Operating Manual
for
VTC-200B, VTC-200C & VTC-300C
(Mazatrol Fusion 640M)
Publication # C557SG0012E

7/99

CAUTION

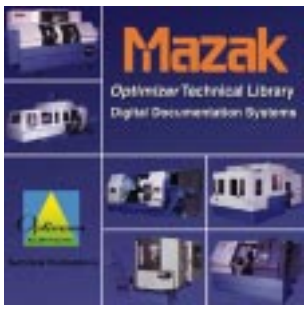
This Manual is published to assist experienced personnel on the operation, maintenance and/or programming of Mazak machine tools.

All Mazak machine tools are engineered with a number of safety devices to protect personnel and equipment from injury or damage. Operators should not, however, rely solely upon these safety devices, but should operate the machine only after fully understanding what special precautions to take by reading the following documentation thoroughly.

Do not attempt to operate or perform maintenance / repair on the machine without a thorough understanding of the actions about to be taken. If any question exists, contact the nearest Mazak service center for assistance.

Certain covers, doors or safety guards may be open or removed to more clearly show machine components. These items must be in place before operating the machine. Failure to comply with this instruction may result in serious personal injury or damage to the machine tool.

This manual was considered complete and accurate at the time of publication, however, due to our desire to constantly improve the quality and specification of all Mazak products, it is subject to change or modification.



Notes:

Sec. 3: Operating NC Unit & Preparation for Auto Operation .....

Sec. 4: Automatic Operation .....

SAFETY PRECAUTIONS

The machine is provided with a number of safety devices to protect personnel and equipment from injury and damage. Operators should not, however, rely solely upon these safety devices, but should operate the machine only after fully understanding what special precautions to take by reading the following documentation thoroughly.

• BASIC OPERATING PRACTICES

DANGER:

1) Some control panels, transformers, motors, junction boxes and other parts have high voltage terminals. These should not be touched or a severe electric shock may be sustained.
2) Do not touch any switches with wet hands. This too, can produce an electric shock.

WARNING:

1) The emergency stop pushbutton switch location should be well known, so that it can be operated at any time without having to look for it.
2) Before replacing a fuse, turn off the main incoming power switch to the machine.
3) Provide sufficient working space to avoid hazardous falls.
4) Water or oil can make floors slippery and hazardous. All floors should be clean and dry to prevent accidents
5) Do not operated any switch without a thorough understanding of the actions about to be taken.
6) Avoid accidental operation of switches.
7) Work benches near the machine must be strong enough to hold materials placed on them to prevent accidents. Articles should be prevented from slipping off the bench surface.
8) If a job is to be done by two or more persons, coordinating signals should be given at each step of the operation. The next step should not be taken unless a signal is given and acknowledged.

CAUTION:

1) In the event of power failure, turn off the main circuit breaker immediately.
2) Use the recommended hydraulic oils, lubricants and grease or acceptable equivalents.
3) Replacement fuses should have the proper current ratings.
4) Protect the NC unit, operating panel, electric control panel, etc. from shocks, since this could cause a failure or malfunction.
5) Do not change parameters or electrical settings. If changes are unavoidable, record the values prior to the change so that they can be returned to their original settings, if necessary.
6) Do not deface, scratch or remove any caution plate. Should it become illegible or missing, order another caution plate from the supplier, specifying the part number shown at the lower right corner of the plate.

• BEFORE POWERING UP

DANGER:

Cables, cords or electric wires whose insulation is damaged can produce current leaks and electric shocks. Before using, check their condition.

WARNING:

1) Be sure the instruction manual and the programming manual are fully understood before operating the machine. Every function and operating procedure should be completely clear.
2) Use approved oil resistant safety shoes, safety goggles with side covers, safe clothes, and other safety protection required.
3) Close all NC unit, operating panel, electric control panel doors and covers.

CAUTION:

1) The power cable from the factory feeder switch to the machine main circuit breaker should have a sufficient sectional area to handle the electric power used.
2) Cables which must be laid on the floor must be protected from hot chips, by using rigid or other approved conduit, so that short-circuits will not occur.
3) Before first time operation of the machine after unpacking it or from being idle for a long period of time (several days or more), each sliding part must be sufficiently lubricated. To do so, push and release the pump button several times until the oil seeps out on the sliding parts. The pump button has a return spring, so do not force it to return.
4) Oil reservoirs should be filled to indicated levels. Check and add oil, if needed.
5) For lubrication points, oil specification and appropriate levels, see the various instruction plates.
6) Switches and levers should operate smoothly. Check that they do.
7) When powering the machine on, turn on the switches in the following order: first the factory feeder switch, then the machine main circuit breaker, and then the control power on switch located on the operating panel.
8) Check the coolant level, and add coolant, if needed.

• **AFTER CONTROL POWER IS TURNED ON**

CAUTION:

When the control power “ON” switch on the operating panel is on, the "READY" lamp on the operating panel should also be on (check to see that it is).

• **ROUTINE INSPECTIONS**

WARNING:

When checking belt tensions, do not get your fingers caught between the belt and pulley.

CAUTION:

1) Check pressure gages for proper readings.
2) Check motors, gear boxes and other parts for abnormal noises.
3) Check the motor lubrication, and sliding parts for evidence of proper lubrication.
4) Check safety covers and safety devices for proper operation.
5) Check belt tensions. Replace any set of belts that have become stretched with a fresh matching set.

• **WARM UP**

CAUTION:

1) Warm up the machine, especially the spindle and feed shaft, by running the machine for 10 to 20 minutes at about one-half or one-third the maximum speed in the automatic operation mode.
2) The automatic operation program should cause each machine component to operate. At the same time, check their operations.
3) Be particularly careful to warm up the spindle which can turn above 4000 rpm.

   If the machine is used for actual machining immediately after being started up following a long idle period, the sliding parts may be worn due to the lack of oil. Also, thermal expansion of the machine components can jeopardize machining accuracy. To prevent this condition, always make sure that the machine is warmed up.

• **PREPARATIONS**

WARNING:

1) Tooling should conform to the machine specifications, dimensions and types.
2) Replace all seriously worn tools with new ones to prevent injuries.
3) The work area should be adequately lighted to facilitate safety checks.
4) Tools and other items around the machine or equipment should be stored to ensure good footing and clear aisles.
5) Do not place tools or any other items on the headstock, turret, covers and similar places (For T/M).

CAUTION:

1) Tool lengths should be within specified tolerances to prevent interference.
2) After installing a tool, make a trial run.

• **OPERATION**

WARNING:

1) Do not work with long hair that can be caught by the machine. Tie it back, out of the way.
2) Do not operate switches with gloves on. This could cause mis-operation.
3) Whenever a heavy workpiece must be moved, if there is any risk involved, two or more people should work together.
4) Only trained, qualified workers should operate forklift trucks, cranes or similar equipment and apply slings.
5) Whenever operating a forklift truck, crane or similar equipment, special care should be taken to prevent collisions and damage to the surroundings.
6) Wire ropes or slings should be strong enough to handle the loads to be lifted and should conform to the mandatory provisions.
7) Grip workpieces securely.
8) Stop the machine before adjusting the coolant nozzle at the tip.
9) Never touch a turning workpiece in the spindle with bare hands, or in any other way.
10) To remove a workpiece from the machine other than by a pallet changer, stop the tool and provide plenty of distance between the workpiece and the tool (for M/C).
11) While a workpiece or tool is turning, do not wipe it off or remove chips with a cloth or by hand. Always stop the machine first and then use a brush and a sweeper.
12) Do not operate the machine with the chuck and front safety covers removed (For T/M).
13) Use a brush to remove chips from the tool tip, do not use bare hands .
14) Stop the machine whenever installing or removing a tool.
15) Whenever machining magnesium alloy parts, wear a protective mask.

CAUTION:

1) During automatic operation, never open the machine door. Machines equipped with the door interlock will set the program to single step.
2) When performing heavy-duty machining, carefully prevent chips from being accumulated since hot chips from certain materials can cause a fire.

• TO INTERRUPT MACHINING

WARNING:

When leaving the machine temporarily after completing a job, turn off the power switch on the operation panel, and also the main circuit breaker.

• COMPLETING A JOB

CAUTION:

1) Always clean the machine or equipment. Remove and dispose of chips and clean cover windows, etc.
2) Make sure the machine has stopped running, before cleaning.
3) Return each machine component to its initial condition.
4) Check the wipers for breakage. Replace broken wipers.
5) Check the coolant, hydraulic oils and lubricants for contamination. Change them if they are seriously contaminated.
6) Check the coolant, hydraulic oil and lubricant levels. Add if necessary.
7) Clean the oil pan filter.
8) Before leaving the machine at the end of the shift, turn off the power switch on the operating panel, machine main circuit breaker and factory feeder switch in that order.

• SAFETY DEVICES

1) Front cover, rear cover and coolant cover.
2) Chuck barrier, tail barrier and tool barrier (NC software).
3) Stored stroke limit (NC software).
4) Emergency stop pushbutton switch.

• MAINTENANCE OPERATION PREPARATIONS

1) Do not proceed to do any maintenance operation unless instructed to do so by the foreman.
2) Replacement parts, consumables (packing, oil seals, O rings, bearing, oil and grease, etc.) Should be arranged in advance.
3) Prepare preventive maintenance and record maintenance programs.

CAUTION:

1) Thoroughly read and understand the safety precautions in the instruction manual.
2) Thoroughly read the whole maintenance manual and fully understand the principles, construction and precautions involved.

• MAINTENANCE OPERATION

DANGER:

1) Those not engaged in the maintenance work should not operate the main circuit breaker or the control power "ON" switch on the operating panel. For this purpose, "Do not Touch the Switch, Maintenance Operation in Progress!" or similar warning should be indicated on such switches and at any other appropriate locations. Such indication should be secured by a semi-permanent means in the reading direction.
2) With the machine turned on, any maintenance operation can be dangerous. In principle, the main circuit breaker should be turned off throughout the maintenance operation.

WARNING:

1) The electrical maintenance should be done by a qualified person or by others competent to do the job. Keep close contact with the responsible person. Do not proceed alone.
2) Overtravel limit and proximity switches and interlock mechanisms including functional parts should not be removed or modified.
3) When working at a height, use steps or ladders which are maintained and controlled daily for safety.
4) Fuses, cables, etc. made by qualified manufacturers should be employed.

• BEFORE OPERATION & MAINTENANCE BEGINS

WARNING:

1) Arrange things in order around the section to receive the maintenance, including working environments. Wipe water and oil off parts and provide safe working environments.
2) All parts and waste oils should be removed by the operator and placed far enough away from the machine to be safe.

CAUTION:

1) The maintenance person should check that the machine operates safely.
2) Maintenance and inspection data should be recorded and kept for reference.

000X717-KY 11/98

WARNING

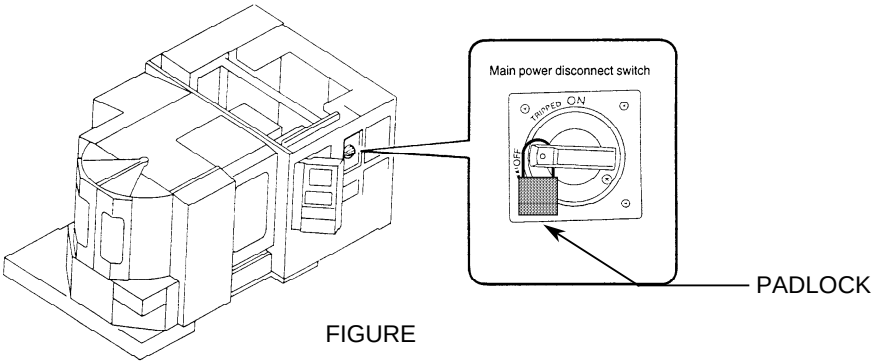
ALWAYS TURN THE MAIN CIRCUIT BREAKER TO THE "OFF" POSITION & USE AN APPROVED LOCKOUT DEVICE WHEN COMPLETING MAINTENANCE OR REPAIRS.

THE LOCKOUT PROCEDURE THAT FOLLOWS IS INTENDED TO SAFEGUARD PERSONNEL & EQUIPMENT DURING MAINTENANCE OPERATIONS, AND, REPRESENTS THE MINIMUM REQUIREMENTS. ANY ACTION SHOULD BE PRECEDED BY A "HAZARD ANALYSIS" TO DETERMINE ANY ADDITIONAL SAFETY PRECAUTIONS THAT MAY BE NECESSARY TO ENSURE THE SAFETY OF PERSONNEL AND EQUIPMENT.

***NOTE*: USE OF THE FOLLOWING LOCKOUT PROCEDURE IS MANDATORY WHEN COMPLETING MAINTENANCE OR REPAIRS.**

LOCKOUT PROCEDURE

1) THE LOCKOUT PROCESS MUST BE PERFORMED BY AUTHORIZED PERSONNEL ONLY.
2) INFORM ALL EFFECTED PERSONNEL OF YOUR INTENT TO LOCKOUT AND SERVICE THE SPECIFIED MACHINE.
3) SHUT OFF MACHINE POWER USING NORMAL SHUT DOWN PROCEDURES.
4) TURN OFF THE MACHINE AND INDIVIDUAL BUILDING CIRCUIT BREAKERS. MAKE SURE ALL STORED ELECTRICAL ENERGY IS RELIEVED. (EG: SPINDLE & AXIS SERVO CONTROLLERS)
5) CONNECT THE LOCKOUT DEVICE AS SHOWN IN FIGURE 1, AND ATTACH THE APPROPRIATE TAG AT THE MACHINE CIRCUIT BREAKER. THE TAG MUST IDENTIFY THE PERSON RESPONSIBLE FOR THE LOCKOUT. THIS WILL ENSURE THAT POWER CANNOT BE RESTORED BY ANYONE ELSE.
6) TEST THE MACHINE TO VERIFY THAT MACHINE SYSTEMS DO NOT OPERATE IN ANY WAY. ONCE TESTING IS COMPLETE, MAKE SURE ALL SWITCHES ARE IN THE "OFF" POSITION. CONFIRM THAT THE LOCKOUT DEVICES REMAIN PROPERLY INSTALLED.
7) COMPLETE THE REQUIRED MAINTENANCE OPERATIONS.
8) MAKE SURE ALL PERSONNEL ARE CLEAR OF THE MACHINE.
9) REMOVE THE LOCKOUT DEVICE. MAKE SURE ALL PERSONNEL ARE AT A SAFE LOCATION BEFORE RESTORING MACHINE POWER.



FIGURE

INSTALLATION PRECAUTIONS

The following subjects outline the items that directly affect the machine installation and start-up. To ensure an efficient and timely installation, please follow these recommendations before calling to schedule a service engineer.

• ENVIRONMENTAL REQUIREMENTS

Avoid the following places for installing the machine:

1) Avoid exposure to direct sunlight and/or near a heat source, etc. Ambient temperature during operation: 0° thru 45°C (32°F to 113°F).
2) Avoid areas where the humidity fluctuates greatly and/or if high humidity is present; normally 75% and below in relative humidity. A higher humidity deteriorates insulation and might accelerate the deterioration of parts.
3) Avoid areas that are especially dusty and/or where acid fumes, corrosive gases and salt are present.
4) Avoid areas of high vibration.
5) Avoid soft or weak ground **(minimum load bearing capacity of 1025 lbs./ft^2)**

• FOUNDATION REQUIREMENTS

For high machining accuracy, the foundation must be firm and rigid. This is typically accomplished by securely fastening the machine to the foundation with anchor bolts. In addition, the depth of concrete should be as deep as possible (minimum 6 - 8 inches). Note the following:

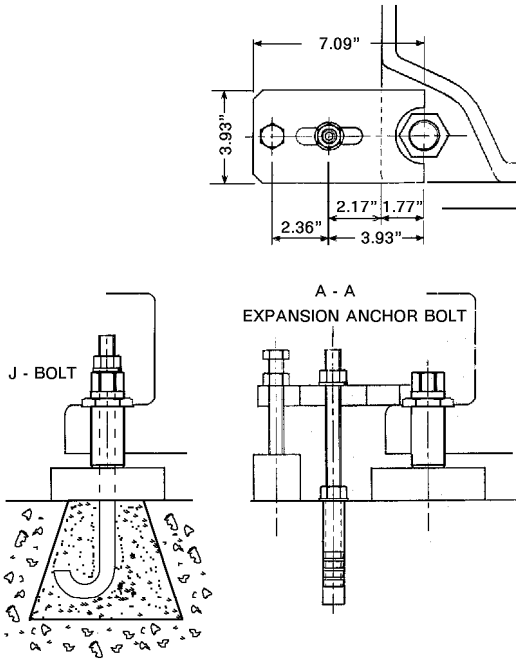
1) There can be no cracks in the foundation concrete or surrounding area.
2) Vibration proofing material (such as asphalt) should be put all around the concrete pad.
3) Form a “cone” in the foundation for J-bolt anchors, or use expansion anchors.
4) With the foundation anchor bolt holes open pour the primary **concrete at a minimum thickness of 6 - 8 inches.** Typically, the concrete must have a **minimum compression rating of 2500 lbs. @ 250 lbs. compression** and strengthened with reinforcing rods. When the concrete has cured, rough level the machine, and install the J-bolts, leveling blocks, etc., and pour grout into foundation bolt holes.
5) Mix an anti-shrinkage agent such as Denka CSA with concrete, or use Embeco grout to fill the foundation bolt holes.
6) In pouring grout, fasten the leveling block base plates with the collar retaining screws to prevent the base plates from dropping. When the grout has completely hardened, level the machine properly, and tighten M24 nuts to secure the machine to the foundation.

Note:

The machine must be anchored to the foundation with J-bolts, expansion bolts or other suitable method.

The machine accuracy and alignment specifications quoted by Mazak can usually be obtained when the minimum foundation requirements are met. However, production of close tolerance parts requires the use of an appropriate certified foundation. Foundations that do not meet certified specifications may require more frequent machine re-leveling and re-alignment, which can not be provided under terms of warranty.

If any of these conditions cannot be met, contact the nearest Mazak service office immediately.



TYPICAL MACHINE HOLD DOWN OPTIONS

• WIRING

1) Use only electrical conductors with performance ratings equivalent or superior.
2) Do not connect any power cables for devices which can cause line noise to the power distribution panel, such as arc welders and high frequency machinery.
3) Arrange for a qualified electrician to connect the power lines.
4) Incoming supply voltage should not deviate more than ±10% of specified supply voltage.
5) Source frequency should be±2 Hz of nominal frequency.

[CAUTION]

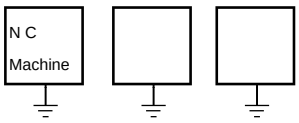
VERIFY THE ACTUAL MACHINE ELECTRICAL POWER REQUIREMENT AND THE MAIN TRANSFORMER RATING (IF APPLICABLE), AS WELL AS THE LOCAL ELECTRICAL CODE BEFORE SIZING AND INSTALLING THE INCOMING POWER WIRING.

PLEASE SEE THE ADDITIONAL CAUTIONS ON THE FOLLOWING PAGE.

• GROUNDING

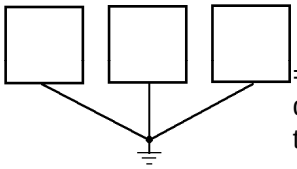
1) An isolated earth ground with a resistance to ground of less than 100 ohms is required. Typically, a 5/8" copper rod, 8 feet long, and no more than 5 feet from the machine, is sufficient. Building grounds or multiple machines grounded to the same ground rod, are not acceptable.
2) The wire size should be greater than AWG (American Wire Gauge) No. 5 and SWG (British Legal Standard Wire Gauge) No. 6.

Desirable Independent Grounding:



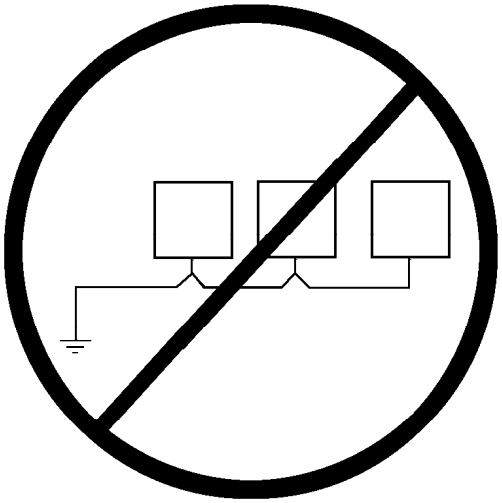
Earth resistance:
Less than 100 ohms

Common Grounds:



Resistance to ground = 100 ÷ the number of devices connected to the grounding (ohms)

Note: Never ground equipment as shown below:



CAUTION

A step-down transformer is *optional* on some machine models. Be certain to verify the transformer Kva rating (where applicable), as well as local electrical code requirements *before* sizing and installing the incoming power wiring.

Machines not equipped with a main transformer are wired for 230 VAC, 3 phase. The end user must supply a step-down transformer where factory electrical power varies more than ± 10% of the 230 VAC rating.

NOTE:

Step-down or voltage regulating transformers are external (peripheral) to the machine tool and are considered the primary input line (source) for the machine. Local electrical code or practice may require a circuit breaker or other switching device for the isolation of electrical power when this type of transformer is used. In such cases, the machine tool end user is required to supply the necessary circuit breaker or switching device.

FAILURE TO COMPLY CAN RESULT IN PERSONAL INJURY AND DAMAGE TO THE MACHINE. IF ANY QUESTION EXISTS, CONTACT THE NEAREST MAZAK SERVICE CENTER FOR ASSISTANCE.

WARNING

MAZATROL CNC CONTROLLERS PROVIDE PARAMETER SETTINGS TO LIMIT SPINDLE RPM. THESE SETTINGS ARE BASED ON THE MAXIMUM SPEED SPECIFIED BY THE CHUCK/ACTUATOR MANUFACTURER.

MAKE SURE TO SET THESE PARAMETERS ACCORDING TO CHUCK SPECIFICATION WHEN INSTALLING A CHUCKING PACKAGE. ALSO, STAMP THE MAXIMUM SPINDLE RPM ON THE CHUCK IDENTIFICATION PLATE LOCATED ON THE MACHINE TOOL COVERS.

REFERENCE THE CNC PARAMETER MANUAL SUPPLIED WITH THE SPECIFIC MACHINE TOOL TO IDENTIFY THE REQUIRED PARAMETERS TO CHANGE.

FAILURE TO COMPLY WITH THESE INSTRUCTIONS COULD RESULT IN DAMAGE TO THE MACHINE, SERIOUS INJURY OR DEATH.

IF ANY QUESTIONS EXIST, CONTACT THE NEAREST MAZAK SERVICE CENTER FOR ASSISTANCE.

WARNING

MAZAK MACHINES ARE ENGINEERED WITH A NUMBER OF SAFETY DEVICES TO PROTECT PERSONNEL AND EQUIPMENT FROM INJURY AND DAMAGE.

DO NOT REMOVE, DISCONNECT, BYPASS OR MODIFY ANY LIMIT SWITCH, INTERLOCK, COVER, OR OTHER SAFETY FEATURE IN ANY WAY, EITHER MECHANICALLY OR ELECTRICALLY.

FAILURE TO COMPLY WITH THESE INSTRUCTIONS COULD RESULT IN DAMAGE TO THE MACHINE, SERIOUS INJURY OR DEATH.

IF ANY QUESTIONS EXIST, CONTACT THE NEAREST MAZAK SERVICE CENTER FOR ASSISTANCE.

WARNING

MAZAK MACHINES ARE ENGINEERED WITH A NUMBER OF SAFETY DEVICES TO PROTECT PERSONNEL AND EQUIPMENT FROM INJURY AND DAMAGE.

MACHINE OPERATOR DOORS AND COVERS ARE DESIGNED TO WITHSTAND ACCIDENTAL IMPACT OF A BROKEN INSERT WHERE A MAXIMUM WEIGHT INSERT AT MAXIMUM TOOL DIAMETER IS RUNNING AT MAXIMUM SPINDLE RPM

NEVER USE A CUTTING TOOL OR TOOL INSERT THAT EXCEEDS MACHINE SPECIFICATIONS OR THAT OF A SPECIFIC TOOL HOLDER ITSELF, WHICHEVER IS LESS. THIS RESTRICTION APPLIES TO DIAMETER, WEIGHT, MAXIMUM SPINDLE RPM, MAXIMUM CUTTING TOOL ROTATION SPEED, ETC.

FOR COMPLETE SPECIFICATIONS, MAKE SURE TO REFERENCE OPERATION, MAINTENANCE AND DETAIL SPECIFICATION DOCUMENTATION SUPPLIED WITH THE MACHINE AND BY THE TOOLING MANUFACTURER.

NOTE: THE MAXIMUM INSERT WEIGHT FOR MAZAK MACHINES IS 20 gf. (0.04 lbs.).

FAILURE TO COMPLY WITH THESE INSTRUCTIONS COULD RESULT IN DAMAGE TO THE MACHINE, SERIOUS INJURY OR DEATH.

IF ANY QUESTIONS EXIST, CONTACT THE NEAREST MAZAK SERVICE CENTER FOR ASSISTANCE.

WARNING

BEFORE STARTING OPERATION, CHECK THAT THE WORKPIECE IS SECURELY MOUNTED IN A VISE OR A SUITABLE FIXTURE. BE CERTAIN THAT THE MOUNTING IS SUFFICIENT TO WITHSTAND CUTTING FORCES DURING WORKPIECE MACHINING.

FAILURE TO COMPLY WITH THESE INSTRUCTIONS COULD RESULT IN DAMAGE TO THE MACHINE, SERIOUS INJURY OR DEATH.

IF ANY QUESTIONS EXIST, CONTACT THE NEAREST MAZAK SERVICE CENTER FOR ASSISTANCE.

WARNING

CONFIRM PROPER WORKPIECE FIXTURING/CLAMPING, TOOL SETUP AND THAT THE MACHINE DOOR IS SECURELY CLOSED BEFORE THE START OF MACHINING.

VERIFY ALL SAFETY PRECAUTIONS OUTLINED IN THIS MANUAL BEFORE USING THE FOLLOWING CUTTING CONDITIONS:

- CUTTING CONDITIONS THAT ARE THE RESULT OF THE MAZATROL FUSION 640 AUTOMATIC CUTTING DETERMINATION FUNCTION
- CUTTING CONDITIONS SUGGESTED BY THE MACHINING NAVAGATION FUNCTION
- CUTTING CONDITIONS FOR TOOLS THAT ARE SUGGESTED TO BE USED BY THE MACHINING NAVAGATION FUNCTION

FAILURE TO COMPLY WITH THESE INSTRUCTIONS COULD RESULT IN DAMAGE TO THE MACHINE, SERIOUS INJURY OR DEATH.

IF ANY QUESTIONS EXIST, CONTACT THE NEAREST MAZAK SERVICE CENTER FOR ASSISTANCE.

Return to Library

MACHINE DOCUMENTATION CUSTOMER EVALUATION

Mazak

The Other Thoroughbred From Kentucky

Your opinion is important to enable us to issue documentation that will fit your needs.
Thank you for taking the time to supply this information.

Date: _____ _____

Machine Type: _____ Machine Serial#: _____ NC Type: _____

Customer: _____ Reported By: _____

Address: _____ Position: _____

_____ Telephone#: _____

Manual Publication #: _____

	Excellent	Good	Adequate	Fair	Poor
How well is the documentation suited to your needs?	☐	☐	☐	☐	☐
Were you able to find the necessary information easily?	☐	☐	☐	☐	☐
How well are the manuals organized?	☐	☐	☐	☐	☐
How easy are the manuals to understand?	☐	☐	☐	☐	☐
Are the illustrations helpful?	☐	☐	☐	☐	☐
Overall, how do you rate the documentation?	☐	☐	☐	☐	☐

What did you like about the documentation? How can it be improved?

RETURN TO: MAZAK Corporation
Technical Publication Dept.
8025 Production Drive
Florence, Kentucky 41042

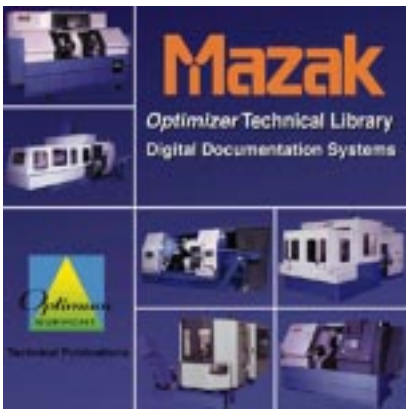
☐☐ ☐☐☐☐ ☐☐

SF-202Xa

Notes:

Section 1:

Introduction



Notes:

TABLE OF CONTENTS
(ABOUT THIS MANUAL)

Page

- MEMO -

1 INTRODUCTION

1-1 List of Related Manuals and Documents

For this model of machine, the following manuals and documents are relevant to this model of machine.

- Operating manual

  Describes operation procedure and other general information on operating the machine.
- Maintenance manual

  Describes the general information on maintenance including inspection and adjustment procedure.
- Programming manual

  Describes the NC programming procedure and related information.
- Tooling manual

  Gives the dimensional drawings of tool shank and retention bolt.
- Parameter list

  Describes the details of parameters.
- Alarm list

  Describes the details of alarms.
- Parts list

  Describes the details of parts and the procedure for purchasing the necessary parts.
- Electric circuit diagrams

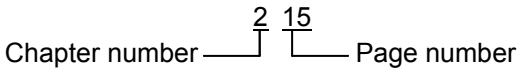
  Shows the wiring diagrams.
- Instruction manuals for optional equipment

  Describes operation procedure and other general information on the operation of the optional equipment.

1-2 Pagination

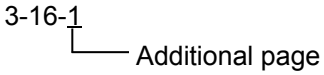
Page numbers are given as two numerals joined by a hyphen. The first number indicates chapter number and the second indicates the page number within that chapter.

Example 1:



If a change, which necessitates altering the number of pages, arises, additional pages will be inserted. The numbers of these pages will be made up by appending a hyphen and sequential numbers to the number of the page before the addition.

Example 2:

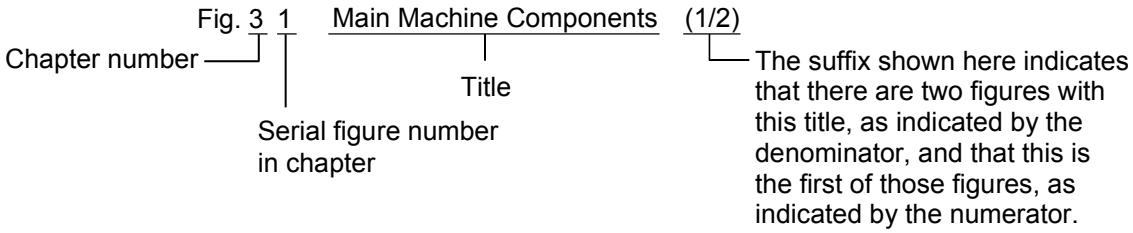


1-3 Numbering of Figures and Tables

1. Figure numbers

Figure numbers consist of the word “Fig.” followed by two numerals joined by a hyphen. The first number indicates the chapter number and the second indicate the serial number in that chapter. The figure number is followed by the title of the figure.

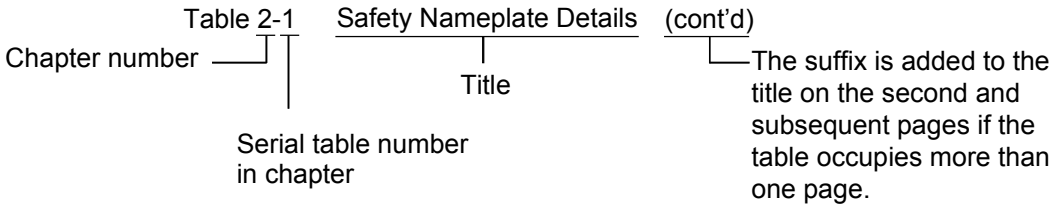
Example 1:



2. Table numbers

Table numbers consist of the word “Table” followed by two numerals joined by a hyphen. The first number indicates the chapter number and the second indicate the serial number in that chapter. The table number is followed by the title of the table.

Example 1:



Some of the figures and tables in this manual are not numbered.

2 SAFETY PRECAUTIONS

The precautions to be observed to ensure safe operation of the machine are described here.

The machine must not be operated until these precautions have been thoroughly understood.

2-1 Rule

1. This section contains the precautions to be observed as to the working methods and states usually expected. Of course, however, unexpected operations and/or unexpected working states may take place at the user site.

During daily operation of the machine, therefore, the user must pay extra careful attention to its own working safety as well as to observe the precautions described below.

2. The meanings of our safety precautions to DANGER, WARNING, and CAUTION are as follows:

DANGER : Failure to follow these instructions could result in loss of life.

WARNING : Failure to observe these instructions could result in serious harm to a human life or body.

CAUTION : Failure to observe these instructions could result in minor injuries or serious machine damage.

2-2 Basic Safety Items

DANGER

- There are high voltage terminals on the electrical control panel, transformer, motors, junction boxes, and other equipment (with a battery warning mark  attached).
- DO NOT touch any of them under any circumstances.
- Make sure that all doors and safety covers are fitted before switching on the power. If any door or safety cover must be removed, cut off and lock the main breaker first.

WARNING

- Memorize the position of the EMERGENCY STOP button so that you can press it immediately from any position.
- Take care not to touch any of the switches accidentally while the machine is in operation.
- Take care not to catch your fingers in the chuck. (For lathes)
- Never touch a rotating workpiece or the tool with your bare hands or any other object.
- To prevent incorrect operation of the machine, carefully check the position of the switches before operation.
- If in the slightest doubt about a procedure, ask the person in charge.
- Always switch off the main power switch on the operation panel and the main power breaker when you are taking a break from work and are going to leave the machine.
- If more than one operator is using the machine, do not proceed to the next step without indicating to the other operator(s) that you are about to do so.
- Do not modify the machine in any way that will affect its safety.
- Do not enter the machine. If you have to enter the machine, take measures such as pressing the EMERGENCY STOP button that disable machine operation.

CAUTION

- If there is a power failure, switch off the main power breaker immediately.
- Do not subject the NC unit, the operation panel, or the electrical control panel to strong shocks.
- Do not change the parameter values, volume values, or other electrical setting values without good reason. If you need to change a value, first check that it is safe to do so, then make a note of the original value so that it can be reset if necessary.
- Do not paint, soil, damage, modify, or remove any of the safety nameplates. If the details become illegible or the nameplate is lost, obtain a replacement by sending the part number stamped at the bottom right-hand corner of the nameplate to Mazak and mount the replacement in the original position. (If a nameplate has been lost, obtain the part number from the “Parts List” supplied separately.)
- Do not open any doors or safety covers while the machine is in automatic operation.
- Stop all machine operations before cleaning the machine or any of the peripheral equipment.
- After a job has been completed, set up each part of the machine so that it is ready for the next series of operations.

2-3 Clothing and Personal Safety

CAUTION

- Tie back long hair which could become entangled in drive mechanisms.
- Wear safety equipment (helmet, goggles, safety shoes, etc.) whenever possible.
- Always wear a helmet if there are any overhead obstacles in the work area.
- Always wear a protective mask when machining magnesium alloys.
- Always use safety shoes with steel toecaps and oil-resistant soles.
- Never wear loose or baggy clothing.
- Always completely fasten buttons and hooks on the arms of clothing to avoid the danger of entanglement in drive mechanisms.
- Always use gloves when loading or unloading workpieces or tools and when removing chips from the work area to protect your hands from sharp edges and heat generated during machining.
- Do not operate the machine while under the influence of drugs with powerful effects, unprescribed drugs, or alcohol.
- Do not operate the machine if you suffer from dizziness or fainting spells.

2-4 Operational Safety

WARNING

- Close all doors and covers of the NC unit, operation panel, electrical control panel, and junction boxes to prevent damage from water, chips, and oil.
- Check all electrical cables for damage to prevent accidents due to current leakage or electric shock.
- Do not handle coolant with bare hands since it is liable to cause irritation. Operators with allergies should take special precautions.
- Do not operate the machine during violent thunderstorms.
- Check the safety covers regularly to make sure that they are properly filled and that they are not damaged. Repair or replace any damaged covers immediately.
- Always obtain assistance in handling loads beyond your capacity.
- Do not operate fork lifts or cranes or carry out slinging work unless you have undergone officially approved training.
- When using fork lifts or cranes, make sure that there are no obstructions in the vicinity of these machines.
- Always use wire rope or slings of a standard suitable for the load to be supported.
- Wait until the machine has come to a complete stop before adjusting the position of the coolant nozzle.
- Do not remove or otherwise interfere with safety equipment such as stop dogs, limit switches, or interlocks in order to increase axis travel.
- Do not operate the machine with any of the safety covers removed.
- Always wear gloves and use a brush to clean chips off the tool tip; never use bare hands.
- Stop all machine operations before mounting or removing a tool.
- Check slings, chains, hoists, and other lifting gear for defects before use. Repair or replace defective gear immediately.

CAUTION

- Do not allow chips to accumulate during heavy-duty cutting since they will be hot and could start a fire.
- Warm up the spindle and all of the axis motion mechanisms before operation. (Operate the machine in the automatic mode for 10 to 20 minutes at maximum spindle speed or at 1/3 to 1/2 maximum axis motion speed.)
- Write the program for the warming-up operation so as to execute the functions of the full range of machine components. If operation is to be performed at spindle speeds in excess of 4000 min^{-1}, pay meticulous attention to the warming-up operation.
- Do not operate the operation panel switches while wearing gloves as this could lead to incorrect operation or other mistakes.
- When one series of operations is completed, switch off the power switch on the NC operation panel, the main power breaker, then the factory power supply.

2-5 Safety Considerations Relating to Holding Workpieces and Tooling

WARNING

- Always use tools which are suitable for the work to be carried out and conform to the machine specifications. Always observe the restrictions on weight of tool insert and tool diameter stipulated in the specifications. [Max. insert weight: 20 gf (0.04 lbs)]
- Replace tools promptly, since badly worn tools can cause accidents and injuries.
- Before starting operation, check that the workpiece is securely mounted and cannot become detached by the cutting forces applied to it during machining.
- Before mounting a cutting tool in the machine, make sure that the tool and its stud bolt are properly secured.
- Do not rotate a cutting tool or holder at speeds higher than the rated speed.
- Check the maximum safe operating speed with the manufacturer.
- Be very careful not to catch your fingers in workpiece holding devices.
- Always use the correct lifting gear for heavy fixtures and workpieces.

CAUTION

- Make sure that cutting tools will not interfere with the workpiece fixture, etc.
- Mount the tool holder in the correct orientation by aligning it with the drive key.
- Occasionally check for looseness after using a milling chuck.
- Make sure that the workpiece is correctly mounted on the pallet or table.
- Check that vices, fixtures, etc., have no overhang with respect to the table.
- Make sure that the tools are arranged in the tool magazine correctly in accordance with the tool data set in the TOOL LAYOUT display.
- Check that the lengths of all tools to be used have been measured.
- Move the tool nose to the initial point (X0, Y0) to check the basic coordinate system.
- Perform a test operation after mounting a cutting tool and workpiece.

2-6 Maintenance Safety

DANGER

- Always switch off the main power breaker and lock it before carrying out any maintenance work. This will eliminate the possibility of the machinery being accidentally started by someone else.
- After the power has been switched off for a short while, check the voltage with a multimeter or similar instrument to make sure that there is no residual voltage. Also discharge the capacitors.

WARNING

- Maintenance of electrical equipment must be carried out by qualified licensed electrical engineers only.
- Do not remove or modify limit switches, proximity switches, or other safety equipment.
- When carrying out maintenance in high places, always use a suitable ladder or a service platform, and always wear a helmet.
- Always switch off the main power breaker and lock it before replacing bulbs or other electrical equipment, and use products with the same specifications as the original.
- Do not start the machine until all of the covers removed for maintenance have been refitted.
- Wipe up any water or oil spills immediately, and keep the maintenance area and the workplace clean and tidy at all times.

CAUTION

- Maintenance work should be carried out by qualified personnel in accordance with the instructions of the person in charge.
- Keep a stock of replacement parts and consumable items.
- Make sure you fully understand how to maintain the machine.
- Do not use compressed air to clean the machine or to remove chips.
- Always use gloves when clearing away chips; never touch chips with bare hands.
- Use only the specified grades of hydraulic oil, lubricating oil, and grease or their equivalents
- When changing a fuse, check that the new fuse has the correct rating. (Using a fuse with too high a rating could result in damage to the equipment.)
- Follow the instructions on the instruction plate concerning the brands of oil to be used, lubrication points, amount to be used, and oil change intervals.
- Check the results of maintenance work in the presence of the person in charge.
- Stop all machine operation before cleaning the machine or the surrounding area.

2-7 Workplace Safety

WARNING

- Immediately remove all water and oil spills from the floor, and dry the floor to prevent accidents.
- Use strong service platforms only and make sure that nothing can slip off them.
- Keep combustible materials well away from the work area and any other place where there are hot chips.
- Always provide sufficient lighting in the work area.
- Always provide sufficient working space and clear access to the machine and peripheral equipment and store tools and other potential obstacles in a prescribed place away from the machine.
- Never place tools or other potentially dangerous objects on top of the headstock, turret tool-post or covers. (For lathes)
- Make sure that the nominal cross-sectional area of the power supply cable between the factory power supply switch and the machine main power breaker is such as to enable a stable supply for operation at the maximum output.
- Protect all cables which will run along the floor from being damaged by chips, since this could cause short-circuiting.

2-8 Safety Considerations Relating to Chip Conveyor

WARNING

Before attempting any kind of work related to the chip conveyor, make sure that the chip conveyor is in a complete stop state with power supply shut off. Ignoring this will cause serious injury.
While the chip conveyor is operating, never touch or extend your hands into any part of the chip conveyor. Ignoring this will cause your hands or fingers to be caught into the chip conveyor and injured seriously.

2-9 Safety Equipment

This machine is provided with various safety equipment to protect the operator and the machines.

The safety equipment include interlock devices and emergency stop switches, as well as doors and covers.

WARNING

- The machine is provided with various devices for the operator's safety.
  Never cancel any of the devices.
  Failure to observe this instruction could result in serious harm to a human life or body.

Arrangement of Safety Devices

No.	Name	Function
1	Operator's door interlock	1. If the operator's door is opened during automatic operation, the machine enters the feed hold state with the spindle stopped. 2. While the operator's door is open, spindle speed is restricted by the preset safety speed.
2	Magazine door interlock	1. If the magazine door is opened while the magazine is operating, the magazine stops operating. 2. While the magazine door is open, the magazine cannot be operated. 3. While the magazine door is open, the magazine stops and the axes stop movements.
3	Emergency stop buttons (2 places)	1. The emergency stop button is provided on each operation panel. 2. When the emergency stop button is pressed, the machine stops in the emergency stop state independent of the selected operation mode (automatic, manual).
4	Safety plate	1. A safety plate is fixed to the machine with screws to give operators and persons nearby warning information that standing near the movable area of equipment and devices is dangerous.

Safety device locations

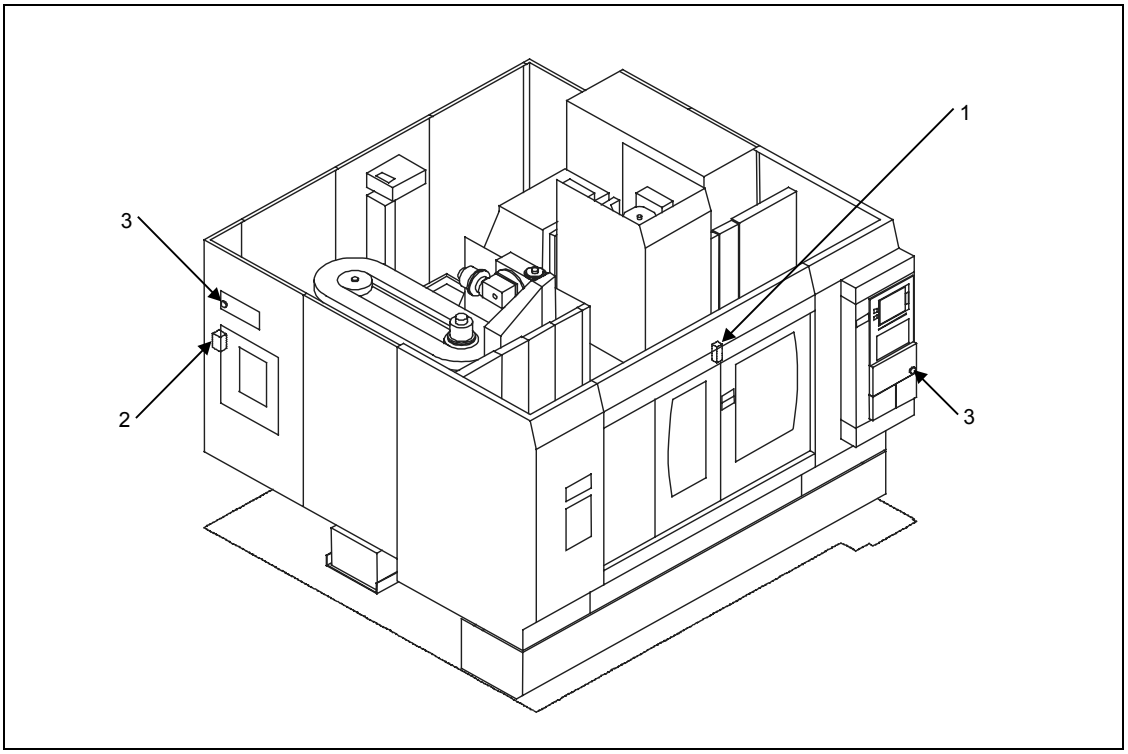


Fig. 2-1 Safety Devices

No.	Name
1	Operator's door interlock
2	Magazine door interlock
3	Pallet changer safety cover and door interlock (2 places)

2-10 Remarks on the Cutting Conditions Recommended by the NC

WARNING

- Before using the following cutting conditions:
  - Cutting conditions that are the result of the Mazatrol Automatic Cutting Conditions Determination Function
  - Cutting conditions suggested by the Machining Navigation Function
  - Cutting conditions for tools that are suggested to be used by the Machining Navigation Function

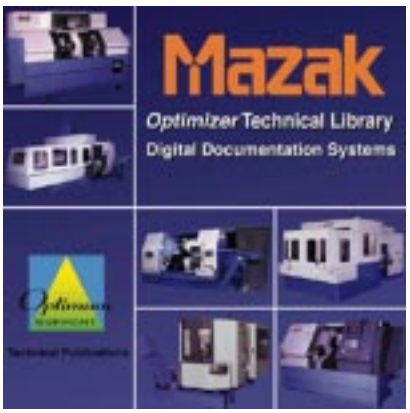
  Confirm that every necessary precaution in regards to safe machine setup has been taken – especially for workpiece fixturing/clamping and tool setup.
- Confirm that the machine door is securely closed before starting machining.
  Failure to confirm safe machine setup may result in serious injury or death.

2-11 Safety Nameplates

Safety nameplates are mounted on the machine to protect the operators and the machine from danger.

Section 2:

Machine Operation



Notes:

TABLE OF CONTENTS
(MACHINE OPERATION)

Page

- MEMO -

1 OUTLINE OF THE MACHINE

This chapter describes the definition of axes and arrangement of major units.

1-1 Axis Definitions

In the manuals supplied with the machine, the coordinate axes are defined as indicated below.

Note: The coordinate axes are defined assuming that an operator is standing in front of the machine.

X-axis	Saddle movements in the right-and-left direction Positive direction (+): Saddle movement to the right Negative direction (-): Saddle movement to the left
Y-axis	Column movements in the back-and-forth direction Positive direction (+): Column movement to the back Negative direction (-): Column movement to the front
Z-axis	Spindle head movements in the up-and-down direction Positive direction (+): Spindle head movement to the top Negative direction (-): Spindle head movement to the bottom

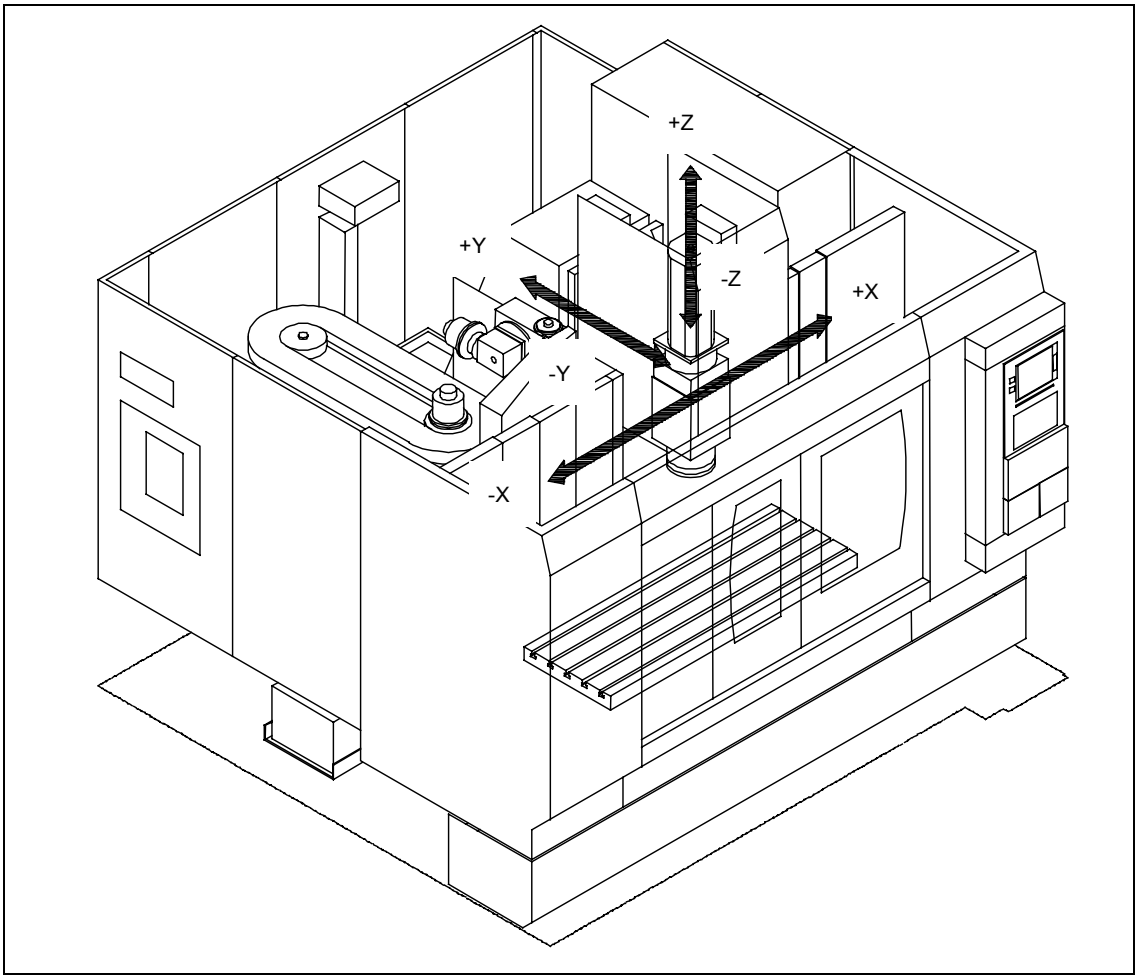


Fig. 1-1 Axis Definitions

1-2 Names and Locations of Machine Units

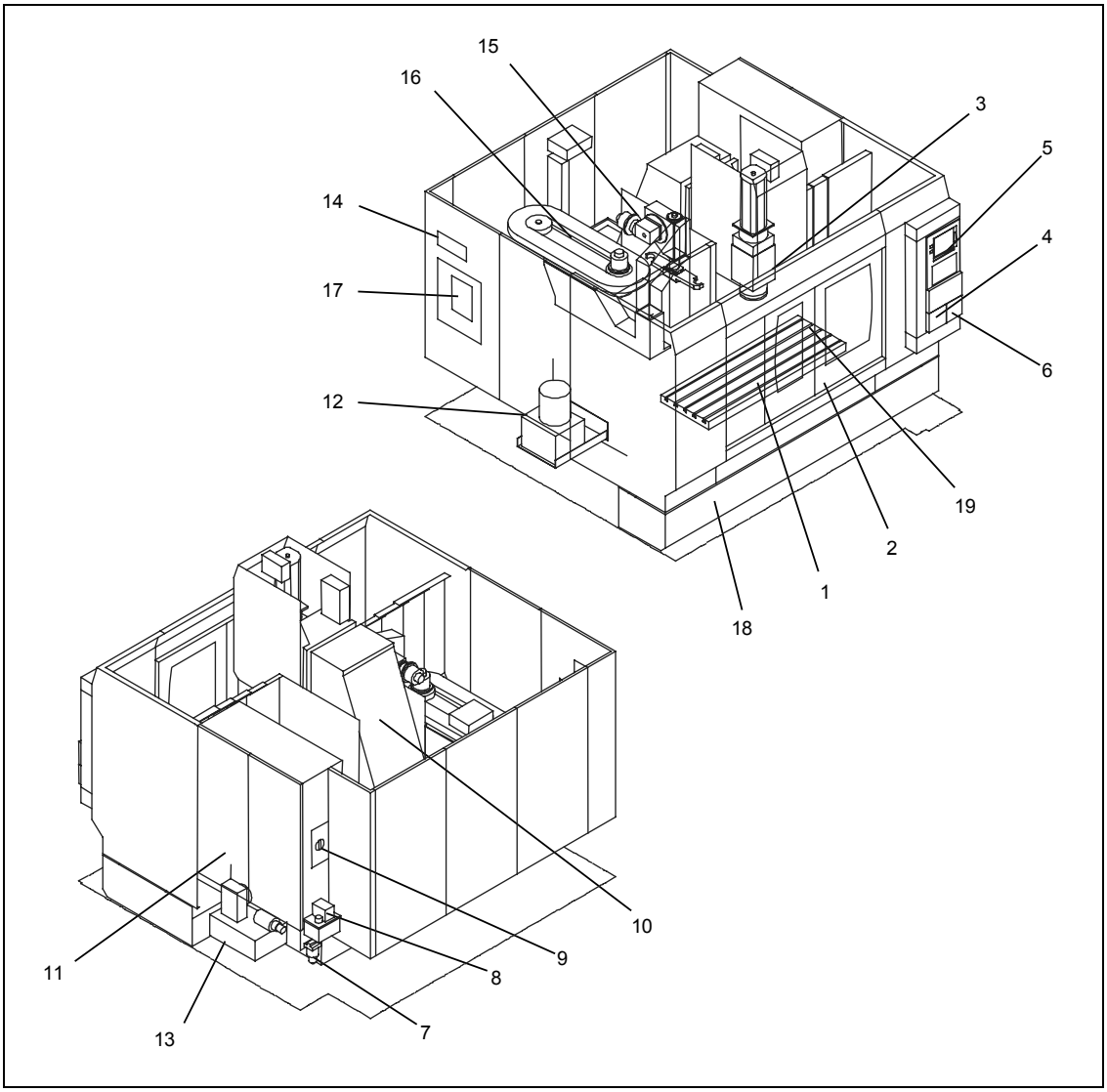


Fig. 1-2 Names and Locations of Machine Units

No.	Name	No.	Name
1	Table	11	Electrical control cabinet
2	Operator's door	12	Hydraulic unit
3	Spindle	13	Spindle cooling fluid unit
4	Auxiliary operation panel	14	Magazine operation panel
5	NC operation panel	15	ATC unit
6	Data input/output panel	16	Magazine
7	Air unit	17	Magazine door
8	Centralized lubrication unit	18	Coolant system
9	Main power disconnect	19	Tool measuring device
10	Column		

2 OPERATION PANELS

This chapter describes the arrangement and function of the switches, indicator lamps, and other controls.

2-1 Locations of Operation Panels

The machine has five operation panels as indicated below.
Refer to Fig. 2-1.

1. NC operation panel

Used to select the display screen, input the data and operation condition, and to perform other NC related operations.
The NC operation panel also has switches used to directly operate the machine in the manual mode.

2. Auxiliary operation panel

Has switches used for controlling the operation other than the operations controlled by the switches on the NC operation panel.

3. Data input/output panel

Used for connecting an external data input/output device.

4. Magazine operation panel

Used for mounting and removing a tool to and from the magazine.

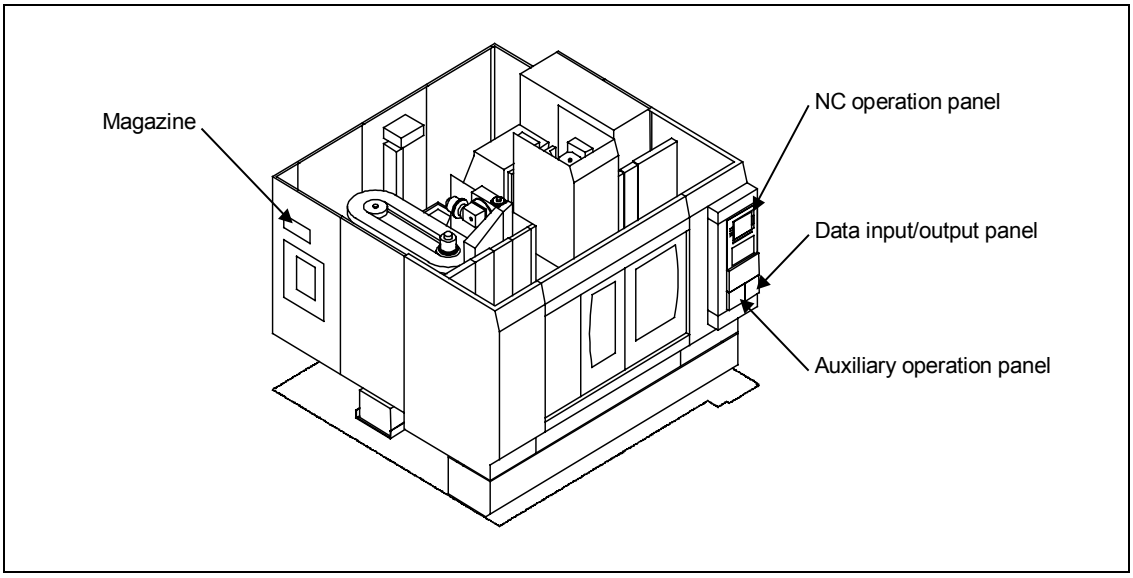


Fig. 2-1 Locations of Operation Panels

Cautions on using the operation panel

- The operation panel can be locked/unlocked using the stay provided at the lower section of the operation panel.
- When moving the operation panel, pay attention so that you will not get your fingers caught by the operation panel.
- When using the operation panel after pulling it out, lock it in the pulled out position. Do not try to move the operation panel while it is locked, otherwise the lock mechanism may be damaged.

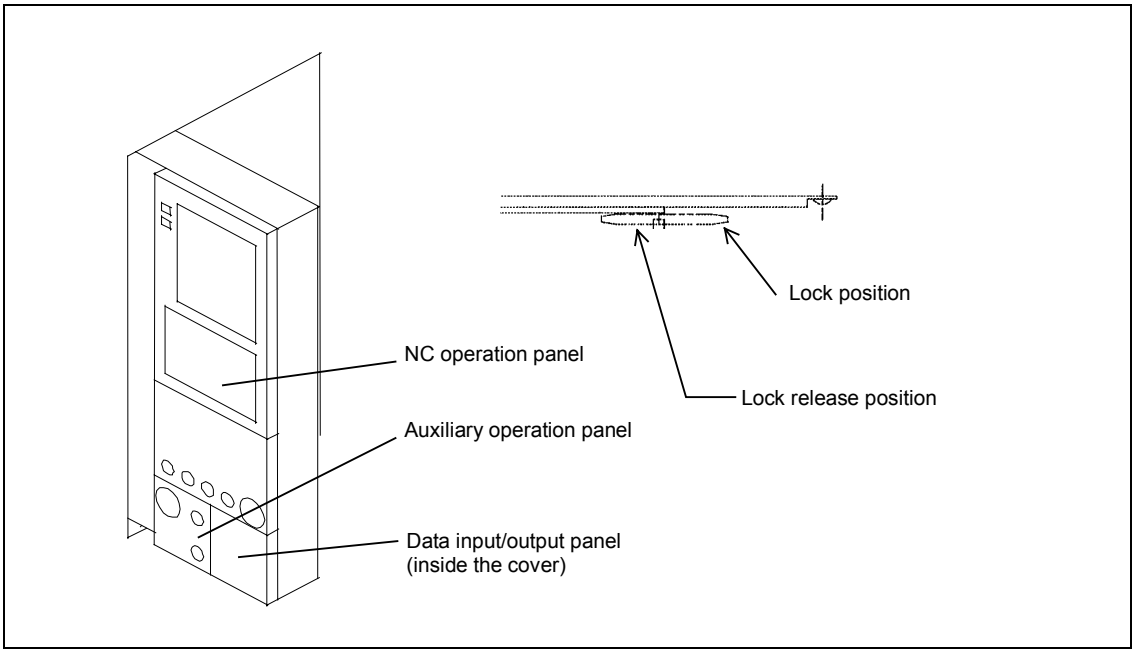


Fig. 2-2 Operation Panel

2-2 NC Operation Panel

WARNING

- Do not operate the switches on the NC operation panel with gloves on. If you operate the switches with gloves on, it could cause troubles such as malfunctioning of the machine.
- To prevent operation errors, make sure you press the switch you intend to press.

For details of NC operation, refer to Part 3 “OPERATING NC UNIT AND PREPARATION FOR AUTOMATIC OPERATION”.

2-2-1 Arrangement of Switches on the NC Operation Panel

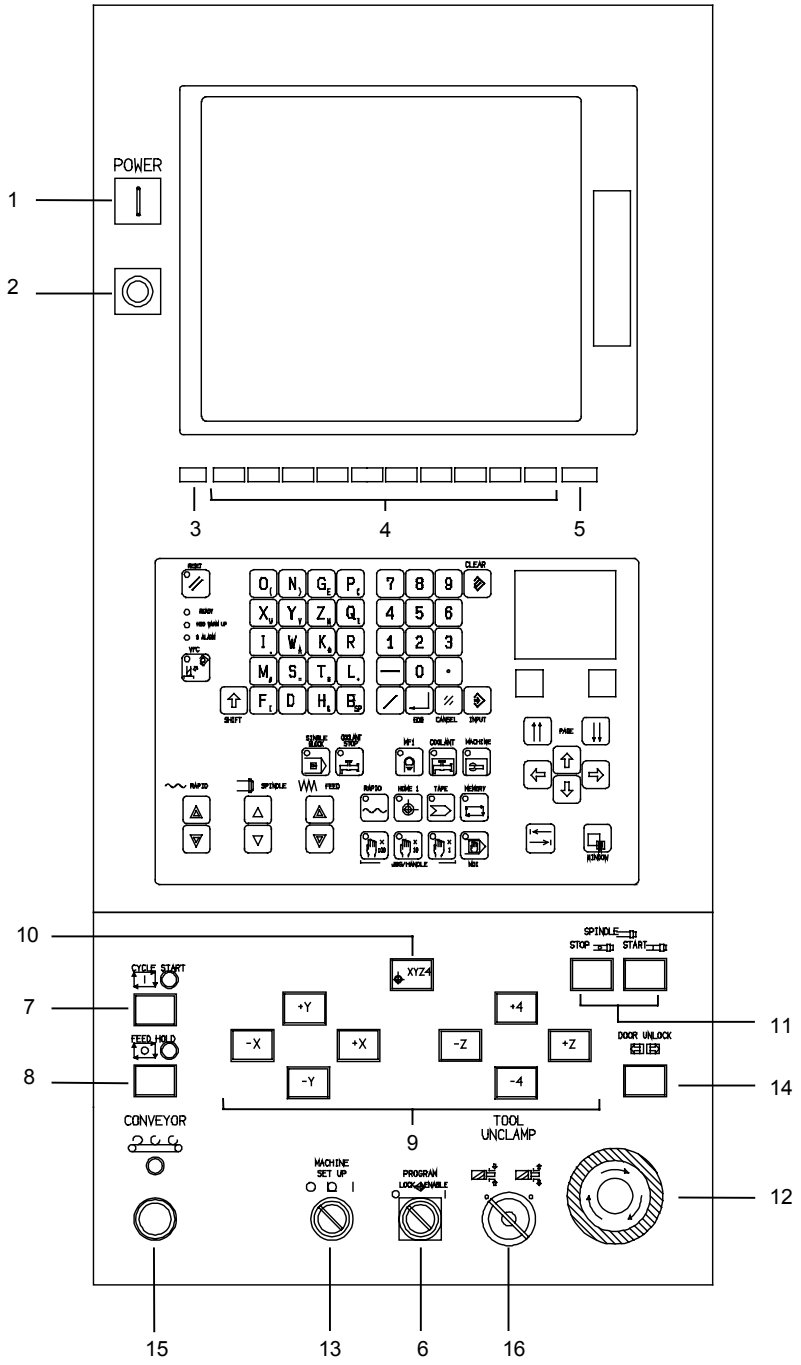


Fig. 2-3 NC Operation Panel

Membrane Keys on NC Operation Panel

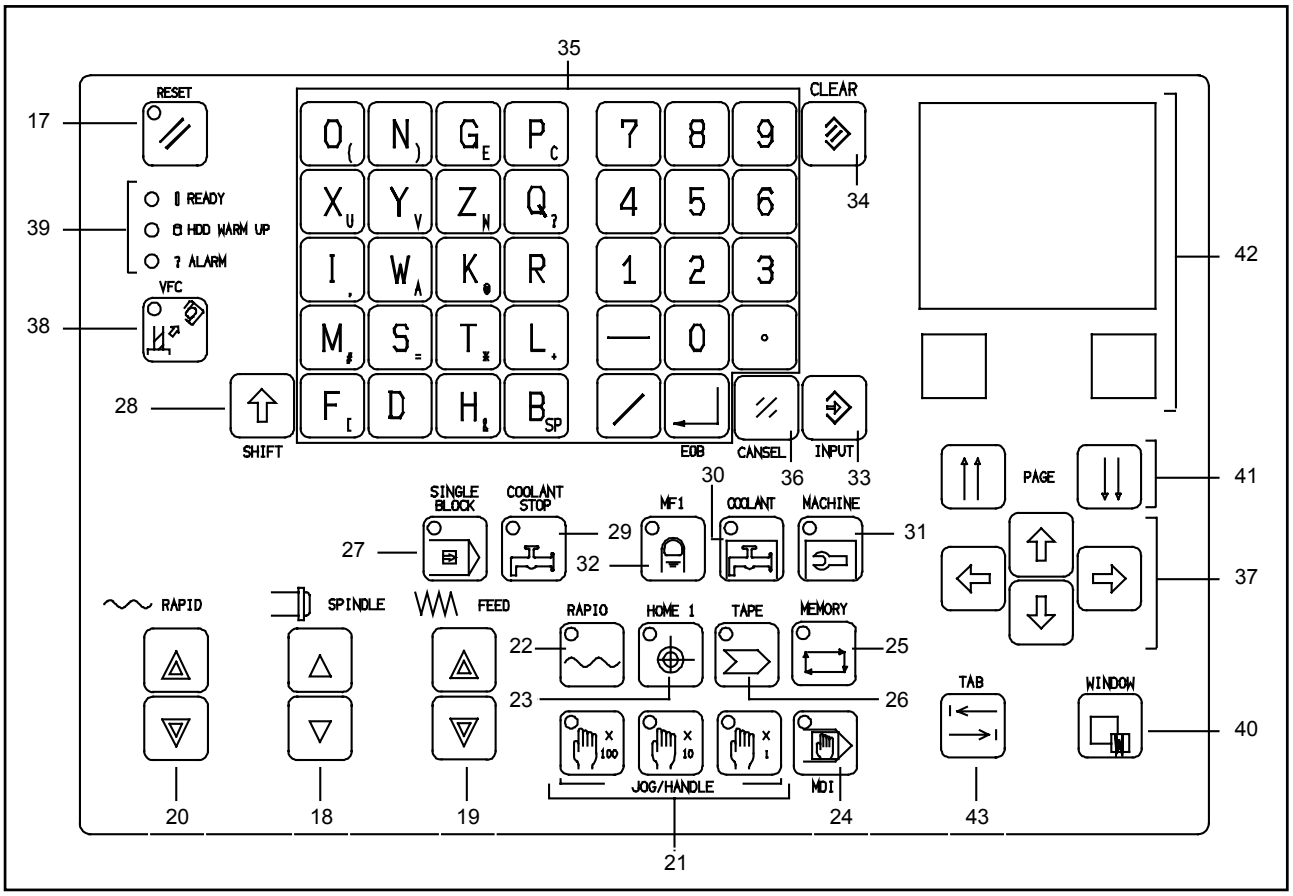


Fig. 2-4 Membrane Keys on NC Operation Panel

2-2-2 Names and Functions of Switches and Other Controls on the NC Operation Panel

Some of the keys on the NC operation panel have small indicator lamps on the top left area.

If the function of such a key is valid, the indicator lamp in the key is lit. If the indicator lamp is unlit, it indicates that the function is invalid.

Example: Indicator lamp ON Indicator lamp OFF

Also, some of the buttons on the NC operation panel have indicator lamps built into them. With such buttons, the indicator lamp lights when the button is pressed or the function of that button is valid. The indicator lamp is unlit when the function of the button is invalid.

Note: The power ON which has the built in indicator lamp differs from the switches with built in indicator lamp explained above.

Table 2-1 Names and Functions of the Switches on the NC Operation Panel

The numbers in the table below correspond to the numbers given in Figs. 2-3 and 2-4.

No.	Name	Function
1	Power ON button	• Used to switch on the power to the NC unit. When this button is pressed, the “READY” indicator lamp comes on several seconds after the pressing of it. The indicator lamp in this button comes on when the main power disconnect is switched to ON, indicating that the power is being supplied to the machine. • The indicator lamp goes off when this button is pressed.
2	Power OFF button	• Used to switch off the power to the NC unit. The indicator lamp in the power ON button comes on when the power OFF button is pressed.
3	Display selector key	• Used to select the screen to be displayed. When this key is pressed, the display selection menu is displayed.
4	Menu keys	• Used to select one of menu items being displayed.
5	Menu selector key	• Used to change the menu presently displayed over to another one.
6	Program protect switch (with key)	• Used to protect the programs and data stored in the NC memory. **I** : Enables writing of the program and data. **O** : Disables writing of the program and data.
7	CYCLE START button (green) CYCLE START indicator lamp (green)	• Used to start the machine in the automatic operation mode. While the machine is operating in the automatic mode, the CYCLE START indicator lamp is lit. The CYCLE START indicator goes off when the automatic operation is interrupted or has been completed.
8	FEED HOLD button (red) FEED HOLD indicator lamp (white)	• Used to suspend axis feed during automatic operation. While the axes are stopped in the feed hold mode, the FEED HOLD indicator lamp is lit. In this state, the CYCLE START indicator lamp is unlit.
9	Axis movement buttons	• Used to move an axis in the manual operation mode. The axis, selected by one of these buttons, keeps moving while the button is held pressed and stops when it is released.
10	All-axis zero-point return button	• Used to return all axes to the first zero-point in the manual operation mode. • The button is valid only after executing the zero-point return operation once. • Press this button after pressing the first zero-point return key (HOME 1), and the axes return to the first zero-point in the following sequence: the Z-axis first returns to the zero-point. Once the Z-axis movement has completed, the X- and Y-axis (and the 4th-axis if additional axis is used) return to the zero-point simultaneously.
11	STOP button 	• Used to stop the spindle rotation in the manual operation mode.
	START button 	• Used to start the spindle rotation in the manual operation mode. • The indicator lamp in the button lights while the spindle is rotating.
12	Emergency stop button	• Used to stop the machine in case of emergency. When this button is pressed, all machine operations are stopped immediately and alarm message “003 EMERGENCY STOP” is displayed. • To reset the emergency stop state, turn the button clockwise (indicated by the arrow symbols) and press the RESET key.
13	MACHINE SET UP switch	• Used to make the door interlock function (the function to prohibit opening of the door) valid or invalid.
14	DOOR UNLOCK button (option)	• Used to unlock the door.
15	CONVEYOR switch (option)	• Used to start and stop the chip conveyor.
16	TOOL UNCLAMP switch	• Used to clamp and unclamp a tool in the spindle in the manual operation mode.

No.	Name	Function
17	RESET key	• Used to reset the NC to the initial state. If an alarm is displayed, eliminate the cause of the alarm and press this key to clear the alarm display.
18	Spindle speed override keys	• Use to adjust the spindle speed. In manual operation: The keys are used to set the required spindle speed. The spindle speed is set in increments of “10” rpm. In automatic operation: The keys are used to change the spindle speed specified in the program in increments of 10% in the range from 0 to 150%. △ : Spindle speed increases. ▽ : Spindle speed decreases. The actual spindle speed is displayed on the screen.
19	Cutting feedrate override keys	• Used to adjust the feedrate. In manual operation: The keys are used to set the required feedrate. In automatic operation: The keys are used to change the feedrate specified in the program in increments of 10% in the range from 0 to 200%. ⧋ : Feedrate increases. ⧍ : Feedrate decreases. The actual spindle speed is displayed on the screen. NOTE: The term “override” means an operation in which spindle speed or feedrate specified in a program is changed to meet the actual cutting conditions.
20	Rapid traverse override keys	• Used to adjust the rapid traverse rate. The override value, set in terms of the percent to the available maximum rapid traverse rate, is displayed on the screen. ⧋ : Override value increases. ⧍ : Override value decreases.
21	Pulse handle feed multiplication ratio setting keys	• Used to select the pulse handle feed mode and the pulse weight (distance to be moved per pulse).  : Selects 0.001 mm (0.001° for the 4th-axis) per division of the pulse handle.  : Selects 0.01 mm (0.01° for the 4th-axis) per division of the pulse handle.  : Selects 0.1 mm (not selectable for the 4th-axis) per division of the pulse handle. • To move an axis by using the pulse handle, press any of these keys and select the axis to be moved with the AXIS SELECT switch. After that turn the pulse handle and the selected axis moves. • If the AXIS SELECT switch is set in the “OFF” position with any of these keys pressed, an axis moves precisely the set distance each time an axis movement button is pressed (step feed mode).
22	RAPID traverse key	• Used to select the rapid traverse mode. If an axis movement button is pressed after pressing this key, the selected axis moves at a rapid traverse rate.
23	HOME 1 key	• Used to return an axis to the first zero-point (machine origin) in the manual operation mode. • Press an axis movement button after pressing this key, and the selected axis returns to the first zero-point.

No.	Name	Function
24	MDI key	• Used to select the MDI mode. Press this key to automatically execute the manually set data.
25	MEMORY key	• Used to select the memory operation mode. Press this key to execute a program stored in the NC memory.
26	TAPE key	• Used to select the tape operation mode. Press this key to execute a program punched on paper tape, stored in an external device such as micro disk. NOTE: This mode of operation is selectable only for EIA/ISO programs.
27	SINGLE BLOCK key	• Used to select the single block mode. Press this key to select the single block mode in which a program is executed block by block.
28	SHIFT key	• Used to enter the address data indicated at the bottom right of an address key.
29	COOLANT STOP key	• Used to switch on and off coolant supply.
30	COOLANT menu key	• Used to display the coolant menu on the screen. Coolant menu is displayed on the screen when the COOLANT menu key is pressed.
31	MACHINE menu key	• Used to display the menu of various machine operations. The machine operation menu is displayed when the MACHINE menu key is pressed.
32	MF1 (machine function) key	• Used as an auxiliary key meeting the selected specifications.
33	INPUT key	• Used to set the data entered and displayed in the data input area. This key must be pressed after entering the data with the data keys.
34	CLEAR key	• Used to cancel the data displayed in the data input area. Press this key if incorrect data has been entered to the data input area. • This is also used to release the machine alarm.
35	Address/numeric data keys	• Used to enter the address data (alphabetic characters) and numeric data.
36	CANCEL key	• Used to cancel the data displayed in the data display area. • Press this key if incorrect data has been set by mistake or to delete the previously set data.
37	Cursor keys	• Used to call the cursor on the screen as well as to move the cursor on the screen. How the cursor moves in response to the pressing of each of the cursor keys is indicated below:  : The cursor moves to the left in the same line. If this key is pressed while the cursor is positioned at the left end, the cursor moves to the right end of the preceding line.  : The cursor moves to the right in the same line. If this key is pressed while the cursor is positioned at the right end, the cursor moves to the left end of the next line.  : The cursor moves up to the preceding line.  : The cursor moves down to the next line.
38	VFC key	• Used update the cutting conditions (surface speed, feedrate) in a program to the values adjusted using the override keys. When this key is pressed after adjusting the spindle speed and/or feedrate during automatic operation by using the override keys, the value in the program are updated to the adjusted value. NOTE: This key is valid only for MAZATROL programs.

No.	Name		Function
39	Machine Status Indicator Lamps ○ READY ○ HDD WARM UP ○ ALARM	READY indicator lamp	• Comes on when the CNC gets ready for controlling the machine.
		HDD WARM UP indicator lamp	• Comes on when the hard disk is warmed by the heater. If ambient temperature is low and the hard disk cannot start correctly, it is warmed by the heater.
		ALARM indicator lamp	• Comes on when an alarm occurred. If an alarm is detected by the CNC, the contents of the detected alarm are displayed on the screen and, at the same time, this indicator lamp lights.
40	WINDOW key		• Used to display a window on the screen.
41	Page keys		• Used to call the previous or next page of a display that consists of more than one page. : The previous page is called. : The next page is called. These keys have different function on the MAZATROL program screen: : The cursor moves to the beginning of the previous unit. : The cursor moves to the beginning of the next unit.
42	Tablet pointer		• Used instead of a mouse or a track ball.
		Pad	Cursor operation is possible by moving your finger on the pad.
		Left button key	Provides the same function as a mouse left button.
		Right button key	Provides the same function as a mouse right button.
43	TAB key		• Provides the same function as a TAB key in a keyboard.

2-3 Auxiliary Operation Panel

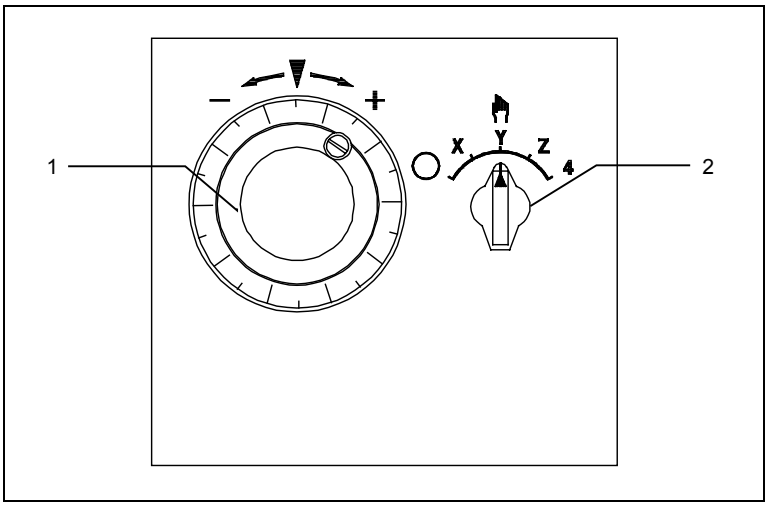


Fig. 2-5 Auxiliary Operation Panel

Table 2-2 Names and Functions of the Switches on the Auxiliary Operation Panel

No.	Name	Function
1	Pulse handle	Used to move the axis selected with the AXIS SELECT switch. Axis movement distance is determined by the rotation amount of the pulse handle.
2	AXIS SELECT switch	Used to select an axis to be operated by using the pulse handle.

2-4 Data Input/Output Panel

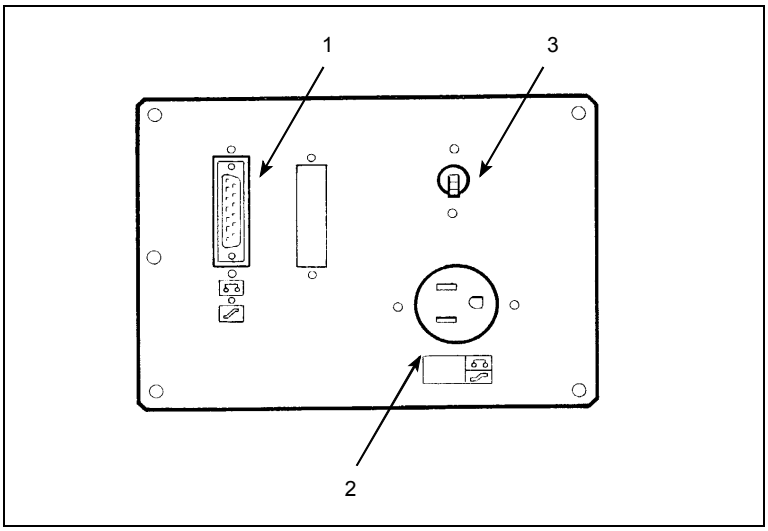


Fig. 2-6 Data Input/Output Panel

Table 2-3 Names and Functions of Connectors on the Data Input/Output Panel

No.	Name	Function
1	Interface	An interface used to connect an external device such as FDD (floppy disk drive), PTP/PTR, and printer.
2	Service outlet (100 V AC)	A service outlet supplying 100 V AC. Do not connect a device whose rating is larger than 2 A.
3	No-fuse breaker	A circuit interrupter having 2 A capacity. This breaker is used to shut off the circuit of 100 V AC service outlet.

2-5 Magazine Operation Panel

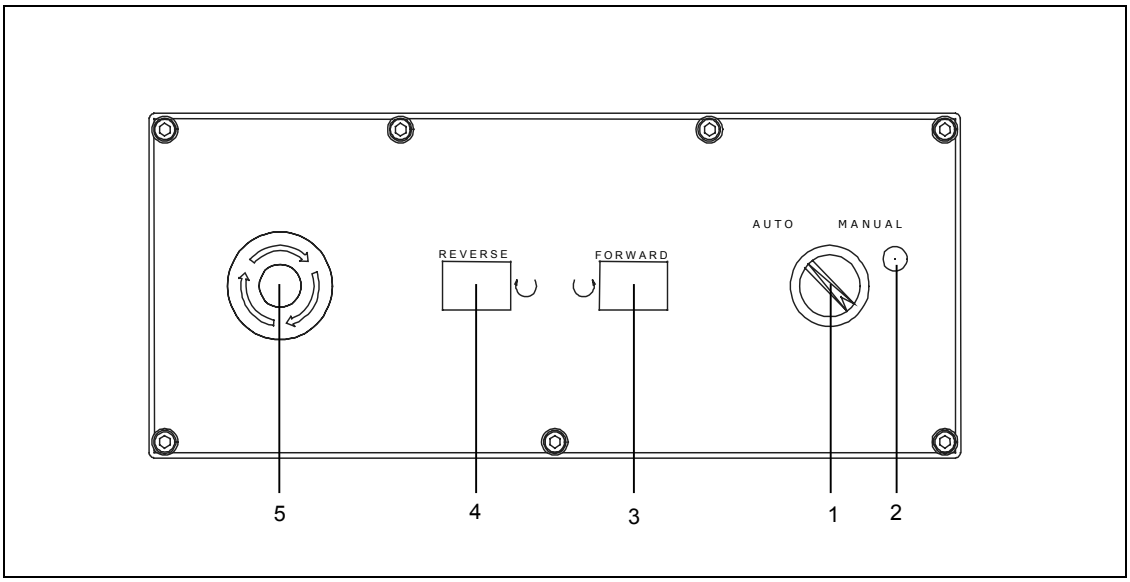


Fig. 2-7 Magazine Operation Panel

Table 2-4 Names and Functions of Switches on the Magazine Operation Panel

No.	Name	Function
1	TOOL SELECT switch	Used to enable manual magazine rotation operation (manual magazine operation interruption).
2	MANUAL indicator lamp	Goes on when the TOOL SELECT switch is placed in the “MANUAL” position. If the machine is in the standby state (manual magazine operation interruption), the indicator lamp flashes.
3	FORWARD button	Used to rotate the magazine in the forward direction. The magazine rotates continuously if the button is held pressed. The button is valid only when the following conditions are satisfied: 1. The TOOL SELECT switch is in the “MANUAL” position. 2. The MANUAL indicator lamp is lit or flashing. 3. The magazine door is closed.
4	REVERSE button	Used to rotate the magazine in the reverse direction. With an exception of the direction in which the magazine rotates, the function and the conditions in which the button is valid are the same as the FORWARD button.
5	EMERGENCY STOP button	Used to stop the machine immediately in an emergency. Press this button if incorrect operation has been attempted, abnormal noise is observed, collision is going to take place, or in other emergency cases.

2-6 Main Power Disconnect Switch

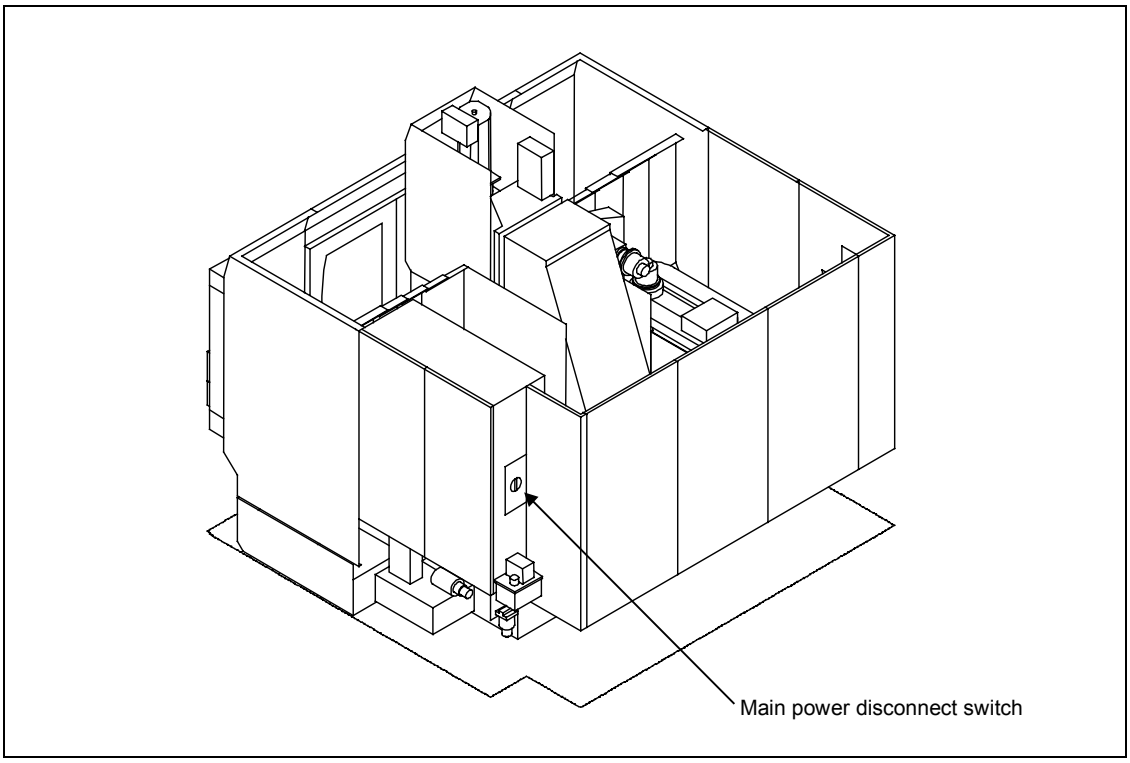


Fig. 2-8 Location of Main Power Disconnect Switch

The functions of the main power disconnect switch are indicated below.

- The power is supplied to the machine via the main power disconnect switch. Therefore, this switch shuts off power supply to the machine if the switch is set to the OFF position.
- In daily operation, this switch should be turned ON first to enable machine operation and turned OFF last to completely stop the machine.
- The switch will be tripped if excessive load current flows. To reset the switch after tripping, turn the switch lever into the RESET position and then back to the ON position.

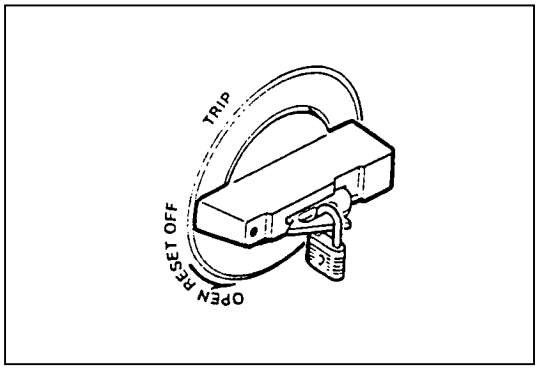


Fig. 2-9 Main Power Disconnect Switch

- The switch has the holes to allow installation of a padlock when the switch is in the OFF position.

WARNING

- When maintenance work must be carried out in the electrical control cabinet or when repairing the machine, be sure to set the main power disconnect switch in the OFF position and lock it in that position with the padlock

2-7 Operator's Door Interlock

2-7-1 MACHINE SET UP Switch (Key-lock Type Switch)

This switch is provided for set up work and test run of a new program. During daily production operation, place the switch in the "**O**" position and remove the key. A responsible person must keep the key.

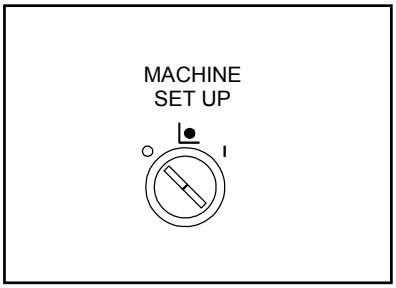


Fig. 2-10 MACHINE SET UP Switch

1. Operator's door interlock specification

Table 2-5 Operator's Door Interlock Specification

Door State	Operation Mode	SET UP Switch Position	
		O	**I**
Open	Manual	The following operations are disabled: - Spindle rotation - ATC operation If an attempt is made to carry out any of these operations, an alarm occurs and the following alarm message is displayed: 232 MACHINE DOOR INTERLOCK NOTE: Axis rapid traverse is permitted at a safe speed.	If an attempt is made to operate the machine using ATC menu keys, an alarm occurs and the following alarm message is displayed: 232 MACHINE DOOR INTERLOCK NOTE: Axis rapid traverse is permitted at a safe speed.
	Automatic	The machine is placed in the feed hold state and the cycle start of the program is not allowed. If the CYCLE START switch is pressed, an alarm occurs and the following alarm message is displayed: 232 MACHINE DOOR INTERLOCK	If the SET UP switch is in the " **I** " position, operation is executed in the single block mode. NOTE: Axis rapid traverse is permitted at a safe speed. The single block mode can be canceled by pressing the CLEAR key after closing the door.
Close ↓ Open	Manual	If the operator's door is opened during spindle rotation or ATC cycle, the operation in execution is stopped and the following alarm message is displayed: 232 MACHINE DOOR INTERLOCK NOTE: Axis rapid traverse is permitted at a safe speed.	If the operator's door is opened during ATC cycle, this operation is stopped. NOTE: Axis rapid traverse is permitted at a safe speed.
	Automatic	If the operator's door is opened during spindle rotation or ATC cycle, the operation in execution is stopped. The axis movement is placed in the feed hold state, and the following alarm message is displayed: 232 MACHINE DOOR INTERLOCK	If the operator's door is opened during ATC cycle, this operation is stopped. Single block mode is permitted at a safe speed. The single block mode can be canceled by pressing the CLEAR key after closing the door.
Close	Manual	There are no restrictions on operation due to the door interlock function.	
	Automatic		

2. Actions to be taken after an occurrence of door interlock function related alarm

To continue the operation after the occurrence of an alarm caused by opening the operator's door, close the door and clear the alarm message. The procedure to be followed to continue the interrupted operation is indicated below.

- In automatic mode:
  Press the cycle start switch.
- In manual mode:
  Complete the interrupted operation by using the menu key or appropriate operation switch.

If the operation has stopped during ATC cycle, complete the cycle using the procedure indicated above.

If it is not necessary to continue the operation after the occurrence of an alarm, simply press the reset key to clear the alarm message.

WARNING

- Before carrying out setup, make sure that the operator's door is closed.
- In operation while the operator's door is open, pay sufficient care to avoid the following:
  - To be hit by scattering chips
  - To be caught or crushed by moving unit
  - To touch rotating tool

3 MANUAL OPERATION

This chapter describes the items to be checked before starting the operation and also manual operation procedure.

3-1 Inspection before Starting Operation

Refer to Fig. 3-1.

＜Procedure＞

(1) Check the oil level for the oil tanks indicated below by the oil level gauges. If the oil level is lower than the specified level, add the specified oil.

For the procedure used to add the oil, refer to the Maintenance Manual.

- Centralized lubrication unit oil tank
- Hydraulic unit oil tank
- Spindle cooling fluid tank
- Coolant tank

(2) Check that the doors and covers indicated below are securely closed.

- Operator's door

  Close the door so that there is no gap.

- Control panel door

  Close the door and secure it in the closed position with the bolt.

- Magazine door

  Lock the door with the handle so that the door cannot be opened.

(3) Make sure that all safety covers and safety devices are installed properly. For the safety covers and safety devices, refer to Part 1, Section 2-9 "Safety Equipment".

(4) Make sure that the compressed air is supplied to the machine. Also make sure that the pressure gauge at the air inlet port indicates 5 kgf/cm^2 (71.1 PSI). If the air pressure is lower than 5 kgf/cm^2 (71.1 PSI), the air source capacity must be increased. If the air pressure is lower than 4 kgf/cm^2 (56.9 PSI), never operate the machine since malfunctions will occur.

(5) Make sure that the pressure gauge of the hydraulic unit indicates 0 kgf/cm^2 (0 PSI).

If the pressure gauge indication is not 0 kgf/cm^2 (0 PSI) while it is stopped, the pressure gauge will be faulty. In this case, it is necessary to change the pressure gauge. For the procedure of changing the pressure gauge, refer to the Maintenance Manual supplied separately.

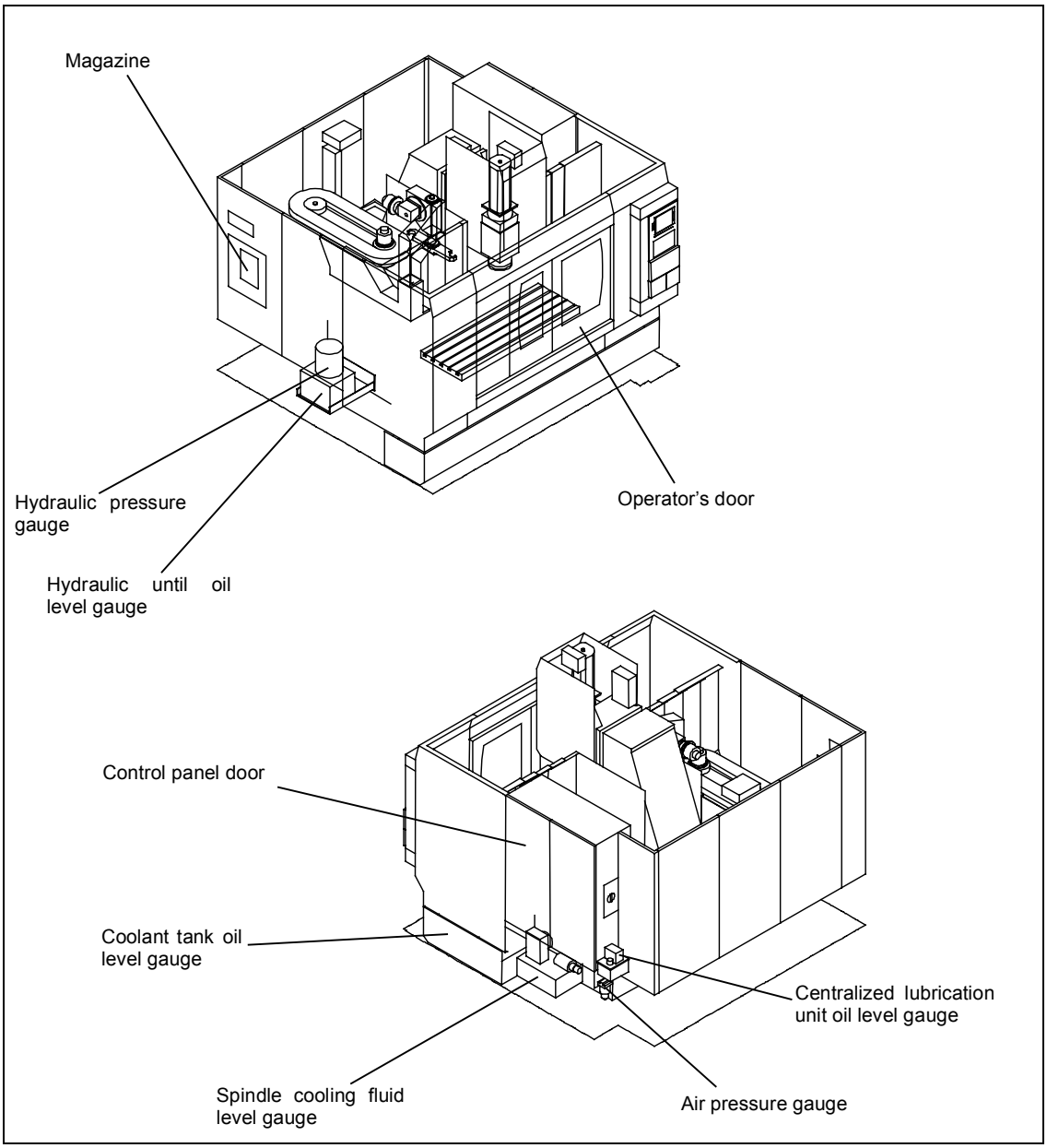


Fig. 3-1 Inspection before Starting Operation

3-2 Switching the Power ON

DANGER

- Cables or wires with damaged insulation could cause current leakage or electrical shocks. Always check that the insulation is sound.
- There are high voltage terminals in the electrical control cabinet, transformers, motors, relay boxes, etc. Never touch these terminals.
- Do not touch the switches with wet hands.

WARNING

- In order to prevent the entry of water, oil, chips, dust, and other foreign matter into the electrical control cabinet and other machine units, make sure to close the doors and covers of the CNC unit and the electrical control cabinet.

CAUTION

- If the instructions on the factory power distribution board are given on such as a safety plate, follow these instructions.

＜Procedure＞

(1) Switch the power ON at the factory power distribution board.

(2) Turn the main power disconnect to the ON position.

→ The fan starts rotating and the  (POWER ON) button on the NC operation panel lights.

(3) Press the  (POWER ON) button on the NC operation panel.

→ The indicator lamp in the  (POWER ON) button goes off.

→ The READY lamp, which indicates that the machine is ready for operation, lights.

→ The POSITION display is given.

→ The 

(first zero-point return) key lights and the following message is displayed.
ZERO RETURN

* For the machine equipped with the absolute position detection specification (option), the 

(memory operation) key lights.

→ The hydraulic unit pump and the centralized lubrication unit pump start.

If an abnormality is found when the hydraulic unit starts, press the  (POWER OFF) button or the emergency stop button. The hydraulic unit stops operating.

If the “EMERGENCY STOP” message is displayed in red at the lower left area of the POSITION screen when the power is switched on, it indicates that one of the emergency stop buttons is locked in the activated position. In this case, clear the emergency stop status and unlock the emergency stop button.

To reset the emergency stop button, turn the emergency stop button in the direction indicated by the arrow symbols given in it. When it is turned approximately 90 degrees, the button comes out to be unlocked.

After that press the 

(RESET) key on the NC operation panel, and the red “EMERGENCY STOP” message is cleared. The hydraulic unit starts operating again.

Note: Before switching on the power for the first time after installation, carry out the inspection that is indicated in the Maintenance Manual.

3-3 Inspection after Switching the Power ON

Refer to Fig. 3-2.

After switching the power ON as described in Section 3-2 “Switching the Power ON”, carry out the following inspection.

- Make sure that the “READY” indicator lamp is lit.
- Make sure that the POSITION screen is displayed.
- Make sure the hydraulic unit pressure and the air pressure regulated by the air unit are correct.

  A. Hydraulic pressure: 70 kgf/cm^2 (996 PSI)

  B. Air pressure: 5 kgf/cm^2 (71.1 PSI)

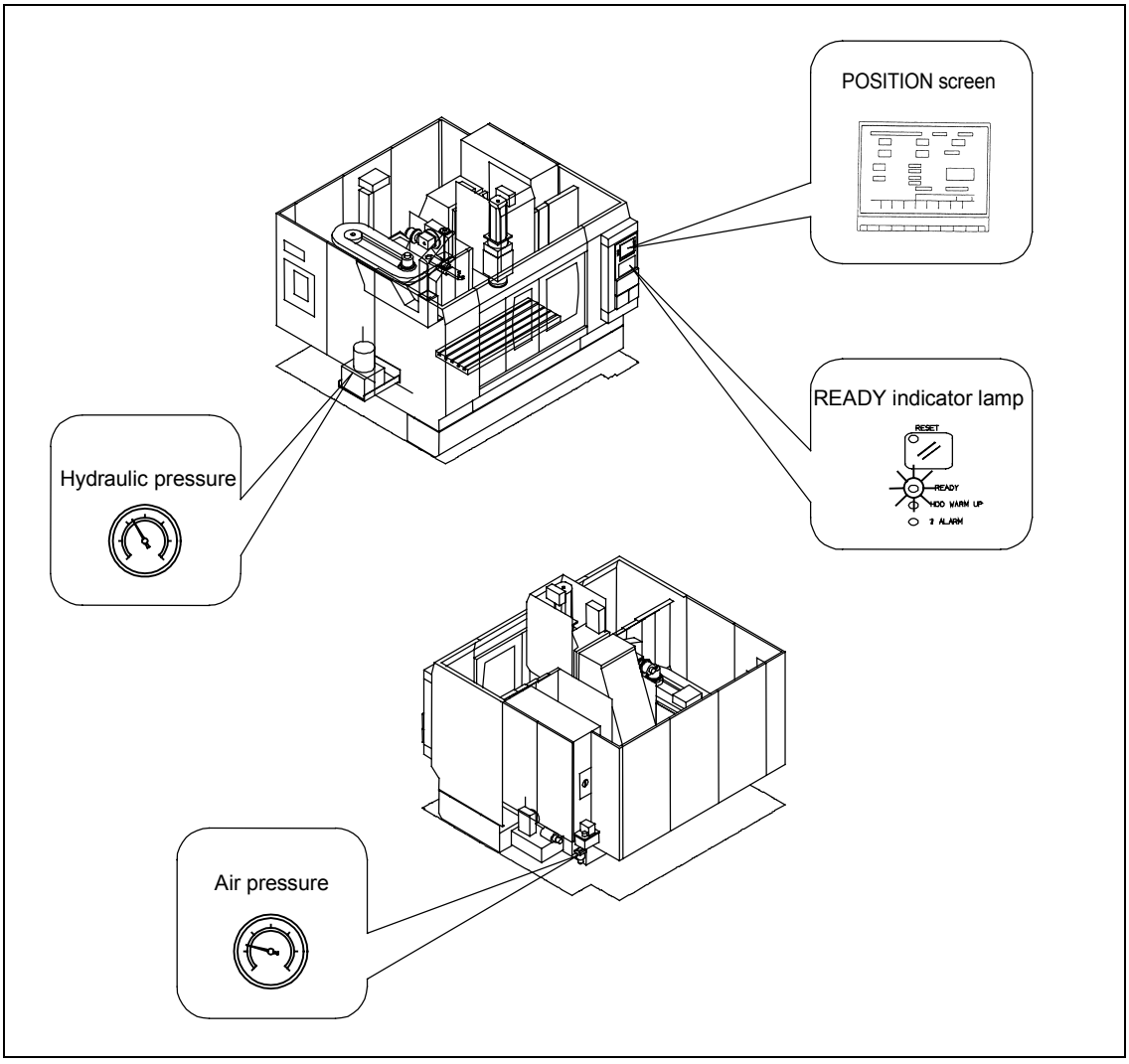


Fig. 3-2 Inspection after Switching the Power ON

3-4 Switching the Power OFF

After finishing day's operation, follow the procedure indicated below to shut off the power.

DANGER

- There are high voltage terminals in the electrical control cabinet transformers, motors, relay boxes, etc. Never touch these terminals.
- Do not touch the switches with wet hands.

＜Procedure＞

(1) Make sure that all machine operations (program operation, input/output with external devices, etc.) have been completed.

(2) Press the  (POWER OFF) button on the NC operation panel.

→ The indicator lamp in the  (POWER ON) button lights.

(3) Turn the main disconnect to the OFF position after the screen display has been turned off completely.

Note: If the main disconnect is turned OFF before the screen display is completely turned off, correct NC functions cannot be guaranteed. It will take about one minute before the screen display is turned off completely.

(4) Shut off the power at the factory power distribution board.

* If the machine is equipped with the automatic power off function (option), refer to the manual prepared separately for the option specifications.

3-5 Zero-Point Return

After switching on the power, it is necessary to carry out the zero-point return operation first to register the machine zero point to the NC.

The stored machine zero point is lost when the power is switched off.

Carry out zero-point return operation for all axes (X-, Y-, Z-, and 4th-axis (option)) to the first zero point.

WARNING

- Do not press the switches while wearing gloves. This could cause machining malfunction and other troubles.
- To prevent machine malfunction, confirm the function of the switches before using them.

CAUTION

- With the machine equipped with the optional absolute position detection function, the machine zero-point data is not lost even when the power is switched off. Accordingly, it is not necessary to carry out the zero-point return operation.
- If battery voltage is lowered, the machine zero-point data may be lost. If the "ZERO RETURN" message is displayed, the machine zero-point data is lost. In this case, carry out the zero-point return operation.

The "ZERO RETURN" message is displayed in green at the lower right area of the POSITION screen when the power is switched on. Carry out the zero-point return operation for the X-, Y-, Z- and 4th-axis (option) by pressing the axis movement button on the NC operation panel.

The procedure for the first zero-point return is indicated below.

＜Procedure＞

(1) Press the 

(first zero-point return) key.
Note that immediately after the power is switched on, the key is in the pressed state. The green lamp lights to indicate that the key has been pressed.

(2) If an axis is close to the zero-point, first momentarily press the -X, -Y, -Z or -4th (optional) key corresponding the axis to be returned to the zero-point.

→ The axis moves in the negative direction and then moves in the positive direction to the zero-point. On completion of the zero-point return cycle, the green zero-point return indicator lamp on the NC operation panel will light. Refer to Fig. 2-3.

Once an axis starts moving in the positive direction, the axis will continue moving to the zero-point even if the axis movement key is released. The axis moves to the zero-point at the zero-point approach speed (0.2 m/min (11.8 IPM)). Refer to Fig. 3-3.

(3) If an axis is not close to the zero-point, momentarily press the -X, -Y, -Z or -4th (optional) key corresponding the axis to be returned to the zero-point first. Next, hold the +X, +Y, +Z, or +4th (optional) key corresponding the axis to be returned to the zero-point pressed. Refer to Fig. 2-3.

- The axis will stop moving if the key is released.
- The key can be released after the axis feedrate is decelerated to the zero-point approach speed (0.2 m/min (11.8 IPM)). The axis continues moving to the zero-point even if the key is released. Refer to Remark 3.

Remark 1: If an axis is moved beyond the travel range by mistake, the soft-limit function operates to stop the axis. If this occurs, the soft-limit alarm indication is displayed at the lower left area in the screen. The procedure to be followed for clearing the soft-limit alarm is explained in section 3-8-4 “Clearing the Soft-Limit Alarm”.

Remark 2: For the machine equipped with the optional absolute position detection function, the first zero-point return operation is not necessary. In this case, the “ZERO RETURN” message is not displayed after switching on power.

Remark 3: Zero-point return speed

In the zero-point return cycle, axis feedrates are controlled in the manner as shown in Fig. 3-3 below.

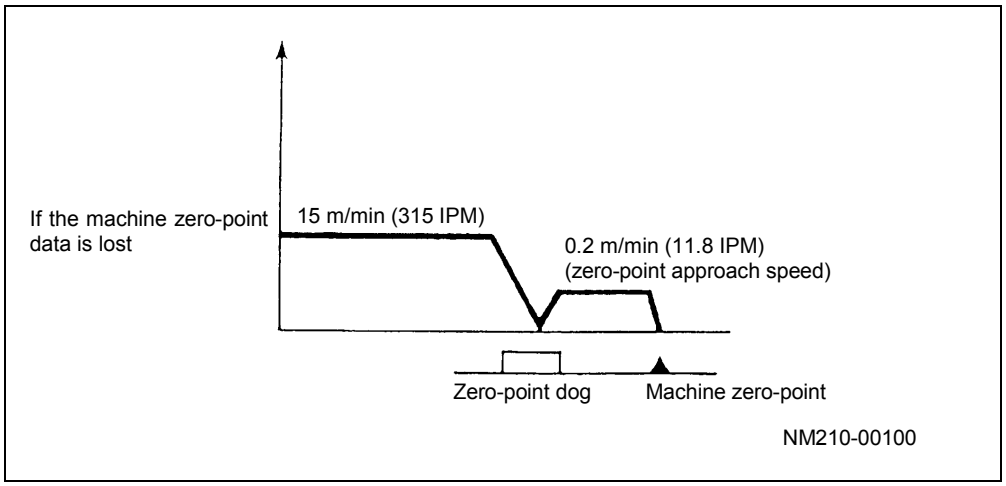


Fig. 3-3 Zero-point Return Cycle (If the machine zero-point data is lost)

- If the machine zero-point data is retained:
  The axis moves to the zero-point at 15 m/min (315 IPM) (*1)
- If the machine zero-point data is lost:

Before the detection of machine zero-point dog:
The axis moves at 15 m/min (315 IPM) (*1)

After the detection of machine zero-point dog:
The axis moves at 0.2 m/min (11.8 IPM) (zero point approach speed) (*2)

*1 : The value set for machine parameter M2.

*2 : The value set for machine parameter N8.

3-6 Warmup Operation

When starting the machine operation after it has been stopped for a long period, it is necessary to warm up the machine in the schedule as indicated below.

Since the machine has not been operated for a long period, the slideways are not lubricated appropriately which will cause stick-slip in axis movement as well as abnormal wear of the slideways if production is started immediately. In addition, thermal expansion must be taken into consideration. At the beginning of machine operation, thermal expansion is not stabilized and thus workpieces machined during such period cannot be finished accurately.

By conducting warm up, the slideways can be appropriately lubricated to ensure smooth and accurate axis movement. It also stabilizes the machine from the viewpoint of thermal expansion to give consistently high accuracy.

Warm up must be conducted before starting daily operation to produce accurate products from the first output. In cold areas, warm must be conducted sufficiently.

Recommended warm up schedule

Warm up period	Approx. 30 minutes
Spindle speed	1000 min^{-1} (rpm) → 3000 min^{-1} (rpm) → 8000 min^{-1} (rpm) → Max. speed
Axis stroke range	Entire axis stroke

CAUTION

- While carrying out warm up, observe the machine operating status, lubrication, and others.

3-7 Spindle Operation

This section describes the procedure used for starting and stopping the spindle and also the procedure for changing the spindle rotating direction.

3-7-1 Starting the Spindle

To start the spindle when carrying out simple machining operation, follow the steps indicated below.

＜Procedure＞

(1) Press one of manual operation mode selector keys.

(2) Adjust the spindle speed by using the spindle override keys.

- The spindle speed can be adjusted in the range from 0 to maximum spindle speed in increments of 10 min^{-1} (rpm).

  △ : Spindle speed increases.

  ▽ : Spindle speed decreases.

- Spindle speed is displayed in units of 10 min^{-1} (rpm).
- The set spindle speed is displayed below the menu on the NC screen.

(3) Press the START (SPINDLE START) button.

→ The spindle starts rotating.

While the spindle is rotating, the indicator lamp in the button remains lit.

3-7-2 Stopping the Spindle

To stop the spindle, follow the step indicated below.

(1) Press the STOP (SPINDLE STOP) button.

→ The spindle stops rotating.

Note 1: The spindle speed displayed below the menu on the NC screen remains unchanged even when the spindle stops rotating.

Note 2: The spindle does not stop rotating even if the SPINDLE STOP button is pressed while in automatic operation.

3-7-3 Jogging the Spindle

To jog the spindle for such as checking the tool mounting conditions in the spindle or measuring the tool nose position, follow the steps indicated below.

＜Procedure＞

(1) Press one of the manual operation mode selector keys.

(2) Select the POSITION screen and press the [SPDL JOG] menu key.

→ The spindle jogs while the button is held pressed, and stops jogging when the key is released.

- Spindle jog speed is 50 min^{-1} (rpm).

3-7-4 Changing the Direction of Spindle Rotation

To change the direction of spindle rotation, follow the step indicated below.

(1) Select the spindle rotation direction by pressing the [SPNDL REV] menu key on the POSITION screen.

- The lamp in the menu is lit: The spindle rotates in the forward direction.
- The lamp in the menu is unlit: The spindle rotates in the reverse direction.

3-8 Feed Operation

3-8-1 Rapid Traverse Operation

To move an axis at a rapid traverse rate for such as mounting a tool to the spindle or carrying out positioning, follow the steps indicated below.

(1) Press the 

(RAPID) key.

→ The rapid traverse mode is selected.

(2) Adjust the rapid traverse rate.

It is possible to adjust the rapid traverse rate with the rapid traverse override keys in the range from 6% to 100% of the maximum traverse rate 100%, 50%, 25%, 12%, 6%

The values of 25%, 12%, and 6% are set for parameters and can be changed if required.

 : The rapid traverse rate increases by one step.

 : The rapid traverse rate decreases by one step.

(3) Move an axis to the required position by pressing the axis movement button.

- The axis is continuously moved while the axis movement button is held pressed and it stops moving when the button is released.
- The actual axis position is displayed on the NC screen such as the POSITION display.

3-8-2 Cutting Feed Operation

To move an axis at a controlled feedrate to carry out simple cutting, follow the steps indicated below.

(1) Select any of the 

(pulse handle feed multiplication ratio setting) keys.

(2) Place the AXIS SELECT switch in the required axis position. (See Fig. 2-5.)

→ The jog (cutting feed) mode is selected.

(3) Adjust the feedrate.

- It is possible to adjust the feedrate in 29 steps of preset feedrate in the range from 0 to 2000 mm/min (0 to 78.74 IPM) by using the cutting feedrate override keys.

   : The feedrate increases by one step.

   : The feedrate decreases by one step.

- The actual feedrate is displayed below the menu on the NC screen.

(4) Move an axis to the required position by pressing the axis movement button.

→ The axis is continuously moved while the axis movement button is held pressed and it stops moving when the button is released.

- The actual axis position is displayed on the NC screen such as the POSITION display.

(5) Upon completion of cutting feed operation, be sure to set the AXIS SELECT switch in the "O" (OFF) position.

3-8-3 Pulse Handle Feed Operation

To move a cutting tool precisely by using the pulse handle, follow the steps indicated below.

(1) Press one of the pulse handle feed multiplication ratio setting keys.

- There are three such keys and they are used to determine axis movement distance per pulse when the pulse handle is used to move an axis.

: Selects 0.001 mm (0.0001 in.) (0.001° for the 4th-axis) per division of the pulse handle.

: Selects 0.01 mm (0.001 in.) (0.01° for the 4th-axis) per division of the pulse handle.

: Selects 0.1 mm (0.01 in.) (not selectable for the 4th-axis) per division of the pulse handle.

(2) Select the axis to be moved.

- Place the AXIS SELECT switch in the required axis position.

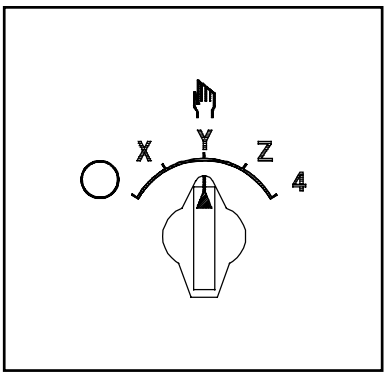


Fig. 3-4

(3) Turn the pulse handle clockwise or counterclockwise.

→ Clockwise rotation:

The axis moves in the positive direction.
Counterclockwise rotation:

The axis moves in the negative direction.

- The selected axis moves at a feedrate corresponding to the pulse handle rotating speed.
- The actual axis position is displayed on the NC screen such as the POSITION display.

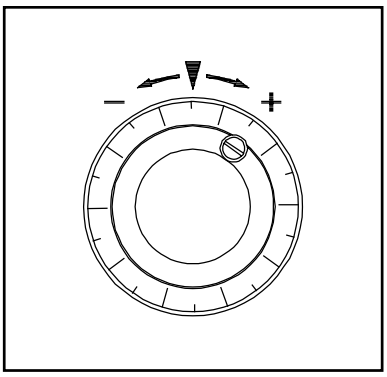


Fig. 3-5

(4) Upon completion of pulse handle feed operation, be sure to set the AXIS SELECT switch in the "O" (OFF) position.

3-8-4 Clearing the Soft-Limit Alarm

1. Soft-limit

The soft-limit refers to the stroke limit that is monitored by the NC.

With X-, Y-, and Z-axis, movable range is defined so that an axis will not move to the “O.T. limits”.

If an attempt is made to move an axis beyond the movable range, soft-limit alarm occurs and further axis movement is made impossible.

The soft-limit becomes valid when the power is switched on.

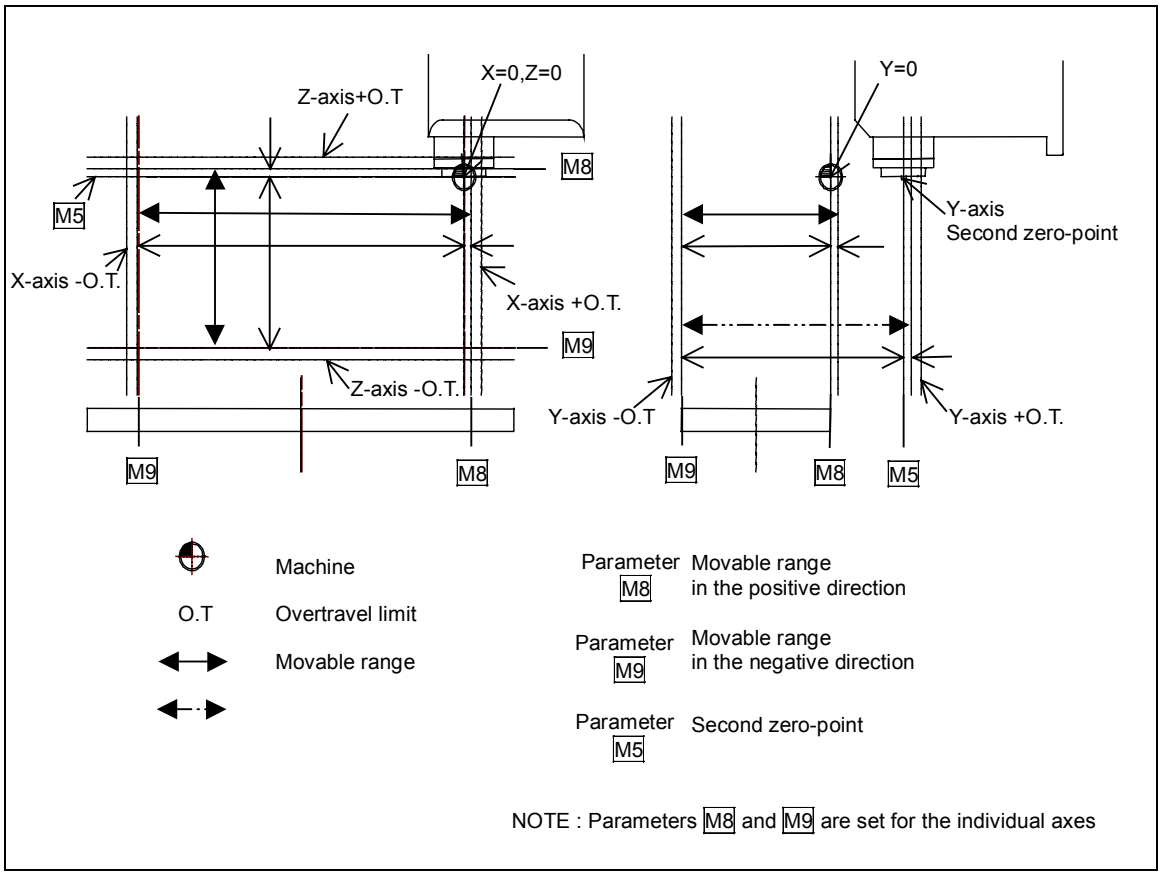


Fig. 3-6 Soft-Limit

CAUTION

- Never try to change the setting for parameters M8 and M9. If setting of these parameters is changed, it could cause serious damage to the machine.

2. Cleaning the soft-limit alarm

If an axis fails to move continuously due to the occurrence of a soft-limit alarm, move the axis in the opposite direction.

Example: Soft-limit alarm has occurred during axis feed of the X-axis in the positive direction by using the pulse handle.

(1) An axis does not move even when the pulse handle is rotated in the +X direction.

The following alarm message is displayed at the lower left area in the screen.

101 SOFT LIMIT +X

(2) Turn the pulse handle in the -X direction.

(3) The axis starts moving in the -X direction and the alarm message indicated above is cleared.

(4) This completes the restoration from the soft-limit alarm state.

It is possible to clear the soft-limit alarm in the following operation modes.

: Manual pulse feed mode (includes cutting feed)

: Rapid traverse mode

: Zero-point return mode (includes second zero-point return)

3-9 Coolant System Operation

1.  (COOLANT MENU) key

The following menu is displayed when this menu key is pressed.

With an exception of [ALL COOL AUTO ON] and [ALL COOL AUTO OFF] menu keys, if a menu key is pressed, the characters in it are highlighted and they return to the normal display when it is pressed once again.

COOLANT AUTO	COOLANT MANUAL							ALL COOL AUTO ON	

								ALL COOL AUTO OFF	

Note 1: All menu items of "△△△ AUTO" (only "COOLANT AUTO" with the standard specification) are valid (menu keys are lit) when the power is turned on.

Note 2: To change the menu display between menu 1 and menu 2, press the menu selection key at the right end.

- [COOLANT AUTO] menu key

  Flood coolant is not supplied even when the M08 command is executed unless this menu key is lit.

- [COOLANT MANUAL] menu key

  When this menu key is pressed, it illuminates and flood coolant is supplied.

- [ALL COOL AUTO ON] menu key

  When this menu key is pressed, all menu keys of "△△△ AUTO" (only "COOLANT AUTO" with the standard specification) are lit and become valid.

- [ALL COOL AUTO OFF] menu key

  When this menu key is pressed, all menu keys of "△△△ AUTO" (only "COOLANT AUTO" with the standard specification) return to the normal display and become invalid.

3-10 ATC Operation

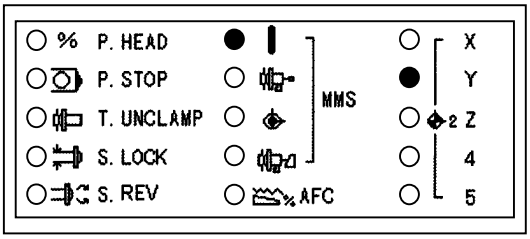
3-10-1 Manual ATC Operation

1. CNC Signal window

(1) Click the [Window] menu on the POSITION screen.

(2) In the pull-down menu, click the [CNC Signal] item.

(3) The CNC Signal window is displayed.



2. Axis movement

A. Move the Z-axis to the second zero-point.

(1) Press the MACHINE (MACHINE MENU) key while the NC is in the manual mode, and the following menu is displayed.

F0		ATC MENU	F3						#2 ZERO RETURN

(2) Press the [#2 ZERO RETURN] menu key.

(3) Hold the +Z (axis movement) key pressed.

(4) If the second zero-point return lamp in the CNC Signal window lights, it indicates that the Z-axis has returned to the second zero point.

B. Move the Y-axis to the second zero-point.

(1) Press the MACHINE (MACHINE MENU) key while the NC is in the manual mode, and the following menu is displayed.

F0		ATC MENU	F3						#2 ZERO RETURN

(2) Press the [#2 ZERO RETURN] menu key.

(3) Hold the +Y (axis movement) key pressed.

(4) If the second zero-point return lamp in the CNC Signal window lights, it indicates that the Y-axis has returned to the second zero point.

3. ATC operation

A. Turning the ATC arm to 52°

The ATC arm cannot be turned to 52° unless the following conditions are satisfied:

- X- and Y-axis are at the second zero point.
- Spindle is oriented.
- Tool is clamped.
- ATC cover is open.

(1) Follow the same step as (1) in item 2. A (to display the MACHINE menu).

(2) Press the [ATC MENU] menu key, and the following menu is displayed.

MAGAZINE FORWARD	MAGAZINE REVERSE	ATC ARM 0 deg.	ATC ARM 52 deg.	ATC ARM 180 CW	ATC ARM 180 CCW	ATC COVR CLOSE	ATC COVR OPEN	UNCLAMP OK	

(3) Keep pressing the [ATC ARM 52 deg.] menu key until the menu lamp comes on.

B. Turning the ATC arm to 180°

The ATC arm cannot be turned by 180° unless the following conditions are satisfied:

- X- and Y-axis are at the second zero point.
- Spindle is oriented.
- Tool is unclamped.
- ATC cover is open.

(1) Follow the same step as (1) in item 2. A (to display the MACHINE menu).

(2) Press the [ATC MENU] menu key, and the following menu is displayed.

MAGAZINE FORWARD	MAGAZINE REVERSE	ATC ARM 0 deg.	ATC ARM 52 deg.	ATC ARM 180 CW	ATC ARM 180 CCW	ATC COVR CLOSE	ATC COVR OPEN	UNCLAMP OK	

(3) Keep pressing the [ATC ARM 180 CW] or [ATC ARM 180 CCW] menu key until the menu lamp comes on.

C. Returning the ATC arm to the stand-by position

The ATC arm cannot be returned to the stand-by position unless the following condition is satisfied:

- Tool is clamped.

(1) Follow the same step as (1) in item 2.A (to display the MACHINE menu).

(2) Press the [ATC MENU] menu key, and the following menu is displayed:

MAGAZINE FORWARD	MAGAZINE REVERSE	ATC ARM 0 deg.	ATC ARM 52 deg.	ATC ARM 180 CW	ATC ARM 180 CCW	ATC COVR CLOSE	ATC COVR OPEN	UNCLAMP OK	

(3) Keep pressing the [ATC ARM 0 deg.] menu key until the menu lamp comes on.

4. Others

A. Perform the ATC cover open operation.

(1) Follow the same steps as (1) and (2) in item 3. A.

(2) Press the [ATC COVR OPEN] menu key.

B. Perform the ATC cover close operation.

(1) Follow the same steps as (1) and (2) in item 3. A.

(2) Press the [ATC COVR CLOSE] menu key.

C. Perform the magazine forward rotation.

(1) Follow the same steps as (1) and (2) in item 3.A.

(2) Press the [MAGAZINE FORWARD] menu key.

D. Perform the magazine reverse rotation.

(1) Follow the same steps as (1) and (2) in item 3.A.

(2) Press the [MAGAZINE REVERSE] menu key.

E. Unclamp the spindle tool.

＜If spindle orientation has not been completed＞

(1) Follow the same steps as (1) and (2) in item 3. A.

(2) Press the [UNCLAMP OK] menu key.

(3) Place the TOOL UNCLAMP switch in the  position.

＜If spindle orientation has been completed＞

(1) The spindle can be unclamped only by the operation in step (3).

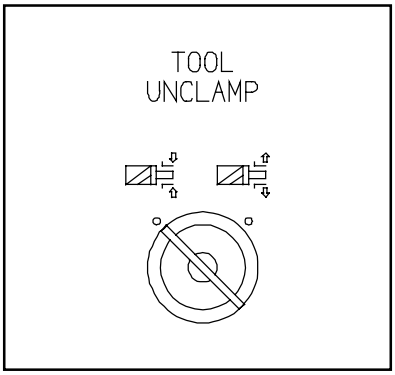


Fig. 3-7 TOOL UNCLAMP switch

F. Clamp the spindle tool.

(1) Place the TOOL UNCLAMP switch in the  position.

G. Orient the spindle nose.

(1) Press the  (MID) key to set the operation to the MDI mode.

(2) Press the [M CODE] menu key.

(3) Input “M19.”

(4) Press the CYCLE START button to start the machine in the automatic operation mode.

3-10-2 Recovering ATC Operation after Interruption

1. Recovering an interrupted ATC operation by pressing the emergency stop button

A. If the emergency stop button has been pressed when the Y-axis and Z-axis were at the second zero-point respectively:

(1) Select the MDI mode by pressing the 

(MDI) key.

(2) Press the [TOOL CHANGE] menu key.

(3) Carry out tool change operation.
Refer to Part 4, section 1-2-3, item 3 “Tool change”.

B. If the emergency stop button has been pressed when the X-axis was moving from the first zero-point to the second zero-point:

(1) Press the 

(first zero-point return) key to select the manual mode.

(2) Confirm the clamp/unclamp of the spindle tool on the CNC Signal window. For the CNC Signal window displaying method, refer to section 3-10-1, item 1 " CNC Signal window."

(3) Press the 

(MACHINE MENU) key, and the following menu is displayed.
This key is not valid unless the operator’s door is closed.

F0		ATC MENU							#2 ZERO RETURN

(4) Press the [FO] menu key, and the following menu is displayed.

ATC ARM MTR CW	ATC ARM MTR CCW	ATC ARM 0 deg.	ATC ARM 52 deg.	ATC ARM 180 CW	ATC ARM 180 CCW			UNCLAMP OK	

(5) When the spindle tool is unclamped (while the ATC arm is turning by 180˚)
Keep pressing the [ATC ARM 180 CW] or [ATC ARM 180 CCW] menu key until the menu lamp comes on.
Perform the tool clamp operation. (Refer to section 3-10-1 4-F "Spindle Tool Clamp.")
Keep pressing the [ATC ARM 0 deg.] menu key until the menu lamp comes on.

(6) When the spindle tool is clamped (while the ATC arm is turning to 52˚ or returning to 0˚)
Keep pressing the [ATC ARM 0 deg.] menu key until the menu lamp comes on.

(7) Press the 

(MACHINE MENU) key.
The normal menu is displayed.

C. If the emergency stop button has been pressed while the magazine was rotating:

- Generally, there are no problems for rotating the magazine even if it is stopped halfway.

(1) Confirm that the ATC arm is positioned at 0˚ (stand-by position).
If the ATC arm is in any position other than the stand-by position, the magazine does not operate. Move the ATC arm to the stand-by position, then turn the magazine to the commanded position.

(2) Manually turn the ATC arm to any position.

2. Recovering an interrupted ATC operation by pressing the reset switch

A. If the reset switch has been pressed when the Y-axis and Z-axis were at the second zero-point respectively:

(1) Follow the same steps as explained in item 1. A.

B. If the RESET switch has been pressed during ATC arm operation (turning):

(1) Follow the same steps as explained in item 1. B.

C. If the reset switch has been pressed while the magazine was rotating:

(1) Follow the same steps as explained in item 1. C.

3. Recovering an interrupted ATC operation by shutting off the power

A. If the power has been shut off when the Y-axis and Z-axis were at the second zero-point respectively:

(1) Carry out the zero-point return for the X-, Y- and Z-axis.

Refer to Part 2, section 3-5, “Zero-point Return”.

B. If the power has been shut off pressed during ATC arm operation (turning):

(1) Follow the same steps as explained in item 1. B.

(2) Carry out the zero-point return for the X-, Y- and Z-axis.

Refer to Part 2, section 3-5, “Zero-point Return”.

C. If the power has been shut off while the magazine was rotating:

(1) Follow the same steps as explained in item 1. C.

(2) Carry out the zero-point return for the X-, Y- and Z-axis.

Refer to Part 2, section 3-5, “Zero-point Return”.

4. Recovering an interrupted ATC operation by feed hold operation

If the FEED HOLD button is pressed during ATC cycle, the FEED HOLD indicator lamp comes on and all axes stop immediately.

Therefore, ATC does not operate if the FEED HOLD button is pressed. However, magazine operation, spindle orientation, and ATC cover operation are continuously performed. The CYCLE START indicator lamp begins flashing when the FEED HOLD button is pressed. Operation can be restarted by pressing the CYCLE START button.

3-11 Chip Conveyor Operation (Optional Specifications)

1. Arrangement and function of the switches

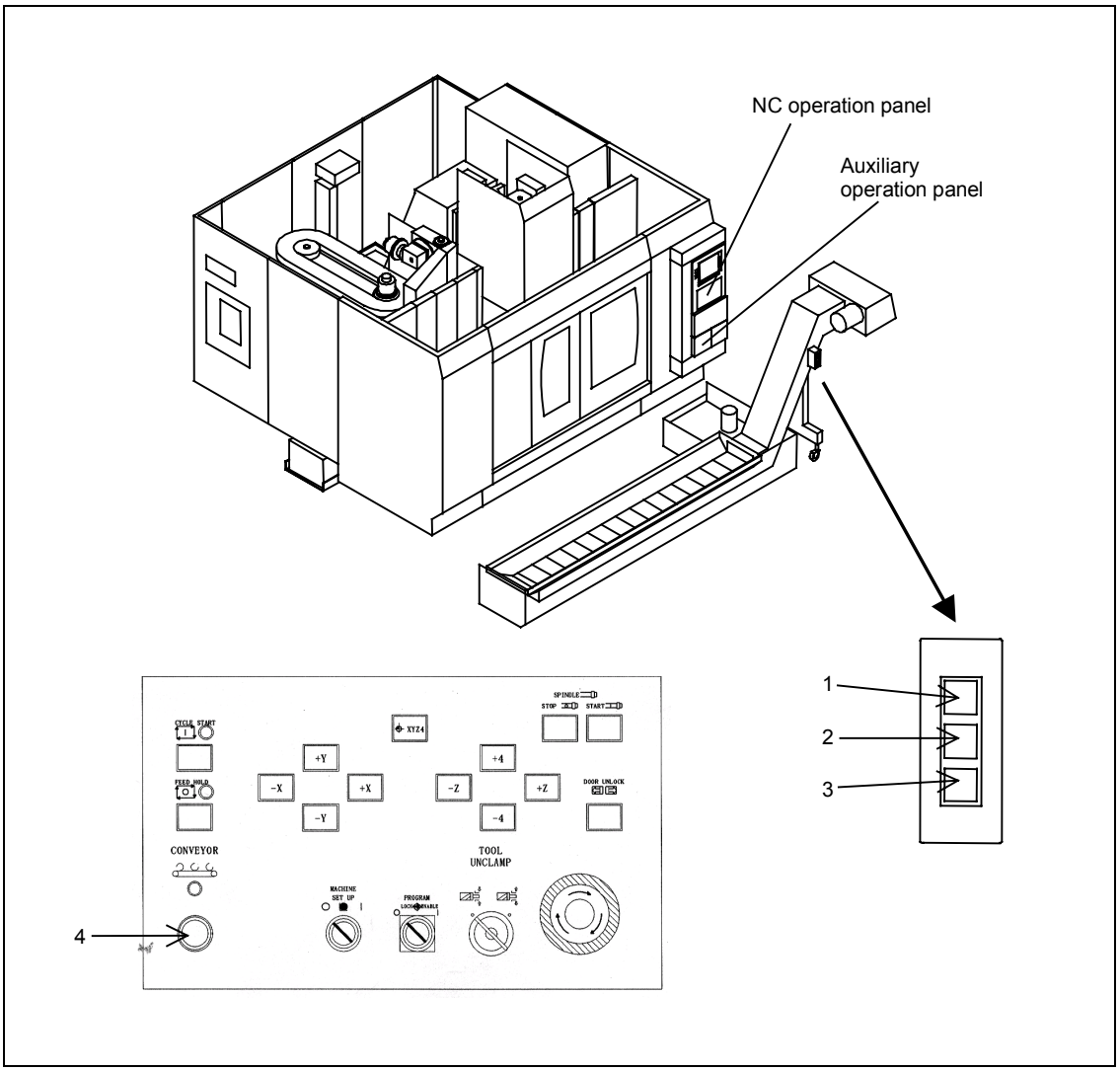


Fig. 3-8 Chip Conveyor Operation

Table 3-1 Function of Chip Conveyor Switches

No.	Name	Function
1	FORWARD	Used to run the chip conveyor belt in the forward direction to discharge chips.
2	STOP	Used to stop the chip conveyor belt.
3	REVERSE	Used to run the chip conveyor belt in the reverse direction.
4	CONVEYOR button (on the auxiliary operation panel)	When the power is supplied to the conveyor, the indicator lamp above the switch is lit.

2. Operation procedure

CAUTION

- In daily operation, the chip conveyor must be kept operating in the forward direction disregarding of whether or not chips are being generated.
- If large volume of chips are put into the chip conveyor at a time, chips will be stuck in the conveyor, causing the conveyor to stop.

＜Procedure＞

(1) Switch the power ON to the machine.

(2) Make sure that the conveyor can be operated safely.

(3) Press the FORWARD switch.

If this occurs, refer to the operating manual supplied with the chip conveyor to recover the chip conveyor operation.

- MEMO -

4 PREPARING FOR MACHINING

4-1 Procedure for Mounting or Removing a Tool in the Tool Magazine

4-1-1 Procedure for Mounting Tools to the Magazine

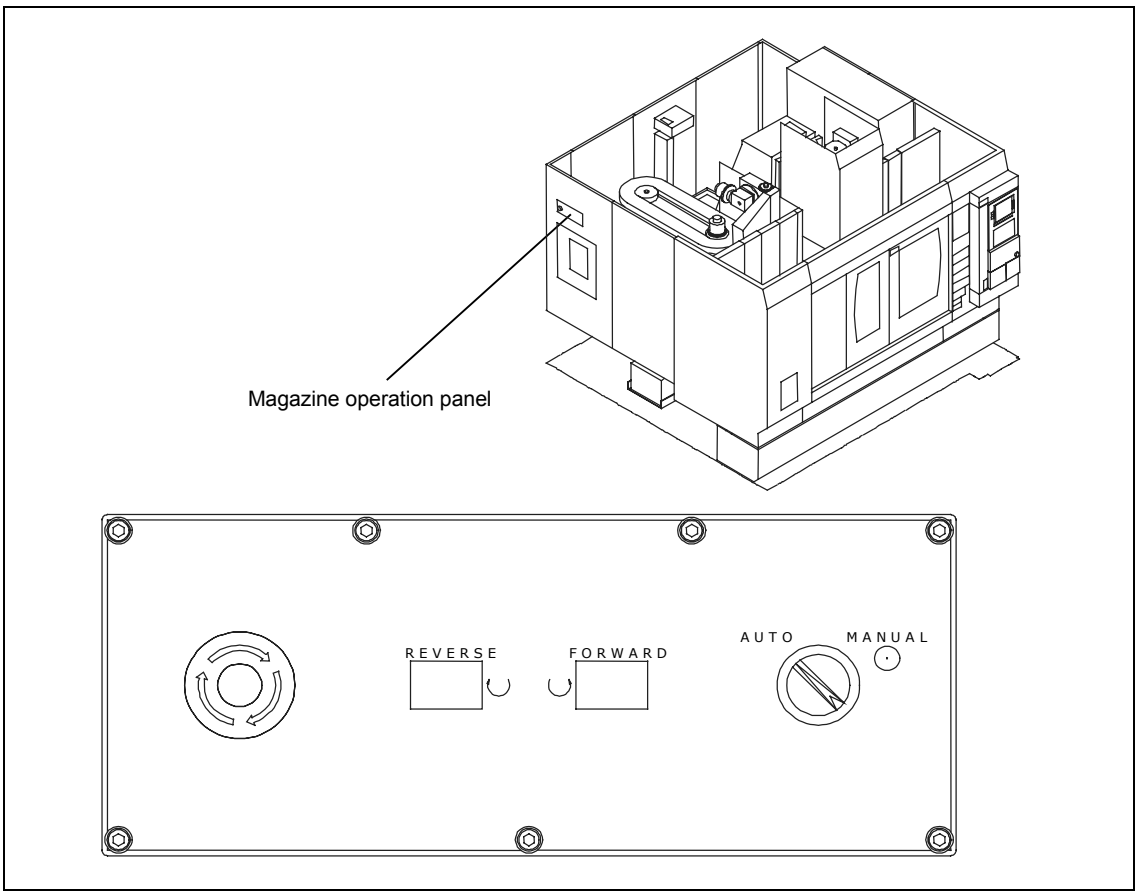


Fig. 4-1 Procedure for Mounting Tools to the Magazine

1. Magazine operation panel

＜Procedure＞

(1) Move the X-axis to the tool mountable position (X-axis stroke negative end) by using the manual pulse handle.

(2) Set the TOOL SELECT switch on the magazine operation panel to the MANUAL position and make sure that the MANUAL indicator lamp comes on continuously or blinks on and off. Refer to the explanation of the switches on the magazine operation panel.

(3) Use the FORWARD or REVERESE button on the magazine operation panel to rotate the magazine. Release the button when the tool pocket for the tool to be mounted reaches the most appropriate position. Note that tools must be mounted in the magazine so that they are well balanced.

(4) Open the magazine door.

(5) Align the tool pocket key with the keyway in the tool and insert the tool into the tool pocket.

(6) Confirm that the tool has been mounted completely in the tool pocket.

(7) Close the magazine door.
If the magazine door is left open, the magazine will not rotate in any operation mode, manual or automatic.

(8) Repeat steps (3) to (7) to mount other tools.

(9) After completing the operation, set the TOOL SELECT switch on the magazine operation panel to the AUTO position.

(10) If the ATC command or magazine rotation command is given while mounting a tool in the automatic mode, the MANUAL indicator lamp begins blinking and the machine enters the standby state waiting for the completion of manual operation. Press the 

(CYCLE START) button on the NC operation panel to continue automatic operation after completing the tool mounting operation.

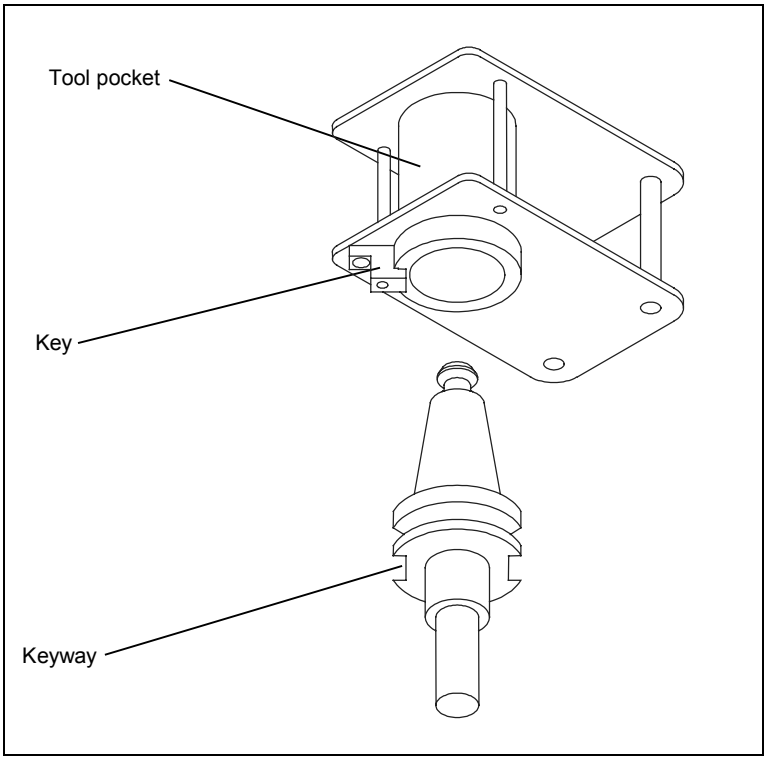


Fig. 4-2 Tool pocket and key

2. Precautions on Mounting the Tools

When mounting a tool such as a back boring bar or fine boring bar which must be shifted at the bottom of finished hole so that the cutting tip will leave a scratch in tool retraction motion, care must be paid in the direction of the cutting bit.

Since the machine shifts the cutting tool in the +Y direction, mount such tools in the magazine in the manner as illustrated below.

Note: For tools that do not require shift after the completion of machining, the cutting bit may be directed in any direction.

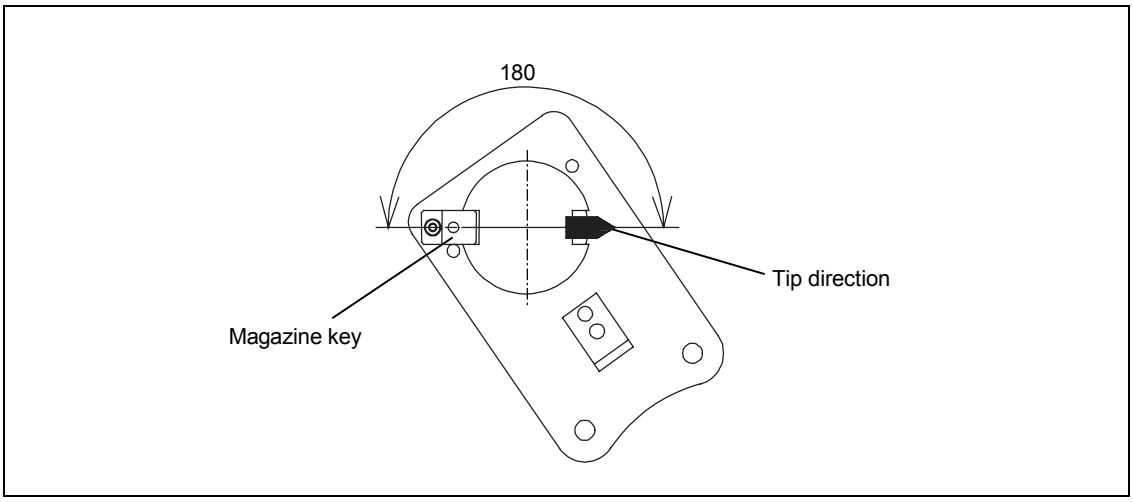


Fig. 4-3

Set the tool tip in the direction 180° from the magazine key.

