Cincom
C12/C16

Basic Manual

Precision Machinery Division
Citizen Watch Co., Ltd.

(Blank page)

Cincom C1216 Basic Manual

0. Table of Contents

This table of contents does not go into details.
Please refer to the table of contents for detailed headline listed on the top of each chapter.

Code No.	C-C1216VI VII VIII IX-770 5E1-0003	MFG No.	C1216/0692 ~	Issue Date	2004.6

(Blank page)

Product Code | C | – | C | 1 | 2 | 1 | 6 | VI | VII | VIII | IX | | | | – | 7 | 7 | 0 |

Document Code | 5 | E | 1 | – | 0 | 0 | 0 | 3 |

1. Preface

Code No.	C-C1216 VI VII VIII IX-770 0E1-0102 0E2-0102	MFG No.	C1216/0359 ~	Issue Date	2001.12

(Blank page)

1.1 Preface

- Every effort has been made to ensure the accuracy of all information in this manual. However, the manual may contain incorrect explanation or typographical errors. If you notice any part unclear, incorrect, or omitted in the manual, please contact Citizen Watch Co., Ltd.

- The contents of this manual may be revised without prior notice.
  This manual applies to only the machine of the machine number shown on the back cover. Do not use manuals written for dealers and reference when operating the machine. Also, do not use this manual for other machines.

- The characteristics, functions, and operations of the machine explained in this manual do not apply to worldwide use. Some illustrations in the manual may not be identical to the actual machine.

- Citizen Watch Co., Ltd. has all copyrights regarding this manual. No part of this document may be reproduced in any form or by any means, electronic, mechanical, or photocopying, without the prior written permission of Citizen Watch Co., Ltd.

- The company names and product names shown in this manual are trademarks or registered trademarks of the companies.

- This machine is a strategic product controlled under the Foreign Exchange and Foreign Trade Control Law of Japan. Therefore, it should not be exported without obtaining export license from the Ministry of International Trade and Industry in accordance with the above Law.

1.2 Readers of This Manual

This manual is intended for the following persons who operate the Cincom series:

- Machine operators
- Programmers
- Persons in charge of scheduled maintenance and repair

Thoroughly read this manual before attempting to operate the machine. In particular, thoroughly read until understanding <Chapter 2 Safety Precautions> and the explanations about safety in other sections of the manual.

1.3 Organization of Instruction Manuals

The instruction manuals consist of a set of three volumes. This manual is one of the three manuals. This section shows the organization of each instruction manual.

Cincom with Guide Bushing Equipped Introduction Manual

Chapter	Title	Description
1	Preface	This chapter gives guidelines for using the instruction manual.
2	Safety Precautions	This chapter explains general notes on safety.
3	Overview of NC Lathe	This chapter explains the overview of the NC lathe. It describes items as prerequisites for operation.
4	Overview of Operation	This chapter explains the overview of Cincom operation.
5	Single-Axis Control Group Programming	This chapter explains single-axis control group programs common to Cincom that are equipped with a guide bushing.
6	Cutting Conditions	This chapter explains standard cutting conditions.
7	Terminology	This chapter explains terms for helping first-time users to understand the contents of the manual.

Cincom Basic Manual

Chapter	Title	Description
1	Preface	This chapter gives guidelines for using the instruction manual.
2	Safety Precautions	This chapter explains the precautions of preparation, operation, and maintenance/checking, the safety devices, and the action to be taken in case of emergency (e.g., fire).
3	Machine Specifications	This chapter shows the parts names of the machine and explains the dimensions and specifications of the machine.
4	Operation Panel	This chapter shows the names of the switches, lamps, and keys on the operation panel and explains their usage.
5	Programming	This chapter explains program functions specific to this product.
6	Screen Functions	This chapter explains the displays and functions of each operation screen.
7	Operation	This chapter explains methods of operating the machine.
8	Tooling	This chapter explains positions for mounting tools.
9	Mounting, Adjustment, and Replacement	This chapter explains methods of mounting/dismounting the standard attachments and methods of replacing consumable parts.
10	Periodical Checking	This chapter explains periodical checking, lubrication, and consumable parts replacement.
11	Troubleshooting	This chapter gives alarm lists and explains response to the alarms.

Cincom Application Manual

Chapter	Title	Description
1	Preface	This chapter gives guidelines for using the instruction manual.
2	Safety Precautions	This chapter explains the precautions of preparation, operation, and maintenance/checking, the safety devices, and the action to be taken in case of emergency (e.g., fire).
3	Machine Structure	This chapter shows the names and configurations of mechanical units.
4	Piping and Wiring Diagrams	This chapter gives piping and wiring diagrams.
5	Ladder Input/Output Lists	This chapter gives ladder input/output lists.
6	Parts Lists	This chapter gives parts lists.
7	Attachment	This chapter explains the setup adjustment, mounting, and dismounting of the accessories and peripheral units.
8	Application Programs	This chapter explains programs whose added value is high.
9	Transfer and Reinstallation of Machine	This chapter explains the method of installing the machine when it is transferred.

1.4 Related Instruction Manuals

The following table lists related instruction manuals:

Title	Description
Alkart Net Instruction Manual	This manual explains methods of operating Alkart net. Alkart net is a system that is connected with our (Citizen Watch Co., Ltd.) computer through the Internet and provides various types of services.
Dedicated Hydraulic Bar Loader Instruction Manual	This manual explains methods of operating the bar loader dedicated to the Cincom series. (Optional)
Instruction Manual of Automatic Measure Compensation device	This manual explains methods of operating the automatic measure compensation device. This device automatically measures workpiece dimensions and makes compensation for errors. (Optional)
Instruction Manual of Automatic Tool Setting device	This manual explains methods of operating the automatic tool Setting device. The setter is a device that automatically performs tool setting of core and diameter. (Optional)

1.5 Notation

The following table explains the symbols used in this manual:

Symbol	Explanation
Bold character	Bold characters are used to emphasize the text. Example: Do not open the door during operation.
<1.1 Preface>	Inequality signs < > indicate the reference section.
START 	This symbol indicates a sheet key and button on the operation panel.
[PRG SEL]	Brackets [ ] indicate buttons on screens and in dialog boxes.
"Processing..."	Double quotation marks " " indicate a character string on screens and in dialog boxes.
Procedure	This indicates the procedure for operation.
Command format **Argument** **Axis control group**	This indicates the syntax of words (commands) for coding programs and the axis control groups for which commands are to be specified.
Note Note(s):	These indicate information that you should keep in mind.
Program sample **Operation sample** Example(s):	These indicate examples and exercises.

(Blank page)

Document Code C – C 1 2 1 6 VI VII VIII IX – 7 7 0

Document Code 0 E 1 – 0 1 0 2

0 E 2 – 0 1 0 2

2. Safety Precautions

2.1 Safety Signs

Be sure to read and understand this chapter and all other applicable chapters of this Manual and all on-product safety signs before preparation, operating, and maintaining this machine.
Each safety sign has the specific signal word indicating the degree of the danger. The following three signal words are provided. Each signal word indicates a particular degree of danger as described below.

DANGER; alerts you to an imminently hazardous situation which, if not avoided, will result in death or serious personal injury.
WARNING; indicates a potentially hazardous situation which, if not avoided, could result in death or serious personal injury.
CAUTION; indicates a potentially hazardous situation which, if not avoided, may result in minor or moderate personal injury and/or possible damage to the machine and its components.

The location and content of the on-product safety signs are on the following pages. Be sure these signs are read and understood.


WARNING

Do not remove or hide any safety sign (warning label). If it is peeling, contact that division of Citizen Watch Co., Ltd. described in this manual.
The warning labels are intended to call user's attention to dangers by indicating the contents of the dangers and further prevent the user's safety form being injured or dead and also the machine from being damaged.

<Figure 2.1-1> shows the locations on which the warning labels are put. <Figure 2.1-2> describes the contents of the warning labels.

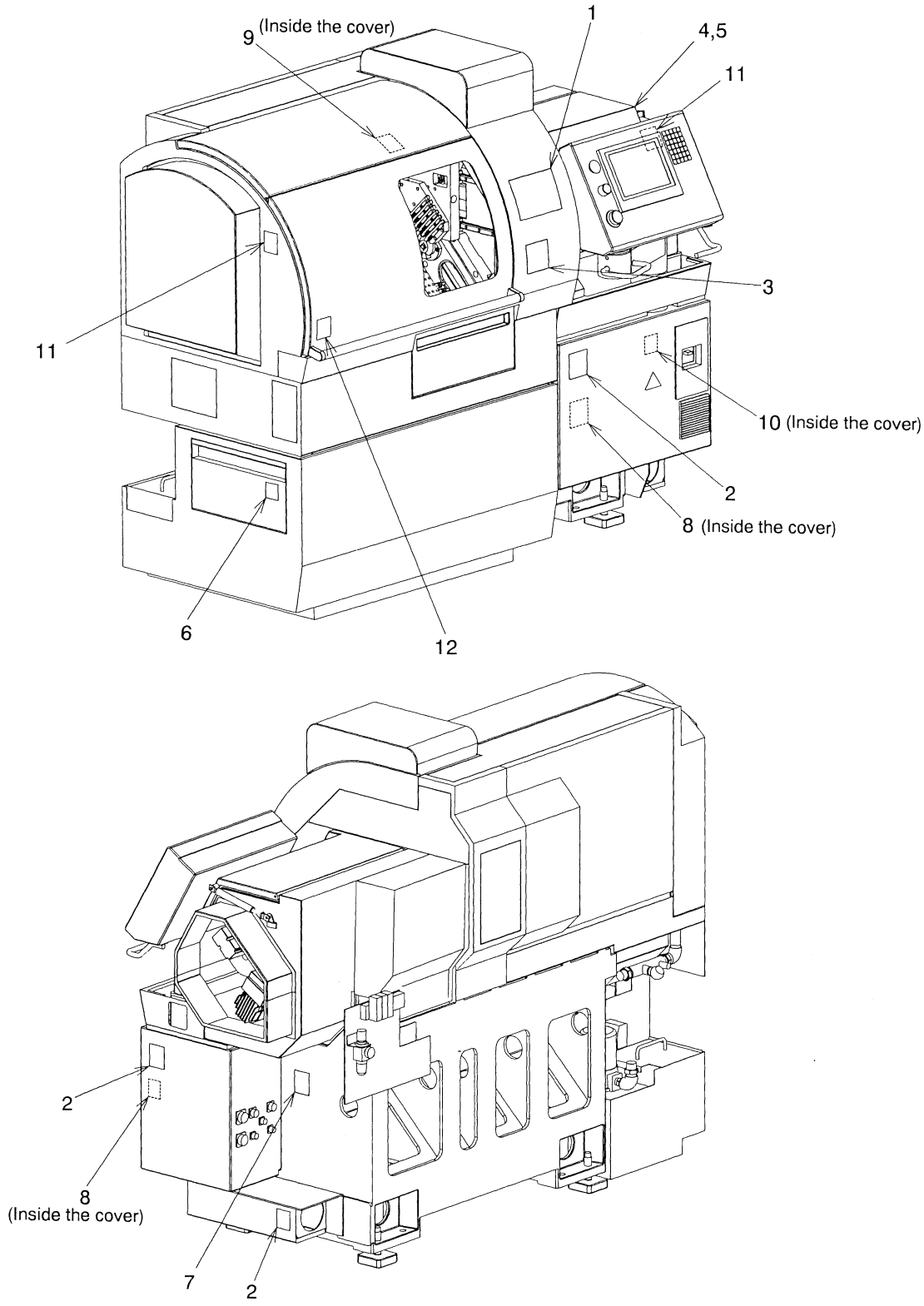


Figure 2.1-1 Location of Warning Labels

1

⚠ WARNING

To prevent serious injury, death or property damage, follow these precautions when operating, inspecting or mintaining the machine:

- Read instruction manual before using machine.
- Do not touch tools or other moving parts when machine is operating.
- Do not operate machine unless all doors and all covers are closed.
- Turn off power at main breaker before opening control unit doors or covers.
- Before operation, be sure all safety devices are working.

⚠ WARNING

To prevent serious injury, death or damage due to fire, do the following:

- Monitor machine when operating.
- Do not use damaged tools.
- Apply sufficient coolant to cutting point during operation.
- Keep flammable items away from machine.
- Be sure all doors, all covers, chip outlet door and product outlet door are closed during operation.
- Regularly remove chip from chip collector and coolant tank.

EW201

2

⚠ WARNING

Hazardous voltage inside. Will shock, burn, or cause death.

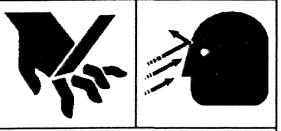
Only qualified personnel totally familiar with electrical circuits and service manual should work inside this enclosure.

Follow Lockout/Tagout.

EW01

3

⚠ WARNING



Moving part, hot chips and hot oil inside.

Keep door closed during operation.

EW02

4

⚠ WARNING

Moving parts inside.

Keep door closed during operation.

EW03

5

⚠ WARNING

Moving parts can cause serious injuries.

Keep hands and body away from moving parts.

EW04

6

⚠ WARNING

Turn off machine before removing chips inside.

Hands or chip remover tool can touch moving parts inside and cause serious injury.

EW05

7

⚠ WARNING

Hazardous voltage. Can shock, burn,or cause death.

Turn off main breaker before connecting or disconnecting coolant pump.

EW07

8

⚠ DANGER

Hazardous voltage inside. Will shock, burn, or cause death.

DO NOT work in this enclosure unless familiar with these electrical circuits and safe servicing procedures.

ED01

9

⚠ WARNING

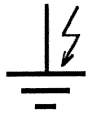
Hot suface can cause bums.

Do not touch when power is on. Wait until surface cools.

EW11

10

⚠ WARNING



GROUND

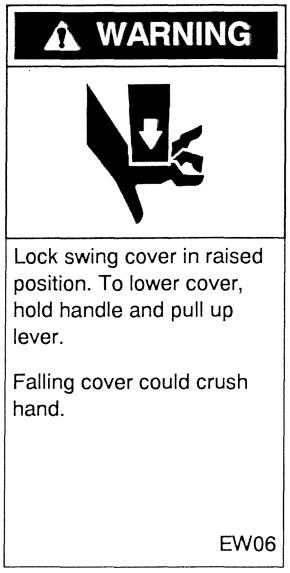
GETTING ELECTRIC SHOCK OR NOISE CAN OCCUR.

THE UNITS MUST BE GROUNDED. (RESISTANCE VALUE MUST BE 100Ω OR LESS)

A-002

Figure 2.1-2 Contents of Warning Labels

11



12

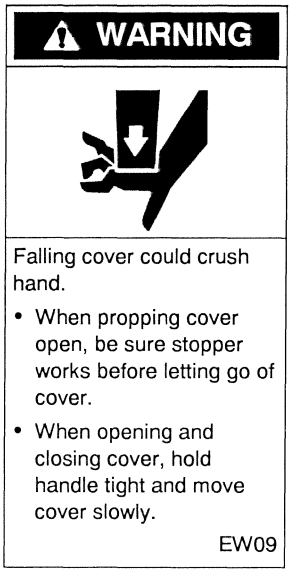


Figure 2.1-2 Contents of Warning Labels

2.2 Emergency Stop Button

The red emergency stop button is located on the operation panel. When there is an emergency situation such as fire, power failure, earthquake, or lightning or if you need to evacuate at once, press this button to stop the operation immediately before you leave the work site. Note, however, that pressing these emergency stop buttons during machine operation might damage a tool as well as the product being processed. To reset the emergency stop state, first verify your safety. Then, turn the locked emergency stop button clockwise to release the lock after confirming the safety of the machine. Also return all the mobile sections of the machine to their return positions and then remove all the workpieces subject to machining from the machine.

<Figure 2.2-1> shows the location of Emergency Stop button.

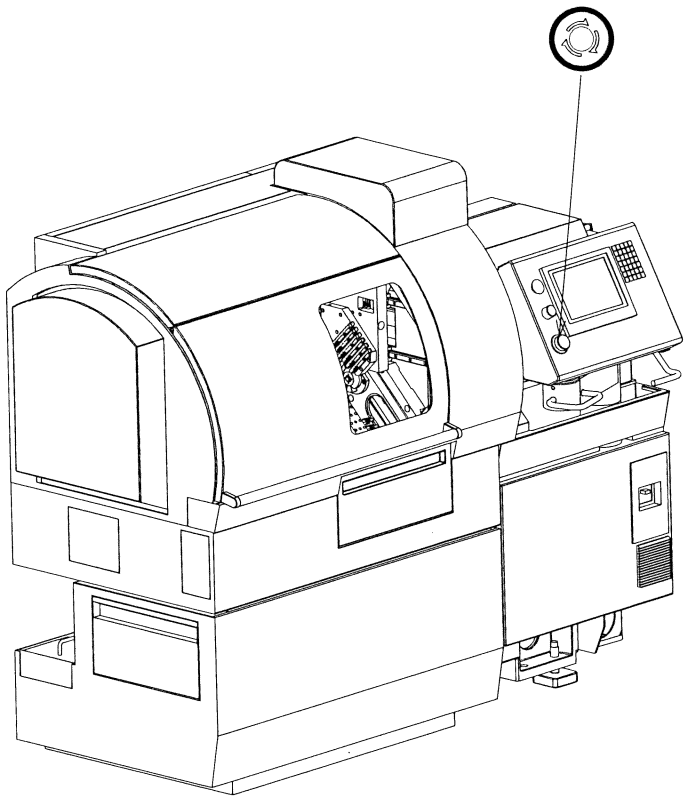


Figure 2.2-1 Location of Emergency Stop Button

2.3 Safety Devices

 WARNING

No safety devices provide complete safety against accidents and hazards.
Be sure to follow the precautions and described in this chapter to operate the machine.
Failure to do so could result in death or serious personal injury.

Cincom provides the following kinds of safety devices to prevent and detect accidents and hazards when operating the machine.
The standard and optional safety devices shown and described on the following pages are installed depending on particular operating needs of the customer.

- Devices to detect any accident that occurs during machine operation.
- Devices to stop the machine operation under an unsafe condition.
- Devices to prevent production of defective products.
- Devices to prevent damage to the machine or tools.

Safety devices are strongly recommended in the following situations:

- When reducing operator's attention such as operating the machine continuously or in night shift.
- When extending the duties of the operator beyond this machine.
- When further reducing the possibility of accidents.

The remainder of this section shows the locations of safety devices and outlines their functions.

Note

Optional safety device can be used only when you purchase it.

<Figure 2.3-1> and <Figure 2.3-2> show the locations on which safety devices are installed.

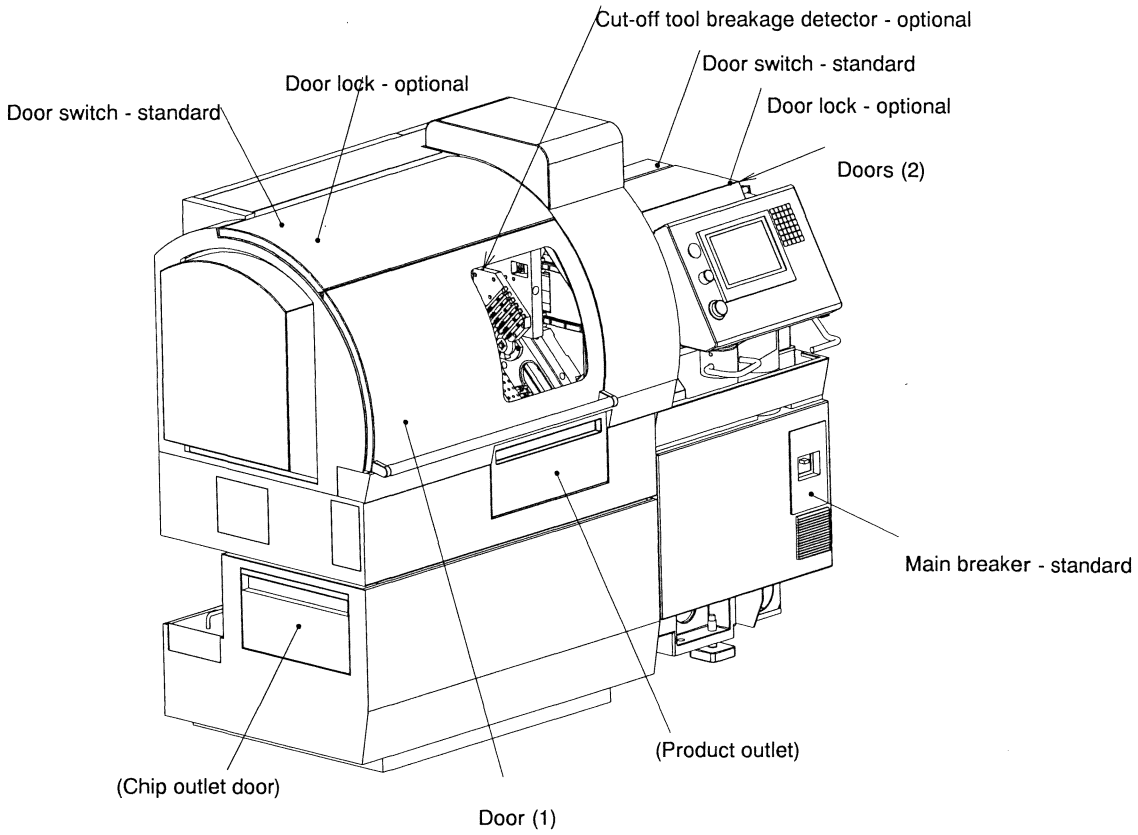


Figure 2.3-1 Locations on which safety devices are installed (viewed from front of machine)

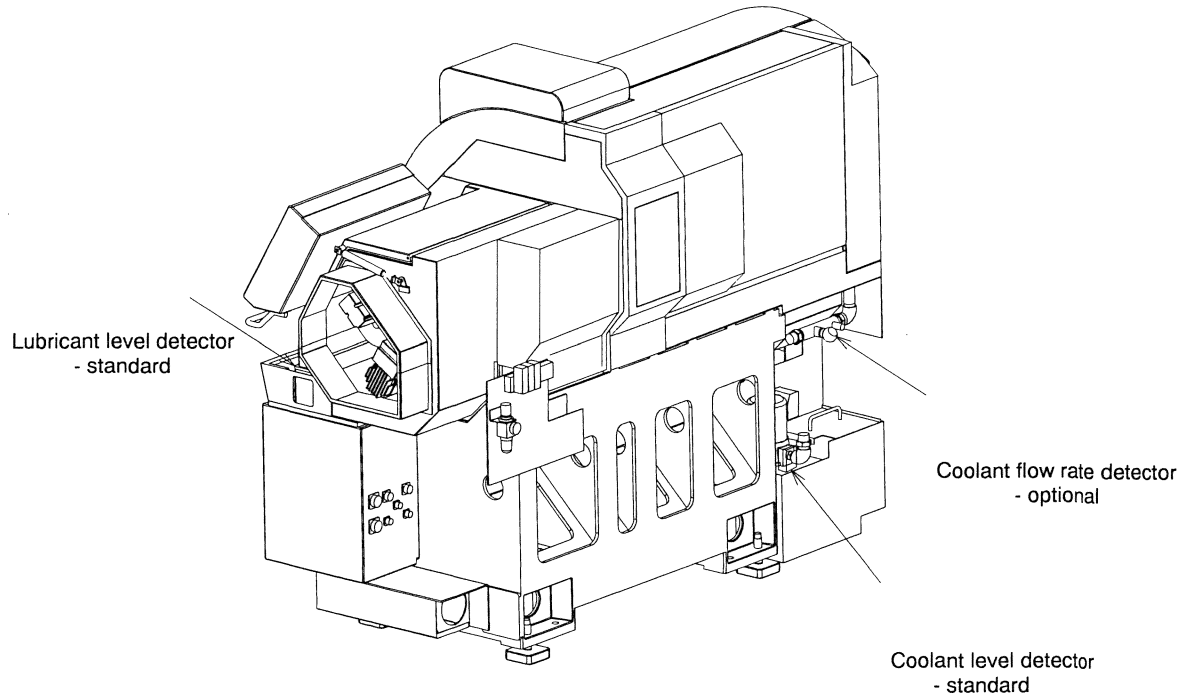


Figure 2.3-2 Locations on which safety devices are installed (viewed from rear of machine)

The following are detailed descriptions of the safety devices:

Door switches - standard

Door switches prevent any person from opening the front left (cutting side) doors (1) and front right (main spindle side) doors (2) during machine operation. See <Figures 2.3-1 and 2.3-2>. The machine can be operated only when all the doors are closed. The machine will be stopped when any of the doors is opened during operation.
In the Handle Feed or Preparation mode, however, operations (excluding the cut-off machining operation) are performed at a speed of up to 2 m per minute even with such doors left open. In addition, manually opening or closing the chuck, manually rotating the spindle, and turning coolant supply on or off are performed whether the doors are open or closed.

Main breaker - standard

The main breaker is automatically turned off to shut down when an overcurrent or a short circuit is detected.

Coolant level detector - standard

This device is installed in the coolant tank and detects the height of the coolant level. When the coolant level gets lower than the limit, the operation of the machine is automatically stopped after completing one cycle to prevent a fire hazard.

Lubricant level detector - standard

This device is installed in the central lubricating oil unit and detects the level of the lubricating oil. When the oil level gets lower than the limit, the operation of the machine is automatically stopped after completing one cycle to prevent machine damage.

Door lock - optional

The door lock disables the door (1) on the front left side (cutting room door) and the door (2) on the front right side (main spindle side door) to be opened during the operation of the machine but enables them to be opened if the machine is stopped (see <Figure 2.3-1>). Door switch is not required when the door lock is mounted.

Cut-off tool breakage detector - optional

A cut-off tool is very easily damaged. If you continue to operate the machine with a damaged cut-off tool, the machine may be damaged or fired. This device detects whether or not material is properly cut. In other words, it indirectly detects whether the cut-off tool is damaged to prevent the above possible dangers. When this device detects a damaged cut-off tool, the operation of the machine is automatically stopped to prevent a fire or machine damage.

Coolant flow rate detector - optional

This device is installed in the coolant supply path to detect the decrease in the flow rate of the coolant. When the coolant flow rate gets lower than the limit, the operation of the machine is automatically stopped to prevent a fire or machine damage.

The following software functions are installed as safety devices in the machine.

Spindle speed change detecting – standard

This function detects excessive changes in spindle speed to prevent machine damage. When the spindle speed fluctuates beyond the predetermined range, the operation of the machine is automatically stopped to prevent the machine from being damaged.
Note, however, that this function must be turned off when the constant surface speed control function is used and during tap and die machining.

Interference check - standard

This function checks for interference between back spindle and opposite tool post. When the function determines the possibility of interference during machine operation, the operation of the machine is stopped to prevent machine damage.

2.4 Specifications of Safety Operations

The start, stop, and speed of the machine are defined as listed in the table below from the viewpoint of safety.

Operation control mode and function	Operating Section	Operation with door being opened	Operation at opening the door
Automatic operation Automatic operation of program Program check Automatic operation of program Handle operation of program MDI operation Preparation for operation Cutting in cut-off machining	Spindle	The operation cannot be started. The operation can be stopped.	The spindle continues to rotate at the specified speed.
	Control axis	The operation cannot be started.	The operation is stopped.
Preparation for operation Start position operation Return position operation Tool selection in cut-off machining Tool selection operation Gang tool return operation Manual tool set operation Positioning point operation Back returning operation	Spindle	–	–
	Control axis	The operation can be started. The operation speed is 2 m/min or less.	The control axis continues the operation at the speed of 2 m/mm or less.
Preparation for operation Chuck open/close operation Automatic adjustment of chuck force Automatic adjustment of guide bushing	Spindle	The operation can be started. The operation speed is 10 min^{-1} or less.	The spindle continues to rotate at the speed of 10 min^{-1} or less.
	Control axis	The operation can be started. The operation speed is 6 m/min or less.	The control axis continues the operation at the speed of 6 m/mm or less.

2.5 General Precautions during Operation

Be sure to follow the notes described below. Failure to do so will result in death or serious personal injury.

- While the main breaker is set to ON, DO NOT make your body contact with or close to the charging section.
- Do not put your fingers or hands into any moving part of the machine during machine operation.

  When the 

key LED on the operation panel lights or flashes, regard the machine status as Operating.

Be sure to follow the notes described below. Failure to do so could result in death or serious personal injury.

- Operate the machine with all the covers closed.
- All the control unit covers and doors must be closed during machine operation. Shutdown the main breaker of the machine before opening the control unit covers and doors.
- Activate all the safety devices during operating the machine.
- DO NOT open all front left and right doors unless the machine is completely stopped.
- Make sure that all front left and right doors are closed and locked (if equipped with locks) and all safety devices are activated before operating the machine.
- To operate the machine, sufficiently understand the operation and visually confirm the operation switch to be used before actually pressing that switch.
- Check the tooling to see that it is securely clamped in place before starting the machine.

Be sure to follow the notes described below. Failure to do so may cause a fire and not only a damage to the property but also a death or serious personal injury.

- Operate the machine where an operator can complete arrangements for fire extinguishing at any time.
- When machining a material combustible (flammable) during machining by cutting, operate the machine where the operator can always monitor the machining procedure.
- Make sure that there is enough coolant in the machine and it is being supplied smoothly to all necessary parts.
- Clean the machine regularly to remove any chips and debris from the cutting area and the chip receiving area. Remove stray chips from the coolant tank as required, depending on cutting condition and type of material being machined.

2.5.1 Before starting the machine

Be sure to follow the notes described below. Failure to do so could result in death, serious personal injury, or damage to the machine.

- Before starting the machine for the first time, you should know the following:
  - The locations of the emergency stop button safety devices, and all front left and right doors.
  - The meaning of all safety signs.
  - How to stop the machine in an emergency situation.
  - What happens to the machine when you operate buttons, switches and keys on the operation panel.
  - Proper shutdown and startup procedures.
  - Procedures for clearing machine troubles.
  - How to shutdown the main breaker of the machine.
- Make sure you are not too tired or sick to operate the machine safely. If you are tired or sick, DO NOT OPERATE THIS MACHINE.
- Wear the proper shop coat (without being slack and/or loose), cap, and safety shoes. Remove gloves, rings, accessories, tie, and those which may possibly be wrapped in the machine from your body.
- Remove the obstacles that may cause slips, falls, and stumbles from the area around the machine.
- Withdraw inflammable chemicals and goods from the area around the machine.
- Clean dirt, oil, and coolant off of the machine, especially on the operation panel.
- Make sure that the machine safety devices are engaged and working properly. DO NOT operate the machine with the safety devices disabled or removed.
- Make sure that the emergency stop button is working properly.
- DO NOT modify the machine and control circuit.

2.5.2 Safety during preparation operation

Be sure to follow the notes described below. Failure to do so causes accidents as in death, serious personal injury, or damage to the machine.

- DO NOT adjust the tools or measure the dimension of the material during machine operation. If adjustment or measurement is necessary, stop the machine first. Then, before adjusting or measuring, make sure that all machine motion has stopped and that the work cycle will not start automatically.
- In the preparation mode, the tool selection or axis move operation can be done if any of the doors on the front left and right sides is opened. Whereas the machine operation speed is suppressed in the situation, take note on the machine fully and do not approach the moving parts of the machine.
- Check whether the chucking force is sufficient for the material.
- Fix the guide bushing by applying proper fixing torque. Failure to do so causes burning or galling.
- Always remove the jig or tool when it is used for the guide bushing or chuck adjustment.
- Confirm that the installed tool does not interfere with the machine in a preparation work such as program installation or tool replacement.
- Confirm the program contents sufficiently. The machine does not have the function of checking of correcting the programs contents automatically.
- DO NOT attempt to perform work that is beyond the specifications of the machine.
- Take coolant flow into consideration when you select the tooling. Select tooling that allows a smooth passage of chips.
- Select the tool fit to the program contents and machining.
- Confirm that proper offset values are set.

