X, Y) = (0, -1.2)
Work 4 (X, Y) = (0, -1.8)
Work 5 (X, Y) = (0, -2.4)
Work 6 (X, Y) = (0, -3.0)
Work 7 (X, Y) = (0, -3.6)
Work 8 (X, Y) = (0, -4.2)

4. Work material setting



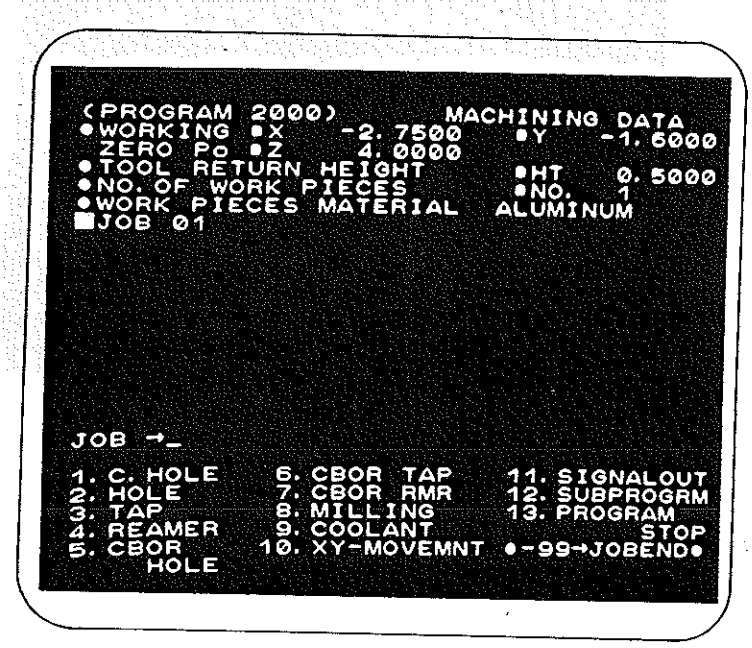
Select desired material from among the registered materials (10 types) by the menu number, and set it.

When work material is not registered, it is necessary to register the material as special material and enter cutting tool/condition.

5. Job

When work material is completely entered, set necessary machining items for each job.

* Job 01 is displayed.



Select the last machining or last shape (except for top face cutting) from among the teaching data area. Items necessary for the machining or shape are requested to be entered.

All machining processes set in the tool patterns are included here.

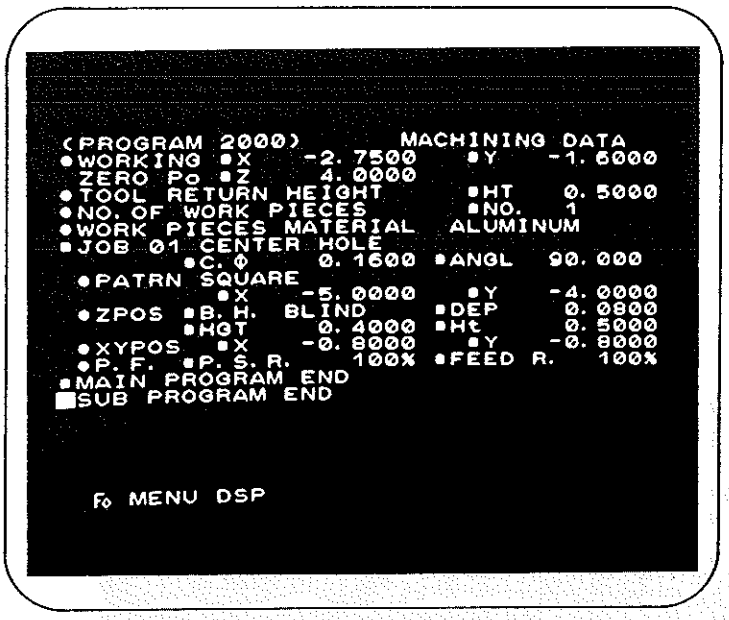
ex) If Chamfering

- Tapping
  - ■ 1. Center drill
  - ■ 2. Drill
  - ■ 3. Chamfering tool
  - ■ 4. Tap

No Chamfering

- ■ 1. Center drill
- ■ 2. Drill
- ■ 3. Tap

is set in the tool pattern, specifying tap and other than 0 for chamfering causes center drill, drill, chamfering tool and tap processes to be carried out in the tapping process.



Set the items necessary to machine in each job.

Set center hole by pressing the [1] and [ENTER] keys.

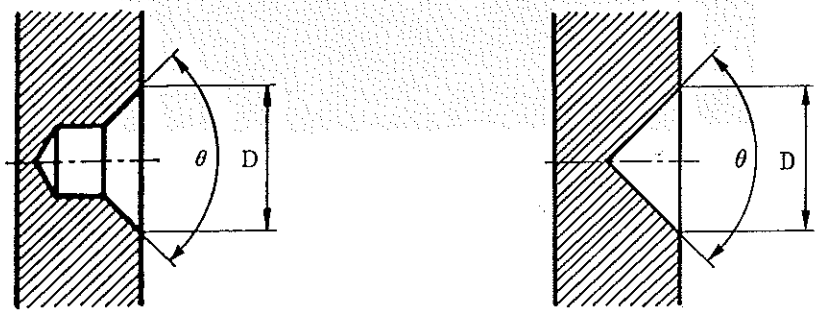


Fig. 5-1

■ C. ϕ (D in Fig. 5-1)

Set the diameter of the work hole.

■ ANGL (θ - in Fig. 5-1)

Set the angle of hole.

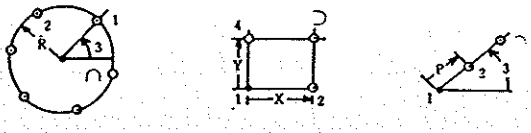
Set the diameter and angle of hole.

Items in a job (pattern, Z axis position, XY position, peripheral speed) are described in Section 5-9.

PATRN: Set work pattern.

Pattern

1. Circle 2. Square 3. Line 4. Point



Select the pattern from above teaching data areas.

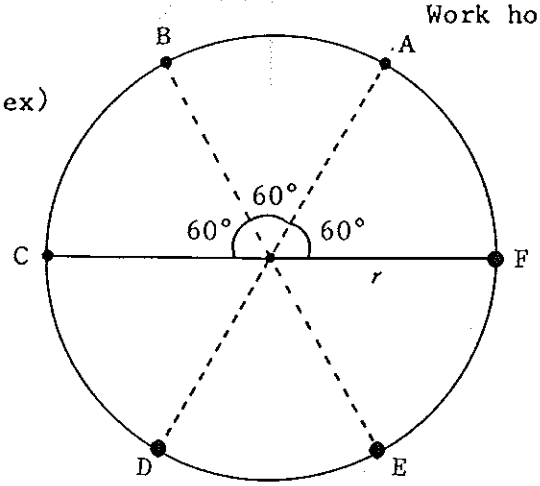
(1) CIRCLE pattern

```
(PROGRAM 2000)          MACHINING DATA
●WORKING ●X    -2.7500    ●Y    -1.6000
 ZERO Po ●Z     4.0000
●TOOL RETURN HEIGHT        ●HT    0.5000
●NO. OF WORK PIECES        ●NO.   1
●WORK PIECES MATERIAL     ALUMINUM
●JOB 01 TAP                ●TORSION RH
 ●THRAD M5.000X0.800       ●DRLD  0.4000
       ●CHAMFR 0.0315
 ●PATRN CIRCLE
       ●N.D.    6.000     ●N.M.   6
       ●S.A.   60.000     ●RADU   2.0000
●ZPOS  ●B.H.  THROUGH      ●DEP    0.4000
       ●HGT    0.4000     ●Ht     0.5000
●XYPOS ●X     -3.2000      ●Y    -2.7500
●P.F.  ●P.S.R.   100%     ●FEED R.  100%
MAIN PROGRAM END

Fo MENU DSP
```

Work hole

ex)



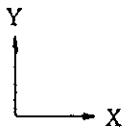


Fig. 1-1

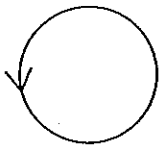
- N.D.: Set whether work is to be performed at the positions obtained by dividing the circle.
  In this case, the circumference must be equally divided.
  In Figure 1-1, the circumference is divided by six.
  A setting value must have a decimal point.
  Maximum setting value: 99.9999

- N.M.: Set the number of the work hole
  In Figure 1-1, work number is 6.

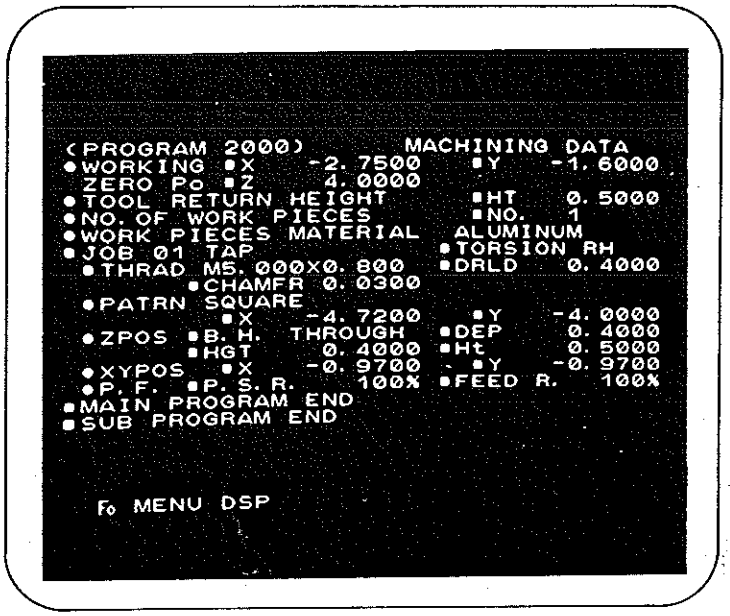
- S.A.: In the XY plain surface, set the angle of the work start position to the X axis.
  In Figure 1-1, when the work sequence is specified in the order of A, B, C, D, E and F, the angle of the work start position to the X axis is specified to 60 degree.

- RADU: Set radius of the circumference pattern.
  r in Figure 1-1

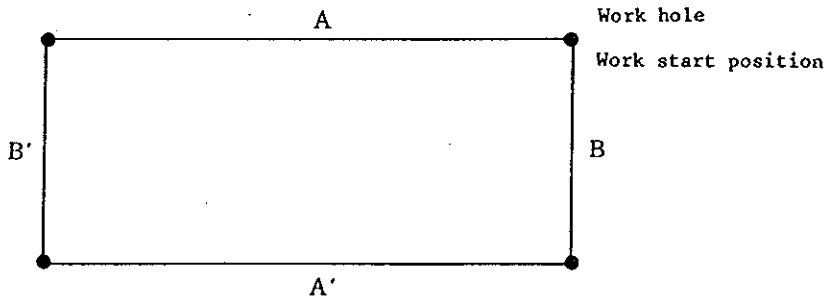
* The working direction of circumference pattern is always specified as counterclockwise direction.



(2) Square pattern

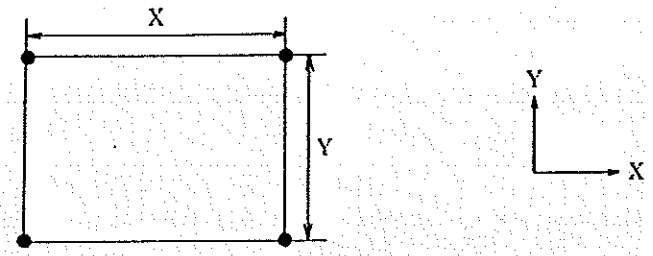


Work is performed on the contacts of the square (square or rectangular). Four-hole work is performed.



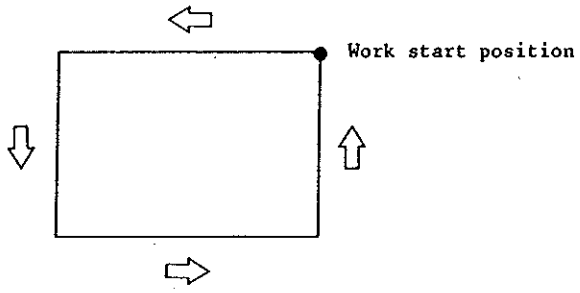
$$\begin{cases} A = A' \\ B = B' \end{cases}$$

■ X, Y: Set pitch in the X direction and in the Y direction of the square pattern.

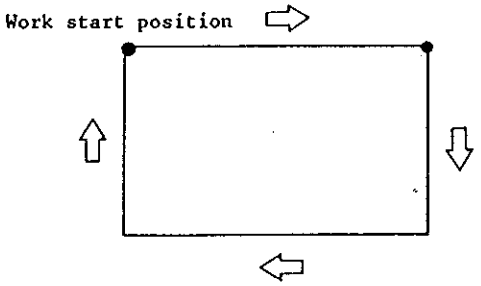


According to work start position and pitch setting, the moving direction is determined.

e x) X -4.72
Y -4.0



X 4.72
Y -4.0



(3) LINE pattern: Perform working in the line.

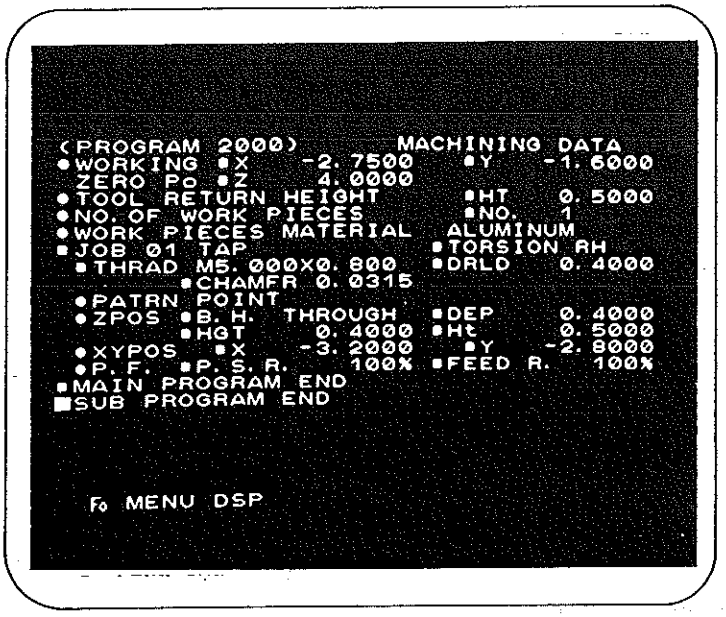
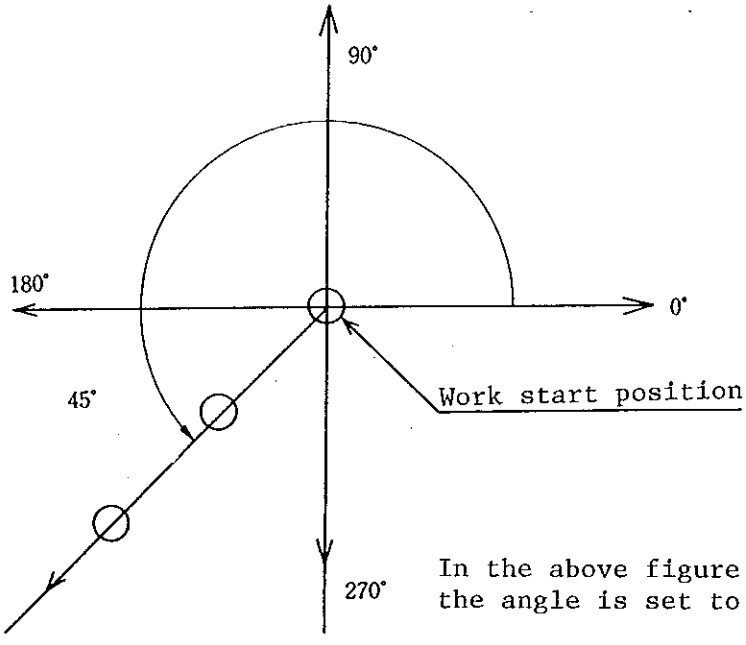


Fig. 3-1

■ N.M.: Set the number of the work hole
In figure 3-1, work number is 3.

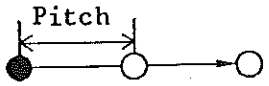
■ S.A.: In the XY plain surface, set the angle between work movement direction (pitch direction) from the work start position and the X axis.

ex)



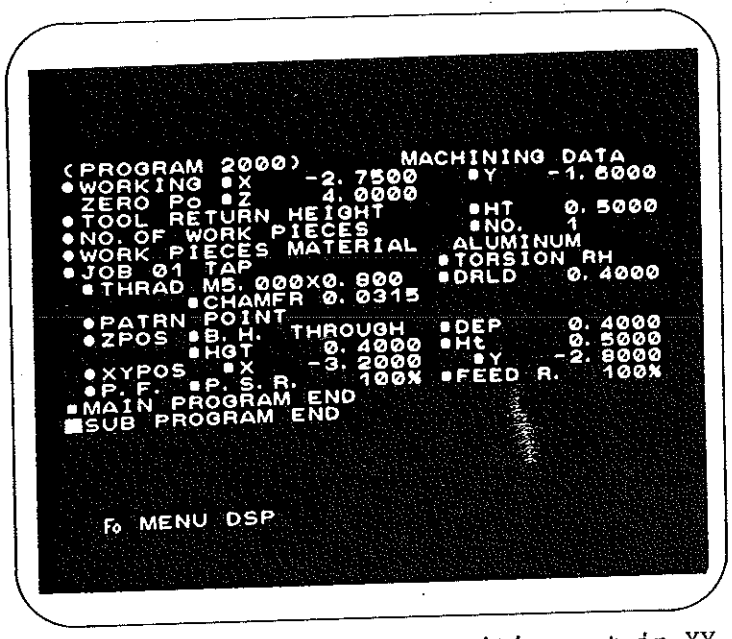
In the above figure,
the angle is set to 225 degree.

■ Pitch: Set the hole work pitch on the straight line pattern.



* Workable only when divided into equal length.
* When pitch is set in the minus direction, moving direction is reversed.

(4) POINT pattern: Work is performed at a position.



Work is performed at the position set in XY position.

Z POS

■ B.H.: Set whether the work hole is a through hole or blind hole. Select data from teaching data areas of: 1. Through 2. Blind

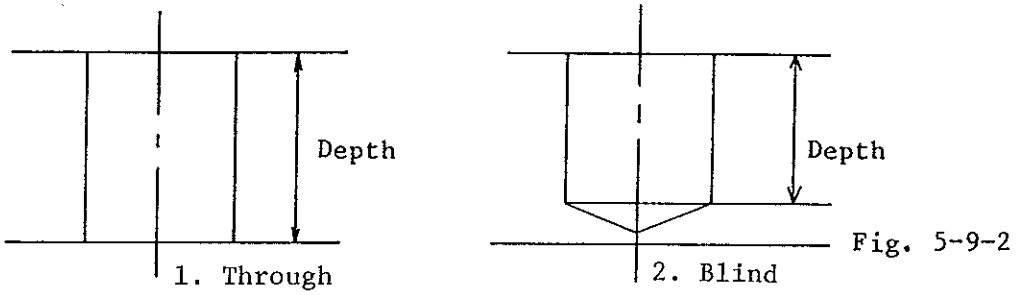
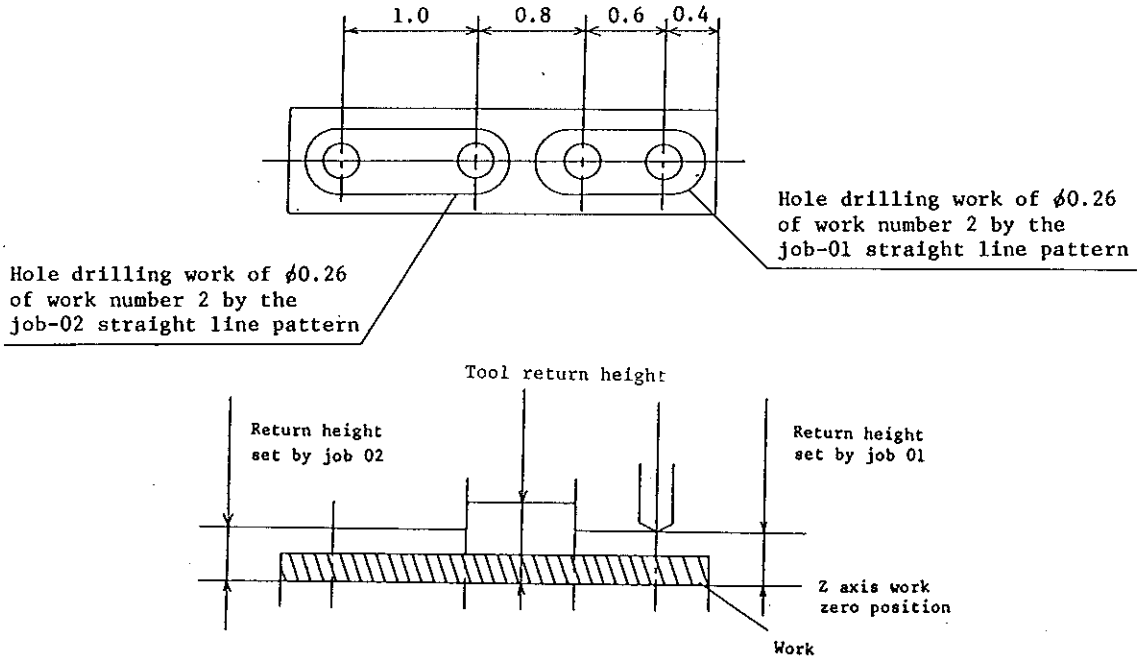


Fig. 5-9-2

■ DEP: Set the depth of the work hole.
In response to the through hole, set the thickness of the work.

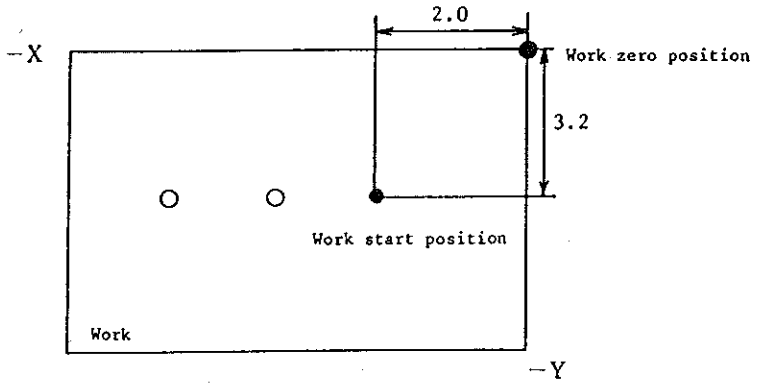
■ HGT: Set the height from the Z axis work zero position to the work top face.
When the work zero position and work top face are the same, set the height to 0.

■ Ht: Set the height of the Z axis returning from one work position to another work position in a job.



XY POS

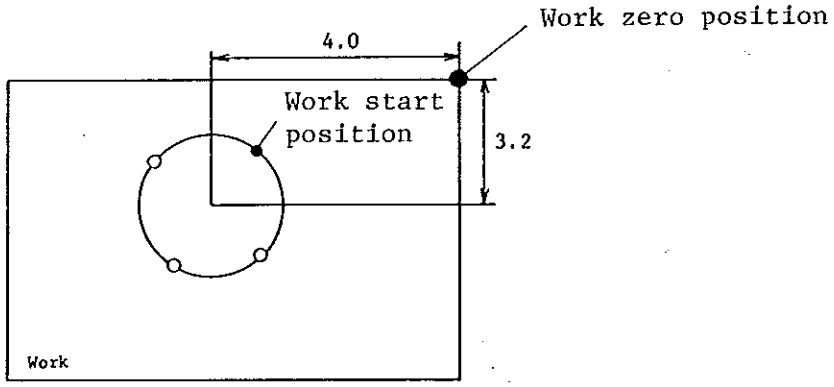
Set the start position of XY work.
Set the position from the XY work zero position.



In this case, the XY position is set as follows:

X - 2.0
Y - 3.2

In response to the CIRCLE pattern, set the center position of the circle from the work zero position.



In this case, the XY position is set as follows:

X - 4.0
Y - 3.2

Peripheral Feed

The memory data specifies the peripheral speed and feed, in percentage, of the cutting condition described in Section 4, Cutting tool/condition of the Program Edit, and the data accelerates or decelerates the peripheral speed and feed. (For details, refer to 7-5-2, Cutting tool/condition.)

Peripheral velocity:

Revolution peripheral speed ft/min $= \dfrac{\text{(Tool diameter (inch))} \times \text{Revolution speed} \times \pi}{12}$

Feed: Feed amount inch/revolution Moving distance per revolution

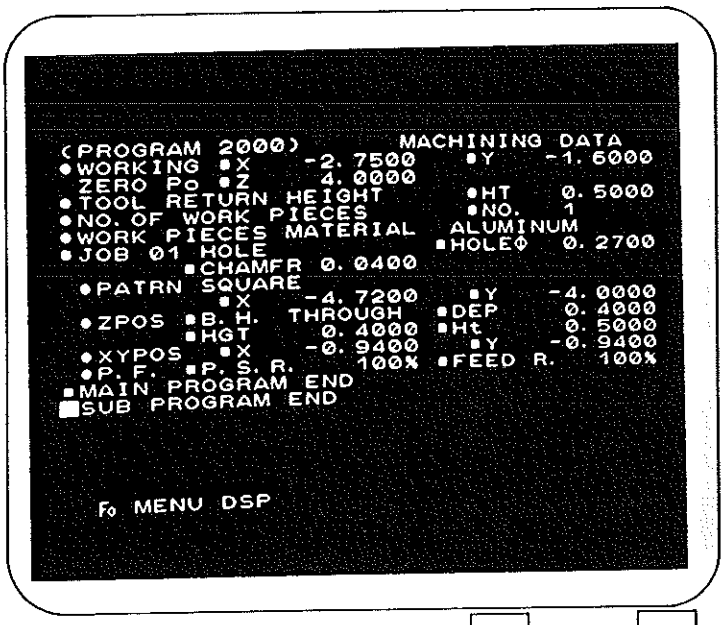
ex) Blind hole work is performed with tap of M8 × 1.25 for material (ADC12) work.
(material ADC12)

Machining type	Registered Tool Name	Peripheral Velocity ft/min	Feed amount inch/revolution
Center Drill	■ Center Drill	■ 98.	■ 0.004
Drill	■ Ø-3 Drill 3-	■ 164. ■ 164.	■ 0.004 ■ 0.008
Tap (Blind)	■ Ø-3 S tap 3-	■ 98. ■ 131.	* *
Tap (Through)	■ Ø-3 P tap 3-	■ 98. ■ 131.	* *

When the machine is delivered, the cutting condition is set as shown above.

In this case, when the peripheral velocity is set to 150% and feed amount to 150%, the peripheral velocity is magnified 1.5 times, or set to 196.5 ft/min. Therefore, the peripheral velocity is set to $12 \times 196.5/(8 \times 25.4) \times \pi = 2383$ min $^{-1}$ (rpm).

5-2. HOLE



Set the hole by pressing the [2] and [ENTER] keys.

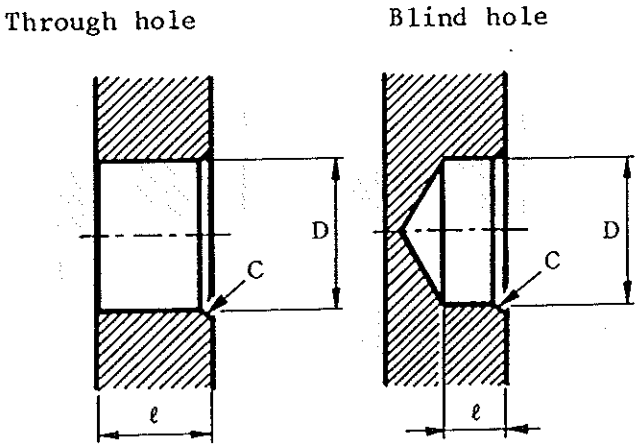


Fig. 5-2

■ HOLE ϕ (D in Fig. 5-2)

Set the diameter of work hole.

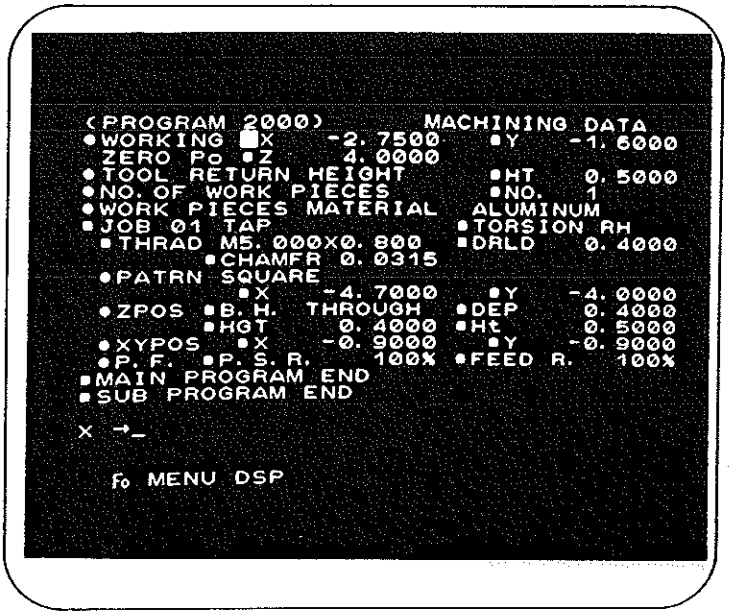
■ CHAMFR (C in Fig. 5-3)

Set the chamfering size.

When chamfering is unnecessary, set the size to 0.

Set the hole diameter and chamfering size.

For the subsequent settings of the pattern, Z-axis position, XY position, peripheral-speed, and feed; see Item 5-1. "Center hole machining".



Set the tap by using the [3] and [ENTER] keys.

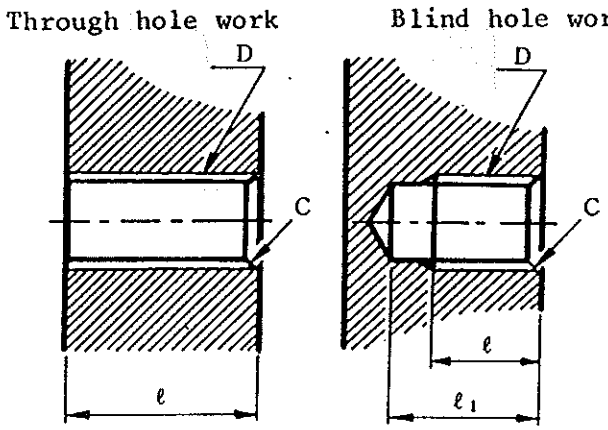


Fig. 5-3

■ TORSION Sets whether the screw is right-handed or left-handed. Data is selected from teaching areas of:

1. right-handed thread 2. left-handed thread.

■ THRAD (D in Figure 5-3) Sets the screw type. Data is selected from teaching data area.

Type	Displayed screw type	Screen display (Abbreviation)
TC-215	1. Metric coarse screw threads	MC
	2. Metric fine screw threads	MF
	3. Unified coarse screw threads	UNC
	4. Unified fine screw threads	UNF
	5. National Pipe Thread Standard	NPS
	6. British Standard Pipe Thread	BSP
	7. British Standard Whitworth	BSW
	8. British Standard Fine Thread	BSF
	9. British Association	BA
TC-225	In addition to the above nine types,	
	10. American Standard Taper Pipe Threads for general use.	NPT

■ Nominal diameter (D in Fig. 5-3) Set the nominal size of screw.

■ Pitch and threads/inch (D in Fig. 5-3)

Metric screw = Pitch

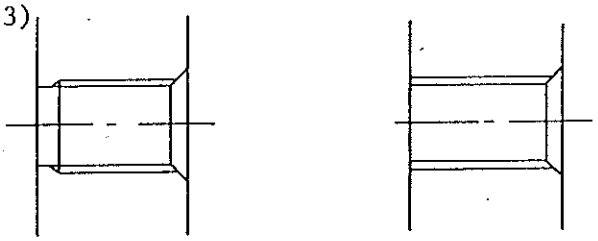
Inch screw = Threads/inch

* When screw type, nominal size, pitch and threads/inch are set, primary hole sizes and hole sizes before tapping in cutting tool/condition (7-5-2) are automatically set.

■ DRLD Sets hole sizes before tapping.

Through hole work

(ℓ in Fig. 5-3)



Ⓐ Ⓑ

The shape after work under the condition of (primary hole depth) > (depth) Ⓐ is different from that under the condition of (primary hole depth) $\leqq$ (depth) Ⓑ.

Blind hole work (ℓ_1 in Figure 5-3)

Taking into account the ineffective length, set the size.

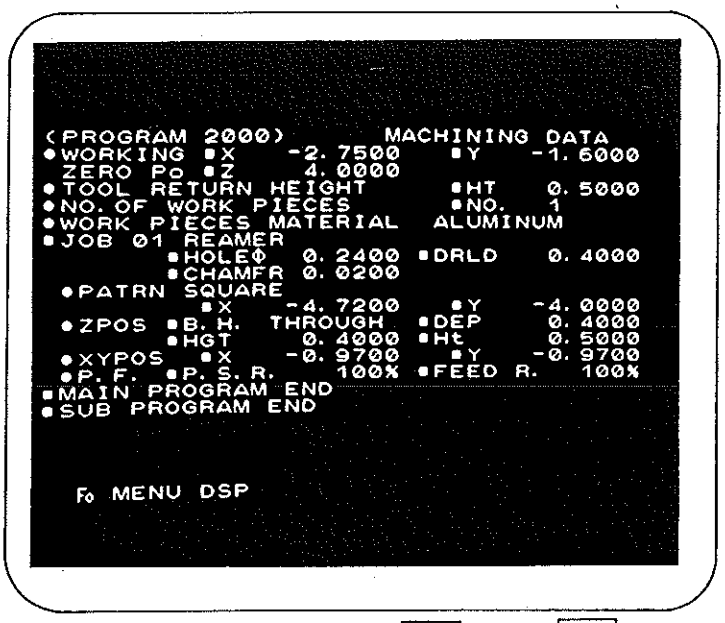
* When the set value of the primary hole depth is less than or equal to the set value of the depth in the counter bore depth, the parameter allowable depth of the hole size before tapping become effective.

(For details, refer to Section 7-2-1, Parameter (Switch).)

■ CHAMFR (C in Figure 5-3) Set chamfering size. When chamfering is unnecessary, set the size to 0.

For the subsequent settings of the pattern, Z-axis position, XY position, peripheral speed, and feed; see Item 5-1. "Center hole machining".

5-4. REAMER



Set reamer by using the [4] and [ENTER] keys.

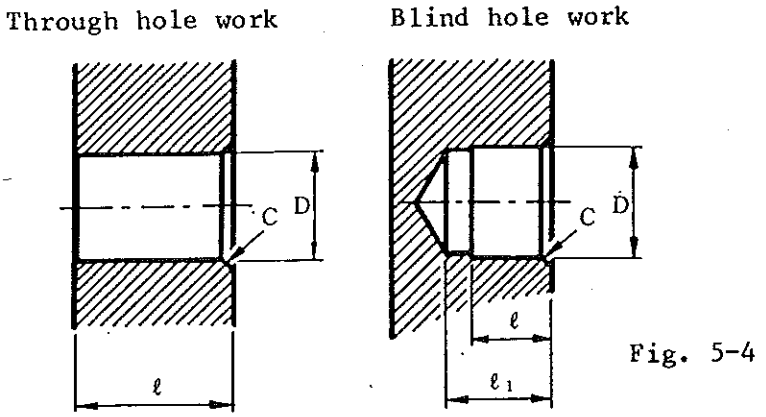
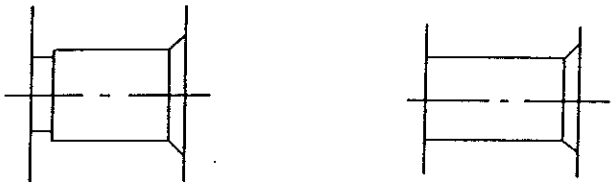


Fig. 5-4

■ HOLE ϕ (D in Figure 5-4) Set the reamer diameter.

* When the reamer diameter and parameter reamer finish allows 1 and 2 (7-2-1) is set, the primary hole drill diameter is automatically set.

■ DRLD Set the primary hole depth of reamer work.
Through hole work (ℓ in Figure 5-4)



Ⓐ Ⓑ

The shape after work under the condition of (primary hole depth) is more than (depth) Ⓐ is different from that under the condition of (primary hole depth) is less than or equal to (depth) Ⓑ.

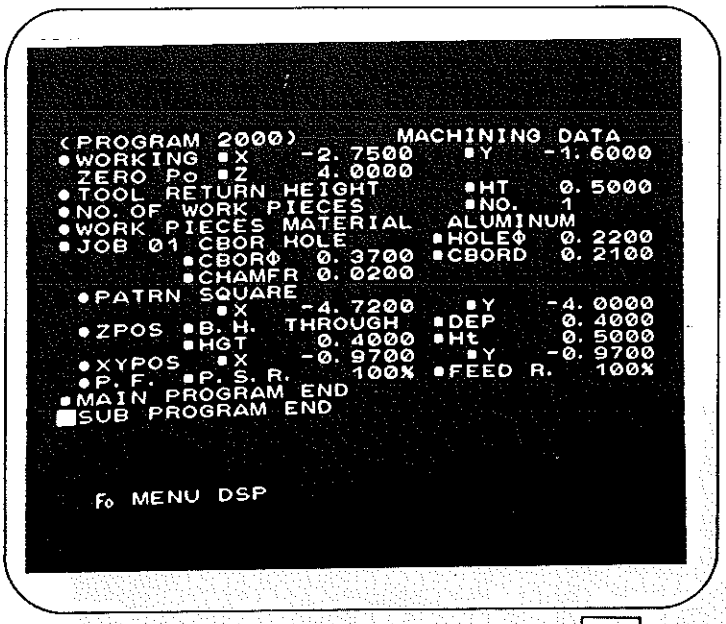
Blind hole work (ℓ_1 in Figure 5-4)
Taking into account ineffective length of the reamer, set the value.

* As for blind hole, when the set value of primary depth is less than or equal to the set value of the depth, the reamer primary allowable depth of the parameter become effective. (For details, refer to Section 7-2-1, Parameter (Switch).)

■ CHAMFR (C in Figure 5-4) Set the chamfering size. When chamfering is unnecessary, set the size to 0.

For the subsequent settings of the pattern, Z-axis position, XY position, peripheral speed, and feed; see Item 5-1. "Center hole machining".

5-5. CBOR HOLE work



Set counter bore hole by using the 5 and ENTER keys.

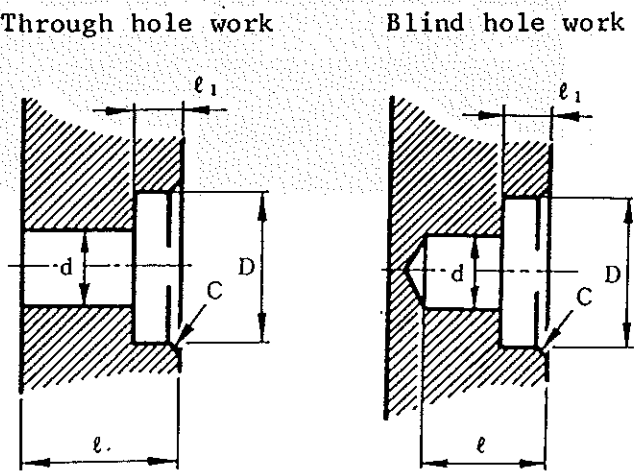


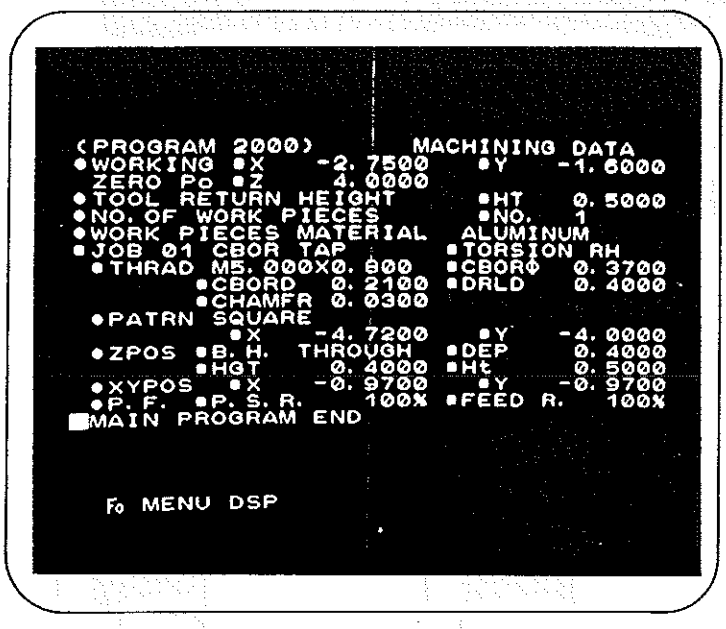
Figure 5-5

- HOLE ϕ (d in Figure 5-5) Set the diameter of the drill hole.
- CBOR ϕ (D in Figure 5-5) Set the counter bore diameter.
- CBDRD (ℓ_1 in Figure 5-5) Set the counter bore depth.

- CHAMFR (C in Figure 5-5) Set the chamfering size. When chamfering is unnecessary, set the size to 0.

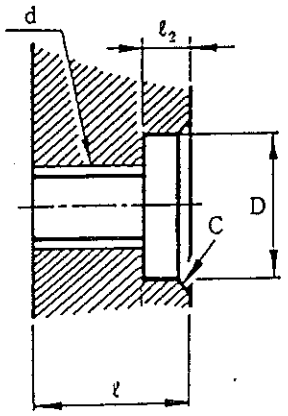
For the subsequent settings of the pattern, Z-axis position, XY position, peripheral speed, and feed; see Item 5-1. "Center hole machining".

5-6. CBOR TAP



Set the counter bore tap by using the [6] and [ENTER] keys.

Through hole work



Blind hole work

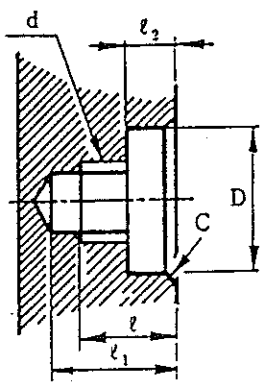


Fig. 5-6

- TORSION

  Sets whether the screw is right-handed or left-handed.

  Data is selected from teaching areas of:

  1. right-handed thread 2. left-handed thread

- THRAD (D in Figure 5-6)

  Sets the screw type.

  Data is selected from teaching data area.

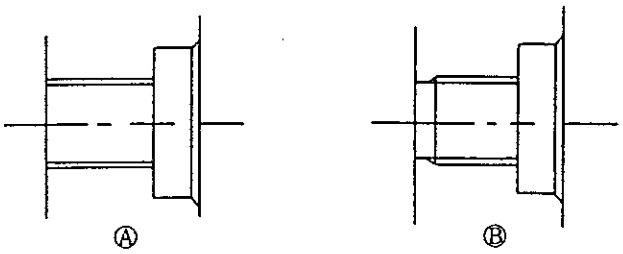
Type	Displayed screw type	Screen display (Abbreviation)
TC-215	1. Metric coarse screw threads	MC
	2. Metric fine screw threads	MF
	3. Unified coarse screw threads	UNC
	4. Unified fine screw threads	UNF
	5. National Pipe Thread Standard	NPS
	6. British Standard Pipe Thread	BSP
	7. British Standard Whitworth	BSW
	8. British Standard Fine Thread	BSF
	9. British Association	BA
TC-225	In addition to the above nine types, 10. American Standard Taper Pipe Threads for general use.	NPT

- Pitch and threads/inch (d in Fig. 5-6)
  Metric screw = Pitch
  Inch screw = Threads/inch

- CBOR ϕ (D in Figure 5-6) Set the counter bore diameter.

- CBORD (ℓ_2 in Figure 5-6) Set the counter bore depth.

- DRLD Set hole sizes before tapping.
  Through hole work (ℓ in Fig. 5-6)



The shape after work under the condition of (primary hole depth) > (depth) (A) is different from that under the condition of (primary hole depth) $\leqq$ (depth) (B).

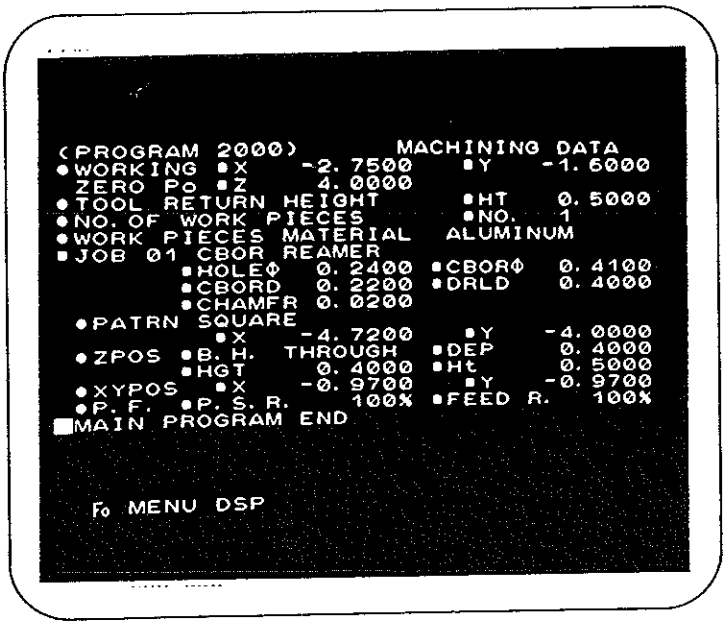
Blind hole work Taking into account the ineffective length, set the size.

* AS for blind hole, when the set value of the primary hole depth is less than or equal to the set value of the depth in the counter , the parameter allowable depth of the hole size before tapping become effective.
(For details, refer to Section 7-2-1, Parameter (Switch).)

■ CHAMFR (C in Figure 5-6) Sets chamfering size. When chamfering is unnecessary, set the size to 0.

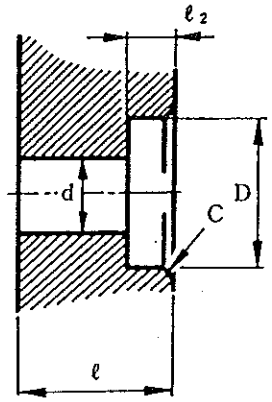
For the subsequent settings of the pattern, Z-axis position, XY position, peripheral speed, and feed; see Item 5-1. "Center hole machining".

5-7. CBOR REAMER work



Set the counter bore reamer by using the [7] and [ENTER] keys.

Through hole work



Blind hole work

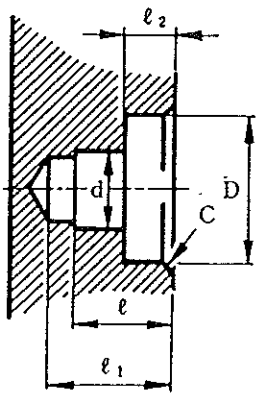


Figure 5-7

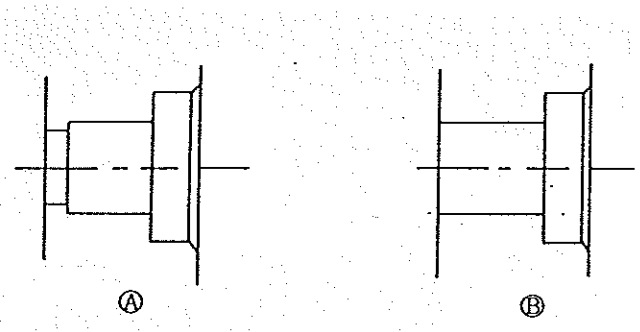
- HOLE ϕ (d in Figure 5-7) Set the reamer diameter.

* According to the setting reamer diameter and setting value of reamer finish allows 1 and 2, the primary drill diameter is automatically set.

- CBOR ϕ (D in Figure 5-7) Set the counter bore diameter.

- CBORD (ℓ_2 in Figure 5-7) Set the counter bore depth.

- DRLD Set the primary hole depth of the reamer work.



The shape after the work under the condition of (primary hole depth) is larger than (depth) Ⓐ is different from that under the condition of (primary depth) is less than or equal to (depth) Ⓑ.

Taking into account ineffective length of the reamer in the blind hole work (ℓ_1 in Figure 5-7), set the value.

* Only in the blind hole, when the setting value of the primary hole depth is less than or equal to the depth, allowable depth of the parameter reamer primary hole becomes effective.

(For details, refer to Section 7-2-1, Parameter (Switch).)

■ CHAMFR (C in Figure 5-7) Set the chamfering size. When chamfering is unnecessary, set the size to 0.

For the subsequent settings of the pattern, Z-axis position, XY position, peripheral speed, and feed; see Item 5-1. "Center hole machining".

5-8. MILLING

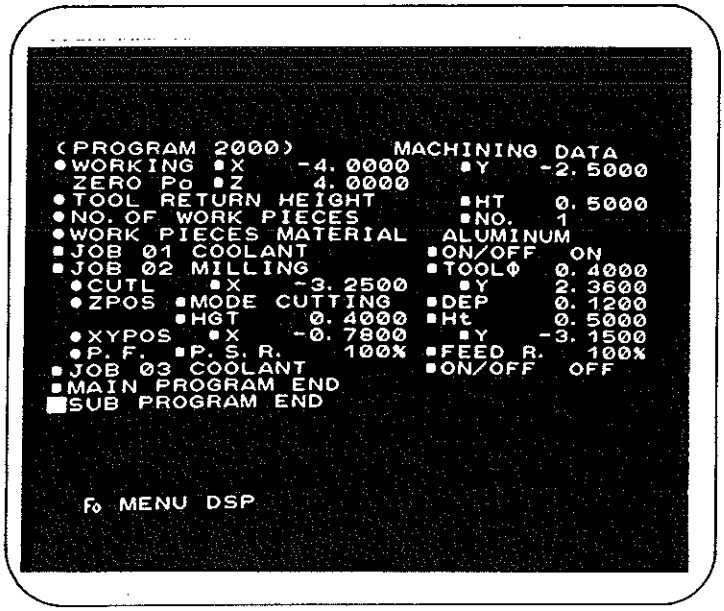
Set the top face cutting by pressing the [8] and [ENTER] keys, and the teaching area will display the following:

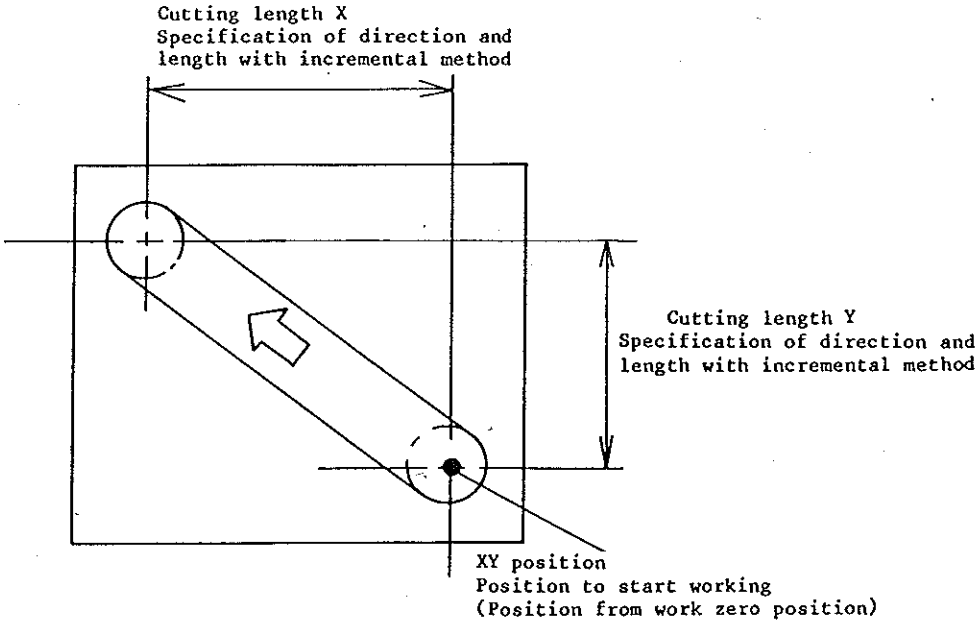
Straight line: plus
R cutting: minus

When tool diameter is set with plus (without sign), straight line cutting is specified, and when tool diameter is set with minus (with minus sign), R cutting is specified. Necessary item is required to be entered.

5-8-1. Straight line cutting (When setting tool diameter without sign)

ex)





■ TOOL ϕ: Set tool diameter. (Wisthout sign)

o Cutting length: Set moving distance of tool.