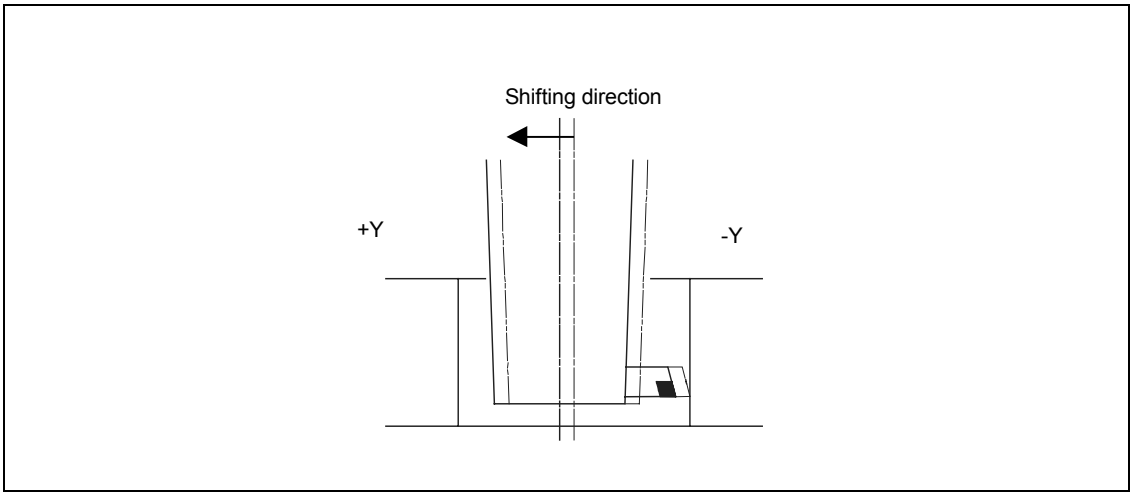


Fig. 4-4

4-1-2 Procedure for Removing Tools from the Magazine

＜Procedure＞

(1) Tools can be removed from the magazine regardless of whether the NC operation panel is set to the manual or automatic mode.

(2) Set the TOOL SELECT switch on the magazine operation panel to the MANUAL position and make sure that the MANUAL indicator lamp comes on continuously or blinks on and off. Refer to the explanation of the switches on the magazine operation panel.

(3) Use the FORWARD or REVERSE button on the magazine operation panel to rotate the magazine. Release the switch when the tool to be removed reaches the most appropriate position.

(4) Open the magazine door.

(5) Remove the tool from the gripper.

(6) Close the magazine door.

Note 1: If the magazine door is left open, the magazine will not rotate in any operation mode, manual or automatic.

(7) Repeat steps (3) to (6) to remove other tools.

(8) After completing the operation, set the TOOL SELECT switch on the magazine operation panel to the AUTO position.

(9) If the ATC command or magazine rotation command is given while removing a tool in the automatic mode, the MANUAL indicator lamp begins blinking and the machine enters the standby state waiting for the completion of manual operation.

Press the 

(CYCLE START) button on the NC operation panel to continue automatic operation after completing the tool removal.

4-1-3 Restrictions on Tools

Only tools that conform to the dimensional restrictions indicated in the diagram below can be mounted in the magazine.

WARNING

- Never use a tool or insert that exceeds the specification. Failure will cause a serious accident.
  The operator door has the strength that can assure safety to the energy effused by a centrifugal force or other external force when an insert mounted around the periphery of a large diameter tool comes off by some reason during a high-speed rotation of the spindle.

Table 4-1 Restrictions on Tools

Number of Tools / Item	Standard specifications	Option
	24	30
Pitch between pockets	95.25 mm (3.75 in)	95.25 mm (3.75 in)
Tool storage capacity	24	30
Tool selection method	Random selection of magazine pocket number, automatic shorter path selection (NOTE 4)	
Tool shank size	NT40	
Retention bolt	Yamazaki ANSI type	
Max. tool diameter	ϕ 80mm (3.15 in)	
Max. tool diameter with adjacent tools	ϕ 110 mm (4.33 in) (NOTE 3)	
Max. tool length	350 mm (13.78 in) (from gage line)	
Max. mass of tool (includes tool holder and retention bolt)	8 kgf (18 lbs) (NOTE 2)	
Magazine drive	AC servo motor	
Tool storage method	Stored in tool clamber	
Max. mass of tools stored in the magazine (NOTE 1)	120 kgf (270 lbs) (Average 5 kg×24)	150 kgf (337.5 lbs) (Average 5 kg×30)

Note 1: The “max. mass of tools stored in the magazine” indicates the allowable weight of the tools when they are set in the magazine so that the magazine is balanced.

Note 2: If a tool exceed a mass of 5 kg, be sure to register this tool as [HEAVY TOOL] (Refer to section 7-1-4 "Registering the tools to be used" in the operating manual " Operating NC Unit and Preparation for Automatic Operation.")

Note 3: If a tool exceeds a diameter of ϕ80 mm, this tool disables the adjacent tool to be mounted. If a tool exceeds a diameter ϕ80 mm, be sure to register this tool as [LARGE L] (large-diameter tool).
(Refer to section 7-1-4 " Registering the tools to be used " in the operating manual " Operating of NC Unit and Preparation for Automatic Operation.")

Note 4: When a tool has been registered as [LARGE L] (large-diameter tool), this tool has its own magazine pocket No.

The tools must be arranged in the magazine taking balance into consideration.

WARNING

- Observe the above restrictions on tools. In registering a tool as [HEAVY TOOL] or [LARGE L], carefully check the tool for correct identification and register the tool correctly, or the machine may be damaged.

4-2 Mounting and Removing Workpieces

4-2-1 Dimensional Restrictions on Workpieces

The following illustrations show the allowable maximum dimensions of workpieces that can be mounted on a table.

A workpiece exceeding these limit values must not be mounted on a table.

There is no interference between tools handled by the ATC and a workpiece not exceeding these limit values.

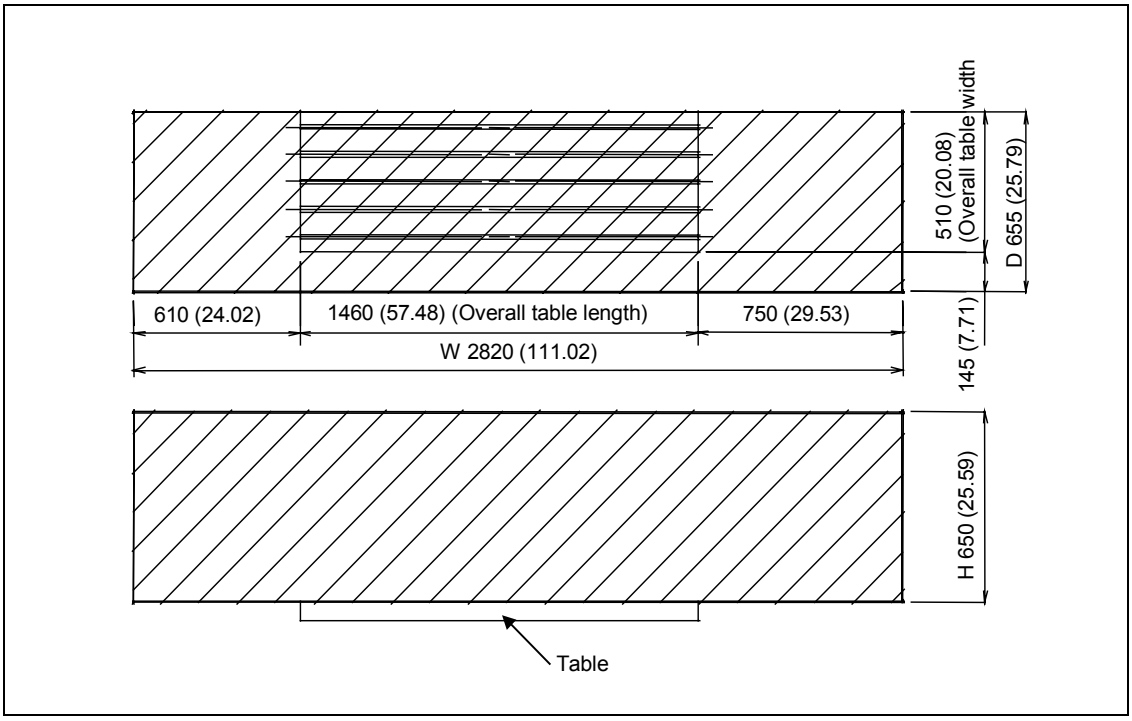


Fig. 4-5 Dimensional Restrictions on Workpieces

4-2-2 Procedure for Mounting and Removing Workpieces

Workpieces can be mounted or removed regardless of the operation mode (manual or automatic) set by the NC operation panel.

＜Procedure＞

(1) Set the SET UP switch (key-lock type switch) to "**O**" position, and remove the key.

(2) Open the operator's door.
When the operator's door is opened, spindle rotation, axis movement and ATC cycle are disabled.

(3) Mount or remove a workpiece to or from the pallet.

WARNING

- Mounting or removing a workpiece must be carried out correctly only while the operator’s door is open.

4-2-3 Measuring Coordinate Values

Refer to the following manual:

Programming Manual (MAZATROL) Part 2, Chapter 6 “COORDINATES MEASUREMENT FUNCTION”

4-3 Preparing Coolant

4-3-1 Selection of Coolant

Check the following points with the supplier when selecting a coolant.

1. Use a water based coolant

   If oil based coolant is used, the coolant discharge quantity will be reduced and its temperature rise will be large, increasing the likelihood of thermal displacement in the machine. In addition, there is a fire hazard associated with oil-based coolant and precautions including the provision of fire-fighting equipment are required if it is used.

2. Take factors such as lubricity, anti-corrosive properties, and anti-foaming properties into account when making the selection.

3. Check if the coolant has any harmful effects on human beings.

   Since water based coolant is likely to cause skin eruptions, pay due attention to the health management of the machine operators.

4. Check if the coolant will cause chemical products such as rubber and plastics to harden or expand.

5. Check if any problems will arise as a result of mixing of the coolant with the recommended lubricating oil for the machine.

   In rare cases a chemical change can result from the mixing of coolant and lubricating oil, causing discoloration and coagulation of the coolant.

4-3-2 Replenishing Coolant

Refer to the following when replenishing coolant.

Maintenance manual
Section 4-3-5, “Changing and Replenishing Coolant”.

4-4 Adjustment of Coolant Nozzle Direction

The coolant nozzle has been tapped to M6.

To change the coolant nozzle direction, screw an M6 bolt in the coolant nozzle and adjust the bolt position.

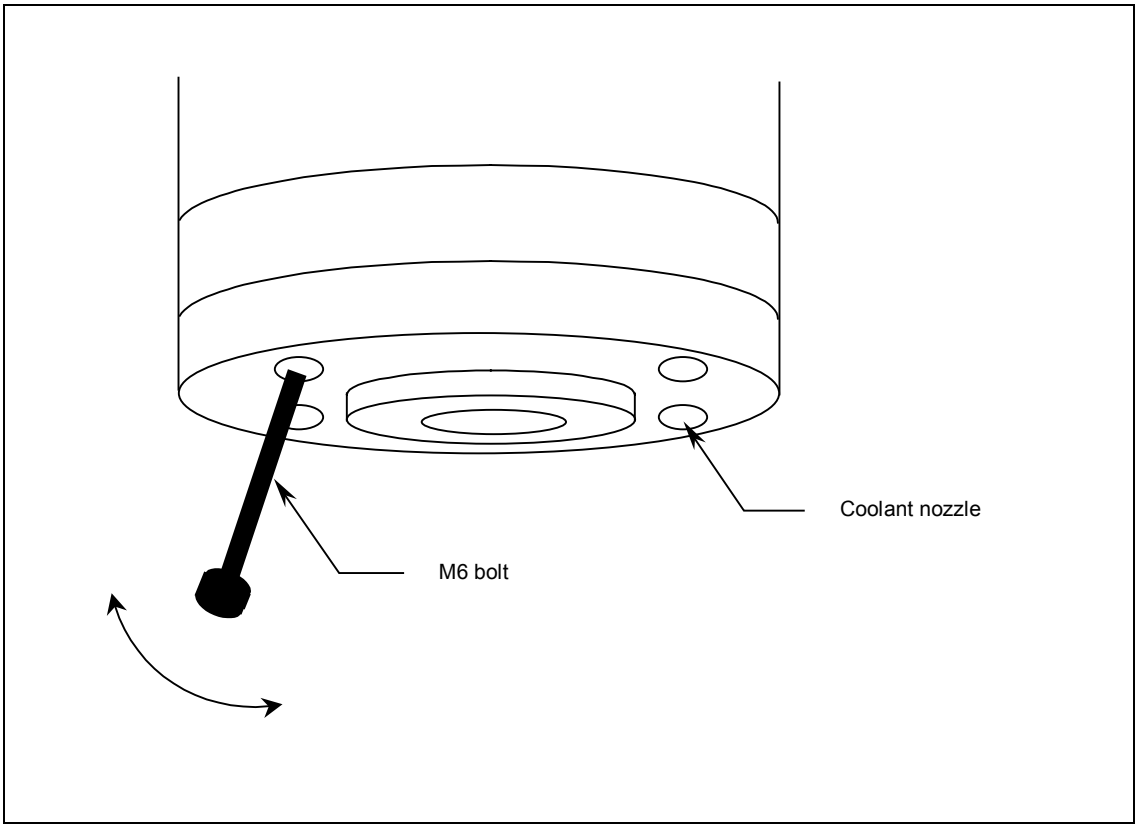


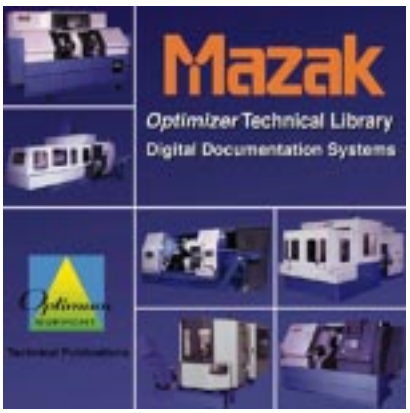
Fig. 4-6 Adjustment of Coolant Nozzle Direction

Before adjusting the coolant nozzle, be sure to confirm that the machine is in full stop.

- MEMO -

Section 3:

Operating NC Unit and Preparation for Auto Operation



Notes:

MAZATROL FUSION 640M NC OPERATION

1 PREPARATION FOR AUTOMATIC OPERATION

To machine workpieces, you must mount the tools for machining to the magazine and prepare jigs. For automatic operation, programs and other various data must be set to the NC.

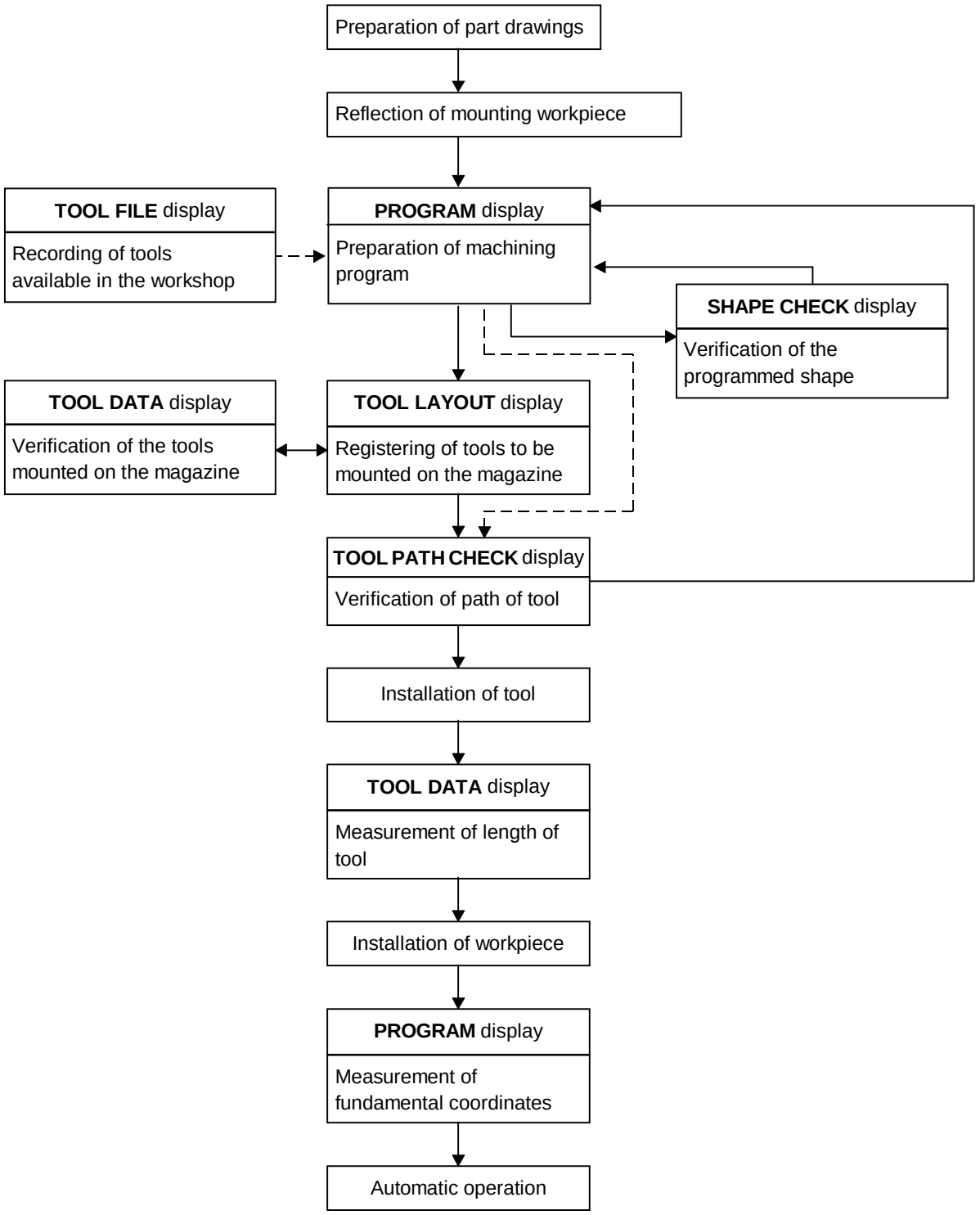
This chapter describes procedures to be carried out before automatic operation.

1-1 Preparation Steps for Automatic Operation

Supposing that the tooling and adjustment of each part for operation have already been completed, the subsequent operational steps are described below.

For execution procedure for automatic operation, see Part 4, **AUTOMATIC OPERATION**.

Note: Carry out the automatic operation always in the following order.



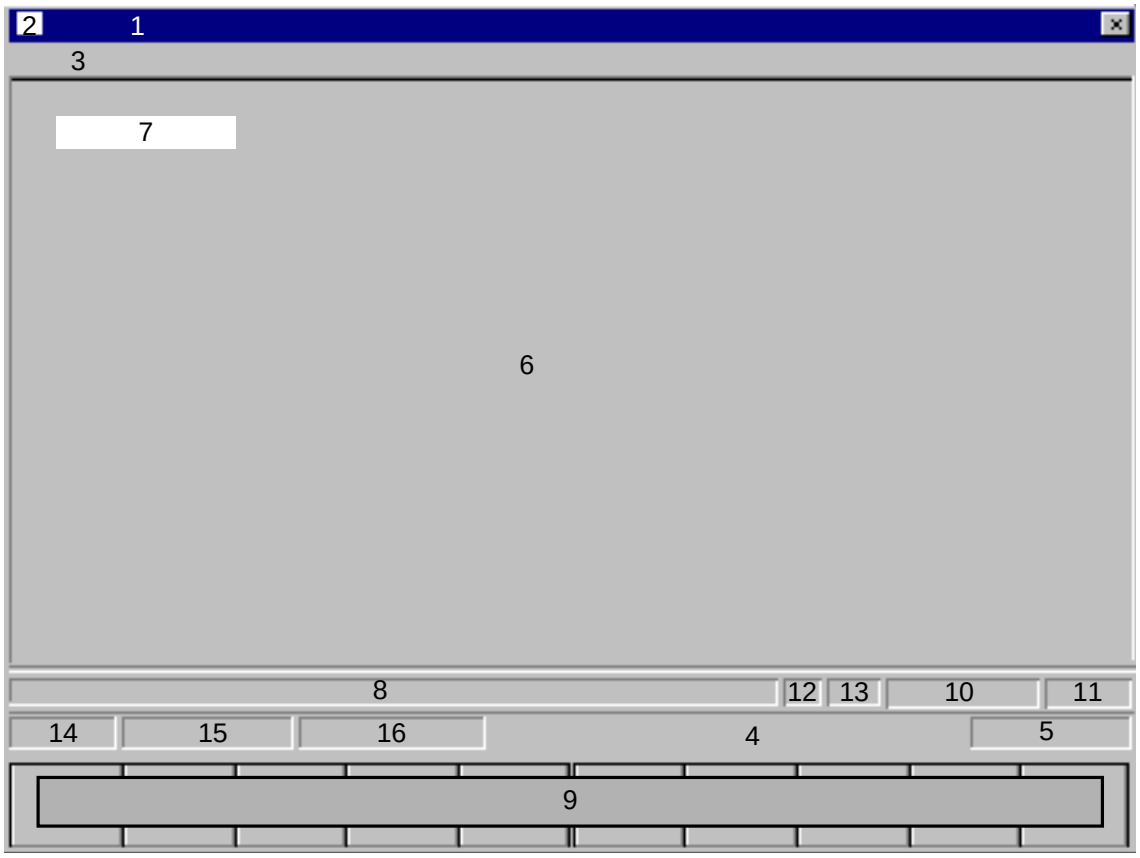
Notes:
1. It is possible to check the tool path even if the tools concerned are not registered on the **TOOL DATA** display. (The tool path is laid out by taking the diameter of the tool = the nominal diameter and the length of the tool = 0).
2. Automatic operation can not be performed if the tools to be used are not registered on the **TOOL DATA** display.

Procedure before performing the machining M3P006

2 DISPLAY OVERVIEW

2-1 Name of the Components of Each Display

The liquid crystal display unit of the operation panel presents a specific display according to the particular requirements. All displays are composed of the display areas shown below.



D735S1001

No.	Name	No.	Name
1	Title area	9	Menu display area
2	Icon display area	10	Date display area
3	Menu bar display area	11	Time display area
4	Message display area	12	Spindle rotational direction display area
5	Data input area	13	Display area of the number of TPS points
6	Data display area	14	Rapid traverse override display area
7	Cursor	15	Spindle speed override display area
8	Alarm display area	16	Cutting feed override display area

1. Title area
   The current display title is displayed.
2. Icon dipslay area
   An image abstractly presenting the current display.
3. Menu bar display area
   Functions common to all displays (Print, Window etc.) can be selected here in general. The menu bar, in particular, can also provide functions proper to specific displays.

4. Message display area
   Directions for operation and data descriptions are displayed in inquiry form.
   The character strings displayed here are referred to as “messages”.
5. Data input area
   Data that has been set using address/numeric data keys is displayed.
6. Data display area
   Data that has been input, data that the NC unit has processed, and other informations specific to the current display are displayed.
7. Cursor
   This is a marking which can be moved in the data display area vertically or horizontally.
   You can set data at the position where the cursor is displayed, or modify or erase the data displayed at the cursor position.
8. Alarm display area
   If erroneous operations are carried out or if machine or NC unit trouble occurs, details of those operations or trouble will be displayed here as an “alarm”.
   If an alarm is displayed, you must remove the cause.
   Refer to the Alarm List for further information on alarms.
9. Menu display area
   The functions of the ten keys (menu keys) located directly below the LCD are displayed. The entire character string displayed here is referred to as a “menu”.
   Each time a menu changes over, the functions of each menu key also change.
10. Date display area
11. Time display area
    Date and time of the current day are displayed respectively.
    The contents of these areas, however, differ from the above to provide special indications proper to some displays.
12. Spindle rotational direction display area
    “REV” appears to indicate the reversed rotation of the spindle during program flow.
13. Display area of the number of TPS points.
    Number of tool stop positions currently stored by the TPS function.
    For details of the TPS function see Section 5-2 in Part 4.
14. Rapid traverse override display area
    Indication of the current rapid traverse speed in percentage (0 to 100) with respect to the maximum value preset in parameter **M1**.
    For details of the rapid traverse override see Section 3-1 in Part 4.
15. Spindle speed override display area.
    Indication of the current spindle speed either in percentage (0 to 150) with respect to the programmed value during automatic operation, or in number of revolutions per minute during manual operation.
    For details of the spindle speed override see Section 3-3 in Part 4.
16. Cutting feed override display area
    Indication of the current feedrate either in percentage (0 to 200) with respect to the programmed value during automatic operation, or in actual value during manual operation.
    For details of the cutting feed override see Section 3-2 in Part 4.

2-2 Types of Displays

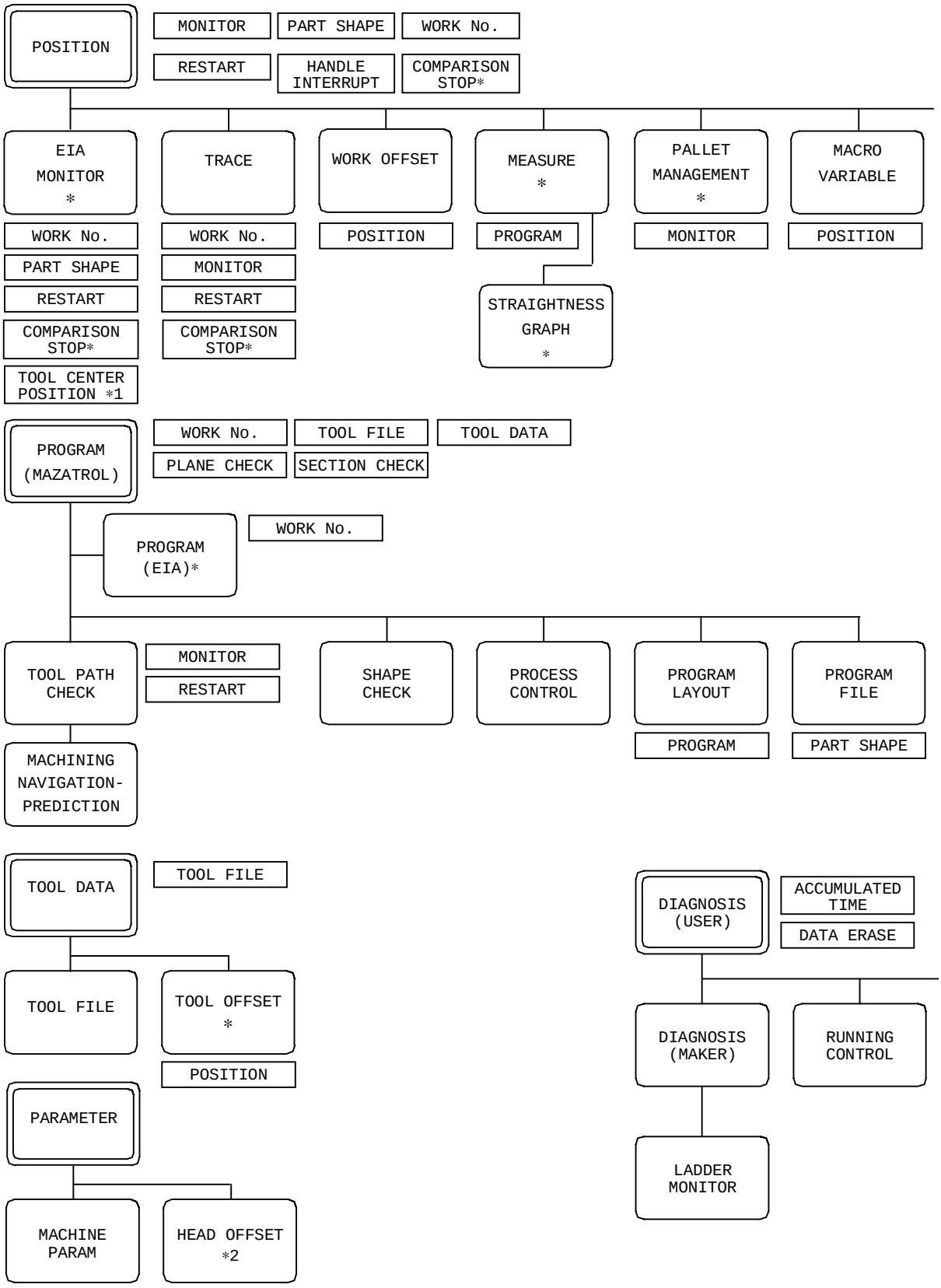
To manufacture a product using the NC machining center, you must register various information, such as tool data, machining program data, etc., in the NC unit in advance. You must also check registered information and the movements of each section of the machine during operation.

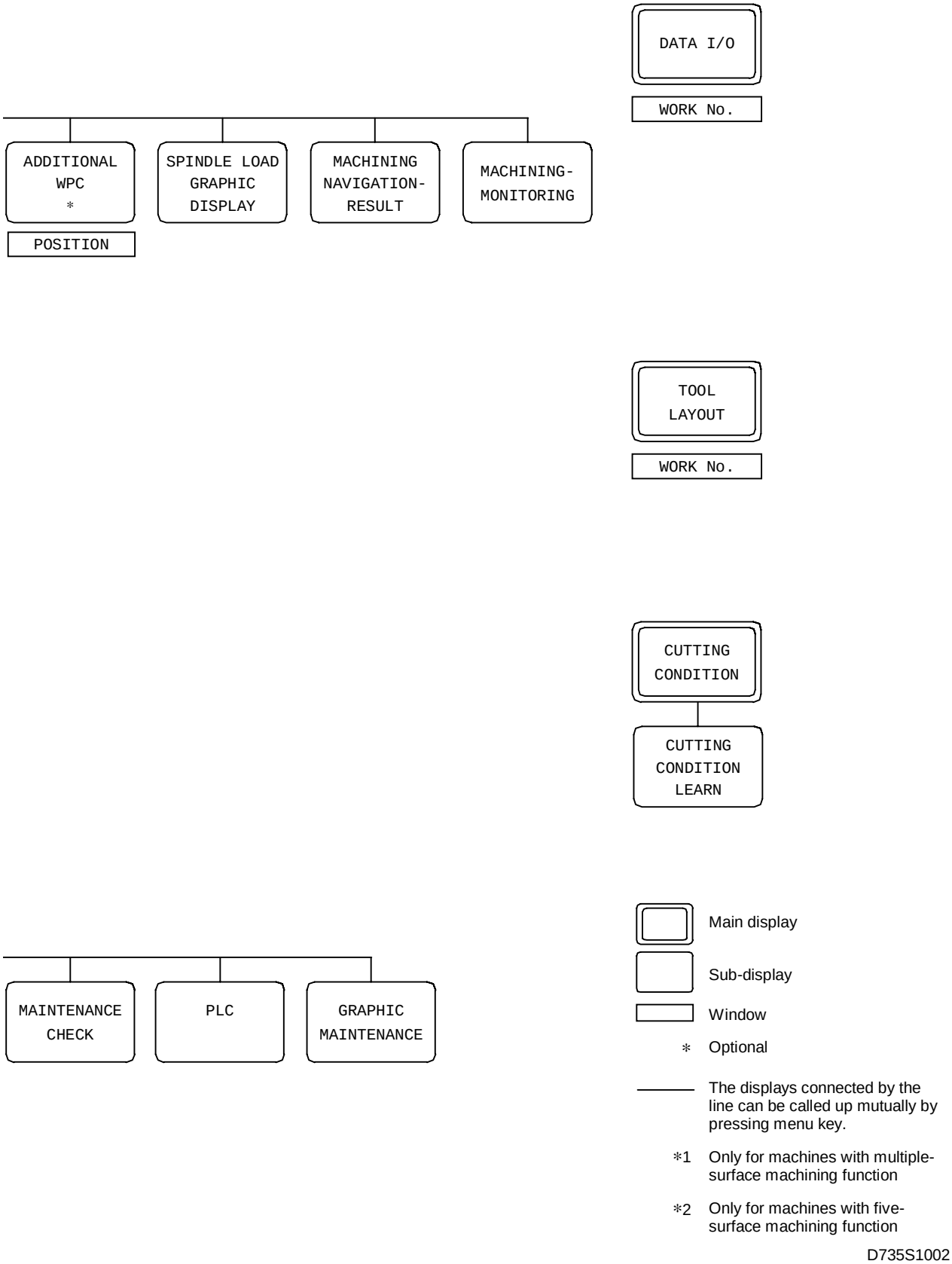
The NC unit provides the following types of displays to allow for a variety of jobs that are required for machine operation:

1. Displays related to the machine operational status
   These displays allows you to check the movements of each section of the machine and the machining status.
2. Displays related to program creation
   Use these displays to create, check, and/or edit a program appropriate for the type of product to be machined.
3. Displays related to tools
   Call these displays to register type and compensation data for the tool you are going to use.
4. Displays related to machining set up
   These displays allows you to carry out data registration required for executing a machining program, and other jobs.
5. Displays related to parameters
   These displays are used to register the control elements (parameters) of the NC unit or machine and the data for automatic decision of cutting conditions.
   Data that YAMAZAKI MAZAK recommends is displayed; you can add or change data as required.
   Refer to the Parameter List for further details.
6. Displays related to data transference between the NC unit and an external hardware unit
   Use these displays to check the registration status of machining programs or to transfer machining program data or other data between the NC unit and an external hardware unit.
7. Displays related to diagnosis
   These displays are presented for diagnosis and maintenance of the NC unit or machine.

2-3 Configuration of Displays

Configuration of displays





D735S1002

2-3-1 DISPLAY MAP display

Function:

The names of the displays that the NC unit can create are listed here.

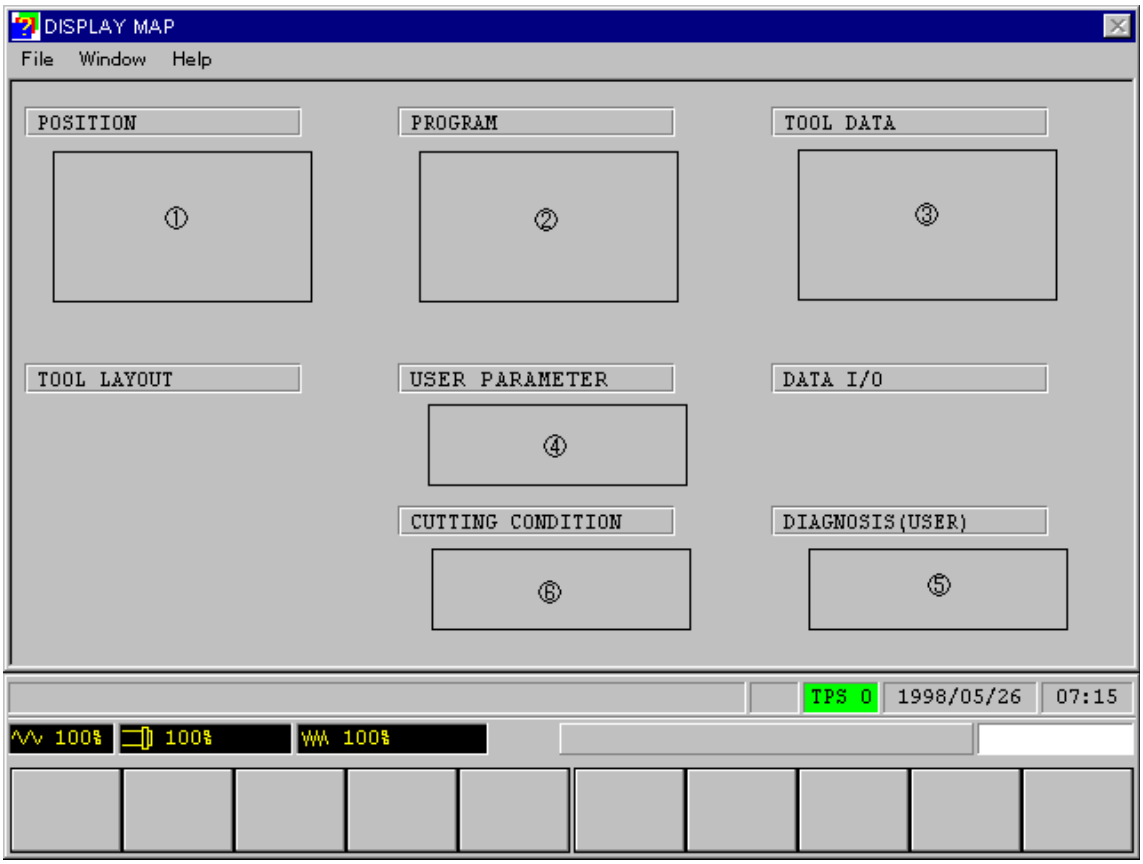
Remarks:

- This display gives a grouped list of display names which can be displayed on the following eight displays:

  - **POSITION** display
  - **PROGRAM** display
  - **TOOL DATA** display
  - **TOOL LAYOUT** display
  - **USER PARAMETER** display
  - **DATA I/O** display
  - **CUTTING CONDITION** display
  - **DIAGNOSIS** display

- Any of the name-listed displays can be directly selected from this display.

1. Data display



D735S1100E

2. Display data description

No.	Item	Unit	Data description
•	POSITION	•	Names of the displays that can be directly selected from the **POSITION** display.
•	PROGRAM	•	Names of the displays that can be directly selected from the **PROGRAM** display.
•	TOOL DATA	•	Names of the displays that can be directly selected from the **TOOL DATA** display.
•	USER PARAMETER	•	Names of the displays that can be directly selected from the **USER PARAMETER** display.
•	DIAGNOSIS	•	Names of the displays that can be directly selected from the **DIAGNOSIS** display.
•	CUTTING CONDITION	•	Names of the displays that can be directly selected from the **CUTTING CONDITION** display.

3. Display operations

Selection of the **DISPLAY MAP** display makes the **DISPLAY MAP** menu item and the name of the immediately previous display assume a reverse display status. For a display whose name is not displayed, however, the name of the display immediately previous to that display changes into reverse display status.
Any of the name-listed displays can be directly selected from this display.

(1) Make the name of the display to be selected assume a reverse display status.

- Press the cursor key and move the cursor to the name of the display to be selected.

(2) Press the **DISPLAY MAP** menu key.

➔ The display the name of which is highlighted will be selected.

- The intended display can also be selected by pressing the INPUT key, instead of the **DISPLAY MAP** menu key.

2-4 Windows

1. Concept of windows

The windows of this NC unit refer to boxes that display operational aid information during the selected display mode.

2. How to display a window

Two types of windows can be displayed: a type that can be displayed by pressing menu key, and a type that can be displayed by selecting through “Window” on the menu bar.

Thus, only the appropriate window can be displayed on each display.

1. Windows opened by pressing menu key
   Windows appearing when the corresponding menu key, such as WORK No. and RESTART, is pressed.
2. Windows selected through “Window” on the menu bar
   Windows appearing when the WINDOW key is pressed and an option is selected from the listing under “Window” on the menu bar.

3. Erasing windows

Basically, windows that have been displayed by the selection from the menu bar or by pressing the corresponding menu key can be erased by repeating the same selection from the menu bar or by pressing that menu key once again, respectively.

Note: Several certain windows will be auto-erased if a menu key or the display selector key is pressed or if numeric data is set.

4. Window list

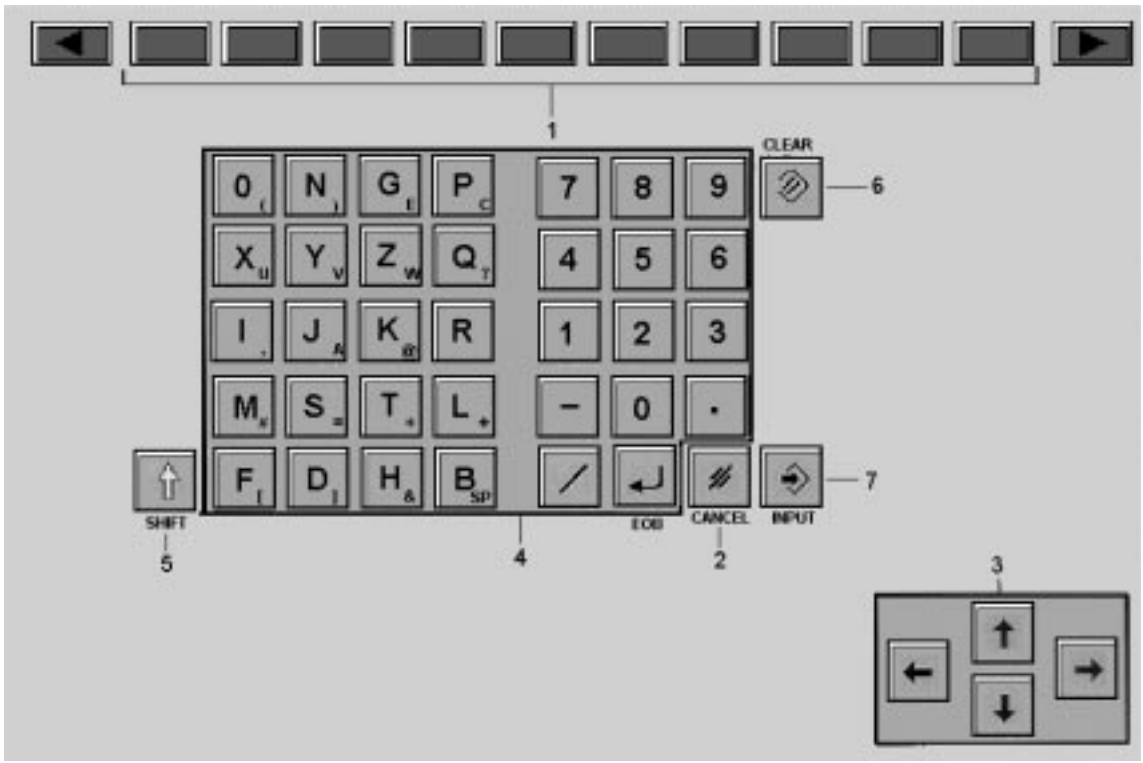
Window name	Description of window display	Displays on which the window can be displayed
WORK No.	A list of program work numbers registered in the NC unit. **Notes:** **1**. If the display of the above data spans over mulitiple pages, the next page or the previous page can be displayed by pressing the page key ⇓ or ⇑, respectively. **2**. The size of the window can be changed by clicking the button displayed at the upper right corner of the window. **3.** Display data in the window will not be changed even if a work number is input in the NC unit using data input/output operation while this window is displayed.	- POSITION - PROGRAM - DATA I/O - EIA MONITOR - TRACE
PART SHAPE	For the **POSITION** display, the machining shape of the program for which a work number searching operation has been performed is displayed only on one XY plane. For the **PROGRAM FILE** display, the desired program can be drawn only on one XY plane.	- POSITION - PROGRAM FILE - EIA MONITOR
MONITOR	The contents of a program for which a work number searching operation has been performed are displayed (that is, for a MAZATROL program, program layout is displayed, or for an EIA/ISO program, program data is displayed). Also, during automatic operation with the **POSITION** display remaining on the monitor, the section currently being executed is displayed in highlighted form. During the tool path check mode, the section currently being checked is displayed in reverse status.	- POSITION - TOOL PATH CHECK
RESTART	Indication of restart position and modal information that corresponds to the restarting section existing during the restart mode after EIA/ISO searching has been completed.	- POSITION - TRACE - EIA MONITOR
HANDLE	Amount of interruption existing during manual pulse interruption	- POSITION
Calculator	Provides functions of a pocket calculator.	
TOOL DATA	Data on the tools registered in the NC unit **Note:** Registered tool data can be displayed only during creation of MAZATROL programs, and only the tool data appropriate for the sequence section indicated by the cursor will be displayed. If the display of the above data spans over multiple pages, the next page or the previous page can be displayed by pressing the page key ⇓ or ⇑, respectively.	- PROGRAM (MAZATROL)

Window name	Description of window display	Displays on which the window can be displayed
POSITION	Same as the current position displayed on the **POSITION** display	- MACRO VARIABLE - TOOL OFFSET - WORK OFFSET - ADDITIONAL WPC
PROGRAM	Display of the MAZATROL program of the unit where the cursor remains	- PROGRAM LAYOUT
TOOL FILE	Tool data that has been registered in the NC unit - Pressing the page key ⇊ displays the next page. - Pressing the page key ⇈ displays the previous page. **Note:** Only tool file data for milling tools are displayed on the **PROGRAM (MAZATROL)** or the **TOOL DATA** display.	- PROGRAM (MAZATROL) - TOOL DATA - CUTTING CONDITION LEARN
NOM-ϕ SELECT	Display of a list of standards for unified/pipe threads - Pressing the page key ⇊ displays the next page. - Pressing the page key ⇈ displays the previous page. **Note:** This window can only be displayed in the MAZATOL programming while the cursor remains set at the required position for input of the nominal diameter for a tapping tool.	- PROGRAM (MAZATROL) - TOOL DATA
TOOL CENTER POSITION (only for machines with multiple-surface machining function)	Moving amount of tool nose and its residual distance in G68 coordinate system	- POSITION
ALARM NAVIGATION	Display of alarm remedial information upon occurrence of alarms A parameter (bit 1 of **F80**) is provided for the selection of whether this window is to be displayed automatically upon occurrence of an alarm. Manual display of the window can be done by the selection through “Help” on the menu bar in any display as well as by pressing the **ALARM NAVI.** menu key on the **GRAPHIC MAINTENANCE** display. **Notes:** **1.** One alarm is described on one page. For multiple simultaneous alarms, use the page keys ⇊ and ⇈ to display the next and previous pages, repectively. **2.** Use the vertical and horizontal scroll bars if the window area is not large enough to display the information in its entirety. **3.** Remedial information is only provided for the alarms No. 200 to No. 399. **4.** The contents of the window remaining opened will not be cleared together with clearence of the alarm itself.	- Any display

- NOTE -

3 DATA SETTING METHODS

Registration of data in the NC unit is accomplished by setting data in each item displayed on the LCD unit.
This chapter describes general methods of setting, erasing and modifying data, and related precautions.
More specific methods of setting data are described in each section for operating display. Refer to the relevant section. You will only need to use the following seven types of keys when setting, erasing or modifying data:



D735S1004

No.	Name	No.	Name
1	Menu keys	5	Shift key
2	Data cancellation key	6	Clear key
3	Cursor keys	7	Input key
4	Address/numeric data keys		

The NC unit has the feature that it uses interactive input. That is, the operator can set data in sequence just by responding to inquiry messages issued by the NC unit. When setting data, check the message display area to ensure that the data you are going to set is exactly what you want to set. Usually, messages are displayed in one of the following two formats:

1) "**********?"
2) "******** <MENU>?" ******* : Details of the data to be set

Set data if the message is displayed in format 1) above.
Select data from the menu if the message is displayed in format 2) above.

3-1 Setting Data and Address

Set data and address using this procedure:

(1) Place the cursor on the intended item.

- Press one of the cursor keys and move the cursor to the intended item.
- If the cursor does not appear on the CRT screen, press one of the cursor keys and then when the cursor is displayed, move it to the intended item.

Pressing the cursor key ↓ or → displays the cursor at the first item on the screen where you can set data.

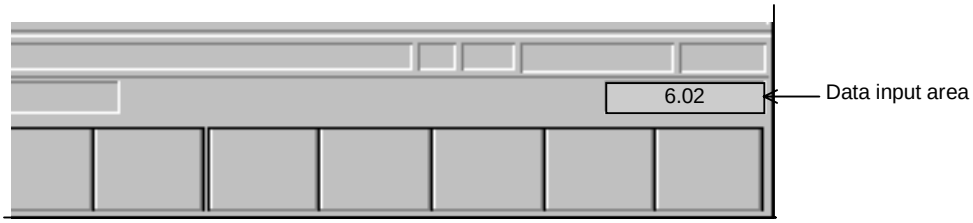
Pressing the cursor key ↑ or ← displays the cursor at the last item on the screen where you can set data.

(2) Specify data with address/numeric data keys.

Example: To set data of “6.02”:

Press [6] [•] [0] [2] in that order.

➔ The selected data is displayed in the data input area.



- If you have carried out wrong key operations and non-intended data has been displayed in the data input area, press the clear key to erase the data, and then specify the intended data.

(3) Press the INPUT key.

➔ The data that has been selected in step (2) above is displayed in the item position where you have set the cursor.

Note 1: Address data setting

Address data can be set by either pressing the address data keys as they are, or pressing [SHIFT] key and then entering the address data.

Address data marked with a large-size character in the middle of each key can be set just by pressing the address data keys as they are.

Address data marked with a small-size character at the right bottom corner must be set by pressing the [SHIFT] key first and then pressing the address data key.

Example: To enter N, press the [N)] key.

To enter), press the [SHIFT] and [N)] keys in that order.

Note 2: Unless data is displayed at the corresponding item within the data display area, the data will not be registered in the NC unit. After data selection using data keys, therefore, the data must be registered by pressing the INPUT key.

3-2 Selecting Menu Data

Use this procedure to set non-numeric/alphabetic data:

(1) Place the cursor on the intended item.

- See the description of step (1) in Section 3-1 above.

(2) Select data item from the menu displayed.

➔ Selected data item is displayed at the position where the cursor has been placed.

Note 1: In the remainder of this manual, selecting data from a menu will be called “menu data selection”.
For the NC unit, all data can be set by “numeric/alphabetic data setting” or “menu data selection”, or by combined use of these two methods.

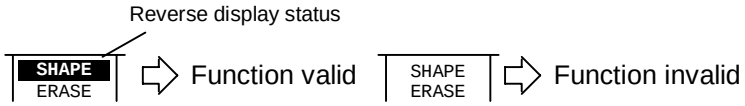
Note 2: After selection from the menu, some specific types of data (such as exponents on the **MACRO VARIABLE** display) may be displayed in the data input area, not directly at the cursor position. If such is the case, the INPUT key must be pressed following execution of step (2).

Note 3: The menu consists of the following three types of terms:

Terms on display selection
Data
Terms that represent a function

Pressing the menu key corresponding to a term representative of a function displays that term in reverse form. Re-pressing the same menu key clears the reverse status of the term.
A reverse display status indicates that the corresponding function is valid.
A normal display status indicates that the corresponding function is invalid.

Example:



3-3 Erasing Data

Erase data as follows:

(1) Place the cursor in the position where the data to be erased is displayed.

- See the description of step (1) in Section 3-1.

(2) Press the CANCEL key.

➔ The data at the cursor position is erased (or cleared to 0).

Note: You may be able to erase data line-by-line or to erase the entire displayed data at one time.
Some types of data, however, cannot be erased by using the CANCEL key.

3-4 Modifying Data

Modify data as follows:

(1) Place the cursor in the position where the data to be modified is displayed.

- See the description of step (1) in Section 3-1.

(2) Set new data.

➔ The data at the cursor position is overridden with the new data.

3-5 Incremental Input

During data setting, you may see the function name **INCRMENT INPUT** in a menu.

Using this function, you can set desired data just by selecting an incremental or decremental value for the original data.
Effectively use this function when increasing or reducing displayed data.

The procedure for using the incremental input function is as follows:

(1) Press the **INCRMENT INPUT** menu key.

➔ The display of **INCRMENT INPUT** changes to reverse display status.

(2) Select an incremental or decremental value for the original data.

Example: To reduce the data of “12.05” by –0.1:

Press [–] [0] [•] [1] in that order.

➔ The selected value is displayed in the data input area.

(3) Press the INPUT key.

➔ The original data changes by the value that was selected in step (2) above.
In the example shown above, the original data changes to “11.95” (= 12.05 – 0.1).

3-6 Saving Data onto the Hard Disk

This NC unit is designed so that machining programs are saved onto the hard disk. If the NC unit is accidentally turned off by a sudden power interruption or other events, entered data may not be saved. During data entry, therefore, perform the following operations as appropriate for data storage:

Press the **PROGRAM COMPLETE** menu key or change the **PROGRAM** display to another one even halfway to save the machining program as far as it is prepared.

If, moreover, the NC unit should be turned off during the operation of the hard disk, saved data may be damaged. Store machining programs, therefore, onto IC memory cards or other media.

4 DISPLAYS RELATED TO THE MACHINE OPERATIONAL STATUS

4-1 POSITION Display

Function:

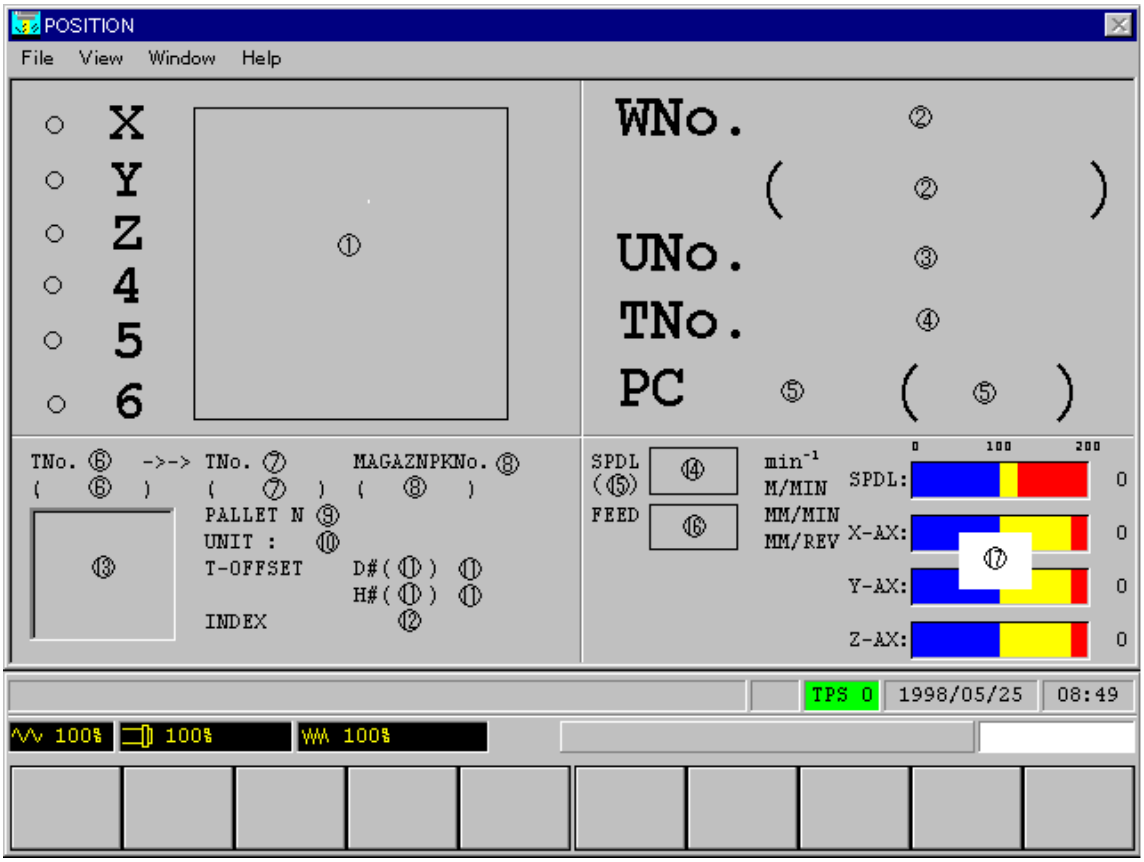
This display shows various types of information about the operational status of the machine and the workpiece-machining status.

Remarks:

When the **POSITION** display is selected, the various functions required for manual operation or automatic operation will be displayed at the menu display area. The usage of each such menu is described in chapters related to manual operation and automatic operation.

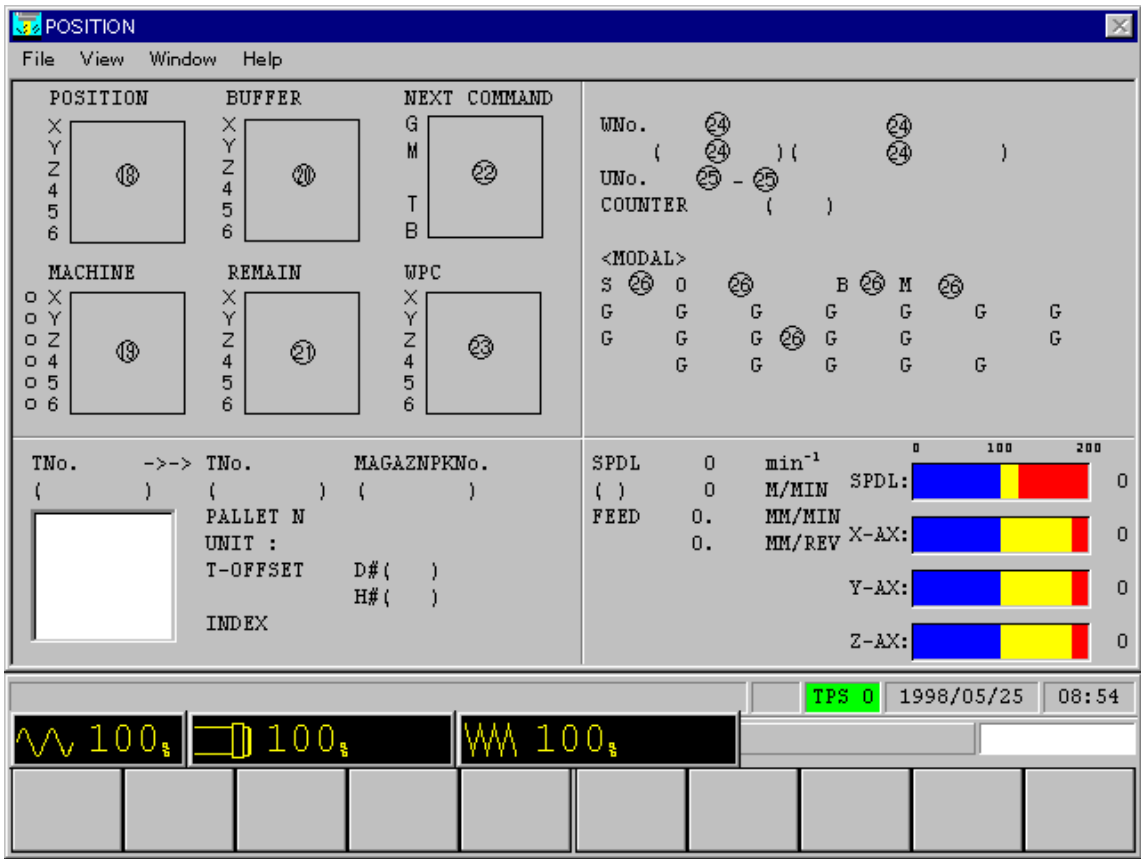
4-1-1 Data display

1. Displayed data when the COMMAND menu item is not reversed



D735S1005E

2. Displayed data when the COMMAND menu item is reversed



D735S1006E

3. Display data description

No.	Item		Unit	Data description
•	Current position	X,Y,Z	mm (inch)	Current position of the tool tip on each axis in the workpiece coordinate system
		4,5,6	mm (inch) or degrees	Current position of the tool tip on each axis in the workpiece coordinate system (when equipped with additional axes)
•	WNo.		—	Work number of the program being executed WNo. **a b** **(a' b')** **a, a'** } Work numbers **b, b'** } Identification of MAZATROL program or EIA/ISO program (For data items **a**' and **b**', data is displayed only during execution of subprograms.)
•	UNo.		—	- If the program being executed is a MAZATROL program: UNo. **a** **a**• Unit number - If the program being executed is an EIA/ISO program: The identification number of the sequence being executed are displayed. UNo. **a** **a**• Sequence number
•	TNo.		—	Number of the tool being mounted in the spindle

No.	Item		Unit	Data description
•	PC		Pieces	Actual quantity of machined workpieces, and the preset number of workpieces to be machined (Parts count) PC **a (b)** **a**• Actual quantity of machined workpieces **b**• Preset number of workpieces to be machined
•	TNo.		—	- The number and name of the tool currently mounted in the spindle - If an eight-digit T-command code is set, the number of the tool being used will be displayed at TNo. The group number will be displayed in parentheses in that case. TNo. **a** **(b)** **a**• Number of the tool currently mounted in the spindle **b**• Name of the tool currently mounted in the spindle (or if an eight-digit T-command code is set, then the group number)
•	TNo.		—	- Number and name of the standby tool - If an eight-digit T-command code is set, then among the entire corresponding command group, only the number of the tool to be used will be displayed at TNo. TNo. **a** **(b)** **a**• Number of the standby tool **b**• Name of the standby tool (or if an eight-digit T-command code is set, then the group number)
•	MAGAZNPKNo.		—	Number of the indexed magazine pocket and name of the tool MAGAZNPKNo. **a** **(b)** **a**• Number of the indexed magazine pocket **b**• Name of the tool
•	PALLET No.		—	Identification number of the pallet being used (This data is displayed only when a patllet change function is provided.)
•	UNIT		—	Name of the program unit currently being executed UNIT: **a** **a**• Name of the unit
•	T-OFFSET		—	Number and amount of the tool diameter and tool length offset T-OFFSET D# **(a) b** H# **(a') b'** **a** • Number of the tool diameter offset **b** • Amount of the tool diameter offset **a'** • Number of the tool length offset **b'** • Amount of the tool length offset
•	INDEX		Degrees	Index angle of the table
•	•		—	Graphic presentation of the tool mounted on the spindle
•	SPDL		rpm	Number of revolutions per minute of the spindle
			m/min (feet/min)	Peripheral speed of the spindle
•	• •		—	Selected spindle gear number
•	FEED		mm/min (inch/min)	Current feed rate per minute
			mm/rev (inch/rev)	Current feed rate per revolution
•	Load meter		%	Load status of each axis
•	POSITION	X,Y,Z	mm (inch)	Current position of the tool tip in the workpiece coordinate system

No.	Item		Unit	Data description
		4,5,6	Degrees or mm (inch)	Current position of the tool tip in the workpiece coordinate system (only when an additional axis is provided)
•	MACHINE	X,Y,Z	mm (inch)	Current position of the machine in the machine coordinate system
		4,5,6	Degrees or mm (inch)	Current position of the machine in the machine coordinate system (only when an additional axis is provided)
•	BUFFER	X,Y,Z	mm (inch)	Moving distance of each axis in the next block to be executed
		4,5,6	Degrees or mm (inch)	Moving distance of an additional axis in the next block to be executed (only when an additional axis is provided)
㉑	REMAIN	X,Y,Z	mm (inch)	The remaining distance of movement of each axis that exists in the block currently being executed
		4,5,6	Degrees or mm (inch)	The remaining distance of movement of an additional axis that exists in the block currently being executed (only when an additional axis is provided)
㉒	NEXT COMMAND G,M,T,B		•	G, M, T, and B codes preset in the next block to be executed
㉓	WPC	X,Y,Z	mm (inch)	Workpiece zero-point position in the machine coordinate system
		4,5,6	Degrees or mm (inch)	Workpiece zero-point position in the machine coordinate system (only when an additional axis is provided)
㉔	WNo.		•	Work number and name of the program being executed WNo. **a b c** **(a' b') (c')** The meaning of data **a**, **b**, **a'** and **b'** is same as in •. Refer to the item •. **c, c'**: Name of the program (For a machine that displays a program name in 48 characters, only the first 16 characters are displayed.)
㉕	UNo.		•	- If the program being executed is a MAZATROL program: UNo. **a – a'** **a** : Number of the unit being executed **a'** : Number of the sequence being executed - If the program being executed is an EIA/ISO program: UNo. **a – a'** **a** : Number of the sequence being executed **a'** : Number of the block being executed
㉖	S, F, B, M, G		•	- Display of modal informations - Title and data of G codes which belong to group 15 and 20 are not displayed.

Note 1: The data items •BUFFER and ㉒NEXT COMMAND are not displayed for the block succeeding such blocks as follows:

- Block including G28
- Block consisting of EOB (End of Block) code only

During execution of a MAZATROL program, the data items are not displayed when the internal control commands are equivalent to the commands above.

Note 2: Commands which do not exist in the program can be displayed for outer corner machining with diameter offsetting, etc. because a connecting tool path is auto-inserted.

4-1-2 Data registration

On the **POSITION** display, the following data can be registered:

1. Parts count

A. Number of machined workpieces (a, item •)

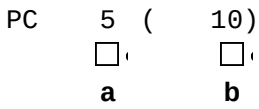
After machining, the total number of machined workpieces will be calculated and displayed at a of item •, if “1” (counting valid) is set in the NUMBER item of the end process in the program.

B. Target number of workpieces to be machined (b, item •)

This value is used for quantitative restriction of workpieces to be machined.
When the total number of machined workpieces reaches the value set here, the machine will stop.

(1) Place the cursor on the data item PC.

- To change the number of machined workpieces, move the cursor to position **a**.
- To set the target number of workpieces to be machined, move the cursor to position **b**.



(2) Set new data.

- Key in new data using the numeric data keys, and press the INPUT key to register the data.

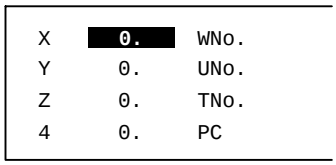
2. Counter setting

The area on the screen where the coordinates of the various axes are displayed is referred to as counter, and modification of the coordinates displayed on the counter is referred to as counter setting.
During the manual operation mode, the coordinates of each axis displayed on the current position counter, item •, can be freely changed as required.

(1) Select the manual operation mode.

(2) Place the cursor on the coordinate data to be changed.

- After calling up the cursor on the screen with the cursor key, move the cursor to the data you want to change.



(3) Set new coordinate data.

4-1-3 Display operations

1. Work number selection

Any of the work numbers displayed on the **POSITION** display (that is, the work numbers of the programs which can be used for automatic operation) can be selected.
The procedure is as follows:

(1) Press the **WORK No.** menu key.

➔ The menu item changes to a reverse display status and a work number listing window is displayed.

(2) Set a work number.

➔ The work number of the program which can be used for automatic operation will be selected.

2. Command display

Machine positions and modal information can be displayed. To check further details, see the item in Subsection **4-1-1 Data display** above.

(1) Press the **COMMAND** menu key.

➔ The menu item changes to a reverse display status and commands are displayed.

- Pressing the menu key once again releases the reverse status of the menu item and creates the normal display.

3. Shape drawing

The shape data in the selected program can be displayed in the window.
See the Section, **2-4 Windows** for further details.

4-2 TRACE Display

Function:

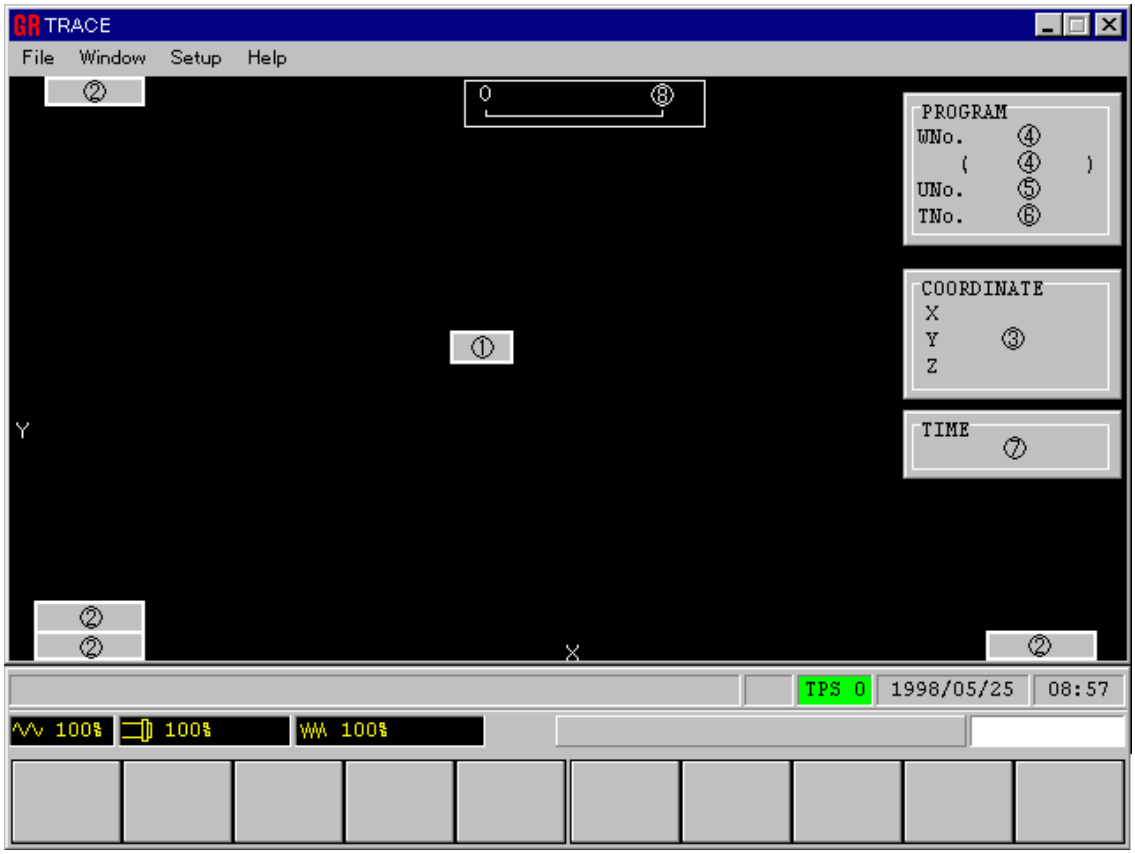
- This display can be used for displaying the operational status of machine in graphic form.
- This display is useful for displaying the tool movements synchronized with actual machine actions including an ATC action.
- Machining shape and path of the tool tip can be displayed on this display. Even in the locked status of machine, the movement of the tool indicated on this display will continue. The running status of the machining program can therefore be checked without actually operating the machine.

Remarks:

- A plane to be used to display machining shape and path of the tool tip can be designated.
- Scale values used for machining shape display and the tool tip path display can be changed.
- Data of the machining shape can be stored.

4-2-1 Data display

1. Displayed data



D735S1007E

2. Display data description

No.	Item	Unit	Data description
•	(Shape)	•	Machining shape that has been defined by the program (For point-machining, this data is displayed with a purple line. For line- or face-machining, this data is displayed with a green line.)
	(Tool path)	•	Path of the tool tip (The path of the tool tip is displayed with a yellow line, and the tool tip itself is displayed with a red • mark.)
•		mm (inch)	Display limit (automatically determined according to the machining shape)
•	X,Y,Z	mm (inch)	Current position of the tool tip in the machine coordinate system
•	WNo.	•	Work number of the program currently being executed.
	( )	•	If a subprogram is currently being executed, the work number of that subprogram will be displayed in parentheses
•	UNo.	•	Number and name of the program unit currently being executed
•	TNo.		Number and name of the tool mounted on the spindle
•	TIME	h: m' s"	Integral time elapsed during automatic cutting (With this data, it is possible to check the actual automatic operation time of the machining program.)
•	0 ⊢——⊣	mm (inch)	Scale value of graphic form (On the initial display, this data is automatically determined according to the machining shape.)

4-1-2 Displaying machining shape and tool tip

(1) Press the **WORK No.** menu key

➔ The display status of **WORK No.** will then be reversed.

(2) Press the numeric key(s) that corresponds to the work number of the program to be executed, and then press the INPUT key.

➔ The input work number will then be displayed in the WNo. item (date portion •) and the reverse-display status of **WORK No.** will be cleared.

(3) Press the **PART SHAPE** menu key.

➔ This will cause the display status of **PART SHAPE** to reverse and the machining shapes to be displayed continuously. After displaying the machining shapes, the reverse-display status of **PART SHAPE** will be cleared.

Note 1: If procedure described in Subsection **4-2-3 Plane select** below is carried out, then it becomes possible to select the plane or 3-D view on which machining shapes are to be displayed.
If no steps of this procedure are carried out, the machining shape available on the XY plane will be displayed.

Note 2: If procedure described in Subsection **4-2-4 Change of scale values** described below is carried out, it becomes possible for available machining shapes to be displayed in enlarged or reduced size.
If no steps of this procedure are carried out, the available machining shapes will be displayed with the scaling factor which has been automatically determined.

(4) Press the CYCLE START button.

➔ The lamp of the CYCLE START button will light up and the path of the tool tip (marked with •) synchronized with the machine action will be displayed.

- After the entire path of the tool tip has been displayed (i.e., after execution of the program has been completed), the lamp of the CYCLE START button will go out.

Note 1: The machining shape that has been displayed in procedural step (3) above can be erased by pressing the **SHAPE ERASE** menu key.

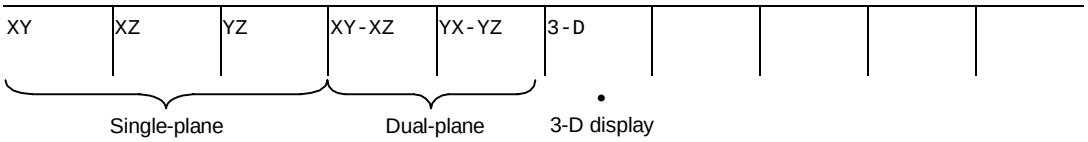
Note 2: The path of the tool tip can be erased by pressing the **PATH ERASE** menu key.

Note 3: If **STORE** is displayed reversely by depression of the **STORE** menu key, the shape data will be preserved permanently to prevent its destruction due to display changeover.

4-1-3 Plane select

(1) Press the **PLANE CHANGE** menu key.

➔ The following menu will then be displayed in the menu display area:



(2) Press the menu key that corresponds to the plane or 3-D view to be selected.

➔ The selected plane or 3-D view will then be displayed in the data display area.

- The particular machining shape or the path of the tool tip will be erased from the screen if plane or 3-D view changeover is carried out when the machining shape and the path of the tool tip are on the display.

(3) Press the **PART SHAPE** menu key.

➔ The available machining shapes will be continuously displayed on the selected plane or 3-D view.

4-1-4 Change of scale values

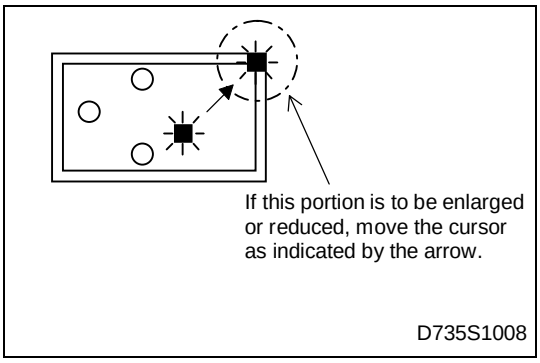
1. If a plane has been selected

The particular machining shape can be displayed in enlarged or reduced size if no plane has been selected (since the necessary XY plane has already been selected automatically) or if a different type of plane has been selected. In addition, a portion of the particular machining shape can be displayed in enlarged or reduced size.

(1) Press the **SCALE CHANGE** menu key.

➔ This will cause the display status of **SCALE CHANGE** to reverse and the cursor to appear in nearly the center of the screen.

(2) Press the cursor key to move the cursor to the central position of that portion to be enlarged or reduced.



Note: This operation is not required if the entire machining shape is to be enlarged or reduced.

(3) Specify new scale data with the appropriate numeric key(s), and then press the INPUT key.

➔ This will cause the reverse-display status of **SCALE CHANGE** to be cleared and the input scale data to be displayed in the scale display area (data portion •).

- Subsequently, the machining shape and the path of the tool tip will be erased automatically.

(4) Press the **PART SHAPE** menu key.

- The available machining shapes will be displayed continuously at the new scaling factor.

2. If 3-D view has been selected

If the **3-D** menu key has been pressed during plane-select operation, the 3-D display viewpoint can be changed. In addition, the entire machining shape can be displayed in enlarged or reduced size. However, partial enlargement or reduction of the machining shape is not possible.

(1) Press the **SCALE CHANGE** menu key.

➔ This will cause the display status of **SCALE CHANGE** to reverse and a cubic to be displayed in nearly the center of the screen.

(2) If the 3-D view is to be enlarged or reduced, carry out step (3) of 1. above.

- Proceed as follows if a change is to be made to the 3-D display viewpoint.

(3) Press the cursor key that corresponds to the direction in which the viewpoint is to be changed.

➔ This will cause erasure of the machining shape and tool-tip path that are currently being displayed.

- Each time the  key is pressed, the cubic rotates as if the viewpoint was shifted upward by five degrees.
- Each time the  key is pressed, the cubic rotates as if the viewpoint was shifted downward by five degrees.
- Each time the  key is pressed, the cubic rotates as if the viewpoint was shifted to the right by five degrees.
- Each time the  key is pressed, the cubic rotates as if the viewpoint was shifted to the left by five degrees.

(4) Press the INPUT key.

➔ The cubic display will then be cleared.

- In addition, the reverse-display status of **SCALE CHANGE** will be released and a new viewpoint registered.

(5) Press the **PART SHAPE** menu key.

➔ The machining shapes existing after the viewpoint has been changed will be displayed continuously.

4-3 EIA MONITOR Display

Function:

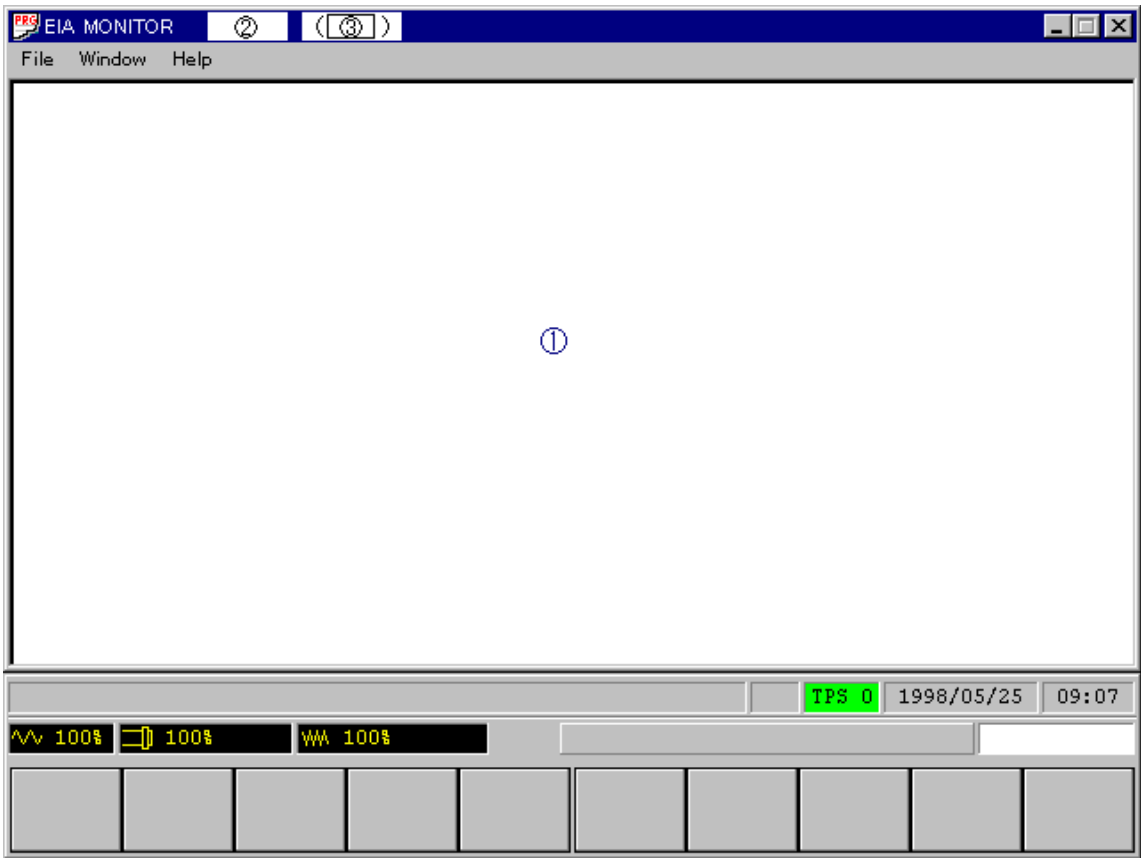
- The currently active block is displayed in a reverse-display status on the program list.
- During block stop, the program section that succeeds the block just completed can be edited.
- The program can be restarted by specifying its starting position using the cursor on the program list. Existing modal information, however, becomes invalid.

Remarks:

- This display is a special one for EIA/ISO programs.
- EIA/ISO function is an optional function.
- This display is not displayed when the machine is not provided with the EIA/ISO option.

4-1-1 Data display

1. Displayed data



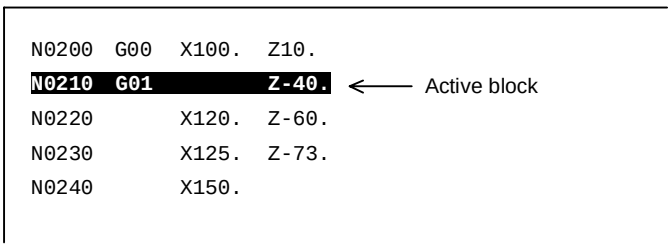
D735S1009E

2. Display data description

No.	Item	Unit	Data description
•	EIA/ISO program	•	Data of the EIA/ISO program. The currently active block is displayed in a reverse-display status.
•	Work number	•	Work number of the EIA/ISO program.
•	Program name	•	Name of the program displayed on the item •.

4-1-2 EIA/ISO monitor function

1. The EIA/ISO program that has been selected on the **POSITION** display are listed on this display.
   During automatic operation, the currently active block is displayed in a reverse-display status (the program scrolls as operation progresses, and the active section is continually displayed on the screen).



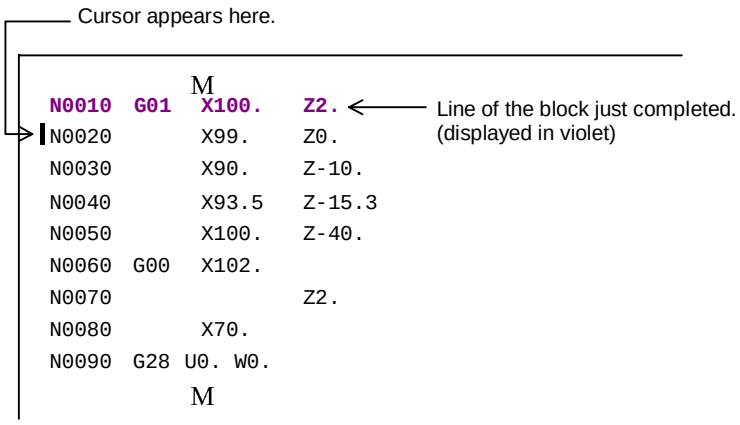
2. Monitoring of subprogram
   If the subprogram is an EIA/ISO program, display will change over to the subprogram.
   If the subprogram is a MAZATROL program, during execution of the MAZATROL program the block which calls subprogram of the main program will remain displayed in a reverse-display status.

4-1-3 Editing functions

(1) Bring the machine to a block stop during automatic operation (for example, press the FEED HOLD button).

(2) Press the **PROGRAM EDIT** menu key.

➔ The line of the block just completed will be displayed in violet, and the cursor will appeare at the beginning of the next line. (The area from the cursor position to the end of the program can now be edited.)



(3) Edit the program.

- Similar editing operations to EIA/ISO program editing operations can be carried out. Refer to the EIA/ISO Programming Manual for further details.

(4) Press the **PROGRAM COMPLETE** menu key.

(5) Execute automatic operation (press the CYCLE START button).

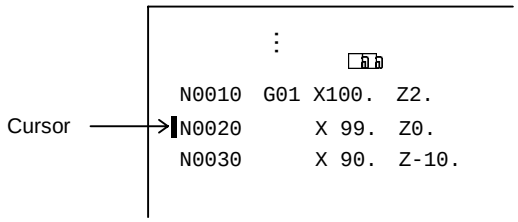
Note 1: Operation will progress according to the edited program.
Extra care is therefore required during program editing.

Note 2: An alarm will result if the CYCLE START button is pressed during program editing on this display. Before pressing the CYCLE START button, be sure to terminate the program editing operation by pressing the **ROGRAM COMPLETE** menu key.

4-1-4 Restart 2 unmodal function

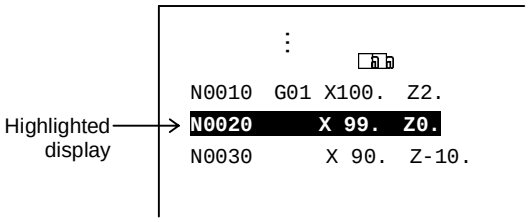
(1) Move the cursor to the block line to be selected.

- The position to be selected can be searched for using the **SEARCH** menu function as required. Refer to the EIA/ISO Programming Manual for the usage of the menu function.



(2) Press the **RESTART2 NONMODAL** menu key.

➔ The line where the cursor is present will change to a reverse-display status and the starting position will be set.



(3) Execute automatic operation (press the CYCLE START button).

➔ Automatic operation will restart from the reverse-display section.

Note 1: To cancel the selection, press the RESET key. The selection cannot be cancelled by changing over the display.

Note 2: To change the selection, repeat steps (1) and (2) above once again.

Note 3: The restart function and the restart 2 unmodal function differ as follows:

Restart function	Restart 2 unmodal function
- Program data is calculated from the head of the program up to the designated block, and during this time, modal information setting, program error checking, etc. are performed. - Also, since the coordinates present before the designated block have been stored, positioning at those coordinates is performed before the designated block is executed.	The entire program data before the designated block is ignored, and the program is restarted from that block, based on the modal information currently valid. Therefore, modal data should be previously designated. Before executing step (1) described above, set required modal data using the MDI mode. See the description on MDI for further details.

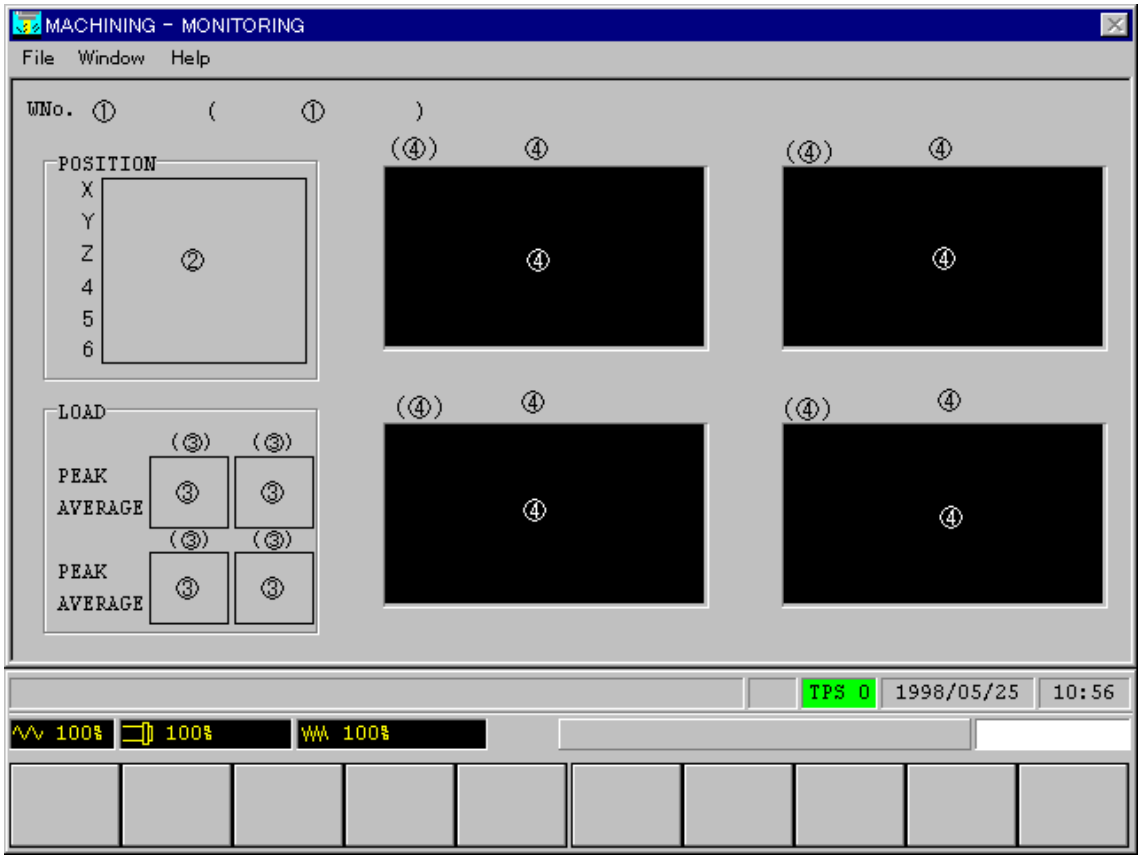
4-4 MACHINING - MONITORING Display

Function:

- Displaying the changes in the machining load of the spindle and each axis, in the spindle speed, and the output characteristics of the spindle.
- Setting the threshold for the cutting-load detection type of drill auto-pecking cycle.

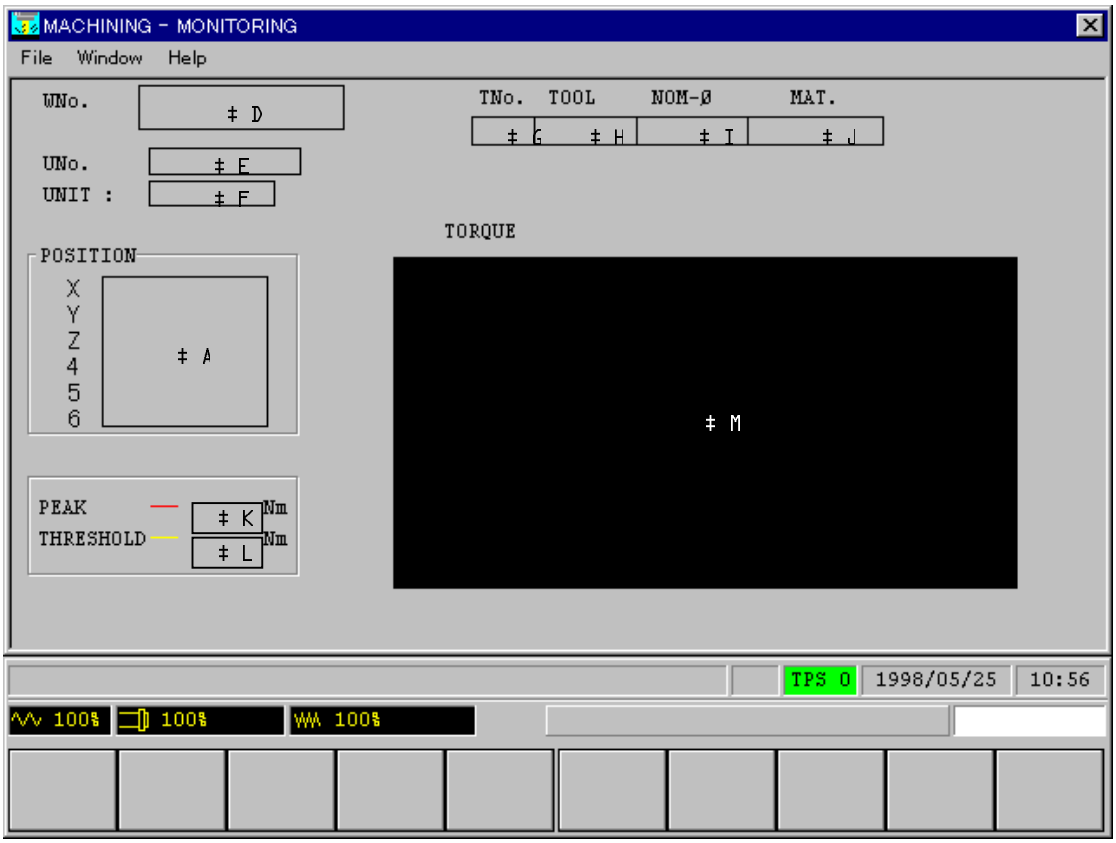
4-4-1 Data display

1. Displayed data with normal display of DRILL MONITOR menu item



D735S1101E

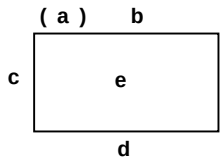
2. Displayed data with reverse display of DRILL MONITOR menu item

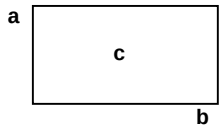


D735S1102E

3. Display data description

No.	Item		Unit	Data description
•	WNo.		•	- Work number of the program being executed WNo. **a b (c)** **a**: Work number **b**: Program type (MAZATROL or EIA/ISO) **c**: Program name
•	POSITION	X,Y,Z	mm (inch)	- Current position of the tool tip in the workpiece coordinate system
		4,5,6	mm (inch) or deg.	- Current position of the tool tip in the workpiece coordinate system (for additional axes)
•	LOAD		% or kW	- Peak and average load values for the monitoring data currently displayed. **(a)** PEAK **b** AVERAGE **c** **a**: Number of the corresponding graph **b**: Peak value **c**: Average value

No.	Item	Unit	Data description
•	Graphic display	•	- Graphic display of spindle output characteristics or changes in machining load of the spindle and feed axes.  **a**: Graph number **b**: Graph title 1. For a graph of spindle output characteristics **c**: Spindle output (kW) **d**: Spindle speed (min^{-1}) **e**: Spindle output characteristic curve (in red) with respect to the spindle speed. The output peak and average output (for the displayed data samplings) are indicated respectively by a red disk • (in a position of the abscissa corresponding to the spindle speed for the output peak) and a white triangle •. 2. For a graph of changes in machining load of the spindle and feed axes **c**: Load (%) **d**: Sampling time (sec) **e**: The load values applied during rapid traverse and cutting feed are respectively displayed in light blue and green lines. White and red lines refer to the average load and peak load during cutting feed.
•	WNo.	—	Work number of the program being executed WNo. **a b c** **(a' b') (c')** **a, a'** } Work numbers **b, b'** } Identification of MAZATROL program or EIA/ISO program **c, c'** } Name of the program * For data items **a**' and **b**', data is displayed only during execution of subprograms. * For machine that displays a program name in 48 character, only the first 16 characters are displayed for items **c** and **c'**.
•	UNo.	•	- If the program being executed is a MAZATROL program: UNo. **a – a'** **a** : Number of the unit being executed **a'** : Number of the sequence being executed - If the program being executed is an EIA/ISO program: UNo. **a – a'** **a** : Number of the sequence being executed **a'** : Number of the block being executed
•	UNIT	—	Name of the program unit currently being executed UNIT: **a** **a**• Name of the unit
•	TNo.	—	Number of the tool being mounted in the spindle
•	TOOL	—	Name (type) of the tool being mounted in the spindle
•	NOM-•	mm (inch)	Nominal diameter and suffix of the tool being mounted in the spindle
•	MAT.	—	Material of the tool being mounted in the spindle
•	PEAK	N•m	Peak value of the machining torque
•	THRESHOLD	N•m	Threshold for pecking by a drill (**Note 1**)

No.	Item	Unit	Data description
•	Graphic display	•	- Graphic display of the machining torque of the spindle.  **a**: Machining torque (N•m) **b**: Sampling time (sec) **c**: Machining torque of the spindle is displayed in a green line. Red and yellow lines refer respectively to the peak value (for the displayed data samplings) and the threshold for pecking by a drill.

Note 1: Threshold is only displayed when a drill is currently mounted on the spindle.

Note 2: The unit "N•m" denotes the torque (torsional moment) of the axis. If a disk with radius of 1 meter, for example, is rotated at a force of 1 newton, a torque of 1 N•m will be imposed on the axis of rotation.

4-1-2 Display modes of the MACHINING - MONITORING display

This display is provided with the following two display modes (to be alternated by the **DRILL MONITOR** menu function):

- Machining-monitoring mode
  Select this mode (with DRILL MONITOR normally displayed in the menu display area) to display the machining load of each axis and the output characteristics of the spindle.
- Drill-monitoring mode
  Select this mode (with DRILL MONITOR highlighted in the menu display area) to display the machining torque of the spindle and to set the threshold for the cutting-load detection type of drill auto-pecking cycle.

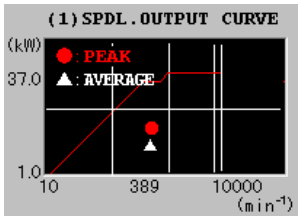
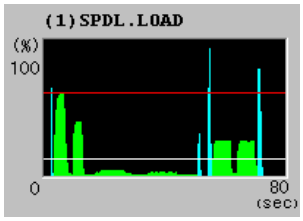
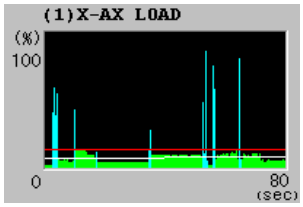
4-1-3 Selecting graphs to be displayed in the machining-monitoring mode

Two or four graphs can be displayed at the same time in the machining-monitoring mode.

- Four-graph display mode
  Four graphs are displayed squarely at the same time. Process monitoring data from the past of a maximum of 80 seconds to the present is displayed in this mode.
- Two-graph display mode
  Two graphs are displayed vertically on the display at the same time. Process monitoring data from the past of a maximum of 160 seconds to the present is displayed in this mode.

Note: Peak and average values displayed are those calculated from the monitoring data currently being displayed graphically.

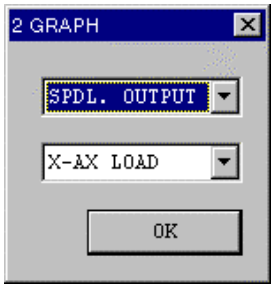
The following data can be displayed graphically in this mode:

	Display item	Display example	Description
1	SPDL. OUTPUT CURVE		- Output characteristics of the spindle. • Red line: Spindle output characteristics against the spindle speed. • Yellow line: Maximum rotational speed for the current tool (only displayed for a valid setting [> 0]). • Red disk •: Crossing point of the maximum output value and the corresponding spindle speed (within the displayed data samplings). • White triangle •: Crossing point of the average output value of the displayed data samplings and the current spindle speed.
2	SPDL. LOAD		- Changes in the machining load of the spindle. • Blue line: Load value applied during rapid traverse. • Green line: Machining load values applied during cutting feed. • Red line: Maximum machining load value applied during cutting feed. • White line: Average machining load value applied during cutting feed.
3	X-AX LOAD Y-AX LOAD Z-AX LOAD		- Changes in the machining load of one of the three fundamental axes. The meaning of lines is the same as for the graph of the spindle machining load.
4	4-AX LOAD 5-AX LOAD 6-AX LOAD		- Changes in the machining load of an additional axis. The meaning of lines is the same as for the graph of the spindle machining load. (Not available for a machine without additional axes)

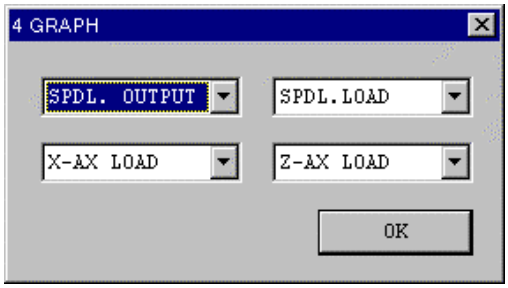
(1) Press the **2 GRAPH** (or **4 GRAPH**) menu key.

➔ The display of the menu item will be reversed and the following dialogue box for graph selection will appear:

(A) 2-graph display mode

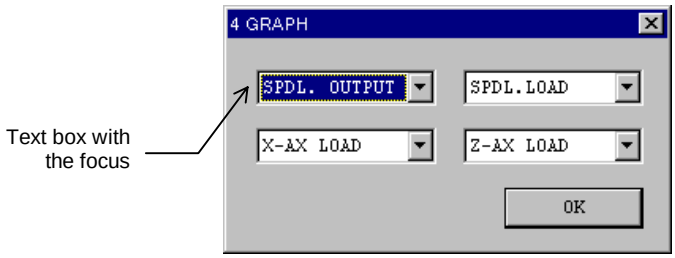


(B) 4-graph display mode



(2) Using the Tab key and the cursor keys, select the type of graph for each display section.

- Move the focus (reverse display) to the text box corresponding to the desired display area by using the Tab key, and then select the type of the desired graph by using the cursor keys.



(3) Press the INPUT key.

➔ The dialogue box for graph selection will be closed, the reverse display of the menu item cleared and the graphic display will be made now in each section according to the settings.

Note 1: The same data cannot be displayed on multiple graph sections. The setting for such a display will cause the alarm **401 ILLEGAL FORMAT** to be given.

Note 2: Selection of graph type can be repeated as often as required in the same display mode.

4-1-4 Setting the threshold in the drill-monitoring mode

In the drill-monitoring mode, the pecking threshold level for the cutting-load detection type of drill auto-pecking cycle can be set monitoring the machining torque of the spindle. This drill auto-pecking cycle will execute the pecking process only when the machining torque exceeds the threshold level. The setting procedure is as follows:

(1) Drill an actual workpiece using the tool for which the threshold is to be set.

- Select a drilling hole depth that does not cause clogging with any chips. The drilling hole depth should not exceed the value of [3 × tool diameter of the drill].

(2) Press the **BORDER VALUE** menu key.

(3) Enter the desired threshold seeing the graph-displayed machining torque and peak torque.

(4) Press the INPUT key.

➔ The entered pecking threshold level is displayed in a yellow solid line on the graph.

➔ The same value is set in the corresponding item on the **TOOL DATA** display.

Note 1: Even if the threshold is changed during execution of the cutting-load detection-type drill auto-pecking cycle, that new value will not become valid until the machining cycle is completed.

Note 2: Do not perform the cutting-load detection type of drill auto-pecking cycle if the drill has a tool diameter of 4 mm or less or is made of a material other than high-speed steel. Otherwise, tool damage may result since the machining torque cannot be detected accurately under those conditions.

4-1-5 Scale change in the drill-monitoring mode

1. Automatic vertical scale change

When a new drill is mounted onto the spindle, the scale for the vertical axis of the cutting torque graph is changed automatically. The cutting resistance is calculated theoretically from the tool diameter of the drill and then the optimum scale is selected automatically from nine types (1, 2, 5, 10, 20, 50, 100, 200 and 500 N•m).

Note 1: The scale will not be changed if a non-drilling tool is mounted onto the spindle.

Note 2: If automatic scale change has not occurred in the past, 10 N•m will be selected as the initial value.

2. Manual vertical scale change

The scale for the vertical axis of the cutting torque graph can be changed to the desired value in manual mode. The procedure is as follows:

(1) Press the **SCALE CHANGE** menu key.

(2) Enter the desired scale.

- The data that can be entered ranges from 0 to 655.35 N•m.

(3) Press the INPUT key.

➔ The graph is cleared and then a new one is displayed with the new scale.

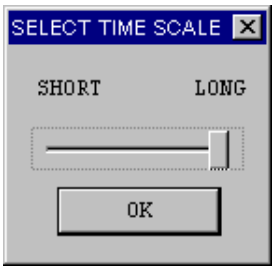
➔ During automatic operation, the graph is cleared temporarily and monitoring commences from the beginning of the graph with the new scale.

4-1-6 Setting the display time scale

The time scale for graphic display can be changed in steps as required for an overview or an enlarged view in detail.

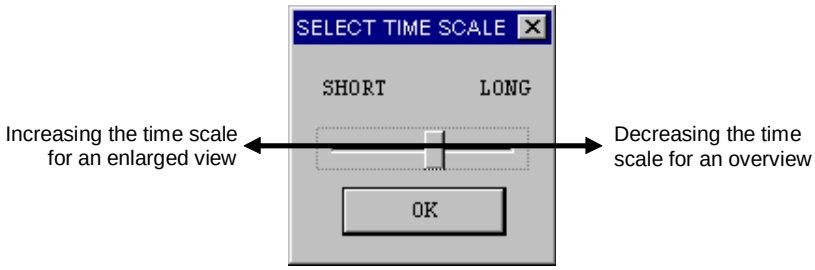
(1) Press the **CHANGE TIME** menu key.

➔ The display of the menu item will be reversed and the following dialogue box for time scale selection will appear:



(2) Using the cursor keys, select a time scale as required.

- The knob displayed in the dialogue box can be moved by pressing a cursor key. Moving the knob to the right (LONG) or the left (SHORT) prolongs or shortens the time which corresponds to the right end of the abscissa for the graphs concerned.



- The display time can be selected from among eight or seven steps (for 2- or 4-graph display mode) as shown below:

LONG ↕ SHORT

Step	Drill-monitoring mode and Machining-monitoring 2-graph display mode	Machining-monitoring 4-graph display mode
1	160	80
2	80	40
3	40	20
4	20	10
5	10	5
6	5	2.5
7	2.5	1.3
8	1.3	

Unit: sec

(3) Press the INPUT key.

➔ The dialogue box for time scale selection will be closed, the reverse display of the menu item cleared and the graphic display will be made now in each section according to the settings.

- NOTE -

5 DISPLAYS RELATED TO MACHINING SET UP

5-1 TOOL LAYOUT Display

Function:

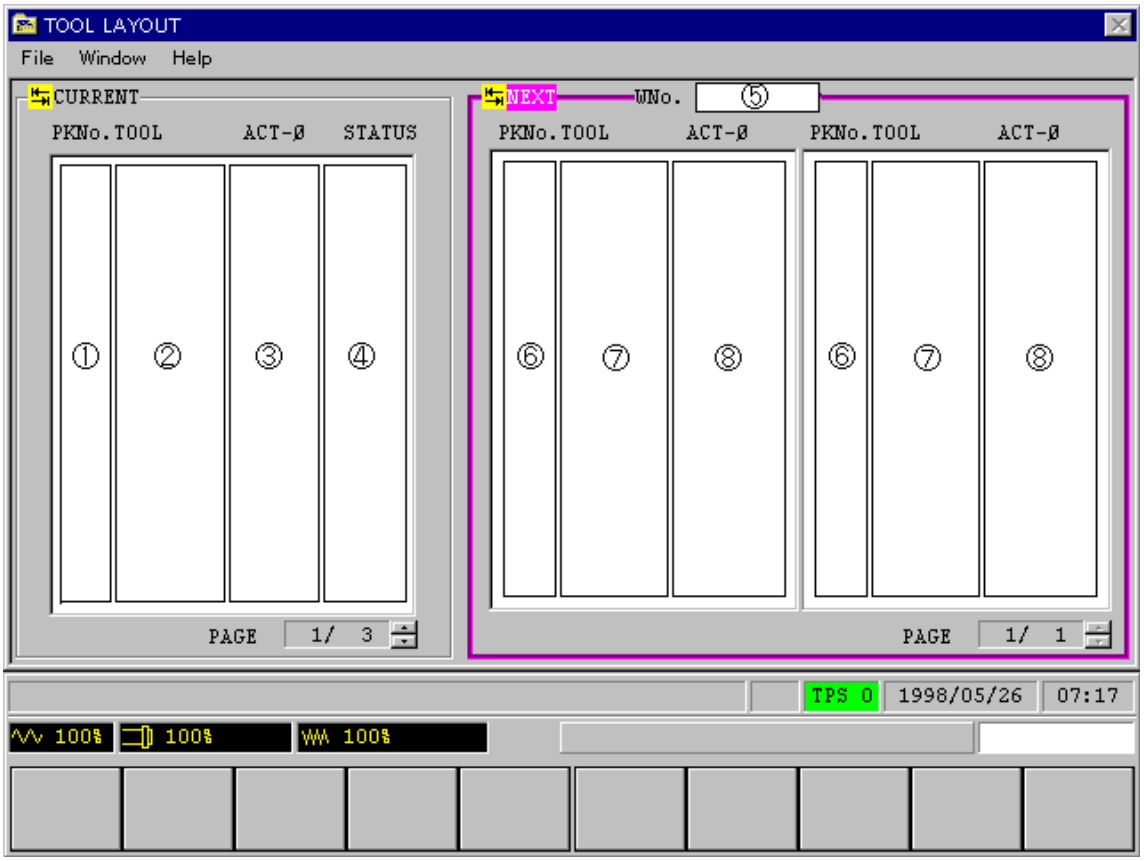
- This display is used on the screen when assigning pocket numbers to the tools to be used for machining or erasing the registered tool names.

Remarks:

- The operation of assigning pocket numbers to the tools to be used is referred to as tool layout.
- The **TOOL LAYOUT** display can be used only when executing a MAZATROL program.

5-1-1 Data display

1. Displayed data



D735S1011E

2. Display data description

NO.	ITEM	UNIT	Data description
•	CURRENT PKNO.	•	Pocket number for the current tools
•	CURRENT TOOL	•	Tool name for the current tools
•	CURRENT ACT-ϕ	mm (inch)	Nominal diameter of the current tools
		•	Suffix (code that identifies tools of the same nominal diameter) for the current tools
•	STATUS	•	Information upon the status of the current tools
•	WNo.	•	Work number of the specified program
•	NEXT PKNo.	•	Pocket number for the next tools
•	NEXT TOOL	•	Tool name for the next tools
•	NEXT ACT-ϕ	mm (inch)	Nominal diameter of the next tools
		•	Suffix (code that identifies tools of the same nominal diameter) for the next tools

Note 1: The CURRENT display indicates the names and nominal diameters of the tools which already are each assigned a pocket number. That is, the CURRENT display indicates the same status of tool registration as with the **TOOL DATA** display for the current program.

Note 2: The NEXT display indicates the names and nominal diameters of all the tools necessary to execute the specified machining program.

5-1-2 Tool layout procedure

(1) Call up the **TOOL LAYOUT** display on the screen.

➔ The names and nominal diameters of the tools which have already been registered will be indicated on the CURRENT display. (No display will be presented if no tools have been registered.)

- The **WORK No.** item will have already been placed in a reverse-display status by the time that the display is called on the screen.

(2) Using the appropriate numeric key(s), specify the work number of the machining program to be executed. Then, press the INPUT key.

➔ This will cause the reverse-display status of **WORK No.** to be cleared and the input work number to be displayed in the WORK No. item (data portion•).

- The names and nominal diameters of all the tools to be used under the program whose work number has been input will be indicated on the NEXT display. Under the PKNo. item, 0 will be displayed for any tool name.

(3) Subsequent operations differ according to several factors such as the registration status of the tools which are already registered on the CURRENT display. However, separate or combined use of the three methods described below makes it possible in any cases to assign pocket numbers to all the tools to be used. Select the most appropriate method according to the particular tool-registration or other status.

A. Pocket-number shift

B. Automatic pocket-number assignment

C. Manual pocket-number assignment

A. Pocket-number shift

If there are the same tools as those which are already registered on the CURRENT display among the tools being indicated on the NEXT display, it is no longer necessary then to assign pocket numbers to such tools. That is, the pocket numbers that have already been assigned can be shifted as they are. This method is referred to as pocket-number shift. Once a pocket-number shift operation has been carried out, it suffices just to assign pocket numbers to the tools for which no shift operations have been carried out.

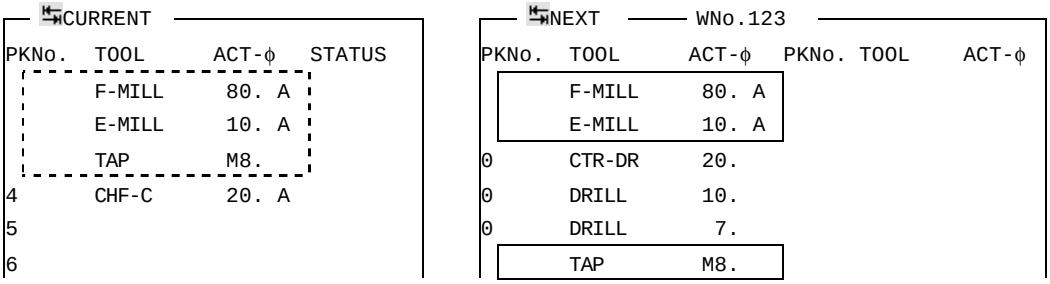
(A-1) Press the **PKNo. SHIFT** menu key.

➔ This will cause the display status of **PKNo. SHIFT** to reverse.

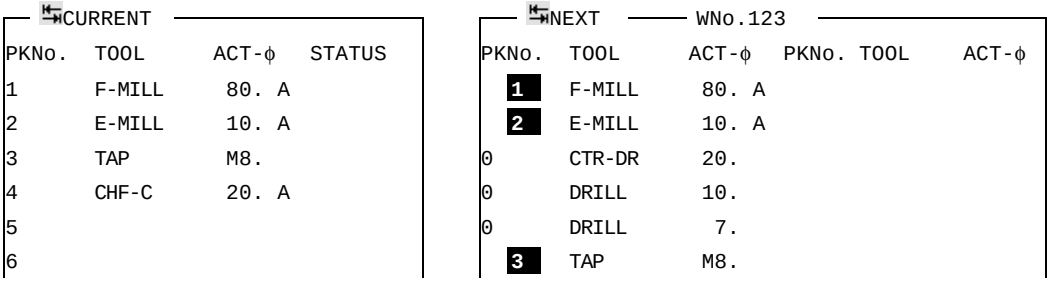
(A-2) Press the INPUT key.

➔ If there are the same tools as those which are already registered on the CURRENT display among the tools being indicated on the NEXT display, then the pocket numbers already assigned under PKNo. of the corresponding tool names will be shifted automatically to the **NEXT** display and will be indicated in reverse-display status.

Example: If the following data is displayed when the work number of a machining program has been input in the procedural step (2) above:



Of the tools being indicated on the NEXT display, a face-mill with a nominal diameter of 80 mm, end-mill with a nominal diameter of 10 mm, and a tap with a nominal diameter of 8 mm are already registered on the CURRENT display (in dotted line). Here, if procedural steps (A-1) and (A-2) are carried out, then the display will change as follows:



The pocket numbers on the CURRENT display will shift automatically and the display status of the pocket numbers on the NEXT display will be reversed.

➔ Subsequently, the reverse-display status of **PKNo. SHIFT** will be cleared.

Note: In the above example, when pocket numbers are assigned to all the remaining tools (that is, a spot drill with a nominal diameter of 20 mm, a drill with a nominal diameter of 10 mm, and a drill with a nominal diameter of 7 mm) by method B. or C. described below, the pocket-number assignment procedure for all the tools will be completed.

B. Automatic pocket-number assignment

This method is one in which, among the tool names being displayed on the NEXT display, all

those with pocket numbers of 0 (undetermined) are automatically assigned serial pocket numbers, starting with the top tool name on the display first.

Note: In this method, the data being indicated on the CURRENT display will be disregarded.

(B-1) Press the **PKNo. ASSIGN** menu key.

➔ This will cause the display status of **PKNo. ASSIGN** to reverse.

(B-2) Press the INPUT key.

➔ Serial pocket numbers will then be automatically assigned to each tool that has a pocket number of 0 (undetermined), starting with the top one on the display first.

Example 1:

When the names of the tools to be used are being indicated on the NEXT display as shown below:

Here, if procedural steps (B-1) and (B-2) are carried out, then the display will change as follows:

NEXT WNo.123

PKNo.	TOOL	ACT-ϕ	PKNo.
0	F-MILL	80.	A
0	E-MILL	10.	A
0	CTR-DR	20.	
0	DRILL	10.	
0	DRILL	7.	
0	TAP	M8.	

NEXT WNo.123

PKNo.	TOOL	ACT-ϕ	PKNo.	TOOL	ACT-ϕ
1	F-MILL	80.	A		
2	E-MILL	10.	A		
3	CTR-DR	20.			
4	DRILL	10.			
5	DRILL	7.			
6	TAP	M8.			

Example 2:

In the example shown below, if pocket numbers are already assigned to the face-mill with a nominal diameter of 80 mm, the end-mill with a nominal diameter of 10 mm, and the tap with a nominal diameter of 8 mm (see the example in the description of “Pocket-number shift” above):

Here, if procedural steps (B-1) and (B-2) are carried out, then the display will change as follows:

NEXT WNo.123

PKNo.	TOOL	ACT-ϕ	PKNo.
1	F-MILL	80.	A
2	E-MILL	10.	A
0	CTR-DR	20.	
0	DRILL	10.	
0	DRILL	7.	
3	TAP	M8.	

NEXT WNo.123

PKNo.	TOOL	ACT-ϕ	PKNo.	TOOL	ACT-ϕ
1	F-MILL	80.	A		
2	E-MILL	10.	A		
4	CTR-DR	20.			
5	DRILL	10.			
6	DRILL	7.			
3	TAP	M8.			

Except the pocket numbers already assigned (i.e., pocket number 1, 2, and 3), serial ones will be assigned starting with the top tool name on the display first.

➔ Subsequently, the reverse-display status of **PKNo. ASSIGN** will be cleared.

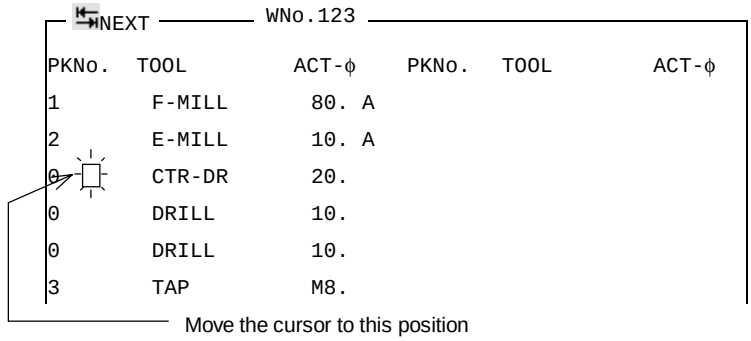
C. Manual pocket-number and tool interference data assignment

This method is one in which the tools being indicated on the NEXT display are respectively assigned serial pocket numbers and tool interference data one by one.

(C-1) Press the cursor key to call the cursor on the screen.

(C-2) Press the the cursor key to move the cursor to the row of the tool name to which a pocket number must be assigned is displayed.

Example: If the screen on the NEXT display is presented as shown below (see the example given in the description of “Pocket-number shift” above) and pocket number 4 and tool interference data are to be assigned to a spot drill with a nominal diameter of 20 mm:



(C-3) Using the appropriate numeric key(s), specify the pocket number to be assigned. Then, press the INPUT key.

➔ The input pocket number will then be displayed in the position where the cursor is blinking.

(C-4) Pressing the cursor key moves cursor to the right of the nominal diameter data suffix.

➔ The following menu will then be displayed.

ORDINARY DIAMETER	LARGE L	MAG DIR + +	MAG DIR - -	SMALL S					

The menu shows types of intra-magazine interference data:

ORDINARY DIAMETER..... Tool which completely fits into that pocket.

LARGE L ..... Tool which interferes with both next tools in the pocket.

MAG DIR + +.....Tool that interferes with the tool of a larger pocket number.

MAG DIR – – ..... Tool that interferes with the tool of a smaller pocket number.

SMALL S..... Diameter of a tool which can be placed in a pocket other than that which interfere with the tool indicated by “MAG DIR +” or “MAG DIR –”.

(C-5) Press the menu key that corresponds to the tool interference data you want to set.

➔ The tool interference data will be displayed at the cursor.

- In the example shown above, press the **LARGE L** menu key.

The display will then change as follows:

NEXT WNo.123

PKNo.	TOOL	ACT-ϕ	PKNo.	TOOL	ACT-ϕ
1	F-MILL	80. A			
2	E-MILL	10. A			
4	CTR-DR	20. L			
0	DRILL	10.			
0	DRILL	7.			
3	TAP	M8.			

Note: By repeating procedural steps (C-2) to (C-5), it is possible to assign the desired pocket numbers to all tools whose pocket number display is 0 (undetermined). However, it is not possible to assign the same pocket number as that which has already been used on the NEXT display.

After pocket numbers have been assigned to all the required tools on the NEXT display by one or more of the methods A., B. and C., proceed with the operations described below.

(4) Press the **LAYOUT FINISH** menu key.

➔ This will cause the display status of **LAYOUT FINISH** to reverse.

(5) Press the INPUT key.

➔ The names of the tools to which pocket numbers have been assigned on the NEXT display will then be indicated on the CURRENT display in order of pocket number. The display on the NEXT display will remain unchanged.

Note: Even if the pocket numbers that have been assigned on the NEXT display are already used on the CURRENT display, the tool names with the former will govern and those with the latter will be erased.

Example: If the data on the CURRENT and NEXT displays is as shown below:

CURRENT

PKNo.	TOOL	ACT-ϕ	STATUS
1	F-MILL	80. A	
2	E-MILL	10. A	
3	TAP	M8.	
4	CHF-C	20. A	
5			
6			

NEXT WNo.123

PKNo.	TOOL	ACT-ϕ	PKNo.	TOOL	ACT-ϕ
1	F-MILL	80. A			
2	E-MILL	10. A			
4	CTR-DR	20.			
5	DRILL	10.			
6	DRILL	7.			
3	TAP	M8.			

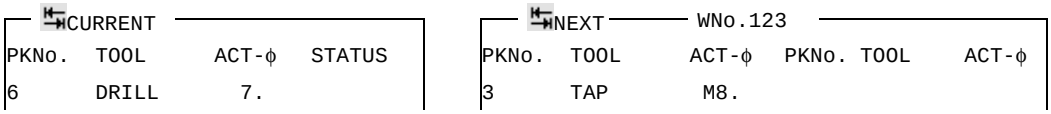
Here, if procedural steps (4) and (5) are carried out, the display will change as follows:

CURRENT

PKNo.	TOOL	ACT-ϕ	STATUS
1	F-MILL	80. A	
2	E-MILL	10. A	
3	TAP	M8.	
4	CTR-DR	20.	
5	DRILL	10.	

NEXT WNo.123

PKNo.	TOOL	ACT-ϕ	PKNo.	TOOL	ACT-ϕ
1	F-MILL	80. A			
2	E-MILL	10. A			
4	CTR-DR	20.			
5	DRILL	10.			
6	DRILL	7.			



That is, the tool names with pocket numbers assigned on the NEXT display will be indicated in order of pocket number on the CURRENT display.
The chamfering cutter with a nominal diameter of 20 mm that has been registered under pocket number 4 will be erased automatically (see **Note** above).

➔ The reverse-display status of **LAYOUT FINISH** will be cleared.

All the tool layout operations have been finished.

Note: If, after you have pressed the **LAYOUT FINISH** menu key following selection of intra-magazine interference data, tools that are likely to interfere with other tools or pockets are found, then the layout operation will stop in the middle and the data display of the tools that are likely to interfere will be highlighted red.

5-1-3 Clearing pocket numbers

If a wrong pocket number has been assigned during tool layout on the NEXT display, carry out the layout operation once again after resetting all the tool pocket numbers to 0 using the following procedure.

(1) Press the **PKNo. CLEAR** menu key.

➔ This will cause the display status of **PKNo. CLEAR** to reverse.

(2) Press the INPUT key.

➔ All the pocket numbers that have been assigned on the NEXT display will be reset to 0 and the reverse-display status of **PKNo. CLEAR** will be released.

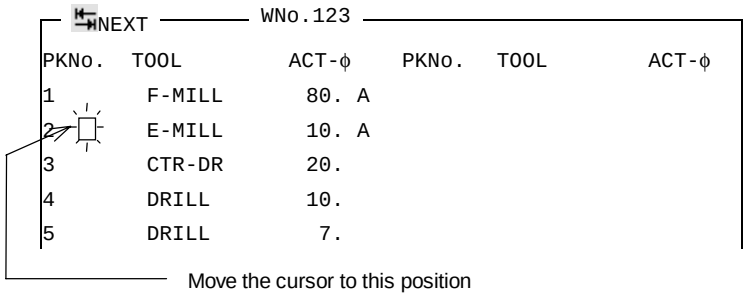
5-1-4 Specifying a spare tool

If the tool data being indicated on the NEXT display includes the data of the tool which needs a spare tool, it is possible to specify the spare tool on this display.

(1) Press the cursor key to call the cursor on the NEXT display.

(2) Press the cursor key to move the cursor to the PKNo. data item of the tool which needs a spare tool.

Example: When the data shown below is indicated on the NEXT display and a spare tool is to be specified for the end-mill with a nominal diameter of 10 mm:



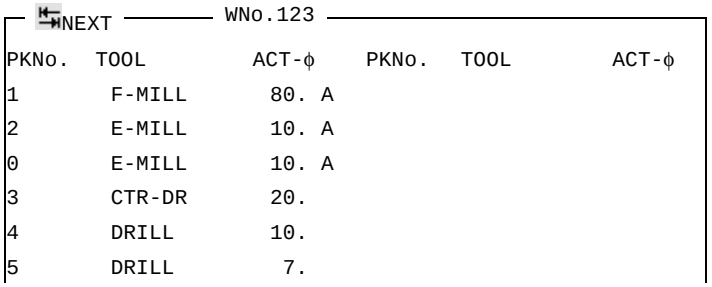
(3) Press the **SPARE T ADDITION** menu key.

➔ This will cause the display status of **SPARE T ADDITION** to reverse.

(4) Press the INPUT key.

➔ The data of a spare tool will be displayed directly under the data of the tool which needs a spare tool.

- In the example above, the display will change as follows:



➔ The reverse-display status of **SPARE T ADDITION** will be released.

Note: The pocket number of a spare tool is displayed as 0 (undetermined), as shown above. Therefore, a pocket number must be assigned to the spare tool after the above operation has been carried out.

5-1-5 Erasing spare tool data

Proceed as follows to erase specified spare tool data:

(1) Press the cursor key to move the cursor to the PKNo. item of the spare tool data to be erased.

- First call the cursor on the screen if it is not being indicated on the NEXT display.

(2) Press the **SPARE T ERASE** menu key.

➔ This will cause the display status of **SPARE T ERASE** to reverse.

(3) Press the INPUT key.

➔ The spare tool data being indicated on the NEXT display will be erased, and the reverse-display status of **SPARE T ERASE** will be released.

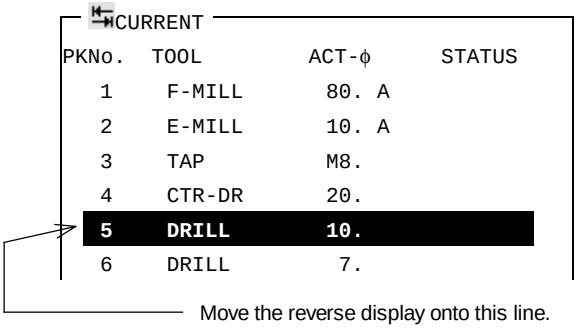
5-1-6 Erasing procedures for registered tool data

All or part of the tool data registered on the CURRENT display of the **TOOL LAYOUT** display, (i.e., the tool data that has already been laid out) can be erased.

1. Partial erasure of registered tool data

(1) Press the cursor key to move the cursor (here: reverse display) to the line of the tool data to be erased.

Example: When tools are registered as shown below and the data of the drill with a nominal diameter of 10 mm is to be erased:



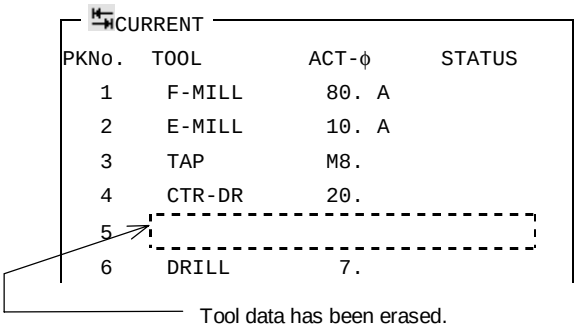
(2) Press the **TOOLDATA ERASE** menu key.

➔ This will cause the display of **TOOLDATA ERASE** to reverse.

(3) Press the INPUT key.

➔ The data of the tool on the reverse-display line will be erased.

- In the example above, the display will change as follows:



➔ The reverse-display status of **TOOLDATA ERASE** will be released.

2. Total erasure of registered tool data

(1) Press the menu selector key to display **LAYOUT CANCEL**.

(2) Press the **LAYOUT CANCEL** menu key.

➔ This will cause the display status of **LAYOUT CANCEL** to reverse.

(3) Specify –9999 with the numeric keys and then press the INPUT key.

➔ The entire tool data that has been registered on the CURRENT display will then be erased and the reverse-display status of **LAYOUT CANCEL** will be released.

5-1-7 Other functions

1. TOOL SEARCH

This menu data should be used to search for a tool name that is registered on the CURRENT display.
The operating procedure is similar to that used for **TOOL SEARCH** of the **TOOL DATA** display.
For details, see the Section **7-1 TOOL DATA Display**.

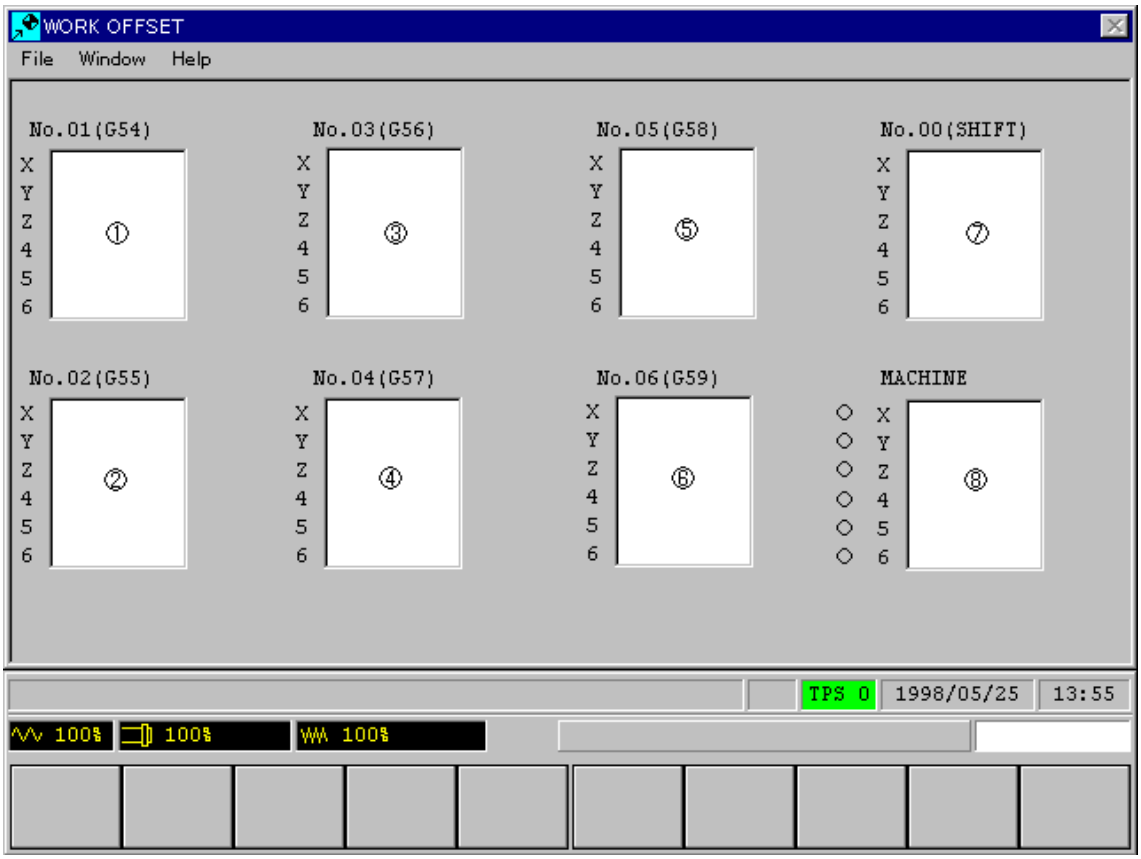
5-2 WORK OFFSET Display

Function:

Setting zero-point offset for six workpiece coordinate systems (G54 to G59) for EIA/ISO programs.

5-2-1 Data display

1. Displayed data



D735S1012E

2. Display data description

No.	Item	Unit	Data description
•	X, Y, Z	mm (inch)	Distance from the machine zero-point to the zero-point in G54 workpiece coordinate system on each axis (Offset of workpiece zero-point)
	4, 5, 6	degrees or mm (inch)	
•	X, Y, Z	mm (inch)	Distance from the machine zero-point to the zero-point in G55 workpiece coordinate system on each axis (Offset of workpiece zero-point)
	4, 5, 6	degrees or mm (inch)	
•	X, Y, Z	mm (inch)	Distance from the machine zero-point to the zero-point in G56 workpiece coordinate system on each axis (Offset of workpiece zero-point)
	4, 5, 6	degrees or mm (inch)	
•	X, Y, Z	mm (inch)	Distance from the machine zero-point to the zero-point in G57 workpiece coordinate system on each axis (Offset of workpiece zero-point)
	4, 5, 6	degrees or mm (inch)	
•	X, Y, Z	mm (inch)	Distance from the machine zero-point to the zero-point in G58 workpiece coordinate system on each axis (Offset of workpiece zero-point)
	4, 5, 6	degrees or mm (inch)	

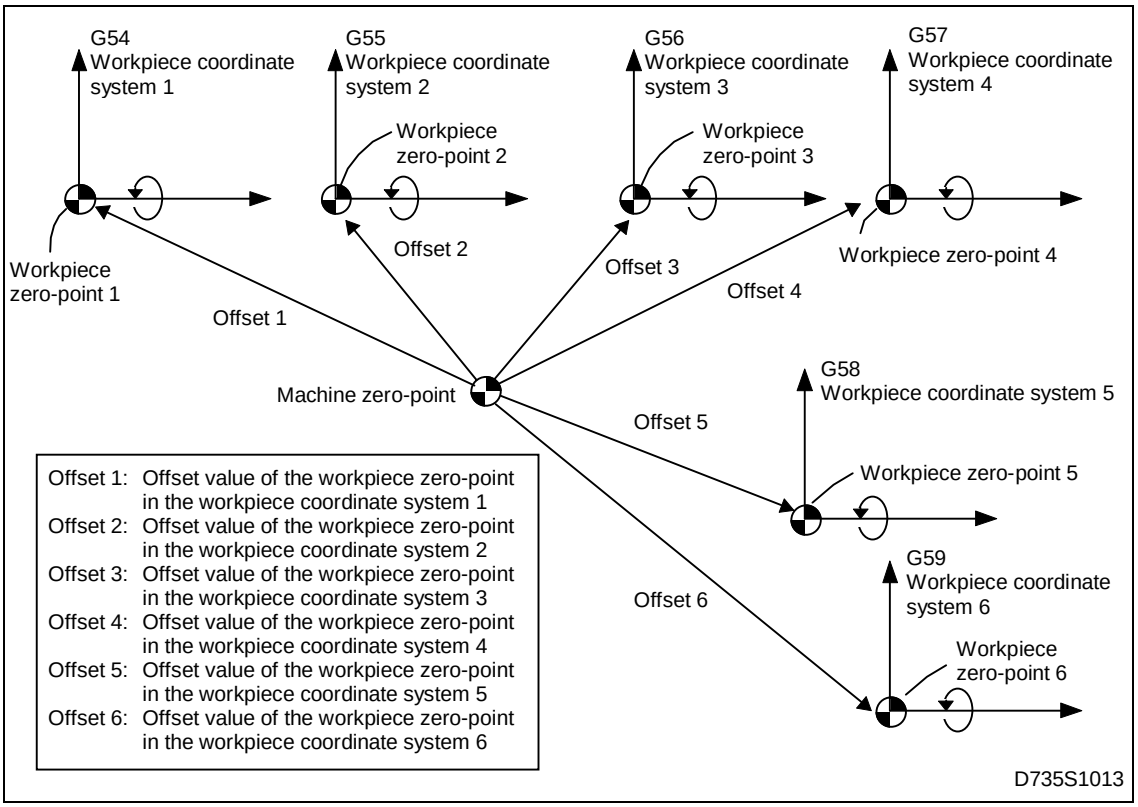
No.	Item	Unit	Data description
•	X, Y, Z	mm (inch)	Distance from the machine zero-point to the zero-point in G59 workpiece coordinate system on each axis (Offset of workpiece zero-point)
	4, 5, 6	degrees or mm (inch)	
•	X, Y, Z	mm (inch)	Shift amount of workpiece zero-point in each workpiece coordinate system.
	4, 5, 6	degrees or mm (inch)	
•	X, Y, Z	mm (inch)	The same as the MACHINE item on the **POSITION** display. See the Section, **POSITION Display**.
	4, 5, 6	degrees or mm (inch)	

5-1-2 Data registration

The following data can be registered on the **WORK OFFSET** display.

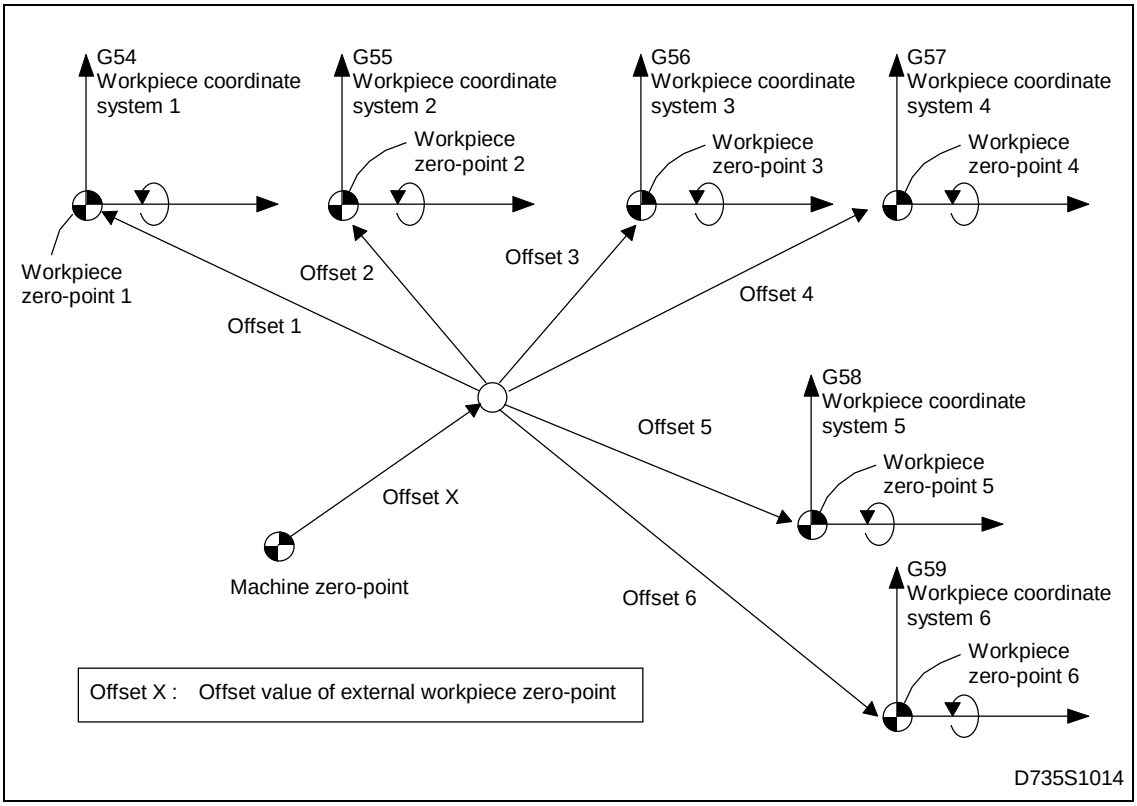
1. Offset value of the zero-point of the workpiece coordinate systems G54 to G59 (items • to •)

Offset value of the workpiece zero-point is the offset value from the machine zero-point to the workpiece zero-point.



2. Shift amount of the workpiece coordinate systems (item •)

Offset value of the external workpiece zero-point for shifting all the workpiece coordinate systems G54 to G59.



3. Procedure for setting offset value of workpiece zero-point

A. If known offset value of workpiece zero-point is to be set

(1) Place the cursor on the axis position of the coordinate system where data setting is required.

- Press the cursor key to call the cursor on the screen. Then, move the cursor to the intended position.

(2) Set the offset value.

- Key in the known data using the numeric data keys, and then press the INPUT key.

(3) Repeat the steps (1) and (2) until entire data of the intended axes have been set.

B. If unknown offset value of workpiece zero-point is to be set

If the offset to be set is unknown, the data can be set using the TEACH (tool-nose position data store) function.

(1) Mount a tool whose length has already been registered.

- For information on tool length, see the Section, **TOOL DATA Display.**

(2) Bring the tool nose to the intended workpiece zero-point.

- Move each axis under manual operation until the tool nose has reached the workpiece zero-point.

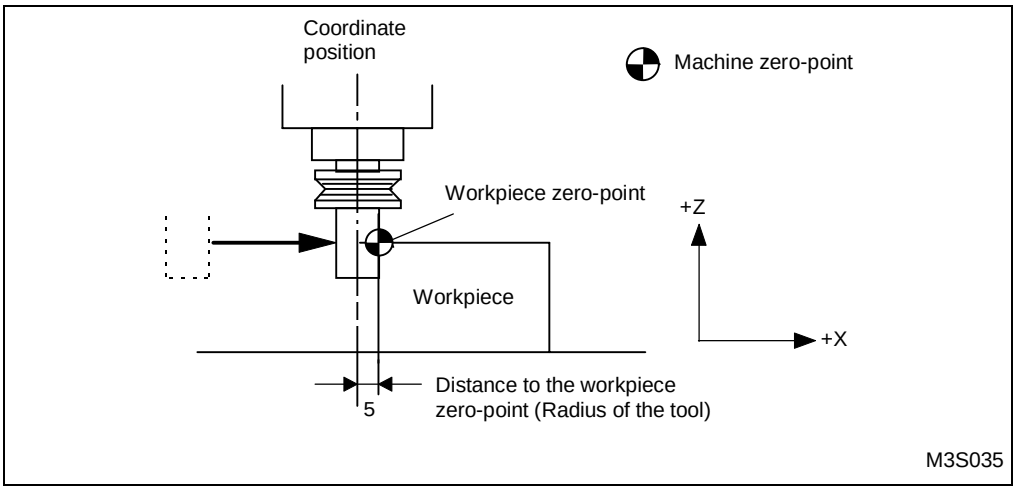
(3) Place the cursor on the intended axis postion of the coordinate system where data setting is required.

- Press the cursor key to call up the cursor on the CRT screen and move the cursor to the axis position of the coordinate system where the data is to be set.

(4) Press the **TEACH** menu key.

➔ Pressing this menu key makes **TEACH** assume a reverse-display status.

(5) Set the distance from the workpiece zero-point to the tool center line (tool nose for Z-axis) in the intended axis direction.



➔ The offset value will be set automatically and the reverse-display status of **TEACH** will be cleared.

4. Setting shift amount of the workpiece coordinate systems

The amount of shifting can be set by setting a workpiece zero-point as shift coordinates, in a similar manner described previously in the item, “3. Procedure for setting offset value of workpiece zero-point”.

Note: Using the **INCRMENT INPUT** menu function, desired data can be set by selecting an incremental or decremental value for the existing data. See the Section, **TOOL DATA Display**, for further details.

5-3 MACRO VARIABLE Display

Function:

This display allows macro variables to be set for the user macros.

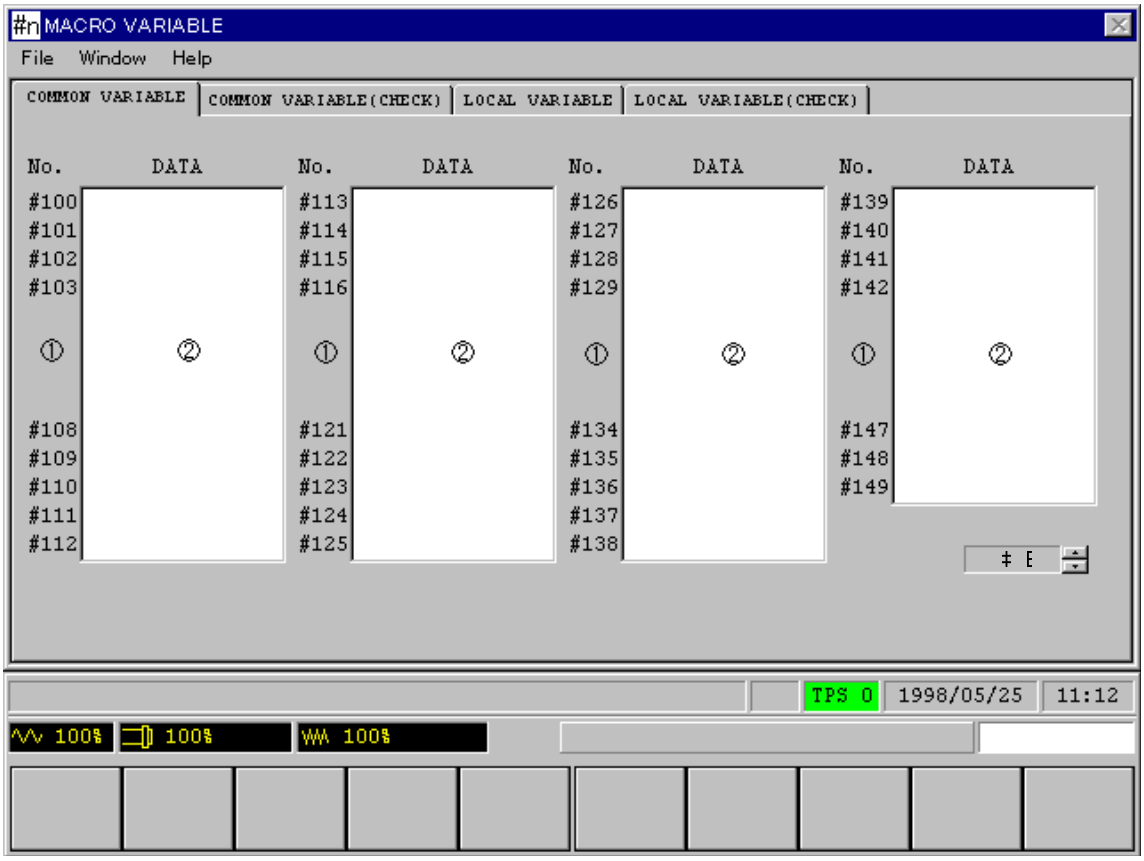
Remarks:

- Two types of macro variables can be used:
  a) Common variables..... Can be used for both the main program and the macro programs that are to be called up from the main program.
  b) Local vairables..... Can only be used in macro programs.
- The macro variable displays consist of the four sub-displays listed below.
  a) **COMMON VARIABLE** display ..... Allows the operator to set the common variables to be used to execute user macros in the automatic operation mode.
  b) **COMMON VARIABLE (CHECK)** display... Allows the operator to set the common variables to be used to carry out tool path checks.
  c) **LOCAL VARIABLE** display..... Allows the operator to check the local variables to be used to execute user macros in the automatic operation mode.
  d) **LOCAL VARIABLE (CHECK)** display ..... Allows the operator to check the local variables to be used to carry out tool path checks.