2.5.3 Safety at machine start

Be sure to follow the notes described below. Failure to do so causes accidents as in death, serious personal injury, or damage to the machine.

- Confirm that any maintenance work is not done.
- Remove the foreign substance (including tools and work towels) within and out of the machine.
- Make sure there is enough oil in the lubricating oil tank and coolant tank.
- Before starting the machine, confirm that the tooling (including tools, tool holder, chuck, and chuck bushing) is mounted securely at the proper positions.
- Close all the doors on the front left and right sides securely.

2.5.4 Safety during automatic operation

To prevent any secondary accident from being triggered by an unexpected incident, such as jammed chips, be sure to monitor the operation status at appropriate intervals during the automatic operation and troubleshoot a failure, if any. When the machine is not monitored at all times, an operator should be placed within a certain distance from the machine so that he can immediately take an appropriate action at the occurrence of a failure in the machine. Always stop the machine before it is repaired.
During operation, close all the doors of the machine for fire prevention.

Regularly Monitor the Machine

- To produce high-quality products and avoid damage to the machine and possible personal injury, monitor the machine at regular intervals for alarm messages, tool wear, coolant flow, etc.

- Take note on the operation status. If an error occurs, stop the machine immediately and take the appropriate action. The major error include chip entanglement, tool break, and workpiece burning.

- Perform the following items periodically. The interval depends on the cutting condition and material. Define the interval according to the usage state.

Remove chips

Too many chips on the tools and materials can greatly reduce the cooling effect of the coolant. Reduced cooling effect can cause a fire, depending on the type of material being machined. Remove chips from the chip receiving area and the cutting area at suitable intervals. When the machine is regularly used, intervals depend on cutting condition, material, etc.

Inspect the Coolant

Monitor the coolant condition and level regularly. Check regularly to make sure that the coolant is discharged smoothly and that the supply to the machining position is adequate. Failure to do so can result in damage to the tools and a possible fire hazard.

Inspect the Tools

Dull, worn, or damaged tools put excessive load on the machine. This can damage the machine and possibly cause a fire. Follow a regular inspection/maintenance schedule for the tools. During machine operation, listen for abnormal sounds and be aware of possible troubles due to damaged, dull, or worn out tools. Also inspect completed workpieces for evidence of damaged, dull, or worn out tools.

Machine stop during operation

DO NOT adjust the tool or measure the size of the workpiece during operation. If done, a serious personal injury may occur. If it is necessary to mount or adjust the tool or measure the size of the workpiece, first stop the machine. Then after confirming that the machine is stopped completely and the machining cycle is not started automatically, start the adjustment or measurement work.

Door closing

During operation, be sure to close the cutting room door, main spindle side door, chip outlet, and product outlet. If a fire occurs due to some cause without any of the door closed, frames may come out from the doors to spread the fire.

2.5.5 Safety during maintenance

Be sure to follow the notes described below. Failure to do so causes accidents as in death, serious personal injury, or damage to the machine.

- Turn off the main breaker before opening the cover or door of the control unit.
- Only qualified maintenance personnel should perform maintenance operations on the machine.
- When more than one engineer maintain the machine, they should always communicate with each other by voices to confirm the safety.
- The electrical components in this machine are high precision devices that can be damaged by excessive force, shock, or vibration. Use caution when you handle all electrical components of the machine.
- Use care when you disconnect connectors. They are easily damaged.

2.5.6 Safety at end of operation

Be sure to follow the notes described below. Failure to do so causes accidents as in damage to the machine.

- To turn off the power of the machine, press  after stopping the machine, confirm that the LCD display disappears and the lamps on the control unit are off, and turn off the main breaker.

2.6 Action of Emergency Situations

Actions and procedure required in emergency situations are explained in this section.

2.6.1 Emergency situations requiring evacuation

Concern human lives first when an emergency situation such as a fire, lightening, or earthquake occurs.

- If you have time before evacuating

  Stop the machine immediately. Press  and confirm that the LCD display disappears and lamps on the control unit are off. Then, turn off the main circuit breaker. Try to get the machine to stop when the tool is not touching the material. Otherwise, the tip of the tool may be damaged in the process of stopping or restarting operation.

- If you do not have time and must evacuate immediately

  Immediately press the Emergency Stop button and turn off the main circuit breaker to stop the machine.

2.6.2 Power failure

If a power failure occurs, illegal data may enter into the machine due to the machine control scheme. After the power is recovered, turn off the main breaker once and then turn on the power of the machine again.
In addition, if a momentary power failure occurs without machine stop and LCD display disappearing, the main circuit breaker must be turned off once. Press  , confirm that the LCD display disappears and lamps on the control unit are off, and turn off the main breaker. Then, turn on the power of the machine again.

2.6.3 Resuming operation

After emergency stop of the machine or a power failure, follow the procedure below to resume operation of the machine:

Operation procedure

1. Inspect the tool sufficiently to check whether it is damaged or worn. If the tool is damaged or worn considerably, replace it with a normal one. Also confirm that the machine can be operated safely in the normal state.
2. Provide the cut-off machining for the workpiece being subject to machining at the occurrence of the emergency situation.
3. Restart the operation of the machine according to <Chapter 7 Operation> in the Basic Manual.

2.7 Notes for Prevention of Accident

2.7.1 Notes for prevention of electric shock accident

Be sure to follow the notes described below. Failure to do so causes accidents as in death or serious personal injury due to an electric shock.

- DO NOT make your body contact with or close to the charging section of the electric device.
- Always turn off the main breaker of the machine before connecting or disconnecting the power cable of the machine. Then, turn off the power breaker in the shop from which the power is supplied to the machine.
- Ground the machine correctly. Refer to <Chapter 9 Relocating the NC Machine> in the Application Manual for details.
- Before opening the cover or the door of the control unit, always turn off the main breaker.

2.7.2 Notes for prevention of injury or death

Be sure to follow the notes described below. Failure to do so causes accidents as in death or serious personal injury.

- Make sure you are not too tired or sick to operate the machine safely. If you are tired or sick, DO NOT OPERATE THIS MACHINE.
- Make sure you wear the proper work clothes (no loose clothing), safety goggles, cap, and safety footwear. Also make sure you remove any gloves, rings, accessories, neckties that may cause you to be caught by the machine.
- DO NOT put your fingers or hands into any moving part of the machine. DO NOT make your body contact with or close to any rotating section of a tool or the machine.
- During the operation of the machine, have the machine cover and all of the doors on the front left and right doors closed. In addition, DO NOT open any of the doors on the front left and right sides unless the machine completely stopped.
- In the preparation mode, the tool selection or axis move operation can be done if any of the doors on the front left and right sides is opened. Whereas the machine operation speed is suppressed in the status, take note on the machine fully and do not approach the moving parts of the machine.
- Be sure to maintain the machine by qualified personnel familiar with the maintenance work.
- If more than one engineer maintain the machine, they should always communicate with each other by voices to confirm the safety.
- DO NOT modify the machine and control circuit.
- Machine transfer requires work using cranes and forklifts and slinging work. Be sure to assign the personnel certified by the public institute to the work.

2.7.3 Notes for prevention of fire

Major causes of occurrence of fire

- The friction between the workpiece and the tool or the metallic section of the machine caused by break or wear of a cutting tool may overheat the machine to be fired.
- Because the coolant is not discharged to the machining point enough, the workpiece is overheated to fire. The following causes may be assumed:
  - The position of the coolant nozzle is incorrect.
  - Chips get caught in the coolant nozzle to move the position of the coolant nozzle.
  - Because of insufficient coolant in the coolant tank, the coolant flow is rather low.
  - Because chips are accumulated in the coolant tank to decrease the quantity of coolant flown into the pump, the coolant flow is rather low.
  - Because the filter in the coolant tank is clogged, the coolant flow is rather low.
  - Chips are accumulated around the machining point.
- Miscellaneous
  - Inflammable coolant or coolant having only low cooling effect is used to cause a fire to occur.
  - Because the temperature is extremely high in the cutting room, the coolant is vaporized to cause a fire to occur.
  - A combustible workpiece (flammable workpiece) is subject to machining to cause a fire to occur.
  - Because no safety devices are operated, any failure cannot be detected to cause a fire to occur.

Major causes of spread of fire

- The occurrence of a fire was found late because the machine was not monitored.
- Frames were spread out of the cutting room because the cutting room door, chip outlet, and/or product outlet were opened.

Notes for fire prevention

Be sure to follow the notes described below. Failure to do so may cause a fire resulting not only a damage to the property but also a death or serious personal injury.

- Monitor the machine status regularly. Then take appropriate action if a malfunction is found.
- To minimize the spread of a fire if it occurs, monitor the operation status to enable the extinguishing work to be done immediately.
- Always check the machine depending on <Chapter 10 Scheduled Maintenance> in the Basic Manual.
- Check the cutting tools to use proper tools which are neither broken nor worn.
- Operate the machine within the range of the specification. See <Chapter 7 Operation> in the Basic Manual.
- Provide machining under the proper cutting conditions.
- Confirm that the quantity of the coolant is enough to be supplied smoothly to the sections requiring the coolant.
- Confirm that the coolant hose is neither twisted nor broken, the connections are not loosened, and chips are not accumulated at the bending sections.
- It is particular dangerous when an error occurs in the electric device of the machine to generate sparks. Stop the machine immediately, turn off the main breaker of the machine, and contact the Cincom service.
- For the machining of a combustible workpiece (flammable workpiece), operate the machine under the condition that an operator always monitors it. Prepare a metal fire extinguisher near the machine.
- Check the cutting tools for any breaks or wears. If a failure is found on a cutting tool, replace it with a normal one. Also check the machining surfaces of products. If they indicate a failure of the tool, replace it with a normal one.
- DO NOT bring fire such as light of cigarette, light of lighter, and sparks close to the machine.
- Confirm the doors on the front left and right sides, chip outlet, and product outlet to close them securely.
- Monitor the quantity of chips in the chip receiver and the coolant tank regularly to remove them.

2.7.4 Notes for prevention of machine damage

Be sure to follow the notes described below. If not, the assets including the machine and machining products may be lost.

- Confirm that the installed tool does not interfere with the machine in a preparation work such as program installation or tool replacement.
- DO NOT modify the machine and control circuit.
- DO NOT provide any machining exceeding over the machine specification.
- Electric parts are extremely precise to be damaged by excess force, shock, or vibration. Take sufficient notes on handling of electric parts.
- Take sufficient notes when connecting or disconnecting the connector because it can easily be damaged.

(Blank page)

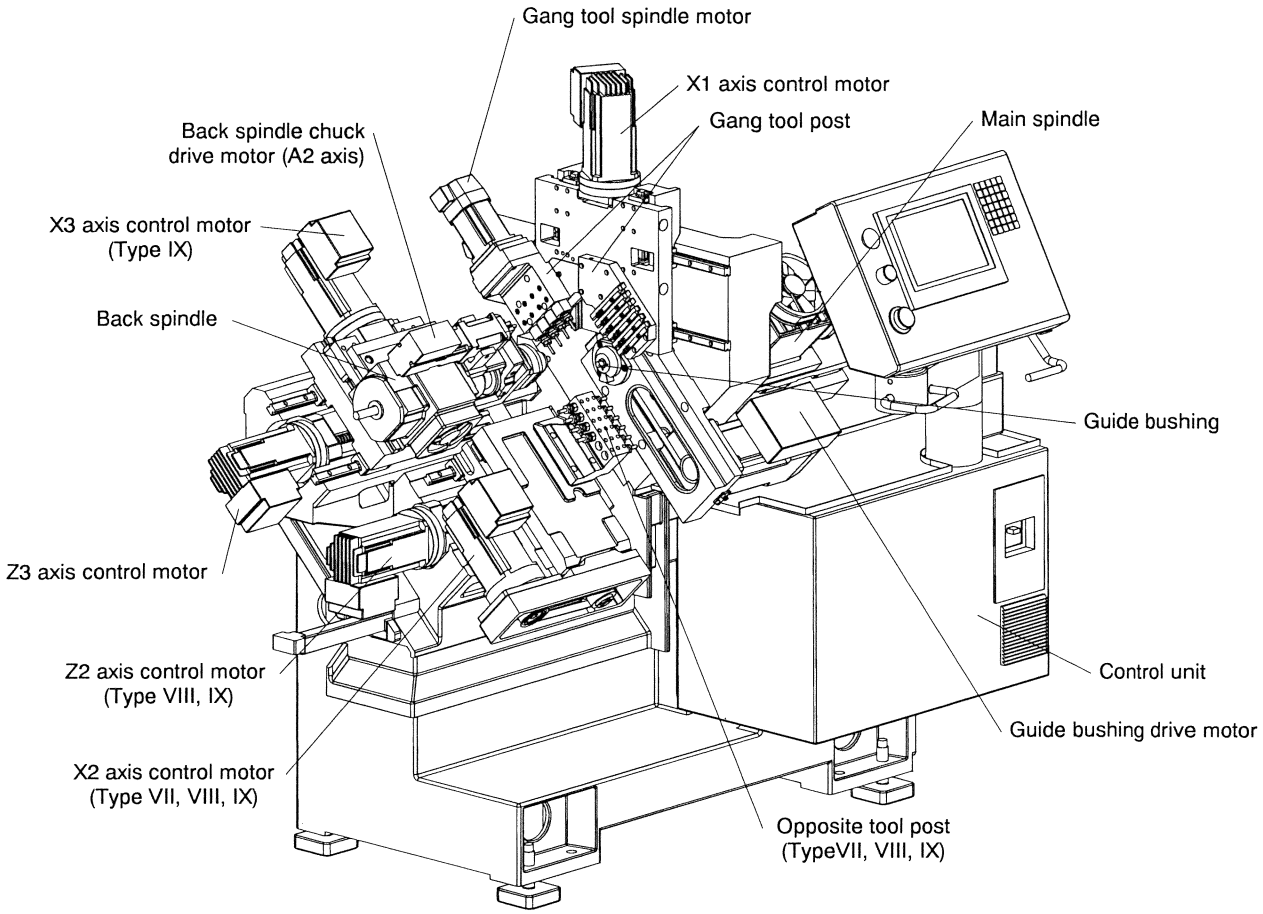
Product Code C – C 1 2 1 6 VI VII VIII IX – 7 7 0

Document Code 0 E 1 – 0 2 0 2

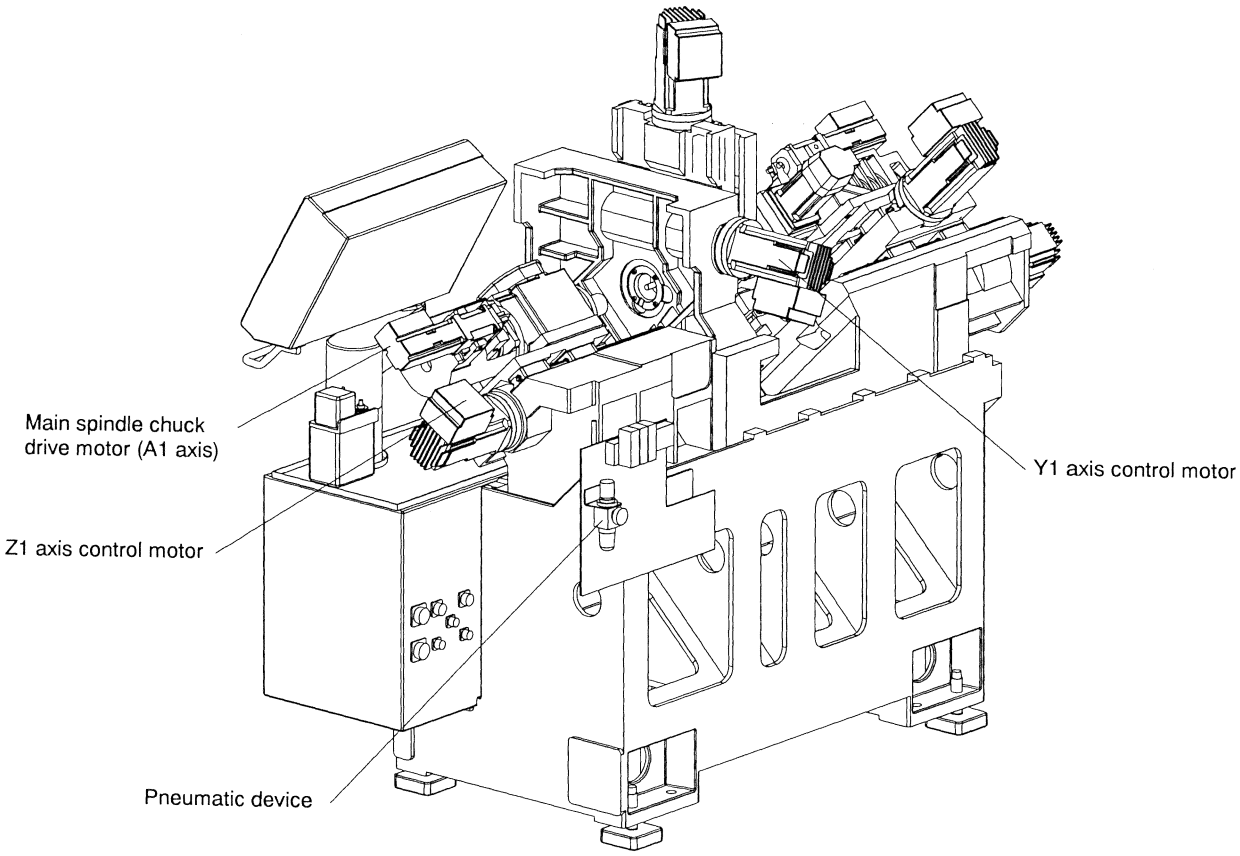
3. Machine Specifications

(Blank page)

3.1 Nomenclature of Machine Components

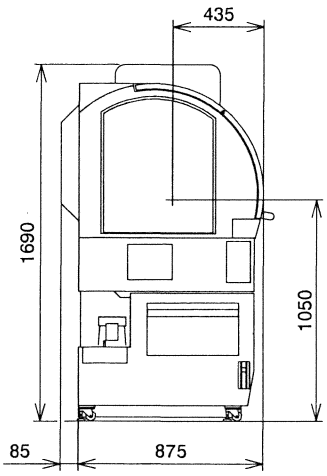


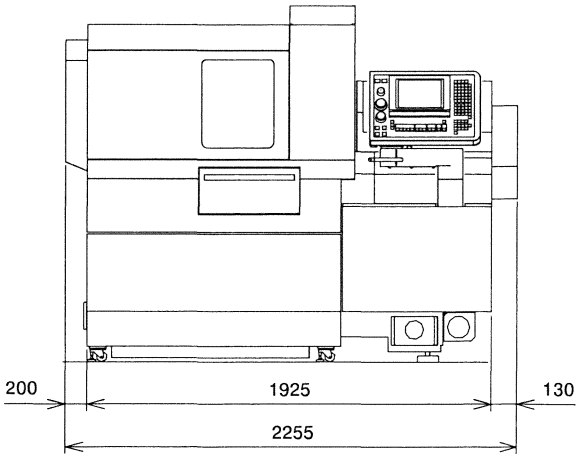
Front view of the machine

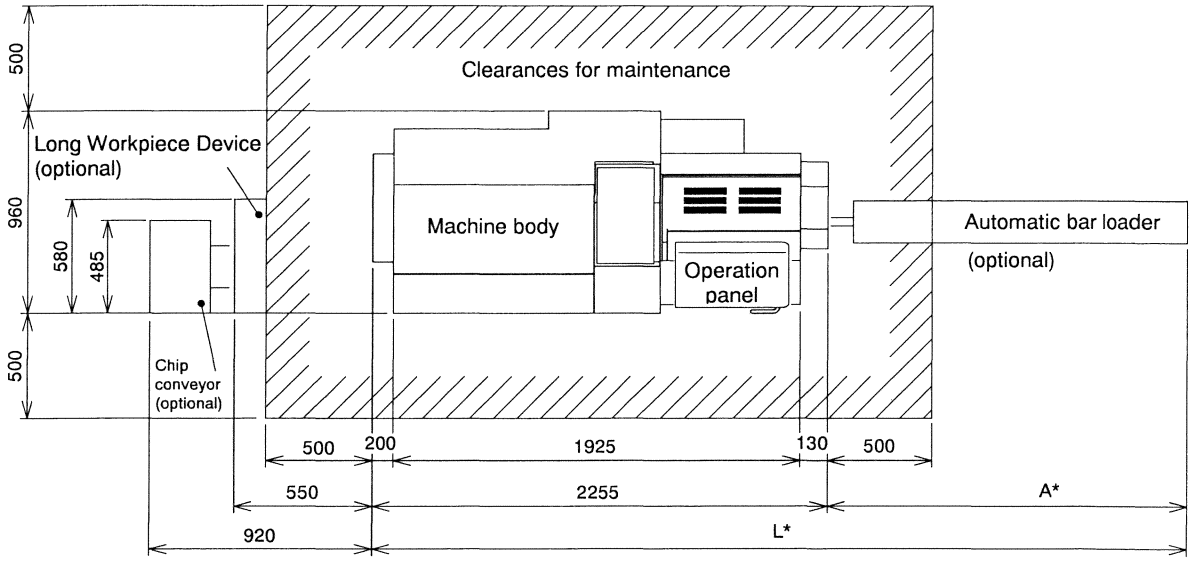


Rear view of the machine

3.2 Outside Dimensions and Layout of the Machine







Manufacturer	Model	A	L
Citizen Watch Co., Ltd.	CAV12C-IS 2.5 m CAV16C-IS 2.5 m	3335	5590
Citizen Watch Co., Ltd.	CAV12C-IS 3.0 m CAV16C-IS 3.0 m	3835	6090
Citizen Watch Co., Ltd.	CAV12C-IS 4.0 m CAV16C-IS 4.0 m	4835	7090

3.3 Machine Specifications

3.3.1 Machine specifications

No.	Item	C12/16				Remarks
		VI	VII	VIII	IX	
1	Maximum machining diameter (Optimum machining diameter)	C12: ϕ 12 mm [ϕ 0.47"] C16: ϕ 16 mm [ϕ 0.63"] (ϕ 4 mm [ϕ 0.16"] to max. machining diameter)				
2	Maximum machining length	200 mm [7.87"] per chuck				The standard workpiece receiver can collect workpieces in length up to 90 mm [3.54"]. Use a long workpiece device (optional) to machine workpieces longer than 90 mm. This device enables the machining and collection of workpieces in length up to 400 mm [15.75"].
3	Maximum front drilling diameter	C12: ϕ 6 mm [ϕ 0.24"] C16: ϕ 8 mm [ϕ 0.31"]				A hole of a diameter greater than 8 mm can be drilled depending on the cutting conditions and the material of workpieces.
4	Maximum front tapping diameter (With a tap or die)	M6 (tap) M4 (die)				The maximum tapping diameter conforms to the cutting tap specifications.
5	Spindle through-hole diameter	C12: ϕ 14 mm [ϕ 0.55"] C16: ϕ 20 mm [ϕ 0.79"]				The through-hole of the chuck sleeve is ϕ 13mm [ϕ 0.51"] / ϕ 17 mm [0.67"] in internal diameter. The maximum diameter of the finger of the bar loader must be smaller than the through-hole diameter of the chuck sleeve.
6	Main spindle speed	C12: 200 to 12,000 min^{-1} C16: 200 to 10,000 min^{-1}				
7	Main spindle speed change	Stepless, S + 5-digit numeric value				
8	Main spindle indexing	15° (1°)				1°: Optional
9	Main spindle C axis	0.001°				Optional
10	Maximum chuck diameter of the back spindle	C12: ϕ 12 mm [ϕ 0.47"] C16: ϕ 16 mm [ϕ 0.63"]				
11	Maximum workpiece length for the front side collection from the back spindle.	90 mm [3.54"]				A long workpiece device (optional) enables the collection of workpieces in length up to 400 mm [15.75"].
12	Maximum workpiece protrusion length from the back spindle	25 mm [0.98"]				Maximum length of a workpiece protrusion from the end face of the back spindle cap nut.
13	Maximum drilling diameter in the back machining process	ϕ 5mm [ϕ 0.20"]				A hole of a diameter greater than 5 mm can be drilled depending on the cutting conditions and the material of workpieces.
14	Maximum tapping diameter in the back machining process	M5				The maximum tapping diameter conforms to the cutting tap specifications.
15	Back spindle speed	200 to 10,000 min^{-1}				
16	Back spindle speed change	Stepless, S + 5-digit numeric value				
17	Back spindle indexing	1°				Optional
18	Back spindle C axis	0.001°				Optional
19	Rotary tool spindle of the gang tool post Maximum drilling diameter Maximum tapping diameter Spindle speed Spindle speed change	 ϕ 5 mm [ϕ 0.20"] M4 200 to 8,000 min^{-1} Stepless, S + 4-digit numeric value				A hole of a diameter greater than 5 mm can be drilled depending on the cutting conditions and the material of workpieces. The maximum tapping diameter conforms to the cutting tap specifications.

No.	Item	C12/16				Remarks
		VI	VII	VIII	IX	
20	Rotary tool spindle of the front/back face machining Maximum drilling diameter Maximum tapping diameter Spindle speed Spindle speed change	—	ϕ 5 mm [ϕ 0.20"] M4 200 to 5,000 min^{-1} Stepless, S + 4-digit numeric value			Optional.A hole of a diameter greater than 5 mm can be drilled depending on the cutting conditions and the material of workpieces. The maximum tapping diameter conforms to the cutting tap specifications.
21	Chuck and guide bushing models Main spindle collet chuck Guide bushing Rego type chuck Back spindle collet chuck	C12: FC096-M [TF16, 40.012, 76-1076] C16: FC261-M [TF20, 40.004, 76-87] C12: WFG541-M/WFG551-M [SD125R-16,208,166.001, B212A] C16: WFG660-M [0201, 61.002, B238] ER11/AR11, ER16/AR16 (Note) C12: FC096-M-K [TF16, 40.012, 76-1076] C16: FC261-M-K [TF20, 40.004, 76-87]				Note: It is not permitted to mount the sleeve for ER16 or AR16 next to the models shown at left. A tool cannot be mounted on the reverse of the station at which the sleeve for ER 16 or AR 16 is mounted. Use for a back spindle of K model with seal provided.
22	Number of tools to be mounted Turning tools on the gang tool post Rotary tools on the gang tool post Front drilling tool Back drilling tool	Up to 17 6 3 4 4 (2-way sleeve)	Up to 22 6 4 6 (Note) 6 (2-way sleeve) (Note)			Note: Number of front/back face machining tool spindles when U121B is mounted: 5 (3 out of 5: rotary) In this case, the maximum number of tools is 20.
23	Tool size Tool (gang tool post) Sleeve	10 × 10 × 120 mm [3/8" × 3/8" × 9/2"] ϕ 19.05 mm [ϕ 3/4"]				
24	Maximum diameter of tool mounted in the rotary tool holder of the gang tool post Drill and end mill	ϕ 7 mm [ϕ 0.28"]				ER11, AR11
25	Maximum diameter of tool mounted on the rotary tool holder for front/back face machining Drill and end mill	—	ϕ 7 mm [ϕ 0.28"]			ER11, AR11
26	Rapid feed rate X1 axis	20 m/min [787.4"/min]				
	Y1 axis	20 m/min [787.4"/min]				
	Z1 axis	20 m/min [787.4"/min]				
	X2 axis	—	20 m/min [787.4"/min]			
	Z2 axis	—		20 m/min [787.4"/min]		
	X3 axis	—			20 m/min [787.4"/min]	
	Z3 axis	20 m/min [787.4"/min]				

No.	Item	C12/16				Remarks
		VI	VII	VIII	IX	
27	Least input increment					
	X1 axis (diameter)	0.001 mm [0.0001"] (0.000 1mm [0.00001"])				The values in parentheses are optional of the submicron function.
	Y1 axis (diameter)	0.001 mm [0.0001"] (0.0001 mm [0.00001"])				
	Z1 axis	0.001 mm [0.0001"] (0.0001 mm [0.00001"])				
	X2 axis (diameter)	—	0.001 mm [0.0001"] (0.0001 mm [0.00001"])			
	Z2 axis	—		0.001 mm [0.0001"] (0.0001 mm [0.00001"])		
	X3 axis	—			0.001 mm [0.0001"] (0.0001 mm [0.00001"])	
	Z3 axis	0.001 mm [0.0001"] (0.0001 mm [0.00001"])				
28	Axis stroke					
	X1 axis	105 mm [4.13"]				
	Y1 axis	202 mm [7.95"]				
	Z1 axis	205 mm [8.07"]				
	X2 axis	—	200 mm [7.87"]			
	Z2 axis	—		75 mm [2.95"]		
	X3 axis	—			66 mm [2.60"]	
	Z3 axis	255 mm [10.04"]				
29	Bar length	2,500 mm [98.43"]				
30	Center height	1,050 mm [41.34"]				
31	Motors					Control motors are as shown below.
	For main spindle drive	1.5/2.2 KW (Rating: Continuous/15 minutes)				Built-in motor
	For guide bushing drive	0. 75/1.5 KW (Rating: Continuous/15 minutes)				Spindle motor
	For back spindle drive	0. 4/0.75 KW (Rating: Continuous/15 minutes)				Built-in motor
	For the tool spindle of the gang tool post	0.75 KW				AC servo motor
	For the tool spindle of front/back face machining	—	0.4 KW			AC servo motor (optional)

No.	Item	C12/16				Remarks
		VI	VII	VIII	IX	
32	Motors X1 axis	0.4 KW				Intelligent servo motor (Ball screw integrated)
	Y1 axis	0.75 KW				Intelligent servo motor (Ball screw integrated)
	Z1 axis	0.75 KW				Intelligent servo motor (Ball screw integrated)
	X2 axis	–	0.4 KW			Intelligent servo motor
	Z2 axis	–		0.75 KW		Intelligent servo motor (Ball screw integrated)
	X3 axis	–			0.75KW	Intelligent servo motor (Ball screw integrated)
	Z3 axis	0.75 KW				Intelligent servo motor (Ball screw integrated)
	For main spindle chuck	0.2 KW				Intelligent servo motor
	For back spindle chuck	0.2 KW				Intelligent servo motor
	For bar loader	0.75 KW				Intelligent servo motor (optional)
	For coolant	0.25 KW				
	For lubrication oil	0.004 KW				
33	Input power capacity	6 KVA				
34	Coolant tank capacity	100 lit				
35	Required floor space	960 mm [37.80"] (depth) 2,255 mm [88.78"] (width) 1,690 mm [66.54"] (height)				
36	Weight	1,600 kg [3.53 klb]	1,650 kg [3.64 klb]	1,675 kg [3.69 klb]	1,700 kg [3.75 klb]	

3.3.2 NC specifications

No.	Item	C12/16				Remarks
		VI	VII	VIII	IX	
1	NC unit	CINCOMSYSTEM M6C				NC unit dedicated to the Cincom C series
2	Display device	10.4-inch color liquid crystal display (LCD)				
3	Displayed language	English (for export)				
4	Spindle	S1, S2, S3, S5	S1, S2, S3, (S4), S5			S1: Main spindle S2: Back spindle S3: Gang tool spindle S4: Front/back face tool spindle S5: Guide bushing drive The tool spindle in parentheses is optional.
5	Control axis (command axis)	X1, Y1, Z1, Z3 (C1, C2) axes	X1, Y1, Z1, X2, Z3 (C1, C2) axes	X1, Y1, Z1, X2, Z2, Z3 (C1, C2) axes	X1, Y1, Z1, X2, Z2, X3, Z3 (C1, C2) axes	All the axes can be controlled simultaneously. The axes in parentheses are optional.
	Control axis (auxiliary axis)	A1, A2 (A3, A7)				
6	Axis control group count	3				An auxiliary axis control group is automatically generated as required.
7	Input code	ISO				
8	Command input system	Incremental or absolute				
9	Feed command system	Feed per rotation or feed per minute (G-code conversion)				
10	Override function Rapid feed Cutting feed	Rotary switch selection100 % at maximum (Standard function)				Only the cutting feed can be increased to 200% (maximum) of the specified value by selecting the setting switch (software switch).
11	Zero point return function	Manual zero point return system (Standard function)				The machine is equipped with the absolute position detection function. Therefore, there is no need to perform operation for zero point return in normal state.
12	On-machine program check function	Manual pulse generator rotation system (Standard function)				When a program is checked on machine, the program execution rate is proportional to the rotation speed of the manual pulse generator. The program can also be run backward by rotating the pulse generator in the minus direction.
13	Manual feed function	Available for all the axes (Standard function)				This function enables the handle to move all the control axes.
14	Manual data input (MDI) function	(Standard function)				This function enables the MDI input and execution of programs.
15	Self-diagnostic function	(Standard function)				This function automatically detects machine alarms and displays alarm messages.
16	Machine status display	(Standard function)				This function displays the machine status.
17	Backup function	(Standard function)				This function saves NC data such as parameters into the hard disk.
18	Operating time display	(Standard function)				This function displays the machine operating time, 1-cycle time, and actual cutting time in 1-cycle operation.
19	Product counter display	Up to 8 digits (Standard function)				The machine enters the 1-cycle stop state when the product counter reaches the specified number of products.