* Set the direction with the incremental method.

ex)

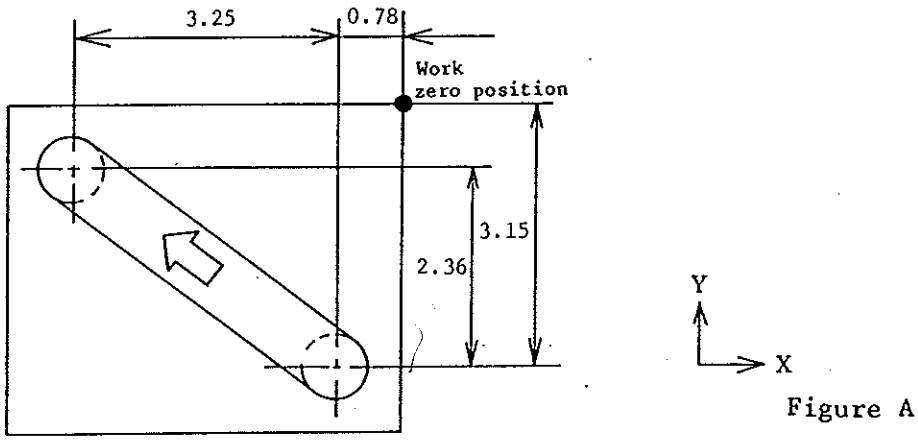


Figure A

When the tool moves in the arrowed direction in Figure A, the cutting length is specified as (X, Y) = (-3.25, 2.36).

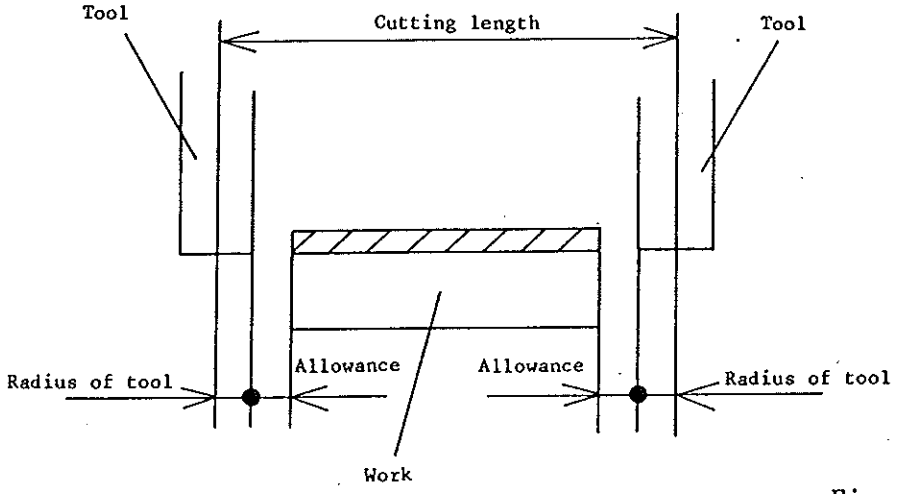


Figure B

(Caution) If the cutting length is set only to the working width when top face of the above portion with slant lines is cut, some portions in the work side may not be cut fully. As shown in the Figure, set the value with allowance.

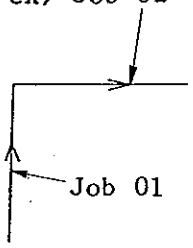
o Z POS

■ MODE: Set whether the in-position check is performed around the intersection point of the X and Y axes at the time of consecutive cutting.

Select the mode from teaching areas of:

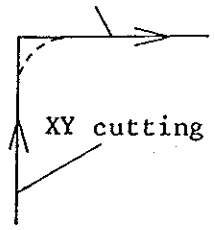
1. Cutting mode 2. Position check mode (In-position check)

ex) Job 02



When job is set as shown in the figure, the mode is specified in the previous job, and job is set to 01 in the figure.

1. Cutting mode



—— Program instruction

--- Actual moving path of the center of tool.

Figure 1

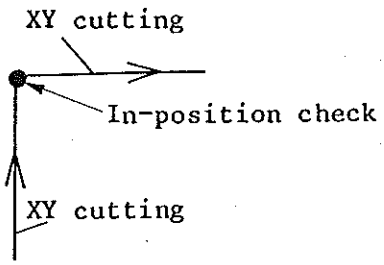
As shown in the figure, corner of sequential cutting movement may be rounded, because one job is shifted to next one (top face cutting) without in-position check.

As the cutting speed is higher or coefficient of acceleration or deceleration is greater, degree of the roundness is greater.

The cutting mode is made valid only when the consecutive XY cutting movement (refer to Fig. 1) is performed.

Therefore, tools moves consecutive corners of "XY cutting movement - Z cutting movement" in the position check mode even if the cutting mode is set.

2. Position check mode



At the end point of the axis movement, reduce the machine speed to zero, then check the machine is placed in the in-position mode (perform the position check), and enter next operation.

- DEP: Set the cutting work depth.

- HGT: Set the height from the Z axis working zero position to the work top face.
  * When the working zero position is set on the work top face, the height is set to 0.

- Ht: Set the Z axis ascending position from working zero position.

  Notice that the return position for the top face cutting work is different from that for other works.
  * When (return position) $\geq$ (work height - depth); Returned to the return position.
  * When (return position) $<$ (work height - depth); Returned to the position of (work height - depth)

Supplementary 1) When top cutting jobs continue and returning position of the previous work and work position of next job are matched, set the depth to 0 allows top cutting job to start from the same position. Therefore, in this case, the top face can be cut without Z axis movement.

Supplementary 2) When more than one workpiece is performed and Machining order is set to 3 (INDV JOB):

(1) When (TOOL RETURN HEIGHT (HT)) $\leqq$ (HGT. height)

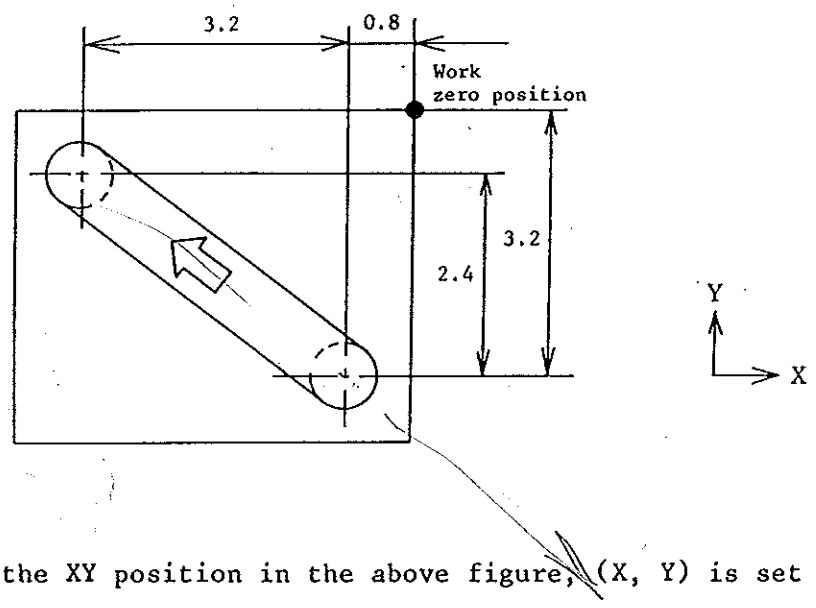
Returned to stop position forward work.

(2) When (TOOL RETURN HEIGHT (HT)) > (HGT)

Returned to TOOL RETURN HEIGHT position

o XY POS: Sets the position where the XY position work starts. Set the position from work zero position.

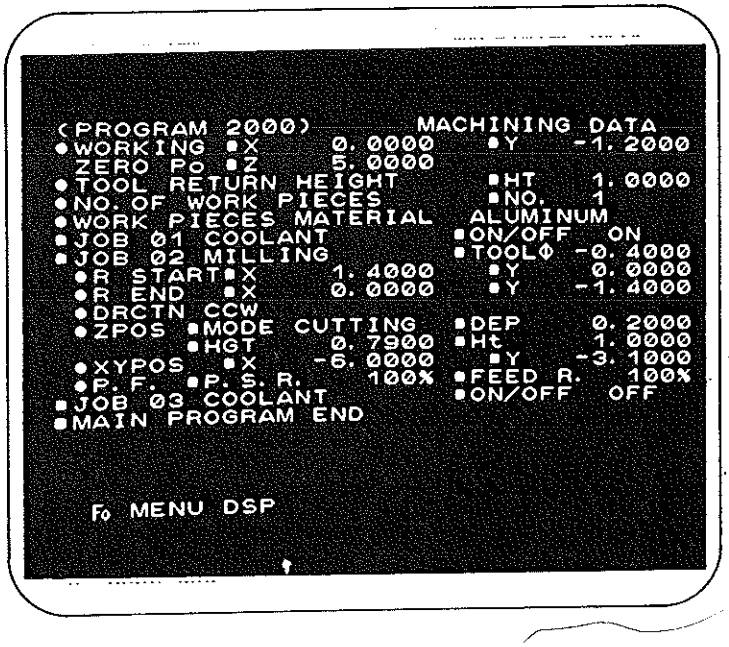
ex)

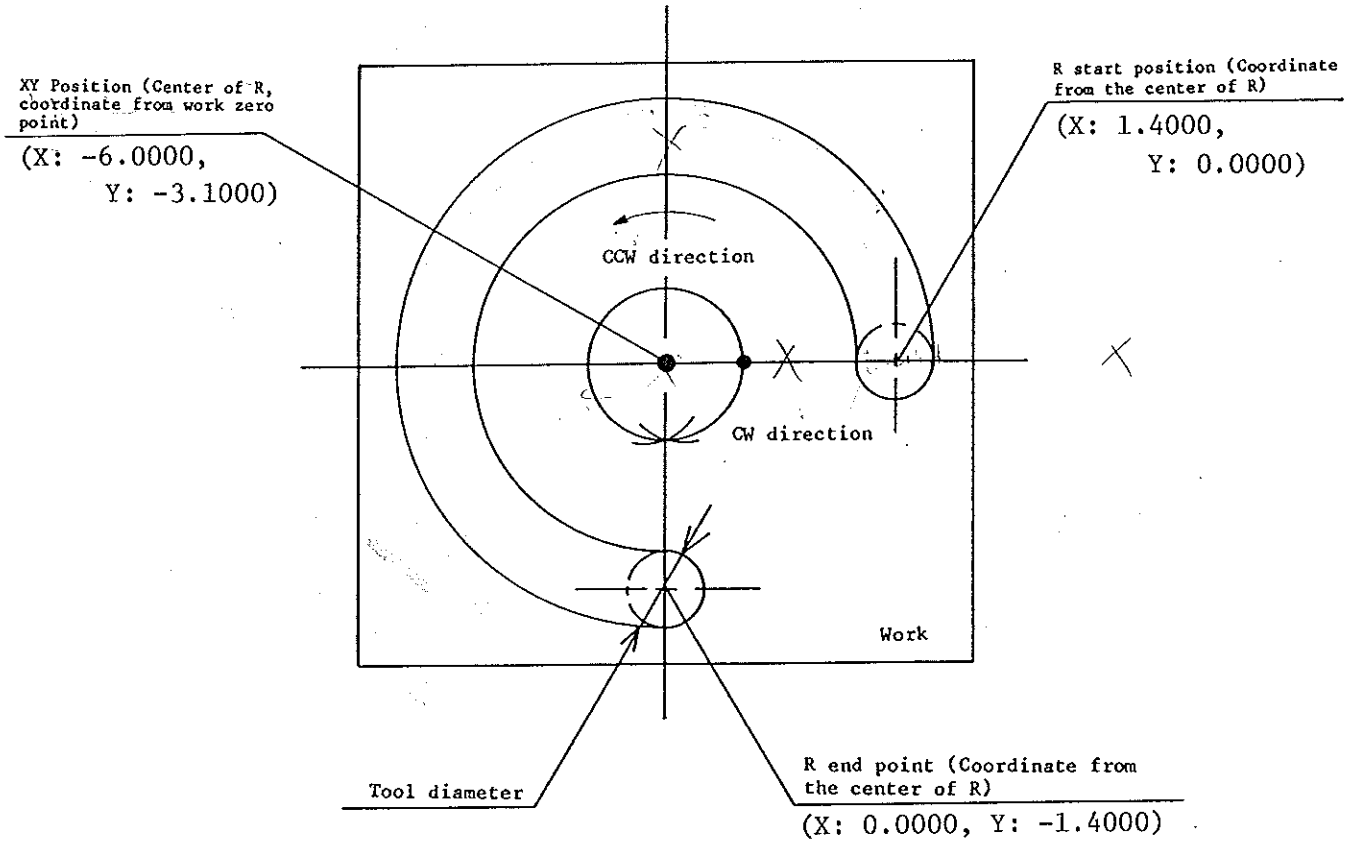


At the XY position in the above figure, (X, Y) is set to (-0.8, -3.2).

o For the peripheral speed and the feed, see Item 5-1. "Center hole machining".

5-8-2. R cutting (When setting tool diameter with minus sign)





■ TOOL ϕ: Set tool diameter with minus sign.

■ R START: Set the X and Y Cutting start position (position from the center of circle).

■ R END: Set the X and Y cutting end position (position from the center of circle).

* When the start point and end point are the same position, an entire circle of 360 degree is formed.

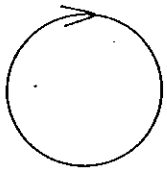
o DRCTN: Set the cutting direction.

Select the direction from teaching data area of:

1. CW 2. CCW

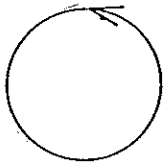
1. CW

Clockwise direction viewed from the top of motor



2. CCW

Counterclockwise direction viewed from the top of motor



o Z POS

■ Mode Set whether the in-position check is performed around the intersection point of the X and Y axes at the time of consecutive cutting work.

Select from the teaching data areas of:

1. Cutting mode
2. Position check mode (In-position check)

(For details, refer to the description of LINE cutting mode.

■ DEP: Set the cutting work depth.

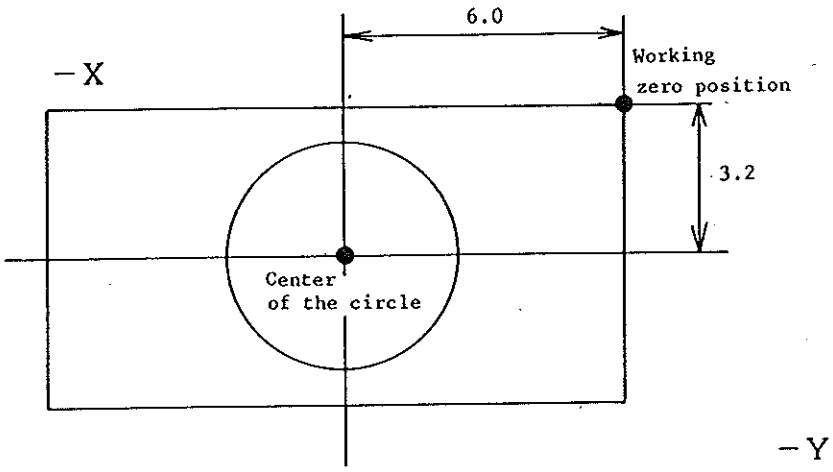
■ HGT: Set the height from working zero position of the Z axis to the work top face.

* When working zero position is set on the work top face, the height is set to zero.

■ Ht: Set the height from working zero position of Z axis. (For details, refer to the return description of the straight line cutting.)

o XY POS: Set the center position of the circle.
Set the position from the working zero position.

ex)



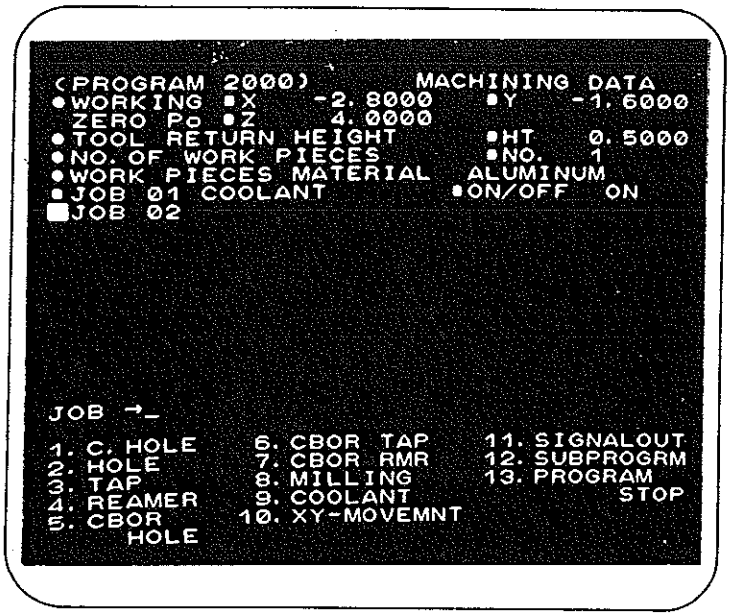
The XY position in the above figure is specified as (X, Y) = (-6.0, -3.2).

* When working zero position is assumed as the center of the circle, the setting value is 0.

o For the peripheral speed and the feed, see Item 5-1. "Center hole machining".

5-9. COOLANT

Set the coolant by pressing the [9] and [ENTER] keys.



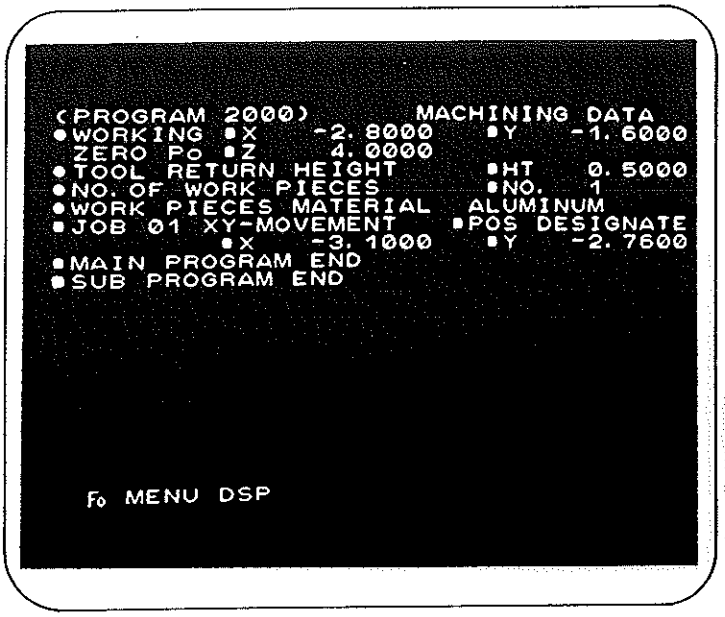
Set whether to open or close the coolant valve.

Data is selected from the teaching data area of:

1. ON 2. OFF

5-10. XY-MOVEMENT

Set the XY movement by pressing the [1] [0] and [ENTER] keys.



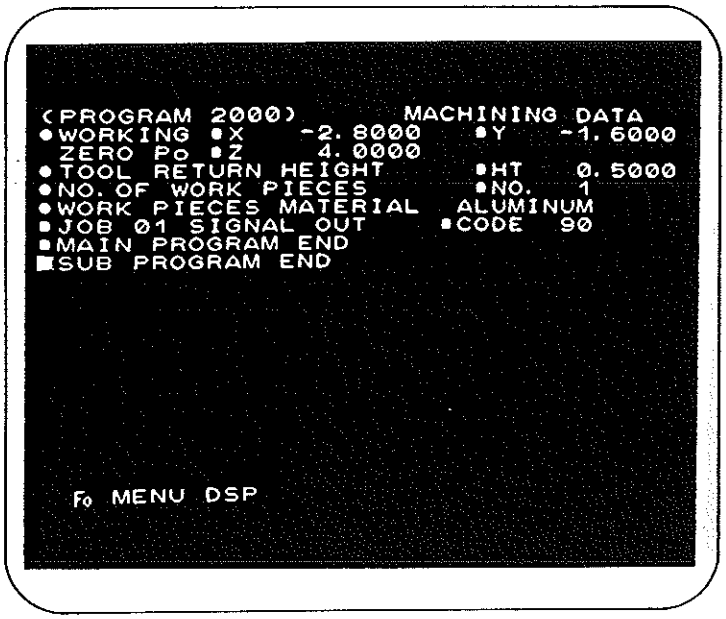
Move the X and Y axes to the position displayed in the teaching data areas of:

1. XY Designation
2. Machining zero return
3. Working zero return

Select data from among the above items.

5-11. SIGNALOUT

Set signal output by pressing the [1] [1] and [ENTER] keys.



When external functions are necessary for the working run, the machine is operated by signal output.

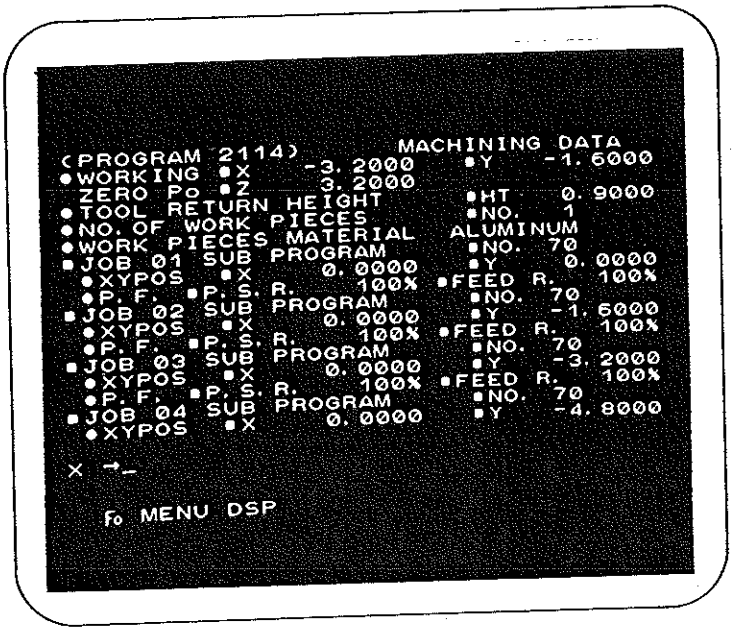
Signals are put out in two-digit BCD codes (binary decimal number). When the FIN signal is returned to the CNC equipment, the operation ends.

Codes 01 to 89 (Exclude 3, 4, 8, 9, and 30.)
Codes 90 to 99 are used for calling the magazine numbers.

Code	Magazine No.	Code	Magazine No.
90	10	95	5
91	1	96	6
92	2	97	7
93	3	98	8
94	4	99	9

5-12. SUB PROGRM

Set subprogram by pressing the [1] [2] and [ENTER] keys.



The machine can edit subprogram as well as main-program.

8-4-4. The subprogram is used to omit a part of program addition to avoid repeating the same work as in the case of sampled program edit and sampled work using subprogram (shown in the above screen).

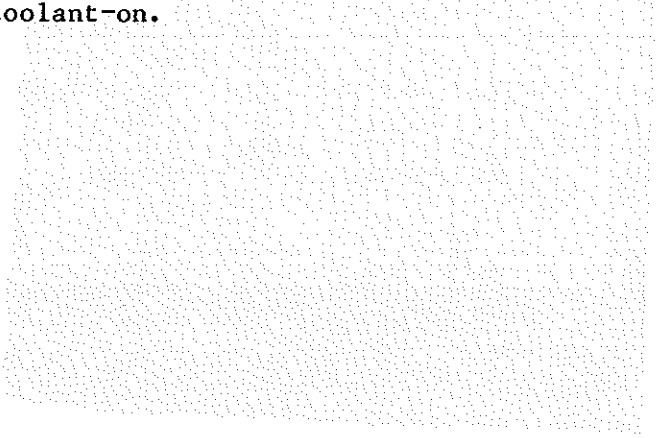
- NO.: Set subprogram number from among 77 to 99. (Work is set later.)
- XY POS: Set the work start position (position from work zero position) of the X and Y axes.
- P.S.R.: Command the peripheral speed for the cutting tool condition with percentage. (See Item 5-1. "Peripheral speed and feed for center hole machining".)
- FEED R: Command the feed rate per revolution for the cutting tool condition with percentage. (See Item 5-1. "Peripheral speed and feed for center hole machining".)

Set the contents of the subprogram process after the main program end. (See Item 5-15. "Subprogram process".)

5-13. PROGRAM STOP

Set the program stop by pressing the [1] [3] and [ENTER] keys.

Program execution can be stopped temporarily. At this time, spindle revolution and cutting oil supply are stopped. Therefore, when cutting oil supply is necessary, set the job for coolant-on.

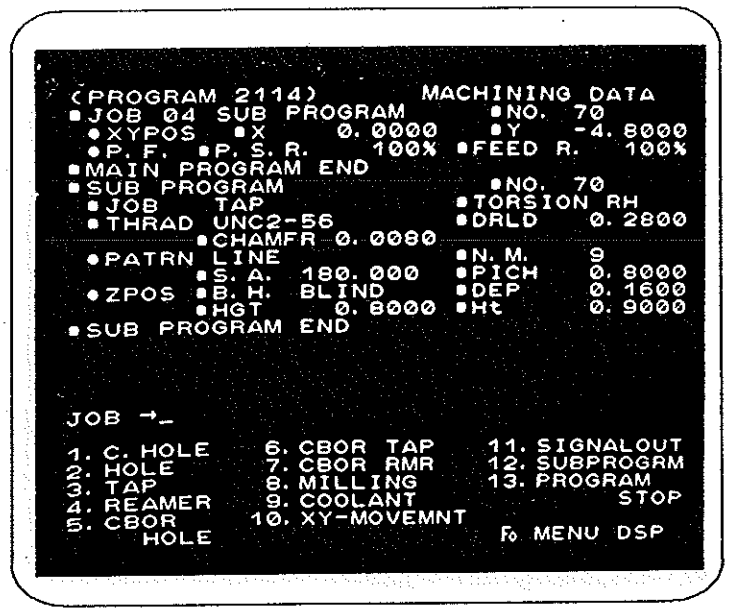


5-14. MAIN PROGRAM END

Set "main program end - 99" when the main-program setting completes.

5-15. SUB PROGRAM

When subprogram processes are included in the main program, set the subprogram number and the items necessary for machining for each process.



The subprogram includes the center hole, hole, tap, reamer, counter bore hole, counter bore tap, counter bore reamer, and face milling processes; and the items are the same with those of the main program.

Set "subprogram end -99" when the subprogram setting completes.

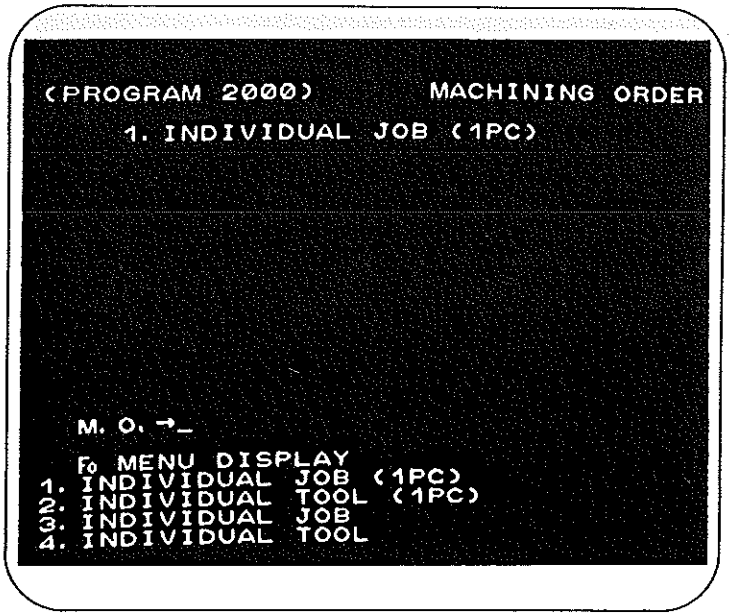
8-3-2. Setting of MACHINING ORDER

Set the machining order of the edited program.

* The setting can be omitted when the machining order data stored in the memory is used.

° To change the machining order

Make the [EDIT] mode and the PROGRAM PROTECT switch ON. Press the [2] and [ENTER] keys to set the number 2 in the menu and set the program number.



The machining order set according to that of the parameter 1 in the memory data is displayed.

Select data from the teaching data area to set it.

When setting a machining order different from that set by parameters, the machining order inherent in this program is set and "****" is displayed.

The setting completes by pressing the [F0]. (The menu display appears.)

- To return the machining order to that of the memory data:
  When setting the machining order in the above operation for change, set "-9999" instead of the menu number. (The asterisks disappear.)

  The operation completes by pressing the [F0]. (The menu display appears.)

1. Individual Job (1 piece)

A piece of work is machined for each job.

According to the machining order set in program and the order of tools for the job, work is machined. When more than one piece of work exists, the program for the first job to the last job is executed for the first work, then the program for the first job to the last job is executed for the second job, and these operations are repeated.

Though execution speed of the machining order for each job is slower than that of other machining orders, in this machining order for each job, the order and moving path are surely grasped.

In addition, no limitation exists in the order of machining data (programs).

ex) Machining pattern: Square and straight

Using tool: (1) Center drill (2) Drill (3) Tap

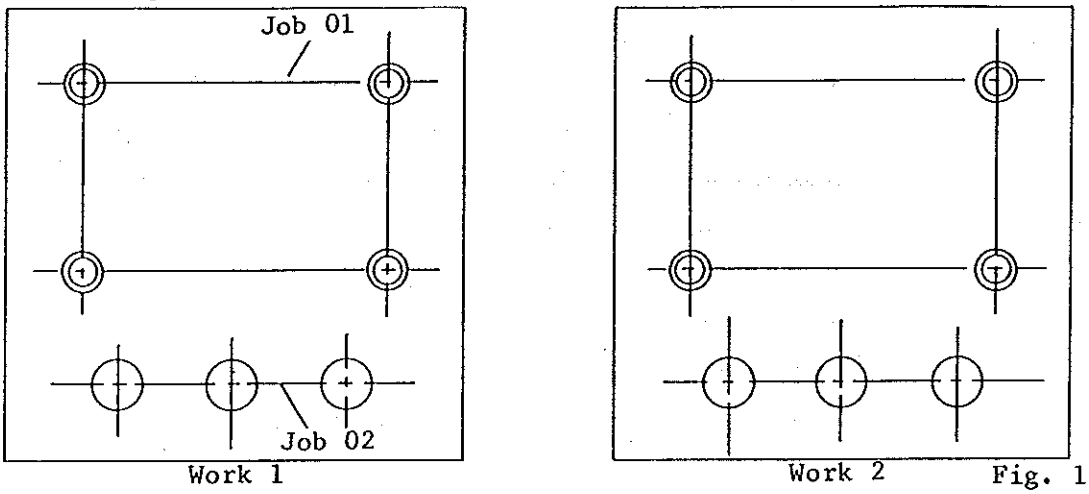
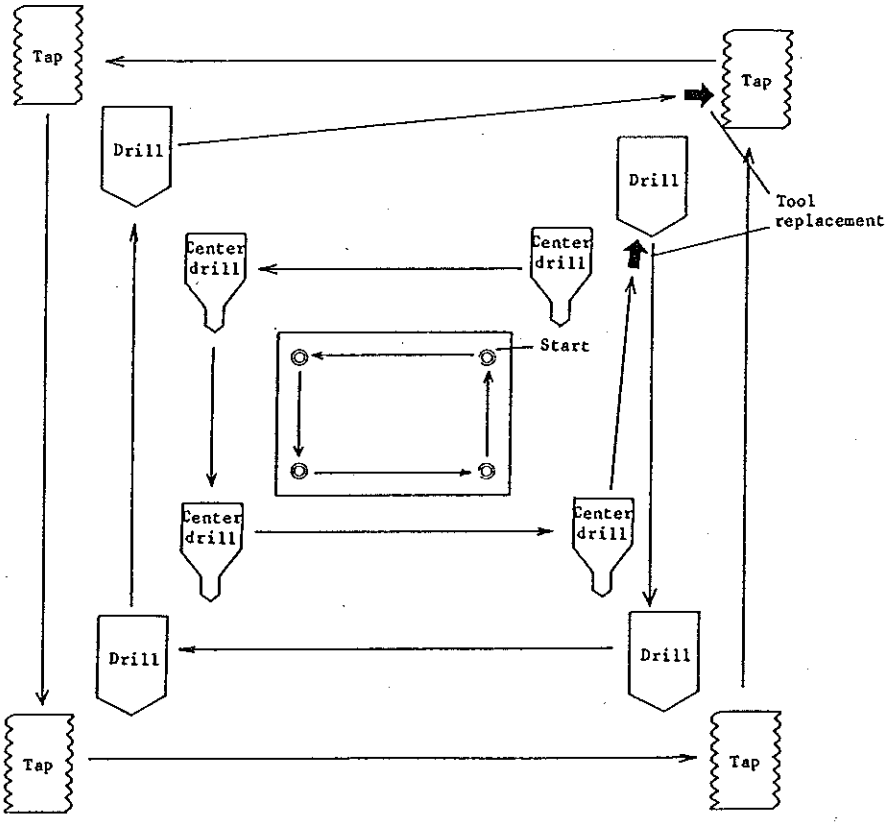


Fig. 1-1

After one piece of work is completed in the machining order, the next work (work 2) is machined.

Job 01: Square pattern



Center drill
(Odd number)
↓
Drill
(Even number)
↓
Tap
(Odd number)

Fig. 1-2

When machining pattern is used, tools in the order of odd numbers move in the setting order. However, tools in the order of even numbers move in the reverse direction. (Figure 1-2)

2. Individual Tool (1PC)

According to the machining order set in the program, work is machined. However, when a tool is used in another job and the tool is to be used in the desired job, the tool can be jumped to the job disregarding of the machining order. When more than one workpiece exists, the first workpiece is completed, then the second workpiece is finished, and these operations are repeated.

When machining pattern is used, all tools are machined in the order of the pattern.
(Toward the arrowed direction in Figure 2-2)

ex) Machining pattern: Square
Using tool: (1) Center drill (2) Drill (3) Tap

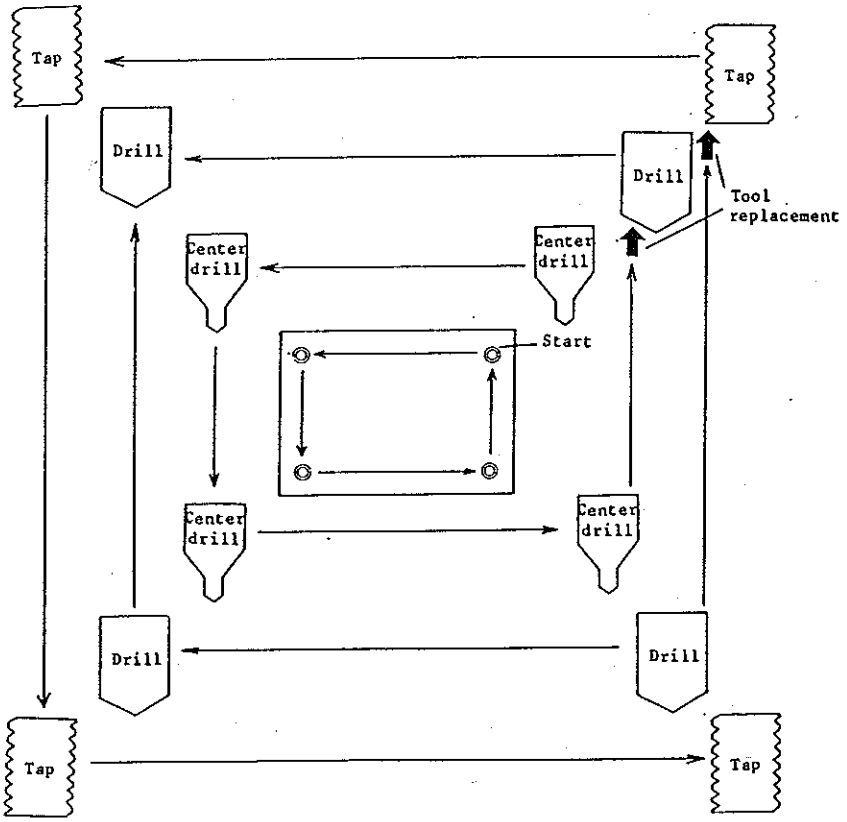


Fig. 2-2

As mentioned above, work is machined. When more than one workpiece exists, one workpiece is completed, then another workpiece is machined.

The execution speed of the machining order for individual tool (1PC) is faster than that for individual job (1PC). However, when more than one workpiece exists, the execution speed of the machine order for individual job is faster. The feature of individual job (1PC) is to finish jobs one by one.

The order of machining data (programs) is limited.

* Non-machining job cannot be entered between machining jobs.

ex) Job 01 Tap
Job 02 External signal output --- Inhibited
Job 02 Drill

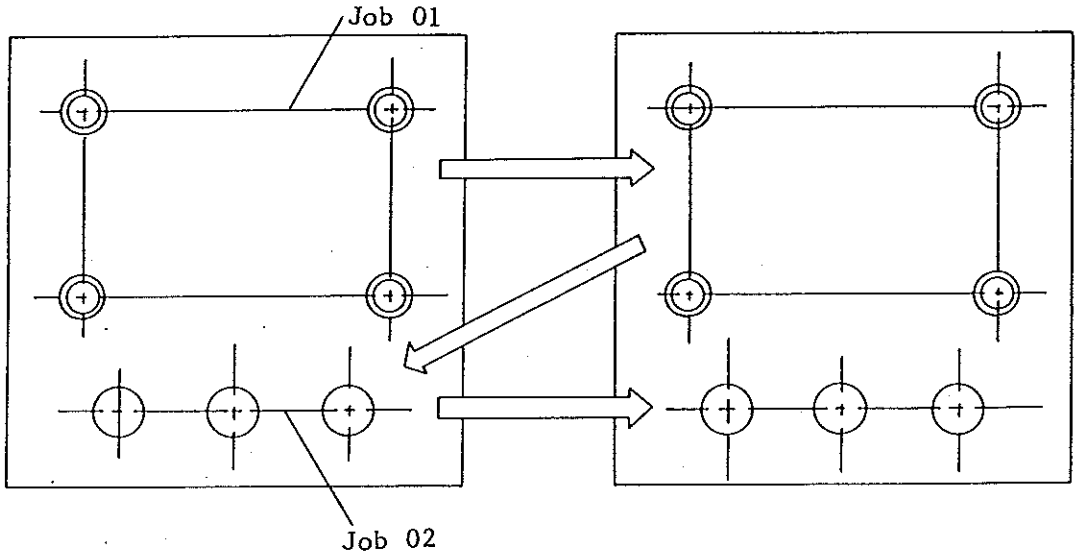
3. Individual Job

All pieces of the same work are machined.

According to the machining order set in the program and the order of tools, workpieces are consecutively machined.

When more than one workpiece exists, the tool of the job in the first work is used, then the same tool of the job in the next work is used. As the machining order of the individual job (1PC), when a work pattern is used, tools with odd number are used in the normal order, and tools with ever number are used in the reverse order.
(1. Refer to Figure 1-2, Individual Job (1PC).)

ex) Machining pattern: Square, straight line
Using tools: (1) Center drill (2) Drill (3) Tap



The workpieces are finished using the tools in the following order: the first tool of job 01 in work 1, the first tool of job 01 in work 2, the second tool of job 01 in work 2, then the second tool of job 01 in work 1.

The execution speed of the machining order of individual job is lower than that of individual tool. However, when more than one workpiece exists, the execution speed of individual job is faster than that of individual job (1PC), and machining order and moving path can be grasped to a certain degree.

The order of machining data (programs) is limited.

* Non-machining job cannot be entered between machining jobs.

ex) Job 01 Tap
Job 02 External signal output --- Inhibited
Job 02 Drill

4. Individual Tool

All workpieces are machined for each tool.

According to the machining order set in the program, work is machined. However, when a tool is used in another job and the tool is to be used in the desired job, the tool can be jumped to the job disregarding of the machining order. (Refer to Figure 2-2, Individual tool.)

When more than one workpiece exists, the first workpiece is completed, then the second workpiece is finished. Therefore, all workpieces are finished at a time.

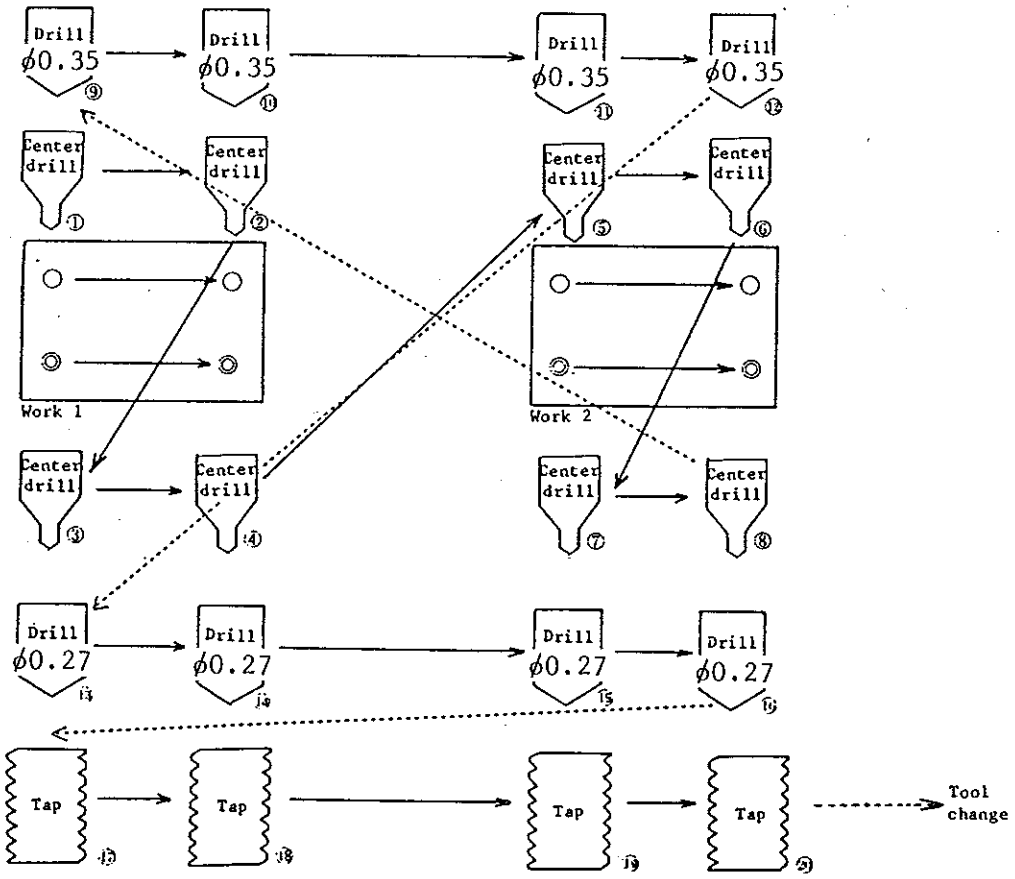
When machining pattern is used, all tools are machined in the order of the pattern.

(Toward the arrowed direction in the figure below.)

ex) Machining pattern: Straight line

Job 01: (1) Center drill (2) Drill ϕ0.35

Job 02: (1) Center drill (2) Drill ϕ0.27 (3) Tap M8



When one workpiece exists, the execution speed of machining order of individual tool is same as that of individual tool (1PC). However, when more than one workpiece exists, the execution speed of machining order of individual tool is fastest.
In addition, the order of machining data (programs) is limited.

* Non-machining job cannot be entered between machining jobs.

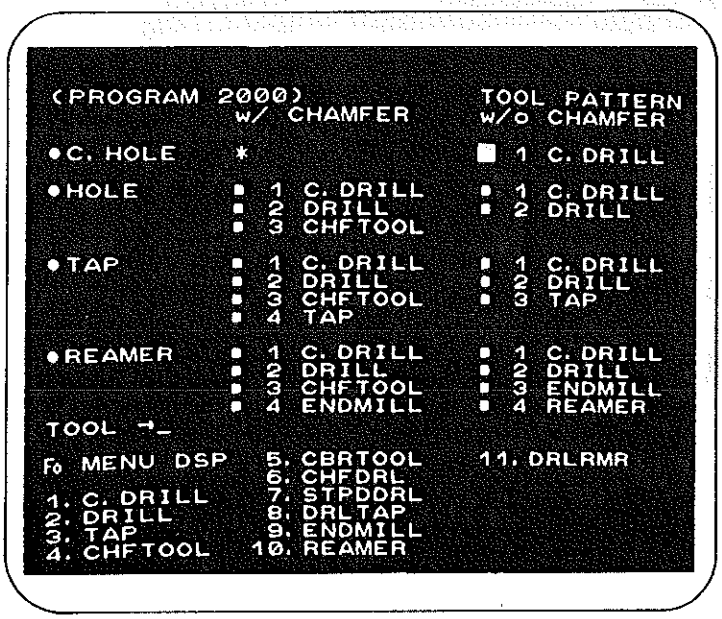
ex) Job 01 Tap
Job 02 External signal output --- Inhibited
Job 02 Drill

8-3-3. Setting of Tool Pattern

Set the tool pattern necessary for the last job of edited program.

When requested to be changed press the [EDIT] key and turn on the PROGRAM PROTECT switch.

Set menu by pressing the [3] switch and set the program number.



The screen displays the tool pattern for cutting tool condition of memory data.

* When tool pattern of the memory data is accepted without change, it can be omitted.

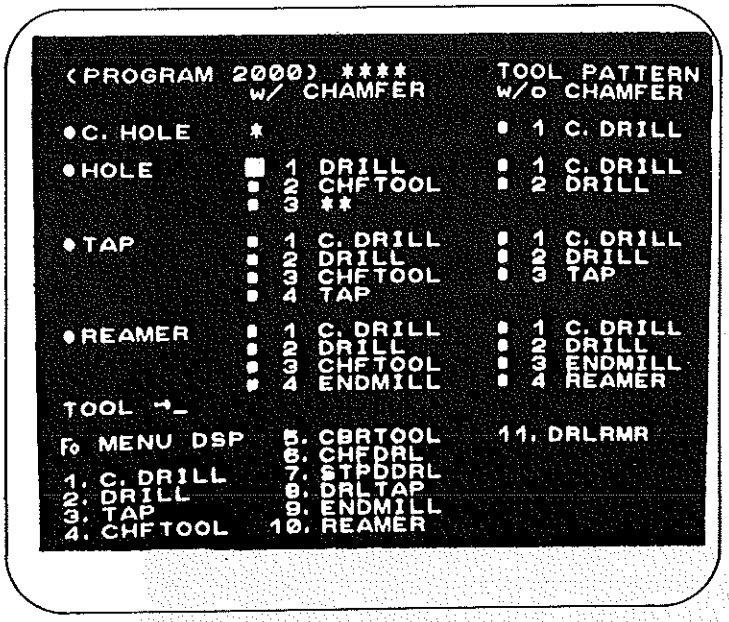
When requested to be changed.

Set the cursor to the position to be changed by using the [⇧ CURSOR] and [⇩ CURSOR] keys, then select the desired tool from the teaching data areas.

(For details, refer to Section 7-5-1, Tool Pattern.)

(Four asterisks (****) is displayed to the right of program number.)

For completion, press the [Fo] key. (The display returns to the menu screen.)



The center drill is omitted from the tool pattern of drill work in screen A.

The asterisks to the right of the program number indicate that the displayed tool pattern is specified only for the program.

When returning to tool pattern of the memory data.
Key in [-9999] in stead of tool when setting the above tool to be changed. (The asterisks well disappear.)

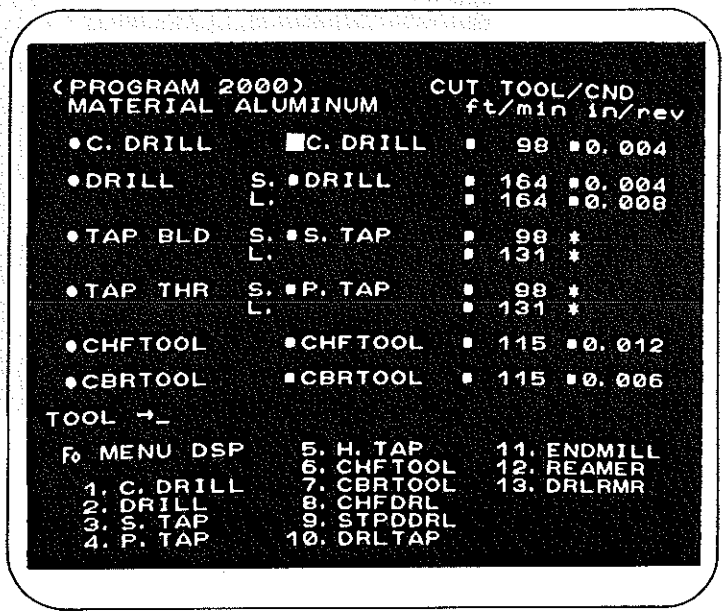
Press the [Fo] key for completion. (The display returns to the menu screen.)

8-3-4. Setting of Cutting Tool/Condition

Set the cutting condition of tools to be used for edited programs.

Press the [EDIT] key and set the memory rewrite switch to on.

Set the menu by pressing the [4] key, and set the program number.



The screen displays the conditions set in the cutting tool/condition for memory data.

* When the cutting tool/condition for memory data is accepted without change, it can be omitted.

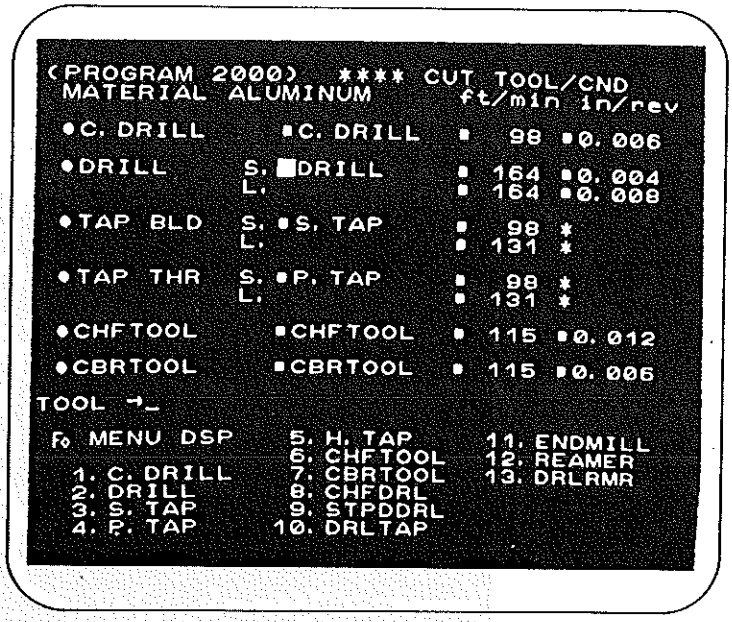
When requested to be changed.
Set the memory rewrite switch to on.

Move the cursor to the position to be changed by using the [CURSOR ⇧] and [CURSOR ⇩] keys, and set the tool change (select data from the teaching data area), and change value of cutting speed and feed amount.

(For details, refer to Section 7-5-2, Cutting Tool/Condition.)

(Four asterisks (****) are displayed to the right of the program number.)

For completion, press the [Fo] key. (The display returns to the menus screen.)



The asterisks to the right of the program number indicate that the displayed cutting tool/condition is specified only for the program.

When returning to the cutting tool/condition of the memory data.
Key in [-9999] in stead of tool, peripheral velocity and feed amount to be changed. (The asterisks well disappear.)

Press the [Fo] key for completion. (The display returns to the menu screen.)

8-3-5. Tool Assign

Assign the tools registered in the tool menu data to the tools to be used for edited program.

1. Information necessary for tool assignment
   (1) Tool pattern
   (2) Information on tool types of the cutting condition data
   (3) Hole sizes before tapping
   (4) A part of parameters
   (5) Machining data
   (6) Tool menu

1-1. Tools used for each work type

Types of tools to be used and types of work are determined by the job contents of work data and tool patterns.

* Tools available for the last shape of machining jobs

Machining	Job
Center hole	Center drill
Hole	Drill, end mill, chamfering drill, stepped drill, drill tap, drill reamer.
Chamfering	Chamfering tool
Counterboring	Counterboring tool, end mill
Ordinary hole finish	End mill, drill, chamfering drill, stepped drill, drill tap, drill reamer
Reamer	Reamer
Tap	Tap
Top face cutting	End mill

In addition, the following cases can be automatically determined according to the tool pattern contents:

(1) Center hole work is not performed without center drill.
(2) Ordinary hole work is not finished when no tool exists for the work in the reamer job.
(3) Multi-work is selected when the number of tools is short compared with machining shapes.

Center hole + face ⟶ Center drill

Hole + face ⟶ Chamfering drill

Hole + counterbor ⟶ Stepped drill

Hole + tap ⟶ Drill tap

Hole + reamer ⟶ Drill reamer

1-2. Available tools

This machine can register 13 types of tools in the tool menu, and the tools can be divided into 10 types.
(Auxiliary code registration)

Center drill	Center drill, Center drills 1 to 9
Drill	Drill, drills 1 to 9
Tap Point tap, Spiral tap, Hand tap	 P tap, P taps 1 to 9 S tap, S taps 1 to 9 H tap, H taps 1 to 9
Chamfering tool	Chamfering tool, chamfering tools 1 to 9
Counterboring tool	Counterboring tool, counterboring tools 1 to 9
Chamfering drill	Chamfering drill, chamfering drills 1 to 9
Stepped drill	Stepped drill, stepped drills 1 to 9
Drill tap	Drill tap, drill tap 1 to 9
End mill	End mill, End mills 1 to 9
Reamer	Reamer, reamers 1 to 9
Drill reamer	Drill reamer, drill reamers 1 to 9

When the tool type set in the tool area in the cutting condition of the relevant material is matched with the type of the tool menu, the tool is used. (having or not having auxiliary code, matching of auxiliary code number)
For tool assignment, be sure to match tools and their names. (Except for taps)

1-3. How to select tools when tools are assigned

When tools are assigned, they are assigned in the order of the jobs. However, when the same tools exist, they are selected as follows:

* Tool matched with the condition and having lower tool number in the tool menu
* Tool matched with the condition and determined that to be used in the previous job

Therefore, it is advisable to register the tools undesired to be used to area with larger number.