Note: The **LOCAL VARIABLE** and **LOCAL VARIABLE (CHECK)** displays only list settings of variables; the data on these two displays cannot be modified.
Refer to the EIA/ISO Programming Manual for further details of user macros.

5-3-1 Data display

1. Displayed data (COMMON VARIABLE, COMMON VARIABLE (CHECK))

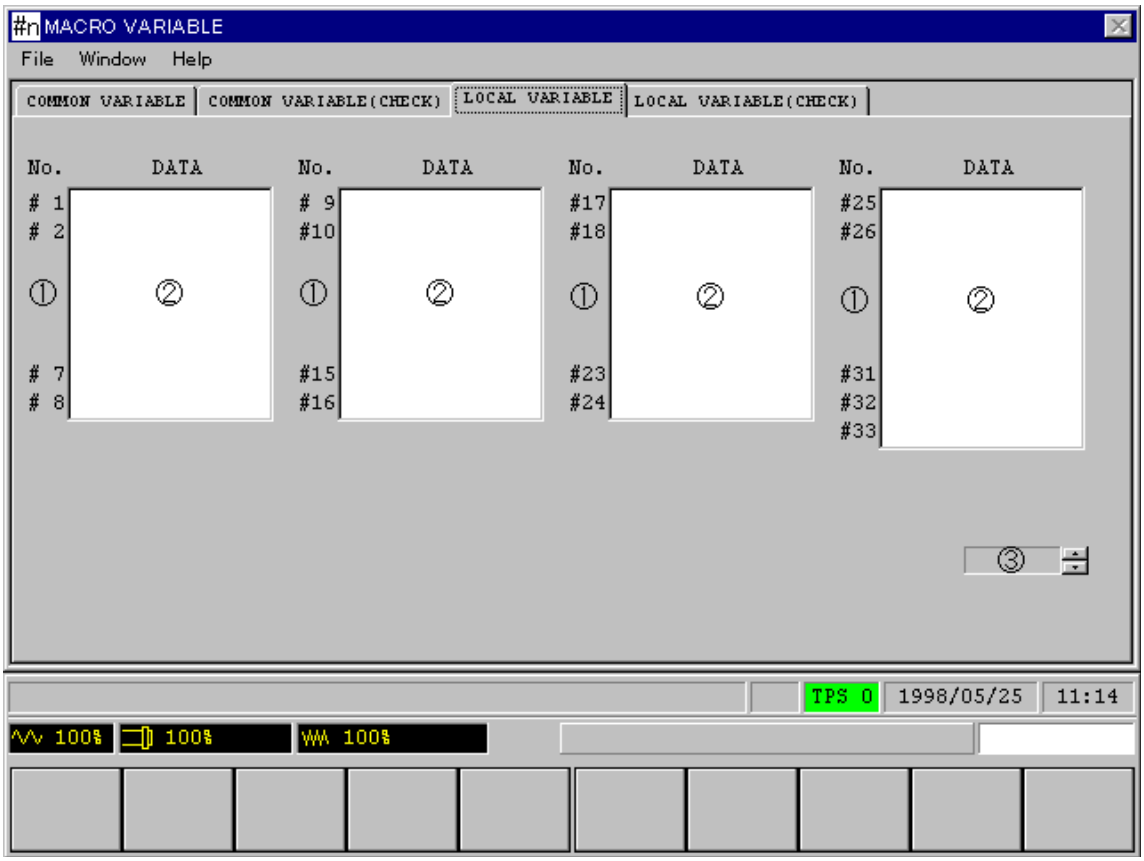


D735S1103E

2. Display data description (COMMON VARIABLE, COMMON VARIABLE (CHECK))

No.	Item	Unit	Data description
•	No.	•	Variable number
•	DATA	•	Registered variable value
•	Page	•	Current page/Total number of pages

3. Data display (LOCAL VARIABLE, LOCAL VARIABLE (CHECK))



D735S1016E

4. Display data description (LOCAL VARIABLE, LOCAL VARIABLE (CHECK))

No.	Item	Unit	Data description
•	No .	•	Variable number
•	DATA	•	Registered variable value
•	Nest	•	Nesting number

5-1-2 Data registration

1. COMMON VARIABLE and COMMON VARIABLE (CHECK) displays

Only data of variables can be registered on the **COMMON VARIABLE** and **COMMON VARIABLE (CHECK)** displays.

(1) Set the cursor on the data item DATA of the variable to be set.

- After calling up the cursor on the display by pressing the cursor key, move the cursor to the data item DATA of the variable to be set.

(2) Set the desired data as the intended variable.

- Enter data using the numeric data keys, and press the INPUT key.

2. LOCAL VARIABLE and LOCAL VARIABLE (CHECK) displays

No data can be registered on the **LOCAL VARIABLE** and **LOCAL VARIABLE (CHECK)** displays.

5-1-3 Display operations

1. COMMON VARIABLE and COMMON VARIABLE (CHECK) displays

1. Pressing the **INCRMENT INPUT** menu key allows the entered data to be added to the current data.
   See the Section, **TOOL DATA Display** for further details.
2. Use the **Exp** menu item to set data in exponential form.

   Example: To set the data 2×10^{-6}, press keys in the following order:

This displays “2. E – 6” under DATA item.

3. The data on the **COMMON VARIABLE** and **COMMON VARIABLE (CHECK)** displays have been registered over multiple pages.

   Pressing the page key  displays the next page.

   Pressing the page key  displays the previous page.

2. LOCAL VARIABLE and LOCAL VARIABLE (CHECK) displays

The data on the **LOCAL VARIABLE** and **LOCAL VARIABLE (CHECK)** displays have been registered on nesting levels 1 to 5 per display. The number displayed on the data item • denotes the nesting level of the current data.

- Pressing the page key  displays data of the next nesting.
- Pressing the page key  displays data of the previous nesting.

5-4 ADDITIONAL WPC Display

Function:

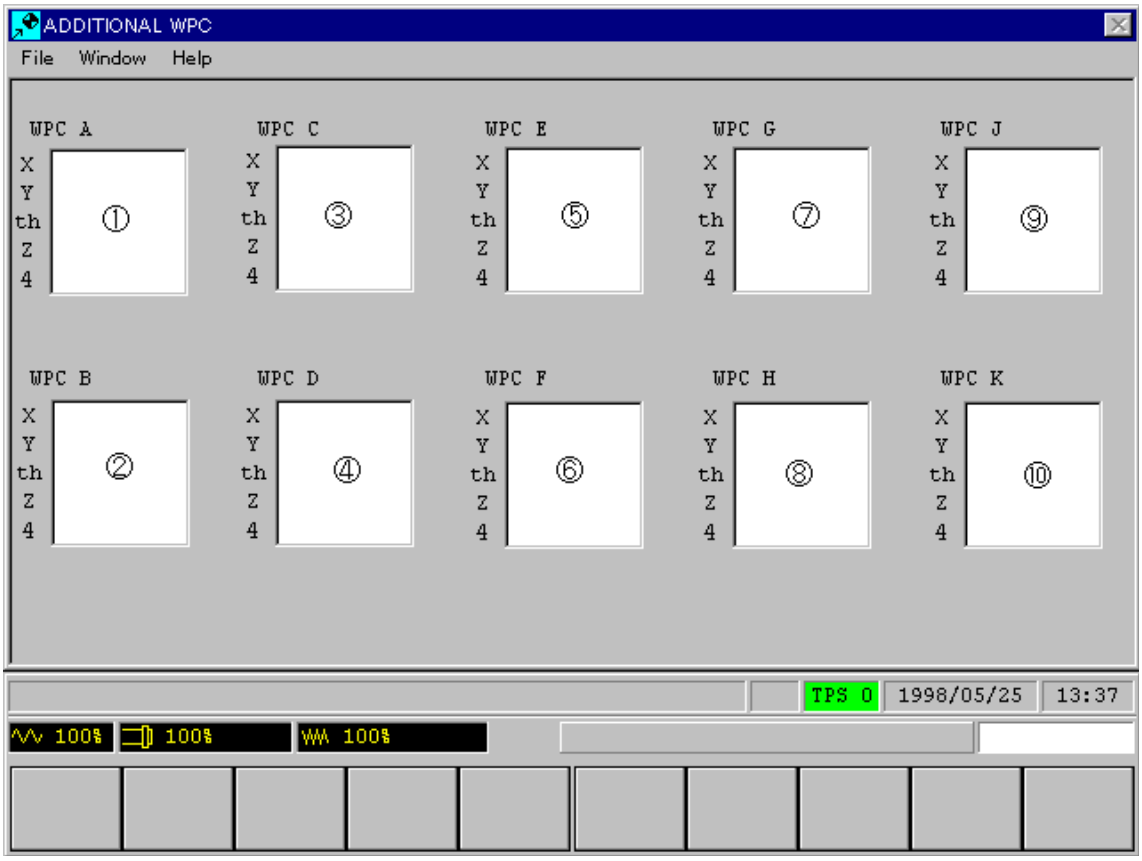
- By using this display, external coordinate data used for the MAZATROL program can be set.

Remarks:

- 10 sets of fundamental coordinates (A through K, except I) (workpiece coordinate system) can be set.
- 10 sets of data to be used can be specified by the appropriate address A through K using the fundamental coordinate unit of MAZATROL program.

5-1-1 Data display

1. Displayed data



D735S1017E

2. Display data description

No.	Item	Unit	Data description
•	X, Y, Z (4)	mm (inch)	Distance from the machine zero-point to the zero-point in workpiece coordinate system A on each axis (Offset of workpiece zero-point)
	th (4)	degrees	
•	X, Y, Z (4)	mm (inch)	Distance from the machine zero-point to the zero-point in workpiece coordinate system B on each axis (Offset of workpiece zero-point)
	th (4)	degrees	
•	X, Y, Z (4)	mm (inch)	Distance from the machine zero-point to the zero-point in workpiece coordinate system C on each axis (Offset of workpiece zero-point)
	th (4)	degrees	

No.	Item	Unit	Data description
•	X, Y, Z (4)	mm (inch)	Distance from the machine zero-point to the zero-point in workpiece coordinate system D on each axis (Offset of workpiece zero-point)
	th (4)	degrees	
•	X, Y, Z (4)	mm (inch)	Distance from the machine zero-point to the zero-point in workpiece coordinate system E on each axis (Offset of workpiece zero-point)
	th (4)	degrees	
•	X, Y, Z (4)	mm (inch)	Distance from the machine zero-point to the zero-point in workpiece coordinate system F on each axis (Offset of workpiece zero-point)
	th (4)	degrees	
•	X, Y, Z (4)	mm (inch)	Distance from the machine zero-point to the zero-point in workpiece coordinate system G on each axis (Offset of workpiece zero-point)
	th (4)	degrees	
•	X, Y, Z (4)	mm (inch)	Distance from the machine zero-point to the zero-point in workpiece coordinate system H on each axis (Offset of workpiece zero-point)
	th (4)	degrees	
•	X, Y, Z (4)	mm (inch)	Distance from the machine zero-point to the zero-point in workpiece coordinate system J on each axis (Offset of workpiece zero-point)
	th (4)	degrees	
•	X, Y, Z (4)	mm (inch)	Distance from the machine zero-point to the zero-point in workpiece coordinate system K on each axis (Offset of workpiece zero-point)
	th (4)	degrees	

5-1-2 Data registration

Data registration procedure on the **ADDITIONAL WPC** display does not differ from that on the **WORK OFFSET** display. See the Subsection **5-2-2 Data registration** of **WORK OFFSET Display**.

5-1-3 Address setting

Using the following menu, set the desired address in ADD. WPC item of the fundamental coordinates unit.

(1) Press the **WPC** menu key on the **PROGRAM** display.

➔ The following fundamental coordinate unit will be displayed.

```
UNo.  UNIT       ADD.WPC     X   Y   th  Z   4
3     WPC-0
```

(2) Move the cursor to ADD. WPC item on the display.

➔ The following menu will be displayed.

G54	G55	G56	G57	G58	G59	A	B	>>>		(a)

- Pressing the menu key >>> changes over to the following menu.

C	D	E	F	G	H	J	K	>>>		(b)

(3) Press the menu key corresponding to the address to be set.

➔ The set address is displayed in the ADD. WPC item.

Note: When the address is set in ADD. WPC item, data previously set in X to 4 items will all be cleared.

5-1-4 Considerations to measurement

1. Storing the tool tip position into the memory

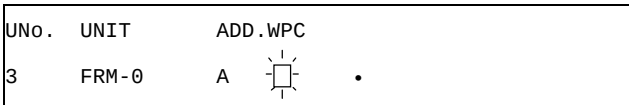
A. ADDITIONAL WPC display

As with the **WORK OFFSET** display, the **ADDITIONAL WPC** display allows the tool tip position to be stored into the memory. Refer to Subsection **5-2-2**.

2. Coordinate measurement

If MMS or MDI•MMS operations are carried out for the fundamental coordinate unit that has an address set in ADD. WPC item, the measured coordinates will be written into that address.

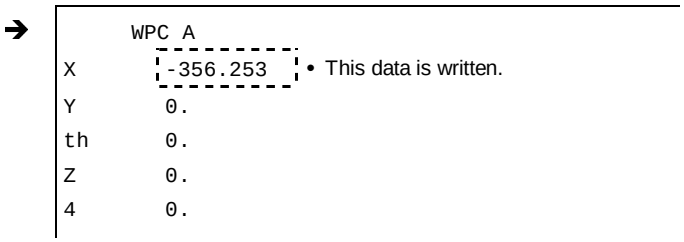
Example: Measurement of X coordinate by MDI•MMS operations



(1) Press the **+X SENSOR** menu key.

(2) Enter 0 and press the INPUT key.

(3) Press the CYCLE START button.



Note 1: Even if an address from G54 to G59 is designated for ADD. WPC item, the modal information in the EIA/ISO program will not be overridden with that address. The MAZATROL coordinate systems will also be used in this case.

Note 2: Although, normally, data can be set in either ADD. WPC item or X to 4 items, transferring a machining program from an external unit enables data to be set in both ADD. WPC and X to 4 items. However, only data in ADD. WPC item becomes valid if data is set in both ADD. WPC item and X to 4 items.

5-5 Manual Measurement

5-5-1 Function overview

Manual measurement allows both dimensional measurement and results display in the manual mode. With manual measurement, therefore, you can easily perform measurements without preparing a program when machining setup or dimensional measurement after machining has taken place. Since it presents coordinates display in addition to dimensional display, the manual measurement function can also be used for workpiece alignment.

5-5-2 Use of manual measurement

The manual measurement capability is designed for measuring work dimensions and provided with alignment function

1. Work dimension measurement

Items below can be measured.

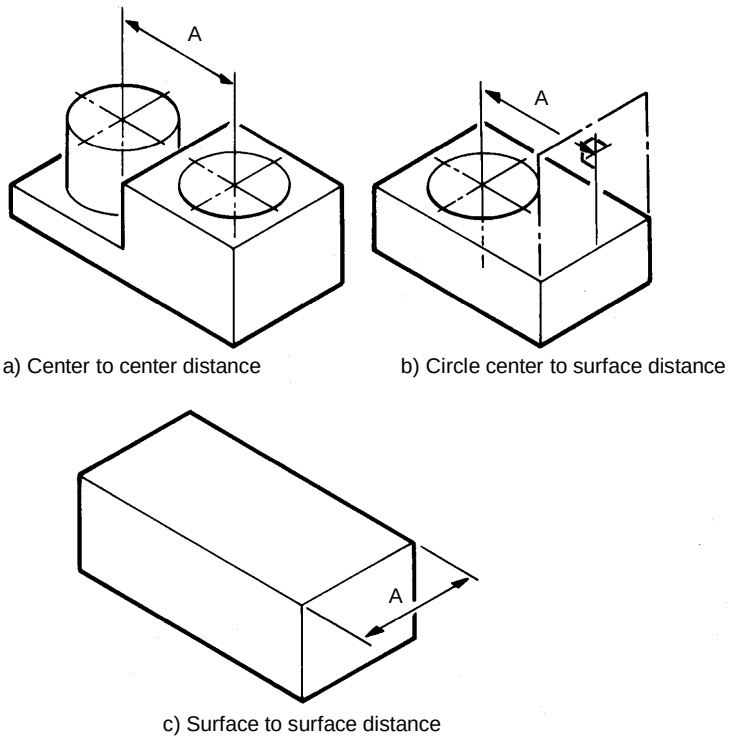
- Diameter of circular shape and center coordinate value
- Surface workpiece coordinate value, slope gradient with respect to machine coordinate axis
- Surface straightness, slope gradient with respect to machine coordinate axis

Measurement will be allowed in combination with the result of three items above.

a) Center to center distance of circular shapes

b) Distance between circular shape and surface

c) Surface to surface distance, parallelism, perpendicularity, slope gradient



a) Center to center distance

b) Circle center to surface distance

c) Surface to surface distance

Distance measurements NM210-00446

2. Alignment function

For alignment with other than manual measurement capability, MDI·MMS function is provided. This is used to measure X-Y, X-Z, Y-Z plane coordinate values. If work origin is at the center of a hole, measurement is impossible. Further, MDI·MMS is exclusively used for program WPC measurement, and so it is not used to make coordinate confirmation only. Although MAZATROL MMS unit can be used for this measurement procedure, program preparation time is required. For alignement with manual measurement capability, coordinate value of measurement point, center coordinate value of circular shape, and slope gradient of surface can be shown, and WPC measurement with work origin at the center of a hole can be accomplished.

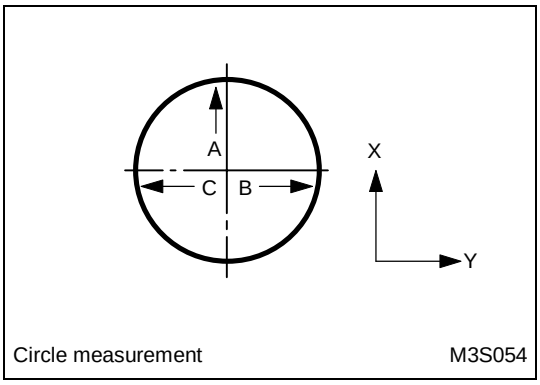
5-1-3 Measurement pattern

Measurement patterns are available in five types as below.

- Circle measurement..... Diameter and center coordinates of hole/boss
- Surface measurement..... Surface coordinates, slope gradient
- Straightness measurement (option).... Straightness between two arbitrary points
- One-point measurement..... Reference surface coordinate (on the measurement axis)
- Two-point measurement..... Groove/Step center coordinate (on the measurement axis)

In addition, combinations of the five measurement patterns above can be used to calculate distance, angle, parallelism, and squareness.

1. Circle measurement (Three measurement points)

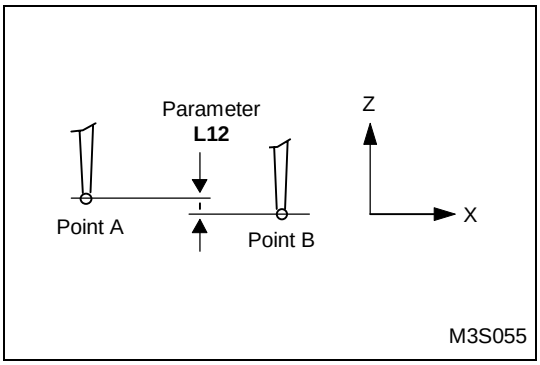


Circle measurement M3S054

A. Measuring procedure

- Measurement points A, B, and C can be positioned arbitrarily. (Three points must be at different positions.)
- Measurement direction (touch sensor movement direction) must be one of the axial directions (simultaneous two axes must not be used). In ±Z direction, however, alarm will be made.
- Circle measurement is allowed only on X-Y plane.

- Z coordinate at points A, B, and C must be identical. If Z coordinate value exceeds set parameter **L12**, alarm will be indicated, but measurement is possible.

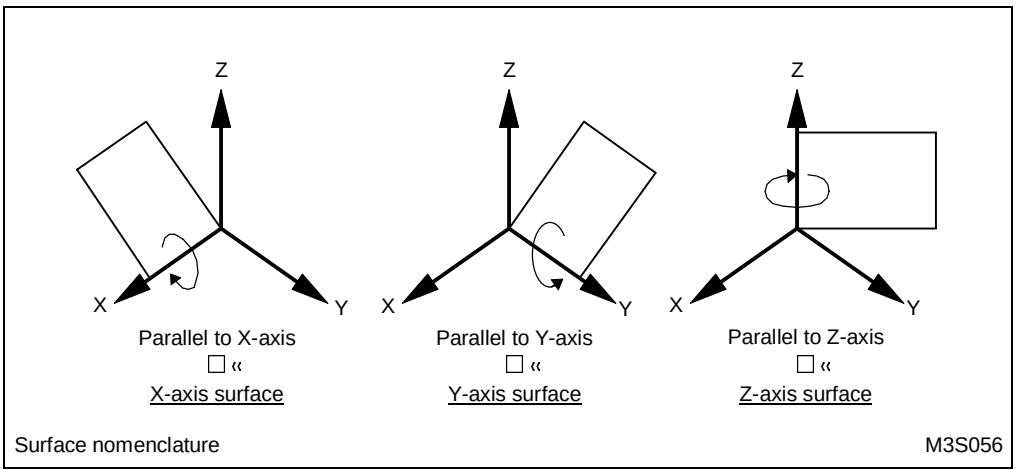


2. Surface measurement

For surface measurement, surface to be measured must be parallel to either of X-, Y- or Z-axes.

A. Surface nomenclature

If the selected surface is parallel to the X-axis, that surface is referred to as the X-axis surface. Likewise, if the selected surface is parallel to the Y-axis or the Z-axis, that surface is referred to as the Y-axis surface or the Z-axis surface, respectively.



B. Selection of measurement surface

Which surface should be measured is determined from measuring JOG direction and coordinates of two points on the surface. The following describes the procedure.

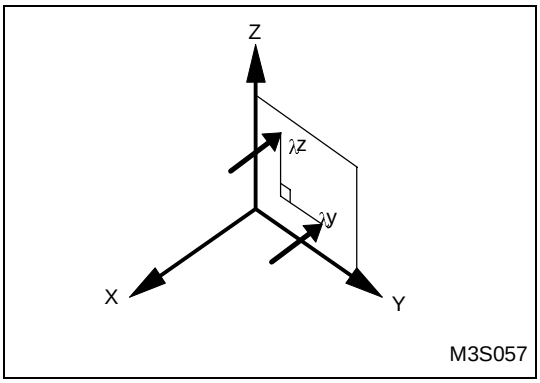
1. Two axial surfaces are determined by JOG direction.
2. Axial surface is determined from comparison of point to point distance in axial component.

Example: JOG in X direction

- X-axis surface is not selected.

Distance component: λz > λy

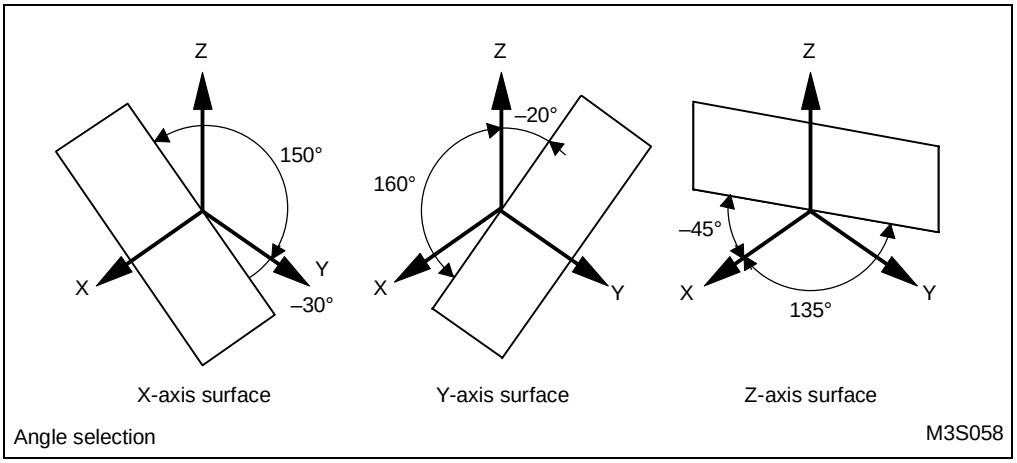
- Y-axis surface is selected



C. Angle selection

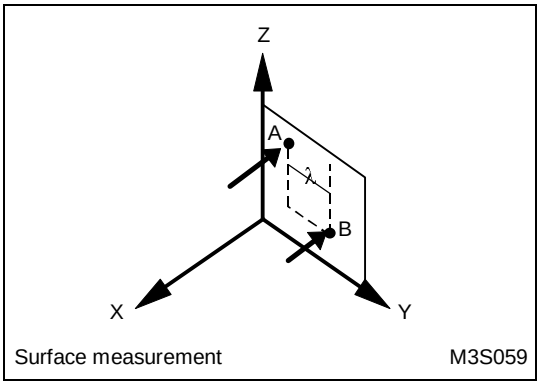
Angular data for various axial surfaces should be as below.

1. X-axis surface ..... CCW angle from +Y-axis on Y-Z plane (150° in example below)
2. Y-axis surface ..... CCW angle from +Z-axis on Z-X plane (160° in example below)
3. Z-axis surface ..... CCW angle from +X-axis on X-Y plane (135° in example below)



D. Measuring procedure

- Measurement points A, B should satisfy parameter **L12** ≥ λ. If **L12** < λ, alarm will be caused, but measurement is possible.



- Measurement direction (touch sensor movement direction) must be one of the axial directions (simultaneous two axes must not be used).
- Measurement movement direction for measurement points A, B must be identical. Different directions will cause alarm.

3. Straightness measurement (option)

A. Selection of axial surface

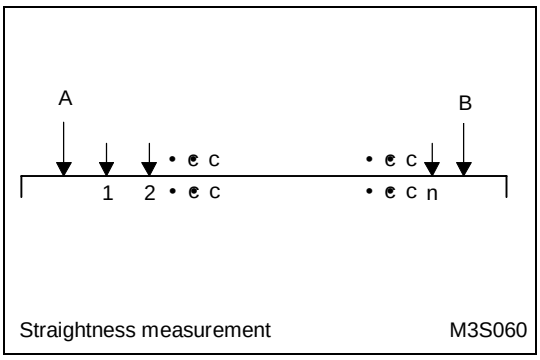
Selection of axial surface is the same as that of surface measurement.

B. Angle selection

Angle must be determined from a line approximated by the least square method.

C. Measuring procedure

- After manual measurement of A and B, points 1 to n will be measured automatically. The number of measurement points n must be entered by a measuring operator. ($1 \leq n \leq 30$)



Straightness measurement M3S060

- Measuring procedure is the same as that of surface measurement.

4. One-point measurement

A. Notes on measurement

- The measurement point can be set at any position.
- After measurement, only the reference plane coordinates in the direction of measurement are displayed.

5. Two-point measurement

A. Axial planes

- Axial planes are the same as those of plane measurement. See item B of preceding paragraph 2.

B. Notes on measurement

- An alarm will result if the first and second axes of measurement differ.
- In that case, processing can be executed from the second-point measurement operation by correcting the measurement axial direction after the alarm has been reset.

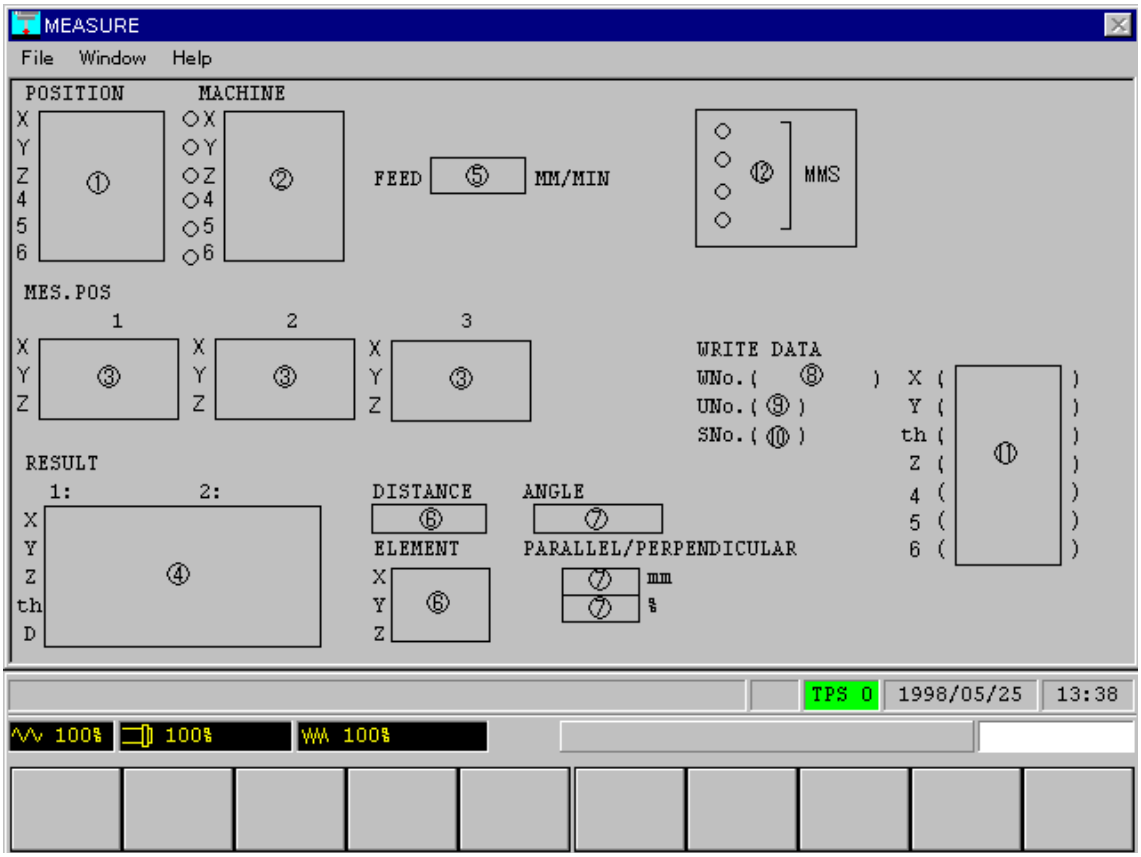
6. Distance, angle, parallelism, squaretness

- Up to two items can be stored as data. Distance, angle, parallelism, and perpendicularity between two data items can be calculated and indicated.
- For calculation of parallelism and perpendicularity, value will be indicated always with data 1 as base.

5-1-4 MEASURE display

Call up the **MEASURE** display to use the manual measurement function.

1. Displayed data



D735S1018E

2. Display data description

No.	Item	Unit	Data description
•	POSITION	mm (inch)	Current position of the tool nose in the workpiece coordinate system
•	MACHINE	mm (inch)	Current position of the machine in the machine coordinate system
•	MES. POS	mm (inch)	Coordinates of the measurement point. See Table A below.
•	RESULT	mm (inch)	Measurement results are displayed. See Table B below.
•	FEED	mm/min (ft/min)	Denotes a jog feed rate.
•	DISTANCE	mm (inch)	Denotes the linear distance derived from two sets of measurement results. No data will be displayed if the distance is not derived from two sets of measurement results. See Table C below.
	ELEMENT	mm (inch)	X-, Y-, and Z-axial components of the distance derived from two sets of measurement results.

No.	Item	Unit	Data description
•	ANGLE	deg.	Denotes the difference in angle between two surfaces. No data will be displayed if the distance is not derived from two sets of measurement results.
	PARALLEL/ PERPENDICULAR	mm %	Degree of parallelism/perpendicularity in “mm”, and the difference in the angles of two planes in “%”. No data will be displayed if the distance is not derived from two sets of measurement results. The degree of parallelism/perpendicularity will be calculated only if the difference in the angles of the two surfaces falls within either the required range of parameter **L13** (degree of parallelism), 180 degrees ± parameter **L13** (degree of parallelism), or 90 degrees ± parameter **L13** (degree of perpendicularity).
•	WNo.	•	Program number or from G54 to G59, added fundamental coordinates (A to K) G54.1P1 to G54.1P48 (option)
•	UNo.	•	Designate either the WPC unit, the five-surface machining unit, or the surface definition unit.
•	SNo.	•	Designate the five-surface machining sequence or the surface definition sequence.
•	Coordinate value to be written	mm (inch)	Input the coordinate data to be written.
•	MMS (Display for touch sensor)		Indications about touch sensor. See Table D below.

Table A

Mode	Circle measurement	Surface measurement	Degree-of-straightness measurement	One-point measurement	Two-point measurement
Points displayed	3	2	2 (Starting/ ending points)	1	2
Correction of the measure-ment results.	Corrections are performed assuming that all measure-ment points are present on the circumference of a circle.	Corrections are performed assuming that the measurement plane is either X-Y, X-Z, or Y-Z.			

Table B

Mode	Circle measurement	Surface measurement	Degree-of-straightness measurement	One-point measurement	Two-point measurement
Measure pattern	XY CIRC, YZ CIRC, ZX CIRC	X PLANE, Y PLANE, Z PLANE	X STRGT., Y STRGT., Z STRGT	X STD, Y STD, Z STD	X CENTR, Y CENTR, Z CENTR
X, Y, Z	Center coordinates (Z is the coordinate of the first measurement point)	Center coordinates of two points	Center coordinates of the starting and ending points	Reference surface coordinate (of only one axis).	Center coordinates in the direction of measurement
th	•	Angle	Angle	•	•
D	Diameter	•	Degree of straightness	•	•

Table C

Distance calculation pattern	Conditions
Circle and circle Circle and Z-axis surface Circle and degree of straightness with respect to the Z-axis surface	None
Circle and X- or Y-axis surface Circle and the degree of straightness with respect to the X- or Y-axis surface	Each surface must be parallel to the Z-axis (within the range of parameter **L13**).
Same axial surface as that of each axis Same degree of straightness as that of each axis	The angles of two surfaces must be equal (within the range of parameter **L13**).
Axial surface different from that of each axis Degree-of-straightness different as that of each axis	Two surfaces must be parallel (within the range of parameter **L13**).

Table D

		Function
1		"ON" will be displayed in reverse form if the touch sensor unit is powered on.
2		"ON" will be displayed in reverse form when the touch sensor is mounted in the spindle and the machine becomes ready for measurement. No data will be displayed if the display of "ON" is displayed by the activation of function 4 described below.
3		"ON" is displayed in reverse form while the touch sensor is being mounted in the spindle. That is, measurement occurs while "ON" remains on the display.
4		If the probes at the end of the touch sensor come into contact with the workpiece or a jig, "ON" will be displayed in reverse form and the display of "ON" of function 2 described above will turn off.

5-1-5 Mode changeover

1. Manual mode

Any manual operation mode can be used during touch sensor positioning movement.

For contact with measurement point, however, only HANDLE/FEED keys  can be used. If skip signal comes on in other modes, alarm will be caused. (Measurement pattern is not cancelled.)

2. TAPE, MEMORY mode ( , )

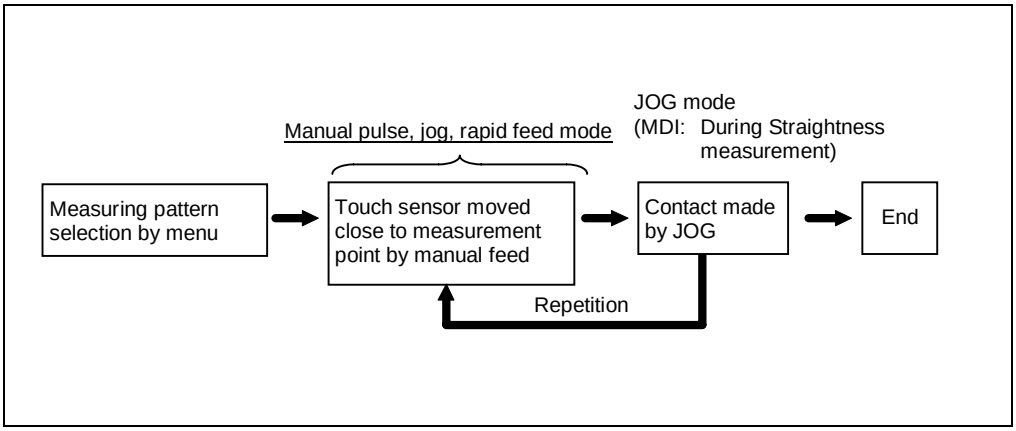
- Measurement is not allowed in this mode. Alarm will be caused if the pattern selection menu key is pressed.
- If TAPE, MEMORY mode is selected during measurement, the measurement operation will be cancelled.

3. MDI mode ()

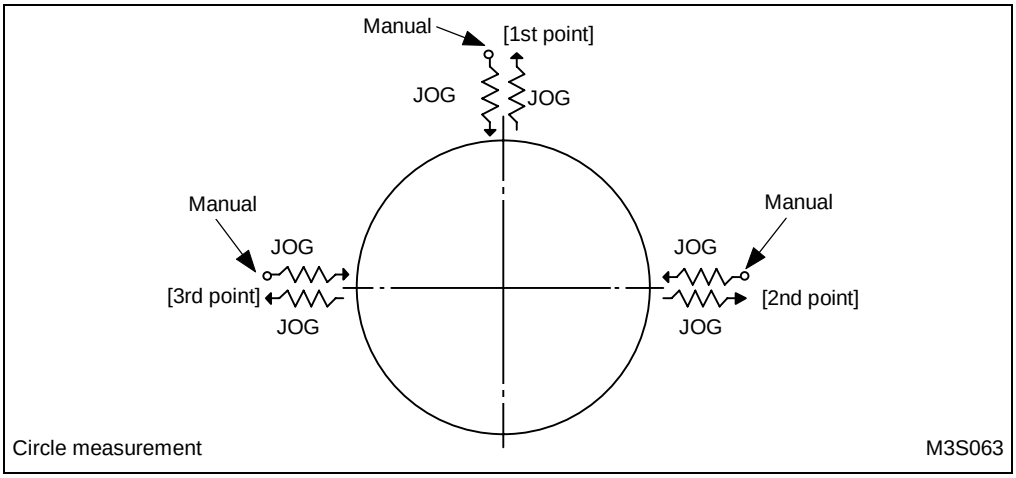
- This can be used only when mode change message is given in straightness measurement. In other cases, identical with other automatic operation modes.

5-1-6 Measuring procedure

Measuring procedure is outlined below.



1. Circle measurement



(1) Press the **CIRCLE MEASURE** menu key.

➔ This displays the message **FIRST POINT MEASURING** and reverses the display status of the menu item.

- The corresponding “Help” window can be displayed by selecting through the “Window” item on the menu bar.

(2) Measure the first point using the jog feed function.

➔ This displays the coordinates of the first measurement point and then displays the message **SECOND POINT MEASURING**.

(3) Measure the second point using the jog feed function.

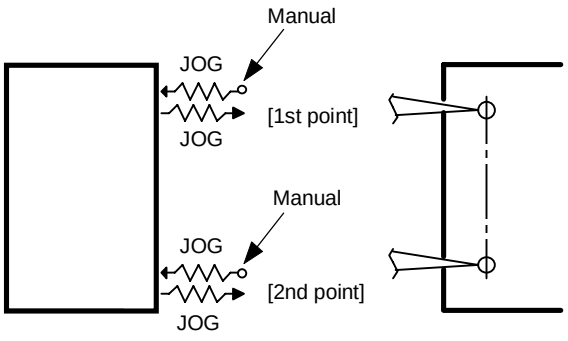
➔ This displays the coordinates of the second measurement point and then displays the message **THIRD POINT MEASURING**.

(4) Measure the third point using the jog feed function.

➔ This displays the coordinates of the third measurement point and the measurement results (center coordinates of the circle).

➔ The reverse-display status of the menu item will be cleared. The Help window will also be cleared if it is displayed.

2. Surface measurement



Surface measurement M3S064

(1) Press the **SURFACE MEASURE** menu key.

➔ This displays the message **FIRST POINT MEASURING** and reverses the display status of the menu item.

- The corresponding “Help” window can be displayed by selecting through the “Window” item on the menu bar.

(2) Measure the first point using the jog feed function.

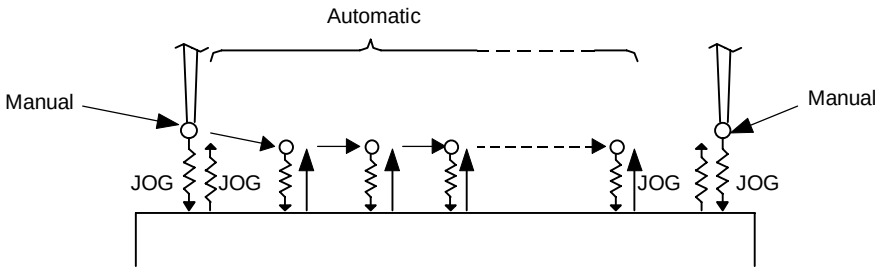
➔ This displays the coordinates of the first measurement point and then displays the message **SECOND POINT MEASURING**.

(3) Measure the second point using the jog feed function.

➔ This displays the coordinates of the second measurement point and the measurement results (center coordinates of the circle).

➔ The reverse-display status of the menu item will be cleared. The Help window will also be cleared if it is displayed.

3. Straightness measurement (option)



Straightness measurement M3S065

(1) Press the **STRAIGHT MEASURE** menu key.

➔ This displays the message **REQUEST STARTING POINT** and reverses the display status of the menu item.

- The corresponding “Help” window can be displayed by selecting through the “Window” item on the menu bar.

(2) Measure the starting point using the jog feed function.

➔ This displays the coordinates of the starting point of measurement and then displays the message **REQUEST FINAL POINT**.

(3) Measure the ending point using the jog feed function.

➔ This displays the coordinates of the ending point of measurement and then displays the message **SELECT MDI MODE**.

(4) Move the sensor away from the measurement surface using the jog feed function.

- When the operation mode is changed over to MDI, the system will display the message **NUMBER OF MEASURING POINTS?**. Input the number of measurement points (any value from 1 to 30) using the numeric keys, and then press the INPUT key.
  The current number of measurement points and the total number of measurement points will then be displayed, and the current number of measurement points will be updated with each automatic measurement operation.
  The message **PUSH CYCLE START BUTTON** will also be displayed at the same time.

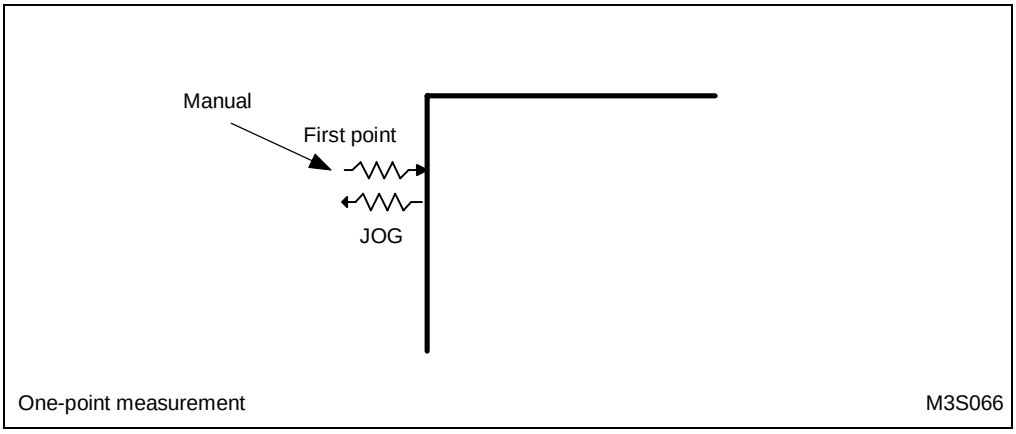
(5) Press the CYCLE START button.

➔ This causes automatic measurement to be repeated the designated number of times.

➔ After measurement, measurement results will be displayed. The reverse-display status of the menu item will be cleared. In addition, the Help window also will be cleared if it is displayed.

Note: To correctly operate the machine, the work number of the manual measurement macroprogram, which will be called up for measuring the degree of straightness, must be the same as the work number that has been set in parameter **L15** (Manual Measurement Macroprogram WNo.).

4. One-point measurement



One-point measurement M3S066

(1) Press the **1 POINT MEASURE** menu key.

➔ This displays the message **MOVE TO THE MEASURING POINT** and reverses the display status of the menu item.

- The corresponding “Help” window can be displayed by selecting through the “Window” item on the menu bar.

(2) Execute the measurement using the jog feed function.

➔ This displays the coordinates and the measurement results.
The reverse-display status of the menu item will be cleared. The Help window will also be cleared if it is displayed.

5. Two-point measurement

Two-point measurement M3S067

(1) Press the **2 POINT MEASURE** menu key.

➔ This displays the message **FIRST POINT MEASURING** and reverses the display status of the menu item.

- The corresponding “Help” window can be displayed by selecting through the “Window” item on the menu bar.

(2) Measure the first point using the jog feed function.

➔ This displays the coordinates of the first measurement point and then displays the message **SECOND POINT MEASURING**.

(3) Measure the second point using the jog feed function.

➔ This displays the measurement results (the center coordinate of the two points).

➔ The reverse-display status of the menu item will be cleared. The Help window will also be cleared if it is displayed.

5-1-7 Other functions

1. Measurement result copying

Measurement results can be copied into the write-data registration column.
Data that has been set in the write-data registration column can be written into the designated position using the function described later in paragraph 3, Data writing.

(1) Press the **COPY RESULT** menu key.
This changes the menu as follows:

RESULT 1-X	RESULT 1-Y	RESULT 1-Z	RESULT 1-th	RESULT 2-X	RESULT 2-Y	RESULT 2-Z	RESULT 2-th		

(2) Press the menu key corresponding to the combination of an identification number (1 or 2) and an axis (X, Y, Z, or th) of the measurement results to be copied, and then press the INPUT key. (Multiple menu items can be selected).

Note: If menu items of the same axis but of different measurement result numbers are selected together, the menu item whose display status has been first reversed will return to the normal status and only the menu item which has been next reversed will have its display status reversed.

➔ The measurement results that have been displayed in reverse form will be copied into the write-data registration column, and the reverse-display status of the menu item will be released.

2. Measurement result clearing

Masurement results can be cleared.

(1) Press the **CLEAR RESULT** menu key.

➔ This displays the message **CLEAR MEASURING RESULT No.?**.

(2) Input the identification number (1, 2, or 0) of the measurement results to be cleared, and then press the INPUT key.

➔ The distance and angle items will then be cleared together with the measurement results. Thus, it will become possible for either of the two points to be measured once again.

- If number 0 is input, both 1 and 2 will be cleared.

3. Data writing

Data that has been set in the write-data registration column (WRITE DATA) can be written into the designated position.

(1) Display the cursor at the write-data registration column by pressing the cursor key.

(2) Set writing destination data.

- Use one of the following three methods to designate the writing destination:

(a) Writing into the fundamental coordinate unit:

`WNo.•   123456M•` — Input the work number of the writing destination program using the numeric keys, and press the INPUT key.

`UNo.•    1•` — Input the unit number within the program where the fundamental coordinate unit is contained, and press the INPUT key.

`SNo.•     •` — Set no data.

(b) Writing into work offsets (WORK OFFSET):

`WNo.•    G54    •` — Set a code from G54 to G59 using the menu.

`UNo.•     •`
`SNo.•     •` — The cursor cannot be moved to ( ) if any code from G54 to G59 is set.

Writing into work offsets (Option)

`WNo.•   G54.1P6•` — Select "G54.1" from the menu. The message **G54.1 P-CODE?** will then be displayed. Input the P number of the writing destination using the numeric keys, and press the INPUT key.

`UNo.•     •`
`SNo.•     •` — The cursor cannot be moved to ( ) if any code from G54.1P1 to G54.1P48 is set.

(c) Writing into additional work offsets (ADDITIONAL WPC):

`WNo.• A         •` — Set a work number from A to K using the appropriate menu.

`UNo.•     •`
`SNo.•     •` — The cursor cannot be moved to ( ) if any code from A to K is set.

(3) Press the **WRITE** menu key.

➔ This displays the message **WRITE <INPUT>?** and reverses the display status of the menu item.

(4) Press the INPUT key.

➔ The coordinate data will then be written into the designated destination. When writing is completed, the reverse-display status of the menu item will be cleared.

- Data that has been set in “th” will be ignored if the coordinate data is written into work offsets.

4. Incremental data input

The data to be written can be input in incremental form.

(1) Press the **INCRMENT INPUT** menu key.

➔ This displays the message **LENGTH TO ADD?** and reverses the display status of the menu item.

(2) Input the data to be added, and press the INPUT key. The input data will then be added to the data indicated by the cursor.

5. Entire data clearing

The entire data under the write-data setting column can be cleared.

➔ Press the **ALL CLEAR** menu key.
This displays the message **ALL CLEAR?** and reverses the display status of the menu item.

(1) Press the INPUT key.

➔ This clears the entire written data and turns off the reverse-display status of the menu item.

6. Window functions

The following window functions are provided in the **MEASURE** display:

A. Help window

This window can be displayed by selecting through “Window” on the menu bar while either the **CIRCLE MEASURE, SURFACE MEASURE, STRAGHT MEASURE, 1 POINT MEASURE,** or **2 POINT MEASURE** menu item remains in the reverse-display status.

B. Work number listing window

This window can be displayed by using the **PROGRAM FILE** menu item while this menu item remains on the display.

C. MAZATROL program window

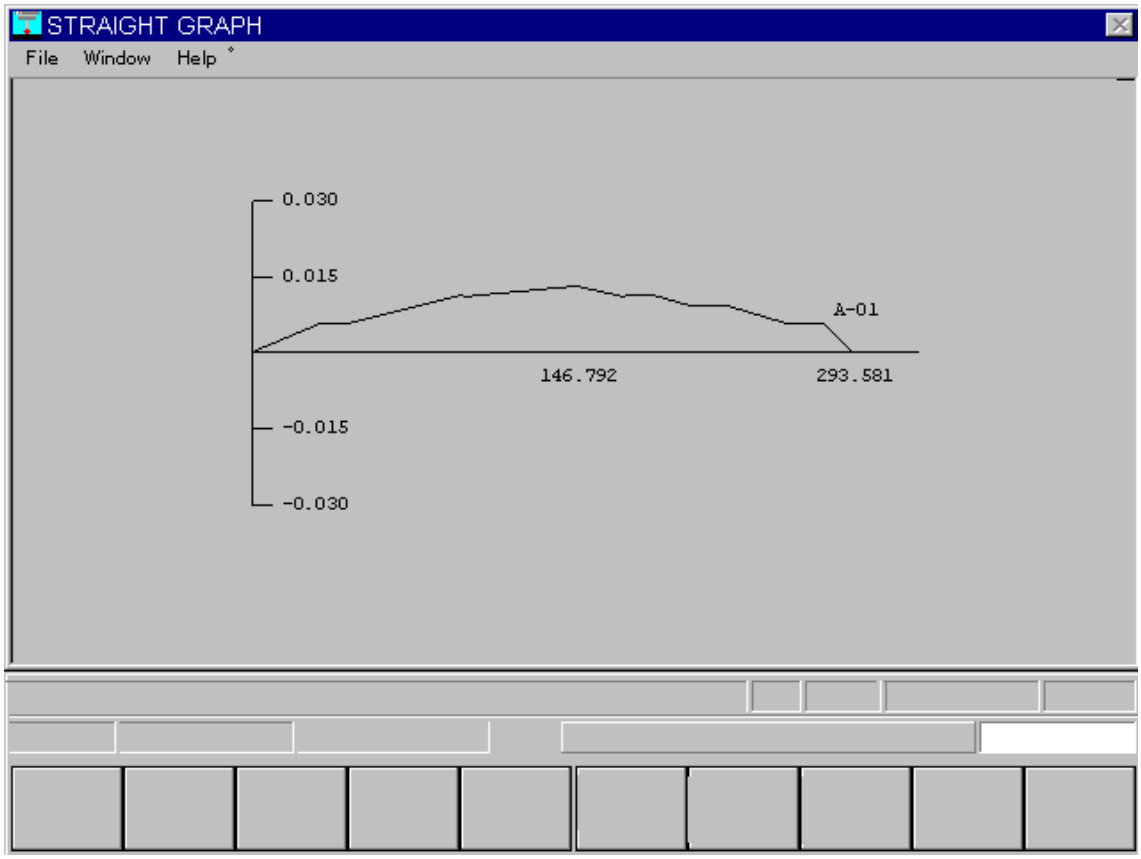
While the work number of a program is designated at WNo. of WRITE DATA and the cursor remains set at WNo., UNo. or SNo., the MAZATROL program window corresponding to the WNo. and UNo. data can be displayed by selecting “Program” through “Window” on the menu bar. The window, however, will not display any data if a unit number that has been selected at UNo. is not included in the program that has been selected at WNo.

5-1-8 Graph of straightness data (option)

All measurement results will be indicated by numeric values on the **MEASURE** display. In addition, straightness data will be graphically shown on a line graph. The measurement results and graph can be copied.

1. Calling up the STRAIGHTNESS GRAPH display

Press the **STRAIGHT GRAPH** menu key on the basic menu of the **MEASURE** display.
The **STRAIGHTNESS GRAPH** display will be displayed.



D735S1019E

2. Initial state of the STRAIGHTNESS GRAPH display

This display is initialized as below according to measurement operation.

1. Straightness measurement data not stored in either “1:” or “2:”
   - Graph frame is only shown.
   - No character is indicated.
2. Straightness measurement data in either “1:” or “2:”
   - Line is drawn on graph with horizontal axis passing start and end points of straightness data.
   - Dimensions are shown.
3. Straightness data in both “1:” and “2:”
   - Graph for data “2:” will be drawn in a manner similar to 2. above.

3. Menu function

A. MEASURE

Direct return from the graph to the **MEASURE** display.

B. PARALLEL PERPNDLR

When surface, straightness measurement data in “1:”, and straightness measurement data in “2:” and parallelism and squareness are calculable between these two data, data “1:” is taken as reference on horizontal axis, on which graph for data “2:” can be drawn. In other cases, this is not available.

C. DRAW GRAPH 1, DRAW GRAPH 2

DRAW GRAPH 1 for showing the first straightness graph, and **DRAW GRAPH 2** for the second straightness graph.

Note: If no data for straightness measurement is specified, these menu functions are not available.

D. NAME INPUT

Comment input is allowed with up to 11 characters of alphabet and numerals at top right on the display.

4. Notes

- Vertical scale is two times the data scale (max).
- Graph coordinate system is identical with machine coordinate system with (+) right direction of X-axis and (+) upward direction of Y-axis.

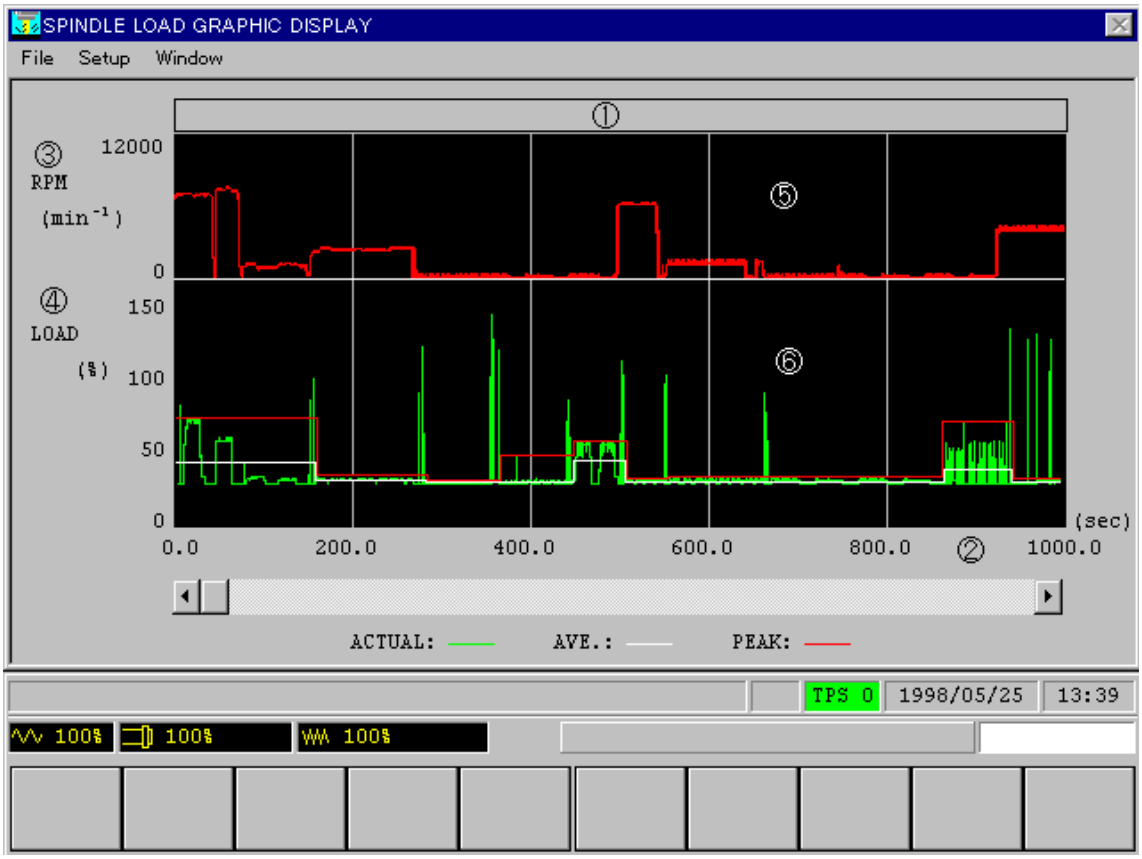
5-6 SPINDLE LOAD GRAPHIC DISPLAY Display

Function:

Displaying the changes in the speed and load of the spindle.

5-6-1 Data display

1. Displayed data



D735S1020E

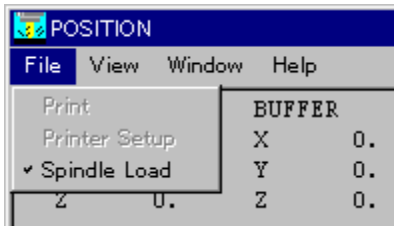
2. Display data description

No.	Item	Unit	Data description
•	Tool number	•	Numbers of the tools mounted on the spindle
•	Axis of abscissa	sec	Period of the graphic display
•	Axis of ordinate (upper)	min^{-1}	Spindle speed
•	Axis of ordinate (lower)	%	Spindle load
•	Graphic display (upper)	•	Display of the changes in the spindle speed in a red line.
•	Graphic display (lower)	•	Display of the changes in actual spindle load in a green line. The average and peak load values can be additionally displayed respectively in white and red lines for each tool.

5-1-2 Displaying spindle load data

(1) Start the sampling of the spindle load data.

- Select the **POSITION** display, press the WINDOW key and select “Spindle Load” through “File” on the menu bar.



(2) Select the display **SPINDLE LOAD GRAPHIC DISPLAY**.

Note 1: To stop sampling, press the WINDOW key once again first. Next after selecting “File” from the menu bar, clear the check-off mark from “Spindle Load” by selecting the “Spindle Load” option anew.

Note 2: Selection of this display during sampling, not after stopping it, will show the data that has been obtained from the start of sampling to the moment of display selection.

Note 3: If sampling is stopped and then started anew, existing data samplings will be erased.

Note 4: Spindle load data can be sampled for a maximum of one hour.

5-1-3 Display mode selection

1. Setting the display time scale

Upon selection of this display, sampled data will initially be displayed in its entirety.

The time scale for graphic display can be changed as required for an enlarged view of a specific period.

(1) Press the **CHANGE TIME** menu key.

➔ The display of the menu item will be reversed and the message **TIME (sec)?** displayed.

(2) Specify the desired time and press the INPUT key.

➔ The graphic display will be remade for the period from 0 (sampling start) to the specified time, and the reverse display of the menu item cleared.

(3) Use the cursor and page keys to select the desired period (time section).

- Press the cursor keys to scroll (move) 1/5 of the current display period (time section) or the page keys to scroll the entire period.

  Use the keys → ↓ ⇊ for the later section (forward scrolling) and ← ↑ ⇈ for the earlier section (backward scrolling).

- This operation can also be performed on the scroll bar below the graph by using the mouse.

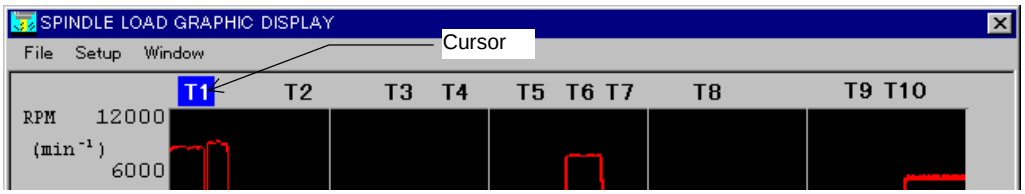
2. Displaying the data of only a specific tool in enlarged form

Data of only a specific tool can be displayed in enlarged form.

(1) Press the **CHANGE TOOL** menu key.

➔ The display of the menu item will be reversed and the message **TOOL No. <CURSOR, INPUT>?** displayed.

➔ The cursor will appear in the display area of tool numbers.



(2) Using the cursor keys, position the cursor on the desired tool number.

(3) Press the INPUT key.

➔ The graphic display will be remade for an enlarged view of the specified tool, and the reverse display of the menu item cleared.

3. Displaying the initial status of the graphic display

Once display period has been changed or only a specific tool has been selected for an enlarged view, temporarily retrieve the initial status of this display to perform a similar modification anew.

(1) Press the **CHANGE DISPLAY** menu key.

➔ The graphic display will return to the initial status (existing when the display was called up).

5-1-4 Displaying additional data related to spindle load

1. Displaying the average spindle load

Average value of the actual spindle load during cutting feed can be calculated and displayed for each tool.

(1) Press the **AVERAGE VALUE** menu key.

➔ The display of the menu item will be reversed and the average load will be additionally indicated in a white line for each tool.

- Press the same menu key once again to clear the display of the average load, and the reverse display of the menu item will be cleared at the same time.

2. Displaying the peak spindle load

Peak value of the actual spindle load during cutting feed can be detected and displayed for each tool.

(1) Press the **PEAK VALUE** menu key.

➔ The display of the menu item will be reversed and the peak load will be additionally indicated in a red line for each tool.

- Press the same menu key once again to clear the display of the peak load, and the reverse display of the menu item will be cleared at the same time

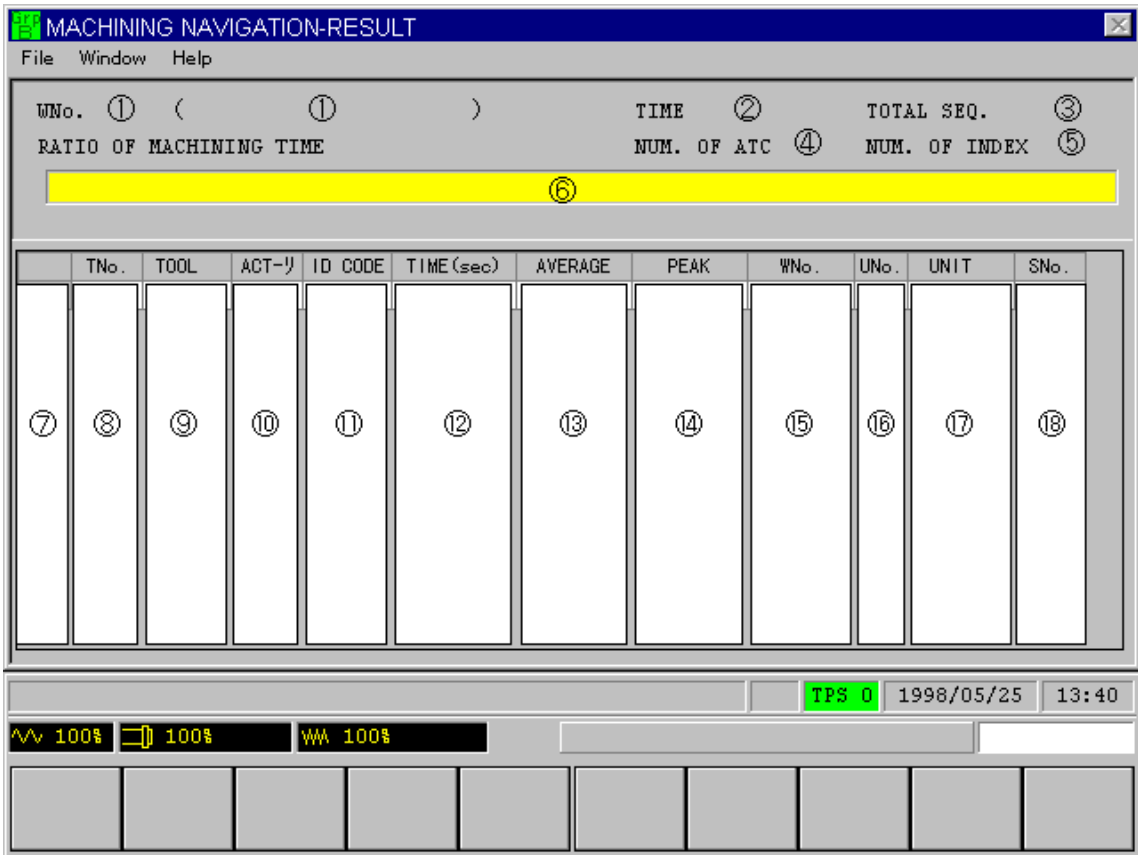
5-7 MACHINING NAVIGATION - RESULT Display

Function:

The machining time and spindle output data are listed for each tool according to the results of actual machine operation.

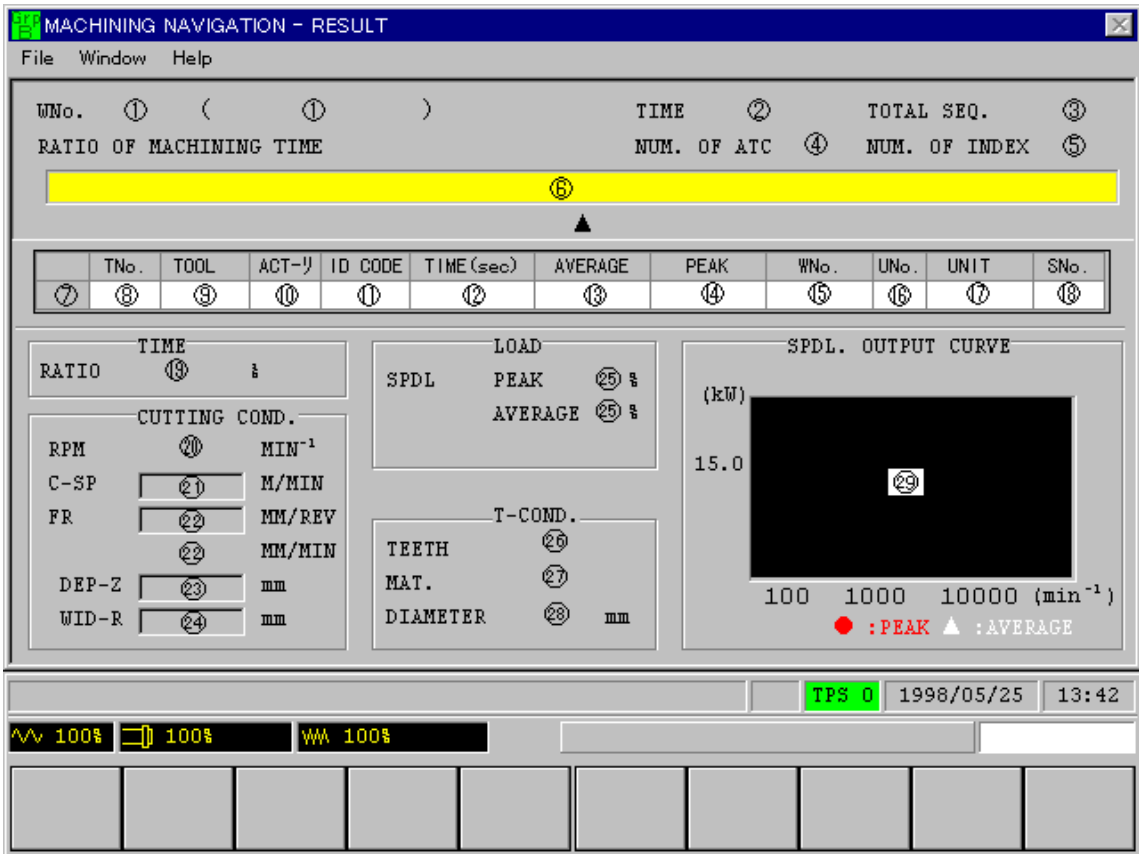
5-7-1 Data display

1. Displayed data with normal display of TOOL DISTINCT menu item



D735S1021E

2. Displayed data with reverse display of TOOL DISTINCT menu item



D735S1022E

3. Display data description

No.	Item	Unit	Data description
•	WNo.	•	Work number of the machining program WNo. **a b (c)** **a** : Work number **b** : Program type (MAZATROL or EIA/ISO) **c** : Program name
•	TIME	H/M/S	Total machining time
•	TOTAL SEQ.	•	Total number of tool sequences in the program
•	NUM.OF ATC	•	Number of ATC operations required for actual machining
•	NUM.OF INDEX	•	Total number of index units in the program
•	RATIO OF MACHINING TIME	•	Color-identified graphic display of the ratio of machining time for each tool
•		•	Serial number of the machining process
•	TNo.	•	Tool number
•	TOOL	•	Tool name
•	ACT-φ	•	Nominal diameter of the tool
•	ID CODE	•	Identification code of the tool
•	TIME (sec)	sec	Machining time
•	AVERAGE	kW	Average value of spindle output
•	PEAK	kW	Peak value of spindle output
•	WNo.	•	Work number of the program

No.	Item	Unit	Data description
•	UNo.	•	Unit number
•	UNIT	•	Unit name
•	SNo.	•	Tool sequence number
•	TIME RATIO	%	Ratio of machining time for each tool sequence
•	RPM	min^{-1}	Spindle speed specified in the tool sequence
㉑	C-SP	m/min (feet/min)	Circumferential speed specified in the tool sequence
㉒	FR	mm/rev (inch/rev)	Feed rate specified in the tool sequence (upper: Feed per revolution)
		mm/min (inch/min)	Feed rate specified in the tool sequence (lower: Feed per minute)
㉓	DEP-Z	mm	Z-axial cutting depth specified in the tool sequence
㉔	WID-R	mm	Radial cutting depth specified in the tool sequence
㉕	SPDL PEAK/AVERAGE	%	Ratio of the peak/average spindle load to the output characteristics, calculated for the tool sequence
㉖	TEETH	•	Number of teeth (cutting edges) of the tool (according to the setting on the TOOL FILE or TOOL DATA display)
㉗	MAT.	•	Material of the tool (according to the setting on the TOOL DATA display)
㉘	DIAMETER	mm (inch)	Diameter of the tool (according to the setting on the TOOL DATA display)
㉙	SPDL. OUTPUT CURVE	•	Spindle output characteristic curve (in red) with respect to the spindle speed, displayed for the tool sequence. The output peak and average output are indicated respectively by a red disk (in a position of the abscissa corresponding to the spindle speed for the output peak) and a white triangle.

5-1-2 Displaying data

The functions of this display cannot be used for a machining program which has not ever been executed actually on the machine.

(1) Execute the desired program in the automatic operation mode.

- Refer to Chapter 1 for the procedure.

(2) Select the **MACHINING NAVIGATION - RESULT** display.

➔ The results of the machine operation will be displayed in the total display mode described in the following subsection.

- No data will be displayed if the automatic operation has not yet been completed at all. In case the automatic operation based on a program of the same work number has already been completed, however, data at that time will be displayed.

5-1-3 Display modes

1. Outline

This display has the following two display modes:

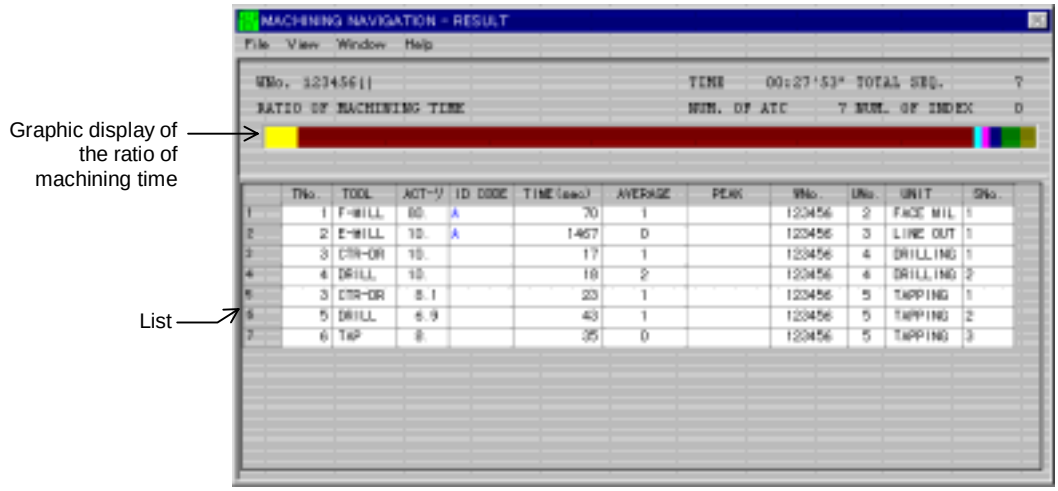
- Total display mode
  The display is always made in this mode when the **MACHINING NAVIGATION - RESULT** display is selected.
  In this display mode, the machining time and average spindle output are listed for each tool sequence in addition to the general information about execution of the program.

- Tool-specific display mode
  In this display mode, which can be selected and canceled by pressing the **TOOL DISTINCT** menu key, data of a specific tool sequence listed up in the total display mode can be selectively displayed in further detail.

2. Changing from the total to the tool-specific display mode

(1) Check that a graphic display of the ratio of machining time for each tool sequence is presented under “RATIO OF MACHINING TIME”.

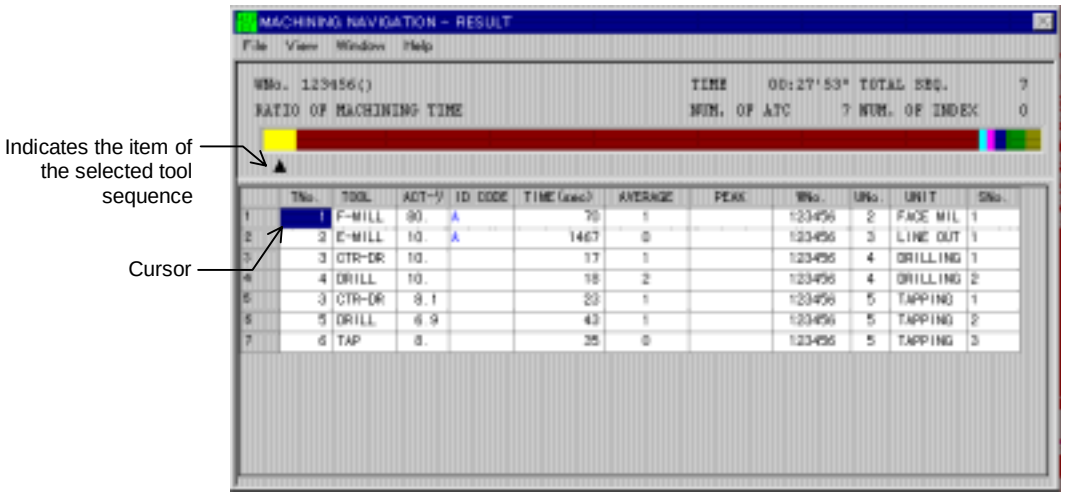
- For a program the execution of which has been completed, the machining time ratio for each tool sequence to the total machining time will be displayed in a color-identified format on a strip graph as shown in the display example below. The items on the strip graph are keyed from left to right, in a one-to-one correspondence, to those line items of the list (machining time and average spindle output for each tool sequence) that are displayed at the bottom of the graph.



(2) Press a cursor key.

➔ The cursor will appear in the “TNo.” column.

➔ A black triangle (•) will be displayed under the graph “RATIO OF MACHINING TIME” to indicate the item of the tool sequence currently selected by the cursor, as shown in the display example below.



(3) Using the cursor keys, position the cursor on the desired tool sequence number.

➔ The black triangle will move correspondingly with the cursor so that the ratio of the machining time of the specific tool sequence to the total machining time can be visually checked.

(4) Press the **TOOL DISTINCT** menu key.

➔ The display of the menu item will be reversed and the display contents will be changed for a detailed view of the specified tool.

- Press the same menu key once again, and the total display mode will be retrieved and the reverse display of the menu item cleared.

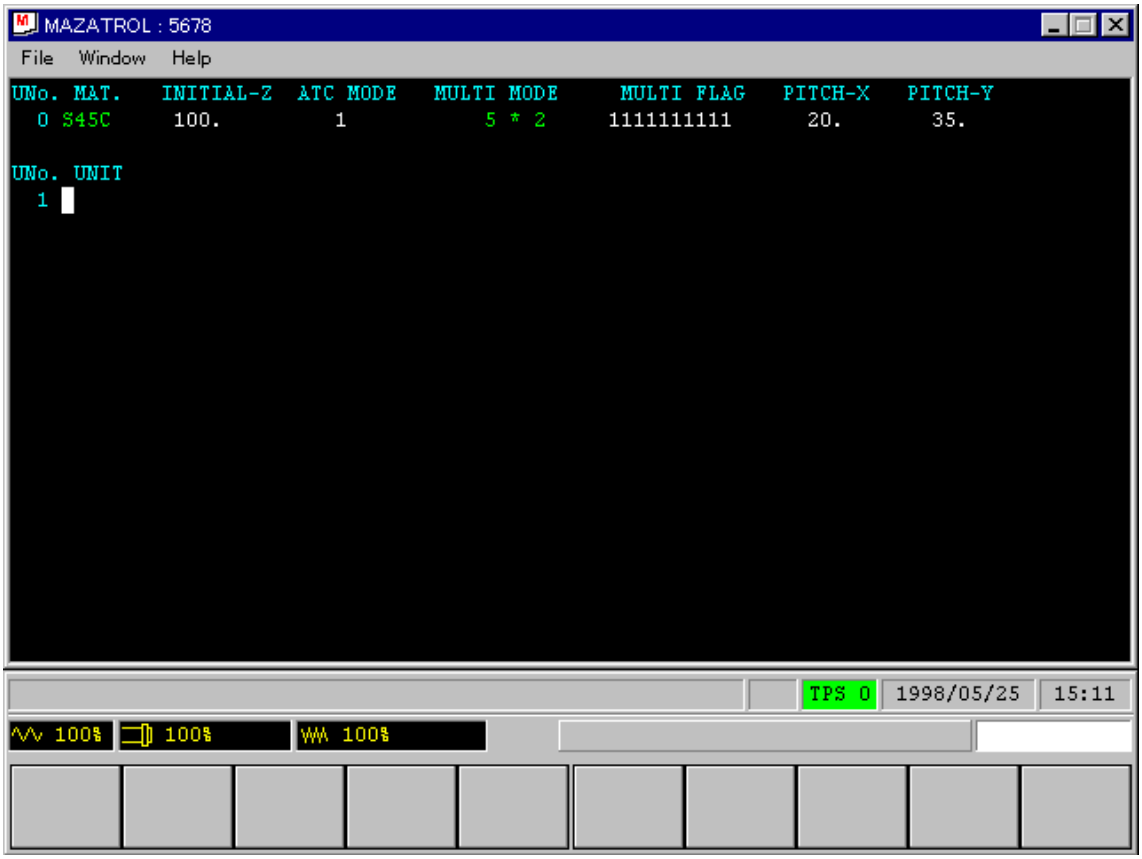
- NOTE -

6 DISPLAYS RELATED TO PROGRAM CREATION

6-1 PROGRAM Display

Refer to the Programming Manual (MAZATROL) when creating MAZATROL programs or the Programming Manual (EIA/ISO) when creating EIA/ISO programs.

An example of display presentation (for MAZATROL programming)



D735S1023E

6-2 TOOL PATH CHECK Display

Function:

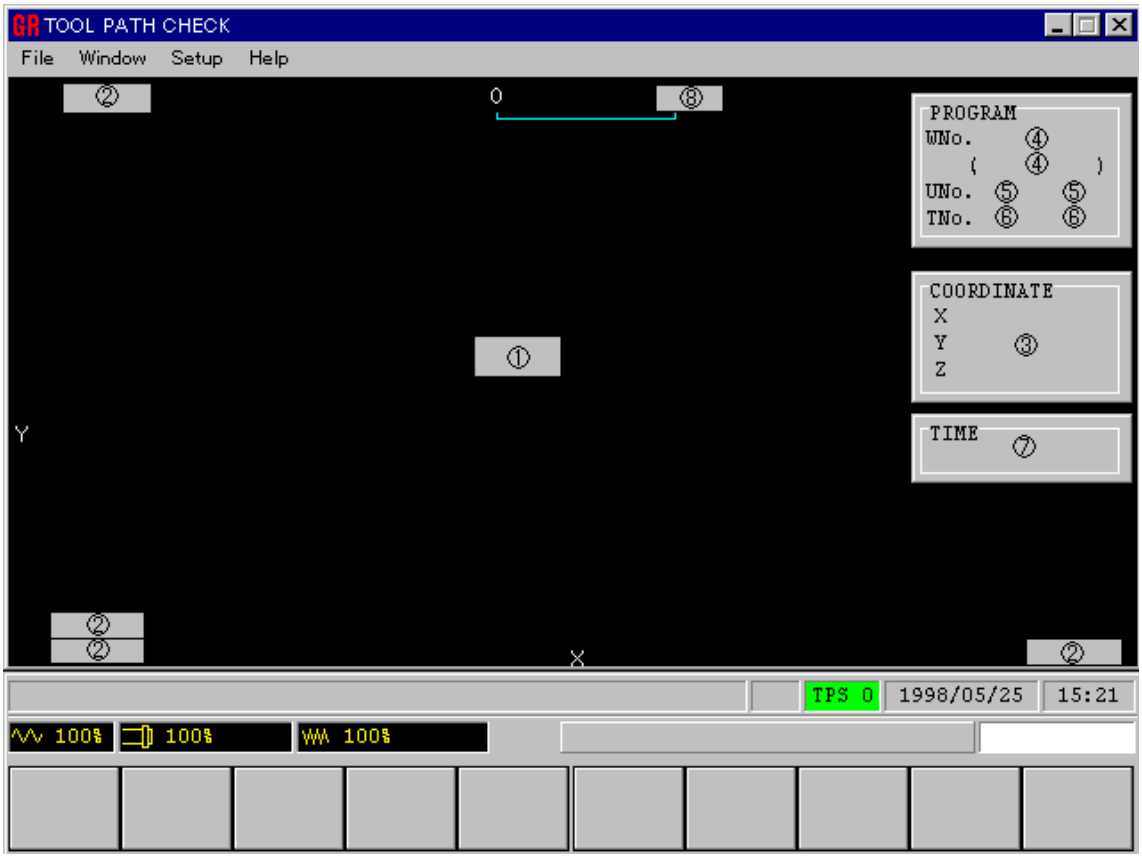
This display is used to display the machining shapes and the tool-tip path that have been defined by the machining program.

Remarks:

- Only the program that has been selected on the **PROGRAM** display can undergo tool path check.
- A plane to be used to display machining shape and path of the tool tip can be designated.
- Scale values used for displaying machining shape and path of the tool tip can be changed.

6-2-1 Data display

1. Displayed data



D735S1024E

2. Display data description

No.	Item	Unit	Data description
•	(Shape)	•	Machining shape that has been defined by the program (For point-machining, this data is displayed with a purple line. For line- or face-machining, this data is displayed with a green line.)
	(Tool path)	•	Path of the tool tip (The path of cutting feed is displayed with a yellow solid line, and the path of rapid feed is displayed with a yellow dotted line.)
•		mm (inch)	Display limit (automatically determined according to the machining shape)
•	X, Y, Z	mm (inch)	Current position of the tool tip in the machine coordinate system
•	WNo.	•	Work number of the program on which the path check is to be made
	( )	•	If the machining program on which the path check is to be made is a subprogram, the work number of that subprogram will be displayed in the parentheses.
•	UNo.	•	Identification number and name of the program unit currently being checked
•	TNo.	•	Identification number and name of the tool whose path is being checked
•	TIME	Hour/minute/ second	Cutting time elapsed during automatic operation
•	0 ├──────┤	mm (inch)	Scale value of graphic form (On the initial display, this data is automatically determined according to the machining shape.)

6-1-2 Displaying machining shape

(1) Press the **PART SHAPE** menu key.

➔ This will cause the display status of **PART SHAPE** to reverse and the machining shapes that have been defined by the program to be displayed continuously.

- Point-machining shapes will each be displayed with a purple line, and line- or face-machining shapes will each be displayed with a green line.
- After displaying machining shapes, the reverse-display status of **PART SHAPE** will be released automatically.

Note 1: Depression of the **PLANE CHANGE** menu key allows either a plane or a 3-D view to be selected for display of a machining shape(s). For the selecting procedure, see the description of display operation (**4-2-3 Plane select**) that is given in Section **4-2 TRACE Display**.
If plane-select operation has not been carried out, the machining shape available on the XY plane will be displayed.

Note 2: Depression of the **SCALE CHANGE** menu key allows display of machining shapes in enlarged or reduced size. In addition, the 3-D display viewpoint can be changed if the 3-D view of a machining shape has been selected by plane selection operation. For the changing procedure, see the description of display operation (**4-2-4 Change of scale values**) that is given in Section **4-2 TRACE Display.**
If this operation has not been carried out, machining shapes will be displayed at the automatically determined scaling factor.

6-1-3 Displaying tool-tip path

1. Continuous display

(1) Press the **PATH CONTINUE** menu key.

➔ This will cause the display status of **PATH CONTINUE** to reverse and the tool-tip path to be displayed continuously.

- The tool-tip path for cutting feed will be displayed with a yellow solid line, and the tool-tip path for rapid feed will be displayed with a yellow dotted line.
- After display of the tool-tip path, the reverse-display status of **PATH CONTINUE** will be released automatically.

2. Intermittent display

(1) Press the **PATH STEP** menu key.

➔ This will cause the display status of **PATH STEP** to reverse and the tool-tip path to be displayed step by step by each depression of the menu key.

- The tool-tip path for cutting feed will be displayed with a yellow solid line, and the tool-tip path for rapid feed will be displayed with a yellow dotted line.
- After display of the entire tool-tip path, the reverse-display status of **PATH STEP** will be released automatically.

6-1-4 Restarting tool path check

Tool path checks can be carried out in the middle of the program.

(1) Press the menu selector key.

(2) Press the **PATH RESTART** menu key.

➔ The display status of the menu will then reverse and a restart data setting window will be opened.

(3) Designate the restarting position of tool path checking.

- How to specify the restarting position of tool path checking is the same as for the restart in the automatic operation mode.
  See the Operating Manual, Part 4, Subsection **1-2-2 Restart operation**, for further details.

(4) Press the **PATH CONTINUE** or **PATH STEP** menu key.

➔ Tool path checking will then start from the designated position.

6-1-5 Block skip

Any block of the EIA/ISO program can be skipped to check the tool path.

(1) Press the menu selector key.

(2) Press the **BLOCK SKIP** menu key.

(3) Press the **PATH CONTINUE** or **PATH STEP** menu key.

How to specify the blocks to be skipped is the same as for the blocks in the automatic operation mode. See the Operating Manual, Part 4, Section **2-5 Block skip**, for further details.

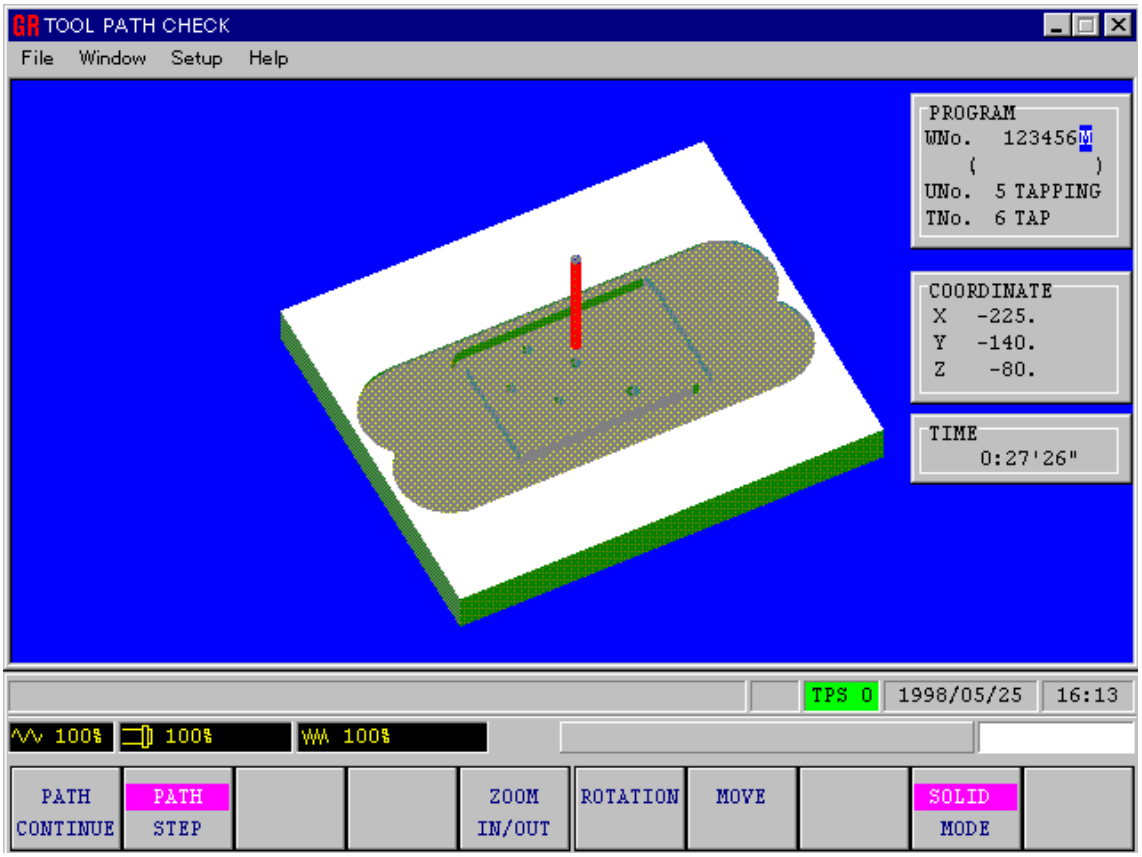
6-1-6 Other menu functions

1. Depression of the **SHAPE ERASE** menu key causes erasure of the particular machining shape.
   Erasure of the machining shape currently being displayed can also be made by pressing this menu key while the machining shape is being drawn on the display.
2. Depression of the **PATH ERASE** menu key causes erasure of the tool-tip path.
   Erasure of the tool-tip path can also be made by pressing this menu key while the tool-tip path is being drawn on the display.
3. The tool-tip path continuous display mode can be changed over to the intermittent display mode if the **PATH STEP** menu key is pressed while the tool-tip path is being displayed continuously (i.e., while **PATH CONTINUE** remains on the display in reverse form).
4. The tool-tip path intermittent display mode can be changed over to the continuous display mode if the **PATH CONTINUE** menu key is pressed while the tool-tip path is being displayed intermittently (i.e., while **PATH STEP** remains on the display in reverse form).
5. It is possible for the display of machining shapes and tool-tip path to be aborted by pressing the clear key.
   This is not possible, however, with the reset key.
6. If the menu display status is changed to reverse form by pressing the **STORE** menu key, the shape data and the tool-tip path data will be preserved even after the display has been changed over to another display.
7. When the **PROGRAM** menu key is pressed, the display will change over to the **PROGRAM** display.

8. Tool-path display using special interpolation functions such as those for spiral-cutting operations do not correspond completely with actual tool movements.

6-1-7 Solid Mode

The tool path specified in a MAZATROL program can be checked in a quasi-actual machining image as shown below by executing the solid-model display function, so that a cutting area error and other abnormalities can be detected in an earlier stage and the test cutting time can be saved correspondingly.

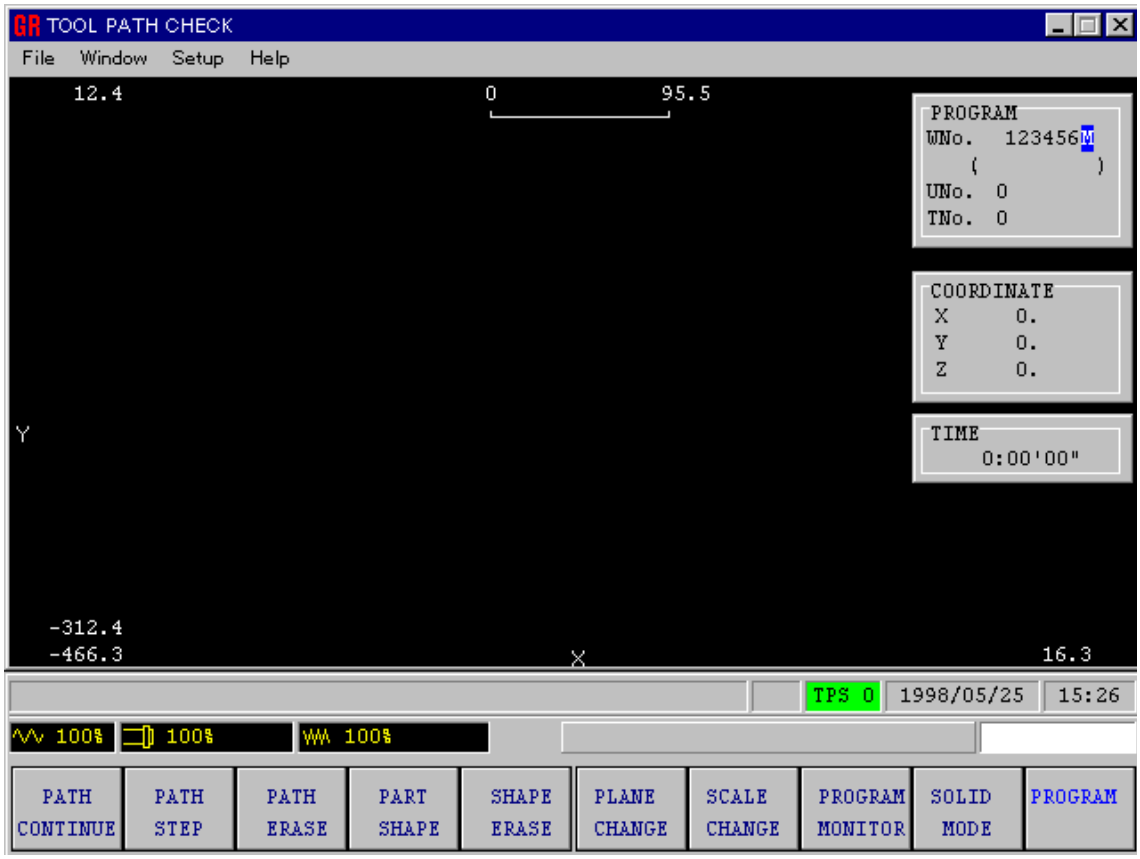


D735S1025E

1. Operation

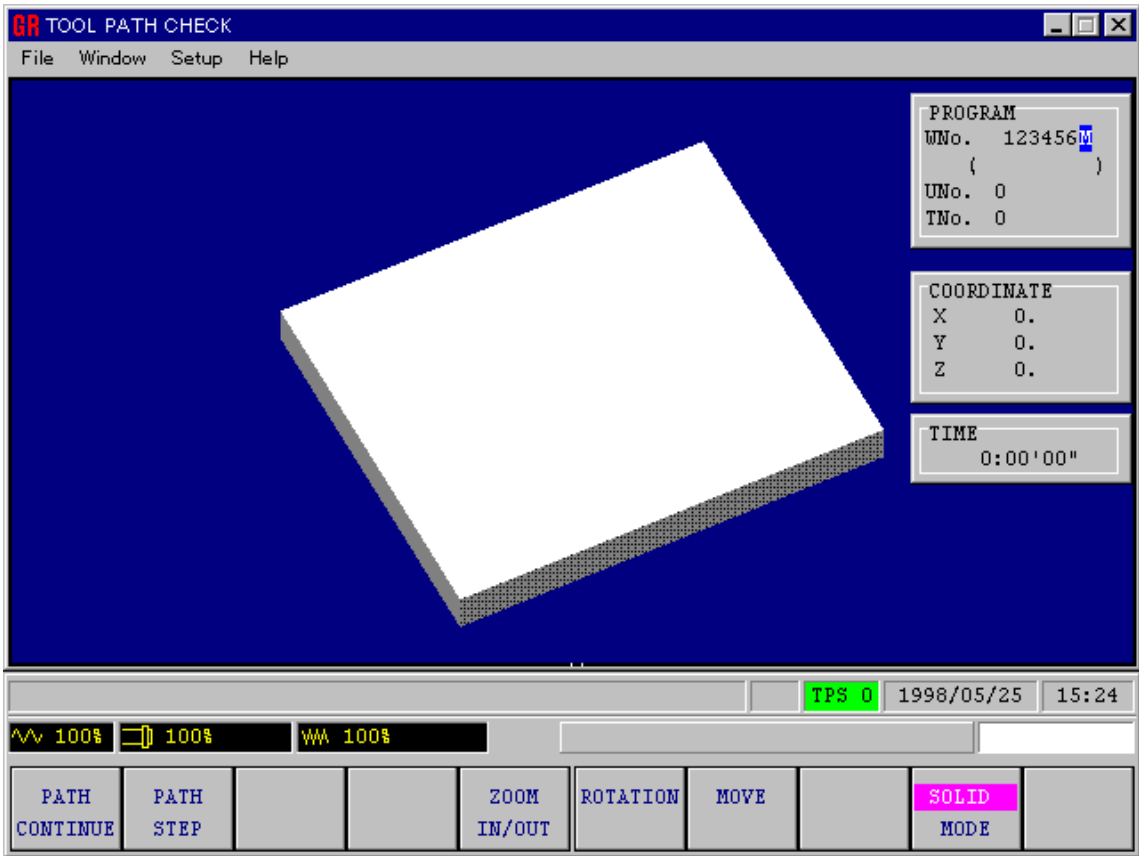
(1) Select the solid mode.

- Press the **SOLID MODE** menu key on the **TOOL PATH CHECK** display.



D735S1026E

➔ A solid-model display will appear as shown in the example below.



D735S1027E

- The rectangular parallelopiped that has been set to the appropriate size according to the selected MAZATROL program is displayed as the material to be machined.

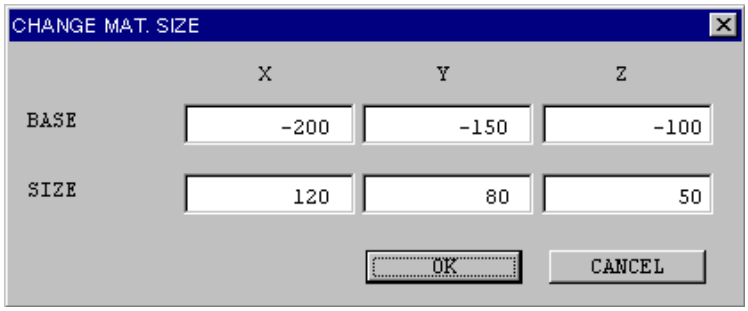
Note 1: Selecting the **SOLID MODE** menu function for an EIA/ISO program will cause an alarm.

Note 2: The material to be machined can only be displayed as a rectangular parallelopiped. Molded materials having a stepped surface cannot be presented as they are.

Note 3: Refer to the next step for the size of the three-dimensional display.

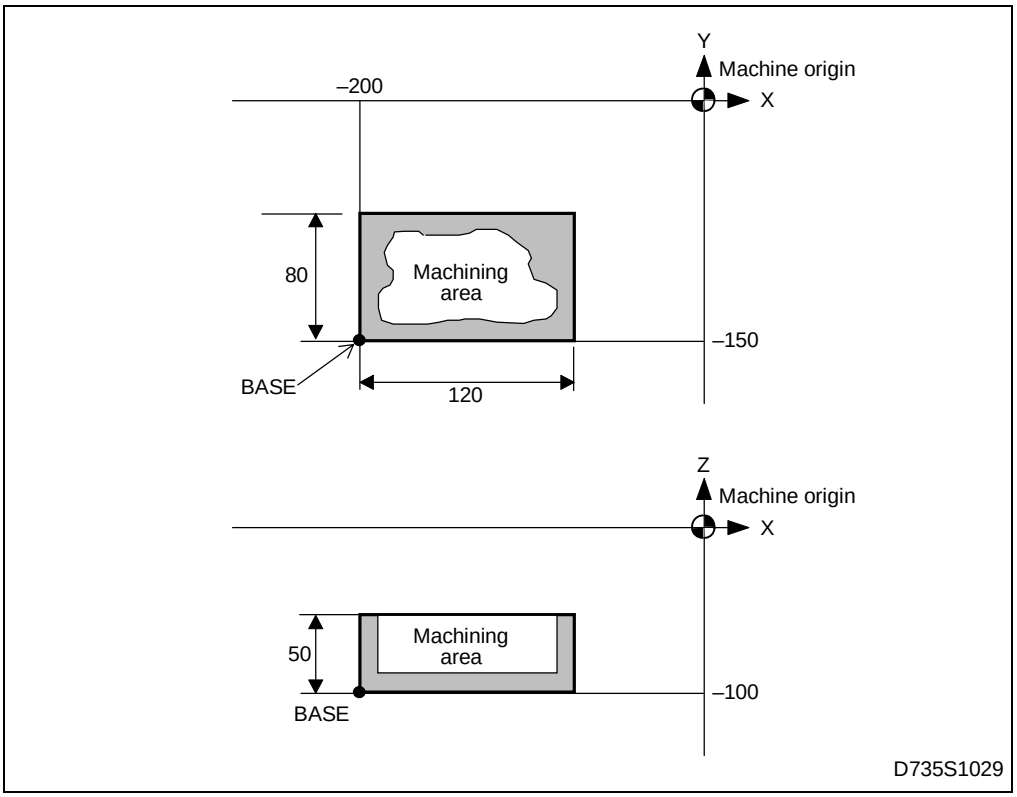
(2) Change the display size of the material as required.

- Select the “Change Material Size” option through “Setup” on the menu bar to display the window for changing the material size. The window initially presents the data of the rectangular parallelopiped that has been set automatically as mentioned above.



* BASE : Position of the material in the machine coordinate system
* SIZE : Size of the material (see the following figure)

D735S1028E



1 Change the coordinates of BASE and the dimensions of SIZE as required by placing the cursor on the desired text box first and then entering the corresponding value.

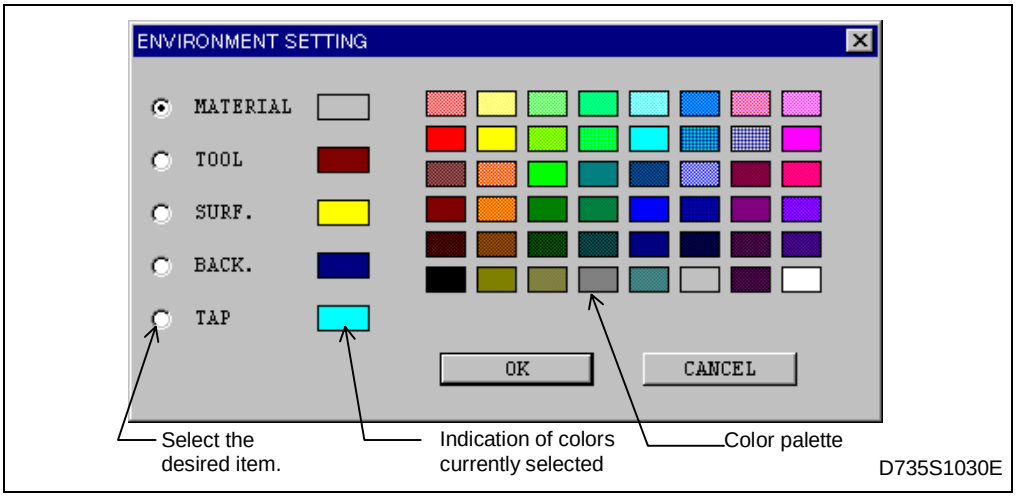
2 Click the OK button with the mouse, and the window will be closed.

3 The figure of the workpiece is now modified and redisplayed according to the entered new values.

4 Click the CANCEL button instead of OK to cancel the entered values in Step •and to keep the original ones unchanged.

(3) Change the display color for each item as required.

- Select the “Environment Setup” option through “Setup” on the menu bar to display the window for changing the display color.



- The following display environment edit function can be executed from the above window:

1 Click an item the display color of which is to be changed: MATERIAL, TOOL, SURF. (machining surface), BACK. (background) or TAP.

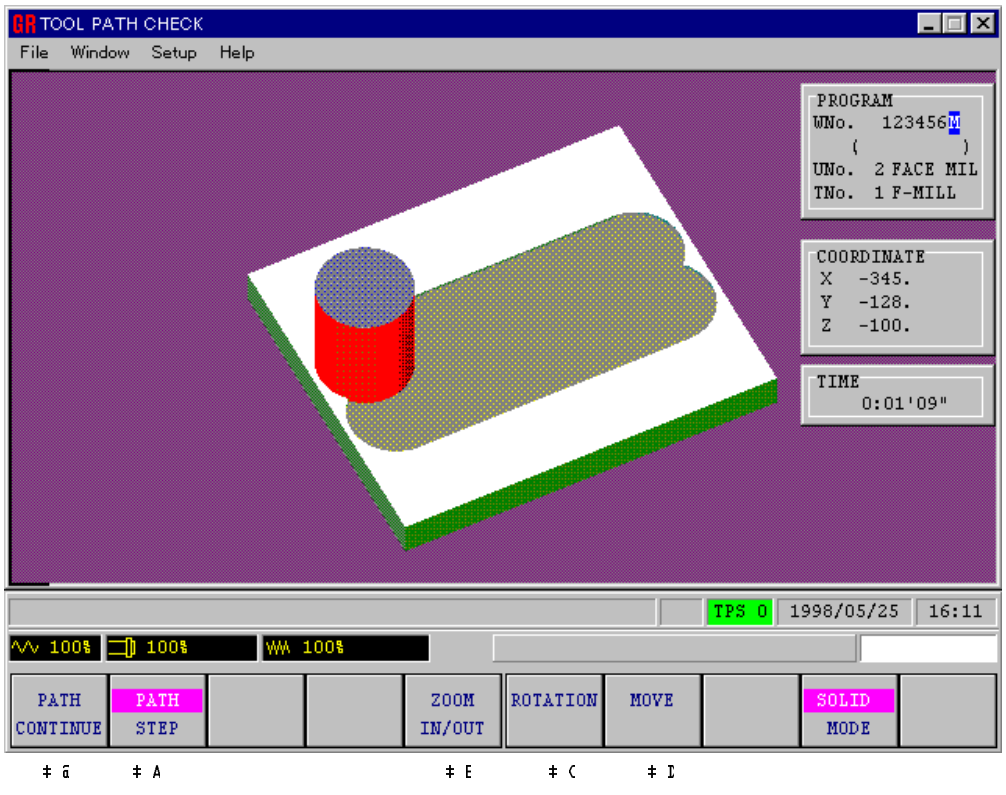
2 Change the color indicated in the box next to the item name by selecting the new color from the palette.

* Default colors are gray for MATERIAL, red for TOOL, yellow (or light blue for tap) for SURF. and blue for BACK.

* Definitely set the color selection by clicking the OK button, and the window will be closed and the display remade according to the new setting. Click the CANCEL button to retrieve the original setting.

(4) Execute the tool path check.

➔ In addition to the solid, machining type (number and name of the current unit), tool type (number and name of the current tool), current coordinates of the tool tip and machining time are simultaneously displayed as shown in the example below.



1 Continuous display of the tool path
PATH CONTINUE: refer to Subsection 6-2-3.

2 Intermittent display of the tool path
PATH STEP: refer to Subsection 6-2-3.

3 Enlargement and reduction of the solid
Press the **ZOOM IN/OUT** menu key and, with the menu item displayed in reverse, use the cursor keys ↑ ↓ to enlarge and reduce the solid given on the display. (See the explanatory table given below under **Note 2** for the function of the cursor keys.)

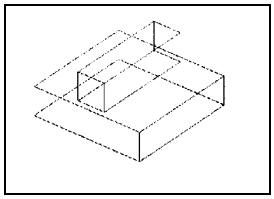
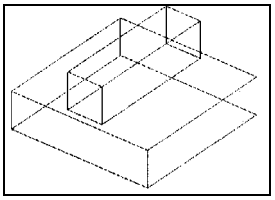
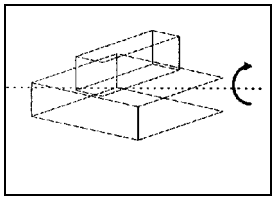
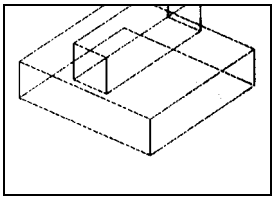
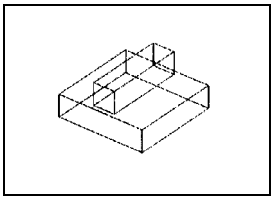
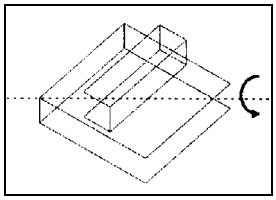
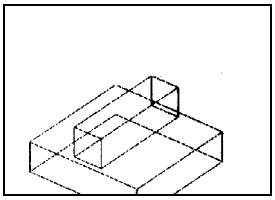
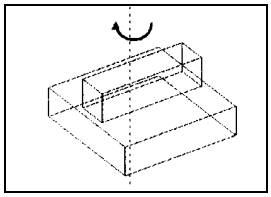
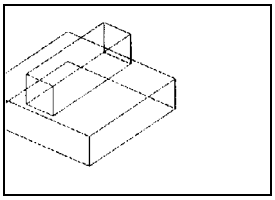
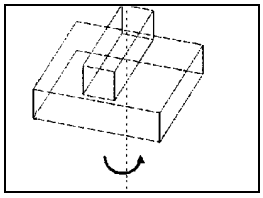
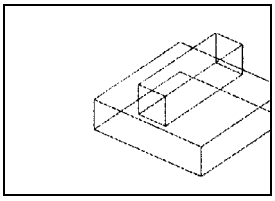
4 Rotation of the solid
Press the **ROTATION** menu key and, with the menu item displayed in reverse, use the cursor keys  to rotate the solid given on the display. (See the explanatory table given below under **Note 2** for the function of the cursor keys.)

5 Movement of the solid
Press the **MOVE** menu key and, with the menu item displayed in reverse, use the cursor keys  to move (shift) the solid given on the display. (See the explanatory table given below under **Note 2** for the function of the cursor keys.)

Note 1: Even during the tool path check, the functions of the menu items **ZOOM IN/OUT**, **ROTATION** and **MOVE** are effective to simultaneously execute the operations •, •and •.

Note 2: The function of the cursor keys for the modes ZOOM IN/OUT, ROTATION and MOVE is explained in the following table.

- Function of the cursor keys for ZOOM IN/OUT, ROTATION and MOVE functions

Current display of the solid			
			
Relationship between cursor keys and menu functions			
Selected function Cursor key	ZOOM IN/OUT	ROTATION	MOVE
↑	 Enlargement	 Right turn around horizontal axis	 Upward
↓	 Reduction	 Left turn around horizontal axis	 Downward
←	No function	 Right turn around Z-axis	 To the left
→	No function	 Left turn around Z-axis	 To the right

D735S1032

The solid display is modified as follows with each key operation:

- ZOOM IN/OUT: Enlarges the solid to 1.5 times or reduces it to 2/3 times.
- ROTATION: Rotates the solid through 15 degrees.
- MOVE: Moves the solid vertically or horizontally through 1/20 of the display area.

(5) Terminate the solid-mode display.

- Press the **SOLID MODE** menu key once again.

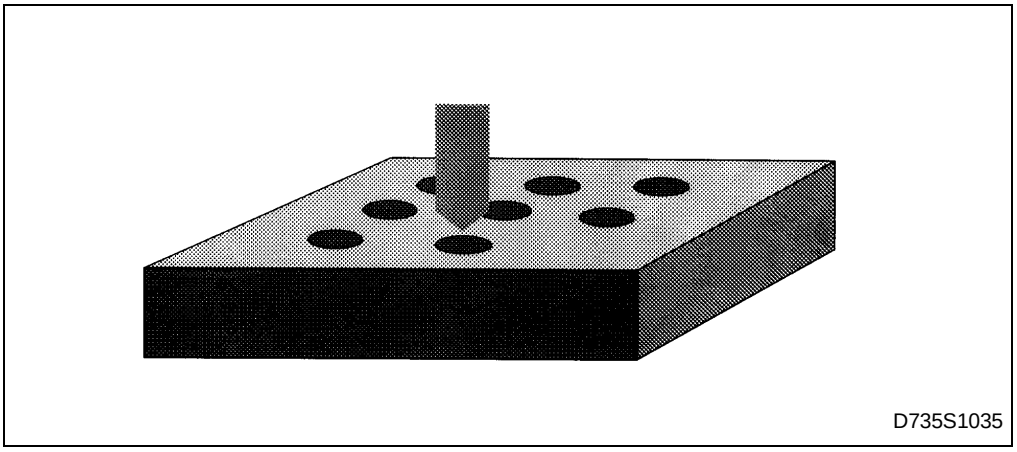
➔ Any operation in the solid mode (tool path drawing, display modification) will be forcibly terminated and the display mode will be reset to the normal mode.

2. Solid mode display for each unit type of MAZATROL programs

The solid mode display for tool path check can only be selected for an MAZATROL program currently designated on the **PROGRAM** display (as the main program). Selction of the solid mode for an EIA/ISO program (as the main program) causes the alarm **440 EIA/ISO PROGRAM DESIGNATED** to be given.
Solid mode display is made as follows for each unit type of MAZATROL programs:

A. Common unit

Multiple-workpiece machining specified in the common unit is only displayed as a repetitive machining in different sections on one and the same workpiece as shown below.

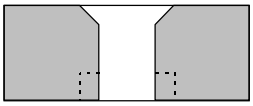
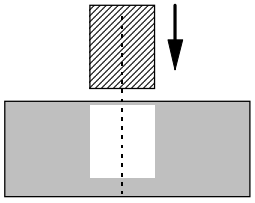
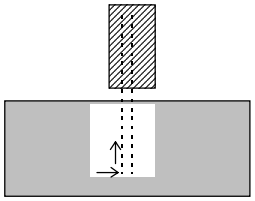


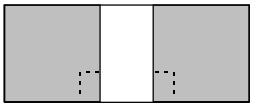
B. Machining unit

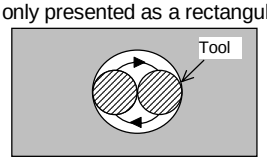
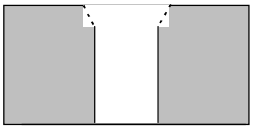
Point machining unit

•: Almost real image of machining and tool
•: Almost real image of machining with somewhat different image of tool
•: Rather different image from actual machining

No.	Unit	Tool	Display	Remarks
1	DRILLING	CTR-DR	•	Tool image displayed 
		DRILL	•	Tool image displayed 
		CHF-C	•	Tool image displayed 

No.	Unit	Tool	Display	Remarks
2	RGH CBOR	CTR-DR	•	Refer to DRILLING unit for tool image displayed.
		DRILL	•	•
		E-MILL	•	Tool image displayed 
		CHF-C	•	Refer to DRILLING unit for tool image displayed.
3	RGH BCB	CTR-DR	•	Refer to DRILLING unit for tool image displayed.
		DRILL	•	•
		CHF-C	•	•
		BK FACE	•	No tool image is displayed for a back facing tool in the solid mode. Machining image displayed is as follows:  * Back facing section is not presented at all.
4	REAMING	CTR-DR	•	Refer to DRILLING unit for tool image displayed.
		DRILL	•	•
		CHF-C	•	•
		CHP VAC	•	No tool image displayed.
		REAM	•	Tool image displayed 
		BOR BAR	•	Tool image displayed  Tool of the same dia. as hole diameter during cutting feed.  Tool of a smaller dia. than hole diameter for returning path. 
		E-MILL	•	Refer to RGH CBOR unit for tool image displayed.
5	TAPPING	CTR-DR	•	Refer to DRILLING unit for tool image displayed.
		DRILL	•	•

No.	Unit	Tool	Display	Remarks
		CHF-C	•	•
		CHP VAC	•	No tool image displayed.
		TAP	•	Tool image displayed  Machining surface in the selected color (Default: light blue)
6-1	BORE T1	CTR-DR	•	Refer to DRILLING unit for tool image displayed.
		DRILL	•	•
		E-MILL	•	Refer to RGH CBOR unit for tool image displayed.
		BOR BAR	•	Refer to REAMING unit for tool image displayed.
		CHF-C	•	Refer to DRILLING unit for tool image displayed.
6-2	BORE S1	CTR-DR	•	Refer to DRILLING unit for tool image displayed.
		DRILL	•	•
		E-MILL	•	Refer to RGH CBOR unit for tool image displayed.
		BOR BAR	•	Refer to REAMING unit for tool image displayed.
		CHF-C	•	Refer to DRILLING unit for tool image displayed.
6-3	BORE T2	CTR-DR	•	Refer to DRILLING unit for tool image displayed.
		DRILL	•	•
		E-MILL	•	Refer to RGH CBOR unit for tool image displayed.
		BOR BAR	•	Refer to REAMING unit for tool image displayed.
		CHF-C	•	Refer to DRILLING unit for tool image displayed.
7	BK-CBORE	CTR-DR	•	Refer to DRILLING unit for tool image displayed.
		DRILL	•	•
		E-MILL	•	Refer to RGH CBOR unit for tool image displayed.
		BOR BAR	•	Refer to REAMING unit for tool image displayed.
		CHF-C	•	Refer to DRILLING unit for tool image displayed.
		B-B BAR	•	No tool image is displayed for a back boring tool in the solid mode. Machining image displaed is as follows:  * Back boring section is not presented at all.
8	CBOR-TAP	CTR-DR	•	Refer to DRILLING unit for tool image displayed.
		DRILL	•	•
		CHF-C	•	•
		CHP VAC	•	No tool image displayed.
		TAP	•	Refer to TAPPING unit for tool image displayed.
9	CIRC MIL	E-MILL	•	Refer to RGH CBOR unit for tool image displayed.
		CHF-C	•	Refer to DRILLING unit for tool image displayed.

No.	Unit	Tool	Display	Remarks
		E-MILL (Tornado cycle)	•	Tool image displayed  Chamfering section is only presented as a rectangular cutting. • Top view  • Sectional view 

- Tapping and reaming tools are presented like an end-milling tool as shown above.

Line machining unit

•: Almost real image of machining and tool

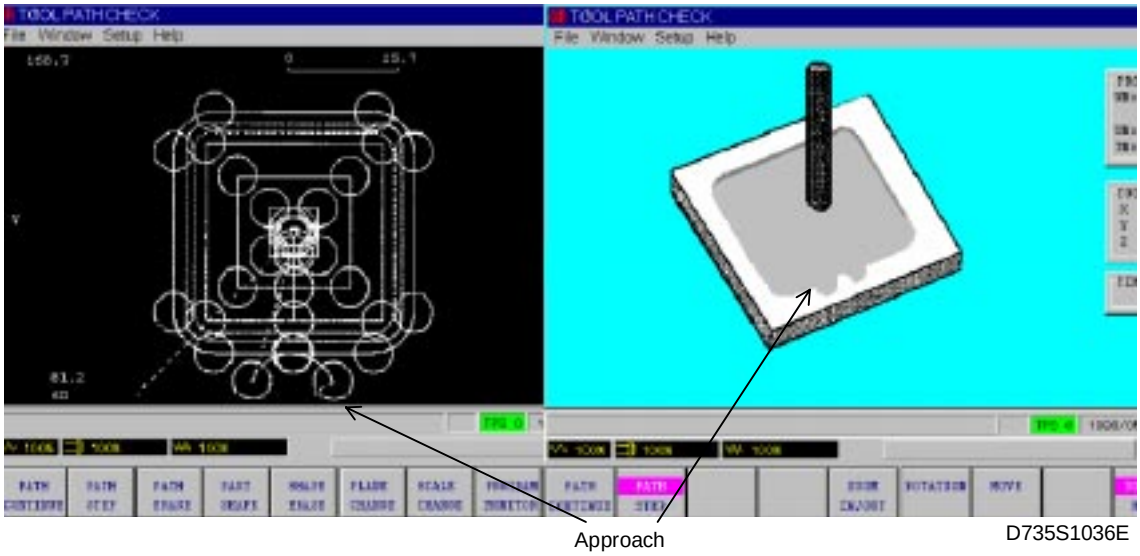
No.	Unit	Tool	Display	Remarks
1	LINE CTR	E-MILL	•	Refer to RGH CBOR unit for tool image displayed.
2	LINE RGT	E-MILL	•	•
3	LINE LFT	E-MILL	•	•
4	LINE OUT	E-MILL	•	•
5	LINE IN	E-MILL	•	•
6	CHMF RGT	CHF-C	•	Refer to DRILLING unit for tool image displayed.
7	CHMF LFT	CHF-C	•	•
8	CHMF OUT	CHF-C	•	•
9	CHMF IN	CHF-C	•	•

Face machining unit

•: Almost real image of machining and tool

No.	Unit	Tool	Display	Remarks
1	FACE MIL	F-MILL	•	Tool image displayed 
2	TOP EMIL	E-MILL	•	Refer to RGH CBOR unit for tool image displayed.
3	STEP	E-MILL	•	Refer to RGH CBOR unit for tool image displayed.
4	POCKET	E-MILL	•	Refer to RGH CBOR unit for tool image displayed.
		CHF-C	•	Refer to DRILLING unit for tool image displayed.
5	PCKT VLY	E-MILL	•	Refer to RGH CBOR unit for tool image displayed.
6	SLOT	E-MILL	•	Refer to RGH CBOR unit for tool image displayed.
7	3D	E-MILL	•	Refer to RGH CBOR unit for tool image displayed.
		B-E-MIL	•	Tool image displayed 

Note: For line/face machining units, the displayed workpiece, generally different from the actual one, may appear as if it were overcut during approach and escape path.



D735S1036E

C. Special unit

- M-CODE unit: Ignored.
- SUB PRO unit (for call of a MAZATROL program):
  The tool path is drawn for the SUB PRO unit in the solid mode according to the machining units (Point, Line, Face) designated in the MAZATROL subprogram.
- SUB PRO unit (for call of an EIA/ISO program):
  The last tool in the flow of the main program is used further for the solid display of the whole EIA/ISO subprogram.

  * Any tool change command in the EIA/ISO subprogram is ignored (the current tool image remains unchanged).
- PALL CHG (pallet change) unit: Ignored.
- INDEX unit: Ignored.

D. MANU PRO unit

The solid display is given with the tool designated in the manual-program unit.

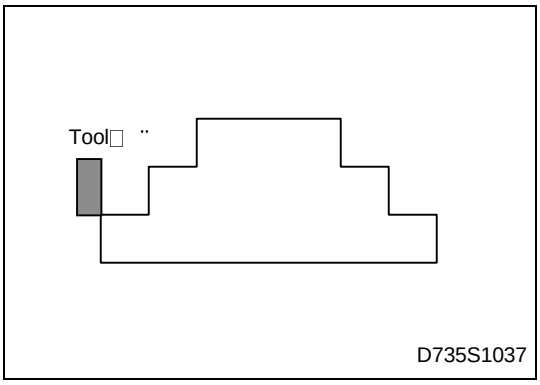
* Refer to the description in “2. Machining unit” for the tool image displayed.

E. MMS unit

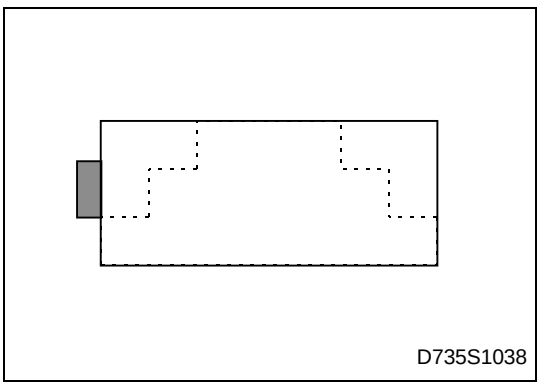
- T. SENS. (Touch sensor) is not displayed, but the position counter indicates the movement.

3. Solid-mode display for a stepped workpiece

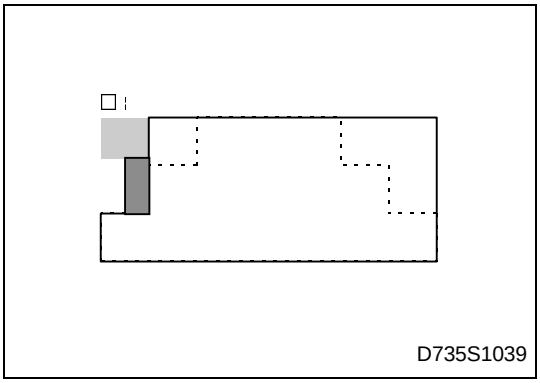
1. A real machining image is as follows:



2. The solid-mode display where the workpiece is always presented as a rectangular parallelopiped including whole machining shape is now as follows:



3. Even a section not covered by a tool image display is surely cleared in the flow of machining as shown below.



- This section is also cleared simultaneously with the feed motion of the displayed tool.

6-3 SHAPE CHECK Display

Function:

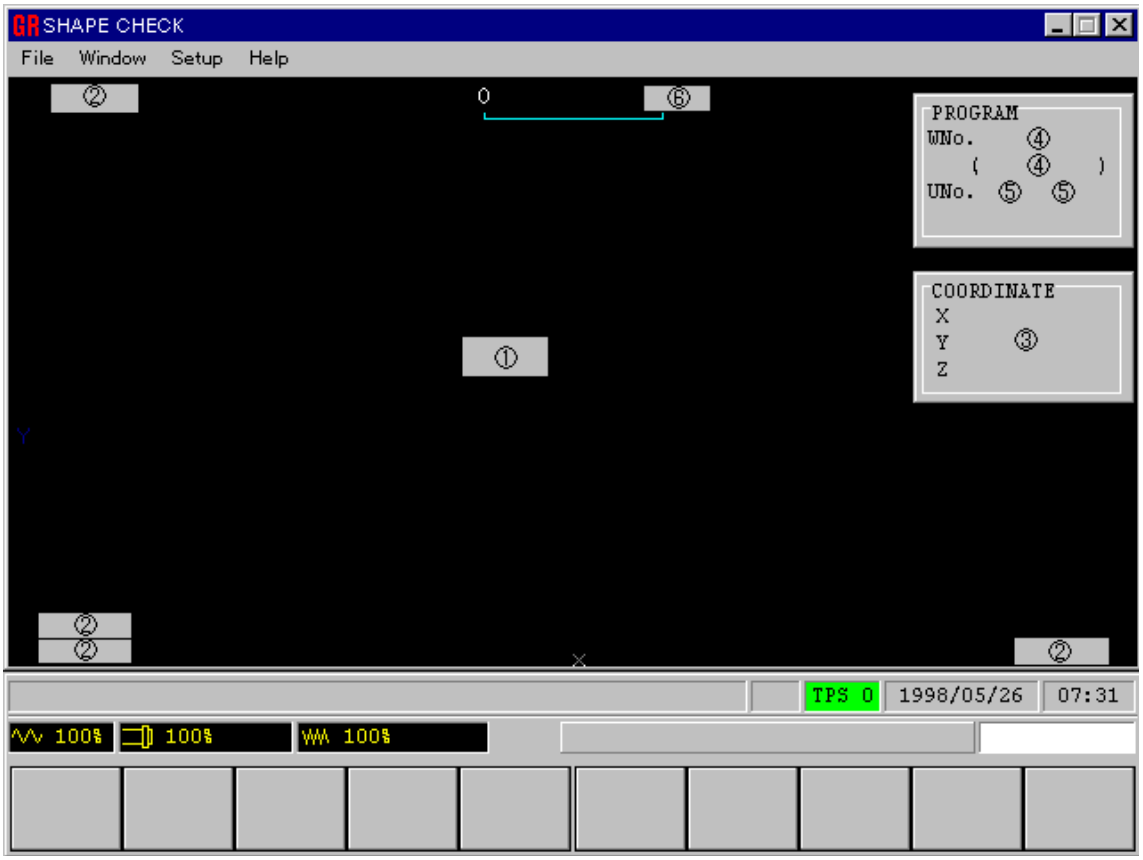
The machining shapes that have been defined using the machining program are displayed. (The machining shapes between one WPC unit and the next WPC unit are displayed.)

Remarks:

- This display can be selected when the cursor is at the unit position during MAZATROL program creation.
- The plane on which a machining shape is to be displayed can be selected.
- Scale value for displaying machining shapes can be changed.

6-3-1 Data display

1. Displayed data



D735S1040E

2. Display data description

No.	Item	Unit	Data description
•	(Shape)	•	Machining shape that has been defined by the program (For point-machining, this data is displayed with a purple line. For line- or face-machining, this data is displayed with a green line.)
•		mm (inch)	Display limit (automatically determined according to the machining shape)
•	X, Y, Z	mm (inch)	Coordinate values of the leading end of the currently displayed machining shape in the workpiece coordinate system
•	WNo.	•	Work number of the program under which the shape check is to be made
•	UNo.	•	Identification number of the program unit under which the shape check is being performed
•	0 ├──────┤	mm (inch)	Scale value displayed in graphic form (On the initial display, this data is automatically determined according to the machining shape.)

6-1-2 Displaying machining shape

1. Continuous display

(1) Press the **SHAPE CONTINUE** menu key.

➔ This will cause the display status of **SHAPE CONTINUE** to reverse and all the machining shapes that have been defined using the program to be displayed continuously.

- After display of the machining shapes, the reverse-display status of **SHAPE CONTINUE** will be cleared.

2. Display on a program unit basis

(1) Press the **SHAPE UNIT** menu key.

➔ This will cause the display status of **SHAPE UNIT** to reverse and the machining shape that has been defined using the first program unit to be displayed.

- If the **SHAPE UNIT** menu key is pressed once again, the machining shape that has been defined using the second program unit will be displayed.
- Similarly, each time the **SHAPE UNIT** menu key is pressed, the machining shape that has been defined using each program unit will be displayed.
- After display of all the machining shapes, the reverse-display status of **SHAPE UNIT** will be cleared.

3. Intermittent display

(1) Press the **SHAPE STEP** menu key.

➔ The display of **SHAPE STEP** will then be reversed, and each time this menu key is pressed, the machining shapes that have been defined using the program will be displayed intermittently.

- After display of all the machining shapes, the reverse-display status of **SHAPE STEP** will be cleared.

6-1-3 Other menu functions

1. If the **PLANE CHANGE** menu key is pressed, it becomes possible to select the plane or 3-D view on which machining shapes are to be displayed. For details of the selecting procedure, see the Subsection **4-2-3 Plane select** that is given in Section **4-2 TRACE Display**. If the plane select is not carried out, the machining shape available on the XY plane will be displayed.
2. If the **SCALE CHANGE** menu key is pressed, it becomes possible to display machining shapes in enlarged or reduced size. In addition, the 3-D display viewpoint can be changed if a 3-D view has been selected during plane-select operation. For details of these operations, see the Subsection **4-2-4 Change of the scaling factor** that is given in Section **4-2 TRACE Display**. If this operation has not been carried out, then machining shapes will be displayed at the automatically determined scaling factor.
3. Pressing the **SHAPE ERASE** menu key causes erasure of the machining shape. In addition, by pressing this menu key, it is possible to erase the machining shape currently being displayed.
4. If the menu display status is changed to reverse form by pressing the **STORE** menu key, the shape data will be preserved even after the display has been changed over to another display.
5. Pressing the **PROGRAM** menu key causes the current display to change over to the **PROGRAM** display.

6-4 PLANE CHECK Window

Function:

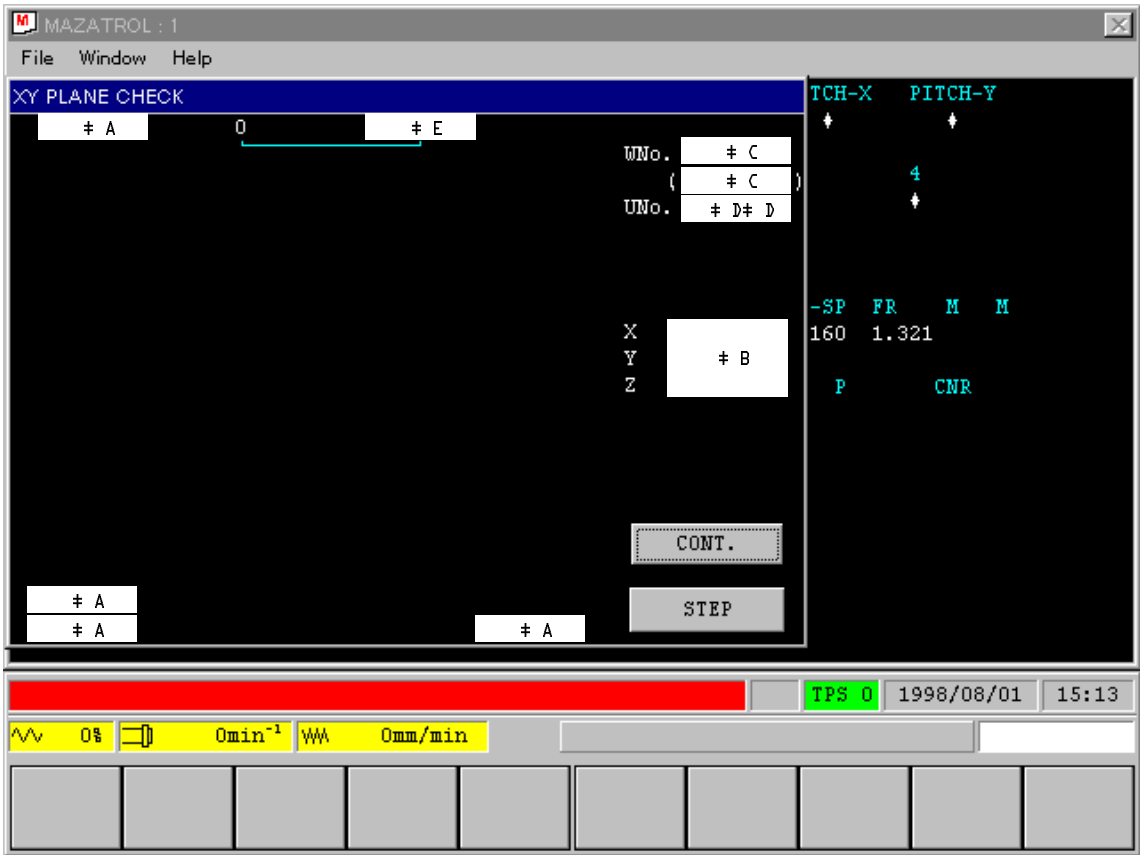
The machining shapes of the program unit are displayed.

Remarks:

- This window can be selected when the cursor is on the shape sequence line during MAZATROL program creation.
- The machining shapes are displayed on the XY plane.

6-4-1 Data display

1. Displayed data



D735S1041E

2. Display data description

No.	Item	Unit	Data description
•	(Shape)	•	Machining shape that has been defined by the program (For point-machining, this data is displayed with a purple line. For line- or face-machining, this data is displayed with a green line.)
•		mm (inch)	Display limit (automatically determined according to the machining shape)
•	X, Y, Z	mm (inch)	Current position of the tool tip in the workpiece coordinate system
•	WNo.	•	Work number of the program under which the plane check is to be made.
•	UNo.	•	Identification number of the program unit under which the plane check is being performed
•	0 ⊢——⊣	mm (inch)	Scaling factor for graphic display (on the primary display, automatically determined according to the type of machining shape)

6-1-2 Displaying machining shape

Press the **PLANE CHECK** menu key to display the PLANE CHECK window.

1. Continuous display

(1) Operate the cursor keys to move the foucus onto the button CONT. in the window and press the INPUT key.

➔ The machining shapes that have been defined in the shape sequence line where the cursor has been blinking on the **PROGRAM** display will be displayed continuously.

2. Intermittent display

(1) Operate the cursor keys to move the foucus onto the button STEP in the window and press the INPUT key.

➔ Each time the INPUT key is pressed, the machining shapes that have been defined in the shape sequence line where the cursor has been blinking on the **PROGRAM** display will be displayed intermittently.

6-1-3 Others

1. Only the machining shape on the XY plane can be indicated in the PLANE CHECK window.
2. The scaling factor for the PLANE CHECK window is automatically determined according to the machining shape. Thus, the operator cannot change the scaling factor.
3. To abort the display of machining shapes, press the CLEAR key.
4. Press the **PLANE CHECK** menu key anew to close the window.

6-5 SECTION CHECK Window

Function:

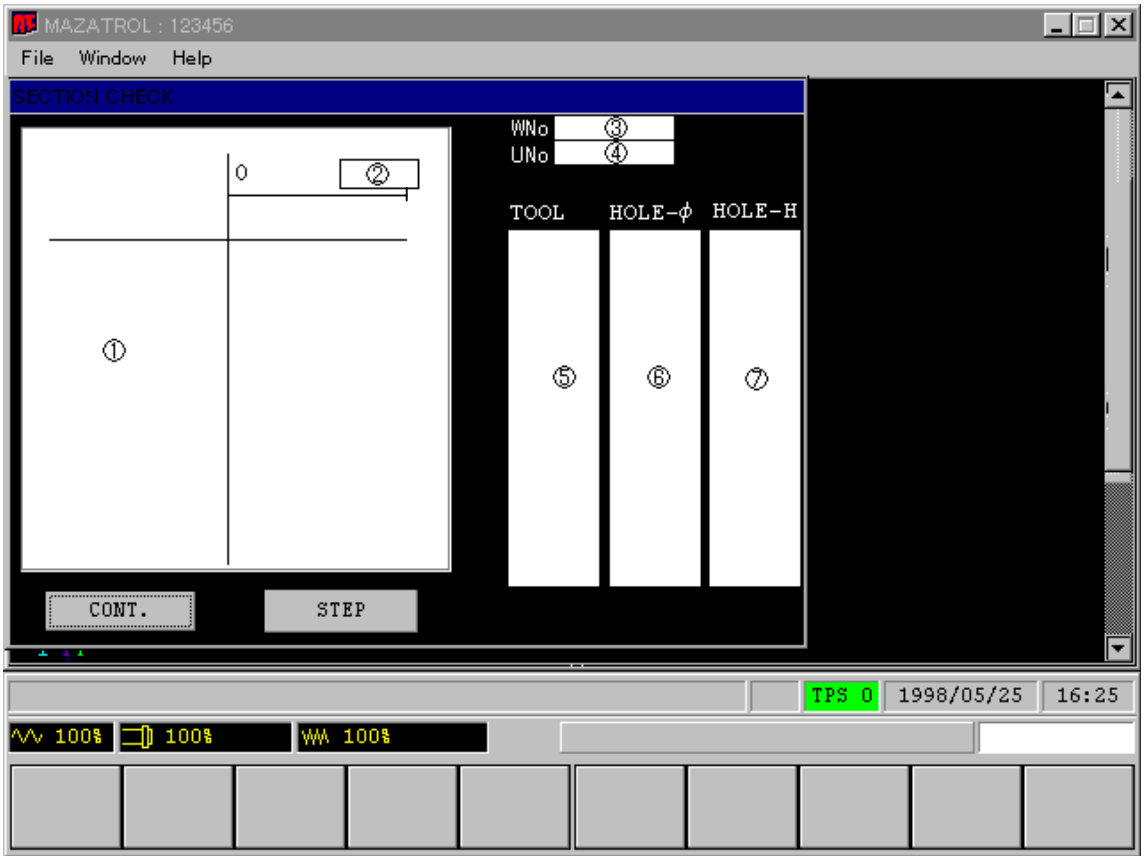
The machining shapes that have been defined on the point-machining shape sequence line are displayed in sectional view.

Remarks:

- This window can be selected when the cursor remains on the point-machining shape sequence line during MAZATROL program creation.
- The sectional shapes are displayed on the XZ plane.

6-5-1 Data display

1. Displayed data



D735S1042E

2. Display data description

No.	Item	Unit	Data description
•	(Sectional shape)	•	Point-machining sectional shape
•	0 ⊢——⊣	mm (inch)	Scaling factor for graphic display (automatically determined according to the type of machining shape)
•	WNo.	•	Work number of the program under which the sectional check is to be made
•	UNo.	•	Identification number and name of the program unit under which the sectional check is being performed
•	TOOL	•	Name of the tool being used during the sectional check
•	HOLE-ϕ	mm (inch)	Diameter of the hole to be drilled (*1)
•	HOLE-H	mm (inch)	Height of the hole to be drilled (*2)

(*1) The amount of chamfering will be displayed in this position if the tool in use is a chamfering tool.

(*2) This data will be displayed if the tool in use is either a spot drill or a chamfering tool.

6-1-2 Displaying sectional shapes

Press the **SECTION CHECK** menu key to display the SECTION CHECK window.

1. Continuous display

(1) Operate the cursor keys to move the foucus onto the button CONT. in the window and press the INPUT key.

- The point-machining sectional shapes that have been defined in the line where the cursor is blinking on the **PROGRAM** display will be indicated continuously.

2. Intermittent display

(1) Operate the cursor keys to move the foucus onto the button STEP in the window and press the INPUT key.

- Each time the INPUT key is pressed, the point-machining sectional shapes that have been defined in the line where the cursor is blinking on the **PROGRAM** display will be indicated intermittently.

6-1-3 Others

1. Only the sectional shape on the XZ plane can be indicated on the SECTION CHECK window.
2. The scaling factor for the SECTION CHECK window is automatically determined according to the machining shape. Thus, the operator cannot change the scaling factor.
3. Press the **SECTION CHECK** menu key anew to close the window.

6-6 PROGRAM FILE Display

Function:

Displaying the work numbers and other data of the programs pre-registered in the NC unit

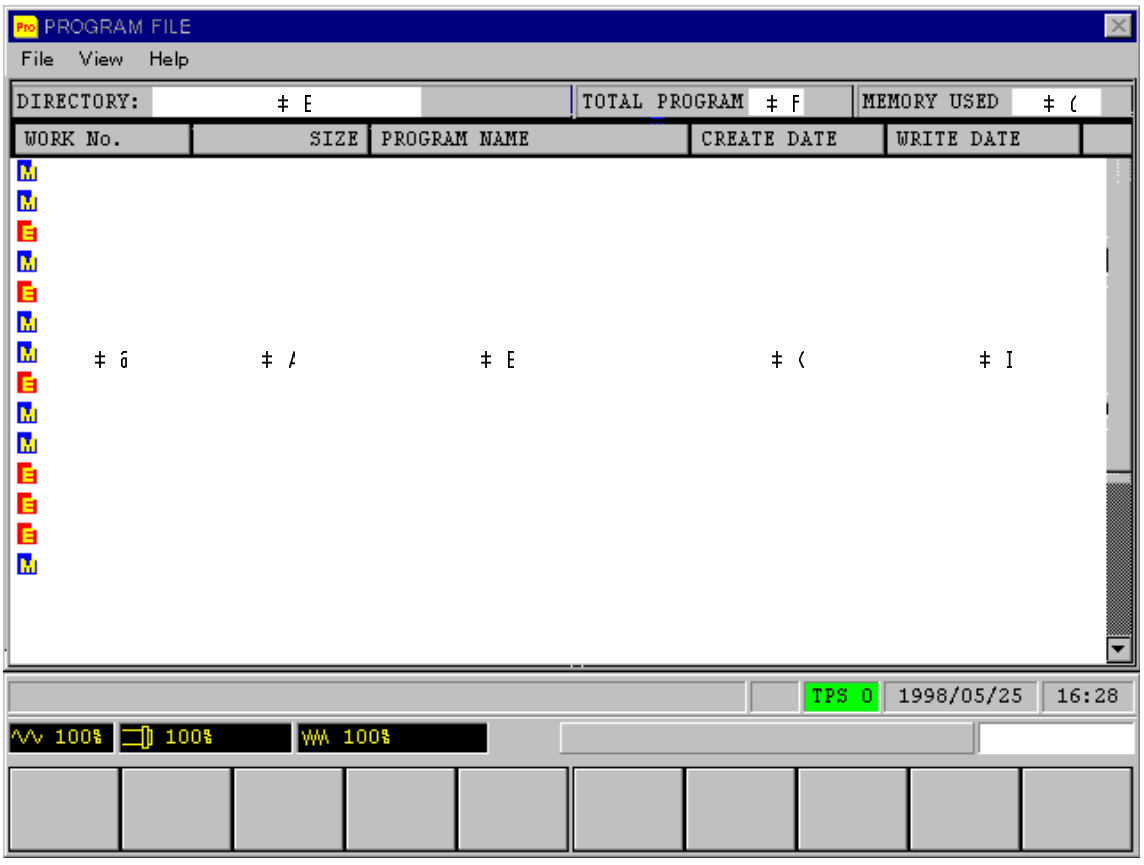
Remarks:

The following operations can be carried out for the pre-registered programs:

- Name assignment
- Work number change
- Erasure
- Program copy
- Converting MAZATROL programs to EIA/ISO programs (option)
- Directory change
- Program transfer

6-6-1 Data display

1. Displayed data



D735S1043E

2. Display data description

No.	Item	Unit	Data description
•	WORK No.	•	- Work number of a pre-registered program, and program type identification (MAZATROL programs or EIA/ISO programs) M: MAZATROL program E: EIA/ISO program
•	SIZE	•	Number of blocks in the registered program
•	PROGRAM NAME	•	Name of the registered program
•	CREATE DATE	Y/M/D	Date when the program was created
•	WRITE DATE	Y/M/D	Date of the last edition of the program
•	DIRECTROY	•	Name of the currently selected directory
•	TOTAL PROGRAM	•	Number of registered programs
•	MEMORY USED	•	Usage ratio of the memory **a** % **a**: Usage ratio (=[Used memory]/[Total memory])

6-1-2 Program name setting

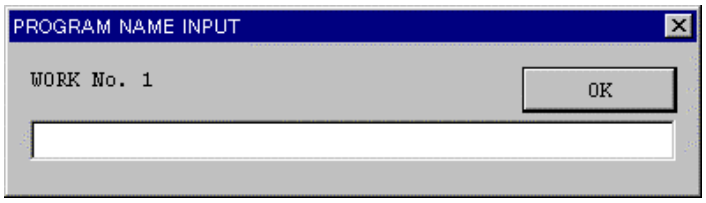
For easy identification of programs pre-registered in the NC unit, a name consisting of 48 alphabetic/numeric letters or less can be assigned to each program.
Even without a program name, however, program execution itself is not affected.