No.	Item	C12/16				Remarks
		VI	VII	VIII	IX	
20	Cycle time check function	(Standard function)				This function checks machine failure by managing the cycle time. The function assumes a machine alarm and stops the machine when the cycle time exceeds 30 minutes.
21	Preparation functions					The functions listed below are provided to support preparation for automatic operation.
	Automatic return to the return position	(Standard function)				The axes automatically return to their return positions (home positions) one after another in the defined order.
	Automatic return to the positioning point	(Standard function)				The axes automatically return their positioning points (home position, or position specified in the machining data).
	Automatic return to the start position	(Standard function)				The axes automatically return to the start position of automatic operation according to the numeric value specified in the machining data that can be set for each workpiece.
	Automatic guide bushing clearance function	For rotary guide bushing (Standard function)				This function automatically adjusts the clearance of the guide bushing.
	Automatic chucking force adjustment function	(Standard function)				This function automatically adjusts the chucking force.
	Automatic cut-off machining function	(Standard function)				This function automatically performs cut-off machining (short cut).
	Material set function	(Option)				This function supports remnant extraction and new material supply in an automatic synchronous bar loader. (Option)
	In-machine tool set function	(Standard function)				This function supports the setting of drilling tools and the gang tool post.
22	Automatic power-off function	(Standard function)				This function shuts down the main circuit breaker to turn off the power when an alarm occurs during automatic continuation operation.
23	Three-dimensional interference check function	(Standard function)				This function monitors interference with a machine component, tool, or material and stops the machine before interference occurs.
24	Tool offset pairs	40				
25	Command to simultaneously specify speed for 4 spindles	(Standard function)				Spindle speed can be specified for up to four spindles simultaneously.
26	Command to simultaneously specify 4 M commands	(Standard function)				Up to four M commands can be specified for a block simultaneously. Restrictions are placed on the use of the M codes that can be specified simultaneously.
27	Axis move command output during axis move	(Standard function)				When an axis move command is executed and the axis reaches the specified position, another axis move command can be issued.
28	Auxiliary function output during axis move	(Standard function)				When an axis move command is executed and the axis reaches the specified position, the M, S, or T command can be issued.

No.	Item	C12/16				Remarks
		VI	VII	VIII	IX	
29	End position specification queuing	(Standard function)				When a specified axis reaches the specified position, another axis can be moved to the end position at the same time.
30	Control/arbitrary axis exchange function	(Standard function)				All the axis control groups are permitted to specify an arbitrary axis and to execute an auxiliary command.
31	Control/arbitrary axis superimpose function	(Standard function)				This function enables pick-off operation by imposing Z3 axis on Z1 axis.
32	Cincom C series dedicated macro	(Standard function)				Dedicated macros (e.g., T code macros) are provided with the C series.
33	Background editing	(Standard function)				While program operation is in progress, another program can be edited.
34	Simultaneous program editing for multiple axis control groups	(Standard function)				The programs of multiple axis control groups can be edited on a screen at the same time.
35	Editing support function					The functions listed below are provided to support editing.
	Calculator function	(Standard function)				This function enables various calculations.
	Code list display	(Standard function)				This function displays lists of available M codes and G codes.
	Cutting condition calculation function	(Standard function)				This function enables you to obtain the spindle speed by entering the cutting conditions.
	Coordinate calculation function	(Standard function)				This function enables you to obtain coordinate values by simply entering specified parameters.
36	Program work area capacity Program storage capacity	(Standard) Program work area capacity Equivalent to 20 m tape [about 8 KB] Program storage capacity Equivalent to 40 m tape [about 16 KB]				Each tape length at left includes the size of the machining data.Each program work area capacity indicates the maximum size of a program that is actually operated. When a subprogram is used, the subprogram size must be added. Each program storage capacity indicates a capacity for saving programs. The program storage capacity is twice as large as the program work area capacity. To increase the work area capacity, add a program work area as an option. The program storage capacity is also increased at the same time.
		(Option/NOP) Program work area capacity Equivalent to 40 m tape [about 16 KB] Program storage capacity Equivalent to 80 m tape [about 32 KB]				
		(Option/NOP) Program work area capacity Equivalent to 80 m tape [about 32 KB] Program storage capacity Equivalent to 160 m tape [about 64 KB]				
		(Option/NOP) Program work area capacity Equivalent to 160 m tape [about 64 KB] Program storage capacity Equivalent to 320 m tape [about 128 KB]				
		(Option/NOP) Program work area capacity Equivalent to 320 m tape [about 128 KB] Program storage capacity Equivalent to 640 m tape [about 256 KB]				

No.	Item	C12/16				Remarks
		VI	VII	VIII	IX	
37	Input/output interface	FDD interface (Standard component)				This connector enables the use of an FDD (Floppy Disk Drive). Consequently, data (e.g., program and machine variables) can be saved in ordinary DOS-format floppy disks. A floppy disk drive is optional.
		PCMCIA card drive connector (Standard component) Provided next to the color liquid crystal display (LCD): Type II × 1 slot (Note) In the operation panel: Type II × 2 slots or type III × 1 slot				You can use PCMCIA cards such as flash memory cards and LAN cards. PCMCIA cards are optional. A modem card (equipped as standard) is already mounted in a slot in the operation panel. Thus, you need to dismount the modem to use a card of type III. Note: You can mount only a flash memory card in the front slot.
		RS232 connector (Standard component)				This connector enables to use the various types of input/output devices.
38	Main spindle speed change detection function	(Standard function)				This function stops the machine when it detects the actual spindle speed exceeds the specified spindle speed by the preset variation rate. The function is useful for preventing overload.
39	Back spindle speed change detection function	(Standard function)				This function stops the machine when it detects the actual spindle speed exceeds the specified spindle speed by the preset variation rate. The function is useful for preventing overload.
40	Main spindle indexing function	15° (Standard function) 1° (Option/NOP)				This function indexes the spindle at 15° intervals and positions the spindle by the holding force of the spindle motor without using any mechanical lock. Indexing the spindle at 1° interval is optional.
	Back spindle indexing function	1° (option/NOP)				
41	Main spindle C axis function	0.001° (Option/NOP)				This function controls the positioning of the spindle at an arbitrary angle while using the spindle motor (for driving the spindle) as the C axis control servo motor. The function positions the spindle by the holding force of the spindle motor without using any mechanical lock.
	Back spindle C axis function	0.001° (Option/NOP)				
42	Constant surface speed control function					This function automatically controls the spindle speed for the tool position so that the workpiece surface speed becomes constant during the cutting process.
	Main spindle	(Standard function)				
	Back spindle	(Option/NOP)				
43	Chasing function					This function enables feed per rotation (mm/rev) and thread cutting with a tool.
	Main spindle	(Standard function)				
	Back spindle	(Option/NOP)				
44	Simplified cut-off tool breakage detection function	(Standard function)				This function gives a speed command to the main spindle when the back spindle finishes workpiece pick-off. The function checks if the back spindle rotates together with the main spindle and determines if the cut-off tool is broken.
45	Corner chamfering/rounding function	(Standard function)				This function simplifies the specification of corner chamfering and corner rounding by using the "C" and "R" command.

No.	Item	C12/16				Remarks
		VI	VII	VIII	IX	
46	Nose R compensation function	(Standard function)				This function makes compensation for the radius of a tool nose by using the G code command. To use this function, store the tool nose radius of each tool in the memory in the same manner as for the tool offsets.
47	Arc radius specification	(Standard function)				This function simplifies arc machining by using the "R" (radius) command.
48	Thread cutting canned cycle	(Standard function)				
49	Tool spindle synchronized tapping	(Option/NOP)				Synchronized tapping can be performed with the tool spindle.
50	Main spindle synchronized tapping	(Option/NOP)				Synchronized tapping can be performed with the main spindle.
51	Back spindle synchronized tapping	(Option/NOP)				Synchronized tapping can be performed with the back spindle
52	Spindle synchronization control function	(Option/NOP)				This function synchronizes the back spindle with the main spindle. The function is useful for pick-off a workpiece.
53	User macro	(Option/NOP)				User macros enable the use of macro programs.
54	Multiple repetitive cycle for turning	(Standard function)				This function enables the use of several types of canned cycles.
55	Canned drilling cycle	(Option/NOP)				This function enables the use of deep hole drilling cycles and boring cycles as canned cycles.
56	Differential rotary tool function	(Option/NOP)				This function enables drilling and tapping by using the difference between two spindle speeds. The function contributes to the decrease of cycle time.
57	Milling interpolation function	(Option/NOP)				This function performs contour control toward the end face of a workpiece by using a linear axis and rotary axis (C axis).
58	Submicron command	(Option)				This command specifies the least input increment with 0.0001 mm.
59	Tool life management I	(Option/NOP)				This function stops the machine when a tool reaches the end of its useful life and indicates its tool number.
60	Tool life management II	(Option/NOP)				This function automatically selects a spare tool when a tool reaches the end of its useful life.
61	Helical interpolation function	(Option)				This function enables the helical interpolation process by gang tool spindle device(end-face rotary tool).
62	Inclined helical interpolation function	(Option)				This function enables inclined helical interpolation process by gang tool spindle device (swing type end-face rotary tool).

Note: NOP: Network Option

3.3.3 Accessory devices

No.	Item	C12/16				Remarks
		VI	VII	VIII	IX	
1	Main spindle chucking device	C12: B12 U910Z (FC096-M) [B12 U9110Z (TF16, 40.012, 76-1076)] C16: M216 U9120Z (FC261-M) [(TF20, 40.004, 76-87)] (Standard component)				Collet chucking device provided with the main spindle.
2	Chuck sleeve for non-conform materials	C12: B12 U610Z (FC907-M) (FC908-M) C16: M216 U6120Z (FC966-M) (FC967-M) (Option)				Main spindle chuck sleeve for machining non-conform materials such as square and hexagonal materials.
3	Rotary guide bushing drive unit	C1216 U40Z (Standard component)				This drive unit synchronizes electrically the synchronous rotary guide bushing with the main spindle. A spindle motor for only driving the unit is used.
4	Fixed guide bushing device	C12: C12 U110Z (WFG541-M) C12 U160Z (WFG551-M 32° taper) [C12 U1110Z (SD125R-16, 208, 166.001, B212A)] C16: C16 U1660Z (WFG660-M) [C16 U1120Z (0201, 61.002, B238)] (Option)				This device supports the guide bushing as stationary. The device is useful for high-precision machining (e.g., materials of relatively small diameters).
5	Synchronous rotary guide bushing device	C12: C12 U210Z (WFG541-M) C12 U260Z(WFG551-M 32° taper) [C12 U2110Z (SD125R-16, 208, 166.001, B212A)] C16: C16 U2660Z (WFG660-M) [C16 U2120Z (0201, 61.002, B238)] (Option)				This device supports the guide bushing while the guide bushing and spindle are being rotated synchronously by the rotary guide bushing device (U40Z). A high-precision high-speed angular ball bearing is used with the device for performing turning precisely.
6	Rotary tool spindle drive unit of the gang tool post	C1216 U31B (Standard component)	C1216 U30B (Standard component)			This drive unit drives a tool spindle, mounted on the gang tool post, which performs drilling or key grooving on the outer circumference of a workpiece. The unit can drive four tool spindles.
7	Rotary tool spindle drive unit for front/back face machining	—	C1216 U121B (Option)			Rotary tool provided in the direction of the end face. This tool enables eccentric hole drilling into the end face.
8	Back spindle device	C12: C12 U40B C16: C16 U40B (Standard component)				This device is able to perform back machining, which is similar to front machining, on the cut-off face (back) while front machining is in progress.
9	Back spindle chucking device	C12: M212 U910B (FC096-M-K) [M212 U9110B (TF16, 40.012, 76-1076)] C16: M216 U9120B (FC261-M-K) [(TF20, 40.004, 76-87)] (Standard component)				Collet chucking device provided with the back spindle.
10	Back chucking device for non-conform materials	C12: M212 U610B (FC907-M-K) (FC908-M-K) C16: M216 U6120B (FC966-M-K) (FC967-M-K) (Option)				Back spindle chuck sleeve for machining non-conform materials such as square and hexagonal materials.
11	Air-driven knock-out device for back machining	C1216 U401B (Standard component)				Workpiece knock-out device driven by air

No.	Item	C12/16				Remarks
		VI	VII	VIII	IX	
12	Motor-driven knock-out device for back machining	C1216 U402B (Option)				Workpiece knock-out device driven by motor
13	Air unit	C1216 U60R (Standard component)				Air unit
14	Knock-out jig for workpiece through-hole	C12: C12 U550B C16: C16 U550B (Option)				This jig prevents chips from going into the back spindle of a workpiece having a through-hole. The workpiece is ejected onto the front side of the back spindle.
15	Single-bar feeder	C1216 U10J (Option)				Single-bar feeder using weight to push a material.
16	Material pipe (up to ϕ4) (up to ϕ10) (up to ϕ12) (up to ϕ16)	 M212 U24J M212 U23J M212 U22J M216 U21J (C16) (Option)				Pipes used with the U10J to feed the material.
17	Back long workpiece machining device Maximum workpiece length	C1216 U410B 400 mm [15.75"] (Option)				Use this device to machine a long workpiece that turns out to be a product longer than 90 mm [3.54"]. The device is equipped with a support pipe and unloads the workpiece from the left side of the machine.
	Pipe for the long workpiece machining device	C12: C12 U4101B C16: C16 U4101B (Option)				Component of the long workpiece machining device. Use this component with U410B.
	Cap nut for the long workpiece machining device	C12: C12 U4102B [C12 U4103B] C16: C16 U4102B (Option)				Component of the long workpiece machining device. Use this component with U410B.
	Workpiece receive shelf for the long workpiece	C1216 U420B (Option)				Component of the long workpiece machining device. Use this component with U410B.
18	Dedicated hydraulic magazine bar loader	C12: CAV12C-IS (2.5 m specifications) (3.0 m specifications) (4.0 m specifications) C16: CAV16C-IS (2.5 m specifications) (3.0 m specifications) (4.0 m specifications) (Option)				Hydraulic magazine bar loader dedicated to the C series. This loader automatically loads bars consecutively. The servo motor controls the feed axis synchronously with the Z axis on the machine side. All control can be done with the operation panel of the machine. The standard stabilizer is also provided at the rear of the main spindle.
19	Component for thin workpiece machining	C12: C12 U4021J C16: C16 U4021J (Option)				Component for thin workpiece machining that is used with the dedicated hydraulic magazine bar loader (4 m specifications)
20	Automatic magazine bar loader	(Option)				This loader automatically loads bars consecutively.
21	Workpiece separator Maximum product length	C1216 U30J 90 mm [3.54"] (Standard component)				This device collects workpieces in the product receiver box after cut-off machining is performed.
22	Workpiece separator Maximum product length	C12: C12 U301J C16: C16 U301J 35 mm [1.38"] (Option)				Component for collecting workpieces in the basket. Use this component with U30J.

No.	Item	C12/16				Remarks
		VI	VII	VIII	IX	
23	Workpiece conveyor	C1216 U31J (Option)				This conveyor sends the products, collected by the workpiece separator, to the left side of the machine.
24	Chip conveyor	C1216 U90J (Option)				This conveyor sends out chips from the machine.
25	Cut-off tool breakage detector	C1216 Y90Z (Option)				This detector checks if a workpiece is cut off after cut-off machining is completed. If a workpiece remains due to a cut-off tool break, the detector automatically stops the machine.
26	Signal lamp	C1216 U80Z (Option)				A signal lamp is mounted on the top of the machine. The lamp works in linkage with an alarm indicator on the operation panel of the machine.
27	3-color signal lamp tower	C1216 U81Z (Option)				A 3-color (green, yellow, and red) signal lamp is mounted on the top of the machine. Green indicates the machine is in continuous operation mode. Yellow indicates the machine is in the cycle-stop state. Red indicates an alarm has occurred.
28	Coolant device Pump type Cartridge-type tank capacity	C1216 U10R 250W dipping type 100 lit. (Standard component)				The coolant level detection function is provided as standard.
29	Coolant flow rate detector	C1216 U52R (Option)				This detector monitors the coolant discharge from the coolant nozzle. The detector automatically stops the machine when it detects the flow rate becoming lower than the setting value.
30	Lubrication device Lubricating oil tank capacity Discharge	(Standard component) 0.8 lit. 2.5cc/30min				This device is used for sliding. The lubrication oil level detection function is provided as standard.
31	Door switch	(Standard component)				A door switch is provided with the splash guard and spindle cover. The machine stops when the door opens. (In case of preparation mode, the machine operation speed is limited.)
32	Earth leakage breaker	Rated current: 30 A (Standard component)				Rated sensitivity current: 30 mA
33	Floppy disk drive device	L51620 U93T (Option)				A floppy disk drive enables you to input/output programs into/from the machine by using floppy disks sold at a store. The floppy disk is useful for saving many programs. It is installed separately from the machine.

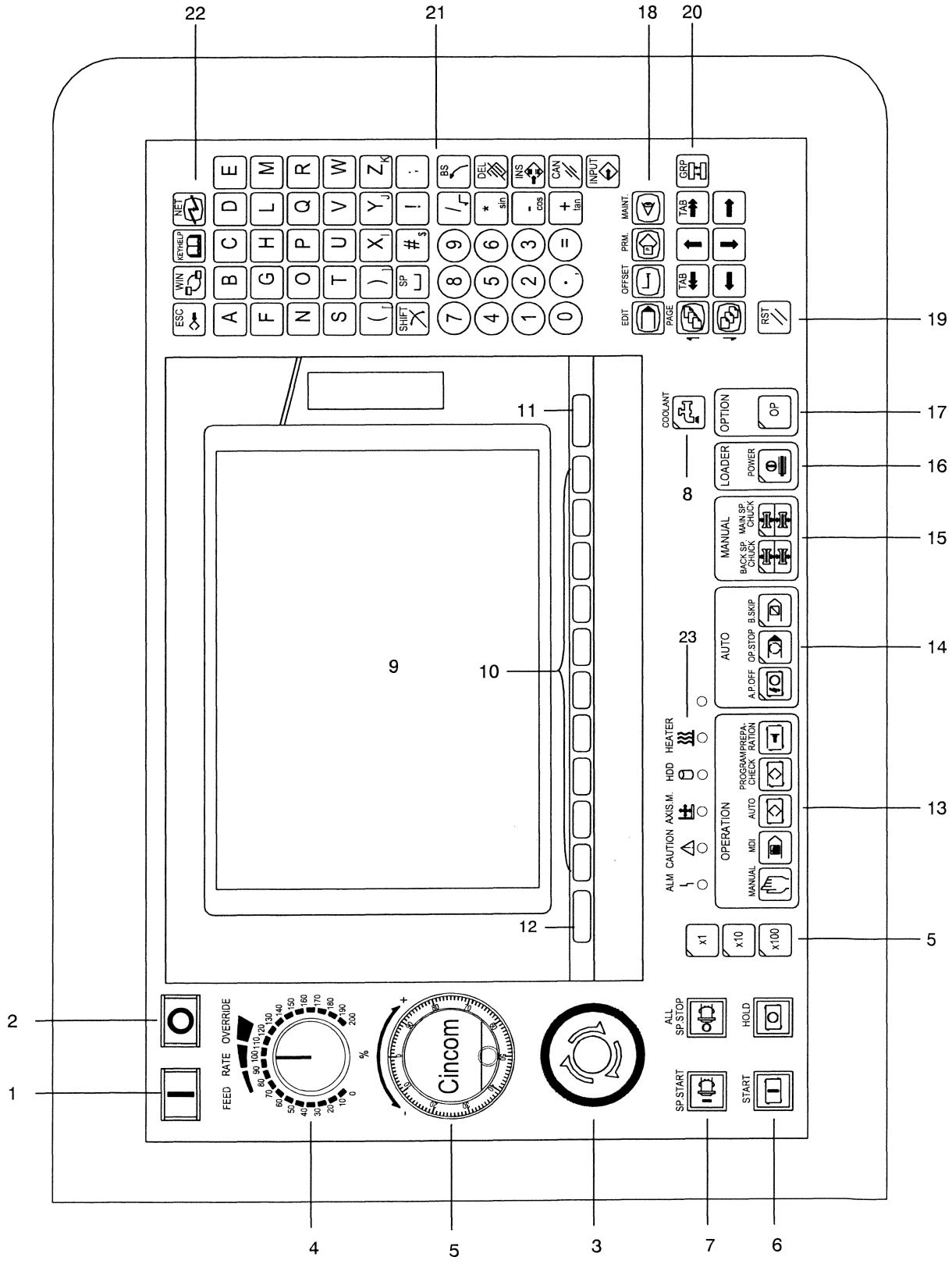
Product Code C – C 1 2 1 6 VI VII VIII IX – 7 7 0

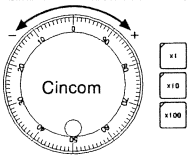
Document Code 5 E 1 – 0 3 0 3

4. Operation Panel

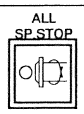
(Blank page)

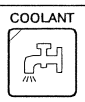
4.1 Switches, Lamps, and Keys on the Operation Panel and their Functions



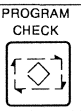
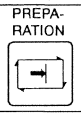
No.	Name	Symbol	Function
1	Power ON switch		This switch turns the NC power on.
2	Power OFF switch		This switch turns the NC power off.
3	Emergency Stop button		Push this button by force in case of emergency during operation. The machine enters the emergency stop state. (To release the switch, turn it clockwise (the direction indicated by the arrows).)
4	Feed Rate Override		This dial changes a feed rate in steps of 10% in the range 0 to 100% (specified feed rate). You can also set a feed rate in the range 0 to 200% on the Set SW screen.
5	Handle Handle Magnification keys		• In the manual operation mode, each axes move in the direction (+ or -) into which the handle is turned. • In the program check mode, a program runs forward when the handle is turned into the plus direction, and it runs backward when the handle is turned into the minus direction. • In the mechanical adjustment mode, the feed axes and the peripheral axes (e.g., chuck, and knock-out axes) move in the direction (+ or -) into which the handle is turned. • The X1, X10, and X100 keys select a handle feed rate in the manual operation mode, and they select a program execution rate in the program check mode.

No.	Name	Symbol	Lamp color	Function
6 Auto-matic opera-tion	START		Green	This key starts automatic operation. The lamp lights during automatic operation. The lamp flashes while the machine is in the temporary stop state during the preparation process or while it is waiting for cut-off operation. The lamp flashes at short intervals while the door is locked.
	HOLD		Red	This key holds automatic operation. The lamp lights while automatic operation is in the hold state. (The lamp also lights when the $1, $2, or $3 axis control group enters the hold state.)

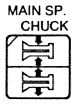
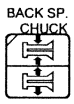
No.	Name	Symbol	Lamp color	Function
7 Manual opera-tio n mode for spindles	ALL SP.STOP (Spindle stop)		Red	Regardless of whether the manual operation mode or automatic operation mode is selected, this button stops the main spindle, back spindle, and tool spindle by the manual operation.
	SP.START (Spindle start)		Green	If you have pressed the ALL SP.STOP button to temporarily stop the spindles in the automatic operation mode or program check mode, this button rotates the spindle, back spindle, and tool spindle at the original speed when pressed. This button is enabled only when the doors are closed while the machine is in the automatic operation hold state or block stop state. The lamp lights while the spindle is rotating. The lamp flashes when the spindle is ready to restart rotating.

No.	Name	Symbol	Lamp color	Function
8 Manual opera-tio n	COOLANT		Orange	This key turns on and off coolant. The lamp lights when coolant is discharged.

No.	Name	Symbol	Function
9	LCD (Liquid Crystal Display)		The LCD screen displays information such as menu contents, coordinate values, programs, offset data, and alarm information.
10	Menu keys		Menus are displayed at the bottom of the LCD screen. A menu key selects the target screen or operation from the menu. The menu key is under the menu.
11	Menu Up/Down Selection key		The keys switch between upper and lower menus. The lower menus become effective when either key is pressed while the upper menus are effective. Similarly, the upper menus become effective when either key is pressed while the lower menus are effective.
12	Menu Up/Down Selection key		

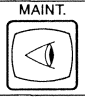
No.	Name	Symbol	Lamp color	Function
13 Opera-ti on modes	MANUAL		Orange	The lamp lights while the manual operation mode is selected. You can manually feed each axes with the handle.
	MDI		Orange	The lamp lights while the MDI mode is selected. You can create and execute programs by the manual data input method.
	AUTO		Orange	The lamp lights while the automatic operation mode is selected. You can execute programs stored in the memory.
	PROGRAM CHECK		Orange	The lamp lights while the program check mode is selected. You can check programs while operating the machine.
	PREPARATION		Orange	The lamp lights while the preparation mode is selected. You can prepare for operation.

No.	Name	Symbol	Lamp color	Function
14 Auto-ma tic opera-tio n	A.P.OFF (Automatic power off)		Orange	The automatic power-off function is enabled while the lamp lights. If an alarm occurs during machine operation in the continuous cycle mode, the function automatically turns the power off.
	OP.STOP (Optional stop)		Orange	The optional stop function is enabled while the lamp lights. The function automatically stops operation after the block specified with "M01" in the program has been executed. The  lamp flashes while the program is in the stopped state. To restart operation, press the START key again. The operation automatically restarts.
	B.SKIP (Block skip)		Orange	The block skip function is enabled while the lamp lights. The function disables the commands in the block with which a slash (/) is written.

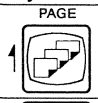
No.	Name	Symbol	Lamp color	Function
15 Manual opera-tio n	MAIN SP.CHUCK		Orange	This key opens and closes the main spindle chuck. The lamp lights when the main spindle chuck "closes". The main spindle chuck repeats opening/closing each time the key is pressed. The key functions is enabled in only the manual operation mode and preparation mode.
	BACK SP.CHUCK		Orange	This key opens and closes the back spindle chuck. The lamp lights when the back spindle chuck "closes". The back spindle chuck repeats opening/closing each time the key is pressed. The key functions is enabled in only the manual operation mode and preparation mode.

No.	Name	Symbol	Lamp color	Function
16 Bar loader	POWER		Orange	This key enables the dedicated magazine bar loader activate. The bar loader is activated while the lamp lights.

No.	Name	Symbol	Lamp color	Function
17 Option				For special specifications

No.	Name	Symbol	Function
18 Screen opera-tio n func-tion s	EDIT		This key displays edit screen such as program creation, registration, search, insertion, and deletion. You can also enter machining data in the edit screen.
	OFFSET		This key displays offset data setting screen.
	PRM. (Parameter)		This key displays parameter setting screen.
	MAINT. (Maintenance)		This key displays interface diagnostic information, software lists, and alarm history.

No.	Name	Symbol	Function
19	RST (Reset)		This key resets NC alarms and machine alarms.

No.	Name	Symbol	Function
20	Page Up		If more than one page exists, this key displays the previous page when pressed.
	Page Down		If more than one page exists, this key displays the next page when pressed.
	Up Arrow		This key moves the cursor one line up on the LCD screen. The cursor continues moving up while the key is held down.
	Down Arrow		This key moves the cursor one line down on the LCD screen. The cursor continues moving down while the key is held down.
	Right Arrow		This key moves the cursor one character position to the right on the LCD screen. The cursor continues moving to the right while the key is held down.
	Left Arrow		This key moves the cursor one character position to the left on the LCD screen. The cursor continues moving to the left while the key is held down.
	Right TAB		This key moves the cursor to the beginning of the next word in the program or input field. The key also moves the cursor to the item on the right of the current setting item.
	Left TAB		This key moves the cursor to the beginning of the previous word in the program or input field. The key also moves the cursor to the item on the left of the current setting item.
	GRP (Group Selection)		This key selects one of groups on a window.

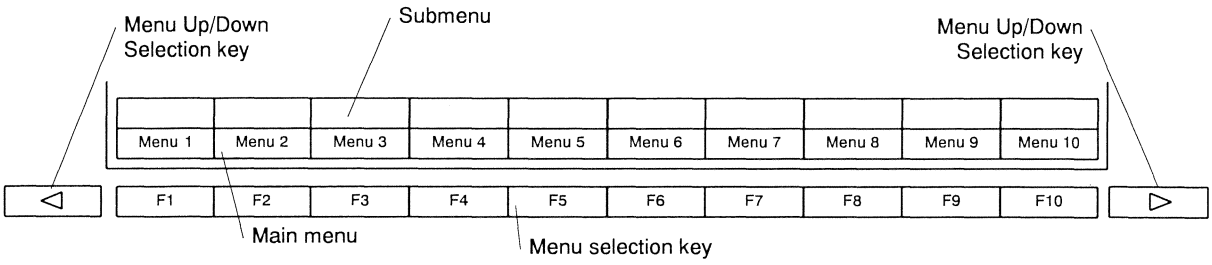
No.	Name	Symbol	Function
21	Alphanumeric keys		These keys enter alphabetic characters, numeric characters, and symbols.
	SP (Space)		This key enters a space (1-character space) each time it is pressed.
	Semicolon (End of block)		A semicolon is the symbol indicating the end of a block. This key enters a semicolon at the end of each block in a program.
	SHIFT		This key enables the input of alphabetic characters, symbols, and other characters printed on the lower portion of the key caps. Press the SHIFT key, then press the target alphanumeric keys.
	CAN (Cancel)		This key deletes 1-block data in a program. The key also deletes one setting data item.
	INPUT		This key registers an edited program in the memory. The key also sets edited setting data.
	INS (Insert)		This key toggles the key entry mode between the insert and overwrite modes.
	DEL (Delete)		This key deletes program data and setting data. The key deletes data following the cursor.
	BS (Backspace)		This key deletes program data and setting data. The key deletes data preceding the cursor.