1-4. Tool Diameter for Primary Hole in the Tap and Reamer (Including Counterboring) Jobs

(1) Hole sizes before tapping

According to tap sizes, drill whose diameter is registered to the hole sizes before tapping.

(2) Reamer primary holes 1 and 2 (Ordinary hole finish)

* Primary hole 1 (Without roughing work or ordinary finish)

(Hole diameter) - (Reamer finish allow 1) = (Tool diameter)

* Assignment for ordinary hole finish

(Hole diameter) - (Reamer finish allow 2) = (Tool diameter)

1-5. Assignment and Machining Order

* If non-cutting job (such as external signal output, coolant and axis movement) exists between cutting jobs in sequence of machining data, only "machining order 1 and individual job (1PC)" can be assigned.

* Since the effective cutting length for the seat depth depends on whether a counterboring tool exists before job, there is a chance that tools can be assigned but they cannot be checked.

1-6. Tool dimension check for tool assignment

(1) Tool length

* Check if the tool is not touched to workpiece or fixture jig when the Z axis is at zero position.
* Check if the table is not cut with the depth.

(2) Life

* Check if life is remained.

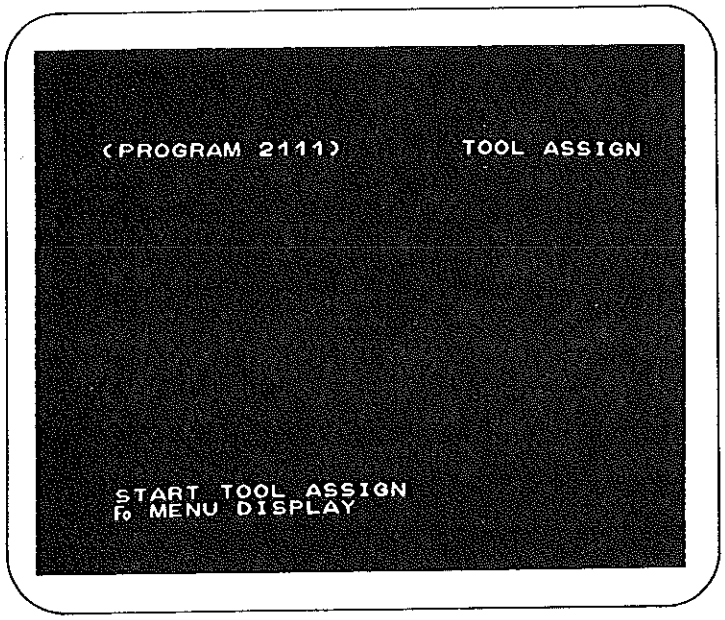
(3) Dimension of each job (diameter, length, angle)

* Center hole job
* Hole job
* Tap job — In the case of blind hole, drill tap cannot be used.
* Reamer job
* Counterboring hole job
* Counterboring tap job — In the case of blind hole, drill tap cannot be used.

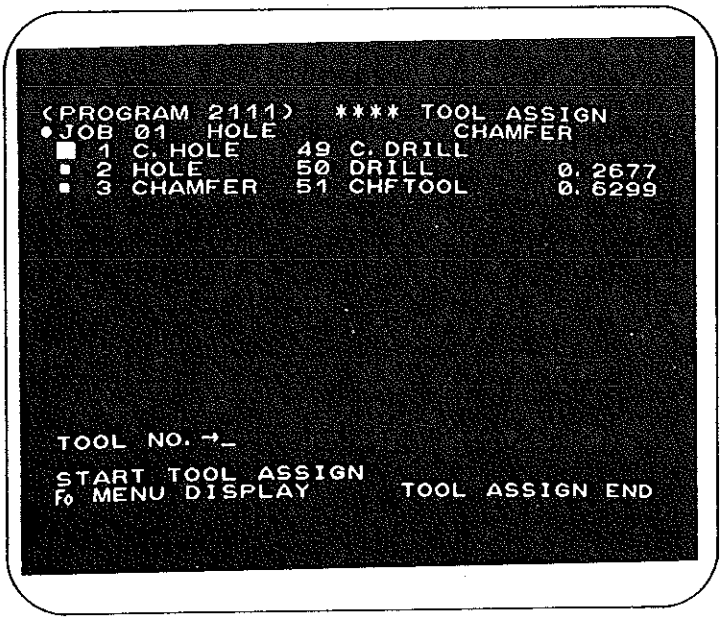
2. Assign operation

Press the [EDIT] key, and set the memory rewrite switch to on.

Set the tool assign by pressing the [5] key, and set the program number.



Press the START key.



Description of screen

■ 2 HOLE 50 DRILL 0.2677

In this screen, drill with tool number 50 in the tool menu and 0.2677 are assigned to the fourth tool (final shape tool) HOLE in job 01.

When message "Tool has been assigned" appears, press the F0 key for completion.
(The display returns to the menu screen.) Set the memory rewrite switch to off.

* For a program which contains only such processes that do not require tools, tool assignment is necessary.

For a log program, program display might require more than one screen page. In this case, move the screen by pressing the ⇩ PAGE key.

o The number of tools to be assigned

Tools necessary for machining data are assigned disregarding the number of tools. However, the number of magazine tools to be assigned is limited to 10. If more than ten magazine tools are assigned, the machine cannot operate normally after the assignment.

Therefore, the following items are necessary to be adjusted sot that the number of tools are within 10.

* Job change
* Forced assignment
* Change of tool list

3. Forced tool assign

When the tool that cannot be assigned in ordinary tool assignment or the substitute is required to be used, or when unnecessary job is omitted, forced tool assignment is made.

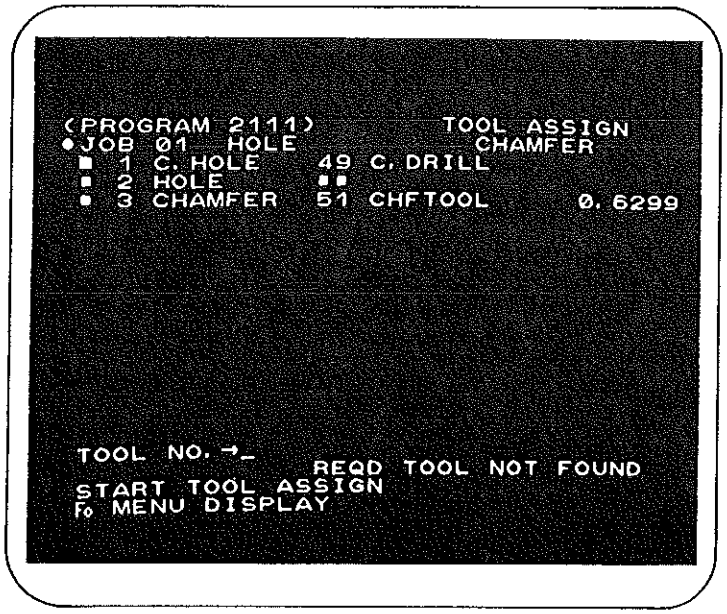
* When forced assign is made, machining data which is entered at the time of program edit is ignored, and data forcibly assigned in the tool menu is given priority.

* When the contents of machining data are not matched with those of tool data in the memory run mode (in the case after assignment or of change of data in the tool menu), data in the tool menu is given priority. When data in the tool menu is changed, check the data.

* Tools which can be forcibly assigned according to machining contents

Machining type	Tool
Center hole	Center drill, drill, chamfering drill, stepped drill, drill tap, drill reamer
Hole	Drill, end mill, chamfering drill, stepped drill, drill tap, drill reamer
Chamfering	Chamfering tool, center drill, drill, chamfering drill, stepped drill, drill tap, drill reamer
Counterboring	Counter bore tool, end mill
Ordinary hole finish	End mill, drill, chamfering drill, stepped drill, drill tap, drill reamer
Reamer	Reamer, end mill
Tap	Tap
Milling	End mill
Center hole + face	Center drill
Hole + face	Chamfering drill
Hole + counterbor	Stepped drill
Hole + tap	Drill tap
Hole + reamer	Drill reamer

3-1. Forced assign operation

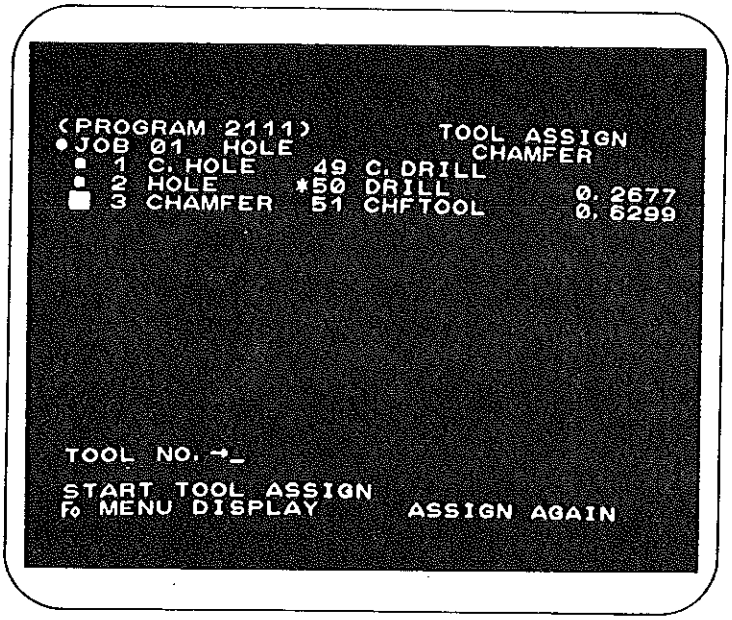


Press the TOOL MENU key and check the tool No. of substitute tool.

Press the PRGRM key (returns the display to the above menu). Set the memory rewrite switch to on.

Move the cursor to the forced assign by using the ⇧ CURSOR and ⇩ CURSOR keys.

Set the tool number of the tool menu.



If forced tool assign is made, tool number is marked with an asterisk '*', and message 'Assign again' is displayed on the screen.

* Then, assignment is completed by the operation of 'forced assignment tool check ⟶ reassignment'.

3-2. Example of Forced Tool Assign (4. Refer to sampled assign display)

(1) Forced assignment of 1 function tool

The forced assignment can be made by inputting the tool number. (See Item 4-(2).)

■	1	Center hole	15 Center drill
■	2	Hole	20 Drill tap
■	3	Tap	
■	4	Chamfering	08 Chamfering tool

When the assignment is complete through the above operation, if "drill 16" is set to the 20 drill tap hole and "1 function tool" is assigned, the assignment of either function tool with the composite function is canceled as follows:

15 Center drill
*16 Drill
* ■ ■
08 Chamfering tool

On the contrary, when the tap "13" is set to the tap of the 20 drill tap and "1 function tool" is assigned, the assignment of either function tool with the composite function is canceled as follows:

15 Center drill
*20 Drill tap
*13 P tap
08 Chanfering tool

(2) Forced assign of multi-function tool

By assigning one tool number (multi-function tool) to two jobs, one tool can be used for two jobs. (Refer to 4-(4).)

(Note) Assign high-order tool to low-order tool.

(3) Deletion

Unnecessary job is deleted so that it cannot be machined. (Refer to 4-(2) and (3).)

(Note) The tools which are forcibly assigned must be checked.

3-3. Release of forced tool assign

All tools that have been forcibly assigned are returned to the original state in the following manner:

Press the PRGRM key (displays the assign screen), and set the PROGRAM PROTECT switch to OFF.

START [-9999].

For completion, press the F0 key. (The display is returned to the menu screen.)

Set the PROGRAM PROTECT switch to ON.

4. Sampled display of assign

(1) (Program 1000) **** Tool Assign

o Job 01 Tap Chamfering

[]	1.	Center hole	01 Center drill
[]	2.	Hole	08 Drill
[]	3.	Chamfering	03 Chamfering tool
[]	4.	Tap	09 S tap

Asterisks "****" at the side of program number indicates that the assign is completed.

(2) (Program 1000) Tool Assign

o Job 01 Tap Chamfering

■	1.	Center hole	01 Center drill
■	2.	Hole	■ ■ Drill ——>Has not been assigned.
■	3.	Chamfering	03 Chamfering tool ——> Has been forcibly assigned.
■	4	Tap	*** ——> Function has been deleted.

(3) (Program 1000) Tool Assign

■	1.	Center hole	01 Center drill
■	2.	Chamfering ——>	Only the hole is machined.
■	3.	Hole 10 Drill tap ——>	Only hole is machined.
■	4.	Tap	*** Tap is not machined.

(4) (Program 1000) Tool assignment

o Job 01 Tap Chamfering

■	1.	Center hole	01 Center drill
■	2.	Chamfering	*** ——> Chamfering is not machined.
■	3.	Hole	*10 Drill tap ——> Drill tap is not made for hole or tap.
■	4.	Tap	*

Without asterisks '****' displayed, reassignment is requested.

5. Tool check

Defectives of forced assign tools are concretely displayed.

5-1. Information necessary for tool check

(1) Tool pattern
(2) Hole size before tapping
(3) Parameter
(4) Machining data
(5) Tool menu

* Notice that the tools for cutting condition are not checked.

5-2. Operation

The program assign screen is displayed.

Move the cursor to the tool to be checked by using the [⇧ CURSOR] and [⇩ CURSOR] keys.

Pressing the [TOOL ASSIGN CHECK] key checks the tool dimension.
Messages such as 'Tool diameter too large' are displayed.

* For checking of jobs without tool number of multi-function tool (refer to Section (3)-2, Chamfering tool in Sampled Assign Display), unassigned jobs, and jobs with too deleted, message 'Required Tool Not Found" is displayed.

o In response to message "Tool Dimension OK", reassign the tool.

o Defective display
If defective exists, a certain message is displayed to make jobs as instructed by machining data. In this case, however, only one message is displayed on the screen, check all messages by using the [ALARM] key.

6. Tool Reassign

If tools are not assigned in one assign operation, reassignment is made. The tools that could not be assigned are forcibly assigned or checked. When reassingment is ready,

(1) press the  key.

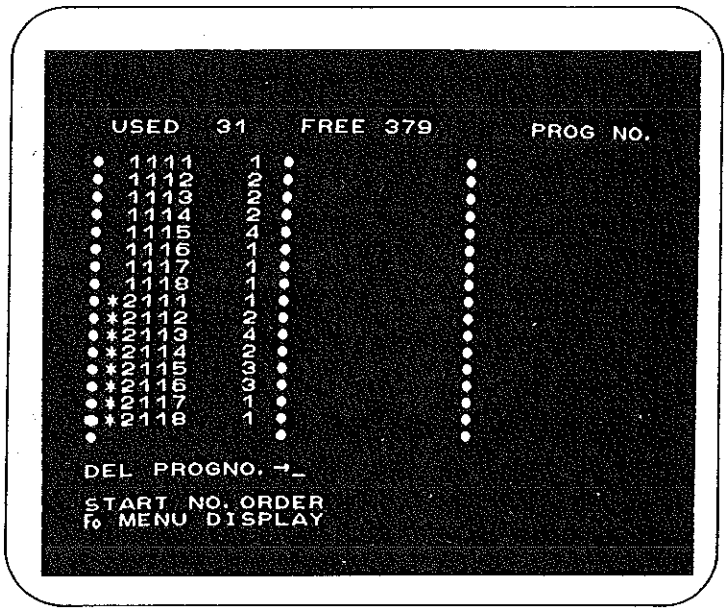
Information on jobs for the forced assignment is remained, and only the jobs without forced assignment are assigned.

(2) [-9999] Press the  key.

All data (including forced assign) is cleared, and assign is made all over again. Therefore, all data by now is erased.

8-3-6. Program and Memory Display: Order

Set program and memory display :order by using the [EDIT] and [6] keys.



* Used memory capacity

The used memory capacity is displayed in units of block. Up to 410 blocks (1 block = 128 characters) can be used.

* Remained capacity

Remained capacity of memory is displayed in units of block.

* List of memory programs

The program number edited just before is indicated by the cursor [].

The program number and the capacity are displayed.

Up to 51 programs can be stored in memory.

On the above screen, programs can be deleted, and the program number can be arranged in order.

(1) Deletion of program

Set the PROGRAM PROTECT switch to OFF.

Enter the program number to be deleted, and delete it.

Press the [F0] key. (The display returns to the menu screen.)

Set the PROGRAM PROTECT switch to ON.

(2) Deletion of whole programs

Set the PROGRAM PROTECT switch to OFF.

Key in [-9999] into program No. (1), and delete the program.

Press the [F0] key. (The display returns to the menu screen.)

Set the PROGRAM PROTECT switch to ON.

(3) Program No. order

Set the PROGRAM PROTECT switch to OFF.

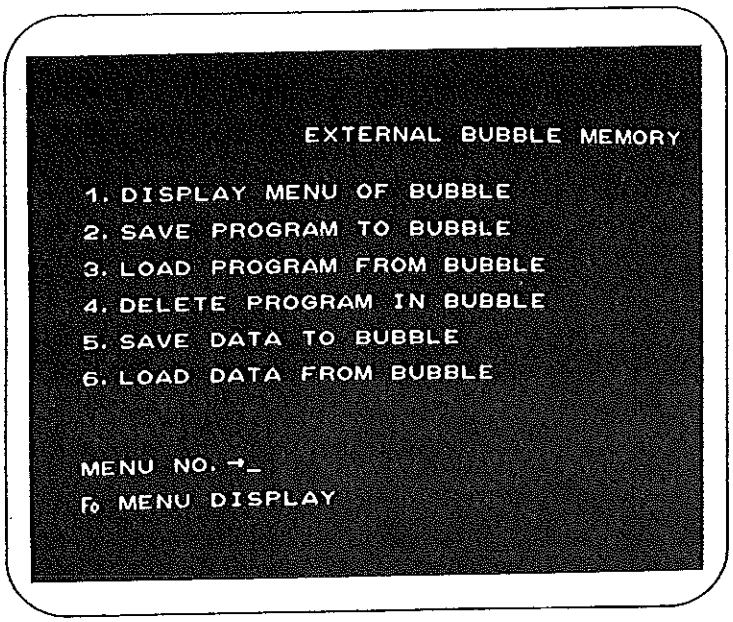
Start the program.

Set the PROGRAM PROTECT switch to ON.

* When this machine is delivered to you, program edit examples (8-4-1 to 8-4-6) and tool data necessary for the programs are stored. Delete programs, if necessary.

8-3-7. Input/Output of External Programs

- o Input of program data from the external memory unit (special accessory, bubble cassette) or the external communication unit (special accessory, Centronics or RS-232C) to this machine and output of program data from the above-mentioned external units to this machine are both possible. These input/output functions assure effective management of program data.
- o To set these external memory unit and external communication unit, use Parameter 4. (Hereinafter the internal block of this machine is abbreviated to internal, and the external memory unit and the external communication unit are sometimes abbreviated to external.)
- o Remember to put the external communication unit into the ready state for communication before starting the key operations.
- o In case the external is a bubble cassette
  Set the Program [EDIT] mode and the Menu Screen [7].

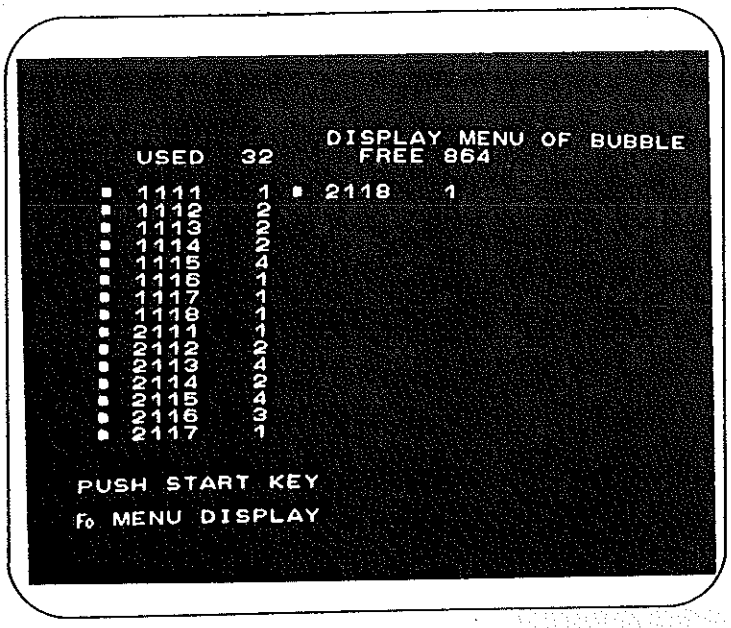


Six external program input/output items are to be displayed on the screen.

1. External Memory Display

Key-in the external memory display [1] and [ENTER] .

Push the [START] key.



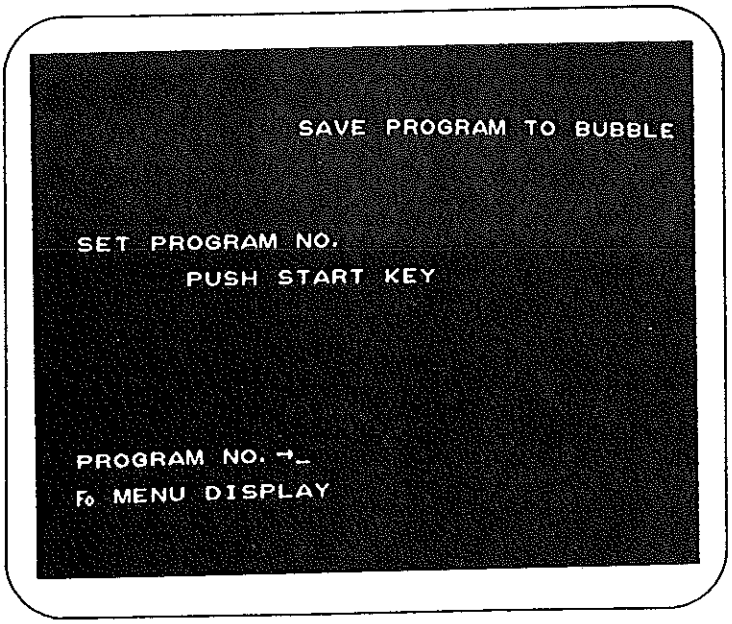
The program memory capacity of the external memory unit (buzzle cassette) is to be displayed on the screen.

2. Program Output to External

The procedure of program write from internal (this machine) to external is as follows.

Put the bubble cassette into the rewrite ready state by setting the [EXT] mode and pushing the [7] key and [ENTER] .

program output to the external [2], and push the [ENTER] key.



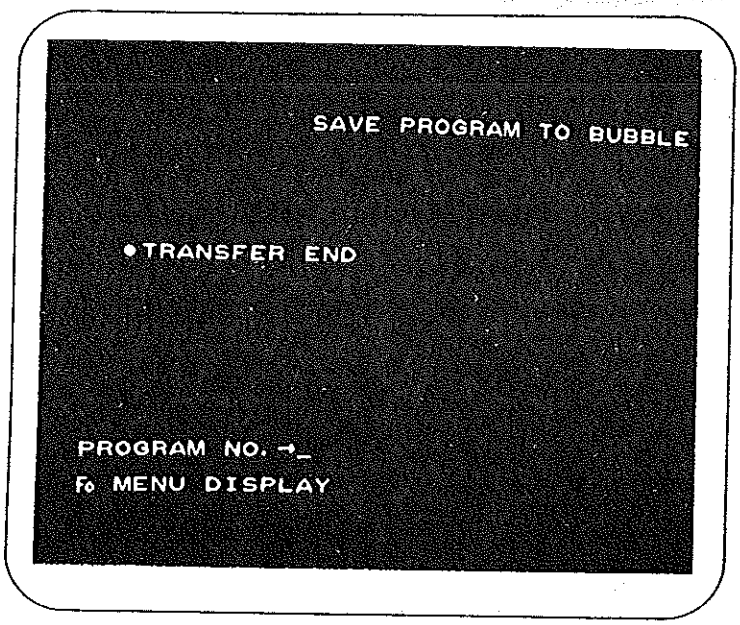
Screen 2-1

Enter the program No. of the internal memory program to be written into the external, and push the [START] key.

1 External program write by changing the program No. of an internal memory program is possible.

Ex.) Key-in [1111] (external), [-], and [2222] (external), and then push the [START] key.
In this case, the No. of the internal memory program changes from 1111 to 2222 when the internal memory program is written into the external.

2 Batch write of the internal programs into the external is possible.
Key-in [-9999], and then push the [START] key.

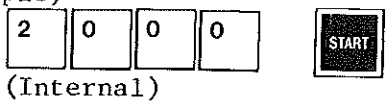


Upon completion of the program write (transfer), such messages as shown left are displayed on the screen.

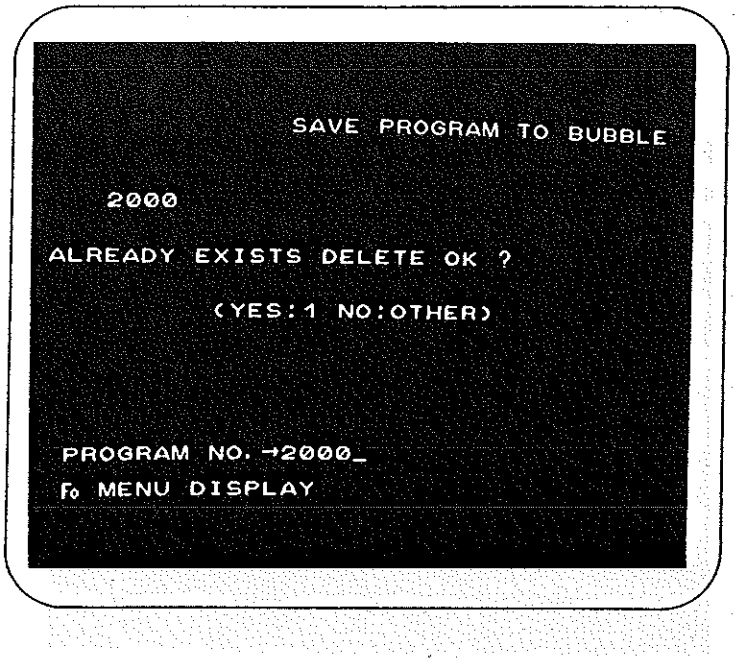
Screen 2-2

If the same program No. as of the internal program to be written already exists in the external, the following messages are displayed on the screen.

Example)



(Internal)



Screen 2-3

(1: To be transferred. Other than 1: Not to be transferred.)

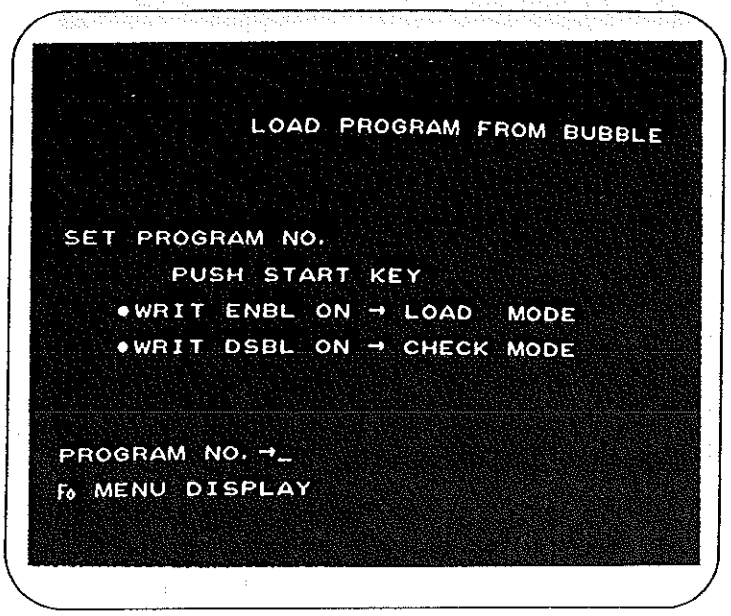
To write the internal program without regard to the messages shown above, enter the [1] key. (In that case, the external memory program No. 2000 is lost.) To stop the program write (transfer), push any other numeric key than 1.

Remember to disable the bubble cassette rewrite after completion of the program transfer operation.

3. Program Input from External

The procedure of external program read or check (confirmation of program transfer) is as follows.

Make the memory rewrite switch operative by setting the Program Edit mode and pushing the [7] key. Set program input from the external [3], and push the [EXE] key.



Screen 3-1

(1) The program is externally loaded.
Make the PROGRAM PROTECT switch OFF.
Input the external memory program number and press the "Start".

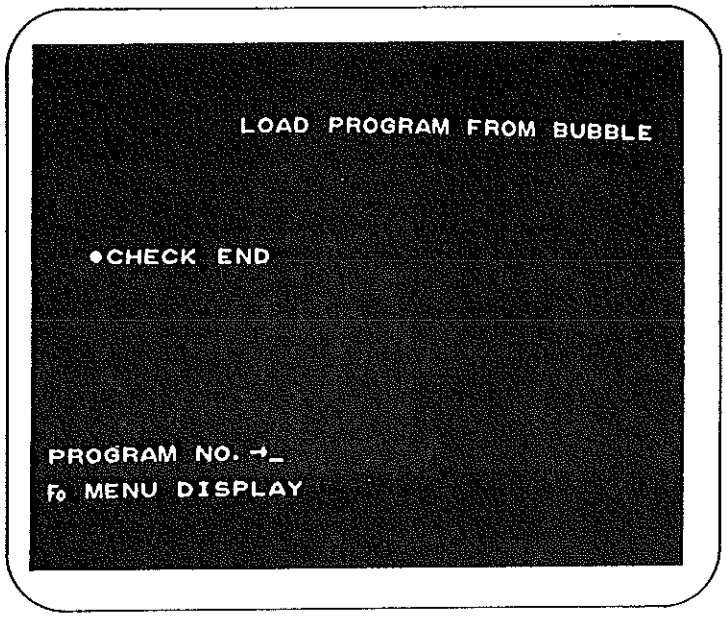
When loading is complete, the message "2. External program output display 2-2" appears.

For loading operation, if the same program number is previously stored, the message "2. External program output display 2-3" appears.
The continue loading the program, key in "1". To stop loading the program, key in a number other than "1".
When the loading is complete, make the PROGRAM PROTECT switch ON.

(2) Loading check

Make the PROGRAM PROTECT switch OFF.

Input the program number to be checked and press the START. It is checked whether or not the internal machining data is the same with the external machining data for the specified program number. When they are the same, the message of the display 3-2 appears. If they are different, the error message appears.



(1) Display 3-2

- It is possible to change the external program number and load the program.

  Key in "1111" (external), [−] and "2222" (internal) and press the START.

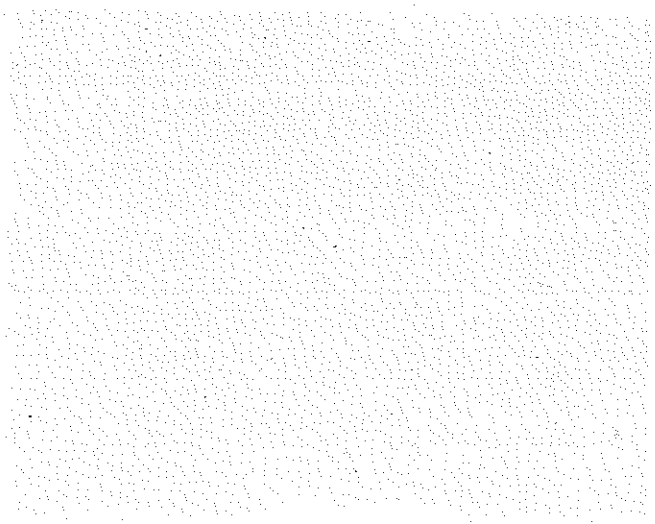
  Change the external program number "1111" to "2222" to load it in the internal memory.

- Partially put the external memory program together to load it in the internal memory.

Key in "1111" (external), [/] , and "2222" (external) and press the [START] .

The external memory program Nos. 1111 through 2222 are loaded in the internal memory. (The program numbers between 1111 and 2222 should be present.)

Same is true for the checking mode.

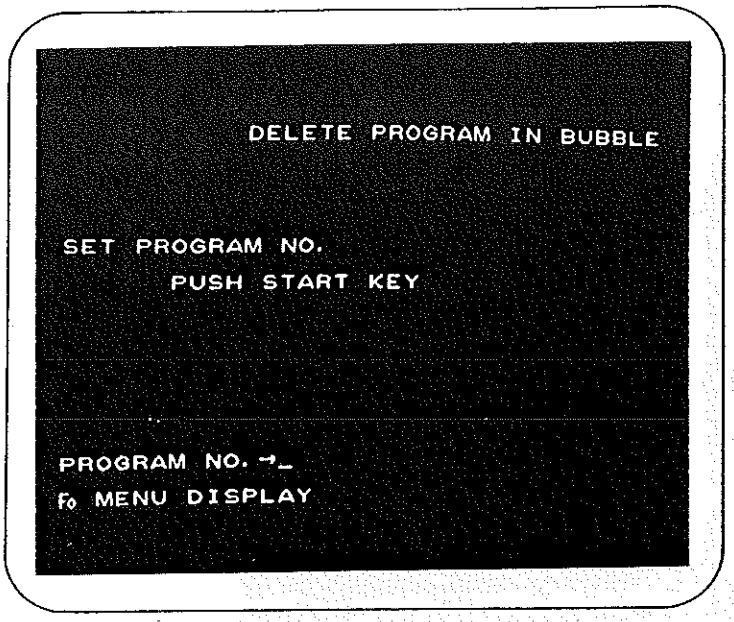


4. External Program Deletion

The procedure of external program deletion is as follows.

Put the bubble cassette into the rewrite ready state by setting the Program [EDIT] mode and pushing the [7] key and [EXE] .

Set the external program deletion from [4], and push the [EXE] key.



Screen 4-1

Enter the Nos. of programs to be deleted while confirming the program Nos. on the external memory display, and then push the [START] key.

To delete all the programs in the external memory by using a virgin bubble cassette, key-in [-9999] and then push the [START] key.

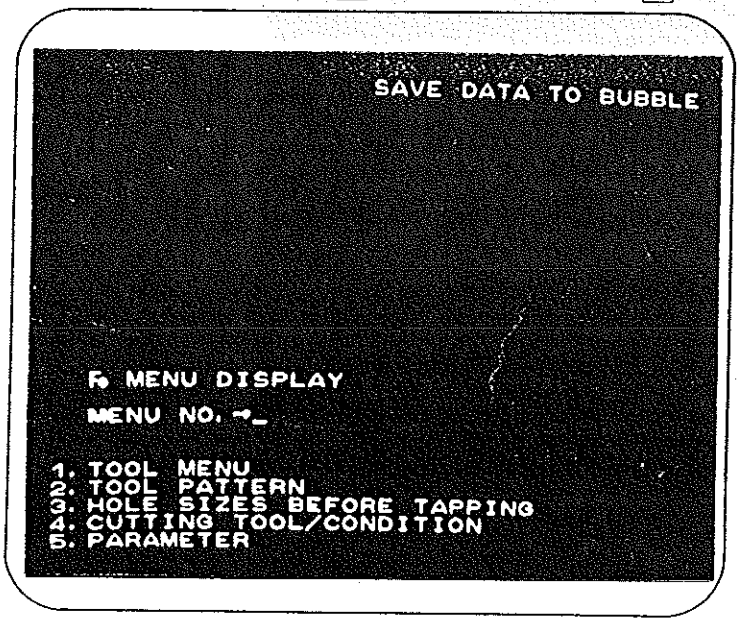
The format function has been prepared for a virgin bubble cassette.

Remember to disable the bubble cassette rewrite after completion of the external memory program deletion.

5. External Data Output

The procedure of internal memory data write into the external is as follows.

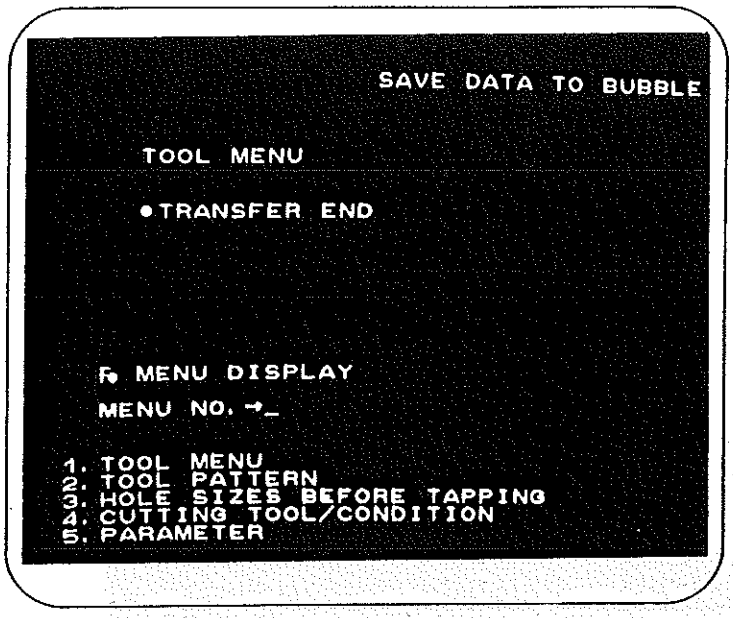
Put the bubble cassette into the rewrite ready state by setting the Program [PRT] mode and pushing the [7] and [INSERT] . Set the data output to the external [5] , and push the [INPUT] key.



Screen 5-1

[INPUT] the menu No. of the memory data to be written into the external, and push the [START] key.

Key-in [1] and [INPUT] , and push the [START] key.



Screen 5-2

The screen shown above indicates that the internal tool menu data was written into the external.
Remember to disable the bubble cassette rewrite after completion of the data write operation.

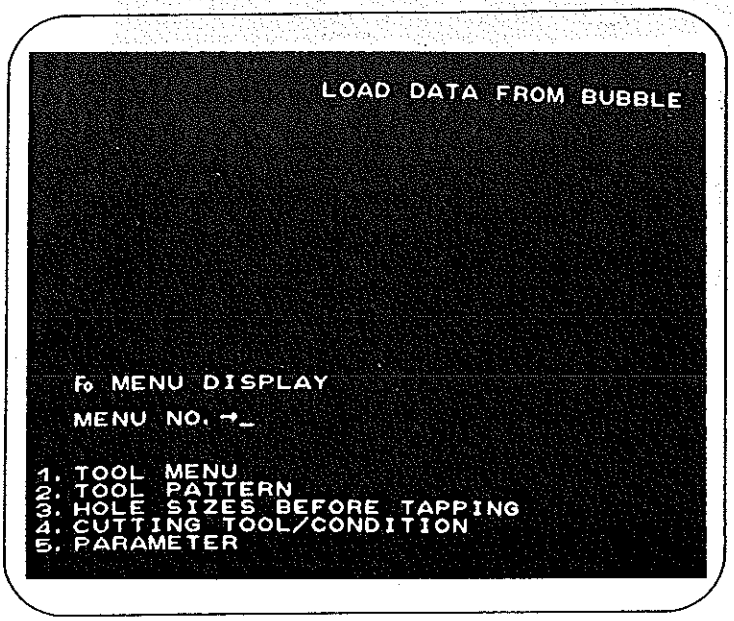
* When data is output to the external, the old data (having existed before the data output) is lost.

6. Data Input from External

The procedure of external memory data read into the external is as follows.

Make the memory rewrite switch operative by setting the [EXT] mode and pushing the key [7] and [DATA] .

Set the data input from the external [6], and push the [DATA] key.



Screen 6-1

[DATA] the menu No. of the memory data to be read into the internal, and push the [INPUT] key.

Upon completion of the data read operation, the same messages as on Screen 5-2 shown in 5. External Data Output are displayed.

Remember to disable the memory rewrite switch after completion of the data read operation.

- In case the external is a computer (RS-232C)

  1. External Memory Display

     The external memory display requests the computer to transfer program Nos. and the numbers of blocks in the programs, and displays the transferred data. It is data on 192 or less programs that this machine can receive from the computer at a time. So, the size of transfer data to be requested must be smaller than that.

     The total capacity is not displayed.

  2. Program Output to External

     The operation is basically the same as of program output to the bubble cassette.

  3. Program Input from External

     The operation is basically the same as of program input from the bubble cassette.

  4. External Program Deletion

     It is impossible to delete the external programs.

  5. Data Output to External

     The operation is basically the same as of data output to the bubble cassette.

  6. Program Input from External

     The operation is basically the same as of data input from the bubble cassette.

- In case the external is a printer (Centronics)

  1. External Memory Display

     The external memory display function is not available.

  2. Program Output to External

     The operation is basically the same as of program output to the bubble cassette.

  3. Program input from External

     Program input from the external is impossible.

  4. External Program Deletion

     It is impossible to delete the external programs.

  5. Data Output to External

     The operation is basically the same as of data output to the bubble cassette.

  6. Data Input from External

     Data input from the external is impossible.

- In case the external is PTP/PTR (RS-232C)

  1. External Memory Display

     The external memory display function is not available.

  2. Program Output to External (In Case of PTP)

     The operation is basically the same as of program output to the bubble cassette. In case the external is PTP, 100 characters of NULL data (reader block) are sent to the external before and after the program output.

  3. Program input from External (In Case of PTR)

     The operation is basically the same as of program input to the bubble cassette. NULL data (reader block) whose size exceeds 60 cm (23.6") causes an error.

  4. External Program Deletion

     It is impossible to delete the external programs.

  5. Data Output to External (In Case of PTP)

     The operation is basically the same as of data output to the bubble cassette. In case the external is PTP, 100 characters of NULL data (reader block) are sent to the external before and after the data output.

  6. Data Input from External (In Case of PTR)

     The operation is basically the same as of data input from the bubble cassette. NULL data (reader block) whose size exceeds 60 cm (23.6") causes an error.

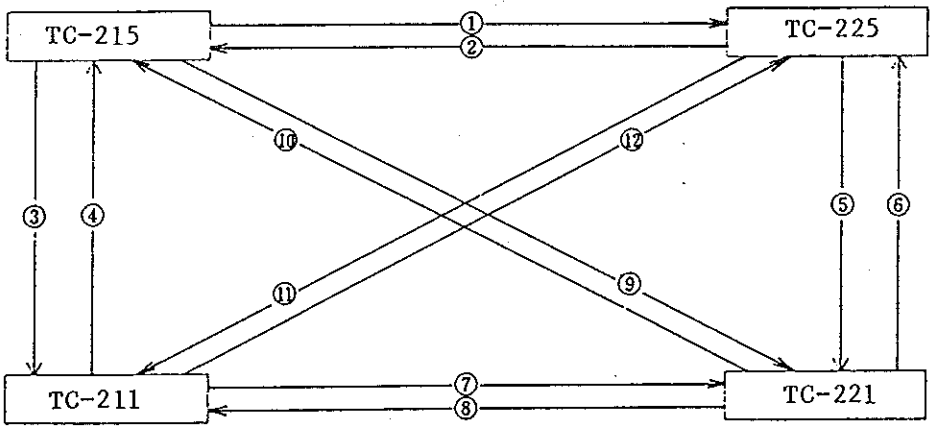
* Data transfer to and from other models

To transfer data to and from other models, pay attention to the following:

(NOTE)

○ : It is possible to transfer the data.
Δ_1 : Set the Tool Life by Manual after transferring the Tool Menu.
Δ_2 : Set the D/A offset value by Manual after transferring the Parameter.
Δ_3 : Parity error may occur because of newly added Parameter.
Input the data to new Parameter area by Manual after transferring the Parameter.
X : Never transmit the data.

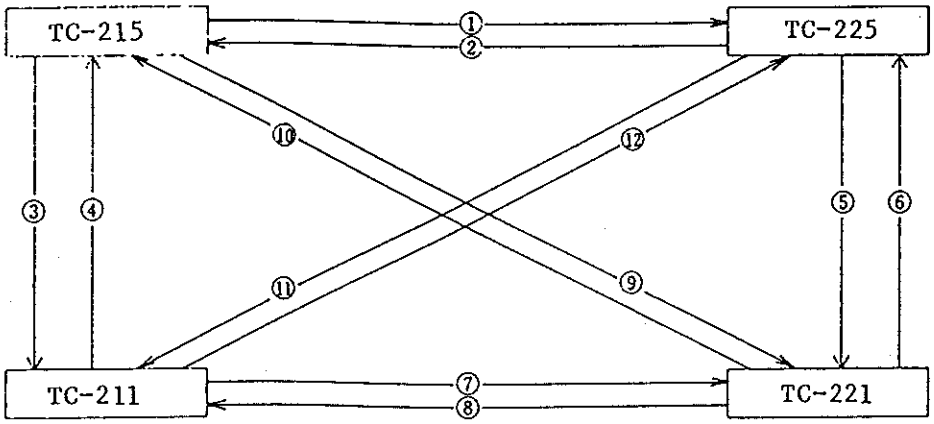
- Bubble memory



* DATA TANSMIT THROUGH BUBBLE CASSETT

	MACHINING DATA	TOOL MENU	TOOL PATURN	HOLE SIZE BEFORE TAPPING	CUTTING/TOOL CONDITION	PARAMETER
TC-215 → TC-225	○	Δ_1	○	×	×	×
TC-225 → TC-215	×	Δ_1	○	×	×	×
TC-215 → TC-211	×	○	○	○	○	Δ_2
TC-211 → TC-215	○	○	○	○	○	Δ_3
TC-225 → TC-221	×	○	○	○	○	Δ_2
TC-221 → TC-225	○	○	○	○	○	Δ_3
TC-211 → TC-221	○	Δ_1	○	×	×	×
TC-221 → TC-211	×	Δ_1	○	×	×	×
TC-215 → TC-221	×	Δ_1	○	×	○	○
TC-221 → TC-215	×	Δ_1	○	×	×	×
TC-225 → TC-211	×	×	○	×	×	×
TC-211 → TC-225	○	Δ_1	○	×	×	×

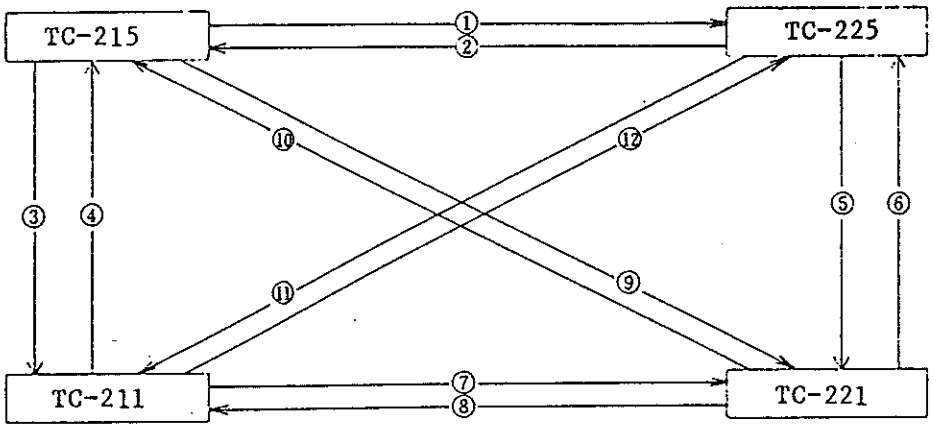
- Paper tape



* DATA TRANSMIT THROUGH PAPER TAPE

	MACHINING DATA	TOOL MENU	TOOL PATURN	HOLE SIZE BEFORE TAPPING	CUTTING/TOOL CONDITION	PARAMETER
TC-215 → TC-225	O	O	O	X	O	X
TC-225 → TC-215	X	O	O	X	O	X
TC-215 → TC-211	X	O	O	O	O	Δ_2
TC-211 → TC-215	O	O	O	O	O	Δ_3
TC-225 → TC-221	X	O	O	O	O	Δ_2
TC-221 → TC-225	O	O	O	O	O	Δ_3
TC-211 → TC-221	O	O	O	X	O	X
TC-221 → TC-211	X	O	O	X	O	X
TC-215 → TC-221	X	O	O	X	O	X
TC-221 → TC-215	X	O	O	X	O	X
TC-225 → TC-211	X	O	O	X	O	X
TC-211 → TC-225	O	O	O	X	O	X

- Computer

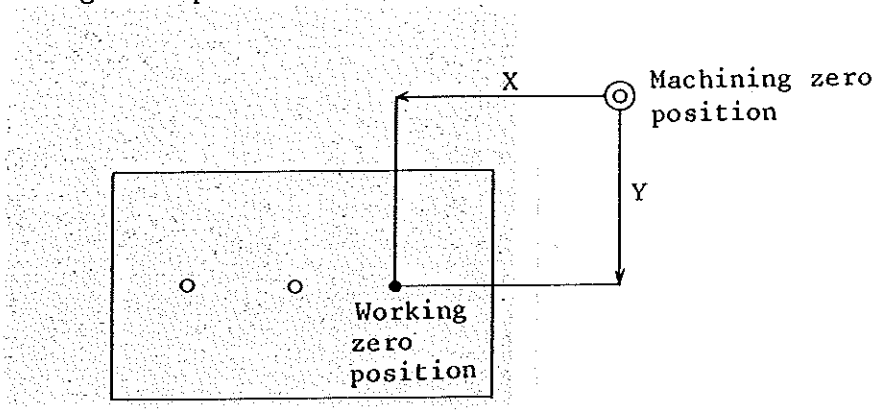


* DATA TRANSMIT THROUGH COMPUTER

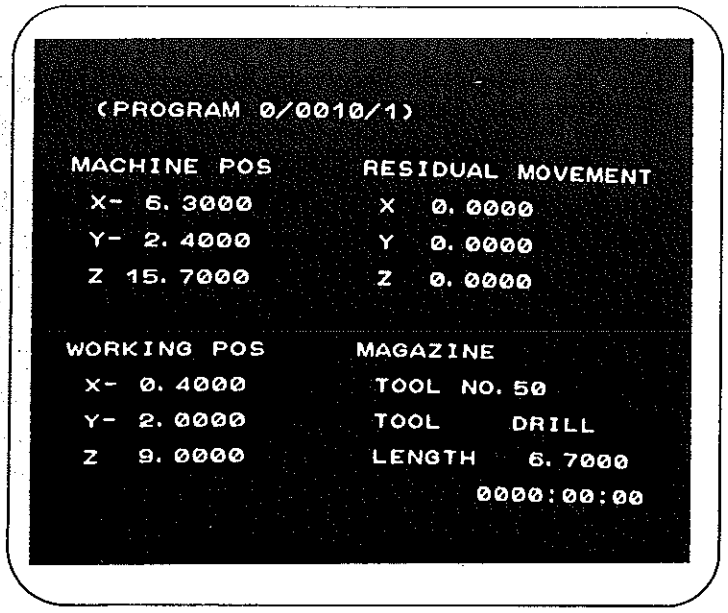
	MACHINING DATA	TOOL MENU	TOOL PATURN	HOLE SIZE BEFORE TAPPING	CUTTING/TOOL CONDITION	PARAMETER
TC-215 → TC-225	O	O	O	X	O	X
TC-225 → TC-215	X	O	O	X	O	X
TC-215 → TC-211	X	O	O	O	O	Δ_2
TC-211 → TC-215	O	O	O	O	O	Δ_3
TC-225 → TC-221	X	O	O	O	O	Δ_2
TC-221 → TC-225	O	O	O	O	O	Δ_3
TC-211 → TC-221	O	O	O	X	O	X
TC-221 → TC-211	X	O	O	X	O	X
TC-215 → TC-221	X	O	O	X	O	X
TC-221 → TC-215	X	O	O	X	O	X
TC-225 → TC-211	X	O	O	X	O	X
TC-211 → TC-225	O	O	O	X	O	X

8-3-8. Setting of Working Zero Position

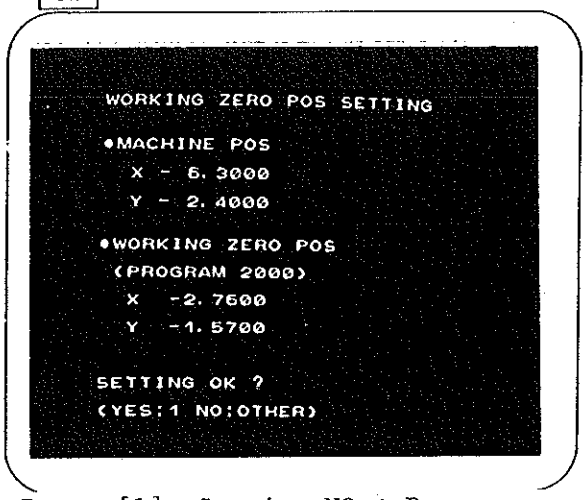
If the coordinate position from the machining zero position to the working zero position (XY in the below figure) is unknown, or if the jig fixing position is changed, the current XY machining coordinate value can be set in the working zero position.



Press the [MANU] key, then press the [M.ZRT] key.
Move the X and Y axes to the working zero position to be set newly.
Check the machining coordinate value of the X and Y axes by using the [POS] key.