(1) Place the cursor on the work number for which a program name is to be input.

- After calling up the cursor on the screen with the cursor key, move the cursor to the work number of the intended program.

(2) Press the **NAME INPUT** menu key.

➔ The display status of **NAME INPUT** changes to reverse and the following window appears.



(3) Set a program name and press the INPUT key.

➔ The name is set on the line where the cursor is presented.

- Set a program name using address/numeric data keys.

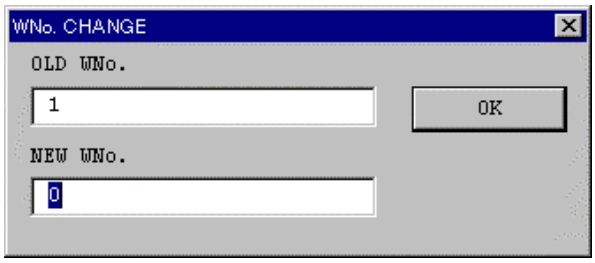
6-1-3 Work number change

A work number is set during creation of a program. The work number of each program pre-registered in the NC unit can be changed as required.

(1) Place the cursor on the work No. to be changed.

(2) Press the **PROGRAM RENUMBER** menu key.

➔ The display status of **PROGRAM RENUMBER** changes to reverse, and the following window appears.



(3) Set a new work number and press the INPUT key.

➔ The old work number is overriden with the new one.

Note: At step (3), if you set the work number of another program pre-registered in the NC unit, alarm **433 SAME PROGRAM EXISTS** will be displayed.

6-1-4 Program erasure

This operation refers to erasing each NC-registered program separately. Carry out this operation if a registered program becomes unnecessary.

(1) Press the **PROGRAM ERASE** menu key.

➔ The display status of **PROGRAM ERASE** changes to reverse.

(2) Set the work number of the program to be erased and press the INPUT key.

➔ The selected program is erased.
However, if the automatic operation work number designated on the **POSITION** display is set and then the INPUT key is pressed, the monitor will display the message **ACTIVE PROGRAM ERASE <INPUT>**. To execute erasure, press the INPUT key once again, or to cancel erasure, press any other key.

Note: If you set the work number of a program not pre-registered in the NC unit at step (2), alarm **405 PROGRAM No. NOT FOUND** will be displayed.

6-1-5 Program copy

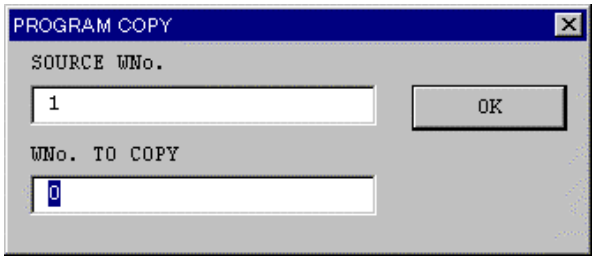
This operation enables the previously registered programs within the NC unit to be copied. Use this function to newly create the similar programs as those which have been registered.

(1) Move the cursor to the line where the work number of the program to be copied is displayed.

- The cursor can be set at any of the items present on the line where the intended work number is displayed.

(2) Press the **PROGRAM COPY** menu key.

- **PROGRAM COPY** will then reverse and the following window will appear.



(3) Enter a new work number, and press the INPUT key.

➔ A program having the same contents as those of the original will be created under the new work number.

Note: At step (3), if you set the work number of another program pre-registered in the NC unit, alarm **433 SAME PROGRAM EXISTS** will be displayed.

6-1-6 All program erasure

This operation refers to erasing all NC-registered programs at one time.

(1) Press the **ALL ERASE** menu key.

➔ The display status of **ALL ERASE** changes to reverse.

(2) Key in –9999 and press the INPUT key.

➔ All the programs pre-registered in the NC unit are erased.

Note: If you set a value other than –9999, alarm **401 ILLEGAL FORMAT** will be displayed.

6-1-7 Converting MAZATROL program to EIA/ISO program

EIA/ISO conversion is a function that converts an NC-registered MAZATROL program into an EIA/ISO program. For details, refer to Section **11-7 EIA/ISO Conversion.**

6-1-8 Opening the shape drawing window

Shape drawing is a function that draws the shape data of an NC-registered program in the window. See the Section, **POSITION Display** to check further details of the drawing function.

(1) Set the cursor on the work number of the program that contains the shape data to be drawn.

- After displaying the cursor on the monitor by pressing the cursor key, move the cursor to the work number of the program that contains the shape data to be drawn.

(2) Press the **PART SHAPE** menu key.

➔ **PART SHAPE** will reverse and the shape drawing window will open.
See the Section, **POSITION Display** to check the types of data displayed in the shape drawing window.

(3) Press the **PART SHAPE** menu key.

- Press the **PART SHAPE** menu key again if the shape drawing window is to be cleared.

➔ **PART SHAPE** will then be displayed in the normal-display status and the shape drawing window will close.

Note: Alarm **465 EIA SHAPE DATA NOT FOUND** will be displayed at step (2) if an EIA/ISO program without shape data is designated.

6-1-9 Directory change

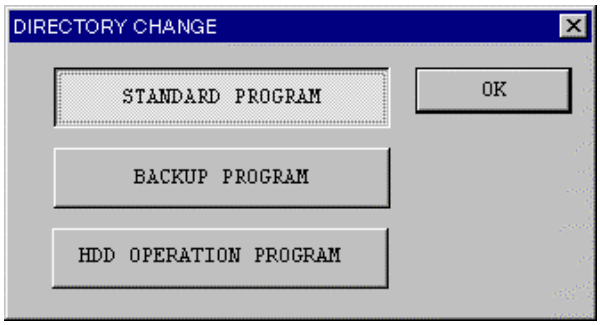
The program storage area in the NC unit is divided into the following three sub-areas:

- STANDARD PROGRAM area:
  Stores programs created on the **PROGRAM** display and used for MEMORY operation.
- BACKUP PROGRAM area:
  Backs up each program.
- HDD OPERATION PROGRAM area:
  Stores programs used for Hard Disk operation.

Use the following procedure to change the area of the programs to be displayed on the **PROGRAM FILE** display:

(1) Press the **DIR. CHANGE** menu key.

➔ The display status of **DIR. CHANGE** changes to reverse and the following window appears:



(2) Using the cursor keys, select the desired storage area.

(3) Press the INPUT key.

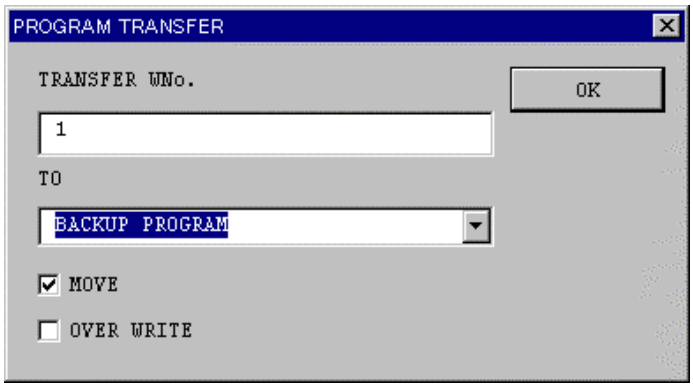
➔ The programs stored in the selected area will be listed on the display.

6-1-10 Program transfer

Use the following procedure to transfer a program from one storage area to another:

(1) Press the **PROGRAM TRANSFER** menu key.

➔ The display status of **PROGRAM TRANSFER** changes to reverse and the following window appears:



(2) Specify the work number of the program to be transferred and select the destination area.

➔ The specified program will be transferred to the selected storage area.

- Using the mouse, click the MOVE check box to clear the program from the source area upon completion of the transfer.
- Using the mouse, click the OVER WRITE check box to allow the data to be overwritten without causing an alarm in case the specified work number should be occupied in the destination area.

6-1-11 Program management functions

For some specific program numbers, erasure and editing of the programs and their call-out on the **PROGRAM** display can be prohibited by using parameters.
The following three functions are available:

- Prohibition of selected-program editing
- Prohibition of selected-program call
- Selected-program erasure

1. Prohibition of selected-program editing

Irrespective of the valid or invalid status of the write protect key, execution of the operations listed in the table below is disabled for the selected program.
The alarm **406 MEMORY PROTECT** will occur if an attempt is made to carry out these operations.

Type of display	Prohibited item
PROGRAM display (MAZATROL, EIA/ISO)	- Modifying the selected program
PROGRAM FILE display	- Erasing the selected program * Execution of ALL ERASE erases all programs, except the selected ones. - Registering the name of the selected program - Renumbering the selected program

2. Prohibition of selected-program call

Display of the contents of the selected program is prohibited.
The alarm **402 ILLEGAL NUMBER INPUT** will occur if an attempt is made to carry out for the selected program the operations listed in the table below.

Type of display	Prohibited item
PROGRAM display (MAZATROL, EIA/ISO)	- Searching for the work number of the selected program - Copying from the selected program (unit copying and shape copying included)
PROGRAM FILE display	- Renumbering the selected program
EIA MONITOR display	- Displaying the selected program for monitoring purposes (No alarms will result even if an attempt is made to call up the selected program.)
DATA I/O display (CMT, DNC, TAPE)	- Saving or punching the selected program * If ALL PUNCH is executed using TAPE mode, the selected program will not be punched. * The alarm **517** (or **587**) **PROG. OPERATION NOT ALLOWED** or **547 TAPE PUNCHER ERROR** will occur if an attempt is made to execute a saving or punching operation under a previously set status of the work number of the selected program. * Loading of the selected program is possible.

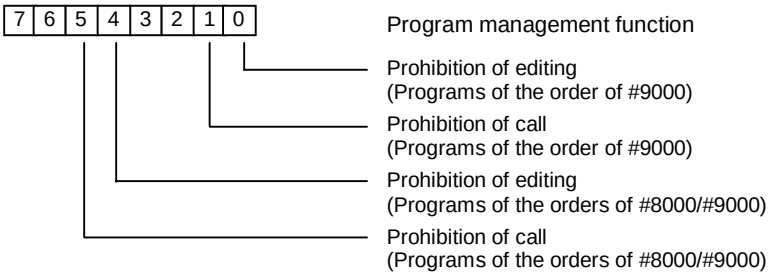
3. Selected-program erasure

All-program erasure, which was possible with the conventional all-program erasure function, has become impossible, because of addition of the new function described previously in 1. above. Only programs other than the selected ones are normally erased by all-program erasure.
Batch erasure of all programs (including the selected ones), however, is required in following cases.

- When installing the system
- If programs within the NC equipment have been destroyed

In those cases, batch erasure of all programs becomes possible by setting 1 in parameter **F82**.

4. Description of parameters

Address	Descriptions
F81	
F82	Total erasing of programs **0:** The programs other than the selected ones erased **1:** All programs erased

6-7 PROCESS CONTROL Display

Function:

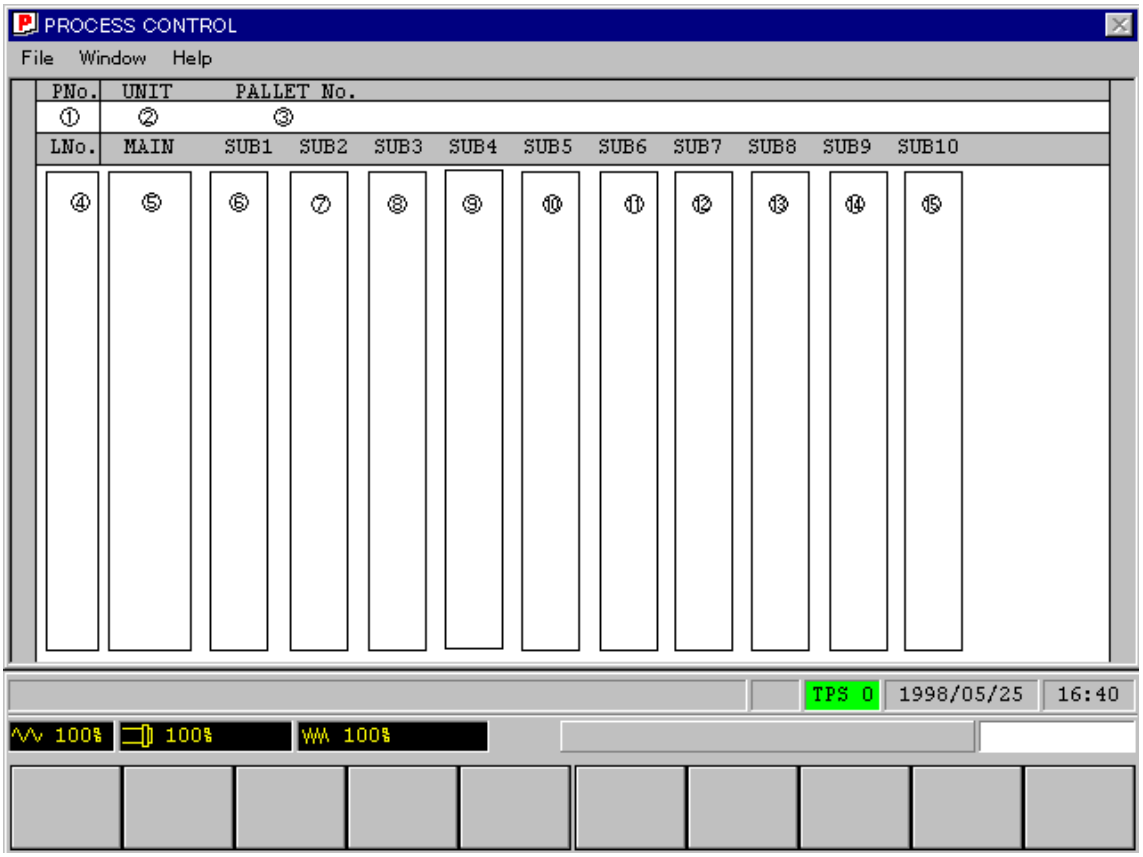
The logical linking status of processes within the program and the configurations of the subprograms within the processes are displayed.

Remarks:

Subprograms that are called up from the EIA/ISO programs will not be displayed on this display.

6-7-1 Data display

1. Displayed data



D735S1044E

2. Displayed data description

No.	Item	Unit	Data description
•	PNo.	•	Process number
•	UNIT	•	Name of the unit used to divide processes
•	PALLET No.	•	Pallet No. **a** (**b**) **a**: Number of the pallet to be used **b**: Number of the pallet to be used next
•	LNo.	•	Layout number
•	MAIN	•	Work number of the main program
•	SUB 1	•	Work number of the subprogram (Nesting 1)
•	SUB 2	•	Work number of the subprogram (Nesting 2)
•	SUB 3	•	Work number of the subprogram (Nesting 3)

No.	Item	Unit	Data description
•	SUB 4	•	Work number of the subprogram (Nesting 4)
•	SUB 5	•	Work number of the subprogram (Nesting 5)
•	SUB 6	•	Work number of the subprogram (Nesting 6)
•	SUB 7	•	Work number of the subprogram (Nesting 7)
•	SUB 8	•	Work number of the subprogram (Nesting 8)
•	SUB 9	•	Work number of the subprogram (Nesting 9)
•	SUB 10	•	Work number of the subprogram (Nesting 10)

6-1-2 Process-number search

In the initial status of the display, control data related to the first process of the specified program will be displayed. If control data related to the other processes of the specified program is to be checked on the screen, use the procedure described below.

(1) Press the **PROC. No. SEARCH** menu key.

➔ This will cause the display status of **PROC. No. SEARCH** to reverse.

(2) Using the appropriate numeric key(s), specify the identification number of the process whose control data is to be checked. Then, press the INPUT key.

➔ The input process number will then be displayed in the PNo. item (data portion •), and the control data of the specified process will be displayed on the screen.

➔ Subsequently, the reverse-display status of **PROC. No. SEARCH** will be released.

6-1-3 Other functions

1. The cursor can be moved up- and downward on a process basis by pressing the respective page key.
2. The display can be switched over to the **PROGRAM LAYOUT** display by pressing the **PROGRAM LAYOUT** menu key.

6-8 PROGRAM LAYOUT Display

Function:

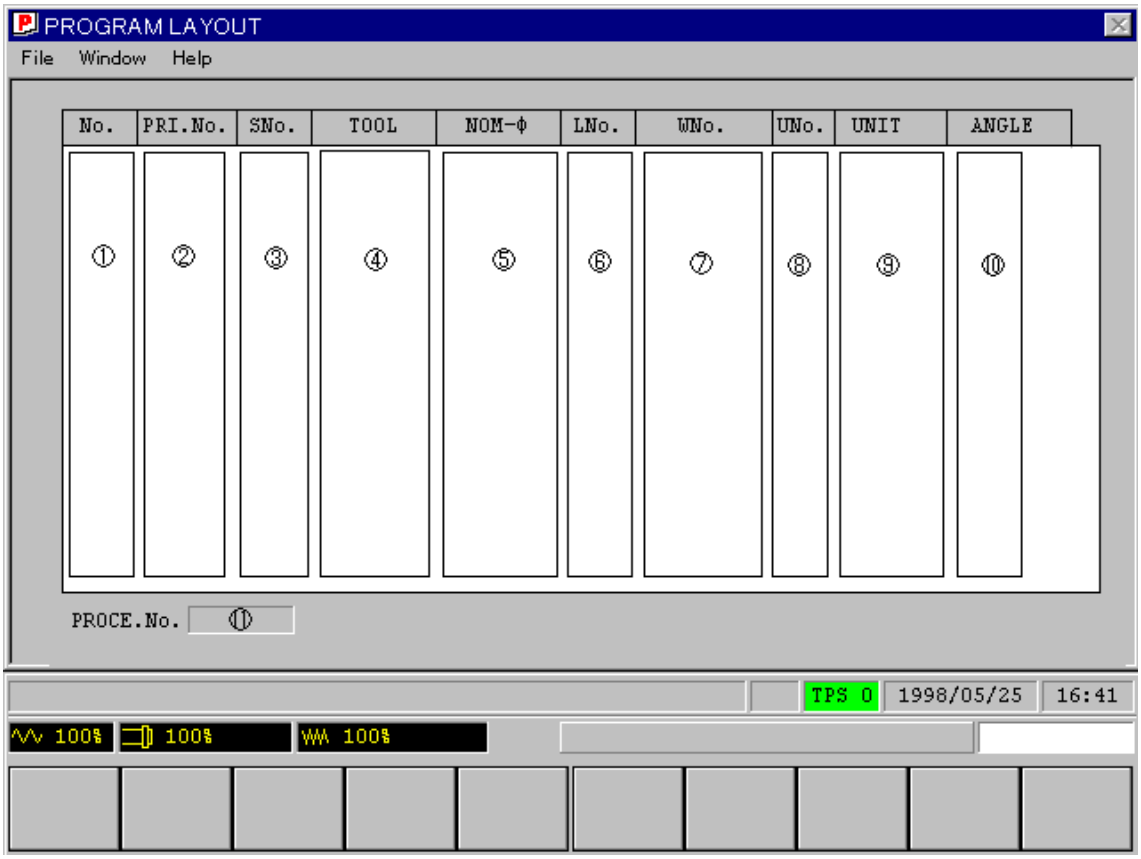
- The tools to be used in the processes of the MAZATROL program are displayed in the order of machining.
- This display can be used to edit a tool priority number and to write the data into the program.

Remarks:

- For both multi-workpiece machining and subprogram looping, only one time of program data is displayed.
- This display can be called up from the **PROGRAM** (MAZATROL) display, the **PROCESS CONTROL** display and the **DISPLAY MAP** display. When this display is called from the **PROGRAM** or **PROCESS CONTROL** display, process layout information for the cursor position is displayed. When the **PROGRAM LAYOUT** display is called from the **DISPLAY MAP** display, the process layout information is displayed from the head of the program.

6-8-1 Data display

1. Displayed data



D735S1045E

2. Display data description

No.	Item	Unit	Data description
•	No.	•	Order of machining
•	PRI. No.	•	Tool priority number
•	SNo.	•	Tool sequence number
•	TOOL	•	Tool name
•	NOM-ϕ	mm (inch)	Nominal diameter of the tool
•	LNo.	•	Layout number
•	WNo.	•	Work number of the program
•	UNo.	•	Unit number
•	UNIT	•	Name of the unit
•	ANGLE	degrees	Table rotation angle
•	PROCE.No.	•	Process number

6-1-2 Priority number setting

Tool priority numbers can be set on a tool sequence basis.

(1) Move the cursor to the PRI. No. item to be set.

(2) Input a machining priority number, and press the INPUT key.

➔ The machining priority number will be set in the position of the cursor.

Note: The operations described later in Subsection **6-8-8 Program write**, must be carried out before the priority number that has thus been edited on this display can be incorporated into the program.

6-1-3 Delay priority designation

A delay machining priority number can be designated during tool priority number setting.

(1) Move the cursor to the PRI. No. item for which delay machining is to be set.

(2) Press the **DELAY PRIORITY** menu key.

(3) Input a delay machining priority number, and press the INPUT key.

➔ The delay machining priority number will be set in the position of the cursor and the reverse-display status of **DELAY PRIORITY** will be released.

6-1-4 Priority number assignment

A tool priority number can be assigned to all tools of the same type within the process.

(1) Move the cursor to the PRI. No. item to be assigned.

(2) Press the **PRI. No. ASSIGN** menu key.

(3) Input a tool priority number, and press the INPUT key.

➔ The tool priority number will be assigned to all tools of the same type within the process, and the reverse-display status of **PRI. No. ASSIGN** will be released.

- Alarm **420 SAME DATA EXISTS** will result if the designated priority number already exists.

6-1-5 Priority number movement

A tool priority number can be moved.

(1) Move the cursor to the PRI. No. item to be moved.

(2) Press the **PRI. No. MOVE** menu key.

(3) Input a destination tool priority number , and press the INPUT key.

➔ The tool priority number will be moved, and the reverse-display status of **PRI. No. MOVE** will be released.

Example:

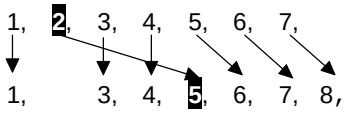
No.	PRI.No.	SNo.	TOOL	NOM-φ
1	1		F-MILL	50.
2	1		F-MILL	50.
3	1		F-MILL	50.
4	**2**		E-MILL	15.
5	2		E-MILL	15.
6	3		E-MILL	12.
7	3		E-MILL	12.
8	3		E-MILL	12.
9	4		DRILL	8.
10	4		DRILL	8.
11	5		DRILL	10.
12	5		DRILL	10.

Reverse display status of **PRI. No. MOVE** Setting “4”

No.	PRI.No.	SNo.	TOOL	NOM-φ
1	1		F-MILL	50.
2	1		F-MILL	50.
3	1		F-MILL	50.
4	4		E-MILL	15.
5	4		E-MILL	15.
6	3		E-MILL	12.
7	3		E-MILL	12.
8	3		E-MILL	12.
9	5		DRILL	8.
10	5		DRILL	8.
11	6		DRILL	10.
12	6		DRILL	10.

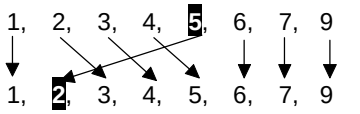
- All source tool priority numbers that have been designated using the cursor are overriden with destination tool priority numbers.
- When the priority number is inserted into the movement destination, the destination priority number and subsequent ones will increment by one if all these numbers are in succession.

1) A train of priority number of



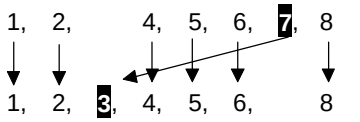
9 will become 9 if 2 is “moved” to 5.

2) A train of priority number of



will become if 5 is “moved” to 2.

3) A train of priority number of



will become if 7 is “moved” to 3, of which priority number has not been designated

* If, in procedural step (1), the cursor is set to a position in which a priority number has not been designated, only the tool sequence priority number at the cursor position will move.

Example:

No.	PRI.No.	SNo.	TOOL	NOM-ϕ
1	1		F-MILL	50.
2	1		F-MILL	50.
3	1		F-MILL	50.
4	[redacted]		E-MILL	15.
5	2		E-MILL	15.
6	3		E-MILL	12.
7	3		E-MILL	12.
8	3		E-MILL	12.
9	4		DRILL	8.
10	4		DRILL	8.
11	5		DRILL	10.
12	5		DRILL	10.

Reverse display status of **PRI. No. MOVE**

Setting "4"

No.	PRI.No.	SNo.	TOOL	NOM-ϕ
1	1		F-MILL	50.
2	1		F-MILL	50.
3	1		F-MILL	50.
4	4		E-MILL	15.
5	2		E-MILL	15.
6	3		E-MILL	12.
7	3		E-MILL	12.
8	3		E-MILL	12.
9	5		DRILL	8.
10	5		DRILL	8.
11	6		DRILL	10.
12	6		DRILL	10.

6-1-6 Priority number all erase

All tool priority numbers for the process can be erased.

(1) Press the **PRI. No. ALL ERAS** menu key.

➔ The display status of **PRI. No. ALL ERAS** will reverse.

(2) Press the INPUT key.

➔ All tool priority numbers will be erased and the reverse-display status of **PRI. No. All ERAS** will be released.

6-1-7 Store

The tool priority number being edited can be stored.
Before changing the display during editing, carry out the storage operation described below. Otherwise, the edited data will be erased when the display is changed.

(1) Press the **STORE** menu key.

➔ The display status of **STORE** will then reverse to indicate that the editing data is stored.

Note 1: If, during storage, the **PROGRAM LAYOUT** display is selected from the **PROGRAM** display, the program layout of the stored process data will be displayed, irrespective of the position of the cursor on the **PROGRAM** display.

Note 2: If, during storage, the **PROGRAM LAYOUT** display is selected from the **PROCESS CONTROL** display, the program layout of the stored process data will be displayed, irrespective of the position of the cursor on the **PROCESS CONTROL** display.

Note 3: In the cases listed below, the message **ERASE LAYOUT DATA Y<1>, N<0>?** is displayed. Input "1" if the layout data is to be erased, or input "0" if the data is not to be erased. The above message will not be displayed if **STORE** is cancelled by pressing the **STORE** menu key once again.

- If an attempt is made to modify the work number on the **PROGRAM** display (that is, if the **WORK No.** menu key is pressed)
- If an attempt is made to select the programming mode on the **PROGRAM** display (that is, if the **PROGRAM EDIT** menu key is pressed)

6-1-8 Program write

Data that has been set in the **PROGRAM LAYOUT** display can be written into the program. After editing, however, the program write operation described below must be carried out to incorporate the data into the program.

(1) Press the **PROGRAM WRITE** menu key.

➔ The display status of **PROGRAM WRITE** will reverse.

(2) Press the INPUT key.

➔ After the data has been written into the program, the reverse-display status of **PROGRAM WRITE** will be released and the display will be made once again in order with the first priority number first.

6-1-9 Simultaneous use of functions

Two of the three functions described above can be used at the same time. Selected two functions are executed at the same time.

	Functions that can be used simultaneously		
	Delay priority	Priority number assignation	Priority number movement
Delay priority		•	•
Priority number assignment	•		×
Priority number movement	•	×	

•: Can be used at the same time
×: Cannot be used at the same time

6-1-10 Program reference window

Selecting the “Program” option through “Window” on the menu bar allows a program reference window to be opened beginning at the unit of the tool sequence indicated by the cursor.

1. The program data displayed in the window cannot be scrolled.
2. Tool priority numbers can be set with the window remaining opened. After the priority number has been specified and the INPUT key pressed, the cursor moves downward to the next line and the window closes.
3. The window will be closed if a page key or a cursor key is pressed.

Note: Tool sequence of SNo. 11 onward cannot be displayed in this window. In order to monitor such a tool sequence, call up the **PROGRAM (MAZATROL)** display.

6-9 MACHINING NAVIGATION - PREDICT Display

Function:

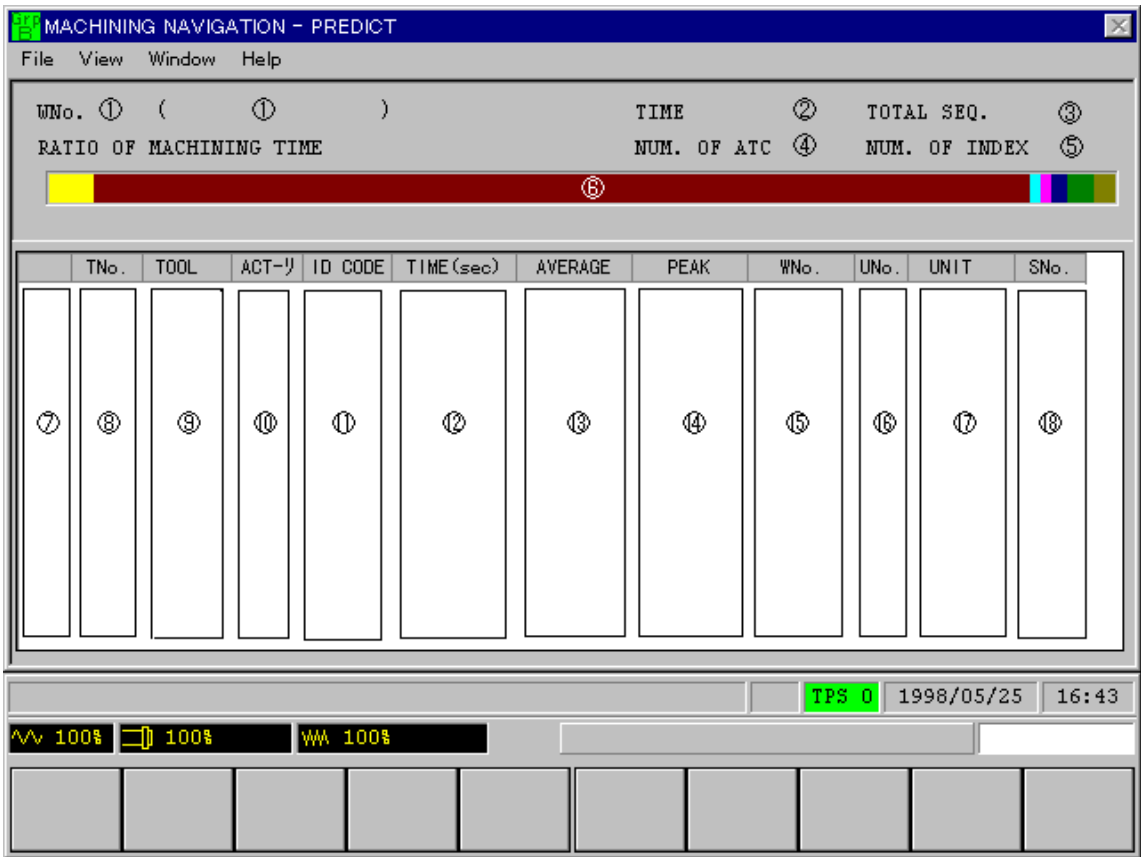
The machining time and spindle output data are listed for each tool according to the estimates made by the checking operation on the **TOOL PATH CHECK** display. The machining time can therefore be reduced effectively.

Remarks:

- The navigation function gives modification data so that cutting conditions can be optimized for each tool.
- Recommended tools are presented to allow more effective machining with each tool type.

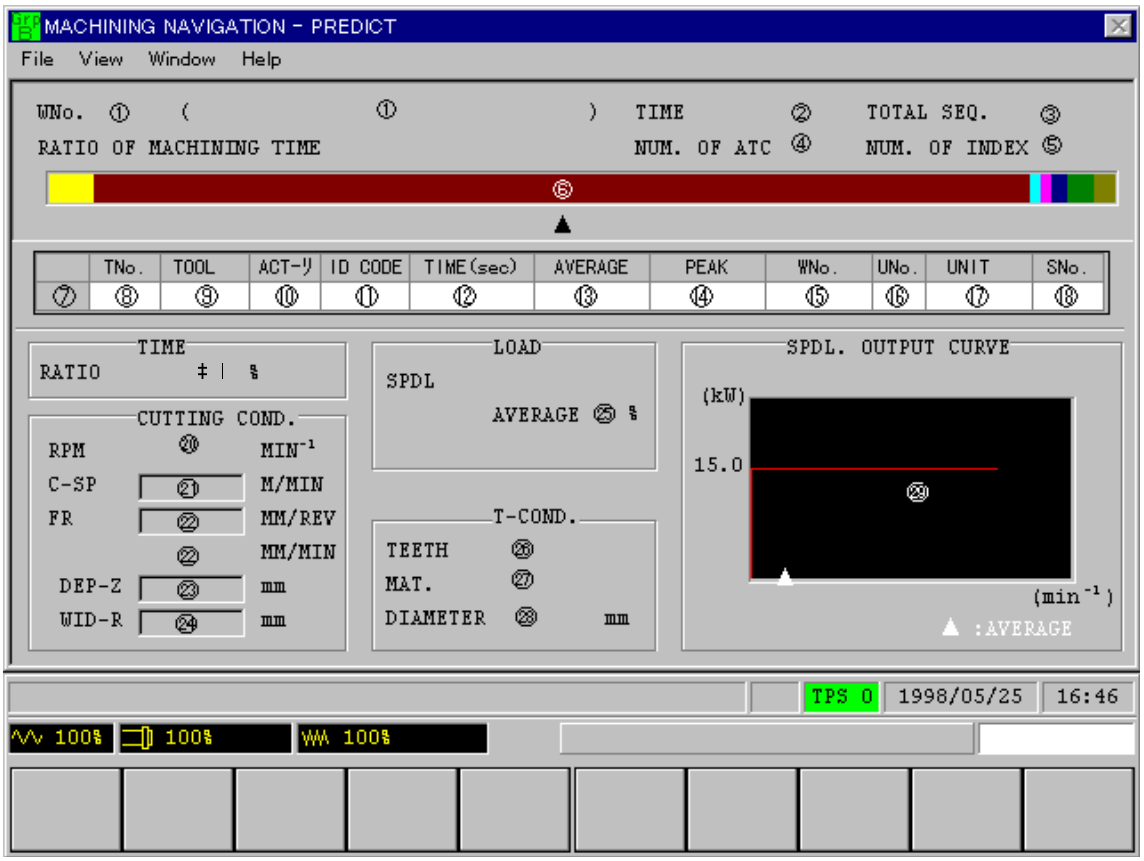
6-9-1 Data display

1. Displayed data with normal display of TOOL DISTINCT menu item



D735S1046E

2. Displayed data with reverse display of TOOL DISTINCT menu item



D735S1047E

3. Display data description

No.	Itme	Unit	Data description
•	WNo.	•	Work number of the machining program currently selected on the PROGRAM display. WNo. **a b (c)** **a** : Work number **b** : Program type (MAZATROL or EIA/ISO) **c** : Program name
•	TIME	H/M/S	Total machining time estimated on the basis of the checking operation on the TOOL PATH display.
•	TOTAL SEQ.	•	Total number of tool sequences in the program
•	NUM. OF ATC	•	Number of ATC operations required for the machining. Not displayed till completion of tool layout.
•	NUM. OF INDEX	•	Total number of index units in the program
•	RATIO OF MACHINING TIME	•	Color-identified graphic display of the ratio of machining time for each tool
•		•	Serial number of the machining process
•	TNo.	•	Tool number
•	TOOL	•	Tool name
•	ACT-φ	•	Nominal diameter of the tool
•	ID CODE	•	Identification code of the tool
•	TIME (sec)	sec	Estimated machining time
•	AVERAGE	kW	Estimated average value of spindle output (see **Note 2.**)

No.	Itme	Unit	Data description
•	PEAK	kW	Not displayed (used only for the MACHINING NAVIGATION - RESULT display)
•	WNo.	•	Work number of the program
•	UNo.	•	Unit number
•	UNIT	•	Unit name
•	SNo.	•	Tool sequence number
•	TIME RATIO	%	Ratio of machining time for each tool sequence
•	RPM	min^{-1}	Spindle speed specified in the tool sequence
㉑	C-SP	m/min (feet/min)	Circumferential speed specified in the tool sequence
㉒	FR	mm/rev (inch/rev)	Feed rate specified in the tool sequence (upper: Feed per revolution)
		mm/min (inch/min)	Feed rate specified in the tool sequence (lower: Feed per minute)
㉓	DEP-Z	mm	Z-axial cutting depth specified in the tool sequence
㉔	WID-R	mm	Radial cutting depth specified in the tool sequence
㉕	SPDL PEAK/AVERAGE	%	Ratio of the peak/average spindle load to the output characteristics, estimated for the tool sequence
㉖	TEETH	•	Number of teeth (cutting edges) of the tool (according to the setting on the TOOL FILE or TOOL DATA display)
㉗	MAT.	•	Material of the tool (according to the setting on the TOOL DATA display)
㉘	DIAMETER	mm (inch)	Diameter of the tool (according to the setting on the TOOL DATA display)
㉙	SPDL.OUTPUT CURVE	•	Spindle output characteristic curve (in red) with respect to the spindle speed, estimated for the tool sequence. The average output is also estimated and indicated by a white triangle.

Note 1: No significant data can be presented on this display till completion of the checking operation for the desired program on the **TOOL PATH CHECK** display.

Note 2: The average value of spindle output is estimated according to the particular workpiece material, and the coefficients used in the calculation of the average output are described and stored for each material in a text file (c:\nm64mdata\eng\cspmat.txt) on the hard disk as shown below.
The indication of the average output will be zero (0) in case the text file contains no descriptions for the workpiece material specified in the program.

c:\nm64mdata\eng\cspmat.txt ← File name
[Material-1]
Name=FC250 ← Workpiece material
Coeff-1=38
Coeff-2=568
Coeff-3=73
Coeff-4=139
Coeff-5=105
Coeff-6=295
Coeff-7=1500
Coeff-8=25

Description of the coefficients used in the calculation of the average output for a workpiece made of FC250. Do not change the values inadvertently.

[Material-2]
Name=FCD450
Coeff-1=31
Coeff-2=568
Coeff-3=73
Coeff-4=142
Coeff-5=105
Coeff-6=294
Coeff-7=1100
Coeff-8=25

[Material-3]
Name=S45C
Coeff-1=23
[Material-4]
[Material-5]

- "eng" in the file name corresponds with the language currently selected for the display mode.
- No registration for a new material can be made arbitrarily since the file only contains coefficients strictly certificated.

6-1-2 Displaying data

The functions of this display cannot be applied for a machining program which has not yet been checked on the **TOOL PATH CHECK** display.

(1) Select the **TOOL PATH CHECK** display.

- The desired program must have been selected beforehand on the **PROGRAM** display.

(2) Carry out the checking operation completely.

- Refer to Subsection 6-2-3 for the checking procedure.

(3) Select the **MACHINING NAVIGATION - PREDICT** display.

➔ The estimates of the machine operation will be displayed in the total display mode described in the following subsection.

- No data will be displayed if the checking operation has not yet been completed at all. In case the checking operation based on a program of the same work number has already been completed, however, data at that time will be displayed.

6-1-3 Display modes

1. Outline

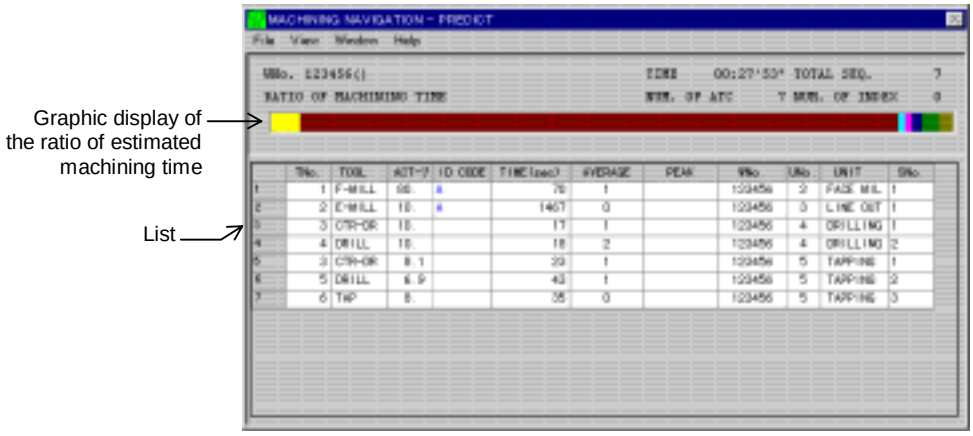
This display has the following two display modes:

- Total display mode
  The display is always made in this mode when the **MACHINING NAVIGATION - PREDICT** display is selected.
  In this display mode, estimated machining time and average spindle output are listed for each tool sequence in addition to the general information about execution of the program.
- Tool-specific display mode
  In this display mode, which can be selected and canceled by pressing the **TOOL DISTINCT** menu key, data of a specific tool sequence listed up in the total display mode can be selectively displayed in further detail. Moreover, cutting conditions specified in the program can be modified through this display.

2. Changing from the total to the tool-specific display mode

(1) Check that a graphic display of the ratio of machining time for each tool sequence is presented under “RATIO OF MACHINING TIME”.

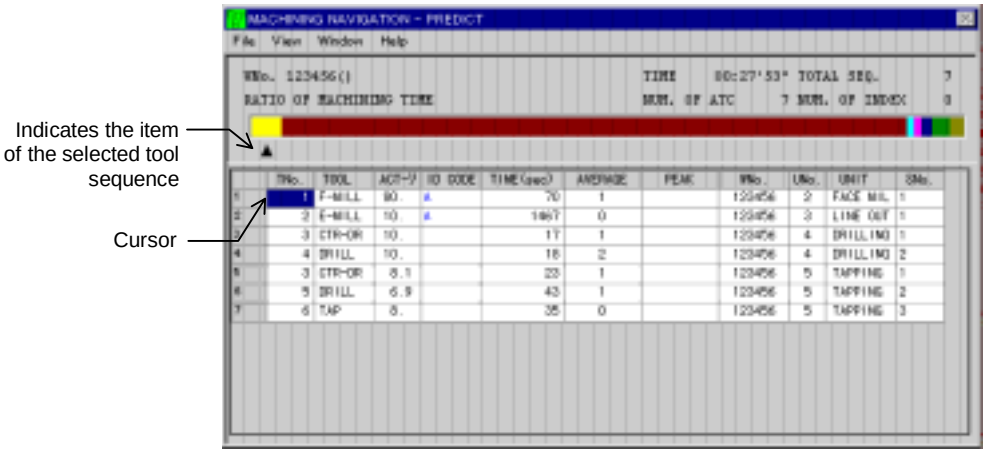
- For a program the tool-path checking of which has been completed, estimated machining time ratio for each tool sequence to the total machining time will be displayed in a color-identified format on a strip graph as shown in the display example below. The items on the strip graph are keyed from left to right, in a one-to-one correspondence, to those line items of the list (estimated machining time and average spindle output for each tool sequence) that are displayed at the bottom of the graph.



(2) Press a cursor key.

➔ The cursor will appear in the “TNo.” column.

- A black triangle (•) will be displayed under the graph “RATIO OF MACHINING TIME” to indicate the item of the tool sequence currently selected by the cursor, as shown in the display example below.



(3) Using the cursor keys, position the cursor on the desired tool sequence number.

➔ The black triangle will move correspondingly with the cursor so that the ratio of the estimated machining time of the specific tool sequence to the total machining time can be visually checked.

(4) Press the **TOOL DISTINCT** menu key.

➔ The display of the menu item will be reversed and the display contents will be changed for a detailed view of the specified tool.

- Press the same menu key once again, and the total display mode will be retrieved and the reverse display of the menu item cleared.

3. NAVIGATE window

In the tool-specific display mode, machining navigation (guidance) information can be displayed in a window to reduce the machining time for the selected tool sequence.
The cutting conditions and know-how for developing the capabilities of your machine to their maximum are supplied in the machining navigation information.

(1) Press the **NAVIGATE** menu key.

➔ The display of the menu item will be reversed and the window for the machining navigation information will be displayed.

Press the same menu key once again, and the window will be closed and the reverse display of the menu item cleared.

WARNING

- Do not follow in actual machining the navigation given here which may not apply to any conditions of tool, jig, workpiece etc. without strictly checking for the operational safety.

4. EXAMPLE window

In the tool-specific display mode, the MAZAK-recommended effective tools are presented in a window for the selected tool sequence.

(1) Press the **EXAMPLE** menu key.

➔ The display of the menu item will be reversed and the window for the recommended effective tools will be displayed.

- Press the same menu key once again, and the window will be closed and the reverse display of the menu item cleared.

WARNING

- Do not use in actual machining the cutting conditions presented here which may not apply to any conditions of tool, jig, workpiece etc. without strictly checking for the operational safety.

6-1-4 Modifying the cutting conditions

In the tool-specific display mode, cutting conditions specified in the **PROGRAM** display can be modified through this display.

Note: Modification and incorporation of the cutting conditions into the program will not immediately update the data in this display. Perform the checking operation anew on the **TOOL PATH CHECK** display for that purpose (modification of the program data on the **PROGRAM** display is of course not required).

1. Modifying the cutting conditions

The cutting conditions specified in the program can be modified in the tool-specific display mode. The modified conditions, however, will not be incorporated into the program until the **WRITE PROGRAM** menu function is executed.

(1) Select the tool-specific display mode.

- Cutting conditions cannot be modified until the tool-specific display mode has been selected for the desired tool sequence.

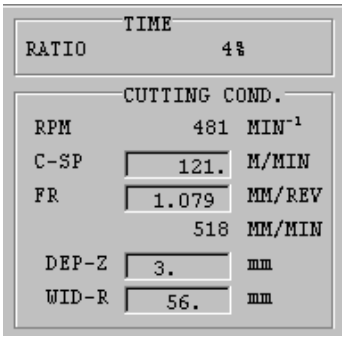
(2) Press the **C-COND CHANGE** menu key.

➔ The display of the menu item will be reversed and the cursor will be placed on the text box “C-SP” in the CUTTING COND. display area, which indicates that the values in this area can now be modified.

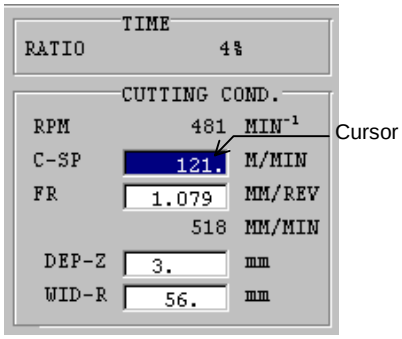
- Press the same menu key once again, and the data modification will be prohibited once again and the reverse display of the menu item cleared.

Example:

(a) With normal display of the menu item



(b) With reverse display of the menu item



Note: It is possible to edit the four cutting conditions displayed in the CUTTING COND. display area (C-SP, FR, DEP-Z and WID-R), of which, however, only those corresponding to the selected tool sequence are subjected to the edition.
Refer to the Programming Manual - MAZATROL - for the description of the data items of the tool sequence.

(3) Using the cursor keys, position the cursor on the desired text box.

- The cursor can only be placed on the available boxes.

(4) Specify new cutting condition and press the INPUT key.

➔ The value selected by the cursor will be overridden by the new data.

2. Cancelling the modification

Before execution of the **WRITE PROGRAM** menu function, the modification data can any time be canceled and the original data retrieved by the following procedure:

(1) Press the **CLEAR DATA** menu key.

➔ The entire text data in the CUTTING COND. display area will be changed to the original data (current settings on the **PROGRAM** display).

- This operation can be performed any time, irrespective of the display status of the **C-COND CHANGE** menu item, unless the **WRITE PROGRAM** menu function has been executed.

3. Incorporating the modification into the program

Carry out the following operation to incorporate the modified cutting conditions into the program:

(1) Press the **WRITE PROGRAM** menu key.

(2) Press the INPUT key.

➔ The modified cutting conditions will be incorporated definitely into the program.

- This operation can be performed any time, irrespective of the display status of the **C-COND CHANGE** menu item.

Note: Once this menu item has been selected, the modification clearing operation described in item 2 above will only retrieve the modified values (the original data will have been cleared from the memory).

- NOTE -

7 DISPLAYS RELATED TO TOOLS

7-1 TOOL DATA Display

Function:

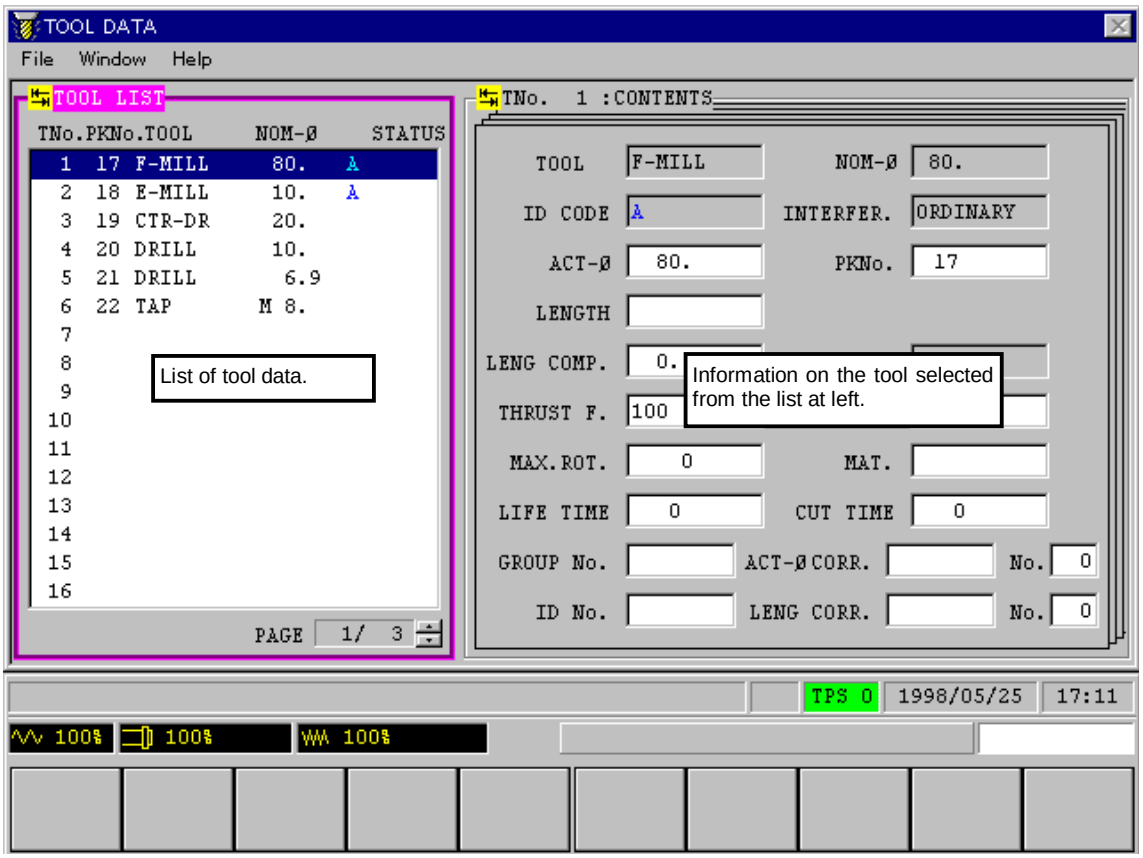
- Registering magazine-mounted tools and setting tool shape data and the data required for management of the tools

Remarks:

- Tool materials data that has been set on this display is used as learning data relating to cutting conditions.
- Tools can also be registered on the **TOOL LAYOUT** display.
- Tool lengths can be manually measured using this display.
- Sections **7-1-2** to **7-1-6** describe the functions to be used for MAZATROL programs, and Sections **7-1-7** to **7-1-11** describe the functions to be used for EIA/ISO programs.

7-1-1 Data display

1. Outline



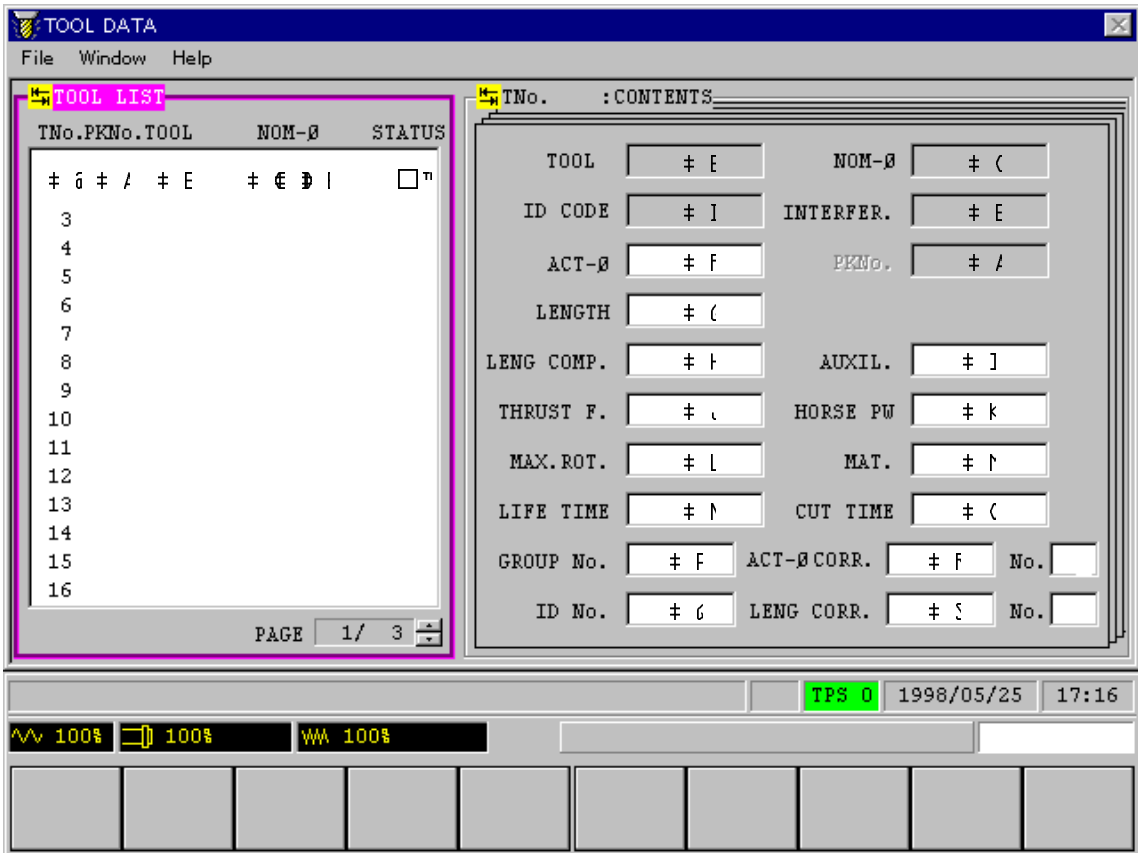
D735S1048E

A list of tool data is displayed at left, and detailed information on the tool selected from the list will be displayed at right. Either of the two areas can be selected by using the Tab key and the current selection is indicated by the reverse display of the title “TOOL LIST” or “TNo. **: CONTENTS” in magenta.

The operation of the cursor keys and page keys differs as follows according to the selected area:

Key	TOOL LIST area	CONTENTS area
Cursor keys	Selection of a tool data line	Selection of a data item
Page keys	Selection of a page of list	Selection of a tool data line on the list page

2. Displayed data



D735S1049E

3. Display data description

No.	Item	Unit	Data description
•	TNo.	•	Pocket number
•	PKNo.	•	Pocket number for random ATC
•	TOOL	•	Tool name
•	NOM-ϕ	mm (inch)	Nominal diameter of the tool
•	ID CODE	•	Suffix (Code that identifies tools of the same nominal diameter)
•	INTERFER.	•	Intra-magazine interference data
•	ACT-ϕ	mm (inch)	Actual diameter of the tool
•	LENGTH	mm (inch)	Tool length
•	LENG COMP.	mm (inch)	Cutting-end position compensation value
•	AUXIL.	•	Number of cutting edges (drill), corner R (end-mill) or tapping method (tap)
•	THRUST F.	%	Thrust force coefficient for Z-axis servomotor
•	HORSE PW	%	Horsepower coefficient for spindle motor
•	MAX.ROT.	min^{-1} (rpm)	Max. spindle speed
•	MAT.	•	Tool material
•	LIFE TIME	Minutes	Life time of the tool
•	CUT TIME	Minutes	Tool operation time

No.	Item	Unit	Data description
•	GROUP No.	•	Group number of the tool (the same group number is assigned to all tools that are the same in both shape and dimensions.)
•	ID No.	•	Tool ID number
•	DIAM.CORR.	mm (inch)	Amount of tool diameter geometric offset
	No.	•	Offset number for tool diameter
•	LENG.CORR.	mm (inch)	Amount of tool length geometric offset
	No.	•	Offset number for tool length
•	STATUS	•	Status of the tool DSBL: disabled (prohibition of use) BRK: broken OVER: tool life over

7-1-2 Tool-data input procedures (for MAZATROL programs)

Usually, when executing a MAZATROL program, the **TOOL LAYOUT** display is to be called up on the screen and then pocket numbers assigned to the tools to be used.
That is, the tools to be used are to be registered in order of pocket number. The **TOOL DATA** display indicates the tools in that order.

Example: If the tools to be used have been registered on the **TOOL LAYOUT** display as shown below:

TOOL LAYOUT display

PKNo.	TOOL	NOM-Ø	
1	F-MILL	80.	A
2	E-MILL	10.	A
3	CTR-DR	20.	
4	DRILL	10.	
5	DRILL	6.9	
6	TAP	M 8.	

The display on the **TOOL DATA** display will then be presented as shown below:

TOOL DATA

File Window Help

TOOL LIST

TNo.	PKNo.	TOOL	NOM-Ø	STATUS
1	1	F-MILL	80.	A
2	2	E-MILL	10.	A
3	3	CTR-DR	20.	
4	4	DRILL	10.	
5	5	DRILL	6.9	
6	6	TAP	M 8.	
7				
8				
9				
10				
11				
12				
13				
14				
15				
16				

PAGE 1/ 3

TNo. 1 :CONTENTS

TOOL	F-MILL	NOM-Ø	80.
ID CODE	A	INTERFER.	ORDINARY
ACT-Ø	80.	PKNo.	1
LENGTH			
LENG COMP.	0.	AUXIL.	
THRUST F.		HORSE PW	100
MAX.ROT.	0	MAT.	
LIFE TIME	0	CUT TIME	0
GROUP No.		ACT-Ø CORR.	No. 0
ID No.		LENG CORR.	No. 0

The shape data necessary to perform actual machining and the data necessary for management purposes must be input for each of the displayed tools. These data are: ACT-ϕ, LENGTH, LENG COMP., AUXIL., THRUST F, HORSE PW, LIFE TIME, CUT TIME , MAT. and MAX. ROT. However, there are data that do not need to be input according to the type of tool. Such data items will be displayed in gray (both title and data box) as shown above, and the cursor will move through such an item to the next data item.

The following describes the data input procedures.

Example: Setting data for the TNo. 2 end-mill shown in the diagram above:
Assume that the **TOOL DATA** display containing the data shown above is already presented on the display.

(1) Press the Tab key as required to select the left column (TOOL LIST) beforehand, and press a cursor key to call up the cursor.

(2) Select TNo. 2.

(3) Press the Tab key to select the right column (CONTENTS).

(1) Press the cursor key to move the cursor to the ACT-ϕ item.

(5) Specify the actual diameter of the tool using the appropriate numeric keys, and then press the INPUT key.

➔ The input value will then be displayed under ACT-ϕ and the cursor will move to the LENGTH item.

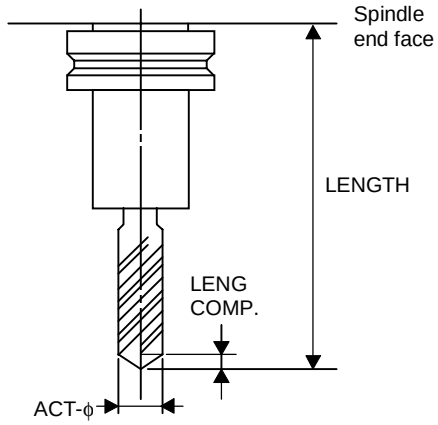
Note: Under the ACT-ϕ item, the same value as that of the NOM-ϕ item will be displayed as shown in the example given. If this value is not to be changed, it is not necessary to execute Steps (4) and (5).

(6) Measure the actual length of the tool.

- For details, see the Subsection **7-1-5 Manual measurement of the tool length** that follows or the Operating manual for machine.

(7) Input the tool length data in the LENGTH item.

(8) Specify the cutting-end position compensation value (LENG COMP.) using the appropriate numeric keys, and press the INPUT key.



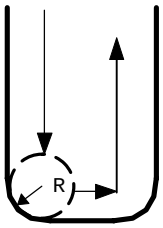
Cutting-end position compensation value for drill M3S020

- Cutting-end position compensation is necessary for drills, back spot-facing tools, back-boring bars, or other tools whose actual cutting-end positions are not at the tips of the tools.

- In the example given above for an end-mill, skip the cursor to the next item by pressing the key because cutting-end position compensation is not necessary.
- The value 0 is preset to each of the LENG COMP., THRUST F., HORSE PW, LIFE TIME, CUT TIME and MAX. ROT. items.
- For drills, setting in “LENGTH” sets the cutting-end position compensation value automatically.
  Automatic setting of the cutting-end position compensation value can be deactivated using the following parameter:
  Parameter **L44**: Automatic setting of the cutting-end position compensation value (0: Valid, 1: Invalid)
- For drills, the appropriate cutting-end position compensation value can be set by setting the nose angle for the tool after pressing the **CUT-ANG INPUT** menu key.

(9) Move the cursor to the AUXIL. item and set the number of cutting edges of the drill, the value of corner R of the end-mill or the method of tapping for the tap.

- Specify the number of cutting edges of the drill using numeric keys.
- Specify the value of corner R of the end-mill using numeric keys.
  Corner R of an end-mill refers to the radius of a corner of an end-mill.
  This date is used in 3-D machining operations.



Comer R of an end -mill M3S021

- For a tapping tool, set the floating tap function or the fixed tap function by pressing the **FLOATING TAP** or **FIXED TAP** menu key.
  Press the **FLOATING TAP** menu key to select “asynchronous” as the type of tapping, or press the **FIXED TAP** menu key to select “synchronous” as the type of tapping.

(10) Move the cursor to the LIFE TIME item, then input tool life time (maximum possible operating time) using the numeric keys, and press the INPUT key.

Note: The NC unit cannot manage the life of that tool if “0” remains set at this item.

(11) Move the cursor to the CUT TIME item, then input tool operating time (cumulative cutting feed time) in minutes, and press the INPUT key.

- The tool cutting time automatically increments with each cutting feed operation of the tool. The display status of the CUT TIME data will reverse when the tool operating time reaches the life time that has been set during step (10) above.

Note: After a new tool has been mounted, input of the tool operating time is not required (“0” is set in the CUT TIME item automatically).

(12) Move the cursor to the MAT. item, and press the menu key corresponding to the type of material to be set.

- If a milling tool that has been registered on the **TOOL FILE** display is registered here, the corresponding type of material set previously on that display will be auto-set. (Only if a suffix is set.)
- The material types presented in the menu here refer to the settings on the **CUTTING CONDITION (W.-MAT./T.-MAT.)** display. See Section 8-1 for further details.
- Data that has been set in the MAT. item is referred to during automatic setting of cutting conditions or used as tutorial data relating to cutting conditions in the MAZATROL program, based on the data that has been registered on the **CUTTING CONDITION** display. See Section 8-2 for further details.

(13) Move the cursor to the THRUST F. item, then input data using the numeric keys, and press the INPUT key.

- Data that the NC unit is to calculate can be set by pressing the **AUTO SET** menu key, instead of setting data using the numeric keys.
- When the **AUTO SET** menu key is pressed, the NC equipment will calculate the maximum permissible load value of the Z-axis servomotor and automatically set the adequate thrust force coefficient (%).
- If you have selected TAP as the tool name and “Synchronous” as the method of tapping, set a returning speed rate (%) for that synchronous tapping tool in this item.
  Returning speed = Returning speed rate/100 × Feed rate
  (All returning speed rates less than 100% are handled as 100%.)

Note 1: If the load on the Z-axis servomotor exceeds the percentage value that has been input to the THR. item, then the cutting feedrate will automatically decrease and the load will be controlled to within its permissible limits.

Note 2: Input of data to the THRUST F. item becomes necessary only for end-mills, drills, chamfering cutters and face-mills.

Note 3: If synchronous tapping cycle machining is required, FIXED TAP must be selected for AUXIL. item of the **TOOL DATA** display.

(14) Move the cursor to the HORSE PW item and specify the value using the appropreate numeric keys, and then press the INPUT key.

- When the **AUTO SET** menu key is pressed instead of specifying the value using numeric keys, the value that is calculated by the NC can be set.
- When the **AUTO SET** menu key is pressed, the NC equipment will calculate the maximum permissible load value of the spindle servo motor and automatically set the adequate horse power coefficient (%).

Note 1: If the load on the spindle servo motor exceeds the percentage value that has been input to the HP item, then the cutting feedrate will automatically decrease and the load will be controlled to within its permissible limits.

Note 2: Input of data to the HP item becomes necessary only for face-mills, end mills, boring bars, drills, backspot-facing tools, backboring bars or chamfering cutters.

(15) Move the cursor to the MAX. ROT. item, then input the maximum rotational speed of the tool using the numeric keys, and press the INPUT key.

Note 1: The maximum rotational speed of the tool will not be limited if “0” remains set in this item.

Note 2: The maximum rotational speed data cannot be set for TOUCH SENSOR and CHIP VACUUM.

Note: Data that has been set during steps (13) and (14) above is valid only for the machine models equipped with AFC. The operations described above are not required for the machine models of standard specifications.

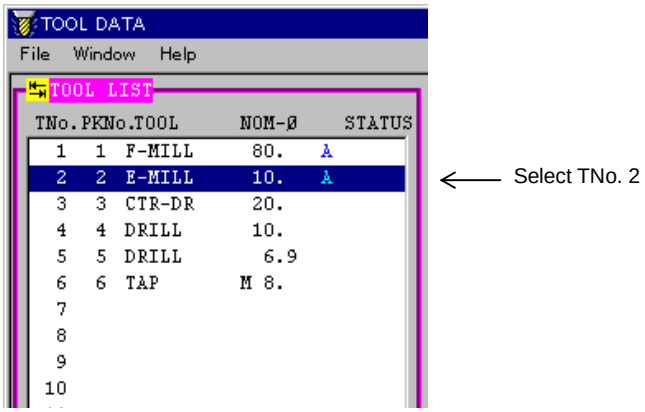
7-1-3 Interchanging pocket numbers (for MAZATROL programs)

The pocket numbers of the tools which have been registered can be interchanged by carrying out the operation described below.

(1) Press the Tab key as required to select the left column (TOOL LIST) beforehand, and press a cursor key to call up the cursor.

(2) Press the cursor key to move the cursor to the line of the tool the pocket number of which is to be interchanged.

Example: When interchanging pocket number 2 of E-MILL and pocket number 4 of DRILL, as shown below:



(3) Press the **EDIT** menu key.

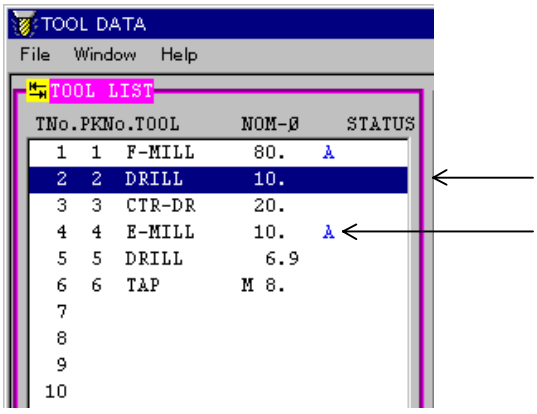
(4) Press the **TOOLDATA MOVE** menu key.

➔ This will cause the display status of **TOOLDATA MOVE** to reverse.

(5) Using the appropriate numeric key(s), specify the interchange destination pocket number. Then, press the INPUT key.

➔ This will cause the pocket number of the end-mill to change to that of the drill, and these two tools will be rearranged in order of pocket number.

- In the above example, specify 4 and press the INPUT key.
  The display on the screen will then change as follows:



7-1-4 Registering the tools to be used

Proceed as follows to register on the **TOOL DATA** display the tools to be used:

(1) Press the Tab key as required to select the left column (TOOL LIST) beforehand, and press a cursor key to call up the cursor.

(2) Press the cursor key to move the cursor to the line of the required TNo.

(3) Press the **EDIT** menu key.

(4) Press the **TOOLDATA ASSIGN** menu key .

➔ This will cause the following menu to be displayed:

ENDMILL	FACEMILL	CHAMFER CUTTER	BALL ENDMILL	OTHER TOOL	TOUCH SENSOR			>>>		(a)

- The menu will change over to the following menu if the menu key >>> is pressed:

CENTER DRILL	DRILL	BACKSPOT FACER	REAMER	TAP	BORING BAR	BACK BOR. BAR	CHIP VACUUM	>>>		(b)

(5) From among the menus (a) and (b) select the name of the tool to be registered and press the corresponding menu key.

➔ The selected tool name will then be displayed under TOOL.

(6) Move the cursor to the NOM-ϕ item of the tool to be registered.

(7) Using the appropriate numeric key(s), specify the nominal diameter of the tool which has been selected in procedural step (5). Then, press the INPUT key

➔ The input value will then be displayed under NOM-ϕ, and the cursor will move to the immediate right of this value.
The following menu will be displayed.

A	B	C	D	E	F	G	H	HEAVY TOOL	>>>	[1]
J	K	L	M	N	P	Q	R	HEAVY TOOL	>>>	[2]
S	T	U	V	W	X	Y	Z	HEAVY TOOL	>>>	[3]

- Pressing the menu key >>> will change the menu in the order of [1] → [2] → [3] → [1].

Note: Menus [1] to [3] are provided for assignment of a suffix (code that identifies tools of the same nominal diameter). For heavy tools, suffix assignment must be made with the menu item **HEAVY TOOL** reversed (by pressing the corresponding menu key).

(8) Select an appropriate suffix from among menus [1] to [3] and press the corresponding menu key.

➔ The selected suffix will then be displayed immediately after the value that has been set in procedural step (7).

➔ The cursor will move to the right of the displayed suffix.
The following menu will be displayed:

ORDINARY DIAMETER	LARGE L	MAG DIR+ +	MAG DIR- -	SMALL S						(c)

- Menu (c) refers to the intra-magazine interference data as follows:

  ORDINARY DIAMETER.....Diameter of a tool which completely fits into the pocket.

  LARGE L.....Tool which interferes with both next tools in the pocket.

  MAG DIR+ +.....Tool that interferes with the tool of a larger pocket number.

  MAG DIR– –.....Tool that interferes with the tool of a smaller pocket number.

  SMALL S.....Diameter of a tool which can be placed in a pocket other than that which interfere with the tool indicated by “+” or “–”.

(9) Press the corresponding menu key contained in menu (c).

➔ The selected type of intra-magazine interference data will be displayed.

Note 1: If shape data or other types of data of the particular tool are to be set following completion of the tool registration operation, then repeat procedural steps (3) through (15) in Subsection 7-1-2.

Note 2: When registering a heavy tool, suffix assignment must be preceded by the selection of the menu item **HEAVY TOOL** in procedural step (8). If a suffix for heavy tools is assigned, this causes the ATC (automatic tool change) speed to decrease, and thus allows safe tool change.

Note 3: Only tools that do not interfere with other tools can be registered.

Note 4: The TNo. display of the tool mounted in the spindle will be highlighted in blue.

7-1-5 Manual measurement of the tool lengths

The actual length of each tool to be used must be measured for setting data under LENGTH on the **TOOL DATA** display. Measurement of the tool lengths can be made in the full-automatic mode, the semi-automatic mode, or the manual mode. Here, the operating procedure for manual tool-length measurement using the LENGTH TEACH function of the **TOOL DATA** display is described. For the full-automatic and semi-automatic tool-length measurement procedures, see the Operating Manual for machine.

(1) Mount on the spindle the tool whose length is to be measured.

- Carry out the tool change operation to mount on the spindle the tool which is accomodated in the magazine pocket. For the operation, see the Operating Manual for machine.

(2) Move the spindle by manual operation and bring the tool tip into contact with the top surface of either the reference block or the workpiece of a pre-determined height.

- For H-type machines, the edge locator of the table functions as reference. For the operation, see the Operating manual for machine.

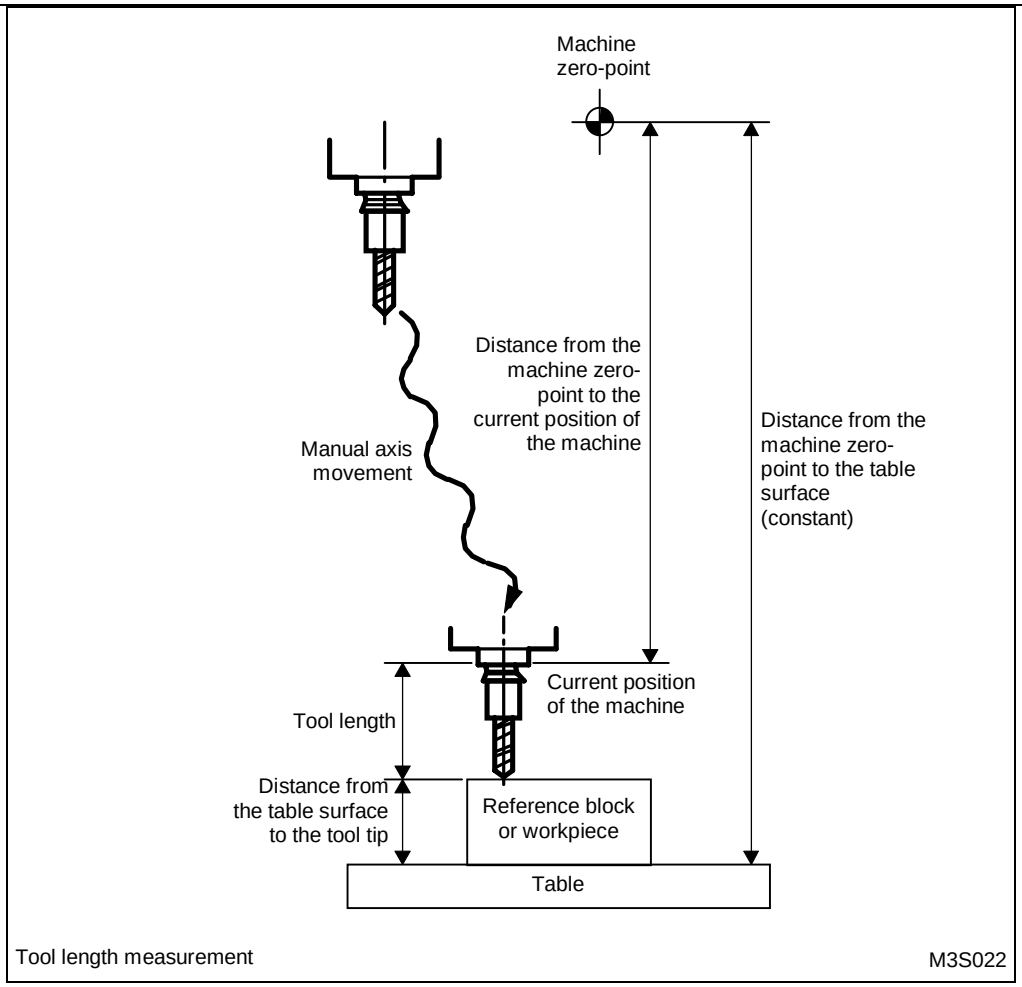
(3) Call up the **TOOL DATA** display on the screen.

(4) Press the **LENGTH TEACH** menu key.

➔ This will cause the display status of **LENGTH TEACH** to reverse. Subsequently, the cursor will appear in the position of the LENGTH item of the tool currently mounted on the spindle.

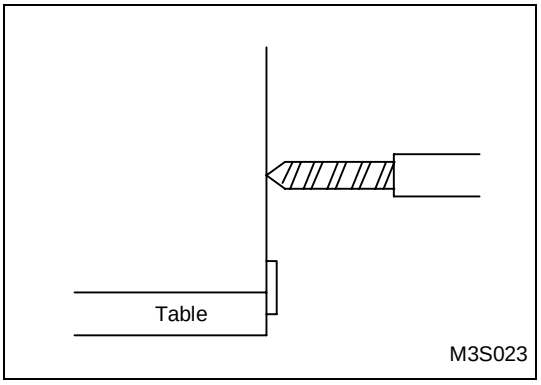
(5) Specify the height of either the reference block or the workpiece using the appropriate numeric key(s), and then press the INPUT key.

➔ The NC equipment will then calculate the length of the tool mounted on the spindle, and the calculated value will be automatically set under LENGTH. (See the following figure.)



Tool length measurement

Note: In the case of the figure shown below, specify “0” and press the INPUT key.



- Since the current position of the machine is stored in the memory of the NC equipment, it will calculate the length of the tool if the height of the reference block or workpiece (i.e., the distance from the table surface to the tool tip) is specified.
- The reverse-display status of **LENGTH TEACH** will be released automatically.

7-1-6 Other functions (for MAZATROL programs)

1. TOOL SEARCH

This menu function should be used to search for a tool name that is registered on the **TOOL DATA** display.

(1) Press the **TOOL SEARCH** menu key.

➔ The following menu will be displayed

ENDMILL	FACEMILL	CHAMFER CUTTER	BALL ENDMILL	OTHER TOOL	TOUCH SENSOR			>>>	(a)

- Pressing the menu key >>> changes the above menu over to the following menu:

CENTER DRILL	DRILL	BACKSPOT FACER	REAMER	TAP	BORING BAR	BACK BOR. BAR	CHIP VACUUM	>>>	(b)

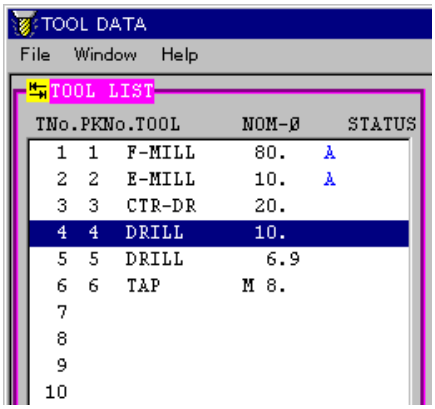
(2) From among menus (a) and (b), select the menu item that corresponds to the tool name to be searched for, and press the appropriate menu key.

➔ This will cause the display status of the selected menu item to reverse.

(3) Using the appropriate numeric key(s), specify the nominal diameter value of the tool name to be searched for. Then, press the INPUT key.

➔ If the tool name to be searched for is already registered, the line of the tool concerned will be displayed and highlighted.

Example: When checking the pocket number under which a drill with a nominal diameter of 10 mm is registered:
After pressing the **DRILL** menu key in procedural step (2), specify 10 and press the INPUT key in procedural step (3).
This will cause the line concerned to be highlighted as shown below.



This shows that the tool to be searched for is registered under pocket number 4.

- If the INPUT key is pressed following this, the NC unit will check whether another corresponding tool exists in the subsequent area, and the line of the tool concerned will be highlighted next. If no corresponding tools can be found any more, then the alarm message **407 DESIGNATED DATA NOT FOUND** will be displayed.

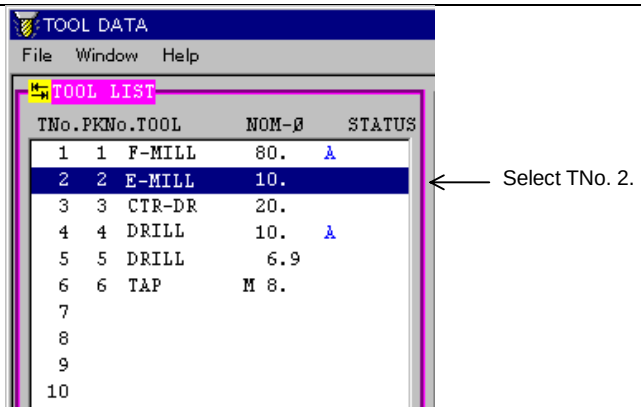
Note: It is also possible to search for the corresponding tool name just by specifying TOOL item. In this case, press only the INPUT key in procedural step (3) without specifying the nominal diameter.

2. TOOLDATA ERASE

Use this menu function to erase tool data registered on the **TOOL DATA** display.

(1) Press the cursor key to select the line of the tool data to be erased.

Example: On the display shown below, if the E-MILL data of tool number 2 is to be erased:



(2) Press the **EDIT** menu key.

(3) Press the **TOOLDATA ERASE** menu key.

➔ This will cause the display status of **TOOLDATA ERASE** to reverse.

(4) Press the INPUT key.

➔ Selected tool data will be erased.

3. TOOLNAME ORDER (displayed only for machines with a random ATC feature)

For machines having a random ATC feature, use this menu function if the tool data previously registered on the **TOOL DATA** display is to be rearranged in order of tool name.

(1) Press the **EDIT** menu key.

(2) Press the **TOOLNAME ORDER** menu key.

➔ This will cause the display of **TOOLNAME ORDER** to reverse.

(3) Press the INPUT key.

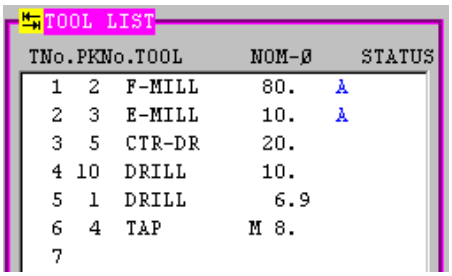
➔ The tool data will be rearranged in order of tool name.
If there are more than one set of tool data having the same name, those sets of tool data will be rearranged in order of the smaller nominal diameter first.

The order of tool name refers to the following order:

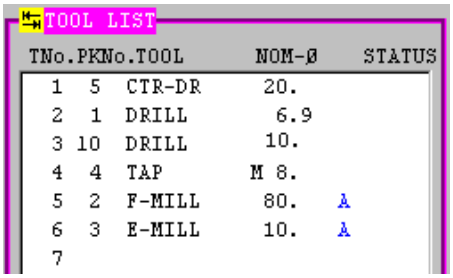
•CTR-DR •DRILL •REAM •TAP (M, U, PT, PF, PS, OTHER, in that order)
•BK FACE •BOR BAR •B-B BAR •CHF-C •F-MILL •E-MILL •OTHER
•CHP VAC •T.SENS •B-E-MIL

Example: To rearrange the tool data on the following display:

(A)



(B)



Note: "In order of tool name" operations cannot be carried out while the spindle has a mounted tool.

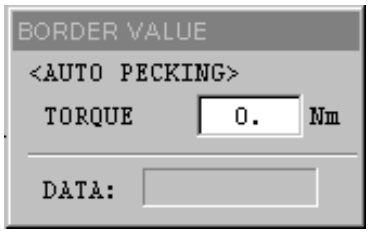
4. BORDER VALUE

Proceed as follows to set the pecking threshold level for a drill:

(1) Using the cursor keys, move the cursor in the TOOL LIST area onto the line of a drill for which the threshold is to be set.

(2) Press the **BORDER VALUE** menu key.

➔ The display of the menu item will be reversed and the following dialogue box will appear:



(3) Using the cursor keys, position the cursor on the text box “TORQUE”.

(4) Enter the required value using the numeric keys and press the INPUT key.

➔ The threshold will be set definitively and the dialogue box will disappear.

5. Setting PKNo. data (Valid only for machine with a random ATC feature)

If your machine has a random ATC feature, use the following procedure to set PKNo. item on the **TOOL DATA** display (PKNo. data can be set even during the write-protected status).

(1) Press the Tab key ⇤⇥ as required to select the left column (TOOL LIST) beforehand, and select the line of the tool required by using the cursor keys.

(2) Press the Tab key ⇤⇥ to select the right column (CONTENTS).

(3) Select the data box “PKNo.” by using the cursor keys, key in the required pocket number and press the INPUT key.

7-1-7 Setting tool data (for EIA/ISO programs)

Setting data in ACT-ϕ CORR. or No. items and LENG CORR. or No. items of this display allows tool designation and tool life management using an EIA/ISO program.

(1) Press the Tab key ⇤⇥ as required to select the left column (TOOL LIST) beforehand, and select the line of the required tool by using the cursor keys.

(2) Press the Tab key ⇤⇥ to select the right colum (CONTENTS).

(3) Select the data box “GROUP No.” by using the cursor keys.

(4) Specify a group number using the numeric keys, and press the INPUT key.

- Any group number from 0 to 99999999 can be set for the tool. Set one and the same group number for multiple tools of the same type (in terms of both shape and dimensions) in order that they shall be used as a spare tool for each other. For details of the spare tool see Subsection 7-1-8.

Note: Group number 0 is not subjected to spare-tool searching.

- When the group number is set, the tool ID number will also be displayed at the same time. To check details of the tool ID number, see **Note 3** that follows.

(1) Select the data box “ACT-ϕ CORR.” by using the cursor keys.

(6) Input tool diameter offset data using the numeric keys, and press the INPUT key.

- When the tool diameter offset data is set, the No. item on the right will become blank to indicate that the item has become invalid.

(7) Select the data box “LENG KORR.” by using the cursor keys.

(8) Input tool length offset data using the numeric keys, and press the INPUT key.

- When the tool length offset data is set, the No. item on the right will become blank to indicate that the item has become invalid.
- For the current tool on the spindle only, tool length offset data can be set using the **LENG-OFS TEACH** menu function. See Subsection 7-1-9 for further details.

(1) Select the data box “No.” on the right of ACT-ϕ CORR.

(10) Input the offset number under which the tool diameter offset data to be referred to has been set on the **TOOL OFFSET** display, and press the INPUT key.

- When the tool diameter offset number is set, the ACT-ϕ CORR. item will become blank to indicate that the item has become invalid.
- See Section 7-3 for further details of the **TOOL OFFSET** display.

(11) Select the data box “No.” on the right of LENG CORR.

(12) Input the offset number under which the tool length offset data to be referred to has been set on the **TOOL OFFSET** display, and press the INPUT key.

- When the tool length offset number is set, the LENG CORR. item will become blank to indicate that the item has become invalid.
- See Section 7-3 for further details of the **TOOL OFFSET** display.

Note 1: Tool length offset data and tool diameter offset data are to be set by either setting the data directly on this display or setting the offset numbers corresponding to the offset data that has been previously set on the **TOOL OFFSET** display.

Note 2: When the tool for which the offset data has been set on this display is to be used during execution of the program, the offset data (or offset number data) that has been set on this display will govern even if another offset data number has been designated in the program.

Example:

Program

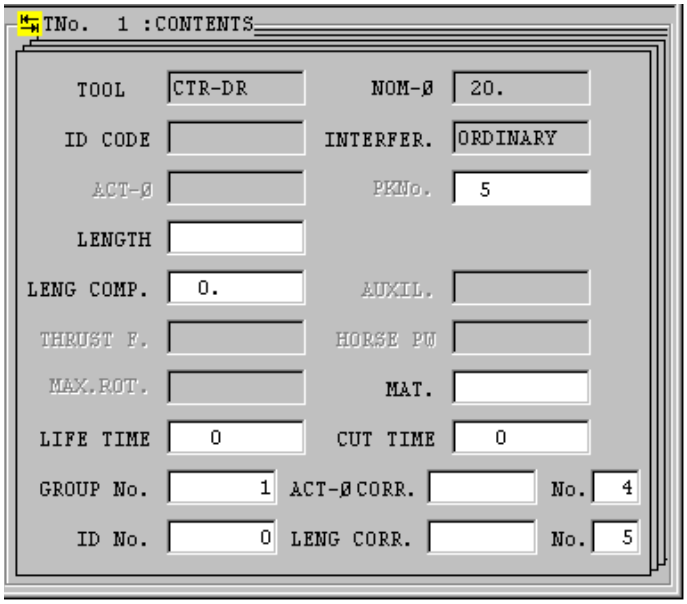
M
H1 ← Tool length offset number
D2 ← Tool diameter offset number
T01T0M06 ← The offset data used for this block will be the data enclosed in ☐ according to the settings as shown below on the **TOOL DATA** display.
M

The offset data enclosed in ⬚ becomes valid according to these blocks.

TOOL OFFSET display

No.	OFFSET
1	10.1
2	10.2
3	10.3
4	10.4
5	10.5

TOOL DATA display



The program is executed according to the tool diameter offset data (No. 4 = 10.4) and tool length offset data (No. 5 = 10.5) that have been set on the **TOOL DATA** display, instead of the tool length offset data (H1 = 10.1) and tool diameter offset data (D2 = 10.2) that have been designated in the program.

Note 3: For a machine that has a tool ID number option, tools can be managed using ID No. data. The data is to be set within either one of the following two ranges, depending on the particular type of tool ID:
- Eight decimal digits (0 to 99999999)
- Eight hexadecimal digits (0 to FFFFFFFF)

For a machine that does not have a tool ID number option, ID No. data does not have a meaning. The data can be freely used as tool identification numbers or for any other purposes. In that case, the data can be set within the range up to eight decimal digits.

Note 4: If data is not displayed under “GROUP No.” or “TOOL” then data can be set under “GROUP No.”, and when any data has been set under “GROUP No.” under that state, then data can be further set under “ID No.”, “ACT-ϕ CORR.” (or “No.”), “LENG CORR.” (or “No.”), “LIFE TIME”, “CUT TIME”, “ACT-ϕ”, “LENGTH”, “LENG COMP.”, “THRUST F.”, “HORSE PW” and “MAX. ROT.”

7-1-8 Setting tool group numbers

If a group number is set for the tools that are to be used during execution of the program, the next spare tool to be used will be auto-selected from tools of the same group number.

1. Selection of spare tools

As listed below, two methods are available to select spare tools. Either is to be selected by setting 0 or 1 in bit 2 of parameter **F84**.

- **F84** bit 2 = 0: Group number scheme
- **F84** bit 2 = 1: Tool name scheme

A. Group number scheme

Tools of the same group number are all regarded as spare ones, and tools that have neither yet reached their lives nor become damaged are automatically selected in order with the smallest tool number first.

B. Tool name scheme

Tools of the same tool name, the same nominal diameter, and the same suffix are all regarded as spare ones, and tools that have neither yet reached their lives nor become damaged are automatically selected in order with the smallest tool number first.

Note: If all spare tools have reached their lives or become damaged, life-expired one having the largest tool number will be selected first.

2. Designation of tool numbers in the EIA/ISO program

As listed below, two methods are available to designate tool numbers in the EIA/ISO program. Either is to be selected by setting 0 or 1 in bit 4 of parameter **F94**.

- **F94** bit 4 = 0: Group number designation scheme
- **F94** bit 4 = 1: Tool number designation scheme

A. Group number designation scheme

The T-command in the program is taken as the command for group number (GROUP No.). Tools with the same GROUP No. as that of the commanded GROUP No. are selected as spare tools.

In this group number designation sheme, T01 in the program is taken as a command for group No. 1. In case the group number “1” has been assigned for TNo. 2 and TNo. 3 on the **TOOL DATA** display, the tool of TNo. 3 is selected if the life of the tool of TNo. 2 has already expired.

B. Tool number designation scheme

The T-command in the program is taken as the command for tool number (TNo.). Tools with the same GROUP No. as that of the commanded TNo. are selected as spare tools.

In this tool number designation sheme, T01 in the program is taken as a command for TNo. 1. In case the group number “2” has been assigned for TNo. 1 and TNo. 4 on the **TOOL DATA** display, the tool of TNo. 4 is selected if the life of the tool of TNo. 1 has already expired.

Note: For both schemes, if there are multiple tools of the same group number, the appropriate tool will be selected according to the particular method of selecting spare tools.

7-1-9 Manual measurement of tool lengths (Setting tool length offset data using the LENG-OFS TEACH menu function)

The **LENG-OFS TEACH** menu function can be used to set tool length data in the LENG CORR.

(1) Manually move the axes, and bring the nose of the tool into contact with the top of either a reference block or a workpiece whose height is predetermined.

- Refer to the Operating Manual of the machine for further details of manual operations.

(2) Press the **LENG-OFS TEACH** menu key.

➔ The display status of the **LENG-OFS TEACH** menu item will then reverse, and the LENG CORR. item of the tool currently mounted on the spindle will be selected.

(3) Input the height of either the reference block or the workpiece, and press the INPUT key.

➔ The NC unit will then calculate the tool length (see the figure given in Subsection 7-1-5), and auto-set the calculated data under the LENG CORR. item.

- The **LENG-OFS TEACH** menu item will return to its normal display status.

7-1-10 Other functions (for EIA/ISO programs)

1. ADD. DATA CANCEL menu function

Use this menu function to erase only GROUP No. and ID No. data.

(1) Press the Tab key as required to select the left column (TOOL LIST) beforehand, and select the line of the required tool by using the cursor keys.

(2) Press the **EDIT** menu key.

(3) Press the **ADD. DATA CANCEL** menu key.

(4) Press the INPUT key.

➔ The added data (GROUP/ID No.) of the tool selected in Step (1) will then be erased.

7-1-11 MAZATROL tool data “INVALIDATION” (Option)

1. Overview

“INVALIDATION” is an option that makes registered MAZATROL tool data on the **TOOL DATA** display invalid during program execution. The invalidated tool data is indicated by **DSBL** under “STATUS” for distinction from valid tool data.
Using this option, you can select tools from all those of the same type or set the tools not to be used without deleting the tool data before executing the program.

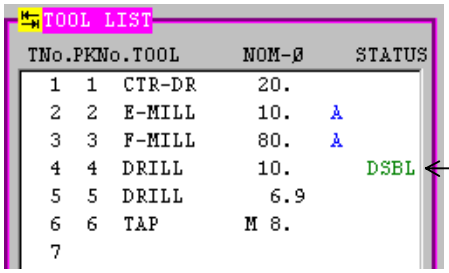
2. Making the tool data valid/invalid

(1) Press the Tab key as required to select the left column (TOOL LIST) beforehand, and select the line of the required tool by using the cursor keys.

(2) Press the **EDIT** menu key.

(3) To make the data selected in (1) valid/invalid, press the menu key **TOOLDATA VALID** or **TOOL DATA INVALID** respectively.

(4) Press the INPUT key.



"DSBL" for indication of an invalid tool

3. Machine action

Only in the following cases, tools designated as invalid (DSBL) on the **TOOL DATA** display are regarded as invalid tools.

A. during automatic operation

B. during tool path check

C. during "tool layout"

A. Automatic operation

The machine action differs as follows between MAZATROL program automatic operation and EIA/ISO program automatic operation:

1. MAZATROL program operation
   Invalid tools are excluded from a list of usable tools. An alarm **626 NO TOOL IN MAGAZINE** will be displayed if an alternative, usable tool of the same type is not present.
2. EIA/ISO program operation
   The MAZATROL tool data invalidation option is inoperative for automatic operation using EIA/ISO programs, since the EIA/ISO program normally operates independently of MAZATROL tool data. Only if MAZATROL tool length and diameter offset data is valid for EIA/ISO program, however, does this option become operative. In this case, if the MAZATROL tool data of the selected tool number is invalid, operation will stop with the display of an alarm **653 ILLEGAL TOOL DESIGNATED** after tool change has been done.

   (a) If MAZATROL tool length and diameter offset data is valid (that is, if tool diameter offset is made valid using bit 7 of user parameter **F92** and tool length data is made valid using bit 3 of user parameter **F93**):
   Operation will stop with the display of an alarm **653 ILLEGAL TOOL DESIGNATED**.

   (b) If MAZATROL tool length and diameter offset data is invalid:
   Normal operation continues.

Note: Operation will also stop with an alarm display, if a block that contains a command for tool change with an invalid tool is found during EIA/ISO search associated with restart.

B. Tool path check is performed

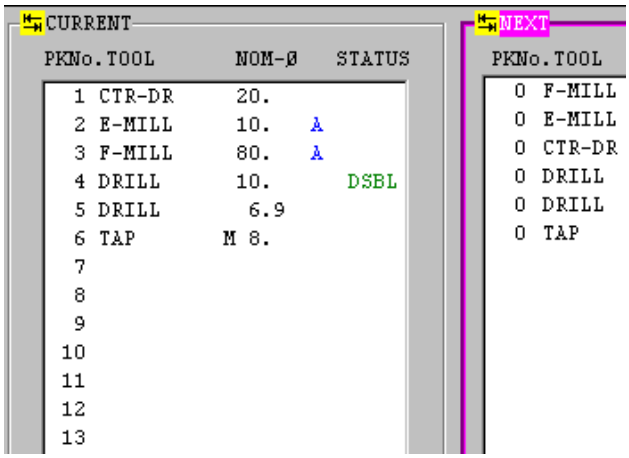
The machine action differs as follows between tool path check using a MAZATROL program and tool path check using an EIA/ISO program:

1. During tool path check using a MAZATROL program
   Invalid tools are not selected. If an alternative, usable tool of the same type is not present, a tool path will be drawn in the absence of a tool (diameter offsetting will use the tool nominal diameter data existing in the program).
2. During tool path check using an EIA/ISO program
   The machine action is similar to that existing during automatic operation.

C. TOOL LAYOUT

All invalid tools will be indicated as “DSBL” on the **TOOL LAYOUT** display.

TOOL LAYOUT display



Note: With the menu function **PKNo. SHIFT**, the tool numbers of invalid tools are not assigned to tools displayed under NEXT.

4. In combination with the optional “GROUP No. assignment”

When the MAZATROL tool data option “GROUP No. assignment” is used, tool commands in an EIA/ISO program are executed according to the group number data of the MAZATROL tool data display.
The machine action during automatic operation (or tool path check) based on an EIA/ISO program, therefore, differs from that occurring with the tool data option “GROUP No. assignment” not being used.
The machine acts as follows according to the particular method of selecting tools:

1. Group number scheme (User parameter **F94**, bit 4 = 0)
   - The machine will stop with the display of an alarm **653 ILLEGAL TOOL DESIGNATED** if the tools of the selected group number are all invalid ones.
   - If the tools of the selected group number include valid ones (life-expired or broken tools included), then these valid tools will be searched for a spare tool.

2. Tool number scheme (User parameter **F92**, bit 4 = 1)
   When the tool of the selected tool number is invalid:

   (a) If a group number has been set, a spare tool search will be made using that group number. The machine action after the search has been made is the same as that described for the group number scheme above.

   (b) If a group number has not been set, an alarm **653 ILLEGAL TOOL DESIGNATED** will result.

Example: Differences in machine action between the two tool selection schemes:

TNo.	GROUP No.	STATUS
1	12345678	DSBL
2		
3	87654321	DSBL
4	87654321	

For the settings on the **TOOL DATA** display as shown above:

Group number scheme	Tool number scheme
Example 1: If `T12345678T0M06` is set: An alarm **653 ILLEGAL TOOL DESIGNATED** results.	**Example 1:** If `T01T0M06` is set: An alarm **653 ILLEGAL TOOL DESIGNATED** results.
Example 2: If `T87654321T0M06` is set: Tool of TNo. 4 is selected.	**Example 2:** If `T03T0M06` is set: Tool of TNo. 4 is selected.

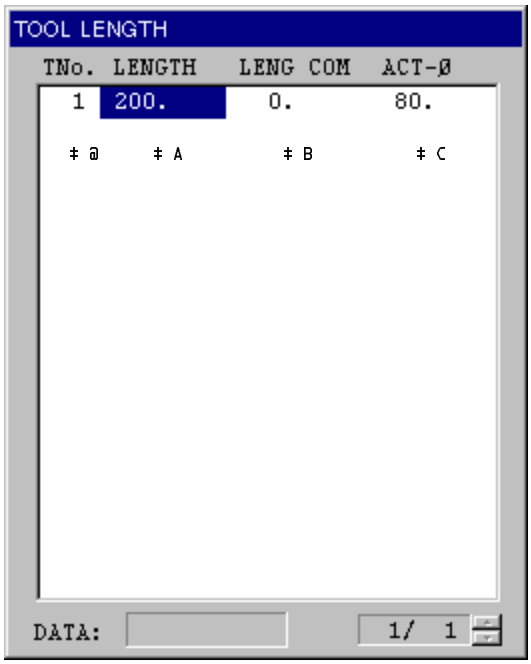
Note:
The machine action during automatic operation and that of tool path check are the same.

7-1-12 Tool length listing window

1. Outline

The tool length listing window is provided on the **TOOL DATA** display in order for the operator to easily enter data of items LENGTH, LENG COMP. and ACT-ϕ for multiple tools without having to select each time the particular tool for the CONTENTS area. The LENGTH TEACH menu function and incremental data input are also available for data entry through the listing window.
Press the **TOOL LENGTH** menu key to call up the listing window, and press the same menu key anew to close it.

2. Description of the display data in the TOOL LENGTH listing window



No.	Item	Unit	Data description
•	TNo.	•	Pocket number
•	LENGTH	mm (inch)	Tool length
•	LENG COM	mm (inch)	Cutting-end position compensation value
•	ACT-ϕ	mm (inch)	Actual diameter of the tool

7-2 TOOL FILE Display

Function:

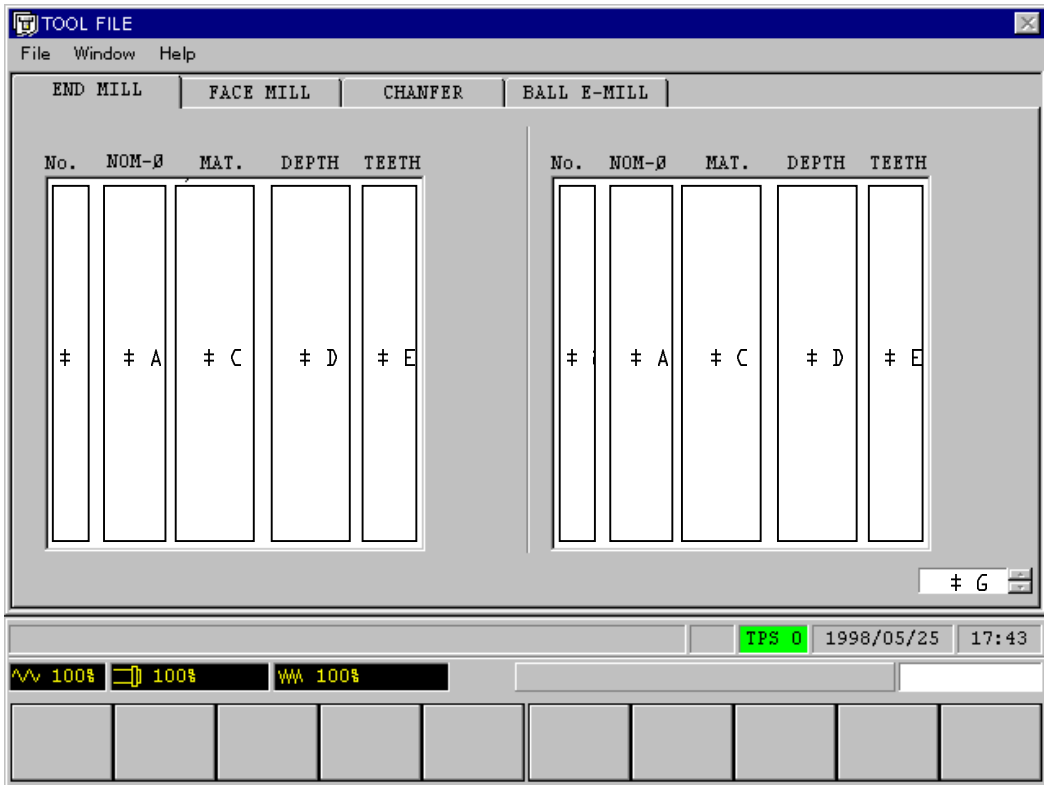
- This display is used when registering the data of the milling tools to be used, such as end-mills, face-mills, chamfering cutters, ball end-mills, etc.

Remarks:

- The **TOOL FILE** display is divided into the following four displays according to the type of milling tool to be used:
  1. TOOL FILE (END MILL) display
  2. TOOL FILE (FACE MILL) display
  3. TOOL FILE (CHAMFER) display
  4. TOOL FILE (BALL E-MILL) display
- Up to a maximum of 256 milling tools can be registered on the **TOOL FILE** display.
- Milling tools that are not registered on this display cannot be used when creating a program or when executing a program in the automatic operation mode. Milling tools on hand, therefore, should be registered on this display.
- If the **TOOL FILE** display has been changed over from another display, the **TOOL FILE (END MILL)** display will be selected automatically.
  If another **TOOL FILE** display should be called up:
  1. Pressing the **FACEMILL** menu key changes the current display over to the **TOOL FILE (FACE MILL)** display.
  2. Pressing the **CHAMFER CUTTER** menu key changes the current display over to the **TOOL FILE (CHAMFER)** display.
  3. Pressing the **BALL ENDMILL** menu key changes the current display over to the **TOOL FILE (BALL E-MILL)** display.
  4. Pressing the **ENDMILL** menu key changes the current display back to the **TOOL FILE (END MILL)** display.

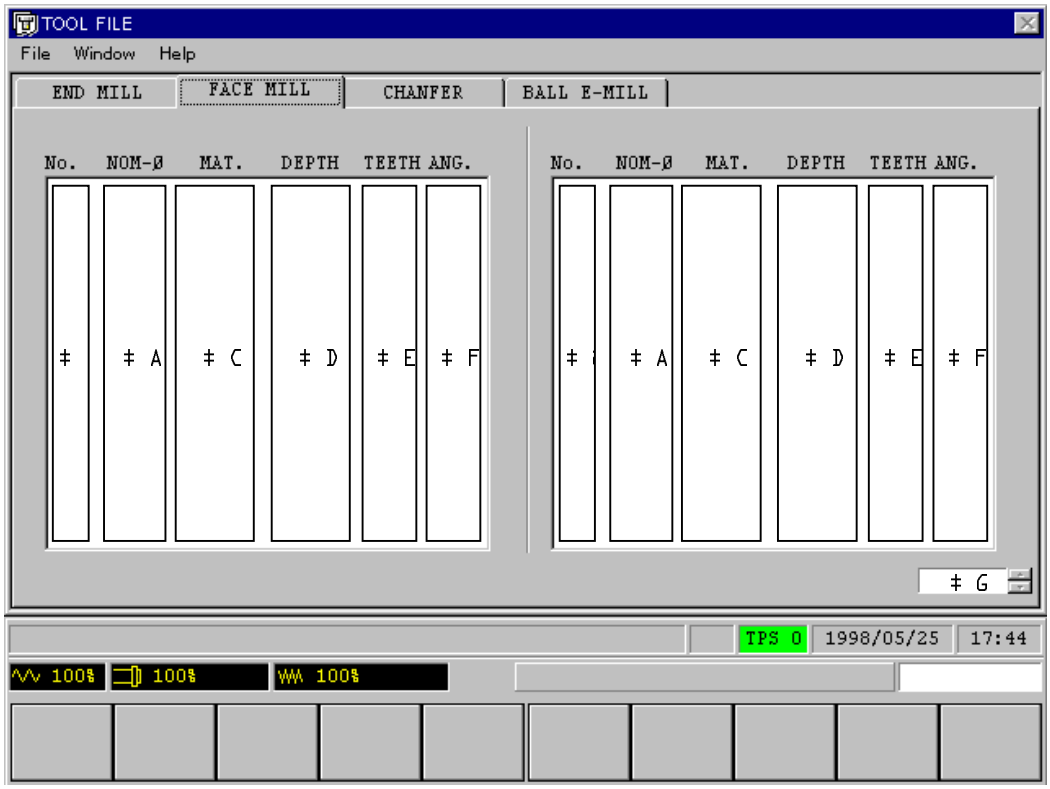
7-1-1 Data display

1. Displayed data of the displays for end-mills and ball end-mills



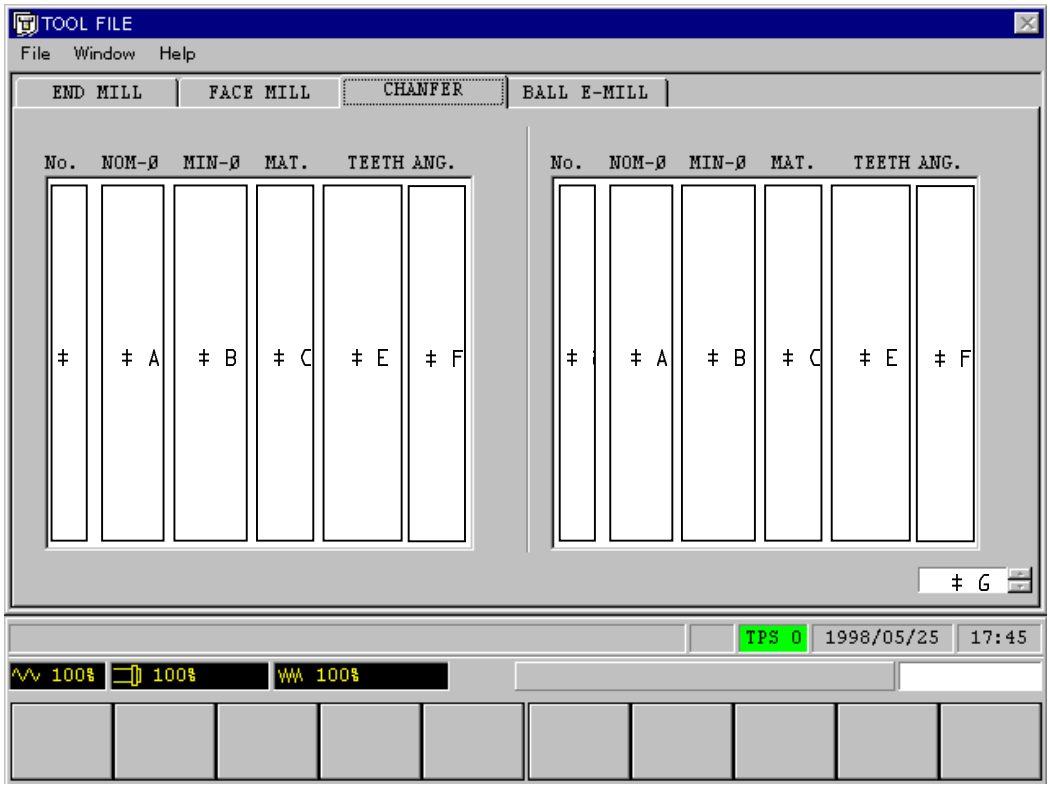
D735S1050E

2. Displayed data of the displays for face-mills



D735S1051E

3. Display data of the display for chamfering cutters



D735S1052E

4. Display data description

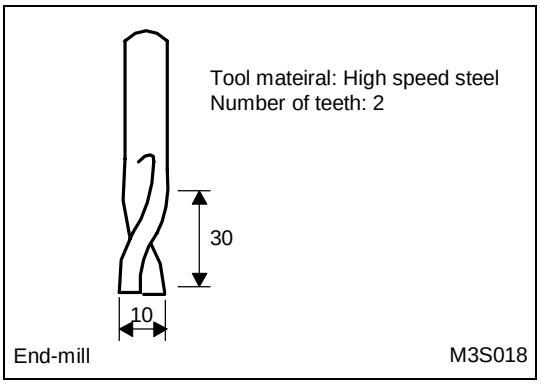
No.	Item	Unit	Data description
•	No.	•	Register number of the milling tool on each display
•	NOM-ϕ	mm (inch)	Nominal diameter of the milling tool
		•	Suffix (code that identifies the milling tools having the same nominal diamater)
•	MIN-ϕ	mm (inch)	Minimal diameter of the chamfering cutter
•	MAT.	•	Material of the milling tool
•	DEPTH	mm (inch)	Maximum depth of cut
•	TEETH	•	Number of teeth of the milling tool
•	ANG.	degrees	Nose angle of the face-mill or chamfering cutter
•	Page	•	Page number and total number of pages **a / b** **a**: Number of the displayed page **b**: Total number of pages

7-1-2 Registering milling tool data

When registering milling tools on the **TOOL FILE** display, the types of data to be set differ according to the types of tools to be used. The setting procedure, however, is the same for all types of data except chamfering cutters.

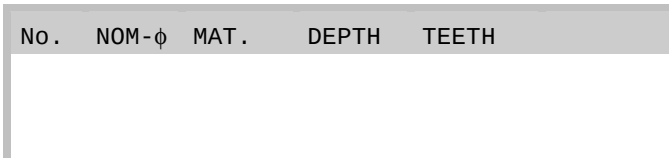
1. Setting a tool other than chamfering cutter

We will register the end-mill shown below as an example.



(1) Call up the **TOOL FILE (END MILL)** display.

➔ The following display will then be presented:

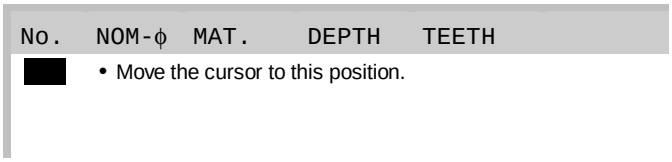


- In this example, assume that no end-mills have yet been registered.

(2) Press the cursor key to call up the cursor on the screen.

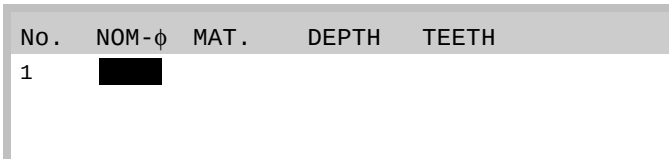
(3) Press the cursor key to move the cursor to the position of the No. item to be registered.

- The cursor can be moved only to the lines of numbers which have been registered and the next line.



(4) Press the INPUT key.

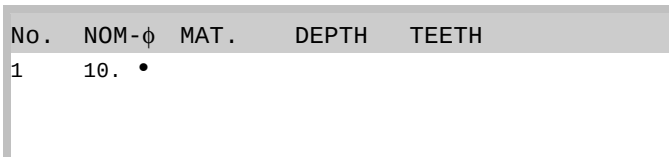
➔ The corresponding number will then be displayed under No., and the cursor will move to the NOM-ϕ item.



(5) Using the numeric keys, specify the nominal diameter of the tool to be registered. Then, press the INPUT key.

- In the example, specify “10” and press the INPUT key.

➔ The input value will then be displayed under NOM-ϕ, and the cursor will move to the right side of that value.



- The following menu for assigning tool ID codes will be presented in the menu display area:

A	B	C	D	E	F	G	H	HEAVY TOOL	>>>	(a)
J	K	L	M	N	P	Q	R	HEAVY TOOL	>>>	(b)
S	T	U	V	W	X	Y	Z	HEAVY TOOL	>>>	(c)

Pressing the menu key >>> will change the menu in order of (a) → (b) → (c) → (a).
With the menu item **HEAVY TOOL** displayed in reverse or normally (changed by pressing the corresponding menu key), a suffix for heavy or normal tools can be respectively selected from among **A** to **Z**.

- The tool ID code refers to the suffix, appended to the nominal diameter value, to identify tools of the same nominal diameter.

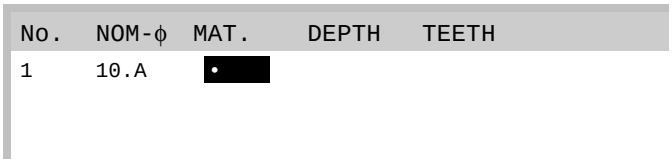
Note: If a suffix for heavy tools is assigned, the ATC speed will automatically decrease and then tool change will be safely made.

(6) Press the menu key that is appropriate to the suffix to be registered.

- In the example, we will assign the suffix A since the tool is the first one with nominal diameter of 10 mm. Press the menu key **A**.
- If it is not necessary to assign a suffix, press the cursor key ➔.

(7) Move the cursor to the MAT. item.

- The list of the materials set on the **CUTTING CONDITION (W.-MAT/T.-MAT.)** display is displayed in the menu display area. Refer to Section 8-1 for further details.

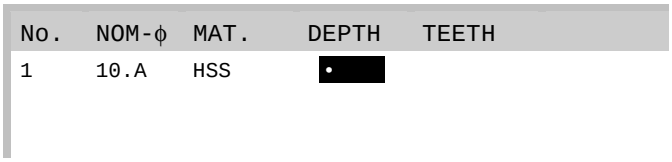


(8) Press the menu key that is appropriate to the tool material to be registered.

- In the example, press the menu key **HSS**.

➔ The selected material will be displayed under MAT.

(9) Move the cursor to the DEPTH item.



(10) Using the numeric keys, specify the maximum depth of cut that can be machined during one cutting operation (i.e., the length of the cutting edge of the end-mill). Then, press the INPUT key.

- In the example given, specify “30” and press the INPUT key.

(11) Move the cursor to the TEETH item.

```
No.   NOM-ϕ  MAT.    DEPTH   TEETH
1     10.A   HSS     30.     •
```

(12) Using the numeric keys, specify the number of teeth of the tool to be registered. Then, press the INPUT key.

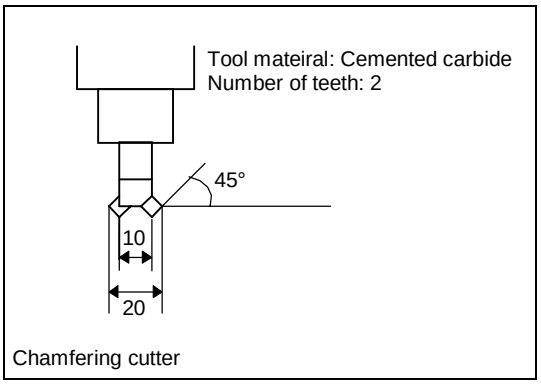
- In the example given, specify “2” and press the INPUT key.

```
No.   NOM-ϕ  MAT.    DEPTH   TEETH
1     10.A   HSS     30.     2
•
```

Note: Use the same registration procedure for face-mills and ball end-mills after calling the **TOOL FILE** display that corresponds to each type of milling tool.

2. Registering chamfering cutter

We will register the chamfering cutter shown below as an example.



Chamfering cutter

(1) Call up the **TOOL FILE (CHAMFER)** display.

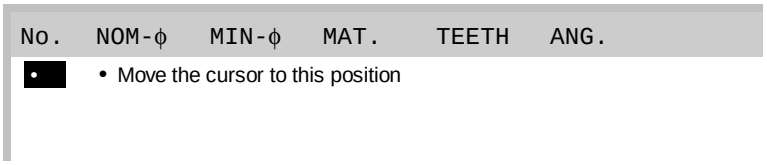
➔ The following display will then be presented.

```
No.   NOM-ϕ   MIN-ϕ   MAT.    TEETH   ANG.
```

- In this example, assume that no chamfering cutters have yet been registered.

(2) Press the cursor key to call up the cursor on the screen.

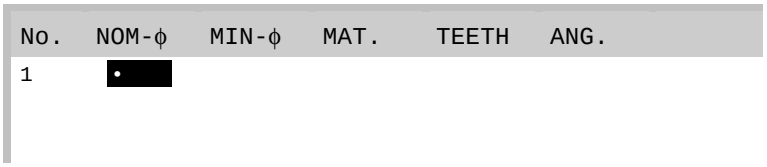
(3) Press the cursor key to move the cursor to the position of the No. item to be registered.



- The cursor can be moved only to the lines of numbers which have been registered and the next line.

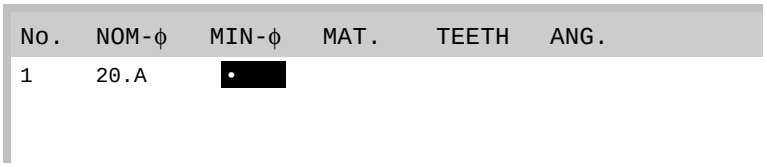
(4) Press the INPUT key.

➔ The corresponding number will then be displayed under No. item, and the cursor will move to the NOM-ϕ item.



(5) Set nominal diameter and suffix data in the same manner as that which is described at steps (5) and (6) of “1. Setting a tool other than chamfering cutter”.

- In the example shown above, if the nominal diameter and the suffix are set as “20” and “A”, respectively, the display will change as follows:



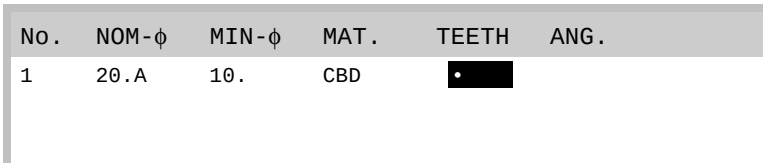
(6) Move the cursor to the MIN-ϕ item.

(7) Input the minimum diameter of the chamfering cutter, and press the INPUT key.

- In the example shown above, input “10” and then press the INPUT key.

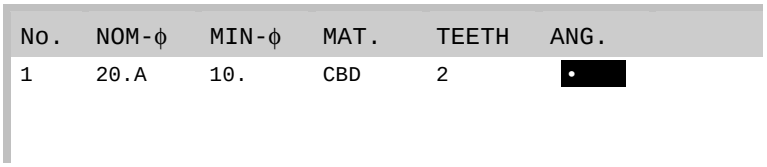
(8) Set the type of material of the tool in the same manner as that which is described at steps (7) and (8) of “1. Setting a tool other than chamfering cutter”.

- In the example shown above, if the type of material is set as “CBD”, the display will change as follows:



(9) Set the number of teeth of the tool in the same manner as that which is described at steps (11) and (12) of “1. Setting a tool other than chamfering cutter ”.

- In the example shown above, if the number of teeth is set as “2”, the display will change as follows:



(10) Move the cursor to the ANG. item.

(11) Input the nose angle of the tool to be registered, and press the INPUT key.

- In the example shown above, specify “45” and then press the INPUT key.

No.	NOM-ϕ	MIN-ϕ	MAT.	TEETH	ANG.
1	20.A	10.	CBD	2	45.

7-1-3 Erasing a part of the registered data

A part of data registered on the **TOOL FILE** display can be erased.

(1) Call up the **TOOL FILE** display on which the data to be partly erased is registered.

Example: Erasing the end-mill data that is registered under No. 3 on the **TOOL FILE (END MILL)** display:

No.	NOM-ϕ	MAT.	DEPTH	TEETH
1	10.A	CBD	30.	2
2	10.B	HSS	10.	2
3	20.A	CBD	20.	4
4	30.A	HSS	15.	3
5	40.A	CBD	20.	4

- In this example, assume that the end-mill data shown above is registered on the display.

(2) Press the cursor key to call the cursor on the screen.

(3) Press the cursor key to move the cursor to the No. item of the tool data to be erased.

No.	NOM-ϕ	MAT.	DEPTH	TEETH
1	10.A	CBD	30.	2
2	10.B	HSS	10.	2
3	20.A	CBD	20.	4
4	30.A	HSS	15.	3
5	40.A	CBD	20.	4

(4) Press the **ERASE** menu key.

➔ This will cause the display of **ERASE** to reverse.

(5) Press the INPUT key.

➔ The data in the line where the cursor is placed will then be erased, and the data on the next line and following lines will then be shifted upwards.

No.	NOM-ϕ	MAT.	DEPTH	TEETH
1	10.A	CBD	30.	2
2	10.B	HSS	10.	2
3	30.A	HSS	15.	3
4	40.A	CBD	20.	4

- After that, the reverse-display status of **ERASE** will be released.

7-3 TOOL OFFSET Display

Function:

This display is used when setting the amount of tool-diameter or tool-position compensation of each tool to be used for an EIA/ISO program.

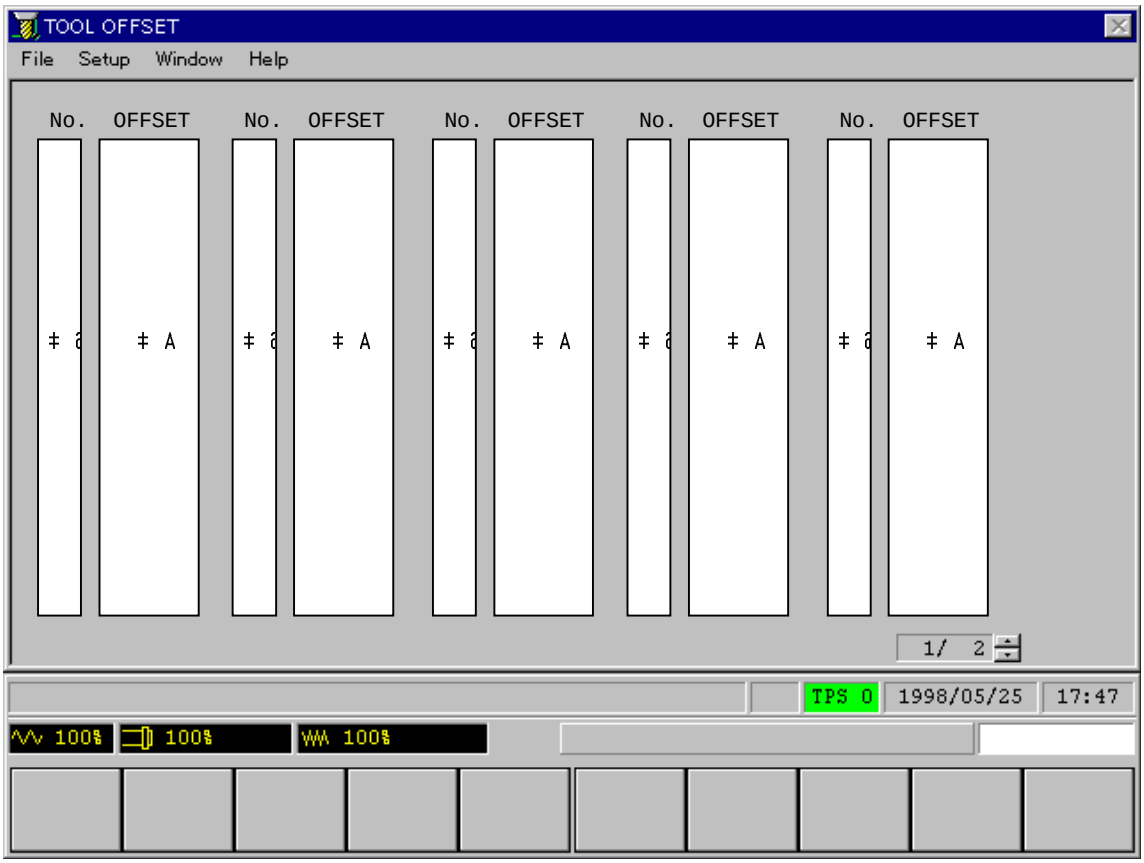
Remarks:

Current position on each axis can be set.

7-3-1 Data display

Type of tool offset differs according to the particular machine model (there are two types of tool offset, type A and type B), and data that will be displayed also differs according to the selected type.

1. Displayed data of type A

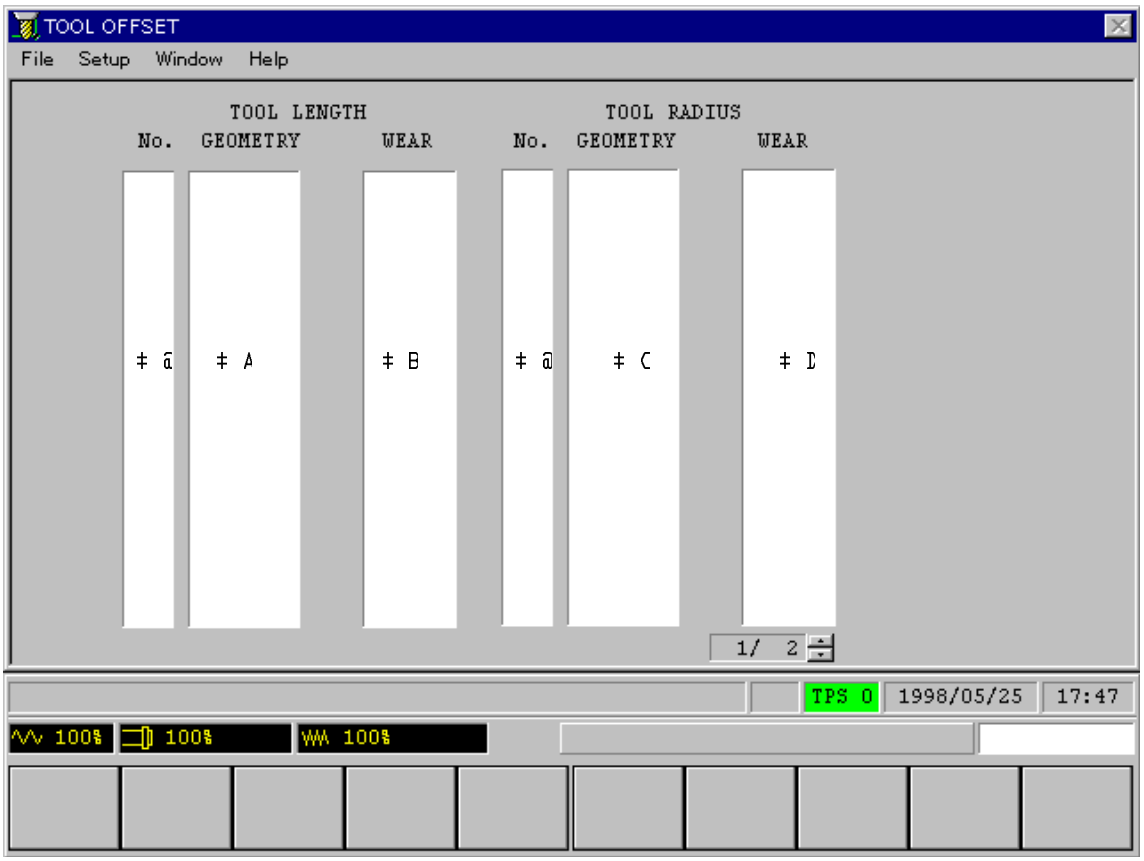


D735S1053E

2. Display data description for type A

No.	Item	Unit	Data description
•	No.	•	Tool offset number for the registered offset amount
•	OFFSET	mm (inch)	Tool offset amount

3. Displayed data of type B



D735S1054E

4. Display data description for type B

No.	Item	Unit	Data description
•	No.	•	Tool offset number for the registered offset amount
•	TOOL LENGTH GEOMETRY	mm (inch)	Amount of geometric compensation for tool length
•	TOOL LENGTH WEAR	mm (inch)	Amount of defacement for tool length
•	TOOL RADIUS GEOMETRY	mm (inch)	Amount of geometric compensation for tool redius
•	TOOL RADIUS WEAR	mm (inch)	Amount of defacement for tool radius

7-1-2 Tool-compensation data setting procedures

Use the following procedures to input data to the OFFSET item of the **TOOL OFFSET** (type A) display or the GEOMETRY and WEAR items of the **TOOL OFFSET** (type B) display:

1. Manual setting procedure

(1) First, press the cursor key to call up the cursor on the screen.

(2) Press the cursor key to move the cursor to the position in which data is to be input.

(3) Specify the desired data with appropriate numeric key(s), and then press the INPUT key.

➔ The input data will then be displayed in the position where the cursor was placed.

2. Automatic setting procedure using the TEACH function

Using the TEACH function of the display, tool-length compensation data can be automatically input to the OFFSET item of the **TOOL OFFSET** (type A) display or the TOOL LENGTH GEOMETRY item of the **TOOL OFFSET** (type B) display.

(1) Move the axes manually and bring the tool tip into contact with the top surface of either the reference block or the workpiece of a predetermined height.

- See the Operating Manual for machine.

(2) Call the cursor on the **TOOL OFFSET** display and move it to the position of the OFFSET item (in the case of type A) or TOOL LENGTH GEOMETRY item (in the case of type B) corresponding to the tool on the spindle.

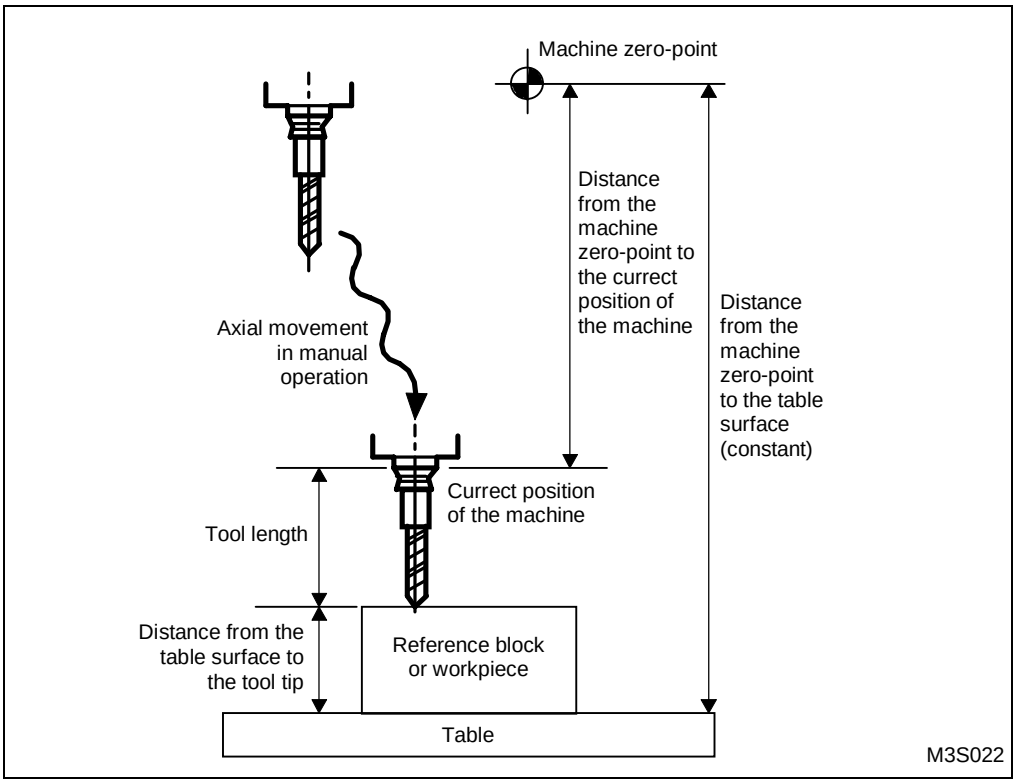
- For details of cursor calling, see the description of step (1) of the “1. Manual setting procedure” above.

(3) Press the **TEACH** menu key.

➔ This will cause the display status of **TEACH** to reverse.

(4) Specify the height of either the reference block or the workpiece using the appropriate numeric key(s), and then press the INPUT key.

➔ The NC equipment will then calculate the tool length, and the calculated value will be automatically set under OFFSET item (in the case of type A) or TOOL LENGTH GEOMETRY item (in the case of type B).



Since the current position of the machine is placed in the memory of the NC equipment, it will calculate the length of the tool if the height of the reference block or workpiece (i.e., the distance from the table surface to the tool tip) is specified.

- The reverse-display status of **TEACH** will be released automatically.

7-1-3 Setting the current position

The current position of the tool nose in the workpiece coordinate system can be set while the POSITION window remains on the **TOOL OFFSET** display. See Section **2-4 Window**, for further details of the POSITION window. The procedure for setting the current position of each axis is described below.

(1) After selecting the manual operation mode, press the WINDOW key, select “Position” through “Window” on the menu bar and press the INPUT key to open the POSITION window.

(2) Move the mouse pointer to the current position data of the desired axis by moving a finger along the touchpad, and click the left mouse button to display the cursor.

(3) Input a new coordinate value using the numeric keys, and press the INPUT key.

(4) Move the mouse pointer to the desired position on the **TOOL OFFSET** display by moving a finger along the touchpad, and click the left mouse button.

(5) Clear the POSITION window from the display by pressing the WINDOW key and performing a similar operation to that in Step (1).

7-1-4 EIA/ISO tool length measurement

1. Outline

- Automatically measured tool lengths can be set in tool offsets automatically.
- Two methods are available to measure tool lengths: fully automatic and semi-automatic.
  Fully automatic tool length measurement allows the following operations to be carried out:
  - Continuous control of a maximum of 13 tools
  - Assignment of any offset number
  - Automatic movement of any tool in X-axial and Y-axial directions (Automatic movement of a tool not having its nose set to the spindle center) [Control parameter: **L94**]

  Semi-automatic tool length measurement allows any offset number to be assigned. [Control parameter: **L95**]
- The EIA/ISO tool length measurement function works with an EIA/ISO option. If, however, bit 3 of parameter **F93** (MAZATROL tool length valid/invalid for EIA/ISO programs) is set to “valid”, this function will not work even when the above-mentioned option is provided.

2. Fully automatic tool length measurement - Operating procedure

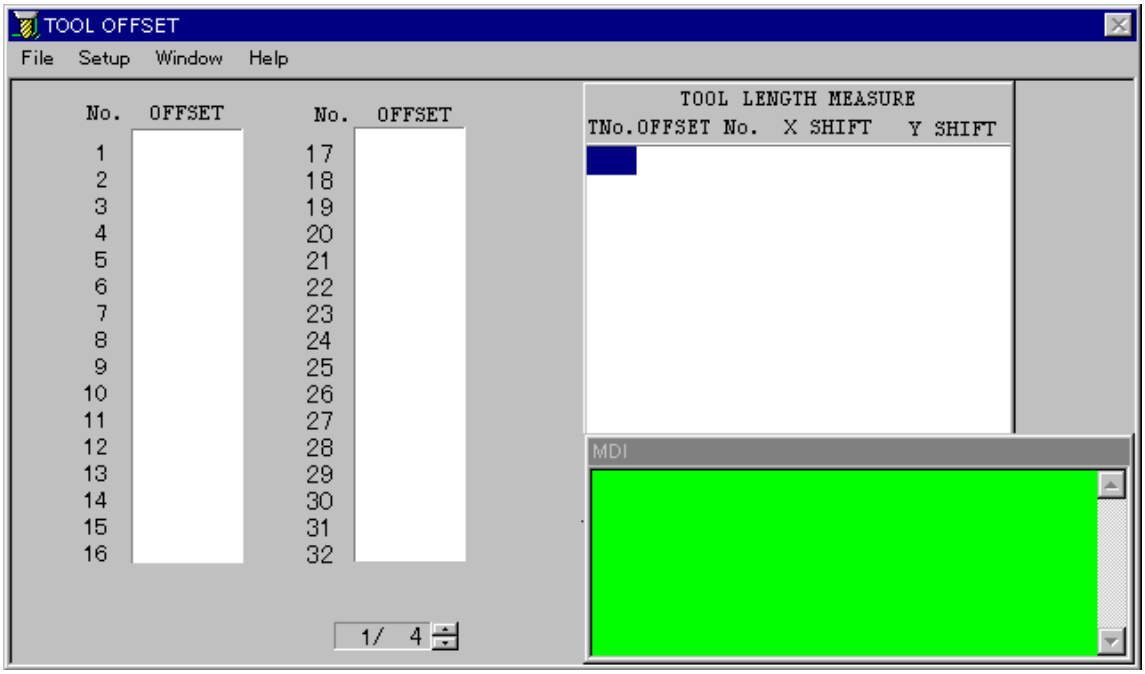
(1) Select the **TOOL OFFSET** display.

(2) Set the MDI mode.

➔ This displays an MDI window.

(3) Press the **T MSR AUTO** menu key.

➔ This reverses the display status of the menu item to make the right-hand half of the display into the setting/display area of tool length measurement data.



D735S1055E

(4) Specify the number of the tool to be measured.

(5) Specify the offset number corresponding to the tool number.

- Automatic offset number setting
  Automatic setting of offset numbers during tool number input becomes possible by setting the required parameter.
  Automatically set offset numbers are keyed to tool numbers (the amount of shifting is set with parameter).

  Parameter **L95**: Automatic offset number setting for offset tool measurement (1: Yes, 0: No)

  Parameter **L96**: The amount of shifting for the offset number that is to be automatically set for offset tool measurement

(6) Move the cursor to specify the desired amounts of movement in X/Y directions.

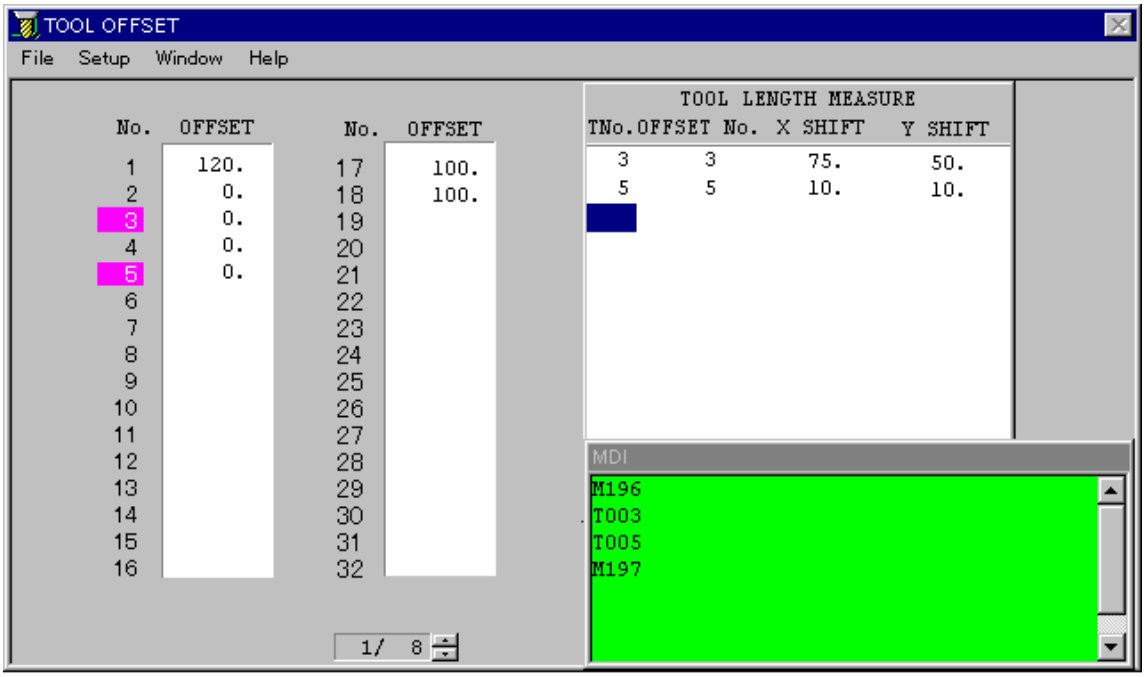
- Specifying the movements of X/Y axial movement
  The required parameter can be used to select whether the items X SHIFT and Y SHIFT are to be displayed.

  Parameter **L94**: Display of X/Y SHIFT items (1: Yes, 0: No)

(7) Press the **MDI START** menu key.

(8) Press the INPUT key

- This displays measurement M- and T-codes in the MDI window and causes the corresponding offset number(s) on the left half of the display to reverse in red.



D735S1056E

(9) While the display of the menu item shown previously at step (3) remains in reverse form, steps (4) to (8) can be repeated for up to a maximum of 13 tools.

(10) Press the CYCLE START button to perform the measurement operation.

➔ Measurement results will then be set under the corresponding offset number.

- In the flow of the measurement for multiple tools, the page containing the data item to be set for each tool is automatically selected and displayed on the left half.

Note: Specified data having a displayed cursor on the right half can be erased for each line using the **ERASE** menu function.
Also, only the data at the cursor position can be deleted using the CANCEL key. The data in the MDI window, however, will remain unchanged. Update the MDI window data by pressing once again the **MDI START** menu key.

3. Semi-automatic tool length measurement - Operating procedure

(1) Select the **TOOL OFFSET** display.

(2) Set the MDI mode.

➔ This displays an MDI window.

(3) Press the **T MSR SEMIAUTO** menu key.

➔ As with fully automatic tool length measurement, the display status of the menu item will then reverse to make the right-hand half of the display into the setting/display area of tool length measurement data.
The number of the tool that is currently mounted on the spindle is displayed at TNo. item.

(4) Specify data under OFFSET No.