No.	Name	Symbol	Function
22	ESC (Escape)		This key cancels setting and returns the value to the original setting value. If a window is displayed on the screen, the key closes the window. If more than one window is displayed, the key closes the windows one after another starting with the top window.
	WIN (Window)		This key selects the focus of multi-windows.
	KEY HELP		This key displays the explanations of keys of the operation panel on the LCD.
	NET (Alkart Net)		This key displays the screen for accessing the Alkart Net.

No.	Name	Symbol	Lamp color	Function
23	ALM lamp (Alarm lamp)	ALM 	Red	This lamp lights or flashes when an alarm occurs. When the lamp lights, turn off the power, eliminate the cause of the alarm, then turn on the power. When the lamp flashes, eliminate the cause of the alarm, then press RST  to return the machine to the operation. If the lamp still flashes, turn off the power, eliminate the cause of the alarm, then turn on the power.
	CAUTION lamp	CAUTION 	Yellow	This lamp flashes when an error occurs. Eliminate the cause of the error, then return the machine to the operation. The lamp also flashes while the zero point return is active.
	AXIS.M.lamp (Axis movement lamp)	AXIS.M. 	Green	This lamp lights while an axis is moving. The lamp flashes while a program is running backward in the program check mode.
	HDD lamp	HDD 	Orange	This lamp lights or flashes while the HDD (Hard Disk Drive) is operating. Do not turn off the main breaker while the lamp lights or flashes. The HDD may fail if the main breaker is turned off.
	HEATER lamp	HEATER 	Yellow	This lamp lights while the heater for the hard disk is working. The lamp lights if the hard disk is too cold when the power is turned on. The heater warms the hard disk until the temperature becomes high enough for operation. The system does not start while the lamp lights. The systems start when the lamp goes off.

4.2 Menu Keys

4.2.1 Operating the menu keys



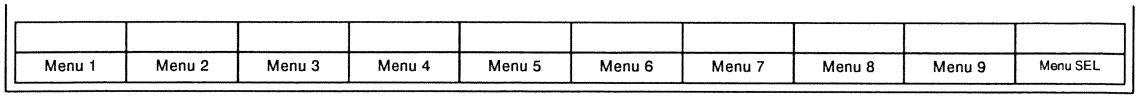
When main menus are displayed:

The menu display field consists of two rows at the bottom of the display screen. The lower row displays main menus. The upper row displays submenus. The main menus are usually displayed solid. In this state, a main menu can be selected with the menu selection key. The Menu Up/Down Selection keys are provided left and right sides of the menu selection keys. The submenus are displayed solid when either key is pressed. In this state, a submenu can be selected with the menu selection key. Use the Menu Up/Down Selection keys to switch between main menus and submenus.

Press the menu selection key under the target menu. The screen, operation, and another screen group will be selected according to the menu.
Each menu is in shape like a button. While a menu is selected, it is debossed as if it were pressed down. While it is not selected, it is embossed as if it were raised in relief.

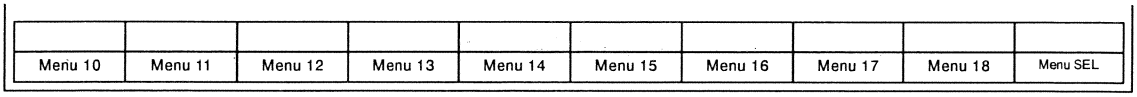
4.2.2 Selecting menus

Some screens have dozens of menus.
However, a screen can display up to ten menus in one row. A menu key called [Menu SEL] is provided on a screen having ten menus or more. All the menus can be displayed sequentially in pages by pressing [Menu SEL].



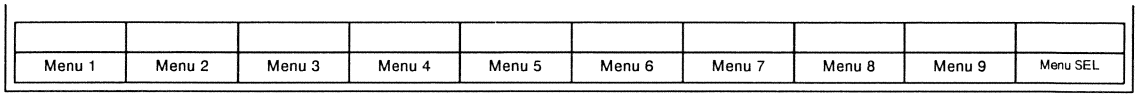
Press the menu key [Menu SEL]. The menus in the next page will be displayed.

Menus in the next page



Press the menu key [Menu SEL] again. If there are no more menus, the menus in the previous page will be returned.

Menus in the previous page



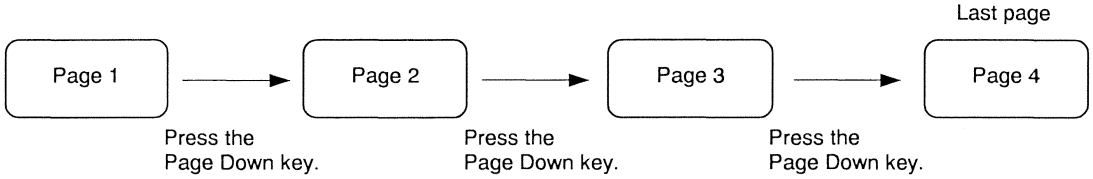
4.3 Page Keys

If a screen has more than one page, press  and 

to feed pages.

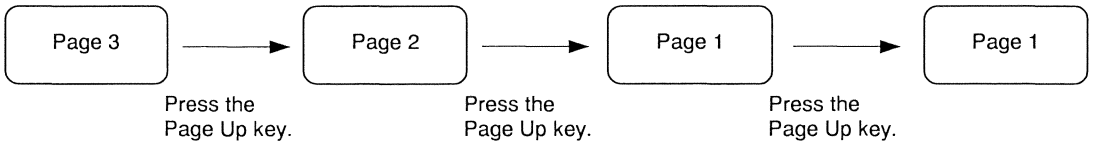
 key

Press the Page Down key to display the next page. While the last page is displayed, pressing this key does not change the page. The last page remains displayed.



key

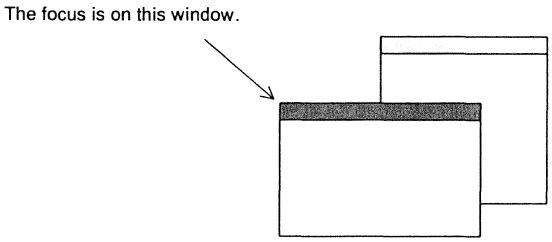
Press the Page Up key to display the previous page. While the first page is displayed, pressing this key does not change the page. The first page remains displayed.



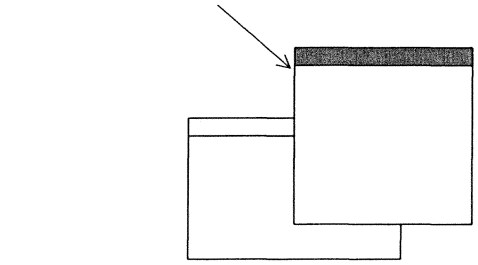
4.4 WIN Key

If more than one window opens on the screen, press WIN to select the focus of window.

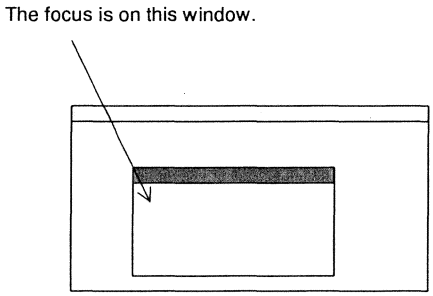
When switching the focus between windows:



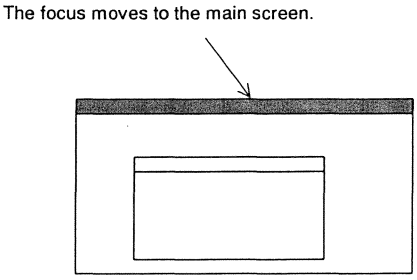
Press WIN.



When switching the focus between a window and the main screen:



Press the WIN key.

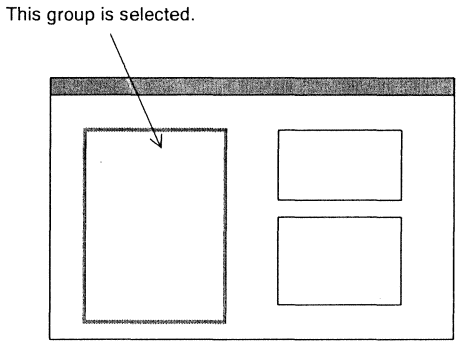


Note

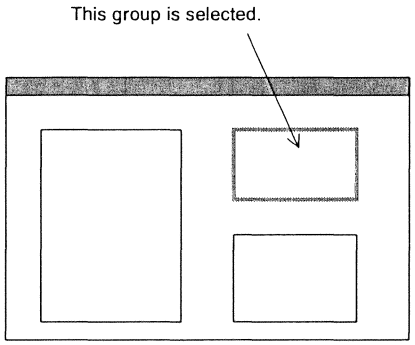
You have to move the focus to the screen on which you are going to select an item or to which you are going to input data.

4.5 Selecting a Group

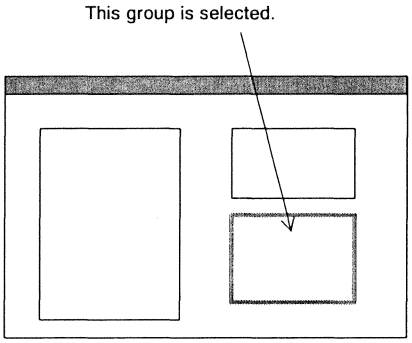
If more then one input or selected group is displayed on the screen, press GRP to select a group.



Press GRP.



Press the GRP key.



Press the Up, Down, Right, and/or Left Arrow keys and the Right and/or Left Tab keys to select an item in the group.

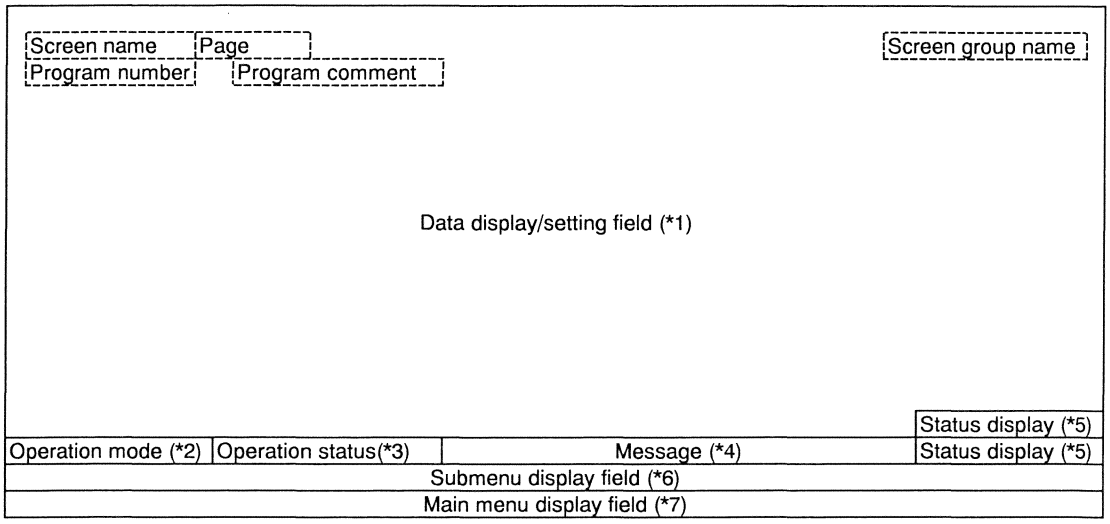
Note

First you have to move the focus to the screen on which you are going to select a group.

4.6 Fields on the Liquid Crystal Display (LCD) and their Functions

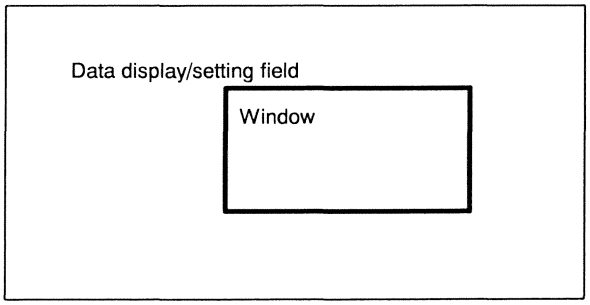
The LCD screen has the following seven fields:

- Data display/setting field (*1)
- Operation mode display field (*2)
- Operation status display field (*3)
- Message display field (*4)
- Status display field (*5)
- Submenu display field (*6)
- Main menu display field (*7)



Data display/setting field

The data display/setting field displays data on the entire screen, or it may also display another data display field (as a window) over its own field. You can input data directly into the data display/setting field.



Operation mode display field

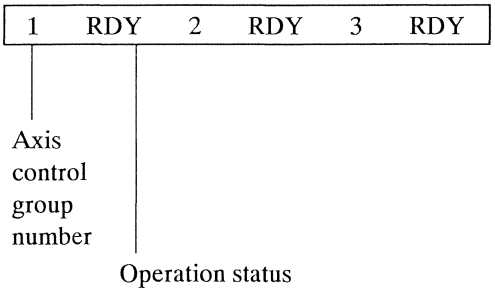
The operation mode display field displays the currently selected operation mode.

Display in the operation mode display field

Symbol	Description
MEM	Memory operation mode
HND	Handle feed mode
MDI	MDI mode
REF	Zero point return mode
JOG	Jog mode

Operation status display field

The operation status display field displays the NC operation status for each axis control group number.



Display in the operation status display field

Symbol	Description
EMG	Emergency stop state
RST	NC reset state
RDY	Ready for operation
(No symbol)	During automatic operation
SYN	Synchronization queuing state
CRS	Axis cross queuing state
STP	Stop state
HLD	Hold state

Relation between operation status display colors and the machine status

Display color	Machine status
Green	Normal
Red	Abnormal

Message display field

The message display field displays alarm messages, machine operation status messages, and operation guidance.

Status display field

The status display field displays the machine operation status.

Display in the status display field

Symbol	Description
OVR/INS	Input state in the edit mode
ABS INPT/INC INPT	Input state in the offset mode
Set COMP/Editing.../Executing...	MDI state

Submenu display field

The submenu display field displays submenus for selecting screens and operation.
This field is usually displayed dimly. Press the Menu Up/Down Selection key. The field will be displayed solid. In this state, you can select a submenu.
While the field is displayed solid, press the menu selection key corresponding to the target submenu. The submenu will be selected.

Main menu display field

The menu display field displays menus for selecting screens and operation.
This field is usually displayed solid.
While the field is displayed solid, press the menu selection key corresponding to the target menu. The menu will be selected.
While the submenu display field is displayed solid, the menu display field is displayed dimly. In this state, you cannot select any menu.
To select a menu, press the Menu Up/Down Selection key to display the menu display field solid.

4.7 Inputting and Editing Data

4.7.1 Moving the cursor

Procedure

Press ↑ ↓ TAB ← TAB →, and/or GRP to move the cursor to the target input field, then press ↑ ↓ → , and/or ← to move the cursor to the target input position.

4.7.2 Inputting data

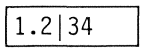
Some data input screens show initial values in input fields when they are displayed.

Insert

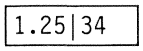
Insert data to the left of the cursor position.
The insert mode is initially set for data input.

Procedure

1. Move the cursor to the insertion position.

2. Press the alphanumeric key to input additional text. (Input 5 in this example.)

3. Press INPUT to set the input value.

   The input value becomes invalid if you press ↑ ↓ TAB ← TAB → GRP or press ESC to erase the screen, or switch the screen before pressing the INPUT key.

Overwrite

The character at the cursor position is replaced with an input character in the overwrite mode.
You can use the overwrite mode only with the MDI screen and program editing screen.

Procedure

1. Move the cursor to the left of the character over which you want to write a new character.

   AB|C

2. Press INS to activate the overwrite mode.
   Confirm that the overwrite mode is indicated at the lower right of the screen.
   The insert and overwrite modes are switched back and forth (e.g., overwrite → insert → overwrite) each time the INS key is pressed. Each mode is enabled until the INS key is pressed.

3. Press the alphanumeric character to enter text. (Input D in this example.)

   ABD|

4. Press INPUT to set the input value.
   The input value becomes invalid if you press ↑ ↓ TAB TAB GRP or press ESC to erase the screen, or switch the screen before pressing the INPUT key.

4.7.3 Deleting data

Deleting the character following the cursor

Delete the character that follows the cursor (that is to the right of the cursor position).

Procedure

1. Move the cursor to the left of the character that you want to delete.

   1.2|34

2. Press DEL.

   The character is deleted, and the characters to the right of the deleted character are shifted one position to the left.

   1.2|4

3. Press INPUT to set the input value.

   The input value becomes invalid if you press ↑ ↓ TAB TAB GRP or press ESC to erase the screen, or switch the screen before pressing the INPUT key.

Deleting the character preceding the cursor

Delete the character that precedes the cursor (that is to the left of the cursor position).

Procedure

1. Move the cursor to the right of the character that you want to delete.

   1.2|34

2. Press BS.

   The character is deleted, and the characters to the right of the deleted character are shifted one position to the left.

   1.34

3. Press INPUT to set the input value.

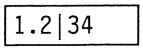
   The input value becomes invalid if you press ↑ ↓ TAB TAB GRP or press ESC to erase the screen, or switch the screen before pressing the INPUT key.

Deleting data in lines

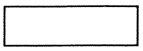
Delete all data on the line on which the cursor is located or all data in the input field in which the cursor is located.

Procedure

1. Move the cursor to the line or input field that you want to delete.

2. Press CAN.
   All the data on the line or in the input field is deleted.

3. Press INPUT to set the deletion.
   The input value becomes invalid if you press ↑ ↓ TAB TAB GRP or press ESC to erase the screen, or switch the screen before pressing the INPUT key.

Deleting data in screens

Delete all data on the displayed screen.

Procedure

1. Display the screen from which you want to delete all data.

2. Press SHIFT, and then CAN.
   All the data on the screen is deleted.

3. Press INPUT to set the deletion.
   The input value becomes invalid if you press ↑ ↓ TAB TAB GRP or press ESC to erase the screen, or switch the screen before pressing the INPUT key.

Product Code C – C 1 2 1 6 VI VII VIII IX – 7 7 0

Document Code 5 E 1 – 0 4 0 2

5. Programming

(Blank page)

5.1 NC Program Structure

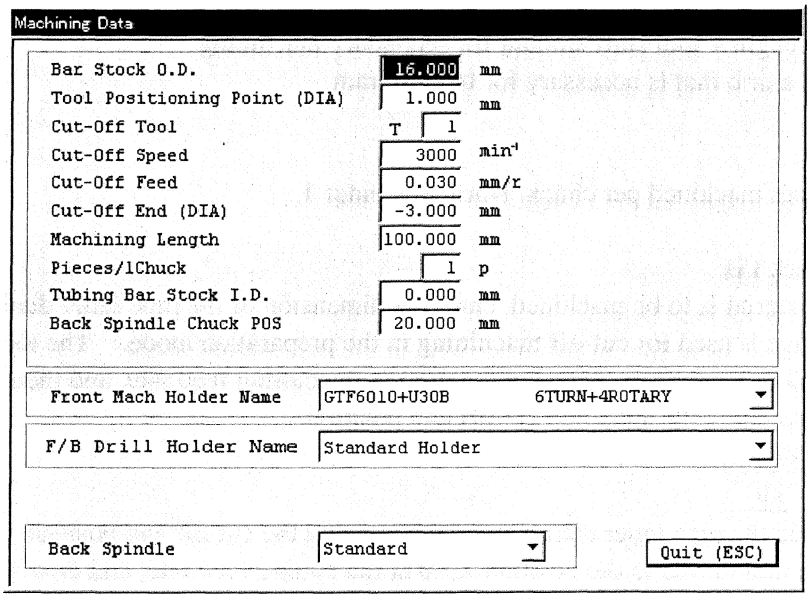
A program consists of a program body and program data.
A program body is a list of machine operation commands in the sequence of machining processes. In general, it is called a program.
Program data is a list of prerequisites for operating the program.
Program data is necessary for machine operation, and it is also useful as a means of operational communication between operators.

- Program
  - Program body ..... Specification of machine operation commands
  - Machining data ..... Settings of requirement for machining
- Tool set data ..... Compensation for errors of mounted tool center, diameter, and longitudinal position

5.1.1 Program number (O number)

To distinguish between programs, a program number is assigned to each program (program body and machining data). The numbers 8000 to 8999 are used with user's custom programs, and the numbers 9000 to 9999 are used with machine manufacturer's custom programs.

5.1.2 Explanation of machining data



(This screen is for type VII, VIII, IX.)

1 Bar Stock O.D.
Enter the dimension of the outer diameter of the material to be machined.

2 Tool Positioning Point (DIA)
Enter a clearance from the material outer diameter at the position to which a tool [T01 to T06 or T11 to T14 (Type VI: T11 to T13)] is moved when it is selected. When another tool is selected, the currently selected tool is moved to this position. In other words, a tool [T01 to T06 or T11 to T14 (Type VI: T11 to T13)] is moved to the tool positioning point when it is selected, and the tool is moved to the tool positioning point when another tool is selected.

3 Cut-Off Tool
T01 is automatically entered.

4 Cut-Off Speed
The tip of a material is cut at the spindle speed in the Preparation mode.

5 Cut-Off Feed
The tip of a material is cut at the feed rate in the Preparation mode.

6 Cut-Off End (DIA)
The X axis is at the end position where cutting the tip of a material is completed in the Preparation mode. The end position is also the start position on the X axis from which the program starts.

7 Machining Length
Enter the maximum move distance of the main spindle required for machining a workpiece.
a. Workpiece length + cut-off tool width or rear turning tool width
b. Workpiece length + tool shift amount for secondary machining
Enter the value a or b that is necessary for the program.

8 Pieces/1Chuck
Number of pieces machined per chuck. Normally, enter 1.

9 Tubing Bar Stock I.D.
When a pipe material is to be machined, enter the dimension of the pipe inner diameter.
The entered value is used for cut-off machining in the preparation mode. The tool moves to the position defined by (pipe inner diameter – a mm) at the cutting feed rate, and then it moves at the rapid feed rate until reaching the specified cut-off end position.

	C12/C16
a	3.0

Example: When the pipe inner diameter is 10.0 mm and the cut-off end position is X–3.0:
The tool moves to the position X7.0 at the cutting feed rate, and then it moves at the rapid feed rate until reaching the end position X–3.0. Be sure to specify the cut-off end position with a negative value.

10 Back Spindle Chuck POS
Enter the value of workpiece protruding from end face of the back spindle cap nut.

11 Front Mach Holder Name
Select the name of the front machining holder mounted on the machine.

12 F/B Drill Holder Name (Type VII, VIII, IX)
Select the name of the front/back drill holder mounted on the machine.

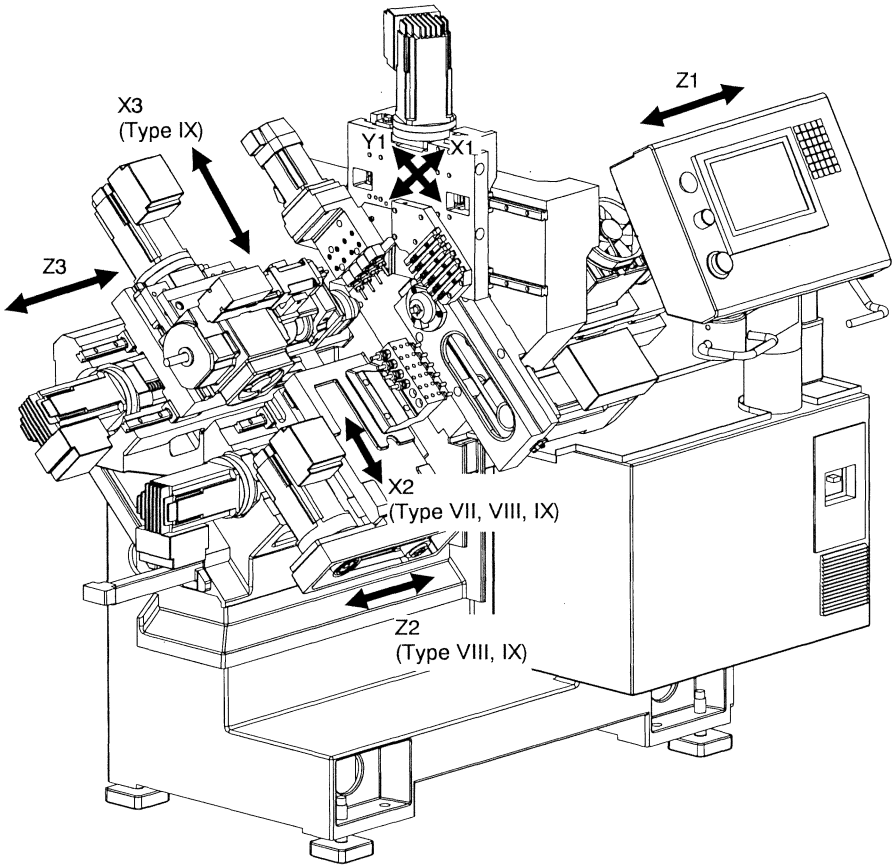
13 Back Spindle
Select the type of the back spindle mounted on the machine. Select either "Standard", "Basket" or "Stabilizer".

CAUTION

Be sure to specify the cut-off end position with a negative value. If the cut-off end position is specified with a positive value, the gang tool may interfere with the bar stock.

5.2 Drive Axes and Multi-axis Control

5.2.1 Drive axes



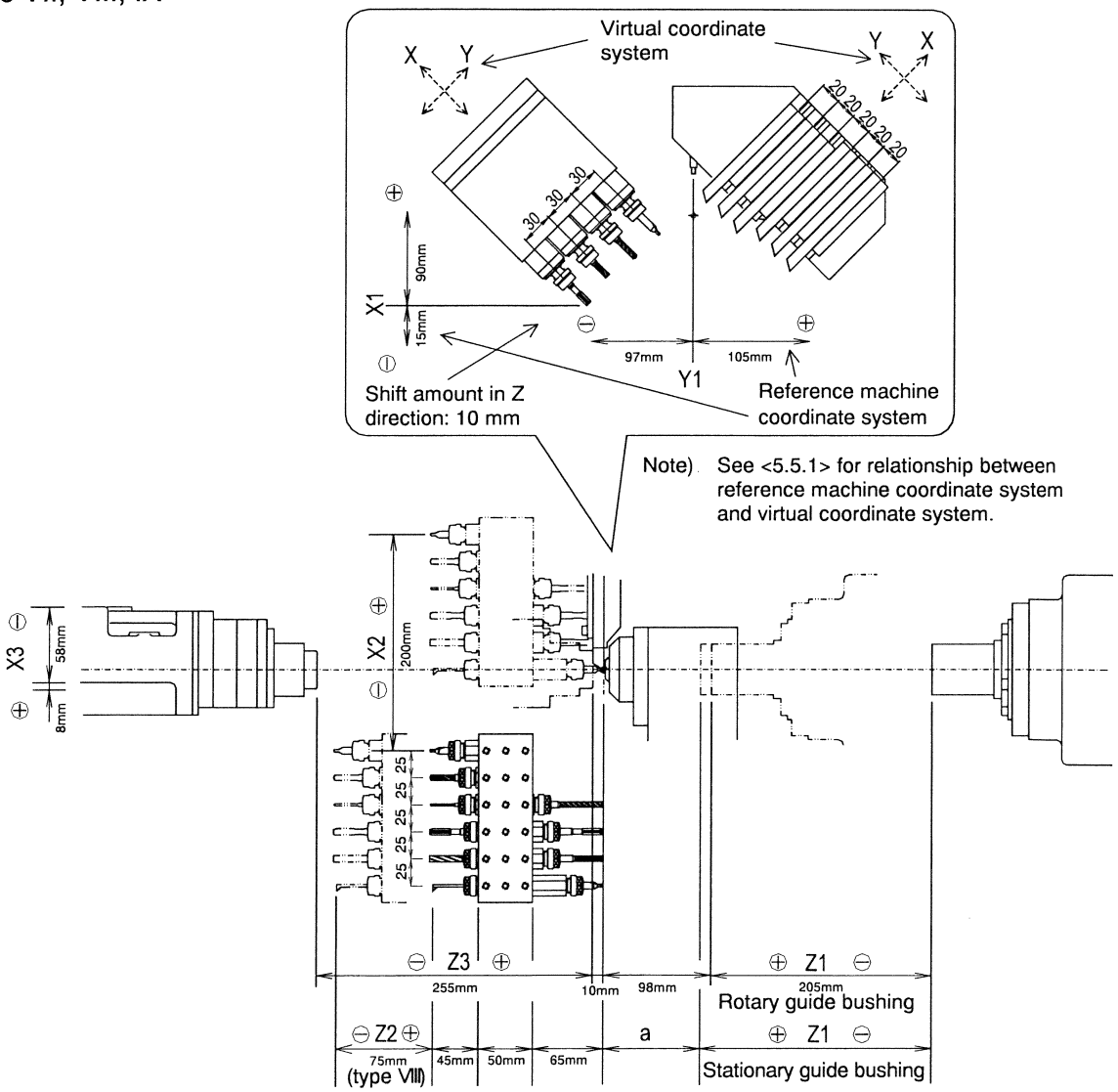
Axis	Name
X1	Gang tool post
Z1	Headstock
Y1	Gang tool post
C1	Main spindle C axis
X2 (Type VII, VIII, IX)	Opposite tool post
Z2 (Type VIII, IX)	Opposite tool post
C2	Back spindle C axis
X3 (Type IX)	Back headstock
Z3	Back headstock

S1 = Main spindle
S2 = Back spindle
S3 = Tool spindle of the gang tool post
(S4 = Tool spindle for front/back face machining) option: Type VII, VIII, IX
S5 = Rotary guide bushing

5.2.2 Coordinate values and signs

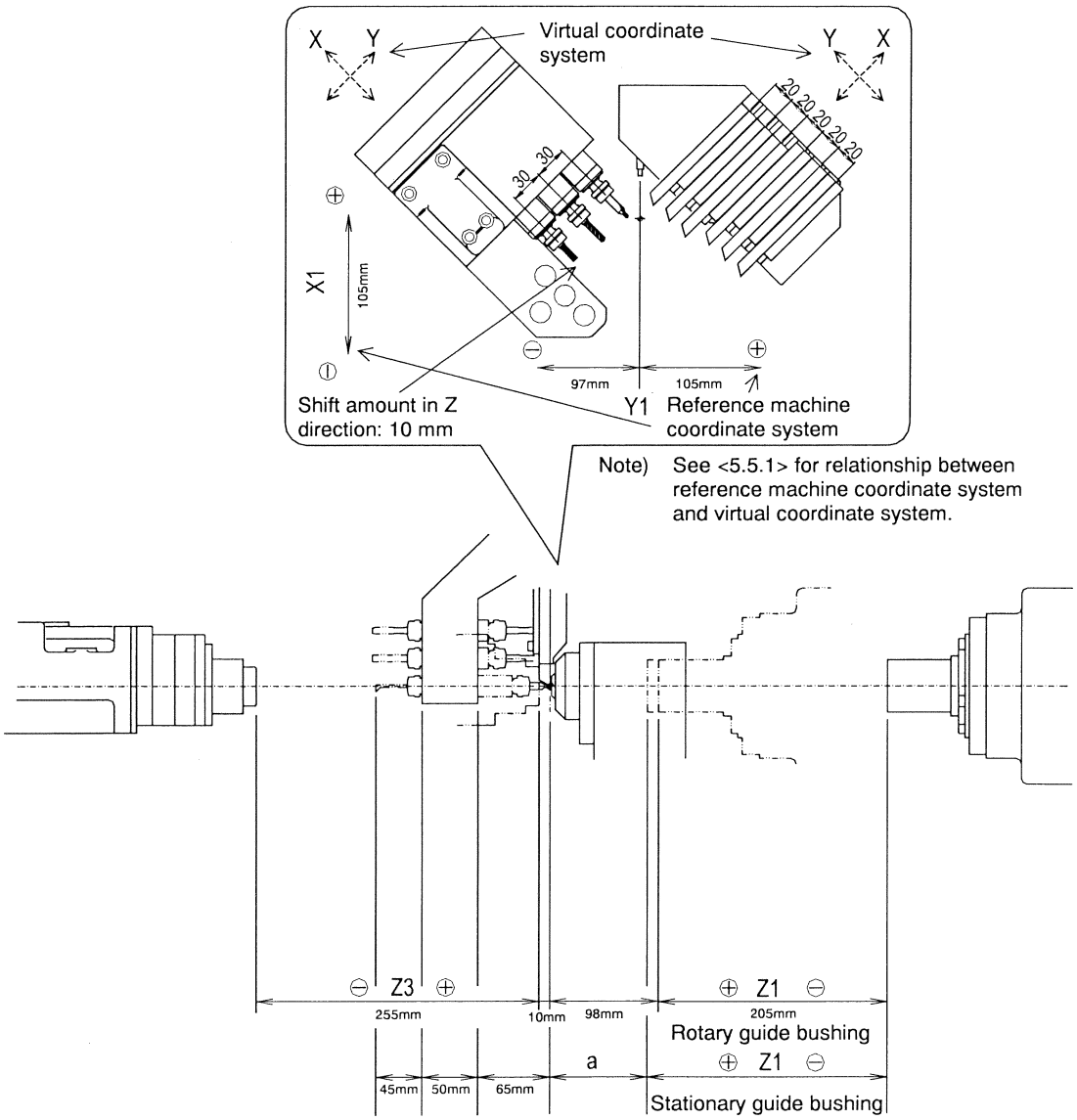
The figure below shows the signs used for operation.
For the X1, X2, X3, and Y1 axes, specify a coordinate value with the diameter that is equivalent to the distance from the center of the material. For the Z1 and Z3 axes, specify a coordinate value with the distance.