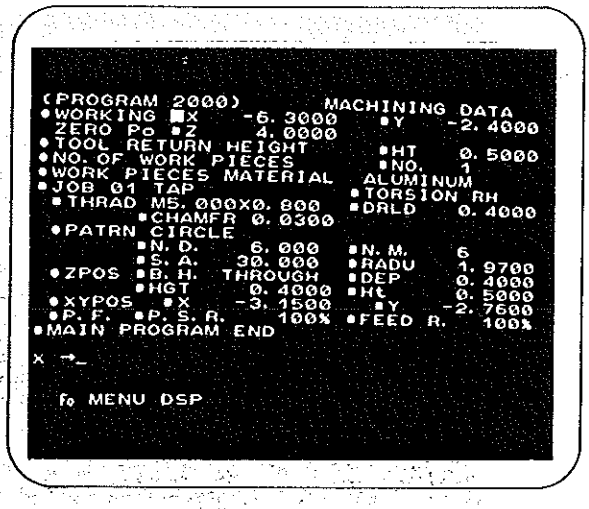


- Set the PROGRAM PROTECT switch to OFF.
  Press the [EDIT] key, [1] key, and [ENTER] key.
  Call the program in which the working zero position is set newly.
  Program No. ex) Set [2000].
  Press the [ZERO WORK SET] .



Setting OK → Press [1] Setting NO → Press any number other than 1
When working zero position (X, Y) = (-2.76, -1.57) of program No. 2000 is changed to (X, Y) = (-6.3, -2.4), press the [1].
In response to defectives, key in any number other than 1.
(The machining data screen is displayed.)
(The working data screen is displayed.)

[1] key in



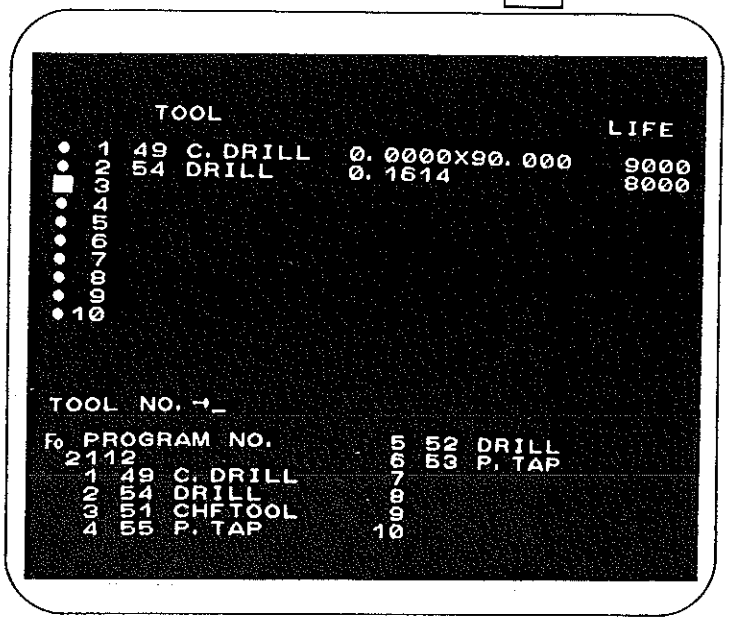
For completion, press the [Fo] key. (The display is returned to the menu screen.)
Set the PROGRAM PROTECT switch to ON.

8-3-9. Setting in the Magazine

When program edit is completed and tool to be used is assigned from the tool list, the magazine number to which the tool is assigned is determined.

Set the PROGRAM PROTECT switch to OFF, press [MDI] key, and press the [MAGAZ] key.

Key-in the [Program No.] and press [ENTER] key.



Tools to be used in the edited program are sequentially displayed in the teaching data area. The tool numbers are then set by moving the cursor to the magazine number in the program display area (data display area).

For details on this setting, refer to Section 7-4, "Magazine Tool".

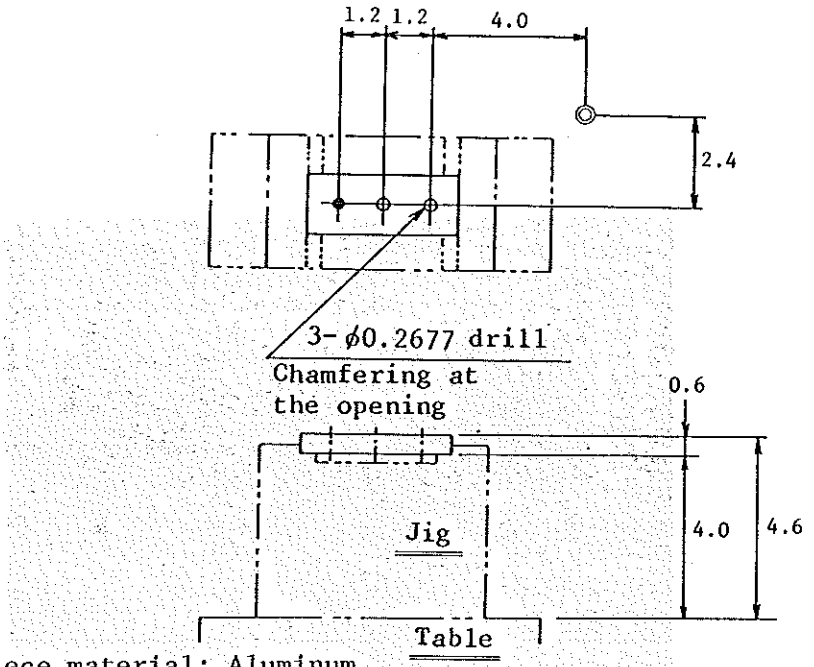
◎ Setting tools in Magazine

In accordance with the above displayed screen for magazine numbers, compare the tool numbers with the magazine number position of this machine, and set the tools in the right position.

For details on this setting, refer to Section 4, "Installation of Tool".

8-4. Program Edit Examples

8-4-1. Examples of Linear Pattern Drilling



Workpiece material: Aluminum

Machining (final): Drilling

Machining order: Memory data

Tool pattern: Memory data

Cutting tool/condition: Memory data

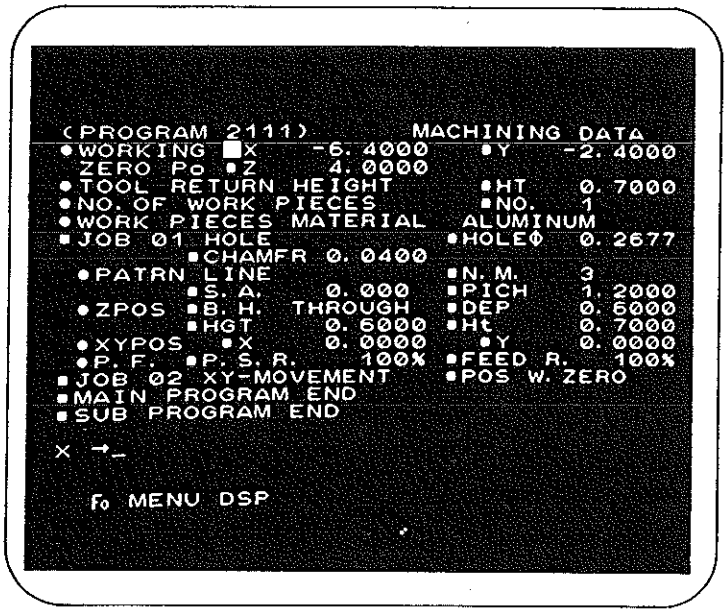
◎ Machining zero position
● Working zero position

Item	Key operation	Contents
Mode key	PROGRAM PROTECT ON OFF 	Set the PROGRAM PROTECT switch from ON to OFF, and checks that the LED for Memory Rewrite Enable comes on.
Menu selection	EDIT 1 ENTER	1. MACHINING DATA
Program No.	2111 ENTER	

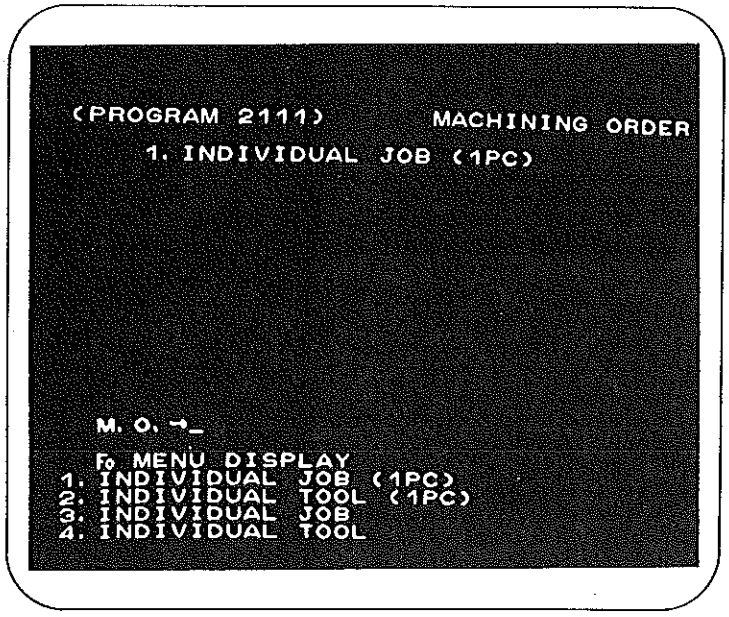
Working zero position X	-6.4 ENTER	X axis position of the working zero position from the machining zero position
Y	-2.4 ENTER	Y axis position of the working zero position from the machining zero position
Z	4.0 ENTER	Z axis position from the table top to the bottom face of the workpiece.
Tool return Height	0.7 ENTER	Z axis returning height moving from one job to the other (Position from the Z axis working zero position)
No. of work pieces	1 ENTER	Number of setting workpieces
Work pieces material	4 ENTER	Selection from the teaching data area 4. ADC12
	Displays job 01. Sets the final machining of job 01 or the final shape	
Job 01	2 ENTER	Selection from the teaching data area 2. Hole
Hole diameter	0.2677 ENTER	Diameter of the drilling
Chamfering	0.04 ENTER	Set to 0 when chamfering is unnecessary.
Pattern	3 ENTER	Selection from the teaching data area 3. Line.

Number of machining	3 ENTER	Number of machining holes
Start angle	0 ENTER	Set angle of the machining start position to the X axis on the XY plane
Pitch	1.2 ENTER	Hole machining pitch on the straight pattern
Z axis position Hole bottom	1 ENTER	Selection from teaching data area 1. Through
Depth	0.6 ENTER	Depth of machining hole
Height	0.6 ENTER	Height from the Z axis working zero position to the workpiece top
Ht	0.7 ENTER	Return height of the Z axis when the Z axis moves from one machining position to the other machining position in a job (Position from the Z axis working zero position)
XY position X	0 ENTER	X axis position at machining start point (Position from the X axis working zero position)
Y	0 ENTER	Y axis position at machining start point (Position from the Y axis working zero position)
Peripheral Speed rate	100 ENTER	100% Instruction of cutting tool condition in percentage (Refer to Section 7-5-2, 'Cutting Tool/Condition".)

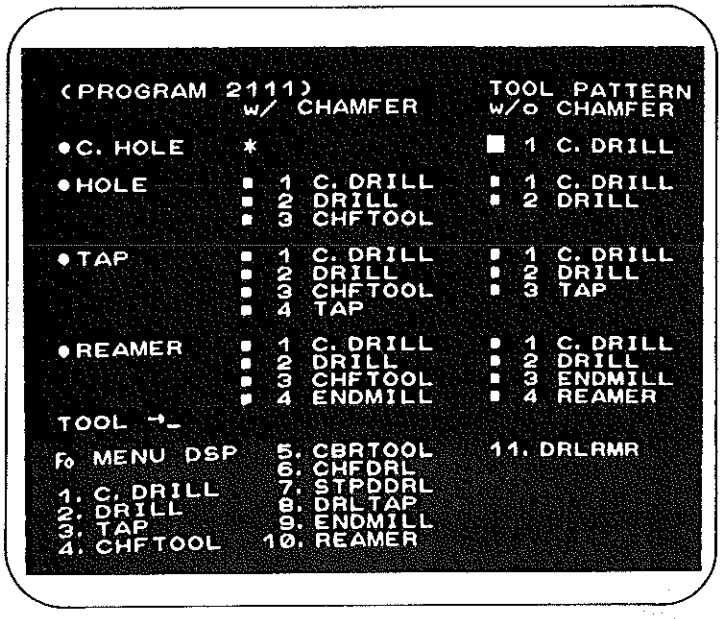
Feed rate	100 ENTER	Instruction of feed amount/revolution of cutting tool condition in percentage (Refer to Section 7-5-2, 'Cutting Tool/Condition".)
	Displays job 02. Sets final machining or final shape of job 02.	
Job 02	10 ENTER	Selection of the teaching data area 10. XY movement
Type of move	3 ENTER	Selection of teaching data area 3. Working zero return
	Displays job 03. Sets final machining or final shape of job 03.	
	-99 ENTER	Selection of Teaching data area -99. JOB END Since a series of operation is completed, ends the main program.
Number	-99 ENTER	Selection from the teaching data area -99. Subprogram END
		Machining input end Returns to the menu screen.

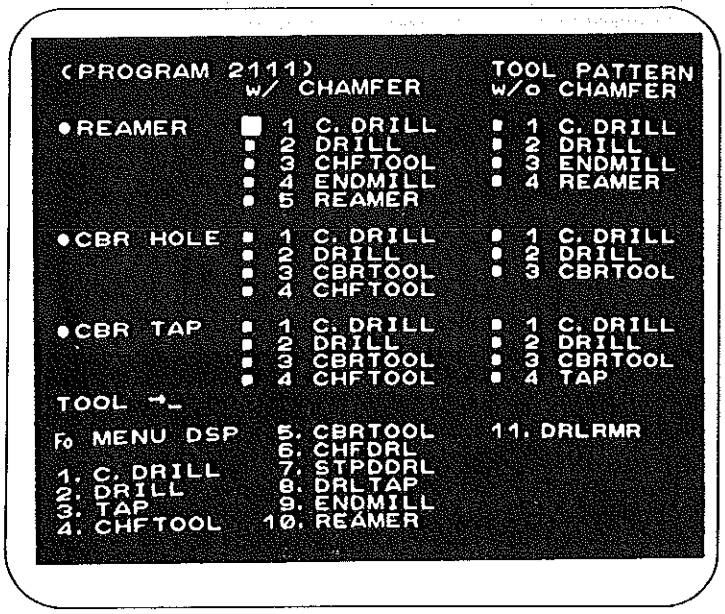


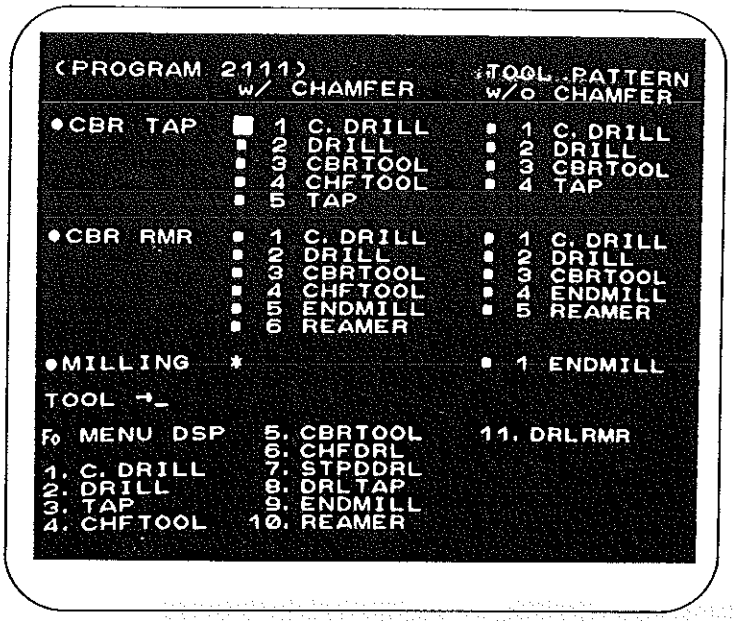
	Fo	The display is returned to the menu screen.
Menu selection	2 ENTER	2. Machining order
Program No.	2111 ENTER	



		Checks displayed memory data. When the data is not appropriate as to the machining order, refer to Section, 8-3-2 Setting of Machining Order. The machining order only for the program can be provided to the machining data.
	F0	The display is returned to the menu screen.
Menu selection	3 ENTER	3. Tool pattern
Program number	2111 ENTER	







Checks displayed memory data. When the use pattern for the set program does not exist, change data by referring to Section 8-3-3, "Tool Pattern". The tool pattern only for the program can be provided to the machining order.

Fo

The display is returned to the menu screen.

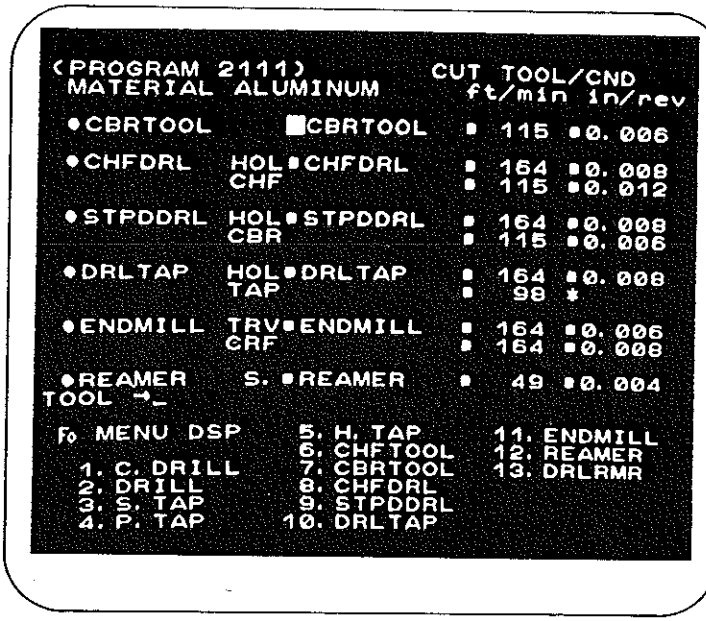
Menu selection

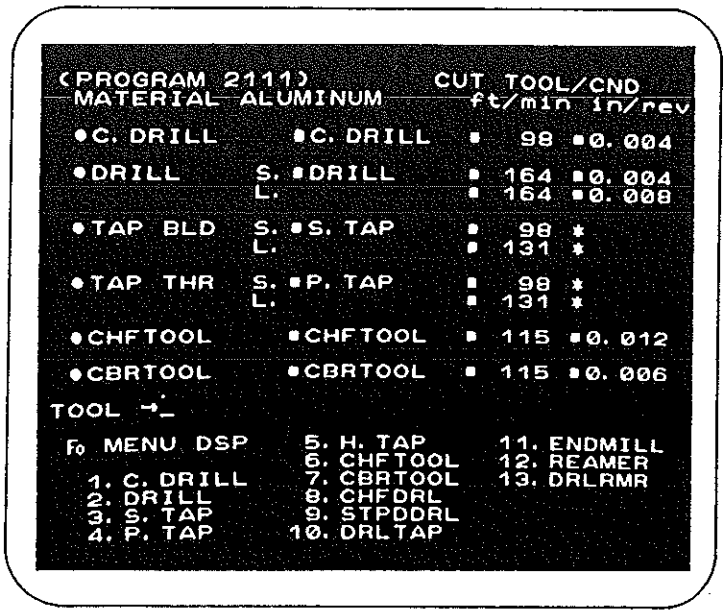
4 ENTER

4. Cutting tool/condition

Program No.

2111 ENTER





When tools are not assigned, refers to Section 8-3-5, 'Tool Assign'.

Checks displayed memory data. When the data is not appropriate for the tool cutting condition of the set program, change data by referring to Section 8-3-4, Cutting Tool/Condition. The tool cutting condition only for the program can be provided.

F0	The display is returned to menu screen.

* Operations for items 2 to 4 can be omitted when it is found that the omission has no bad effects on data set in advance.

Menu selection

5 ENTER

5. Tool assign

Necessary tools are automatically selected or set according to the program contents, tool menu, and cutting tool condition.

Use tool (Center drill, drill, chamfering tool)

ex) The following tools are registered on the tool menu:

Tool No. Tool Name	49 Center drill	50 Drill	51 Chamfering tool
Point angle	0	135	*
Small diameter	0	*	0.02
Small diameter length	0	*	*
Center angle	90	*	90.
Ineffective length	*	*	0
Major diameter nominal size	0.4	0.2677	0.6299
Pitch/thread per pitch	*	*	*
Flute length	*	1.2	*
Torsion	*	*	*
Overall length	6.0	6.7	6.0
Tool life	9000	9000	9000

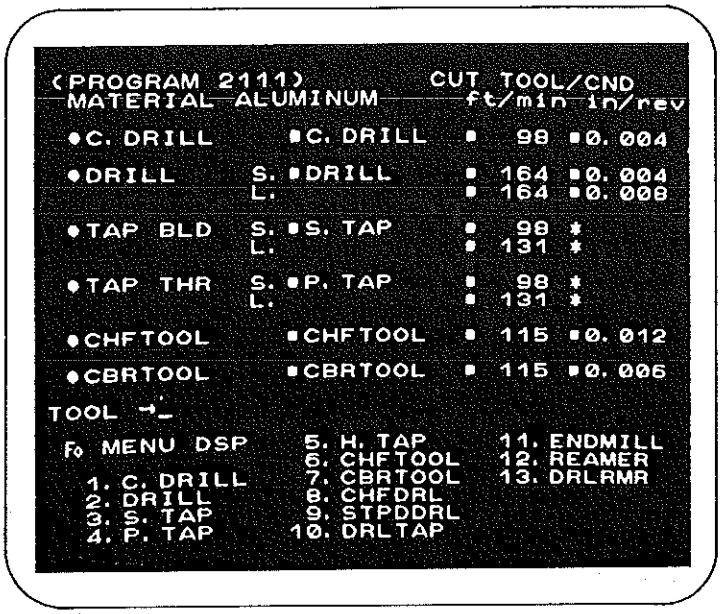
(Refer to 7-3. Tool Menu)

Program No.

2111 ENTER

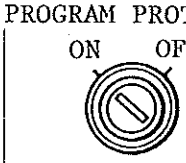
START

(Refer to Section 8-3-5, 'Tool Assign'.)



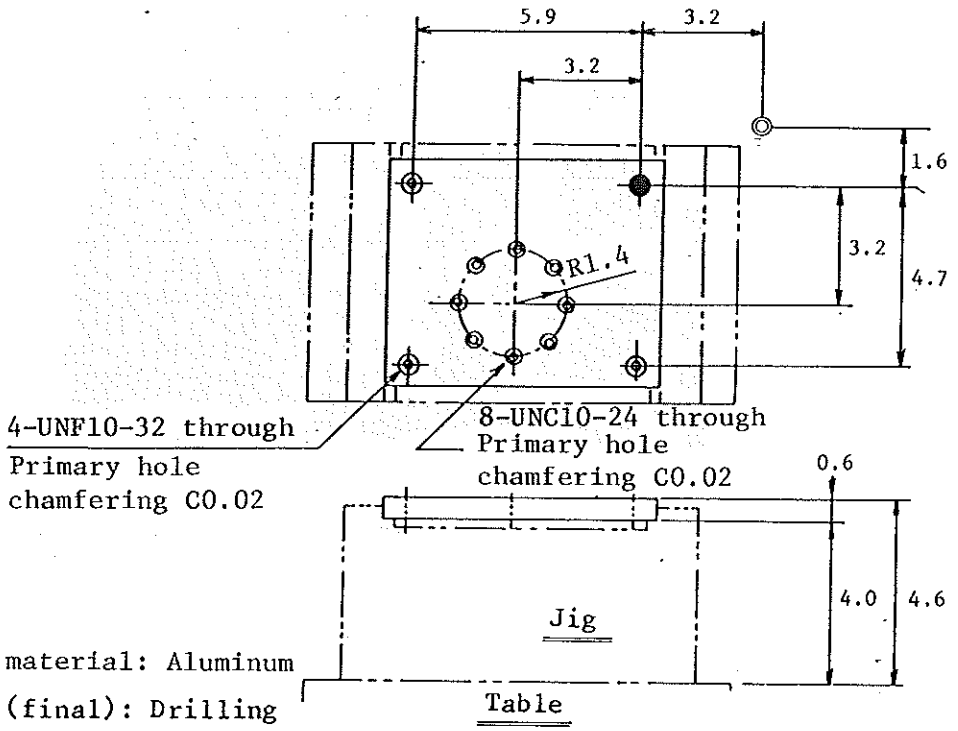
When tools are not assigned, refers to Section 8-3-5, 'Tool Assign'.

Tool Assignment end
The display is returned to menu screen.



Set the PROGRAM PROTECT switch from OFF to ON, and check that the LED for memory rewrite ENABLE goes off.

8-4-2. Drilling and Tapping using patterns



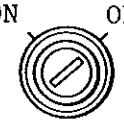
Workpiece material: Aluminum

Machining (final): Drilling

Machining order: Memory data

Tool pattern: Memory data

Cutting tool/condition: Memory data

Item	Key operation	Contents
Mode key	PROGRAM PROTECT ON OFF 	Switches the Program protect switch from ON to OFF, and checks that the LED for Memory rewrite Enable comes on.
	EDIT	
Menu selection	1 ENTER	1. Machining data
Program No.	2112 ENTER	

Working zero position X	-3.2 ENTER	X axis position of the working zero position from the machining zero position
Y	-1.6 ENTER	Y axis position of the working zero position from the machining zero position
Z	4.0 ENTER	Z axis position from the table top to the bottom face of the workpiece.
Tool return height	0.7 ENTER	Z axis returning height moving from one job to the other (Position from the Z axis working zero position)
Number of workpieces	1 ENTER	Number of setting workpieces
Work pieces material	4 ENTER	Selection from the teaching data area

Displays job 01.
Sets the final machining or final shape of job 01

Job 01	3 ENTER	Selection from the teaching data area 3. Tap
Torsion	1 ENTER	Selection from the teaching data area 1. Right hand thread
Thread	4 ENTER	Selection from the teaching data area 4.Unified fine screw threads

Nominal diameter	10 ENTER	Nominal size of screw
Thread/inch	32 ENTER	Screw pitch
Drilling depth	0 ENTER	Drill machining depth
Chamfering	0.02 ENTER	When chamfering is unnecessary, set the size to 0.
Pattern	2 ENTER	Selection from the teaching data area 2. Square pattern
X	-5.9 ENTER	Square pattern pitch in the direction of X
Y	-4.7 ENTER	Square pattern pitch in the direction of Y
Z axis position Hole bottom	1 ENTER	Selection from the teaching data area 1. Through
Depth	0.6 ENTER	Machining hole depth
Height	0.6 ENTER	Height from the Z axis machining zero position to the top face of workpiece

Ht	0.7 ENTER	Return height of the Z axis when the Z axis moves from one machining position to the other machining position in a job (Height from the Z axis working zero position)
XY position X	0 ENTER	X axis position at machining start point (Position from the X axis working zero position)
Y	0 ENTER	Y axis position at machining start point (Position from the Y axis working zero position)
Peripheral speed rate	100 ENTER	100% Instruction of peripheral velocity for cutting tool condition in percentage (Refer to Section 7-5-2, 'Cutting Tool/Condition".)
Feed rate	100 ENTER	100% Instruction of feed amount of the cutting tool condition per revolution in percentage (Refer to Section 7-5-2, 'Cutting Tool/Condition".)

Displays job 02.
Set the final machining or final shape of job 02.

Job 02	3 ENTER	Selection form the teaching data area 3. Tap
Torsion	1 ENTER	Selection from the teaching data area 1. Right-handed thread
Thread	3 ENTER	Selection from the teaching data area 3. Unified coarse thread
Nominal diameter	10 ENTER	Nominal size of screw

Thread/inch	24 ENTER	Screw threads per inch
Drilling depth	0 ENTER	Drill machining depth
Chamfering	0.02 ENTER	When chamfering is unnecessary, set the size to 0.
Pattern	1 ENTER	Selection from the teaching area 1. Circle pattern
No. of division	8. ENTER	Number of division of circumference (decimal point required)
Number of machining	8 ENTER	Number of machining workpieces
Start angle	90. ENTER	Set angle of the machining start position to the X axis on the XY plane
Radius	1.4 ENTER	Radius of circumference pattern
Z axis position Hole bottom	1 ENTER	Selection from teaching data area 1. Through
Depth	0.6 ENTER	Working hole depth

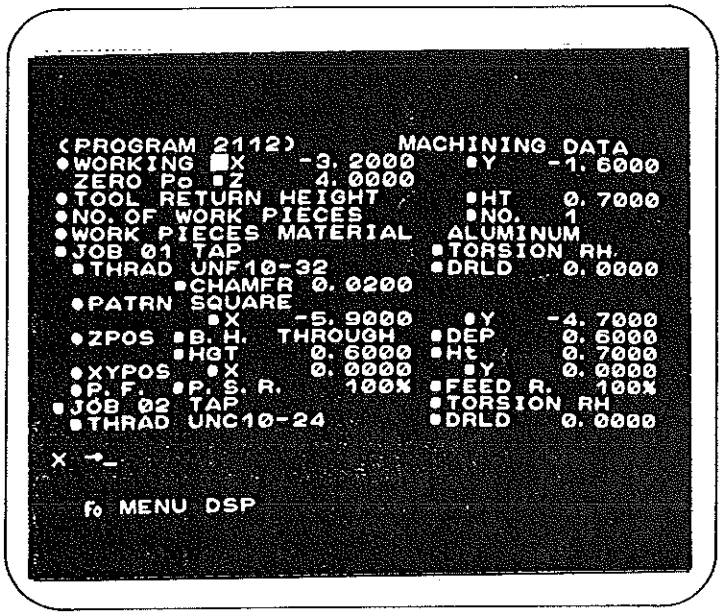
Height	0.6 ENTER	Height from the Z axis working zero position to the workpiece mount face
Ht	0.7 ENTER	Return height of the Z axis when the Z axis moves from one machining position to the other machining position in a job (Height from the Z axis working zero position)
XY position X	-3.2 ENTER	X axis position of the center of the circle (Position from the X axis working zero position)
Y	-3.2 ENTER	Y axis position of the center of the circle (Position from the Y axis working zero position)
Peripheral speed rate	100 ENTER	200% Instruction of peripheral velocity for cutting tool condition in percentage
Feed rate	100 ENTER	200% Instruction of feed amount of the cutting tool condition per revolution in percentage

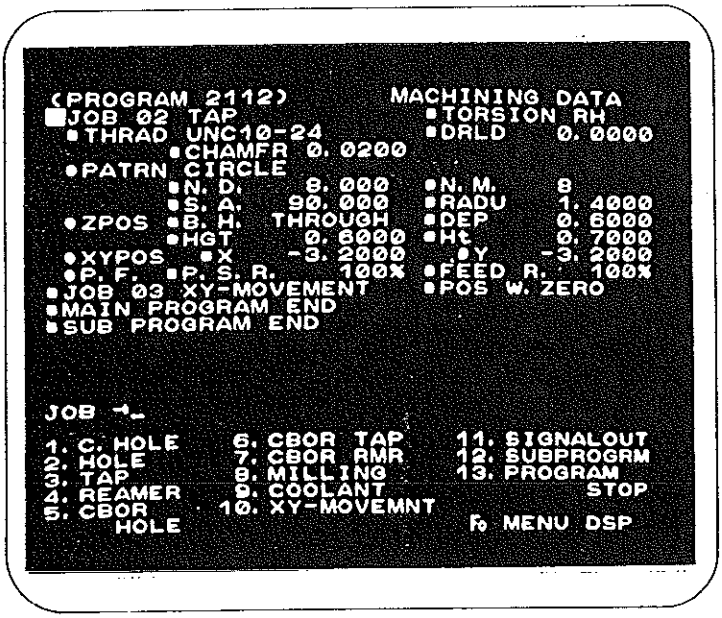
Displays job 03.
Set the final machining or final shape of job 03.

Job 03	10 ENTER	Selection from the teaching data area 10. XY position
Position of move	3 ENTER	Selection from the teaching data area 3. Working zero return

Displays job 04.
Set the final machining or final shape of job 04.

	-99 ENTER	Selection from the teaching data area -99. Main program END Since a series of program ends, end the main program.
Number	-99 ENTER	Selection from the data teaching area -99. Subprogram END





Fo	Machining input end The display is returned to the menu screen.

Since memory data is used for items 2 to 4 in the menu screen, the operation is unnecessary.
(Refer to Section 8-4-1, Examples of Straight Pattern Drilling.)

Menu selection

5 [ENTER]

5. Tool assign

Set necessary tool according to the program contents, tool menu, and cutting tool condition.

Using tools (Center drill, drill, chamfering tool, tap)

Tool No. Tool Name drill	49 Center tool	51 Chamfering	52 Drill	53 P tap	54 Drill	55 P tap
Point angle	0	*	135	*	135	*
Small diameter	0	0.02	*	*	*	*
Small diameter length	0	*	*	*	*	*
Center angle	90.	90.	*	*	*	*
Ineffective length	*	0	*	1.5	*	1.5
Major diameter nominal size	0.4	0.6299	0.1535	UCN10	0.1614	UNF10
Pitch/thread per pitch	*	*	*	24	*	32
Effective cutting length	*	*	0.78	0.9	0.8	0.9
Torsion direction	*	*	*		*	
Tool length	6.0	6.0	5.5	RH 5.5	5.5	RH 5.5
Life	9000	9000	9000	9000	9000	9000

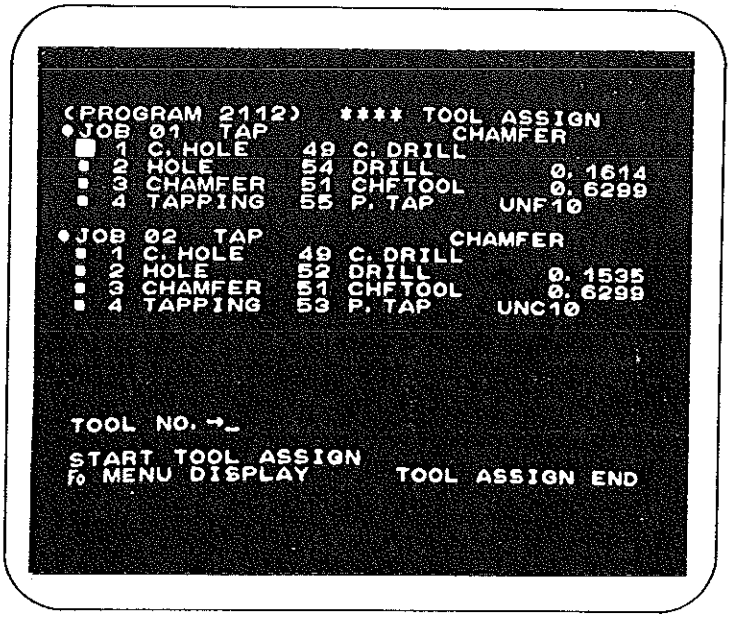
(Refer to Section 7-3, Tool Menu.)

Program No.

2112 [ENTER]

[START]

(Refer to 8-3-5, 'Tool Assign'.)

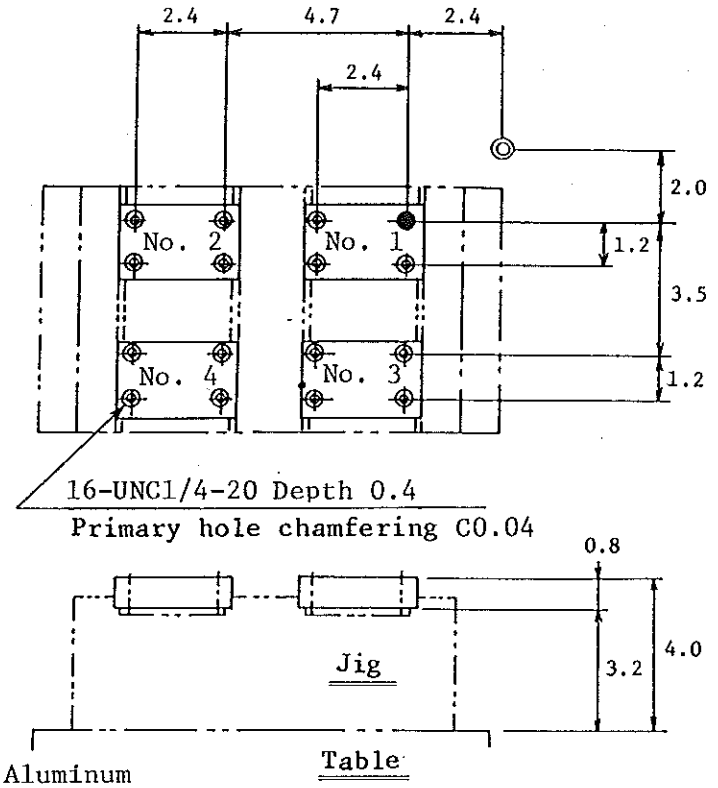


When tools are not assigned, refer to Section 8-3-5, "Tool Assign".

Tool assign end.
The display is returned to the menu screen.

Switch the Program protect switch from OFF to ON, and check that the LED for memory rewrite switch goes off.

8-4-3. Example of Tapping for Multiple Workpiece



Workpiece material: Aluminum

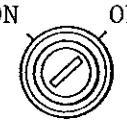
Machining (final): tap

Machining order: Memory data

Tool pattern: Memory data

Cutting tool/condition: Memory data

◎ Machining zero position
● Working zero position

Item	Key operation	Contents
	PROGRAM PROTECT ON OFF 	Switches the Program protect switch from ON to OFF, and checks that the LED for Memory Rewrite Enable comes on.
Mode selection	EDIT	
Menu selection	1 ENTER	1. Machining data
Program No.	2113 ENTER	

Working zero position X	-2.4 ENTER	X axis position of the working zero position from the machining zero position
Y	-2.0 ENTER	Y axis position of the working zero position from the machining zero position
Z	3.2 ENTER	Z axis position from the table top to the bottom face of the workpiece.
Tool return Height	0.9 ENTER	Z axis returning height moving from one job to the other (Position from the Z axis working zero position)
No. of workpieces	4 ENTER	Number of setting workpieces
No. 2 X	-4.7 ENTER	Value from the work standard position (working zero position) to the second work standard position
Y	0 ENTER	
No. 3 X	0 ENTER	Value from the work standard position (working zero position) to the third work standard position
Y	-3.5 ENTER	
No. 4 X	-4.7 ENTER	Value from the work standard position (working zero position) to the fourth work standard position

Y	-3.5 ENTER	
Work pieces material	4 ENTER	Selection from the teaching data area 4. ADC 12
	Displays job 01. Sets the final working or final shape of job 01.	
Job 01	3 ENTER	Selection from teaching data area 3. Tap
Torsion	1 ENTER	Selection from teaching data area 1. Right hand thread
Thread	3 ENTER	Selection from teaching data area 3. Unified coarse screw threads
Nominal size	1/4 ENTER	Nominal diameter of screw
Thread/inch	20 ENTER	Thread/inch
Drilling depth	0.6 ENTER	Drill work depth
Chamfering	0.04 ENTER	When chamfering is unncessary, set the size to 0.
Pattern	2 ENTER	Selection form the teaching data area 2. Square pattern

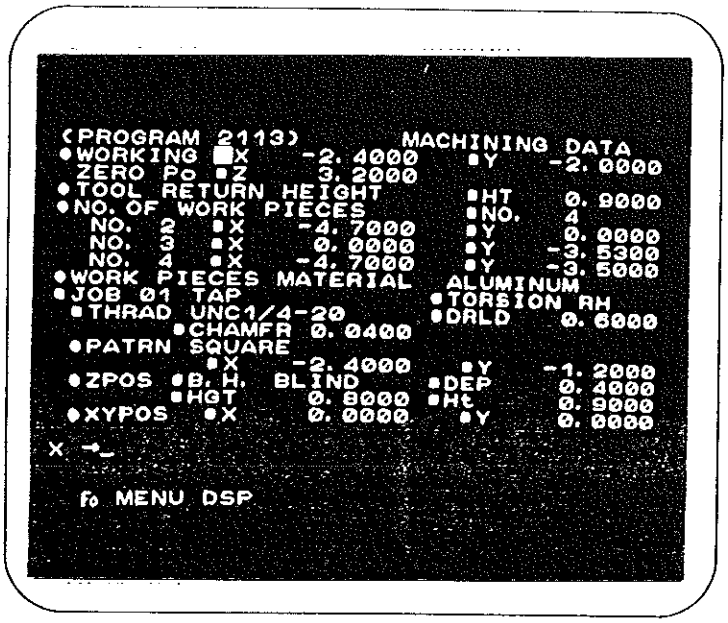
X	-2.4 ENTER	X direction pitch of the square pattern
Y	-1.2 ENTER	Y direction pitch of the square pattern
Z axis position Hole bottom	2 ENTER	Selection from the teaching area 2. Blind
Depth	0.4 ENTER	Machining hole depth
Height	0.8 ENTER	Height from the Z axis working zero position to the top face of the workpiece
Ht	0.9 ENTER	Returning height of the Z axis when the Z axis moves from a working position to another working position in a job. (Position from the Z axis working zero position)
XY position X	0 ENTER	X axis position at the first workpiece machining start position (Position from the X axis working position)
Y	0 ENTER	Y axis position at the first workpiece machining start position (Position from the Y axis working position)
Peripheral speed rate	100 ENTER	100% Instruction of peripheral velocity of the cutting tool condition in percentage (Refer to Section 7-5-2. "Cutting Tool/Condition".)
Feed rate	100 ENTER	100% Instruction of feed amount per revolution of cutting tool condition in percentage (Refer to Section 7-5-2, "Cutting Tool/Condition".)

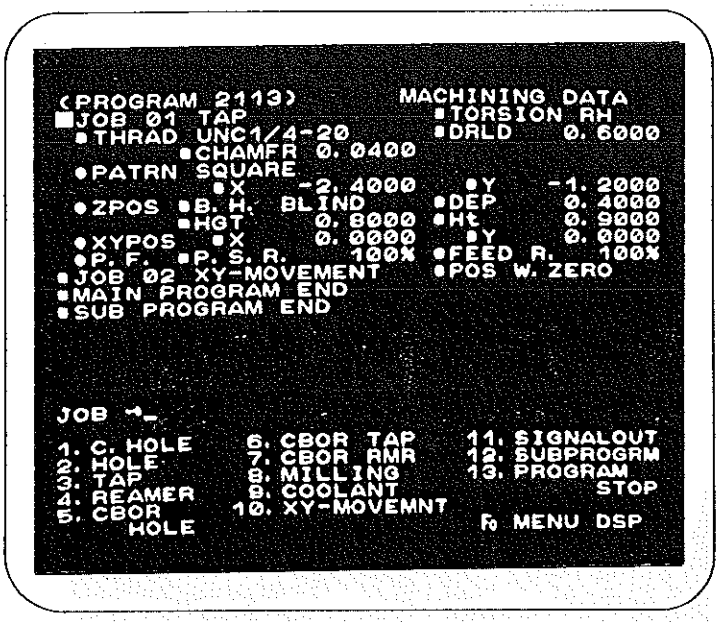
Displays job 02.
Set the final machining or final shape of job 02.

Jog 02	10 ENTER	Selection from the teaching data area 10. XY movement
Position of move	3 ENTER	Selection form the teaching data area 3. Working zero return

Displays Job 03.
Sets the final machining or final shape of job 03.

	-99 ENTER	Selection from the teaching data area -99. JOB END Since a series of operations are completed, end the main program.
Number	-99 ENTER	Selection from the data area -99. Subprogram END





* Since memory data is used for items 2 to 4 in the menu screen, the operation is unnecessary.
(Refer to Section 8-4-1, "Example of Straight Line Pattern Drilling".)

F0

Machining input end
The display is returned to the menu screen.

Menu screen

5 ENTER

Tool Assign

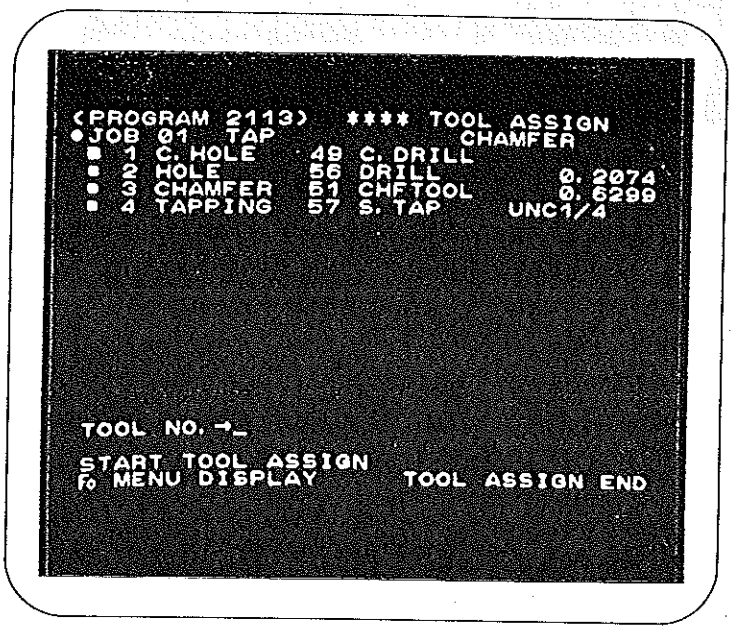
Necessary tools are automatically selected or set according to program contents, tool condition, and cutting tool condition.

Available tools (center drill, drill, chamfering tool, tap)
The following tools are registered to the tool menu:

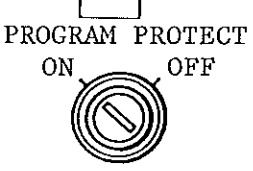
Tool No. Tool Name	49 Center drill	51 Chamfering tool	56 Drill	57 S tap
Point angle	0	*	135.	*
Small diameter	0	0.04	*	*
Small diameter length	0	*	*	*
Center angle	90.	90.	*	*
Ineffective length	*	0	*	1.5
Major diameter nominal size	0.4	0.6299	0.2074	UNC1/4
Pitch/thread per pitch	*	*	*	1.0
Effective cutting length	*	*	0.9	1.0
Torsion direction	*	*	*	
Tool length	6.0	6.0	6.0	RH 6.0
Life	9000	9000	9000	9000

(Refer to Section 7-3, "Tool Menu".)

Program No. | 2113 [ENTER] | [START] | (Refer to Section 8-3-5, "Tool Assign".)



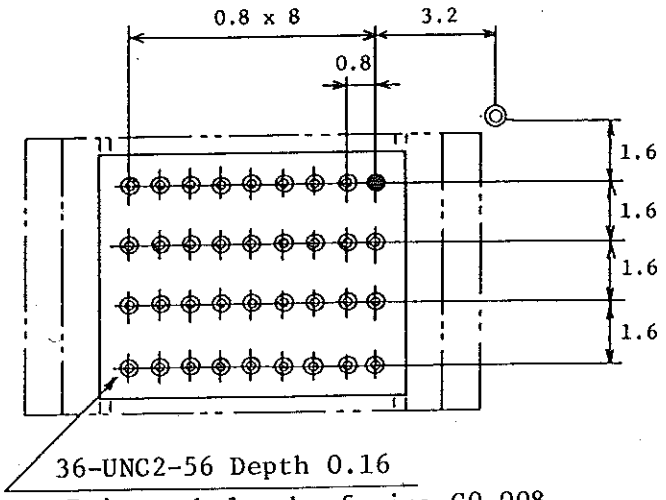
[F0]



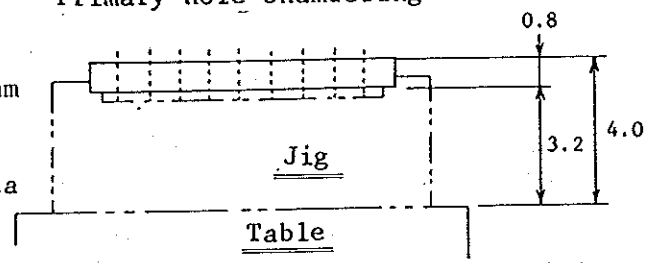
Set the Program protect switch from OFF to ON, and check that the LED of memory rewrite comes on.

When tools are not assigned, refer to Section 8-3-5, "Tool Assign".)

8-4-4. Examples of Tapping on Subprogram



Workpiece material: Aluminum
Machining (final): Tap
Machining order: Memory data
Tool pattern: Memory data
Cutting tool/condition: Memory data



Item	Key operation	Contents
	Memory rewrite ENABLE DISABLE 	Switches the memory rewrite switch from DISABLE to ENABLE, and checks that the LED for Memory Rewrite Enable comes on.
Mode key	EDIT	
Menu No.	1 ENTER	1. Machining data
Program No.	2114 ENTER	

Working zero position X	-3.2 ENTER	X axis position of the working zero position from the machining zero position
Y	-1.6 ENTER	Y axis position of the working zero position from the machining zero position
Z	3.2 ENTER	Z axis position from the table top to the bottom face of the workpiece.
Tool return Height	0.9 ENTER	Z axis returning height moving from one job to the other (Position from the Z axis working zero position)
Number of workpieces	1 ENTER	Number of setting workpieces
Work pieces material	4 ENTER	Selection from the teaching data area 4. ADC 12

Displays job 01.
Sets the final machining of job 01 or the final shape

Job 01	12 ENTER	Selection from the teaching data area 12. Subprogram
Number	70 ENTER	Set subprogram 70 to be set later.
XY position X	0 ENTER	X axis position at machining start point (Position from the X axis working zero position)
Y	0 ENTER	Y axis position at machining start point (Position from the Y axis working zero position)

Peripheral speed rate	100 ENTER	100% Instruction of cutting tool condition in percentage (Refer to Section 7-5-2, 'Cutting Tool/Condition".)
Feed rate	100 ENTER	100% Instruction of feed amount per revolution of cutting tool condition in percentage (Refer to Section 7-5-2, 'Cutting Tool/Condition".)

Displays Job 02
Sets the final machining or final shape of job 02.

Job 02	12 ENTER	Selection from teaching data area 12. Sub program
Number	70 ENTER	Set subprogram 70 to be edit later.
XY position X	0 ENTER	X axis position at machining start point (Position from the X axis working zero position)
Y	-1.6 ENTER	Y axis position at machining start point (Position from the Y axis working zero position)
Peripheral speed rate	100 ENTER	100% Instruction of peripheral velocity of cutting tool condition in percentage
Feed rate	100 ENTER	100% Instruction of feed amount per revolution of cutting tool condition in percentage

Displays job 03.
Set the final machining or final shape of job 03.

Job 03	12 ENTER	Selection from teaching data area 12. Sub program
Number	70 ENTER	Set subprogram 70 to be edit later.
XY position X	0 ENTER	X axis position at machining start point (Position from the X axis working zero position)
Y	-3.2 ENTER	Y axis position at machining start point (Position from the Y axis working zero position)
Peripheral speed rate	100 ENTER	100% Instruction of peripheral velocity of cutting tool condition in percentage
Feed rate	100 ENTER	100% Instruction of feed amount per revolution of cutting tool condition in percentage

Displays job 04.
Set the final machining or final shape of job 04.

Job 04	12 ENTER	Selection from teaching data area 12. Sub program
Number	70 ENTER	Set subprogram 70 to be edit later.
XY position X	0 ENTER	X axis position at machining start point (Position from the X axis working zero position)

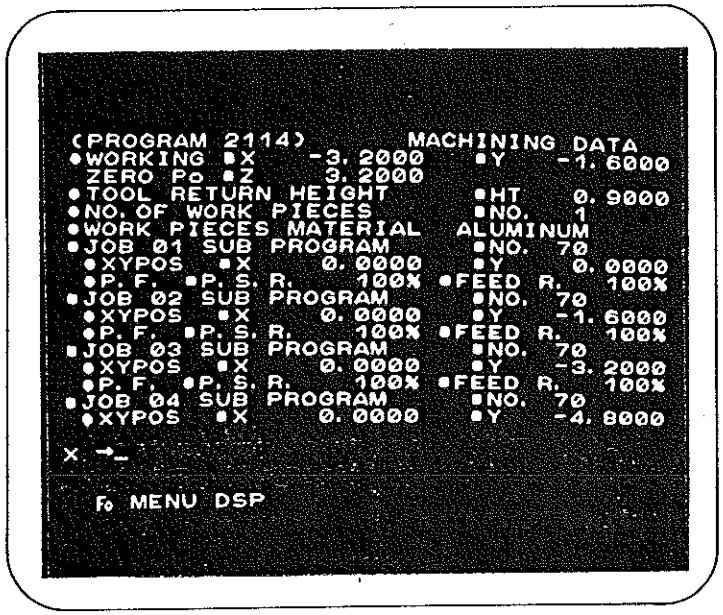
Y	-4.8 ENTER	Y axis position at machining start point (Position from the Y axis working zero position)
Peripheral Peripheral speed rate velocity	100 ENTER	100% Instruction of peripheral velocity of cutting tool condition in percentage
Feed rate	100 ENTER	100% Instruction of feed amount per revolution of cutting tool condition in percentage

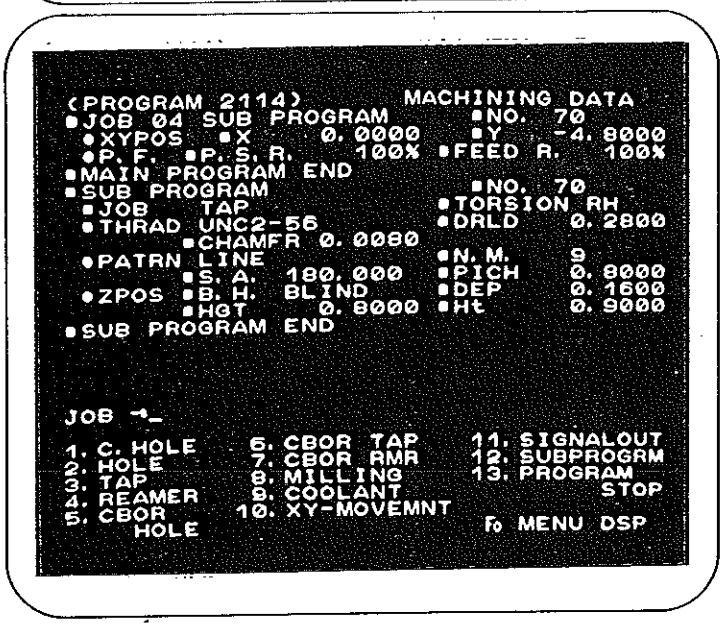
Displays job 05.
Set the final machining or final shape of job 05.