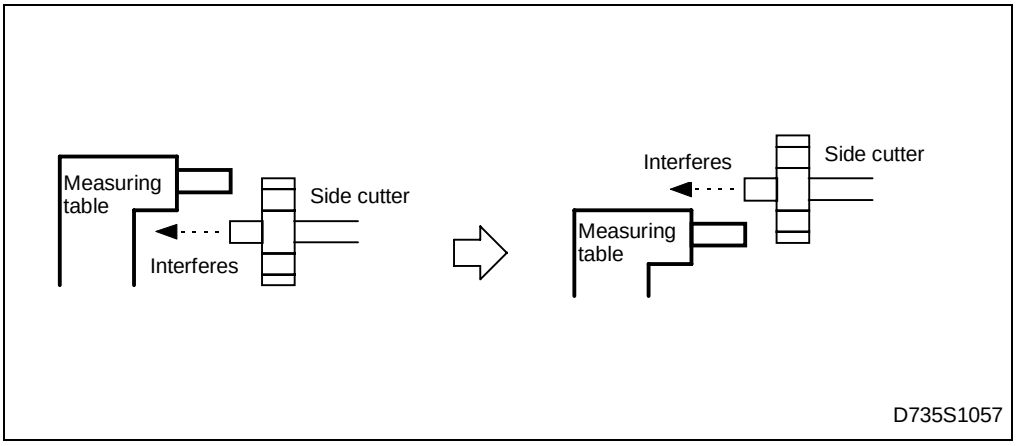
(5) Press the INPUT key.

➔ This displays measurement M-codes in the MDI window and releases the reverse display

status of the menu item.
If an offset number is unset, no measurement M-codes will be displayed in the MDI window.

(6) Press the CYCLE START button to perform the measurement operation.

- Use semi-automatic tool length measure-ment if, as shown below, tool interference is to be considered for beforehand.



4. Remarks

- This function is not available for a multi-nose tool that requires setting of multiple tool lengths for one pocket.
- Parameter **L96** (the parameter for setting the amount of shifting based on an offset number) should be set to a value more than 0. After shifting, an alarm will occur if the maximum usable offset number is exceeded.
- This function cannot be used to measure the tool length of a touch sensor.

8 DISPLAYS RELATED TO PARAMETERS

8-1 CUTTING CONDITION (W.-MAT./T.-MAT.) Display

Function:

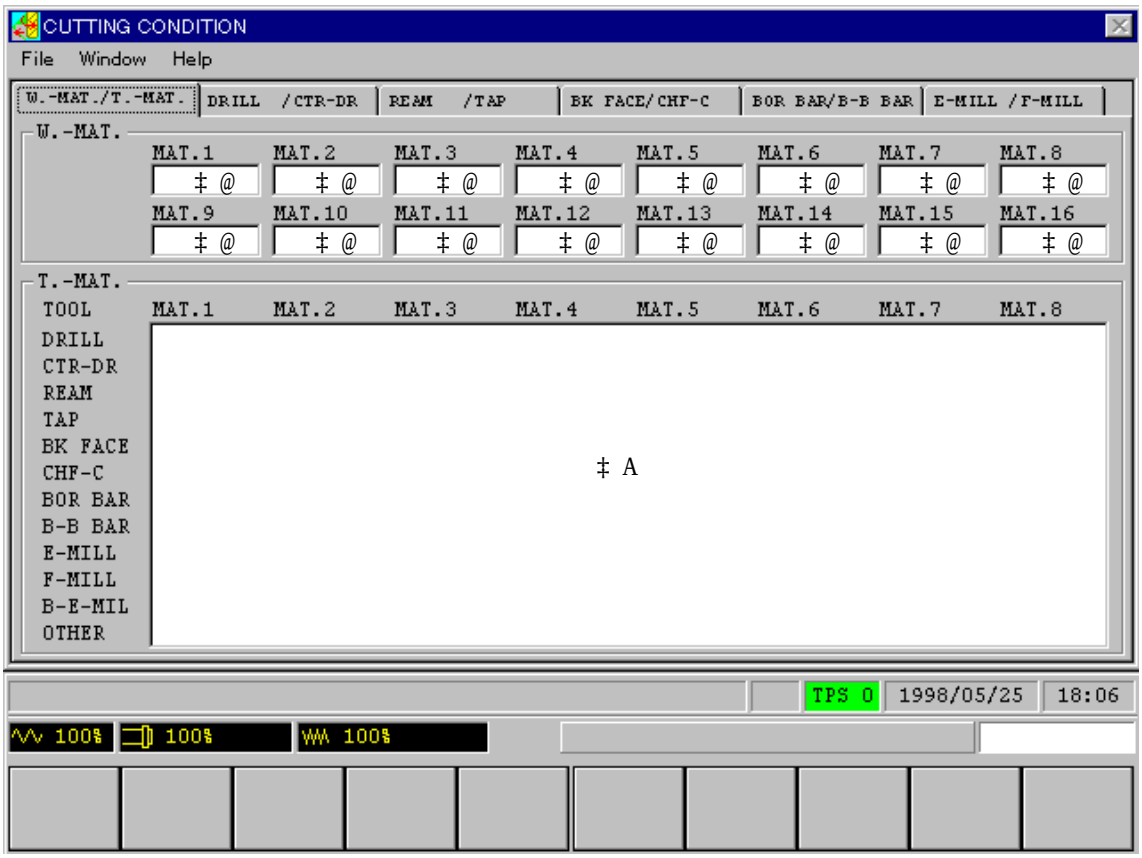
Registering the materials types of workpieces and tools that are to be selected for automatic setting of cutting conditions on the **PROGRAM (MAZATROL)** display and the **TOOL FILE** display

Remarks:

- Data that has been registered on this display can be used only for a MAZATROL program.
- Initially, YAMAZAKI-recommended data is displayed. You can change or add data as required.

8-1-1 Data display

1. Displayed data



D735S1058E

2. Display data description

No.	Item	Unit	Data description
	W.-MAT.		Workpiece material name
	T.-MAT		Tool material name

8-1-2 Registering data

The **CUTTING CONDITION (W.-MAT./T.-MAT.)** display allows the names of the materials of workpieces and tools to be registered using character strings of alphabetic characters and arabic numerals.

(1) Display the cursor by pressing the cursor key.

(2) Move the cursor to any of the workpiece materials data items from MAT.1 to MAT.16 or the tool materials data items from MAT.1 to MAT.8.

(3) Input the appropriate name using the alphabetic and/or numeric keys, and then press the INPUT key.

- Up to eight characters can be set. The maximum number of workpiece materials types is sixteen, and that of tool materials types is eight for each tool.
- Press the CANCEL key to delete the input name.

Note 1: Two parameters are provided for automatic development of the tool sequence in the MAZATROL program. One is **D14** (decisive factor for drilling aluminum materials) and one is **D15** (decisive factor for drilling non-aluminum materials). Which of the two parameters will be auto-selected depends on the selected workpiece material. Workpiece material 6 is regarded as aluminium. Parameter **D14** will be used if the workpiece material 6 is selected in the MAZATROL program. Parameter **D15** will be used if another workpiece material is selected.
Refer to the Programming Manual (MAZATROL) for further details of automatic tool development. See the Parameter List for further details of the parameters.

Note 2: The workpiece material names registered in this display are cross-referred to those in the file (\nm64mdata\eng\cspmat.txt) containing the data for the calculation of the average output indicated on **the MACHINING NAVIGATION - PREDICT** display. Changing a workpiece material name in one storage area requires the same operation for the other. Refer to item of “3. Display data description” in Subsection 6-9-1.

8-1-3 Read and Write of text file

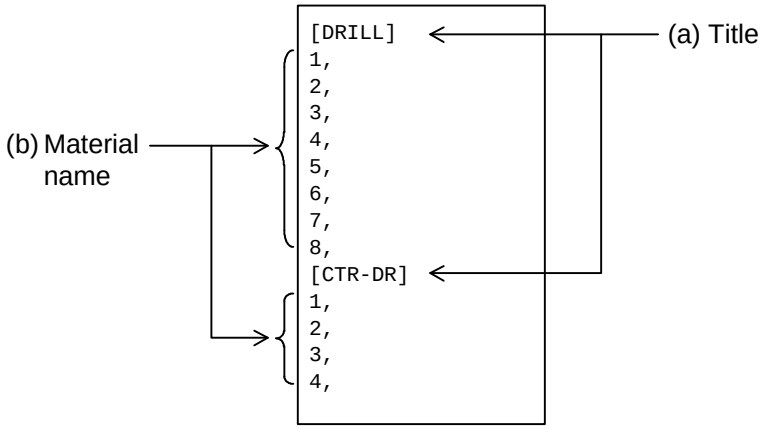
The workpiece and tool material names (W.-MAT. and T.-MAT.) in the cutting conditions can be taken in and out as a text file of the hard disk. Use the item “File” on the menu bar to perform these operations.

1. File saving and file loading

- File saving
  The workpiece and tool material names indicated on the **CUTTING CONDITION** display are saved onto the hard disk as a text file (\nm64data\eng\material.txt).
- File loading
  The text file (\nm64data\eng\material.txt) of the workpiece and tool material names saved on the hard disk is loaded into the NC unit.

Note: “eng” in the file name correspoonds with the language currently selected for the display mode.

2. Format of the text file



(a) Title
The title denotes the category of the material data described under it as listed below.

Title	Meaning	Title	Meaning
WORK	Workpiece	BOR BAR	Boring bar
DRILL	Drill	B-B BOR	Back-boring bar
CTR-DR	Center drill	E-MILL	End-mill
REAM	Reamer	F-MILL	Face-mill
TAP	Tap	B-E-MILL	Ball-end-mill
BK FACE	Back spot-facing tool	OTHER	Other types of tool
CHF-C	Chamfering cutter		

Note: The characters enclosed in brackets [ ] are used as the keyword to construct the file. Do not enter any comment on this character line. Otherwise, the data may not be read correctly during file loading.

(b) Material name
Materials are described.

```
2,
```

The first numeral denotes the material number corresponding to the title given above. The material name can be entered after the comma “,”.

Example 1:

File

```
%WORK%
1,S45C
2,...
```

"S45C" is registered as workpiece material No. 1.

Note 1: The material name can consist of a maximum of eight half-size characters (or four normal-size characters).

Note 2: The character string of the material name must not begin or end with a half-size null code. This code can only be used within the character string.

Example 2: Leading/trailing half-size null code.

File

```
%WORK%
1,␣S45C␣
2,....
```

An example of incorrect description
(␣ denotes a half-size null code.)

Example 3: Half-size null code within a character string

File

```
%DRILL%
1,HSS␣B
2,...
```

An example of correct description

Note 3: Do not enter one and the same material name repeatedly under the same category.

Example 4:

File

```
%WORK%
1,S45C
2,S45C
3,FC
4,S45C
```

An example of incorrect entry
(Loading of this file only registers "S45C" and "FC" as workpiece material No. 1 and 3 respectively; entry of No. 2 and 4 is ignored.)

8-2 CUTTING CONDITION Display

Function:

Registering the fundamental data for automatic setting of the cutting conditions on the **PROGRAM (MAZATROL)** display and the **TOOL FILE** display.

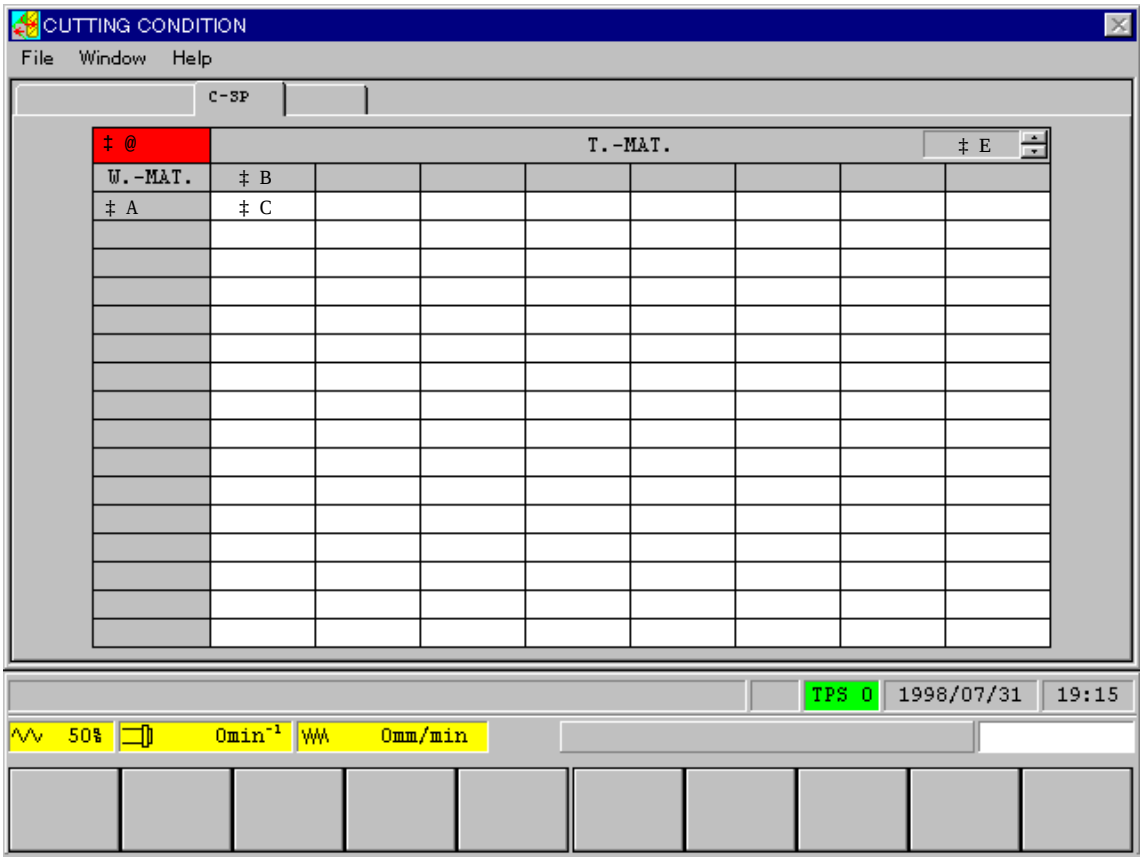
Remarks:

- The **CUTTING CONDITION** display consists of the following two sub-displays containing the surface velocity and feed rate data for each tool type:
  1. **CUTTING CONDITION (C-SP)** display
  2. **CUTTING CONDITION (FR)** display
- Data that has been registered on this display is used only for MAZATROL programs.
- Data recommended by Yamazaki Mazak is pre-registered prior to shipping of the machine. The registered data can be modified as required.
- The cutting conditions to be used for the MAZATROL program are automatically calculated from the registered data within this display, the **CUTTING CONDITION (W.-MAT./T.-MAT.)** display, and the **TOOL DATA** display, the registered data within the program, and the settings of parameters. Refer to the Programming Manual for calculation expressions.

Note: The **CUTTING CONDITION (C-SP)** display will be auto-selected when any other display is changed over to the **CUTTING CONDITION** display.

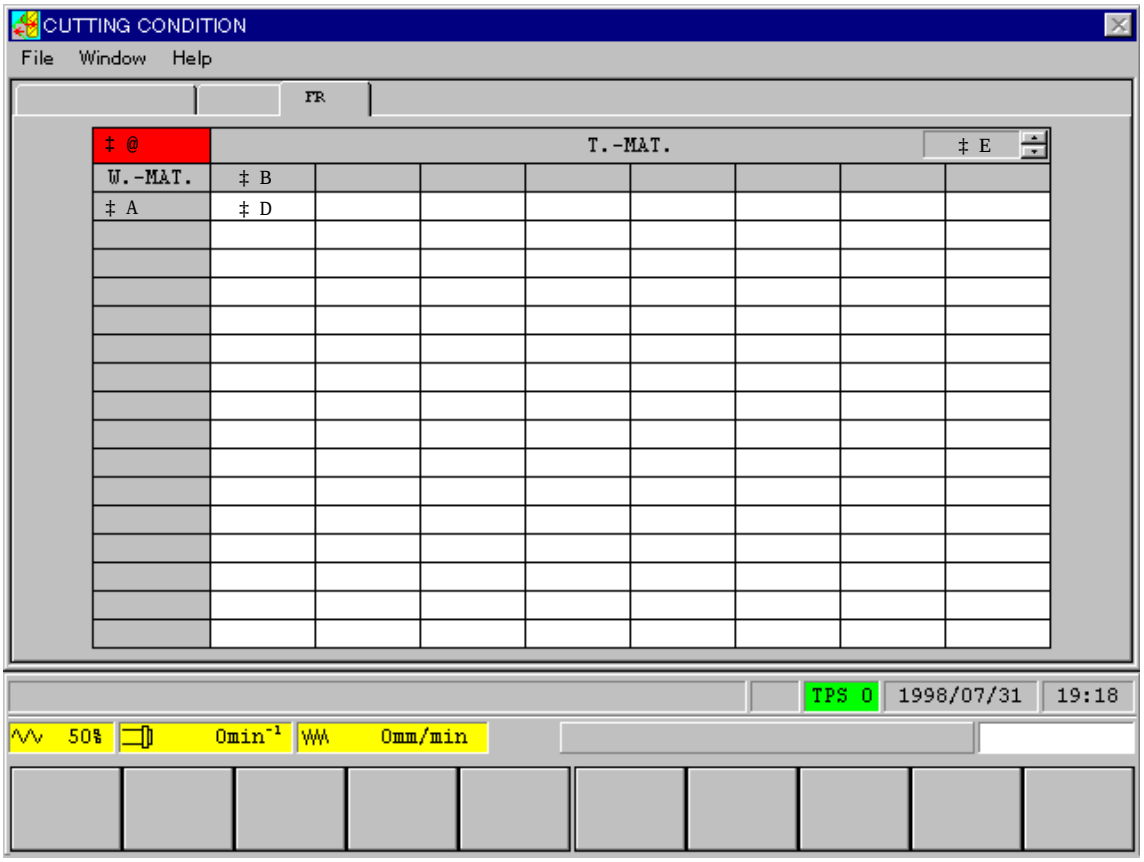
8-2-1 Data display

A. CUTTING CONDITION (C-SP) display



D735S1082E

B. CUTTING CONDITION (FR) display



D735S1083E

2. Display data description

No.	Item	Unit	Data description
	Tool name		Name (type) of the currently selected tool
	W.-MAT.		Name of the material of the workpiece
	T.-MAT.		Name of the material of the tool
	C-SP	m/min (ft/min)	Surface velocity for the workpiece material
	FR	mm/rev (inch/rev)	Feed rate for the workpiece material
	Page		Current page and the total number of pages **a** / **b** **a**: Current page **b**: Total number of pages

8-2-2 Data registration

The following data can be registered for each tool on the **CUTTING CONDITION** display:

1. Surface velocities for each workpiece material type registered on the **CUTTING CONDITION (W.-MAT./T.-MAT)** display
2. Feed rates for each workpiece material type registered on the **CUTTING CONDITION (W.-MAT./T.-MAT)** display

Cutting conditions data differs according to the selected type of milling tool, type of workpiece material, and type of tool material. The method of setting the data, however, is the same. Here, how to set data is described below taking the following conditions as an example:

1. Assume that “FC” and “HSS” are already registered under MAT.1 of W.-MAT. and MAT.1 of DRILL item, respectively, on the **CUTTING CONDITION (W.-MAT./T.-MAT)** display.

```
W.-MAT.
          MAT.1    MAT.2    MAT.3
          FC
          MAT.9    MAT.10   MAT.11
T.-MAT.
TOOL      MAT.1    MAT.2    MAT.3
DRILL     HSS
CTR-DR
```

2. Set the C-SP and FR data of the drilling tool for the workpiece material “FC” to “110 (m/min)”

(1) Call up the **CUTTING CONDITION (C-SP)** display.

➔ The following display will then be made:

```
DRILL                          T.-MAT.
W.-
MAT.        HSS     BALANCE    CARBID
                               E
FC
```

- This example assumes that only “FC” and “HSS” are already registered as the workpiece material type and the drilling tool material type, respectively, on the **CUTTING CONDITION (W.-MAT./T.-MAT)** display.

(2) Display the cursor by pressing the cursor key.

(3) Using the cursor keys, move the cursor to the HSS column of the drilling tool for the workpiece material “FC”. (Use the page keys to enter the C-SP data for other types of tool.)

```
DRILL                          T.-MAT.
W.-
MAT.        HSS     BALANCE    CARBID
                               E
FC          ■
            ↑
            └── Move the cursor to this position
```

(4) Input a surface velocity using the numeric keys, and press the INPUT key.

- In the example shown above, “110” is to be set.

(5) Next, call up the **CUTTING CONDITION (FR)** display.

➔ The following display will then be made:

```
DRILL                          T.-MAT.
```

W.-MAT. FC	HSS	BALANCE	CARBID E

(6) Using the cursor keys, move the cursor to the HSS column of the drilling tool for the workpiece material “FC”. (Use the page keys to enter the FR data for other types of tool.)

DRILL			T.-MAT.
W.-MAT. FC	HSS	BALANCE	CARBID E

Move the cursor to this position

(7) Input a feed rate using the numeric keys, and press the INPUT key.

- In the example shown above, set “2” and press the INPUT key.

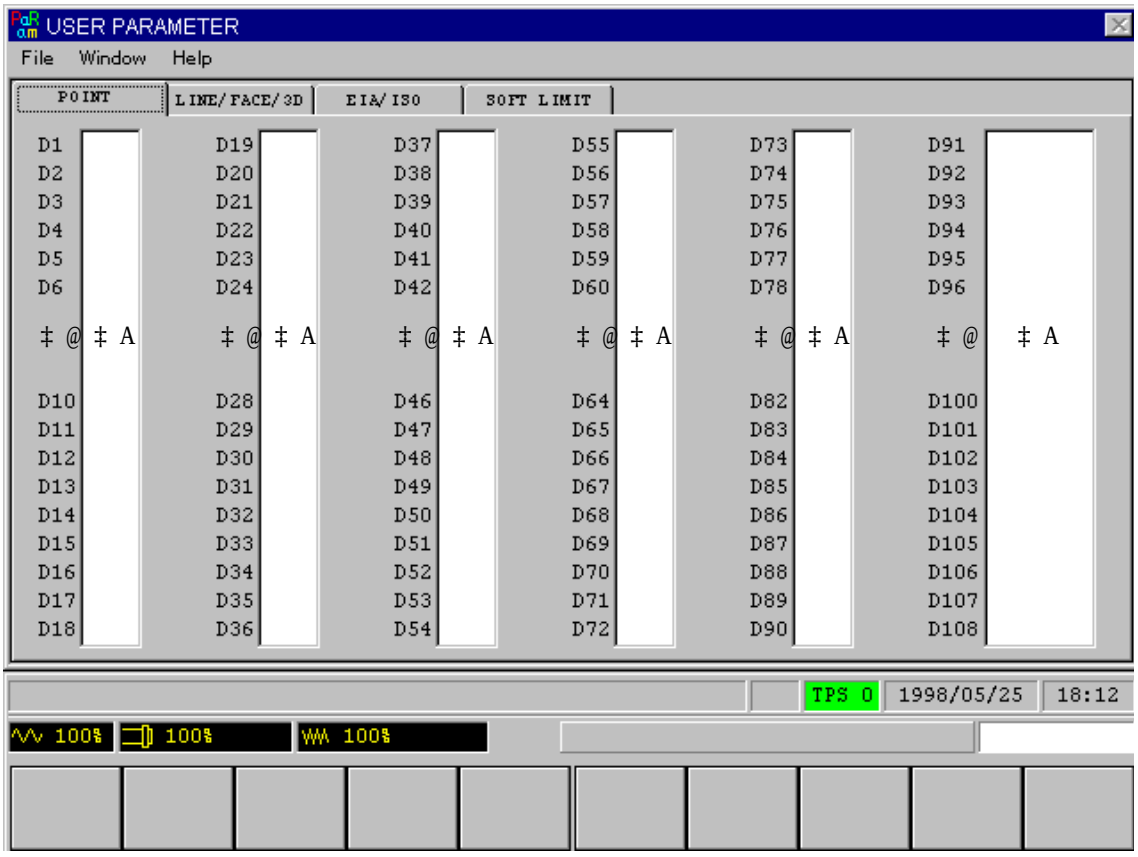
8-3 USER PARAMETER Display

Function:

Registering and displaying user parameters

8-3-1 Data display

1. Displayed data



D735S1060E

2. Display data description

No.	Item	Unit	Data description
	Address		Address of various user parameters

	Data		Data of various user parameters

Note 1: For further details, see the Prameter List.

Note 2: Confirm the contents of parameters before changing them.

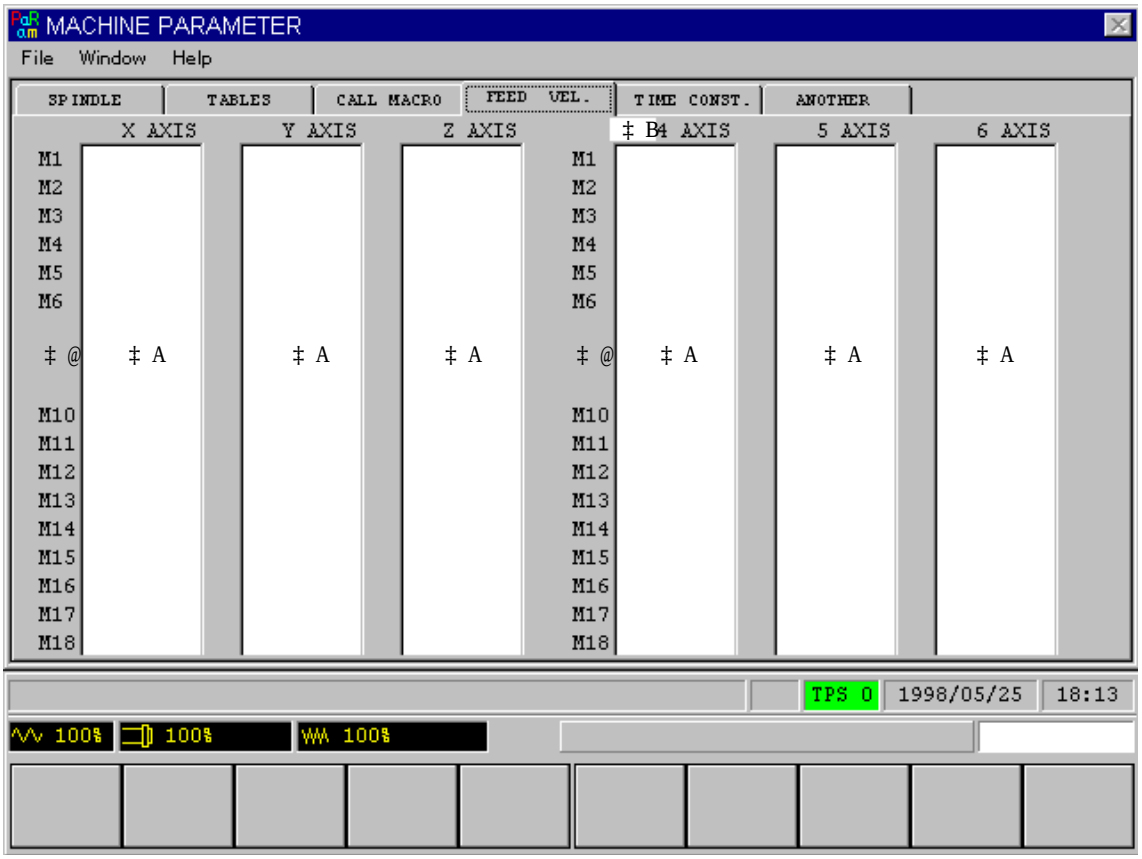
8-4 MACHINE PARAMETER Display

Function:

Registering and displaying machine parameters

8-4-1 Data display

1. Displayed data



D735S1105E

2. Display data description

No.	Item	Unit	Data description
	Address		Address of various machine parameters
	Data		Data of various machine parameters
	Axis name		Axis name **Note:** According to parameter number, it is not displayed.

Note 1: For further details, see the Parameter List.

Note 2: Confirm the contents of parameters before changing them.

- NOTE -

9 DISPLAY RELATED TO DATA STORAGE

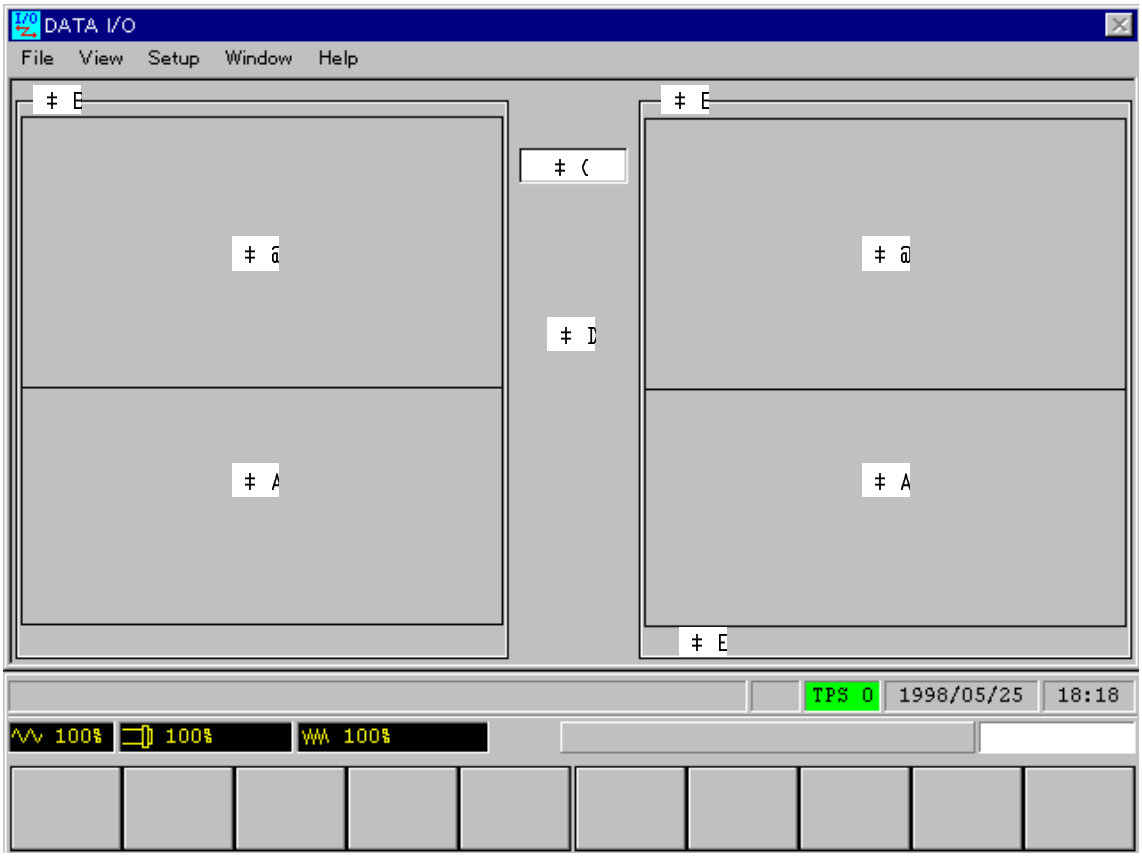
9-1 DATA I/O Display

Function:

This display is used to transfer programs or other types of data between the NC unit and an external unit.

9-1-1 Data display

1. Displayed data



D735S1062E

2. Display data description

No.	Item	Unit	Data description
•	Work number	•	Work number of program for transference and identification of MAZATROL programs or EIA/ISO programs M: MAZATROL program E : EIA/ISO program
•	Data type	•	Data other than programs for transference
•	Unit name	•	Name of unit for transference
•	I/O mode	•	Data transfer mode (loading, saving, etc.)
•	Display of execution	•	Display that indicates data is being input or output. During data input or output, display according to transfer mode being executed (•) is made.
•	CONVERT	•	Display that indicates an M PLUS program is being converted into a program for the NC unit.

9-1-2 DATA I/O operations (CARD)

The following four data input/output operations can be executed using a memory card.

1. Loading
   The program data or other types of data prestored on a memory card are registered into the NC unit.
2. Saving
   The program data or other types of data pre-registered in the NC unit are stored onto a memory card.
3. Directorying
   The data type prestored on a memory card are displayed.
4. Formatting
   A memory card is formatted (initialized).

The operating procedure of each operation listed above is described below.
The following description assumes that a memory card unit has already been connected to the NC unit and that memory card setting is also completed.

CAUTION

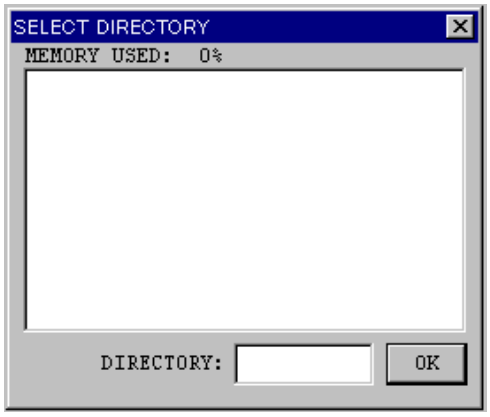
- Only use IC memory cards of the ATA specification. Failure to observe this instruction could result in machine damage.

1. Selecting the directory

For loading, saving and directorying with a memory card, moreover, selection of the required directory must be done beforehand by the following procedure:

(1) Press the **DIR. SELECT** menu key.

➔ **DIR. SELECT** will change to a reverse-display status and the following window will appear.



(2) Key in the name of the required directory and press the INPUT key.

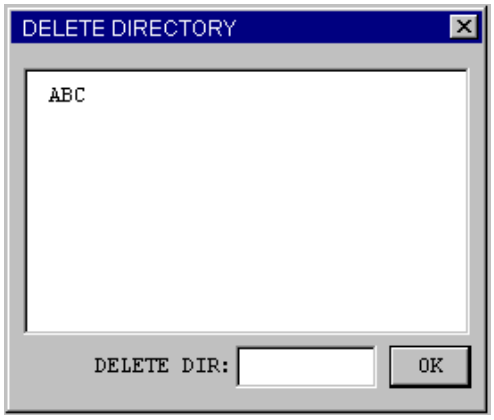
- Use the alphabetic and numeric keys to input the name.

2. Deleting the directory

To delete a directory (with all the files thereunder) on the memory card, proceed as follows:

(1) Press the **DIR. DELETE** menu key.

➔ **DIR. DELETE** will change to a reverse-display status and the following window will appear.



(2) Key in the name of the directory to be deleted and press the INPUT key.

- Use the alphabetic and numeric keys to input the name.

Note: All the files stored under the specified directory will be deleted by the above operation.

3. Loading

(1) Press the **CARD** menu key.

(1) Press the **NC ← CARD LOAD** menu key.

➔ LOAD will be displayed on the I/O mode item.

(3) Select the data to be transferred.

<For program data transfer>

- Move the cursor to the work number item of (CARD) column and input the work number of the program to be transferred. The work number will change to a reverse-display status after setting the work number.
- Up to 32 programs can be selected.

<For non-program data transfer>

- Move the cursor to the intended data in data type item of (CARD) column and press the INPUT key. The data name will change to a reverse-display status. It indicates that the data is to be transferred.

<For erasing the data that has been set>

- Carry out the following operations to erase the entire data that has been set:

1) Press the **ALL ERASE** menu key.

➔ **ALL ERASE** will change into a reverse-display status.

2) Press the INPUT key.

➔ The reverse-display status of the entire data will be cleared and all the work number will become blank.

(4) Press the **START** menu key.

➔ **START** will change into a reverse-display status and data transfer will begin.

- The display color will change at the position where the data being transferred is displayed.

- The highlighted-display status of the data that has been transferred will be cleared. The same data will be displayed in the (NC) column.
- After data transfer, the reverse-display status of **START** will be cleared.

Note 1: You can select using a parameter whether loading of a program whose work number is the same as that of a program pre-registered within the NC unit, is to be enabled. For further details of parameters, see the Parameter List.
To enable such a program for which the loading operation is not available by parameter setting, to be loaded, the work number of the related pre-registered program must be changed prior to loading.

Note 2: A listing window for the work numbers of all programs registered previously in the NC unit can be displayed by pressing the **PROGRAM FILE** menu key. While the window remains on the display, the display status of the **PROGRAM FILE** menu reverses.
Pressing the **PROGRAM FILE** menu key once again clears the reverse-display status of the menu item and closes the window.

Note 3: Data transfer will be continued even if the existing display is changed over to another display during the data transfer process. However, the data being transferred cannot be overridden.
This also applies when a transfer mode other than LOAD has been selected.

Note 4: Data transfer will halt if the reverse-display status is cleared by pressing the **START** menu key during the data transfer process. Data transfer cannot be restarted from the position in which the transfer process has halted.
This also applies when a transfer mode other than LOAD has been selected.

4. Saving

(1) Press the **CARD** menu key.

(1) Press the **NC** → **CARD SAVE** menu key.

➔ SAVE will be displayed on the I/O mode item.

(3) Select the data to be transferred.

- The data input procedure is the same as that of LOAD. Input the data in the (NC) column. (See procedural step (3) of “3. Loading”.)

(4) Press the **START** menu key.

➔ **START** will change to a reverse-display status and data transfer will begin.

- The display color will change at the position where the data being transferred is displayed.
- The data that has been transferred will change to a normal display status and the same data will be displayed in the (CARD) column.
- After data transfer, the reverse-display status of **START** will be cleared.

5. Directorying

(1) Press the **CARD** menu key.

(2) Press the **CARD CONTENT** menu key.

➔ DIR will be displayed on the I/O data item.

(3) Press the **START** menu key.

➔ **START** will change into a reverse-display status, and what type of data is stored on the memory card will be displayed in the (CARD) column on the display.

- If program data is stored on the external media, the corresponding work number will be displayed on the work number item.
- If non-program data is stored on the external media, the corresponding item will be displayed in a reverse-display status.
- After entire data display is completed, the reverse-display status of **START** will be cleared.

6. Formatting

(1) Press the **CARD** menu key.

(2) Press the **CARD FORMAT** menu key.

➔ FORMAT will be displayed on the I/O mode item.

(3) Key in "–9999" and press the INPUT key.

(4) Press the **START** menu key.

➔ **START** will change into a reverse-display status and initialization will begin.

- Upon completion of the initialization, the reverse-display status of **START** will be cleared.

7. Selecting the HDD OPERATION PROGRAM area

Use the following procedure to select the HDD OPERATION PROGRAM area, instead of the STANDARD PROGRAM area, for data input/output operations with an IC memory card:

(1) Press the **HD AREA** menu key.

➔ **HD AREA** will change into a reverse-display status.

- The HDD OPERATION PROGRAM area will remain selected for the I/O operations with the IC memory card until the reverse display of **HD AREA** is cleared by pressing again the same menu key.

9-1-3 DATA I/O operations (CMT)

The following four data input/output operations can be executed using a cassette tape or a microdisk.

1. Loading
   The program data or other types of data prestored on a cassette tape (CMT) or a microdisk are registered into the NC unit.
2. Saving
   The program data or other types of data pre-registered in the NC unit are stored onto a cassette tape or a microdisk.
3. Comparing
   The comparison is made between the data prestored on a cassette tape or a microdisk and the data pre-registered in the NC unit.
4. Directorying
   The data type prestored on a cassette tape or a microdisk are displayed.

The operating procedure of each operation listed above is described below.
The following description assumes that a cassette tape deck or a MAZAK microdisk unit has already been connected to the NC unit and that cassette tape or microdisk setting is also completed.

1. Loading

(1) Press the **CMT I/O** menu key.

(2) Press the **LOAD NC ← CMT** menu key.

➔ LOAD will be displayed on the I/O mode item.

(3) Select the data to be transferred.

<For program data transfer>
- Move the cursor to the work number item of (CMT) column and input the work number of the program to be transferred. The work number will change to a reverse-display status after setting the work number.
- Up to 32 programs can be selected.

<For non-program data transfer>
- Move the cursor to the intended data in data type item of (CMT) column and press the INPUT key. The data name will change to a reverse-display status. It indicates that the data is to be transferred.

<For erasing the data that has been set>
- Move the cursor to the data to be erased and press the data cancellation key. When the data to be erased is a program, then the work number will become blank and the reverse-display status will be cleared. In case of non-program data, the reverse-display status will be cleared.
- Carry out the following operations to erase the entire data that has been set:
  1) Press the **ALL ERASE** menu key.
     ➔ **ALL ERASE** will change into a reverse-display status.
  2) Press the INPUT key.
     ➔ The reverse-display status of the entire data will be cleared and all the work number will become blank.

(4) Press the **START** menu key.

➔ **START** will change into a reverse-display status and data transfer will begin.

- The display color will change at the position where the data being transferred is displayed.
- The highlighted-display status of the data that has been transferred will be cleared. The same data will be displayed in the (NC) column.
- After data transfer, the reverse-display status of **START** will be cleared.

Note 1: You can select using a parameter whether loading of a program whose work number is the same as that of a program pre-registered within the NC unit, is to be enabled. For further details of parameters, see the Parameter List.
To enable such a program for which the loading operation is not available by parameter setting, to be loaded, the work number of the related pre-registered program must be changed prior to loading.

Note 2: A listing window for the work numbers of all programs registered previously in the NC unit can be displayed by pressing the **PROGRAM FILE** menu key. While the window remains on the display, the display status of the **PROGRAM FILE** menu reverses.
Pressing the **PROGRAM FILE** menu key once again clears the reverse-display status of the menu item and closes the window.

Note 3: It is possible to load a program that has been created using the MAZATROL M PLUS (M PLUS program). In that case, the M PLUS program will be automatically converted into a program used for the NC unit of the machine. It is not possible, however, to load any other types of data than programs even if they have been stored in the M PLUS.
While the NC unit is reading in an M PLUS program, the message CONVERT will be displayed at the lower right corner of the display.

Note 4: Data transfer will be continued even if the existing display is changed over to another display during the data transfer process. However, the data being transferred cannot be overridden.
This also applies when a transfer mode other than LOAD has been selected.

Note 5: Data transfer will halt if the reverse-display status is cleared by pressing the **START** menu key during the data transfer process. Data transfer cannot be restarted from the position in which the transfer process has halted.
This also applies when a transfer mode other than LOAD has been selected.

Note 6: If data saving is interrupted, then although all previously set items will have their display status reversed for the next directory, only data that has been transferred down to interruption will have actually been saved.

2. Saving

(1) Press the **CMT I/O** menu key.

(2) Press the **SAVE NC → CMT** menu key.

➔ SAVE will be displayed on the I/O mode item.

(3) Select the data to be transferred.

- The data input procedure is the same as that of LOAD. Input the data in the (NC) column. (See procedural step (3) of “1. Loading”.)

(4) Press the **START** menu key.

➔ **START** will change to a reverse-display status and data transfer will begin.

- The display color will change to purple at the position where the data being transferred is displayed.
- The data that has been transferred will change to a normal display status and the same data will be displayed in the (CMT) column.
- After data transfer, the reverse-display status of **START** will be cleared.

Note 1: When saving the entire NC data onto a microdisk, divide the storage area of the microdisk into two segments, not into six segments. Division into six segments may result in the lack of capacity.

Note 2: The **PAGE SAVE** menu item will be displayed if the display status of the **PROGRAM FILE** menu item is reversed by pressing the **PROGRAM FILE** menu key during the saving transfer mode.

Note 3: Pressing the **PAGE SAVE** menu key during data setting allows all work numbers displayed in the work number listing window to be automatically set.
If the **PAGE SAVE** menu key is pressed during data input/output, alarm **422 MEMORY PROTECT (I/O BUSY)** will be displayed. This alarm will also be displayed if the COMPARE transfer mode is selected.

3. Comparing

(1) Press the **CMT I/O** menu key.

(2) Press the **COMPARE NC = CMT** menu key.

➔ COMPARE will be displayed on the I/O mode item.

(3) Select the data to be compared with data pre-registered within the NC unit.

- The data input procedure is the same as that of “1. Loading”. Input the data in the (NC) column. (See procedural step (3) of “1. Loading”.)

(4) Press the **START** menu key.

➔ **START** will change into a reverse-display status and data comparing will begin.

- The display color will change to purple at the position where the data being compared is displayed.
- The data that has been compared will change to a normal display status and the same data will be displayed in the (CMT) column.
- If the reverse-display status of **START** is cleared without displaying an alarm message, then this indicates that the entire data you have selected agrees with data pre-registered within the NC unit.
- If disparities exist in the data being compared, an alarm message **511 PROGRAM DATA NOT SAME** will be displayed and the comparison process will be aborted.
  This state indicates that data being displayed in purple does not agree with data pre-registered within the NC unit.

4. Directorying

(1) Press the **CMT I/O** menu key.

(2) Press the **CMT CONTENTS** menu key.

➔ DIR will be displayed on the I/O data item.

(3) Press the **START** menu key.

➔ **START** will change into a reverse-display status, and what type of data is stored on the cassette tape (CMT) or the microdisk will be displayed in the (CMT) column on the display.

- If program data is stored on the external media, the corresponding work number will be displayed on the work number item.
- If non-program data is stored on the external media, the corresponding item will be displayed in a reverse-display status.
- After entire data display is completed, the reverse-display status of **START** will be cleared.

Note: Executing a Directorying operation prior to Loading or Comparing operations enables data selection time to be saved.

Example 1: Do the following to load the entire data that is stored on a cassette tape or a microdisk:

(1) Call the stored data out on the display by carrying out a directorying operation.

(2) Press the **LOAD NC ← CMT** menu key.

(3) Press the **START** menu key.

Example 2: Do the following to load part of the data stored on a cassette tape or a microdisk:

(1) Call the stored data out on the display by carrying out a directorying operation.

(2) Press the **LOAD NC ← CMT** menu key.

(3) Set the cursor on the data not to be loaded and press the data cancellation key. Or set 0 for program not to be loaded.

(4) Press the **START** menu key.

5. Setting of communications parameters

A. Setting of parameters

Parameters for data transfer (baud rate, etc.) can be set on this display by the following procedure:

(1) Press the **PARAM.** menu key.

➔ **PARAM.** will change to a reverse-display status and parameters for data transfer will be presented in a window.

(2) Set the parameters as required.

- Refer to the Parameter List for details of the parameters.

(3) Press the INPUT key or click [OK] in the window.

➔ The reverse-display status of **PARAM.** will be cleared and the window will disappear.

B. Setting of detailed parameters

(1) Press the **PARAM. DETAIL** menu key.

➔ **PARAM. DETAIL** will change to a reverse-display status and parameters for data transfer will be presented in a window.

(2) Set the parameters as required.

- Refer to the Parameter List for details of the parameters.

(3) Press the INPUT key or click [OK] in the window.

➔ The reverse-display status of **PARAM. DETAIL** will be cleared and the window will disappear.

9-1-4 DATA I/O operations (TAPE)

The following six data input/output operations can be executed using a paper tape or a microdisk.

1. Loading
   A program or non-program data of all those prestored on a paper tape or a microdisk is registered in the NC unit.
2. All loading
   All programs and non-program data prestored on a paper tape or a microdisk are registered in the NC unit.
3. Punching
   A program or non-program data of all those pre-registered within the NC unit is stored on a paper tape or a microdisk.
4. All punching
   All programs and non-program data pre-registered within the NC unit are stored on a paper tape or a microdisk.
5. Comparing
   A program or non-program data of all those prestored on a paper tape or a microdisk is compared with a program or non-program data pre-registered within the NC unit.
6. All comparing
   All programs and non-program data prestored on a paper tape or a microdisk are compared with the programs and non-program data pre-registered within the NC unit.

Note: All the data other than EIA/ISO programs are I/O-operated in the three-digit G-code format, the details of which are described in the Programming Manual (MAZATROL).

The operating procedure for the functions listed above is described below. The following description assumes that a paper tape reader/puncher or a MAZAK microdisk unit has already been connected to the NC unit and that paper tape or microdisk setting is also completed.

1. Loading

(1) Press the **TAPE I/O** menu key.

(1) Press the **LOAD NC ← TAPE** menu key.

➔ LOAD will be displayed on the I/O mode item.

(3) Select the data to be transferred.

- Refer to procedural step (3) of “1. Loading” in Subsection 9-1-3.

(4) Press the **START** menu key.

➔ **START** will change into a reverse-display status, and data transfer will begin.

- The display color will change to purple at the position in which the data being transferred is displayed.
- After data transfer, the reverse-display status of **START** will be cleared.

Note: The precautions described in **Notes 1**, **2**, **4** and **5** in "1. Loading" of **9-1-3 DATA I/O operations (CMT)** are valid for this loading operation.

2. All loading

(1) Press the **TAPE I/O** menu key.

(1) Press the **ALL LOAD NC ← TAPE** menu key.

➔ ALL LOAD will be displayed on the I/O mode item.

(3) Press the **START** menu key.

➔ **START** will change into a reverse-display status, and data transfer will begin.

- After data transfer, the reverse-display status of the **START** will be cleared.

3. Punching

(1) Press the **TAPE I/O** menu key.

(2) Press the **PUNCH NC → TAPE** menu key.

➔ PUNCH will be displayed on the I/O mode item.

(3) Select the data to be transferred.

- See the description of procedural step (3) of "1. Loading". Set data in the (NC) column.

(4) Press the **START** menu key.

➔ **START** will change into a reverse-display status, and the tape puncher will start operating.

- The display color will change to purple at the position in which the data being punched out is displayed.
- The name of the data that has been transferred will change to a normal display status and the same data name is displayed in the (TAPE) column.
- After all the selected data have been punched on the paper tape, the reverse-display status of **START** will be cleared.

4. All punching

(1) Press the **TAPE I/O** menu key.

(2) Press the **ALL PUNCH NC →• TAPE** menu key.

➔ ALL PNC will be displayed on the I/O mode item.

(3) Press the **START** menu key.

➔ **START** will change into a reverse-display status, and the tape puncher will start operating.

- After all the pre-registered programs and other data have been punched on the paper tape, the reverse-display status of **START** will be cleared.

5. Comparing

(1) Press the **TAPE I/O** menu key.

(2) Press the **COMPARE NC = TAPE** menu key.

➔ COMPARE will be displayed on the I/O mode item.

(3) Select the data to be compared.

- See the description of procedural step (3) of “1. Loading”.

(4) Press the **START** menu key.

➔ **START** will change into a reverse-display status, and data comparison will begin.

- The display color will change to purple at the position in which the data being compared is displayed.
- The name of the data that has been compared will change to a normal display status and the same data name will be displayed in the (NC) column.
- Clearing of the reverse-display status of **START** without an alarm message indicates that the contents of all the selected data agree with those of the pre-registered data.
- If disparities exist in the data being compared, an alarm message **511 PROGRAM DATA NOT SAME** will be displayed and comparison will be aborted.
  This indicates that the contents of the data displayed in purple do not agree with the contents of the pre-registered counterpart.

6. All comparing

(1) Press the **TAPE I/O** menu key.

(2) Press the **ALLCOMP NC = TAPE** menu key.

➔ ALL CMP will be displayed on the I/O mode item.

(3) Press the **START** menu key.

➔ **START** will change into a reverse-display status, and data comparison will begin.

- Clearing of the reverse-display status of **START** without an alarm message indicates that the contents of all data stored on the paper tape agree with those of the data pre-registered within the NC unit.
- If disparities exist in the data being compared, an alarm message **511 PROGRAM DATA NOT SAME** will be displayed and comparison will be aborted.

7. Program-name tape input/output

When a program is transferred using tape I/O function, the name of the program which is set on the **PROGRAM FILE** display can be assigned to that program during transfer operation.

A. Transfer conditions

1. Input or output of program names is possible only if the program-name tape I/O function has been made valid using bit 2 of parameter **TAP26**.
   See the Parameter List, for parameter details.
2. The following functions presented on the **DATA I/O** display (TAPE) can be used to transfer program names to or from a tape.
   - Loading
   - All loading
   - Punching

- All punching
- Comparing
- All comparing

B. Tape format

The name of the program transferred to or from a tape must be designated as follows:
Set the control-out code "(" after the number with O, then set the program name and the control-in code ")", in that order.

Example: `O00001234 (COMMENT1234)`

•
Program name

C. Notes

1. Only the codes listed in the ASCII codes list can be used as program names.
   If an ASCII code other than 20, 2D, 2E, 30 to 39, or 41 to 5A is entered as the program name, the ASCII code will be converted into a space code (&20) and registered in the internal memory of the NC unit.
2. The maximum number of name characters which can be stored into the NC internal memory during the program-name loading operation is 48. If, therefore, the name to be loaded consists of 49 characters or more, all those excess characters will be ignored.

8. Setting of communications parameters

A. Setting of parameters

Parameters for data transfer (baud rate, etc.) can be set on this display by the following procedure:

(1) Press the **PARAM.** menu key.

➔ **PARAM.** will change to a reverse-display status and parameters for data transfer will be presented in a window.

(2) Set the parameters as required.

- Refer to the Parameter List for details of the parameters.

(3) Press the INPUT key or click OK in the window.

➔ The reverse-display status of **PARAM.** will be cleared and the window will disappear.

B. Setting of detailed parameters

(1) Press the **PARAM. DETAIL** menu key.

➔ **PARAM. DETAIL** will change to a reverse-display status and parameters for data transfer will be presented in a window.

(2) Set the parameters as required.

- Refer to the Parameter List for details of the parameters.

(3) Press the INPUT key or click OK in the window.

➔ The reverse-display status of **PARAM. DETAIL** will be cleared and the window will disappear.

9-1-5 DATA I/O operations (FLOPPY)

The following four data input/output operations can be executed using a floppy disk.

1. Loading
   The program data or other types of data prestored on a floppy disk are registered into the NC unit.
2. Saving
   The program data or other types of data pre-registered in the NC unit are stored onto a floppy disk.
3. Directorying
   The data type prestored on a floppy disk are displayed.
4. Formatting
   A floppy disk is formatted (initialized).

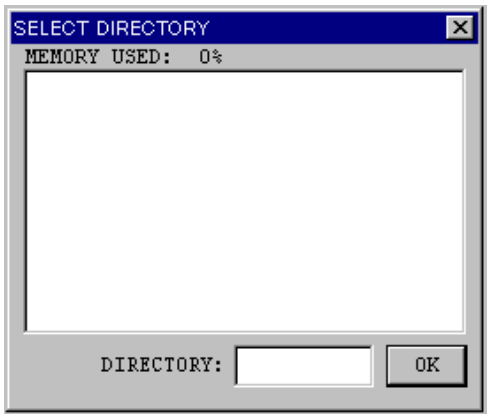
The operating procedure of each operation listed above is described below.
The following description assumes that a floppy disk unit has already been connected to the NC unit and that floppy disk setting is also completed.

1. Selecting the directory

For loading, saving and directorying with a floppy disk, moreover, selection of the required directory must be done beforehand by the following procedure:

(1) Press the **DIR. SELECT** menu key.

➔ **DIR. SELECT** will change to a reverse-display status and the following window will appear.



(2) Key in the name of the required directory and press the INPUT key.

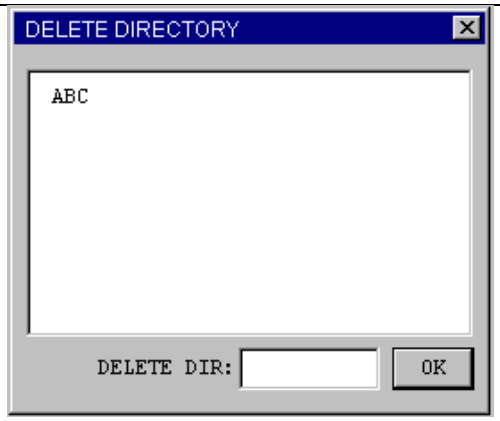
- Use the alphabetic and numeric keys to input the name.

2. Deleting the directory

To delete a directory (with all the files thereunder) on the floppy disk, proceed as follows:

(1) Press the **DIR. DELETE** menu key.

➔ **DIR. DELETE** will change to a reverse-display status and the following window will appear.



(2) Key in the name of the directory to be deleted and press the INPUT key.

- Use the alphabetic and numeric keys to input the name.

Note: All the files stored under the specified directory will be deleted by the above operation.

3. Loading

(1) Press the **FLOPPY** menu key.

(2) Press the **LOAD NC ← FD** menu key.

➔ LOAD will be displayed on the I/O mode item.

(3) Select the data to be transferred.

<For program data transfer>

- Move the cursor to the work number item of (FLOPPY) column and input the work number of the program to be transferred. The work number will change to a reverse-display status after setting the work number.
- Up to 32 programs can be selected.

<For non-program data transfer>

- Move the cursor to the intended data in data type item of (FLOPPY) column and press the INPUT key. The data name will change to a reverse-display status. It indicates that the data is to be transferred.

<For erasing the data that has been set>

- Carry out the following operations to erase the entire data that has been set:
  1) Press the **ALL ERASE** menu key.
     ➔ **ALL ERASE** will change into a reverse-display status.
  2) Press the INPUT key.
     ➔ The reverse-display status of the entire data will be cleared and all the work number will become blank.

(4) Press the **START** menu key.

➔ **START** will change into a reverse-display status and data transfer will begin.

- The display color will change at the position where the data being transferred is displayed.
- The highlighted-display status of the data that has been transferred will be cleared. The same data will be displayed in the (NC) column.
- After data transfer, the reverse-display status of **START** will be cleared.

Note: The precautions described in **Notes 1**, **2**, **4** and **5** in “1. Loading” of **9-1-3 DATA I/O operations (CMT)** are valid for this loading operation.

4. Saving

(1) Press the **FLOPPY** menu key.

(2) Press the **SAVE NC → FD** menu key.

➔ SAVE will be displayed on the I/O mode item.

(3) Select the data to be transferred.

- The data input procedure is the same as that of LOAD. Input the data in the (NC) column. (See procedural step (3) of "3. Loading".)

(4) Press the **START** menu key.

➔ **START** will change to a reverse-display status and data transfer will begin.

- The display color will change at the position where the data being transferred is displayed.
- The data that has been transferred will change to a normal display status and the same data will be displayed in the (FLOPPY) column.
- After data transfer, the reverse-display status of **START** will be cleared.

5. Directorying

(1) Press the **FLOPPY** menu key.

(2) Press the **CONTENTS FD** menu key.

➔ DIR will be displayed on the I/O data item.

(3) Press the **START** menu key.

➔ **START** will change into a reverse-display status, and what type of data is stored on the floppy disk will be displayed in the (FLOPPY) column on the display.

- If program data is stored on the external media, the corresponding work number will be displayed on the work number item.
- If non-program data is stored on the external media, the corresponding item will be displayed in a reverse-display status.
- After entire data display is completed, the reverse-display status of **START** will be cleared.

6. Formatting

(1) Press the **FLOPPY** menu key.

(2) Press the **FORMAT FD** menu key.

➔ FORMAT will be displayed on the I/O mode item.

(3) Key in "–9999" and press the INPUT key.

(4) Press the **START** menu key.

➔ **START** will change into a reverse-display status and initialization will begin.

- After initialization, the reverse-display status of **START** will be cleared.

Note 1: Only double-sided high-density (2HD) floppy disks can be used.

Note 2: Formatting is executed at 1.44 MB.

9-1-6 DATA I/O operations (DNC)

The following two data input/output operations can be executed using a host CPU or other NC.

1. Loading
   Non-program data or programs pre-registered within a host CPU or the NC unit of other

machines are transferred to the NC unit of this machine.

2. Saving
   Non-program data or programs pre-registered within the NC unit of this machine are transferred to a host CPU or the NC unit of other machines.

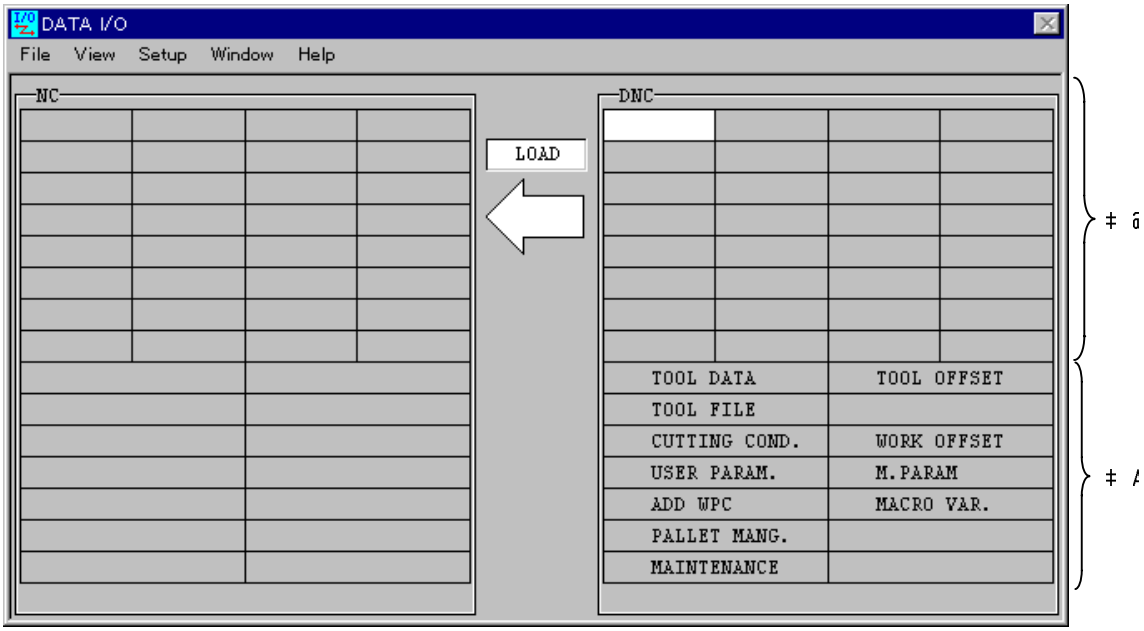
The operating procedure for the operations listed above is described below. The following description assumes that the NC unit of this machine has already been connected to the NC unit of other machines or a host CPU.

1. Loading

(1) Press the **DNC I/O** menu key.

(2) Press the **LOAD NC ← HOST** menu key.

➔ LOAD will be displayed on the I/O mode item.



D735S1063E

(3) Select the data to be transferred.

<For program data transfer>

- Move the cursor to the work number (•) of the (DNC) column and input the work number of the program to be transferred. The work number will change to a reverse-display status after setting the work number.
- Up to 32 programs can be selected.

<For non-program data transfer>

- Move the cursor to the intended data in data type item (•) of the (DNC) column and set the file number to be transferred. The data will change to a reverse-display status after setting data.

<For erasing the set data>

➔ For erasing procedure, see the step (3) in “1. Loading” of Subsection **9-1-3 DATA I/O operations (CMT)**.

(4) Press the **START** menu key.

➔ **START** will change into a reverse-display status, and data transfer will begin.

- The display color will change to purple at the position in which the data being transferred is

displayed.

- The data that has been transferred will change to a normal display status and the same data will be displayed in the (NC) column.
- After data transfer, the reverse-display status of **START** will be cleared.

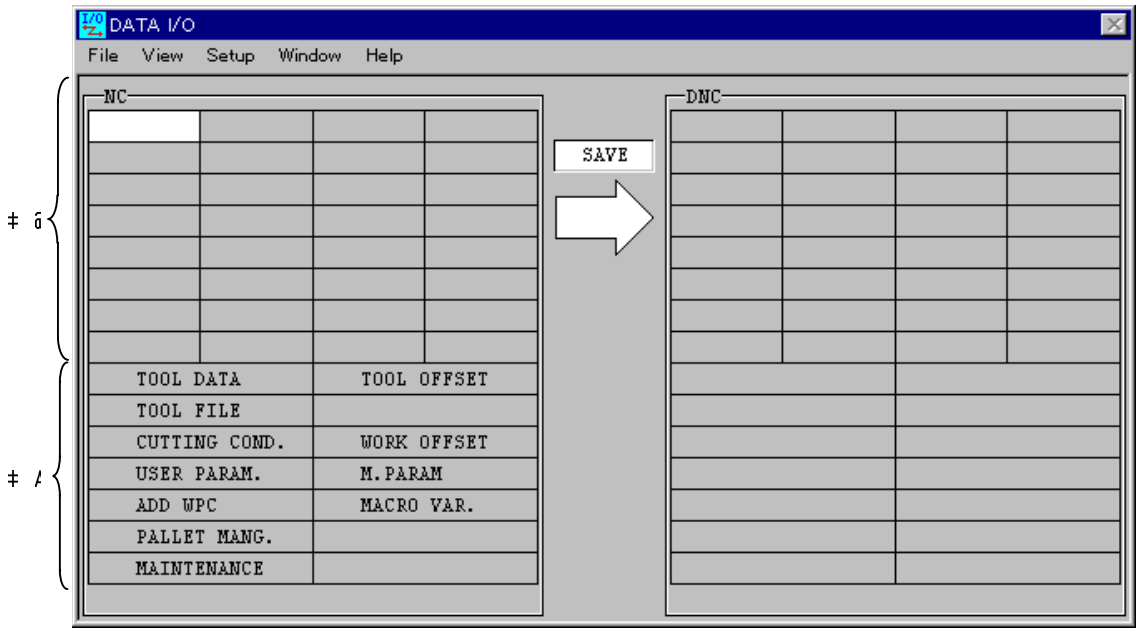
Note: All the precautions in **Notes** in "1. Loading" of Subsection **9-1-3 DATA I/O operations (CMT)** are valid for this loading operation.

2. Saving

(1) Press the **DNC I/O** menu key.

(2) Press the **SAVE NC → HOST** menu key.

➔ SAVE will be displayed on the I/O mode item, and the display will change as follows:



D735S1064E

(3) Select the data to be transferred.

- The data selection procedure is the same as that of the loading operation. Set data in the (NC) column. For further details, see the step (3) in "1. Loading".

(4) Press the **START** menu key.

➔ **START** will change into a reverse-display status, and data transfer will begin.

- The display color will change to purple at the position in which the data being transferred is displayed.
- The data that has been transferred will change to a normal display status.
  The same data will be displayed in the (DNC) column.
- After data transfer, the reverse-display status of **START** will be cleared.

3. Setting of communications parameters

A. Setting of parameters

Parameters for data transfer (baud rate, etc.) can be set on this display by the following procedure:

(1) Press the **PARAM.** menu key.

➔ **PARAM.** will change to a reverse-display status and parameters for data transfer will be

presented in a window.

(2) Set the parameters as required.

- Refer to the Parameter List for details of the parameters.

(3) Press the INPUT key or click [OK] in the window.

The reverse-display status of **PARAM.** will be cleared and the window will disappear.

B. Setting of detailed parameters

(1) Press the **PARAM. DETAIL** menu key.

➔ **PARAM. DETAIL** will change to a reverse-display status and parameters for data transfer will be presented in a window.

(2) Set the parameters as required.

- Refer to the Parameter List for details of the parameters.

(3) Press the INPUT key or click [OK] in the window.

➔ The reverse-display status of **PARAM. DETAIL** will be cleared and the window will disappear.

9-1-7 DATA I/O operations (HARD DISK)

The following three data input/output operations can be executed using the hard disk.

1. Loading
   The program data or other types of data prestored on the hard disk are registered into the NC unit.
2. Saving
   The program data or other types of data pre-registered in the NC unit are stored onto the hard disk.
3. Directorying
   The data type prestored on the hard disk are displayed.

Note: Data communication with the HDD OPERATION PROGRAM area cannot be achieved by the HARD DISK function of the **DATA I/O** display.

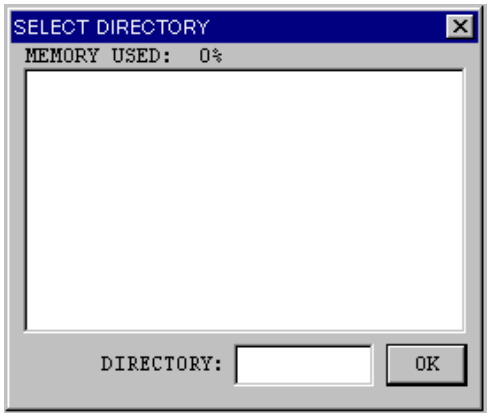
The operating procedure of each operation listed above is described below.

1. Selecting the directory

For loading, saving and directorying with the hard disk, selection of the required directory must be done beforehand by the following procedure:

(1) Press the **DIR. SELECT** menu key.

➔ **DIR. SELECT** will change to a reverse-display status and the following window will appear.



(2) Key in the name of the required directory and press the INPUT key.

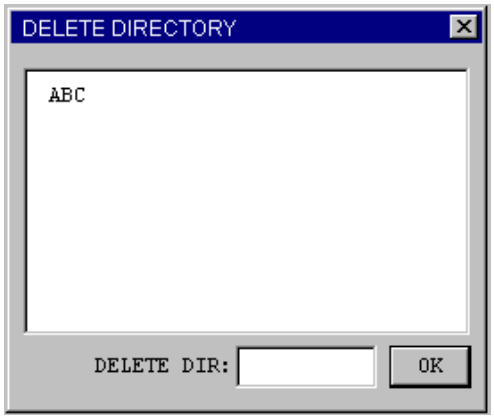
- Use the alphabetic and numeric keys to input the name.

2. Deleting the directory

To delete a directory (with all the files thereunder) on the hard disk, proceed as follows:

(1) Press the **DIR. DELETE** menu key.

➔ **DIR. DELETE** will change to a reverse-display status and the following window will appear.



(2) Key in the name of the directory to be deleted and press the INPUT key.

- Use the alphabetic and numeric keys to input the name.

Note: All the files stored under the specified directory will be deleted by the above operation.

3. Loading

(1) Press the **HARD DISK** menu key.

(2) Press the **NC ← HD LOAD** menu key.

➔ LOAD will be displayed on the I/O mode item.

(3) Select the data to be transferred.

<For program data transfer>

- Move the cursor to the work number item of (HARD DISK) column and input the work number of the program to be transferred. The work number will change to a reverse-display status after setting the work number.
- Up to 32 programs can be selected.

<For non-program data transfer>

- Move the cursor to the intended data in data type item of (HARD DISK) column and press the INPUT key. The data name will change to a reverse-display status. It indicates that the data is to be transferred.

<For erasing the data that has been set>

- Carry out the following operations to erase the entire data that has been set:

1) Press the **ALL ERASE** menu key.

➔ **ALL ERASE** will change into a reverse-display status.

2) Press the INPUT key.

➔ The reverse-display status of the entire data will be cleared and all the work number will become blank.

(4) Press the **START** menu key.

➔ **START** will change into a reverse-display status and data transfer will begin.

- The display color will change at the position where the data being transferred is displayed.
- The highlighted-display status of the data that has been transferred will be cleared. The same data will be displayed in the (NC) column.
- After data transfer, the reverse-display status of **START** will be cleared.

Note: The precautions described in **Notes 1**, **2**, **4** and **5** in "1. Loading" of **9-1-3 DATA I/O operations (CMT)** are valid for this loading operation.

4. Saving

(1) Press the **HARD DISK** menu key.

(2) Press the **NC → HD SAVE** menu key.

➔ SAVE will be displayed on the I/O mode item.

(3) Select the data to be transferred.

- The data input procedure is the same as that of LOAD. Input the data in the (NC) column. (See procedural step (3) of "3. Loading".)

(4) Press the **START** menu key.

➔ **START** will change to a reverse-display status and data transfer will begin.

- The display color will change at the position where the data being transferred is displayed.
- The data that has been transferred will change to a normal display status and the same data will be displayed in the (HARD DISK) column.
- After data transfer, the reverse-display status of **START** will be cleared.

5. Directorying

(1) Press the **HARD DISK** menu key.

(2) Press the **HD CONTENT** menu key.

➔ DIR will be displayed on the I/O data item.

(3) Press the **START** menu key.

➔ **START** will change into a reverse-display status, and what type of data is stored on the hard disk will be displayed in the (HARD DISK) column on the display.

- If program data is stored on the external media, the corresponding work number will be displayed on the work number item.
- If non-program data is stored on the external media, the corresponding item will be displayed in a reverse-display status.
- After entire data display is completed, the reverse-display status of **START** will be cleared.

9-2 Program Transfer from the Host PC to the Memory Card

9-2-1 Outline

This section explains how to transfer machining programs, under an Ethernet-based network environment, from the host PC to the memory card inserted into the front slot on the M640 operating panel.

Use of the method explained below eliminates the need for the host PC to have any memory card handling equipment, and only the related data input/output functions of the M640 are needed to write program files onto the memory card.

The examples shown in this section assume the following environment where the host PC and the M640 are already connected to the network:

- Host PC .....A personal computer having installed Microsoft Windows95/98 or Microsoft Windows NT4.0,
- Network connection.....Ethernet connection based on Windows functions,
- Network protocol .....Microsoft network and Microsoft network sharing services.

Note: The host PC can use an operating system (OS) other than the above or the host PC itself may be a workstation if the program files can surely be transferred to the folder within the hard disk (HD) of the M640. See the OS-accompanying manuals for detailed information about transferring program files.

The procedure for transferring programs is outlined below.

(1) Modifying the HD-operating program storage folder of the M640 into a sharable folder.

(2) Copying program files from the host PC into the HD-operating program storage folder of the M640.

(3) Selecting the **DATA I/O** display on the M640 and copying program files from the HD into the memory card.

9-2-2 Detailed description

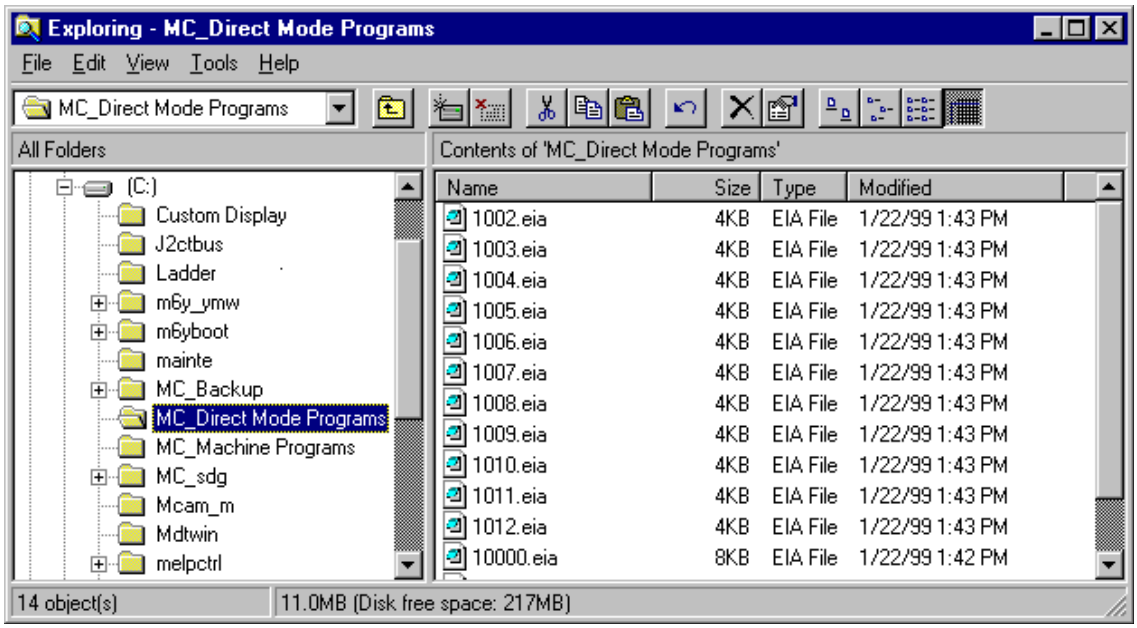
1. Modifying the HD-operating program storage folder into a sharable folder

The HD-operating program storage folder within the HD of the M640 can be made readable and writable from another Windows personal computer present on the network by modifying the attributes of that folder. This task is called folder sharing, and the corresponding folder is referred to as a sharable folder.

The procedure is as follows:

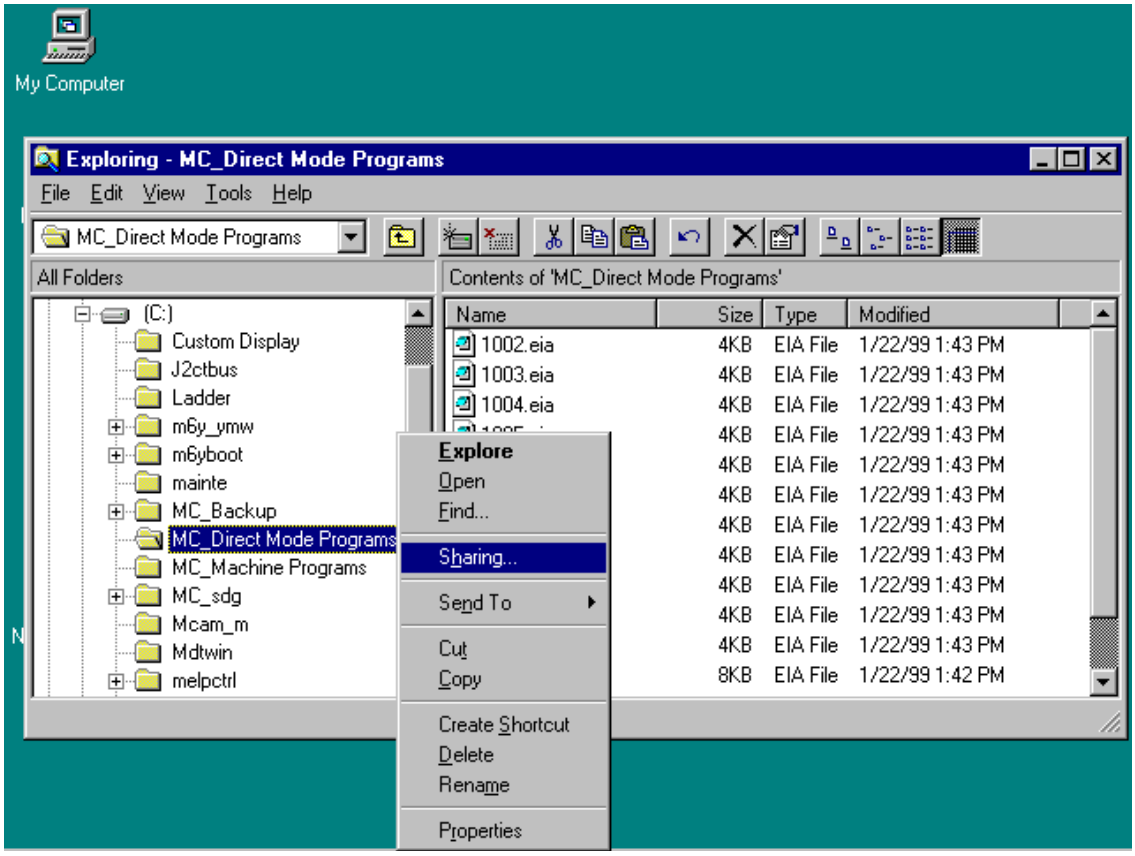
(1) Start the Windows Explorer.

(2) Select the HD-operating program storage folder by clicking “MC_Direct Mode Programs”.



D735S1107E

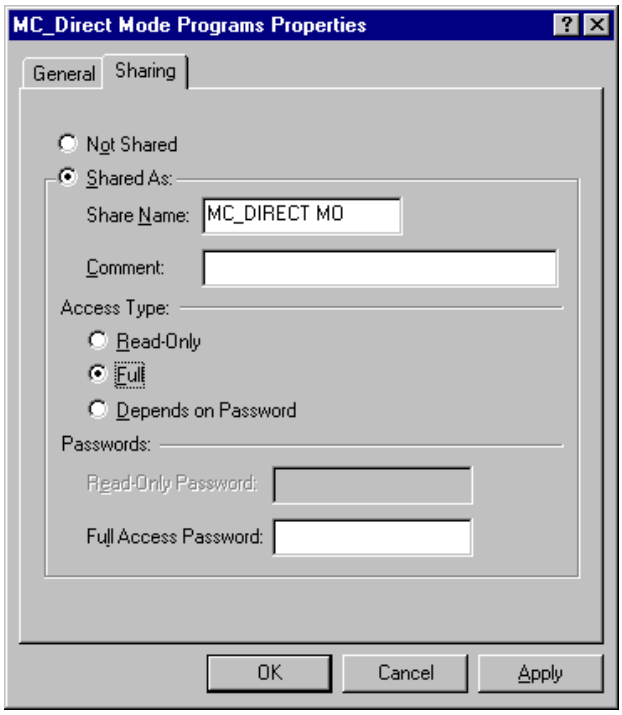
(3) Click the right-hand button of the mouse on “MC_Direct Mode Programs” to display the pop-up menu.



D735S1108E

(4) By clicking the left button of the mouse, select “Sharing ...” from the pop-up menu to display the Properties dialogue box for the sharing.

(5) Set data as follows in the Sharing Properties dialogue box:

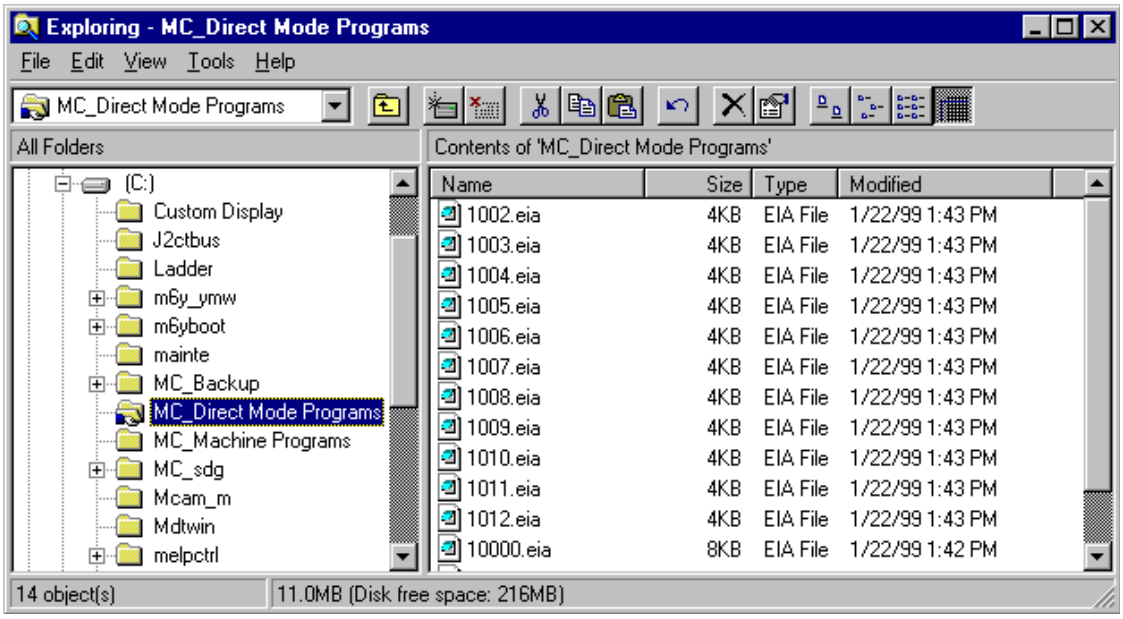


D735S1109E

- Sharing.....Select “Shared As”.
- Share Name .....Leave the automatically set name (MC_DIRECT MO) as it is.
- Access Type.....Select “Full”.
- Passwords .....Skip setting.

Note: Although any share name can be entered, it is recommended for users unfamiliar with Windows to use the automatically set name.
It is likewise recommended for users unfamiliar with Windows not to set any password.

(6) Close the Sharing Properties dialogue box by clicking the left button of the mouse on the “OK” button to complete the sharing process for the folder.



D735S1110E

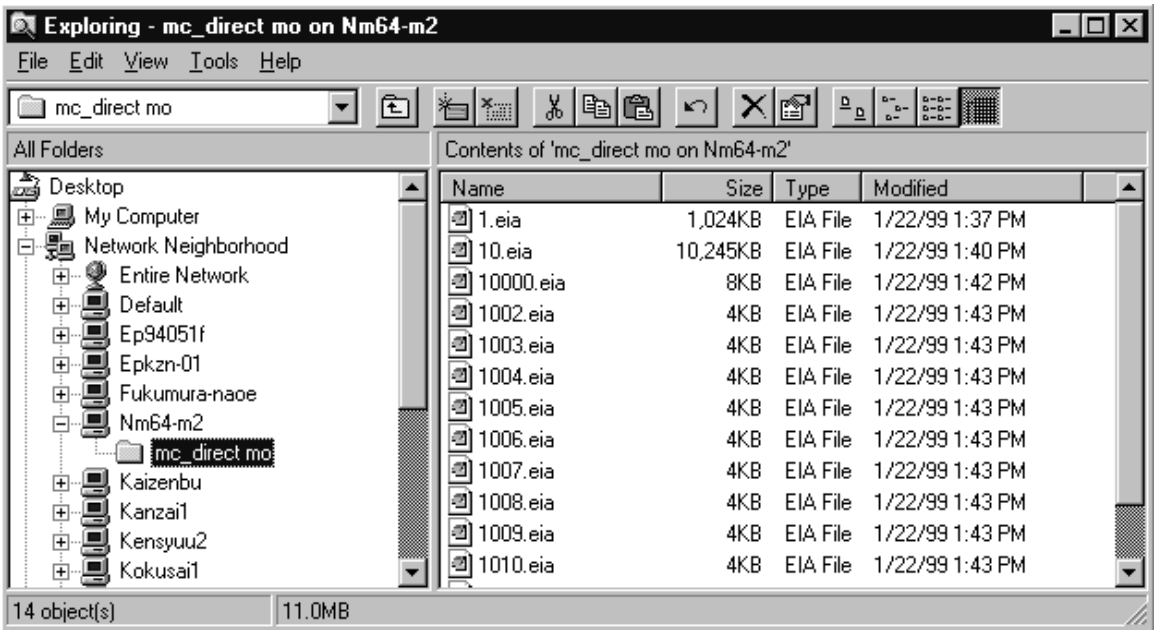
When sharing succeeds, the icon for the HD-operating program storage folder will change as shown above.

2. Copying program files from the host PC into the HD-operating program storage folder

The Windows Explorer allows program files within the host PC to be copied into the HD-operating program storage folder of the M640.

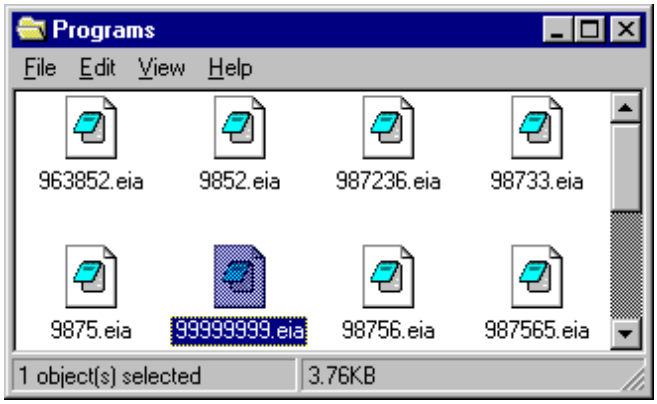
This section introduces the procedure for accessing from a Windows95-containing host PC the M640 HD-operating program storage folder that was modified into a sharable folder as described in Section 1 above and then copying an existing program file from any folder within the host PC.

(1) Activate the Windows Explorer of the host PC to display the sharable HD-operating program storage folder of the M640 located on the network. The share name mentioned in Section 1 above will be displayed as “mc_direct mo”. (The capitalized names of the sharable folders on the network will usually be displayed in lower-case letters.)

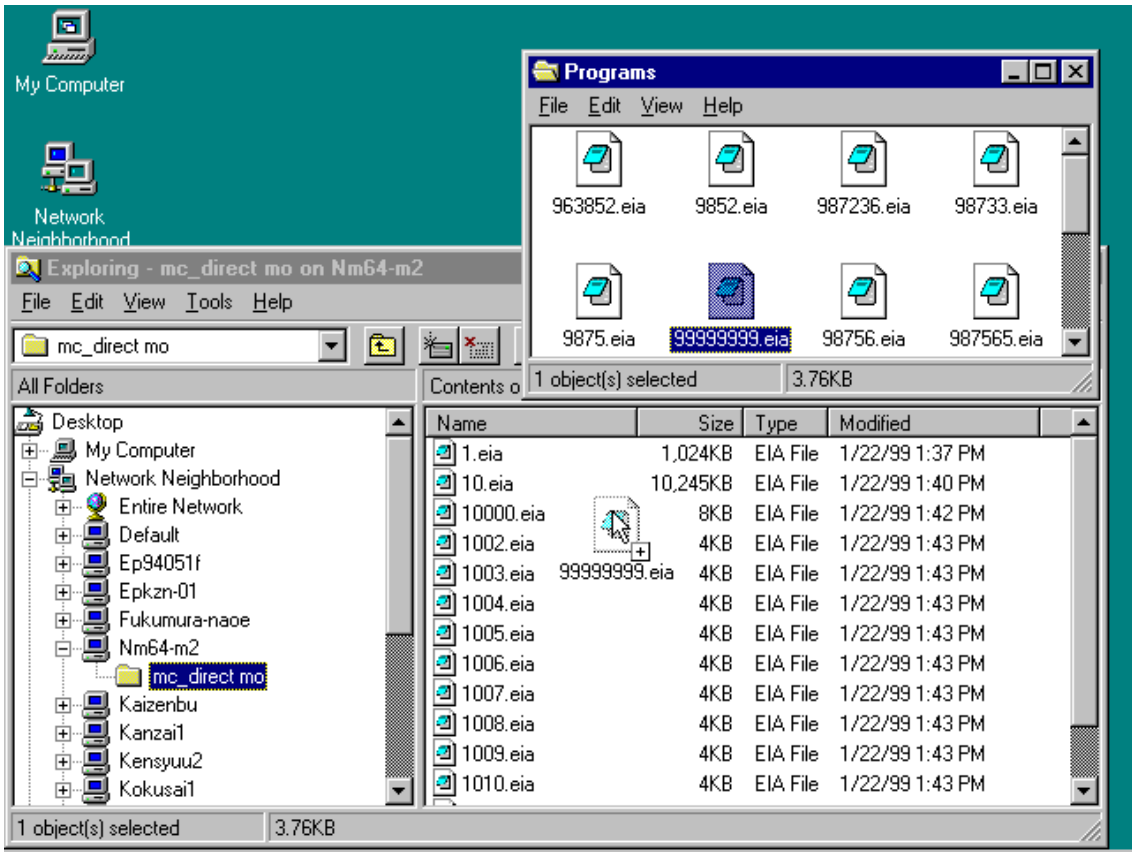


D735S1111E

(2) Select “My Computer” or the Windows Explorer and display the contents of the desired folder within the HD of the host PC. (The example below shows the results of processing from the selection of “My Computer” to the display of the contents of “Programs” folder.)



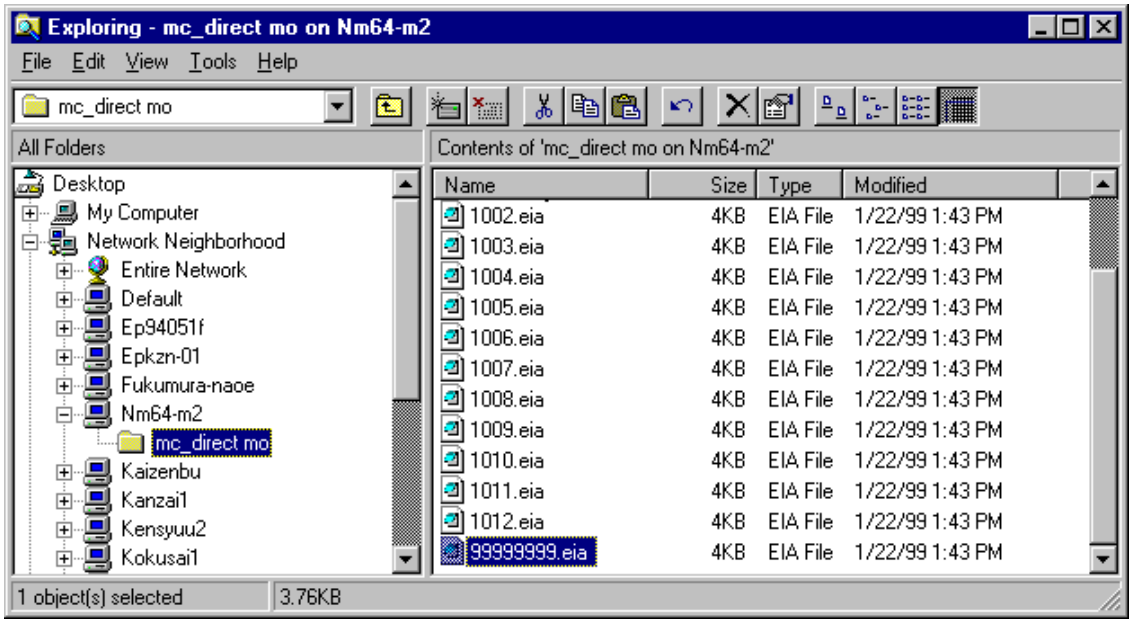
(3) Position the mouse pointer on the desired program in the displayed folder, press and hold down the left button of the mouse, move the pointer into the Contents area of the HD-operating program storage folder, as displayed in Step (1) above, on the Windows Explorer, and release the mouse button. The selected program file within the host PC will then be copied into the HD-operating program storage folder of the M640. This operation is referred to as the drag-and-drop method.



D735S1112E

Note 1: The copying in Step (3) above will come to a halt in case the folder of destination contains a program file having the same name as that of the selected file. The system will then display a window to ask whether the existing file is to be overridden with the selected one.

Note 2: Before copying a large-size program file, make sure that the HD of the M640 has a sufficient available area. Always keep the largest possible area free by deleting unnecessary programs from the HD-operating program storage folder on the **PROGRAM FILE** display. Refer to Section 6-6, PROGRAM FILE Display, for the deleting procedure. Use the Help function of the Windows95 system to check the total residual capacity of the M640-mounted HD.



D735S1113E

3. Copying program files from the HD-operating program storage folder onto the memory card

The procedure is as follows:

(1) Press the **DATA I/O** menu key.

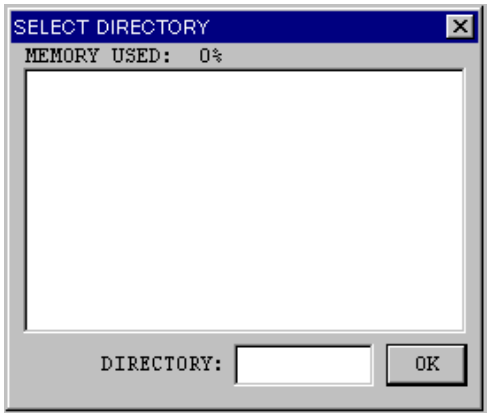
(2) ress the **CARD** menu key.

(3) Press the **HD AREA** menu key.

➔ **HD AREA** will change to a reverse-display status to indicate that data input/output operations can be performed on the HD-operating program storage folder.

(4) Press the **DIR. SELECT** menu key.

➔ **DIR. SELECT** will change to a reverse-display status and the following window will be opened:



(5) Key in the name of the desired directory and then press the INPUT key.

- Use the alphanumeric keys to enter the directory name.

Note: Specify “MCICPROG” as the directory name if programs for the IC memory card operation (optional function) are to be transferred. The IC memory card operation can only refer to the programs stored under that directory name.
Refer to Section 11-6 for further details of the IC memory card operation.

(6) Press the **NC → CARD SAVE** menu key.

(7) Specify the program(s) to be transferred.

(8) Press the **START** menu key.

➔ **START** will change to a reverse-display status to indicate that transfer of the program has been started.

- The reverse-display status of **START** will be cleared upon completion of the program transfer.

Note: Before copying a program onto the IC memory card, make sure that the IC memory card has a sufficient available area. Take great care when copying a program of a large capacity, in particular. If the size of the desired program file within the HD-operating program storage folder is larger than the capacity of the IC memory card to be used, it is not possible to transfer that program file.

- NOTE -

10 DISPLAYS RELATED TO DIAGNOSIS

The NC unit provides the following diagnostic displays and windows:

1. **DIAGNOSIS (USER)** display [ALARM, ALARM HISTORY, PLC SIGNAL]
2. **MAINTENANCE CHECK** display
3. **PLC** display
4. **RUNNING CONTROL** display
5. **DIAGNOSIS (MAKER)** display [VERSION, DIAGNOSIS MONITOR, OPTION, SERVO MONITOR*, SPINDLE MONITOR*, HARDWARE MONITOR*, LADDER MONITOR*]
6. DATA ERASE window
7. ACCUMULATED TIME window

Description of displays marked with an asterisk (*) is omitted because these displays are reserved for YAMAZAKI MAZAK servicemen.

10-1 DIAGNOSIS (USER) – ALARM Display

Function:

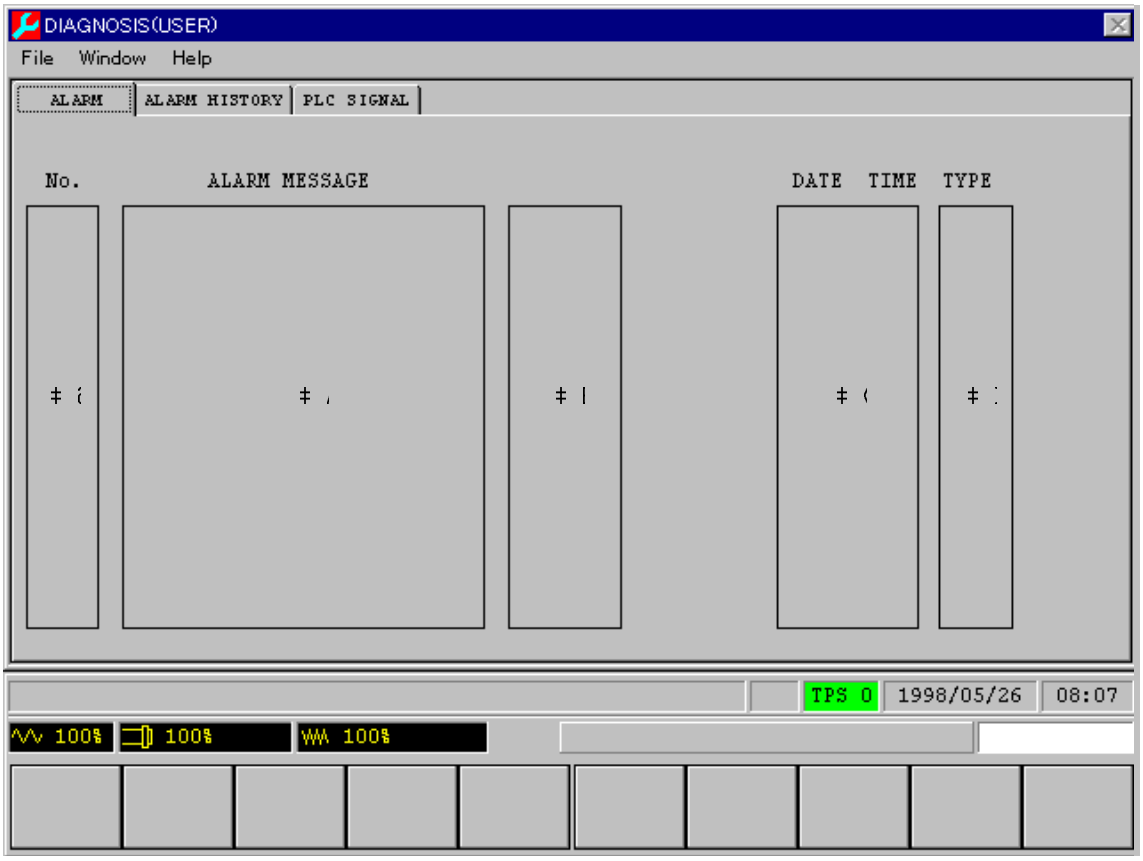
- Displaying the number, details, etc. of alarms currently occuring

Remarks:

- Even if more than one alarm occurred, only one of the alarm messages will appear on other displays. Selection of this display, however, gives you further detailed information about all alarms whose causes have not been removed.
- Refer to the Alarm List for further details of alarms and their resetting methods.

10-1-1 Data display

1. Displayed data



D735S1065E

2. Display data description

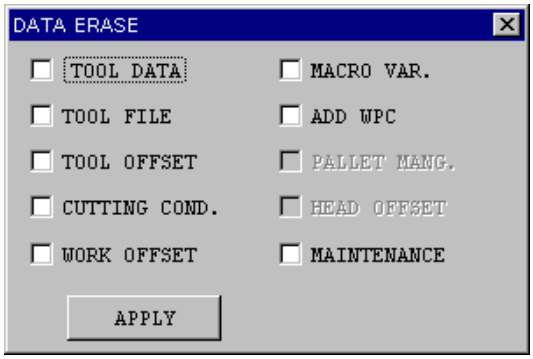
No.	Item	Unit	Data description
•	No.	•	Alarm number
•	ALARM MESSAGE	•	Alarm contents
•		•	Section where the alarm occurred (**a**, **b**, **c**) **a**: Work number **b**: Unit number **c**: Sequence number **Note:** The data may not be displayed for certain types of alarms.
•	DATE TIME	M.D H:M	Date and time when the alarm occurred
•	TYPE	•	Source of the occurrence of the alarm

3. DATA ERASE window

This window is used to erase the entire data of, or to initialize, the particular display.

(1) Press the **DATA ERASE** menu key.

➔ The display of the menu item will be reversed and the DATA ERASE window will appear.



(2) Select the display(s) to be initialized.

- Using the cursor keys, move the dotted frame onto the name of the display to be initialized and press the INPUT key to put a check-off mark ☑ on the left-side box.
- To cancel the selection, press the CANCEL key with the required display name put in the frame to clear the left-side box of the check-off mark.

(3) Using the cursor keys, set the dotted frame on “APPLY” and press the INPUT key.

(4) Key in ″ –9999″ and press the INPUT key.

➔ The selected display(s) will be initialized.

4. ACCUMULATED TIME window

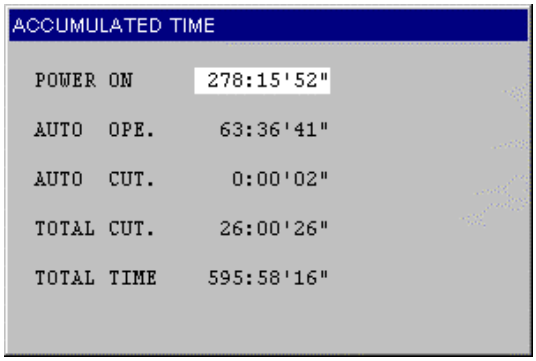
A. Setting various cumulative time data

The four types of cumulative time data listed below can be set on the **DIAGNOSIS** display.

1. POWER ON
2. AUTO OPE.
3. AUTO CUT.
4. TOTAL CUT.

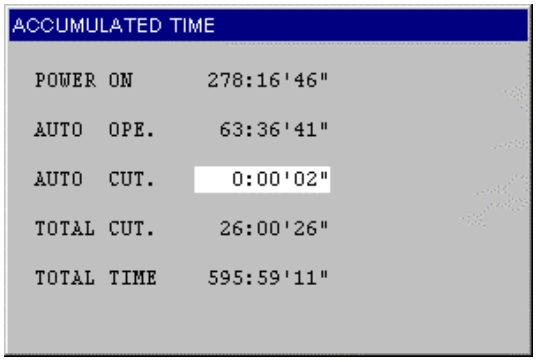
(1) Select the “Accumulated Time” option through “Window” on the menu bar.

➔ The ACCUMULATED TIME window will appear with the cursor placed on the POWER ON item.



(2) Using the cursor key, move the cursor to the line of the desired item.

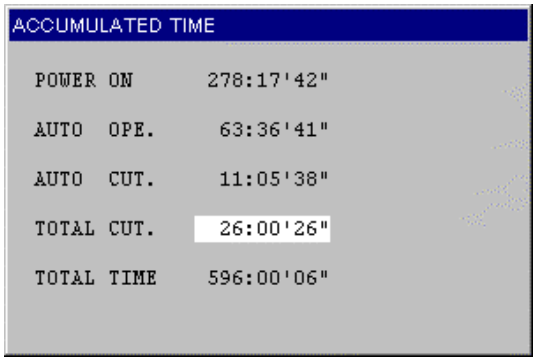
Example: If the cumulative time of automatic cutting is to be set



(3) Specify the time, and press the INPUT key.

Example: To set 11: 05’ 38’’, press keys in the following order:

[1] [1] [•] [5] [•] [3] [8] [→]



(4) Repeat steps (2) and (3) above for all other items for which cumulative time data is to be set.

(5) Select the “Accumulated Time” option anew through “Window” on the menu bar.

➔ The ACCUMULATED TIME window will then disappear.

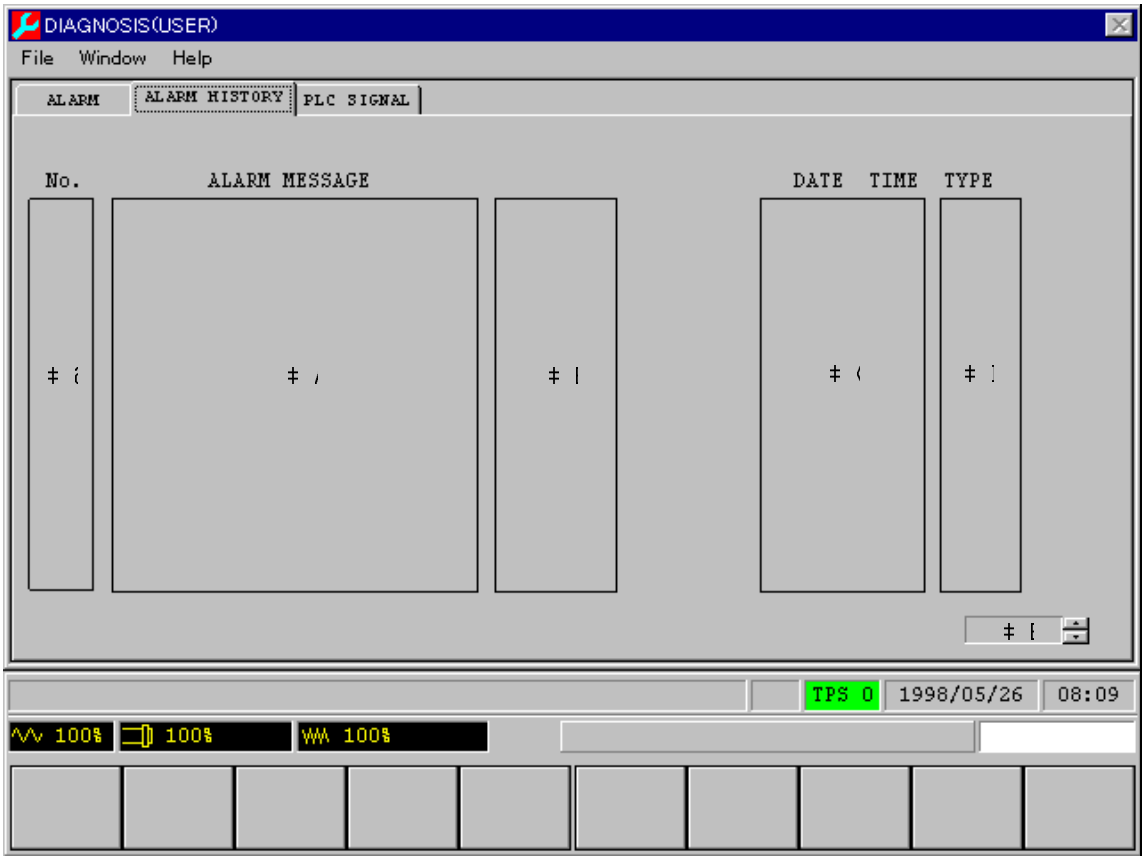
10-2 DIAGNOSIS (USER) – ALARM HISTORY Display

Function:

Displaying the details, the places of occurrence, etc. of all alarms that have occurred up to now, in order of the latest alarm first.

10-2-1 Data display

1. Displayed data



D735S1066E

2. Display data description

No.	Item	Unit	Data description
•	No.	•	Alarm number
•	ALARM MESSAGE	•	Contents of alarm
•		•	Section where the alarm occured **(a , b , c)** **a**: Work number **b**: Unit number **c**: Sequence number **Note:** May not be displayed for the particular alarm.
•	DATE TIME	M.D H:M	Date and time when the alarm occurred
•	TYPE	•	Source of occurrence of the alarm
•	Page	•	Current page and the total number of pages **a / b** **a**: Current page **b**: Total number of pages

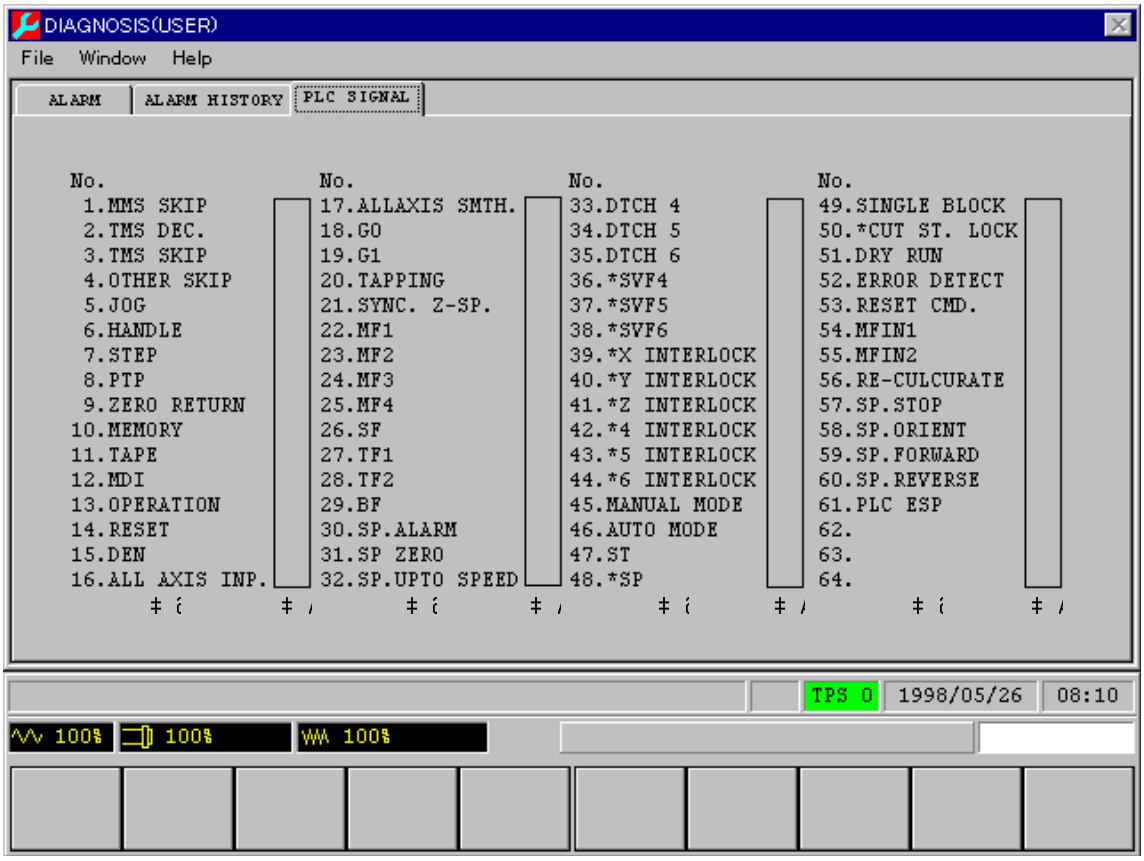
10-3 DIAGNOSIS (USER) – PLC SIGNAL Display

Function:

Displaying ON/OFF status of machine signals

10-1-1 Data display

1. Displayed data



D735S1067E

2. Display data description

No.	Item	Unit	Data description
•	Signal name	•	Singal name
•	0/1	•	ON/OFF status of signals 0: OFF 1: ON

10-4 MAINTENANCE CHECK Display

Function:

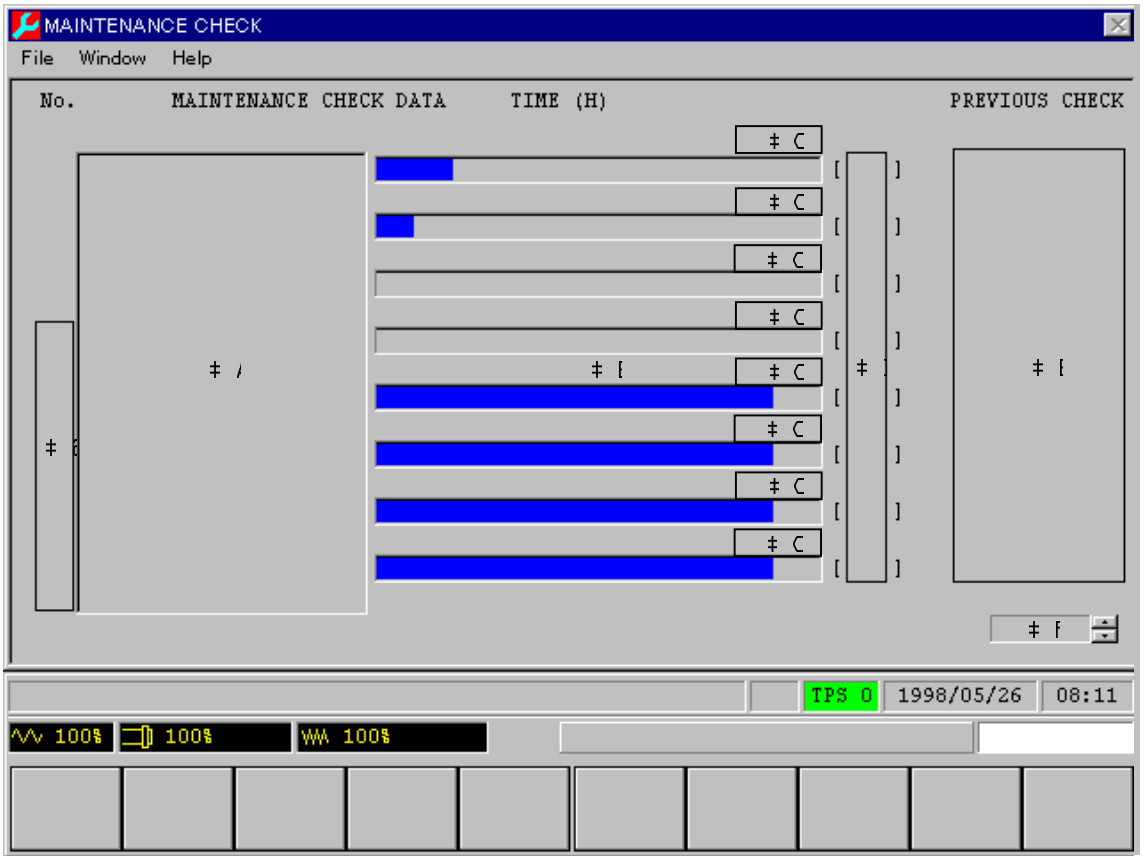
- Periodic check items, the last date of checking, and the time that has elapsed since the last date of checking are displayed.
- Periodic check items can be registered.

Remarks:

The machine will not stop even if the display of the time which has elapsed since the last date of checking exceeds the target time.

10-1-1 Data display

1. Displayed data



D735S1068E

2. Display data description

No.	Item	Unit	Data description
•	No.	•	Check number
•	MAINTENANCE CHECK DATA	•	Name of a periodic check item
•	TIME	%	Graphical representation of the rate of the current time to the target time
•	Target time	Hour	Target time for the periodic check item
•	Current time	Hour	Current time for the periodic check item
•	PREVIOUS CHECK	'Y. M. D	The last date of periodic check item
•	Page	•	Page number

3. Registering the data for regular check items

A. Presetting the target time or the current time

(1) Move the cursor to the data item where data is to be preset.

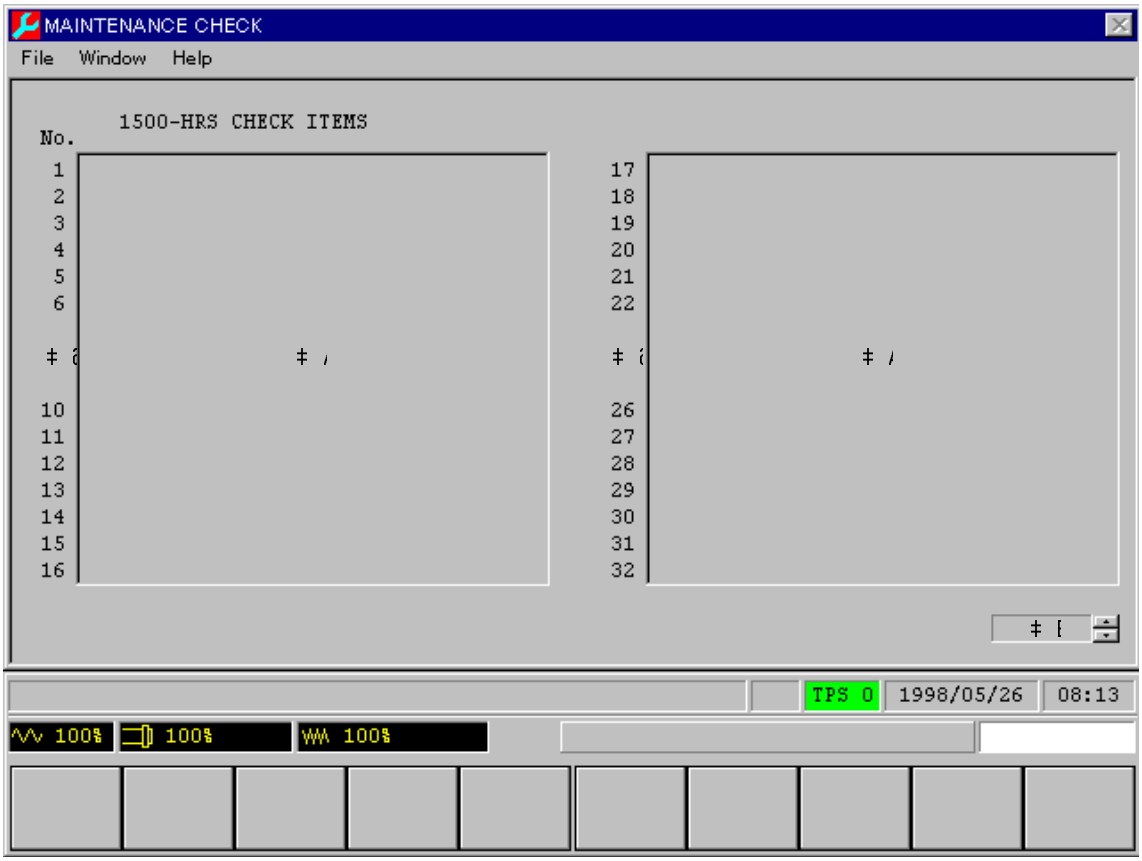
(2) Enter time.

Example: Specify “96” and press the INPUT key to input 96.

Note: Entry of the current time updates the data of data item • (PREVIOUS CHECK).

10-1-2 Data display for long-term check items (1500 and 3000 hours check items)

1. Displayed data



D735S1069E

2. Display data description

No.	Item	Unit	Data description
•	No .	•	Check number
•	Check items	•	Long-term check item
•	Page	•	Page number

10-1-3 Display operations

The display status of data item will reverse when the current time reaches the target time. After completion of checking, move the cursor to “No.” using the cursor key and then press the INPUT key. The display of data item will then return from the reverse status to the normal status.

10-1-4 Entry of the check items to be displayed

The check items to be displayed cannot be entered directly on the **MAINTENANCE CHECK** display. This subsection describes the procedure for adding or deleting the check items to be displayed.

1) Save the factory-set data of display items beforehand into the file.
2) Execute the File Write function to create a file under the specified directory (see Item 2 below).
3) Edit the text file as required (see Item 1 below).
4) Execute the File Read function to fetch the data onto the display (see Item 2 below).

1. Registering the data of the check items

Using a commercially available editor, open the text file of the following format under the specified directory (\nm64mdata\eng\maintenance.txt).

```
[urcom];  .......... Header of periodic check items
;   ............... Comment line
1,CHECK GUIDE WAY LUB-OIL
 M
24,
[mei15];  .......... Header of 1500-hrs check items
1,CHANGE HYDRAULIC UNIT OIL
 M
32,
[mei30];  .......... Header of 3000-hrs check items
1,CHECK MACHINE LEVEL
 M
32,
```

Note: "eng" in the directory name corresponds with the language currently selected for the display mode.

A. Data format of the periodic check items

```
[urcom] Periodic check items
1, CHECK GUIDE WAY LUB-
OIL
 M
24,
```

- Up to 24 items can be registered.
- Set the description after the serial number and comma.

Note: The description can consist of up to 40 characters (a two-byte code is counted as two characters). Excess items or characters will be ignored.

B. Data format of the long-term check items

```
[mei15] 1500-hrs check items
1, CHANGE HYDRAULIC
UNIT OIL
 M
32,

[mei30] 3000-hrs check items
1, CHECK MACHINE LEVEL
 M
32,
```

- Up to 32 items can be registered.
- Set the description after the serial number and comma.

Note: The description can consist of up to 40 characters (a two-byte code is counted as two

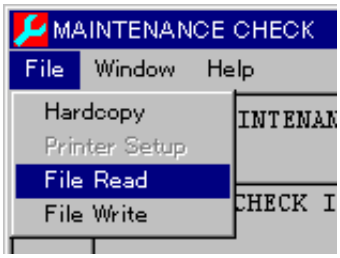
characters). Excess items or characters will be ignored.

2. Read and Write of check items' text file

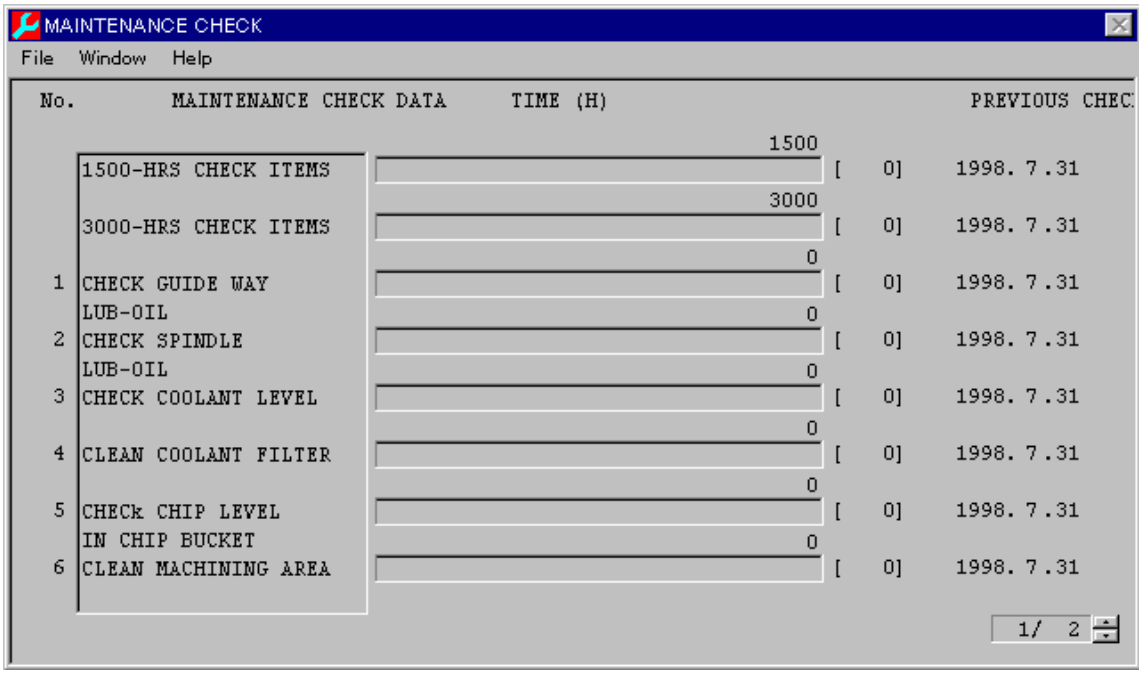
All the check items presented on the **MAINTENANCE CHECK** display can be read in from, and written into, the specified text file. Use a commercially available editor to prepare a text file containing the desired check items to be read in and presented on the display.

A. File Read

Using a commercially available editor, prepare a text file (containing the desired check items) and copy it into the specified directory.



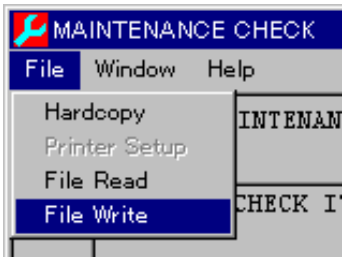
Select the option "File Read" through the item "File" on the menu bar to start data loading. The contents thus read in will be displayed as shown below upon completion of loading.



D735S1084E

B. File Write

Select the option "File Write" through the item "File" on the menu bar to save the display data. A text file containing the current display data will be created under the specified directory.



10-5 PLC Display

Function:

Displaying informations provided for the individual machine.

10-5-1 Data display

1. Displayed data

D735S1070E

Note: The data on this display differs from machine to machine. Refer to the Manuals for the machine.

10-6 RUNNING CONTROL Display

Function:

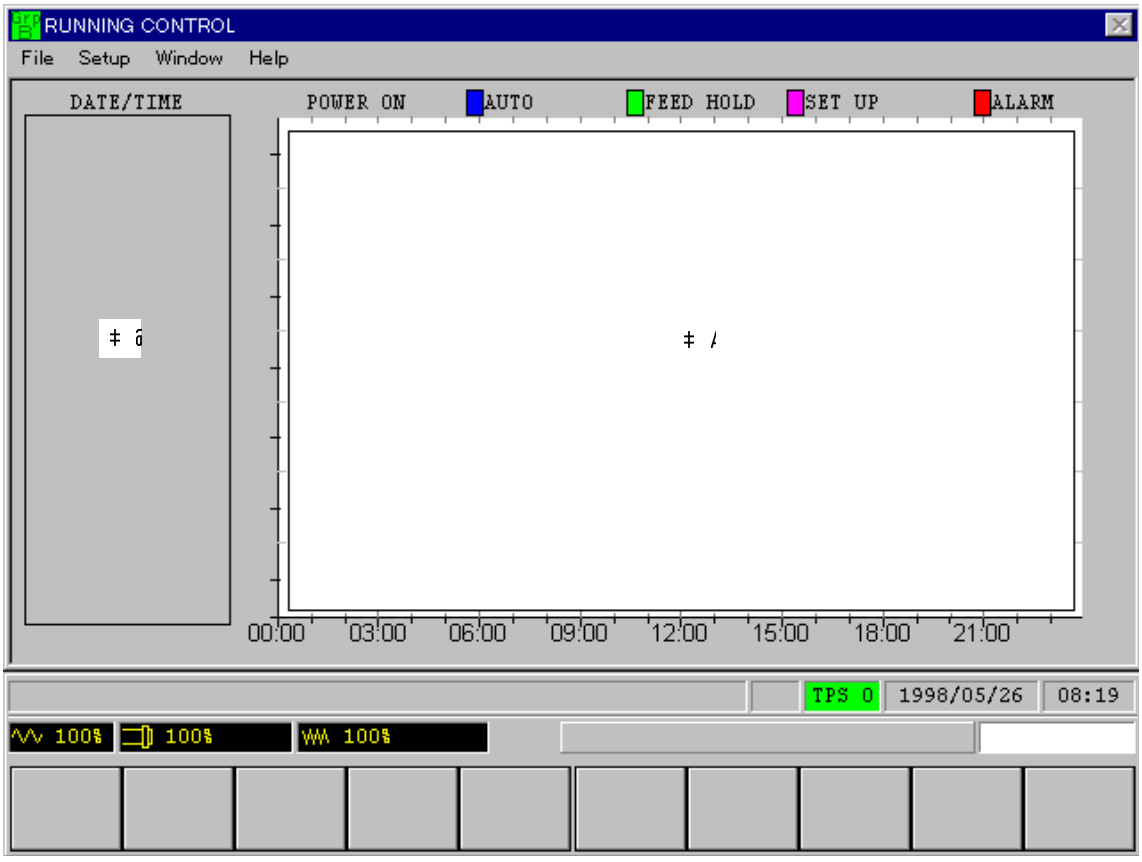
Displaying the machine operational status.

Remarks:

- The machine operational status for the last seven days (one week) is displayed.
- The operational status during power-on time is displayed in distinction between the following four divisions:
  1. Running in automatic operation
  2. Stopping in automatic operation
  3. Setting-up in automatic operation
  4. Stopping due to an alarm
- Program numbers and machining results (numbers of finished workpieces) on the selected date can be displayed.
- Change in speed and load (%) of the spindle on the selected date can be displayed.

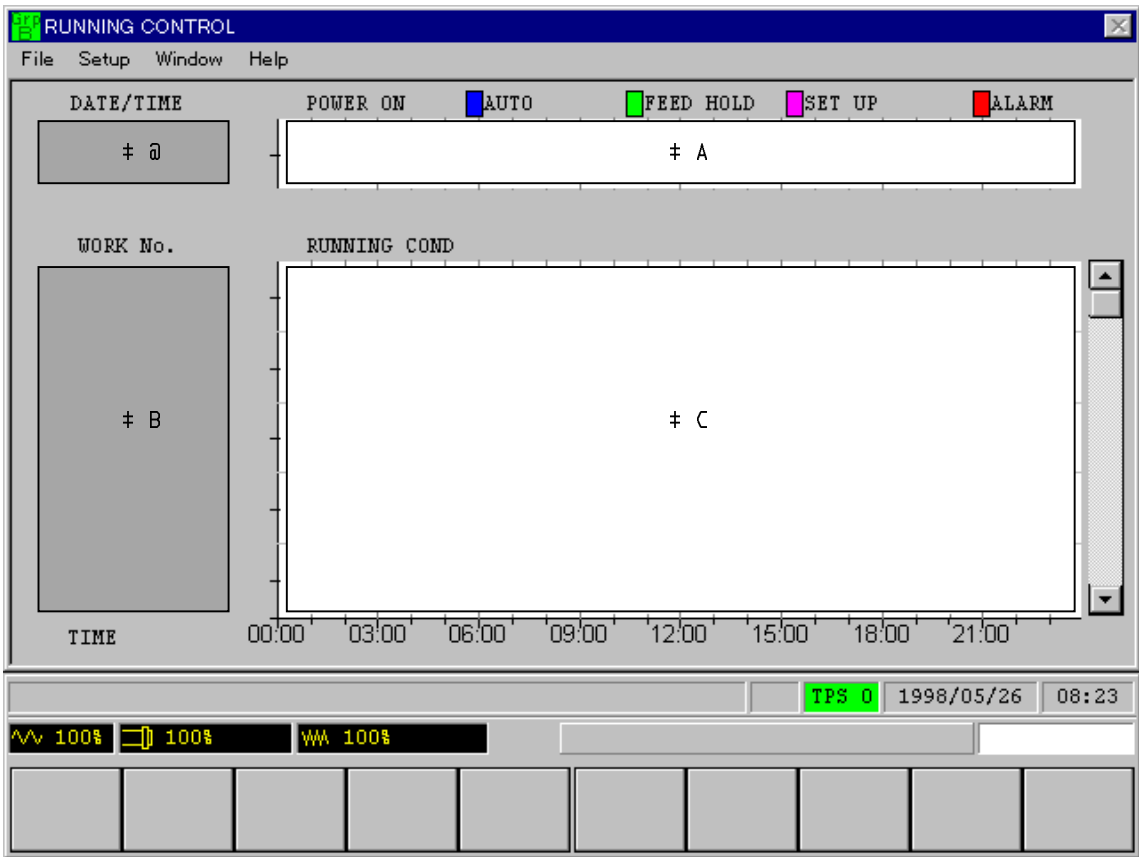
10-6-1 Data display

1. Displayed data with normal display of RUNNING COND. and SPINDLE LOAD menu items



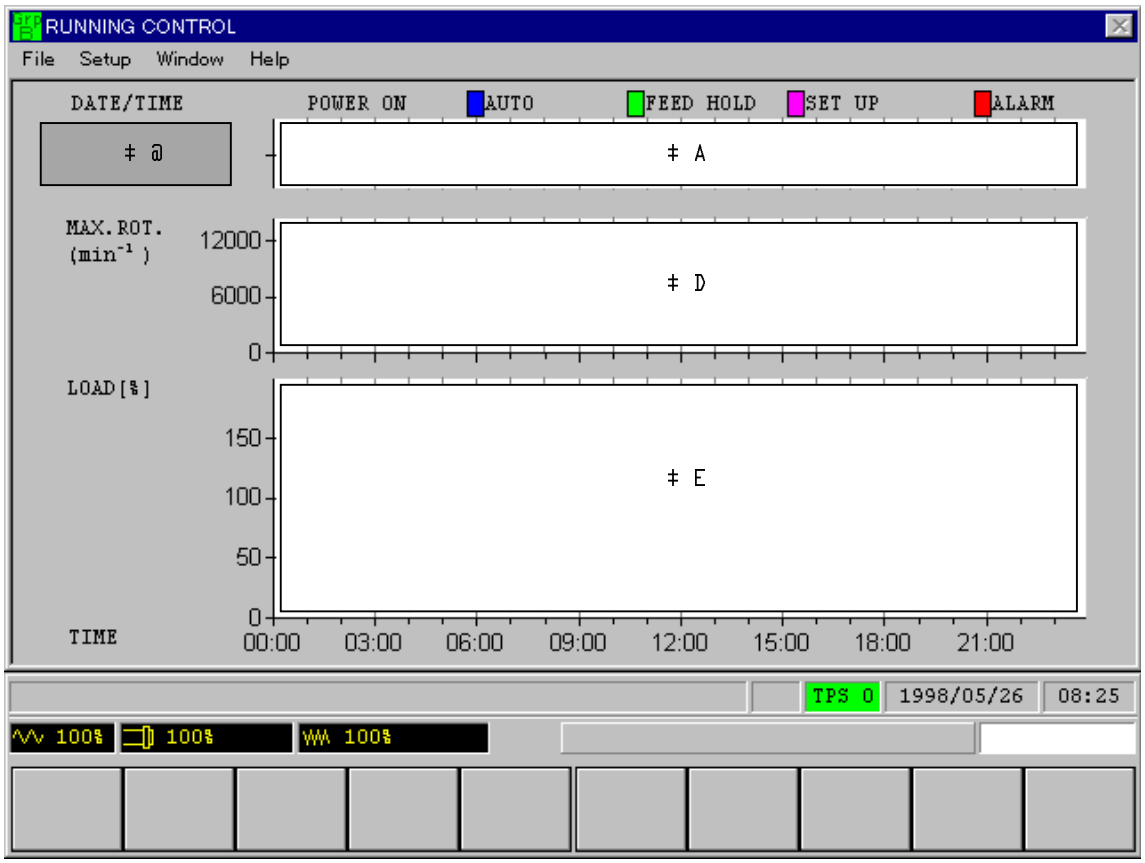
D735S1071E

2. Displayed data with reverse display of RUNNING COND. menu item



D735S1072E

3. Displayed data with reverse display of SPINDLE LOAD menu item



D735S1073E

4. Display data description

No.	Item	Unit	Data description
•	DATE/TIME	•	- Date and cumulative power-on time for the operational status display **a** (**b**) **c** **a** : Date **b** : Day of the week **c** : Cumulative power-on time
•	Graphic display of operational status	•	- The operational status is displayed as follows on a strip graph: Division / Color / Description AUTO / blue / The time for which the CYCLE START lamp was on. FEED HOLD / green / The time for which the machine remained in feed-hold or single-block stop. ALARM / red / The time for which the automatic operation was disabled due to an alarm. SET UP / purple / The time for which power was on under other states. - Blank areas (no strip display) refer to power-off time.
•	WORK No.	•	- Work number of the machining program and machining result. **a** (**b**) **c** **a** : Work number **b** : Program type (MAZATROL or EIA/ISO) **c** : Number of finished workpieces
•	RUNNING COND.	•	- Strip graphs of the operational status are displayed for each work number. Refer to • above for the division method.

No.	Item	Unit	Data description
•	MAX.ROT.	min^{-1}	- Graphic display of change in spindle speed. - The change in spindle speed during power-on is displayed in a red line.
•	LOAD	%	- Graphic display of change in spindle load. - The changes in spindle load during rapid traverse and cutting feed are displayed respectively in a light-blue and a green line.

10-1-2 Display modes

This display has the following three display modes:

1. Total display
   The display is always made in this mode when the **RUNNING CONTROL** display is selected. The operational status is displayed totally for the last seven days (one week) from the current date.
   After selecting a date displayed in this mode, a further detailed information upon the operational status can be obtained for that date in another display mode.
2. Display for each work number
   In this mode, the operational status on the date which was selected in the total display mode can be displayed for each machining program.
   Since this mode provides the display of the number of finished workpieces, it can be readily checked what quantity was machined with what program on the particular date.

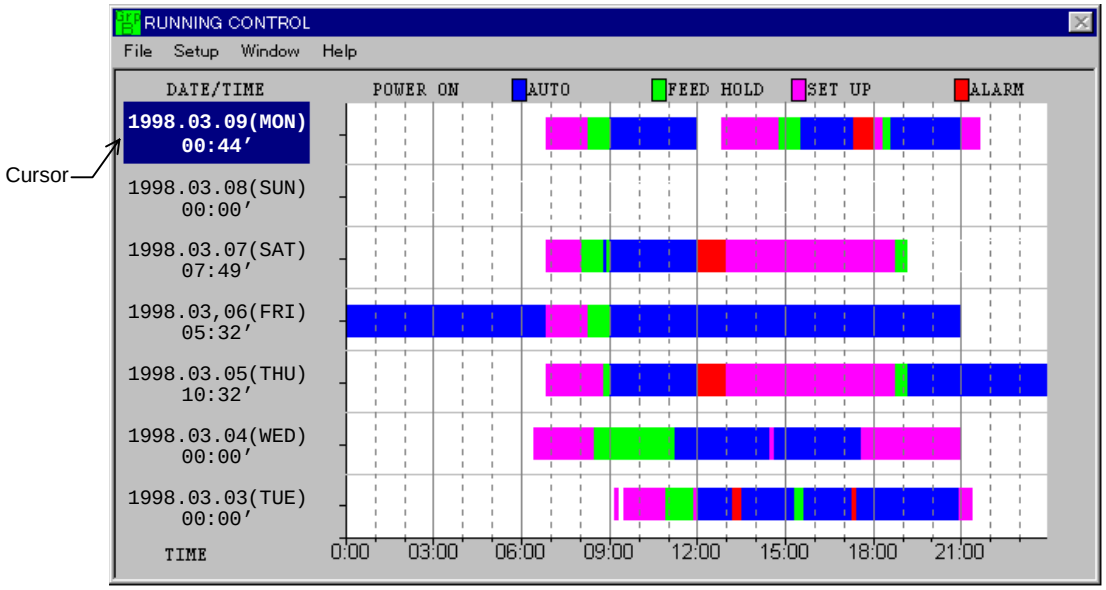
Note: The machining quantity displayed here only refers to the value of “parts count” automatically taken on the corresponding command in the particular program. Refer to the Programming Manual for the parts counting.

3. Spindle load display
   In this mode, the changes in spindle speed and load on the date which was selected in the total display mode can be displayed on a graph.

1. Changing from the total to the display mode for each work number

(1) Call up the cursor in the column of DATE/TIME by pressing a cursor key.

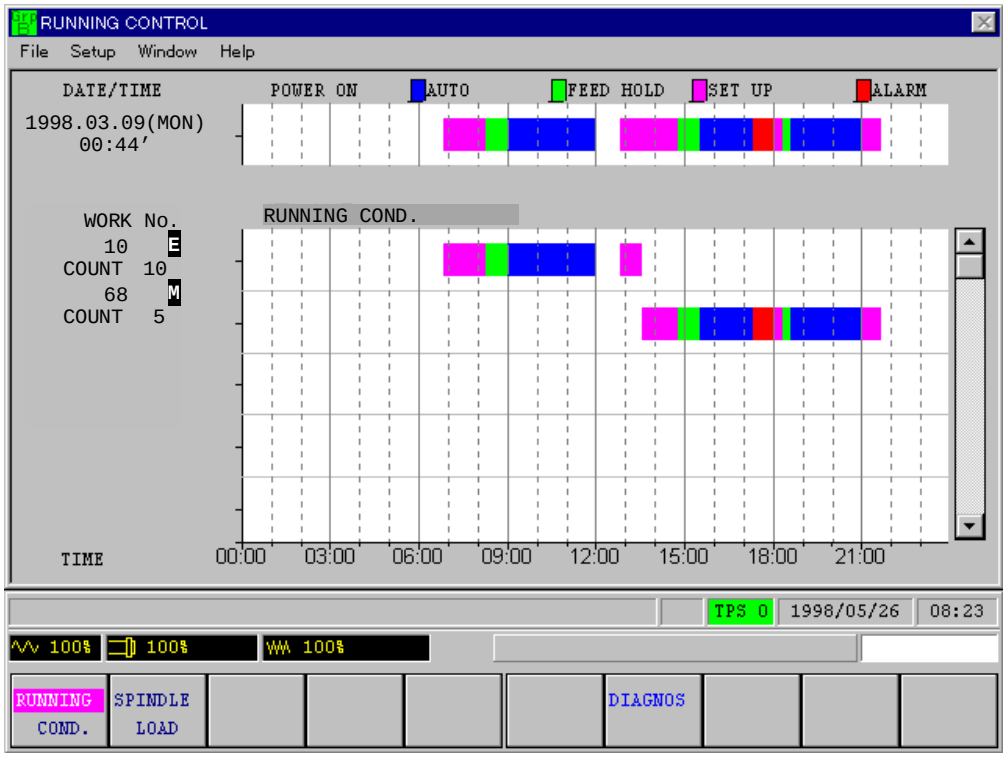
- The cursor here refers to a rectangular section displayed in reverse, as shown below, in the DATE/TIME column.



(2) Position the cursor on the desired date by using the cursor keys.

(3) Press the **RUNNING COND.** menu key.

➔ The display of the menu item will be reversed and the display mode will change to the display for each work number.



- Up to five machining programs can be displayed at the same time. Scrolling for the sixth program onward can be done by pressing the page key.

- Pressing the **RUNNING COND.** menu key anew will clear the reverse display of the menu item to retrieve the total display mode.

2. Changing from the total to the display mode for spindle load

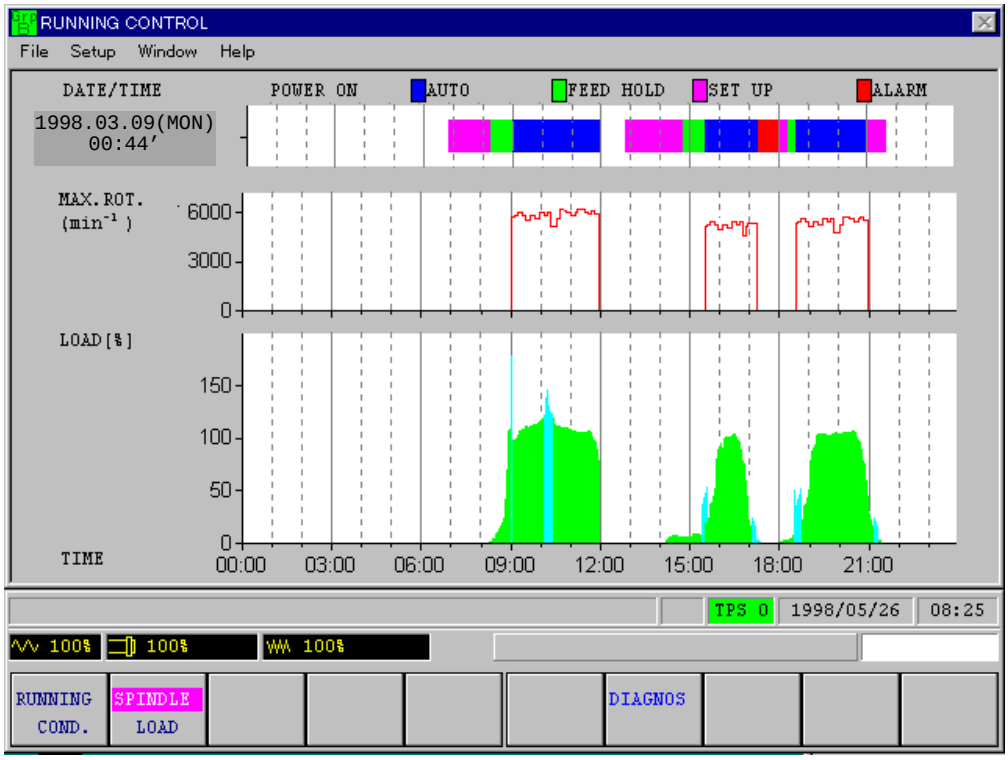
(1) Call up the cursor in the column of DATE/TIME by pressing a cursor key.

- The cursor must be called up first, as is the case with the change to the display mode for each work number.

(2) Position the cursor on the desired date by using the cursor keys.

(3) Press the **SPINDLE LOAD** menu key.

➔ The display of the menu item will be reversed and the display mode will change to the display of spindle load.



- Pressing the **SPINDLE LOAD** menu key anew will clear the reverse display of the menu item to retrieve the total display mode.