Type VII, VIII, IX



	C12	C16
X1 stroke	105.0	105.0
Z1 stroke (stationary guide bushing)	225.0	215.0
Z1 stroke (rotary guide bushing)	205.0	205.0
Y1 stroke	202.0	202.0
X2 stroke	200.0	200.0
Z2 stroke (Type VIII)	75.0	75.0
X3 stroke (Type IX)	66.0	66.0
Z3 stroke	255.0	255.0
a	78.0	88.0

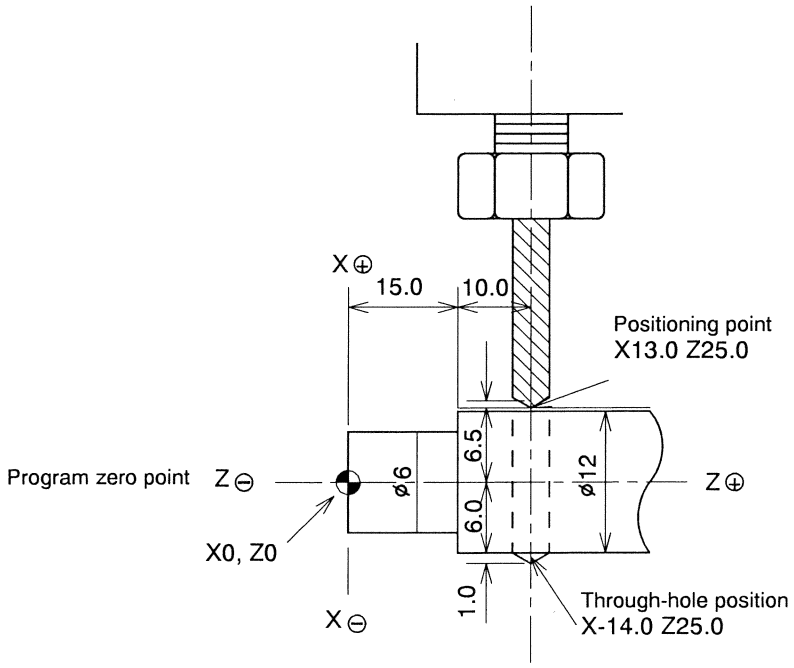
Type VI



	C12	C16
X1 stroke	105.0	105.0
Z1 stroke (stationary guide bushing)	225.0	215.0
Z1 stroke (rotary guide bushing)	205.0	205.0
Y1 stroke	202.0	202.0
Z3 stroke	255.0	255.0
a	78.0	88.0

5.2.3 Coordinate values and signs for secondary machining

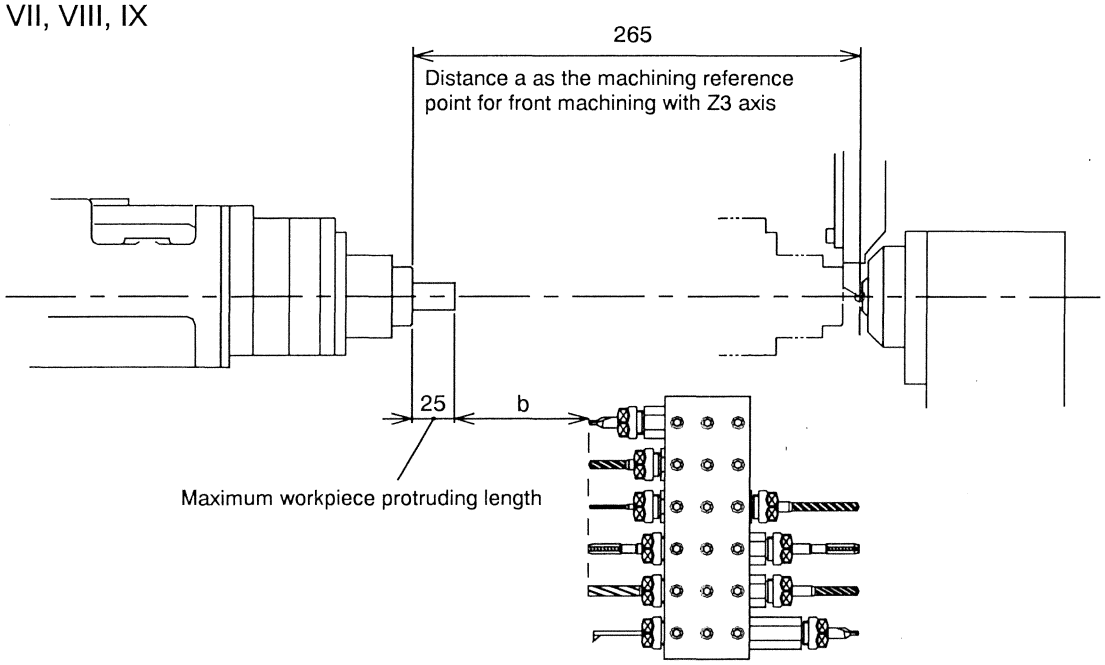
The same concept as for turning applies to coordinate values and signs for secondary machining. Be sure to specify an X coordinate value with the diameter. As for signs, specify X coordinates as X⊕ on the side on which the tool exists, and specify Z coordinates as Z⊕ on the side on which the workpiece exists.



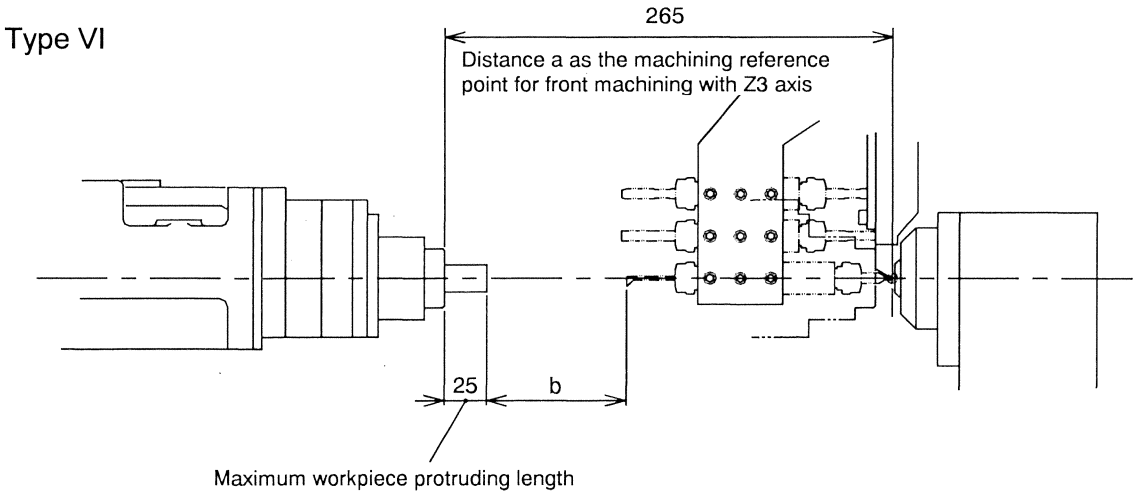
5.2.4 Coordinate values and signs for back machining

The figure following shows the positional relation between the back spindle, the back machining tools, and the workpiece chucked by the back chuck.

Type VII, VIII, IX

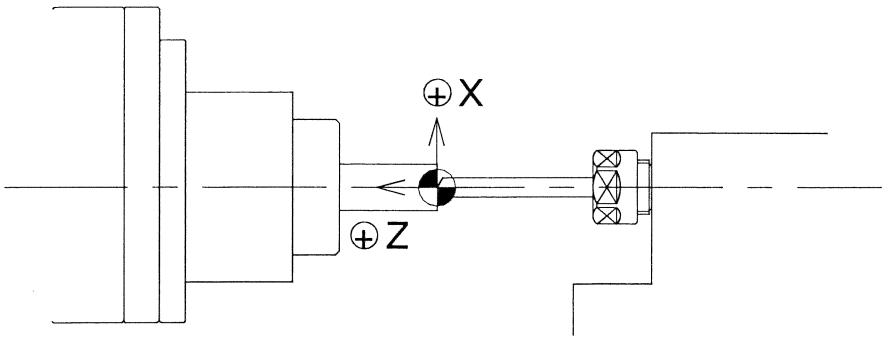


Type VI



	C12/C16 type VI VII	C12/C16 type VIII IX
A Maximum workpiece protruding length	25.0	25.0
a	265.0	265.0
b	80.0	5.0

Specify the coordinate values in the program assuming that the workpiece is fixed.



Notes

- The maximum workpiece protrusion from the end face of the back spindle is 25 mm (A in the table above). If the workpiece protrusion is longer than 25 mm, the workpiece interferes with another tool during tool selection, or the workpiece interferes with the workpiece separator.
- For the "Back Spindle Chuck POS" in the machining data, enter the workpiece protrusion from the back spindle. (Workpiece protrusion = entire workpiece length – back chuck position specified in the program)
  Entering no values makes the distance between the end face of the back spindle and the back drilling tool zero, causing the protruded workpiece to interfere with a tool for back turning.

5.3 Machining Patterns

To simplify a program used with a multi-axis machine, operations (machining) are grouped by purpose. An operational (machining) group is called a machining pattern. The following eight machining patterns are available: The machining pattern commands should be specified for all the three axis control groups.

- Machining pattern cancel, Free machining in three axis control groups (G600)
  G600 cancels the machining patterns G610 to G670. (Power-on state)
  Use this machining pattern command to perform machining with no particular machining pattern specified.

- $1 single machining, Front/back simultaneous machining (G610): Type VII, VIII, IX
  This machining pattern command enables single machining with a front drill for front machining in $1, or front/back simultaneous machining with the front drill in $1 and a back drill for back machining in $2. In the case of type IX, X2 axis (opposite tool post) and X3 axis (back spindle) are superimposed, and enables ID machining for front and back simultaneously.

- Outer diameter/inner diameter simultaneous machining (G620): Type VIII, IX
  This machining pattern command enables the following two types of machining at the same time while superimposing the Z2 axis on the Z1 axis:
  Outer diameter machining with a gang tool for front machining in $1
  Inner diameter machining with a front drill for front machining in $2

- Front/back parallel machining (G630): Type VII, VIII, IX
  Use this machining pattern to perform front machining and back machining independently. The machining pattern command enables machining with a gang tool for front machining in $1 and machining with a back drill for back machining in $2.

- 3-line simultaneous machining (G640): Type VIII, IX
  This machining pattern command enables the following three types of machining at the same time while superimposing the Z2 axis on the Z1 axis and the Z3 axis on the Z2 axis:
  Outer diameter machining with a gang tool for front machining in $1
  Inner diameter machining with a front drill for front machining in $2
  Inner diameter machining with a back drill for back machining in $3
  In the case of type IX, X2 axis (opposite tool post) and X3 axis (back spindle) are superimposed, and enables ID machining for front and back simultaneously.

- Pick-off, Center-support (G650)
  This machining pattern enables the back spindle to pick off workpiece and support the long workpiece.

- Front/back simultaneous machining (G660): Type VI
  This machining pattern command enables front /back simultaneous machining with a vertical front drill for front machining in $1 and a vertical back drill for back machining in $2.

- Sequential machining in a single axis control group (G670)
  Specify this machining pattern command to create a program using only $1. The machining pattern command enables machining with a gang tool, a front drill, and a back drill, product collection, and pick-off by using only $1.

Axis control groups

The axes of a multi-axis machine are grouped by operational purpose. The axis groups are called axis control groups.
Create a program for each axis control group. The programs of the axis control groups are executed when the machine is started.

Superimpose control

If the superimpose control function (G650 machining pattern) is specified to an axis control group, its member axes that have been operating with different coordinates system will operate with synchronizing to the superimposed coordinate system.
For example, with G650 (Z1-Z3) superimposed, Z1 is the reference axis and Z3 is the superimposed axis. In a program, you can specify Z3 coordinate values on the Z1 coordinate for synchronous or asynchronous operation.

5.3.1 Machining patterns, axes of axis control groups, and superimpose axes

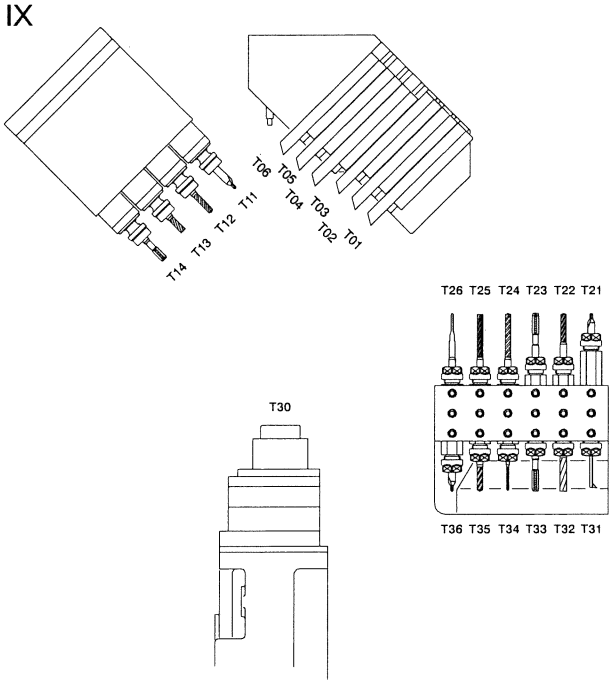
The table below assigns the selectable tool and the axis control group that corresponds to the axis you want to move in the machining patterns.

Machining pattern	Command code	Axes of axis control groups			Superimpose axis
		$1	$2	$3	
Machining pattern cancel Free machining in three axis control groups	G600	T0100 to T0600 T1100 to T1400 (Type VII, VIII, IX) T1100 to T1700 (Type VI) T2100 to T2600 (Type VII, VIII, IX) T3100 to T3600 (Type VII, VIII, IX) T3000 T5400 to T5700 (Type VI) X1, Z1, Y1, C1	T0100 to T0600 T1100 to T1400 (Type VII, VIII, IX) T1100 to T1700 (Type VI) T2100 to T2600 (Type VII, VIII, IX) T3100 to T3600 (Type VII, VIII, IX) T3000 T5400 to T5700 (Type VI) X2, Z2, C2	T0100 to T0600 T1100 to T1400 (Type VII, VIII, IX) T1100 to T1700 (Type VI) T2100 to T2600 (Type VII, VIII, IX) T3100 to T3600 (Type VII, VIII, IX) T3000 T5400 to T5700 (Type VI) X3, Z3	
$1 single machining Front/back simultaneous machining (Type VII, VIII, IX)	G610	T2100 to T2600 X2, Z1, C1	T3100 to T3600 X3, Z3, C2		X2-X3 (Type IX)
Outer diameter/inner diameter simultaneous machining (Type VIII, IX)	G620	T0100 to T0600 T1100 to T1400 X1, Z1, Y1, C1	T2100 to T2600 X2, Z2		Z1-Z2
Front/back parallel machining (Type VII, VIII, IX)	G630	T0100 to T0600 T1100 to T1400 X1, Z1, Y1, C1	T3100 to T3600 X2, Z3, C2		
3-line simultaneous machining (Type VIII, IX)	G640	T0100 to T0600 T1100 to T1400 X1, Z1, Y1, C1	T2100 to T2600 X2, Z2	T3100 to T3600 X3, Z3, C2	X2-X3 (Type IX) Z1-Z2 Z2-Z3
Pick-off Center support	G650	T3000 X1, Z1, Y1, C1	T3000 X2, Z3, C2	T3000	Z1-Z3
Front/back simultaneous machining (Type VI)	G660	T1400 to T1700 X1, Z1, Y1, C1	T5400 to T5700 Z3, C2		
Sequential machining in a single axis control group	G670	T0100 to T0600 T1100 to T1400 (Type VII, VIII, IX) T1100 to T1700 (Type VI) T2100 to T2600 (Type VII, VIII, IX) T3100 to T3600 (Type VII, VIII, IX) T3000 T5400 to T5700 (Type VI) X1, Z1, Y1, C1	X2, Z2, C2	X3, Z3	
1-cycle stop (No machining pattern)	M2	X1, Z1, Y1, C1	X2, Z2, C2	X3, Z3	

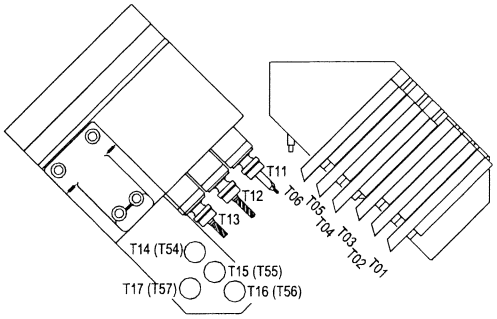
Notes

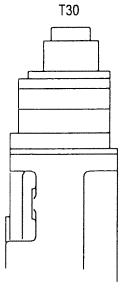
- Do not execute the machining pattern commands G600 to G670 in the MDI mode.
- The above table shows the axes of axis control groups after the machining pattern is specified. After tool selection, the axes of axis control groups that correspond to the selected tool numbers become effective.
- Type VI does not have X2, Z2 and X3 axes.
- Type VII does not have Z2 and X3 axes.
- Type VIII does not have X3 axis.

Tool post of type VII, VIII, IX



Tool post of type VI





5.3.2 Machining pattern flow

The machining pattern commands should be specified for all the three axis control groups.

Program sample

$1	$2	$3
G610 ····· $1 single machining	G610 ····· $1 single machining	G610 ····· $1 single machining
:	:	:
G630 ····· Front/back parallel machining	G630 ····· Front/back parallel machining	G630 ····· Front/back parallel machining
:	:	:
G650 ····· Pick-off	G650 ····· Pick-off	G650 ····· Pick-off
:	:	:
G600 ····· Machining pattern cancel	G600 ····· Machining pattern cancel	G600 ····· Machining pattern cancel
:	:	:
M56		
G999	G999	G999
N999	N999	N999
M02	M02	M02
M99	M99	M99
%	%	%

Note

Be sure to cancel the coordinate system shift command and compensation command before switching the machining pattern.

5.3.3 Machining pattern cancel, Free machining in three axis control groups (G600)

G600 cancels the machining patterns G610 to G670. (Power-on state) Use this machining pattern command to perform machining with no particular machining pattern specified.

Command format

$1	$2	$3
G600	G600 U0 W0	G600 U0 W0
X1 Z1 Y1 C1	X2 Z2 C2	X3 Z3

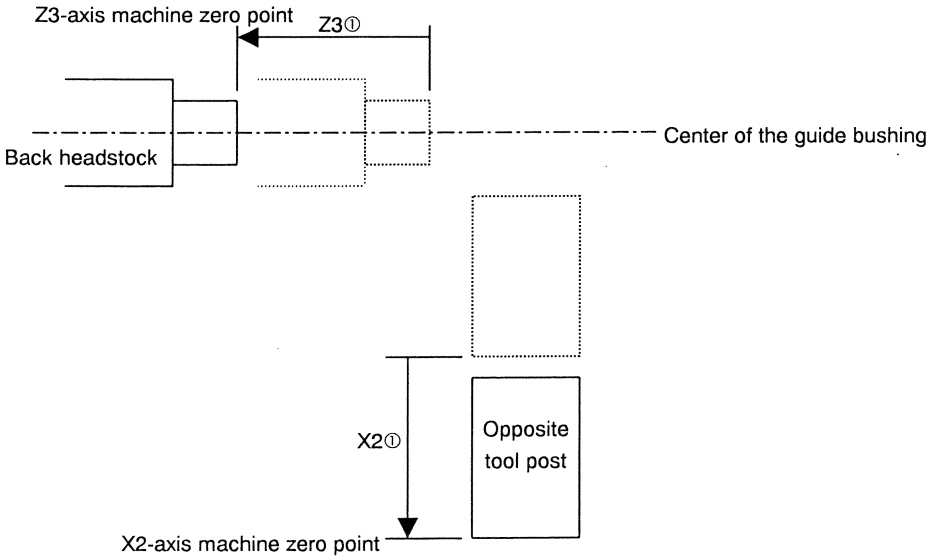
Axis control group

Specify this command for all the axis control groups $1, $2, and $3.

Arguments

- $1: The gang tool post (X1 axis) does not move.
- $1: The gang tool post (Y1 axis) does not move.
- $2 U0: The opposite tool post (X2 axis) does not move. The axis moves to the return position unless the argument is specified.
- $2 W0: The opposite tool post (Z2 axis) does not move. The axis moves to the forward end position unless the argument is specified. (Type VIII, IX)
- $3 U0: The back headstock (X3 axis) does not move. The axis moves to the return position unless the argument is specified. (Type IX)
- $3 W0: The back headstock (Z3 axis) does not move. The axis moves to the return position unless the argument is specified.

Operation sample



- The circled numbers indicate the operation sequence for each tool post.

Macro specification

Command code	G600		
Name	Machining pattern cancel, free machining in three axis control groups		
Axis control group	$1	$2	$3
Axes of axis control group	X1, Z1, Y1, C1	X2, Z2, C2	X3, Z3
Superimpose	–	–	–
Coordinate system	–	–	–
Argument	The gang tool post (X1 axis) does not move. The gang tool post (Y1 axis) does not move.	U0: The opposite tool post (X2 axis) does not move. The axis moves to the return position unless the argument is specified. W0: The opposite tool post (Z2 axis) does not move. The axis moves to the forward end position unless the argument is specified.	U0: The back headstock (X3 axis) does not move. The axis moves to the return position unless the argument is specified. W0: The back headstock (Z3 axis) does not move. The axis moves to the return position unless the argument is specified.
Spindle with which synchronous feed is enabled	Main spindle	–	–
Spindle with which constant surface speed control is enabled	Main spindle	–	–
Cutting block interlock	–	–	–
T command	Gang tool post, front/back drilling and back spindle T0100 to T0600, T1100 to T1400 (Type VII, VIII, IX) T1100 to T1700 (Type VI) T2100 to T2600, (Type VII, VIII, IX) T3100 to T3600 (Type VII, VIII, IX) T3000 T5400 to T5700 (Type VI)	Gang tool post, front/back drilling and back spindle T0100 to T0600, T1100 to T1400 (Type VII, VIII, IX) T1100 to T1700 (Type VI) T2100 to T2600, (Type VII, VIII, IX) T3100 to T3600 (Type VII, VIII, IX) T3000 T5400 to T5700 (Type VI)	Gang tool post, front/back drilling and back spindle T0100 to T0600, T1100 to T1400 (Type VII, VIII, IX) T1100 to T1700 (Type VI) T2100 to T2600, (Type VII, VIII, IX) T3100 to T3600 (Type VII, VIII, IX) T3000 T5400 to T5700 (Type VI)
Others	–	–	–

Program sample

Front/back simultaneous machining → $1 single machining → Front/back parallel machining → Pick-off

$1	$2	$3
:	:	
	G44 Back spindle feed per rotation ON	
M160 ····· S1 cutting start interlock ON	M162 S2 cutting start interlock ON	
!2L1	!1L1	
T2100 R1 (X2 Z1 C1) ···· Front/back simultaneous machining	T3100 S1 (Z3 C2) ···· Front/back simultaneous machining	
!2L2	!1L2	
G00 Z–0.5 T21	G00 Z–0.5	
:	:	
!2L3	!1L3	
T2200 R1 (X2 Z1 C1) ···· Front/back simultaneous machining	T3200 S1 (Z3 C2) ···· Front/back simultaneous machining	
!2L4	!1L4	
G00 Z–0.5 T22	G00 Z–0.5	
:	:	
!2L5	!1L5	
T2300 R1 (X2 Z1 C1) ···· $1 single machining		
G00 Z–0.5 T23		
:		
M147 ····· The opposite tool post retracts.		
!2L6	!1L6	
T0200 (X1 Z1 Y1 C1) ···· Front/back parallel machining	T3400 (X2 Z3 C2) ···· Front/back parallel machining	
G00 Z17.0 Z–0.5 T02	G00 Z–0.5 T34	
:	:	
T0300 (X1 Z1 Y1 C1) ···· Front/back parallel machining	T3500 (X2 Z3 C2) ···· Front/back parallel machining	
:	:	
	M147 ····· The opposite tool post retracts.	
	G43 ····· Back spindle feed per rotation OFF	
	M163 ····· S2 cutting start interlock OFF	
	M25	
	M33 ····· Product collection	

Superimposed machining for type VIII (without tool set longitude data)

$1	$2	$3
:	:	:
!2!3L1	!1!3L1	!1!2L1
T0300	T2600 Z1 K2	T3600 Z2
!2!3L2	!1!3L2	!1!2L2
:	:	:
T0400	!3L3	!2L3
:	T2400 Z1 K2	T3400 Z2
	!3L4	!2L4
	:	:

Superimposed machining for type VIII (with tool set longitude data)

$1	$2	$3
:	:	:
!2!3L1	!1!3L1	!1!2L1
T0300		
!2!3L2	!1!3L2	!1!2L2
	T2600 Z1 K2	
!2!3L3	!1!3L3	!1!2L3
		T3600 Z2
!2!3L4	!1!3L4	!1!2L4
:	:	:
T0400	!3L5	!2L5
:	T2400 Z1 K2	T3400 Z2
	!3L6	!2L6
	:	:

Notes

- To perform machining with superimposition, specify the queuing command before and after the selection of the tool numbers in the T01's, T20's and T30's. At this time, do not specify an offset number for the last 2 digits of the 4-digit T code. Specified the offset number in the 2-digit T command after tool selection.
- After superimposition of the axes, specify the queuing command before and after the selection of the tools of tool numbers in the T20's and T30's.
- Specify the same value for the first digits of the tool numbers in the T20's and T30's.

Superimposed machining for type IX (without tool set longitude data)

$1	$2	$3
:	:	:
!2!3L1	!1!3L1	!1!2L1
T0300	T2600 Z1 K2	T3600 Z2 X2 I3
!2!3L2	!1!3L2	!1!2L2
:	:	:
T0400	T2400 Z1 K2	T3400 Z2 X2 I3
:	:	:

Superimposed machining for type IX (with tool set longitude data)

$1	$2	$3
:	:	:
!2!3L1	!1!3L1	!1!2L1
T0300		
!2!3L2	!1!3L2	!1!2L2
	T2600 Z1 K2	
!2!3L3	!1!3L3	!1!2L3
		T3600 Z2 X2 I3
!2!3L4	!1!3L4	!1!2L4
:	:	:
T0400	T2400 Z1 K2	T3400 Z2 X2 I3
:	:	:

Notes

- To perform machining with superimposed, specify the queuing command before and after the selection of the tool numbers in the T01's, T20's and T30's. At this time, do not specify an offset number for the last 2 digits of the 4-digit T code. Specified the offset number in the 2-digit T command after tool selection.
- X3 axis stroke is shorter than X2 axis when the selection of the tool numbers in the T20's and T30's, OT may cause.

$1	$2	$3
!2L7	!1L7	
G99 M03 S1=[ ] M24 S2=[ ]		
T0100		
G00 X17.0 Z30.0 T01		
G650 ····· Pick-off	G650 ····· Pick-off	G650 ···· Pick-off
:	:	:

The values in parentheses indicate the axes to which the tool belongs. Programming is not necessary.

Notes

- When performing machining with G600 (free machining in three axis control groups), you need to specify cutting block interlock ON/OFF for each spindle by using the relevant M code (M160 to M167). See Section <5.8.2 Cutting start interlock enabled/disabled (M84, M85, and M160 to M167)>.
  When performing back machining with the back spindle, you need to specify back spindle feed per rotation ON/OFF by using the relevant G code (G44 and G43). See Section <5.6.2 Back spindle feed per rotation>.
- Queuing is required in the following cases:
  1. Before and after selection of a tool number in the T20's and T30's for front/back simultaneous machining
  2. Immediately before switching between machining patterns (e.g., from front/back simultaneous machining to $1 single machining or front/back parallel machining)
  3. Before and after selection of a tool number in the T10's and T50's for front/back simultaneous machining.
  4. Before and after selection of a tool number in the T01's, T20's and T30's for superimpose machining.
- When having created a program using $2 or $3 with G600 (free machining in three axis control groups) specified, use G650 to perform pick-off operation. When having created a program using only $1, you can use G650 or T3000A1 to perform pick-off operation. See Sections <5.3.8 Pick-off, Center support (G650)> and <5.5.6 Back spindle>.
- When front/back simultaneous machining is performed with the tools of tool numbers in the T20's and T30's, the core is adjusted for front machining. To adjust the core for back machining, specify B3 in the argument for tool numbers in the T20's. See Section <5.5.4 Front drill (T2100 to T2600)>.
- When front/back simultaneous machining is performed with the tools of tool numbers in the T10's and T50's, the core is adjusted for front machining. To adjust the core for back machining, specify B5 in the argument for the numbers in the T10's.
  See section <5.5.3 Tools on the gang tool post>.
- Each axis control group should hold the feed-per-rotation command and feed-per-minute command as modal functions.
- G600 is a modal G code that is enabled until another machining pattern command is executed.
- Type VI does not have X2, Z2 and X3 axes.
- Type VII does not have Z2 and X3 axes.
- Type VIII dose not have X3 axis.