	-99 ENTER	Selection from the teaching data area -99. Main program END
Subprogram Number	70 ENTER	
Job	3 ENTER	Selection from the teaching data area 3. Tap
Torsion	1 ENTER	Selection from the teaching data area 1. Right-handed thread
Thread	3 ENTER	Selection from teaching data area 3. Unified coarse screw threads
Nominal diameter	2. ENTER	Nominal diameter of screw

Thread/inch	56 ENTER	Thread/inch
Drilling hole depth	0.28 ENTER	Depth of drill machining
Chamfering	0.008 ENTER	When chamfering is unnecessary, set 0.
Pattern	3 ENTER	Selection from the teaching data area 3. Linear pattern
Work number	9 ENTER	Number of work holes
Start angle	0 ENTER	Set the angle of machining start position to the X axis on the XY plane.
Pitch	-0.8 ENTER	Hole machining pitch on the linear pattern
Z axis position Hole bottom	2 ENTER	Selection form the teaching data area 2. Blind
Depth	0.16 ENTER	Depth of machining hole
Height	0.8 ENTER	Height from the Z axis machining zero position to the top face of workpiece

Ht	0.9 ENTER	Returning height of the Z axis when the Z axis moves from one machining position to another machining position in a job. (Position from the Z axis working position)
Sub program No.	-99 ENTER	Selection from the teaching data area -99. Subprogram END





Fo	Machining input end The display is returned to the menu screen.

	* Since memory data is used for items 2 to 4 in the menu screen, the operation is unncessary. (Refer to Section 8-4-1, "Examples of Linear Pattern Drilling".)	
Menu Screen	5 ENTER	5. Tool assign Necessary tools are automatically selected or set according to the program contents, tool condition, and cutting tool condition.

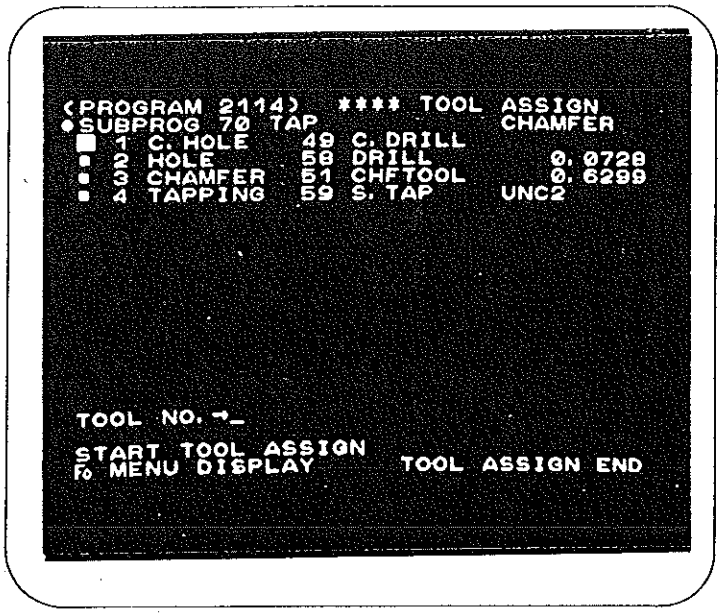
Available tool (Center drill, drill, chamfering tool, tap)

The following tools are registered in the tool menu:

Tool No. Tool Name	49 Center drill	51 Chamfering tool	58 Drill	59 S tap
Point angle	Ø	*	135.	*
Small diameter	Ø	0.02	*	*
Small diameter length	Ø	*	*	*
Center angle	9Ø.	9Ø.	*	*
Ineffective length	*	Ø	*	2.5
Major diameter nominal size	0.4	0.6299	0.0728	UNC2
Pitch/thread per pitch	*	*	*	56
Effective cutting length	*	*	0.35	0.24
Torsion direction	*	*	*	
Tool length	6.0	6.0	5.1	RH 5.5
Life	9ØØØ	9ØØØ	9ØØØ	9ØØØ

(Refer to Section 7-3, "Tool Menu".)

Program No.	2114 ENTER	
	START	(Refer to Section 8-4-5, "Tool Assign".)



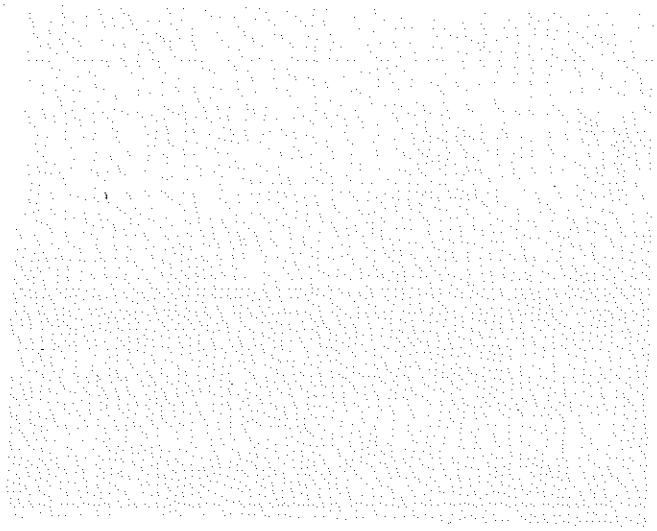
When tools are not assigned, refer to Section 8-3-5, "Tool Assign".)

F0

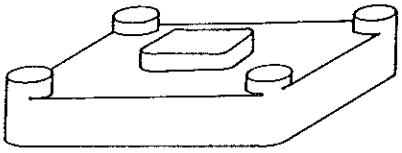
Tool assign end.
The display is return to the menu screen.

PROGRAM PROTECT
ON OFF

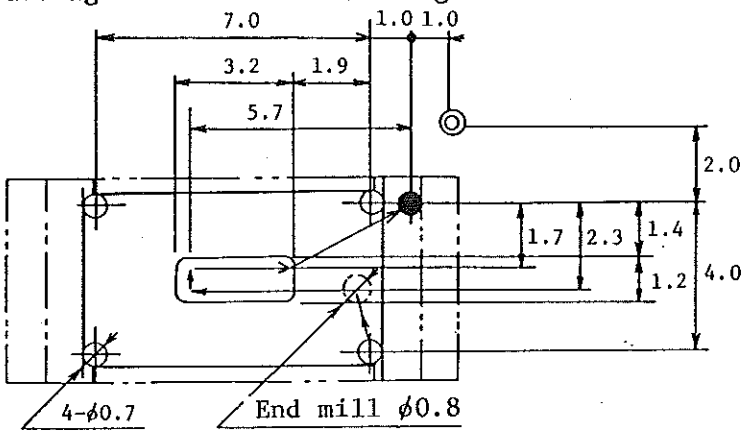
Switch the Program protect switch from OFF to ON, and check that the LED for the memory goes off.



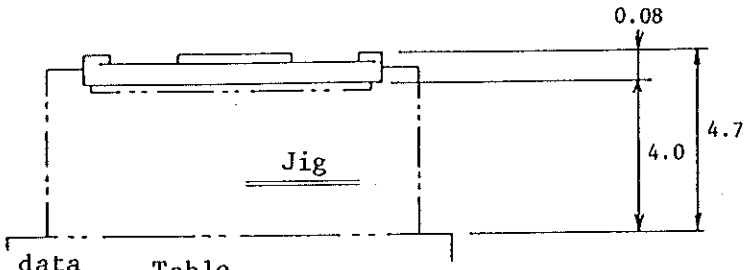
8-4-5. Example of Surface Cutting with End Mill (Straight line)



Working sample



Workpiece material: Aluminum
Machining (final): End mill
Machining order: Memory data
Tool pattern: Memory data
Cutting tool/condition: Memory data



◎ Machining zero position
● Working zero position

* For supplementary explanation, refer to the last portion of the machining examples.

Supplementary Explanation 8-4-5. Surface Cutting with End Mill

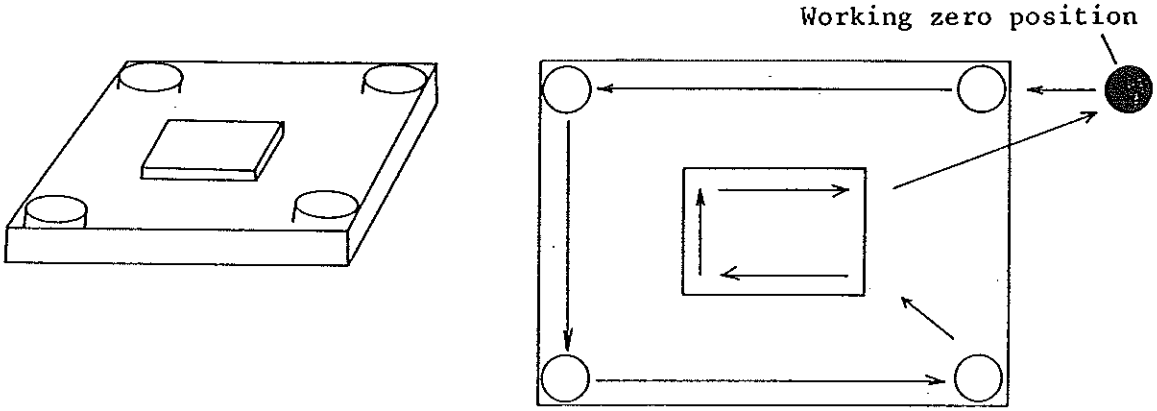


Fig. B

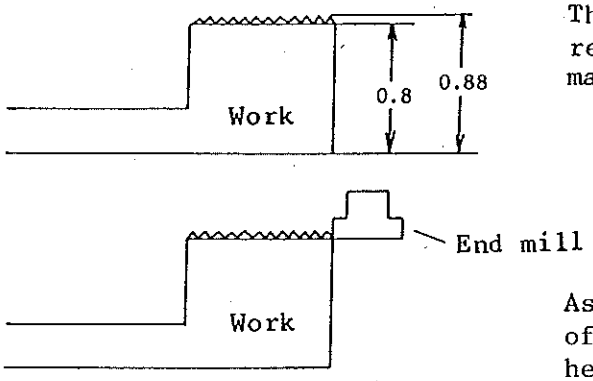


Fig. C

The accurate method is taken by omitting returning operation and shortening the machining time.

JOb 01
Depth 0.008
Height 0.88
Ht 0.8

As shown in Figure C, work at the height of 0.88 is cut with 2. When the returning height for the next operation is set to 0.8, the machine does not return and shifts to the next work, as shown in Figure D.

Job 02
Depth 0
Height 0.8
Ht 0.8

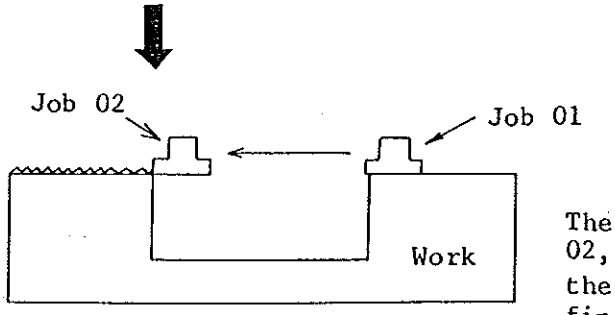


Fig. D

The depth of cutting is not set in job 02, and the returning height is set to the same level as previous job. (In the final job, it is set higher.)

When the top face cutting is completed, the inter-job returning height is ignored, and returning height is applied to the job. Therefore, set the desired returning position.

Item	Key operation	Contents
	PROGRAM PROTECT ON OFF 	Switch the Program protect switch from ON to OFF, and check that the LED for Memory Rewrite Enable comes on.
Mode key	EDIT	
Menu Selection	1 ENTER	1. Machining data
Program No.	2115 ENTER	
Working zero position X	-1.0 ENTER	X axis position of the working zero position from the machining zero position
Y	-2.0 ENTER	Y axis position of the working zero position from the machining zero position
Z	4.0 ENTER	Z axis position from the table top to the bottom face of the workpiece.
Tool return Height	2.0 ENTER	Z axis returning height moving from one job to the other (Height from the Z axis working zero position)
Number of workpieces	1 ENTER	Number of setting workpieces
Work pieces material	4 ENTER	Selection from the teaching data area 4. ADC 12
	Displays job 01. Sets the final machining or the final shape of job 01	
Job 01	8 ENTER	Selection from the teaching data area 8. Milling
Tool diameter	0.875 ENTER	Diameter of end mill tool

Cutting length X	-1.0 ENTER	Feed amount of cutting toward the X axis (Specify the direction by the incremental method.)
Y	0 ENTER	Feed amount of cutting toward the Y axis (Specify the direction by the incremental method.)
Z position mode	1 ENTER	Selection from the teaching data area 1. Cutting mode
Depth	0.08 ENTER	Cutting depth
Height	0.88 ENTER	Height form the Z axis working zero position to the top face of the workpiece
Ht	0.8 ENTER	Returning height of the Z axis at the time of working machining completion (Height from the Z axis working zero position)
XY position X	0 ENTER	X axis position at machining start point (Position from the X axis working zero position)
Y	0 ENTER	Y axis position at machining start point (Position from the Y axis working zero position)
Peripheral speed rate	200 ENTER	200% Instruction of peripheral velocity of cutting tool condition in percentage (Refer to Section 7-5-2, "Cutting Tool/Condition".)
Feed rate	200 ENTER	200% Instruction of feed amount per revolution of cutting tool condition in percentage (Refer to Section 7-5-2, "Cutting Tool/Condition".)

	Displays Job 02 Sets the final machining or final shape of job 02.	
Job 02	8 ENTER	Selection from the teaching data area 8. Milling
Tool diameter	0.875 ENTER	Diameter of end mill tool
Cutting length X	-1.0 ENTER	Feed amount of cutting toward the X axis (Specify the direction by the incremental method.)
Y	0 ENTER	Feed amount of cutting toward the Y axis (Specify the direction by the incremental method.)
Z position mode	1 ENTER	Select from the teaching data area 1. Cutting mode
Depth	0 ENTER	Cutting depth (Refer to the supplementary.)
Height	0.8 ENTER	Height from the Z axis working zero position to the top face of the workpiece (Refer to the supplementary.)
Ht	0.8 ENTER	Returning height of the Z axis at the time of working machining completion (Height from the Z axis working zero position)
XY position X	-7.0 ENTER	X axis position at machining start point (Position from the X axis working zero position)
Y	0 ENTER	Y axis position at machinig shart point (Position from the Y axis working zero position)
Peripheral speed rate	200 ENTER	200% Instruction of peripheral velocity of cutting tool condition in percentage

Feed rate	200 ENTER	200% Instruction of feed amount per revolution of cutting tool condition in percentage

Displays job 03.
Sets the final machining of final shape of job 03.

Job 03	8 ENTER	Selection from the teaching data area 8. Milling
Tool diameter	0.875 ENTER	Diameter of end mill tool
Cutting length X	0 ENTER	Feed amount of cutting toward the X axis (Specify the direction by the incremental method.)
Y	-1.0 ENTER	Feed amount of cutting toward the Y axis (Specify the direction by the incremental method.)
Z position mode	1 ENTER	Selection from the teaching data area 1.Cutting mode
Depth	0 ENTER	Cutting depth
Height	0.8 ENTER	Height from the working zero position to the top face of the workpiece
Ht	0.8 ENTER	Returning height of the Z axis when machining operation ends (Height from the Z axis working zero position)

XY position X	-8.0 ENTER	X axis position at machining start point (Position from the X axis working zero position)
Y	-3.0 ENTER	Y axis position at machining start point (Position from the Y axis working zero position)
Peripheral speed rate	200 ENTER	200% Instruction of peripheral velocity of cutting tool condition in percentage
Feed rate	200 ENTER	200% Instruction of feed amount per revolution of cutting tool condition in percentage

Displays job 04.
Sets the final machining or final shape of job 04.

Job 04	8 ENTER	Selection from the teaching data area 8. Milling
Tool diameter	0.875 ENTER	Diameter of end mill tool
Cutting length X	1.0 ENTER	Feed amount of cutting toward the X axis (Specify the direction by the incremental method.)
Y	0 ENTER	Feed amount of cutting toward the Y axis (Specify the direction by the incremental method.)
Z position mode	1 ENTER	Selection from the teaching data area 1. Cutting mode

Depth	0 ENTER	Cutting depth
Height	0.8 ENTER	Height from the working zero position to the top face of the workpiece
Ht	0.8 ENTER	Returning height of the Z axis when machining operation ends (Position from the Z axis working zero position)
XY position X	-2.0 ENTER	X axis position at machining start point (Position from the X axis working zero position)
X	-4.0 ENTER	Y axis position at machining start point (Position from the Y axis working zero position)
Peripheral speed rate	200 ENTER	200% Instruction of peripheral velocity of cutting tool condition in percentage
Feed rate	200 ENTER	200% Instruction of feed amount per revolution of cutting tool condition in percentage

Displays job 05.
Sets the final machining or final shape of job 05.

Job 05	8 ENTER	Selection from the teaching data area 8. Milling
Tool diameter	0.875 ENTER	Diameter of end mill tool

Cutting length X	-3.7 ENTER	Feed amount of cutting toward the X axis (Specify the direction by the incremental method.)
Y	0 ENTER	Feed amount of cutting toward the Y axis (Specify the direction by the incremental method.)
Z position mode	1 ENTER	Selection from the teaching data area 1. Cutting mode
Depth	0 ENTER	Cutting depth
Height	0.8 ENTER	Height from the working zero position to the top face of the workpiece
Ht	0.8 ENTER	Returning height of the Z axis when machining operation ends (Height from the Z axis working zero position)
XY position X	-2.4 ENTER	X axis position at machining start point (Position from the X axis working zero position)
Y	-2.3 ENTER	Y axis position at machining start point (Position from the Y axis working zero position)
Peripheral speed rate	200 ENTER	200% Instruction of peripheral velocity of cutting tool condition in percentage
Feed rate	200 ENTER	200% Instruction of feed amount per revolution of cutting tool condition in percentage

Displays job 06.
Sets the final machining or final shape of job 06.

JOb 6	8 ENTER	Selection from the teaching data area 8. Milling
Tool diameter	0.875 ENTER	Diameter of end mill tool
Cutting length X	0 ENTER	Feed amount of cutting toward the X axis (Specify the direction by the incremental method.)
Y	0.6 ENTER	Feed amount of cutting toward the Y axis (Specify the direction by the incremental method.)
Z position mode	1 ENTER	Selection from the teaching data area 1. Cutting mode
Depth	0 ENTER	Cutting depth
Height	0.8 ENTER	Height from the working zero position to the top face of the workpiece
Ht	0.8 ENTER	Returning height of the Z axis when machining operation ends (Height from the Z axis working zero position)
XY position X	-6.1 ENTER	X axis position at machining start point (Position from the X axis working zero position)
Y	-2.3 ENTER	Y axis position at machining start point (Position from the Y axis working zero position)

Peripheral speed rate	200 ENTER	200% Instruction of peripheral velocity of cutting tool condition in percentage
Feed rate	200 ENTER	200% Instruction of feed amount per revolution of cutting tool condition in percentage

Displays job 07.
Sets the final machining or final shape of job 07.

Job 7	8 ENTER	Selection from the teaching data area 8. Milling
Tool diameter	0.875 ENTER	Diameter of end mill tool
Cutting length X	3.7 ENTER	Feed amount of cutting toward the X axis (Specify the direction by the incremental method.)
Y	0 ENTER	Feed amount of cutting toward the Y axis (Specify the direction by the incremental method.)
Z position mode	1 ENTER	Selection from the teaching data area 1. Cutting mode
Depth	0 ENTER	Cutting depth
Height	0.8 ENTER	Height from the working zero position to the top face of the workpiece

Ht	2.0 ENTER	Returning height of the Z axis when machining operation ends (Height from the Z axis working zero position)
XY position X	-6.1 ENTER	X axis position at machining start point (Position from the X axis working zero position)
Y	-1.7 ENTER	Y axis position at machining start point (Position from the Y axis working zero position)
Peripheral speed rate	200 ENTER	200% Instruction of peripheral velocity of cutting tool condition in percentage
Feed rate	200 ENTER	200% Instruction of feed amount per revolution of cutting tool condition in percentage

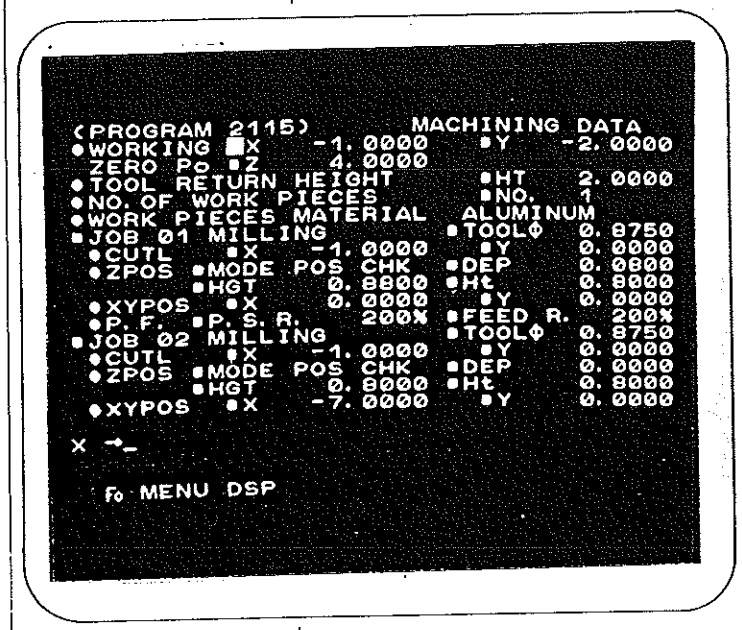
Displays job 08.
Sets the final machining or final shape of job 08.

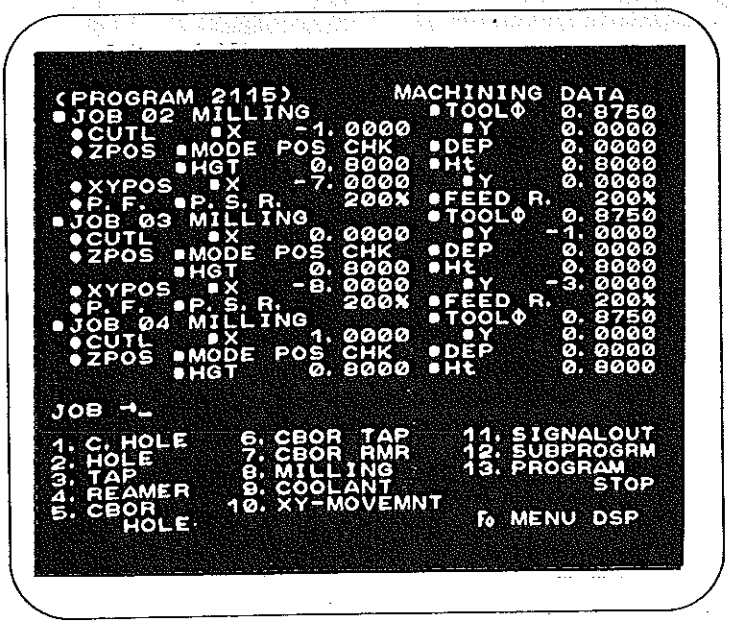
Job 08	10 ENTER	Selection from the machining data 10. XY designation
Position of move	3 ENTER	Selection from the machining data 3. Working zero return

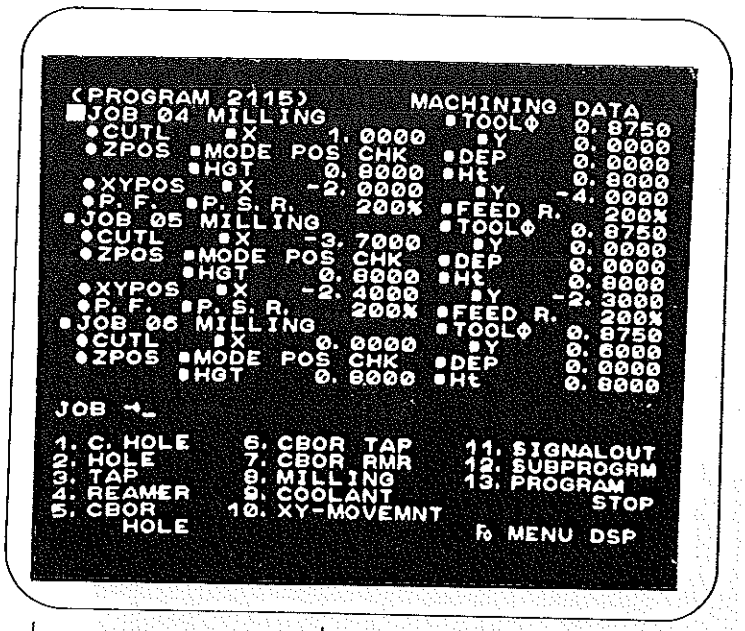
Displays job 09.
Displays the final machining or the final shape of job 09.

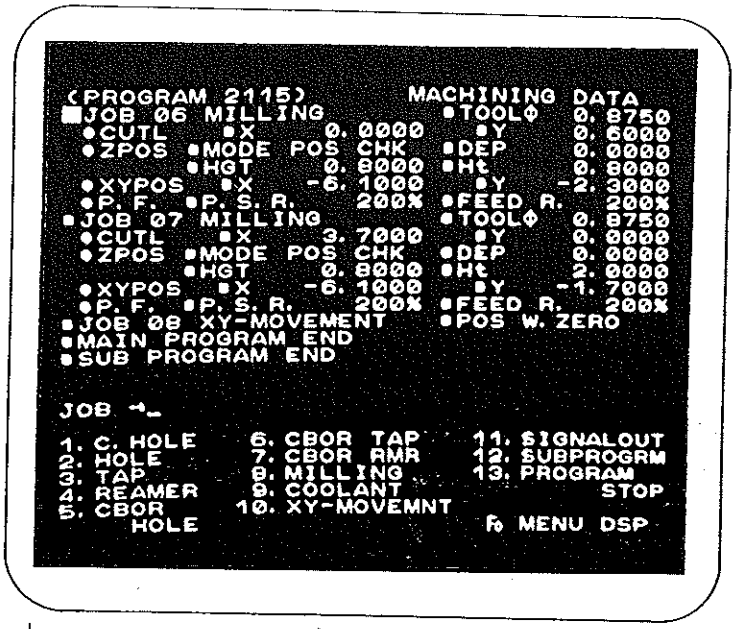
-99 ENTER	Selection from the teaching data area -99. Main program END Since a series of operations are completed, ends the main program.

Number	-99 ENTER	Selection from the teaching data area -99. Subprogram END









Fo	Machining input end The display is returned to the menu screen.

* The memory data is used for items 2 to 4 in the menu screen, the operation is unnecessary.
(Refer to Section 8-4-1, "Examples of Linear Pattern Drilling.)

Menu screen

5 ENTER

5. Tool assign

Necessary tools are automatically selected or set according to the program contents, tool condition, and cutting tool condition.

Available tool (End mill)

The following tools are registered to the tool menu.

Tool Number Tool Name	60 End mill
Point angle	*
Small diameter	*
Small diameter length	*
Center angle	*
Ineffective length	*
Major diameter nominal size	0.875
Pitch/thread per pitch	*
Effective cutting length	1.5
Torsion direction	*
Tool length	5.1
Life	5000

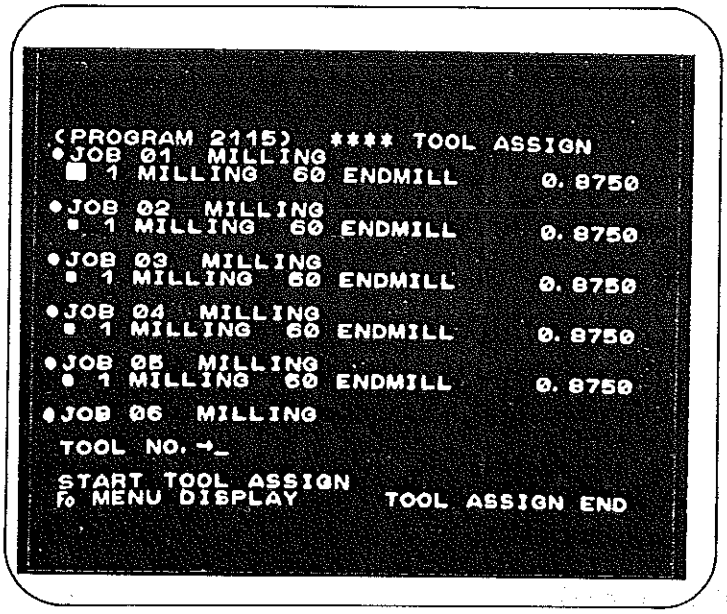
(Refer to Section 7-3, "Tool Menu".)

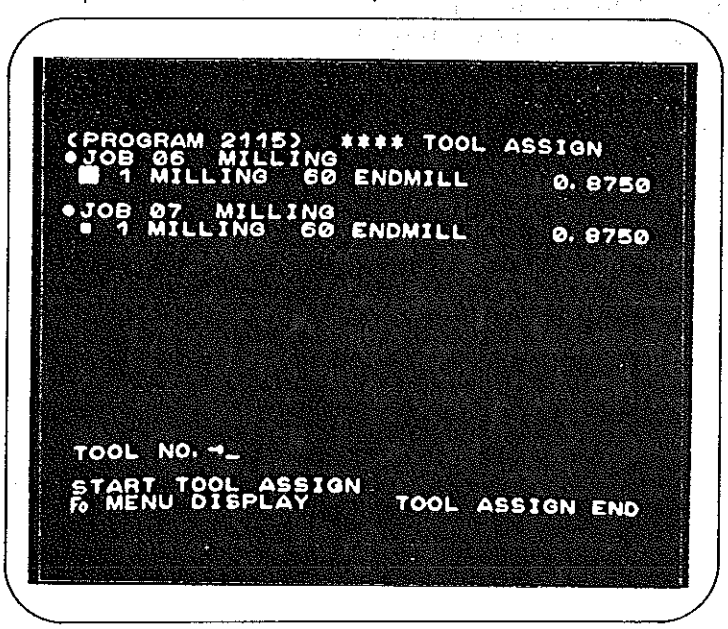
Program No.

2115 ENTER

START

(Refer to Section 8-3-5, "Tool Assign".)





F0

Tool assign end.
The display is returned to the menu screen.

PROGRAM PROTECT

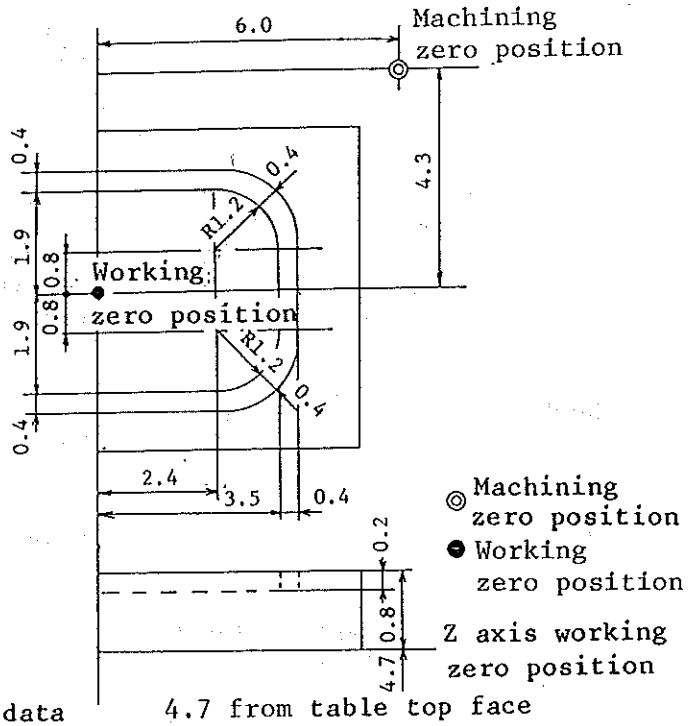
ON OFF

When tools are not assigned, refer to Section 8-3-5, "Tool Assign.")

Switch the Program protect switch from OFF to ON, and check that the LED for memory rewrite switch goes off.

8-4-6. Examples of Surface Cutting With End Mill (R Cutting)

Machining zero position
(Center position of the spindle when the table is at the zero position.)



Workpiece material: Aluminum
Machining (final): End mill
Machining order: memory data
Tool pattern: memory data
Cutting tool/condition: Memory data

Item	Key operation	Contents
Mode key	PROGRAM PROTECT ON OFF 	Switch the Program protect switch from ON to OFF, and check that the LED for the memory rewrite ENABLE comes on.
		
Menu selection	1 ENTER	1. Machining data
Program No.	2116 ENTER	

Working zero position X	-6.0 ENTER	X axis position of the working zero position from the machining zero position
Y	-4.3 ENTER	Y axis position of the working zero position from the machining zero position
Z	-4.7 ENTER	Height of the Z axis from the table top to the bottom face of the workpiece
Tool return Height	1.0 ENTER	Z axis returning height from one job to the other (Position from the Z axis working zero position)
Number of workpieces	1 ENTER	Number of setting workpieces
Work pieces material	4 ENTER	Selection from the teaching data area 4. ACD12
	Displays job 01. Displays the final machining or the final shape of job 01.	
Job 01	9 ENTER	Selection from the teaching data area 9. Coolant
ON/OFF	1 ENTER	Selection from the teaching data area 1. On
	Displays job 02. Displays the final machining or the final shape of job 02.	
Job 02	8 ENTER	Selection from the teaching data area 8. Milling

Tool diameter	0.4 ENTER	Diameter of end mill tool Linear cutting
Cutting length X	2.8 ENTER	Feed amount of cutting toward the X axis (Specify the direction by the incremental method.)
Y	0 ENTER	Feed amount of cutting toward the Y axis (Specify the direction by the incremental method.)
Z position mode	1 ENTER	Selection from the teaching data area 1. Cutting mode
Depth	0.2 ENTER	Cutting depth
Height	0.8 ENTER	Height from the working zero position to the top face of the workpiece
Ht	0.6 ENTER	Returning height of the Z axis when machining operation ends (Height from the Z axis working zero position)
XY position X	-0.4 ENTER	X axis position at machining start point (Position from the X axis working zero position)
Y	2.2 ENTER	Y axis position at machining start point (Position from the Y axis working zero position)
Peripheral speed rate	100 ENTER	100% Instruction of peripheral velocity of cutting tool condition in percentage (Refer to Section 7-5-2, "Cutting Tool/Condition".)

Feed rate	100 ENTER	100% Instruction of feed amount per revolution of cutting tool condition in percentage (Refer to Section 7-5-2, "Cutting Tool/Condition".)
	Displays job 03. Displays the final machining or final shape of job 03.	
Job 03	8 ENTER	Selection from the teaching data area 8. Milling
Tool diameter	-0.4 ENTER	Diameter of end mill tool R cutting
R start X	0 ENTER	X axis position from R cutting start point (Position from the center of circle)
Y	1.4 ENTER	Y axis position from R cutting start point (Position from the center of circle)
R end X	1.4 ENTER	X axis position from the R cutting end position (Position from the center of circle)
Y	0 ENTER	Y axis position from the R cutting end position (Position from the center of circle)
Direction	1 ENTER	Selection from the teaching data area 1. CW
Z axis position Mode	1 ENTER	Selection from the teaching data area 1. Cutting mode

Depth	0 ENTER	Cutting depth
Height	0.6 ENTER	Height from the working zero position to the top face of the workpiece
Ht	0.6 ENTER	Returning height of the Z axis when machining operation ends (Height from the Z axis working zero position)
XY position X	2.4 ENTER	X position of the center of circle (Position from the X axis working zero position)
Y	0.8 ENTER	Y position of the center of circle (Position of the Y axis working zero position)
Peripheral feed Peripheral velocity	100 ENTER	100% Instruction of peripheral velocity of cutting tool condition in percentage
Feed rate	100 ENTER	100% Instruction of feed amount per revolution of cutting tool condition in percentage
	Displays job 04. Displays the final machining or final shape of job 04.	
Job 04	0.4 ENTER	Selection from the teaching data area 8. Milling
Tool diameter	10. ENTER	Diameter of end mill tool Linear cutting
Cutting length X	0 ENTER	Feed amount of cutting toward the X axis (Specify the direction by the incremental method.)

Y	-1.6 ENTER	Feed amount of cutting toward the Y axis (Specify the direction by the incremental method.)
Z position mode	1 ENTER	Selection from the teaching data area 1. Cutting mode
Depth	0 ENTER	Cutting depth
Height	0.6 ENTER	Height from the working zero position to the top face of the workpiece
Ht	0.6 ENTER	Returning height of the Z axis when machining operation ends (Height from the Z axis working zero position)
XY position X	3.8 ENTER	X axis position at machining start point (Position from the X axis working zero position)
Y	0.8 ENTER	Y axis position at machining start point (Position from the Y axis working zero position)
Peripheral speed rate	100 ENTER	100% Instruction of peripheral velocity of cutting condition in percentage
Feed rate	100 ENTER	100% Instruction of feed amount per revolution of cutting tool condition in percentage
	Displays job 05. Displays the final machining or final shape of job 05.	
Job 05	8 ENTER	8. Milling

Tool diameter	-0.4 ENTER	Diameter of end mill tool R cutting
R start point X	1.4 ENTER	X axis position from R cutting start point (Position from the center of circle)
Y	0 ENTER	Y axis position from R cutting start point (Position from the center of circle)
R end point X	0 ENTER	X axis position from the R cutting end position (Position from the center of circle)
Y	-1.4 ENTER	Y axis position from the R cutting end position (Position from the center of circle)
Direction	1 ENTER	Selection from the teaching data area 1. CW
Z position mode	1 ENTER	Selection from the teaching data area 1. Cutting mode
Depth	0 ENTER	Cutting depth
Height	0.6 ENTER	Height from the working zero position to the top face of the workpiece
Ht	0.6 ENTER	Returning height of the Z axis when machining operation ends (Height from the Z axis working zero position)

XY position X	2.4 ENTER	X position of the center of circle (Position from the X axis working zero position)
Y	-0.8 ENTER	Y position of the center of circle (Position of the Y axis working zero position)
Peripheral speed rate	100 ENTER	100% Instruction of peripheral velocity of cutting condition in percentage
Feed rate	100 ENTER	100% Instruction of feed amount per revolution of cutting tool condition in percentage
	Displays job 06. Displays the final machining or final shape of job 06.	
Job 06	8 ENTER	Selection from the teaching data area 8. Milling
Tool diameter	10. ENTER	Diameter of end mill tool Linear cutting
Cutting length X	-2.8 ENTER	Feed amount of cutting toward the X axis (Specify the direction by the incremental method.)
Y	0 ENTER	Feed amount of cutting toward the Y axis (Specify the direction by the incremental method.)
Z position mode	1 ENTER	Selection from the teaching data area 1. Cutting mode
Depth	0 ENTER	Cutting depth

Height	0.6 ENTER	Height from the working zero position to the top face of the workpiece
Ht	1.0 ENTER	Returning height of the Z axis when machining operation ends (Height from the Z axis working zero position)
XY position X	2.4 ENTER	X axis position at machining start point (Position from the X axis working zero position)
Y	-2.2 ENTER	Y axis position at machining start point (Position from the Y axis working zero position)
Peripheral speed rate	100 ENTER	100% Instruction of peripheral velocity of cutting condition in percentage
Feed rate	100 ENTER	100% Instruction of feed amount per revolution of cutting tool condition in percentage

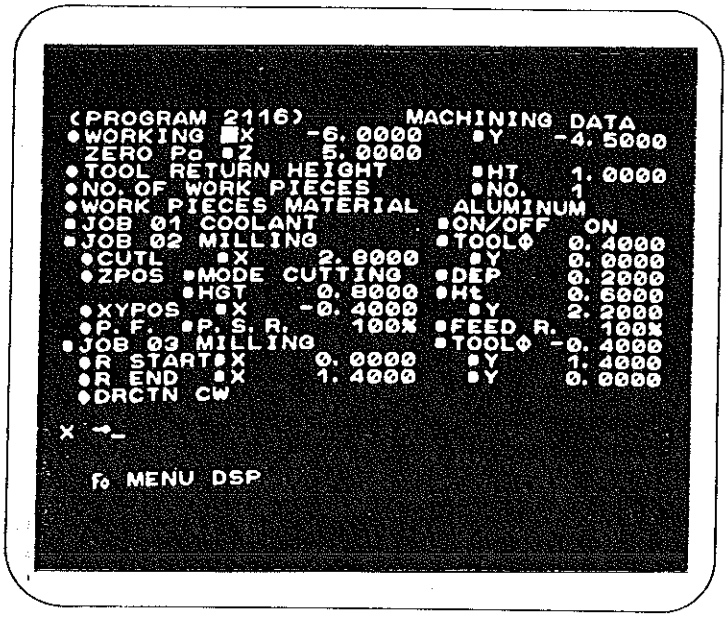
Displays job 09.
Sets the final machining or final shape of job 09.

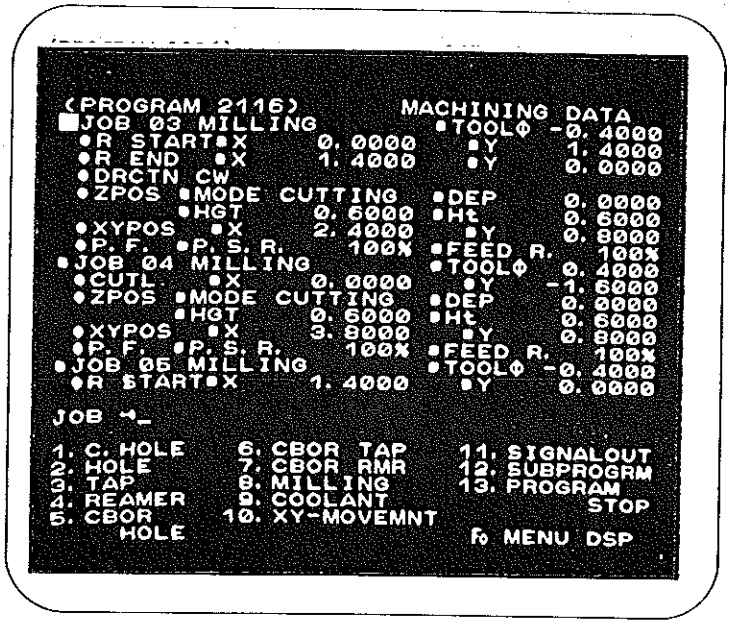
Job 09	9 ENTER	Selection from the teaching data area 9. Coolant
ON/OFF	2 ENTER	Selection from the teaching data area 2. Off

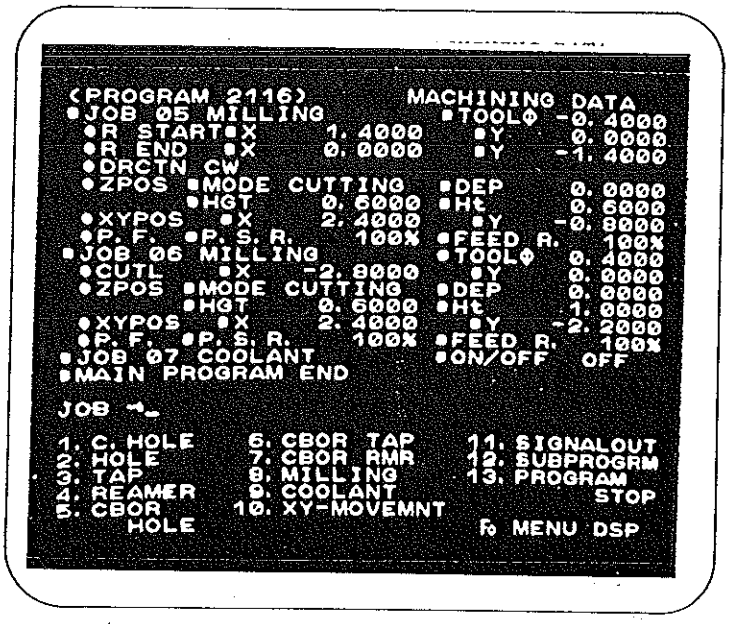
Displays job 10.
Sets the final machining or final shape of job 10.

-99 ENTER	Selection from the teaching data area -99. Main program END

		Since a series of operations are completed, end the main program.
Number	-99. ENTER	Selection from the teaching data area -99. Subprogram END







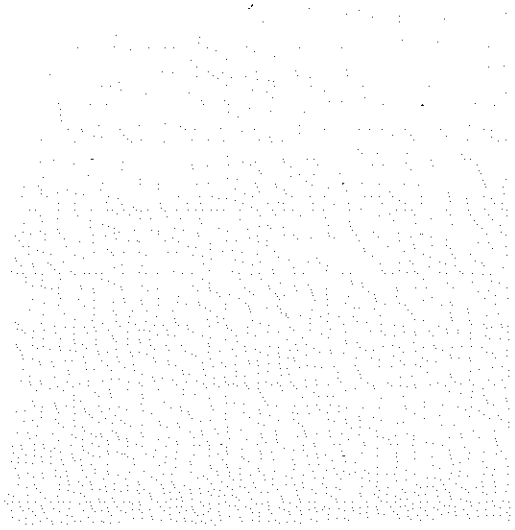
	Fo	Work input end The display is returned to the menu screen.

* Since the memory data is used for items 2 to 4 in the menu screen, no operation is necessary.
(Refer to Section 8-4-1, "Linear Pattern Drilling.)

Menu screen	5 ENTER	5. Tool assign Necessary tools are automatically selected and set according to the program contents, tool condition, and cutting tool condition.

Available tool (end mill)
ex) The following tools are registered to the tool menu.

Tool No. Tool Name	48 End mill
Point angle	*
Small diameter	*
Small diameter length	*
Center angle	*
Ineffective length	*
Major diameter nominal size	0.4
Pitch/thread per pitch	*
Effective cutting length	0.8
Torsion direction	*
Tool length	5.5
Life	9000

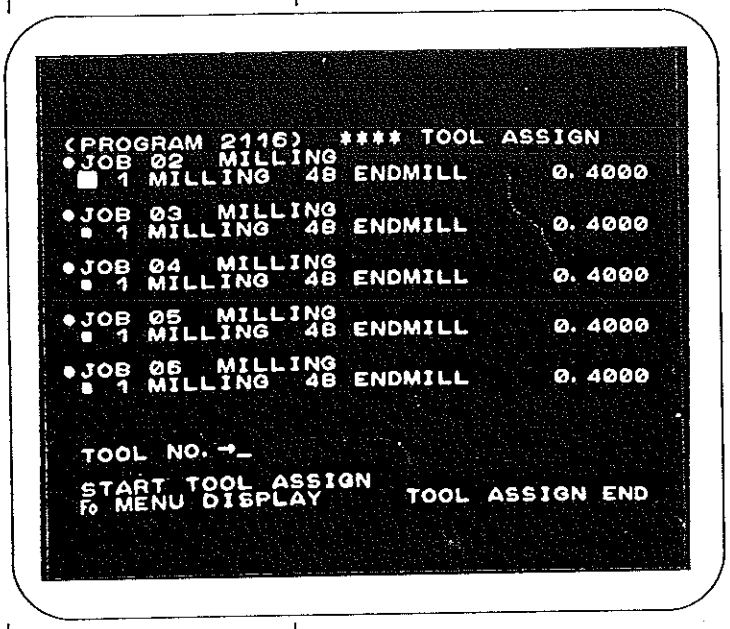


(Refer to Section 7-3. Tool Menu.)

Program No. 2116 ENTER

START

(Refer to Section 8-3-5. "Tool Assign".)



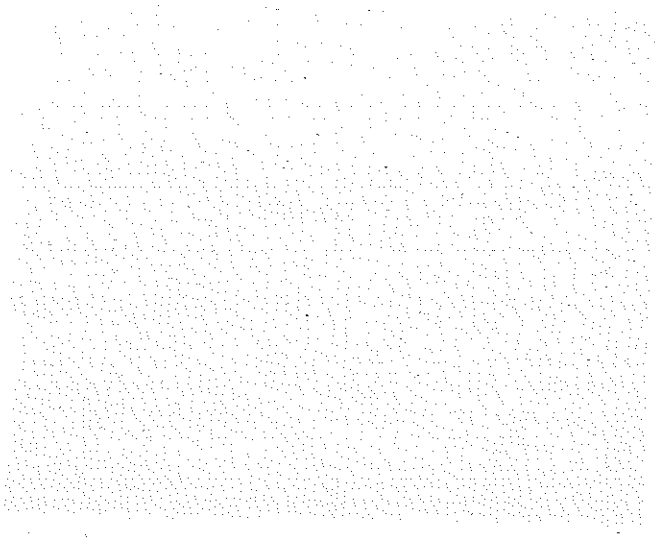
F0

Tool assign end.
The display is returned to the menu screen.

When tools are not assigned, refer to Section 8-3-5. "Tool Assign".)

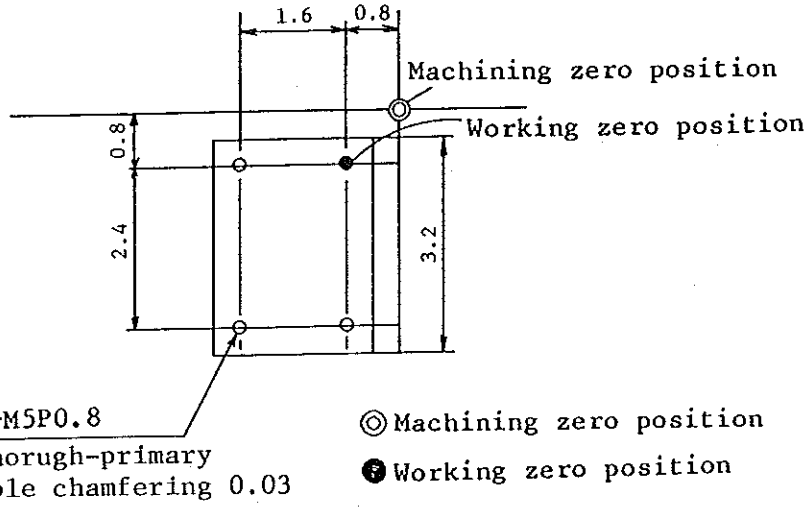
PROGRAM PROTECT
ON OFF

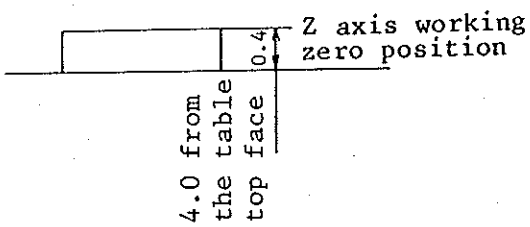
Switch the Program protect switch from OFF to ON, and check that the LED for memory rewrite ENABLE goes off.



8-4-7. Examples of Schedule Machining

Example of Program Edit (A)





Workpiece material: Aluminum

Machining (final): Tap

Machining order: memory data

Tool pattern: memory data

Cutting tool/condition: Memory data

Item	Key operation	Contents
	PROGRAM PROTECT ON OFF 	Switch the PROGRAM PROTECT switch from ON to OFF, and check that the LED for the memory rewrite ENABLE comes on.
Mode key	EDIT	
Menu selection	1 ENTER	1. Machining data
Program No.	1117 ENTER	

Working zero position X	-0.8 ENTER	X axis position of the working zero position from the machining zero position
Y	-0.8 ENTER	Y axis position of the working zero position from the machining zero position
Y	4.0 ENTER	Height of the Z axis from the table top to the bottom face of the workpiece
Tool return Height	0.5 ENTER	Z axis returning height from one job to the other (Position from the Z axis working zero position)
Number of workpieces	1 ENTER	Number of setting workpieces
Work pieces material	4 ENTER	Selection from the teaching data area 4. Aluminum

Displays job 01.
Displays the final machining or the final shape of job 01.