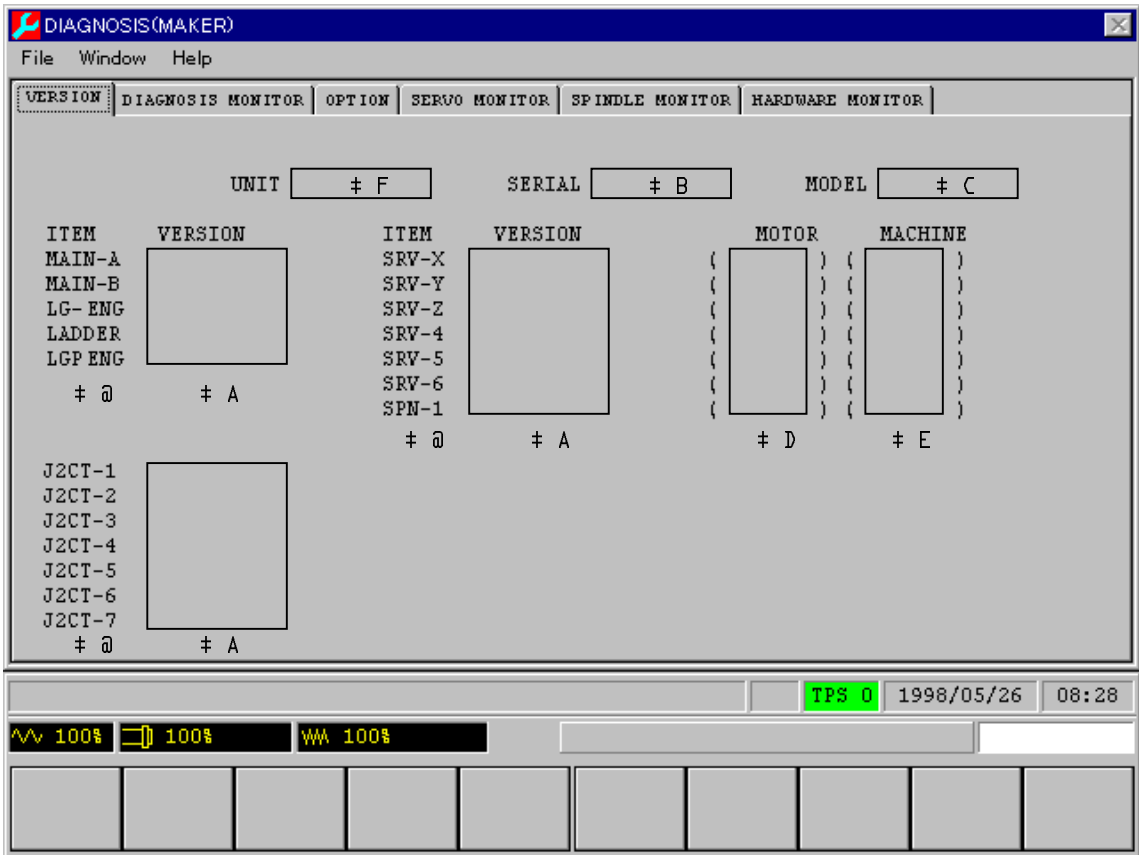
10-7 DIAGNOSIS (MAKER) – VERSION Display

Function:

Displaying versions of NC unit and servo axes

10-1-1 Data display

1. Displayed data



D735S1074E

2. Display data description

No.	Item	Unit	Data description
•	ITEM	•	Name of NC unit and servo axes
•	VERSION	•	Version
•	SERIAL	•	Serial number of NC unit
•	MODEL	•	Model name of NC unit
•	MOTOR	•	Model name of detector
•	MACHINE	•	Model name of detector
•	UNIT	•	Type name of NC unit

Note: Version of NC unit refers to MAIN-A.

10-8 DIAGNOSIS (MAKER) – DIAGNOSIS MONITOR Display

Function:

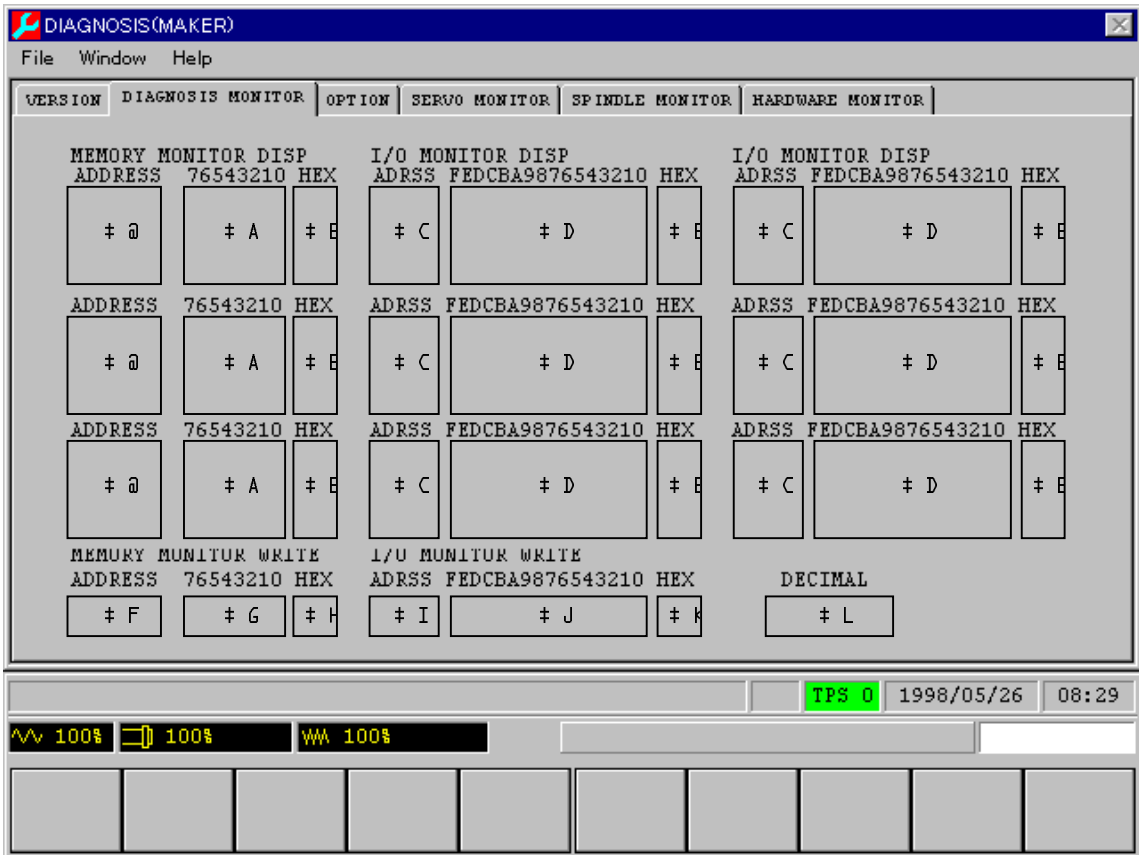
- Display of memory status in the NC unit
- You can write data into the memory.

Remarks:

Please follow the instructions of YAMAZAKI MAZAK servicemen when writing data into the memory.

10-1-1 Data display

1. Displayed data



D735S1075E

2. Display data description

No.	Item	Unit	Data description
•	ADDRESS	•	Display of memory address
•	Data	•	Data display in binary for the address in item •
•	Data	•	Data display in hexadecimal for the address in item •
•	ADRSS	•	Display of I/O address
•	Data	•	Data display in binary for the address in item •
•	Data	•	Data display in hexadecimal for the address in item •
•	ADDRESS	•	Address when writing data into memory
•	Data	•	Data display in binary for the address in item •
•	Data	•	Data display in hexadecimal for the address in item •

No.	Item	Unit	Data description
•	ADRSS	•	Address when writing data into I/O address
•	Data	•	Data display in binary for the address in item •
•	Data	•	Data display in hexadecimal for the address in item •
•	Data	•	Data display in decimal for the address in item •

10-1-2 Display operations

1. Monitoring memory status

(1) Place the cursor on the ADDRESS column of MEMORY MONITOR DISP.

Example:

```
MEMORY MONITOR DISP
ADDRESS    76543210  HEX
MH00000000 00000000  00
MH00000001 00110000  30
MH00000002 00011110  1E
MH00000003 00011010  1A
```

(2) Set the address to be monitored.

Example: Set 89ABC.

```
MEMORY MONITOR DISP
ADDRESS    76543210  HEX
MH00089ABC 11111111  FF
MH00089ABD 00101000  28
MH00089ABE 00100001  21
MH00089ABF 01101110  6E
```

2. Monitoring I/O memory status

(1) Place the cursor on the ADRSS column item of I/O MONITOR DISP.

```
I/O MONITOR DISP
ADRRS FEDCBA9876543210 HEX
X0000 0000000000000000 0000
X0010 0000000000000000 0000
X0020 0000000000000000 0000
X0030 0000000000000000 0000
```

(2) Set the address to be monitored.

Example: Set Y120.

```
I/O MONITOR DISP
ADRRS FEDCBA9876543210 HEX
Y0120 0000000000000000 0000
Y0130 0000000000000000 0000
Y0140 0000000000010000 0010
Y0150 0000100000000000 0800
```

3. Writing data into the memory

(1) Place the cursor on the ADDRESS item of MEMORY MONITOR WRITE.

```
MEMORY MONITOR WRITE
ADDRESS    76543210  HEX
MW00000000 00000000  00
```

(2) Set the address for which the data should be written.

Example: Set 123456.

```
MEMORY MONITOR WRITE
ADDRESS    76543210  HEX
MW00123456 11111111  FF
```

(3) Move the cursor onto the HEX item of MEMORY MONITOR WRITE.

Example:

```
MEMORY MONITOR WRITE
ADDRESS    76543210  HEX
NW00123456 11111111  FF
```

(4) Set the data to be written.

Example: Set 1A.

```
MEMORY MONITOR WRITE
ADDRESS    76543210  HEX
NW00123456 00011010  1A
```

Note: Please follow the instructions of YAMAZAKI MAZAK servicemen when writing data into the memory.

4. Writing data in hexadecimal into the I/O memory

(1) Place the cursor on the ADRSS item of I/O MONITOR WRITE.

Example:

```
I/O MONITOR WRITE
ADRSS FEDCBA9876543210  HEX  DECIMAL
X0000 0000000000000000  0000 0
X0010 0000000000000000  0000 0
```

(2) Set the address for which the data should be written.

Example: Set X120.

```
I/O MONITOR WRITE
ADRSS FEDCBA9876543210  HEX  DECIMAL
X0120 0000000000000000  0000 0
X0130 0000000000000000  0000 0
```

(3) Move the cursor onto the HEX item of I/O MONITOR WRITE.

Example:

```
I/O MONITOR WRITE
ADRSS FEDCBA9876543210  HEX  DECIMAL
X0120 0000000000000000  0000 0
X0130 0000000000000000  0000 0
```

(4) Set the data to be written.

Example: Set F1.

```
I/O MONITOR WRITE
ADRSS FEDCBA9876543210  HEX
X0120 0000000011110001  00F1
```

Note: Write data in accordance with the directions of a YAMAZAKI MAZAK serviceman.

5. Writing data in decimal form into the I/O memory

(1) Carry out the same operations as those of steps (1) and (2) which are to be taken when writing data in hexadecimal form into the I/O memory.

Example: Set X120.

```
I/O MONITOR WRITE
ADRSS FEDCBA9876543210  HEX  DECIMAL
X0120 0000000000000000  0000 0
X0130 0000000000000000  0000 0
```

(2) Move the cursor onto the DECIMAL item of I/O MONITOR WRITE.

Example:

```
I/O MONITOR WRITE
ADRSS FEDCBA9876543210  HEX  DECIMAL
X0120 0000000000000000  0000 0
X0130 0000000000000000  0000 0
```

(3) Press the **LONG** menu key only if the data to be written falls outside the range from –32768 to 32767.

➔ The display status of the **LONG** menu item will then reverse and the numeral on the second line will be cleared.

Example:

```
I/O MONITOR WRITE
ADRSS FEDCBA9876543210  HEX  DECIMAL
X0120 0000000000000000  0000 0
X0130 0000000000000000  0000
```

(4) Input the data to be written.

Example: Set 32768.

```
I/O MONITOR WRITE
ADRSS FEDCBA9876543210  HEX  DECIMAL
X0120 1000000000000000  8000 32768
X0130 0000000000000000  0000
```

Note: Write data in accordance with the directions of a YAMAZAKI MAZAK serviceman.

10-9 DIAGNOSIS (MAKER) – OPTION Display

Function:

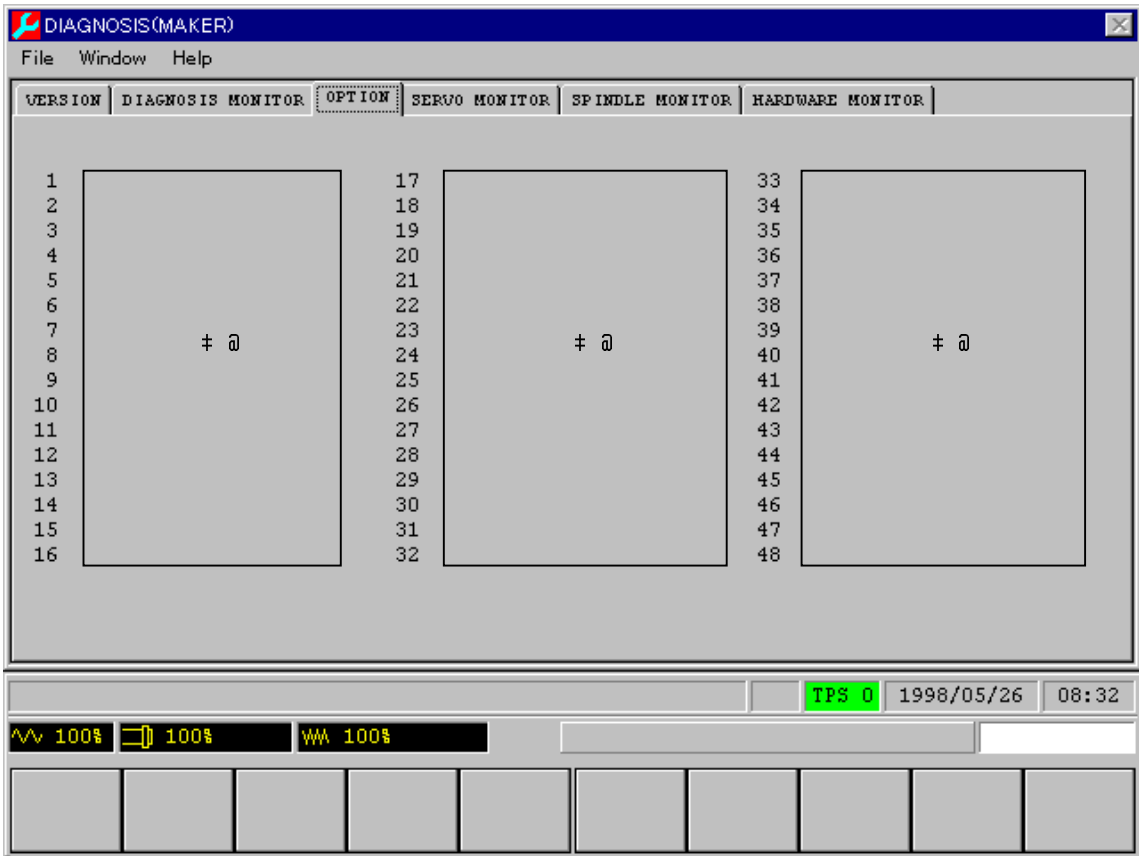
Displaying all optional functions for the NC unit

Remarks:

All terms are displayed in English on this display.

10-9-1 Data display

1. Displayed data



D735S1076E

2. Display data description

No.	Item	Unit	Data description
•	Function name	•	Display of whether each of optional functions for the NC unit is available or not.

- NOTE -

11 OPTION

11-1 Machining Management Functions

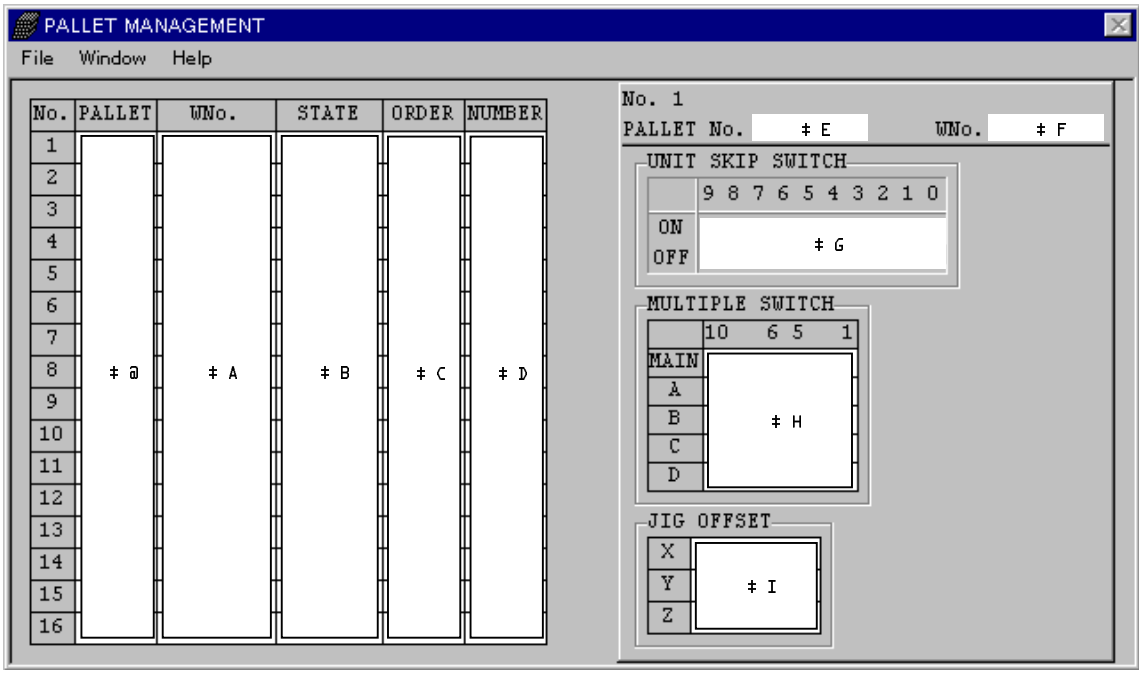
11-1-1 General

The machining management functions are designed so that machining management information for each pallet can be managed by saving the information into a **PALLET MANAGEMENT** display area and pallet ID information separate from the machining program data field. This allows the machining program sequence to be changed according to the status of the workpiece without updating the program. These functions are therefore effective for multi-pallet types of machines, in particular, that have recently come into widespread use.

1. Scheduled-operation function
   Allows the machining sequence to be designated for each pallet on the **PALLET MANAGEMENT** display. This eliminates the need for program updating.
2. External unit skipping function
   Allows the MMS unit and subprogram unit of a machining program to be designated on the **PALLET MANAGEMENT** display as the program units to be skipped.
3. External multi-piece machining function
   Allows a multi-piece machining process to be included in a machining program by designating that process on the **PALLET MANAGEMENT** display.
4. Jig offsetting function
   Allows offsetting data from a reference pallet to be designated on the **PALLET MANAGEMENT** display. Thus, constant machining accuracy can be achieved even when workpieces of the same type are machined using multiple pallets.

11-1-2 PALLET MANAGEMENT display

1. Displayed data



D735S1077E

Displayed data for each function is described below.

11-1-3 Scheduled-operation function

1. Data of the PALLET MANAGEMENT display

Data displayed for the scheduled-operation function (• through •) is described below.

Display data description (1/4)

No.	Data name	Description
•	PALLET	Number of pallets that is initially set for the particular machine specifications.
•	WNo.	Work number of the machining main program to be used for each pallet.
•	STATE	The machining status is displayed for each pallet. (Blank) ..... Indicates that machining for the pallet will not be performed. "•" is displayed under ORDER. READY ..... Indicates that the pallet is ready for machining. CUTTING ..... Indicates that machining is in progress. FINISHED..... Indicates that machining has been completed. (Alarm No.) ..... Identifies the alarm that has occurred during machining. UNFINISH ..... Indicates that machining has been aborted since the RESET key was pressed during machining.
•	ORDER	The machining order is displayed for each pallet. Machining will not be performed if the pallet number has a displayed "•" or UNFINISH.
•	NUMBER	The total count of times of machining up to that time is displayed. The maximum count is 9999.

2. Data editing methods

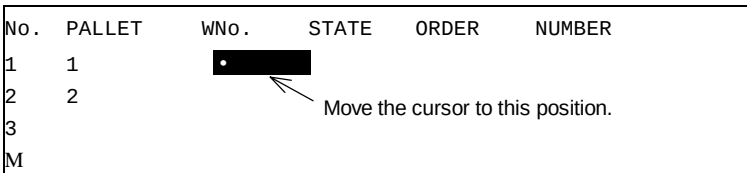
WNo., STATE, ORDER, NUMBER and multiple process numbers must be set for editing scheduled-operation data. These methods are described below.

A. Setting WNo.

Set the work numbers of the main programs to be executed for each pallet.

(1) Press the **EDIT** menu key on the **PALLET MANAGEMENT** display.
So many pallets as predetermined for the particular machine specifications will then be displayed.

(2) Move the cursor to the WNo item. of the intended pallet using the cursor key.



(3) Using the appropriate numeric key(s), enter the work number of the machining program (main routine) which corresponds to the pallet. Then press the INPUT key .

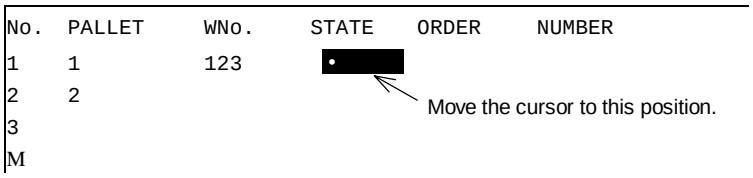
Example: To set WNo. 123, press [1] [2] [3] [INPUT] in this order.

➔ The work number is set.

B. Setting data in the STATE item

Set the status of machining on each pallet.

(1) Move the cursor to the STATE item using the cursor key.

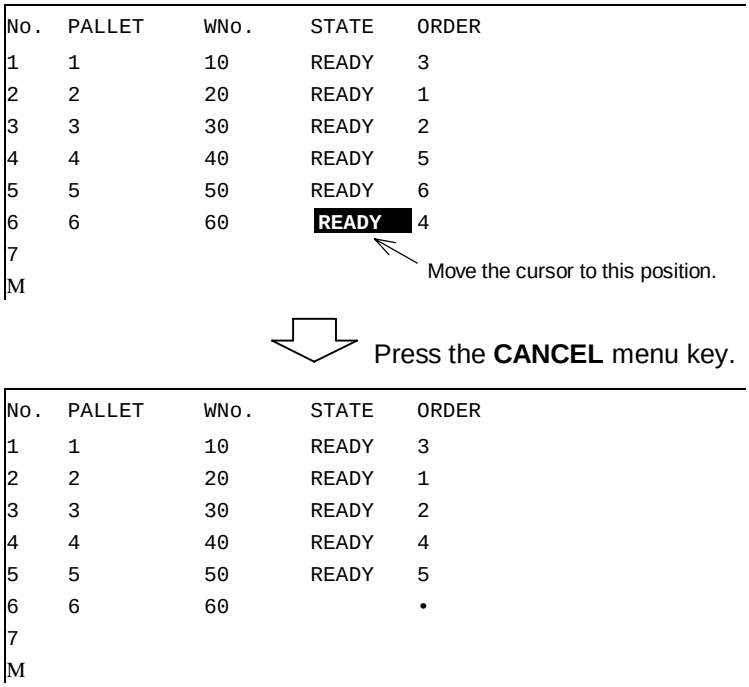


➔ The inquiry message **PALLET STATE <MENU>?** is displayed.

(2) Press the **SET END (READY)** menu key and set READY if operational setup for palletizing has been finished and the pallet is ready for machining.

- If READY is set, the machining order will also be set automatically. Data displayed under ORDER item at this time will be the number next to that of the last pallet having a displayed READY.
- To cancel READY present under STATE item, press the **CANCEL** menu key. STATE item will become blank, "•" will be displayed under ORDER item, and the data under ORDER item for other pallets will be converted.

Example: Canceling READY on the line PALLET 6.



C. Changing data in ORDER

When changing the machining order that has already been set, move the cursor to the ORDER item, enter new data using the appropriate numeric key(s) and press the INPUT key. The ORDER item data for other pallets are auto-changed.

Example: Updating the data of ORDER in line PALLET 4 to “1”.

No.	PALLET	WNo.	STATE	ORDER
1	1	10	READY	5
2	2	20	READY	1
3	3	30	READY	2
4	4	40	READY	**4**
5	5	50	READY	3
6	6	60	READY	6
7				
M				

To update the data of ORDER in line PALLET 4 to “1”:

Press [1] and [INPUT].

No.	PALLET	WNo.	STATE	ORDER
1	1	10	READY	5
2	2	20	READY	2
3	3	30	READY	3
4	4	40	READY	1 ← The data updates to “1”.
5	5	50	READY	4
6	6	60	READY	6
7				
M				

D. Setting data in NUMBER item

Data from 0 to 9999 can be set and the parts count will be performed from the set data at the end of machining.

Note 1: If a data greater than 9999 is set under NUMBER item, the parts count will not be performed at the end of machining.

Note 2: Parts count will be performed irrespective of the data set under NUMBER item in end unit.

E. Setting multiple process

If the machining program for one pallet is structured for each workpiece and/or each process separately, multiple main programs can be set by setting multiple processes.

<Setting multiple process>

(1) Move the cursor to the position of the pallet number for which multiple processes are to be executed.

Example: Setting multiple processes for PALLET 3:

No.	PALLET	WNo.	STATE	ORDER
1	1	10	READY	5
2	2	20	READY	2
3	**3**	30	READY	3
4	4	40	READY	1
5	5	50	READY	4
6	6	60	READY	6
7				
M				

Move the cursor to this position.

(2) Press the **PLURAL PROCESS** menu key.

(3) Press the INPUT key.

➔ For the PALLET No. specified in procedure (1), multiple processes are set.

No.	PALLET	WNo.	STATE	ORDER
1	1	10	READY	5
2	2	20	READY	2
3	3		READY	•
4	3	30	READY	3
5	4	40	READY	1
6	5	50	READY	4
7	6	60	READY	6

M

<Erasing multiple process setting>

(1) Move the cursor to the position of that process of all those previously set for the multi-process pallet that is to be erased.

Example:

No.	PALLET	WNo.	STATE	ORDER
1	1	10	READY	5
2	2	20	READY	2
3	**3**			
4	3	30	READY	3
5	4	40	READY	1
6	5	50	READY	4
7	6	60	READY	6

M

Move the cursor to this position.

(2) Press the **ERASE** menu key**.**

(3) Press the INPUT key.

No.	PALLET	WNo.	STATE	ORDER
1	1	10	READY	5
2	2	20	READY	2
3	3	30	READY	3
4	4	40	READY	1
5	5	50	READY	4
6	6	60	READY	6
7				

M

F. ALL ERASE

This function clears the entire schedule that has been set (A to E).

<Setting method>

Press the **ALL ERASE** menu key. When the message **ALL ERASE <–9999 INPUT>?** is displayed, enter "–9999" and then press the INPUT key.
The display will then be initialized.

Use this function also for initialization in the initial status of the NC system (that is, immediately after the NC system has been loaded).

3. Starting scheduled operation and its progress display

The procedure for starting scheduled operation and the display of the progress of scheduled operation are described below.

A. Starting procedure and progress display

(1) Enter necessary data on the **PALLET MANAGEMENT** display.
See "2. Data editing methods", for further details.

(2) Make the scheduled-operation mode effective using an external switch. The message SCH. OPERAT will be displayed in the reverse-display status in the lower part of the display.

(3) Press the CYCLE START button. Machining will begin with the workpiece of machining order (ORDER) number 1 and the status will change into CUTTING.

(4) When machining is completed, FINISHED and • will be displayed under STATE and ORDER items, respectively, and new data for other pallets will be displayed under ORDER item.

Example: Change of progress display

No.	PALLET	WNo.	STATE	ORDER
1	1	10	READY	4
2	2	20	READY	2
3	3	30	CUTTING	1
4	4	40	READY	5
5	5	50	READY	3
6	6	60	READY	6
7				
M				

Machining PALLET 3 is completed.
STATE of PALLET 2 is changed to "CUTTING".

No.	PALLET	WNo.	STATE	ORDER
1	1	10	READY	3
2	2	20	CUTTING	1
3	3	30	FINISHED	•
4	4	40	READY	4
5	5	50	READY	2
6	6	60	READY	5
7				
M				

(5) The machining process that corresponds to the pallet number having a newly displayed 1 under ORDER item will start.

B. Precautions

1. Editing in the scheduled-operation mode

   - Even during scheduled operation, editing can be done on the **PALLET MANAGEMENT** display, except for the line of 1 under ORDER item (machining now in progress).
     See “2. Data editing methods”, for further details of editing.
   - If the machining for the pallet of CUTTING is finished during EDIT mode, the machine will be stopped. In this case, press the **EDIT END** menu key, then press CYCLE START button to continue machining.

2. Restarting in the scheduled-operation mode
   Restarting in the scheduled-operation mode can be done in the following case only:

   - If the work number for ORDER 1 is the same as that displayed on the **POSITION** display.

   Note: If the UNIT SKIP SWITCH is ON, execution of that unit will be skipped and the immediately succeeding unit will be executed.

3. Other precautions

   - Pressing the RESET key changes CUTTING to UNFINISH. To perform machining for the pallet anew, set READY under STATE.
   - If an alarm occurs during machining, the alarm will be processed as follows:

     1) The alarm number is displayed under STATE.
     2) • is displayed under ORDER, and the machining order is updated.
     3) If the alarm is of such a type that machining can be continued, the machining operation will start from the machining program preset for the next process. If machining cannot be continued, it will terminate on occurrence of the alarm.

Note: Do not set a multi-process program that enables machining to be done only after the preprocess has been completed. If such a program is set and an alarm occurs, the previous process will remain aborted and the next process will begin. Use a subprogram if such programming is required.

CAUTION

- The external switch must be turned off when machining is to be done in the normal mode. If machining is done in the scheduled-operation mode, scheduled operation will be carried out with automatically updated work numbers.
  When SCH. OPERAT is displayed in the reverse-display status in the lower part of the **POSITION** display, then scheduled-operation is in effect.
  When SCH. OPERAT is not displayed on the **POSITION** display, then normal-mode operation is in effect.

11-1-4 External unit skipping function

1. Data of the PALLET MANAGEMENT display.

Display data description (2/4)

No.	Data name	Description
•	PALLET No.	The pallet number corresponding to data • is displayed.
•	WNo.	The work number corresponding to data • is displayed.
•	UNIT SKIP SWITCH	The program is controlled according to ON (skipping), OFF (no skipping) of the bit corresponding to a skip number from 0 to 9 in that program.

2. Entering data

Set UNIT SKIP SWITCH as follows to activate the external unit skipping function.

(1) Press the **EDIT** menu key on the **PALLET MANAGEMENT** display.

(2) Press the cursor key to move the cursor to the position of the intended pallet number.

(3) Press the **PLT DATA EDIT** menu key.

➔ The cursor will be moved to the UNIT SKIP SWITCH item.

(4) Move the cursor to the position of the intended switch number.

(5) Pressing **ON** or **OFF** menu key completes UNIT SKIP SWITCH setting.

(6) Press the **EDIT END** menu key on the **PALLET MANAGEMENT** display.

(7) Set either from “0” to “9” in the $ field of the subprogram unit on the **PROGRAM** display. Refer to the Programming Manual (MAZATROL) for setting procedure.

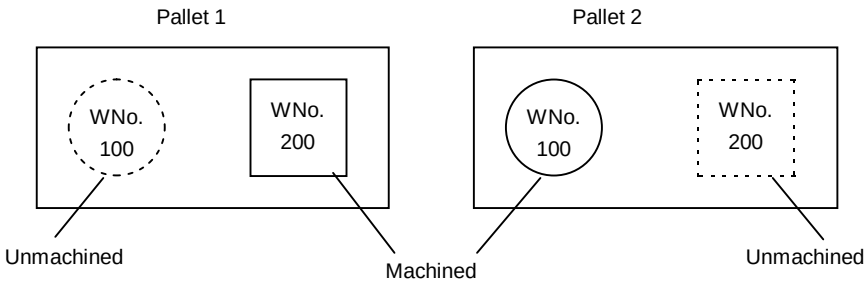
3. Starting operation

See 3. of Subsection **11-1-3**. The operating procedure is the same.

4. Operation

A specific example is shown below.

Example:



M3S045

Program WNo. 1 (Main program)

```
UNo.   UNIT     X         Y         θ         Z     4
1      WPC-1    -300      -300      0         -400  0
UNo.   UNIT     WORK.No.          $         REPEAT
2      SUB PRO  100               1         1
UNo.   UNIT     WORK.No.          $         REPEAT
3      SUB PRO  200               2         1
```

PALLET MANAGEMENT display

```
No.  PALLET  WNo.  STATE  ORDER  NUMBER
 1     1      1   READY    1      0
 2     2      1   READY    2      0
 3     3
 4     4
    ⟨
    ⟩
```

```
No. 1
PALLET No. 1          WNo. 1

 UNIT SKIP SWITCH
     9 8 7 6 5 4 3 2 1 0
 ON                  •
 OFF • • • • • • • •   •
```

```
No. 2
PALLET No. 2          WNo. 1

 UNIT SKIP SWITCH
     9 8 7 6 5 4 3 2 1 0
 ON                •
 OFF • • • • • • •   • •
```

- For pallet 1, since UNIT SKIP SWITCH 1 is set to ON, UNo. 2 (that is, subprogram No. 100) will be skipped.
- For pallet 2, since UNIT SKIP SWITCH 2 is set to ON, UNo. 3 (that is, subprogram No. 200) will be skipped.

11-1-5 External multi-piece machining function

1. Data of the PALLET MANAGEMENT display.

Display data description (3/4)

No.	Data name	Description
•	MULTIPLE SWITCH	Multi-piece machining will be performed if the bits corresponding to the multi-piece machining flag (MULTI FLAG) on the program are correspondingly set.

2. Entering data

Set MULTIPLE SWITCH as follows to activate the external multi-piece machining function.

(1) Press the **EDIT** menu key.

(2) Press the cursor key to move the cursor to the position of the intended pallet number.

(3) Press the **PLT DATA EDIT** menu key.

(4) Set the cursor to the MAIN, A, B, C, or D data item of MULTIPLE SWITCH.

(5) Press data key 0 or 1 to enter a 10-bit number of 0s or 1s.

(6) Set the switches by repeating procedures (4) and (5).

(7) Press the **EDIT END** menu key.

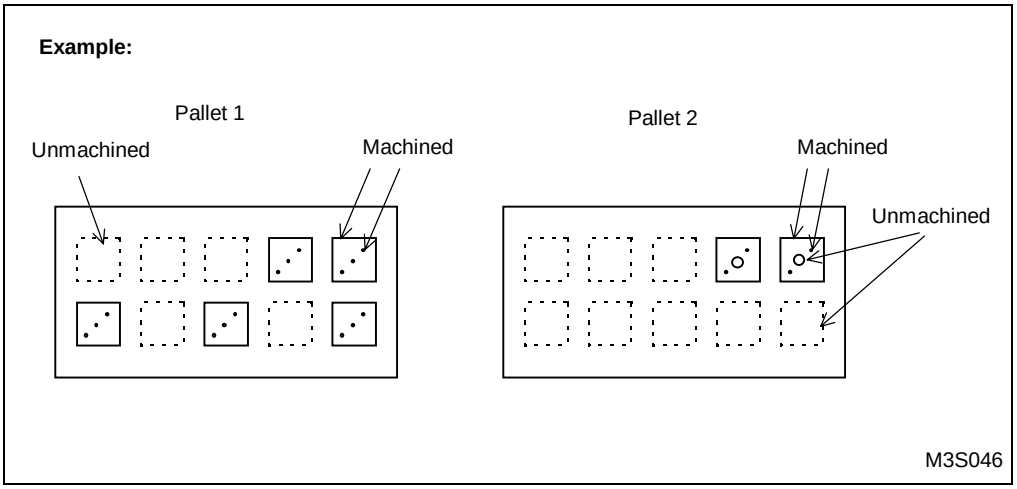
(8) If you have selected either one from A to D in step (4) above, set that selected switch in the $ ield of the subprogram unit on the **PROGRAM** display. Refer to the Programming Manual (MAZATROL) for the setting procedure.

3. Starting operation

See 3. of Subsection **11.1.3**. The operating procedure is the same.

4. Operation

An operation example is shown below.



Program WNo. 1 (Main program)

UNo.	MAT	INITIAL-Z	ATC MODE	MULTI MODE	MULTI FLAG	PITCH-X	PITCH-Y
0	CST IRN	50	1	5 * 2	1100010101	10	10
UNo.		X	Y	θ	Z	4	(a)
1	WPC-1	-300	-300	0	-400	0	
UNo.							
2	LINE OUT•						
SNo.							
1	E-MILL•						
FIG							
1	SQR•						
UNo.	UNIT	WORK No.	$	REPEAT			
3	SYB PRI	100	A	1			
UNo.	UNIT	CONTI.	NUMBER				
4	END	0	1				

Program WNo. 100 (Subprogram)

UNo.	MAT	INITIAL-Z	ATC MODE	MULTI MODE	MULTI FLAG	PITCH-X	PITCH-Y
0	CST IRN	50	1	OFFSET	•	•	•
OFS	X	Y	θ	Z			
1	10	10					
2	20	20					
3	30	30					
UNo.							
1	DRILLING•						
SNo.							
1	CTR-DR•						
2	DRILL•						
FIG		Z	X	Y			
1	PT	0	0	0	•		
UNo.	UNIT	CONTI.	NUMBER				
2	END	1	0				

Equivalent to MULTI FLAG 0000000111

PALLET MANAGEMENT display

No.	PALLET	WNo.	STATE	ORDER	NUMBER
1	1	1	READY	1	0
2	2	1	READY	2	0
3	3				
4	4				
5					
6					
7					

No. 1
PALLET No. 1 WNo. 1

UNIT SKIP SWITCH

	9	8	7	6	5	4	3	2	1	0
ON										
OFF	•	•	•	•	•	•	•	•	•	•

MULTIPLE SWITCH

	10	6	5	1
MAIN	0 0 0 0 0	0 0 0 0 0		
A	0 0 0 0 0	0 0 0 0 0		
B	0 0 0 0 0	0 0 0 0 0		
C	0 0 0 0 0	0 0 0 0 0		
D	0 0 0 0 0	0 0 0 0 0		

No. 2
PALLET No. 2 WNo. 1

UNIT SKIP SWITCH

	9	8	7	6	5	4	3	2	1	0
ON										
OFF	•	•	•	•	•	•	•	•	•	•

MULTIPLE SWITCH

	10	6	5	1
MAIN	1 1 0 0 0	0 0 0 0 0		
A	0 0 0 0 0	0 0 0 0 0		
B	0 0 0 0 0	0 0 0 0 0		
C	0 0 0 0 0	0 0 0 0 0		
D	0 0 0 0 0	0 0 0 0 0		

The machining is executed according to the common data which is included both in MAIN and MULTI FLAG of the main program, or both in A to D (set in $ of the subprogram unit) and MULTI FLAG of the subprogram.

- Pallet 1

(Main program)	MAIN	0 0 0 0 0 0 0 0 0 0	(If all bits are 0s, they will all be handled the same as 1s.
	MAIN	1 1 1 1 1 1 1 1 1 1	
	MULTI FLAG	1 1 0 0 0 1 0 1 0 1	
	Common	1 1 0 0 0 1 0 1 0 1	→ (a) (Main program)
(Sub program)	A	0 0 0 0 0 0 0 0 0 0	(If all bits are 0s, they will all be handled the same as 1s.)
	A	1 1 1 1 1 1 1 1 1 1	
	MULTI FLAG	0 0 0 0 0 0 0 1 1 1	
	Common	0 0 0 0 0 0 0 1 1 1	→ (b) (Subprogram)

- Pallet 2

(Main program)		
MAIN		1 1 0 0 0 0 0 0 0 0
MULTI FLAG		1 1 0 0 0 1 0 1 0 1
Common		1 1 0 0 0 0 0 0 0 0
(Sub program)		
A		0 0 0 0 0 1 0 1 0 1
MULTI FLAG		0 0 0 0 0 0 0 1 1 1
Common		0 0 0 0 0 0 0 1 0 1

11-1-6 Jig offsetting function

1. Data of the PALLET MANAGEMENT display

Display data description (4/4)

No.	Data name	Description
•	JIG OFFSET	The amounts of offset from the reference point of a reference pallet at a table angle of 0 degrees are input to X, Y, and Z with plus or minus signs.

2. Entering data

Set JIG OFFSET as follows to activate the jig offsetting function.

(1) Press the **EDIT** menu key.

(2) Press the cursor key to move the cursor to the position of the intended pallet number.

(3) Press the **PLT DATA EDIT** menu key.

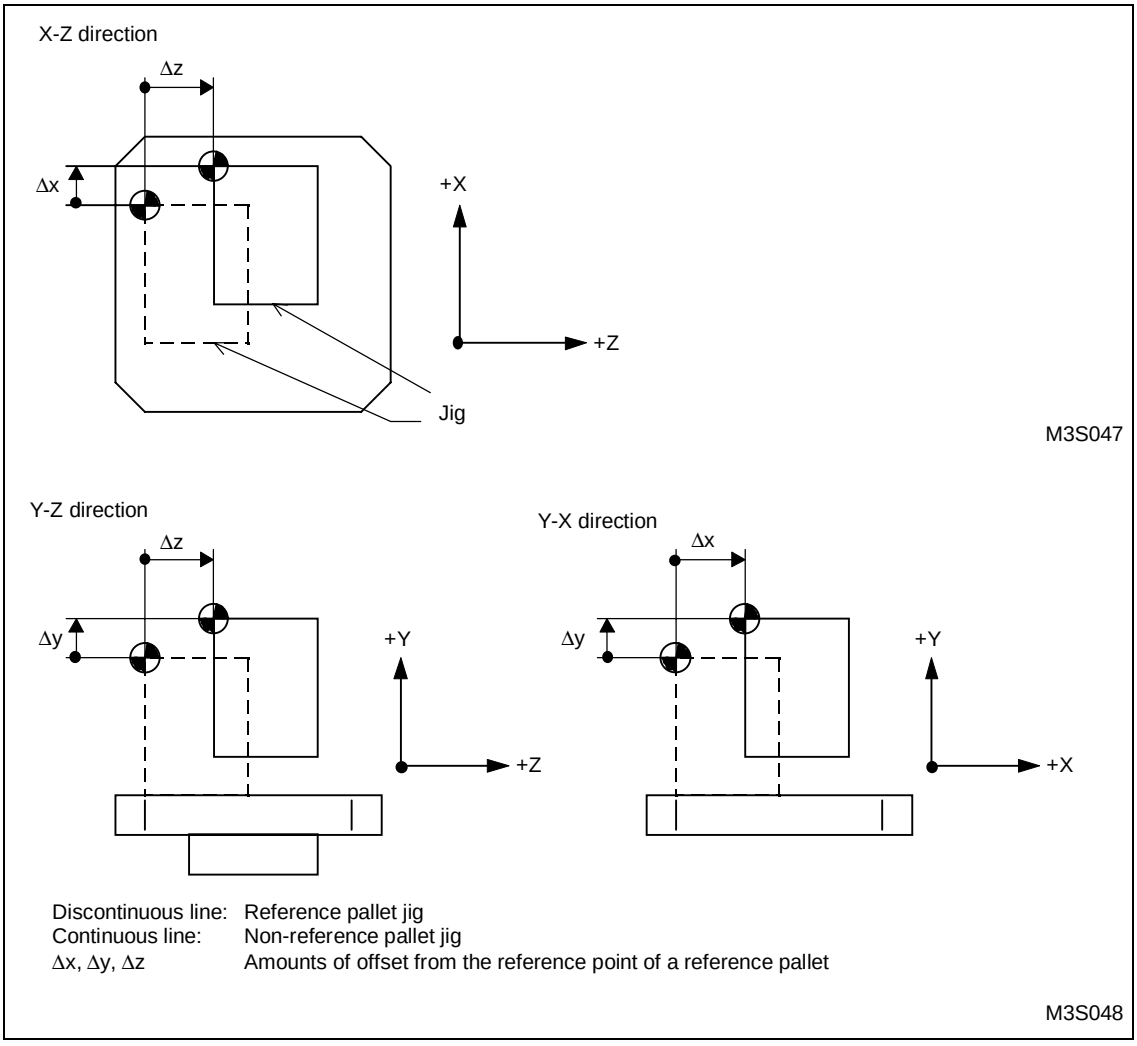
(4) Set the cursor to the X, Y or Z data item of JIG OFFSET.

(5) Using the numeric key(s), set the amount of offset from the reference point of a reference pallet and press the INPUT key.

(6) Press the **EDIT END** menu key.

3. Meaning of jig offset data

When, as shown below, the reference point differs between the reference pallet jig (indicated by a discontinuous line) and the intended jig (indicated by a continuous line), set the amounts of offset at a table angle of 0 degrees (Δx, Δy, Δz) as the offsetting data for the jig you are going to use. The offsetting data thus set will be added to the program coordinates and the machining program will operate on the coordinate system having the added data.



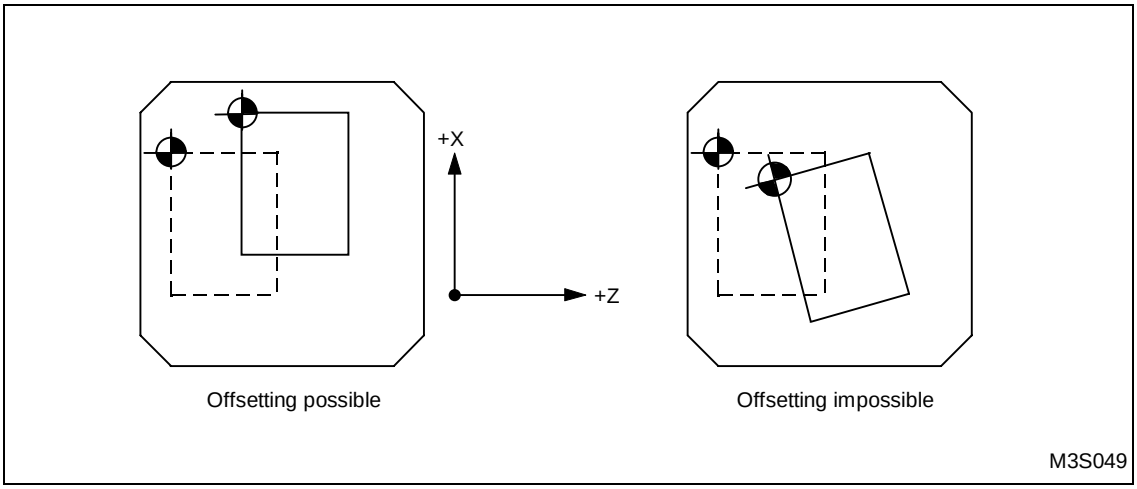
4. Notes

Strictly observe the following notes when using the jig offsetting function:

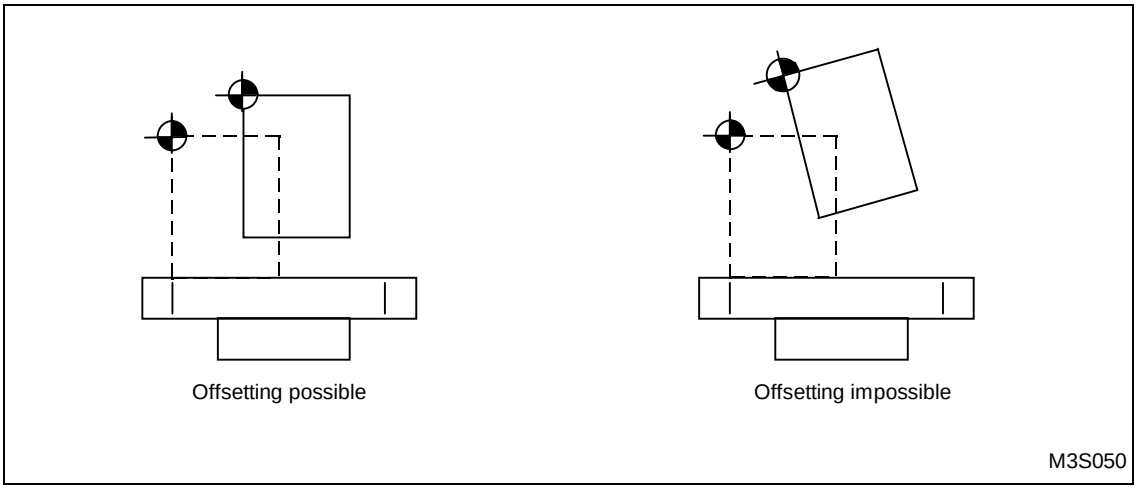
1. Data containing the jig offsetting data is displayed in the field of WPC (workpiece coordinates) of the **POSITION** display.
2. Set jig offsetting data to 0 if the jig offsetting function is not to be used.
3. Jig offsetting will not be performed if the table is rotated using the B code commands of manual programs or EIA programs. Jig offsetting function, however, is available for the table rotation by an INDEX unit.

4. Since the jig offsetting function only corrects any offsets in each axis direction, no corrections are performed for any offsets in the rotational direction of each axis.

   1) X-Z direction



   2) Y direction



5. During MMS measurement, data not containing the jig offsetting data is written into the program coordinate data area.

11-1-7 Parameters

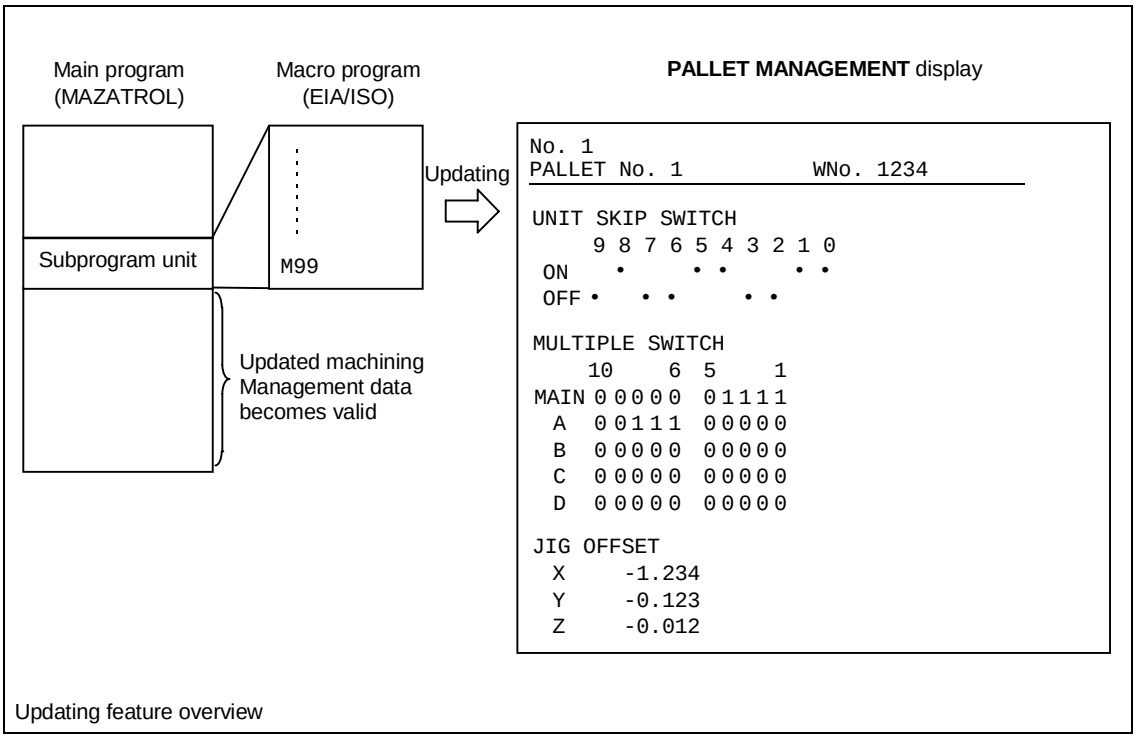
The following lists parameters related to the machining management functions:

Parameter	Setting value	Description
L54	0	The machining management functions become invalid.
	1	Valideated for operation of MAZATROL-FMS with pallet ID-function.
	2	Machining management operation for multiple-pallet system is selected.
L53		On the **PALLET MANAGEMENT** display, work numbers are:
	1	Displayed
	0	Not displayed

11-1-8 Machining management data writing macro-program

1. Overview

The UNIT SKIP SWITCH data, MULTIPLE SWITCH data, and JIG OFFSET data that are saved within the machining management feature can be updated using macro-programs.
Updating needs special system variables. For macroprograms, not only setting data with those variables, but also special setting for updating is required.
This feature becomes valid only when the main program is a MAZATROL program.
Updating will not occur if an updating program is executed alone. Updating program must be called using a subprogram unit of a MAZATROL program.



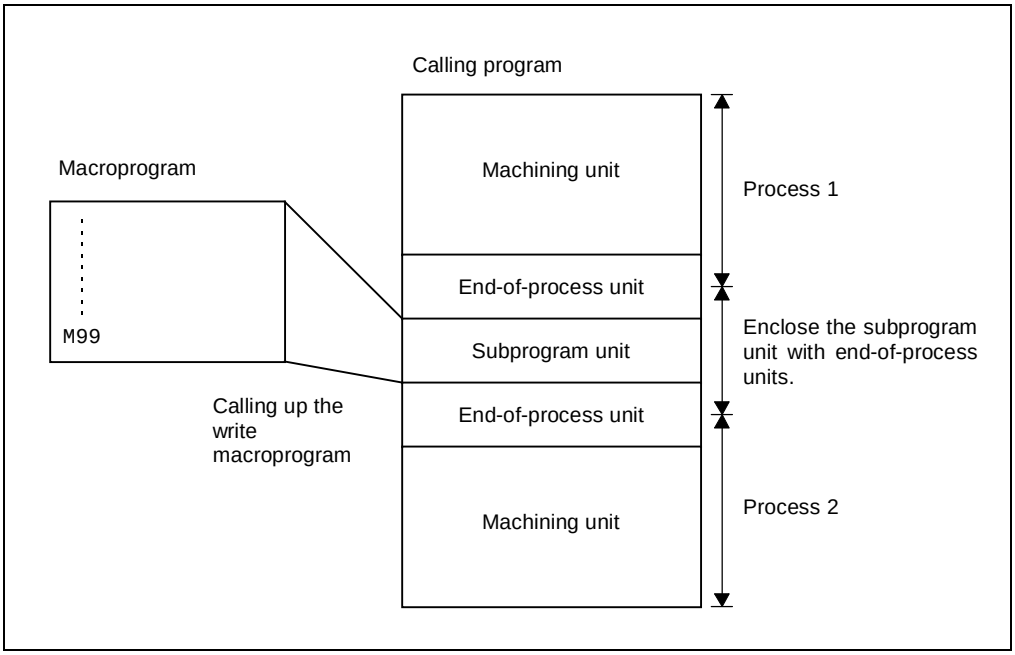
Updating feature overview

2. Making an updating program

Make an updating program into a configuration that allows a macro-program to be called using a subprogram unit of a MAZATROL program.

A. MAZATROL program to be used for call

A calling program must be made as follows. Otherwise, the machine will operate incorrectly.



Make a calling program so that each subprogram unit is executed between end-of-process units, as shown in the diagram above.
Also, set the subprogram units as follows:

<Setting the subprogram units>

(1) During creation of a MAZATROL program, press the **OTHER** menu key. (For operation, refer to the subsection “Procedure for calling up the special mode unit” in the Programming Manual (MAZATROL).

(2) Press the **SUB PROGRAM** menu key.

➔ The cursor blinks at WORK No. item.

UNo.	UNIT	WORK No.
•	SUB PRO	

- The menu display will change over to the following menu:

MEASURE MACRO									

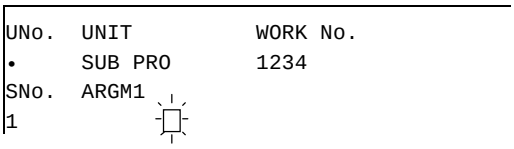
(3) Press the **MEASURE MACRO** menu key.

(4) Set a write macro program number using the numeric keys and press the INPUT key.

➔ The work number is displayed yellow in the WORK No. item.

UNo.	UNIT	WORK No.
•	SUB PRO	1234

(5) When using an argument(s), move the cursor to the ARGM item and set an argument(s) using the numeric keys.



<Supplementary description>

The following lists restrictions, and the reasons, as to the making of a calling program:

Restriction 1 The write macro calling subprogram unit must end with an end-of-process unit.

Reason The NC performs data check and other pre-processing operations on a machining program process basis. If a write macro-program is not separated as one process, therefore, pre-processing will be performed on all subsequent machining units including skipped units before write operations are performed. This will result in an abnormal machine action even if the UNIT SKIP SWITCH data is set to ON.

Restriction 2 The **MEASURE MACRO** menu key must be pressed to set a subprogram unit workpiece number.

Reason The NC pre-reads the selected machining program. When machining management data is to be written, pre-reading must be stopped immediately. The NC stops the pre-reading operation on execution of the subprogram unit that you have set after pressing the **MEASURE MACRO** menu key.

Note: Special processing must be done to restart from the stopped status of pre-reading by the NC. Do not use the **MEASURE MACRO** menu function, except when the usage is clearly indicated.

B. Write macro-program

<System variables for machining management data>

Writing uses special system variables. System variables for various types of machining management data are listed in the table below.

Machining management data and system variables

Data name	System variables number	Setting
UNIT SKIP SWITCH	#50443	Decimal, without decimal point
MULTIPLES SWITCH MAIN	#50441	Decimal, without decimal point
MULTIPLES SWITCH A	#50435	Decimal, without decimal point
MULTIPLES SWITCH B	#50433	Decimal, without decimal point
MULTIPLES SWITCH C	#50431	Decimal, without decimal point
MULTIPLES SWITCH D	#50429	Decimal, without decimal point
JIG OFFSET X	#50449	(mm), decimal, with decimal point
JIG OFFSET Y	#50447	(mm), decimal, with decimal point
JIG OFFSET Z	#50445	(mm), decimal, with decimal point

Note: Since UNIT SKIP SWITCH data and MULTIPLE SWITCH data are both binary bit data, they must be converted into decimal data when set.

Example: To write 0000001111 into the UNIT SKIP SWITCH data area, set the data as follows: #50443 =15; (15 = 2^3 × 1• • • 2^2 × 1• • • 2 × 1• • • 1)

System variables for various types of data have been listed above. Writing is not performed just by

setting data for these variables. To let the NC perform write operations, system variables for write control must be further set. Write control system variables are listed in the table below.

<Write control system variables>

Data name	System variables number	Setting
Write data valid flag	#50467	Decimal, without decimal point bit 0: UNIT SKIP SWITCH valid bit 1: MULTIPLE SWITCH MAIN valid bit 2: MULTIPLE SWITCH A valid bit 3: MULTIPLE SWITCH B valid bit 4: MULTIPLE SWITCH C valid bit 5: MULTIPLE SWITCH D valid bit 6: JIG OFFSET X valid bit 7: JIG OFFSET Y valid bit 8: JIG OFFSET Z valid
Write start request flag	#50499	Decimal, without decimal point bit 1: Write start request

Note 1: Since system variables #50467 and #50499 are both decimal data, binary bit data must be converted into decimal data when setting.

Example:

To write only UNIT SKIP SWITCH data and MULTIPLE SWITCH A, set the data as follows:

#50467 = 5 (= 000000101)

Note 2: Since other features also use it, system variable #50499 must be set as follows for identification:

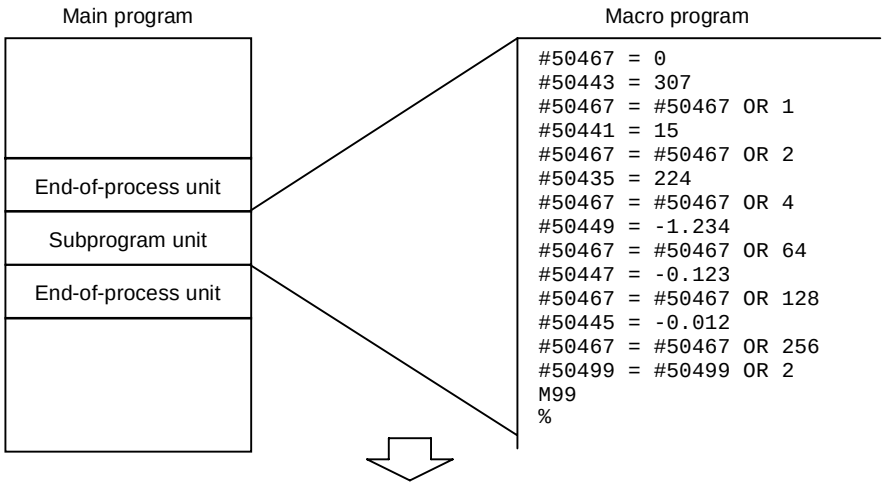
#50499 = #50499 OR 2

Note 3: System variable #50499 must be set at the end of the macro-program.

<Sample: Write macro-program>

```
                                            9 8 7 6 5 4 3 2 1 0
                                              •     • •     • •
UNIT SKIP SWITCH         ON
                                        • : 0 1 0 0 1 1 0 0 1 1 • j=307• Decimal•
MULTIPLE SWITCH          MAIN               0 0 0 0 0 0 1 1 1 1    = 15• Decimal•
MULTIPLE SWITCH          A                  0 0 1 1 1 0 0 0 0 0   =224• Decimal•
JIG OFFSET               X                         -1.234
JIG OFFSET               Y                         -0.123
JIG OFFSET               Z                         -0.012
```

To write data as shown above, make a macro-program for updating the machining management data as follows:



PALLET MANAGEMENT display

```
UNIT SKIP SWITCH
     9 8 7 6 5 4 3 2 1 0
 ON    •     • •     • •
 OFF •   • •     • •

MULTIPLE SWITCH
    10    6  5    1
MAIN 0 0 0 0 0  0 1 1 1 1
  A  0 0 1 1 1  0 0 0 0 0
  B  0 0 0 0 0  0 0 0 0 0
  C  0 0 0 0 0  0 0 0 0 0
  D  0 0 0 0 0  0 0 0 0 0

JIG OFFSET
  X     -1.234
  Y     -0.123
  Z     -0.012
```

3. Machine action

Use the following parameter to select whether the write feature is to be made valid or invalid;

Machine parameter **L52**	Set “0” to make the write feature invalid. Set “1” to make the write feature valid.

Even when the write feature is made valid, writing will be performed only when external CPU operation or scheduled operation (see **Note 2**) is performed. If the write feature is made invalid or if wrong macro-program data is set, writing will not occur (the machine will stop together with the display of an alarm **192 EXECUTION IMPOSSIBLE**).

Note 1: Even when the alarm occurs and writing is not performed, the machine action will not stop if the **MEASURE MACRO** menu key was not pressed when setting subprogram unit for calling the write macro-program.

Note 2: External CPU operation and scheduled operation refer to the following types of operation:

- External CPU operation
- Automatic operation with machine parameter **L54** = 1
  (Pallet-ID-based operation controlled by a CPU external to the NC)
- Scheduled operation
  Schedule-mode-based automatic operation with machine parameter **L54** = 2

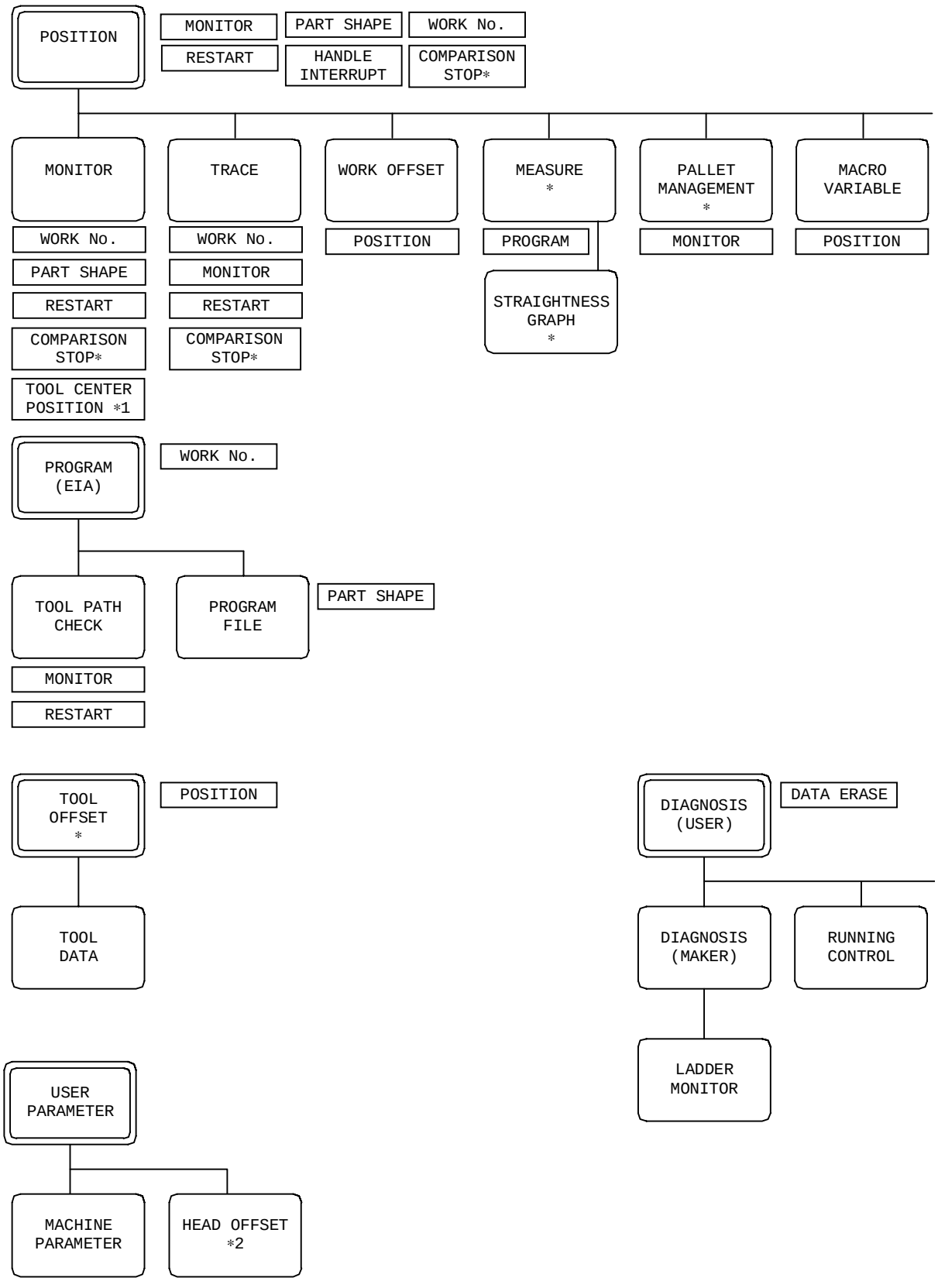
11-2 Function Relating EIA/ISO Program (Option)

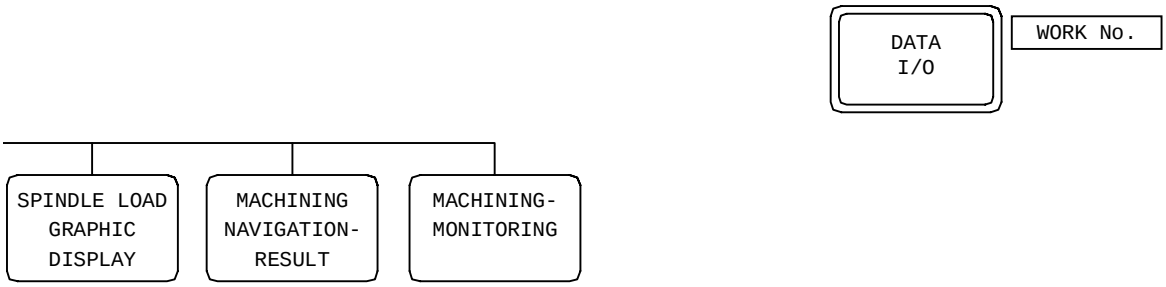
EIA/ISO ONLY menu item will be displayed on the **DIAGNOSIS (ALARM)** if an EIA/ISO programming option is added. Pressing the menu key will reverse the display status of the menu item and modify the display configuration and various data items into ones that can be more conveniently used for EIA/ISO program.
Pressing the menu key once again will clear the reverse-display status of the menu item and return it to the normal status.

11-2-1 Configuration of displays

If the **EIA/ISO ONLY** menu item is reversed, the displays that the NC unit provides are of the configuration shown in the following pages.

Configuration of displays - when **EIA/ISO ONLY** is reversed -





Main display

Sub-display

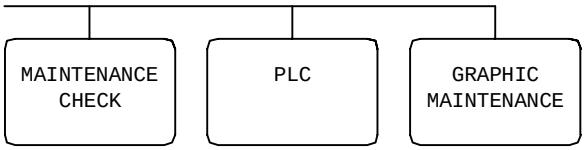
Window

* Optional

The displays connected by the line can be called up mutually by pressing menu key.

*1 Only for machines with multiple-surface machining function

*2 Only for machines with five-surface machining function

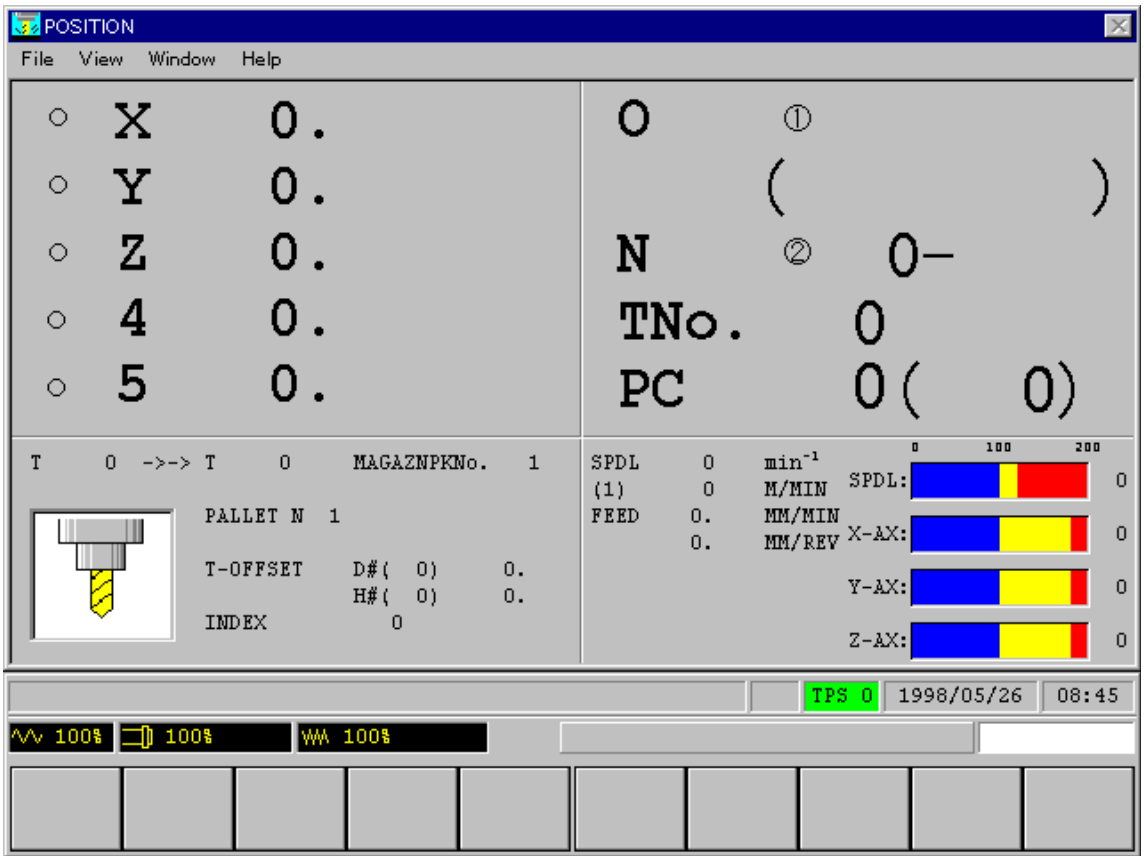


D735S1078

11-2-2 POSITION display

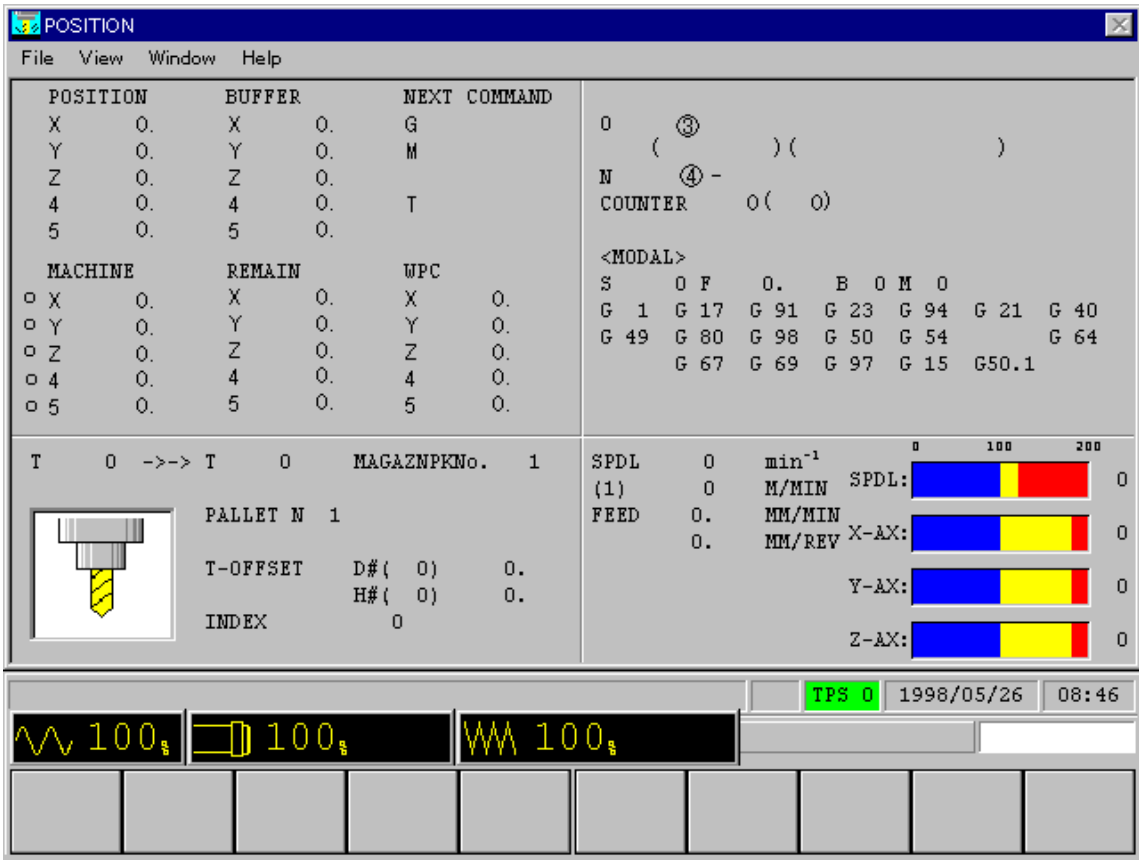
POSITION display is changed as follows when the **EIA/ISO ONLY** menu item is reversed.

1. Displayed data when the COMMAND menu item is not reversed



D735S1079E

2. Displayed data when the COMMAND menu item is reversed



D735S1080E

3. Display data description

No.	Item	Unit	Data description
•	O	•	Work number of the program being executed O **a b** (**a' b'**) **a**, **a'**: Work numbers **b**, **b'**: Identification of MAZATROL or EIA/ISO program (For data items **a'** and **b'**, data is displayed only during execution of subprograms.)
•	N	•	N **a** **a**: Sequence number
•	O	•	Work number and name of the program being executed O **a b c** (**a' b'**) (**c'**) The meaning of data **a**, **b**, **a'** and **b'** is same as in •. Refer to the item •. **c, c'**: Name of the program (For a machine that displays a program name in 48 characters, only the first 16 characters are displayed.)
•	N	•	N **a – a'** **a**: Number of the sequence being executed **a'**: Number of the block being executed

11-3 Workpiece Measurement Printout System

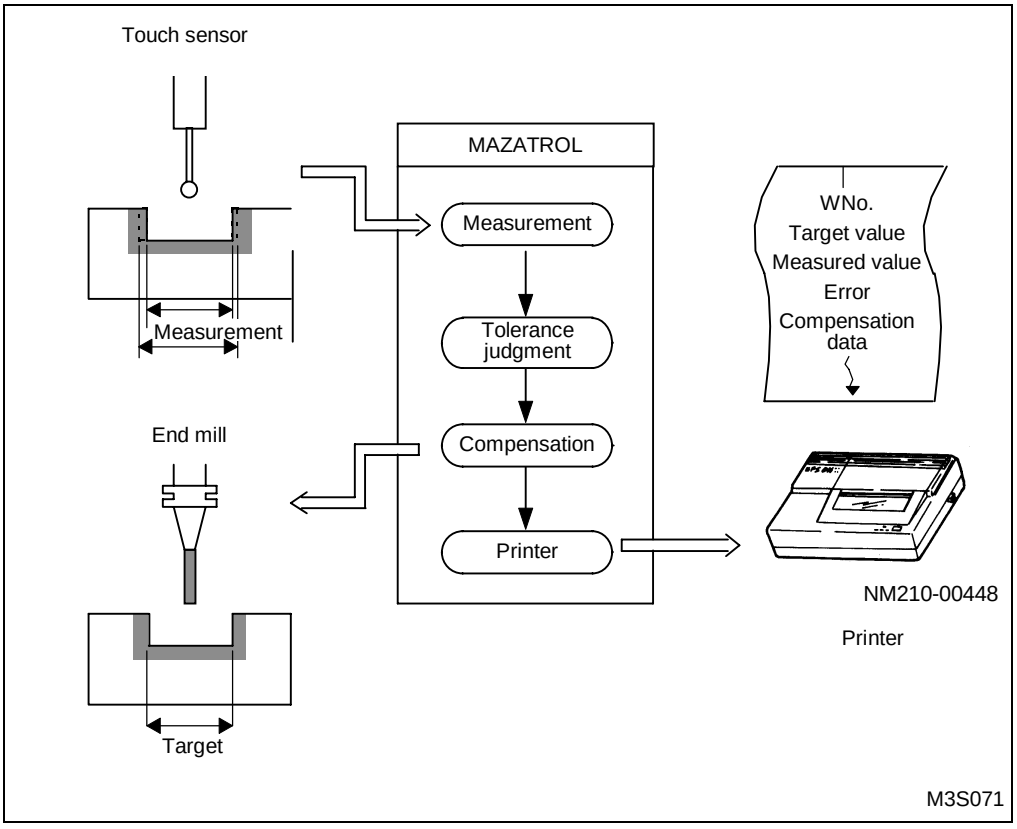
11-3-1 Function overview

The workpiece measurement printout system has been developed to take automatic measurements of hole and boss diameter, groove and step width, center to center distance, step differences and other distances between randomly selected points, angles of slant and the like, and to provide printouts of the results of these measurements.

When the printout is made, the system will compare the actual measured value with the target value and judge whether or not the results fall within the tolerance range. It will also output the results of this judgment. Further, it will also automatically conduct tool data compensation when measurement is made of a surface machined by an end mill.

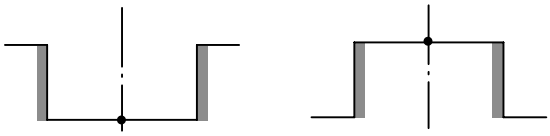
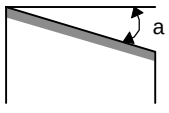
These functions can be used by embedding a designation for calling a macro program for the execution of workpiece measurement printout within normal machining programs that have been written either in MAZATROL programming language or in EIA/ISO code. This type of application will allow the preservation of initial machining accuracy as well as the rapid discovery of any non-conforming parts.

The optional function “EIA/ISO code input” must be present to use this system.



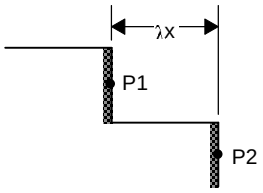
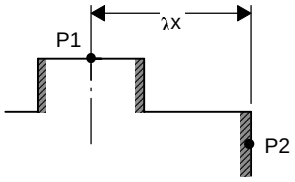
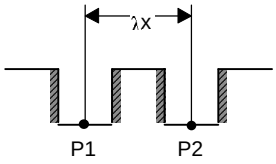
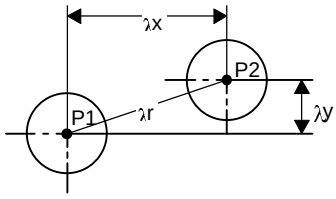
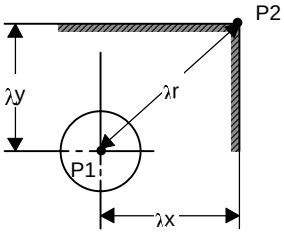
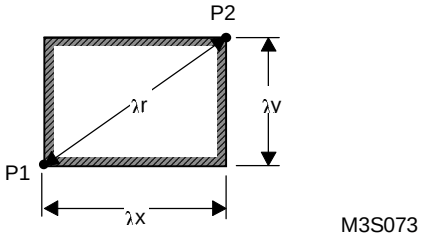
1. Measurement function

Random workpiece lengths or angles are measured by the touch sensor tool. A wide variety of measurement patterns are available for selection. Patterns available are described in the following charts.

Measurement group	Measurement patterns	
A. Diameter/width measurement: Designate center coordinate data and target diameter and width.	ID and OD	 Inner diameter Outer diameter
	Groove and step width	 Groove width Step width
B. Measurement between 2 random points: Any two points can be selected and measurement can be made between them. For surface, groove or step center, hole or boss center or corner measurement. Depending on the type of point combination selected, measurement can be made for step difference, from center to center, and between hole center and surface.	Surface	 X and Y surfaces Z surface
	Groove and step center	 Groove center Step center
	Hole and boss center	 Hole center Boss center
	Outside and inside corners	 Outside corner Inside corner
C. Slant measurements: Measurement of two points on the surface	 Angle of indication	* Angle can be measured on the X-Y surface; the X-Z surface; or the Y and Z surface

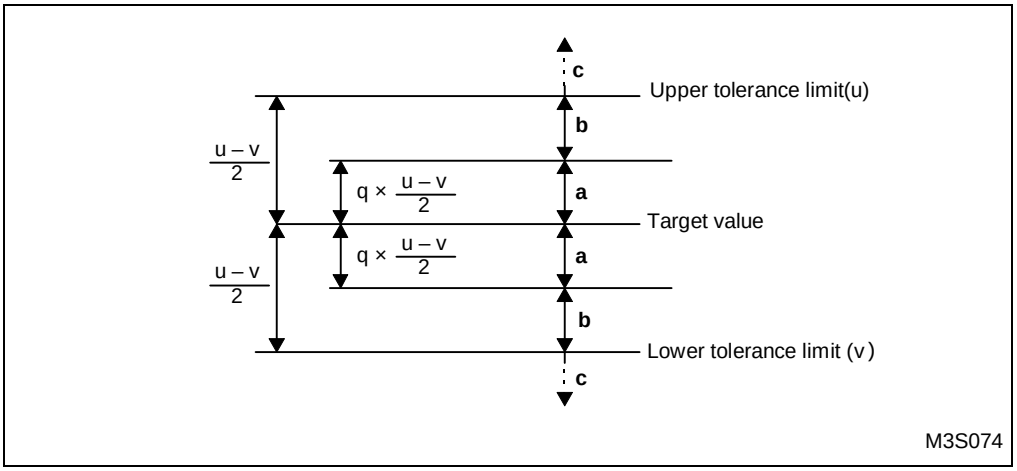
M3S072

Examples of measurement between two randomly selected points:

X-surfcace to X-xurface (step)	Step center X to X-surface
	
Groove center X to groove center X	**Hole center to holde center (distance between centers)**
	
Hole center to outside corner	**Outside corner to outside corner**
	

- The number of machining cycles to be conducted between measurements is designated by a parameter setting.

2. Tolerance judgment function



Tolerance judgment is classified in three stages according to the upper tolerance limit (u), the lower tolerance limit (v) and the compensation range setting rate (q).

Note 1: The value of “q” is designated by an argument in the workpiece measurement initial setting program (WNo. 9100). (Refer to 1. of Subsection **11-3-5**.)

Note 2: The values of “u” and “v” are designated by arguments in the tolerance judgment printout program (WNo. 91• 5). (Refer to 4., 9. and 10. of Subsection **11-3-5**.)

Note 3: Range **a**: Tolerance judgment is “OK” and no compensation is necessary.

Rngee **b**: Tolerance judgment is “OK” but an alarm message will be printed out. If offset tool designation has been made, compensation will be conducted towards the target value.

Range **c**: An alarm will be given, and no compensation will be made. The target value will automatically be established to fall exactly in the middle of the range established between the upper and lower tolerance limits.

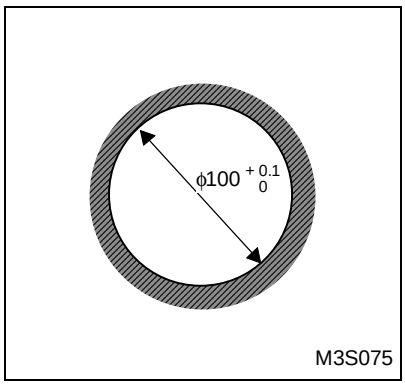
Example:

- ID measurement arguments `X•Y•Z•D100.`
- Tolerance judgment arguments `U0.1V0`

  ⇨ 100.05 is target value.

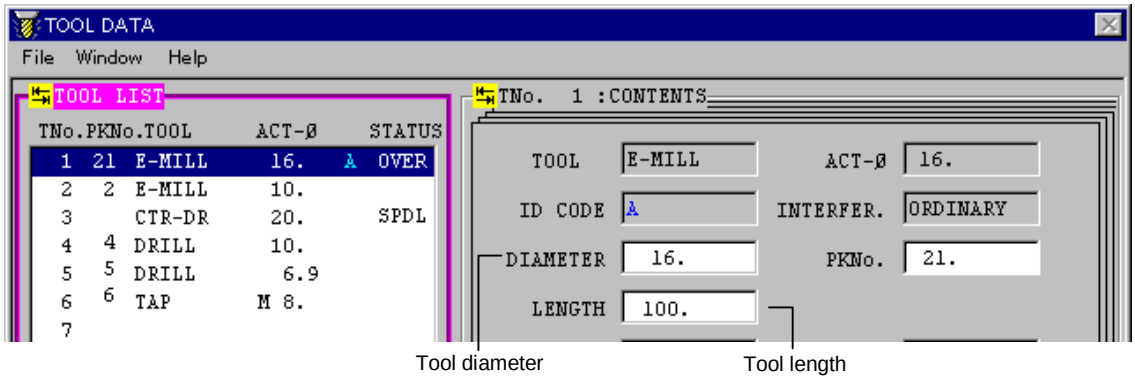
When q = 0.4

c	100.1	(+0.05)
b	100.07	(+0.02)
a	100.05	(Targer value)
a	100.03	(–0.02)
b	100.	(–0.05)
c		

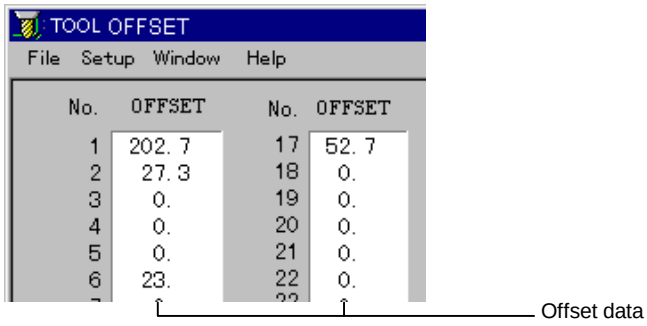


3. Compensation function

TOOL DATA display



TOOL OFFSET (TYPE A) display



Designation of the tool number or the offset number of the end mill that was used on the measured surface allows compensation to be made in the relevant tool diameter, tool length and offset data. Compensation for OFFSET DATA can not be made when setting TOOL OFFSET in type B. Designation of whether compensation is to be made to the TOOL DATA value or to the TOOL OFFSET DATA value is made by parameter setting.

1. Diameter and width measurement
   Compensation direction is determined automatically. The system compares the target value with the measured value, and makes compensation along the minus direction if undercutting has been detected, and compensation along the positive direction if overcutting has been detected.
2. Measurement between 2 randomly selected points
   The compensation direction cannot be determined automatically, and it is thus necessary to supply an argument that can be used for selection. If no argument has been supplied, compensation will be made along the positive direction. See the explanations of the various programs for further details.
3. Slant measurement
   The compensation function is not available for this type of measurement.

4. Printout function

The printout function can be controlled in any of three different ways, depending on parameter settings. These are indicated below:

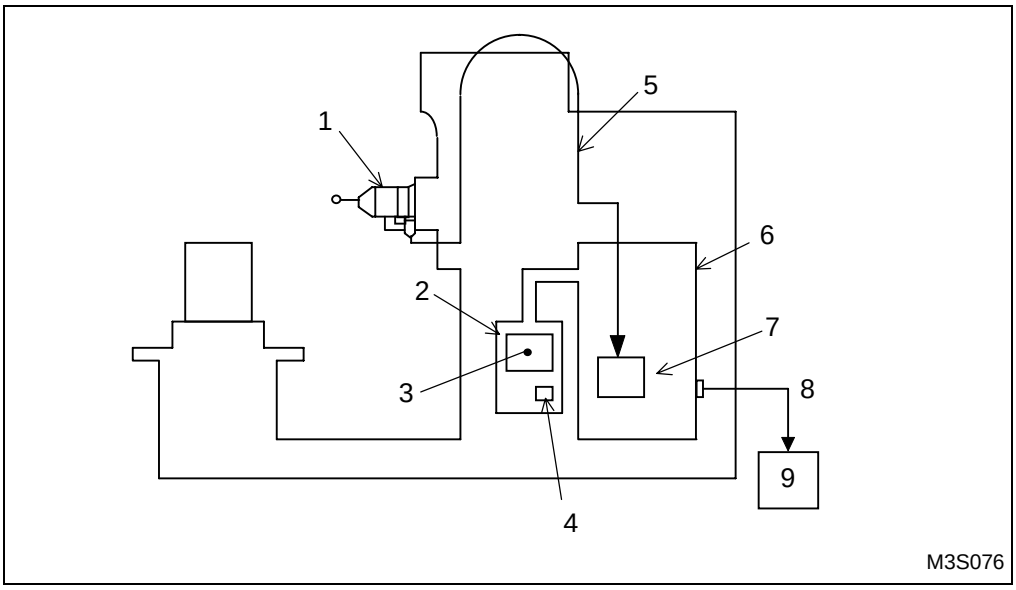
1. Printout not required
   This selection is made when the system is to be operated with the measurement, tolerance judgment and compensation functions only, or when printer failure or some other reason makes it impossible to use the printer. Measurement results can be verified at the **COMMON VARIABLE** display even if no printout is to be made.
2. Printout each measurement
   This selection should be made if it is necessary to make a printout of every measurement taken, regardless of results.
3. Printout compensation
   This selection should be made if it is necessary to print out only the results of tolerance judgments when error has been found to fall within the compensation range or to exceed tolerance. If this selection is made, however, the results of the first measurement taken will be printed out regardless of results. (This selection is controlled by the results of tolerance judgments, and has no relationship to whether the offset tool designation is made or not).

A maximum of eight items can be included in the printout for a single measurement.
An example follows. (Example assumes metric input and surface point to hole/boss center point measurement.)

```
                                /RESULT OF MEASURING
1) Work number, workpiece count ..... WNO 5678 COUNT 1
2) Target value, measurement pattern ..... MARK DATA FACE-HOL/BOS 60.000
3) Measured value, direction of measurement ..... MEASURED DATA X 60.053
4) Tolerance range ..... TOLERANCE 0.100
5) Error, tolerance judgment results ..... CHECK OK/WARNING 0.053
6) Compensation tool No. or compensation offset No. ..... OFFSET NO. 30
7) Pre-compensation tool data or offset data ..... OFFSET DATA 10.000
8) Post-compensation tool data or offset data ..... NEW OFFSET DATA 9.947
```

11-1-2 System configurations

1. Hardware



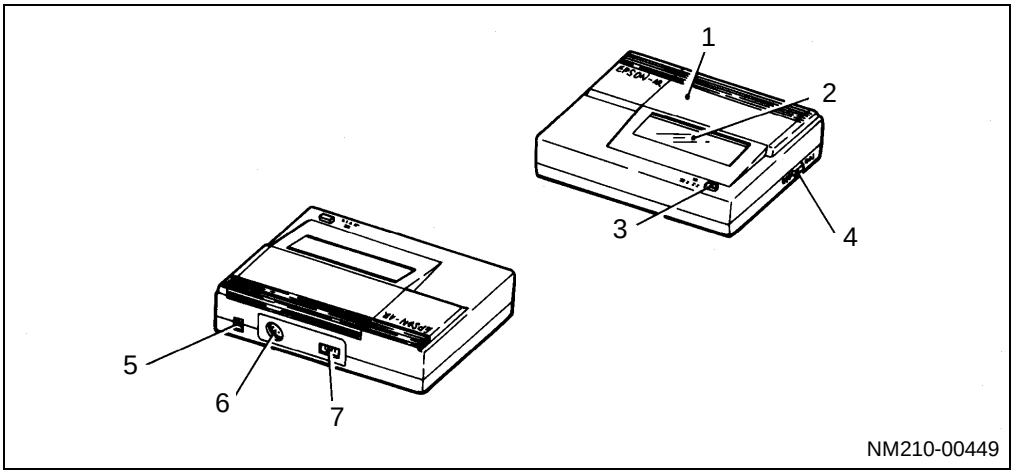
No.	Part name	No.	Part name
1	Touch sensor tool	6	NC unit (MAZATROL)
2	Pendant operation box	7	MMS receiver
3	LCD display	8	Printer cable
4	Signal check panel	9	Printer
5	Signal cable		

The workpiece measurement printout function is composed of the following components: a touch sensor tool that is brought into contact with the measurement surface; a MAZAK measuring system (MMS) receiver that receives contact signals from the touch sensor; the MAZATROL NC unit, which reads coordinate values when contact is made between the touch sensor and the measurement surface; and a printer, which prints out the results of the measurement.
See the Programming Manual, for further details on the touch sensor tool.

2. Printer

A. Printer for this system

The printer used in this system is an Epson P-40 printer, which is interfaced with the NC unit using PTP/PTR connector.



No.	Part name	No.	Part name
1	Printer cover	5	Power connector
2	Paper cutter	6	Interface connector
3	PF switch	7	DIP switches
4	Power switch		

B. Principal specifications

Item	Description
Type of printing	Heat sensitive serial dot matrix printer
Printing speed	45 cps (for normal characters in 40 col. mode)
Lines per second	Approximately 1.2 lines per second
Paper for use	Epson thermal roll paper (P40TRP), thermal roll paper (luminous black); width 112 mm/ roll length 11 meters
Power supply	Internal..... UN3 NiCd batteries External..... AC adaptor available
Dimensions and weight	46 (H) × 216 (W) × 128 (D) mm, 650 g

See the printer operating manual for a more detailed specifications.

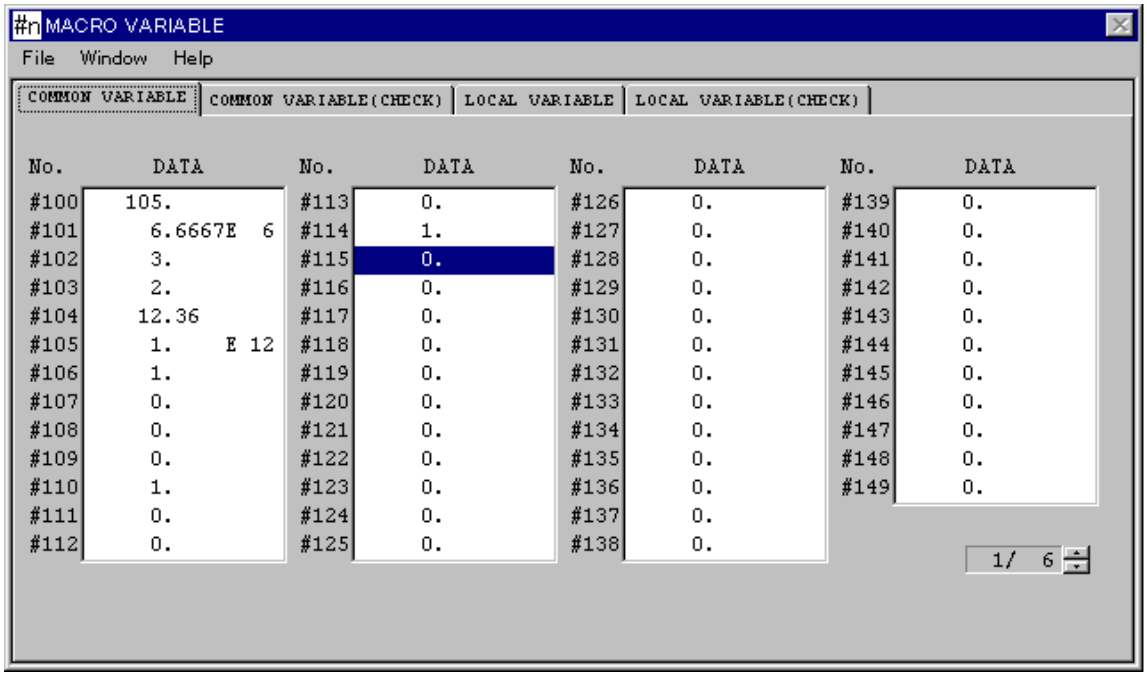
C. Setting of printer DIP switches and NC parameters

1. Set the following NC detailed parameters as required:
   - **DPR1** to **DPR16**
2. Set the printer DIP switches following the printer operation manual.
   - Data length: 7 bit
   - Parity: EVEN
   - Baud rate: Set the same value as that of NC parameter **DPR1**.

11-1-3 Parameter registration

Before attempting to utilize the workpiece measurement printout function, it is necessary to first register in the common variables the parameter settings which designate the data management system to be used and the compensation amount of touch sensor tool. For data management system parameters, refer to 2. of this subsection.

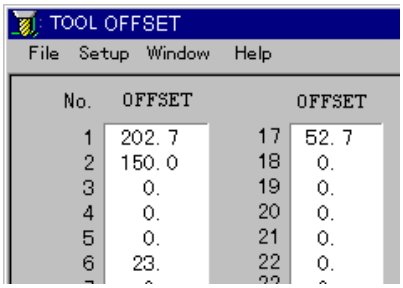
1. COMMON VARIABLE display



D735S1081E

Set the parameters for data management system on the **COMMON VARIABLE** display.
Refer to Section **5-3 MACRO VARIABLE Display** for display data description and setting method.

2. Data management system parameters

Item	Address	Value to register
Mesurement cycle	#500	Register number of cycles in which measurement is to be made. If #500 is set at “0”, no measurement will be made at all.
Printout categories	#501	0. = No printout 1. = Printout each measurement 2. = Printout first measurement, then subsequent measurements if compensation is made or tolerance level is exceeded
Compensation categories	#502	0. = Conduct tool data compensation 1. = Conduct tool offset data compensation 2. = Conduct tool data compensation for actual tool (***Note**)
Touch sensor tool number	#503	Tool number registration of touch sensor tool **Example:** Tool No. 15 : #503 = 15.
Offset number	#504	Set a touch sensor tool length offset register No. **Example:** Set a touch sensor tool length offset register No. at macro variable #504, and input the touch sensor tool length in the number. (Input value may be the same as the touch sensor tool length given on the **TOOL DATA** display.) **TOOL OFFSET** display  If offset register No. 2 is used for a touch sensor tool length of 150, set the value 150 as shown above, and besides set offset No. 2 at macro variable #504. #504 = 2.
MAZATROL tool data Effective/Ineffective	#510	When **F93** bit 3 (MAZATROL tool data tool length Effective) is made effective, set #510 = 1. Set #510 = 0. except the above.
ATC type	#511	0 ... H-400 type Model which requires 2nd zero point return (G30) for ATC 1 ... FH type Model which doesn't require 2nd zero point return (G30) for ATC
Touch sensor tool nominal radius	#554	Assuming nominal diameter of 5 mm: Metric .....#554 = 2.5 Inch .....#554 = 0.0984

Refer to subection **7-1-11** for MAZATROL tool data tool length/tool diameter Effective (in EIA/ISO program).

F92 bit 7 ON: MAZATROL tool data tool diameter Effective

F93 bit 3 ON: MAZATROL tool data tool length Effective

F94 bit 2 ON: Tool length compensation cancel is not performed by G28, G30.

(When **F93** bit 3 is set to ON, be sure to set it to “compensation cancel is not performed”.)

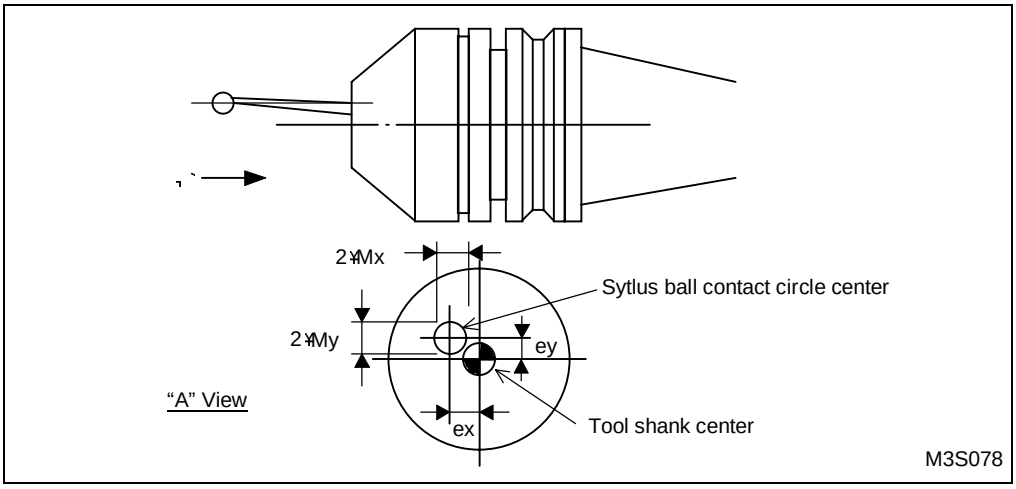
F84 bit 0 ON: Tool offset value involved into the counter during EIA/ISO.

***Note:** If the life of the actual tool is expired, tool data compensation is executed to the spare tool. By using bit 7 of parameter **F80**, it can be selected, which of both tool data compensations is executed:

0: Compensation for tool length/tool diameter (MAZATROL tool data)

1: Compensation for tool data (EIA/ISO)

3. Touch sensor tool compensation amount



Eccentricity compensation values “ex” and “ey” and measurement compensation values “Mx” and “My” must be registered as common variables before any attempt is made to use the touch sensor to measure a workpiece.
Execution of calibration measurement by the MAZATROL MMS (automatic centering system) will cause these compensation amounts to be displayed as “**MACHINE PARAMETER No. 3**” at the parameter display. Input the values displayed at addresses **L1** through **L4** as common variables #550 through #553, respectively.

Compensation amount	Parameter for mesurement			Common variable		
	Address	Metric	Inch	Address	Metric	Inch
Eccentricity compensation amount ex	L1	–58	–23	#550	–0.0058	–0.0023
Eccentricity compensation amount ey	L2	1690	67	#551	0.169	0.0067
Measurement compensation amount Mx	L3	24150	9510	#552	2.415	0.0951
Measurement compensation amount My	L4	24277	9590	#553	2.437	0.0959

Refer to the Programming Manual (MAZATROL) for further information on the calibration measurement operation.

11-1-4 Program configuration

The workpiece measurement printout function is used by adding a macroprogram call command to the main program. Before this program is called out, however, it is necessary to designate the workpiece coordinate system.
Tool exchange commands for the touch sensor are executed by the workpiece measurement initial setting program, and they do not have to be added to the main program.

1. Coordinate system setting

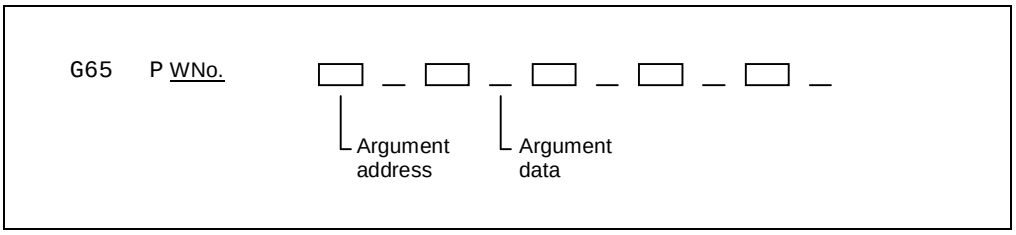
Coordinate system setting for workpiece measurement should be made according to the procedures used with the MAZATROL programming language. In other words, the coordinate value for the Z-axis should be the distance from the spindle nose at the machine zero-point and the length of the touch sensor tool should be registered as tool data.

EIA/ISO program: Set with codes G54 through G59 and G92.

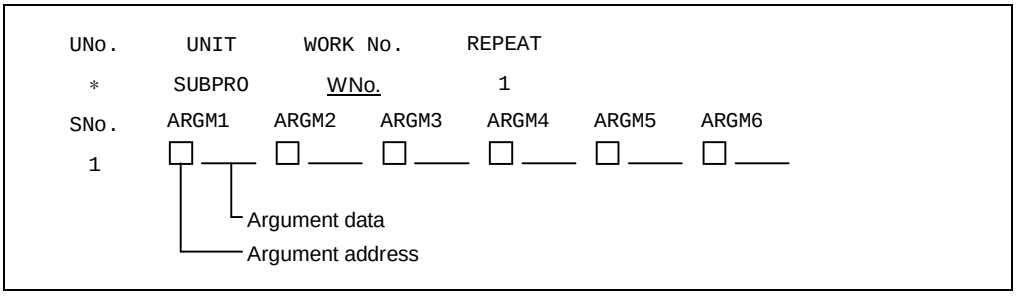
MAZATROL program: Set with basic coordinate unit. However, "ϕ" values will not be effective. It should also be noted that the miscellaneous coordinate unit will also be ineffective.

2. Macro program calls

- EIA/ISO program: Use G65 user macro unmodal call.



- MAZATROL program: Use subprogram unit.



The argument data are effective down to four decimal places. Any fraction will be rounded off to four decimal places.

3. Macro program

The workpiece measurement printout system is made up of the 15 macroprograms indicated in the following table. The program necessary for the measurement pattern to be executed should be selected.

Group	Work No.	Type of measurement	Argument address
Initial setting	9100	Workpiece measurement initial setting	WRKQ
Measurement A (Diameter and width)	9110	ID/OD measurement	MXYZD
	9111	Groove/step width measurement	MXYZD
	9115	Tolerance judgment printout A	UVS
Measurement B (Measurement between two random points)	9120	Surface measurement	MXYZ
	9121	Groove center/step center measurement	MXYZD
	9122	Hole center/boss center measurement	MXYZD
	9123	Corner measurement	MXYZI
	9125	Tolerance judgment printout B	EUVSB
Measurement C (Slant)	9130	Slant measurement	FMXYZ
	9135	Tolerance judgment printout C	IAUV
Sub-routines	9140	Data check sub-routine	••
	9141	Measurement movement sub-routine	••
	9142	Tolerance judgment and compensation sub-routine	••
	9145	Printout sub-routine	••

The initial setting program (Work No. 9100) is always necessary when using the work measurement function. Select the desired measurement pattern from measurement groups A, B or C. The tolerance judgment printout program (Work No. 91•5) should be the one from the group from which the measurement pattern has been selected. Sub-routines are used within the system, and cannot be directly used in a user program.

A. Diameter and width measurement

Measurement Group A

9100 ..... Workpiece measurement initial setting
9110, 9111 ..... Diameter or width measurement
9115 ..... Tolerance judgment printout A

B. Measurement between 2 random points

Measurement Group B

9100 ..... Workpiece measurement initial setting
9120, 9121, 9122, 9123 ..... Measurement point 1 coordinate value
9120, 9121, 9122, 9123 ..... Measurement point 2 coordinate value
9125 ..... Tolerance judgment printout B

A total of 10 different combinations is possible for workpiece numbers 9120 through 9123, depending on the designation of arguments at measurement points 1 and 2.

No.	Program combination	Measurement pattern	Direction	Compensation setting (**Note 1**)
1	9120 - 9120	Surface to surface	X, Y, Z	•
2	9120 - 9121	Surface to groove/step center	X, Y	•
3	9120 - 9122	Surface to hole/boss center	X, Y	•
4	9120 - 9123	Surface to corner	X, Y	•
5	9121 - 9121	Groove/step center to groove/step center	X, Y	×
6	9121 - 9122	Groove/step center to hole/boss center	X, Y	×
7	9121 - 9123	Groove/step center to corner	X, Y	•
8	9122 - 9122	Hole/boss center to hole/boss center	X, Y, R (**Note 2**)	•
9	9122 - 9123	Hole/boss center to corner	X, Y, R (**Note 2**)	•
10	9123 - 9123	Corner to corner	X, Y, R (**Note 2**)	•

Note 1: The following symbols are used in this chart:
- • : All measurement systems can be compensated
- • : Center measurement taken as standard for compensation
- × : Compensation designation not possible

Note 2: X, Y and R ($=\sqrt{X^2 + Y^2}$) tolerance judgment printouts can be selected for each measurement at Nos. 8, 9 and 10. (More than one printout is also possible.)

Measurement Group B

9100.....Workpiece measurement initial setting

9122, 9123.....Measurement point 1; hole/boss center or corner coordinate values measurement

9122, 9123.....Measurement point 2; hole/boss center or corner coordinate values measurement

9125 (X).....X-axis direction tolerance judgment printout B

9125 (Y).....Y-axis direction tolerance judgment printout B

9125 (R).....Distance on X-Y plane $\sqrt{X^2 + Y^2}$ tolerance judgment printout B

C. Slant measurements

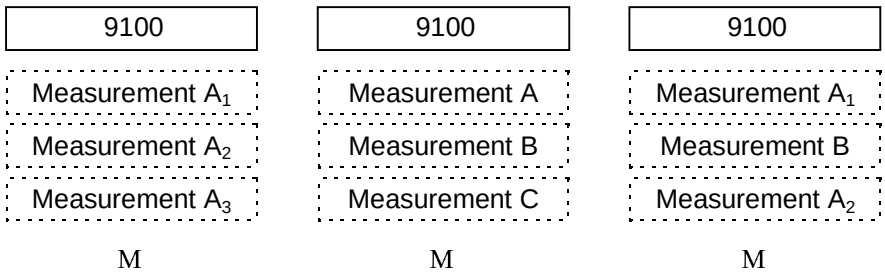
Measurement Group C

9100.....Workpiece measurement initial setting

9130, 9135.....Slant measurement; tolerance judgment print-out C (always pair these two)

D. Multiple group measurements

- It is possible to designate measurements to be made from each group (A, B, and C) within the same main program. In such cases, the workpiece measurement initial setting program is necessary only once, at the beginning.



Workpiece measurement will be discontinued if an alarm is generated as a result of tolerance judgment.

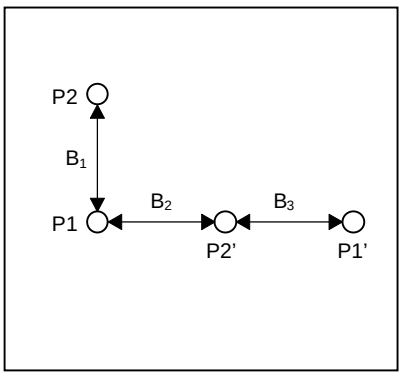
- Group B measurements are normally composed of a cycle containing three programs:

1) Measurement point 1 coordinate value measurement
2) Measurement point 2 coordinate value measurement
3) Tolerance judgment printout

When more than one location is being measured, however, and one measurement point is to be duplicated, it is possible to make tolerance judgments from only one coordinate value measurement from the second measuring cycle on.

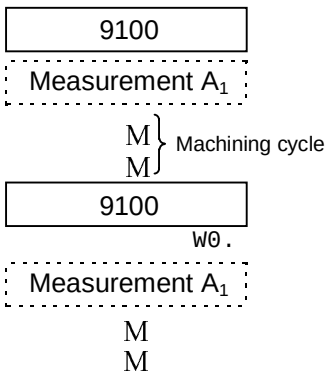
B_1
- 9100 ¥¥¥¥¥¥¥¥¥¥ Measurement point 1 (P1) coordinate value measurement
- 9120 - 9123 ¥¥¥¥ Measurement point 2 (P2) coordinate value measurement
- 9120 - 9123 ¥¥¥¥ Measurement point 2 (P2)
- 9125 ¥¥¥¥¥¥¥¥¥¥ Printout of toelerance judgment between points P1 and P2

B_2
- 9120 - 9123 ¥¥¥¥ Measurement point 2 (P2') coordinate value measurement
- 9125 ¥¥¥¥¥¥¥¥¥¥ Printout of toelerance judgment between points P1 and P2'



B_3
- 9120 - 9123 ¥¥¥¥ Measurement point 1 (P1') coordinate value measurement
- 9125 ¥¥¥¥¥¥¥¥¥¥ Printout of toelerance judgment between points P1' and P2'

- When machining is executed between measuring cycles:



WNo. 9100 includes the touch sensor automatic tool change function as well as the workpiece count function, and for this reason argument "Ww" (normally this is the workpiece number) should be set at "0" for the second and all subsequent instances of 9100, so that the count function will not be executed. Other arguments are unnecessary if they are unchanged from the initial 9100 appearance, although it is also possible to change their settings.

E. Examples for calling up a macroprogram

EIA/ISO programs

```
〈
G54 Coordinate system setting
〈
```

Workpiece offset G54
X -360.
Y –450.
Z -600.
4 0

```
G65P9100W1200.R5.K3.Q0.5          ---------  Workpiece measurement initial setting
G65P9110M101.X200.Y200.Z-5.D100.  ---------  Measurement (example is ID)
G65P9115U0.1V-0.1S2.              ---------  Tolerance judgment, printout
〈
```

Note:
G90/G91 and F-code modal information is updated according to workpiece measurement.

MAZATROL program language program

```
〈
UNo.    UNIT    X      Y      th     Z      4
**      WPC - •  –360.  –450.  θ0     –600.  0
〈

UNo.    UNIT      WORK No.  REPEAT
**      SUBPRO    9100E     1
SNo.    ARGM 1    ARGM 2    ARGM 3  ARGM 4   ARGM 5   ARGM 6
1       W1200.    R5.       K3.     Q0.5
UNo.    UNIT      WORK No.  REPEAT
**      SUBPRO    9110E     1
SNo.    ARGM 1    ARGM 2    ARGM 3  ARGM 4   ARGM 5   ARGM 6
1       M101.     X200.     Y200.   Z-5.     D100.
UNo.    UNIT      WORK No.  REPEAT
**      SUBPRO    9115E     1
SNo.    ARGM 1    ARGM 2    ARGM 3  ARGM 4   ARGM 5   ARGM 6
1       U0.1      V-0.1     S2.
```

11-1-5 Explanations of macro programs

1. Workpiece measurement initial setting (WNo. 9100)

This program is responsible for the designation of common data to be used in workpiece measurement, the initializing of common variables, workpiece counting management and the automatic tool change operation for the touch sensor. It should always be called out immediately before the beginning of any measuring movement.

EIA/ISO	G65	P9100	Ww	Rr	Kk	Qq	
MAZATROL	UNo. * SNo. 1	UNIT SUBPRO ARGM 1 Ww	WORK No. 9100E ARGM 2 Rr	REPEAT 1 ARGM 3 Kk	ARGM 4 Qq	ARGM 5	ARGM 6

Ww: Work number of main program

Example: 4078 Printout

```
WNO 4078 COUNT 10
MARK DATA HOL/BOS DIA 100,000
MEASURED DATA R 100.799
```

However, if this is used in two or more locations in the main program, the setting for the second and all subsequent times should be “W=0”.

- With the exception of cases in which “W = 0” the counter will advance by 1 each time the program is called up, and the counter will automatically be cleared to 0 when the work number (value “w”) changes. If it is necessary to clear the counter to 0 without changing work numbers, set common variables #135 and #136 to 0.
- “Ww” designates a work number, but no check is made to ensure that this does not differ from the actual work number.
- This is used in the heading section of the printout, so it is also possible to use a different code setting at the user's discretion.

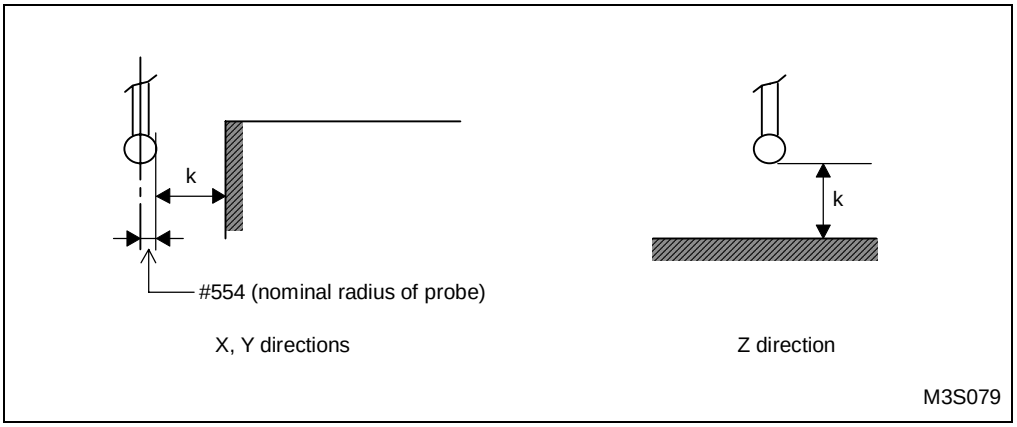
Rr: Initial point Z used for measurement movement (workpiece coordinate value)

- After an automatic tool change has been conducted for the touch sensor, positioning will be made to the point designated by “Rr” and after this is done feed will be made at a rate of 3000 mm/min (or 120 inch/min). The measurement feed rate is made at a parameter setting (**K41**: G31 skip feed). Because measurement is normally made successively, after one measurement cycle has been completed, return is made to point R, which means that a zero return command should be made for the final measurement (see the various measurement programs).

Kk: Measurement feed amount (positive value)

- Measurement feed amount “Kk” is not calculated from the center of the spindle, but rather represents the distance from the tip of the probe to the surface to be measured.

This data is not effective during slant measurement.



When performing slant measurement, set the measurement start point in such a way that it will be within 15 mm (0.6 inch) of the measurement point.

Qq: Compensation field setting rate (0 to 1.0)
Set ratio of the range that will not result in compensation conducted, with respect to tolerance width 1.0.

Note: All argument data should contain decimal points.

2. Inner diameter and outer diameter measurements (WNo. 9110)

Diameters are calculated by the measurement of four points on a circle. Measurement is made first along the X-axis, then along the Y-axis.

EIA/ISO	G65	P9110	Mm	Xx	Yy	Zz	Dd
MAZATROL	UNo.	UNIT	WORK No.	REPEAT			
	*	SUBPRO	9110E	1			
	SNo.	ARGM 1	ARGM 2	ARGM 3	ARGM 4	ARGM 5	ARGM 6
	1	Mm	Xx	Yy	Zz	Dd	

Mm: Measurement pattern:

1) m = 1.: ID measurement
2) m = 2.: OD measurement
3) m = 3.: ID measurement II

If a zero point return operation is to be made after all measurement operations have been completed, these values should all be added “100.” and, then, set as follows:
m = 101.; m = 102.; m = 103.

Xx: Center point of circle, X coordinate (workpiece coordinate value)

Yy: Center point of circle, Y coordinate (workpiece coordinate value)

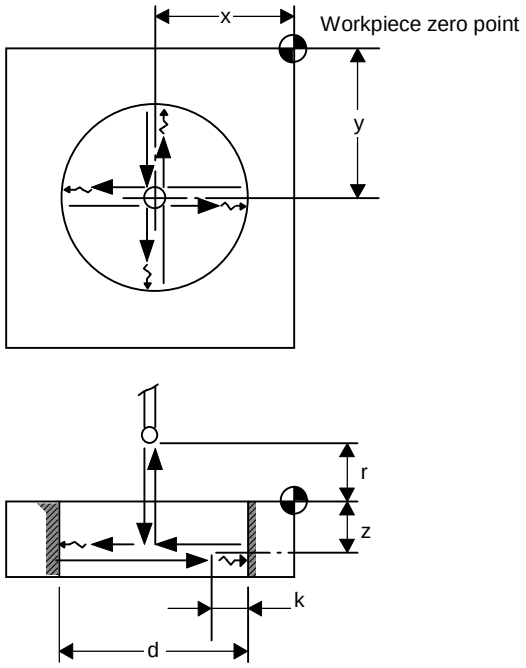
Zz: Measurement point, Z coordinate (workpiece coordinate value)

Dd: ID/OD target value

All argument data should contain decimal points.

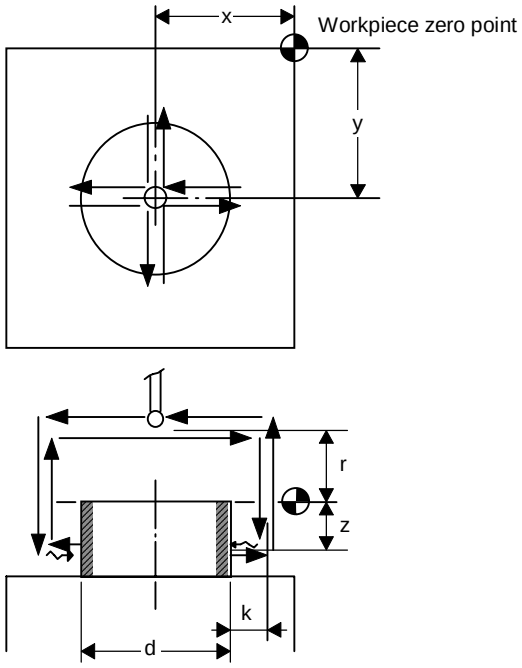
[Measurement operations]

m = 1.	ID Measurement



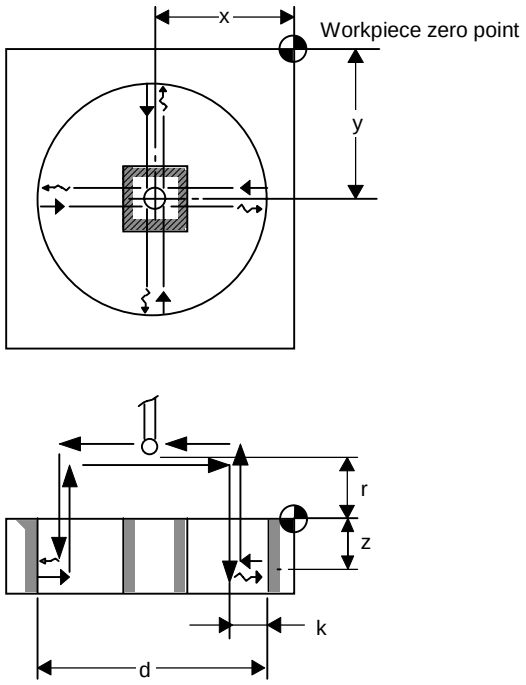
M3S082

m = 2.	OD measurement



M3S083

m = 3.	ID Measurement II



M3S084

- Measurement pattern is determined in accordance with the value of argument “m” as shown in the accompanying diagrams.
- The values of “r” and “k” are designated in the workpiece measurement initial setting program (WNo. 9100).
- ID measurement II is used when there is a step within a circle, such as in PCKT MT machining.

3. Groove width and step width measurements (WNo. 9111)

Groove and step width are calculated by measuring both ends of the groove or step section (along either the X-axis or the Y-axis direction)

EIA/ISO	G65	P9111	Mm	Xx	Yy	Zz	Dd
MAZATROL	UNo. * SNo. 1	UNIT SUBPRO ARGM 1 Mm	WORK No. 9111E ARGM 2 Xx	REPEAT 1 ARGM 3 Yy	 ARGM 4 Zz	 ARGM 5 Dd	 ARGM 6

Mm: Measurement pattern:

1) m = 1.: X-axis groove width measurement
2) m = 2.: Y-axis groove width measurement
3) m = 3.: X-axis step width measurement
4) m = 4.: Y-axis step width measurement

If a zero point return operation is to be made after all measurement operations have been completed, add “100” to these values and, then, set as follows:
m = 101.; m = 102.; m = 103; m = 104.

Xx: When m = 1. or 3.: Center of groove/step, X coordinate (workpiece coordinate value)
When m = 2. or 4.: Measurement point X coordinate (workpiece coordinate value)

Yy: When m = 1. or 3.: Measurement point Y coordinate (workpiece coordinate value)
When m = 2. or 4.: Center of groove/step, Y coordinate (workpiece coordinate value)

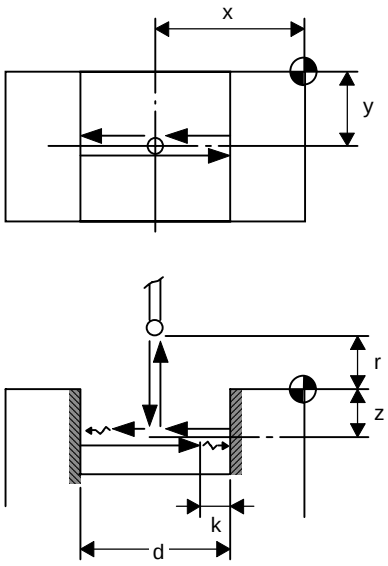
Zz: Measurement point, Z coordinate (workpiece coordinate value)

Dd: Groove or step width target value

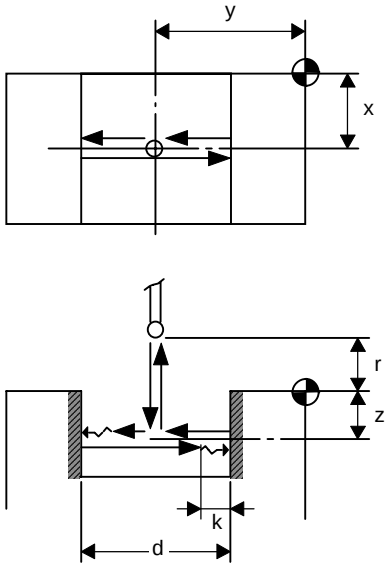
All argument data should contain decimal points.

[Measurement operations]

m = 1.	X-axis groove width measurement	m = 2.	Y-axis groove width measurement

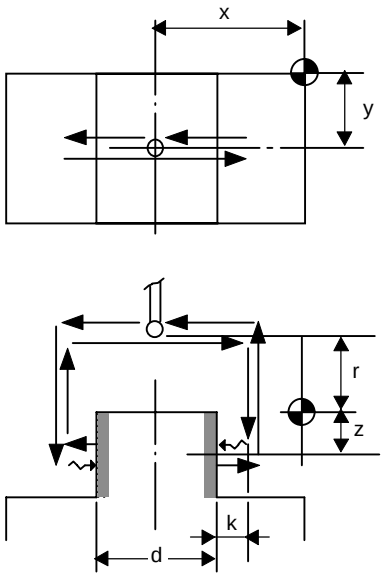


M3S085

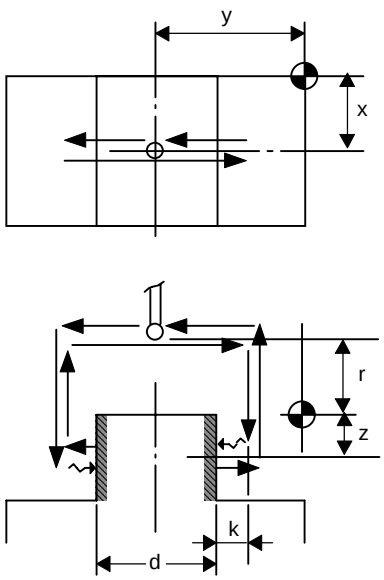


M3S085

m = 3.	X-axis step width measurement	m = 4.	Y-axis step width measurement



M3S085



M3S085

- Values of “r” and “k” are designated in the workpiece measurement initial setting program (WNo. 9100).

4. Tolerance judgment printout A (WNo. 9115)

This program compares the results of the measurements of inner diameters, outer diameters, groove widths and step widths with the target values; makes a tolerance judgment on the basis of this comparison, and prints out the results of this judgment. Tool diameter compensation can also be made, depending on the amount of error in measured values in respect to the target values.

EIA/ISO	G65	P9115	Uu	Vv	Ss			
MAZATROL	UNo.	UNIT	WORK No.	REPEAT				
	*	SUBPRO	9115E	1				
	SNo.	ARGM 1	ARGM 2	ARGM 3	ARGM 4	ARGM 5	ARGM 6	
	1	Uu	Vv	Ss				

Uu: Upper tolerance limit

Vv: Lower tolerance limit

If measurement results are to be printed out without the performance of a tolerance judgment, the setting of these first two values should be “u = v = 0”.

Ss: The Tool No. or Offset No. for which compensation is to be made.

- Designation in a case where compensation category #502 = 0. and tool number is to be set:
  Example:
  Tool No. 20 • S = 20.
- Designation in a case where compensation category #502 = 1. and offset number is to be set:
  Exemple:
  Offset No. 20 • S = 20.
- Designation in a case where compensation category #502 = 2. and tool number or group number is to be set:
  Select tool number or group number using bit 4 of parameter **F94.**
- If compensation is unnecessary or not possible, make no input for “Ss” or set at “S = 0”.

5. Surface measurement (WNo. 9120)

This program allows measurement of a surface workpiece coordinate. It is combined with other measurement patterns to calculate distance to the surface.

EIA/ISO	G65	P9120	Mm	Xx	Yy	Zz	
MAZATROL	UNo. * SNo. 1	UNIT SUBPRO ARGM 1 Mm	WORK No. 9120E ARGM 2 Xx	REPEAT 1 ARGM 3 Yy	 ARGM 4 Zz	 ARGM 5	 ARGM 6

Mm: Measurement point categories, measurement pattern (designated with three digits:•••).

- Hundreds integer: Establishes return point category
  0: R-point return after measurement operation is completed
  1: Zero-point return after measurement operation is completed
- Tens integer: Establishes measurement point category
  1: Measurement point 1
  2: Measurement point 2
- Ones integer: Establishes measurement pattern
  1: X surface measurement, measurement feed +X
  2: X surface measurement, measurement feed –X
  3: Y surface measurement, measurement feed +Y
  4: Y surface measurement, measurement feed –Y
  5: Z surface measurement, measurement feed –Z

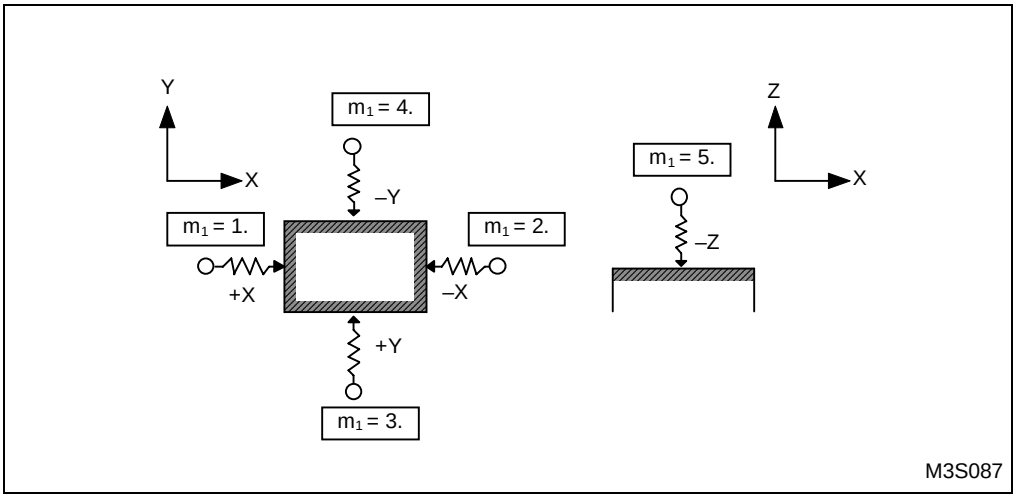
Xx: Measurement point X coordinate (workpiece coordinate value)

Yy: Measurement point Y coordinate (workpiece coordinate value)

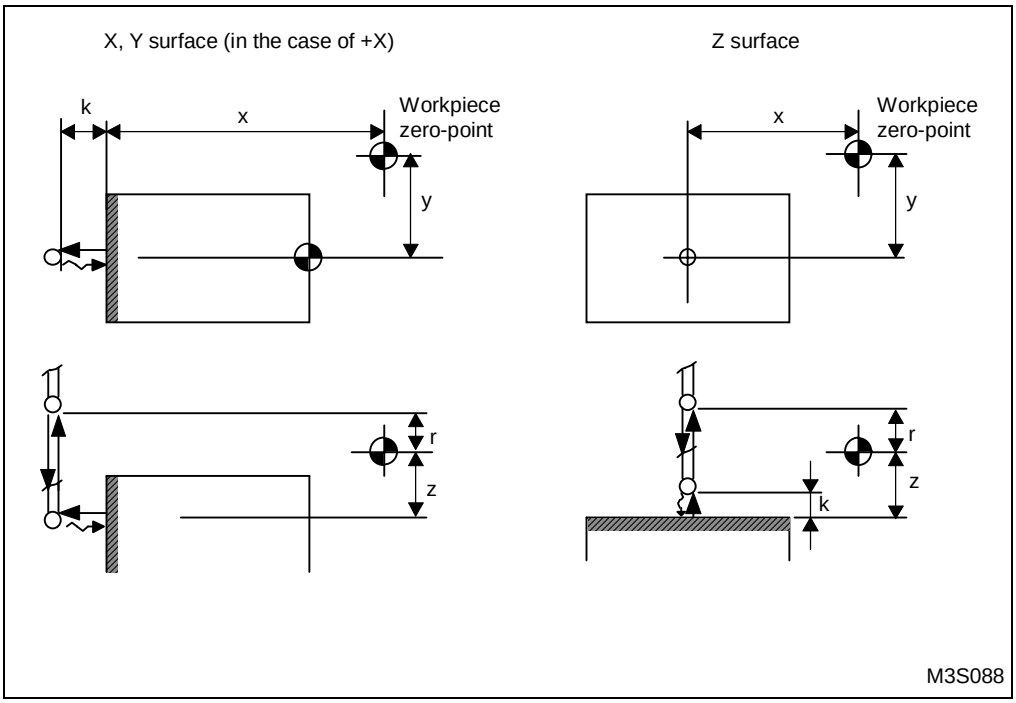
Zz: Measurement point Z coordinate (workpiece coordinate value)

All argument data should contain decimal points.

[Measurement patterns (“m_1” corresponds to the ones integer of argument m)]



[Measurement operations]



Values of “r” and “k” are designated in the workpiece measurement initial setting program (WNo. 9100).

6. Groove center and step center measurement (WNo. 9121)

Ths program measures both edges of grooves and step sections (in the X-axis or Y-axis direction) and calculates the center coordinates of the groove or step section from these measurements. It is combined with other measurement patterns to calculate the distance to the groove or step section center.

EIA/ISO	G65	P9121	Mm	Xx	Yy	Zz	Dd
MAZATROL	UNo.	UNIT	WORK No.	REPEAT			
	*	SUBPRO	9121E	1			
	SNo.	ARGM 1	ARGM 2	ARGM 3	ARGM 4	ARGM 5	ARGM 6
	1	Mm	Xx	Yy	Zz	Dd	

Mm: Measurement point categories, measurement pattern (designated with three digits:•••).

- Hundreds integer: Establishes return point category (identical to surface measurement)
- Tens integer: Establishes measurement point category (identical to surface measurement)
- Ones integer: Establishes measurement pattern:
  1: X-axis groove center measurement
  2: Y-axis groove center measurement
  3: X-axis step center measurement
  4: Y-axis step center measurement

Xx: For measurement patterns 1 and 3: Groove/step center X coordinate (workpiece coordinate value)
For measurement patterns 2 and 4: Measurement point X coordinate (workpiece coordinate value)

Yy: For measurement patterns 1 and 3: Measurement point Y coordinate (workpiece coordinate value)
For measurement patterns 2 and 4: Groove/step center Y coordinate (workpiece coordinate value)

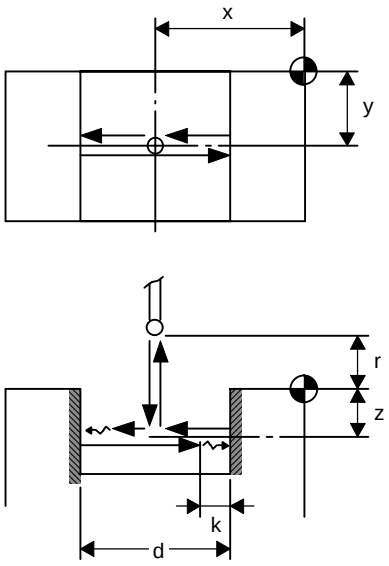
Zz: Measurement point Z coordinate

Dd: Groove width, step section width

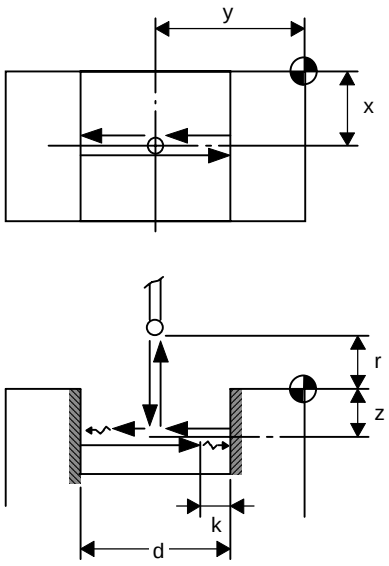
All argument data should contain decimal points.

- Measurement operations
  (“m_1” corresponds to the ones integer in argument “m”)

m_1 = 1.	X-axis groove center measurement	m_1 = 2.	Y-axis groove center measurement

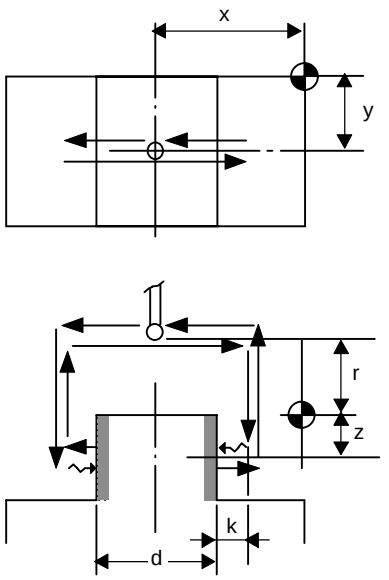


M3S089

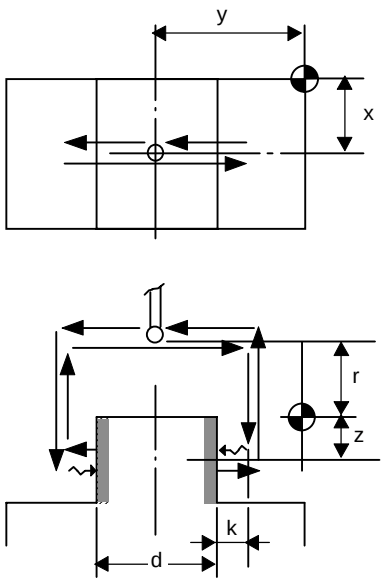


M3S089

m_1 = 3.	X-axis step section center measurement	m_1 = 4.	Y-axis step section center measurement



M3S090



M3S090

- Values of “r” and “k” are designated in the workpiece measurement initial setting program (WNo. 9100).

7. Hole center and boss center measurement (WNo. 9122)

This program calculates center coordinates (X and Y) by the measurement of four points on a circle. Measurement is made first along the X-axis, then along the Y-axis. It is combined with other measurement patterns to calculate the distance to a hole or boss center.

EIA/ISO	G65	P9122	Mm	Xx	Yy	Zz	Dd
MAZATROL	UNo. •	UNIT SUBPRO	WORK No. 9122E	REPEAT 1			
	SNo. 1	ARGM 1 Mm	ARGM 2 Xx	ARGM 3 Yy	ARGM 4 Zz	ARGM 5 Dd	ARGM 6

Mm: Measurement point categories, measurement pattern (designated with three digits:•••).

- Hundreds integer: Establishes return point category (identical to surface measurement)
- Tens integer: Establishes measurement point category (identical to surface measurement)
- Ones integer: Establishes measurement pattern:
  1: Hole center measurement
  2: Boss center measurement
  3: Hole center measurement II (movement on initial point level)

Xx: Circle center X coordinate (workpiece coordinate value)

Yy: Circle center Y coordinate (workpiece coordinate value)

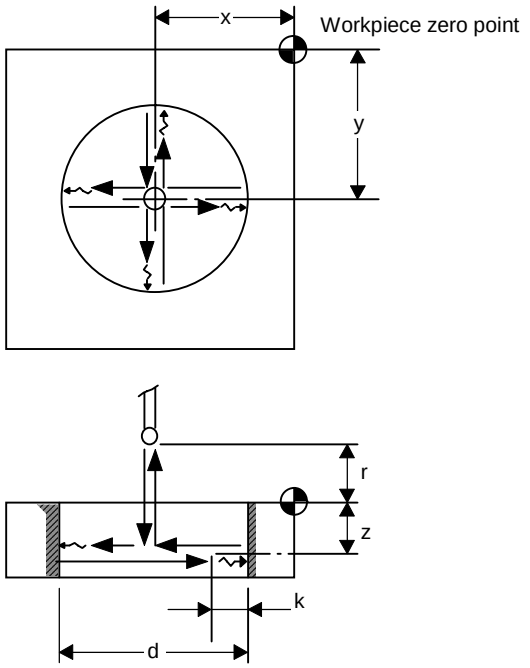
Zz: Measurement point Z coordinate (workpiece coordinate value)

Dd: Inner diameter/Outer diameter

All argument data should contain decimal points.

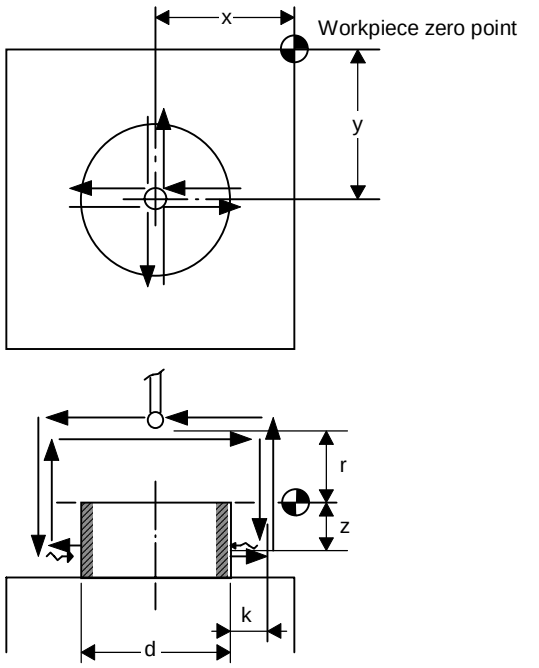
- Measurement operations
  ("m_1" corresponds to the ones integer in argument "m")

m_1 = 1.	Hole center measurement



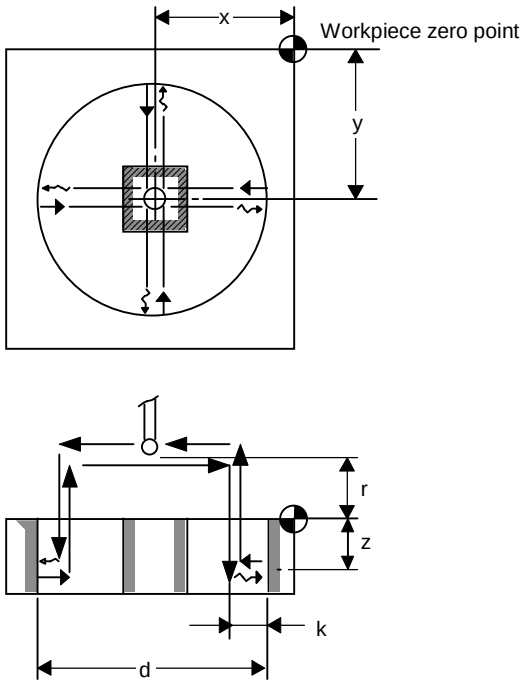
M3S082

m_1 = 2.	Boss center measurement



M3S083

m_1 = 3.	Hole center measurement II



M3S084

- Measurement pattern is determined in accordance with the value of argument "m_1" as shown in the accompanying diagrams.
- The values of "r" and "k" are designated in the workpiece measurement initial setting program (WNo. 9100).
- Hole center measurement II is used when there is a step within a circle, such as in PCKT MT machining.

8. Outside and inside corner measurement (WNo. 9123)

This program performs continuous measurement of an X surface and a Y surface to calculate the corner coordinate value (X and Y). It is used in combination with other measurement patterns to calculate the distance to the corner.

EIA/ISO	G65	P9123	Mm	Xx	Yy	Zz	Ii
MAZATROL	UNo. •	UNIT SUBPRO	WORK No. 9123E	REPEAT 1			
	SNo. 1	ARGM 1 Mm	ARGM 2 Xx	ARGM 3 Yy	ARGM 4 Zz	ARGM 5 Ii	ARGM 6

Mm: Measurement point categories, measurement pattern (designated with three digits:•••).