5.3.4 $1 single machining, Front/back simultaneous machining (G610): Type VII, VIII, IX

This machining pattern command enables single machining with a front drill for front machining in $1, or front/back simultaneous machining with the front drill in $1 and a back drill for back machining in $2. In the case of type IX, X2 axis (opposite tool post) and X3 axis (back spindle) are superimposed, and enables ID machining for front and back simultaneously.

Command format

$1	$2	$3
G610 U0 V0	G610 U0 S1 X2	G610 U0 W0
X2 Z1 C1	X3 Z3 C2	–

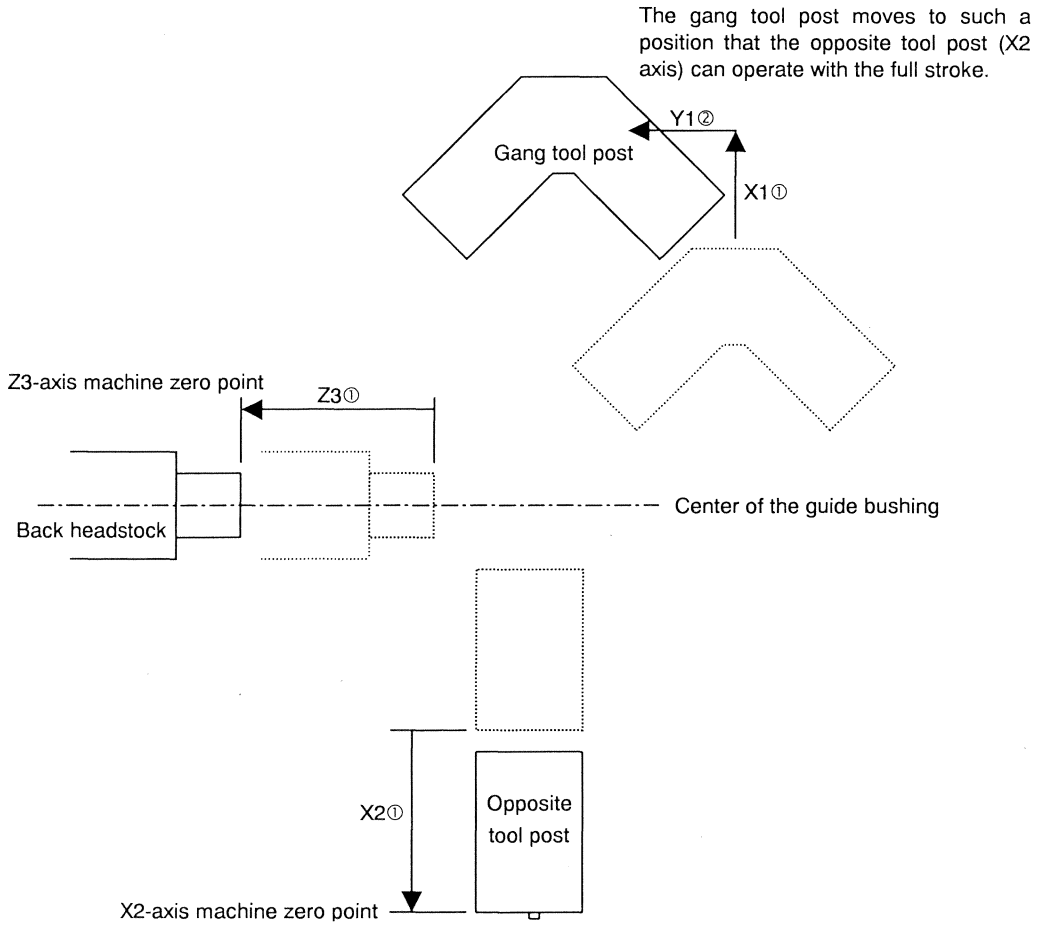
Axis control groups

Specify this command for all the axis control groups $1, $2, and $3.

Arguments

- $1 U0: The gang tool post (X1 axis) does not move. The axis moves to the retract position unless the argument is specified.
- $1 V0: The gang tool post (Y1 axis) does not move. The axis moves to the retract position unless the argument is specified.
- $2 U0: The opposite tool post (X2 axis) does not move. The axis moves to the return position unless the argument is specified.
- $2 S1: Specify this argument to perform front/back simultaneous machining (with tools of tool numbers in the $1T20's and $2T30's). Execute G226 S0 to switch from front/back simultaneous machining to $1 single machining.
- $2 X2: Back spindle (X3 axis) is superimposed on opposite tool post (X2 axis). It is specified when front and back ID machining simultaneously. When X2 argument is specified, it changes to front/back simultaneous machining automatically, don't need to specify S1 argument. When change from front/back simultaneous machining to $1 single machining, specify G266 X3. (Type IX)
- $3 U0: The back headstock (X3 axis) does not move. The axis moves to the return position unless the argument is specified. (Type IX)
- $3 W0: The back headstock (Z3 axis) does not move. The axis moves to the return position unless the argument is specified.

Operation sample



- The circled numbers indicate the operation sequence for each tool post.
- The retract positions of X1 and Y1 axes are X180.0 and Y–180.0 in the machine coordinate system after the virtual X1 and Y1 axes are canceled. While the X1 and Y1 axes are at the retract positions, the opposite tool post X2 axis (tools of tool numbers in the T20's) can freely operate with its full stroke.

Macro specification

Command code	G610		
Name	$1 single machining and front/back simultaneous machining		
Axis control group	$1	$2	$3
Axes of axis control group	X2, Z1, C1	X3, Z3, C2	–
Superimpose	–	–	–
Coordinate system	–	–	–
Argument	U0: The gang tool post (X1 axis) does not move. The axis moves to the retract position unless the argument is specified. V0: The gang tool post (Y1 axis) does not move. The axis moves to the retract position unless the argument is specified.	U0: The opposite tool post (X2 axis) does not move. The axis moves to the return position unless the argument is specified. S1: Specify this argument to perform front/back simultaneous machining (with tools of tool numbers in the $1T20's and $2T30's). Execute G226 S0 to switch from front/back simultaneous machining to $1 single machining. X2: Back spindle (X3 axis) is superimposed on opposite tool post (X2 axis). It is specified when front and back ID machining simultaneously. When change from front/back simultaneous machining to $1 single machining, specify G266 X3. (Type IX)	U0: The back headstock (X3 axis) does not move. The axis moves to the return position unless the argument is specified. W0: The back headstock (Z3 axis) does not move. The axis moves to the return position unless the argument is specified.
Spindle with which synchronous feed is enabled	Main spindle	Back spindle	–
Spindle with which constant surface speed control is enabled	Main spindle	Back spindle	–
Cutting block interlock	Main spindle Front/back face tool spindle	Back spindle	–
T command	Front drilling tool T2100 to T2600	Back drilling tool T3100 to T3600	–
Others	–	–	–

Program sample

$1 single machining (G610) → front/back parallel machining (G630)

$1	$2	$3
: G610 T2100 (X2 Z1 C1) : T2200 (X2 Z1 C1) : G630 T0300 (X1 Z1 Y1 C1) :	: G610 G630 T3200 (X2 Z3 C2) :	: G610 G630

The values in parentheses indicate the axes to which the tool belongs. Programming is not necessary.

Front/back simultaneous machining (G610) → $1 single machining (G610) →front/back parallel machining (G630)

$1	$2	$3
G610 T2100 (X2 Z1 C1) : T2200 (X2 Z1 C1) : !2L1 T2300 (X2 Z1 C1) : G630 T0300 (X1 Z1 Y1 C1) :	G610 S1 T3100 (Z3 C2) : T3200 (Z3 C2) : G226 S0 ····· Front/back simultaneous machining cancel !1L1 G630 T3200 (X2 Z3 C2) :	G610 G630

The values in parentheses indicate the axes to which the tool belongs. Programming is not necessary.

Notes

- To perform front/back simultaneous machining, specify the S1 (or X2) argument for G610$2. Specifying S1 enables automatic queuing at selection of the tools of tool numbers in the T20's and T30's. To select a tool after executing the G610 command, select the tools of tool numbers in the T20's and T30's. In the case of type VII and VIII, be sure to specify the same number as the first digit of the tool numbers of the T20's and T30's.
- To cancel front/back simultaneous machining, execute G226 S0 (or G226 X3). In this case, queuing is necessary after execution of G226 S0 (or G226 X3).
- When front/back simultaneous machining is performed, the center is adjusted for front machining. To adjust the center for back machining, specify B3 in the argument for tool numbers in the T20's. See Section <5.5.4 Front drill (T2100 to T2600)>.
- Each axis control group should hold the feed-per-rotation command and feed-per-minute command as modal functions.
- G610 is a modal G code that is enabled until another machining pattern command is executed.

CAUTION

Operating the X2 axis of $1 in front/back simultaneous machining (G610) may cause interference with the tool of a tool number in the T30's of $2.

5.3.5 Outer diameter/inner diameter simultaneous machining (G620): Type VIII, IX

This machining pattern command enables the following two types of machining at the same time while superimposing the Z2 axis on the Z1 axis:
Outer diameter machining with a gang tool for front machining in $1
Inner diameter machining with a front drill for front machining in $2

Command format

$1	$2	$3
G620	G620 U0 W0	G620 U0 W0
X1 Z1 Y1 C1	X2 Z2	–

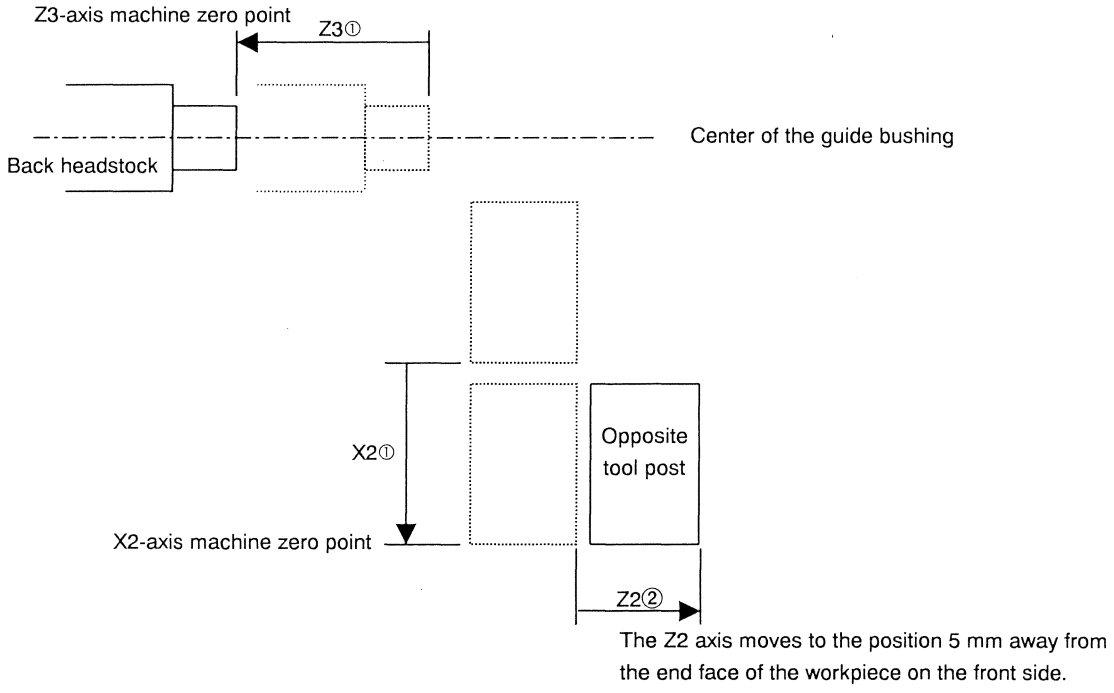
Axis control groups

Specify this command for all the axis control groups $1, $2, and $3.

Arguments

- $1: The gang tool post (X1 axis) does not move.
- $1: The gang tool post (Y1 axis) does not move.
- $2 U0: The opposite tool post (X2 axis) does not move. The axis moves to the return position unless the argument is specified.
- $2 W0: The opposite tool post (Z2 axis) does not move. The Z2 axis moves to the position 5.0 mm away from the end face of the workpiece on the front side unless the argument is specified.
- $3 U0: The back headstock (X3 axis) does not move. The axis moves to the return position unless the argument is specified. (Type IX)
- $3 W0: The back headstock (Z3 axis) does not move. The axis moves to the return position unless the argument is specified.

Operation sample



- The circled numbers indicate the operation sequence for each tool post.

Macro specification

Command code	G620		
Name	Outer diameter/inner diameter simultaneous machining		
Axis control group	$1	$2	$3
Axes of axis control group	X1, Z1, Y1, C1	X2, Z2	–
Superimpose	–	Z2 superimposed on Z1	–
Coordinate system	–	–	–
Argument	The gang tool post (X1 axis) does not move. The gang tool post (Y1 axis) does not move.	U0: The opposite tool post (X2 axis) does not move. The axis moves to the return position unless the argument is specified. W0: The opposite tool post (Z2 axis) does not move. The Z2 axis moves to the position 5.0 mm away from the end face of the workpiece on the front side unless the argument is specified.	U0: The back headstock (X3 axis) does not move. The axis moves to the return position unless the argument is specified. W0: The back headstock (Z3 axis) does not move. The axis moves to the return position unless the argument is specified.
Spindle with which synchronous feed is enabled	Main spindle	Main spindle	–
Spindle with which constant surface speed control is enabled	Main spindle	Main spindle	–
Cutting block interlock	Main spindle	Main spindle	–
T command	Gang tool T0100 to T0600 T1100 to T1400	Front drilling T2100 to T2600	–
Others	–	–	–

Program sample

$1	$2	$3
: G620 T0300 (X1 Z1 Y1 C1) :	: G620 T2600 (X2 Z2) : T2500 (X2 Z2) :	G620

The values in parentheses indicate the axes to which the tool belongs. Programming is not necessary.

Notes

- Each axis control group should hold the feed-per-rotation command and feed-per-minute command as modal functions.
- G620 is a modal G code that is enabled until another machining pattern command is executed.

5.3.6 Front/back parallel machining (G630): Type VII, VIII, IX

Use this machining pattern to perform front machining and back machining independently. This machining pattern command enables machining with a gang tool for front machining in $1 and machining with a back drill for back machining in $2.

Command format

Type VII

$1	$2	$3
G630	G630 U0	G630 W0
X1 Z1 Y1 C1	X2 Z3 C2	–

Type VIII, IX

$1	$2	$3
G630	G630 U0 S[ ]	G630 U0 W0
X1 Z1 Y1 C1	X2 Z3 C2	–

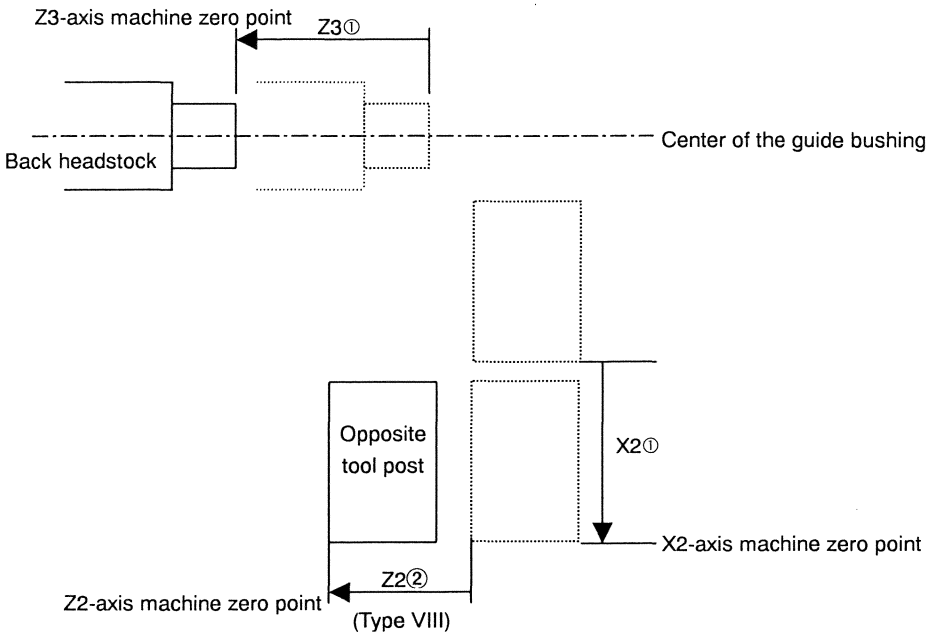
Axis control groups

Specify this command for all the axis control groups $1, $2, and $3.

Arguments

- $1: The gang tool post (X1 axis) does not move.
- $1: The gang tool post (Y1 axis) does not move.
- $2 U0: The opposite tool post (X2 axis) does not move. The axis moves to the return position unless the argument is specified.
- $2 S[ ]: Specify the amount of shift from the machine zero point of the opposite tool post (Z2 axis). The Z2 axis shifts the amount specified by the S argument. Specify the argument if the protrusion of the workpiece chucked by the back spindle chuck exceeds 25.0 mm or if the protrusion of the tool nose from the back drilling holder (T3100 to T3600) goes beyond the standard tool nose position. If macro software Ver 005-001 or later is used, the Z2 axis is automatically shifted in the following case: the protrusion of the tool nose from the back drilling holder (T3100 to T3600) goes beyond the standard tool nose position, and the protrusion is set as the value for tool setting in the longitudinal direction. However, the S argument is valid if it is specified. (Type VIII, IX)
- $3 U0: The back headstock (X3 axis) does not move. The axis moves to the return position unless the argument is specified. (Type IX)
- $3 W0: The back headstock (Z3 axis) does not move. The axis moves to the return position unless the argument is specified.

Operation sample



- The circled numbers indicate the operation sequence for each tool post.

Macro specification

Command code	G630		
Name	Front/back parallel machining		
Axis control group	$1	$2	$3
Axes of axis control group	X1, Z1, Y1, C1	X2, Z3, C2	–
Superimpose	–	–	–
Coordinate system	–	–	–
Argument	The gang tool post (X1 axis) does not move. The gang tool post (Y1 axis) does not move.	U0: The opposite tool post (X2 axis) does not move. The axis moves to the return position unless the argument is specified. S□□□□: Specify the amount of shift from the machine zero point of the opposite tool post (Z2 axis). (Type VIII, IX)	U0: The back headstock (X3 axis) does not move. The axis moves to the return position unless the argument is specified. W0: The back headstock (Z3 axis) does not move. The axis moves to the return position unless the argument is specified.
Spindle with which synchronous feed is enabled	Main spindle	Back spindle	–
Spindle with which constant surface speed control is enabled	Main spindle	Back spindle	–
Cutting block interlock	Main spindle Gang tool spindle	Back spindle Front/back face tool spindle	–
T command	Gang tool T0100 to T0600 T1100 to T1400	Back drilling tool T3100 to T3600	–
Others	–	–	–

Program sample

$1	$2	$3
: G630 ···· Front/back parallel machining T0200 (X1 Z1 Y1 C1) G00 X12.0 Z-0.5 T02 G01 Z15.0 F0.1 :	: G630 ···· Front/back parallel machining T3600 (X2 Z3 C2) G00 Z-0.5 T36 G01 Z3.0 F0.03 :	: G630 ···· Front/back parallel machining :

The values in parentheses indicate the axes to which the tool belongs. Programming is not necessary.

Notes

- Each axis control group should hold the feed-per-rotation command and feed-per-minute command as modal functions.
- Specify commands to rotate the main spindle in $1, and specify commands to rotate the back spindle in $2.
- G630 is a modal G code that is enabled until another machining pattern command is executed.

CAUTION

When using G630 as a machining pattern, the front drilling tool may be installed on the opposite side of the back drilling tool (e.g., T3600 and T2600, or T3500 and T2500). If the back machining using the back drilling tool and front machining using the gang tool are performed simultaneously, the gang tool or front workpiece may interfere with the front drilling tool.

5.3.7 3-line simultaneous machining (G640): Type VIII, IX

This machining pattern command enables the following three types of machining at the same time while superimposing the Z2 axis on the Z1 axis and the Z3 axis on the Z2 axis:
Outer diameter machining with a gang tool for front machining in $1
Inner diameter machining with a front drill for front machining in $2
Inner diameter machining with a back drill for back machining in $3
In the case of type IX, X2 axis (opposite tool post) and X3 axis (back spindle) are superimposed, and enables ID machining for front and back simultaneously.

Command format

$1	$2	$3
G640	G640 U0 W0	G640 U0 W0
X1 Z1 Y1 C1	X2 Z2	X3 Z3 C2

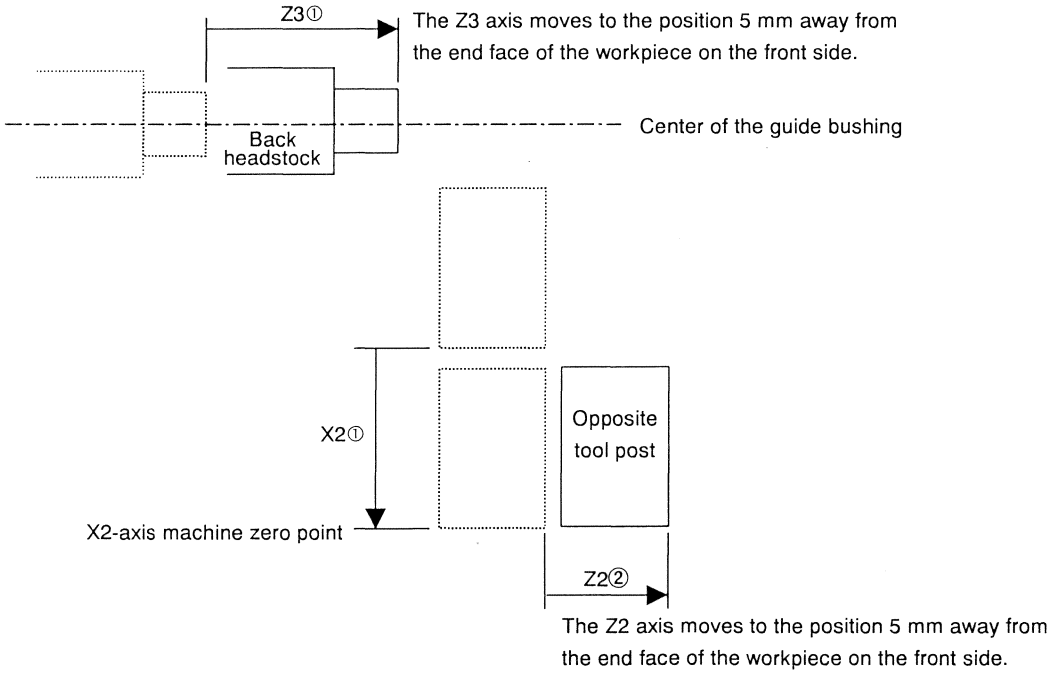
Axis control groups

Specify this command for all the axis control groups $1, $2, and $3.

Arguments

- $1: The gang tool post (X1 axis) does not move.
- $1: The gang tool post (Y1 axis) does not move.
- $2 U0: The opposite tool post (X2 axis) does not move. The axis moves to the return position unless the argument is specified.
- $2 W0: The opposite tool post (Z2 axis) does not move. The Z2 axis moves to the position 5.0 mm away from the end face of the workpiece on the front side unless the argument is specified.
- $3 U0 The back headstock (X3 axis) does not move. The axis moves to the return position unless the argument is specified. (Type IX)
- $3 W0: The back headstock (Z3 axis) does not move. The Z3 axis moves to the position 5.0 mm away from the end face of the workpiece on the front side unless the argument is specified.

Operation sample



- The circled numbers indicate the operation sequence for each tool post.

Macro specification

Command code	G640		
Name	3-line simultaneous machining		
Axis control group	$1	$2	$3
Axes of axis control group	X1, Z1, Y1, C1	X2, Z2	X3, Z3, C2
Superimpose	–	Z2 superimposed on Z1	X3 superimposed on X2 Z3 superimposed on Z2
Coordinate system	–	–	–
Argument	The gang tool post (X1 axis) does not move. The gang tool post (Y1 axis) does not move.	U0: The opposite tool post (X2 axis) does not move. The axis moves to the return position unless the argument is specified. W0: The opposite tool post (Z2 axis) does not move. The Z2 axis moves to the position 5.0 mm away from the end face of the workpiece on the front side unless the argument is specified.	U0: The back headstock (X3 axis) does not move. The axis moves to the return position unless the argument is specified. W0: The back headstock (Z3 axis) does not move. The Z3 axis moves to the position 5.0 mm away from the end face of the workpiece on the front side unless the argument is specified.
Spindle with which synchronous feed is enabled	Main spindle	Main spindle	Back spindle
Spindle with which constant surface speed control is enabled	Main spindle	Main spindle	Back spindle
Cutting block interlock	Main spindle	Main spindle	Back spindle
T command	Gang tool T0100 to T0600 T1100 to T1400	Front drilling T2100 to T2600	Back drilling T3100 to T3600
Others	–	–	–

Program sample

$1	$2	$3
: G640 T0300 (X1 Z1 Y1 C1) :	: G640 T2600 (X2 Z2) : T2500 (X2 Z2) :	: G640 T3600 (X3 Z3 C2) : T3500 (X3 Z3 C2) :

The values in parentheses indicate the axes to which the tool belongs. Programming is not necessary.

Notes

- Each axis control group should hold the feed-per-rotation command and feed-per-minute command as modal functions.
- G640 is a modal G code that is enabled until another machining pattern command is executed.

5.3.8 Pick-off, Center support (G650)

This machining pattern enables the back spindle to re-chuck (pick off) workpiece and support the long workpiece.

Command format

$1	$2	$3
G650	G650	G650 W0
X1 Z1 Y1 C1	Z3 C2	–

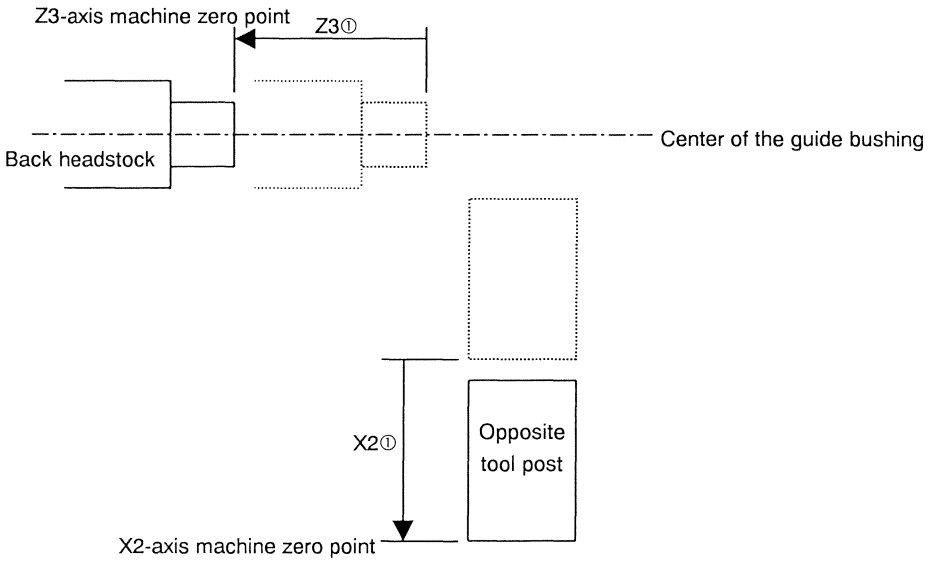
Axis control groups

Specify this command for all the axis control groups $1, $2, and $3.

Arguments

- $1: The gang tool post (X1 axis) does not move.
- $1: The gang tool post (Y1 axis) does not move.
- $2: The opposite tool post (X2 axis) moves to the return position (unconditionally).
- $3 W0: The back headstock (Z3 axis) does not move. The axis moves to the return position unless the argument is specified.

Operation sample



- The circled numbers indicate the operation sequence for each tool post.
- Set the work coordinate system for the Z3 axis so that the end face of a workpiece is adjusted to 0 for front machining.

Macro specification

Command code	G650		
Name	Pick-off, center support		
Axis control group	$1	$2	$3
Axes of axis control group	X1, Z1, Y1, C1	Z3, C2	–
Superimpose	–	–	Z3 superimposed on Z1
Coordinate system	–	–	–
Argument	The gang tool post (X1 axis) does not move. The gang tool post (Y1 axis) does not move.	Moves the tool on the opposite tool post (X2 axis) to the return position.	W0: The back headstock (Z3 axis) does not move. The axis moves to the return position unless the argument is specified.
Spindle with which synchronous feed is enabled	Main spindle	Back spindle	–
Spindle with which constant surface speed control is enabled	Main spindle	Back spindle	–
Cutting block interlock	Main spindle Gang tool spindle	Back spindle Front/back face tool spindle	Back spindle Front/back face tool spindle
T command	Back spindle T3000	Back spindle T3000	Back spindle T3000
Others	–	–	–

Program sample

$1	$2	$3
:		:
:		:
G99 M03 S1=1500 M24 S2=1500		:
T0100		:
G0 X17.0 Z30.0		
G650 ····· Pick-off	G650 ····· Pick-off	G650 ····· Pick-off
	G0 Z-0.5	
	G98 G1 Z10.0 F3000	
	G4 U0.5	
	M15	
!2L1	!1L1	
G01 X-1.0 F0.03		
G600	G600	G600
X-3.0	:	:
:	:	:

Notes

- Execute the pick-off command (G650). At this time, the back spindle moves to the position determined by the front workpiece coordinates, superimpose the Z3 axis on the Z1 axis and execute the axis move command.
- Position the Z1 axis at cut-off position before specifying the G650 command.
- During pick-off operation (G650), you can move the Z3 axis to machine zero point by specifying the G231 command. See <Section 5.7.4 Pick-off cancel (G231)>.
- When having created a program using only $1, you can also perform pick-off operation by using T3000A1. See Section <5.5.6 Back spindle>.

5.3.9 Front/back simultaneous machining (G660): Type VI

This machining pattern command enables front/back simultaneous machining with a vertical front drill for front machining in $1 and a vertical back drill for back machining in $2.

Command format

$1	$2	$3
G660	G660	G660 W0
X1 Z1 Y1 C1	Z3 C2	–

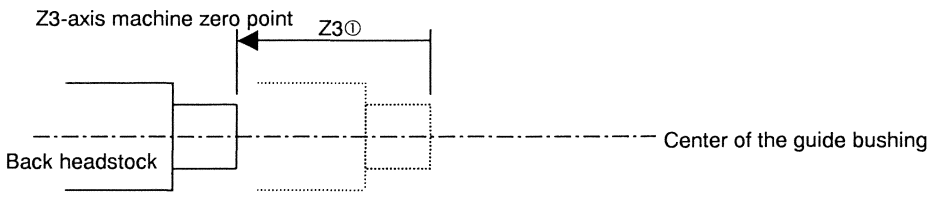
Axis control groups

Specify this command for all the axis control groups $1, $2, and $3.

Arguments

- $3 W0: The back headstock (Z3 axis) does not move. The axis moves to the return position unless the argument is specified.