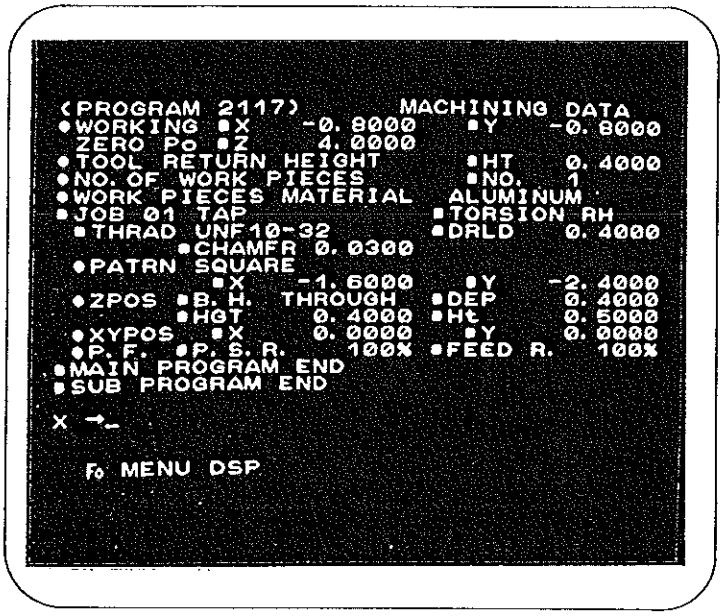
Job 01	3 ENTER	Selection from the teaching data area 3. Tap
Torsion	1 ENTER	Selection from the teaching data area 1. Right-handed thread
Thread	4 ENTER	Selection from the teaching data area 4.
Nominal diameter	10 ENTER	Nominal diameter of thread.

Thread/inch	0 32 ENTER	Thread pitch
Drilling hole depth	0.4 ENTER	Drill machining depth
Chamfering	0.03 ENTER	When chamfering is unnecessry, set to 0.
Pattern	2 ENTER	Selection from the teaching data area 2. Square pattern
X	-1.6 ENTER	Pitch of the square pattern in the X direction
Y	-2.4 ENTER	Pitch of the square pattern in the Y direction
Z axis position Hole bottom	1 ENTER	Selection from the teaching data area
Depth	0.4 ENTER	Depth of machining hole
Height	0.4 ENTER	Height from the Z axis working zero position to the top face of the workpiece
Ht	0.5 ENTER	Returning height of the Z axis when the axis moves from a working position to another working position in a job (Height from the Z axis working zero position)
XY position X	0 ENTER	X axis position at machining start point (Position from the X axis working zero position)
Y	0 ENTER	Y axis position at machining start point (Position from the Y axis working zero position)
Peripheral speed rate	100 ENTER	100% Instruction of peripheral velocity of cutting tool condition in percentage (Refer to Section 8-5-2, "Cutting Tool/Condition".)

Feed rate	100 ENTER	100% Instruction of feed amount per revolution of cutting tool condition in percentage (Refer to Section 8-5-2, "Cutting Tool/Condition".)

Displays job 02.
Sets the final machining or final shape of job 02.

	-99 ENTER	Selection from teaching data area -99. Main program END Since a series of operations are completed, the main program is ended.
Number	-99 ENTER	Selection from the teaching data area -99. Subprogram END



* Since the memory data is used for items 2 to 4 in the menu screen, the operation is unnecessary.
(Refer to Section 8-4-1, "Examples of Linear Drilling".)

Fo	Working input end The display is returned to the menu screen.

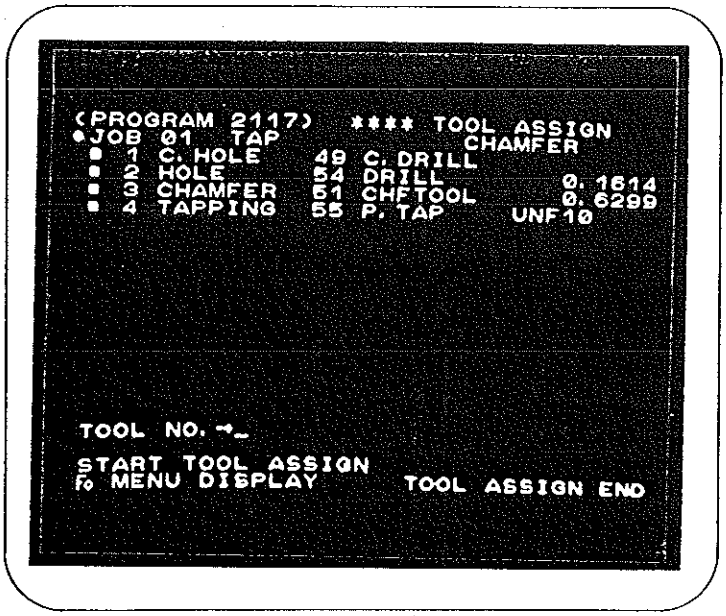
Menu screen	5 ENTER	5. Tool Assign Necessary tools are set according to the program contents, tool menu, and cutting tool condition.

Available tools (Center drill, drill, chamfering tool, P tap)

Tool No. Tool Name	49 Center drill	51 Chamfering tool	54 Drill	55 P tap
Point angle	0	*	118.	*
Small diameter	0	0.02	*	*
Small diameter length	0	*	*	*
Center angle	90.	90.	*	*
Ineffective length	*	0	*	0.06
Major diameter nominal size	0.4	0.6299	0.17	UNF10
Pitch/thread per pitch	*	*	*	32
Effective cutting length	*	*	0.8	0.88
Torsion direction	*	*	*	RH
Tool length	6.0	6.0	5.5	5.5
Life	9000	9000	9000	9000

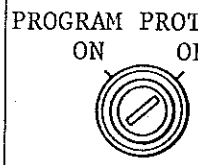
Note) Tool No. 55 is a general type of point tap.
(Refer to Section 7-3, "Tool Menu".)

Program No.	1117 ENTER	
	START	Refer to Section 8-3-5, "Tool Assign".
	F0	The display is returned to the menu screen.
	PROGRAM PROTECT ON OFF	Switch the memory Protect switch from OFF to ON, and check that the LED for memory rewrite ENABLE goes off.



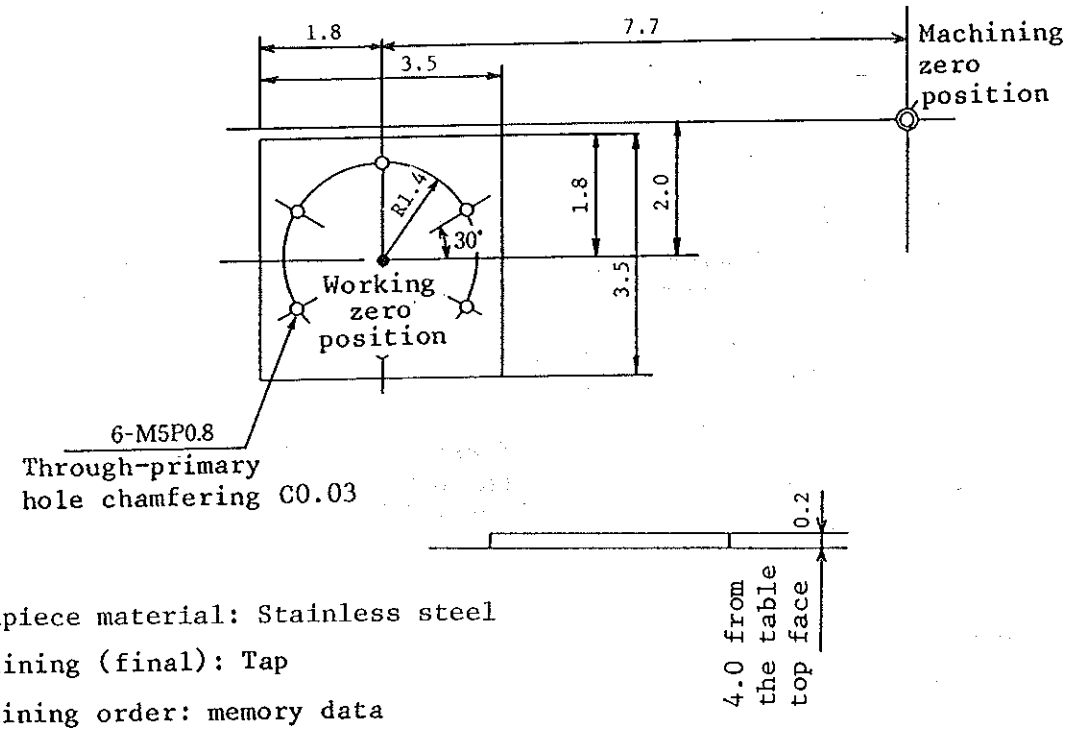
When tools are not assigned, refer to Section 8-3-5, "Tool Assign".)

The display is returned to the menu screen.



Switch the memory Protect switch from OFF to ON, and check that the LED for memory rewrite ENABLE goes off.

Example of Program Edit (B)



Workpiece material: Stainless steel

Machining (final): Tap

Machining order: memory data

Tool pattern: memory data

Cutting tool/condition: Program characteristic

Item	Key operation	Contents
	PROGRAM PROTECT ON OFF 	Switch the PROGRAM PROTECT switch from ON to OFF, and check that the LED for the memory rewrite ENABLE comes on.
Mode key	EDIT	
Menu selection	1 ENTER	1. Machining data
Program No.	1118 ENTER	

Working zero position X	-7.7 ENTER	X axis position of the working zero position from the machining zero position
Y	-2.0 ENTER	Y axis position of the working zero position from the machining zero position
Z	4.0 ENTER	Height of the Z axis from the table top to the bottom face of the workpiece
Tool return Height	0.3 ENTER	Z axis returning height from one job to the other (Position from the Z axis working zero position)
Number of workpieces	1 ENTER	Number of setting workpieces
Work pieces material	5 ENTER	Selection from the teaching data area 5. Stainless steel

Displays job 01.
Displays the final machining or the final shape of job 01.

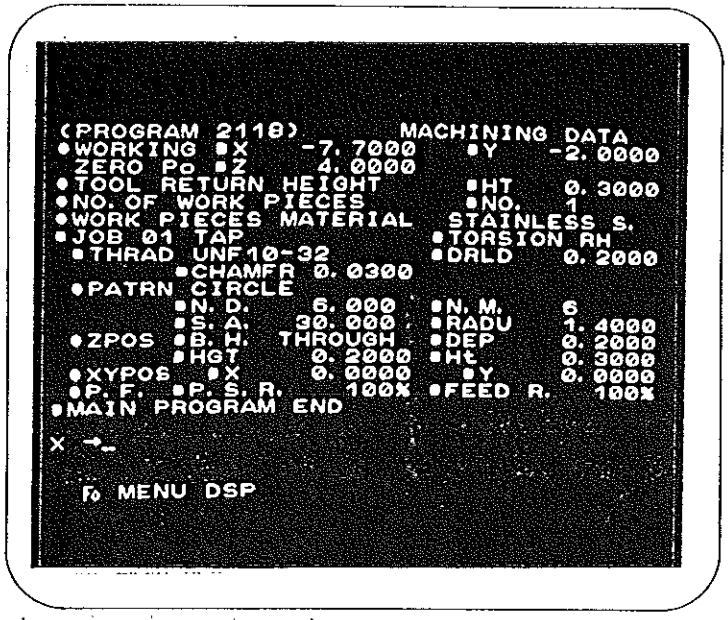
Job 01	3 ENTER	Selection from the teaching data area 3. Tap
Torsion	1 ENTER	Selection from the teaching data area 1. Right-handed thread
Thread	4 ENTER	Selection from the teaching data area 4.
Nominal diameter	10 ENTER	Nominal diameter of screw

Thread/inch	32 ENTER	Thread pitch
Drilling hole depth	0.2 ENTER	Drill machining depth
Chamfering	0.03 ENTER	When chamfering is unnecessry, set to 0.
Pattern	1 ENTER	Selection from the teaching data area 1. Circumference pattern
Number of division	6. ENTER	Set the number of division of the circumference (Decimal point required)
Machining number	6 ENTER	Number of machining holes
Start angle	30. ENTER	Set the angle of machining start position to the X axis on the XY plane
Radius	1.4 ENTER	Circumference pattern
Z axis position Hole bottom	1 ENTER	Teaching data area 1. Through
Depth	0.2 ENTER	Depth of machining hole
Height	0.2 ENTER	Height from the Z axis working zero position to the top face of the workpiece
Ht	0.3 ENTER	Returning height of the Z axis when the axis moves froma working position to another working position in a job (Height from the Z axis working zero position)
XY position X	0 ENTER	X axis position at machining start point (Position from the X axis working zero position)
Y	0 ENTER	Y axis position at machining start point (Position from the Y axis working zero position)

Peripheral speed rate	100 ENTER	100% Instruction of peripheral velocity of cutting tool condition in percentage (Refer to Section 8-5-2, "Cutting Tool/Condition".)
Feed rate	100 ENTER	100% Instruction of feed amount per revolution of cutting tool condition in percentage (Refer to Section 8-5-2, "Cutting Tool/Condition".)

Displays job 02.
Sets the final machining or final shape of job 02.

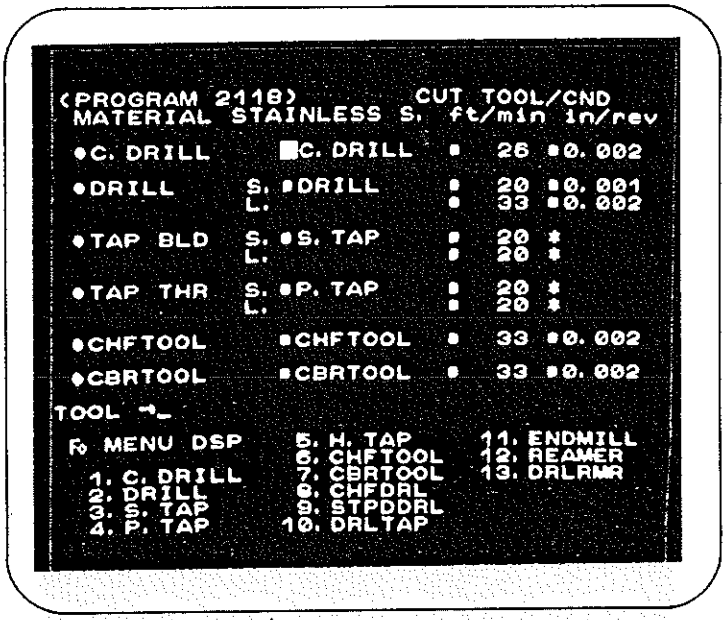
-99 ENTER	Selection from teaching data area -99. Main program END Since a series of operations are completed, the main program is ended.
Number -99 ENTER	Selection from the teaching data area -99. Subprogram END



	Fo	Machining input end The display is returned to the menu screen.

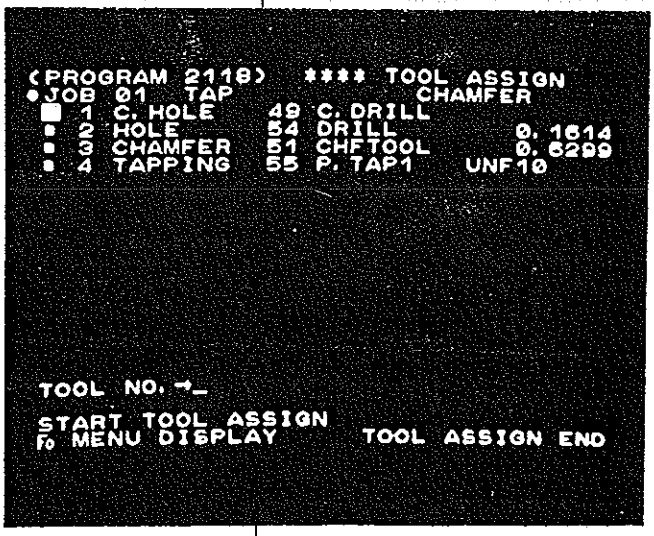
* Since the memory data is used for items 2 and 3 in the menu screen, the operation is unnecessary.
(Refer to Section 8-4-1, "Examples of Linear Drilling".)

Menu No.	4 ENTER	4. Cutting tool/condition
Program No.	1118 ENTER	



	Fo	Machining data input end. The display is returned to the menu screen.

	⇧ CURSOR ⇩ CURSOR	Set the cursor to the tap-through tool.
P Tap	4-1 ENTER	P tap-1 Set new auxiliary code. (Refer to Section 7-5-2, "Cutting Tool/Condition")
	Fo	The display is returned to the menu screen.
Menu No.	5 ENTER	5. Tool assign Necessary tools can are set according to the program contents, tool menu, and cutting tool condition.



		(When tools are not assigned refer to Section 8-3-5 "Tool Assign")
	Fo	Tool assigned. The display is returned to the menu screen.
	PROGRAM PROTECT ON OFF	Switch the PROGRAM PROTECT switch from OFF to ON, and check that the LED for memory rewrite ENABLE goes off.

Available tools (Center drill, chamfering tool, drill, P tap-1)

Tool No. Tool Name	49 Center drill	51 Chamfering tool	54 Drill	55 P tap
Point angle	0	*	135	*
Small diameter	0	0.02	*	*
Small diameter length	0	*	*	*
Center angle	90.	90.	*	*
Ineffective length	*	0	*	0.06
Major diameter nominal size	0.4	0.6299	0.17	UNF10
Pitch/thread per pitch	*	*	*	32
Effective cutting length	*	*	0.8	0.88
Torsion direction	*	*	*	RH
Tool length	6.0	6.0	5.5	5.5
Life	9000	9000	9000	9000

Note) Tool No. 56 is a point tap for stainless.
(Refer to Section 8-3, "Tool Menu".)

Program No. 1118 ENTER

START

(Refer to Section 8-3-5, "Tool Assign".)
Tool Assign END.

F0

Return to menu display.

PROGRAM PROTECT
ON OFF

Switch the Program protect switch to OFF to ON, and check that the LED for memory rewrite ENABLE goes off.

* If use schedule program refer to 9 of memory run.

9. Memory Run

9-1. Contents of Memory Run

By calling the programs edited as described in the previous subparagraph or external memory program, the following operation can be made in memory run mode. However, different data is used for the external memory program, assign the tools again.

For calling the program, key in the [PRGRM] and press the [ENTER] key.

* The schedule program can be called by setting the program No. to 0.

1. Dry run

Dry run allows quick checking of the called program by accelerating only the jog feedrate (rapid feedrate in the manual run) in test run.
Another function is providing better safety operation by setting offset amount by use of the Z axis as parameter 1.

2. Cycle run

The called programs can be consecutively executed.

3. Single run

- o The called program allows operation per action.
- o ATC makes one action with Z axis ascent, magazine index (table positioning at the same time), and Z axis descend. Care must be taken especially when the long tool is set in the magazine, because the table of the next job is positioned during the magazine index.
- o Tapping makes one action with one returning.
- o Step operation of drilling or tapping makes one action when it reaches the foremost end under command, irrespective of the number of step.

4. The job in machining can be confirmed by having the program under memory run displayed. Unless run mode is changed, each data can be confirmed by commanding display control keys. If not required, the display can be deleted.

5. Program can be edited during memory run (setting of machining data, machining order, tool pattern and cutting tool/condition).

   * In memory run, some screen displays may take several seconds depending on program contents.

o Preparation for memory run

Check to see the following information for memory run:

(1) Program data
(Machining data, machining order, tool pattern, cutting tool/condition, tool assignment)
(2) Tool menu
(3) Part of parameters
(Dimension after cutting through, position before work, etc.)

After the information is checked, set the tools in the magazine.

9-2. Preparation and Operation

Remember to confirm the following information before starting Memory Run.

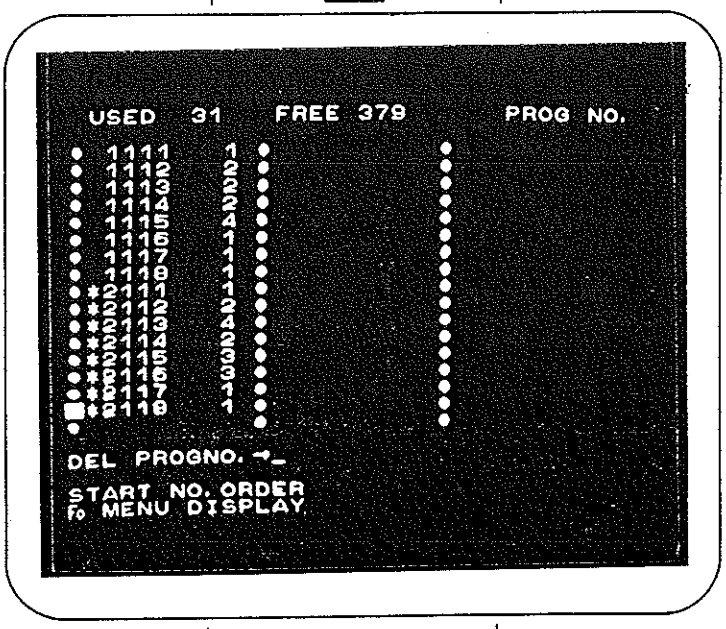
(1) Program data
(Machining data, machining order, tool patterns, cutting tools, cutting conditions, tool assignment)

(2) Tool list

(3) Relative parameters
(Cutting depth after piercing, stop position before work piece, etc.)

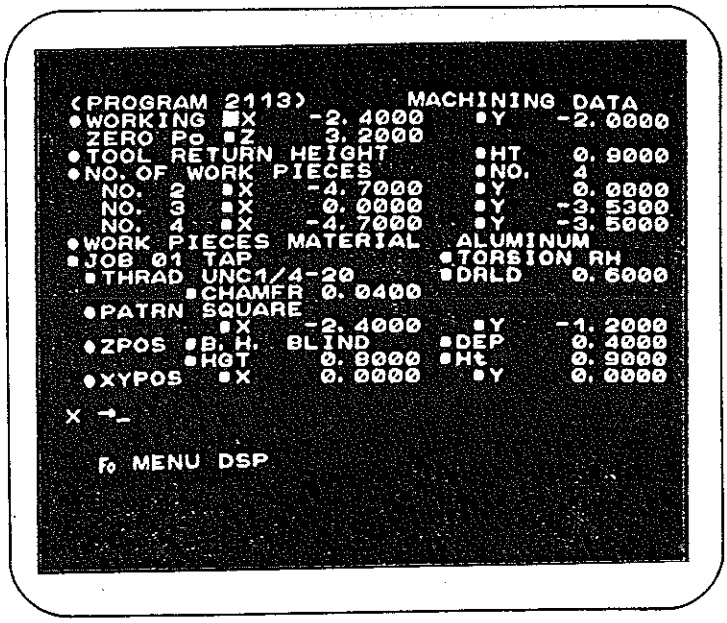
After confirming the above-mentioned information, set the tools to the magazine.
After completion of the tool setting, call the programs in the Memory Run mode, Single Run mode, and Dry Run mode, and check the running condition of the started machine. If everything was found normal, start the Cycle Run.

Item	Key operation	Contents
Mode key	EDIT	
Menu No.	6 ENTER	6. Program memory display: order



The memory run is performed in this screen. Check for program No. (1113 in this screen).

	F0	Return to menu display.
Menu screen	1 ENTER	1. Machining data
Program No.	2113 ENTER	

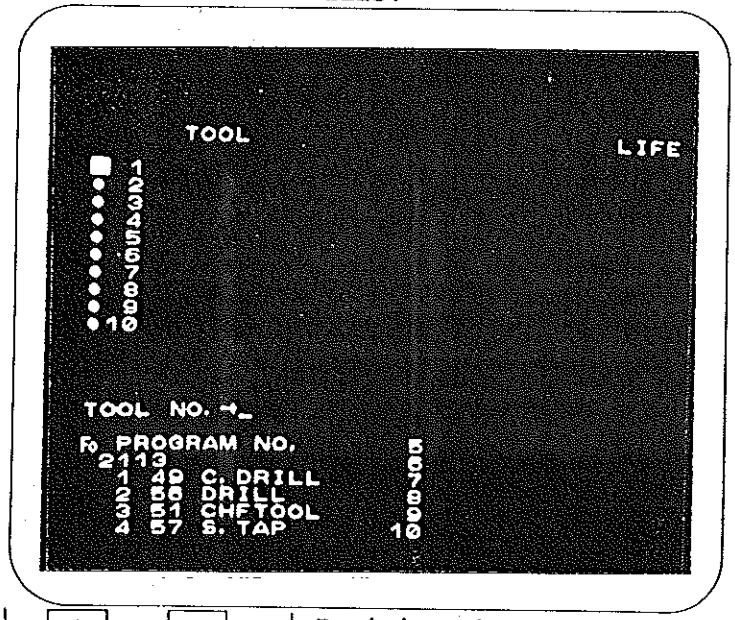


Check the program contents in this screen.

Item	Key operation	Contents
	Fo	

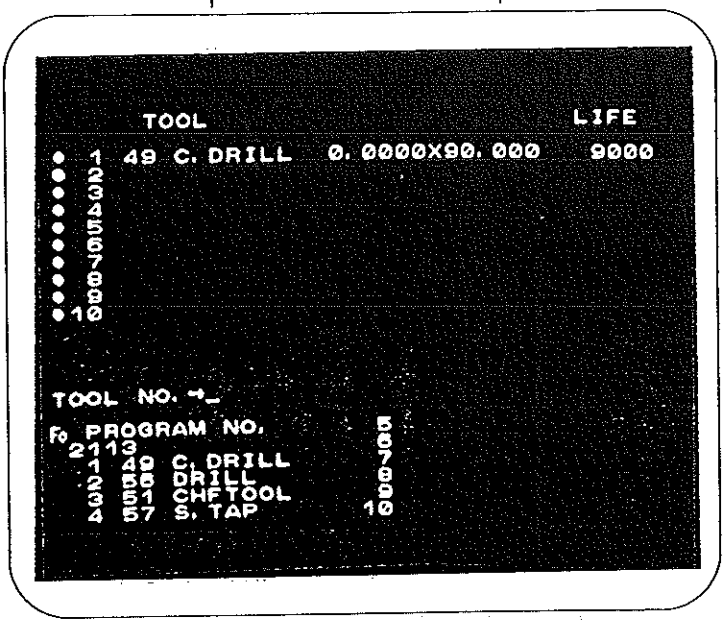
Check data according the above procedure.

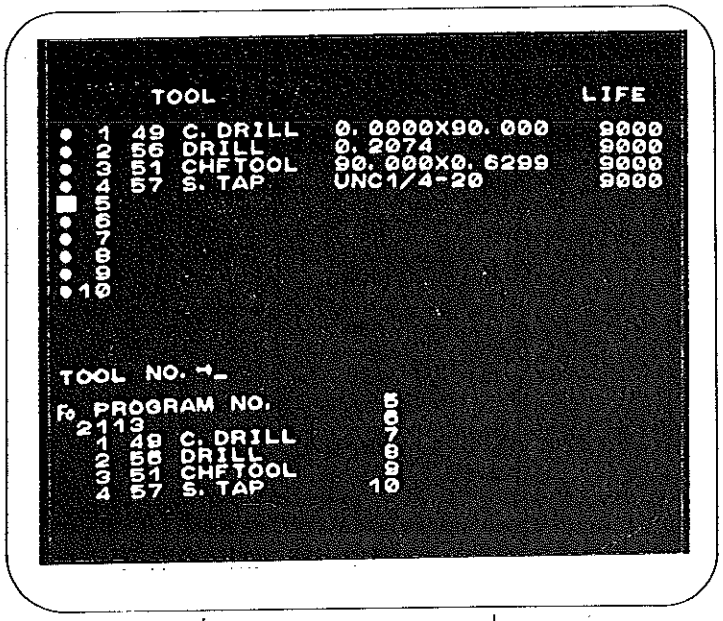
Mode key	MDI PROGRAM PROTECT ON OFF	Switch the PROGRAM PROTECT from ON to OFF, and check that LED for the memory rewrite ENABLE comes on.
Display control key	MAGAZ	
Program No.	2113 ENTER	Displays data area displays the tool which has been assigned to program No. 2113.

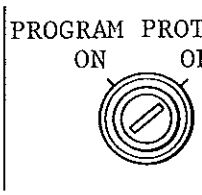


	⇧ CURSOR ⇩ CURSOR	Position the cursor at the magazine No. to be set. (Any number is allowed. In the above display, the cursor is positioned at the 1.)
Tool No.	49 ENTER	Center drill with tool No. 49 is set at magazine (1).

Item	Key operation	Contents
		As mentioned, tools displayed in the display data area are set in the magazine.





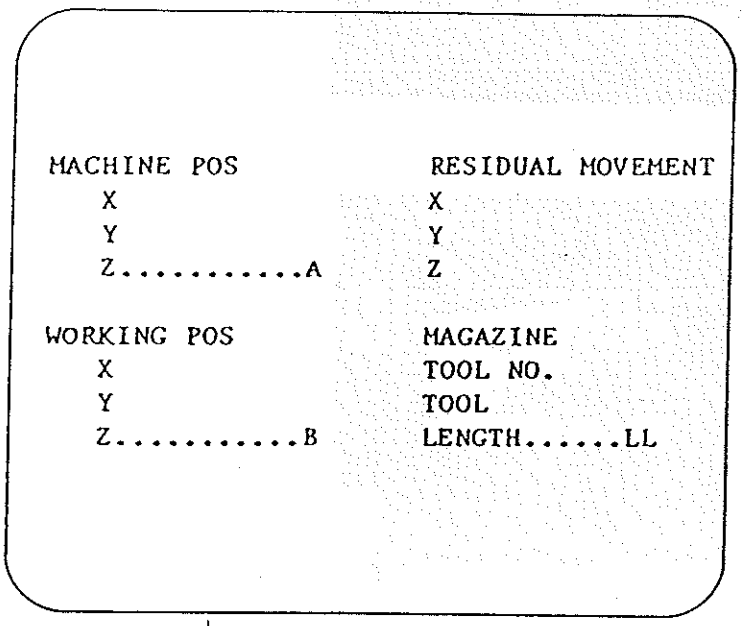


Switch the PROGRAM PROTECT switch from OFF to ON, and check that the LED for the memory rewrite enable goes off.

Magazine tools are installed in the magazine as set. (For details on installation, refer to V-4, 'Tool Installation and Removal".)

* When heavy tool is installed in the magazine, take care for the balance of the tools.

Item	Key operation	Explanation
Mode key	MEM	
Program No.	1113 ENTER	Check whether the tools assigned by program editing and set to the magazine smoothly work without trouble or not when the machine was driven by the called program, and whether wrong tools are not set to the magazine. Then confirm the working condition of the machine in Dry Run, and check whether every tool is free from interference trouble by means of the position display and the Single Run screen.
	DRY RUN, SINGL RUN	
	START	
	POS	

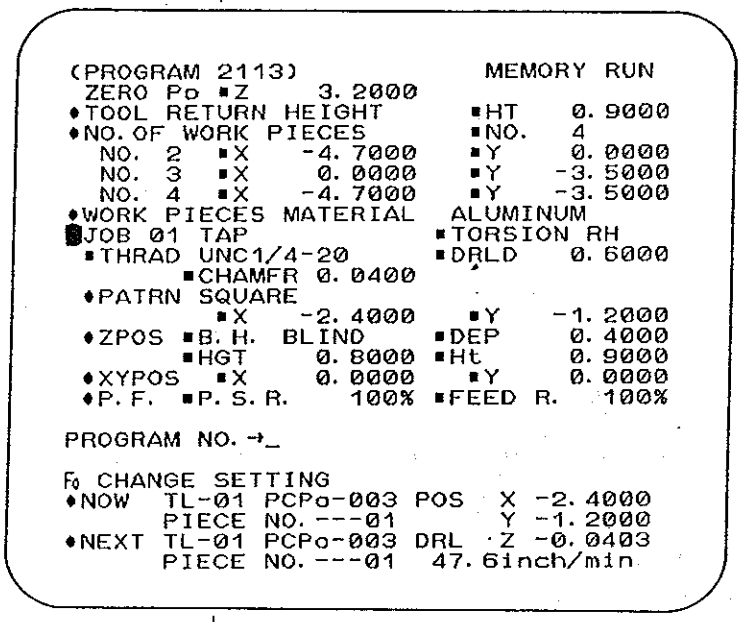


A: Coordinate position of the machine.
LL: Length of the tool presently set to the spindle.
B: Coordinate position of work (tool head position)
B = A - LL - (Z work zero point in program)

(Note) While in Dry Run, the values of A and B are kept offset as much as the dry run offset values.

The distance between the work coordinate position B shown above and the Z axis work coordinate position shown below on the Single Run screen denotes the next moving distance of the Z axis. It is possible, therefore, to continue the checking work while measuring the space between the tool head and the top face of the work.

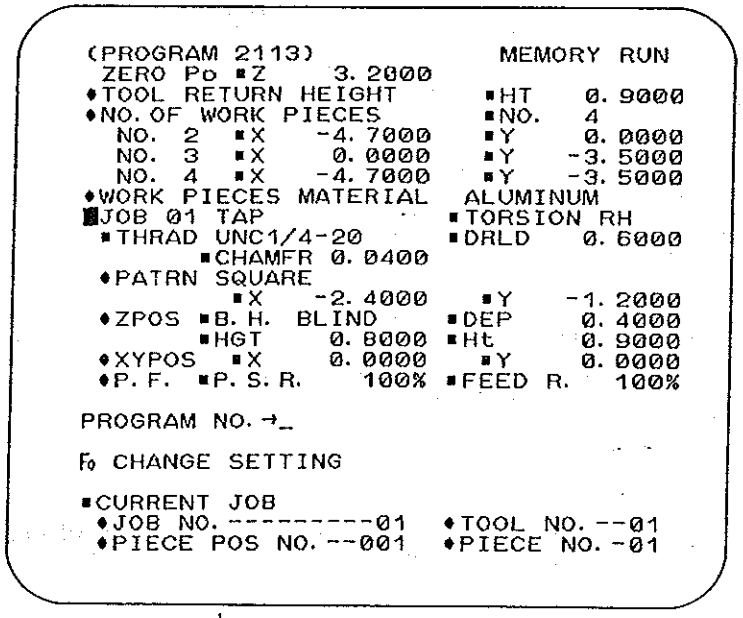
Item	Key operation	Explanation
Mode key	SINGL RUN	For program display.
	START	For one cycle program run.



Confirm the contents of operation while observing the information displayed in the lower part of the screen.

- o Tool : To indicate the sequence No. (order of tool patterns) of the tool to be used in that process.
- o Work position : To indicate the sequence No. of the machining position.
- o Contents of work: To indicate the contents of work (positioning, spindle, cutting feed, etc.) per 1 cycle operation.
- o Work No. : To indicate the sequence No. of the work presently under machining.

Item	Key operation	Explanation
Mode key	CYCL RUN START	In case the contents of the program shown on the preceeding page were found normal, confirm that the cursor and the process indicated in the bottom part of the screen are positioned at the program head.



o Process : To indicate the No. of the process presently at work. (It is also indicated by the cursor.)

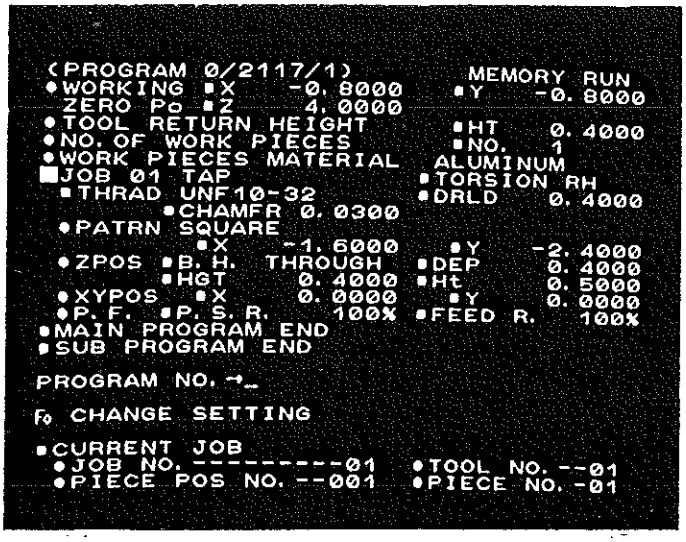
o Tool : To indicate the sequence No. (order of tool patterns) of the tool to be used in that process.

o Work position: To indicate the sequence No. of the machining position.

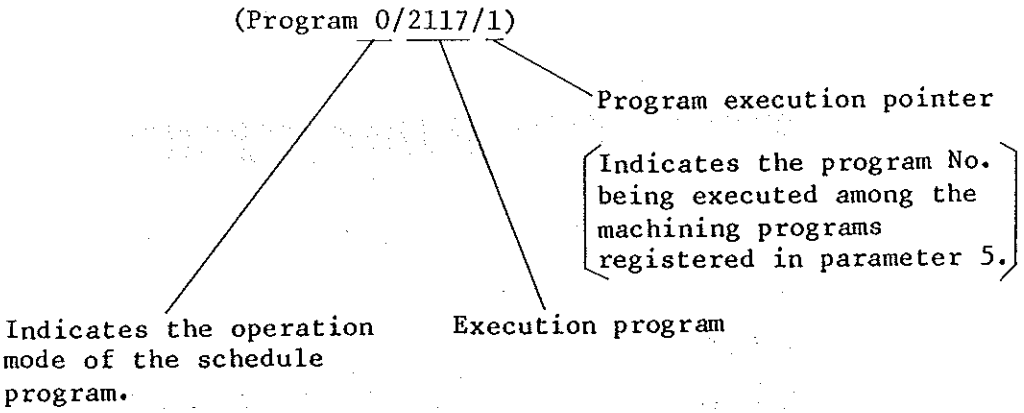
o Work No. : To indicate the sequence No. of the work presently under machining.

Schedule program run

When program No. is set to [0] in the memory run mode, the schedule program is called. (In this case, program is assumed to be set in parameter 5.)



When the machine enters into the memory run mode for the schedule program, the execution program No. displayed on the upper left of the screen is different from that of the memory run mode for ordinary programs, as follows:



Pressing the START key activates the operation. Programs are executed one by one.

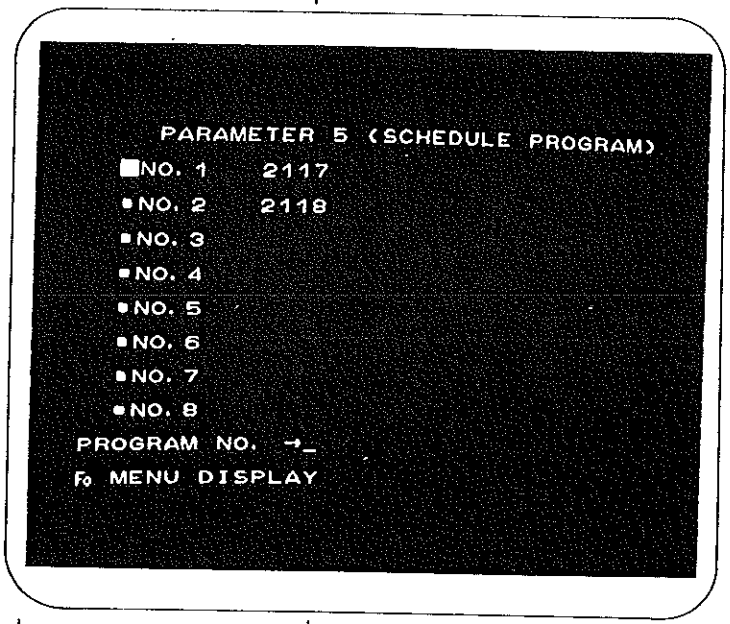
Except for the above description, this operation is the same as the ordinary memory run.

Operation Using the Schedule Program

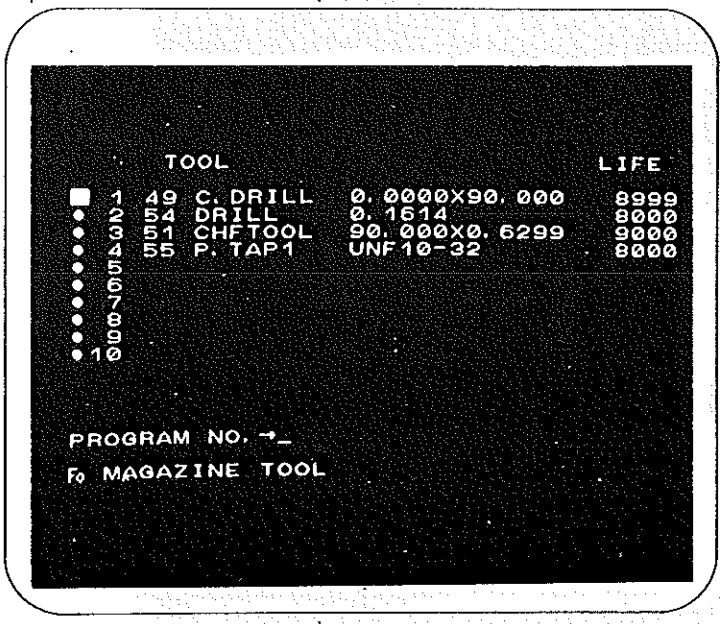
Materials are machined according to the examples of program edit using the schedule program.

Preparation for the schedule program operation

Item	Key operation	Contents
	PROGRAM PROTECT ON OFF 	Turn on the PROGRAM PROTECT switch to confirm that the "WRIT ENBL" LED is goes off.
Mode key		
Screen control key		
	5 	5. Schedule program
Program No.	1117 	
Program No.	1118 	



	Fo	Menu screen
Screen control key	MAGAZ	
Program No.	2117 ENTER	
	⇧ CURSOR ⇩ CURSOR	Move the cursor to a proper magazine No., and set the tool NO. of the assigned tool.
	Tool No. ENTER	Set all the tools assigned to program No. 2117 in the magazine.



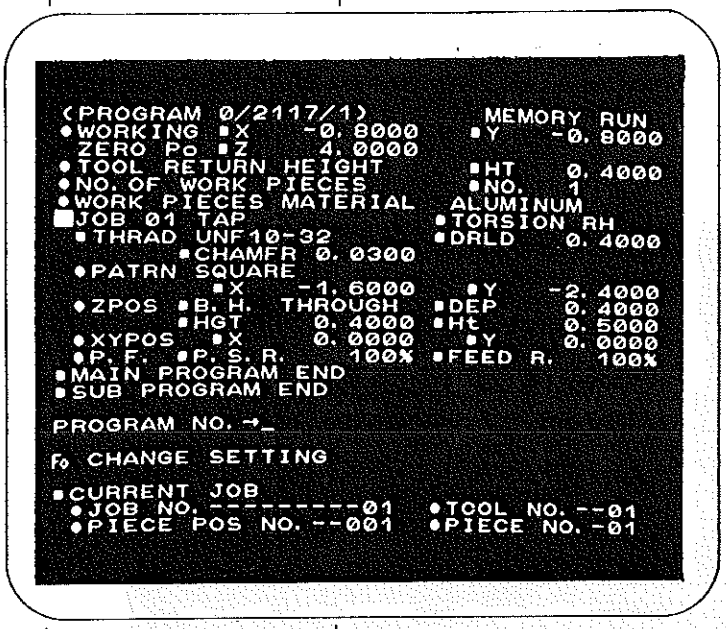
	Fo	
Program No.	2118 ENTER	
	⇧ CURSOR ⇩ CURSOR	Move the cursor to a magazine No. other than the above magazine No., and set the tool number of the assigned tool.
Tool No.	Tool No. ENTER	Set all the tools assigned to program No. 2118 in the magazine.
	PROGRAM PROTECT ON OFF	Switch the program protect switch to ON to OFF, and check that the LED for memory rewrite ENABLE goes off.

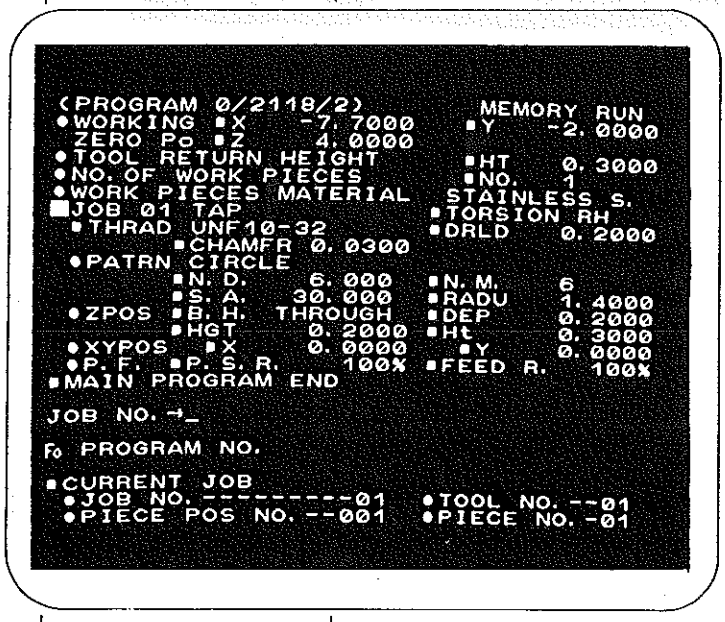
Schedule Program Run

Mode key — MEM

Program No. — 0 ENTER

START





9-2. Operation of Memory Run

1. ATC makes one action with Z axis ascend, magazine index (table positioning at the same time), and Z axis descend. Care must be taken when long tool is set in the magazine, because the table of the next job is positioned during the magazine index.

2. When the machining data is different from that for tool menu due to forced assignment, change of tool menu after assignment, and others, the data for tool menu are given priority.

3. Since information on assigned tools contains only tool numbers, data on the tool menu is used as it is even when the data is changed. Therefore, notice that change of tool types and others must take into account the assigned program after tools are set in the tool menu.

4. Determination of the portion before cutting

1) The position before cutting is determined to be the position before work, and the machine starts cutting.
When the position before work > returning position, the position is determined to be the returning position.

2) * When more than one workpiece exists and machining positions of the workpieces are different, and
* When the inter-job returning position is not matched with the height of the next job:
The higher position of the inter-job returning position and stop position before work is determined.

5. Positioning operation for returning after machining

a) Returning operation when the machining pattern area.
When (work height) $\geqq$ (returning position),
the machine returns to the stop position before work. (In the ordinary operation, the machine returns to the returning position.)

b) Returning operation when the machining pattern ends

When (work height) $\geqq$ (inter-job returning position), the machine returns to the stop position before work. (In the ordinary operation, the machine returns to the inter-job returning position.)

6. Cutting Condition and Upper Limit of the Speed

1) The revolution speed is calculated by the peripheral velocity of the cutting condition. However, when difference exists between the diameter of the machining data and the diameter of the tool data in the tool menu, the diameter of the tool data is given priority.

When the results of the calculation is higher than the highest speed of the machine, the machine rotates at its highest speed. (However, tapping speed may be limited by the feedrate.)

2) The feed rate is obtained by multiplying the above calculated revolution speed by feed amount per revolution specified by the cutting condition.

When the calculated result is higher than the maximum cutting feedrate of the machine, the machine rotates at the maximum cutting feedrate of the machine.

7. Cautions for Tool Operations

(1) Center drill

* Center drill with small diameter. Only when the center hole is machined, the machining depth is specified as small diameter length.
* Center drill without small diameter length. Only when the center hole is machined, the depth depends on the hole diameter of the job.

When (hole diameter) $\geqq$ 0.16 (inch):
Machined so that the diameter at the mouth is 0.16 inch.

When (Hole diameter) $<$ 0.16 (inch):
Machined so that the diameter at the mouth is same size as the hole diameter.

(2) Counter bore tool

* Before the counter bore tool with small diameter is used for machining a hole, a larger hole must be machined by the tool. (Otherwise, the operation stops before the work.)
* When the counter bore tool with small diameter is used, the point of the small diameter stops at the stop position before the work and machines at the job feed rate.

(3) Chamfering drill and stepped drill

Drill tap and drill reamer

When hole and face, hole and seat, hole and tap, and hole and reamer are machined at the same time, the material is cut with the small diameter length, cutting condition is changed, then face, seat, tap and reamer are machined.

9-3. Halt and Restart

o Halt

Press the HOLD key in memory run, the machine will halt at the position (during ATC operation or after tapping is completed).

Use the following key operation to stop the spindle.

1. MANU S.STOP

o Restart

o Restart when the machine starts from the stop position (while spindle is revolving)

Check the lamp display for the memory run, cycle or single run.

o Restart when the machine starts from the stop position (while spindle is at stop)

In the manual mode, set the cursor to the spindle revolution speed by suing the ⇧ CURSOR and ⇩ CURSOR key, and set the revolution speed nearest to the program revolution speed with △ and ▽ keys.

Press the CYCL RUN or SINGL RUN key in the memory run mode.

Press the START key.

o Restart when the machine restarts from the head position of the program

By pressing the [RESET] key, program is returned to the head position, and the cursor on the display screen indicates the first job. (However, in the dry run, the program is not returned to the head position, and the former state is maintained.)

Under part of the screen, current job, tool, machining position and work No. will be displayed. Check whether these are all at head, and then use the following key operation.

[MEM] mode,

Set the program No. and press the [ENTER] key. (This operation can be omitted when the program number is the same as that on the display.)

Press the [CYCL RUN] or [SINGL RUN] key, and press the [START] key.

In the schedule program, the program is returned to the head of the execution program by pressing the [RESET] key. Therefore, the program execution pointer is unchanged.

o Restart from the different position from the stop position or position on the way of program

In the memory run mode,

Press the [CYCL RUN] or [SINGL RUN] key,

Set the [PROGRAM NO.], then press the [ENTER] key. (When the program No. is the same as that on the display, it can be omitted.)

Set the start point by the [F0] key.

Set the [PROGRAM NO.] to start, the [TOOL ORDER] to be used for the job, the order of [START POSITIONs] in the job, and [WORK NO.] to start.

The machine moves to the machining start position set in the above operation, then executes the unexecuted portion of the displayed program.

* When the machining order is set in the individual job, the machining position moves in the ascending order by odd number of the tool, and the machining position moves in the descending order by even number.

* When the schedule program is executed, O , program No. and program execution pointer is set in the program No.

9-4. Program Edit in Memory Run

In memory run, program can be edited only for the following items:

Machining data
Machining order
Tool pattern
Cutting tool/condition

Set the EDIT mode while pressing the MEM key, confirm the lighting of the pilot lamp located upper right of the memory run mode key, and edit programs. (For program edit, refer to Section V-8, "Program Edit".)

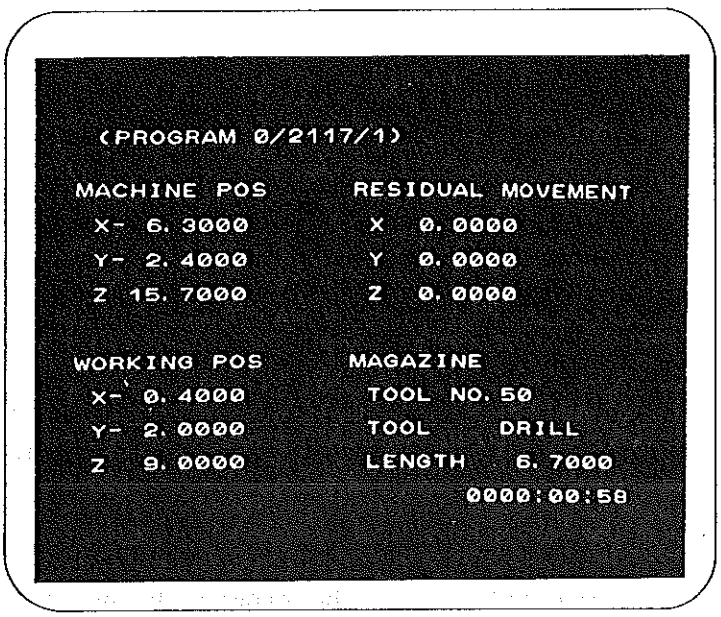
Since the operation cannot be performed after the tool assignment, edit programs after the completion of memory run.

* When the RESET key is pressed, operations stop immediately and the program being executed is returned to the head, and all the programs in the course of edit are reset.

(If program editing has been carried out using the program stored in memory, the block called and edited are reset.)

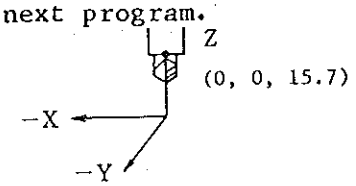
10. Position Display

Portion of axes, using tools, and machining time in the operation can be displayed.



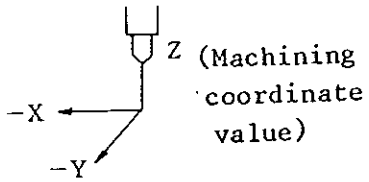
Program number display
With the position display in the memory run, the program No. in operation is displayed.
The previous program No. is displayed until the operation ends and the machine executes the next program.

Machining coordinate value
The X and Y axes are close to the machining zero position, and the Z axis is positioned from the table top face to the end face of the spindle.



ex) The dimension of the Z axis zero position for parameter is assumed to be 15.7.

Working coordinate position
Value from working zero position set arbitrarily. The Z axis indicates the top position of the tool.



Residual amount of movement
The residual amount of movement (unexecuted movement value) in the memory run and key-in run is displayed.

Magazine

Tool No.: Of the tool numbers registered in the tool list, the registration number of the tool currently set in the spindle is displayed.

Tool: The above tool name is displayed.

Tool length: Tool length of the above tool is displayed.