- Hundreds integer: Establishes return point category (identical to surface measurement)
- Tens integer: Establishes measurement point category (identical to surface measurement)
- Ones integer: Establishes measurement pattern:
  - 1: Outside corner; measurement direction +X, +Y
  - 2: Outside corner; measurement direction –X, +Y
  - 3: Outside corner; measurement direction –X, –Y
  - 4: Outside corner; measurement direction +X, –Y
  - 5: Inside corner; measurement direction –X, –Y
  - 6: Inside corner; measurement direction +X, –Y
  - 7: Inside corner; measurement direction +X, +Y
  - 8: Inside corner; measurement direction –X, +Y

Xx: Corner X coordinate (workpiece coordinate value)

Yy: Corner Y coordinate (workpiece coordinate value)

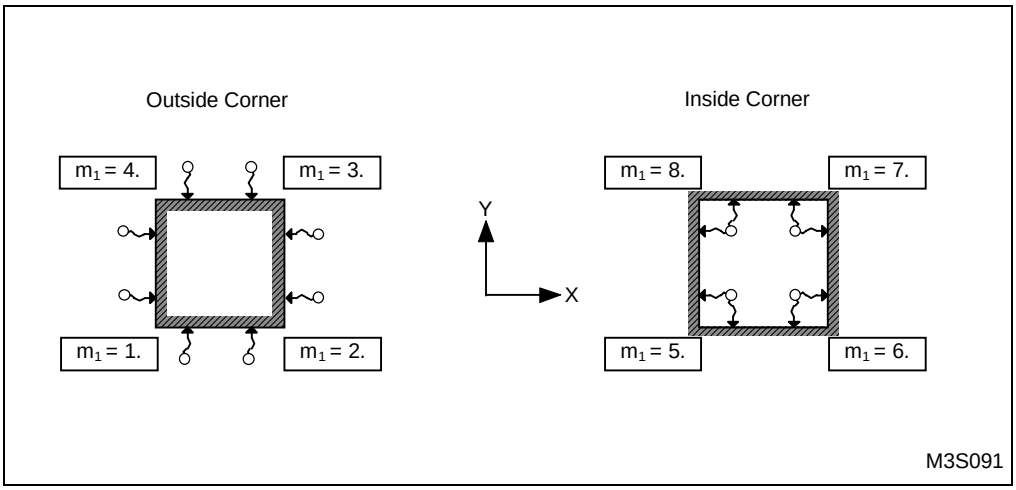
Zz: Measurement point Z coordinate (workpiece coordinate value)

Ii: Distance from corner to measurement point (plus value)

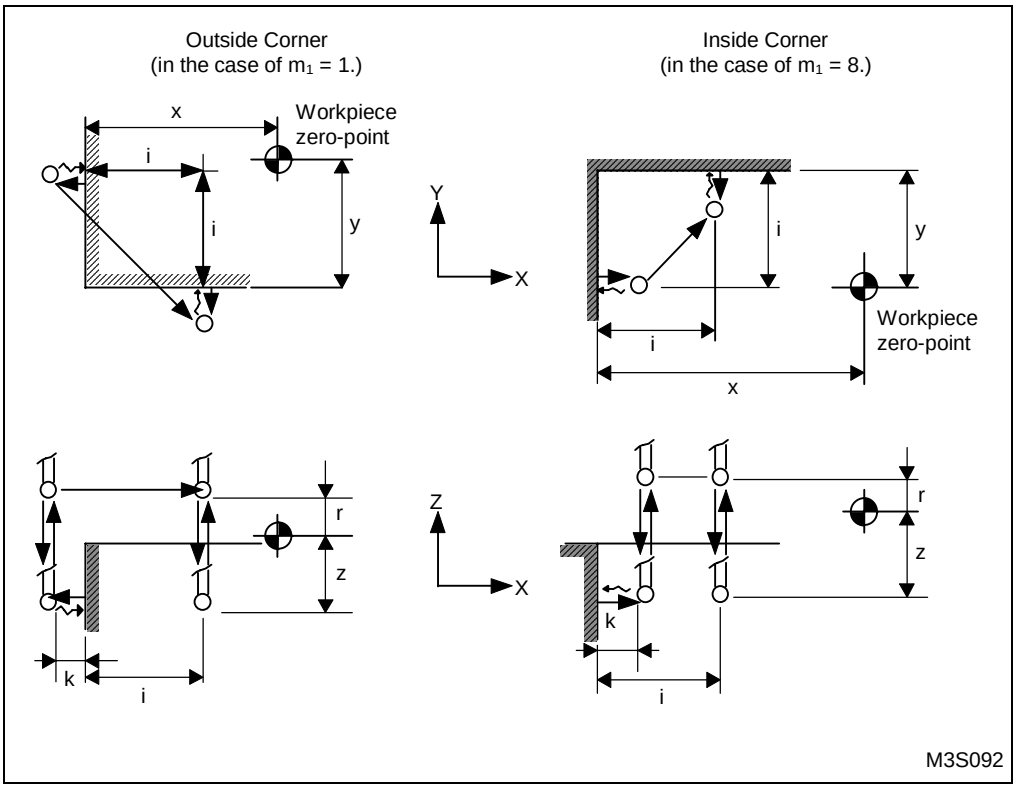
All argument data should contain decimal points.

[Measurement patterns]

("m_1" corresponds to the ones integer of argument m)



[Measurement operations]



- Values of "r" and "k" are designated in the workpiece measurement initial setting program (WNo. 9100).

9. Tolerance judgment printout B (WNo. 9125)

This program compares the distances between measurement point 1 and measurement point 2 as measured in combination with coordinate value measurements (WNo. 9120 through 9123) with the target values; makes a tolerance judgment on the basis of this comparison, and prints out the results of this judgment. Tool diameter and tool length compensation can also be made, depending on the amount of error in measured values in respect to the target values.

EIA/ISO	G65	P9125	Ee	Uu	Vv	$s	Bb
MAZATROL	UNo.	UNIT	WORK No.	REPEAT			
	•	SUBPRO	9125E	1			
	SNo.	ARGM 1	ARGM 2	ARGM 3	ARGM 4	ARGM 5	ARGM 6
	1	Ee	Uu	Vv	Ss	Bb	

Ee: Distance and axis direction designations
- e = 1.: Distance along X-axis
- e = 2.: Distance along Y-axis
- e = 3.: Distance along Z-axis
- e = 4.: Distance on the X-Y plane ($\sqrt{X^2 + Y^2}$)

Uu: Upper tolerance limit

Vv: Lower tolerance limit

If measurement results are to be printed out without the performance of a tolerance judgment, the setting of these two values should be “u = v = 0”.

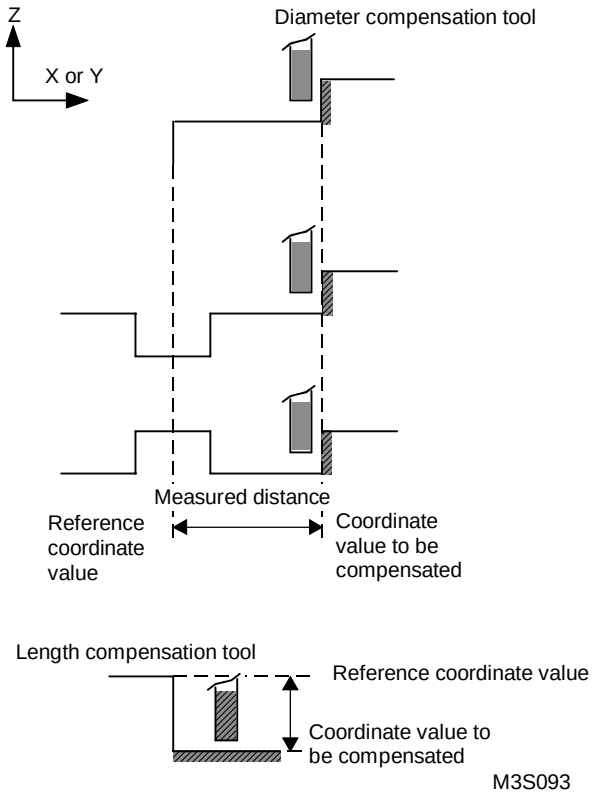
Ss: The Tool No. or Offset No. for which compensation is to be made.
- Designation in a case where compensation category #502 = 0. and tool number is to be set:
  Example:
  Tool No. 20 • S = 20.
- Designation in a case where compensation category #502 = 1. and offset number is to be set:
  Example:
  Offset No. 20 • S = 20.
- Designation in a case where compensation category #502 = 2. and tool number or group number is to be set:
  Select tool number or group number using bit 4 of parameter **F94.**
- If compensation is unnecessary or not possible, make no input for “Ss” or set at “S = 0”.

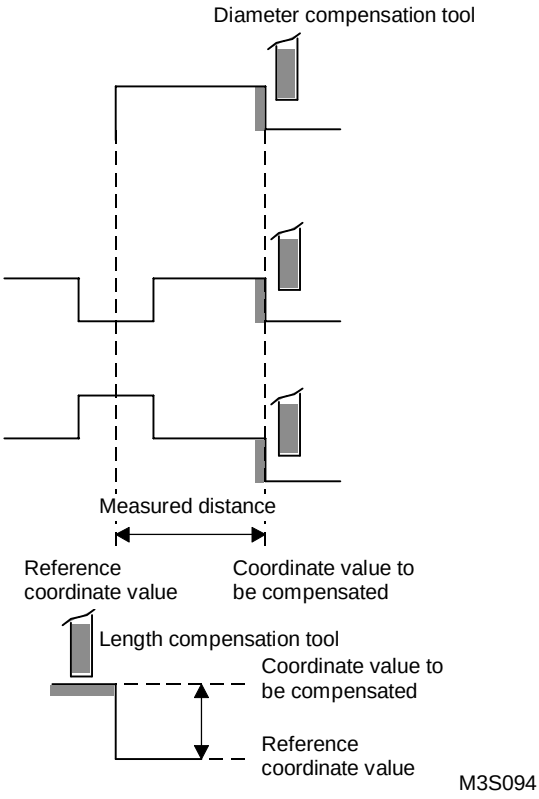
Bb: Compensation amount conversion coefficient
This data is effective only when compensation is made. If no input is made, it will be treated as “b = 1”. (Refer to the following figures.)

All argument data should contain decimal points.

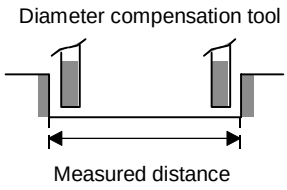
- Compensation operation

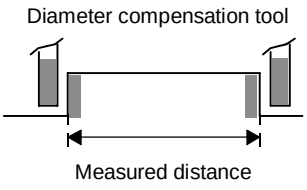
b = 1.	Inner compensation as seen from reference point	b = –1.	Outer compensation as seen from reference point





b = 0.5	Inner compensation on both surfaces	b = –0.5	Outer compensation on both surfaces





10. Slant measurement and tolerance judgment printout C (WNo. 9130/9135)

This program measures two points on the workpiece surface and calculates the angle of slant in respect to the coordinate axis; compares the results to a target value to make a tolerance judgment, and prints out the result of this judgment.

EIA/ISO	G65	P9130	Ff	Mm	Xx	Yy	Zz
	G65	P9135	Ii	Aa	Uu	Vv	
MAZATROL	UNo.	UNIT	WORK No.	REPEAT			
	•	SUBPRO	9130E	1			
	SNo.	ARGM 1	ARGM 2	ARGM 3	ARGM 4	ARGM 5	ARGM 6
	1	Ff	Mm	Xx	Yy	Zz	
	UNo.	UNIT	WORK No.	REPEAT			
	•	SUBPRO	9135E	1			
	SNo.	ARGM 1	ARGM 2	ARGM 3	ARGM 4	ARGM 5	ARGM 6
	1	Ii	Aa	Uu	Vv		

Ff: Selection of surface
f = 1.: X-Y surface angle measurement
f = 2.: X-Z surface angle measurement
f = 3.: Y-Z surface angle measurement

Mm: Designation of measurement direction
m = 1.: Measurement feed direction is +X
m = 2.: Measurement feed direction is –X
The above two settings cannot be made when f = 3.
m = 3.: Measurement feed direction is +Y
m = 4.: Measurement feed direction is –Y
The above two settings cannot be made when f = 2.
m = 5.: Measurement feed direction is –Z
The above setting cannot be made when f = 1.
When a zero-point return operation is to be performed after the measurement operation has been completed, add “100” to each of the above settings and set as follows:
m = 101.; m = 102.; m = 103.; m = 104.; m = 105.

Xx: Measurement starting point X coordinate (workpiece coordinate value)

Yy: Measurement starting point Y coordinate (workpiece coordinate value)

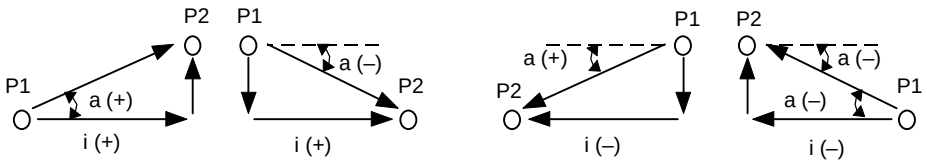
Zz: Measurement starting point Z coordinate (workpiece coordinate value)

Ii: Movement amount in axial direction from measurement point 1 to measurement point 2
The movement axis is automatically determined according to the combination of “f” (surface) and “m” (measurement direction) arguments.

Aa: Target angle (degrees): Set at $-45. \leq a \leq 45.$ (see **Note** below)

- The reference coordinate axis for angle calculation is automatically determined according to the combination of “f” (surface) and “m” (measurement direction) arguments.
- The value assigned to | i | should be as large as possible.

- The coordinate value of measurement starting point 2 is automatically determined according to the measurement starting point, the axis movement amount and the target angle. It is therefore especially important to take care about signs of axis movement amount “i” and target angle “a”.



Signs of axis movement amount “1” and target angle “a” M3S101

f: Surface	m: Measurement direction	i: Movement amount	a: Target angle
f = 1. X-Y surface	m = 1, 2 X-axis	Y-axis	Angle with Y-axis
	m = 3, 4 Y-axis	X-axis	Angle with X-axis
f = 2. X-Z surface	m = 1, 2 X-axis	Z-axis	Angle with Z-axis
	m = 5 Z-axis	X-axis	Angle with X-axis
f = 3. Y-Z surface	m = 3, 4 Y-axis	Z-axis	Angle with Z-axis
	m = 5 Z-axis	Y-axis	Angle with Y-axis

Uu: Upper tolerance limit (degrees)

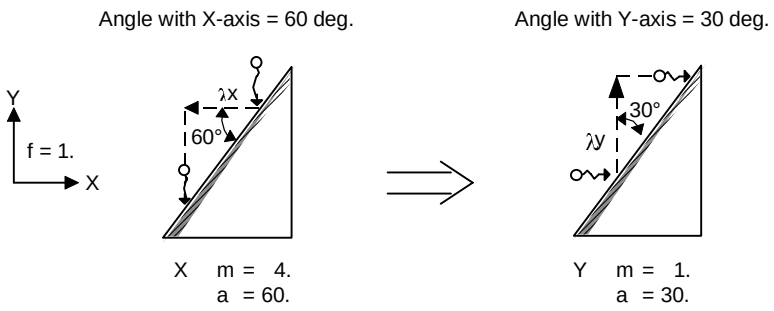
Vv: Lower tolerance limit (degrees)

If tolerance check is not necessary, set as “u = v = 0”

Note:

If angle is greater than 45 degrees, change measurement direction.

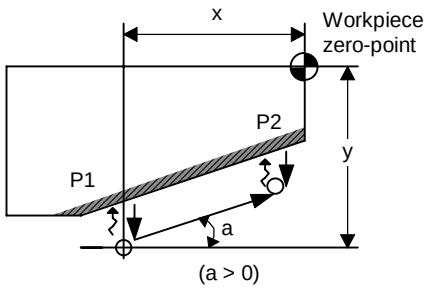
Example:

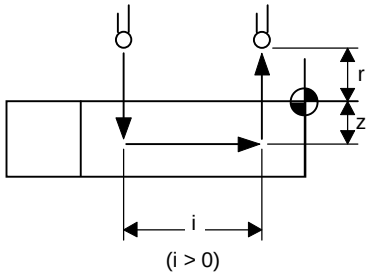


M3S097

[Measurement operations]

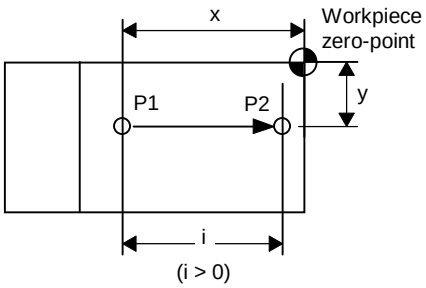
When f = 1. and m = 3.

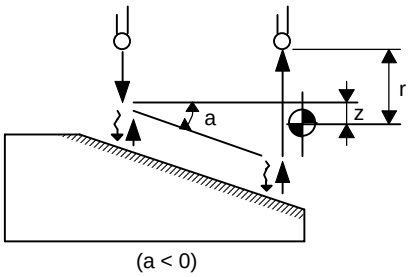




M3S098

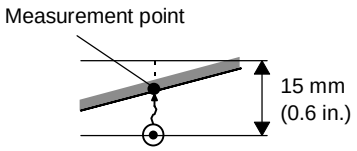
When f = 2. and m = 5.



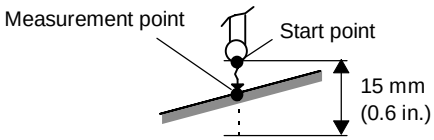


M3S099

- The measurement start point should be set within 15 mm (0.6 in.) of the measurement point.



Measurements on X- and Y-axes

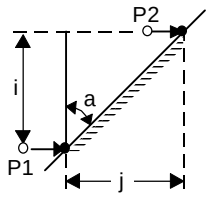
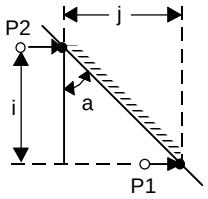
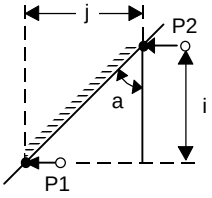
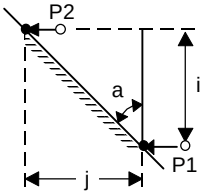
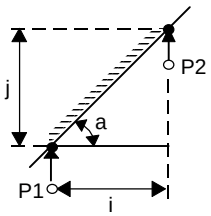
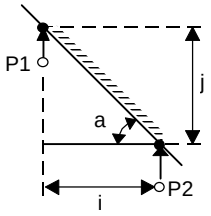
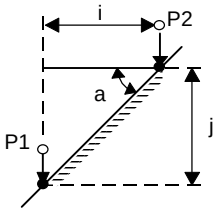
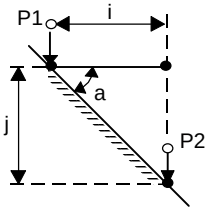


Measurements on Z-axis

M3S100

[Defining the measured angle]

1. f = 1.; X-Y surface

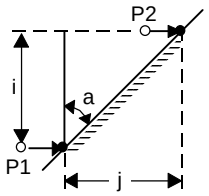
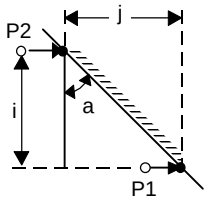
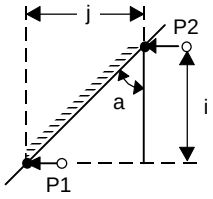
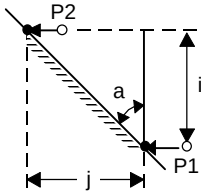
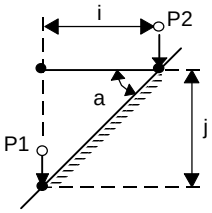
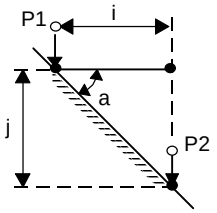
Measurement direction	a > 0	a < 0
m = 1. +X		
m = 2. –X		
m = 3. +Y		
m = 4. –Y		 M3S102

“P1” is the measurement start point designated by arguments “x” “y” and “z”. “P2” is determined automatically according to the values of arguments “i” and “a”.
In the above example, i > 0, but if the relationship were i < 0, then the positions of P1 and P2 would be reversed.
The measured angle is determined by the following equation:

$$\tan^{-1}\frac{j}{i}$$

(j = measurement point 2 coordinate value – measurement point 1 coordinate value)

2. f = 2.; X-Z surface

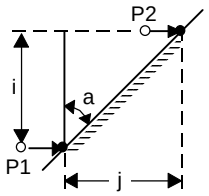
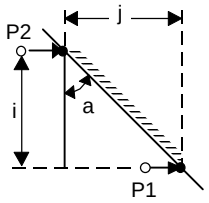
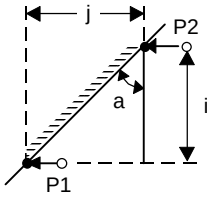
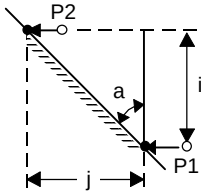
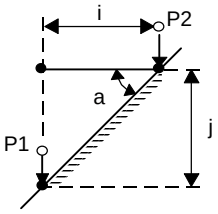
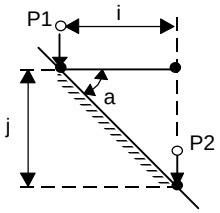
Measurement direction	a > 0	a < 0
m = 1. +X		
m = 2. –X		
m = 5. –Z		 M3S103

“P2” is determined automatically according to the values of arguments “i” and “a”.
In the above example, i > 0, but if the relationship were i < 0, then the positions of P1 and P2 would be reversed.
The measured angle is determined by the following equation:

$$\tan^{-1}\frac{j}{i}$$

(j = measurement point 2 coordinate value – measurement point 1 coordinate value)

3. f = 3.; Y-Z surface

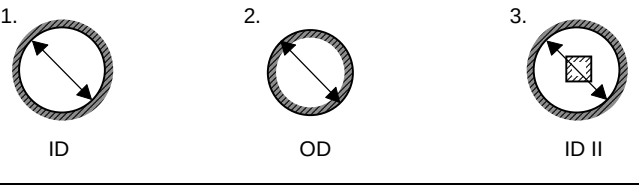
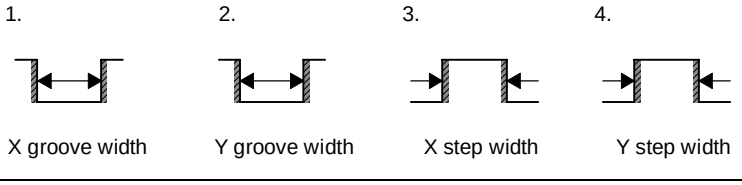
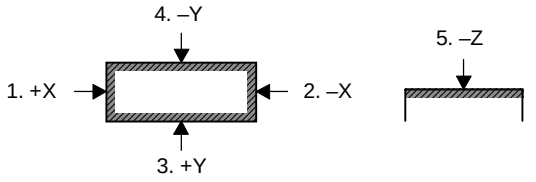
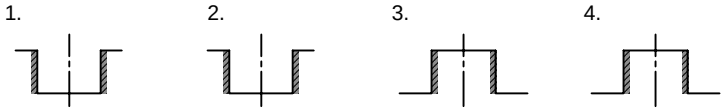
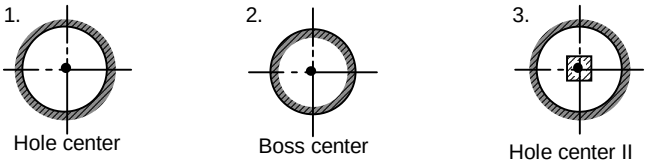
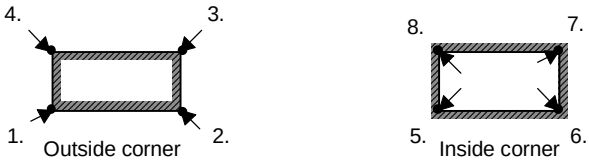
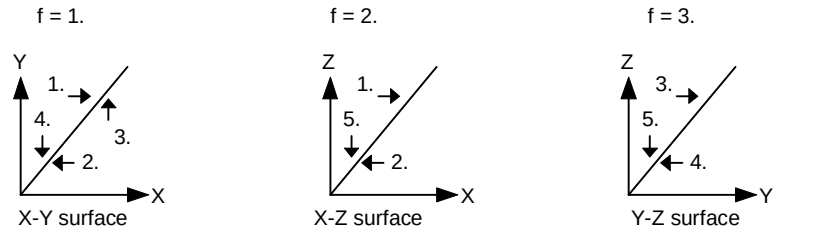
Measurement direction	a > 0	a < 0
m = 3. +Y		
m = 4. –Y		
m = 5. –Z		 M3S103

“P1” is the measurement start point designated by arguments “x” “y” and “z”.
“P2” is determined automatically according to the values of arguments “i” and “a”.
In the above example, i > 0, but if the relationship were i < 0, then the positions of P1 and P2 would be reversed.
The measured angle is determined by the following equation:

$$\tan^{-1}\frac{j}{i}$$

(j = measurement point 2 coordinate value – measurement point 1 coordinate value)

11-1-6 Program and measurement pattern chart

Group	Work No.	Measurement patterns (m_1)
A	9110	 M3S104
	9111	 M3S105
B	9120	 M3S106
	9121	 M3S107
	9122	 M3S108
	9123	 M3S109
C	9130 (9135)	 M3S110

11-1-7 Output of measurement results

1. Contents of printout

The contents of the printout are automatically controlled by the type of data processing system being used. A maximum of 8 items can be included in one print cycle.

Format (Differs by contents of the printout)

Millimeter: Integer: up to four figures;
Decimal: down to three decimal places.

Inch: Integer: up to four figures;
Decimal: down to four decimal places.

A. Work No. and workpiece count

The work number designated by the argument W in workpiece measurement initial setting (WNo. 9100) and workpiece count number are printed.
An example follows:

```
WNO•1234•COUNT 1
```

B. Target values

The printout includes a comment which indicates the automatically calculated target value (“MARK DATA”) and the measurement pattern.

The format of this comment is indicated in the following table:

Measurement pattern	Comment
Inner diameter/outer diameter	MARK DATA HOL/BOS DIA
Groove/step width	MARK DATA GRV/STP WIDTH
Surface to surface	MARK DATA FACE-FACE
Surface to groove/step center	MARK DATA FACE-GRV/STP
Surface to hole/boss center	MARK DATA FACE-HOL/BOS
Surface to corner	MARK DATA FACE-CNR
Groove/step center to groove/step center	MARK DATA GRV/STP-GRV/STP
Groove/step center to hole/boss center	MARK DATA GRV/STP-HOL/BOS
Groove/step center to corner	MARK DATA GRV/STP-CNR
Hole/boss center to hole/boss center	MARK DATA HOL/BOS-HOL/BOS
Hole/boss center to corner	MARK DATA HOL/BOS-CNR
Corner to corner	MARK DATA CNR-CNR
X-Y surface slant	MARK DATA SLANT X-Y
X-Z surface slant	MARK DATA SLANT X-Z
Y-Z surface slant	MARK DATA SLANT Y-Z

C. Measured values

The actually measured lengths and angles are also printed with the comments according to the format shown below.

Length	Angle	Comment
Length on X-axis	Angle with X-axis	MEASURED DATA X
Length on Y-axis	Angle with Y-axis	MEASURED DATA Y
Length on Z-axis	Angle with Z-axis	MEASURED DATA Z
ID/OD or X-Y surface length ($\sqrt{X^2 + Y^2}$)	••	MEASURED DATA R

D. Tolerance ranges

The following value is printed out:
(upper tolerance limit - lower tolerance limit) / 2.
The example below assumes an upper tolerance limit of u = 0.3 and a lower tolerance limit of v = 0:

```
TOLERANCE 0.150
```

E. Error and judgment

The value representing “measured value” – “target value” is printed, along with the results of the tolerance judgment.

Format examples are indicated below:

Error judgement results	Comment
Within tolerance/outside compensation range	CHECK OK
Within tolerance/in compensation range	CHECK OK/WARNING
Outside tolerance range	CHECK LIMIT OVER ALARM

F. Compensation tool No./compensation offset No.

Either the tool number or the offset number designated by argument “Ss” will be printed.

Tool No. for tool No. 25: TOOL No. 25
Offset No. for offset No. 25: OFFSET No. 25

G. Pre-compensation tool data/offset data

This function prints out the value of the data listed in F. above before offset compensation is made. If the data to be printed is tool data, it will all refer to tool diameter with the exception of Z-axis step difference measurement, which will be tool length.

For tool data: TOOL DATA 20.000
For offset data: OFFSET DATA 10.000

H. Post-compensation tool data/offset data

This function prints the new data as listed in F. above, after offset compensation has been made.

For tool data: NEW TOOL DATA 19.950
For offset data: NEW OFFSET DATA 9.975

- Items printed

•: Printout ×: Not printout

No.	Item	Tolerance not indicated	Componsation indicated/ not indicated Judgment results outside compensation	Compensation made
A	Workpiece No. and workpiece count	•	•	•
B	Target value	•	•	•
C	Measured value	•	•	•
D	Tolerance range	×	•	•
E	Error and judgment	×	•	•
F	Compensation tool number/offset number	×	×	•
G	Pre-compensation tool data/offset data	×	×	•
H	Post-compensation tool data/offset data	×	×	•

Note: For compensation category #502 = 2., the following items are not output.
F: Compensation tool number/offset numberz
G: Pre-compensation tool data/offset data
H: Post-compensation tool data/offset data

2. Measurement results and display on screen

Under normal circumstances measurement results are verified by the printout, but they can also be verified at the **COMMON VARIABLE** display if no printout command has been made.
This display can also be used for measurements between two random points if the purpose is to verify the measurement coordinates themselves and not the distance between coordinates.

Address	Data contained at address	Address	Data contained at address
#135	Work No.	#162	X coordinate value of measurement point 1
#136	Workpiece count	#163	Y coordinate value of measurement point 1
#145	Target value	#164	Z coordinate value of measurement point 1
#146	Measured value	#168	X coordinate value of measurement point 2
#147	Tolerance range ((u – v)/2)	#169	Y coordinate value of measurement point 2
#148	Error: Measured value – target value (#146 – #145)	#170	Z coordinate value of measurement point 2
#149	Offset number or tool number*		
#150	Pre-compensation offset data or tool data*		
#151	Post-compensation offset data or tool data*		
#152	Tolerance judgment OK range (q × #147)		
#153	Judgment results: 1. OK 2. WARNING 3. Tolerance over ALARM		

*: For compensation category #502 = 2., these items are not output.

11-1-8 Alarm displays

An NC alarm will be generated in the event that a programming error or a malfunction in the touch sensor prevents an accurate measurement from being made or in case of tolerance over state. An alarm message will be displayed on the CRT screen if an alarm is generated.

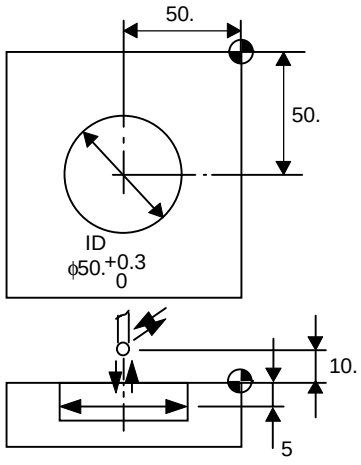
Alarm No./Message	Remarks
3901 PARAMETER ERROR	Data processing system parameter value is not correct.
3902 INITIALZ ARGUMENT ERROR	Argument data is not correct in workpiece measurment initial setting program.
3903 MEASURE ARGUMENT ERROR	Argument data is not correct in measurement operation program.
3904 JUDGE ARGUMENT ERROR	Argument data is not correct in tolerance judgment printout program.
3905 PATTERN GROUP MISS MATCH	Mismatch between tolerance judgment printout program and measurement pattern group
3906 ARGUMENT E ERROR	The value of “Ee” in measurement B tolerance judgment does not match the measurement pattern.
3907 CORRECTION APPOINT ERR.	A compensation command has been made in measurement B tolerance judgment program for an item for which compensation is not possible.
3910 TOLERANCE OVER	Results of tolerance judgment show a tolerance over state.
3912 MMS NOT TOUCH	Tool does not touch the work during touch sensor measurement operation.

Note 1: Refer to the Alarm List for other alarms.

Note 2: If a block preceded by a block for calling workpiece measurement macro is designated for restart position of EIA/ISO programs, alarm No. 3910 will result at the calling block during restart position search, unless all the blocks for calling work No. 9100 to 9145 are provided with skip code /n and the corresponding block-skip function is validated before restart position search.

11-1-9 Program and printout example

1. The following example illustrates tool diameter compensation made for Tool No. 20 as the result of ID measurement.



M3S111

Program (EIA/ISO)

M

```
G65P9100W1234.R10.K3.Q0.5
G65P9110M101.X-50.Y-50.Z-5.D50.
G55P9115U0.3V0.S20.
```

M

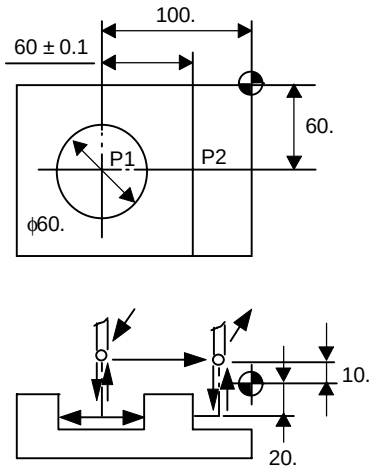
Print out

```
%%
   /RESULT OF MEASURING/
  WNO  1234  COUNT 1

MARK DATA HOL/BOS DIA      50.150
MEASURED DATA R            50.064
TOLERANCE                   0.150
CHECK OK/WARNING           -0.086
TOOL NO                    20
TOOL DATA                  10.000
NEW TOOL DATA               9.914
          END
```

The values q = 0.5; d = 50.; u = 0.3; and v = 0 would cause an alarm to be generated if the measured value was greater than 50.3 or less than 50. If the result were 50.075 < measured value < 50.225, the result would be CHECK OK and no compensation would be made.

2. The following example illustrates compensation made for Offset No. 30 as the result of measurements of hole center as reference and surface.



M3S112

Program (EIAISO)

```
M
     G65P9100W5678.R10.K4.Q0.5
(P1) G65P9122M11.X-100.Y-60.Z-20.D60.
(P2) G65P9120M122.X-40.Y-60.Z-20.
     G65P9125E1.U0.1V-0.1S30.B-1.
M                              (*)
```

Printout

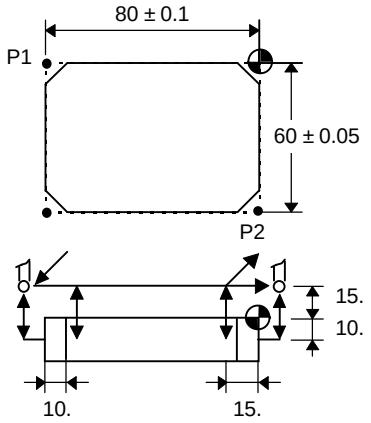
```
%%
   /RESULT OF MEASURING/
  WNO  5678  COUNT 1
```

MARK DATA FACE-HOLE/BOS	60.000
MEASURED DATA X	60.053
TOLERANCE	0.100
CHECK OK/WARNING	0.053
OFFSET NO	30
OFFSET DATA	10.000
NEW OFFSET DATA	9.947

```
          END
```

* If the value “B–1.” were not designated, the new offset data would become 10.053.

3. The following example illustrates a measurement made between two outside corner points and a tolerance judgment made along the X-axis and the Y-axis.



M3S113

Program (EIA/ISO)

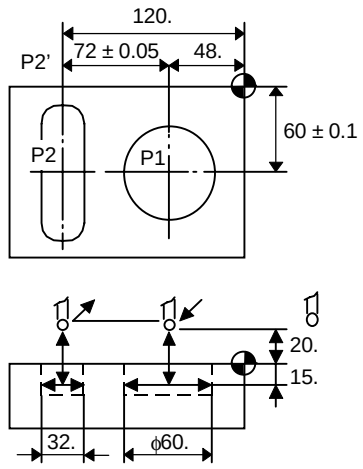
```
M
     G65P9100W2468.R15.K5.Q0.5
(P1) G65P9123M14.X-80.Y0.Z-10.I15.
(P2) G65P9123M122.X0.Y-60.Z-10.I15.
     G65P9125E1.U0.1 V-0.1
     G65P9125E2.U0.05 V-0.05
M
```

Print out

```
%%
    /RESULT OF MEASURING/
     WNO  2468  COUNT 1
MARK DATA CNR-CNR          80.000
MEASURED DATA X            80.047
TOLERANCE                   0.100
CHECK OK/WARNING            0.047
          END
%%
MARK DATA CNR-CNR          60.000
MEASURED DATA Y            60.051
TOLERANCE                   0.050
CHECK LIMIT OVER ALARM      0.051
          END
```

The above example could also apply a step width measurement.

4. Continuous measuring of X-axis distance between hole center and groove center, and Y-axis distance between hole center and plane.



M3S114

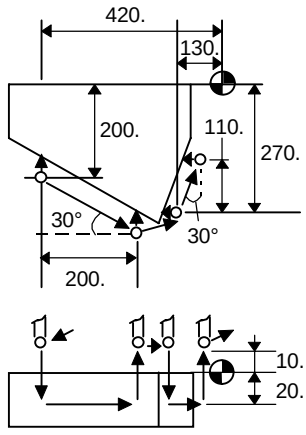
Program (EIA/ISO)

```
M
     G65P9100W3690.R20.K3.Q0.5
(P1) G65P9122M11.X-48.Y-60.Z-15.D60.
(P2) G65P9121M121.X-120.Y-60.Z-15.D32.
     G65P9125E1.U0.05 V-0.05
(P2')    G65P9120M124.X-120.Y0.Z-15.
     G65P9125E2.U0.1V-0.1
M
```

Print out

```
%%
    /RESULT OF MEASURING/
      WNO  3690  COUNT 1
MARK     DATA     GRV/STP-      72.000
HOL/BOS
MEASURED DATA X                71.963
TOLERANCE                       0.050
CHECK OK/WARNING               -0.037
            END
%%
MARK DATA FACE-HOLE/BOS        60.000
MEASURED DATA Y                59.952
TOLERANCE                       0.100
CHECK LIMIT OVER ALARM         -0.048
            END
```

5. Measuring slant angle between two planes on X-Y plane
   Tolerance: ±0.2 deg.



M3S115

Program (EIA/ISO)

```
M
G65P9100W1496.R10.K5.Q0.5
G65P9130F1.M3.X-420.Y-200.Z-20.
G65P9135I200.A-30.U0.2V-0.2
G65P9130F1.M102.X-130.Y-270.Z-20.
G65P9135I110.A30.U0.2V-0.2
M
```

Print out

```
%%
        /RESULT OF MEASURING/
      WNO  1496  COUNT 1

MARK DATA SLANT X-Y          -30.000
MEASURED DATA X              -29.925
TOLERANCE                      0.200
CHECK OK                       0.075
            END
%%
MARK DATA SLANT X-Y           30.000
MEASURED DATA Y               29.937
TOLERANCE                      0.200
CHECK OK                      -0.063
            END
```

Argument K is not effective for this function. However, specify a positive value.

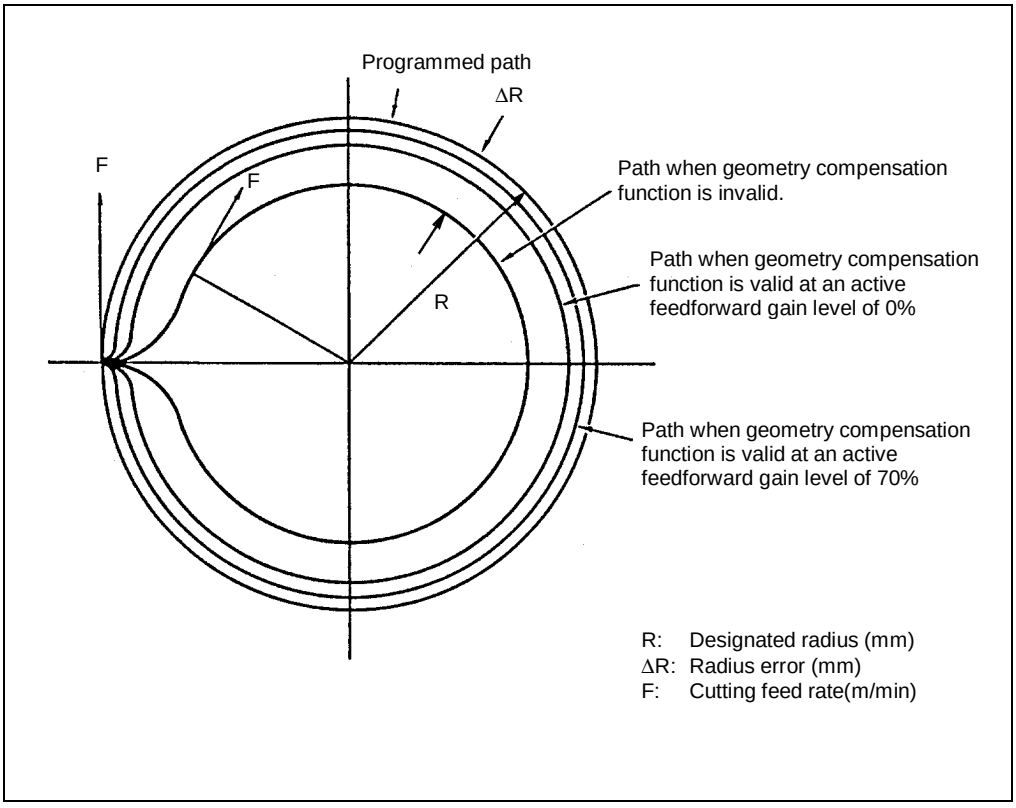
11-4 Geometry Compensation Function

11-4-1 Function general

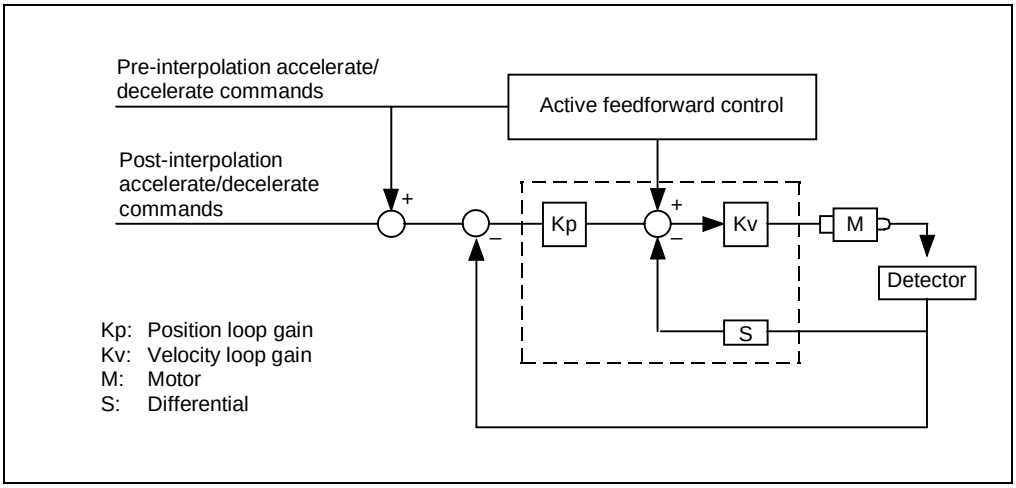
Machining at rapid feed will generally degrade a geometry accuracy in proportion to the feed rate. For example, delayed follow-up of smoothing circuit and servo system occurs in circular cutting, making a smaller workpiece than commanded circle radius.
Our CNC unit allows conventional geometry errors to be reduced by using geometry compensation function. The geometry compensation function is composed of the following four functions:

1. Pre-interpolation acceleration/deceleration
2. Feedforward control
3. Precise vector compensation
4. Optimum corner deceleration

In circular cutting, geometry errors from smoothing circuit can be reduced by pre-interpolation acceleration/deceleration and those from delayed follow-up of servo system can be reduced by feedforward control.

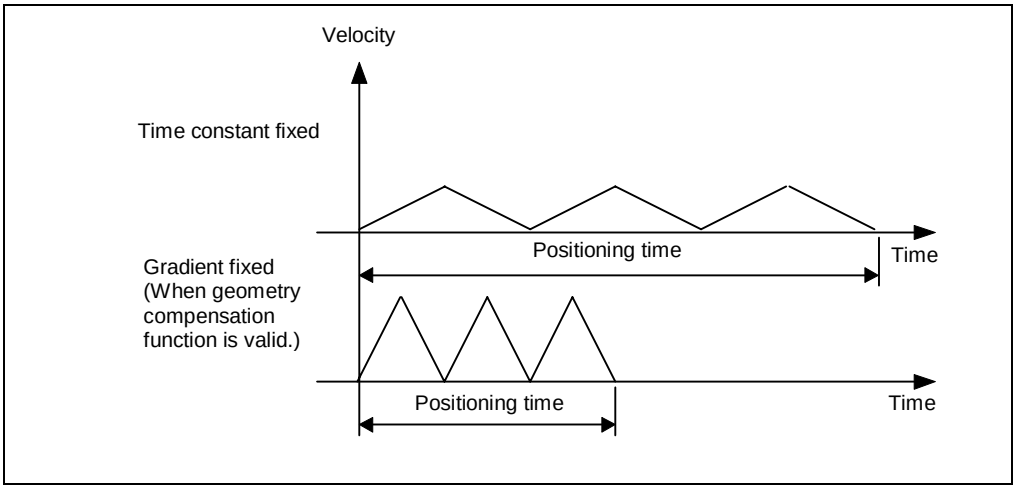


Active feedforward control

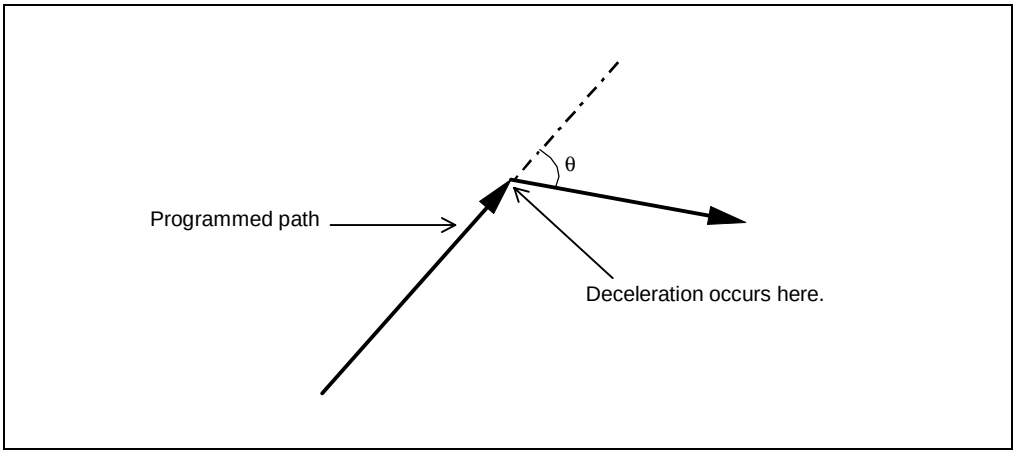


Note: Active feedforward control is valid only when the geometry compensation function is valid.

Also, the time required for the positioning of very small line segments using command G00 can be reduced since gradient-fixed acceleration/deceleration occurs automatically.
The positioning time changes as follows:



Optimum corner deceleration may occur, de-pending on the angle, θ, of connection point A. This allows machining into highly precise edges.



11-4-2 Detailed description

1. Programming format

A. MAZATROL program

For MAZATROL programs the geometry compensation function becomes automatically valid through the setting of user-parameter **F72**.

User-parameter	**F72** = 1: Geometry compensation function valid
	F72 = 0: Geometry compensation function invalid

It is also possible to validate/invalidate the geometry compensation function by setting M153/M154 under M-code article either of tool sequence data in a machining unit or of a manual program mode unit. But it is not the case if M153/M154 is given in an M-code unit.

M153 : Geometry compensation function valid

M154 : Geometry compensation function invalid

Sample program

```
UNo.  MAT       INITIAL-Z     ATC MODE    MULTI MODE   MULTIFLAG    PITCH-X  PITCH-Y
0     FC        50.           0           OFF          •            •        •
UNo.  UNIT      ADD.WPC X          Y          th        Z          4
1     WPC-0     •       −227.556   −165.792   0.        −241.778   0.
UNo.  UNIT      DEPTH   SRV-Z    BTM      WAL   FIN-Z     FIN-R
2     FACE MIL  0.      2.       4        •     0.6       •
SNo.  TOOL      NOM-ϕ  No.   APRCH-X   APRCH-Y   TYPE  ZFD   DEP-Z  WID-R  C-SP   FR   M   M
R 1   FACE MIL  100.A       ?         ?         XBI   •     1.4    70.    110    2.1      154
F 2   FACE MIL  100.        ?         ?         XBI   •     •      70.    110    0.84
FIG   PTN       P1X/CX  P1Y/CX  P3X/R   P3Y     CN1     CN2     CN3     CN4
1     SQR       0.      0.      150.    −150.
UNo.  UNIT      DEPTH   SRV-Z    BTM      WAL   FIN-Z     FIN-R     INTER-R   CHMF
3     POCKET    10.     10.      4        4     0.6       0.6       0.        0.
SNo.  TOOL      NOM-ϕ  No.   APRCH-X   APRCH-Y   TYPE  ZFD   DEP-Z  WID-R  C-SP  FR      M  M
R 1   E-MILL    20.         ?         ?         CW    G01   9.4    14.    82    0.159      154
F 2   E-MILL    14.         ?         ?         CW    G01   •      9.8    86    0.448
FIG   PTN       P1X/CX  P1Y/CX  P3X/R   P3Y     CN1     CN2     CN3     CN4
1     SQR       45.     −85.    105.    115.    R 10.   R 10.   R 10.   R
                                                                        10.
UNo.  UNIT      CONTI.  NUMBER   ATC      X     Y     Z     4     ANGLE
4     END       0       0        0        0.    0.    0.    0.    ••
```

Note: If the program shown above is executed in the automatic operation mode with parameter **F72** = 1, the geometry compensation function will be made invalid for rough machining of face milling (with face mill of ϕ100 A) and pocket milling (with end mill of ϕ20).

B. EIA/ISO Program

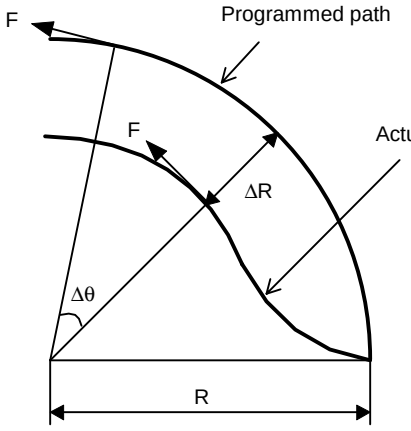
G61.1		Geometry compensation function valid
G61	(Exact stop check mode)	Geometry compensation function invalid (Command codes of the same group cancel the mode.)
G62	(Automatic corner override)	
G63	(Tapping mode)	
G64	(Cutting mode)	

Note: In the case of EIA/ISO programs, it is always necessary to give the G-code command to make the geometry compensation function valid since the setting of parameter **F72** is not effective for them.

Sample program

```
N001  G61.1                Geometry compensation function valid
      G01X100.F1000
      X100.Y-100.
      X-100.Y-100.
      X-100.
      X-100.Y100.
      X100.Y100.
      G64                  The mode is cancelled (becomes invalid).
```

2. Pre-interpolation acceleration/deceleration



R: Designated radius (mm)/(inch)
ΔR: Radius error (mm)/(inch)
Δθ: Angle error (rad)
F: Cutting feed rate (m/min)/(inch/min)

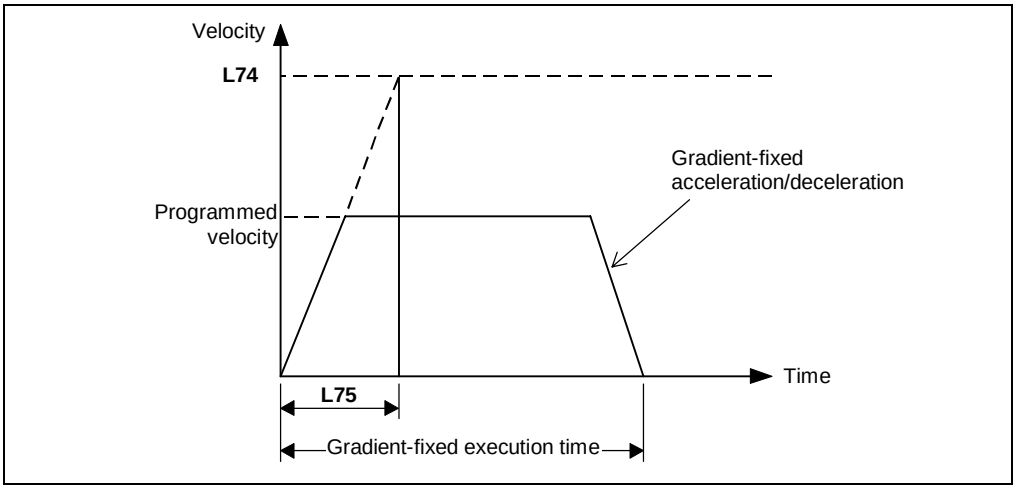
The radius error, ΔR, is calculated as follows:

Post-interpolation acceleration/ deceleration	Exponential	$\Delta R = \frac{1}{R} \times \left(\frac{1}{2}Ts^2 + \frac{1}{2}Tp^2\right) \times \left(\frac{F \times 10^3}{60}\right)^2$ (mm)
	Linear	$\Delta R = \frac{1}{R} \times \left(\frac{1}{24}Ts^2 + \frac{1}{2}Tp^2\right) \times \left(\frac{F \times 10^3}{60}\right)^2$ (mm)
Pre-interpolation acceleration/deceleration (When geometry compensation function if effective)		$\Delta R = \frac{1}{R} \times (\bullet + \frac{1}{2}Tp^2(1 - Kf^2)) \times \left(\frac{F \times 10^3}{60}\right)^{\bullet 2} \bullet mm \bullet$

Note: Under the inch system, $\left(\frac{F}{60}\right)$ acts as a substitute for $\left(\frac{F \times 10^3}{60}\right)$

Ts : The preset time constant (sec) for the smoothing circuit = Parameter **N5/**1000
Tp :=Position loop time constant (sec) = 1/parameter **SV003**
Kf : Feedforward gain = Parameter **S4/**100

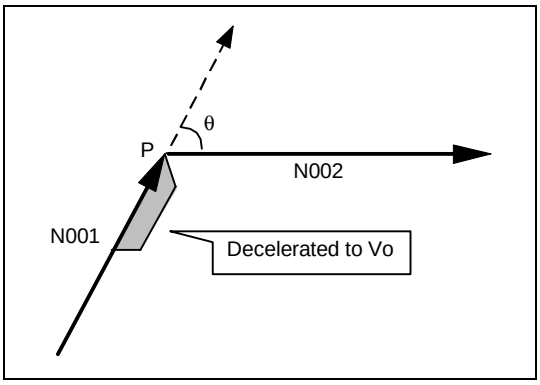
The acceleration/deceleration pattern is shown below.



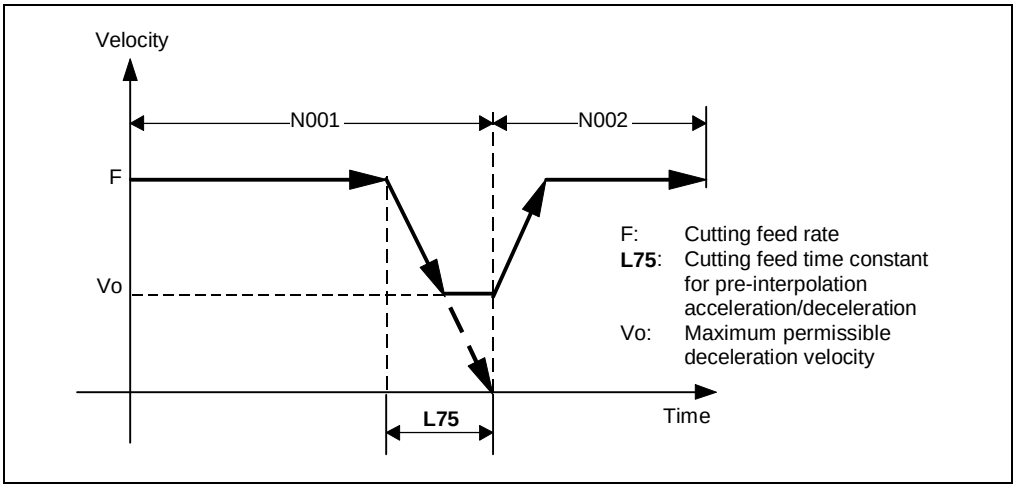
3. Optimum corner deceleration

Machining with high edge precision can be done by decelerating the machine at corners.

Before it arrives at corner point P, the machine is automatically decelerated to achieve its maximum permissible deceleration value, Vo.



The deceleration pattern is shown below.

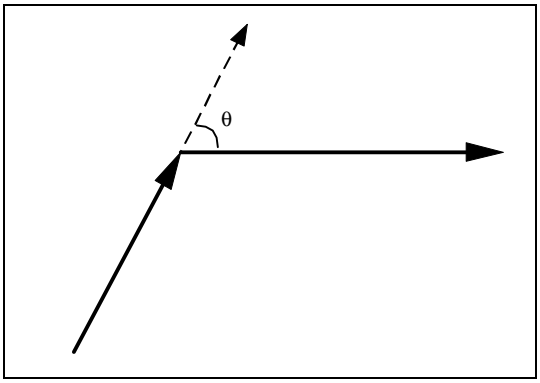


The value of Vo can be approximately given in the following expressions:

$Vo = \Delta V / \sqrt{2\,(1 - \cos\theta)}$

$\Delta V = (\mathbf{L74}/\mathbf{L75}) \times \Delta t$ Δt: Sampling frequency during interpolation (= 7.1 msec)

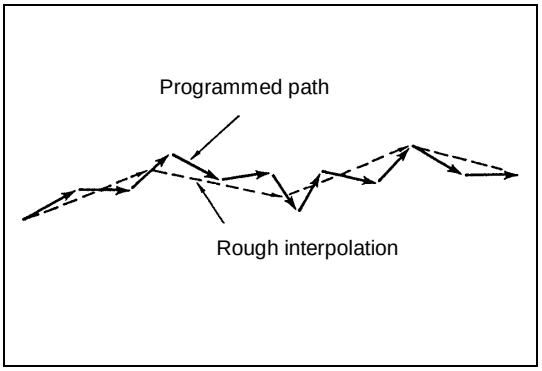
If the intersection angle, θ, of the path of the current block and that of the next one is the value of parameter **L77** or less, these two paths will be judged smooth and no deceleration will occur. If this angle is in excess of the value of parameter **L77** the paths will be judged nonsmooth and optimum corner deceleration will be performed.



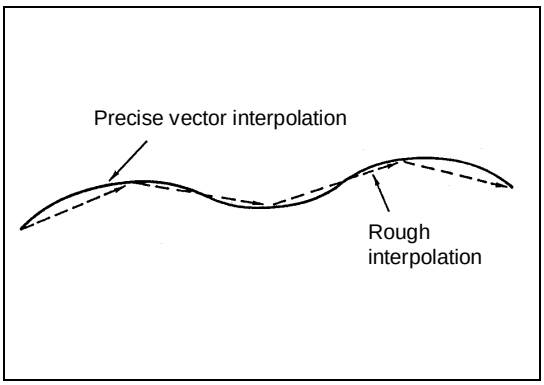
Note: The angle θ will be handled as five degrees if parameter **L79** is set to 0.
Although any value from 0 to 30 degrees can be set using parameter **L77**, setting of too large an angle value may cause machine vibration and/or rough surface machining, even if that value is an allowable one.

4. Precise vector interpolation

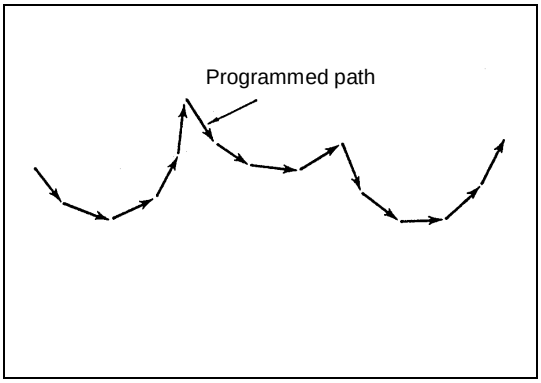
Although a very small line segment forms a very small block-to-block angle and thus provides smooth interpolation, it may not become possible for optimal corner deceleration to be achieved. This, however, becomes possible with a vectorial accuracy interpolation function since smoother interpolation can be obtained.
(Precise vector interpolation is valid only for the geometry compensation function.)



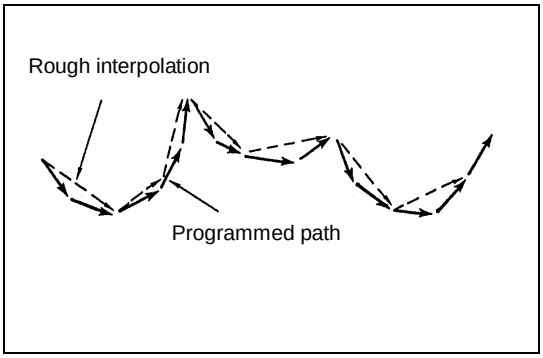
If interpolation has been programmed, rough interpolation is performed prior to precise vector interpolation.



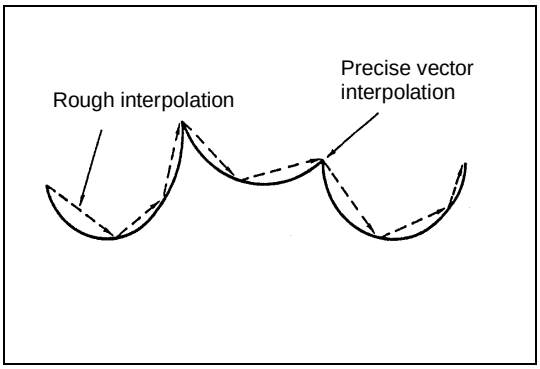
After rough interpolation, precise vector interpolation is performed.



As shown on the left, the programmed path may not be smooth,



If this is the case, the velocity will be reduced and optimum corner deceleration will occur.



Reduction of the velocity moves the tool almost as programmed, thus allowing precise machining.

5. Accuracy coefficient

Under corner- and arc-machining commands, optimal corner deceleration and arc feed rate limitation provide automatic deceleration to compensate for any errors in the accuracy of machining. In that case, machining accuracy can be further improved by specifying an accuracy coefficient in the program since further deceleration is achieved at corners and at arc sections.

Command format

,K__	Specifies the rate of reduction of the corner deceleration speed and the arc feed rate limitation in percentage terms.

<Sample program 1>

```
N001  G61.1
N200  G1X_Y_,K30   <-- If a corner deceleration command or an arc-interpolation command has
N300     X_Y_         been specified, machining will begin decelerating to 70% of the default
N400  ...          value at this block onward.
```

<Sample program 2>

```
N001  G61.1
N200  G2I-10.,K30  <-- Only this block is where machining decelerates to 70% of the default value.
N300  G1X10.,K0    <-- The accuracy coefficient is cancelled for this block onward automatically.
N400  ...
```

The accuracy coefficient is cancelled automatically in the following cases:

- The case that the reset key is pressed
- The case that a shape offset cancellation command (G64) is issued
- The case that command code ,K0 is set

Note 1: An accuracy coefficient cannot be specified in MAZATROL programs.

Note 2: If a value from 1 to 100 is specified as accuracy coefficient K in a program that includes a corner deceleration command and an arc-interpolation command, the machining time will increase, compared with its normal value.

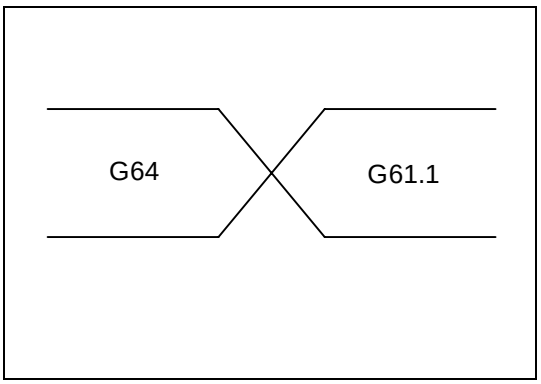
11-1-3 Hold Status

		G61.1
Initial status	Power on	×
Reset command	M02, M30, M00, reset	×
Reset operation	NC reset switch, external reset	×
	Reset and initialize	×
Block interruption	Mode change (auto/manual), feed hold	•
Block stop	Single block	•
Emergency stop	Emergency stop switch, external emergency stop	•
NC alarm	Servo alarm	•
OT	Hardware OT	•
Emergency stop cancel	Emergency stop switch, external emergency stop	×

•: No change ×: Cancel

11-1-4 Notes

1. This feature works only under optional specifications for geometry compensation function. A program error (Alarm **935 NO PRE-INTERP ACCEL/DECEL OPT.**) will result if programming is done using this feature under the standard specifications for usual machining.
2. Pre-interpolation acceleration/deceleration is valid for the G61.1 command block onward.



3. The feedrate which has been set using command F is limited to the value obtained by the following expression:
   Maximum cutting feedrate **M3** × Speed limit magnification **L76**
   If the value that has thus been obtained is in excess of the rapid feedrate value of **M1**, the **M1** value will become the actual upper limit of feedrate.
4. See Subsection 11-4-6 for arc-machining speed limitation.

11-1-5 Parameters

Parameters related to shape compensation are listed below.
See the Parameter List for further details of the parameters.

L74	Cutting feed rate for pre-interpolation acceleration/deceleration
L75	Cutting feed time constant for pre-interpolation acceleration/deceleration
L76	Speed limit magnification for geometry compensation function
L77	Angle for corner deceleration during optimum corner deceleration
S4	Feedforward gain
J53	Work number of the program to be called up using an M-code
J54	M-code to be used for call
J57	Work number of the program to be called up using an M-code
J58	M-code to be used for call
F72	High-accuracy mode for MAZATROL programs
K107	Acceleration/deceleration filtering magnification

11-1-6 Appendix

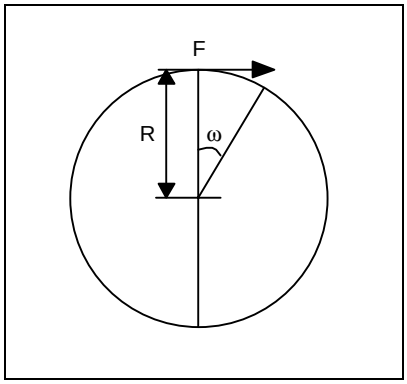
Appended data concerns theoretical values obtained from the conditions of commanded radius, feedforward gain, etc. They may differ from measured values in actual machining depending on cutting conditions, machine accuracy, etc. Apply it as a general criterion for test cuts and others.

1. Maximum feed rate in circular interpolation when geometry compensation function is valid

Maximum feed rate in circular interpolation when geometry compensation is valid can be determined from the following equation.

$\alpha = F \times \omega$ ..... •

$F = R \times \omega$ ..... •



From equations • and •

$$F = \sqrt{R \times \alpha} = \sqrt{R \times \frac{\mathbf{L74}/60}{\mathbf{L75} \times 10^{-3}}} \times 60 \text{ ..... •}$$

F : maximum feed rate (mm/min)
R : Circle radius (mm)
L74 : Cutting feed rate (mm/min)
L75 : Cutting feed time constant

Each maximum feed rate in radius from equation • is as follows:

Parameter **L74** = 30000 (mm/min)
L75 = 400 (msec)

Maximum feed rate for circle radius

Radius (mm)	15	25	50	100	150
Maximum feed rate (m/min)	8.216	10.607	15.000	21.213	25.981

2. Maximum feed rate to keep a geometry error within 10 μm in circular interpolation

Maximum feed rate obtained from a geometry error in circular interpolation can be determined from the following equation.

A. When geometry compensation function is valid

$$F \times 10^3 = \sqrt{\frac{\Delta R \times R}{\frac{1}{2} Tp^2 (1 - Kf^2)}} \times 60 \text{ ..... •}$$

F : Maximum feed rate (m/min)
Tp : Position loop time constant (sec) (= 1/**SV003**)
Kf : Feedforward gain (= **S4**/100)
ΔR : Geometry error (mm)
R : Circle radius (mm)

B. When geometry compensation function is invalid

(In exponential acceleration/deceleration system)

$$F \times 10^3 = \sqrt{\frac{\Delta R \times R}{\frac{1}{2}Tp^2 + \frac{1}{2}Ts^2}} \times 60 \quad \cdots\cdots\cdots\cdots\cdots\cdots \bullet$$

F : Maximum feed rate (m/min)
Tp : Position loop time constant (sec) (= 1/**SV003**)
Kf : Feedforward gain (= **S5**/100)
ΔR: Geometry error (mm)
R : Circle radius (mm)

Each maximum feed rate within 10 μm in geometry error from equations • and • is as follows:

Radius (mm)	15	25	50	100	150
Geometry compensation function valid (m/min)	1.534	1.980	2.801	3.961	4.851
Geometry compensation function invalid (m/min)	0.911	1.177	1.664	2.353	2.882

Parameter **SV003** = 33 (rad/s)
S4 = 70 (%)
N5 = 20 (msec)
ΔR = 10 (μm)

3. Maximum cutting feed in circular interpolation when geometry compensation function is available and geometry error (Theoretical value).

L74	3000	(mm/min)
L75	400	(msec)
Feedforward gain	0.7	
Geometry error	0.01	(mm)
Time constant (G61.1)	0.4	(sec)
Position loop time constant	0.03	(sec)
Time constant (G64)	0.02	(sec)

R (mm)	Geometry compensation function valid			Normal cutting mode	
	Maximum feed 1 (m/min)	Error (mm)	Maximum feed 2 (m/min)	Error (mm)	Maximum feed (m/min)
150	25.981	0.287	4.851	0.028	2.882
100	21.213	0.287	3.961	0.028	2.353
50	15.000	0.287	2.801	0.028	1.664
25	10.607	0.287	1.980	0.028	1.177
15	8.216	0.287	1.534	0.028	0.911
5	4.743	0.287	0.886	0.028	0.526

L74	30000	(mm/min)
L75	400	(msec)
Feedforward gain	0.85	
Geometry error	0.01	(mm)
Time constant (G61.1)	0.4	(sec)
Position loop time constant	0.03	(sec)
Time constant (G64)	0.02	(sec)

R (mm)	Geometry compensation function valid			Normal cutting mode	
	Maximum feed 1 (m/min)	Error (mm)	Maximum feed 2 (m/min)	Error (mm)	Maximum feed (m/min)
150	25.981	0.156	6.576	0.052	2.882
100	21.213	0.156	5.369	0.052	2.353
50	15.000	0.156	3.797	0.052	1.664
25	10.607	0.156	2.685	0.052	1.177
15	8.216	0.156	2.080	0.052	0.911
5	4.743	0.156	1.201	0.052	0.526

L74	30000	(mm/min)
L75	400	(msec)
Feedforward gain	0.99	
Geometry error	0.01	(mm)
Time constant (G61.1)	0.4	(sec)
Position loop time constant	0.03	(sec)
Time constant (G64)	0.02	(sec)

R (mm)	Geometry compensation function valid			Normal cutting mode	
	Maximum feed 1 (m/min)	Error (mm)	Maximum feed 2 (m/min)	Error (mm)	Maximum feed (m/min)
150	25.981	0.011	24.556	0.726	2.882
100	21.213	0.011	20.050	0.726	2.353
50	15.000	0.011	14.178	0.726	1.664
25	10.607	0.011	10.025	0.726	1.177
15	8.216	0.011	7.765	0.726	0.911
5	4.743	0.011	4.483	0.726	0.526

Remark 1: - Maximum feed 1 when geometry compensation function is available represents the maximum feed in circular interpolation.

- Maximum feed 2 when geometry compensation function is available represents the maximum feed to keep a geometry error within 10 μm.
- Error when geometry compensation is available represents a geometry error at the maximum feed 1.

Remark 2: - Maximum cutting feed in normal machining mode represents the maximum feed to keep a geometry error within 10 μm.

- Error in normal machining mode represents a geometry error at the maximum feed 2 when geometry compensation function is available.

Note: An increased feed forward gain causes a decreased geometry error. However, it causes vibration to the machine, which may deteriorate the surface roughness of cutting surface.

11-5 Hard Disk Operating Function

11-5-1 Outline

With the hard disk (HD) operating function, any large-size EIA/ISO program presaved within a special folder of the HD contained in the NC unit can be executed for automatic operation while transferring that program to the NC memory.

11-5-2 Preparation for HD operation

To execute HD operation, it is required first to create a machining program and save it into a special folder of the built-in HD. Create and then save (copy) the desired program into the folder shown below by either the Explorer accompanying the Windows95 system, or another commercially available software.

Folder for HD operation programs: C:\MC_Direct Mode Programs\

11-5-3 Selection of operation mode

1. Selection between HD and conventional TAPE operation modes

The HD operating function is provided as the alternative to the conventional TAPE operating function. Use the following parameter to select between the provided operating functions as required for actual operational conditions:

Address	Name		Program type	Description
	Unit	Setting range	Conditions	
F40	TAPE mode selection		•	The operating function to be selected for the TAPE operation mode. 0: Conventional TAPE operating function 1: HD operating function 2: IC memory card operating function
		0, 1, 2	Immediate	

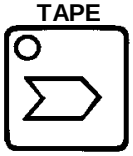
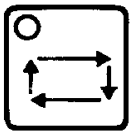
When the HD operating function is selected, the date and time display area will be used for the indication of the special operation mode, as shown below, on the **POSITION** and **TRACE** displays. The indication is given, indeed, during actual HD operation, bot not during selection of the MDI or a manual operation mode nor during execution of a program stored in the NC memory.

2. HD operation modes

The HD operating function can be used in the following two modes:

1. Direct operation mode
   A program stored on the HD operates as the main program.
2. Subprogram operation mode
   A program stored on the HD is called up as the subprogram from a main program registered in the NC memory.

Press the TAPE key to select the direct operation mode, and the MEMORY key to use the subprogram operation mode of the HD operating function.

TAPE 	MEMORY 
TAPE operation key	MEMORY operation key

3. Restrictions concerning the••operating function

General restrictions on the HD operation are as follows:

- Only programs of the EIA/ISO format can be used for the HD operation.
- Programs stored on the HD cannot be checked for tool path and machining shape on the **TOOL PATH** display.
- Programs stored on the HD cannot be edited on the **PROGRAM (EIA)** display. Use a commercially available editor capable of editing text files.
- Programs stored on the HD cannot be presented on the **EIA MONITOR** display.
- The RESTART 2 NONMODAL menu function cannot be applied to a program stored on the HD.
- Programs stored on the HD cannot be subjected to the external work number search.

Further particular restrictions for each HD operation mode are as follows:

<u>For direct operation mode</u>

- The HD main program cannot call another HD program as a subprogram. On a subprogram-call command from the HD main program can only the NC memory be searched for, and an alarm will be given if the specified program is not found there.
- An NC subprogram called up from the HD main program cannot call another HD program as the further subroutine.
- An alarm will be caused if an " M99" command (used for return from subprogram) is given in the HD main program.

<u>For subprogram operation mode</u>

- The HD subprogram cannot call another HD program as a subprogram. On a further subprogram-call command from the HD subprogram can only the NC memory be searched for, and an alarm will be given if the specified program is not found there.
- Sequence number specification for subprogram call by " M98H∗∗" is not effective in calling up

an HD program. The HD subprogram is always executed from the beginning.

- Repeat time specification for subprogram call is not effective in calling up an HD program. The HD subprogram is always executed only one time for one call command.
- The subprogram call from the NC main program preferably selects a program in the NC memory to that on the HD in case the specified work number should have been assigned to the both programs.
- With MAZATROL programs in the NC memory, use a subprogram unit or " G65" (non-modal user macro call) in a manual-program unit for the HD subprogram operation mode.

11-1-4 Program selection for HD operation

This subsection describes the operating procedure for selecting the program in each mode for HD operating function.

1. Direct operation mode

(1) Select the machine operating mode for the direct HD operation mode.

- Press the TAPE operation key  .

(2) Select the **POSITION** display on the screen.

- This is required for selecting the program to be run and checking the operational status of the machine.

(3) Specify the desired program stored on the HD.

- Proceed as follows to specify the program:

1) Press the **WORK No.** menu key.
The display of the menu item is reversed and the work-number listing window appears.

2) Select the desired program from among those stored on the HD.
To specify the program of work No. 100, for example:

Example 1: Press the keys 1 0 0 in this order.

Example 2: Press a cursor key to call up the cursor, move the cursor onto the displayed work number "100" in the window, and then press the INPUT key .

Carry out the above procedural steps, referred to as "HD work number search", to start the direct HD operation.

2. Subprogram operation mode

(1) Select the machine operating mode for the subprogram operation mode.

- Press the MEMORY operation key  .

(2) Select the **POSITION** display on the screen.

- This is required for selecting the program to be run and checking the operational status of the machine.

(3) Specify the desired main program stored in the NC memory.

- Select, or perform work number search for, an NC program (stored in the NC memory) including the subprogram-call command for the desired HD program.
- The operational procedure required here is just the same as that for normal MEMORY operation. No HD subprogram operation, however, takes place if the correct command for calling the HD program is not given in the specified NC program.

11-1-5 Restart in the HD operation mode

The restart function is restricted for the HD operation as follows:

- The RESTART 2 NONMODAL menu function cannot be applied to an HD program.
- The restart function can be executed in the subprogram operation mode only when the selected NC main program is of the EIA/ISO format.

The restarting procedure for the HD operation is also the same as that for normal automatic operation. Refer to 1-2-2 in the Operating Manual, Part 4 “Automatic Operation”.

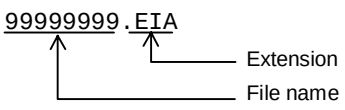
11-1-6 Format for HD operation programs

The programs on the HD to be used for the HD operation must be of the text file format. Use a commercially available text editor capable of editing text files to create the desired programs.

Note: Only programs of the EIA/ISO format can be used for the HD operation.

1. File name

Designate the file of a program to be stored on the HD for HD operation as directed below. The file names here serve as the work numbers to be searched for in actual machine operation.

<table>
<tr><td colspan="2"> 
</td></tr>
<tr><td>File name</td><td>Assign a work number.
- The work number must consist of only numerals (0 to 9).
- The work number may be up to eight digits long.
- The work number must begin with a non-zero numeral.
The leading zero of the file name is always ignored, and thus the NC unit cannot distinguish the following two names, for example, from one another:
1234567.EIA
01234567.EIA
} Both denote work number □1234567□h</td></tr>
<tr><td>Extension</td><td>Identification of an EIA/ISO program
- All programs for the HD operation must have this extension assigned.</td></tr>
</table>

2. Programming format

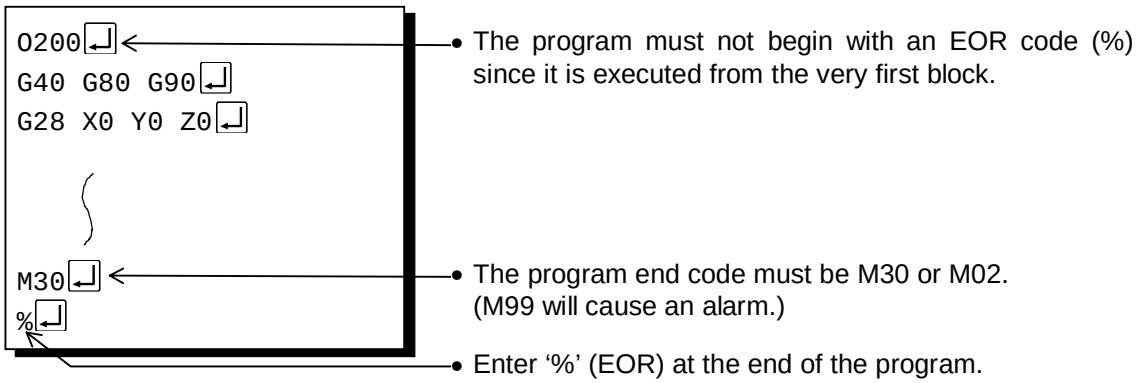
Create HD operation programs in the following text file format:

- One program must be created for one text file.
- The program must not begin with an EOR code (%) since it is executed from the very first block.
- Enter a line feed code (by pressing the return key) at the end of each block. A semicolon ‘•’ must not be used. An alarm will result if a semicolon is included in the program.
- Macro data cannot be used.
- Only characters listed in the table below may be used. Two-byte codes (such as normal-size characters or uni-codes) are not admissible.

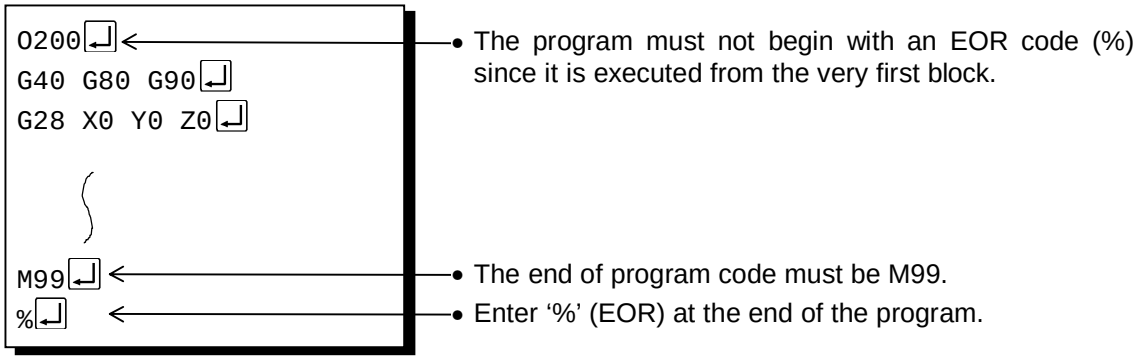
Type	Usable characters	Remarks
Alphabet	- ‘A’•‘Z’ (Capitals only)	- Only one-byte codes can be used. - End of block is to be specified by CR (Carriage Return) or LF (Line Feed).
Numeral	- ‘0’•‘9’	
Symbol	- ‘•’ (Decimal point) - ‘%’ (EOR) - ‘•’•‘•’ (Parentheses)	
Others	- Space code - Line feed code (EOB)	

3. Example of programming

Programs for direct operation mode



Programs for subprogram operation mode



11-1-7 Alarms concerned

No.	Message	Type of error	Stopped status	Clearing procedure	Display
405	PROGRAM No. NOT FOUND (, ,)	A	L	S	Blue
Cause	An attempt has been made to select a program with the work number not regitered on the HD or in the NC memory.				
Action	Check and select the correct work number.				
842	SUB PROGRAM NESTING EXCEEDED (WNo., NNo., BNo.)	B	I	O	Red
Cause	1. An attempt has been made to execute in the direct operation mode an HD program inclucing an “M99” code. 2. An attempt has been made to call from an HD program another HD program as the subprogram.				
Action	1. Do not enter the code “M99” in an HD program for the direct HD operation. 2. The HD main program must not include the subprogram call command for another HD program.				
844	PROGRAM No. NOT FOUND (WNo., NNo., BNo.)	B	I	O	Red
Cause	An attempt of subprogram call has been made for a program with the work number not registered on the HD or in the NC memory.				
Action	Register the desired subprogram on the HD. Or specify the correct work number of an existing program (on the HD or in the NC memory) in the subprogram call command.				

11-6 IC Memory Card Operating Function

11-6-1 Outline

With the IC Memory Card (abbreviated to CARD in this section) operating function, any large-size EIA/ISO program presaved within a special folder of the CARD set into the slot on the front of the NC operating panel can be executed for automatic operation while transferring that program to the NC memory.

11-6-2 Preparation for CARD operation

To execute CARD operation, it is required first to create a machining program and save it into a special folder of the CARD. Create and then save (copy) the desired program into the folder shown below by either the Explorer accompanying the Windows95 system, or another commercially available software.

Folder for CARD operation programs: \MCICPROG\

CAUTION

- Only use IC memory cards of the ATA specification. Failure to observe this instruction could result in machine damage.

11-6-3 Selection of operation mode

1. Selection between CARD and conventional TAPE operation modes

The CARD operating function is provided as the alternative to the conventional TAPE operating function. Use the following parameter to select between the provided operating functions as required for actual operational conditions:

Address	Name		Program type	Description
	Unit	Setting range	Conditions	
F40	TAPE mode selection		•	The operating function to be selected for the TAPE operation mode. 0: Conventional TAPE operating function 1: HD operating function 2: CARD operating function
		0, 1, 2	Immediate	

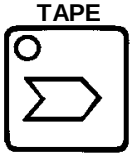
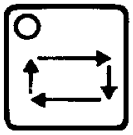
When the CARD operating function is selected, the date and time display area will be used for the indication of the special operation mode, as shown below, on the **POSITION** and **TRACE** displays. The indication is given, indeed, during actual CARD operation, bot not during selection of the MDI or a manual operation mode nor during execution of a program stored in the NC memory.

2. CARD operation modes

The CARD operating function can be used in the following two modes:

1. Direct operation mode
   A program stored on the CARD operates as the main program.
2. Subprogram operation mode
   A program stored on the CARD is called up as the subprogram from a main program registered in the NC memory.

Press the TAPE key to select the direct operation mode, and the MEMORY key to use the subprogram operation mode of the CARD operating function.

TAPE 	MEMORY 
TAPE operation key	MEMORY operation key

3. Restrictions concerning the CARD operating function

General restrictions on the CARD operation are as follows:

- Only programs of the EIA/ISO format can be used for the CARD operation.
- Programs stored on the CARD cannot be checked for tool path and machining shape on the **TOOL PATH** display.
- Programs stored on the CARD cannot be edited on the **PROGRAM (EIA)** display. Use a commercially available editor capable of editing text files.
- Programs stored on the CARD cannot be presented on the **EIA MONITOR** display.
- The RESTART 2 NONMODAL menu function cannot be applied to a program stored on the CARD.
- Programs stored on the CARD cannot be subjected to the external work number search.

Further particular restrictions for each CARD operation mode are as follows:

For direct operation mode

- The CARD main program cannot call another CARD program as a subprogram. On a subprogram-call command from the CARD main program can only the NC memory be searched for, and an alarm will be given if the specified program is not found there.
- An NC subprogram called up from the CARD main program cannot call another HD program as the further subroutine.
- An alarm will be caused if an " M99" command (used for return from subprogram) is given in the CARD main program.

For subprogram operation mode

- The CARD subprogram cannot call another CARD program as a subprogram. On a further sub-program-call command from the CARD subprogram can only the NC memory be searched for, and an alarm will be given if the specified program is not found there.
- Sequence number specification for subprogram call by " M98H∗∗" is not effective in calling up a CARD program. The CARD subprogram is always executed from the beginning.
- Repeat time specification for subprogram call is not effective in calling up a CARD program. The CARD subprogram is always executed only one time for one call command.
- The subprogram call from the NC main program preferably selects a program in the NC memory to that on the CARD in case the specified work number should have been assigned to the both programs.
- With MAZATROL programs in the NC memory, use a subprogram unit or " G65" (non-modal user macro call) in a manual-program unit for the CARD subprogram operation mode.

11-1-4 Program selection for CARD operation

This subsection describes the operating procedure for selecting the program in each mode for CARD operating function.

1. Direct operation mode

(1) Select the machine operating mode for the direct CARD operation mode.

- Press the TAPE operation key  .

(2) Select the **POSITION** display on the screen.

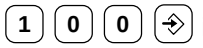
- This is required for selecting the program to be run and checking the operational status of the machine.

(3) Specify the desired program stored on the CARD.

- Proceed as follows to specify the program:

1) Press the **WORK No.** menu key.
The display of the menu item is reversed and the work-number listing window appears.

2) Select the desired program from among those stored on the CARD.
To specify the program of work No. 100, for example:

Example 1: Press the keys  in this order.

Example 2: Press a cursor key to call up the cursor, move the cursor onto the displayed work number "100" in the window, and then press the INPUT key .

Carry out the above procedural steps, referred to as "CARD work number search", to start the direct CARD operation.

2. Subprogram operation mode

(1) Select the machine operating mode for the subprogram operation mode.

- Press the MEMORY operation key  .

(2) Select the **POSITION** display on the screen.

- This is required for selecting the program to be run and checking the operational status of the machine.

(3) Specify the desired main program stored in the NC memory.

- Select, or perform work number search for, an NC program (stored in the NC memory) including the subprogram-call command for the desired CARD program.
- The operational procedure required here is just the same as that for normal MEMORY operation. No CARD subprogram operation, however, takes place if the correct command for calling the CARD program is not given in the specified NC program.

11-1-5 Restart in the CARD operation mode

The restart function is restricted for the CARD operation as follows:

- The RESTART 2 NONMODAL menu function cannot be applied to a CARD program.
- The restart function can be executed in the subprogram operation mode only when the selected NC main program is of the EIA/ISO format.

The restarting procedure for the CARD operation is also the same as that for normal automatic operation. Refer to 1-2-2 in the Operating Manual, Part 4 “Automatic Operation”.

11-1-6 Format for CARD operation programs

The programs on the CARD to be used for the HD operation must be of the text file format. Use a commercially available text editor capable of editing text files to create the desired programs.

Note: Only programs of the EIA/ISO format can be used for the CARD operation.

1. File name

Designate the file of a program to be stored on the CARD for CARD operation as directed below. The file names here serve as the work numbers to be searched for in actual machine operation.

<table>
<tr><td colspan="2">99999999.EIA
Extension
File name</td></tr>
<tr><td>File name</td><td>Assign a work number.
- The work number must consist of only numerals (0 to 9).
- The work number may be up to eight digits long.
- The work number must begin with a non-zero numeral.
The leading zero of the file name is always ignored, and thus the NC unit cannot distinguish the following two names, for example, from one another:
1234567.EIA
01234567.EIA } Both denote work number 1234567</td></tr>
<tr><td>Extension</td><td>Identification of an EIA/ISO program
- All programs for the CARD operation must have this extension assigned.</td></tr>
</table>

2. Programming format

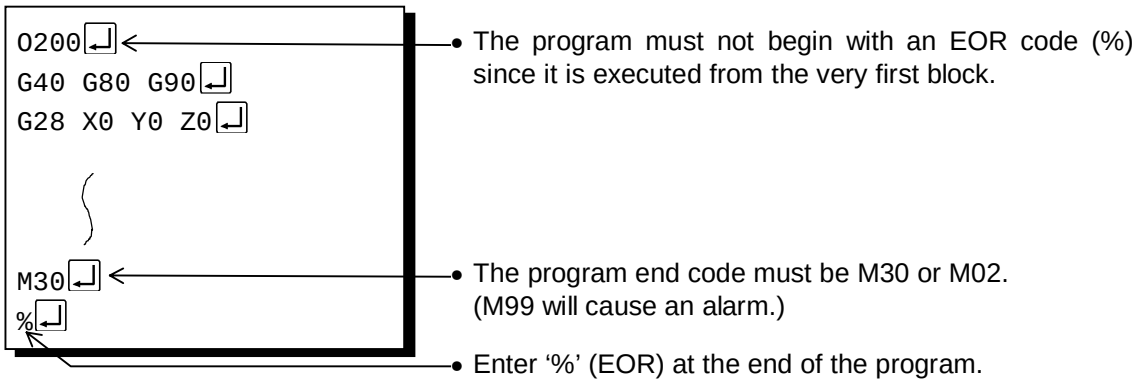
Create CARD operation programs in the following text file format:

- One program must be created for one text file.
- The program must not begin with an EOR code (%) since it is executed from the very first block.
- Enter a line feed code (by pressing the return key) at the end of each block. A semicolon ';' must not be used. An alarm will result if a semicolon is included in the program.
- Macro data cannot be used.
- Only characters listed in the table below may be used. Two-byte codes (such as normal-size characters or uni-codes) are not admissible.

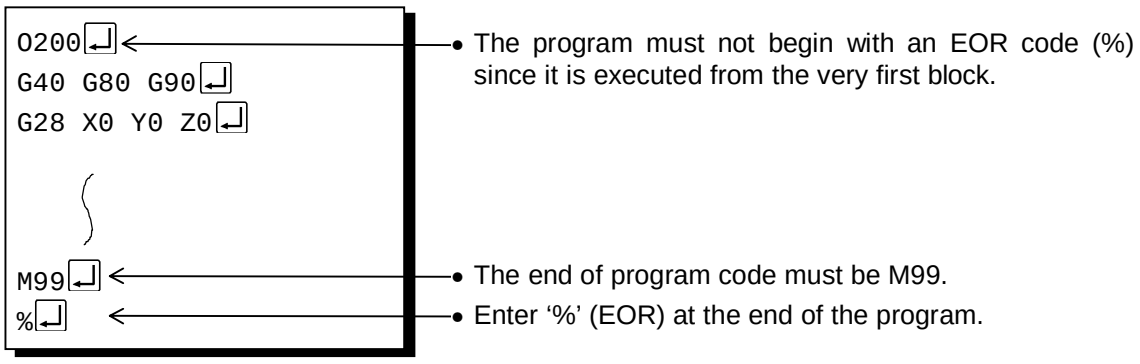
Type	Usable characters	Remarks
Alphabet	- 'A'•'Z' (Capitals only)	- Only one-byte codes can be used. - End of block is to be specified by CR (Carriage Return) or LF (Line Feed).
Numeral	- '0'•'9'	
Symbol	- '•' (Decimal point) - '%' (EOR) - '•'•'•' (Parentheses)	
Others	- Space code - Line feed code (EOB)	

3. Example of programming

Programs for direct operation mode



Programs for subprogram operation mode



11-1-7 Alarms concerned

No.	Message	Type of error	Stopped status	Clearing procedure	Display
405	PROGRAM No. NOT FOUND (, ,)	A	L	S	Blue
Cause	An attempt has been made to select a program with the work number not regitered on the CARD or in the NC memory.				
Action	Check and select the correct work number.				
842	SUB PROGRAM NESTING EXCEEDED (WNo., NNo., BNo.)	B	I	O	Red
Cause	1. An attempt has been made to execute in the direct operation mode a CARD program inclucing an “M99” code. 2. An attempt has been made to call from a CARD program another CARD program as the subprogram.				
Action	1. Do not enter the code “M99” in a CARD program for the direct CARD operation. 2. The CARD main program must not include the subprogram call command for another CARD program.				
844	PROGRAM No. NOT FOUND (WNo., NNo., BNo.)	B	I	O	Red
Cause	An attempt of subprogram call has been made for a program with the work number not registered on the CARD or in the NC memory.				
Action	Register the desired subprogram on the CARD. Or specify the correct work number of an existing program (on the CARD or in the NC memory) in the subprogram call command.				
895	IC CARD I/O BUSY (, ,)	A	I	O	Red
Cause	An attempt has been made to execute the IC memory card operation during data I/O operation with an IC memory card.				
Action	Execute the IC memory card operation after stop or completion of the data I/O operation with an IC memory card.				
938	NO IC CARD MODE OPTION (, ,)	A	K	P	Red
Cause	An attempt has been made to execute the IC memory card operation although the optional function of IC memory card operation is not available.				
Action	This operation cannot be executed because the optional function of IC memory card operation is not available.				

11-7 EIA/ISO Conversion

11-7-1 Outline

Any MAZATROL program that has been created using the M640M can be converted into an EIA program by EIA/ISO conversion. Conversion into an EIA program makes machining possible even with a machine not having a MAZATROL NC.

Two output schemes are available for an EIA/ISO converted program. Differences between these two schemes in specifications are listed in the table below. Both schemes require respective specific options. If both options are provided, select either output scheme using bit 2 of parameter **F88**.

Differences in specifications

<table>
<tr><th colspan="2"></th><th>EIA conversion 1</th><th>EIA conversion 2</th></tr>
<tr><td colspan="2">Bit 2 of parameter F88 when both options are provided</td><td>1</td><td>0</td></tr>
<tr><td rowspan="3">Point-machining</td><td>- Counterboring
- Boring (Roughness level 2 to 9)
- Chamfering (Cycle 2)
- End-milling (Cycles 2, 3)
- Tornado cycle
- MAZAK precision high-speed tornado boring
- Planet tapping</td><td colspan="2">Output with a code from G0 to G4</td></tr>
<tr><td>Synchronous tapping</td><td colspan="2">Output with G84/G74 or G84.2/G84.3 (Parameter-selectable)</td></tr>
<tr><td>Other point-machining patterns</td><td>Output with a code from G0 to G4</td><td>Output of a fixed-cycle code from G73 to G87</td></tr>
<tr><td colspan="2">Feature</td><td>For point machining, since a code from G0 to G4 is issued auto-matically, workpieces can be machined even with a machine not having a fixed-cycle option. Operation is the same as for the original MAZATROL program.</td><td>For point machining, the length of the program decreases since a fixed-cycle code is issued automatically.</td></tr>
</table>

To execute an EIA/ISO converted program using other machines, edit the program according to the particular NC specifications or machine configuration. Before machining a workpiece, edit the functions listed in the table below and then perform checks by cutting test pieces.

Functions required for editing

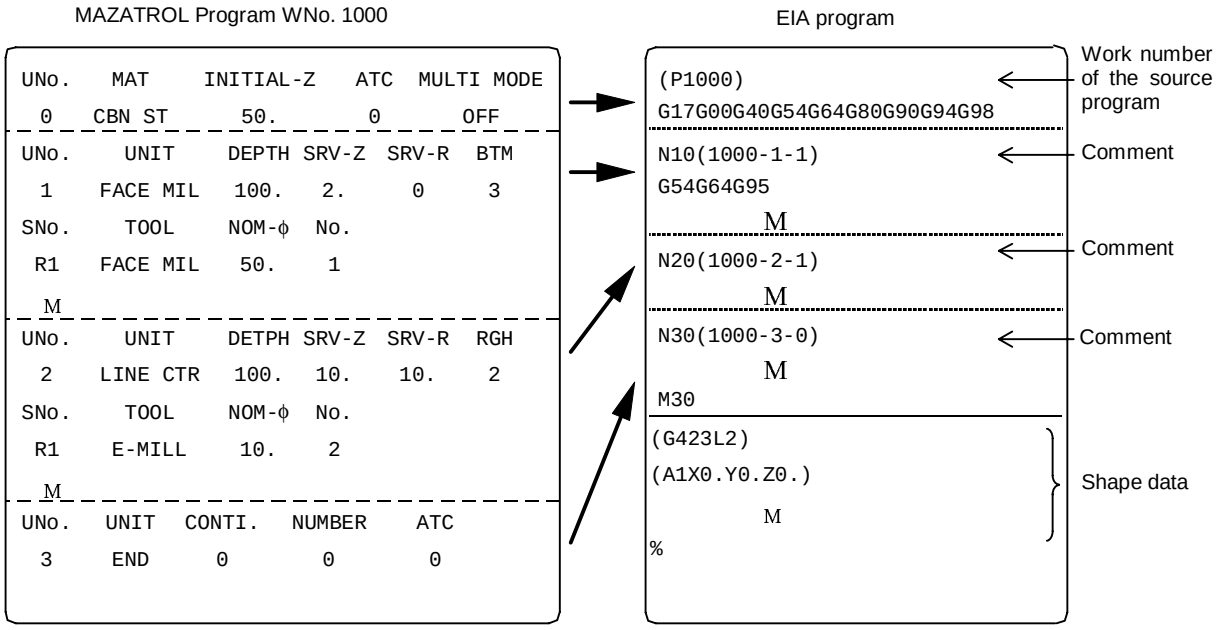
Function	Editing
Workpiece coordinate system rotation (G92.5)	- Since workpiece coordinate system rotation is a characteristic function of the M640M/M PLUS/M-32, this function cannot be used for other machines. Therefore, after changing the θ of the WPC unit into 0, convert the program into EIA form and delete G92.5 from the EIA program. - For the M-32, rewrite G92.5 into G92.1.
M-code (M_ _)	- Edit M-codes into the form appropriate for the intended machine. - During EIA/ISO conversion, a maximum of four M-codes are issued automatically for one block. The number of M-codes which can be included in one block differs according to the particular configuration of the machine. Therefore, edit M-codes into the form appropriate for the intended machine.
MMS (G65 P macro call)	- Since it is a characteristic function of the M640M/M PLUS, this function cannot be used for other machines. Therefore, delete the command code.
Positioning to tool change position (G30)	- Edit this command code into the command code appropriate for the intended machine.
Tool change command (M6T_T_)	- Edit this command code into the tool change command code appropriate for the intended machine.
Geometry compensation (G61.1)	- Edit this command code according to the particular machine specifications.

11-7-2 EIA conversion program format

The format of program conversion from MAZATROL into EIA is shown below. The work number of the source program is assigned on the first line and comments are automatically inserted for each tool sequence. These comments take the format of “source program work number - unit number - sequence number”.

Whether shape data (the data for drawing the shape of the workpiece) is to be added to the end of the EIA program can be selected using bit 0 of parameter **F89**. To add shape data, set this bit to “1”. Set the bit to “0” if addition is not required.

The shape of the workpiece can only be drawn for the M640M.



11-7-3 Output data unit

Coordinate data is always output with a decimal point. If the data is 0, then 0 is output. If the least significant digit in data with a decimal point is 0, then the digit is omitted.

Digit number of decimal fraction

	Metric (mm)	Inch
Micron	3 digits	4 digits
Submicron	4 digits	5 digits

Example 1: 0.000 • 0

Example 2: 10.100 • 10.1

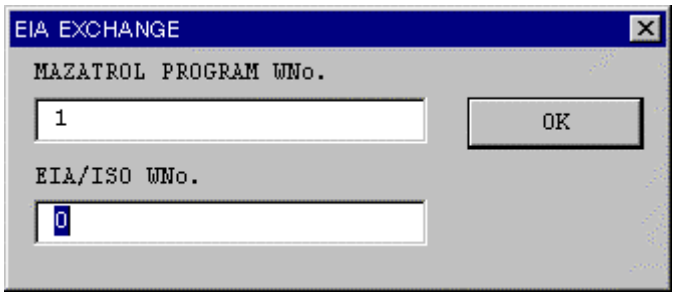
11-1-4 Operating procedure

(1) Move the cursor on the **PROGRAM FILE** display to the line where the work number of the program to be converted is displayed.

- The cursor can be set at any of the items present on the line where the work number of the intended MAZATROL program is displayed.

(2) Press the **EIA/ISO CONVERT** menu key.

➔ **EIA/ISO CONVERT** will reverse and the following window will appear on the display.



(3) Enter a new work number for the conversion destination, and press the INPUT key.

- The conversion can also be started by clicking the OK button in the window, instead of pressing the INPUT key.
- With the window opened (before pressing the INPUT key), the source program number can be changed after selecting the upper text box “MAZATROL PROGRAM WNo.” with the Tab key or the mouse.
- During conversion, EIA/ISO CONVT is displayed above the message display area.
- When conversion is completed, the corresponding EIA/ISO program will be created under the new work number and the message EIA/ISO CONVT will disappear.
- Press the **EIA/ISO CONVERT** menu key once again to stop converting.

Note: During step (2) above, alarm **440 EIA/ISO PROGRAM DESIGNATED** will be displayed if an EIA/ISO program has been designated in step (1).
During step (3) above, alarm **433 SAME PROGRAM EXISTS** will be displayed if another NC-registered program is designated.
During step (3) above, alarm **418 EIA/ISO CONVERTING** will be displayed if an attempt is made to change, delete, or copy the work number of the conversion source program with an EIA/ISO CONVT message remaining on the display.

11-1-5 Supplementary description and precautions for conversion of MAZATROL programs into EIA/ISO programs

MAZATROL programs operate considering the machine status existing during the start of processing. Programs that have been converted into EIA/ISO ones, on the other hand, are those which considered the machine status existing when they were converted into the EIA/ISO format. Therefore, in order to operate other machines, to change tools in the same machine, or to modify tool data, the EIA/ISO converted program must be edited.

1. The parameters that were valid during conversion are considered during output of an EIA/ISO converted program. EIA/ISO converted program will not be affected even if modifications are performed on a parameter that is used in a MAZATROL program.

2. Parameter **F97** can be used to select the type of coordinate system to be used for an EIA/ISO converted program.

F97	Coordinate system
1	G54
2	G55
3	G56
4	G57
5	G58
6	G59

(Codes A to J or codes G54.1P1 to G54.1P48 cannot be selected)

3. To select external WPC using the WPC unit of the conversion source program, set any command code from G54 to G59. Command codes A to J or G54.1P1 to G54.1P48 cannot be set.

4. Data that has been measured using the MMS unit is written into the external WPC (G54 to G59) of the WPC unit. Since external WPC does not include angle θ, measured inclinations of workpieces cannot be subjected to EIA/ISO conversion.

5. For a tool change, data is output in the format of "M6T target tool T next tool".
M6T0T0 is output if the tool to be used is not registered on the **TOOL DATA** display.

6. Shape data can be assigned to an EIA/ISO converted program.

Converted EIA/ISO program

```
        M
G91G30XYZ
G90
M6T7T8
        M
M30
(............
 ...........
 ...........)
```

Shape data

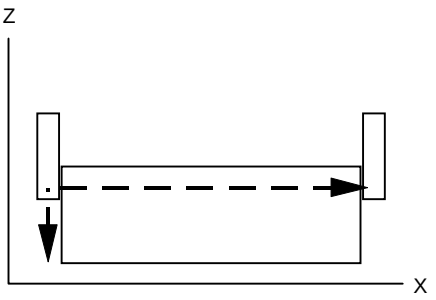
With the shape data, the corresponding shape of machining can be displayed on the **TRACE** and **TOOL PATH** displays. Use bit 0 of parameter **F89** to select whether shape data is to be assigned.

F89 bit 0 = 0 : Does not assign shape data.
= 1 : Assigns shape data.

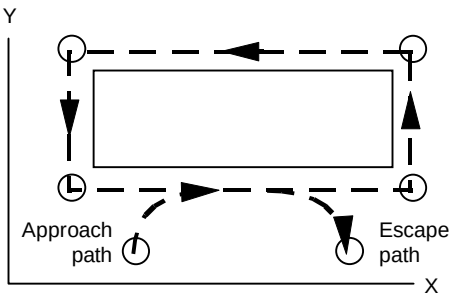
7. If the program to undergo EIA/ISO conversion has overlapping sections, the total program size can be reduced by taking one of those overlapping sections as a subprogram and calling it up multiple times.
A sequence number is set on the beginning line of the subprogram, and M99 on the ending line. M98P_H_ is set in the main program to call up the subprogram.
Whether a subprogram is to be created can be selected by setting bit 0 of parameter **F88**. Set the bit to "1" for creation, or "0" for no creation. A subprogram number can be assigned by the value of parameter **F99** + main program number.
Absolute-data command G90 is used to output a subprogram.

(a) X- and Y-axis line machining, groove machining, and face machining (F-MILL (bidirectional cut), TOP EMIL) are set as a subprogram.

Example:



Z-axial cutting is set as the mainprogram.



X- and Y-axial machining are set as a subprogram.

Main program WNo. 1001

```
(P1000)
G17G0G40G54G64G90G94
   M
N10(1000-1-1)
M98P1002H10
G1Z-2.
   M
M98P1002H10
G0Z100.
   M
N20(1000-2-1)
M98P1002H20
G0Z-2.
   M
N30(1000-3-0)
   M
M30
```

Subprogram WNo. 1002

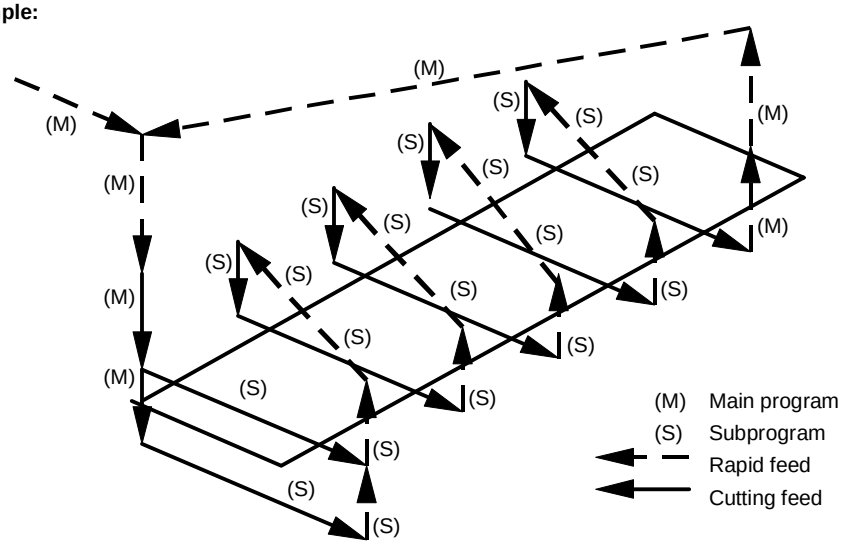
```
N10(1000-1-1)
G0X5.Y2.
G1X50.
   M
G0X60.Y60.
M99
   M
N20(1000-2-1)
G2X10.Y10.R5.
G1X50.
   M
G2X40.Y10.R5.
M99
```

For each machining unit

For each machining unit

(b) Face machining is set as a subprogram (F-MILL (unidirectional), STEP, POCKET, PCKT MT, PCKT VLY)

Example:



Main program WNo. 2001

```
(P2000)
G17G00G40G54G64G90G94G98
N10(2000-1-1)
G80
          M
G1Z  Cutting face height (1)
#  Variable number = Cutting face height (1)
M98P2002H10
G0Z  Initial point
  X ____ Y ____
G1Z  Cutting face height (2)
#  Variable number = Cutting face height (2)
M98P2002H10
G0Z  Initial point
          M
M30
```

Subprogram WNo. 2002

```
N10(2000-1-1)
G1X _____
G0Z  Initial point
  X _____ Y _____
G1Z#  Variable number
  X _____
G0Z#  Initial point
  X _____ Y _____
G1Z#  Variable number
  X _____
       M
M99
```

The value set in the parameter **F98** is output as a macro variable number.

Note 1: If a program is to be subjected to subprogram call with a subprogram unit, that program is not set as a subprogram.

Note 2: Subprogram technique cannot be applied to manual program mode unit.

Note 3: Subprogram technique cannot be applied to multi-workpiece machining .

Note 4: Subprogram technique cannot be applied to three-dimensional machining.

8. For EIA conversion 2, since the point-machining unit is converted into a standard fixed-cycle code, the machine does not operate as minutely as with the original MAZATROL program.

   Operational characteristics in that case are listed below.

   - Cutting clearances always become equal to the setting of parameter **D41**.
   - Deceleration for escape is not valid for drilling.
   - Tapping tools do not move upward during R-point returning.
   - The returning-speed override function is not valid for tapping, reaming, or boring.
   - Returning at the bottom of a hole is not valid for back-boring.
   - For boring and back-boring, the spindle always rotates forward (M03) after initial point/R-point returning.
   - During synchronous tapping, pecking does not occur (if bit 1 of parameter **F89** is set to “1”).
   - The cutting depth does not decrease in steps during drilling.

9. For a line-machining pattern (LEFT, RIGHT, IN, or OUT) or a face-machining pattern (wall finishing), use of bits 4 and 5 of parameter **F88** makes diameter offset (G41, G42) valid for the program that is to be obtained by EIA/ISO conversion. When any of these two offset G-codes is set, the number of D denotes the PKNo. of the tool data.

Diameter offset patterns

	Dia. offset G-code output **F88** bit 4	Dia. offset inclusion in path **F88** bit 5	Use of the same tool as for EIA conversion	Remarks
Pattern 1	ON	OFF	Not required	General EIA pattern
Pattern 2	OFF	ON	Required	Same path as for MAZATROL
Pattern 3	ON	ON	Required	Wear compensation using the G-code

11-1-6 Parameter and alarm lists

Parameters relating to EIA/ISO conversion are listed below.

Parameter list

Parameter	Function	Setting	Description
F88 bit 0	Subprogram creation	0/1	0: Subprogram creation invalid 1: Subprogram creation valid
F88 bit 2	Point-machining G0-G4 output	0/1	0: Fixed-cycle G-code output (EIA conversion 2) 1: G0-G4 code output (EIA conversion 1)
F88 bit 3	Workpiece offset writing	0/1	0: Output invalid 1: G10/G92.5 output valid
F88 bit 4	Tool diameter offset G-code output	0/1	0: G-code output invalid 1: Diameter offset G-code output valid
F88 bit 5	Tool diameter offset inclusion in path	0/1	0: Offset inclusion in path invalid 1: Offset inclusion in path valid
F89 bit 0	Shape data addition	0/1	0: Shape data addition invalid 1: Shape data addition valid
F89 bit 1	Synchronous tapping G-code selection	0/1	0: G84/G74 output 1: G84.2/G84.3 output
F97	Coordinate system selection	1 - 6 Others	1 - 6: G54-G59 output Others: G54 output
F98	Common variable ID number	100 - 199 500 - 699 Others	Set one variable ID number to be used for the main program/subprogram. Others: #100 output
F99	Subprogram number	1 -	Specify the offset from the main work number.

Note: Bit 2 of **F88** is valid only when both options (EIA conversion 1 and EIA conversion 2) are provided.

Alarms relating to EIA/ISO conversion are listed below.

Alarm list

Error No.	Error message	Description	Remedy
613	DATA MISSING IN WPC UNIT	Although the program to be converted into an EiA program contains an MMS unit, a code from G54 to G59 has not been set in the added WPC.	Select a coordinate system by setting a code (G54-G59) in the added WPC.
657	ILLEGAL NUMBER INPUT	Although the program to be converted into an EIA program contains an MMS unit, the G54.1 coordinate system or a code (A to H, J, or K) has been selected for the added WPC.	Select a coordinate system by setting a code (G54-G59) in the added WPC.
703	PROCESS DEFINITION ERROR	A unit not supported under the specifications, such as five-face machining or H/V face definition, has been set.	Delete the unsupported unit.

Section 4:

Automatic Operation

Notes:

Fusion 640M Auto Operation

Notes:

1 AUTOMATIC OPERATION EXECUTION

Automatic operation of the machine according to previously created programs and data is referred to as automatic operation.

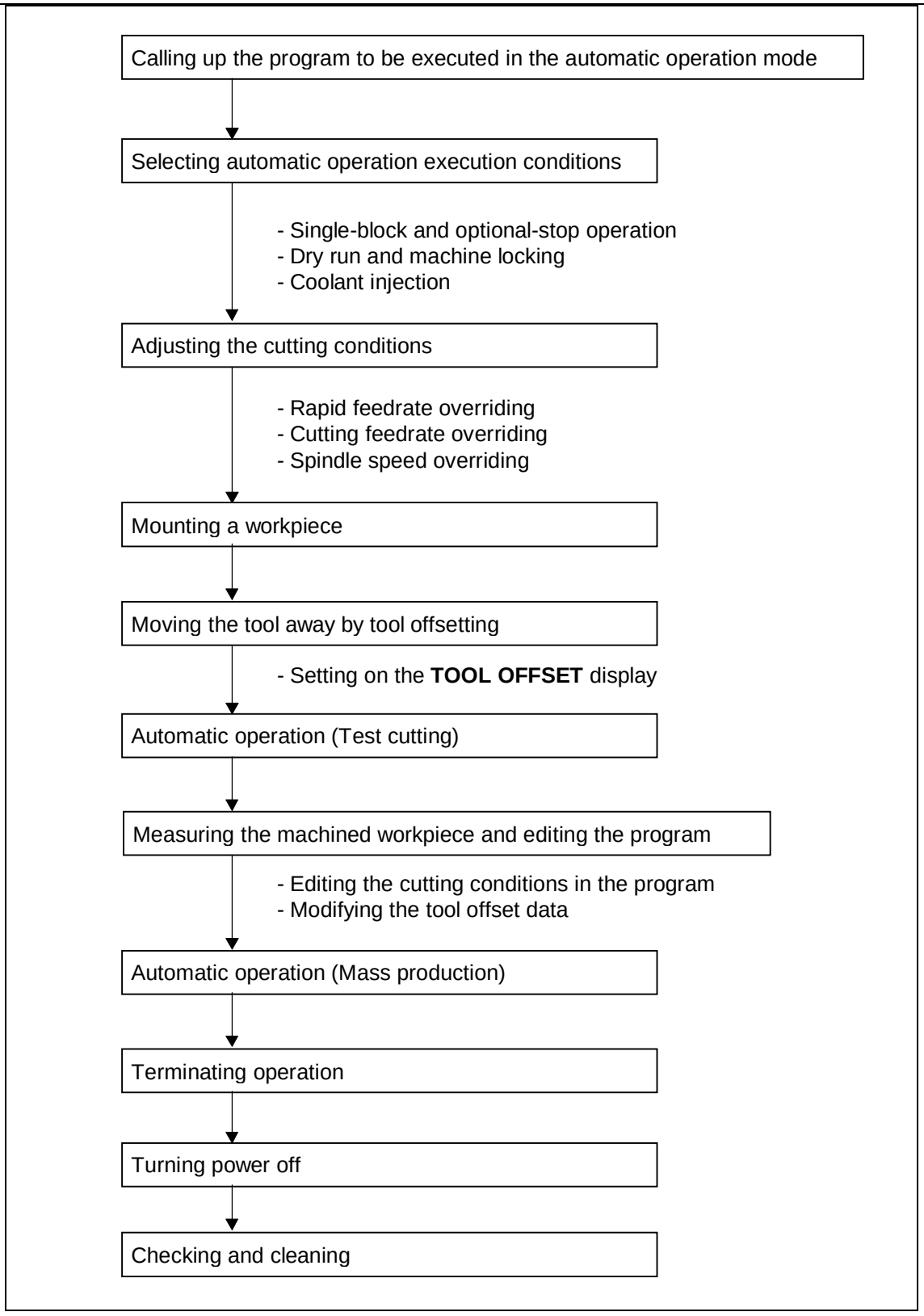
Automatic operation becomes possible just by selecting the program to be executed and then starting it, provided that errors are not present in the program or data. No other operations are required.

During automatic operation, however, since it is possible to stop the machine action, to change the cutting conditions, or to generate hand interrupts, these operations may become necessary.

This chapter describes automatic operation execution procedures.

1-1 Automatic Operation Execution Procedure

The description given below assumes that tooling, machine adjustment, programming, and automatic operation data setting have already been completed. Here, general working procedures to be used after completion of those operational setup jobs are described.



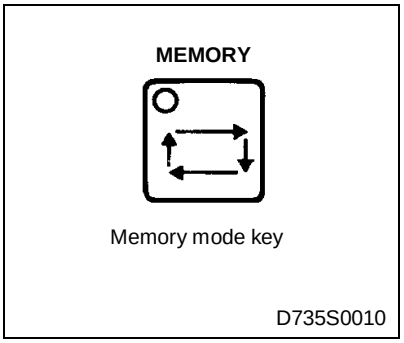
1-2 Types of Automatic Operation Mode

Automatic operation can be executed using the programs registered within the NC unit or EIA/ISO programs stored on paper tapes, microdisks, or other such external media.

The machine can also be made to automatically execute non-program data that the operator may have set as required.

1. Memory operation mode

The memory operation mode is used to execute a program registered in the NC memory.
This mode is selected by pressing the MEMORY key shown in the figure.

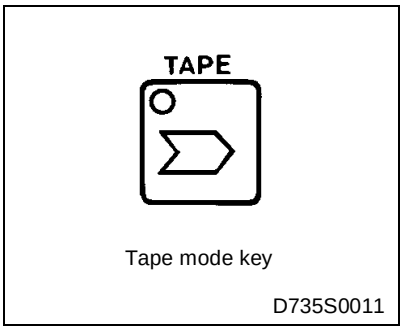


Memory mode key

D735S0010

2. Tape operation mode

The tape operation mode is used if a registered EIA/ISO program within an external unit is to be both loaded and executed at the same time.
The tape operation mode will be selected when the lamp of the TAPE key shown in the figure is turned on by pressing this key.

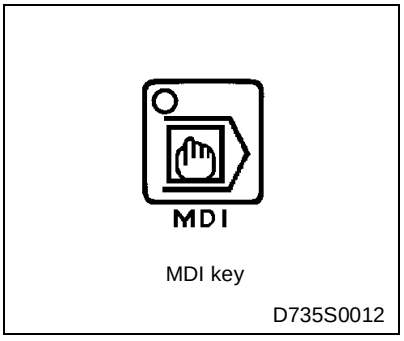


Tape mode key

D735S0011

3. MDI mode

The MDI mode is used to let the machine execute nonprogram data.
This mode is selected by pressing the MDI key shown in the figure.



MDI key

D735S0012

1-2-1 Automatic operation

A. Memory operation mode (to execute the programs registered within the NC unit)

(1) Press the memory mode key 

.

(2) Select a program to be executed. (This operation is referred to as “work number search.”)

- Carry out the following procedure.

1) Call either one of the following displays:
POSITION display
TRACE display
EIA MONITOR display

2) Press the **WORK No.** menu key.

➔ **WORK No.** will change into a reverse display status and the work number window will be displayed.

1) Set the work number of the program to be executed.

Example: To execute the program of work number 100:

[Procedure 1]

Press the keys 1 0 0  in that order.

[Procedure 2]

Press any cursor key to display the cursor in the window and set the cursor to work number 100. Then, press the INPUT key  .

(3) Select automatic operation execution conditions.

- The menu that contains the menu items shown below is referred to as “execution conditions selection menu for automatic operation”. And the key shown below is referred to as “execution conditions selection key for automatic operation”.

Selecting these items and key enables you to determine under what conditions the machine is to be operated in auto mode.

OPTIONAL STOP: If this item is selected, automatic operation can be stopped at M01 position within the program.

DRYRUN: If this item is selected together with the auxiliary (MF1) key, automatic operation can be started at a manually adjusted feedrate instead of the feedrate specified in the program.

MACHINE LOCK: If this item is selected together with the auxiliary (MF1) key, automatic operation can be started without actually performing any axis movements.

WORK LIGHT: If this item is selected, the illumination lamps mounted on the machine can be turned on.

BLOCK SKIP: If this item is selected, any block of the program can be skipped during program execution. This item can be used only if the program is an EIA/ISO program.

ATC STOP: If this item is selected, the machine action will stop upon completion of the machining with the tool currently used.

Z AXIS CANCEL: If this item is selected together with the auxiliary (MF1) key, automatic

operation can be performed with no mouvement of the Z-axis.

TUNING SV PARAM: If this item is selected, the NC unit automatically estimates the characteristics of the machine and the motor after automatic operation, then optimizes these characteristics according to actual machining conditions so that tool path errors do not occur, and sets the optimum characteristic data in the required parameter automatically.

MST LOCK: If this item is selected together with the auxiliary (MF1) key, automatic operation can be performed with the M-, S-, T- and B-codes totally skipped.

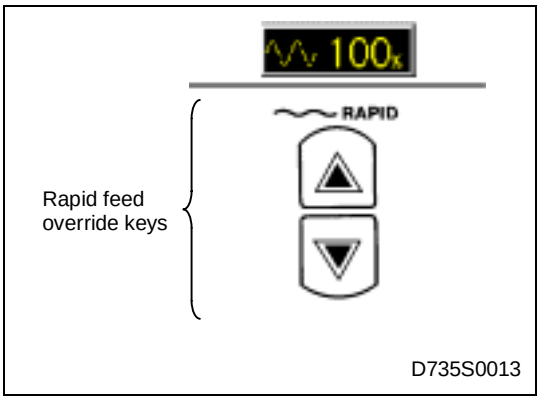
: If this key is pressed, the program can be executed in steps of one block.

Remark 1: During the automatic operation execution, it is possible to select or cancel the execution conditions.

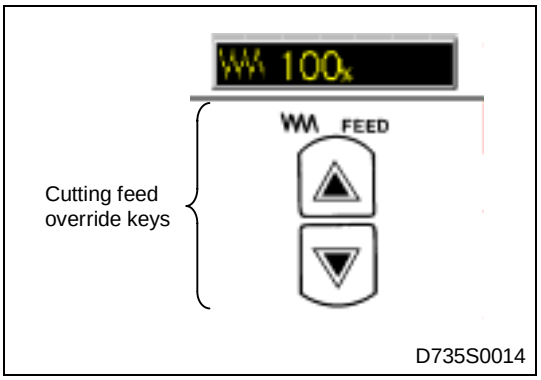
Remark 2: See Chapter 2, “SELECTING EXECUTION CONDITIONS”, for a detailed description of the usage of the above menu items and key.

(4) Adjust the cutting conditions.

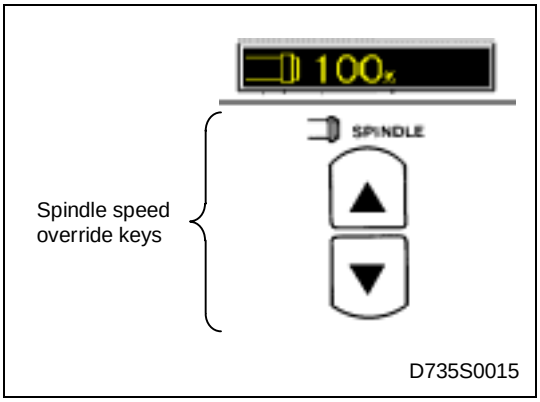
- Perform the following procedure only when adjustments are to be made to the programmed cutting conditions (feedrate and peripheral speed of the spindle).
- Adjust the rapid feedrate using the rapid feed override keys.



- Adjust the cutting feedrate using the cutting feed override keys.



- Adjust the peripheral speed (number of revolutions) of the spindle using the spindle speed override keys.



- During automatic operation, the cutting conditions can be adjusted using the override keys.
- See Chapter 3, “CHANGING THE CUTTING CONDITIONS”, for a detailed description of the usage of the override keys.

Note 1: The cutting conditions specified in the program will remain unchanged even after they have been adjusted using the override keys. Use the VFC function if their overriding is required. See the section of “VFC Function” for details.

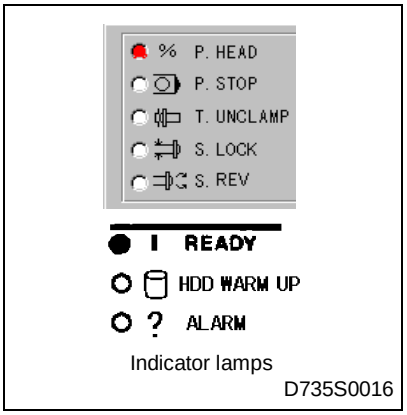
Note 2: The cutting feed override becomes 100% during threading or tapping. After these processes have been completed, the original, preadjusted value is restored automatically.

Note 3: Separate data are stored for automatic operation and manual operation.

(5) Press the reset key 

.

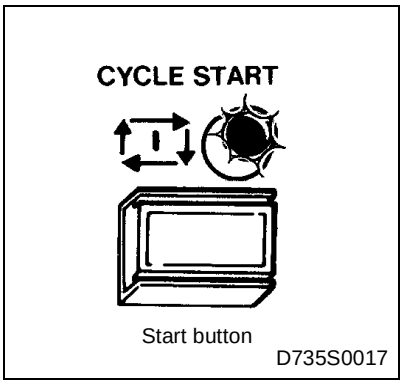
➔ The starting part of the program is called and the P. HEAD (program head) indicator in the machine status indicator window will turn red.



(6) Press the CYCLE START button.

➔ The automatic operation will start.

- The CYCLE START button lamp stays on while automatic operation is in progress, and turns off when automatic operation terminates.



Start button D735S0017

Note: Before pressing the CYCLE START button, make sure that the READY indicator lamp is on.

B. Tape operation mode (to execute EIA/ISO programs stored on external media)

(1) Connect the external unit to the NC unit.

- For connection between the NC unit and the paper tape unit or microdisk unit you are going to use, refer to the relevant Operating Manual for the external unit.

(2) Press the tape mode key 

.

(3) Call the **POSITION** display.

- This display call is required for you to check the intended program and the operational status of the machine.

(4) Select automatic operation execution conditions.

- The selection method is the same as that used when executing the programs registered within the NC unit. (foregoing procedure “To execute the programs registered within the NC unit”.)

(5) Adjust the cutting conditions.

- The adjustment method is the same as that used when executing the programs registered within the NC unit. (foregoing procedure “To execute the programs registered within the NC unit”.)

(6) Call the starting part of the program.

- The calling method is the same as that used when executing the programs registered within the NC unit. (foregoing procedure “To execute the programs registered within the NC unit”.)

(7) Start automatic operation.

- The starting method is the same as that used when executing the programs registered within the NC unit. (foregoing procedure “To execute the programs registered within the NC unit”.)

1-2-2 Restart operation

Select the RESTART menu function if machining that has been aborted for some reason is to be restarted from the corresponding portion of the program.
The procedure up to activation of the restart operation is described below.
The following description assumes that operations, such as a search for the work number of the program you may wish to restart, selection of execution conditions, etc. have already been completed. See the subsection of “Automatic operation” for more details of these operations.

Restart from a part of a subprogram is limited to the following cases:

- The case where both the main program and the subprogram come under the category of MAZATROL programs.
- The case where both the main program and the subprogram come under the category of EIA/ISO programs.

For example, if the main program comes under the category of MAZATROL programs and the subprogram comes under the category of EIA/ISO programs, then it is not possible to restart from that subprogram.

1. To restart a MAZATROL program

A. Restarting the normal MAZATROL programs

(1) Press the reset key 

.

➔ The starting part of the program is called and the P. HEAD (program head) indicator lamp will come on.
Make sure at this time that the READY indicator lamp is on.

Indicator lamps
D735S0016

(2) Press the **RESTART** menu key.

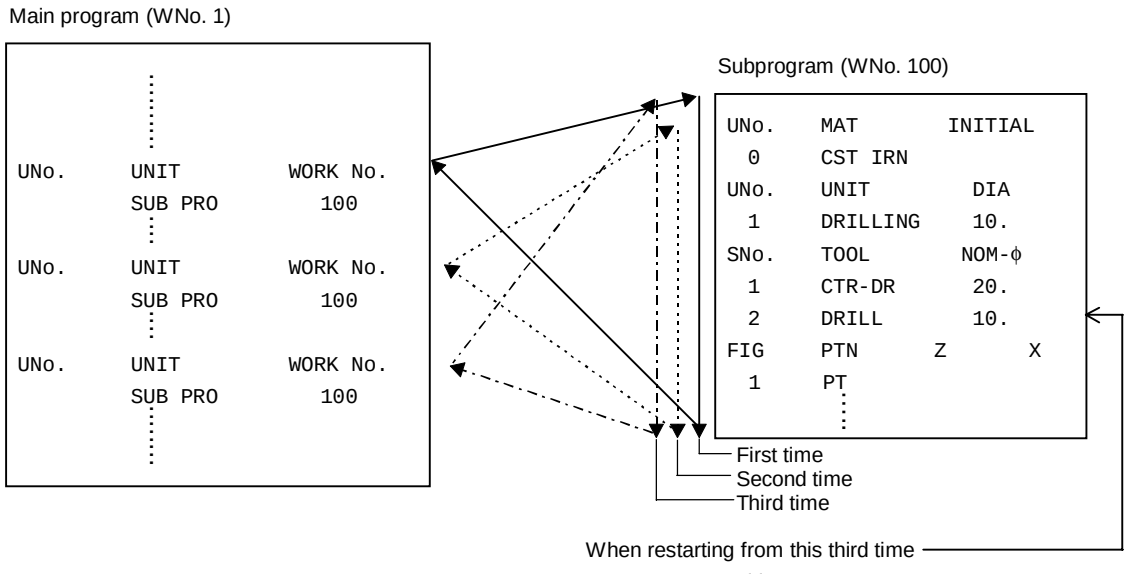
➔ The restart command data window will be displayed at the right side of the bottom of the screen.

(3) Input restart command data.

D735S0018E

- Work No.
- Unit No.
- Sequence No.
- If the unit specified in • is repeatedly used in the program, specify how many times of use are required to reach the intended position for restart.

Example: When it is desired to restart from the third time of subprogram use position indicated by the arrow (•):



1) SUB PROG. No.

Restarting from the subprogram is to take place.

Input the work number of that subprogram by pressing [1], [0], [0] and [⇨], in that order.

2) UNIT No.

Restarting from the drill unit of the subprogram is to take place.

Input 1 by pressing [1] and [⇨], in that order.

3) SEQUENCE No.

Restarting from the drill sequence of the drill unit is to take place.

Input 2 by pressing [2] and [⇨], in that order.

4) REPEAT NUMBER

Restarting from the position in the subprogram by the third use is to take place.

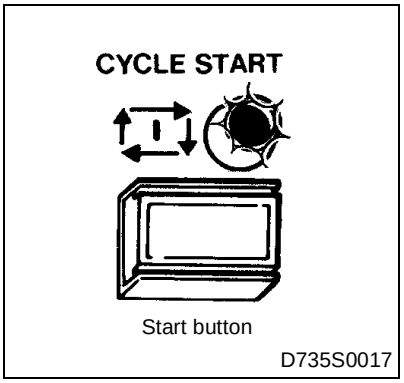
Input 3 by pressing [3] and [⇨], in that order.

Here, if 1, for example, is input, automatic operation will restart from the drill sequence of the drill unit in the subprogram (WNo. 100) by the first use.

(4) Press the CYCLE START button.

➔ The program execution will restart from the specified restart position.

- The CYCLE START button lamp will go out when program execution ends.



Start button

D735S0017

Note: Setting of the unit number:
It is not possible to restart from the following unit. An alarm will occur.

- Fundamental-coordinate (WPC) units
- Auxiliary-coordinate (OFS) units
- Index units
- M-code units with no same-tool priority numbers assigned
- Subprogram units
- End unit

When restarting, a tool sequence is usually to be set as the restart position.

B. Restarting the MAZATROL programs that contain a multi-machining command

The coordinate offsetting in MAZATROL programs that contain a multiple-machining command must be regarded as analogous to subprogram call during data setting for “RESTART NUMBER”.

The other (setting) items are the same as described in **A** above.

Example 1: If program structure is as shown below:

Main program (WNo. 10)

UNo.	SNo.	
0		SAME unit OFS-1 OFS-2
1		WPC unit
2	1 2	DRILLING unit CTR-DR DRILL FIG: PT
3		SUB PRO unit (WNo. 11)
4	1 2	DRILLING unit CTR-DR DRILL FIG: PT
5		SUB PRO unit (WNo. 11)
6		END unit

Subprogram (WNo. 11)

UNo.	SNo.	
0		SAME unit OFS-3 OFS-4
1	1	POCKET unit E-MILL FIG: SQR
2		END unit

The machining order is as follows:

		Order number		Tool	Work number and unit number of the program to be executed
		(1)	OFS-1	Spot drill	WNo. 10, UNo. 2
		(2)	OFS-2	Spot drill	
	(1st time)	(3)	OFS-1	Drill	
A	(2nd time)	(4)	OFS-2	Drill	
	[1st time]	(5)	(OFS-1)+(OFS-3)	End-mill	WNo. 11, UNo. 1 (First call of the subprogram)
	[2nd time]	(6)	(OFS-1)+(OFS-4)	End-mill	
	[3rd time]	(7)	(OFS-2)+(OFS-3)	End-mill	
	[4th time]	(8)	(OFS-2)+(OFS-4)	End-mill	
		(9)	OFS-1	Spot drill	WNo. 10, UNo. 4
		(10)	OFS-2	Spot drill	
	(3rd time)	(11)	OFS-1	Drill	
	(4th time)	(12)	OFS-2	Drill	
	[5th time]	(13)	(OFS-1)+(OFS-3)	End-mill	WNo. 11, UNo. 1 (Second call of the subprogram)
B	[6th time]	(14)	(OFS-1)+(OFS-4)	End-mill	
	[7th time]	(15)	(OFS-2)+(OFS-3)	End-mill	
	[8th time]	(16)	(OFS-2)+(OFS-4)	End-mill	

A: To restart from machining order (4), input data as follows:

1) SUB PROG. No. = 10
2) UNIT No. = 2
3) SEQUENCE No. = 2
4) REPEAT NUMBER = 2

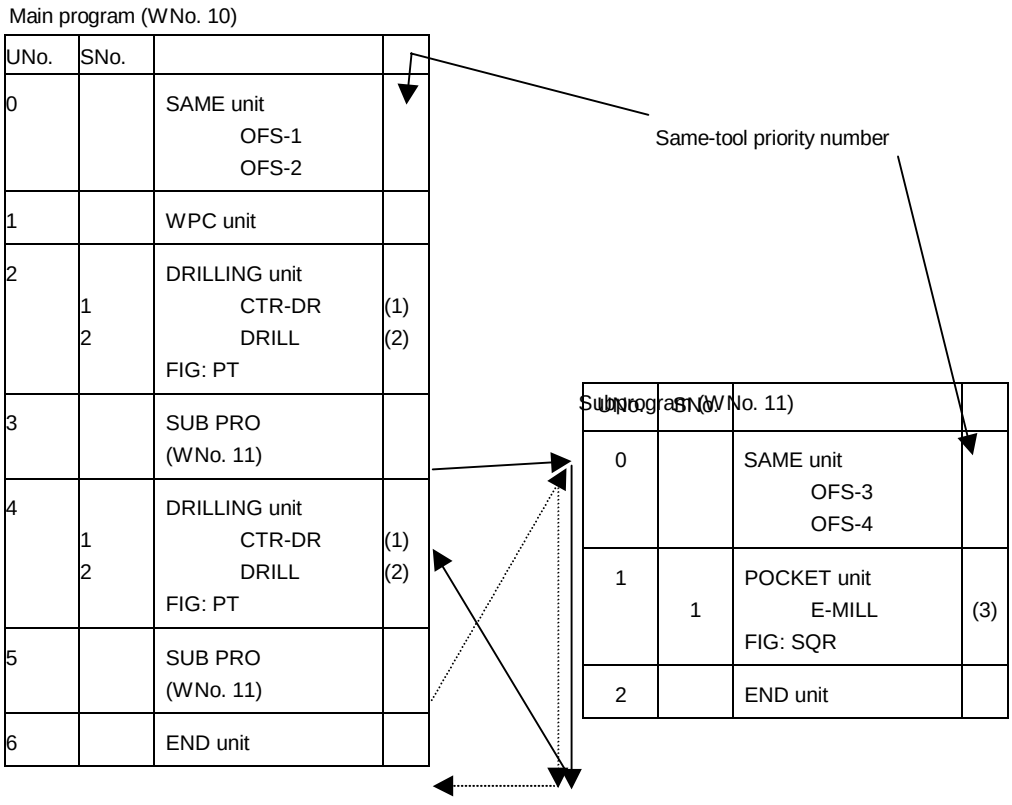
Although restarting from the subprogram is not to take place, the offset operation must be regarded as analogous to the call of subprogram.

B: To restart from machining order (14), input data as follows:

1) SUB PROG. No. = 11
2) UNIT No. = 1
3) SEQUENCE No. = 1
4) REPEAT NUMBER = 6

One call of the subprogram causes the four offset operations. Set, therefore, REPEAT NUMBER = 6 if restarting is to take place from the second offset operation of the subprogram by the second call.

Example 2: If the same-tool priority numbers are assigned as follows in the same program structure as in Example 1 above:



The machining order is as follows.

		Order number		Tool	Work number and unit number of the program to be executed
		(1)	OFS-1	Spot drill	WNo. 10, UNo. 2
		(2)	OFS-1	Spot drill	WNo. 10, UNo. 4
		(3)	OFS-2	Spot drill	WNo. 10, UNo. 2
		(4)	OFS-2	Spot drill	WNo. 10, UNo. 4
	(1st time)	(5)	OFS-1	Drill	WNo. 10, UNo. 2
	(2nd time)	(6)	OFS-1	Drill	WNo. 10, UNo. 4
A'	(3rd time)	(7)	OFS-2	Drill	WNo. 10, UNo. 2
	(4th time)	(8)	OFS-2	Drill	WNo. 10, UNo. 4
	[1st time]	(9)	(OFS-1)+(OFS-3)	End-mill	(First call of the subprogram)
	[2nd time]	(10)	(OFS-1)+(OFS-4)	End-mill	WNo. 11, UNo. 1
	[3rd time]	(11)	(OFS-1)+(OFS-3)	End-mill	(Second call of the subprogram)
B'	[4th time]	(12)	(OFS-1)+(OFS-4)	End-mill	WNo. 11, UNo. 1
	[5th time]	(13)	(OFS-2)+(OFS-3)	End-mill	(First call of the subprogram)
	[6th time]	(14)	(OFS-2)+(OFS-4)	End-mill	WNo. 11, UNo. 1
	[7th time]	(15)	(OFS-2)+(OFS-3)	End-mill	(Second call of the subprogram)
	[8th time]	(16)	(OFS-2)+(OFS-4)	End-mill	WNo. 11, UNo. 1

A': To restart from machining order (7), input data as follows (**A** and **A'** are of the same type of machining):

For 1), 2) and 3) see the statement in Example 1.

4) REPEAT NUMBER of subprogram call = 3

B': To restart from machining order (12), input data as follows (**B** and **B'** are of the same type of machining):

For 1), 2), and 3), see the statement in Example 1.

4) REPEAT NUMBER of subprogram call = 4

As have been described in the above examples, even if automatic operation is to be restarted from the same type of machining, the required number of offset operations depends on the specification of the same-tool priority. Thus, different data must be input to the REPEAT NUMBER item (number of subprogram calls).
When restarting is to be done under a program that contains the multiple-machining command, the number of offset operations should be checked carefully to set data in the REPEAT NUMBER item (number of subprogram calls).

2. To restart an EIA/ISO program

(1) Press the reset key 

.

➔ The starting part of the program is called and the P. HEAD (program head) indicator lamp will come on.
Make sure at this time that the READY indicator lamp is on.

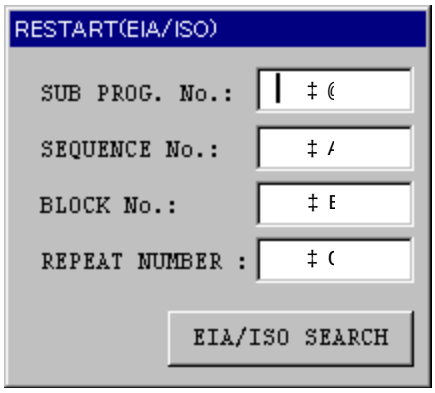
Indicator lamps

D735S0016

(2) Press the **RESTART** menu key.

➔ The restart command data window will be displayed at the right side of the bottom of the screen.

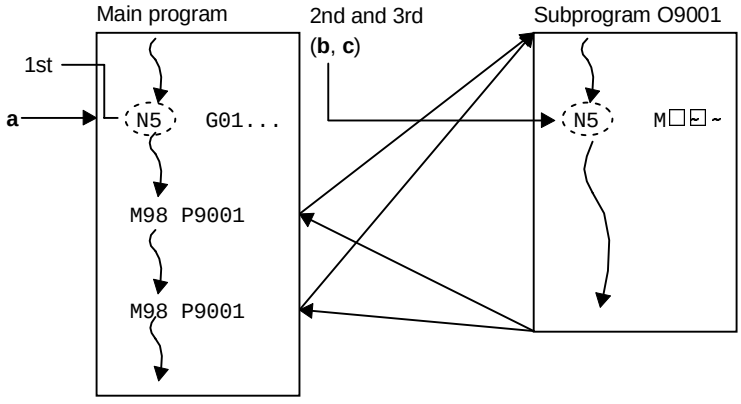
(3) Input restart command data.



D735S0019E

- Work No.
- Sequence No.
- Block No.
- If the sequence specified in • is repeatedly used in the program, specify how many times of use are required to reach the intended position for restart.

Example: If the program is constructed as follows:



In this example, sequence number 5 is used in three different positions.

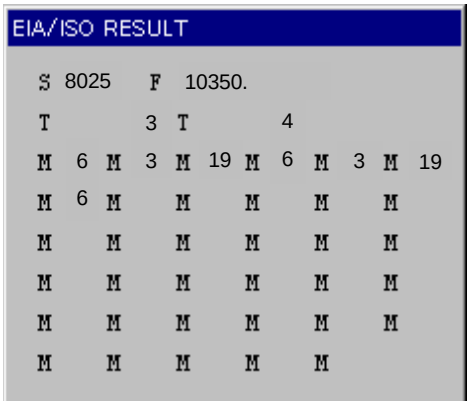
Therefore:
Input “1” to restart from position **a**.

Input “2” to restart from position **b**.
Input “3” to restart from position **c**.

(4) Make sure that all the data items • to •above have been filled in and press the INPUT key.

➔ A search will be made for the restart position.

- After the search is completed, the current status of S-, F-, T- and M-codes resulting from the restart position search will be indicated in the window.



D735S0020E

Note: If the restart position was not found, then an alarm message will be displayed. In that case, the restart command data must be checked for errors.

(5) Press the CYCLE START button.

3. Limitations and notes on restarting

1. Before restarting, data of the items listed below must be made to coincide with those which are required for the restarting position.
   1) For EIA/ISO programs
      - Pallet number
      - M-, S-, T-, and B-codes
   2) For MAZATROL programs
      - Pallet number
      - Index angle
      - Position of the 4th axis

   Note: For programs using the M-code index table, it is possible to restart only from the machining program unit whose table angle is 0 degrees.
2. All the data items must be filled in with data when setting restart data.
3. Under the EIA/ISO programs, the machine moves first to the machining restart position using dry run.