Operation sample



- The circled numbers indicate the operation sequence for each tool post.

Macro specification

Command code	G660		
Name	Front/back simultaneous machining		
Axis control group	$1	$2	$3
Axes of axis control group	X1, Z1, Y1, C1	Z3, C2	–
Superimpose	–	–	–
Coordinate system	–	–	–
Argument			W0: The back headstock (Z3 axis) does not move. The axis moves to the return position unless the argument is specified.
Spindle with which synchronous feed is enabled	Main spindle	Back spindle	–
Spindle with which constant surface speed control is enabled	Main spindle	Back spindle	–
Cutting block interlock	Main spindle	Back spindle	–
T command	Vertical drill tool T1400 to T1700	Vertical drill tool T5400 to T5700	–
Others	–	–	–

Program sample

Front/back simultaneous machining (G660)

$1	$2	$3
G660 T1400 (X1 Z1 Y1 C1) : T1600 (X1 Z1 Y1 C1) :	G660 T5400 (Z3 C2) : T5600 (Z3 C2) :	G660

The values in parentheses indicate the axes to which the tool belongs. Programming is not necessary.

Notes

- Specifying the G660 enables automatic queuing at selection of the tools of tool numbers in the T10's and T50's. To select a tool, select the tools of tool numbers in the T10's and T50's. At this time, be sure to specify the same number as the first digit of the tool numbers of the T10's and T50's.
- When front/back simultaneous machining is performed, the center is adjusted for front machining. To adjust the center for back machining, specify B5 in the argument for tool numbers in the T10's. See Section <5.5.3 Tools on the gang tool post>.
- Each axis control group should hold the feed-per-rotation command and feed-per-minute command as modal functions.
- G660 is a modal G code that is enabled until another machining pattern command is executed.

5.3.10 Sequential machining in a single axis control group (G670)

Specify this machining pattern command to create a program using only $1. The machining pattern command enables machining with a gang tool, a front drill, and a back drill, product collection, and pick-off by using only $1.

Command format

$1	$2	$3
G670	G670 U0	G670 U0 W0
X1 Z1 Y1 C1	–	–

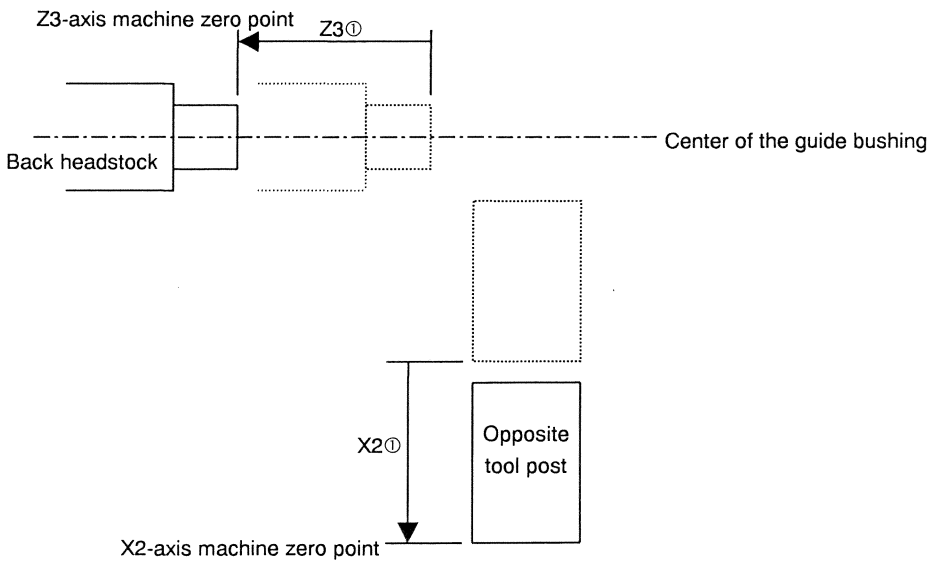
Axis control groups

Specify this command for all the axis control groups $1, $2, and $3.

Arguments

- $1: The gang tool post (X1 axis) does not move.
- $1: The gang tool post (Y1 axis) does not move.
- $2 U0: The opposite tool post (X2 axis) does not move. The axis moves to the return position unless the argument is specified. (Type VII, VIII, IX)
- $3 W0: The back headstock (X3 axis) does not move. The axis moves to the return position unless the argument is specified. (Type IX)
- $3 W0: The back headstock (Z3 axis) does not move. The axis moves to the return position unless the argument is specified.

Operation sample



- The circled numbers indicate the operation sequence for each tool post.

Macro specification

Command code	G670		
Name	Sequential machining in a single axis control group		
Axis control group	$1	$2	$3
Axes of axis control group	X1, Z1, Y1, C1	–	–
Superimpose	–	–	–
Coordinate system	–	–	–
Argument	The gang tool post (X1 axis) does not move. The gang tool post (Y1 axis) does not move.	U0: The opposite tool post (X2 axis) does not move. The axis moves to the return position unless the argument is specified.	U0: The back headstock (X3 axis) does not move. The axis moves to the return position unless the argument is specified. W0: The back headstock (Z3 axis) does not move. The axis moves to the return position unless the argument is specified.
Spindle with which synchronous feed is enabled	Main spindle	–	–
Spindle with which constant surface speed control is enabled	Main spindle	–	–
Cutting block interlock	Main spindle Back spindle Gang tool spindle Front/back face tool spindle	–	–
T command	Gang tool, front/back drilling, and back spindle T0100 to T0600, T1100 to T1400, (Type VII, VIII, IX) T1100 to T1700, (Type VI) T2100 to T2600, (Type VII, VIII, IX) T3000 T3100 to T3600 (Type VII, VIII, IX) T5400 to T5700 (Type VI)	–	–
Others	–	–	–

Program sample

$1	$2	$3
: G670 ···· Sequential machining in a single axis control group T2100 R1 (X2 Z1 C1) G00 Z–0.5 T21 G01 Z3.0 F0.03 : M147 ···· The opposite tool post retracts. T0200 (X1 Z1 Y1 C1) G00 X17.0 Z–0.5 T02 G01 Z15.0 F0.1 : M23 S2=2500 G44 M151 ···· The gang tool post retracts. T3600 (X2 Z3 C2) G00 Z–0.5 T36 G01 Z3.0 F0.03 : M147 ···· The opposite tool post retracts. G43 M25 M33 ····· Sequential operation of pick-off and product collection G99 M03 S1=[ ] M24 S2=[ ] T0100 (X1 Z1 Y1 C1) G0 X17.0 Z30.0 T01 T3000 A1 E10.0 S3000 ·· Pick-off G01 X–3.0 F0.03 G600 :	G670 ···· Sequential machining in a single axis control group G600	: G670 ···· Sequential machining in a single axis control group G600

The values in parentheses indicate the axes to which the tool belongs. Programming is not necessary.

Notes

- While the G670 command (sequential machining in a single axis control group) is in progress, the gang tool post retracts to a safety position in the following conditions:
  The previously selected tool number is of a tool on the gang tool post (T0100 to T0600 or T1100 to T1400), and a tool on the opposite tool post (T2100 to T2600 or T3100 to T3600) is currently specified.
  While the G670 command (sequential machining in a single axis control group) is in progress, the opposite tool post (X2 axis) retracts to the machine zero point in the following conditions:
  The previously selected tool number is of a tool on the opposite tool post (T2100 to T2600 or T3100 to T3600), and a tool on the gang tool post (T0100 to T0600 or T1100 to T1400) is currently specified.
  If you do not want to retract the tool post, specify R0 for the argument of each T code.
  (The R0 argument is supported in macro Ver 002-001 and later.)
- When having created a program using the G670 command (sequential machining in a single axis control group), you can use G650 or T3000 A1 to perform pick-off operation. See Sections <5.3.8 Pick-off, Center support (G650)> and <5.5.6 Back spindle>.
- Each axis control group should hold the feed-per-rotation command and feed-per-minute command as modal functions.
- G670 is a modal G code that is enabled until another machining pattern command is executed.

5.4 S Functions (S Codes)

The S functions specified in the following formats are called speed functions. The S functions are used to specify the speeds of the main spindle, the back spindle, the tool spindle of the gang tool post, and the tool spindle of the front back face machining.

Command format

S1 = ☐	Main spindle
S2 = ☐	Back spindle
S3 = ☐	Tool spindle of the gang tool post
S4 = ☐	Tool spindle of the front/back face machining (Type VII, VIII, IX)

Calculate the spindle speed from the following formula. Round the calculation result to the nearest whole integer.

$$N = \frac{V}{\pi D} \times 1000$$

N: Speed (min^{-1})
V: Cutting speed (m/min)
D: Workpiece diameter (mm) (With drilling: Hole diameter)
π: Circular constant (≈ 3.14)

Command ranges of the S codes

Spindle no.	Name	Speed C12	Speed C16
S1	Main spindle	200 to 12000min^{-1}	200 to 10000min^{-1}
S2	Back spindle	200 to 10000min^{-1}	200 to 10000min^{-1}
S3	Tool spindle of gang tool post	200 to 8000min^{-1}	200 to 8000min^{-1}
S4	Tool spindle of the front/back face machining	200 to 5000min^{-1}	200 to 5000min^{-1}

Note

S4 (Tool spindle of front/back face machining) is optional.

M codes

Name	Forward	Reverse	Stop	S command
Main spindle	M03	M04	M05	S1 = ☐
Back spindle	M23	M24	M25	S2 = ☐
Tool spindle of gang tool post	M58	M59	M60	S3 = ☐
Tool spindle of the front face machining	M80	M81	M82	S4 = ☐
Tool spindle of the back face machining	M74	M75	M76	S4 = ☐

5.5 T Functions (T Codes)

The T function determines a tool position where the tool can machine the workpiece. T codes for tool selection are as follows. T0100 to T0600, T1100 to T1400 (Type VII, VIII, IX), T1100 to T1700 (Type VI) and T5400 to T5700 (Type VI) are for tools on the gang tool post. T2100 to T2600 (Type VII, VIII, IX) are for tools on the front drilling. T3100 to T3600 (Type VII, VIII, IX) are for tools on the back drilling. T3000 is for the back spindle.
A tool is positioned by the "T□□ΔΔ" command.
The queuing position of a tool on the gang tool post can be set arbitrarily as the "Tool Positioning Point (DIA)" in the machining data.
To machine thin material, set at least about 0.2 mm for the tool queuing point.
Specify the "T□□ΔΔ" command with a 4-digit number. The first two digits correspond to the tool number. The last two digits correspond to the compensation number. Specify 00 as a compensation number in the compensation cancel state.

Command format

T□□ΔΔ

Argument

- □□: Up to 23 tool (Type VII, VIII, IX) numbers can be specified, which is the total of tools (T0100 to T0600, T1100 to T1400, T1100 to 1700, and T5400 to T5700) on the gang tool post, tools (T2100 to T2600) on the front drilling, tools (T3100 to T3600) on the back drilling, and the back spindle (T3000).
- ΔΔ: Compensation number of tool nose wear

5.5.1 Virtual coordinate system

Virtual X-Y control

If a tool post having a vertical axis (motor X axis) and a horizontal axis (motor Y axis) is used, the virtual X-Y control function moves a tool placed under the following conditions when an infeed direction and vertical direction are specified. The tool is placed with which a nose position and an angle are specified for the orthogonal coordinate system (called a reference machine coordinate system) that is parallel to the motor axis. This machine uses this function with the gang tool post.
The command coordinate system, whose reference point and axial direction are different from those of the reference machine coordinate system, is called a virtual coordinate system. Axes specified in the virtual coordinate system are called virtual axes (X axis called virtual X axis, and Y axis called virtual Y axis).

Relationship between the virtual coordinate system and operation modes:

Mode	Virtual coordinate system	Description
Zero point return	OFF	Any selected axis is treated as a motor axis.
Mechanism adjustment	OFF	Any selected axis is treated as a motor axis.
Preparation	ON	A virtual coordinate system is set in preparation to move the gang tool post. However, when the machine enters this mode, the virtual coordinate system of the previously selected tool is set.
Program check	ON	A virtual coordinate system is set when a T code is specified in a program. However, when the machine enters this mode, the virtual coordinate system of the previously selected tool is set.
Automatic operation	ON	A virtual coordinate system is set when a T code is specified in a program. However, when the machine enters this mode, the virtual coordinate system of the previously selected tool is set.
MDI	ON	A virtual coordinate system is set when a T code is specified in the MDI command. However, when the machine enters this mode, the virtual coordinate system of the previously selected tool is set.
Manual operation	ON	The virtual coordinate system of the previously selected tool is set because this mode is used to move an axis with the handle. This virtual coordinate system cannot be switched by selecting a tool number. However, the plus and minus directions at selection of X and Y axes are displayed with the icons.

Software limits when virtual axes are used:

Software limits (OT+, OT–) are set in the reference machine coordinate system. They are also set in a virtual coordinate system separately. A virtual axis can move in the area where the move-permitted areas of the reference machine coordinate system and the virtual coordinate system overlap with each other.

Feed rate

A feed rate turns out to be the synthesized speed of two motor axes. For example, if an angle for the reference point of a virtual axis is 45°, the rapid feed rate is the motor-axis speed $\times \sqrt{2}$.

Coordinate display

A virtual coordinate system also has machine coordinates and work coordinates. The machine coordinates in the virtual coordinate system are positioned where the standard tool nose position of a selected tool is adjusted to the center of the main spindle. Setting a coordinate system does not change the machine coordinates. However, selecting another tool changes the original machine coordinates to the machine coordinates of the selected tool. The Position Data screen displays the machine coordinates of the virtual coordinate system. The Zero Return screen and the Mechanism Adjustment screen display the machine coordinates of the motor axis (the same machine coordinates as for other axes).

5.5.2 Tool mounting positions and the types of machining

The following table shows the tool mounting positions and the types of machining when GTF5010 (standard tool holder) is used:

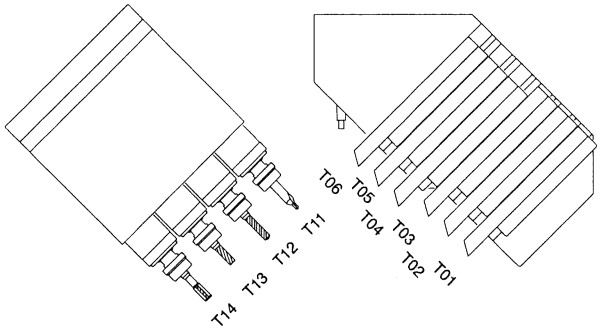
Type VII, VIII, IX

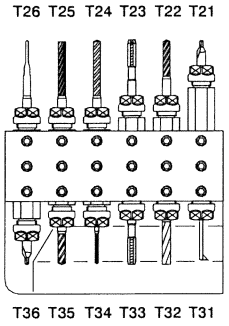
Mounting position	Tool no.	Outer diameter machining	Tool spindle	Inner diameter machining in front machining	Pick-off	Inner diameter machining in back machining
Gang tool post	01	T0100	–	–	–	–
	02	T0200	–	–	–	–
	03	T0300	–	–	–	–
	04	T0400	–	–	–	–
	05	T0500	–	–	–	–
	06	T0600	–	–	–	–
	11	–	T1100	–	–	–
	12	–	T1200	–	–	–
	13	–	T1300	–	–	–
	14	–	T1400	–	–	–
Front drilling tool	21	–	–	T2100	–	–
	22	–	–	T2200	–	–
	23	–	–	T2300	–	–
	24	–	–	T2400	–	–
	25	–	–	T2500	–	–
	26	–	–	T2600	–	–
Back spindle	30	–	–	–	T3000	–
Back drilling tool	31	–	–	–	–	T3100
	32	–	–	–	–	T3200
	33	–	–	–	–	T3300
	34	–	–	–	–	T3400
	35	–	–	–	–	T3500
	36	–	–	–	–	T3600

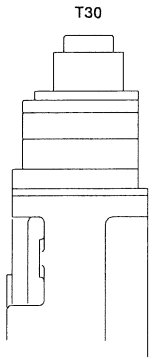
Type VI

Mounting position	Tool no.	Outer diameter machining	Tool spindle	Inner diameter machining in front machining	Pick-off	Inner diameter machining in back machining
Gang tool post	01	T0100	–	–	–	–
	02	T0200	–	–	–	–
	03	T0300	–	–	–	–
	04	T0400	–	–	–	–
	05	T0500	–	–	–	–
	06	T0600	–	–	–	–
	11	–	T1100	–	–	–
	12	–	T1200	–	–	–
	13	–	T1300	–	–	–
	14	–	–	T1400	–	–
	15	–	–	T1500	–	–
	16	–	–	T1600	–	–
	17	–	–	T1700	–	–
	30	–	–	–	T3000	–
	54	–	–	–	–	T5400
	55	–	–	–	–	T5500
	56	–	–	–	–	T5600
	57	–	–	–	–	T5700

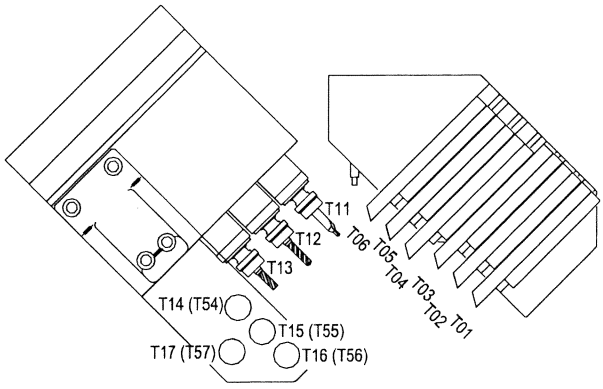
Tool post of type VII, VIII, IX

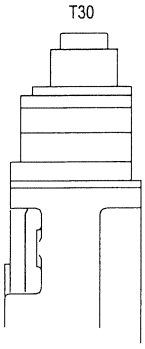






Tool post of type VI





5.5.3 Tools on the gang tool post

Command format

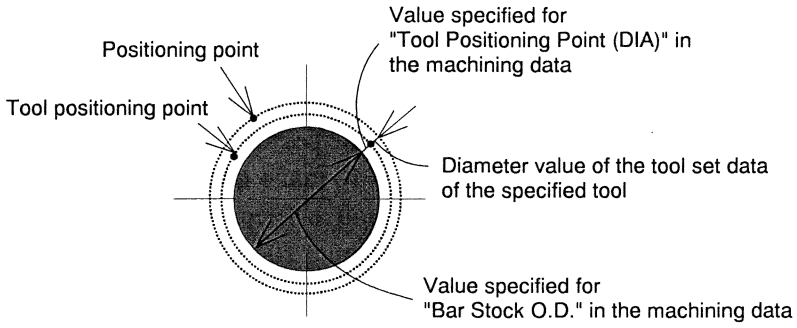
T□□ΔΔ H[ ] U[ ] V[ ] W[ ] Q1 K0, 3 R0 B5

Argument

- H[ ]: Specify the distance by which the currently machining tool moves from the material outer diameter specified in the machining data. If the H argument is not specified, the tool moves to the position ("Tool Positioning Point (DIA)" + maximum diameter value of tool set data). (As for the vertical face holder (T1400 to T1700) of type VI, this argument is invalid.)
- U[ ]: Specify the X1 axis work coordinate after tool selection. If the X argument is not specified, the tool moves to the positioning point.
- V[ ]: Specify the Y1 axis work coordinate after tool selection. If the Y argument is not specified, the tool moves to the work zero point.
- W[ ]: Specify the Z1 axis work coordinate after tool selection. The Z axis does not move unless the Z argument is specified.
- Q1: Specify this argument to select a tool without moving the currently machining tool to the queuing point. If the Q1 argument is not specified, the machine selects a tool after the currently machining tool moves to the position ("Tool Positioning Point (DIA)" + maximum diameter value of tool set data).
- K0: Specify this argument to select a tool for free machining in three axis control groups (G600) without using the Z1 axis. The Z1 axis is used unless the argument is specified. The argument must be specified for pre-selection.
- K3: Specify this argument to use the Z3 axis with the machining pattern G600/G670. The Z1 axis is used if the K3 argument is not specified.
- R0: Specify this argument not to move the opposite tool post (X2 axis) while the G670 command (sequential machining in a single axis control group) is in progress if the previously selected tool number is of a tool on the opposite tool post (T2100 to T2600 or T3100 to T3600). The opposite tool post (X2 axis) retracts to the machine zero point if the R0 argument is not specified. (The R0 argument is supported in macro Ver 002-001 and later.)
- B5: Specify this argument to add the center data for back machining (with a tool T5400 to T5700). Default: The center data specified for front machining (with a tool T1400 to T1700) is used. This argument function in the case of type VI.

What is a positioning point?

Positioning point = value specified for "Bar Stock O.D." in the machining data + value specified for "Tool Positioning Point (DIA)" in the machining data + the diameter value of the tool set data of the specified tool



Note

Setting and calling of tools for secondary machining that are mounted on the gang tool post:
Observe the following if a tool for secondary machining mounted on the gang tool post is beyond the tool nose reference position in the diametrical direction (direction of X1 axis):
Set the protrusion length from the tool nose reference position for the diameter value of the tool set data that corresponds to the tool number.
In this case, you do not have to set the protrusion length from the tool nose reference position for the "Tool Positioning Point (DIA)" in the machining data. For the tool positioning point in the machining data, generally set about 1.0 mm, or set 1.0 mm or more depending on the tool to be mounted.
After calling the tool mounted beyond the tool nose reference position, you do not have to shift the coordinate system of the diametrical direction in the program. The tool is moved to the positioning point when the calling is completed.

Operation sample

- T□□ΔΔ

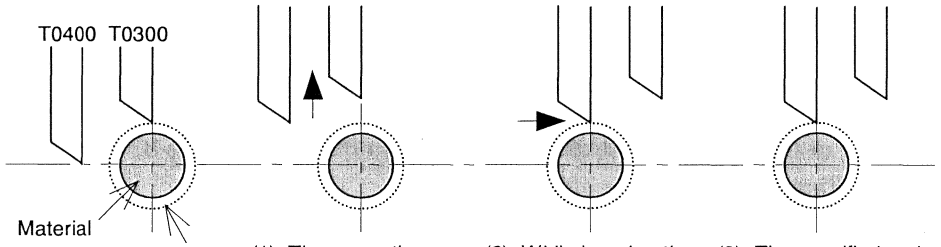
  Selection of a tool T0100 to T0600 while a tool T0100 to T0600 is selected

The tool nose position of a tool T0400 to T0600 is shifted 10.0 mm (radius value) from the tool nose position of a tool T0100 to T0300 into the axial direction. Therefore, if a command to select a tool T0400 to T0600 is issued while a tool T0100 to T0300 is selected, the tool T0400 to T0600 is called after the tool T0100 to T0300 has moved to the position determined by (Bar Stock O.D. + Tool Positioning Point (DIA) + tool shift amount 10 mm + maximum diameter value of tool set data of T0100 to T0600). Conversely, if a command to select a tool T0100 to T0300 is issued while a tool T0400 to T0600 is selected, the tool shift amount 10 mm is not added.

Operation when a tool T0400 to T0600 is selected while a tool T0100 to T0300 is selected

(1) The currently selected tool moves to the position ("Tool Positioning Point (DIA)" + tool shift amount 10.0mm + maximum diameter value of tool set data) at the rapid feed rate.
(2) While keeping the X1 axis at the position (1), the machine moves the specified tool to the position (Y1 axis) at the rapid feed rate.
(3) The specified tool moves to the positioning point at the rapid feed rate.

Example: T0300 → T0400



(1) The currently selected tool moves to the position ("Tool Positioning Point (DIA)" + tool shift amount 10mm + maximum diameter value of tool set data).

(2) While keeping the position ("Tool Positioning Point (DIA)" + tool shift amount 10mm + maximum diameter value of tool set data), the machine moves the specified tool to the position.

(3) The specified tool moves to the positioning point.

Operation when a tool T0100 to T0300 is selected while a tool T0400 to T0600 is selected

(1) The currently selected tool moves to the position ("Tool Positioning Point (DIA)" + maximum diameter value of tool set data) at the specified feed rate.
(2) While keeping the X1 axis at the position (1), the machine moves the specified tool to the position (Y1 axis) at the rapid feed rate.
(3) The specified tool moves to the positioning point at the rapid feed rate.

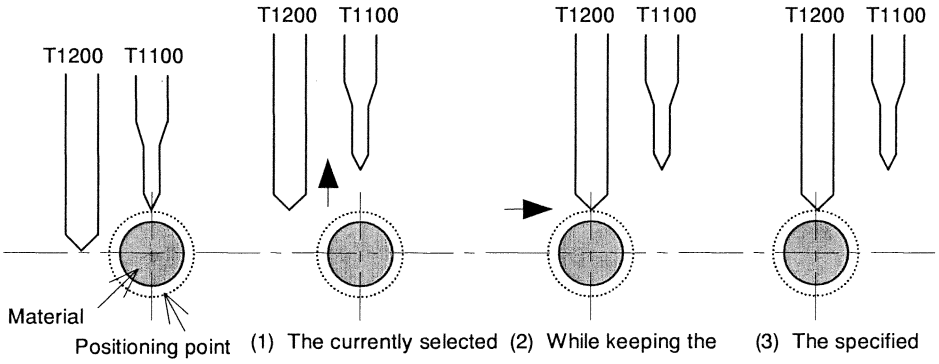
Selection of a tool T1100 to T1400 (Type VI: T1100 to T1300) while a tool T1100 to T1400 (Type VI: T1100 to T1300) is selected

A tool T1100 to T1400 is mainly used as a rotary tool for secondary machining. When you want to make a through-hole in a workpiece, mount the rotary tool at a position shifted from the standard tool nose into the axial direction. In this case, set the shift amount (diameter value) as the diameter value of tool set data.

Operation when a tool T1100 to T1400 is selected while a tool T1100 to T1400 is selected

(1) The currently selected tool moves to the position ("Tool Positioning Point (DIA)" + maximum diameter value of tool set data) at the specified feed rate.
(2) While keeping the X1 axis at the position (1), the machine moves the specified tool to the position (Y1 axis) at the rapid feed rate.
(3) The specified tool moves directly to the positioning point at the rapid feed rate.

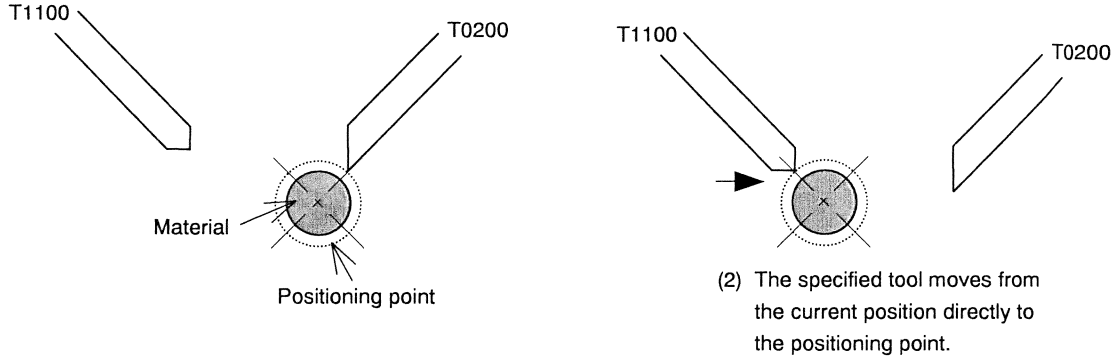
Example: T1100 → T1200 (Set the shift amount as the diameter value of tool set data of T1200.)



Selection of a tool T1100 to T1400 (Type VI: T1100 to T1300) while a tool T0100 to T0600 is selected, or selection of a tool T0100 to T0600 while a tool T1100 to T1400 (Type VI: T1100 to T1300) is selected

(1) The currently selected tool moves to the position ("Tool Positioning Point (DIA)" + diameter value of selected tool set data) at the rapid feed rate.
(2) The specified tool moves directly to the positioning point at the rapid feed rate.

Example: T0200 → T1100



Selection of a tool T1400 to T1700 while a tool T0100 to T1300 is selected. (Type VI)

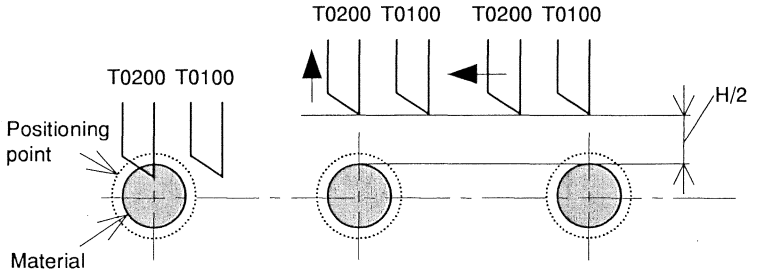
The specified tool-bit moves center of workpiece directly, when the vertical face holder (T1400 to T1700) is selected.

Selection of a tool T0100 to T1300 while a tool T1400 to T1700 is selected. (Type VI)

The specified tool moves to the positioning point directly, when a other tool is selected while the vertical face holder (T1400 to T1700) is selected. (=Q1)

- T□□ΔΔ H□

(1) The currently selected tool moves to the position ("Bar Stock O.D." in the machining data + value (diameter) specified by the H argument) at the rapid feed rate.
(2) While keeping the X1 axis at the position (1), the machine moves the specified tool to the position (Y1 axis) at the rapid feed rate.

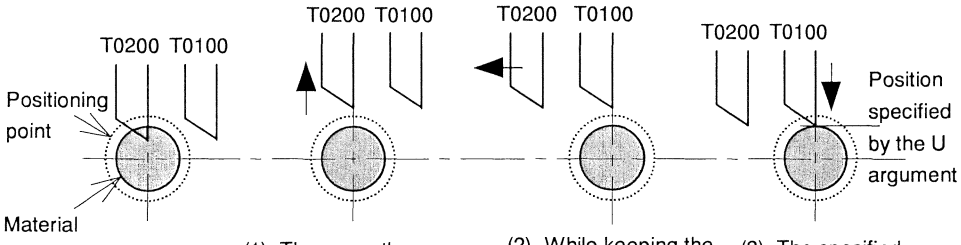


(1) The currently selected tool moves to the position ("Bar Stock O.D." + value specified by the H argument).

(2) While keeping the position "value specified for the material outer diameter + value specified by the H argument", the machine moves the specified tool to the position.

- T□□ΔΔ U□

(1) The currently selected tool moves to the position ("Bar Stock O.D."+ "Tool Positioning Point (DIA)" + maximum diameter value of tool set data) at the rapid feed rate.
(2) While keeping the X1 axis at the position (1), the machine moves the specified tool to the position (Y1 axis) at the rapid feed rate.
(3) The specified tool moves directly to the position specified by the U argument at the rapid feed rate.



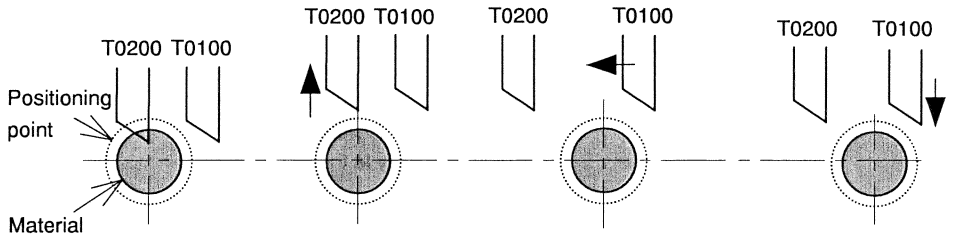
(1) The currently selected tool moves to the position ("Bar Stock O.D."+ "Tool Positioning Point (DIA)" + maximum diameter value of tool set data).