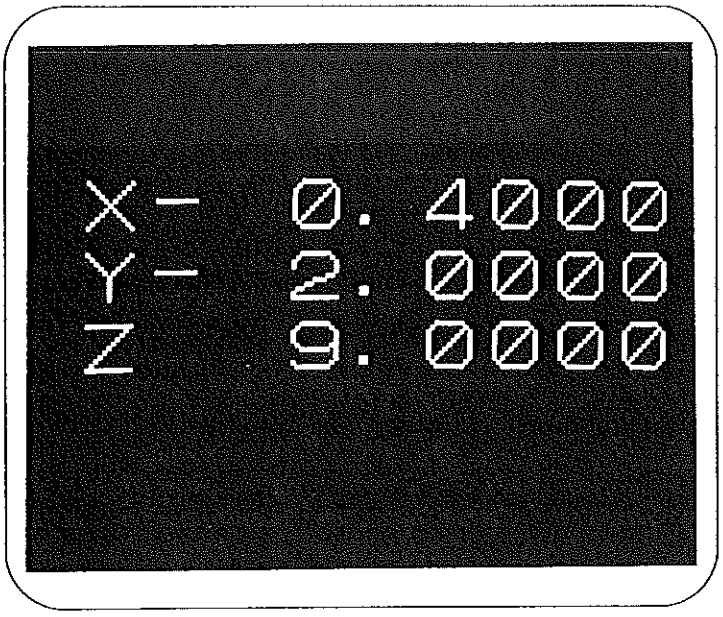
Automatic operation time

The total time of memory run is displayed.

The time is initialized by the operation time of parameter 1.

Machining coordinate values of the X, Y and Z axes in the position display screen can be magnified for display.

Press either of the ⇧CURSOR or ⇩CURSOR key.

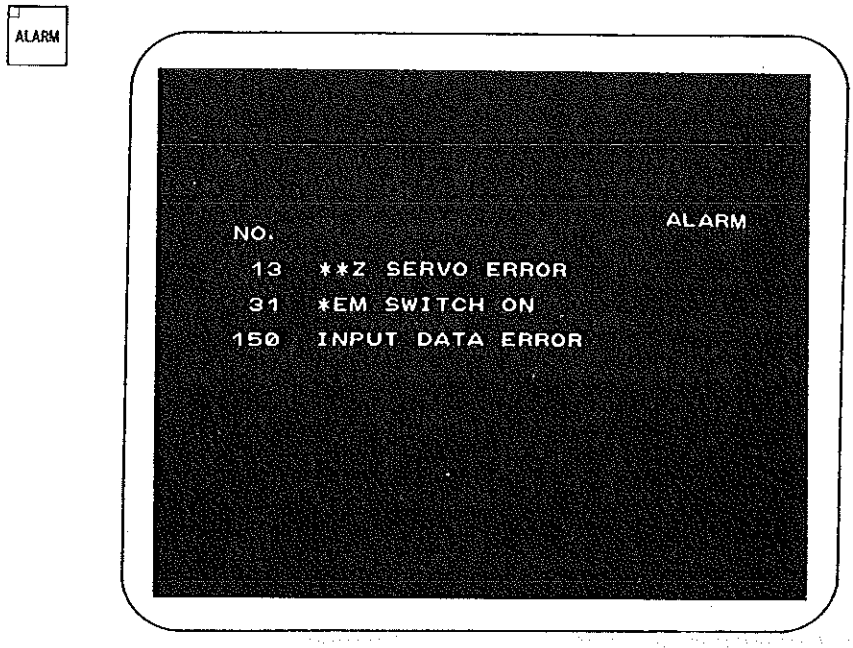


Working coordinate values are magnified.

By pressing the ⇧CURSOR or ⇩CURSOR key, the display is returned to the original screen.

11. Alarm display

When an alarm occurs, all causes of the error are displayed.



Error numbers and messages are displayed.

When several errors simultaneously occur, error numbers and messages are displayed up to 9 respectively in order of the major error. (The priority can be judged through the number of asterisks on the error.)

In other displays, however, only the message of the most important error flickers when several errors simultaneously occur.

Restore the error according to the displayed error code and the error restoring method by referring to the error code table (VIII. Error code) in this manual.

VI Optional accessories and relevant components

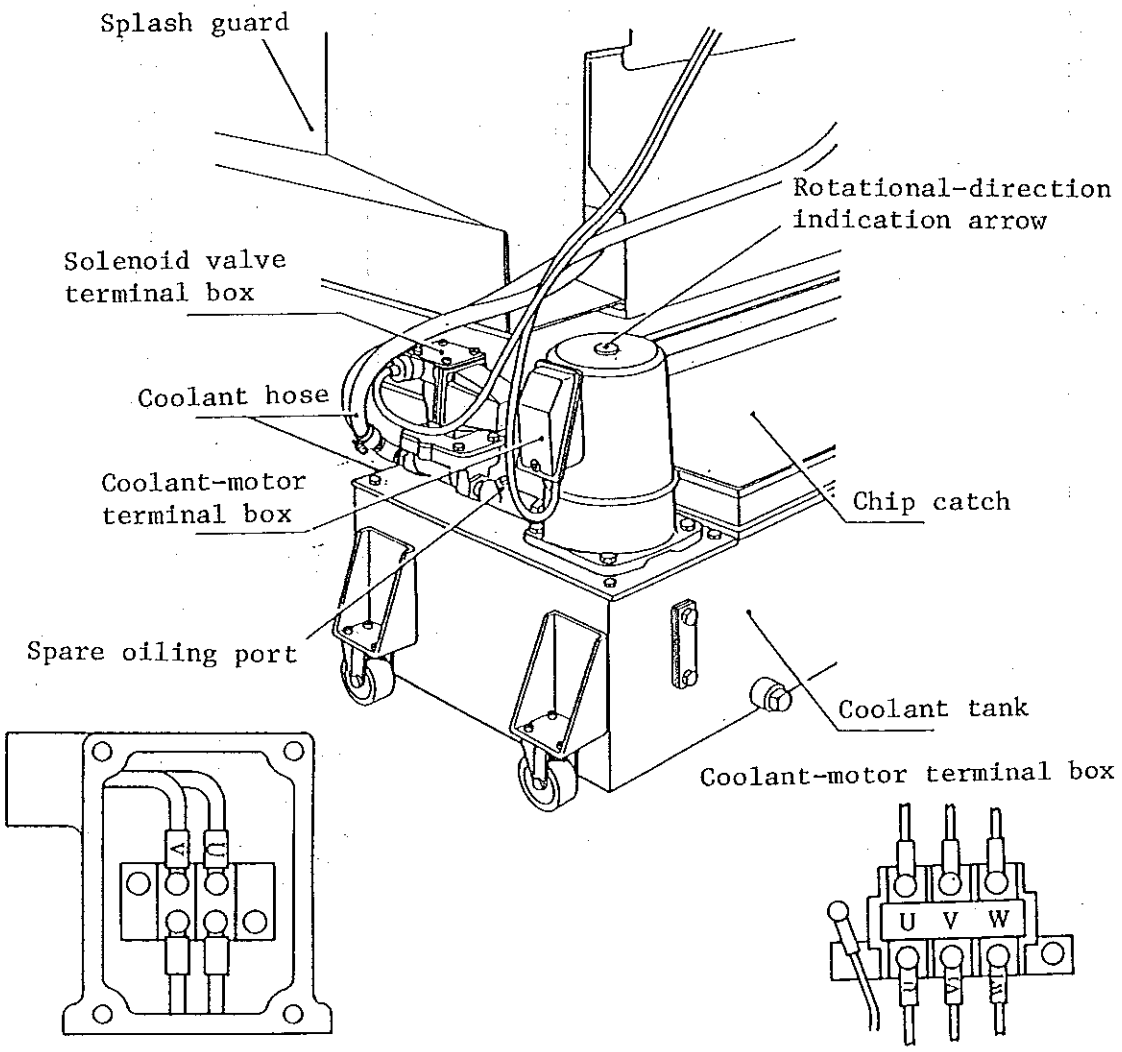
Optional accessories and relevant components

1. Coolant equipment and solenoid valve

Turn on/off the coolant equipment with the coolant motor switch in the control case.

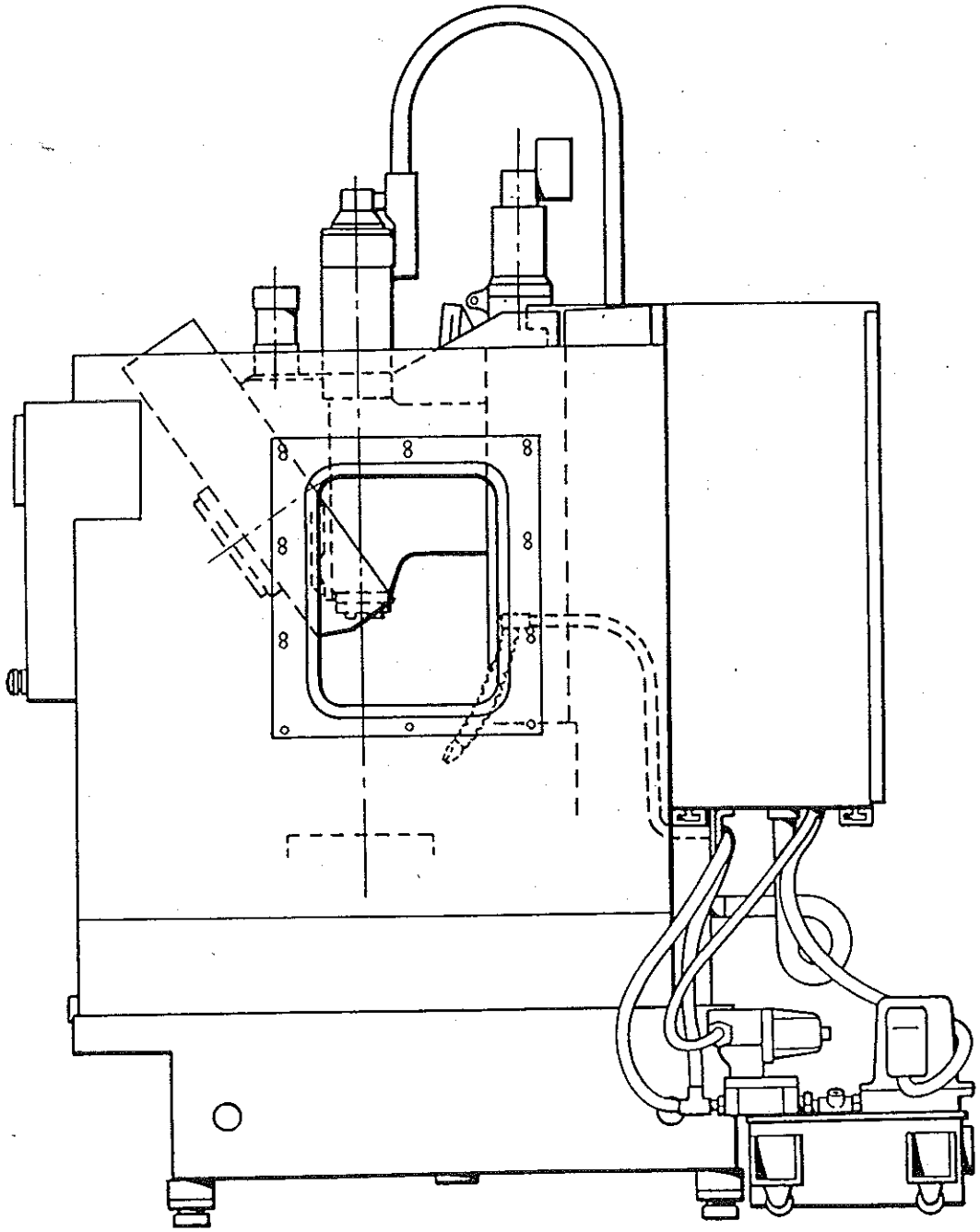
1-1. Installation

Install the coolant tank at the back of the base so that it will be aligned with the chip catch groove.
Connect the control-case parts with the coolant motor and solenoid valve of the coolant equipment according to the description of the external output terminal block as shown below. After connecting them, turn on the power supply and the coolant-motor switch in the control case to confirm that the coolant motor rotates in the direction of the indication arrow.



1-2. Coolant nozzles and their connection

Connect the coolant hoses to the right and left coolant nozzles installed on the side cover. Adjust the nozzle position so that coolant can be constantly supplied to the machining point of the work.

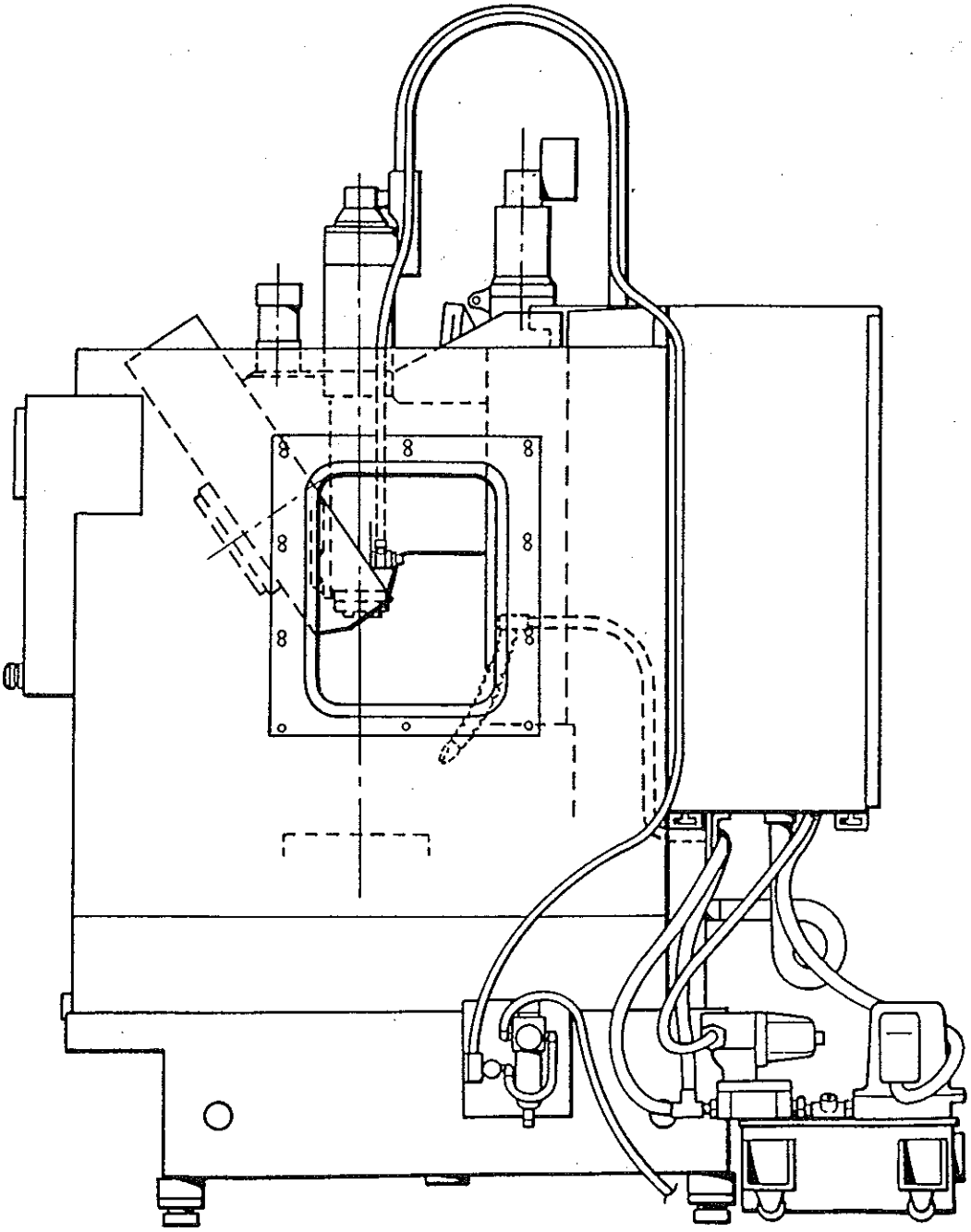


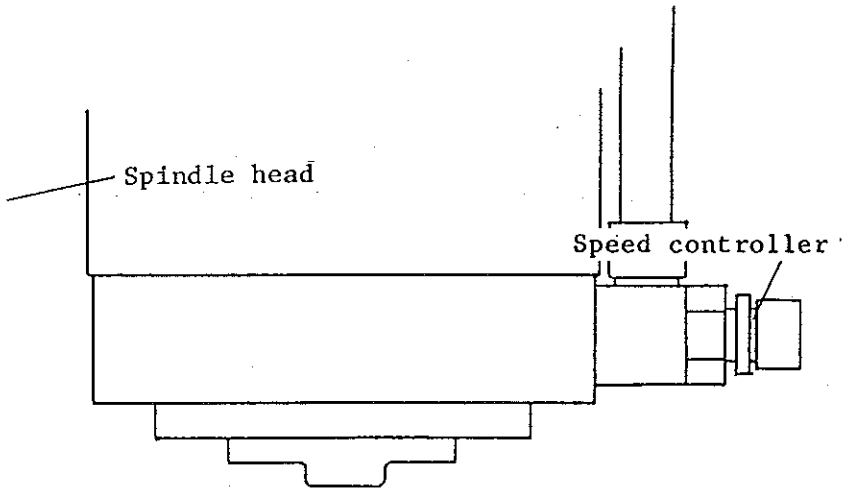
2. Installation of spindle-taper airblast equipment

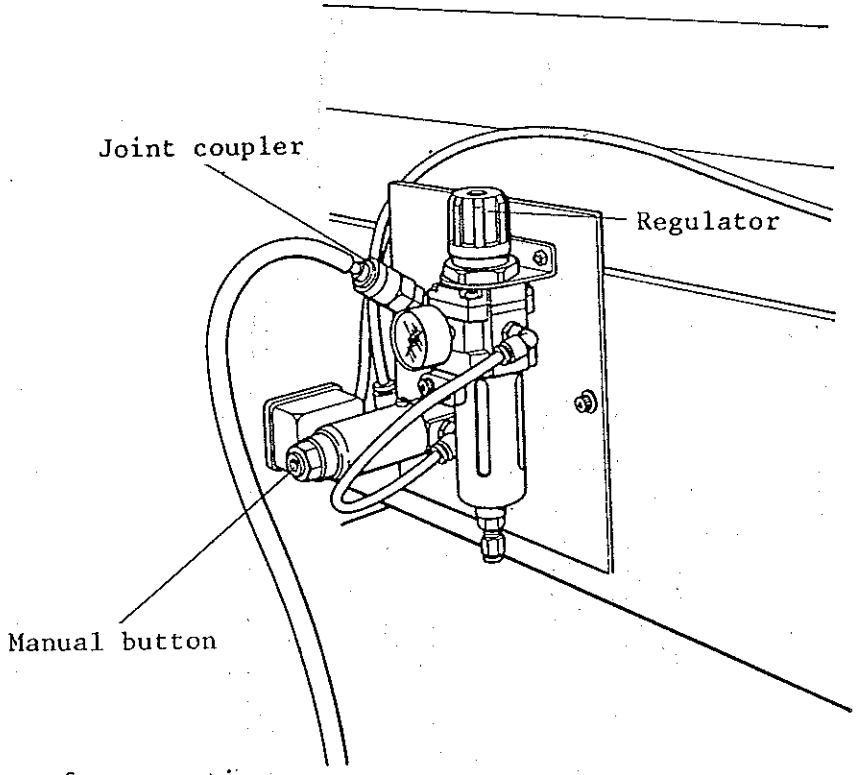
This equipment eliminates foreign matters collected on the taper portion to protect it by jetting air with the pressure of 0.4 to 0.5 Mpa (4 to 5 kg/cm^2) from the inside of the spindle taper portion during ATC operation.

Secure the air tube at the right of the spindle head using the saddle.

Make wiring according to the external I/O terminal block.





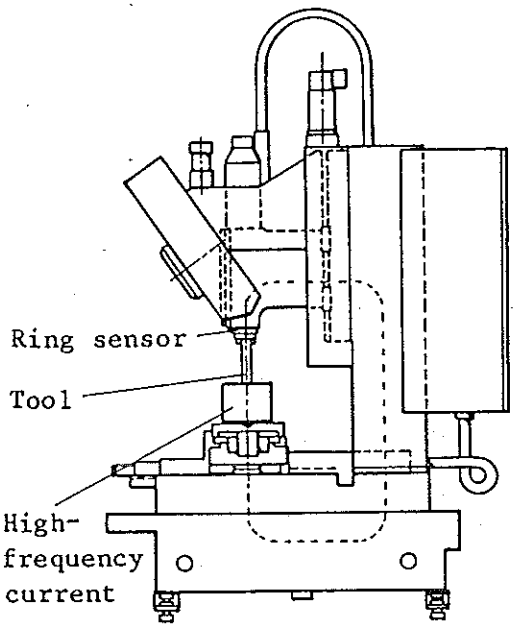


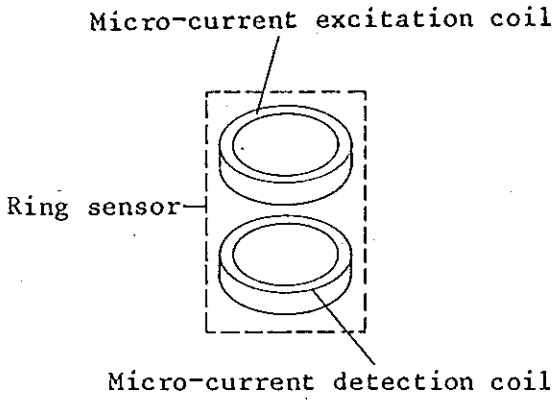
2-1 Cautions for operation

1. Use the air source with the capacity of 500 L/min (at the atmospheric pressure) or more.
2. Use clean dry air.
3. Remove the filter joint coupler and drain the air pipe before starting the operation.
4. Adjust the regulator knob so that the air pressure will be approx. 0.5 MPa (5 kg/cm^2) (green mark) and fully release the speed controller.
5. It is possible to jet air at any position on the Z-axis by pressing the manual button.

3. Tool breakdown detector

The detector uses the ring sensor system.





This system detects whether or not the conductive tool contacts a conductive work as shown in the above figure. Therefore, the system cannot be applied to non-conductive works including resin works.

This equipment checks whether or not the tool contacts the work (conductivity check) at the cutting-feed lower limit of the tool in every machining process excluding the face milling, and stops the machine to display the tool breakdown error unless the tool contacts the work.

Fix the cable at the left of the spindle head with the saddle. Make wiring according to the description of external I/O terminal block.

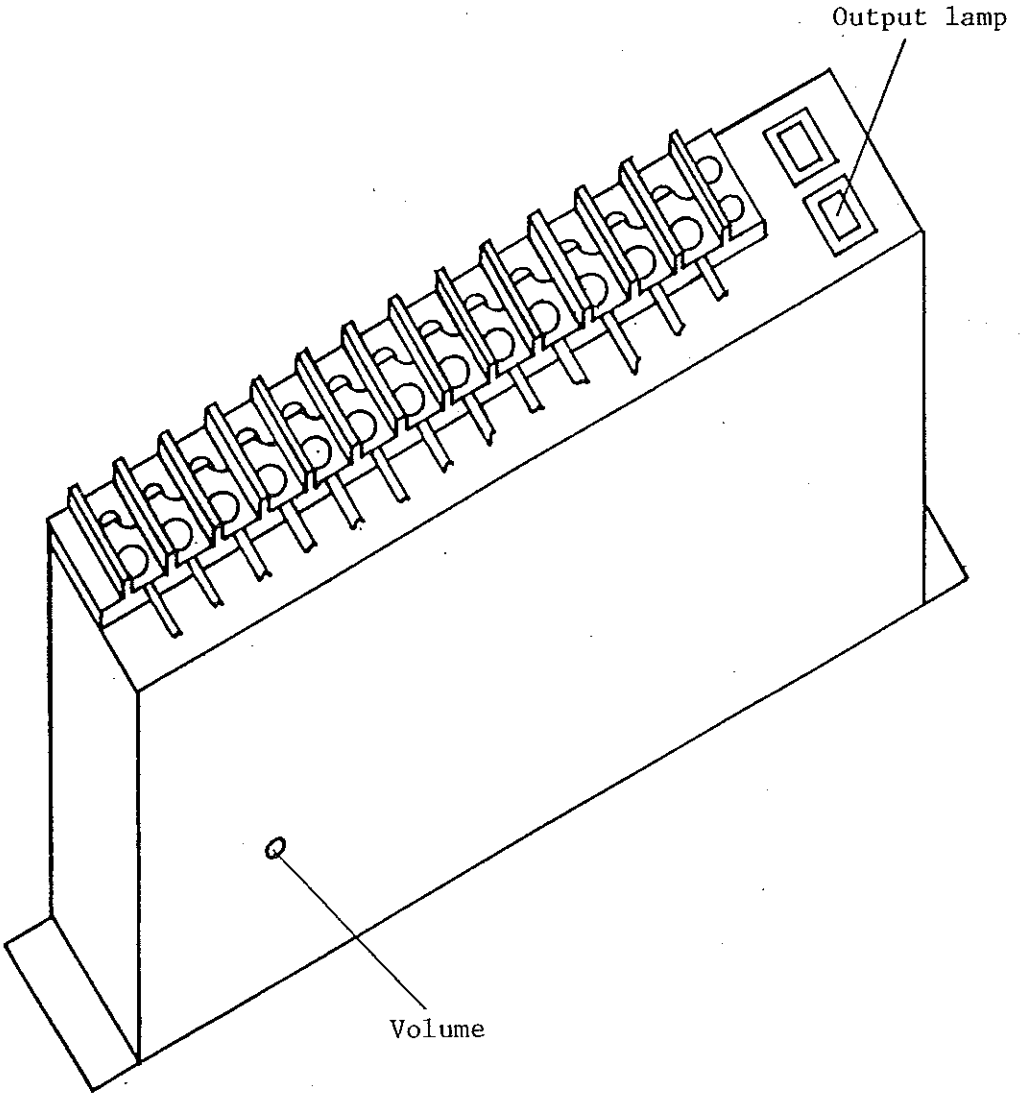
o How to adjust sensitivity (The method slightly depends on coolant and tool used.)

(1) Run the spindle at the speed of 1,000 rpm.

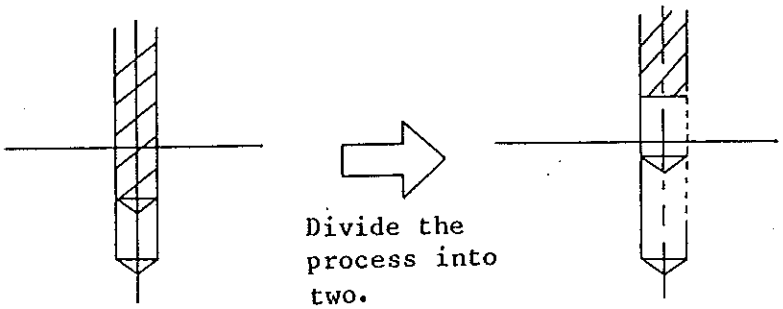
(2) Turn the dial clockwise until the output lamp goes on.

(3) After the lamp goes on, turn the dial counterclockwise until the lamp goes out.

(4) Further turn the dial counterclockwise by 360°.



Note: If the tool is broken (when a short drill is used) as shown in the figure below, the tool breakdown cannot be detected. Therefore, it is recommended to divide the drilling process into two processes -- shallow drilling and deep drilling, and prepare the program to detect the tool breakdown in the shallow drilling process.



If the tool is broken during machining a work, the tool breakdown cannot be detected because of conductivity.

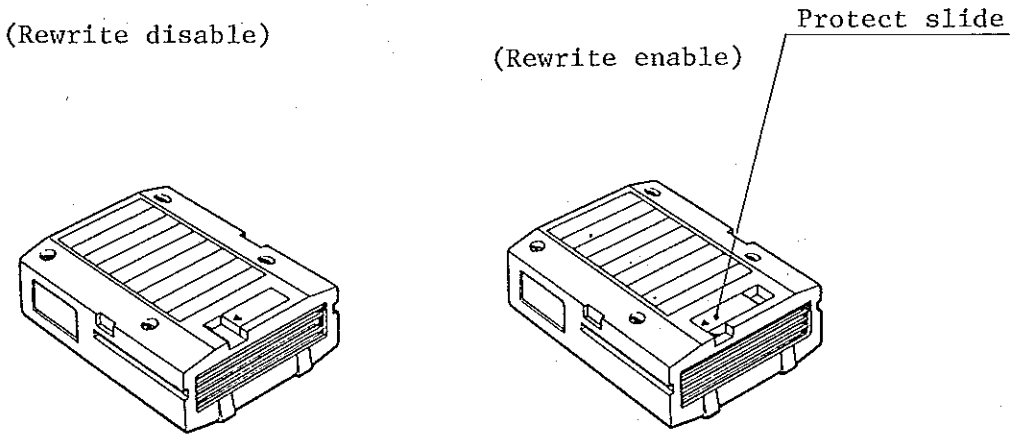
For shallow drilling, if the tool is broken, the tool breakdown can be detected because of no conductivity.

4. Bubble cassette and Bubble memory unit

How to handle Bubble cassette

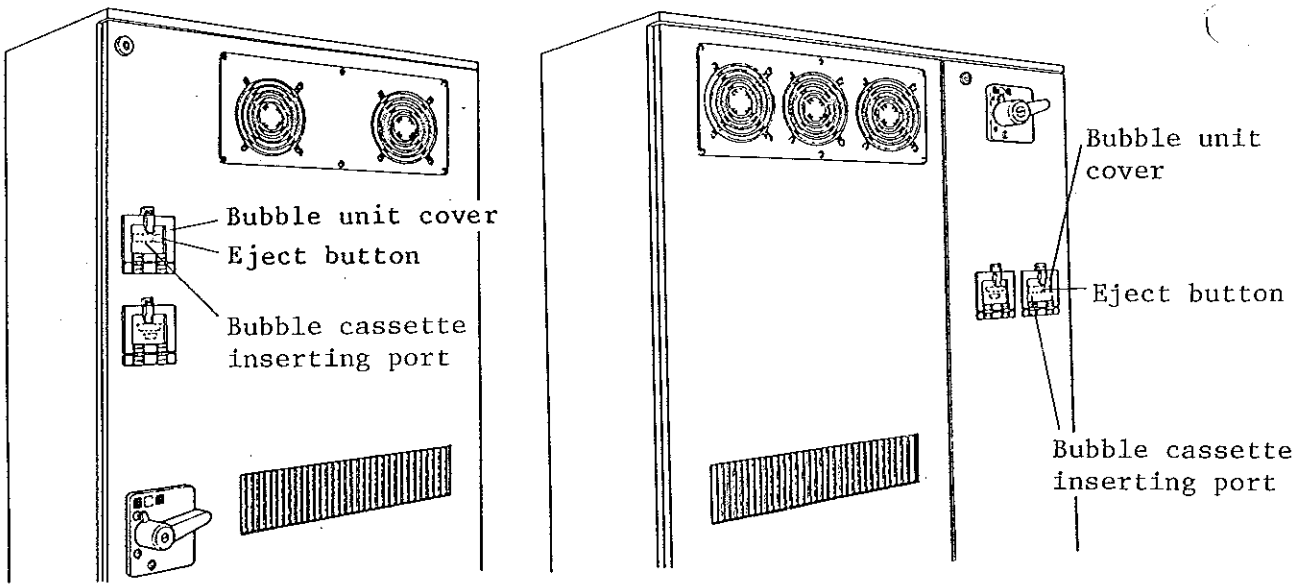
For the output (write) the edited program or data to the bubble cassette, change the write protect slide position as shown in the figure at bottom right. Set the bubble cassette to the holder, securely close the cover, and confirm that the write-protect indicator lamp (green) goes out.

Maker: FUJITSU
Type: FBM-C128GA



Bubble memory unit

To take out the bubble cassette, open the cover and press the eject button as shown below.



4-1 How to install bubble memory unit

(1) Remove the iron plate cover at the bubble cassette entrance on the outside of the door.

(2) Install the movable bubble-memory unit cover.

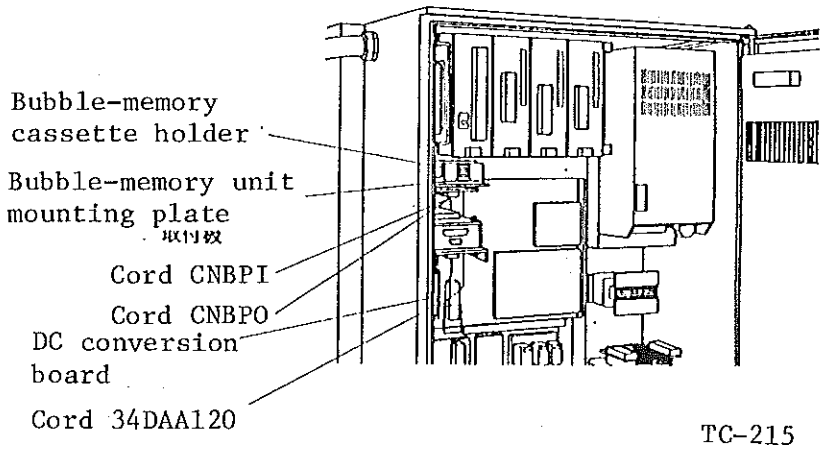
(3) Screw the bubble-memory unit mounting plate at the inside of the door.

(4) Screw the DC conversion board.

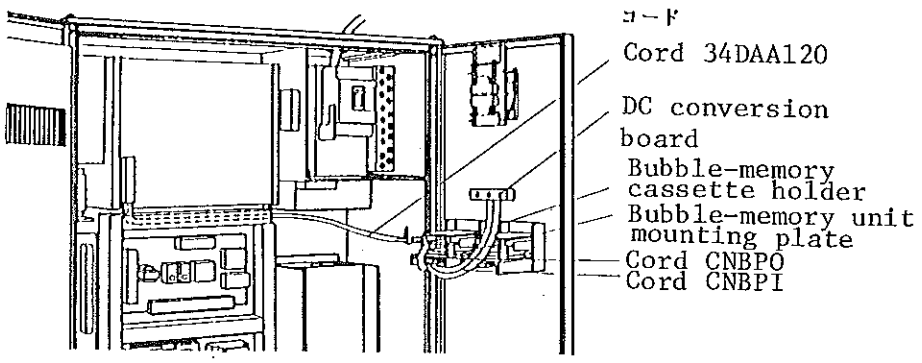
(5) Connect the cord 34DAA120 according to the figure. (When connecting the cord, match the mark "▽" on the connector.)

(6) Connect the cords CNBPOK and CNBPIK according to the figure. (Check the number of pins on each connector.)

(7) Connect the cord CNBPIK to the terminal block. Match the other cord with the mark tube.



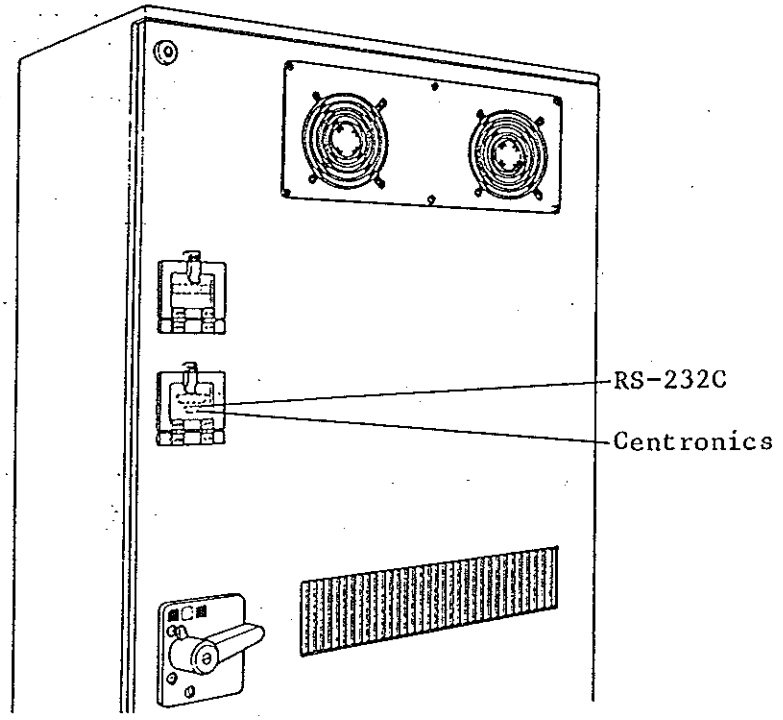
TC-215



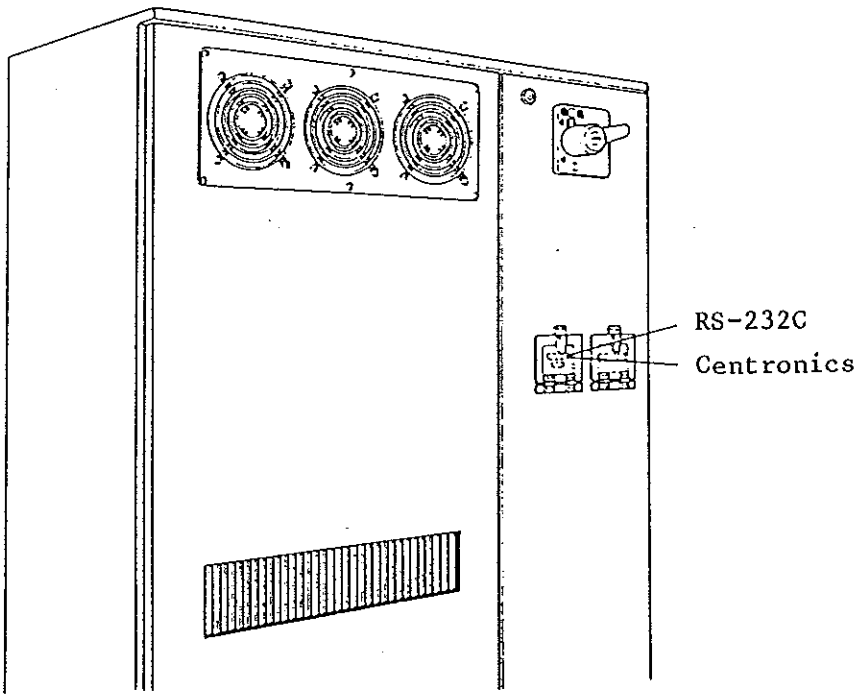
TC-255

5. EXTERNAL COMMUNICATION UNIT

This machine can output various kinds of data by a Centronics interface (printer), and can input and output various kinds of data by RS-232C interfaces (computer, PTP (paper tape puncher)/PTR (paper tape reader)).



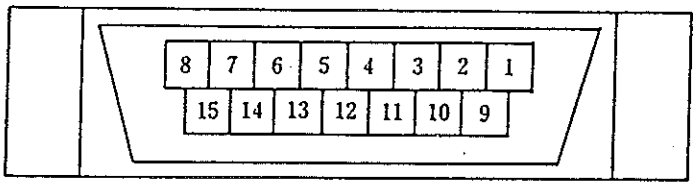
TC-215



TC-225

5-1. Centronics Interface

5-1-1. Connection Method of Centronics Interface



Connector: RDAB-15S-LN (Hirose)
(Connector on cable side: HDAB-15P)
(Connector on cable side: Cable case HDA-CTH)

Pin Assignment

Pin No.	Signal Name	Signal flowing Direction	Remarks
1	STROB	→	Strobe
2	DATA1	→	Transmission data
3	DATA2	→	Transmission data
4	DATA3	→	Transmission data
5	DATA4	→	Transmission data
6	DATA5	→	Transmission data
7	DATA6	→	Transmission data
8	DATA7	→	Transmission data
9	DATA8	→	Transmission data
10	ACK	←	Acknowledge
11	BUSY	←	Busy
12	FAULT	←	Fault
13	N.C.		No connection
14	SG		Signal ground
15	N.C.		No connection

Each arrow mark directed to the right-hand side (←) indicates the flowing direction of an outgoing signal from this machine, and each arrow mark directed to the left-hand side (→) indicates the flowing direction of an incoming signal into this machine.

5-1-2. Printer Specification

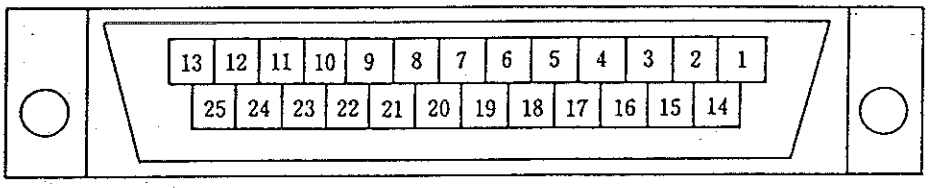
Remember to use a printer whose interface conforms to the Centronics Standard and whose printing line capacity is 60 characters/line or more.

5-2. RS-232C Interface

5-2-1. Communication System of RS-232C (Serial)

- o Full duplex asynchronous system
- o Control line system
- o Code 8 (bit code)
- o Transfer rate: 300 to 4800 bps (to be set by Parameter 4)

5-2-2. Connection Method of RS-232C Interface



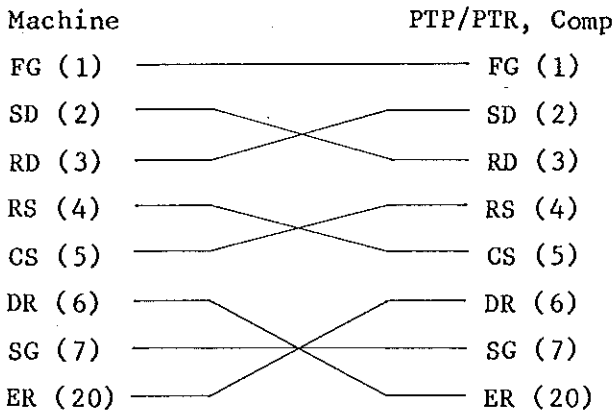
Connector: RDBB-25S-LN (Hirose)
(Cable side: Connector HDBB-25P)
(Cable side: Plug case HDB-CTH)

Pin Assignment

Pin No.	Signal Name	Signal flowing Direction	Remarks
1	FG		Frame ground
2	SD	→	Send data
3	RD	←	Receive data
4	RS	→	Request to send
5	CS	←	Ready for sending
6	DR	←	Data set ready
7	SG		Signal ground
20	ER	→	Data terminal ready

Each arrow mark directed to the right-hand side (←) indicates the flowing direction of an outgoing signal to this machine, and each arrow mark directed to the left-hand side (→) indicates the flowing direction of an incoming signal into this machine.

Signal Cable Connection Method



(Note) PTP ..... Paper tape puncher
PTR ..... Paper tape reader

Cautions for using the serial interface

This unit is equipped with the serial interface based on the RS-232C. This interface can be connected with and exchange information with only the units including the paper tape reader/punch and the external program editing equipment which are recommended or designed by us.

To exchange information using the serial interface, both units should be matched in the protocols for:

* Signal speed
* Signals constituting a character
* Signal to temporarily stop or restart receiving characters
* Detection of signal error
* Determination of a piece of information
* Determination of start/end of information exchange
* Limitation of time to wait for response from the associate unit

For these protocols, see "Section V 7-2-4 Parameter 4", "Section VI 5-2 RS-232C interface", and "Section VIII Error code" in the instruction manual for TC-215 and TC-225.
Note that, because the RS-232C is the specifications only for the mechanical and electrical characteristics of connectors, no information can be exchanged through the same RS-232C interfaces without the above protocol.

5-2-3. Protocol for Connection with Computer

This machine always serves as a host, and communication starts when REQBLK was sent from this machine to the computer.

1 The machine sends ⟨ REQBLK ⟩ to the computer.

2 In case data transfer from the machine to the computer or from the computer to the machine is possible, the computer returns ⟨ACK⟩ in replay.

3 o In case of data transfer from machine to computer
As soon as the computer gets ready for data reception, the computer urges the machine to send data by returning ⟨SYN⟩.

o In case of data transfer from computer to machine
The computer returns ⟨ REQBLK⟩ in reply.

(Note) The ⟨length⟩ of ⟨REQBLK⟩ from the machine is meaningless, and the ⟨command⟩ of ⟨REQBLK⟩ from the computer is also meaningless.

The machine returns ⟨ACK⟩ in reply, and when the machine gets ready for data reception, it urges the computer to send data by returning ⟨SYN⟩.

4 After arrival of ⟨SYN⟩, the data output side sends ⟨DATABLK⟩.

5 The data receiving side receives ⟨DATABLK⟩, and if everything is in order, it returns ⟨ACK⟩.

6 After sending all the data, the data sending side outputs EOD.

7 The data receiving side returns ACK.

This ACK return from the data receiving side completes the data transfer.

(Note 1) If NAK was returned instead of ACK, the data transfer is further repeated seven times, and if NAK was returned seven times, the operation comes to an abnormal end.

(Note 2) The control character of each block terminates the CR code (ODH).

(Note 3) If NAK was returned instead of EOD in the above 6, it causes an error.

5-2-4. Protocol for Data Transfer with Computer Data Transfer Format

⟨REQBLK⟩ ==> ⟨command⟩ ⟨file name⟩ ⟨,⟩ ⟨length⟩

⟨DATABLK⟩ ==> ⟨:⟩ ⟨block number⟩ ⟨data⟩ ⟨;⟩ ⟨check sum⟩

⟨command⟩ ==> L (Load) ⟨SP⟩ .... Data transfer to machine

S (Save) ⟨SP⟩ .... Data transfer from machine

⟨file name⟩ ==> Character string of 8 bytes

(PROGRAM NO) O****...- Machining data

(TOOL LIST) L.....- Tool menu

(PATTERN) P.....- Tool pattern

(CONDITION) C.....- Cutting condition

(HOLE) H.....- Hole sizes before tapping

(PARAMETER) R.....- Parameter

(DIRECTORY) _D.....- Directory

(Note 1) The symbol "." denotes an SP.

(Note 2) The symbol "*" denotes a numeric character.

⟨length⟩ ==> A five-digit decimal number to be right justified, and the rest to be filled with '0' or SP. (ASCII code)

(Note) The unit of length is 128 bytes.
(1 length = 128 bytes)

⟨block number⟩ ==> A four-digit decimal number. (ASCII code)

(Note) Not to be checked in this machine.

⟨data⟩ ==> To be transferred per 128 bytes.

(Note) When sending the directory, O**** length is to be repeated, and when it falls short of 128 bytes, the rest is to be filled with SP or '0'. (1 length: 3 bytes, Unit of directory sending: 128 bytes)

⟨check sum⟩ ==> Three-digit decimal expression of one's complement of the least significant 1 byte of the sum of codes between ":" and ",". (ASCII code)

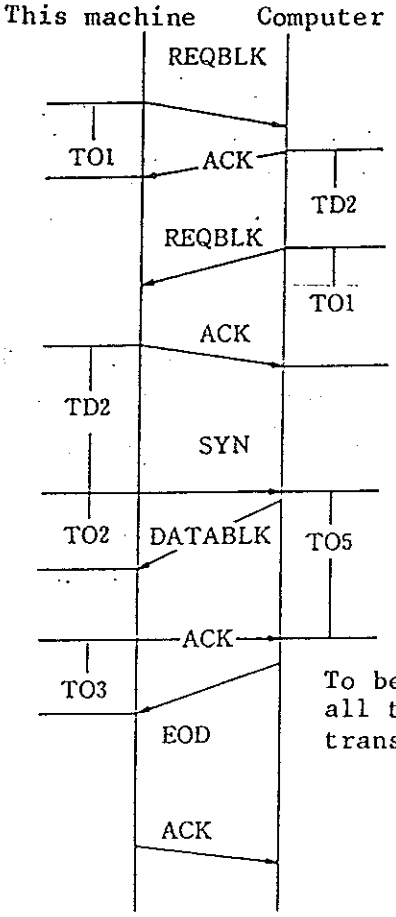
Control Character Code

ACK --- 06H
NAK --- 15H
SYN --- 16H
EOD --- 03H
CK --- 0DH
WAIT --- 14H
SP --- 20H

(Note) If data requested in data transfer from the computer to the machine does not exist, the REQBLK length is returned as 0, and when SYN was received thereafter, EOD shall be returned.

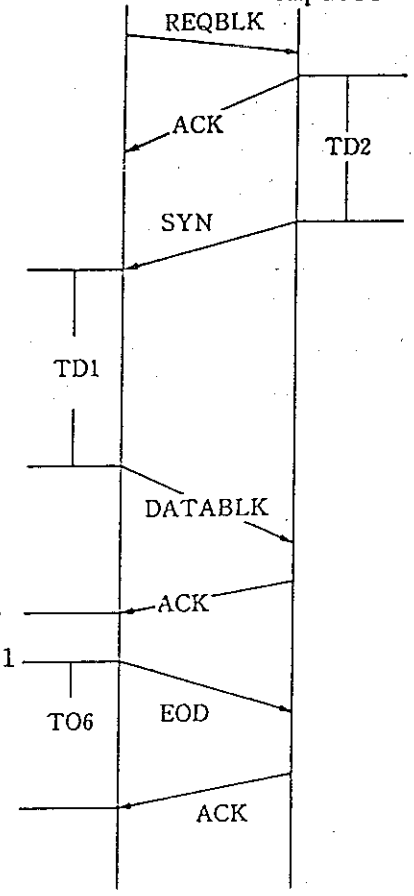
5-2-5. Sequence of Data Transfer between This Machine and Computer

o Data Transfer to Machine

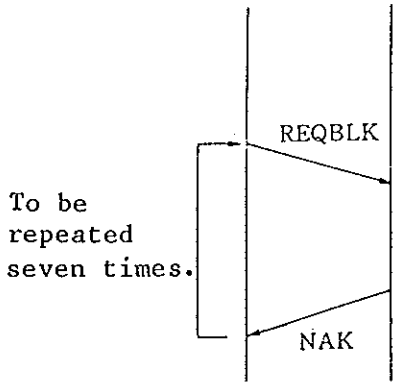


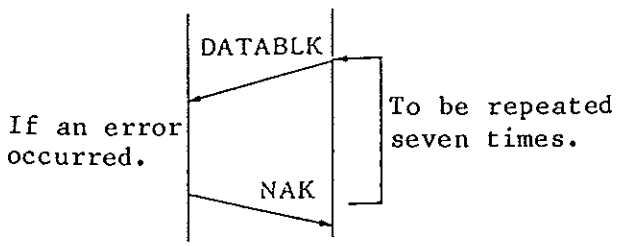
To be repeated until all the data is transferred.

o Data Transfer from Machine

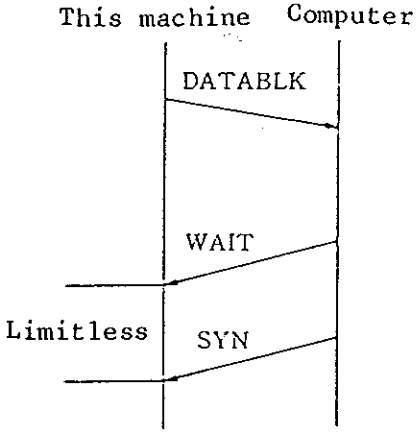


o In Case of Transfer Error





o Data Block Sending



(Note) WAIT is returned only in data transfer, and SYN is returned in every case.

5-2-6. Timing in Sequence of Data Transfer between This Machine and Computer

1. Time Out

TO1	REQBLK send	==> ACK receive	60 sec.
TO2	SYN/ACK send	==> DATABLK/EOD receive	30 sec.
TO3	SYN/ACK send	==> DATABLK/EOD receive	30 sec.
TO5	DATABLK send	==> ACK receive	60 sec.
TO6	EOD send	==> ACK receive	60 sec.

2. Delay Time

TD1	SYN/ACK receive	==> DATABLK/EOD	0.5 sec.
TD2	ACK send	==> SYN send	0.5 sec.

5-2-7. Operation of PTP and PTR

The operation method is basically the same as in external program input/output operation of TC. Remember to put the external unit (PTP, PTR, printer, or computer) into the ready state before starting the operation of this machine.

1. When the START key was pushed, the DR (data set ready) input of this machine must be kept active.

2. In case the external unit is PTP, the CS (ready for sending) input on the machine side must be kept active. If the PTP side made the CS (ready for sending) input inactive on account of 'buffer full' or for some other reason, make it active once again within 10 seconds.

3. In case the external unit is PTR, send data when the machine made the RS (request to send) output active, and stop the data transmission when the machine made the RS output inactive. Unless data transmission starts within 90 seconds after the RS output, the time-out error (external memory error 8) is displayed on the screen.

Remember to use such a PTP or PTR as meets the abovementioned conditions. So far as the PTP or PTR meets these conditions, cross connection of the cable is not necessary. (Closely check the PTP or PTR specification.)

5-2-8. Data Output to PTP

The data list is as follows.