1-1-3 MDI operation

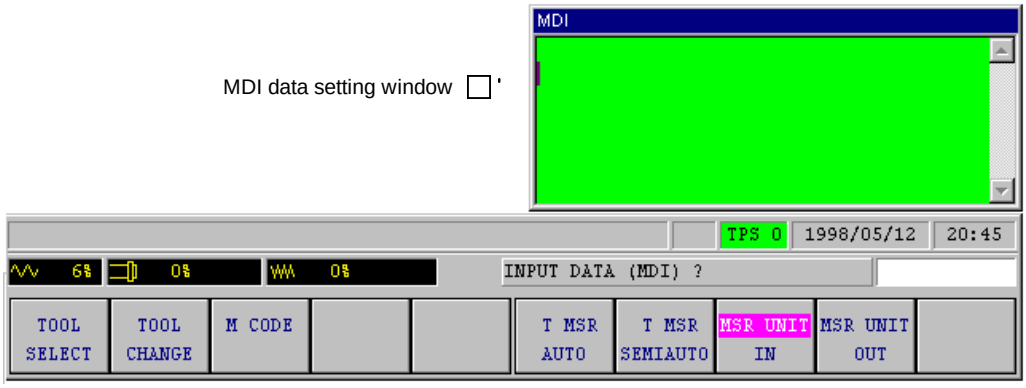
MDI operation refers to the operation in which data that corresponds to an EIA/ISO program is set by operator as required and executed. MDI operation is used to check particular machine operation, to carry out operational setup, to execute simple EIA/ISO program, etc.

1. Basic operation

(1) Press the MDI key 

.

➔ The following MDI data setting window will be displayed in reverse display status at the right side of the bottom of the screen. (The example below shows the MDI data line on the **POSITION** display.)



D735S0021E

(2) Input MDI data.

- Input data using the appropriate address and numeric data keys.

Note 1: Input data in the same format as an EIA/ISO program. Up to 247 characters can be set. (An EOB code (line feed code) requires two characters.)

Note 2: To erase input data, following two methods are used:

- Pressing the clear key  causes the preceding character to be cleared.
- To cancel the entire data that has been set, press the data cancel key 

.

(3) Press the CYCLE START button.

➔ The lamp of the CYCLE START button will be turned on and the data that was set in procedural step (2) will be executed.

- After the entire data has been processed, the CYCLE START button lamp will turn off and the set data will be cleared.

2. Tool select

This operation should be carried out to move the tool accommodated in a magazine pocket to the tool change position.
The operation procedure is described below.

(1) Make sure that the MDI operation mode is selected.

- If the lamp of the MDI key 

is lit, then this indicates that the MDI operation mode is already selected. If the lamp is unlit, press the MDI key.

(2) Press the **TOOL SELECT** menu key.

➔ This will cause the display of **TOOL SELECT** to reverse and the message **POCKET NUMBER?** to be displayed.

(3) Using the appropriate numeric key(s), specify the identification number of the pocket that accommodates the tool to be moved to the tool change position. Then, press the INPUT key.

➔ "T••• M149" will then be displayed in the MDI data setting window and the reverse-display state of **TOOL SELECT** will be released. (••• above denotes the input pocket number.)

(4) Select the conditions for executing the TOOL SELECT operation.

- See the description of procedural step (3) of the item A in Section 1-2-1 "Automatic operation".

(5) Press the CYCLE START button.

➔ This will cause the start button lamp to light up and the tool that was specified in procedural step (3) above to start moving to the tool change position.

3. Tool change

This operation should be carried out to mount a tool from the magazine onto the spindle.

If this operation is carried out with another tool being mounted on the spindle, then this tool will be automatically returned to the required pocket and then the specified tool will be mounted on the spindle.

The procedure for tool change operation is described below.

(1) Make sure that the MDI operation mode is selected.

- If the lamp of the MDI key  is lit, then this indicates that the MDI operation mode is already selected. If the lamp is unlit, press the MDI key.

(2) Press the **TOOL CHANGE** menu key.

➔ This will cause the display of **TOOL CHANGE** to reverse and the message **POCKET NUMBER?** to be displayed.

(3) Using the appropriate numeric key(s), specify the identification number of the pocket that accommodates the tool to be mounted on the spindle. Then, press the INPUT key  .

➔ "Txxx T0 M6" will then be displayed in the MDI data setting window and the reverse-display state of **TOOL CHANGE** will be released. (xxx denotes the specified pocket number, and "T0" denotes the identification number of the next tool specified. In the MDI operation mode, the next tool command is generally displayed as "T0".)

Example: To mount on the spindle the tool accommodated in pocket number 12:

Press keys , , and , in that order.

➔ This will cause "T012 T0 M6" to be displayed in the MDI data setting window.

(4) Select the conditions for executing the TOOL CHANGE operation.

- See the description of procedural step (3) of the item A in Section 1-2-1 "Automatic operation".

(5) Press the CYCLE START button.

➔ This will cause the start button lamp to light up and the tool that was specified in procedural step (3) above to be mounted on the spindle.

- When the tool has been mounted on the spindle, the start button lamp will go out.

4. Setting the M-code

Two methods are available for setting the M-code in the MDI data setting window. One is by using the menu keys, and the other, by specifying the M-code using the appropriate numeric keys. The procedure for setting the M-code is described below.

(1) Make sure that the MDI operation mode is selected.

- If the lamp of the MDI key  is lit, then this indicates that the MDI operation mode is already selected. If the lamp is unlit, press the MDI key.

(2) Press the **M CODE** menu key.

➔ This will cause the message **M CODE (MDI)?** to be displayed and the following "M-code (auxiliary-function code) menus" to appear in the menu display area:

01 OPT STOP.	03 SPNDL FWD	04 SPNDL REV	05 SPNDL STOP	07 MIST STOP	08 FLOOD COOLANT	09 OFF COOLANT	50 AIR BLAST		>>>	(a)

- Each time the >>> menu key is pressed, the menu changes over as follows:

00 PROG STOP	19 SPNDL ORIENT	35 T-BRK DETECT	38 SPNDL GEAR L/M	39 SPNDL GEAR H	33 OUT MSR UNIT	34 IN MSR UNIT			>>>	(b)
16 OPEN ATC CVR	15 CLOSE ATC CVR	11 TOOL UNCLAMP	10 TOOL CLAMP	53 CHIP VAC ON	09 CHIP VAC OFF				>>>	(c)

That is, menu changeover occurs in order of (a)•(b)•(c)•(a)•(b)•....

(3) Set the M-code.

A. Setting the M-code by using the menu keys

Select the function to be used, and press the menu key appropriate to that function.

➔ "M••" will then be displayed in the MDI data setting window.
(•• denotes the identification number of the selected M-code.)

Example: If the **19 SPNDL ORIENT** menu key is pressed, "M19" will be displayed in the MDI data setting window.

B. Setting the M-code by using the appropriate numeric keys

Input the identification number of the desired M-code by pressing the appropriate numeric keys and then press the INPUT key.

➔ "M•••" will then be displayed in the MDI data setting window.
(••• denotes the identification number of the selected M-code.)

Example: To set M71 (pallet No. 1 selection):

Press keys 7, 1, and →.

➔ This will cause "M071" to be displayed in the MDI data setting window.

- With this method, up to three digits are available for one M-code.
- For details of the functions, refer to the Programming Manual (MAZATROL Programming) and the Operating manual of the machine.

Remark: When executing the M-code which has been set as described above, first verify the

execution conditions and then press the start button. (For further details, see the description of procedural step (3) of Section 1-2-1, “Auto mode operation”).

5. Measuring unit IN/OUT

Except some models, the measuring unit must be moved out of its storage position before the tool length can be measured automatically or semi-automatically. In addition, after measurement has been completed, the measuring unit must be replaced in its storage position. The operating procedures for moving the measuring unit in or out (MSR UNIT IN or OUT) are described below. See the following Item 6, “Tool-length measurement”, for details of tool-length measurement.

A. Measuring unit OUT

(1) Make sure that the MDI operation mode is selected.

- If the lamp of the MDI key  is lit, then this indicates that the MDI operation mode is already selected. If the lamp is unlit, press the MDI key.

(2) Press the **33 OUT MSR UNIT** menu key.

➔ “M33” will be displayed in the MDI data setting window.

(3) Press the CYCLE START button.

➔ This will activate the start button lamp and move the measuring unit out of its storage position.

- When the measuring unit has reached the measurement position, the start button lamp will go out and the display of **33 OUT MSR UNIT** will be reversed.

B. Measuring unit IN

(1) Make sure that the MDI operation mode is selected.

- If the lamp of the MDI key  is lit, then this indicates that the MDI operation mode is already selected. If the lamp is unlit, press the MDI key.

(2) Press the **34 IN MSR UNIT** menu key.

➔ “M34” will be displayed in the MDI data setting window.

(3) Press the CYCLE START button.

- This will activate the start button lamp and move the measuring unit back to its storage position.
- When the measuring unit has reached the storage position, the start button lamp will go out and the display of **34 IN MSR UNIT** will be reversed.

6. Tool-length measurement

Before starting a machining operation, it is necessary to measure the lengths of the tools that have been mounted in the tool magazine and then to input the measured data to the TOOL LENGTH item of the **TOOL DATA** display.
This operation is called “tool-length measurement”, for which are available the following three methods:

A. Full-automatic tool-length measurement

In the MDI operation mode, the length of a tool in the tool magazine can be measured automatically. The measured data will be automatically set in the TOOL LENGTH item of the **TOOL DATA** display. However, the tools whose lengths can be measured using this method are limited to tools whose tips are on the centerline of the spindle, that is, spot drills, drills, reamers, and taps.

B. Semi-automatic tool-length measurement

It is possible to semi-automatically measure the lengths of tools whose tips are not on the centerline of the spindle, that is, end-mills, face-mills, chamfering cutters, boring bars, etc. The measured data will be automatically set in the TOOL LENGTH item of the **TOOL DATA** display.

C. Manual tool-length measurement

It is possible with the TEACH function to manually measure the lengths of all types of tools.

The measuring unit must be used for methods A and B, but not be used for method C.
The operating procedures for full-automatic tool-length measurement and semi-automatic tool-length measurement are described below. For details of the manual tool-length measurement procedure and measured-data setting procedure, refer to Section 7-1 “TOOL DATA Display” in PART 3.

Note: Tool-length measurement for EIA/ISO program
On machines with the EIA/ISO option, the results of full- and semi-automatic tool-length measurement can directly be registered in data items on the **TOOL OFFSET** display, under the following conditions:

- Parameter **F93** bit 3 = 0, and
- Menu function of **EIA/ISO ONLY** is selected.

Refer to Section 7-3, “TOOL OFFSET Display” in PART 3 for further details.

A. Full-automatic tool-length measurement

(1) In the MDI operation mode, move the measuring unit out of its storage position.

Remark: Depending on the model of machine being used, the measuring unit may have been locked in the measurement position. If this is the case, then it is not necessary to carry out the “measuring unit OUT” operation (nor “IN” operation).

(2) Call the **TOOL DATA** display on the screen.

(3) Press the **T MSR AUTO** menu key.

➔ This will cause the display of **T MSR AUTO** to reverse and the message **TOOL NUMBER?** to be displayed.

(4) Using the appropriate numeric key(s), specify the identification number (pocket number) of the tool whose length is to be measured. Then press the INPUT key.

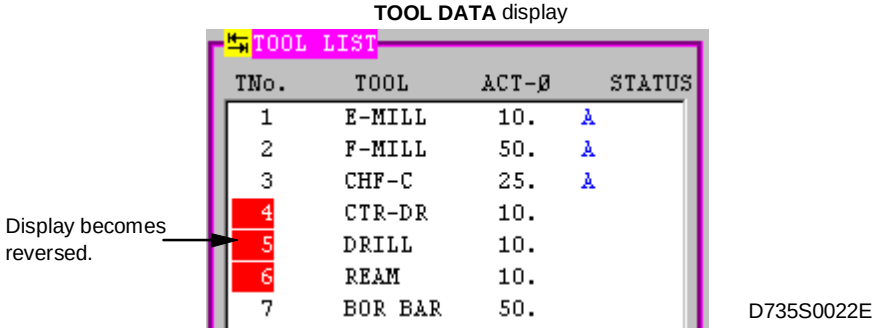
- The number of tools that can be specified at one time is up to 39.
- When the tool number is input, “`M196 T••• T××× .... M197`” will be displayed in the MDI data setting window. (••• and ××× denote the tool numbers that have been input.)

Example: If the lengths of the tools with tool numbers 4, 5, and 6 (tools accommodated in pocket numbers 4, 5, and 6) are to be measured:

Press keys 4, →, 5, →, 6, and →, in that order.

➔ This will cause "M196 T004 T005 T006 M197" to be displayed in the MDI data setting window.

TOOL DATA display



(5) Press the CYCLE START button.

➔ This button action causes the start button lamp to light up and the lengths of the tools which were specified in procedural step (4) above to be measured in sequence. (Tool change occurs automatically.)

- Upon completion of measurement of each tool length, the measured values are automatically indicated in the TOOL LENGTH item of the **TOOL DATA** display and the reverse-display state of the TNo. item is released.
- After the lengths of all the specified tools have been measured, the start button lamp goes out and the reverse-display state of **T MSR AUTO** is released.

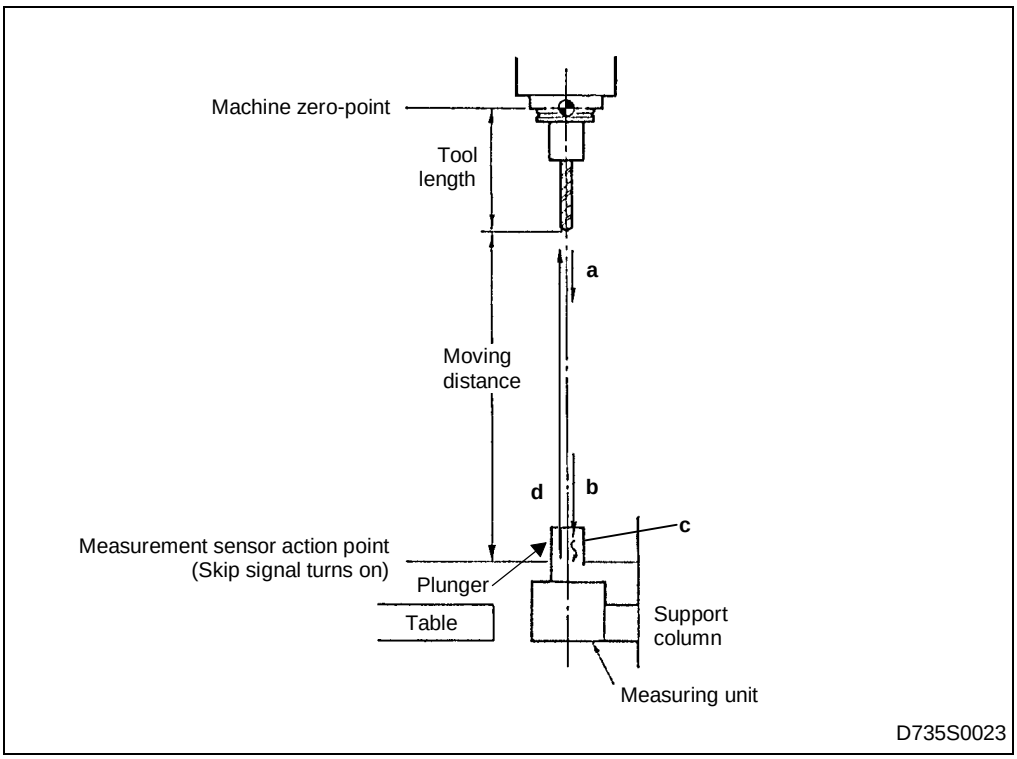


Fig. 1-1 Machine action in full-automatic tool length measurement

Machine action in full-automatic measurement

1) The specified tool is automatically mounted on the spindle.

2) The tool moves downward along an extension of the centerline of the measuring unit. (See **a**.)

3) The tool tip touches the top face of the plunger, forcing it downward. (See **b**.)

4) The downward moving speed of the tool decreases and the tool tip pushes the plunger down to the measurement sensor action point. (See **c**.)

5) When the skip signal turns on, the tool starts moving upward and returns to the machine zero-point. (See **d**.)

6) The NC equipment measures the distance the tool has moved, and calculates the length of the specified tool.

(6) Move the measuring unit back into its storage position.

Note: Do not touch any numeric key on the operating panel while full-automatic tool-length measurement is in progress. Otherwise, measured data may become invalid.

B. Semi-automatic tool-length measurement

(1) Mount on the spindle the tool whose length is to be measured.

- Use procedural steps (1) through (5) of Item 3, “Tool change”.

(2) Move the measuring unit out of its storage position.

(3) Call the **TOOL DATA** display on the screen.

(4) Bring the tool tip close to the plunger of the measuring unit by manual operation. (Action **a** in Fig. 1-2)

Note: Bring the tool tip close to the centerline of the plunger to increase the accuracy.

(5) Press the MDI key to switch over from the manual mode to the MDI operation mode.

(6) Press the **T MSR SEMIAUTO** menu key.

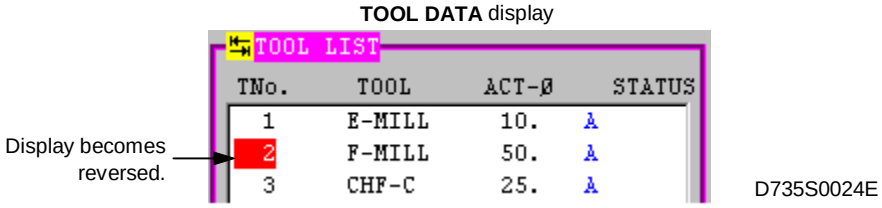
➔ This will cause the display of **T MSR SEMIAUTO** to reverse and the message **MEAS. WITH SPINDLE TOOL <INPUT>?** to be displayed.

(7) Press (input key).

➔ Subsequently, “M198” will be displayed in the MDI data setting window.

Example: If the tool accommodated in pocket number 2 has been mounted on the spindle in procedural step (1):

TOOL DATA display



D735S0024E

(8) Press the CYCLE START button.

➔ The machine acts in the same way as in full-automatic tool-length measurement, and the length of the tool which has been mounted on the spindle will be measured automatically.

- Upon completion of measurement of the tool length, the measured value will be automatically displayed in the TOOL LENGTH item of the **TOOL DATA** display and the reverse-display state of the TNo. item will be released.
- At the same time, the start button lamp will go out and the reverse-display state of **T MSR SEMIAUTO** will be released.

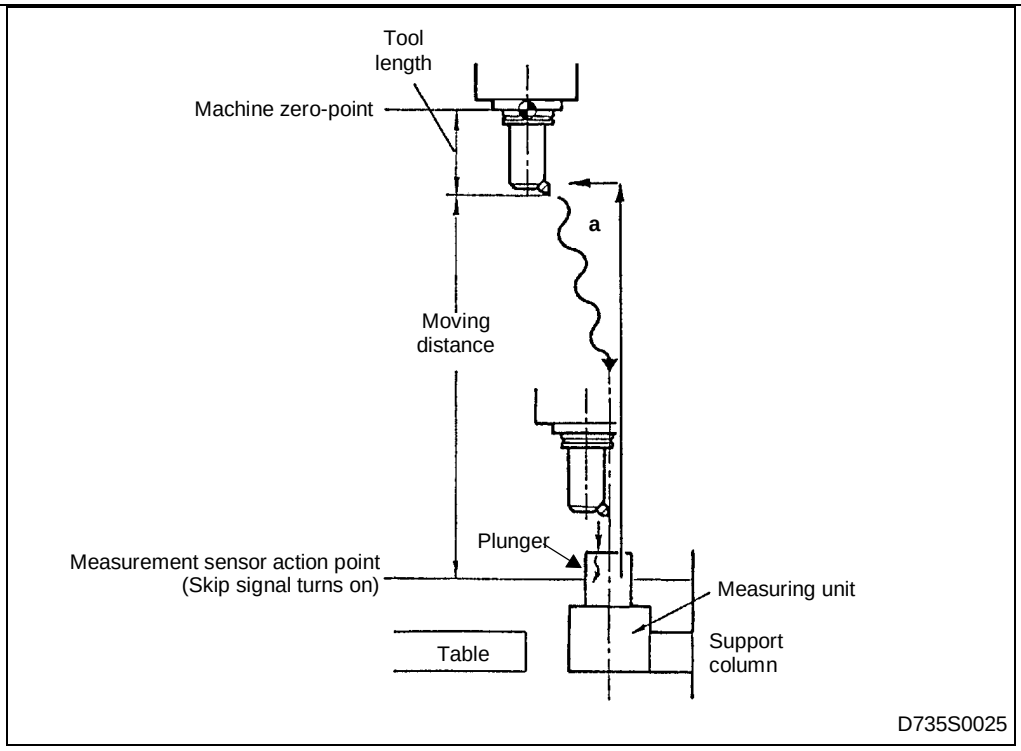


Fig. 1-2 Machine action in semi-automatic tool length measurement

(9) Move the measuring unit back into its storage position.

2. Setting the pallet No.

This operation, which is required only when an automatic pallet change function is provided, should be carried out to register the identification number of the pallet provided in the machine.
Since registered pallet number will still remain in memory of the NC equipment even after power has been turned off, usually, this operation should be carried out only when using pallets for the first time or when a manual pallet-changing operation has been carried out.

(1) Make sure that the MDI operation mode is selected.

- If the lamp of the MDI key  is lit, then this indicates that the MDI operation mode is already selected. If the lamp is unlit, press the MDI key.

(2) Press the menu changeover key several times (located to the right of the menu keys) to call the following menu on the screen:

					PALLET No.SET		SP.TOOL No.SET	NXT TOOL No.SET

(3) Press the **PALLET No. SET** menu key.

- This key action will reverse the display of **PALLET No. SET**, presenting a display of the **PALLET No.?** message.

(4) Using the appropriate numeric key(s), specify the pallet number to be registered. Then, press the INPUT key.

➔ The input pallet number will be displayed in the PALLET No. item of the **POSITION** display.

Example: If pallet No. 2 is to be registered:

Press keys [2] and [⇨], in that order.

3. Setting the spindle-tool No.

The identification number of the tool currently mounted on the spindle is to be registered by this operation. The identification number of the tool currently mounted on the spindle will still remain stored within the memory of the NC equipment even after power has been turned off.

This operation should be carried out only when tools have been manually changed, when the tool has been manually dismounted from, or mounted onto the spindle.

(1) Make sure that the MDI operation mode is selected.

- If the lamp of the MDI key  is lit, then this indicates that the MDI operation mode is already selected. If the lamp is unlit, press the MDI key.

(2) Press the menu changeover key several times (located to the right of the menu keys) to call the following menu on the screen:

					PALLET No.SET		SP.TOOL No.SET	NXT TOOL No.SET

(3) Press the **SP. TOOL No. SET** menu key.

➔ This key action will reverse the display of **SP. TOOL No. SET**, presenting a display of the **TOOL NUMBER?** message.

(4) Using the appropriate numeric key(s), specify the identification number of the tool currently mounted on the spindle. Then, press the INPUT key.

➔ The input tool number and tool name will be displayed in the (left) TNo. item of the **POSITION** display.

- Subsequently, the reverse-display state of **SP. TOOL No. SET** will be released.

Example: If the tool with tool number 5 has been manually mounted on the spindle:

Press keys 5 and INPUT, in that order.

Remark: If the tool has been manually dismounted from the spindle and no tool is currently mounted on it, the tool number 0 should be input.

Note: The magazine pocket corresponding to the tool number which was input in procedural step (4) must be left empty.
If an ATC operation was aborted by pressing the RESET key or the emergency stop button, the tool number of the tool currently mounted on the spindle may have been erased from the memory. In such a case, therefore, check the tool number being displayed in the TNo. item of the **POSITION** display for actuality.

4. Setting the next-tool No.

The identification number of the tool currently in the standby state is to be registered by this operation.
The identification number of the tool currently in the standby state will still remain stored within the memory of the NC equipment even after power has been turned off. This operation should be carried out in case such as those where tools have been manually changed.

The operation procedure is basically the same as that for the item 8 “Setting the spindle-tool No.”. For this operation, press the **NXT TOOL No. SET** menu key instead of the **SP. TOOL No. SET** menu key.

5. Group-number specification for tool management

A. Tool change with group-number specification

In a tool-management system which applies group numbers for T-code specification in EIA/ISO programming, the MDI tool-change operation using the TOOL CHANGE menu function can also be managed by specifying 8-digit group numbers.

(1) Make sure that the MDI operation mode is selected.

- If the lamp of the MDI key  is lit, then this indicates that the MDI operation mode is already selected. If the lamp is unlit, press the MDI key.

(2) Press the **TOOL CHANGE** menu key.

➔ This will cause the display of **TOOL CHANGE** to reverse and the message **SAME TOOL GROUP No.?** to be displayed.

(3) Using the appropriate numeric key(s), specify the group number of the tool to be mounted on the spindle. Then, press the INPUT key.

- "`Txxxxxxxx T0 M6`" will then be displayed in the MDI input window and the reverse-display state of **TOOL CHANGE** will be released. (xxxxxxxx denotes the input group number, and "T0" denotes the identification number of the next tool specified.)

For the following operation steps (4) and (5), perform the same operation as described in the item 3 "Tool change".

B. Next-tool command with group-number specification

The next-tool command can be executed by specifying a group number in the free-format for the MDI operation mode.

Example: If a tool registered under group number 12 is to be mounted on the spindle and a tool of group number 13 is to be indexed as the next tool:
Enter "`T00000012 T00000013 M6`" on the MDI data setting window.

C. Tool select with group-number specification

In a tool-management system which applies group numbers for T-code specification in EIA/ISO programming, the MDI tool-select operation using the TOOL SELECT menu function can also be managed by specifying 8-digit group numbers.

(1) Make sure that the MDI operation mode is selected.

- If the lamp of the MDI key  is lit, then this indicates that the MDI operation mode is already selected. If the lamp is unlit, press the MDI key.

(2) Press the **TOOL SELECT** menu key.

➔ This will cause the display of **TOOL SELECT** to reverse and the message **SAME TOOL GROUP No.?** to be displayed.

(3) Using the appropriate numeric key(s), specify the group number of the tool to be moved to the tool change position. Then, press the INPUT key.

- "`T••••••••M149`" will then be displayed in the MDI data setting window and the reverse-display state of **TOOL SELECT** will be released. (•••••••• above denotes the input group number.)

For the following operation steps (4) and (5), perform the same operation as described in the item 2 "Tool select".

D. Parameters related to TOOL CHANGE and TOOL SELECT

L51	0:	Pocket-number specification
	1:	Group-number specification
F94 bit 4	0:	Group number
(Tool searching)	1:	Tool (pocket) number

If the EIA/ISO option is not provided, only pocket-number specification is valid, irrespective of the value of parameter **L51**.

Note: If the value “0” (group number specification) is set in parameter **F94** bit 4, group-number specification is valid even during automatic operation.

Remark: Register group numbers by setting them in the data item GROUP No. of the **TOOL DATA** display.

2 SELECTING EXECUTION CONDITIONS

The following functions are available with the execution conditions selection menu or key for automatic operation:

1) Optional Stop
2) Dry Run
3) Machine Lock
4) Work Light
5) Block Skip (for EIA/ISO programs only)
6) Single Block
7) Comparison Stop
8) Z-axis Cancel
9) MST Lock
10) ATC Stop
11) TUNING SV PARAM

These functions can be selected for all automatic operation modes (memory mode, tape operation mode, and MDI mode).

During automatic operation, these functions can also be selected or canceled.

Functions 3) and 4) listed above can also be selected during the manual operation mode.

Function 11) above cannot be selected in the MDI mode.

The execution conditions selection menu for automatic operation can be displayed while any one of the following displays stays on the CRT screen:

- **POSITION** display
- **TRACE** display
- **EIA MONITOR** display
- **PALLET MANG.** display

Note: If the execution conditions selection menu is not displayed when one of these displays has been selected, then press the menu changeover key.

A description of each function and its usage follows.

2-1 Optional Stop

Optional Stop refers to the function with which the automatic operation can be optionally stopped at the end of a block containing M01 code.

Select this function when you want to decide step by step whether to continue or to discontinue the automatic operation.

2-1-1 Procedure

Set M01 (optional stop) in the position where automatic operation is to be halted.

- For MAZATROL programs, the M-code can be set in the following positions:
  M-code unit: Data item “M1” through “M8”
  Machining unit: Sequence data item “M”
- For EIA/ISO programs, the M-code can be set in any block.

(1) Press the **OPTIONAL STOP** menu key.

➔ **OPTIONAL STOP** will change into a reverse display status to indicate that the Optional Stop function has become active.

Remark: Performing the operation step (1) again makes this function inactive.

2-1-2 Description

1. If the above procedure is followed by automatic operation, it will stop when the execution of the block containing M01 is completed.
2. Pressing the CYCLE START button causes automatic operation to be restarted from the next block.
3. Procedural step (1) above can also be carried out during automatic operation.

2-2 Dry Run

Dry Run refers to the function that allows you to execute a program using user-adjusted feedrate data, instead of using the feedrate data specified in the program.

Select this function when checking the machining status.

2-2-1 Procedure

(1) Press the **DRYRUN** menu key while pressing the auxiliary key 

on the operation panel.

➔ **DRYRUN** will change into a reverse display status to indicate that the Dry Run function has become active.

- At the same time, the cutting feed override display will change over to the display for the manual operation mode.

Remark: Performing the operation step (1) again makes this function inactive.

Example:

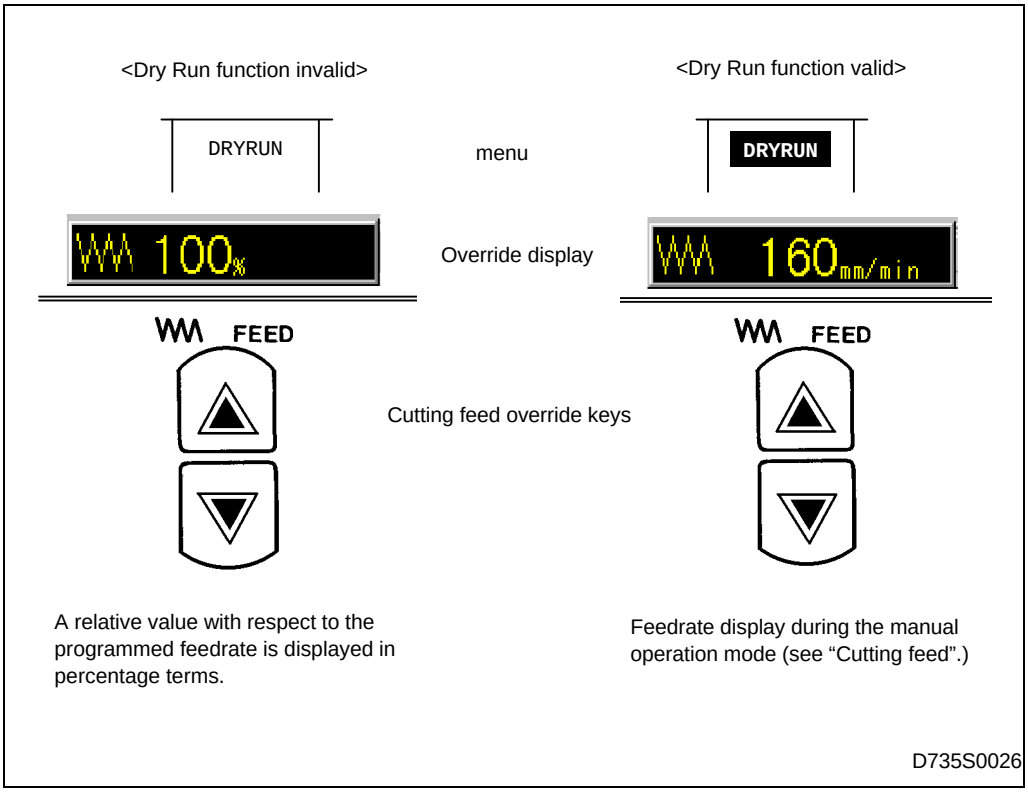


Fig. 2-1 Menu, override display and cutting feed override key state for the Dry Run function valid/invalid

(2) Adjust the feedrate using the cutting feed override keys.

2-2-2 Description

- When the above procedure is followed by automatic operation, the program will be executed using the adjusted feedrate data.
- Carrying out procedural step (1) above during automatic operation nullifies the programmed feedrate and the manual cutting feedrate will be selected.

2-3 Machine Lock

Machine Lock refers to the function that runs the program with no axis movement.

Selecting this function makes the current-position counter indicate the programmed axis movements, even though they do not actually take place.

Use this function when the program execution status is to be checked.

2-3-1 Procedure

(1) Press the **MACHINE LOCK** menu key while pressing and holding down the auxiliary key 

on the operation panel.

➔ MACHINE LOCK will then change into a reverse display status to indicate that the Machine Lock function has become active.

Remark: Performing the operation step (1) again makes this function inactive.

2-3-2 Description

1. When the above procedure is followed by automatic operation, the program will be executed without actual movements of all axes excepting the spindle.
2. Selecting the Machine Lock function during automatic operation brings automatic operation to single-block stop at the end of the block being executed. If automatic operation is subsequently restarted, then the succeeding portion of that program will be executed with the machine remaining locked.
   Do not cancel the Machine Lock function while the machine is in automatic operation.
   If the Machine Lock function is canceled during automatic operation with the machine remaining locked, actual movements of the axes will begin at the end of the block being executed.

However, under a machine locked status, the machine position that the NC unit stores in its memory will more or less differ from the actual machine position. After the Machine Lock function has been canceled, therefore, a collision of the machine can arise since the axis movement will be carried out in such a disturbed coordinates system.

2-4 Work Light

Work Light refers to the function that turns on or off the illumination lamps mounted on the machine.

2-4-1 Procedure

(1) Press the **WORK LIGHT** menu key.

➔ **WORK LIGHT** will change into a reverse display status, turning the lamps on.

Remark: Performing the operation step (1) again makes this function inactive.

2-5 Block Skip

Block Skip refers to the function with which the specific blocks can be optionally skipped during program execution.

This function is only valid for EIA/ISO programs.

2-5-1 Procedure

Include a slash in the starting part of the block(s) to be optionally skipped.

Example: /G

(1) Press the **BLOCK SKIP** menu key.

➔ **BLOCK SKIP** will then change into a reverse display status to indicate that the Block Skip function has become active.

Remark: Performing the operation step (1) again makes this function inactive.

2-5-2 Description

1. When the above procedure is followed by automatic operation, the program will run skipping through the designated block(s).
2. Slash can be set only at the head of the block or after the sequence number.
   For further details see the Programming Manual (EIA/ISO Programming).

2-6 Single Block

Single Block refers to the function used to execute a program on a block-by-block basis.

Select this function to check the machine actions and the machining status.

2-6-1 Procedure

(1) Press the 

key.

➔ The key lamp will turn on to indicate that the Single Block function has become active.

Remark: Performing the operation step (1) again makes this function inactive.

2-6-2 Description

1. When the above procedure is followed by automatic operation, the CYCLE START button lamp will go out when the first block of the program is executed. At the same time, automatic operation will stop.
   When automatic operation is restarted by pressing the CYCLE START button once again, the next block will be executed and automatic operation will stop once again.

   In this manner, once Single Block has been selected, automatic operation restarts each time the CYCLE START button is pressed, and stops each time one block is executed.
2. When the above procedure is performed during automatic operation, it will stop at the end of the block being executed. The Single Block function will become active when automatic operation is restarted by pressing the CYCLE START button.
   The stop of automatic operation at the end of the block being executed is referred to as single-block stop.

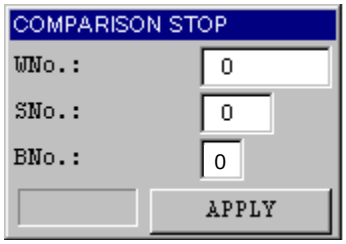
2-7 Comparison Stop

Comparison Stop refers to the function with which the automatic operation can be placed into the single-block stop in any position as occasionally specified with work No., sequence No. and block No.

2-7-1 Setting procedure

(1) Press the **COMPARSN STOP** menu key.

➔ This reverses the display status of **COMPARSN STOP** and the comparison stop window will be displayed at the left side of the bottom of the screen.



D735S0027E

Remark: Performing the operation step (1) again makes this function inactive.

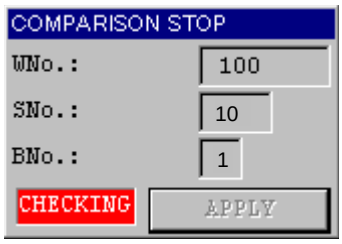
(2) First, specify the stop position with the work No. of the program.

(3) Then, specify the stop position with the sequence No.

(4) Finally, specify the stop position with the block No.

(5) Press the INPUT key .

- The word “CHECKING” will be displayed at the left bottom of the comparison stop window.

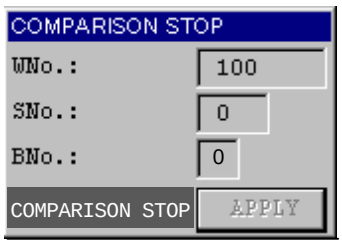


D735S0028E

2-7-2 Description

1. Comparison stop

On completion of the above procedure, start the automatic operation. The machine will come to a single-block stop when execution of the specified block is completed.



D735S0029E

2. Setting conditions

1. Comparison stop can be set or cancelled even during automatic operation.
2. Setting of comparison stop will be automatically cancelled in the following cases:

- The machine operation has come to the comparison stop correctly as specified.
- The system has entered the reset status (at the end of a cycle or by pressing the RESET key).

3. Comparison stop is only available in MEMORY, TAPE , HARD DISK or IC memory card operation mode.

3. Detailed description

1. If the program has multiple blocks which correspond with the specified sequence No. and block No., comparison stop will occur at the first of such blocks.
2. Comparison stop function does not include checking for the presence of the specified block in the respective program.
3. The sequential operation of tapping cannot be brought half-way to a comparison stop.
4. The setting of comparison stop is not valid for a block which is currently executed.
5. Only the setting of 0 for the block No. can be omitted.
6. The head block containing a program No. cannot be effective for comparison stop. In other words, the setting of 0 for both sequence and block Nos. is ineffective.
7. Setting the work No. of a MAZATROL program will cause an alarm (**439 MAZATROL PROGRAM DESIGNATED**).
8. Comparison stop for a subprogram which is called multiple times will occur on the first call.
9. The sequential operation of high-speed machining cannot be brought half-way to a comparison stop.
10. If a block for fixed-cycle call is specified, comparison stop will occur on completion of the first block of fixed-cycle macroprogram.
11. Whichever block in a series of macro statements is specified, comparison stop will always occur at the first block.

    Example: In case of NNo. = 1 and BNo. = 2:

```
     G28 X0 Y0 Z0
     G01 X50.
N01  #100 = 1. ..... Actual stop position
     #101 = 2.
     #102 = 3. ..... Specified position
N02  M02
```

    The designation of any block of the above-mentioned type, however, can be validated for comparison stop by setting parameter **F93** bit 6 (User-macro-statement processing for single-block operation) to “1”.

12. Even if the block succeeding a block to be skipped is specified, comparison stop will occur at the skipped block.

    Example: In case of NNo. = 1 and BNo. = 2 (with **BLOCK SKIP 1** selection)

```
     G28 X0 Y0 Z0
N01  G91 G01 X10. F1000
     /1   X10. ..... Actual stop position
          Y10. ..... Specified position
          Z10.
N02  M02
```

The designation of such a block as mentioned above, however, can be validated for comparison stop by setting parameter **F93** bit 6 to “1”.

13. Even if the succeeding block of M98 in the calling program is specified during comparison stop after the M99 block of the called program, the restarted operation will not come to a comparison stop at the required position.

Example:

WNo. 100 (Main program)	WNo. 200 (Subprogram)
`G28 X0Y0Z0`	`N01 X10.`
`G91 G01 X10. F1000`	`X10.`
`M98 P200`	`X10.`
`M98 P200`.....Specified position	`X10.`
`M02`	`M99` ..... Specification during comparison stop here

2-8 Z-axis Cancel

Z-axis Cancel is a function that allows automatic operation to be performed with the Z-axis remaining locked in place. This function should be selected when checking the movements of the X-axis and Y-axis. Even when this function is selected, all machine actions (including the Z-axis movement) will also be displayed on the screen.

2-8-1 Procedure

(1) Press the **Z AXIS CANCEL** menu key while pressing the auxiliary key 

.

➔ The display of **Z AXIS CANCEL** will reverse.

Remark: Performing the operation step (1) again makes this function inactive.

2-8-2 Description

1. When the automatic operation is started on completion of the above procedure, the specified program will be executed with the Z-axis remaining locked in place.
2. If procedural step (1) is carried out during automatic operation, it will stop after completion of the block being executed (single-block stop), and then the start button lamp will go out. If the start button is subsequently pressed again, automatic operation will restart, at which time the Z-axis Cancel function will be operative.

Note: After the Z-AXIS CANCEL function has been cleared, the actual position of the Z-axis may differ from the programmed position of the Z-axis.

2-9 MST Lock

MST Lock is a function that allows the M-, S-, T- and B-codes to be disregarded during automatic operation. When checking the contents of a program, this function should be selected.

The following codes, however, will be implemented even when this function is selected:

- M00 (Program stop)
- M01 (Optional stop)
- M02 (Program end)
- M30 (Reset and rewind)
- M98 (Subprogram call)
- M99 (Subprogram end)

2-9-1 Procedure

(1) Press the **MST LOCK** menu key while pressing the auxiliary key 

.

➔ The display of **MST LOCK** will reverse.

Remark: Performing the operation step (1) again makes this function inactive.

2-9-2 Description

1. When the CYCLE START button is pressed on completion of the above procedure, the specified program will be executed with the axis movement and M-, S-, T- and B-code functions remaining locked.
2. All the programmed machine actions will be displayed on the screen.
3. If procedural steps (1) is carried out during automatic operation, it will stop after the end of the block being executed (single-block stop), and then the start button lamp will go out.
   If the start button is subsequently pressed again, automatic operation will restart, at which time the MST LOCK function will be operative.

Note: After the MACHINE LOCK function has been cleared, the actual operational status of the machine may differ from the corresponding contents of the program.

Remark: If MST LOCK function is selected together with the MACHINE LOCK function, the program is run only inside the NC unit and no action of the machine will be carried out. Use these functions together when the program is to be checked in the first stage.

2-10 ATC Stop

If the ATC (automatic tool change) STOP menu function is selected during automatic operation, the machine action will stop at the ATC position after the machining with the tool currently used has been completed.

If the ATC STOP menu function is selected during ATC, then the ATC operation will be completed and the machine action will stop immediately before the next ATC operation. Automatic operation can be restarted by pressing the CYCLE START button after cancellation of the ATC STOP menu function.

Operating actions associated with ATC operation slightly differ according to the machine models. For details, refer to the Operating Manual for your machine.

2-11 TUNING SV PARAM

TUNING SV PARAM is a function that lets the NC unit automatically estimate the characteristics of the machine and the motor after automatic operation, then optimize these characteristics according to actual machining conditions so that tool path errors do not occur, and set the optimum characteristic data in the required parameter automatically. Thus, the corresponding axes are driven accurately.

Select this function in the following cases:

1. When mounting a large-mass workpiece and machining it at high accuracy
2. When calculating the optimum data at any specific position of the machine
3. When highly accurate machining is required

To use this function, the Shape Correction Option and the MAZAK Precision Rapid Boring Tornado Option are required. Alarm **936: OPTION NOT FOUND (4, 0, 0)** will occur if this function is selected for a machine not provided with the above two options.

2-11-1 Operation procedure

(1) Select an axis characteristics estimation program on the **POSITION** display. (See Note 2 below.)

(2) Move the spindle to a position near the actual machining location of the table using the manual pulse feed keys and the pulse handel. However, there is the need to move the Z-axis away from the workpiece so that they do not interfere.

(3) Select memory mode and then select **TEST** from the following menu:

WORK No.	BLOCK SKIP	TEST	OPTIONAL STOP	DRYRUN	MACHINE LOCK	WORK LIGHT			RESTART

(4) Select **TUNING SV PARAM**.

MST LOCK	ATC STOP	Z AXIS CANCEL	OPTIONAL STOP	DRYRUN	MACHINE LOCK	WORK LIGHT			TUNING SV PARAM

➔ The display mode of the menu will reverse to indicate that the TUNING SV PARAM function has become valid.

Remark: This function will become invalid when step (4) above is repeated once again.

Note 1: Since the axis characteristics differ slightly according to the particular on-table position of the axis, move the spindle to a position near the actual machining location of the table before executing the characteristics estimation program. When the Precision Rapid Boring Tornado Cycle is specified to be executed at multiple positions, move the spindle to a position either near the circle of the highest degree of priority, or in the center of the table before executing the characteristics estimation program.

Note 2: Provide the following program for the axis characteristics estimation program:

1. The recommended circular operation is 25 seconds or less in total starting time, 250 mm (9.84 in.)/sec^2 in acceleration, and 5 m/min or more in feed rate.
   The combination of the feed rate and circle radius for an acceleration of at least 250 mm (9.84 in.)/sec^2

Feed rate / Radius	5 m/min	7 m/min	10 m/min
25 mm (0.98 in.)			
50 mm (1.97 in.)			
100 mm (3.94 in.)			

$a = F^2/r$

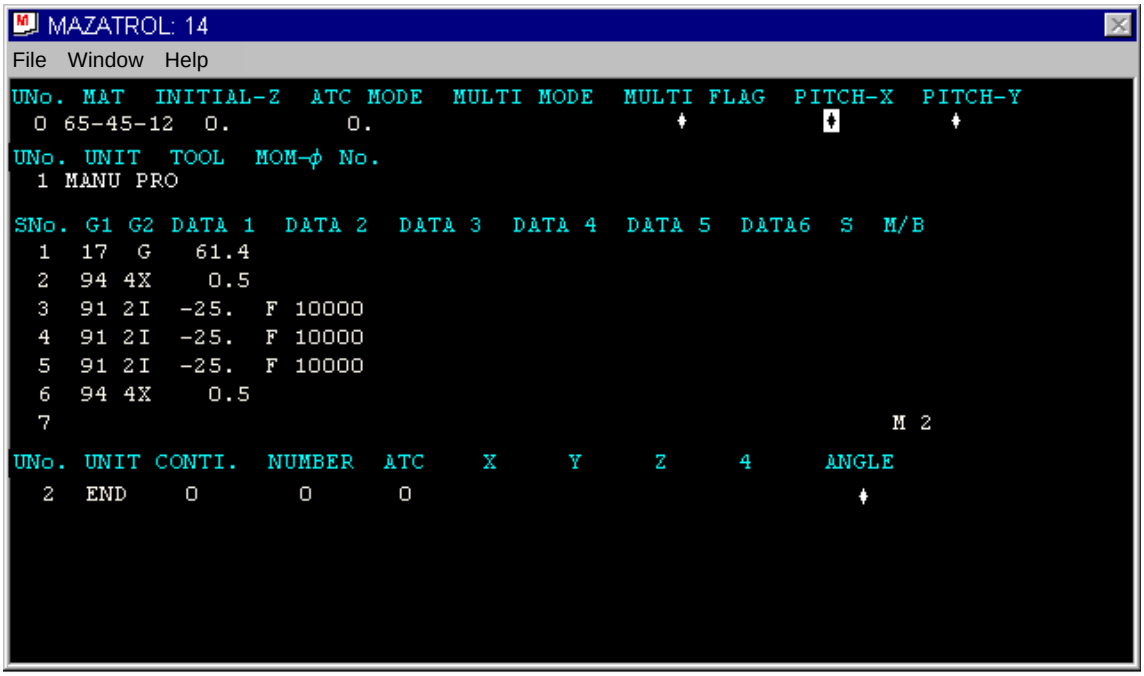
a: Acceleration (mm/sec^2)
r: Radius of the circle (mm)
F: Feed rate (mm/min)

2 M02 or M03 is required for characteristic data calculation and parameter setting.

(1) For EIA program:
Circular operation of 25 mm (0.98 in.) in radius and 10 m (393.7 in.) in feed rate is repeated three times on the X-Y plane.

```
G17
G61.4
G94G04X1.
G91G02I-25.F10000
G91G02I-25.F10000
G91G02I-25.F10000
G94G04X1.
M30
%
```

(2) For MAZATROL:
Circular operation of 25 mm (0.98 in.) in radius and 10 mm (0.39 in.) in feed rate is repeated three times on the X-Y plane in the manual program mode unit.



D735S0051E

2-11-2 Function description

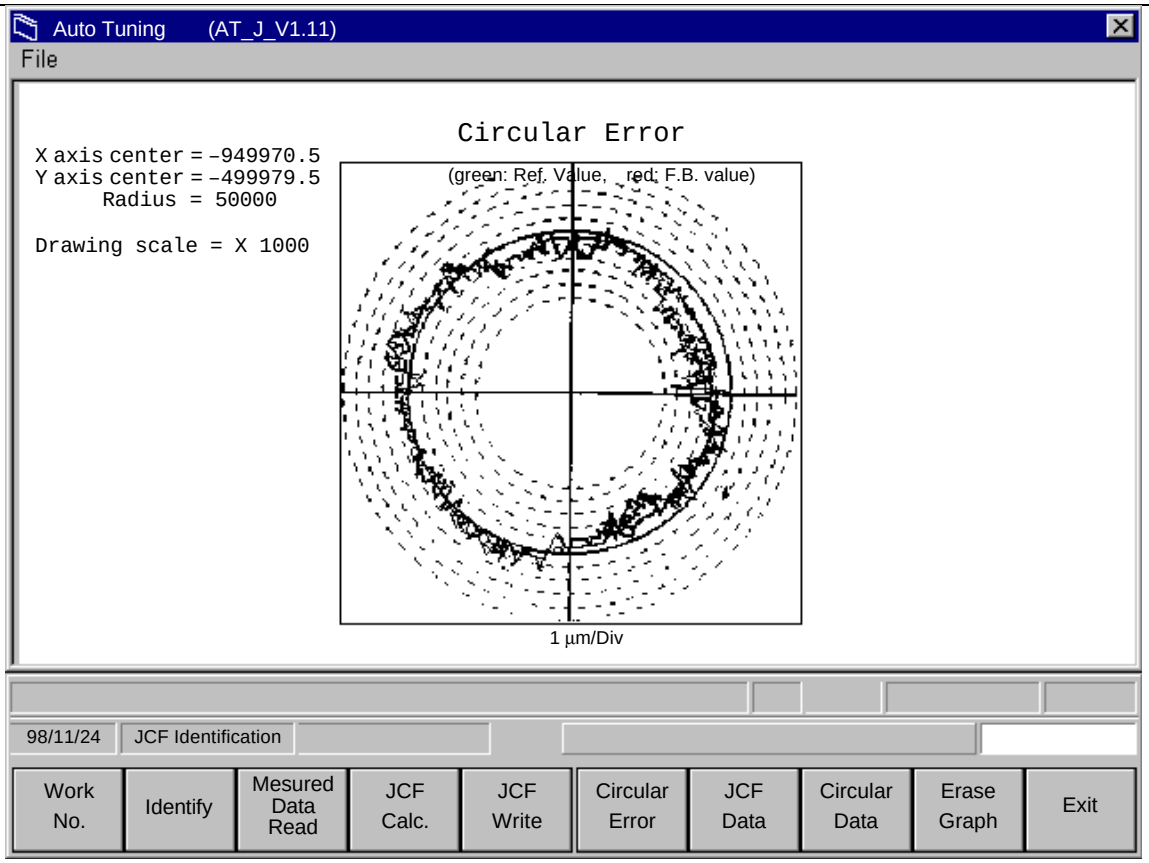
After the above operations, when automatic operation is started, the characteristics estimation program will be executed.

By this program the NC unit estimates the characteristics of the machine and the motor, then calculates the optimum characteristic data for machining, and sets the data in the required parameter. After data setting, the reverse display of TUNING SV PROGRAM is cleared.

Note 1: Set the cutting override to the same value as an actual machining value.

Note 2: Repeating the TUNING SV PARAM function two to three times is recommended. The greater number of times of repetition gives more accurate characteristic data for machining.

Note 3: Setting bit 0 of the **F82** parameter to “1” lets the NC unit calculate the characteristic data in the program, then set the data in the parameter, and display the command data and motor movement as a circular error graph on the display.



D735S0052E

When the **Exit** menu function is selected using the mouse pointer, the window will close and the reverse display of TUNING SV PARAM will be cleared.

- NOTE -

3 CHANGING THE CUTTING CONDITIONS

During automatic operation, especially during test cutting, the cutting parameters specified in the program may require changing according to the shape of cutting chips, the status of its flyabout, the load status display of the spindle and axis driving motors on the **POSITION** display, the degree of as-finished surface roughness of the workpieces, and other factors.

This chapter describes how to modify cutting parameters using the override keys and how to update cutting parameters using the VFC function.

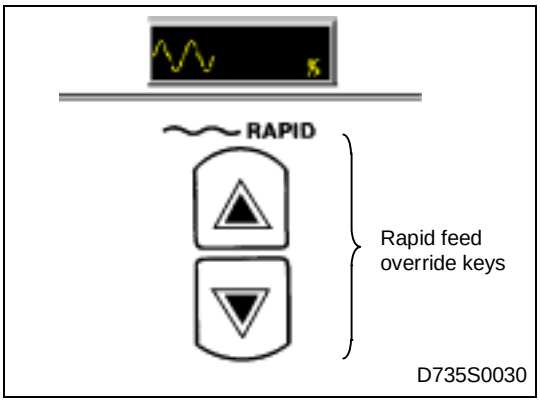
3-1 Rapid Feed Override

Using the rapid feed override keys, the rapid feedrate during automatic operation can be adjusted to within the range from 0 to 100 percent of the maximum rapid feedrate specified by parameter **M1**. The values of R0, R1 and R2 are set in other parameters.

3-1-1 Procedure

(1) Press the rapid feed key ▲ or ▼.

- The rapid feedrate increases in the order of R0 • R1 • R2 • 50% • 100% each time the ▲ key is pressed, and continuously increases when this key is pressed and held down.
- The rapid feedrate decreases in the order of 100% • 50% • R2 • R1 • R0 each time the ▼ key is pressed, and continuously decreases when this key is pressed and held down.



Example: If the maximum feedrate for each axis is preset as follows:

30000 mm/min for the X-axis
30000 mm/min for the Y-axis
24000 mm/min for the Z-axis

Overriding 50% changes actual rapid feeds into:

X-axis: 30000 × 50/100 = 15000 mm/min,
Y-axis: 30000 × 50/100 = 15000 mm/min,
Z-axis: 24000 × 50/100 = 12000 mm/min.

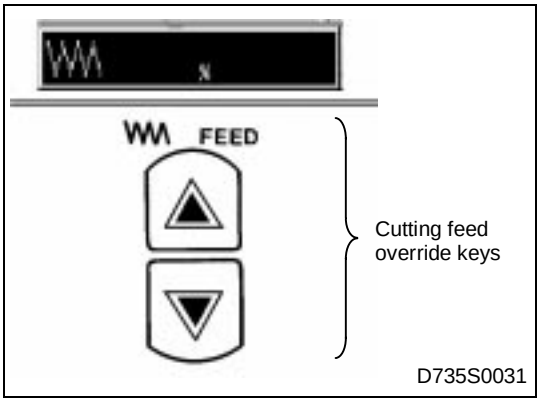
3-2 Cutting Feed Override

Using the cutting feed override keys, the cutting feedrate specified in the program can be changed within the range from 0 to 200 percent.

3-2-1 Procedure

(1) Press the cutting feed override key  or  .

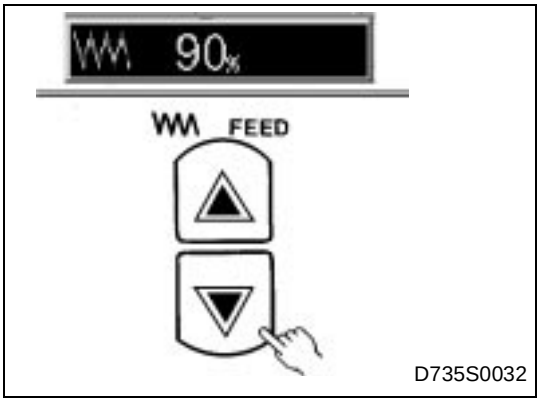
- The cutting feedrate increases in 10% steps each time the  key is pressed, and continuously increases when this key is pressed and held down.
- The cutting feedrate decreases in 10% steps each time the  key is pressed, and continuously decreases when this key is pressed and held down.
- The current cutting feedrate relative to the original value is displayed in percentage terms on the screen.



Example: If the programmed feedrate is 1000 mm/min:

Overriding this value using the percentage value shown in the figure will change the feedrate into:

1000 × 90/100 = 900 mm/min.



Note: The cutting feedrate will not change even if the cutting feed override keys are pressed while a tapping cycle is in progress.

Remark: Axis feeding will not occur if 0 is displayed on the screen.

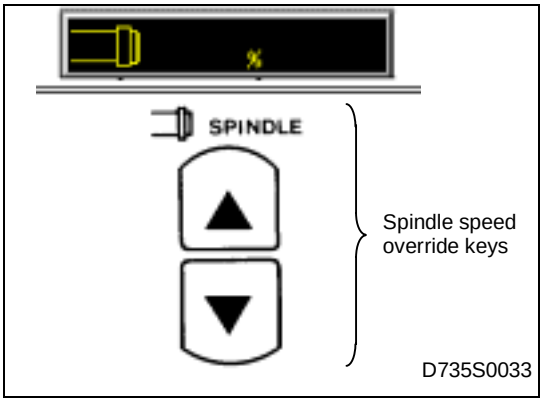
3-3 Spindle Speed Override

Using the spindle speed override keys, the programmed rpm (revolutions per minute) value of the spindle can be changed within the range from 0 to 150 percent of the original value.

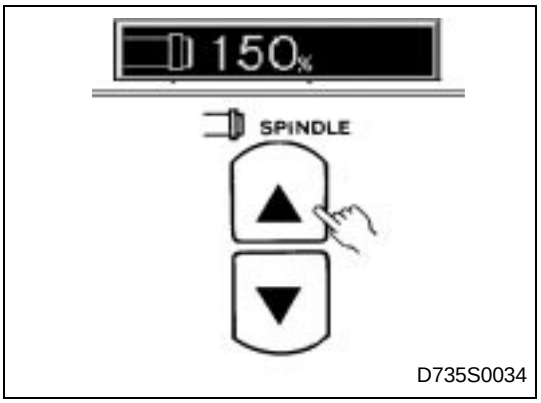
3-3-1 Procedure

(1) Press the spindle speed override key ▲ or ▼.

- The spindle speed increases in 10% steps each time the ▲ key is pressed, and continuously increases when this key is pressed and held down continuously decreses when this key is pressed and held down.
- The spindle speed decreases in 10% steps each time the ▼ key is pressed, and held down.
- The current spindle speed relative to the programmed value is displayed in percentage terms on the screen.



Example: If the programmed spindle speed is 2000 min^{-1}(rpm):
Overriding 150% as shown in the figure will change the spindle speed into:
2000 × 150/100 = 3000 min^{-1}(rpm).



Note 1: The spindle will not rotate if 0 is displayed on the screen.
When synchronous feeding, neither occurs axis feeding.

Note 2: The spindle speed will not change even if the spindle speed override keys are pressed while a tapping cycle is in progress.

Note 3: Even if the spindle speed specified using the spindle speed override keys is in excess of the maximum allowable value, the actual spindle speed will not become higher than the maximum allowable value.

3-4 VFC Function

VFC refers to the function that automatically overrides the cutting conditions in the actual program, such as feedrates, peripheral speeds (or number of revolutions), with the new data that have been set using the override keys.

The VFC function can be used only for MAZATROL programs.

The cutting conditions that can be overriden using the VFC function are limited to the cutting feedrates and spindle peripheral speeds (or number of revolutions).

3-4-1 Procedure

(1) Change the cutting feedrate and peripheral speed as required during automatic operation.

- See 3-2, “Cutting Feed Override” and 3-3, “Spindle Speed Override”.

(2) Press the VFC key 

.

➔ The VFC key lamp will then light up.

3-4-2 Description

1. When machining in the tool sequence being executed is completed, the cutting feedrate and peripheral speed that have been changed in procedural step (1) above will be automatically written into the program, and the VFC key lamp will go out.

Example: When the cutting feedrate and peripheral speed have been specified in the program as follows:

Cutting feedrate: 20 mm/min

Peripheral speed: 2 m/min

After changing these values as follows in procedural step (1),

Cutting feedrate: 150% (i.e., 20 × 150/100 = 30 mm/min)

Peripheral speed: 120% (i.e., 2 × 120/100 = 2.4 m/min)

Press the VFC key.

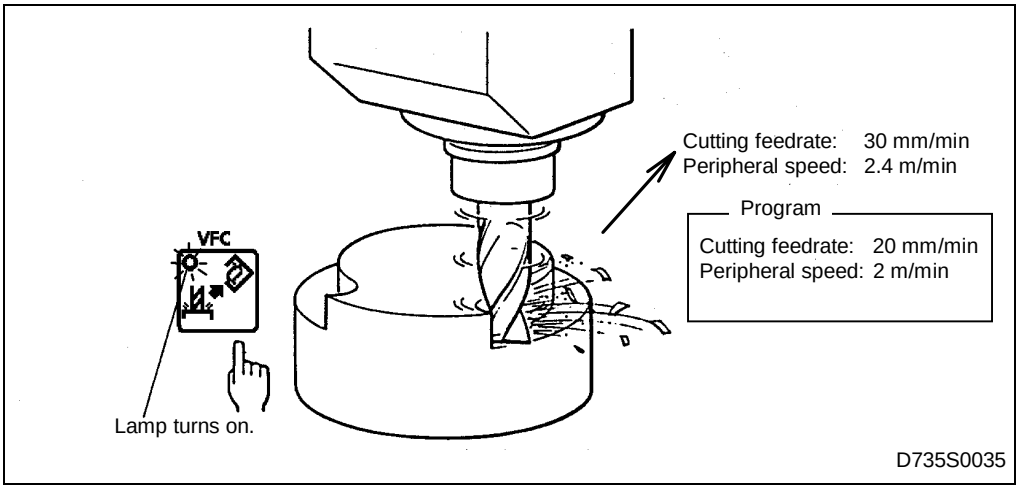


Fig. 3-1 Reserved status of updating the cutting conditions data in the program — VFC

By this time, the actual cutting feedrate and peripheral speed will have already been changed, but not yet the corresponding values in the program.

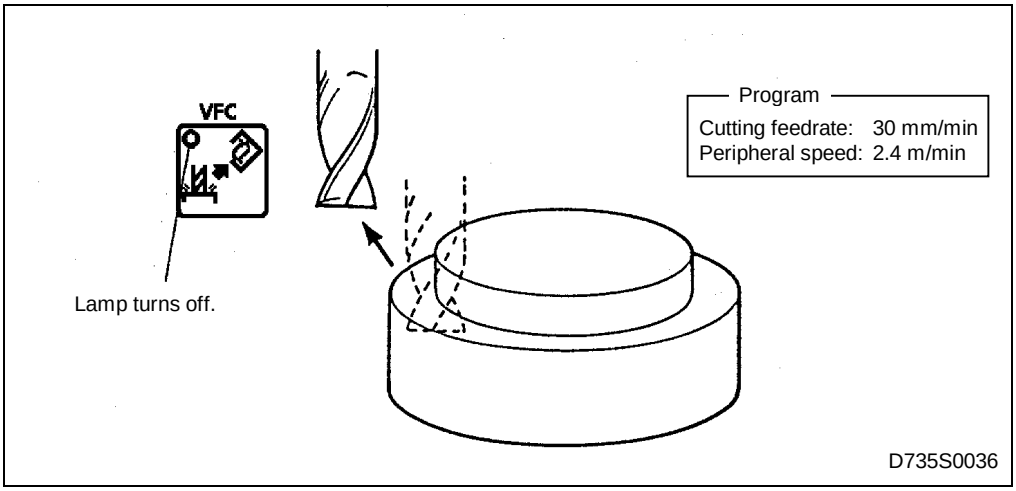


Fig. 3-2 Updating the cutting conditions data in the program — VFC

Upon completion of machining in the tool sequence being executed when the VFC key was pressed, the corresponding values in the program will be automatically overridden.

2. The VFC function is invalid for EIA/ISO programs.
3. The VFC function becomes invalid if any of the cutting feed override keys or spindle speed override keys is pressed while the VFC key lamp stays lit. That is, overriding of the values in the program does not occur.
4. When the corresponding values in the program are overridden with the new values and the VFC key lamp goes out, the cutting feedrate display and the spindle speed display will automatically return to 100 (100%).
5. Even if during automatic operation feed-hold stop is executed, the VFC function will remain active.
6. The VFC function is inactive if the Dry Run function is active.

3-5 AFC Function (Option)

The AFC (Adaptive Feedrate Control) is a function that provides automatic control of the feedrate to prevent overloading of the spindle motor and the Z-axis servo motor. After detecting the cutting current load on the spindle motor or Z-axis servo motor, this function compares the current load value with the corresponding presetting (relative value in percentage terms with respect to the rated load) on the **TOOL DATA** display and then automatically changes the feedrate if any deviation from the preset value is found.

The actual current load varies according to the loaded states of the motors and the state of the cutting edge; however, in order not to allow the equipment to operate because of these variations alone, the current load is continually supervised with a certain tolerance of the current load presetting. The feedrate is changed if this tolerance is overstepped. If five seconds have elapsed since the feedrate was reduced to 50% in order to maintain the current load presetting, this condition is judged as an overloaded one and operation comes to a temporary stop (feed-hold stop).

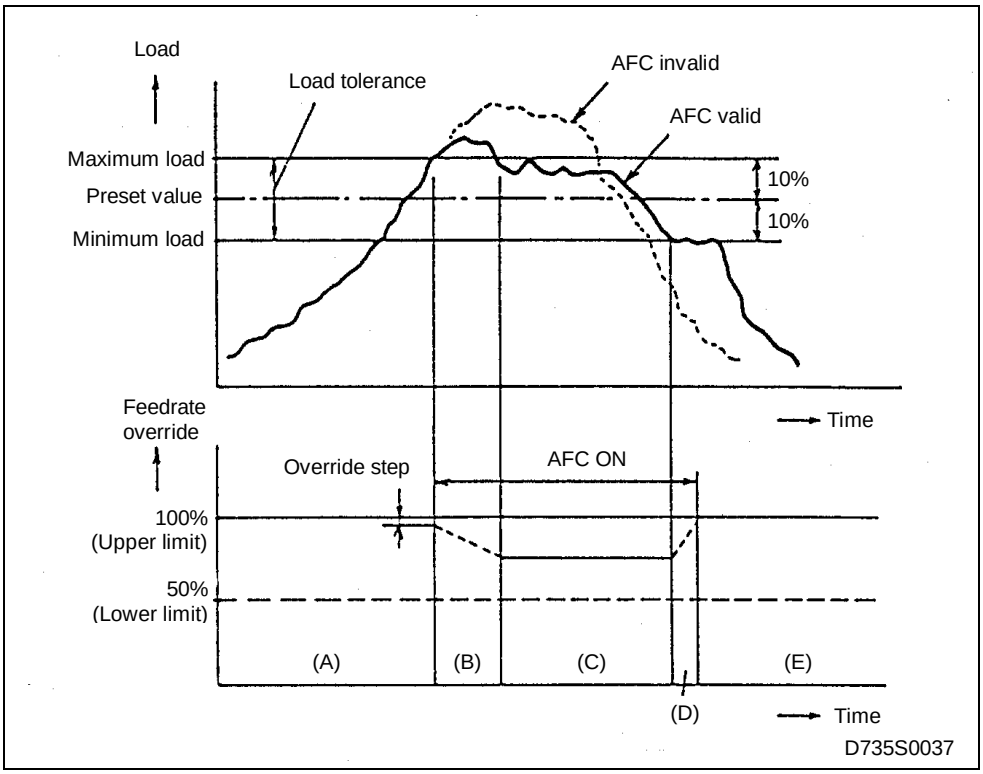


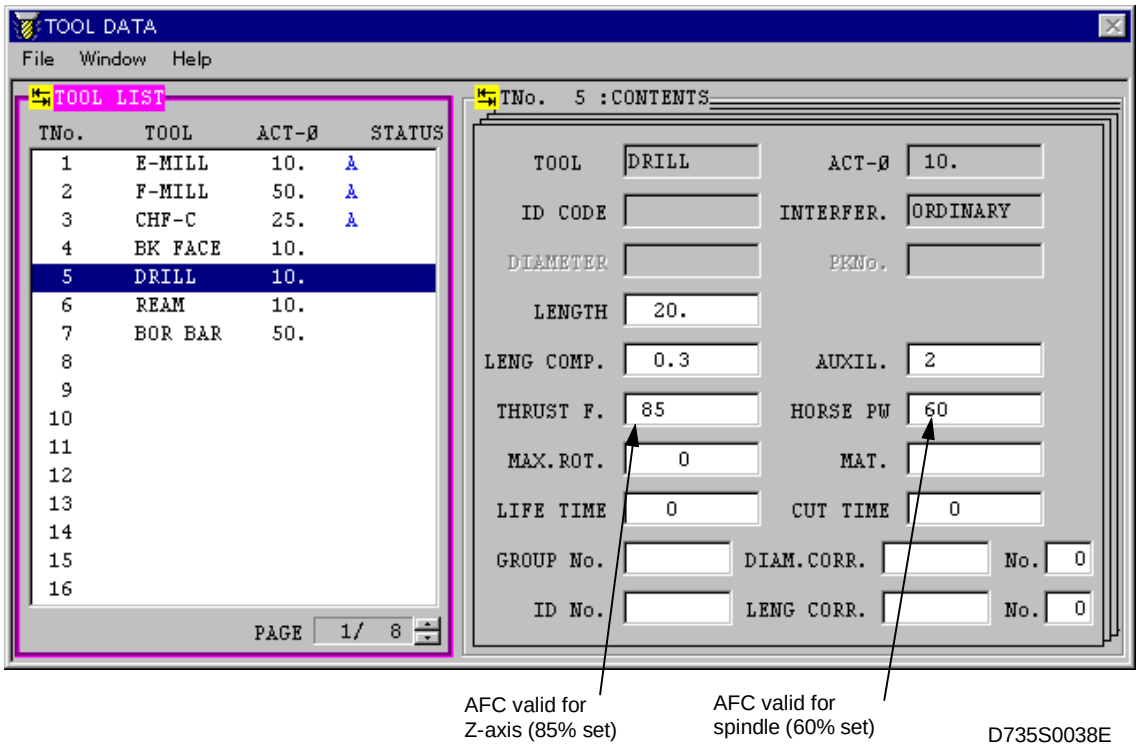
Fig. 3-3 Load tolerance and feedrate

Area	Loaded state	Feedrate override
A	Load < Maximum load	The feedrate override value of 100% is maintained.
B	Load ≥ Maximum load	The feedrate override value is stepped down depending on the load (clamped at a minimum of 50%) so that “load < maximum load”.
C	Load < Maximum load	The feedrate override value appropriate for the actual load is set automatically. Feed-hold stop occurs if the feedrate override value of 50% is forced for 5 seconds in succession.
D	Load ≤ Minimum load	The feedrate override value is stepped up (clamped at a maximum of 100%).
E	Load < Minimum load	The feedrate override value of 100% is maintained.

3-5-1 Setting the desired current load value

Set the desired current load value in percentage terms on the **TOOL DATA** display.
If the current load on the spindle motor is to be supervised, set the desired horsepower value (HP) (this renders the AFC function valid for the spindle).

If the current load on the Z-axis motor is to be supervised, set the desired thrust force value (THR) (this renders the AFC function valid for the Z-axis).



Remark: The AFC function is invalid if the horsepower or thrust force is set equal to zero.

1. Setting of the desired value through trial cutting

Carry out a trial cutting operation using the machining program for which the AFC function is to be executed, and then read the load values displayed on the **POSITION** display and take the displayed value (or a value close to it) as the load value to be set. Use of this calculation method allows highly accurate setting of the current load value.

2. Setting of the desired value through auto-setting

Depression of the **AUTO SET** menu key on the **TOOL DATA** display with the cursor set on the items “THR.” and “HP” allows automatic setting of the horsepower and thrust force values. This auto-setting, however, is just a simplified one that is performed according to the diameter of the tool used. Therefore, the auto-set values will usually be larger than the values set through trial cutting.

Note: The auto-setting is not available for tools of types B-E-MILL (ball end-mill) and OTHER (special cutter).

3-5-2 Tools available with AFC and AFC supervisory modes

The tools for which AFC can be executed and the modes of AFC supervision are listed below.

Tool name	AFC supervisory mode
DRILL (Drill)	Spindle and Z-axis AFC (Simultaneous)
E-MILL (End-mill)	
OTHER (Special cutter)	
F-MILL (Face-mill)	
CHF-C (Chamfering cutter)	
BOR BAR (Boring bar)	Spindle AFC only
BK FACE (Back-spot facer)	
B-B BAR (Back-boring bar)	

The following tools are not available with AFC and AFC supervisory modes.

1. CTR-DR (Center drill)
2. REAM (Reamer)
3. TAP (Tap)
4. T. SENS (Touch sensor)
5. CHP VAC (Chip vacuum)
6. B-E-MIL (Ball end-mill)

3-5-3 Functional operation

1. Spindle AFC function

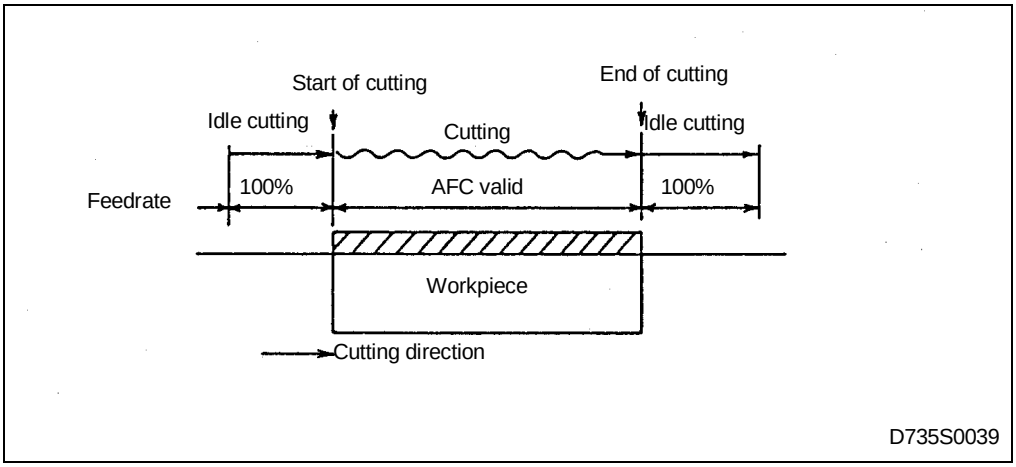


Fig. 3-4 Spindle AFC function

Note: When the spindle AFC function starts operating, the corresponding lamp in the machine status indicator window lights up.

2. Z-axis AFC function

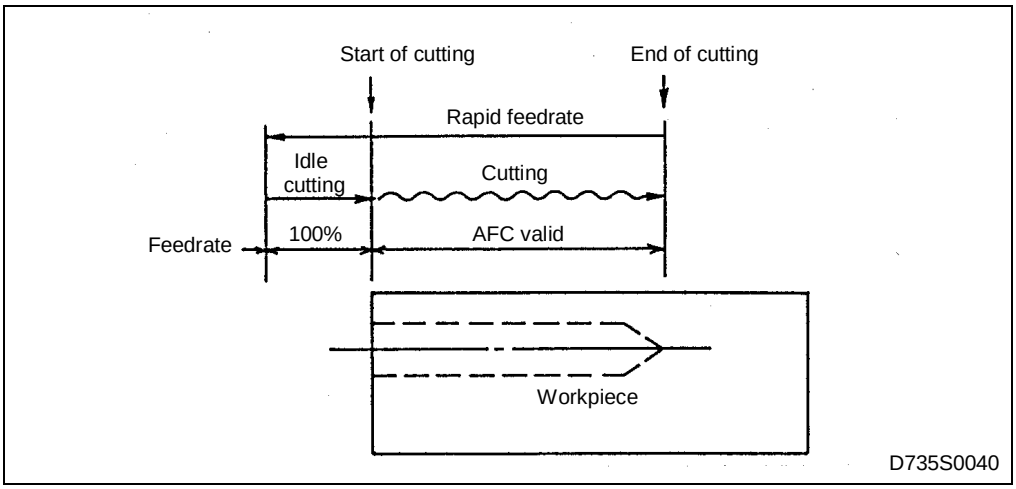
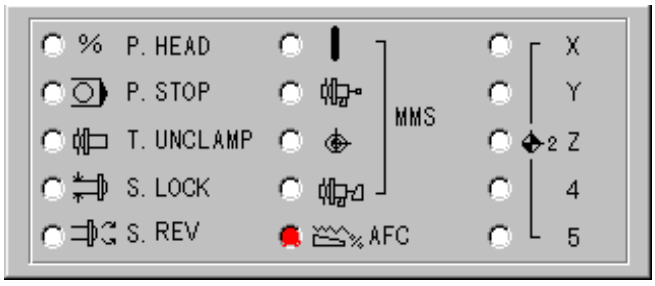


Fig. 3-5 Z-axis AFC function

Note: When the Z-axis AFC function starts operating, the corresponding lamp in the machine status indicator window lights up.

3. Indicator lamps on the machine status indicator window

Note: During AFC operation, the feedrate is automatically overridden, but the override display on the screenl does not change.



D735S0041

- NOTE -

4 STOPPING AUTOMATIC OPERATION

All machine actions must be emergently stopped if any abnormalities occur during automatic operation, such as in the case where the tool is likely to contact the workpiece, the jig, etc.

When machining has been executed to a certain extent, it may be also necessary to bring automatic operation to a halt so that you can check the finished status of the workpieces.

This section describes several types of methods of stopping automatic operation.

Select the best-suited stop method according to the particular machine or machining status and your requirements.

Of the several methods, those indicated as 4-1, 4-2 and 4-3 below, in particular, are most important methods for safety purposes.

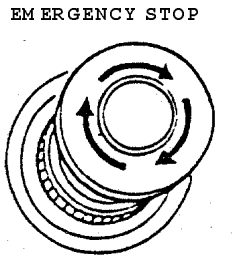
In emergencies, immediately carry out one of these methods.

4-1 Emergency Stop

4-1-1 Emergency Stop

(1) Press the EMERGENCY STOP switch.

➔ All machine actions will then stop immediately and an alarm message **EMERGENCY STOP** will be displayed.



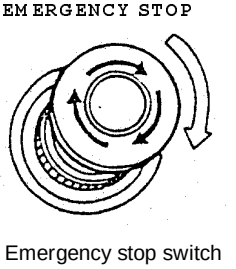
Emergency stop switch
D735S0042

4-1-2 Releasing the emergency stop status

If automatic operation has been brought to an emergency stop, it is possible to carry out manual operations or restart automatic operation just by releasing the emergency stop status using the following procedure:

(1) Turn the EMERGENCY STOP switch clockwise (direction of the arrow).

- Turning the EMERGENCY STOP switch clockwise and then releasing the switch will return it to the original position from the depressed state.



Emergency stop switch
D735S0043

(2) Press the RESET key 

.

➔ The alarm message **EMERGENCY STOP** will disappear.

- The READY (ready for operation) indicator lamp will turn on to indicate that the emergency stop status has been released.

Note: This procedure can be different according to the machine model.

4-2 Reset

(1) Press the RESET key 

.

➔ The machine actions (such as the actions of the spindle, axes, etc.) will stop immediately.

Note: After a reset operation has been carried out, the program will restart from the beginning, not from the stop position if automatic operation is restarted by pressing the CYCLE START button.
Select the Restart mode if automatic operation is to be restarted from the aborted portion of the program.

Remark: The system of coordinates as the initial state after resetting (using the RESET key) is selected as follows according to the related parameter:

F95 bit 7 = 0: G54 system after resetting

= 1: The same system after as before resetting

4-3 Feed Hold

(1) Press the FEED HOLD button.

➔ The actions of the axes will stop immediately.

- The action of the spindle will not stop even when this feed hold operation is carried out.

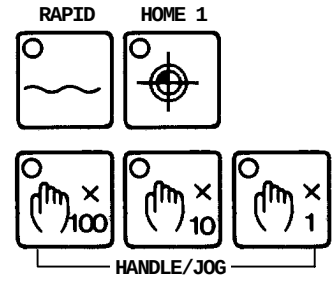
Remark: When the CYCLE START button is pressed following the feed hold operation, axis feeding will restart and the succeeding portion of the program will be executed through to completion.

4-4 Stopping by Changeover to a Manual Operation Mode

(1) Press one of the manual operation mode selector keys.

➔ The machine actions (movements of the spindle, axes, etc.) will stop immediately.

Remark: Changing back to the original automatic operation mode and then pressing the CYCLE START button will let automatic operation restart.



Manual operation mode selector keys

D735S0044

4-5 Single-block Stop

Carrying out one of the two methods described below will stop all machine actions when the block being executed terminates.

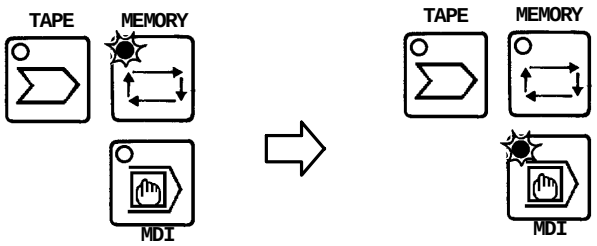
Such a stop is referred to as “single-block stop”.

4-5-1 Single-block stop by changing within automatic operation modes

(1) Change the existing automatic operation mode over to another automatic operation mode.

➔ Machine actions will stop at the end of the block being executed.

Example: If the program is being executed in the MEMORY mode:
Press the MDI key.



D735S0045/46

Note: Single-block stop does not occur if the TAPE key is pressed.

Remark: Changing back to the original automatic operation mode and then pressing the CYCLE START button will let automatic operation restart from the next block.

4-5-2 Single-block stop using the SINGLE BLOCK key

(1) Press the 

key.

➔ The key lamp turns on to indicate that the Single Block function has become active and the machine actions will stop at the end of the block being executed.

4-6 Program Stop

(1) Set previously M00 (M-code for program stop) in the program to be executed.

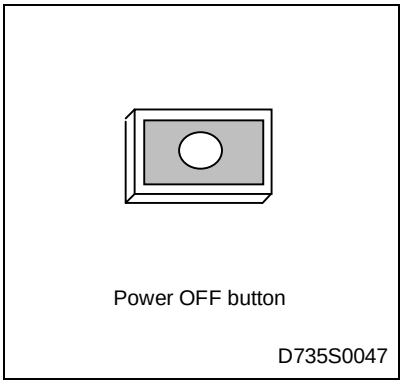
- Automatic operation will always stop at the position of M00 in the flow of program execution.

Remark: Automatic operation can be restarted by pressing the CYCLE START button.

4-7 Stopping by Turning the Power Off

(1) Press the power OFF button.

➔ This will stop all machine actions and turn off the power to the NC unit.
However, if the spindle is rotating it will take some time for the machine actions to stop, since the spindle will continue rotating due to its inertia.



Note: Using this method causes shocks to the NC unit and the machine, and trouble can result from repeated use.
Therefore, use this method as the last available means only.

Remark: When the power of the NC unit is turned off, the power of the machine will be turned off automatically.

5 MANUAL INTERRUPTION

Interrupting automatic operation for you to carry out manual operations is referred to as manual interruption.
Carry out manual interruption in cases that the tool path specified in the programs is to be shifted, that a damaged tool-tip is to be replaced, etc.
To carry out manual interruption, first interrupt automatic operation using one of the following three methods:

- Feed hold
- Stopping by changeover to a manual operation mode
- Single-block stop

If you do not wish to use these three methods, carry out manual interruption after automatic operation has been stopped by Program Stop or Optional Stop.

5-1 Handle Interruption (Option)

Handle interruption refers to the operation by which, during automatic operation, axis movement is carried out using the manual pulse handle until the tool path specified in the program is shifted to the desired pattern.

5-1-1 Handle interruption

(1) Interrupt automatic operation.
- See the section of “Feed Hold” or “Single-block Stop”.

(2) Carry out axis movement(s) through the necessary distance using the manual pulse handle.
- For details of the axis-shifting procedures, see the description of “Manual pulse feed”.

(3) Press the CYCLE START button.
➔ The automatic operation will restart in the coordinates system that is shifted through the distance of the axis movement carried out in procedural step (2) above.

5-1-2 Clearing the manual handle interruption

(1) Interrupt the automatic operation.

(2) Carry out either one of the following two operations.
- Press the RESET key.
  In this case, the program will return to its starting part.
- Carry out an axis movement by the same distance in the opposite direction to that of the manual interruption.

Remark 1: For the zero-point return, however, on the axis on which shifting by the handle interruption was carried out, the shift distance data will temporarily be cancelled. And it will become effective anew for the next movement on the same axis away from the zero-point.

Remark 2: The shift distance data will be also cleared, when the power OFF button is pressed.

5-2 TPS Function

TPS refers to the function that lets the NC unit store the tool-tip position during manual interruption into the memory of the unit.

With this function, those positions of the tool-tip at up to four different points during manual-interruption axis movement can be stored into the memory of the NC unit.

Therefore, even after axis movement during manual interruption, automatic operation can be restarted without disturbing the tool path specified in the program.

This function comes in handy during automatic operation if a tool is to be replaced because of damage to the tool-tip or the presence of swarf entangling the workpiece.

The following gives three examples of application of the TPS function and related machine actions.

5-2-1 Storing the data of only the stop point of automatic operation

(1) Abort automatic operation.

(2) Change the automatic-operation mode over to any manual mode.

- This can be accomplished simply by pressing one of the manual-mode selector keys (HANDLE ×100, HANDLE ×10, HANDLE ×1, RAPID, HOME 1).
- This changeover is not required if automatic operation has been aborted with a manual-mode selector key in procedural step (1) above.

(3) Press the **TPS** menu key.

➔ This reverses the display status of TPS, the data of TPS 0 in the right bottom on the display becomes 1 and the tool-tip position will be stored into the memory of the NC unit.

(4) Manually move the tool.

➔ The reverse display status of **TPS** will be released when tool movement is started.

(5) Change the operation mode back to the original one.

(6) Press the CYCLE START button.

➔ This will cause the start button lamp to light up and the tool tip to return to the position existing when the **TPS** menu key was pressed and then, automatic operation will restart from that position.

Example: If, as shown below, the tool has been moved in order of **a**•**b**•**c**•N in procedural step (4):

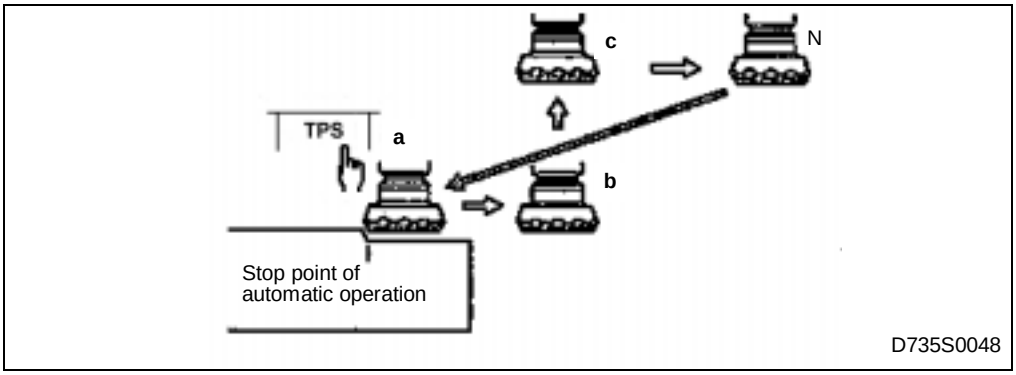


Fig. 5-1 Tool path

When the start button is pressed, the tool will linearly move from N to **a** at the feedrate existing when automatic operation was stopped.

5-2-2 Storing the data of both the stop point of automatic operation and the passage point(s) existing during manual interruption

(1) Abort automatic operation.

(2) Change the automatic-operation mode over to any manual mode.

- See the description of procedural step (2) in Section 5-2-1 above.

(3) Press the **TPS** menu key.

- See the description of procedural step (3) in Section 5-2-1 above.

(4) Manually move the tool, and press the **TPS** menu key during tool movement.

➔ The number on TPS 0 in the right bottom on the display increases by 1, and the tool-tip position at that time will be stored into the memory of the NC equipment.

- The reverse display status of **TPS** will be released next time the tool is moved.

- For the detailed tool-movement procedure, see 5-1, “Handle Interruption”.

(5) Change the operation mode back to the original one.

(6) Press the CYCLE START button.

➔ The start button lamp will then light up, and the tool tip will move through the position existing when the **TPS** menu key was pressed in step (4) and will finally return to the point at which the automatic operation was stopped and then, automatic operation will restart from the position in which it was stopped.

Example: If, as shown below, the tool has been moved in order of **a**•**b**•**c**•**d**•**e**•N in procedural step (4) and the **TPS** menu key has been pressed at the points **a**, **b**, **c** and **d**:

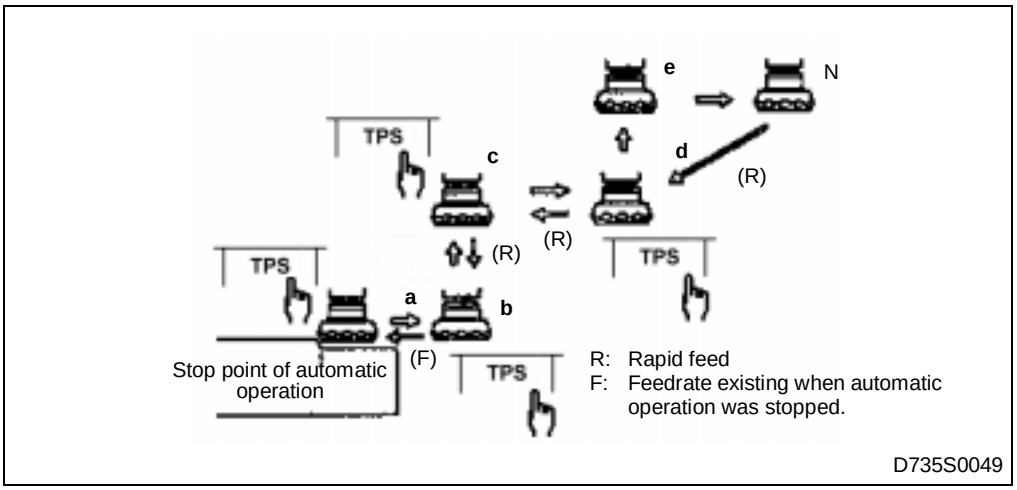


Fig. 5-2 When pressing the TPS menu key at stop point and passage points

When the start button is pressed, the tool will move at the current rapid feedrate from point N to **d** directly, then from **d** through **c** to **b**. Then, the tool will move linearly from **b** to **a** at the feedrate existing when automatic operation was stopped.

5-2-3 Storing the data of only the passage point(s) existing during manual interruption

Carry out the same operation except the procedural step (3), as described in Section 5-2-2 above.

Example: If, as shown below, the tool has been moved in order of **a**•**b**•**c**•**d**•**e**•N and the TPS menu key has been pressed at the points **c** and **d**:

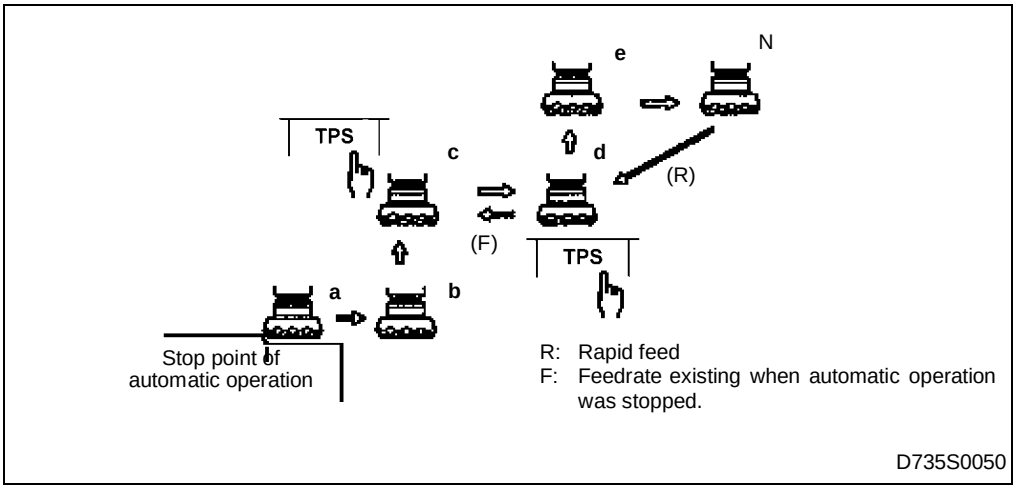


Fig. 5-3 When pressing the TPS menu key only at the passage points during manual interruption

When the start button is pressed, the tool will move at the rapid feedrate from point N directly to **d**. Then, the tool tip will move linearly from **d** to **c** at the feedrate existing when automatic operation was stopped.

Note: Automatic operation will restart from the position where the TPS key was pressed for the first time in the manual interruption (**c** in above example). That is, the restarted automatic operation will be performed on the basis of that workpiece coordinate system to which the programmed system is shifted in accordance with the distance from the abort position of automatic operation to the position of the first depression of the TPS menu key.

5-2-4 General remarks on TPS function

1. Operating procedure of Subsection 5-2-3 above must be carried out only when automatic operation is to be restarted on the basis of a shifted system of coordinates.
   Operating procedure Subections 5-2-1 or 5-2-2 must be carried out when automatic operation is to be restarted in such a manner that the tool tip follows the path specified in the program.
2. Do not apply the 1. nor the 2. zero-point return mode to interrupt the automatic operation under control of tool-length or -diameter offset.
   Otherwise, the tool offsetting may not correctly be performed any more in the restarted automatic operation.

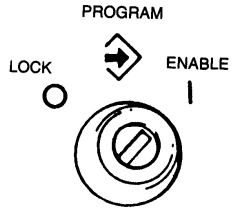
6 OPERATIONS POSSIBLE DURING AUTOMATIC OPERATION

6-1 Data Rewriting

6-1-1 Data writing conditions

Data setting and other operations on each display are restricted by the operational status of the machine at a particular time. Also, while program override lock/enable switch remains set to LOCK (writing disabled), writing or updating the current program or several other types of data cannot be performed.

This section gives a listing as to what types of data can be set, and what types of display operations can be carried out, according to the particular operational status (during execution of automatic operation, during automatic operation mode, or during manual operation mode) of the machine as well as according to the status of the program override lock/enable switch.



Program override switch
D735S0051

•: Possible
×: Impossible
•: Impossible only for the program being executed.
•: When parameter **L57** is 0: Impossible 1: Possible

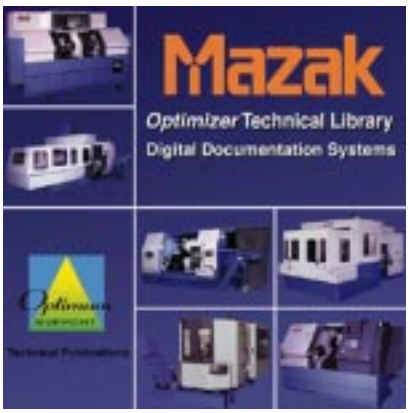
Display	Data item	Operation	PROGRAM LOCK/ENABLE switch status						Remarks
			ENABLE			LOCK			
			Autom. mode		Manu. mode	Autom. mode		Manu. mode	
			Machn. running	Machn. halted		Machn. running	Machn. halted		
POSITION	Workpc. qty		•	•	•	•	•	•	
	Desired qty		•	•	•	•	•	•	
		Counter set	×	×	•	×	×	•	
		Work No. search	×	•	•	×	•	•	
TRACE		Work No. search	×	•	•	×	•	•	
		Shape display	•	•	•	•	•	•	
EIA MONITOR		Program editing	• (*)	•	•	×	×	×	(*) Possible only during single-block or feed-hold stop
PROGRAM		Display of program	•	•	•	•	•	•	
		Creation/edition of program	•	•	•	×	×	×	

•: Possible
×: Impossible
•: Impossible only for the program being executed.
•: When parameter **L57** is 0: Impossible 1: Possible

Display	Data item	Operation	PROGRAM LOCK/ENABLE switch status						Remarks
			ENABLE			LOCK			
			Autom. mode		Manu. mode	Autom. mode		Manu. mode	
			Machn. running	Machn. halted		Machn. running	Machn. halted		
PROGRAM FILE	Program name		•	•	•	×	×	×	
		Work No. change	•	•	•	×	×	×	
		Prg. deletion	•	•	•	×	×	×	
		All prgs. deletion	×	•	•	×	×	×	
PROCESS		Process No. search	•	•	•	•	•	•	
PRG. LAYOUT		Lay-out	•	•	•	×	×	×	
PATH CHECK		Display of shape/path	•	•	•	•	•	•	
TOOL LAYOUT	Entire data		×	•	•	×	×	×	
TOOL DATA	TOOL NOM-ϕ ACT-ϕ LENGTH THR. HP		• (*)	•	•	•	•	•	(*) Possible only for tools not in use and during execution of an EIA/ISO program.
TOOL FILE	Entire data		×	•	•	×	×	×	
TOOL OFFSET	Entire data		•	•	•	•	•	•	
PARAMETER	Entire data		×	•	•	×	×	×	
WORK OFFSET	Entire data		•	•	•	•	•	•	
MACRO VARIABLE	Common var.		•	•	•	×	×	×	
	Common var. (Check)		×	•	•	×	×	×	
	Local var.		×	•	•	×	×	×	
	Local var. (Check)		×	•	•	×	×	×	
DATA I/O		LOAD (prg.)	•	•	•	•	•	•	CMT I/O Memory card Floppy
		LOAD (other)	×	•	•	×	•	•	
		SAVE, COMPARE, CONTENTS	•	•	•	•	•	•	
		LOAD, ALL LOAD	•	•	•	•	•	•	TAPE
		PUNCH, ALL PUNCH, COMPARE, ALL COMP, TAPE SEARCH, REWIND	•	•	•	•	•	•	
		LOAD (prg.)	•	•	•	•	•	•	DNC
		LOAD (other)	×	•	•	×	•	•	
		SAVE	•	•	•	•	•	•	

Section 5:

Appendix



Notes:

TABLE OF CONTENTS
(APPENDIX)

Page

- MEMO -

1 STANDARD MACHINE SPECIFICATIONS

1-1 Major Machine Specifications - VTC-200B

Item				Unit	B-type
Capacity	Table working area		Right and left	mm (in)	1460 (57.48)
			Back and forth	mm (in)	510 (20.08)
	Saddle right-and-left stroke		X-axis	mm (in)	1120 (44.09)
	Column back-and-forth stroke		Y-axis	mm (in)	510 (+250 ATC stroke) (20.08(+9.84ATC))
	Spindle head up-and-down stroke		Z-axis	mm (in)	510 (20.08)
	Distance from the spindle nose to the top of table		Min.	mm (in)	180 (7.09)
			Max.	mm (in)	690 (27.17)
	Distance from the center of spindle to the front of column			mm (in)	550 (21.65)
	Max. allowable table load mass			kgf (lbs)	800 (1800)
Spindle	Spindle nose type			-	7/24 taper,No,40
	Standard specifications	Spindle speed		min^{-1} (rpm)	40～10000
		Main motor (AC inverter motor)	10 min. rating	kW (HP)	15 (20)
			30 min. rating	kW (HP)	11 (15)
			Continuous rating	kW (HP)	7.5 (10.2)
	Option	Spindle speed		min^{-1}	-
		Main motor (AC inverter motor)	10 min. rating	kW (HP)	-
			30 min. rating	kW (HP)	-
			Continuous rating	kW (HP)	-
Feedrate	Rapid traverse rate			mm/min (IPM)	30000 (1181.1)
	Cutting feedrate			mm/min (IPM)	1～8000 (0.039~314.96)
Table	Table working area		Right and left	mm (in)	1460 (57.48)
			Back and forth	mm (in)	510 (20.08)
	T-slot	Nominal size (ISO R299)		mm (in)	18 (0.71)
		Number of slots		-	5
		Interval of slots		mm (in)	100 (3.94)
	Max. table load mass			kgf (lbs)	800 (1800)
	Distance from the bottom of machine to the top of table			mm (in)	750 (29.53)

Item			Unit	B-type
ATC, tool magazine	Tool storage capacity	Standard specifications	-	24
		Option	-	30
	Tool selection system		-	Random selection by magazine pocket number Automatic shorter path selection
	Tool shank type		-	MAS BT-40
	Max. tool diameter	with adjacent tools	mm (in)	ϕ 80 (3.15)
		without adjacent tools	mm (in)	ϕ 110 (4.33)
	Max. tool mass (including tool shank and pull stud bolt)		kgf (lbs)	8 (18)
	Max. tool storage capacity of magazine	Standard specifications	kgf (lbs)	120 (270)
		30-tool specifications	kgf (lbs)	150 (377.5)
	Max. tool length (from gauge line)		mm (in)	350 (13.78)
	Tool change time (chip-to-chip) (MAS)	5 kg or less	sec	4.7
		8 kg or less	sec	5.0
Tanks	Lubricating oil tank capacity		L (gal (US))	3 (0.79)
	Spindle cooling oil tank capacity		L (gal (US))	20 (5.28)
	Hydraulic unit tank capacity		L (gal (US))	10 (2.64)
	Coolant tank capacity (standard specifications)		L (gal (US))	250 (66.05)
Accuracy	Positioning accuracy (pitch error compensation function used)	X-, Y-, Z-axis	—	±0.005mm (±0.0002 in) / entire stroke
	Repeatability	X-, Y-, Z-axis	mm (in)	±0.0015 (±0.00006)
Machine Dimen-sions	Machine dimensions (standard specifications)	Height (from floor level)	mm (in)	2664 (104.88)
		Width	mm (in)	2972 (117.01)
		Depth	mm (in)	2820 (111.02)
	Weight (standard specifications)		kgf (lbs)	6100 (13.725)
Power Source	Voltage (3 phases)		—	AC200/220V±10%
	Frequency		—	50/60Hz±1Hz
	Capacity (standard specifications)	10 min. rating	kVA	32.7
		Continuous rating	kVA	21.4
Air Source	Pressure		—	0.5Mpa (5kgf/cm^2 (71.15PSI) or more 0.9Mpa (9kgf/cm^2 (128.07PSI) or less
	Capacity (standard specifications)		—	140L/min (4.9ft^3/min) or more

Note: The figures indicated in the machine plates shall be applied if ones in the manual may differ.

1-2 Stroke Diagram and Spindle Interference Diagrams - VTC-200B

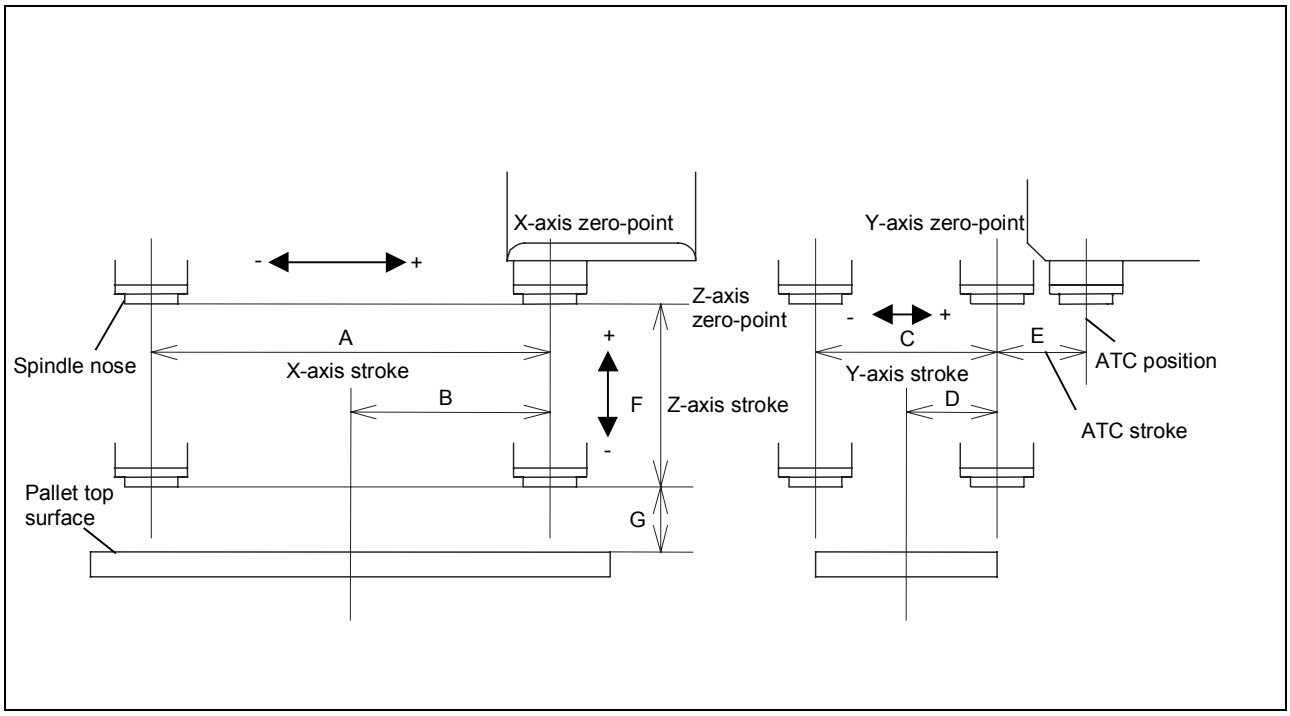


Fig. 1-1 Stroke Diagram

Unit: mm (in.)

Stroke	VTC-200B
A	1120 (44.09)
B	560 (22.05)
C	510 (20.08)
D	255 (10.04)
E	250 (9.84)
F	510 (20.08)
G	180 (7.09)

1-3 Spindle Speed-Power-Torque Diagrams

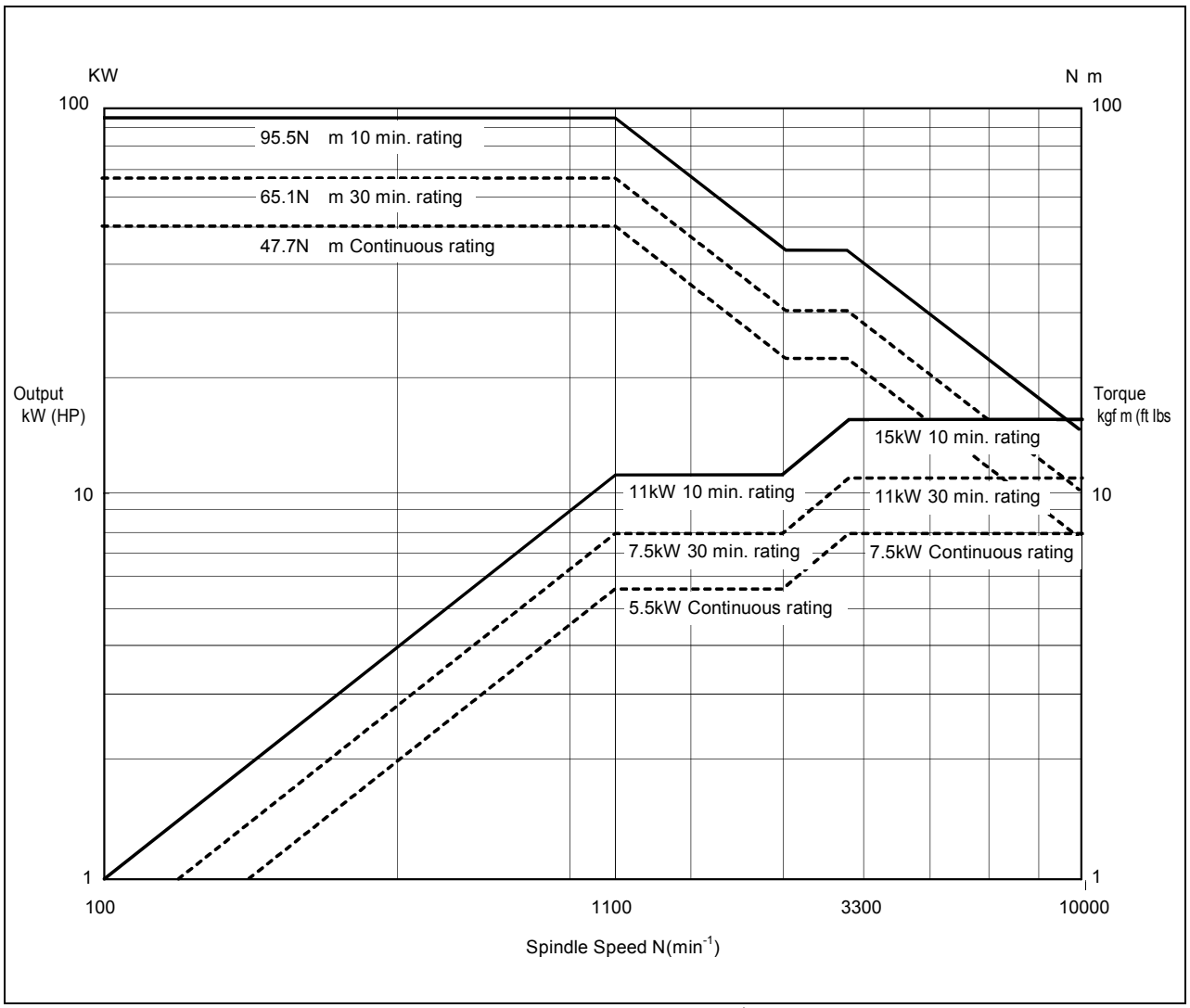


Fig. 1-2 Spindle Speed-Power-Torque Diagrams (10000 min^{-1} (rpm) specification)

1-4 Table Dimensions

VTC-200B

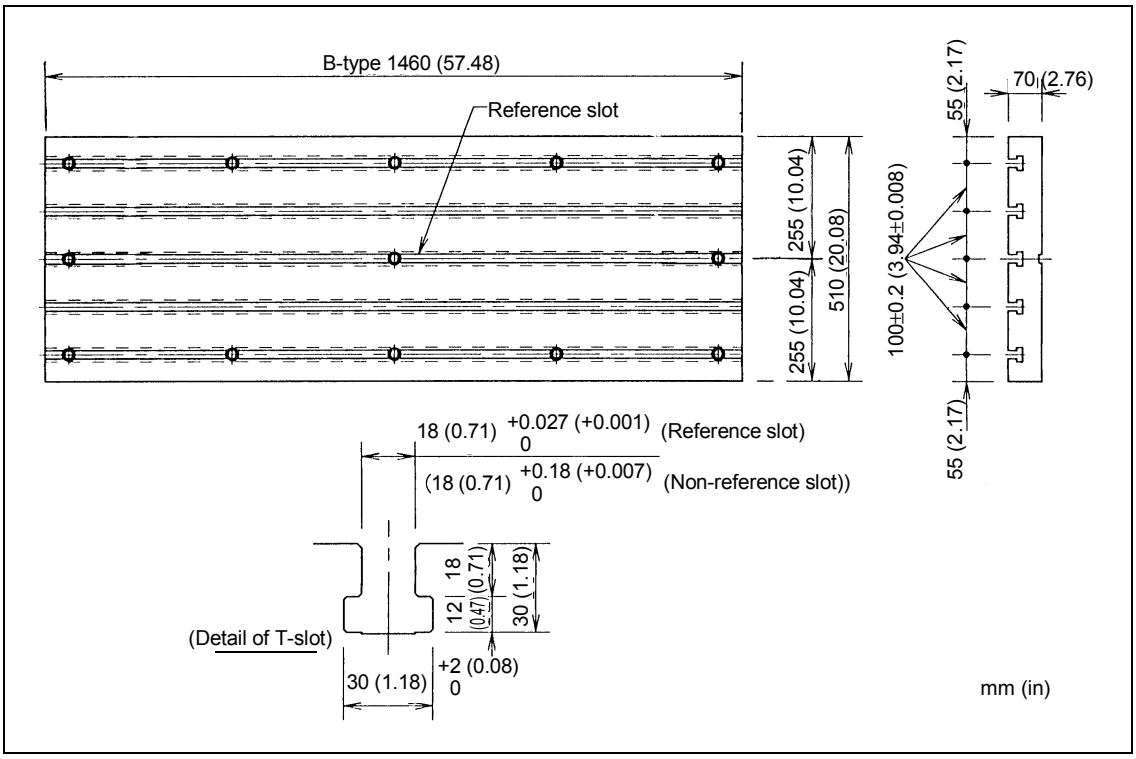


Fig. 1-3 Table Dimensions

1-5 Sub table Dimensions

VTC-200B

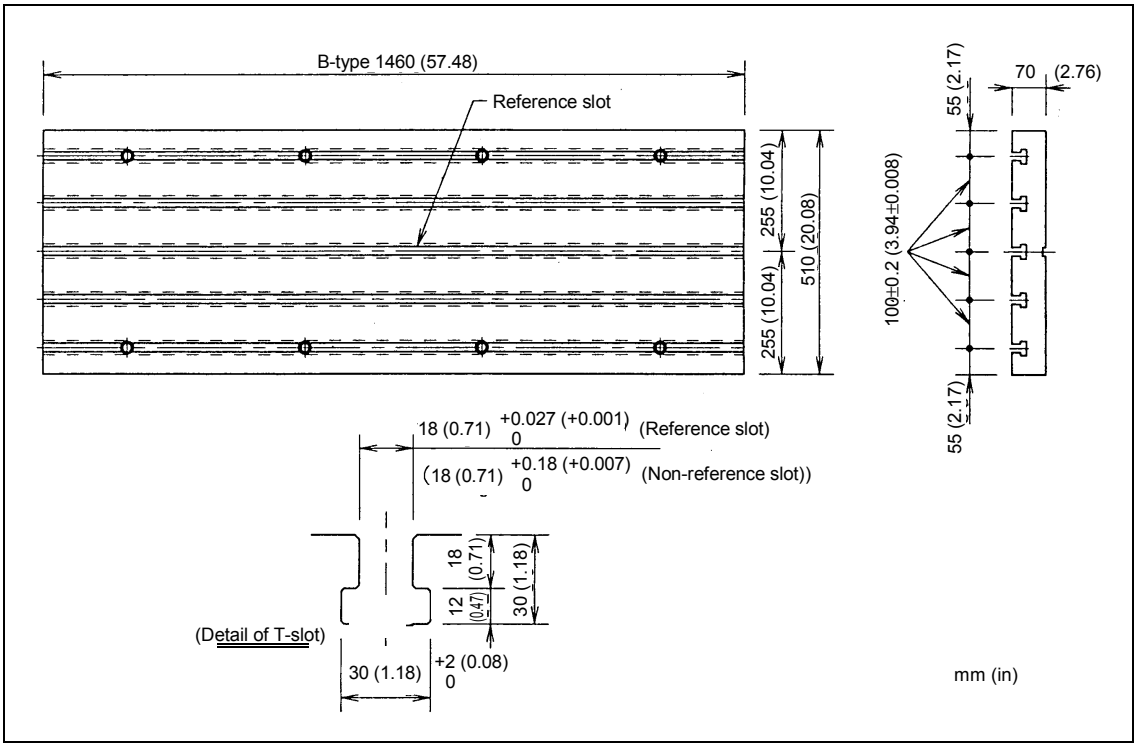


Fig. 1-4 Sub table Dimensions

1-6 Workpiece Interference Diagram

VTC-200B

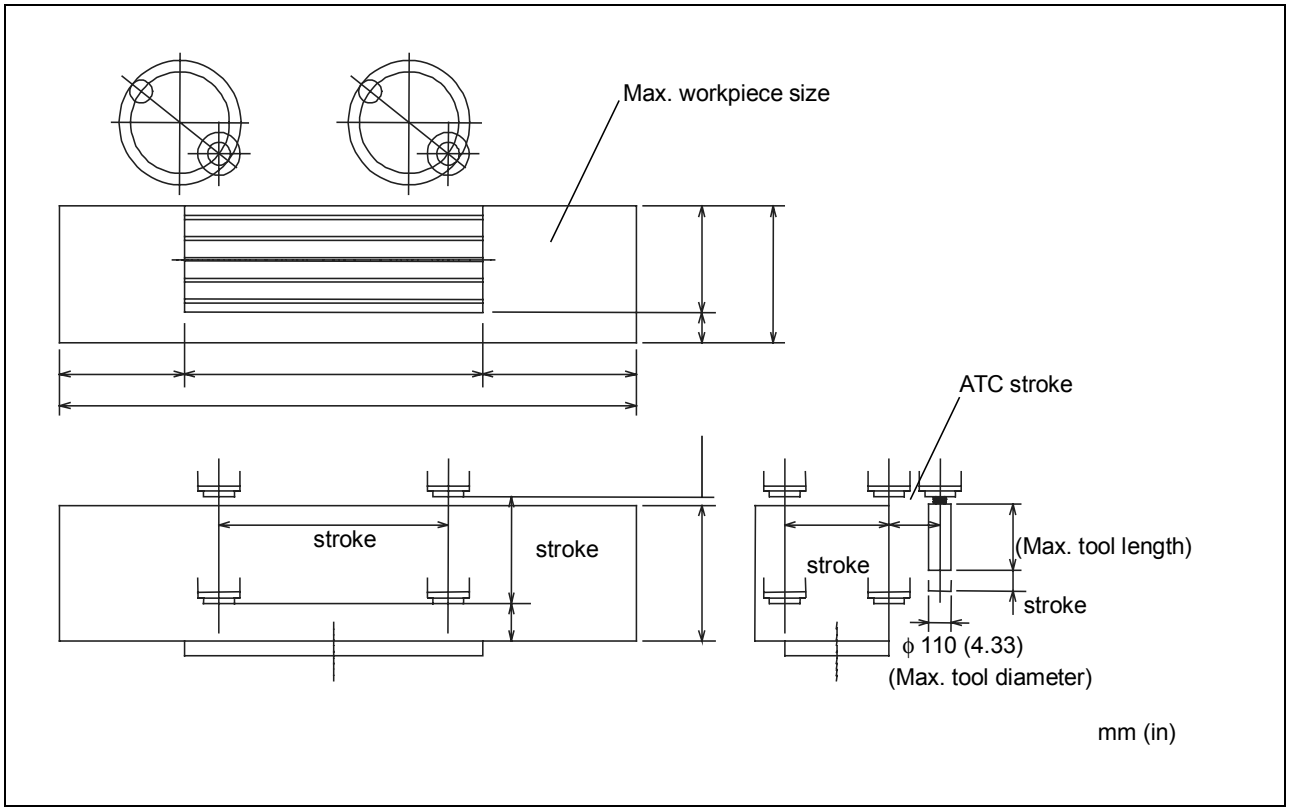


Fig. 1-5 Workpiece Interference Diagram

2 ALARM LIST - VTC-200/300 (MAZATROL FUSION 640M)

If an alarm listed below occurs, it can be cleared by pressing 

(clear) or 

(reset) key after eliminating the cause of alarm.

Alarm No. (SQ Coil No.)	Alarm Name	Description	Machine Status	Sensor Detecting the Alarm
200 (F0)	HYDRAULIC UNIT PRESSURE DOWN	Low hydraulic pressure state has continued for more than 2 seconds.		
202 (F2)	AIR PRESSURE DOWN	Low pressure for spindle lubrication	Feed hold Spindle stop	SP10
203 (F3)	SPINDLE OIL PRESSURE DOWN	Low pressure of the spindle lubrication oil (pressure switch)		
204 (F4)	SPINDLE OIL LOW	Low spindle lubricating oil level		
206 (F6)	SPINDLE LUB. CHILLER MALF.	Alarm with the spindle lubricating oil chiller unit.		
207 (F7)	IRREGULAR SLIDEWAY OIL PRESSURE	Low slideway lubricating oil pressure		
208 (F8)	SLIDEWAY OIL LOW	Low level of lubrication oil in the centralized lubrication unit tank	Only display of message	Centralized lubrication unit SL4 X03
209 (F9)	SLIDEWAY OIL MALFUNCTION	Alarm with the centralized lubrication unit (for the pressure error of the centralized lubrication unit)	Feed hold Spindle stop	Centralized lubrication unit SL4 X9A
211 (F11)	SPINDLE CONTROLLER MALFUNCTION	Alarm with the spindle controller	Feed hold Spindle stop	Spindle controller X243
212 (F12)	MAGAZINE DRUM MALFUNCTION	Alarm with the magazine controller	Magazine rotation disabled	
214 (F14)	ILLEGAL TOOL DESIGNATED	Tool with tool data specified as invalid is commanded.	Magazine rotation disabled	
217 (F17)	THERMAL TRIP	A thermal relay of auxiliary equipment motor is tripped. The circuit protector in the 100 V line is tripped.	Feed hold Spindle stop	KM10, KM11, FR4, FR76, QF7, QF8, QF13 X1A
218 (F18)	CONVEYOR THERMAL TRIP	The thermal relay of chip conveyor or conveyor drive motor is tripped.	Only display of alarm	FR13, FR113, X1B
219 (F19)	MAIN TRANSFORMER OVERHEAT	The main transformer is exceeding 120°C (248°F).	Single block Spindle stop	Main transformer X10
222 (F22)	AFC OVERLOAD	Spindle load cannot be lowered even when an override value is lowered using the AFC function.	Feed hold Spindle stop	
223 (F23)	TOOL MEAS. UNIT NOT IN POS	An attempt is made to execute semi-automatic tool length measurement while the tool length measurement base is not in place.		

Alarm No. (SQ Coil No.)	Alarm Name	Description	Machine Status	Sensor Detecting the Alarm
224 (F24)	ILLEGAL TOOL FOR TOOL LGTH MEAS	1. Tool measuring cycle is executed for a touch sensor or chip removing tool. 2. An attempt is made to execute automatic tool length measurement cycle for the following types of tools: Back boring tool, boring tool, back spot facing tool, chamfering tool, face milling cutter and special tool. Tool length measurement cycle does not start.	Tool length measurement cycle does not start.	
225 (F25)	TOOL LIFE OVER	Accumulated tool used time executed the set tool life time.	Stop in the single block mode if M58 is executed while this alarm message is displayed. To continue operation, press the CLEAR key and then press the CYCLE START button.	
226 (F26)	TOOL BREAKAGE	Tool breakage is detected in the tool breakage detection cycle called by M35. Broken tool is loaded to the spindle.	Spindle block	
227 (F27)	SIMULTANEOUS M CODE OPERATION	An M code which cannot be processed simultaneously with the command given in the same block is specified.	Operation does not start and the machine stops.	
228 (F28)	ILLEGAL M CODE	An M code which is not supported by the selected specifications is specified.	No answer signal is received and the machine stops.	
229 (F29)	INVALID SCREEN FOR CYCLE START	An attempt is made to start the memory or tape mode operation (cycle start) from any screen other than the following screens: POSITION, TRACE and EIA MONITOR	The machine does not start, resulting in feed hold.	
230 (F30)	ILLEGAL MMS UNIT	A necessary signal is not output due to a trouble with the MMS unit.	Feed hold	MMS ERROR X17 MMS LOW BATT X18
231 (F31)	EXTERNAL CONTROLLER ALARM	Alarm of an external controller (programmable controller, etc.) (optional)		
232 (F32)	MACHINE DOOR INTERLOCK	The machine door has been opened.	Feed hold Spindle stop Stop in the single block mode	SQ9 X0B
233 (F33)	MAGAZINE DOOR INTERLOCK	While the machine is operating in the automatic mode, the magazine door opens, or while the door is open, the manual tool select is commanded.	Feed hold Magazine rotation disabled	SQ31 X13
234 (F34)	PALLET CHANGER DOOR INTERLOCK	The pallet changer safety door has been opened, or the door lock has been released. (Option)		
235 (F35)	MAGAZINE MANUAL INTERRUPT	The MAGAZINE SELECT switch (MAGAZINE INTERRUPT switch) is enabled.	Display of message (Automatic mode is suspended.)	SA4 X16
236 (F36)	HOLD PALLET CHANGE	The PALLET CHANGE HOLD switch on the pallet changer operation panel is set to ON. (6P/C)		

Alarm No. (SQ Coil No.)	Alarm Name	Description	Machine Status	Sensor Detecting the Alarm
237 (F37)	MAGAZINE DOOR OPENED	The magazine door is open.	Only display of message	SQ31 X13
238 (F38)	AUTO TOOL CHANGE STOP	The [ATC STOP] menu key has been pressed and the ATC is stopped.	Stop just before the ATC cycle. Press the [ATC STOP] menu key to turn it off and press the CYCLE START button to continuously execute the ATC cycle.	
240 (F40)	OPERATION RESTART UNFINISHED	An attempt is made to start the operation before completing the EIA restart procedure.	Feed hold	
241 (F41)	SPINDLE ROTATION ABNORMAL	The zero speed signal remains ON although the spindle rotation command has been executed.	Feed hold Spindle stop	Y24, X244
242 (F42)	SPINDLE ORIENT TIME OVER	The spindle orientation check signal is not turned on in 10 seconds after the execution of the spindle orientation command.	Operation stops.	Y2D6, X246
243 (F43)	SPINDLE ORIENT CANCEL TIME OVER	The spindle orientation check signal remains on in 1 second after turning off of the spindle orientation command.	Operation stops.	Y2D6, X246
244 (F44)	ZERO SIGNAL OFF IN SPDL ORIENT	The spindle zero speed check signal goes off while the spindle orientation check signal is on.	Feed hold	Y2D6, X244
250 (F50)	TOOL LENGTH EXTEND SENSOR MALF.	The measuring unit base out end sensor is not turned on although the out command has been executed, or the sensor is not turned off although the in command has been executed (5 seconds).		
251 (F51)	TOOL LENGTH RETRACT SENSOR MALF.	The measuring unit base in end sensor is not turned on although the in command has been executed, or the sensor is not turned off although the out command has been executed (5 seconds).		
252 (F52)	DECELERATION SENSOR MALF.	During tool length measuring cycle, the skip signal has been output although the skip deceleration signal is not output.	Spindle stop	SQ43 X17C
253 (F53)	SKIP SENSOR MALFUNCTION	During tool length measuring cycle, the skip signal is not output although the skip deceleration signal has been output.	Spindle stop	SQ42 X17D
256 (F56)	TOOL CLAMP SENSOR MALFUNCTION	The tool clamp sensor is not turned on although the tool clamp command is executed, or the tool clamp sensor is not turned off although the tool unclamp command is executed. (5 seconds)	Spindle stop	SQ11 X91
257 (F57)	TOOL UNCLAMP SENSOR MALFUNCTION	The tool unclamp sensor is not turned on although the tool unclamp command is executed, or the tool unclamp sensor is not turned off although the tool clamp command is executed. (5 seconds)	Spindle stop	SQ10 X90
258 (F58)	ATC COVER SENSOR (OPEN) MALF.	The ATC cover open sensor is not turned on although the ATC cover open command is executed, or the ATC cover open sensor is not turned off although the ATC cover close command is executed. (10 seconds)	Spindle stop	SQ22 X82

Alarm No. (SQ Coil No.)	Alarm Name	Description	Machine Status	Sensor Detecting the Alarm
259 (F59)	ATC COVER SONSOR (CLOSE) MALF.	The ATC cover close sensor is not turned on although the ATC cover close command is executed, or the ATC cover close sensor is not turned off although the ATC cover open command is executed. (10 seconds)	Spindle stop	SQ23 X83
266 (F66)	PALLET CLAMP SENSOR MALF.	The pallet clamp sensor is not turned on although the pallet clamp command is executed, or the pallet clamp sensor is not turned off although the pallet unclamp command is executed.		
267 (F67)	PALLET UNCLAMP SENSOR MALF.	The pallet unclamp sensor is not turned on although the pallet unclamp command is executed, or the pallet unclamp sensor is not turned off although the pallet clamp command is executed.		
268 (F68)	MAGAZINE IN POS SENSOR MALF.	The magazine in-position signal is not turned on although magazine rotation has been completed. Two minutes or more have passed since the issuance of the magazine rotation command.	Spindle stop	Magazine controller
270 (F70)	X AXIS ORIGIN RETURN UNFINISH	ATC, tool length measurement cycle, pallet change cycle, etc. is executed although the X-axis has not been returned to the zero point.	Attempted operation does not start.	
271 (F71)	Y AXIS ORIGIN RETURN UNFINISH	ATC, tool length measurement cycle, pallet change cycle, etc. is executed although the Y-axis has not been returned to the zero point.	Attempted operation does not start.	
272 (F72)	Z AXIS ORIGIN RETURN UNFINISH	ATC, tool length measurement cycle, pallet change cycle, etc. is executed although the Z-axis has not been returned to the zero point.	Attempted operation does not start.	
273 (F73)	4 AXIS ORIGIN RETURN UNFINISH	ATC, tool length measurement cycle, pallet change cycle, etc. is executed although the 4th-axis has not been returned to the zero point.	Attempted operation does not start.	
274 (F74)	SPDL CANNOT ROT (UNCLAMPED TOOL)	The spindle start command is executed although the tool has not been clamped.	Spindle rotation disabled	SQ10, X90, SQ11, X91
275 (F75)	SPDL CANNOT ROT (SPECIAL TOOL)	The spindle start command is executed although a special tool (touch sensor, chip removal tool) is mounted in the spindle.	Spindle rotation disabled	
276 (F76)	SPDL CANNOT ROT (ATC ARM POS)	The spindle cannot be started since the ATC arm is not in correct position.	Spindle rotation disabled	
280 (F80)	NO ORIENT (UNCLAMPED TOOL)	The spindle orientation command is executed although the tool has not been clamped.	Spindle orientation disabled	SQ10, X90 SQ11, X91
281 (F81)	NO SPDL ORIENT (SPECIAL TOOL)	The spindle orientation command is executed although a special tool (touch sensor, chip removal tool) is mounted in the spindle.	Spindle orientation disabled	
282 (F82)	NO ORIENT (ATC ARM POS. ERROR)	The spindle orientation command is executed although the ATC arm is not in correct position.	Spindle orientation disabled	

Alarm No. (SQ Coil No.)	Alarm Name	Description	Machine Status	Sensor Detecting the Alarm
283 (F83)	NOT UNCLAMPED (NOT STOP SPINDLE)	The TOOL UNCLAMP button is pressed while the [SPDL JOG] menu key is held pressed or while the spindle rotating.	Tool unclamp is disabled.	Y300
284 (F84)	NOT UNCLAMPED (SPDL ORI. UNFIN)	The TOOL UNCLAMP button is pressed although the spindle orientation has not been completed.	Tool unclamp is disabled.	Y2D6 X246
285 (F85)	NOT UNCLAMPED (NOT MANUAL MODE)	The TOOL UNCLAMP button is pressed in other than the manual mode.	Tool unclamp is disabled	
286 (F86)	NOT AUTO MODE (UNCLAMPED TOOL)	An attempt is made to change the operation mode to automatic while the tool is unclamped.	Mode switching is disabled.	SQ11, X91, SQ10, X90, SA1, X10C
287 (F87)	NOT AUTO MODE (MAG. MANUAL OPER)	An attempt is made to change the operation mode to automatic while manually setting or removing a tool at the magazine.	Mode switching is disabled.	SA4 X16
289 (F89)	NO TOOL LOAD (ARM/SFTER POS ERR)	The tool load command is executed although the ATC arm is not in the correct position.		
290 (F90)	NO TOOL LOAD (MAGAZINE POS ERR)	The tool load command is executed although the magazine is not in the correct position.	Attempted operation is disabled.	
291 (F91)	NO TOOL UNLOAD (TOOL IN MAG.)	The tool unloading command is executed although a tool exists at the tool change position in the magazine.		
292 (F92)	NO UNLOAD (ARM/SHIFTER POS ERR)	The tool unload command is executed although the ATC arm is not in the stand-by position.		
293 (F93)	NO TOOL UNLOAD (MAG. POS ERROR)	The tool unload command is executed although the magazine is not in the specified position.		
294 (F94)	NO TOOL SELECT (INCORRECT TNo.)	A tool number greater than the magazine capacity is specified or No. 0 is specified although No. 0 pocket does not exist in the magazine.	Attempted operation is disabled.	
295 (F95)	NO TOOL SELECT (ARM/SFT POS ERR)	The magazine rotation command is executed although the ATC arm is not in the stand-by position.	Attempted operation is disabled.	
296 (F96)	NO TOOLCHANGE (AXIS NOT ATC POS)	The tool change command is executed although the axis is not in the ATC position.	Attempted operation is disabled.	
302 (F102)	ATC ARM DRIVER MALF.	Alarm with the ATC arm controller	ATC arm stops.	ATC arm controller
303 (F103)	ATC ARM 0° MALF.	The 0° completion signal is not turned on although the ATC arm 0° command is executed.	Attempted operation is disabled.	ATC arm controller
310 (F110)	CAN'T ATC COVER CLOSE	The ATC cover close command is executed although the ATC arm is not in the stand-by position.	Attempted operation is disabled.	
313 (F113)	MMS SKIP SIGNAL MALF.	The touch sensor skip signal is turned on when the axis movement is in the rapid traverse.	Emergency stop	
314 (F114)	PTP ERROR	The fixed point positioning is specified in either ATC, pallet change or tool length measurement, but the fixed point positioning completion signal is not turned on even after 10 minutes passed.		

Alarm No. (SQ Coil No.)	Alarm Name	Description	Machine Status	Sensor Detecting the Alarm
316 (F116)	SET UP SWITCH WAS SELECTED	The automatic mode is selected when the SET UP switch is ON.		SA3 X10A
317 (F117)	CAN'T PLUS MOTION	After the power supply is turned on, the axis movement in the positive direction is attempted without trying to move the axis in the negative direction at all.	Attempted operation is disabled.	
320 (F120)	CAN'T ATC (M.LOCK OR Z.NG)	The ATC command is executed when the machine lock or Z-axis neglect is ON.	Cancel the machine lock or Z-axis neglect, and restart the operation in the cycle start mode.	
321 (F121)	CAN'T ATC (INTERFERE IN MG.)	The tool change command is executed for the pockets adjacent to a large-diameter tool.	Attempted operation is disabled.	
325 (F125)	DOOR INTERLOCK (SINGLE BLOCK)	Automatic operation is attempted when the machine SET UP switch is in the "**I**" position.	Single block operation	
326 (F126)	DOOR INTERLOCK (AT ATC)	The machine door opens while the ATC is in operation when the machine SE UP switch is in the "**O**" position.	Operation stops.	
327 (F127)	RESTART (CYCLE START PB.)	Release the alarm 326 or 234, and restart the operation by pressing the CYCLE START button. The alarm 326 turns to the alarm 232 after the ATC ends.	Display of message.	
370 (F170)	NC TABLE UNCLAMP SENSOR MALF.	The unclamp sensor is not turned on although the NC rotary table unclamp command is executed, or the unclamp sensor is not turned off although the NC rotary table clamp command is executed (for 10 seconds).	Operation stops.	XE
371 (F171)	5 AXIS UNCLAMP SENSOR MALF.	The unclamp sensor is not turned on although the NC tilt axis unclamp command is executed, or the unclamp sensor is not turned off although the NC tilt axis clamp command is executed (for 10 seconds).	Operation stops.	
373 (F173)	5 AXIS ORIGIN RET. UNFINISHED	ATC, tool length measurement or the like is attempted before the 5th-axis zero-point return is completed.	Operation stops.	
374 (F174)	DOOR INTERLOCK SWITCH MALF.	The SAFETY switch of the machine door interlock is malfunctioned.	Feed hold Spindle stops.	
		An error is detected in the machine door lock confirmation signal when the machine is in operation. An error is detected in the door lock confirmation signal when the machine door is opened/cosed.	Emergency stop	
376 (F176)	DESIGNATION WORK NO. NOT FOUND	The warm-up operation program No. has not been registered in the memory or different from parameter settings.	Warm-up operation is disabled.	Check the registration of the parameters and program
377 (F177)	DESIGNATION WORK NO.=0,EXCESS	The warm-up operation program No. has been set to 0 or larger than 9999.	Warm-up operation is disabled.	Check the registration of the parameters.

Alarm No. (SQ Coil No.)	Alarm Name	Description	Machine Status	Sensor Detecting the Alarm
378 (F178)	WARMING UP !!	Indication of the machine status during warm-up (not an error).	Indication of the machine status	
381 (F181)	OPERATING SW.CONDITION ERROR	Any of the following operation switches have already been turned on before the power supply is turned on: ・CYCLE START button ・SPINDLE START button ・AXIS TRAVEL buttons (X, Y, Z, 4th and 5th) ・DOOR UNLOCK button ・TOOL UNCLAMP switch ・MAGAZINE FORWARD/REVERSE button	Emergency stop	
382 (F182)	POWER CUT OFF MALF.	When the machine door is open, the power supply to the spindle is not shut off. (The spindle power shutoff confirmation signal is not turned on although the machine door is open.)	Emergency stop (The emergency status is maintained even if the power supply to the NC unit is turned off.)	
		An error is detected in the emergency stop circuit. (The relay contact of the software emergency stop is not turned off in case of emergency stop.)		
		The power supply to the spindle is not shut off in case of emergency stop. (The power shutoff confirmation signal is not turned on in case of emergency stop.)		
		An error is detected in the magazine door interlock circuit. (The safety relay contact is not turned on although the magazine door is opened.)		
384 (F184)	WARM UP ERROR	The warm-up operation does not end within a specified time due to other alarm or the like in the warm-up operation mode.	Halt	Other alarm

- MEMO -

3 M CODE LIST - VTC-200/300 (MAZATROL FUSION 640M)

M Code	Function
M00	Program stop Stops reading and execution of a program; the P.STOP in the CNC Signal window goes on. The spindle also stops.
M01	Optional stop Executes the same function as the program stop function provided that the OPTIONAL STOP menu is selected (valid). If the menu is not selected (invalid), the M01 command is ignored.
M02	End of program Stops the spindle, coolant, air blast and oil mist. The NC functions are all reset and the machine stops. In an operation mode other than tape, the M02 command executes the same function as the function called by the M30 (reset and rewind) command.
M03	Spindle CW start Starts the spindle in the forward direction.
M04	Spindle CCW start Starts the spindle in the reverse direction.
M05	Spindle stop Stops the spindle, which was started by M03 or M04.
M06	Tool change Returns the spindle tool to the magazine and mounts the specified tool to the spindle. If the next tool command is specified, the next tool is loaded from the magazine to the ATC arm. Program examples: 1. M06 T** T△△ ↑ ↑ Specified tool Next tool 2. T**; M06; T△△ Machining program M06; 3. M06 T** ↑ Valid/invalid is selected by parameter setting. (Standard: Invalid)
M07	Mist coolant ON (option) Starts the oil-mist coolant system.
M08	Flood coolant ON Starts the flood coolant system to discharge coolant from the nozzles.
M09	All coolant OFF Turns off all coolant systems and air blast system. Note: It is not allowed to specify a coolant supply M code preceding M09 in the same block.
M10	Tool clamp Clamps a tool.
M11	Tool unclamp Unclamps a tool. For this operation, the spindle must be stopped in the oriented position.
M15	ATC cover close Closes the ATC cover.

M Code	Function
M16	ATC cover open Opens the ATC cover.
M19	Spindle oriented stop Stops the spindle in the oriented stop position (tool change angular position).
M23	Error detect ON Select the exact stop mode. Note: For details of the exact stop mode, refer to the instruction manual for the NC.
M24	Error detect OFF Cancels the M23 mode.
M30	Reset and rewind Calls the same function as M02; stops the machine and resets the NC functions. The program is rewound to the start of the program.
M33	Tool length measurement unit extend (option) Moves out the tool length measurement unit to the measuring position for tool breakage detection or tool length measurement cycle.
M34	Tool length measurement unit retract (option) Moves in the tool length measurement unit to the retracted position for tool breakage detection or tool length measurement cycle.
M35	Tool breakage recovery (Valid only for Mazatrol programs) (option) Calls the tool breakage detection cycle. Note: In the first ATC cycle after the execution of the M35 command, the selected tool is checked for breakage. For details of this function, refer to the instruction manual for the NC (Mazatrol).
M43 M44 M45	Index command (option) Rotates the M-code index table.
M46	4th axis table unclamp (option) Unclamps the 4th-axis table. Unclamps the 4th-axis table during the 4th-axis feed and clamps the 4th-axis table when 4th-axis feed stops. In complicated machining, automatic clamping may cause trouble to machining. By executing the M46 command, the 4th-axis table remains unclamped until the M47 command is executed or the M46 command is reset, causing no trouble to machining.
M47	4th axis table clamp (option) Clamps the 4th-axis table.
M48	M49 (override cancel) cancel Cancels the override cancel function which is specified by the M49 command. Note: In the M48 mode, the override function is valid.
M49	Override cancel Cancels the override function (feedrate override and spindle override). Once the override function is canceled, spindle rotation and axis feed are executed at the spindle speed and feedrate specified in a program. Note: When power is switched on or after emergency stop, the NC is in the M48 mode.
M50	Air blast ON (option) Starts the air blast system. Note: To stop the air blast system, execute M09.
M51	Through-spindle coolant (option) Discharges coolant through the center hole of a tool if an oil-hole attachment or a through-spindle coolant unit is used. The M51 command is canceled at the ATC command. When the through-spindle coolant unit is used, through-spindle air is discharged for 6 seconds to clean the spindle at the M09 command or in the ATC operation.

M Code	Function
M52	Tap coolant ON (option) Discharges tap coolant if the tap coolant unit is installed. Note: To top the tap coolant, execute M09.
M58	Tool life check Causes single block stop if tool life over alarm occurs with the tool mounted in the spindle. Note: To continue the operation, press the CLEAR (CLEAR) key and the CYCLE START (CYCLE START) button.
M90	Mirror image cancel Cancels the mirror image function called by M91, M92, or M93.
M91	Mirror image X-axis ON Calls the X-axis mirror image function. Note: Axis motion direction is reversed for the X-axis.
M92	Mirror image Y-axis ON Calls the Y-axis mirror image function. Note: Axis motion direction is reversed for the Y-axis.
M93	Mirror image B-axis (4th) ON (option) Calls the B-axis (4th-axis) mirror image function. Note: Axis motion direction is reversed for the B-axis (4th-axis).
M98	Sub-program call-out Calls a sub-program. Note: For details of this function, refer to the instruction manual for the NC.
M99	Return to main program Indicates the end of a sub-program. Note: For details of this function, refer to the instruction manual for the NC.
M120	Automatic power shut-off Shuts off the power to the machine if the AUTO POWER OFF switch is set to the “ON” position.
M132	Through-spindle air ON Discharges through-spindle air.
M139	Heavy tool cancel Cancels the heavy tool mode.
M140	Heavy tool designation Designates the heavy tool mode.
M149	Magazine rotation Selects the tool in the magazine. M149 T△△ selects No. △△ tool.
M160	M161, M162 cancel Sets the spindle speed arrived completion level at 85% of the specified spindle speed.
M161	Spindle speed arrived setting at 70% Sets the spindle speed arrived completion level at 70% of the specified spindle speed.
M162	Spindle speed arrived setting at 50% Sets the spindle speed arrived completion level at 50% of the specified spindle speed.
M169	M170 cancel Makes the AFC function valid. (Normally, the AFC function is valid.)
M170	AFC cancel Cancels the AFC function.

M Code	Function
M195	Tool breakage detection ON (option) Executes a tool breakage detection cycle. This command is automatically specified right before the ATC cycle when M35 is entered in a MAZATROL program. Usually, this command is not used by the customer.
M196	Tool length measurement mode ON (option) Turns on the tool length measurement mode.
M196	Tool length measurement mode ON (option) Turns on the tool length measurement mode.
M197	Tool length measurement mode OFF (option) Once the M196 is executed, tool length is measured and registered by executing "T△△" command; the specified tool is mounted to the spindle by the ATC, whose length is measured and registered to the tool data. The tool length measurement unit retracts when the M197 command is executed. Note 1: For tools such as face mill which do not have the cutting tip at the spindle center line, tool length measurement is not possible. Note 2: M196 and M197 are automatically specified by the NC when "tool length measurement automatic" is executed in the MDI mode.
M198	Semi-automatic tool length measurement (option) Measures the length of the tool mounted in the spindle. In this mode, tool length measurement is possible for tools which do not have the cutting tip at the spindle center line since measurement cycle starts from the present position. Note: M198 is automatically specified by the NC when "tool length measurement automatic" is executed in the MDI mode.
M199	Full-auto tool length measurement (option) Measures the length of the tool mounted in the spindle.

Note 1: Simultaneous processing impossible M codes

Mazatrol permits the entry of up to four M codes in a single block. However, if M codes which cannot be processed simultaneously are entered in the same block, the following message is displayed and operation is suspended.

227 SIMULTANEOUS M CODE OPERATION

- This alarm occurs in the following cases:
- Any combination of M03, M04, M05 and M19
- M15 and M16 (opposing operation)
- M23 and M24 (opposing operation)
- M33 and M34 (opposing operation)
- M48 and M49 (opposing operation)
- M06 and M149 (opposing operation)
- M195 to M198, specified with other M code

Note 2: When entering more than one M code in the same block, there is a restriction on entry order when entering M codes related with coolant and air blast system.

An M code starting the coolant system or air blast system must not be entered before M09.

To enter M08 before M09, for example, enter M08 in a block preceding the M09 block.

M08 M09 M51 → M08
M09 M51 .....

- MEMO -