⟨feed⟩ ⟨%⟩ ⟨:⟩ ⟨file name⟩ ⟨data⟩ ⟨LF⟩ ⟨%⟩ ⟨feed⟩
⟨feed⟩ --- NULL data of 100 characters
⟨%⟩ --- 25H
⟨:⟩ --- 3AH
⟨file name⟩ --- See Clause 5 and Clause 6. (There is no directory.)
⟨LF⟩ --- OAH

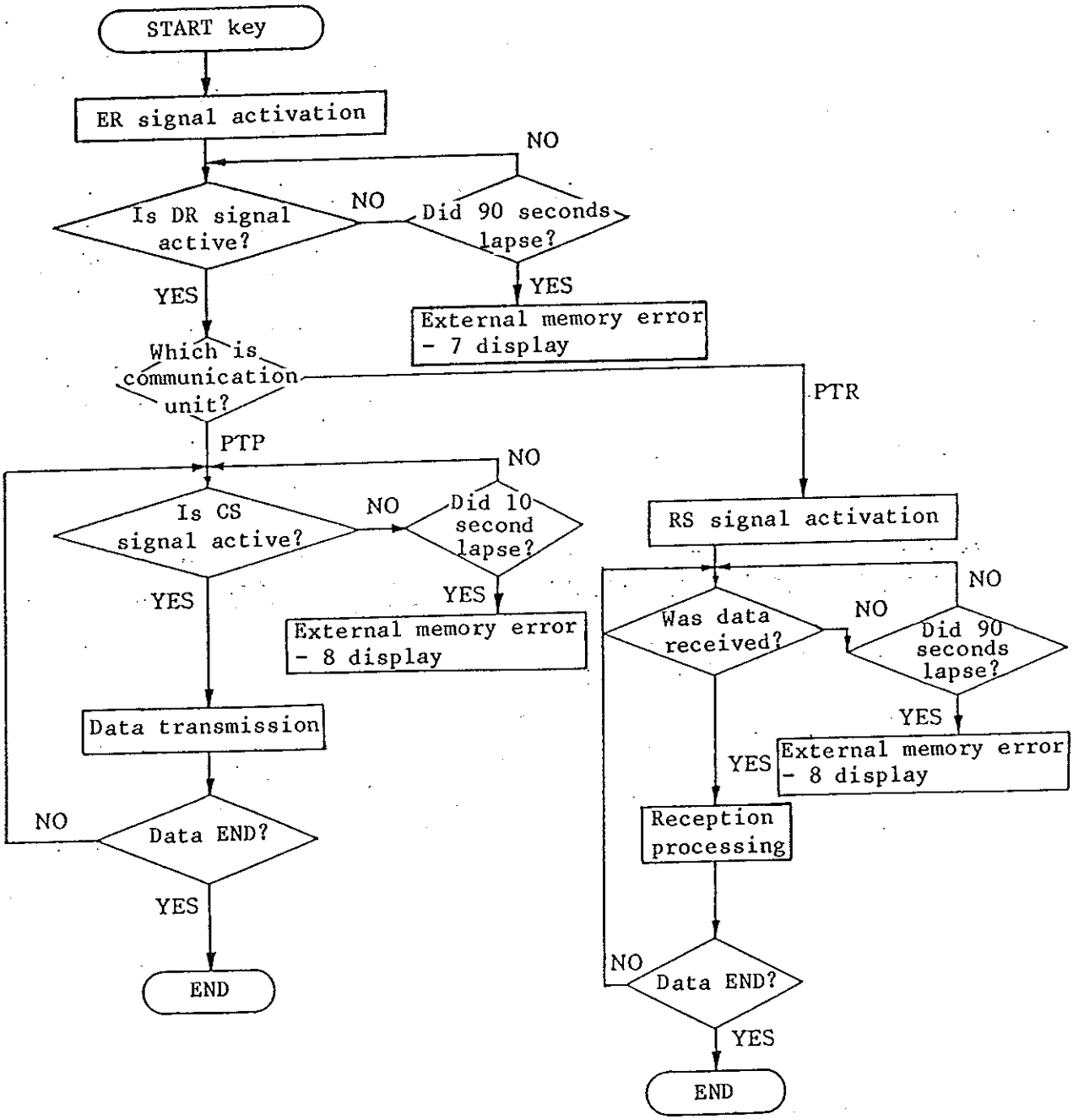
5-2-9. Data Input from PTR

The data list is the same as shown in 5-2-8.

(Note 1) A feed part whose length exceeds 60 cm (23.6") causes an error.

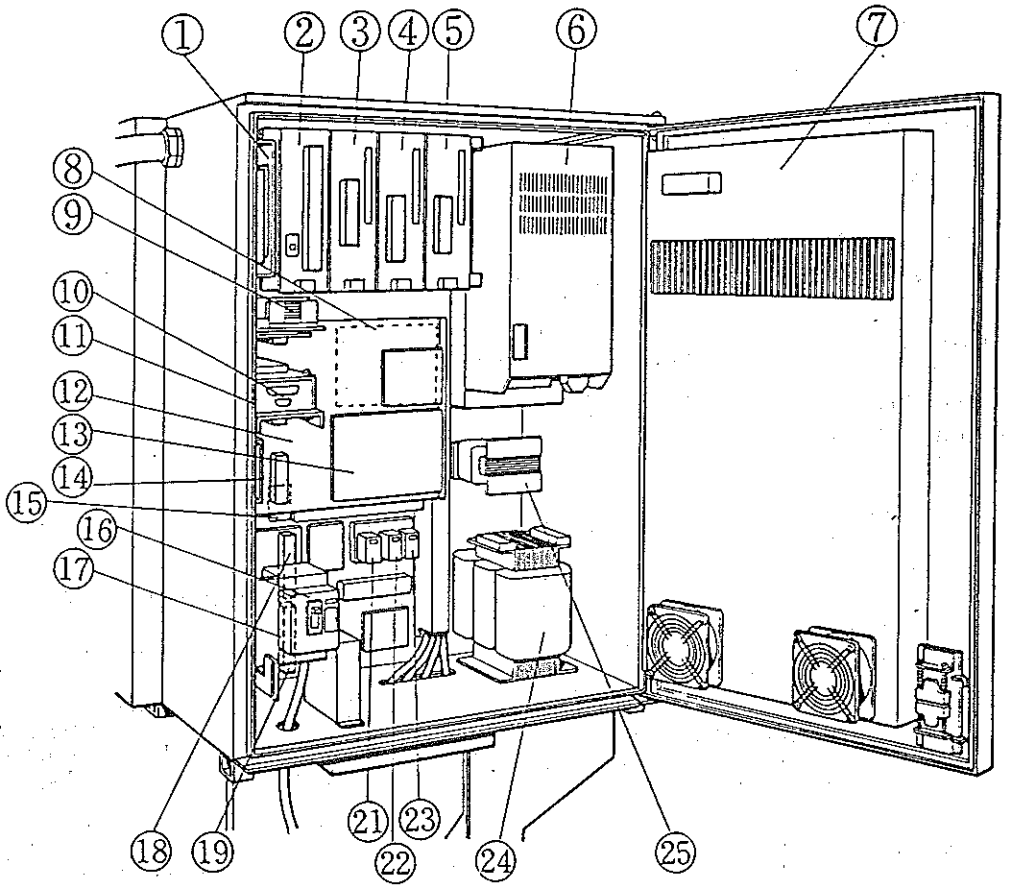
(Note 2) For the paper tape, use an eight-level perforated paper tape of 1 inch in width (conforming to JIS C6243).

5-2-10. Flow of Communication with PTP and PTR



6. Layout of components in control case

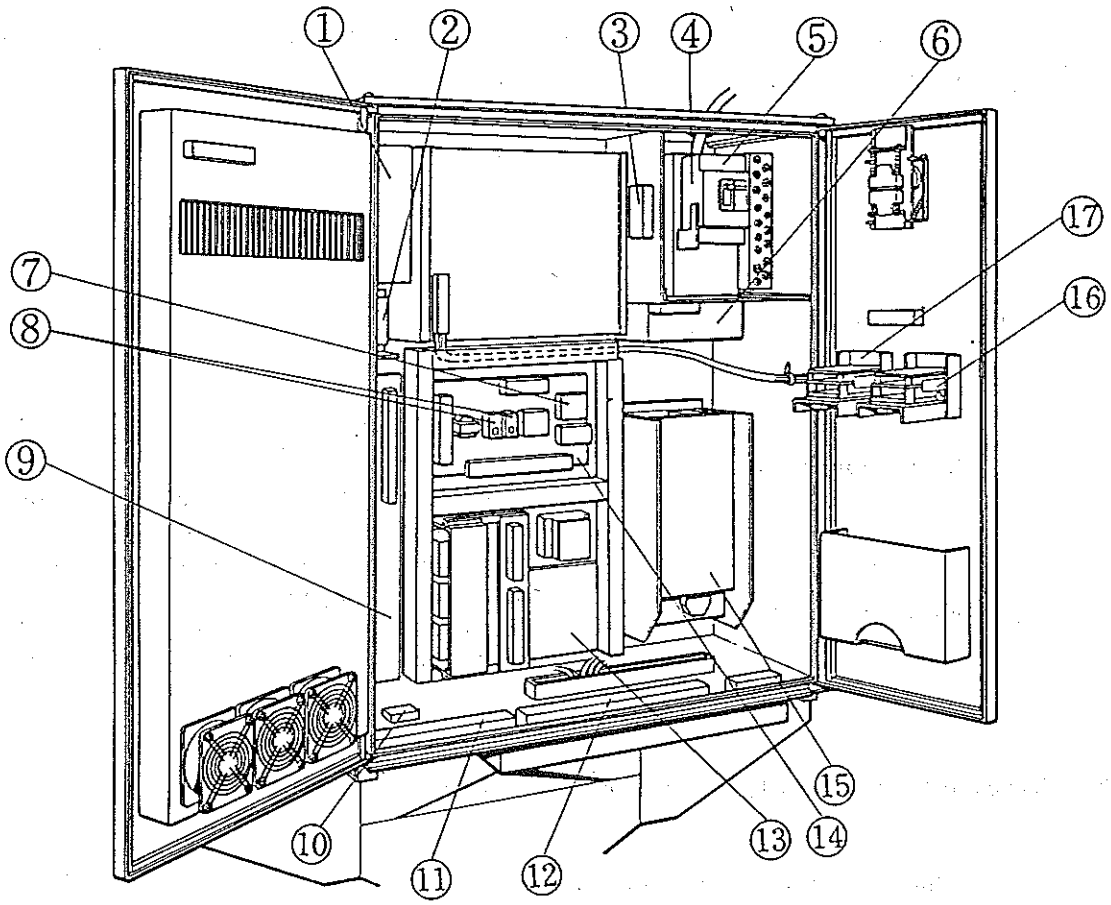
6-1. TC-215



① AVR1 (Switching power supply for P5V, P15V, and N15V)
② Servo power supply
③ Z-axis servo driver
④ Y-axis servo driver
⑤ X-axis servo driver
⑥ Spindle servo driver
⑦ Cleaning unit
⑧ DB unit
⑨ Bubble memory unit (Optional accessory)
⑩ RS232C terminal (Optional accessory)
⑪ DC conversion board
⑫ NC board
⑬ Memory board
⑭ Battery holder
⑮ Control transformer
⑯ No-fuse breaker
⑰ Noise filter
⑱ External output terminal
⑲ External input terminal
㉑ Coolant motor circuit protector
㉒ ATC spindle motor circuit protector
㉓ DB circuit protector
㉔ Power transformer
㉕ Serial reactor

6-2. TC-225

Layout of components in control case

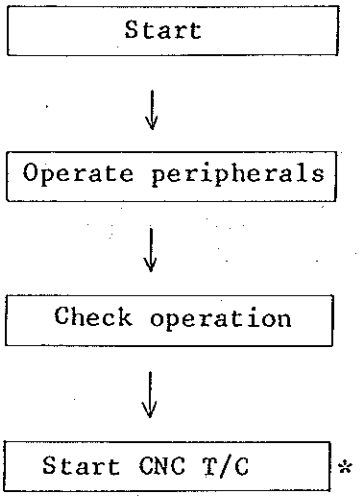


① Relay unit board
② AVR2
③ Dry battery holder
④ No-fuse breaker
⑤ Power connection terminal block
⑥ Noise filter
⑦ Coolant relay
⑧ Circuit protector
⑨ X-, Y-, and Z-axis servo drivers
⑩ Coolant terminal block
⑪ External output terminal
⑫ External input terminal
⑬ ATC control unit
⑭ Power unit
⑮ Spindle servo driver
⑯ RS-232C equipment (Optional accessory)
⑰ Bubble memory unit (Optional accessory)

7. CNC T/C Peripherals Connection

1. How to externally start

To start the CNC T/C after peripherals including a jig installed around the CNC T/C operate:



*1. Connect the N.O. contact with the CNC T/C input board terminal Nos. 1 and 2. (Dry contact to which no external voltage is applied).

*2. External the CNC T/C closes and then opens the contact, the program starts.

2. External feed hold, resetting, and emergency stop of CNC T/C

	Operation during manipulation	Restarting of program	Program operation during machining
Feed hold	*Only the feed stops.	Starting operation is allowed.	Operation continues according to the program.
Resetting	*All operations stop.	Starting operation is allowed.	Operation starts with the first process of the programs.
Emergency stop	All operations stop.	Zero position return is necessary.	Through the machining process is stored, no operation can be continued during tapping.

*1. The ATC tapping does not stop until it is completed or the spindle coolant does not stop.

*2. The ATC tapping does not stop until it is completed.

3. M code

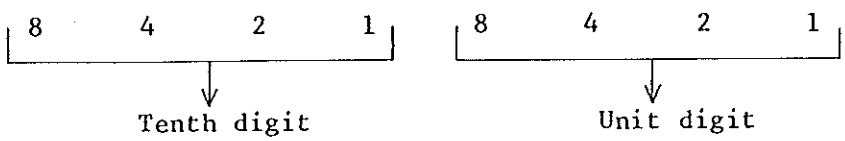
(1) When the M code is input during program operation, the machining order is executed for Individual JOB (INDV JOB).

(2) When two or more works are machined with the program for Individual JOB, the peripherals operate for each work. When controlling a jig with the M code, use INDV JOB-1PC (1 piece)

(3) The M code is output to the output terminal block with the binary-coded decimal number.

(Example)

M code	Output signal name							
	M28	M24	M22	M21	M18	M14	M12	M11
M80	ON	OFF	OFF	OFF	OFF	OFF	OFF	OFF
M63	OFF	ON	ON	OFF	OFF	OFF	ON	ON
M17	OFF	OFF	OFF	ON	OFF	ON	ON	ON
M59	OFF	ON	OFF	ON	ON	OFF	OFF	ON

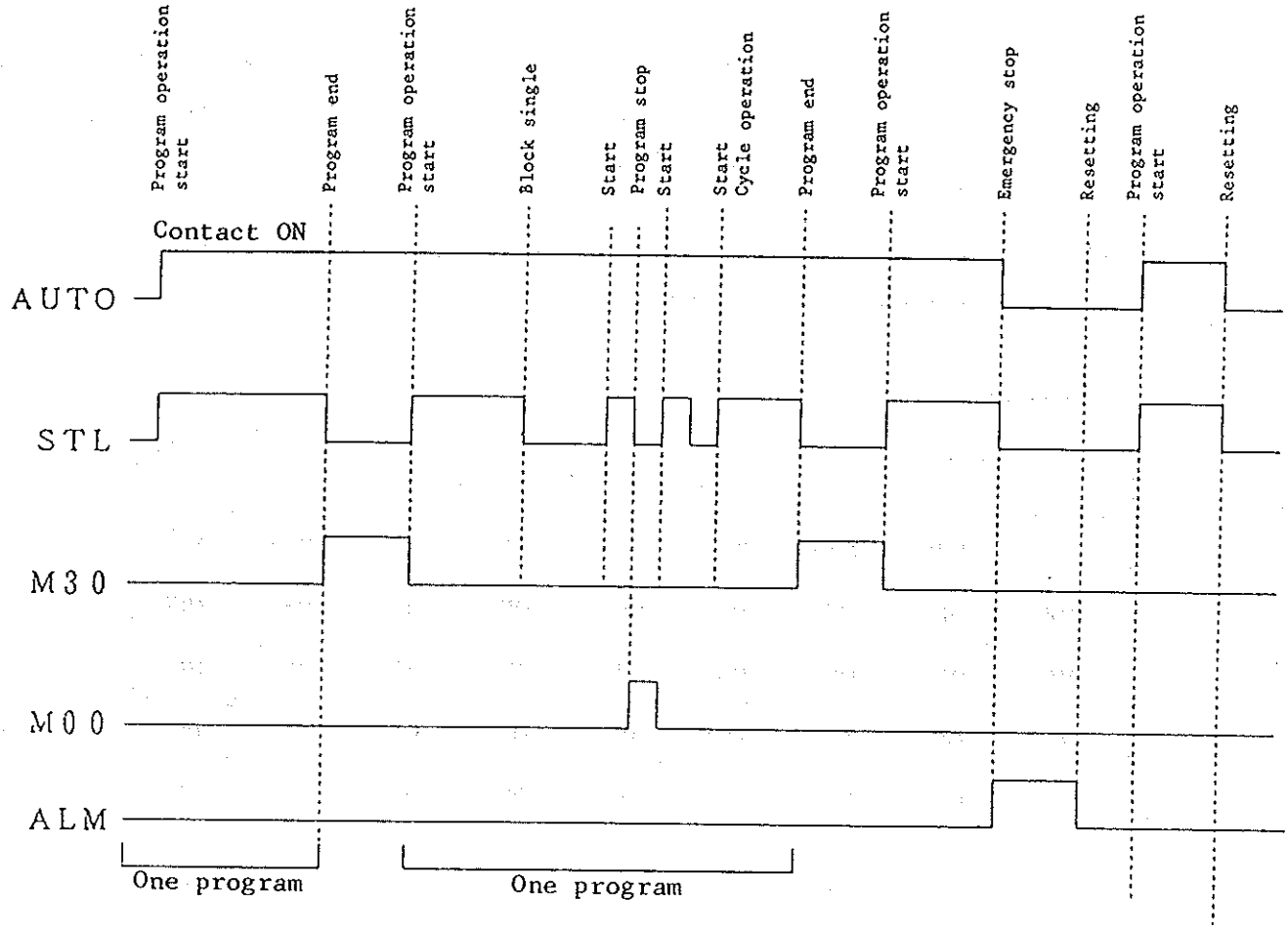


(4) The MF signal is output 20 ms after the M code is output. When using different types of M codes, make interlock after the MF signal is output.

(5) When the operation of peripherals is completed by the M code, keep inputting the external signal completion until 50 sec elapses after the MF signal is turned off.

4. How to check CNC T/C status with peripherals

Because the CNC T/C has the signals including AUTO, YGEO, STL, M30, and ALM; it is possible to check the status by combining these signals.



5. External program selection

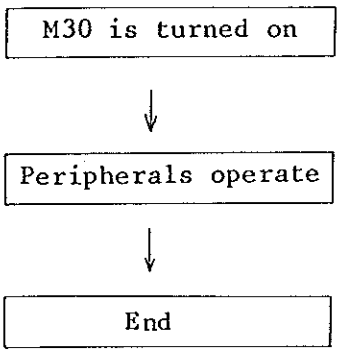
Start the external program selection by inputting the hexadecimal numbers of the program numbers 1 through 15 into PRO1 through PRO8.

(Example)

	PRO1	PRO2	PRO4	PRO8
Program number 0001	ON	OFF	OFF	OFF
Program number 0004	OFF	OFF	ON	OFF
Program number 009	ON	OFF	OFF	ON
Program number 0012	OFF	OFF	ON	ON

6. How to operate peripherals after CNC T/C ends program operation

When the CNC T/C ends the program operation, the M30 signal is output.

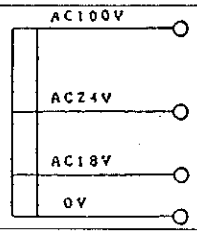


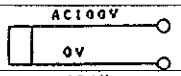
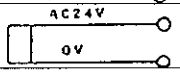
7. Loads to be connected with the machine

Be sure to provide loads with a noise absorption device, surge killer, spark killer, or ZNR. (Spark killer CR-10201: Made by OKAYA DENKI, ZNR ENB221D-14A: Made by FUJI ELECTRIC CO., LTD.)

When using a diode for the DC circuit, use the diode with the dielectric strength of 100 V and the current capacity of 1 A by paying attention to the polarity. The wiring to this equipment shall be isolated from any motor power cable or solenoid wiring.

TC-215

	Voltage	Capacity	Terminal No. of seal	Remarks
	AC 100 V	Up to 1A by totaling the both	159 - 160	
	AC 24 V		159 - 161	
	AC 18 V		159 - Transformer AC 18 V	
	DC 24 V	Up to 0.1 A	P24 - GND	Pay attention to the polarity.
	Three-phase 200 VAC	Up to 250 W	Ue - Ve - We	For coolant motor
*	AC 100 V	Up to 0.5 A	153 - 154	For coolant valve
*	AC 100 V	Up to 0.5 A	163 - 164	For spindle air-blow valve

	Voltage	Capacity	Terminal No. of seal	Remarks
	AC 100 V	1A	159 - 160	
	AC 24 V	3A	147 - 148	
	DC 24 V	Up to 0.6 A	2 - 55	Pay attention to the polarity.
	Three-phase 200 VAC	Up to 250 W	UC - VC - WC	For coolant motor
*	AC 100 V	Up to 1 A	153 - 154	For coolant valve
*	AC 100 V	Up to 0.5 A	149 - 150	For spindle air-blow valve

Note: The power with the sign "*" is turnd on/off by the program of this machine.

VII Action for different cases

How to take action for the following cases:

1. Machine stops during memory operation.

   (1) Check this operation after setting to the single run mode.

   (2) Check if each operation of the ATC (Automatic Tool Changer) is complete. (Checking of motors and limit switches for input and output signals on the I/O display)

   (3) Check if each operation of auxiliary functions is complete. (Checking of signals to and from the auxiliary units on the I/O display)

   (4) Reset key is erroneously pressed.

   (5) Tool life expires.
   When the program in execution terminates, it is impossible to restart the operation before tool life is newly set.

   (6) Check the set value of the parameter-2 in-position width.

   (7) Check if the program stops.

2. How to perform manual operation during memory operation

   (1) Press the "single operation" key with block (during operation) before starting manual operation.
   However, consider the following three operations as one operation respectively:

   i) Operation until tool change is complete in ATC operation

   ii) Operation until the step returns to the home position for normal and reverse rotations in tapping

   iii) Operation until the step reaches the end position in drilling

   (2) Perform manual operation.
   When the spindle speed (rpm) is changed (including stop and reverse revolution), reset to the original state.

   (3) Press the mode key "memory (MEM)".

   (4) Press the "single run (SINGL RUN)" and "start" keys.

   (5) Check if the operation is made according to the program and then press the "cycle run (CYCL RUN)" and "start" keys.

3. How to machine with manual operation

(1) Press the mode key "manual operation".
(2) Set the cursor to the spindle speed.
(3) Press the key "spindle normal rotation (S·CW)" (or "spindle reverse rotation (S·CCW)") to specify the speed with the [△] or [▽] key.
(4) Set the cursor to the cutting speed.
(5) Press the "JOG key" to specify the feed rate with [△] or [▽] key.
(6) Move the table with the "Axis select key".
(7) Press the "spindle stop (S.STOP)" key to stop the operation.

4. How to return the step to the working zero position during memory operation

(1) Stop the machine by pressing the "STOP" or "RESET" key.
(2) Press the "manual mode (MANU)" or "MDI" key.
(3) Return the step with the "WORKING ZERO RETURN" key.
Or set the "Designate position" or "WORKING ZERO RETURN" by MDI mode and press the "start" key.

5. Machine does not work a certain type of machining

* Delete the tool for the function (item displayed at left) when assigning tools, and no machining for the function is made. Confirm the above with the assignment display.
Note: To simultaneously make machining by combining functions, delete no tool.

6. How to set a short tool position or no tool position before moving the table since the current tool contacts the work because of the tool length when moving the table.

* Because it is possible to directly call the magazine for the code of 90 to 99 with the external output signal, use the procedure to avoid the above trouble. However, be careful of setting the tool to a magazine because this machine can be operated for the tool with any magazine number even if the same program is used.

7. The message "OK" appears in checking tools though no too can be assigned.

* Check if the tool for the machining concerned is set to the tool pattern.

* Check if the tool is correctly set to the type-of-tool column for the cutting condition.

8. The reset-key LED remains lit.

* Check the set value of parameter-2 in-position width.

9. No power is applied though turning on the power supply of the operation panel by connecting the power cord and turning on the breaker handle at the CNC control box.

* Check if the LED to indicate power trouble is turned on (the voltage of 15% or more the rated voltage is applied).

10. The alarm lamp to indicate recharging of the battery is turned on.

* The lamp goes on when the voltage of the dry cell for memory backup (in the control box) is reduced to the rated value or less. In this case, replace the three size-AA dry cells with new ones immediately.

VIII Error code

Error code

Since the error number and message are displayed on the screen when an error occurs in control of this machine, reset the error with a proper method according to the table below.

The error message includes the following three levels of meaning according to the number of asterisks "*":

(1) Message with double asterisk "* *" -- Because this is a serious trouble, eliminate the cause of the error before restating the operation and then switch ON the power supply again. It is necessary to operate the key. [M.ZRT]

(2) Message with single asterisk "*" -- It is possible to reset the error with the [RESET] key. (It may be necessary to operate the [M.ZRT] key.)

(3) Message with no asterisk -- It is possible to continue the operation through the correct procedure.
The error message is shown as a typical message. For details, make confirmation by pressing the [ALARM] key.

* Error code table

	No.	Error message	Cause	How to recover
Overall system error	0	**TURN OFF POWER	The parameter is changed.	Turn OFF and then turn ON power.
	1	**CPU 1 ERROR 1	An error occurs in the master-board CPU.	Turn OFF and then turn ON power. If the error recurs, contact us.
	2	**CPU 1 ERROR 2		
	3	**CPU 2 ERROR 1	An error occurs in the slave-board CPU.	
	4	**CPU 2 ERROR 2		
	5	**RAM ERROR 1	It is impossible to read or write the master -board memory.	
	6	**ROM AREA ERROR 1	An error is found in the master-board ROM data.	
	8	**POWER FAILURE	The voltage drop of +5 and ± 15 V lines of the DC power supply	Check and repair the DC power supply unit, turn OFF and then turn ON power.
	9	**THERMAL ERROR	The coolant thermal relay operates.	Turn off the power to manually reset the thermal relay in the control panel and turn ON the power again.

	No.	Error message	Cause	How to recover
	10	**ABNORMAL TEMP.	The temperature of the regenerative braking resistance of any one of X-,Y-, and Z-axis motors isincreased.	Stop the operation to allow 10 to 20 min for cooling, and turn OFF power, and then turn ON power.
Overall system error	11	**X SERVO ERROR	An error occurs in the servo power supply. The power supply is over-loaded, or the signal line or power cable is disconnected.	Turn off the power to eliminate the cause of the error and then turn it on again.
	12	**Y SERVO ERROR		
	13	**Z SERVO ERROR		
	14	**SPINDLE SRVO ERR	The temperature of the spindle-motor regenerative braking resistance is abnormally increased.	
	15	**SPINDLE OVERHEAT	The spindle-motor temperature exceeds the specified value.	Stop the spindle to allow 10 to 20 min for cooling, and turn OFF power, and then turn ON power.
	16	**RAM ERROR 2	It is impossible to read or write the slave-board RAM.	Turn OFF and then turn ON power. If the error recurs, contact us.
	17	**ROM ERROR 2	An error is found in the slave-board ROM data.	
	21	*DEVIATION ERROR X	The deviation value of the X-axis motor exceeds the parameter value or the X-axis is overloaded.	Press the "RESET" key and check parameter deviation. It is necessary to operate the "MACHINE ZERO RETURN" key.
	22	*DEVIATION ERROR Y	The deviation value of the Y-axis motor exceeds the parameter value or the Y-axis is overloaded.	
	23	*DEVIATION ERROR Z	The deviation value of the Z-axis motor exceeds the parameter value or the Z-axis is overloaded.	

	No.	Error message	Cause	How to recover
Overall system error	24	*DEVIATION ERROR SP	The spindle-motor deviation exceeds the parameter value or the spindle is overloaded.	Press the "RESET" key, and check parameter deviation. It is necessary to operate the "MACHINE ZERO RETURN".
	25	*OVERRUN (X)	The X-axis overruns in (+) direction.	Move the X-axis out of the overrun area by pressing the "RELSE" and "RESET" keys together to set the "MANU" mode and then pressing the "-X" key. It is necessary to operate the "MACHINE ZERO RETURN" key.
	26	*OVERRUN (-X)	The X-axis overruns in (-) direction.	Move the X-axis out of the overrun area by pressing the "RELSE" and "RESET" keys together to set the "MANU" mode and then pressing the "+X" key. It is necessary to operate the "MACHINE ZERO RETURN" key.
	27	*OVERRUN (Y)	The Y-axis overruns in (+) direction.	Move the Y-axis out of the overrun area by pressing the "RELSE" and "RESET" keys together to set the "MANU" mode and then pressing the "-Y" key. It is necessary to operate the "MACHINE ZERO RETURN" key.
	28	*OVERRUN(-Y)	The Y-axis overruns in (-) direction.	Move the Y-axis out of the overrun area by pressing the "RELSE" and "RESET" keys together to set the "MANU" mode and then pressing the "+Y" key. It is necessary to operate the "MACHINE ZERO RETURN" key.

	No.	Error message	Cause	How to recover
Overall system error	29	*OVERRUN (Z)	The Z-axis overruns in the positive direction.	Move the Z-axis out of the overrun area by pressing the "RELSE" and "RESET" keys together to set the "MANU" mode and then pressing the "-Z" key. It is necessary to operate the "MACHINE ZERO RETURN" key.
	30	*OVERRUN (-Z)	The Z-axis overruns in the negative direction.	Move the Z-axis out of the overrun area by pressing the "RELSE" and "RESET" keys together to set the "MANU" mode and then pressing the "+Z" key. It is necessary to operate the "MACHINE ZERO RETURN" key.
	31	*EM SWITCH ON	The emergency-stop button is pressed.	Eliminate the cause of emergency stop, reset the emergency stop button and press the "RESET" key. It is necessary to operate the "MACHINE ZERO RETURN" key.
Error caused to entire system	33	**METHOD (TOOL)	Tools of different system in operation are displayed.	Delete all data by selecting parameter 2 of MDI operation mode. Then turn on and off power supply. After that re-set all data or read from bubble memory.
	34	**METHOD (CUTTING)	Cutting conditions of the different system in operation are set.	Press the RESET key to clear the alarm state and delete all data by selecting parameter 2 (system). Then re-set all data or read from bubble memory.

	No.	Error message	Cause	How to recover
	35	*METHOD (TAP HOLE)	Tap drill hole dia. of different system in operation is displayed.	Same as above.
Error caused to entire system	36	*METHOD (PROGRAM)	Program of different system in operation	Set correct program number. In case this alarm occurs with the machine other than the machine provided with inch and metric system selection function, data would have been destroled. In this case programs should be deleted. (In PROGRAM EDIT mode, pressing RESET key is necessary.)
	37	*METHOD (MAGAZINE)	Tools for different system in operation are installed in magazine.	Confirm tools in magazine and remove unnecessary tools. (Pressing RESET key is necessary.)
	41	*POSITION ERROR (X)	It is impossible to check if the dog is turned off during "zero-position return"	It is impossible to check if the dog is turned off during "ZERO-POSITION RETURN".
	42	*POSITION ERROR (Y)		
	43	*POSITION ERROR (Z)	When the Z-axis is in the ATC operation area it is manually fed.	Cancel the error display with the "reset" key and move the Z-axis out of the ATC operation area. If the error recurs in other casez, contact us.

	No.	Error message	Cause	How to recover
Overall system error	45	*COMMAND ERROR	The spindle speed command is output when the Z-axis is in the ATC operation area.	Cancel the error display with the "RESET" key and move the Z-axis out of the ATC operation area. If the error recurs in other cases, contact us.
	46	*ATC ERROR	After the magazine rotates, it is not found in the specified magazine number. Though the Z-axis moves to the ascending limit, the ATC zero-position limit switch is not turned on.	Check the magazine number and the operation of each limit switch (especially the limit switch at the ascending limit) using the I/O table for correction and press the "RESET" key.
	47	*SYSTEM ERROR 1	An error occurs in the system.	Press the "RESET" key to cancel the error display. If the error recurs, contact us.
	51	*SYSTEM ERROR 2	Though the error signal is detected, the error concerned is not found.	Press the "RESET" key. If the error recurs, contact us.
Error during key-in operation	61	SPDL RPM NOT DSGN	Though the spindle speed command is output in the key-in operation, the speed concerned is not found.	Press the "RESET" key and command rpm.
	62	*ILLEGAL AXIS	The X-, Y-, and Z-axes are simultaneously assigned in the key-in operation, or the Z- and X-axes or the Z- and Y-axes are simultaneously assigned in the operation.	Press the "RESET" key and command Z-axis alone or both the X- and Y-axes.

	No.	Error message	Cause	How to recover
Error during key-in operation	63	*AXIS NOT DSGNTD	Though the axis moving command is output in the key-in operation, no axis data is found.	Press the "RESET" key and command the axis data.
	64	*X OR Y DSGN IN TAP	When tapping is assigned, neither X- nor Y-axis cannot be assigned.	Press the "RESET" key and give normal designation.
	65	*Z NOT DSGN IN TAP	No Z-axis is assigned for tapping during key-in operation.	Press the "RESET" key and give Z-axis designation.
	66	*FDRATE NOT SPEFD	The feed rate of 0 to 0.16 inch/min is assigmed or no speed is set.	Press the "RESET" key and changing the cutting condition for memory operation.
	67	*PRESS RESET KEY	The parameter is changed.	Press the "RESET" key.
Error during memory operation	80	*ABNORMAL OPERATION	An error is found in the operation cycle.	Press the "RESET" key. If the error recurs, contact us.
	81	*ZERO RTN NOT DONE	Operation is started before "MACHINE ZERO RETURN" is made.	Press the "RESET" key and make "MACHINE ZERO RETURN".
	82	*NUT READY FOR OPE	No assigned program is found or no tool is assigned. Or no tool is present in the magazine.	Press the "RESET" key and take necessary action.
	84	*TOOL ASIGN NOT DONE	Tool assignment is uncompleted.	Press the "RESET" key and assign tools.

	No.	Error message	Cause	How to recover
Error during operation	85	*NO TOOL IN MGZNE	The tool on magazine tool display is deleted or changed in operation.	Press the "RESET" key and set the tool to the magazine tool display correctly.
	86	*N.M. NOT SPECIFIED	The number of machining processes for pattern is set to 0.	Press the "RESET" key and set the number of machining processes for pattern.
	87	*RPM NOT SPECIFIED	The peripheral speed for cutting condition is not set or too small during memory operation.	Press the "RESET" key and set the cutting condition correctly.
	88	*FDRATE NOT SPECFD	No positioning, cutting feed, or tapping feed is assigned during key-in operation.	Press the "RESET" key and designate feed-rate.
	89	*NO PECK DATA	The step data for the cutting condition is set to 0 or not set during memory operation.	Press the "RESET" key and set the step data for cutting condition.
	90	*IMPROPER TOOL	The type of tool is improper for the machining function or the type of tool in the tool table is changed after assignment of tool.	Press the "RESET" key and reassign tools.
	92	*NO TOOL IN MENU	The tool is not found in the tool menu.	Press the "RESET" key and enter the tool in the tool menu.
	93	*TOOL LNGTH NOT SET	The tool length is set to 0 or not set.	Press the "RESET" key and set the tool length.

	No.	Error message	Cause	How to recover
Error during operation	94	*DRILL DIA NOT SET	The outside or smaller diameter of drills is set to 0 or not set in the tool menu.	Press the "RESET" key and set the drill diameter in the tool menu.
	95	*TAP SIZE ERROR	The denominator of size for inch screw is expressed as 0. Example: 1/0	Press the "RESET" key and set the nominal size of tap correctly.
	96	*NO PITCH DATA	The pitch data in the tool table is set to 0 or not set for tapping.	Press the "RESET" key and set the pitch data.
	97	*R OR L HAND?	No screw-direction data is set in the tool table.	Press the "RESET" key and set the screw direction data.
	98	*TOOL LIFE END	The service life value of the tool in service reaches 0.	Press the "RESET" key and replacing the tool and resetting the tool life value.
	99	*TIME UP	No completion signal is sent back though the external signal read time of parameter is over after the external signal is output.	Press the "RESET" key and send back the externally-read completion signal within the limit time or changing the parameter value.
	100	*STROKE LIMIT OVER (The direction for over limit is also displayed.)	The unit moves exceeding the limit value during the key-in or memory operation.	Press the "RESET" key and correct the program. (Check the stroke of the parameter 2.)

	No.	Error message	Cause	How to recover
Error during operation	101	*N.D. NOT SPECIFIED	The number of pattern divisions is assigned as 0.	Press the "RESET" key and set the number of pattern divisions.
	102	*TOOL BROKEN ERROR	The tool breakdown signal is detected.	Replace the broken-down tool with new one before pressing the "RESET" key. If the tool is not broken down, adjust the sensitivity of the controller on the tool breakdown detector.
	103	*CHG TOOL DG WK	The tool menu for the tool already analyzed is changed during operation.	Press the "RESET" key and change the menu before assigning the tool and starting memory operation.
	104	*CHG CUT COND DG WK operation	The cutting condition for machining program is changed during operation.	Press the "RESET" key and change the condition before assigning the tool and starting memory operation.
	105	*STRKE OVER (The direction of over stroke is also displayed.)	The unit exceeds the stroke during manual, key-in, or memory operation.	Press the "RESET" key. Then correct the program for key-in and memory operations.
Overall system error	121	*PARITY (MAGZ)	The magazine-tool area data is destroyed.	Press the "RESET" key and set the data again.
	122	*PARITY (TOOL)	The tool-table data is destroyed.	Press the "RESET" key and delet the whole data in each data area to reset the error. (See Item "DELETION" for different cutting conditions.)
	123	*PARITY (PTRN)	The tool-menu data is destroyed.	

	No.	Error message	Cause	How to recover
Overall system error	124	*PARITY (CUTC)	The cutting-condition data is destroyed.	Press the "RESET" key and delete the whole data in each data area to reset the error. (See Item "DELETION" for different cutting conditions.)
	125	*PARITY (HOLE)	Data for the hole size before tapping is destroyed.	
	126	*PARITY (PARM)	The parameter data is destroyed.	Press the "RESET" key and set the parameter again.
Error during edition	127	*PARITY (PROGRAM)	The program data is destroyed.	Press the "RESET" key and delete the program to edit it again.
	128	*PROGRAM ERROR	The program directory is destroyed.	
	129	*DATA NOT FOUND	The separation mark of data in the program is lost.	
	130	*MEMORY OVERFLOW	Every data area or directory is already used.	Press the "RESET" key and delete the program to prepare free area.
	131	*MATERIAL NOT DSGN	No work material is set.	Press the "RESET" key and set the work material.
	150	*INPUT DATA ERROR	The data out of assigned area is set.	Set the data in the assigned area.

	No.	Error message	Cause	How to recover
Error during edition	151	INPUT CHARACTER ERR	Data other than allowable characters is set.	Set the data for allowable character
	152	*DIGITS TOO MANY	Data exceeding the allowable number of digits is set.	Set the data with the allowable number of digits.
	153	MINUS SIGN ASSIGNED	Minus sign is set though minus data is not allowed.	Set plus sign.
	154	DECIMAL ASSIGNED	Data with a decimal point is set which should be integral data.	Set integral data.
	155	MEMORY PROTECTED	Data is changed of which memory is unrewritable.	Make the memory in rewrite enable before changing the data.
	156	MENU NUMBER ERROR	A menu number other than the assigned number is assigned.	Assign the menu number within the assigned range.
	157	ALTER POSITION ERR	The data which should not be changed is changed.	Move the cursor to the data which may be changed before changing the data. (To change a job, delete the current job or inserting a new job).
	158	DELETE IMPOSSIBLE	Data is deleted during insertion. (When a job is being inserted, the data is deleted.)	Stop the insertion by pressing the "INSRT" key again or delete data after insertion is complete.

	No.	Error message	Cause	How to recover
Error during edition	159	INSERT POSITION ERR	Data is inserted to the position other than the position where data can be inserted. (Only the job or subprogram can be inserted.)	Move the cursor to the position where data can be inserted using the [↑] and [↓] keys to insert data.
	160	DELETE POSITION ERR	Data is deleted at the position other than that where data can be deleted. (Only a job or subprogram can be inserted.)	Move the cursor to the position where data can be deleted using the [↑] and [↓] keys to delete data.
	161	WRITE DISBLE	A program is edited though the memory is rewrite disable.	Set in memory rewrite enable and renew program edition.
	162	PROG NO. NOT SPECFD	No program number is assigned.	Assign a program number.
	163	TOO MANY PROGRAMS	The 52nd program is edited.	Delete the existing program and renew program edition.
	164	TOO MANY JOBS	The 70th job is created.	To edit a new job, edit a subprogram directly. For a job other than a new job, insert the job by deleting other job.
	165	SUBPRG ALRDY DSGNTD	A subprogram is edited using the existing subprogram number.	Edit the subprogram with other number or with the number of which subprogram is deleted.

	No.	Error message	Cause	How to recover
Error during edition	166	PRGRM IN EXECUTION	The program during memory operation is edited.	Press the "RESET" key to stop the operation and then edit the program.
	167	MEMORY RUNNING	An operation is made which is not allowed during memory operation. (Menu for external memory and tool assignment)	Stop the operation before execution.
	168	WORK DATA INCOMPL	When all number of works is not input, some number is changed.	Input all number of works, and then change a number.
	169	THRD DATA INCOMPL	When all screw data are not input, some data is changed.	Input all screw data and then change some data.
	170	PATRN DATA INCOMPL	When all pattern data of machining data are not input, some data is changed.	Input all pattern data and then change some data.
	171	W.ZERO INCOMPLETE	When all data of X, Y, and Z for working zero position are not input during new editing, the cursor movement is requested.	Complete in put all working zero position X, Y and Z data.
	172	XYPOS DATA INCOMPL	When all data are not input in assigning the X and Y positions for X and Y movement, the cursor movement is requested.	Input all data of X-/Y-axes for X-/Y-position assignment.

	No.	Error message	Cause	How to recover
Error during assignment	180	REQD TOOL NOT FOUND	No tool corresponding to machining function is found during assignment of tool.	Check the following and correct data if necessary: machining data, tool pattern, hole size before tapping, cutting tool, cutting condition, contents of tool menu, bottom-hole margin of depth, and heights of jig and work.
	181	TOO MANY TOOLS ASGD	11 or more tools are assigned.	Adjust the number of tools with key-in operation so that it will be 10 or less, or change the program.
	182	NO JOB	The program with no job.	
Message during checking tools	183	DEPTH SHALLOW	When chamfering is made with the chamfering tool, the tip of the tool contacts the bottom of hole.	Check the machining data and the contents of the tool menu and correct data if necessary.
	184	SHALLOW CONTRBORE	When chamfering is made with the chamfering tool, the tip of the tool contacts the seat surface.	
	185	WORK HEIGHT IMPRPR	When the work height and cutting depth are calculated, the results shows the value smaller than the table height.	
	186	S.DIA TOO SMALL	The smaller diameter of the tool is insufficient for the machining data.	
	187	TL LGTH NOT SPECFD	No tool length is set.	

	No.	Error message	Cause	How to recover
Message during checking tools	188	TL LIFE NOT SPECFD	No tool life time is set.	Check the machining data and the contents of the tool menu and correct data if necessary.
	189	TOOL TOO LONG	The tool length is too long.	
	190	S.DIA TOO LONG	The smaller diameter is too large.	
	191	IMPROPER S.DIA	The smaller diameter is incorrect.	
	192	S.DIA L TOO LONG	The smaller-diameter length is too long.	
	193	IMPROPER S.DIA L	The smaller-diameter length is incorrect.	
	194	S.DIA L TOO SHORT	The smaller-diameter length is too short.	
	195	S.DIA L, INEF L ERR	The margin of depth is different from the smaller-diameter length or ineffective length.	
	196	TOOL TOO SHORT	The tool length is too short.	
	197	TOOL DIA TOO SMALL	The tool diameter is too small.	
	198	IMPROPER TOOL DIA	The tool diameter is incorrect.	
	199	IMPROPER CENTER ANG	The center angle is incorrect.	
	200	IMPROPER CHAMF ANG	The angle of chamfer is incorrect.	
	201	FLUTE TOO SHORT	The flute length is too short.	
	202	TORSION IMPROPER	The torsion direction is incorrect.	
	203	THREAD IMPROPER	The type of screw thread is incorrect.	

	No.	Error message	Cause	How to recover
Message during checking tools	204	THREAD DIA IMPROPER	The screw thread dia. is incorrect.	Check the machining data and the contents of the tool menu and correct data if necessary.
	205	PITCH IMPROPER	The pitch or threads per inch is incorrect.	
	206	PRIMARY DEPTH INSUF	The margin of depth is too small.	
	207	NO TOOL NUMBER	The tool with no number is checked.	
	208	NO DRILL THRU	Blind hole or too short smaller-diameter length	

	No.	Error message	Cause	How to recover
External input/output error	220	CHECK ERROR	The designated external program data does not match the internal program data in the machine.	Check whether the internal program matches the external program or not by means of editing. If found matched, delete the transferred program and check the error by transferring the program once again.
	221	BUBBLE PROTECTED	The protect slide of the bubble cassette is kept at the protect position.	Unlock the protect of the bubble cassette.
	222	NO BUBBLE	The bubble cassette is not found, or is not correctly set.	Correctly set the bubble cassette.
	223	NO FORMAT IN BUBBLE	Program transfer to as unused cassette was tried, or the cassette data is abnormal.	Set the external memory program delete mode, and format the cassette. (-9999 [EXE])
	224	REQD PRG NOT FOUND	The program to be transferred is not found.	Check the designated program No. by program No. display.
	225	PROG ALREADY EXISTS	A program of the same No. exists in the destination of transfer.	Change the program No., or delete the program of the same No.
	226	NO FREE AREA	There is no enough area to store the program to be transferred.	Delete unnecessary programs, or prepare a new cassette.
	227	EXT BUBBLE ERROR	An undefined command was sent.	Push the [RELSEkey, and execute the operation once again. If the same error repeatedly occurs, the bubble memory itself is considered defective.
		EXT BUBBLE ERROR 2	The bubble cassette is out of order.	Replace the bubble cassette.
		EXT BUBBLE ERROR 3	The bubble cassette is out of order.	Replace the bubble cassette.

	No.	Error message	Cause	How to recover
External input/output error	227	EXT BUBBLE ERROR 4	The bubble cassette is out of order.	Replace the bubble cassette.
		EXT BUBBLE ERROR 5	Communication between TC and the bubble memory was not correctly executed.	Push the [RESET] key, and execute the operation once again. If the same error repeatedly occurs, the bubble memory itself is considered defective.
		EXT BUBBLE ERROR 6	The bubble cassette is kept write disabled.	Put the bubble cassette into the write enable state, and execute the operation once again.
		EXT BUBBLE ERROR 7	No cassette has been set to the bubble memory.	Set the cassette to the bubble memory, and execute the operation once again.
		EXT BUBBLE ERROR 8	Communication between TC and the bubble memory was not correctly executed.	Push the [RESET] key, and execute the operation once again. If the same error repeatedly occurs, the bubble memory itself is considered defective.
		EXT BUBBLE ERROR 9	The bubble cassette is out of order.	Replace the bubble cassette.
		EXT BUBBLE ERROR 10	The bubble cassette is out of order.	Replace the bubble cassette.
		EXT BUBBLE ERROR 11	The bubble cassette is out of order.	Replace the bubble cassette.
		EXT BUBBLE ERROR 12	The bubble cassette is out of order.	Replace the bubble cassette.
		EXT BUBBLE ERROR 13	The door of the holder unit was opened during communication with the bubble memory, and the operation was put to an emergency stop.	Check the door, and execute the operation once again.

	No.	Error message	Cause	How to recover
External input/output error	227	EXT BUBBLE ERROR 14	An error occurred in the bubble memory during the input data check.	Push the [RESET] key, and execute the operation once again. If the same error repeatedly occurs, the bubble memory itself is considered defective contact us.
		EXT BUBBLE ERROR 15	Communication between TC and the bubble memory was not correctly executed.	Push the [RESET] key, and execute the operation once again. If the same error repeatedly occurs, the bubble memory itself is considered defective contact us.
	228	EXT CONNECT ERROR	The external input/output 7 was selected in the program edit mode within 7 or 8 seconds after the power input, or the wiring cable has not been connected with the device for external input/output, or the parameter destination is not the external device.	Cut off the power source, and check the connected condition of the wiring cable. If the wiring connection was found normal, input the power source again check the external input/output 7 in the program edit mode after 7 or 8 seconds after the power input.
	229	PERIPHERAL ERROR (error in TC interior)	* Some other command data than those designated was input from TC.	Push the [CAN] key, and execute the operation once again.

	No.	Error message	Cause	How to recover
External input/output error	229	PERIPHERAL ERROR 2 (caused by incorrect format)	* Some other code than ACK or NAC was returned during data transfer. * Some other code than SYN was returned after WAIT. * Some other code than ACK was returned after EOD. * The location of ":", "%", and "LF" for data input from the reader is abnormal. * The NULL data input is longer than 60 cm (23.6"). * The location of ":" and ";" for data input from the computer is abnormal. * The format for directory input from the computer is incorrect.	In case the external unit is a computer, push the [CAN] key and check whether the program on the computer side conforms to the protocol or not. In case the external unit is a PTR, set the tape once again so that the NULL data part becomes shorter than 60 cm.
		PERIPHERAL ERROR 3 (caused by invalid type of data)	* The PTR or the computer does not the designated data.	Push the [CAN] key, and set the same data as requested by TC.
		PERIPHERAL ERROR 4 (caused by invalid type of data)	* The PTR or the computer does not send the designated data.	Push the [CAN] key, and set the data of the same program No. as of the data requested by TC.
		PERIPHERAL ERROR 5 (caused by different number of data)	* The number of REQBLK data (17 including CR) or the number of DATABLK data (138 including CR) is abnormal. * The number of machining data input from TC is abnormal (not a multiple of 128).	In case the external unit is a computer, push the [CAN] key and check whether the program on the computer side conforms to the protocol or not. In case the external unit is a PTR, set the tape once again so that the NULL data part becomes shorter than 60 cm.

	No.	Error message	Cause	How to recover
External input/output error	229	PERIPHERAL ERROR 6 (error in TC interior)	* Invalid command sequence was designated by TC.	Push the [CAN] key, and execute the operation once again.
		PERIPHERAL ERROR 7 (communication unit unit inactive)	* The DC input was not made active within 90 seconds.	In case the external unit is computer, push the [CAN] key and turn on the ER signal on the computer side. In case of PTR or PTP, turn on the power supply. If the same error message is still displayed, check the cable connection.
		PERIPHERAL ERROR 8 (time out)	* Data was not input within the limit time. * Data was not output within the limit time.	In case the external unit is a computer, push the [CAN] key and check whether the program on the computer side conforms to the protocol or not. In case the external unit is a PTR, check whether the control line system is usable or not. If usable, check the cable connection.
		PERIPHERAL ERROR 9	* The ROM or RAM of the RS board is out of order. * The CPU of the RS board is out of order. (LED1 is lighting.)	Replace the RS board.
		PERIPHERAL ERROR 10 (requested data not found in computer)	The requested data is not found in the computer.	Push the [CAN] key. not found in the computer.
		PERIPHERAL ERROR 11 (NAK returned from computer seven times)	NAK was returned from the computer repeatedly seven times.	Push the [CAN] key, and check whether the program on the computer side conforms to the protocol or not.

	No.	Error message	Cause	How to recover
		PERIPHERAL ERROR 12 (caused by capacity overflow or invalid number of blocks)	* The program is too long (121 blocks or more). * The directory is too long (13 blocks or more). * The number of blocks of input data from the computer is invalid.	In case the external unit is a computer, push the [CAN] key and check whether the computer conforms to the protocol or not.
External input/output error	229	PERIPHERAL ERROR 13 (DATABLK check sum error)	DATABLK check sum error	In case the external unit is a computer, push the [CAN] key and check whether the computer conforms to the protocol or not.
		PERIPHERAL ERROR 14 (serial transmission error)	* A parity error, framing error, or overrun error occurred during data reception from the computer or the reader.	Push the [CAN] key and check whether the baud rate, parity, data length (8 bits), and stop bit on the computer, PTR, or PTP side match the parameters or not.
		PERIPHERAL ERROR 15 (error in TC interior)	A parity error occurred during data input from TC.	Push the [CAN] key, and execute the operation once again.

	No.	Error message	Cause	How to recover
Error during operation	240	TOOL ALRDY REGISTRD	The same tool is already entered in the magazine.	Check the tool number.
	241	NO PROGRAM	The assigned program is absent.	Check the program number.
	242	NO DATA IN MENU	The assigned data is not found in the tool menu.	Assign the tool with the tool number listed in the tool menu.
	243	NO SUBPROGRAM	Though the subprogram is called, the number corresponding to the subprogram is not found.	Enter the subprogram or delete calling.
	244	MACHINING ORD ERROR	The machining order is set other than Nos. 1 through 4. For the order Nos. 2 through 4, a job excluded from the machining job is present between machining jobs.	Set the machining order correctly.
	245	JOB WITHOUT TOOL	A machining job is present which uses no tool.	Delete the job.
	246	NO TOOL IN MAGAZINE	The tool necessary for memory operation is not entered in the magazine.	Enter the necessary tool in the magazine.
	247	INADEQUATE TOOL	The tool is not suitable for the machining.	Set the tool suitable for the machining.
	248	NO. OF WRKPCS SPECFD	The number of works is set to 0 or not set.	Set the number of works in the program edit mode.
	249	MDI RUNNING	Memory operation is started at the time of hold during key-in operation.	Press the "RESET" key to stop the operation and then start memory operation.

	No.	Error message	Cause	How to recover
Error during operation	250	MEMORY RUNNING	Key-in operation is started at the time of hold during memory operation.	Press the "RESET" key to stop the operation and then start key-in operation.
	251	NON SCHEDULE PRGRAM.	No program necessary for the schedule program of parameter 5 is entered.	Enter the parameter-5 schedule program and then start memory operation.

690084000
8712 ②