(2) While keeping the position "Tool positioning point + maximum diameter value of tool set data", the machine moves the specified tool to the position.

(3) The specified tool moves to the position specified by the U argument.

- T□□ΔΔ V□

(1) The currently selected tool moves to the position ("Bar Stock O.D."+ "Tool Positioning Point (DIA)" + maximum diameter value of tool set data) at the rapid feed rate.
(2) While keeping the X1 axis at the position (1), the machine moves the position specified by the V argument (Y1 axis) at the rapid feed rate.
(3) The specified tool moves directly to the positioning point at the rapid feed rate.



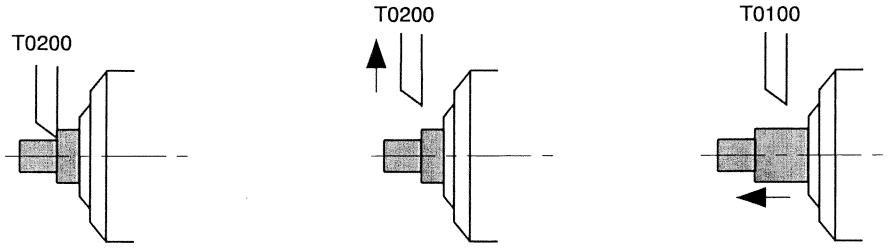
(1) The currently selected tool moves to the position ("Bar Stock O.D."+ "Tool Positioning Point (DIA)" + maximum diameter value of tool set data).

(2) While keeping the position "Tool positioning point + maximum diameter value of tool set data", the machine moves the position specified by the V argument.

(3) The specified tool moves to the positioning point.

- T□□ΔΔ W□

(1) The currently selected tool moves to the position ("Bar Stock O.D."+ "Tool Positioning Point (DIA)" + maximum diameter value of tool set data) at the rapid feed rate.
(2) While keeping the X1 axis at the position (1), the machine moves the specified tool to the position (Y1 axis) at the rapid feed rate.
(3) The specified tool moves to the positioning point ("Bar Stock O.D." + "Tool Positioning Point (DIA)" + diameter value of tool set data of the specified tool), and the Z1 axis moves directly to the position specified by the Z argument at the rapid feed rate.

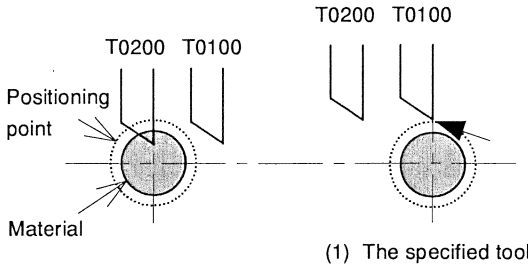


(1) The currently selected tool moves to the position ("Bar Stock O.D."+ "Tool Positioning Point (DIA)" + maximum diameter value of tool set data).

(3) The specified tool (X1 axis) moves to the positioning point, and the Z1 axis moves to the position specified by the W argument.

- T□□ΔΔ Q1

(1) The specified tool moves from the current position directly to the positioning point at the rapid feed rate.



(1) The specified tool moves from the current position directly to the positioning point.

Notes

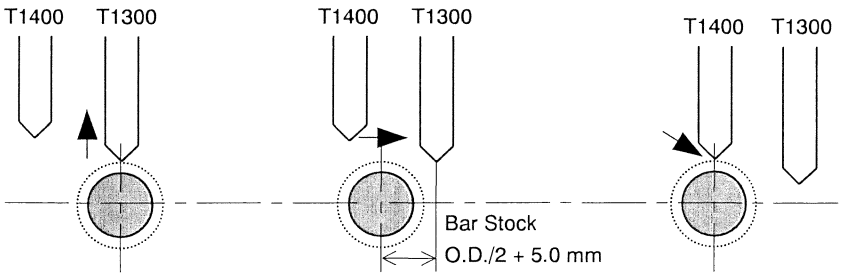
- The operation sample uses GTF 6010 + U30B holder.
- The following explains the relationship between Bar Stock O.D. in machining data, Tool Positioning Point (DIA), and maximum diameter value of tool set data when U30B is mounted.

Set the value depending on the tool number:	With T1100 to T1300: Bar Stock O.D. in machining data + Tool Positioning Point (DIA) + maximum diameter value of tool set data =	a ≤ 82.0 mm
	With T1400:	a ≤ 54.0 mm

If a ≥ 54.0 mm when a tool T1100 to T1300 is used, operation at selection of the tool T1400 is performed for avoiding the OT area as follows:

Selection of the tool T1400 while a tool T1100 to T1300 is selected

(1) The currently selected tool moves to the position ("Tool Positioning Point (DIA)" + maximum diameter value of tool set data) at the rapid feed rate.
(2) While keeping the X1 axis at the position (1), the machine moves the tool T1300 at the rapid feed rate until the center of the tool has passed the position determined by (Bar Stock O.D./2 (radius value) + 5.0 mm (drill diameter).
(3) The specified tool moves directly to the positioning point at the rapid feed rate.



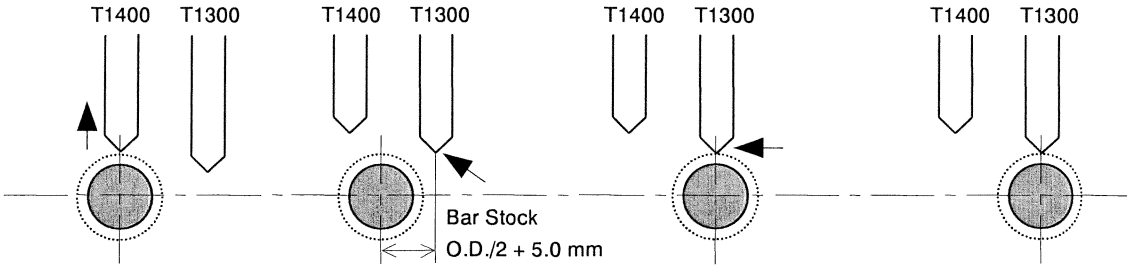
(1) The currently selected tool moves to the position ("Tool Positioning Point (DIA)" + maximum diameter value of tool set data).

(2) While keeping the position "Tool Positioning Point (DIA) + maximum diameter value of tool set data," the machine moves T1300 until it goes beyond the position "Bar Stock O.D. /2 + 5.0 mm."

(3) The specified tool moves from the current position directly to the positioning point.

Selection of a tool T1100 to T1300 while the tool T1400 is selected

(1) T1400 moves to the "Tool Positioning Point (DIA)" at the rapid feed rate.
(2) X1 axis moves directly to the position determined by "Tool Positioning Point (DIA) + maximum diameter value of tool set data" and Y1 axis moves the tool T1300 at the rapid feed rate until the center of the tool goes beyond the position determined by the "Bar Stock O.D./2 (radius value) + 5.0 mm (drill diameter)."
(3) While keeping the X1 axis at the position (2), the machine moves the specified tool to the position (Y1 axis) at the rapid feed rate.
(4) The specified tool moves directly to the positioning point at the rapid feed rate.



(1) T1400 moves to the "Tool Positioning Point (DIA)" at the rapid feed rate.

(2) X1 axis moves directly to the position "Tool Positioning Point (DIA) + maximum diameter value of tool set data" and Y1 axis moves T1300 beyond the position "Bar Stock O.D./2 + 5.0 mm."

(3) While keeping the position "Tool positioning point + maximum diameter value of tool set data", the machine moves the specified tool to the position.

(4) The specified tool moves from the current position to the positioning point.

5.5.4 Front drill (T2100 to T2600): Type VII, VIII, IX

Command format

T□□ΔΔ U[ ] R0, 1 C[ ] B3 Z1 K2

Argument

- U[ ]: Specify the X2 axis work coordinate after tool selection. It the U argument is not specified, the tool moves to the center (0) of workpiece. (The U1 argument is supported in macro Ver 002-001 and later.)
- R0: Specify this argument not to move the gang tool post while the G670 command (sequential machining in a single axis control group) is in progress if the previously selected tool number is of a tool on the gang tool post (T0100 to T0600 or T1100 to T1400). The gang tool post retracts to a safety position if the R0 argument is not specified. (The R0 argument is supported in macro Ver 002-001 and later.)
- R1: A front drill is selected after the gang tool post has moved to a safe position. The safe position means that the opposite tool post (X2 axis) can freely operate with its full stroke as long as the gang tool post is there.
- C[ ]: Specify the address of C axis to be used with machining.
  C1: C1 axis is used. (Treated as an axis of the axis control group)
  C2: C2 axis is used. (Treated as an axis of the axis control group)
  Default: No C axis is used. (Not treated as an axis of the axis control group)
- B3: Specify this argument to add the center data for back machining (with a tool T3100 to T3600). Default: The center data specified for front machining (with a tool T2100 to T2600) is used.
- Z1: Specify this argument to superimpose the Z2 axis on the Z1 axis for free machining in three axis control groups (G600). (The Z1 axis is a reference axis, and the Z2 axis is a superimposed axis.) In general, specify the Z1 and K2 arguments together. (Type VIII, IX)
- K2: Specify this argument to use the Z2 axis for free machining in three axis control groups (G600). The Z1 axis is used unless the argument is specified. (Type VIII, IX)

Note

- If you want to adjust the center for back machining in front/back simultaneous machining which is performed with the tools of tool numbers in the T20's and T30's, specify B3 in the argument for tool numbers in the T20's. The center is usually adjusted for front machining.
- The U argument is invalid in front/back simultaneous machining which is performed with the tools of tool numbers in the T20's and T30's for free machining in three axis control groups (G600).
- To perform machining with superimposed (the Z2 axis superimposed on the Z1 axis, the Z3 axis superimposed on the Z2 axis, and the X3 axis superimposed on the X2 axis), specify the queuing command before and after the selection of the tools of T01 and tool numbers in the T20's and T30's for free machining in three axis control groups (G600). At this time, do not specify an offset number for the last 2 digits of the 4-digit T code. Specify the offset number in the 2-digit T command after tool selection. In the case of type VIII, after superimposition of the axes, specify the queuing command before and after the selection of the tools of tool numbers in the T20's and T30's. Specify the same value for the first digits of the tool numbers in the T20's and T30's. (Type VIII, IX)

5.5.5 Back drills (T3100 to T3600): Type VII, VIII, IX

Command format

T□□ΔΔ U[ ] W[ ] Q1 B2 S1 R0 X2 Z2 I3 K2

Argument

- U[ ]: Specify the X axis work coordinate after tool selection. If the U argument is not specified, the tool moves to the center of workpiece. (The U argument is supported in macro Ver 002-001 and later.)

- W[ ]: Specify the position to which the back spindle retracts. (Distance from the retract position of the back spindle) If the W argument is not specified, the X2 axis starts moving when the back spindle (Z3 axis) has moved away 5.0 mm from the tool nose position where the maximum protrusion is specified.

- Q1: Specify this argument to select a tool without moving the currently machining tool (back spindle: Z3 axis) to the retract position. Only the X2 axis operates. If the Q1 argument is not specified, the X2 axis starts moving when the back spindle (Z3 axis) has moved away 5.0 mm from the tool nose position where the maximum protrusion is specified.

- Q3: Specify this argument to select a tool after moving the currently machining tool (back spindle: Z3 axis) to the retract position. (The Q3 argument is supported in macro Ver 005-001 and later.)

- B2: Specify this argument to add the center data for front machining (with a tool T2100 to T2600). Default: The center data specified for back machining (with a tool T3100 to T3600) is used.

- S1: Specify this argument to perform front/back simultaneous machining with the tools of tool numbers in the T20's and T30's by using the G600 command (free machining in three axis control groups).

- R0: Specify this argument not to move the gang tool post while the G670 command (sequential machining in a single axis control group) is in progress if the previously selected tool number is of a tool on the gang tool post (T0100 to T0600 or T1100 to T1400). The gang tool post retracts to a safety position if the R0 argument is not specified. (The R0 argument is supported in macro Ver 002-001 and later.)

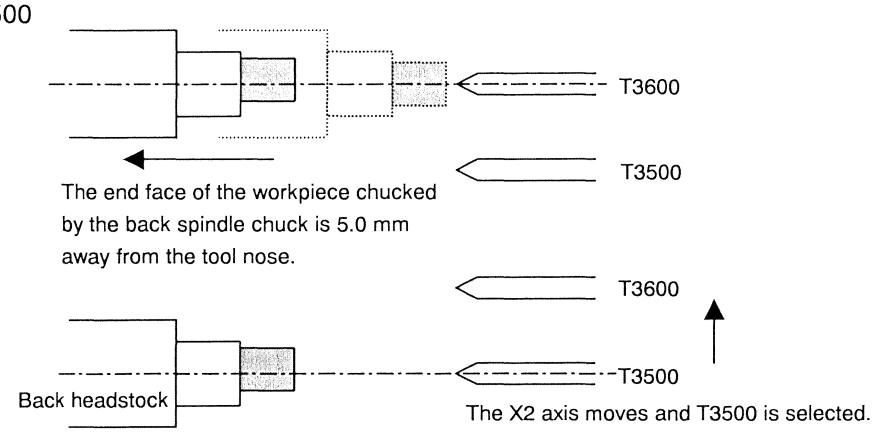
- X2: Specify this argument to superimpose the X3 axis on the X2 axis for free machining in three axis control groups (G600). In general, specify the X2 and I3 argument is specified. (Type IX)

- Z2: Specify this argument to superimpose the Z3 axis on the Z2 axis for free machining in three axis control groups (G600). (Type VIII, IX)

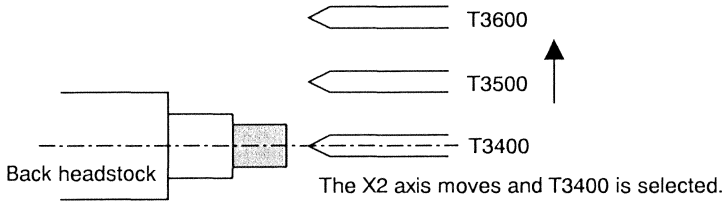
- I3: Specify this argument to use the X3 axis for free machining in three axis control groups (G600). The X2 axis is used unless the argument is specified. (Type IX)

- K2: Specify this argument to use the Z2 axis for free machining in three axis control groups (G600). The plus direction of the Z2 axis is reversed (mirror image). (Type VIII, IX)

Example: T3500



Example: T3400 Q1

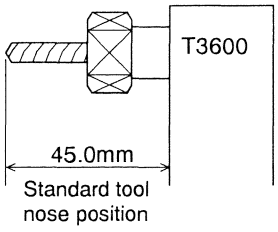


Note

- If you want to adjust the center for back machining in front/back simultaneous machining which is performed with the tools of tool numbers in the T20's and T30's, specify B3 in the argument for tool numbers in the T20's. The center is usually adjusted for front machining.
- The U argument is invalid in front/back simultaneous machining which is performed with the tools of tool numbers in the T20's and T30's for free machining in three axis control groups (G600).
- To perform machining with superimposition (the Z2 axis superimposed on the Z1 axis, the Z3 axis superimposed on the Z2 axis and the X3 axis superimposed on the X2 axis), specify the queuing command before and after the selection of the tools of T01 and tool numbers in the T20's and T30's for free machining in three axis control groups (G600). At this time, do not specify an offset number for the last 2 digits of the 4-digit T code. Specify the offset number in the 2-digit T command after tool selection. In the use of type VIII, after superimposition of the axes, specify the queuing command before and after the selection of the tools of tool numbers in the T20's and T30's. Specify the same value for the first digits of the tool numbers in the T20's and T30's. (Type VIII, IX)

CAUTION

If mounting a back drilling tool (T3100 to T3600) with the tool nose position beyond the standard position, be sure to set the protrusion from the standard tool nose position with a negative value, which is used for tool setting in the longitudinal direction. Failure to do so may cause interference.



5.5.6 Back spindle

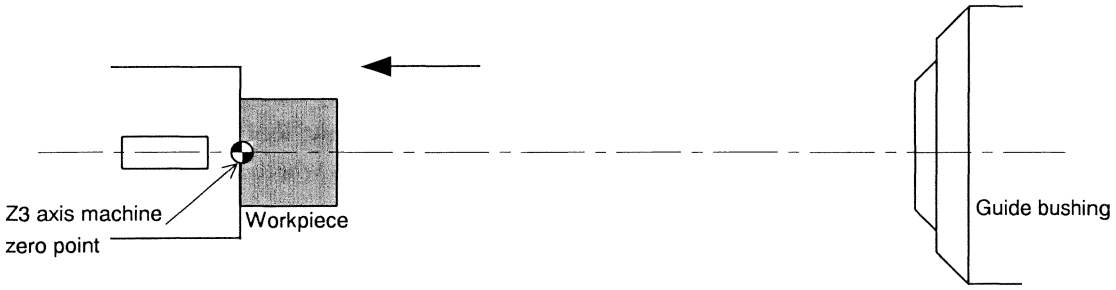
Command format

T3000 A1 D[ ] E[ ] S[ ] Z1

Operation sample

T3000

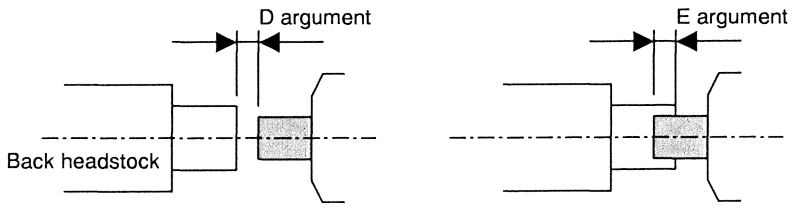
When selection of the back spindle is specified, the back headstock (Z3 axis) moves to the retract position.



Argument

- A1: When having created a program using only $1, specify this argument to perform pick-off operation. The axes of the axis control group are X1, Z1, Y1, and C1.
  This argument are enabled when the G670 command ($1 single machining) is executed.
- D[ ]: If you have specified the A1 argument, specify the position to which the back spindle (Z3 axis) advances to perform pick-off operation at the rapid feed rate. If this argument is not specified, the back spindle advances to the position 5.0 mm before the end face of a workpiece on the main spindle side.
- E[ ]: If you have specified the A1 argument, specify the position where the back spindle (Z3 axis) chucks a workpiece. An alarm is issued unless the E argument is specified.
- S[ ]: If you have specified the A1 argument, specify a feed rate (per minute) at which the back spindle (Z3 axis) advances to chuck a workpiece. An alarm is issued unless the S argument is specified.
- Z1: Specify this argument to superimpose Z3 axis (superimposed axis) on Z1 axis (reference axis).

Operation sample



5.5.7 Vertical face drill (T5400 to T5700): Type VI

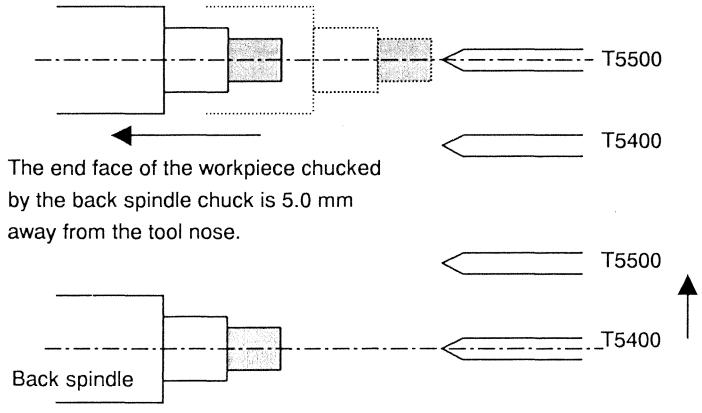
Command format

T□□ΔΔ U□□□ V□□□ W□□□ Q1 B1 S1

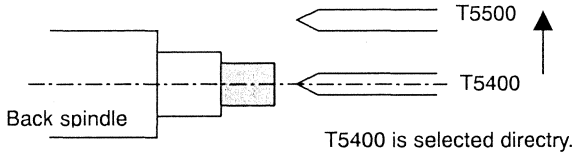
Argument

- U□□□: Specify the X1 axis work coordinate after tool selection. It the U argument is not specified, the tool moves to the center of workpiece.
- V□□□: Specify the Y1 axis work coordinate after tool selection. It the U argument is not specified, the tool moves to the center of workpiece.
- W□□□: Specify the position to which the back spindle retract. (Distance from the retract position of the back spindle) If the W argument is not specified, the tool is selected when the back spindle (Z3 axis) has moved away 5.0 mm from the tool nose position where the maximum protrusion is specified.
- Q1: Specify this argument to select a tool without moving the currently machining tool (back spindle Z3 axis) to the retract position. If the Q1 argument is not specified, the tool is selected when the back spindle (Z3 axis) has moved away 5.0 mm from the tool nose position where the maximum protrusion is specified.
- Q3: Specify this argument to select a tool after moving the currently machining tool (back spindle: Z3 axis) to the retract position.
- B1: Specify this argument to add the center data for front machining (with a tool T1400 to T1700). Default: The center data specified for back machining (with a tool T5400 to T5700) is used.
- S1: Specify this argument to perform front/back simultaneous machining with the tools of tool numbers in the T10's and T50's by using the G600 command (free machining in three axis control groups).

Example: T5400



Example: T5400 Q1

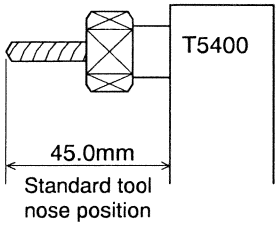


Notes

- If you want to adjust the center for back machining in front/back simultaneous machining which is performed with the tools of tool number in the T10's and T50's specify B5 in the argument for tool number in the T10's. The center is usually adjusted for front machining.
- The U, V arguments are invalid in front/back simultaneous machining which is performed with the tools of tool numbers in the T20's and T30's.

CAUTION

If mounting a Vertical face drill (T5400 to T5700) with the tool nose position beyond the standard position, be sure to set the protrusion from the standard tool nose position with a negative value, which is used for tool setting in the longitudinal direction. Failure to do so may cause interference.



5.5.8 T codes and arguments for each machining pattern

Machining pattern T code	Selectable tool Argument		T code		Argument																
			4 digits	2 digits	H	Q1	Q3	U	V	W	R1	R0	X	Z	C	B	S	A1	D	E	I	K
Machining pattern cancel Free machining in three axis control groups (G600)	$1 $2 $3	Tool on gang tool post	√	√	√	√	–	√	√	√	–	–	–	–	–	–	–	–	–	–	–	√
		Front drilling	√	√	–	–	–	–	–		√	–	–	√	√	√	–	–	–	–	–	√
		Back drilling	√	√	–	√	√	–	–	√	–	–	–	√	–	√	√	–	–	–	–	√
		Back spindle	√	√	–	–	–	–	–	–	–	–	–	√	–	–	√	√	√	√	–	–
		Vertical front drilling	√	√	–	–	–	√	√	√	–	–	–	–	–	√	–	–	–	–	–	–
		Vertical back drilling	√	√	–	√	√	√	√	√	–	–	√	–	–	√	√	–	–	–	√	–
$1 single machining Front/back simultaneous machining (G610) Type VII, VIII, IX	$1	Front drilling	√	√	–	–	–	–	–	–	√	–	–	–	√	√	–	–	–	–	–	–
	$2	Back drilling	√	√	–	√	√	–	–	√	–	–	–	–	–	√	√	–	–	–	–	–
Outer diameter/inner diameter simultaneous machining (G620) Type VIII, IX	$1	Tool on gang tool post	√	√	√	√	–	√	√	√	–	–	–	–	–	–	–	–	–	–	–	–
	$2	Front drilling	√	√	–	–	–	–	–	–	–	–	–	–	√	√	–	–	–	–	–	–
Front/back parallel machining (G630) Type VII, VIII, IX	$1	Tool on gang tool post	√	√	√	√	–	√	√	√	–	–	–	–	–	–	–	–	–	–	–	–
	$2	Back drilling	√	√	–	√	√	–	–	√	–	–	–	–	–	√	√	–	–	–	–	–
3-line simultaneous machining (G640) Type VIII, IX	$1	Tool on gang tool post	√	√	√	√	–	√	√	√	–	–	–	–	–	–	–	–	–	–	–	–
	$2	Front drilling	√	√	–	–	–	–	–	–	–	–	–	–	√	√	–	–	–	–	–	–
	$3	Back drilling	√	√	–	–	–	–	–	√	–	–	–	–	–	√	–	–	–	–	–	–
Pick-off Center support (G650)	$1	Back spindle	√	√	–	–	–	–	–	–	–	–	–	√	–	–	√	√	√	√	–	–
	$2		√	√	–	–	–	–	–	–	–	–	–	√	–	–	–	–	–	–	–	–
	$3		√	√	–	–	–	–	–	–	–	–	–	√	–	–	–	–	–	–	–	–
Front/back simultaneous machining (G660) Type VI	$1	Vertical front drilling	√	√	–	–	–	√	√	√	–	–	–	–	–	√	–	–	–	–	–	–
	$2	Vertical back drilling	√	√	–	√	√	–	–	√	–	–	–	–	–	–	–	–	–	–	–	–
		Tool on gang tool post	√	√	√	√	–	√	√	√	–	√	–	–	–	–	–	–	–	–	–	√
		Front drilling	√	√	–	–	–	–	–	–	√	√	–	–	√	√	–	–	–	–	–	–
					–	√	√	–	–	√	–	√	–	–	–	√	√	–	–	–	–	–
					–	–	–	–	–	–	–	–	–	√	–	–	√	√	√	√	–	–
					–	–	–	√	√	√	–	–	–	–	–	√	–	–	–	–	–	–
					–	√	√	√	√	√	–	–	–	–	–	√	√	–	–	–	–	–

not permitted

ents shown in the above table.
Special G Codes 1 and 2>.
spindle (T3000) can be specified only in $1.

6340

2609 1744

5.6 G Functions (G Codes)

A G function is expressed with G followed by a 2-digit number. It is also called a preparation function. G functions are provided for the NC unit to control axes (X1, Z1, Y1, X2, and Z3).

Command format

G□□

Examples: G00 Rapid feed positioning
G01 Linear interpolation

5.6.1 G code list

Once the G functions of groups A, D, E, F, H, L, and O are specified, they are enabled until another G function of the same group is specified. They are called modal G codes. The G functions of group C are enabled only when specified.

G code list

G code	Function	Group	Reference
*G00	Rapid feed positioning	A	Instruction Manual 5.4
G01	Linear interpolation	A	Instruction Manual 5.4
G02	Circular interpolation (clockwise)	A	Instruction Manual 5.4
G03	Circular interpolation (counterclockwise)	A	Instruction Manual 5.4
G04	Dwell	C	Instruction Manual 5.4
G12.1	Milling interpolation ON (optional)	L	Application Manual 8.5
G13.1	Milling interpolation OFF (optional)	L	Application Manual 8.5
G16	Y-Z cylindrical plane selection	E	
G17	X-Y plane selection	E	Basic Manual 5.10.3
*G18	Z-X plane selection	E	Basic Manual 5.10.3
G19	Y-Z plane selection	E	Basic Manual 5.10.3
G28	Reference point return	C	
G32	Single Point thread cutting	A	
*G40	Tool nose radius compensation cancel	J	Application Manual 8.2
G41	Tool nose radius compensation left ON	J	Application Manual 8.2
G42	Tool nose radius compensation right ON	J	Application Manual 8.2
*G43	Back spindle feed per rotation OFF	K	Basic Manual 5.6.2
G44	Back spindle feed per rotation ON	K	Basic Manual 5.6.2
G50	Coordinate system setting/Main spindle speed clamp setting	C	Instruction Manual 5.13 5.19
G68	X2 axis mirror image ON	O	Basic Manual 5.6.3
*G69	X2 axis mirror image OFF	O	Basic Manual 5.6.3
G76	Multiple repetitive threading cycle	H	
G79	Face drilling cycle (optional)	H	
G80	Synchronized tapping mode OFF (optional)	H	Application Manual 8.7
G84	Face cross tapping mode ON (optional)	H	Application Manual 8.7
G88	Synchronized cross tapping mode ON (optional)	H	Application Manual 8.7

G code	Function	Group	Reference
G90	Longitudinal turning canned cycle	H	
G92	Thread cutting canned cycle	H	Instruction Manual 5.14
G96	Constant surface speed control ON	F	Instruction Manual 5.19
*G97	Constant surface speed control OFF	F	Instruction Manual 5.19
G98	Millimeters per minute feed	D	
*G99	Millimeters per revolution feed	D	

Notes

- You can specify G codes of different groups together in the same block.
- You cannot specify a G code and T□□□□ together in the same block.
- You cannot execute a 3-digit G code in the MDI mode.
- Single block operation is disabled while G32, G76, G79, G84, G88, G90, or G92 is enabled. If any of the G codes is called during single block operation, the operation continues until the G code is canceled (until G0 or G1 or G2, G3 is executed). If the axis control group $1, $2, or $3 enters such a state, all the other axis control groups operate continuously.
- The * prefixed to G codes indicate the specification in the power-on state.

5.6.2 Back spindle feed-per-rotation

The feed-per-rotation command can be specified for the back spindle.

Command format

G43	Back spindle feed-per-rotation OFF
G44	Back spindle feed-per-rotation ON

In general, the feed-per-rotation speed is determined according to feed rate of the main spindle. If G44 is specified when machining workpieces with the back spindle, the feed-per-rotation for the specified axis control group is determined by the feed rate of the back spindle.
When a machining pattern is switched, the machining pattern command automatically turns ON/OFF the back spindle feed-per-rotation.
The following table shows back spindle feed-per-rotation ON/OFF for each machining pattern:

Machining pattern	$1	$2	$3
Machining pattern cancel, Free machining in three axis control groups (G600)	OFF	OFF	OFF
$1 single machining, Front/back simultaneous machining (G610) Type VII, VIII, IX	OFF	ON	OFF
Outer diameter/inner diameter simultaneous machining (G620) Type VIII, IX	OFF	OFF	OFF
Front/back parallel machining (G630) Type VII, VIII, IX	OFF	ON	OFF
3-lines simultaneous machining (G640) Type VIII, IX	OFF	OFF	ON
Pick-off (G650)	OFF	ON	OFF
Front/back simultaneous machining (G660) Type VI	OFF	ON	OFF
Sequential machining in a single axis control group (G670)	OFF	OFF	OFF

Note

Specify back spindle feed per rotation ON/OFF when machining workpieces with the back spindle by executing the G600 command (free machining in three axis control groups) or G670 command ($1 single machining).

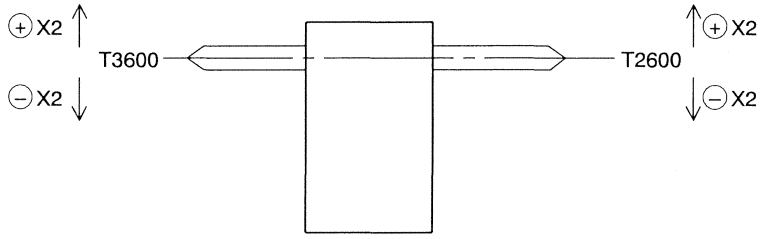
5.6.3 X axis mirror image

G68 and G69 are provided to reserve the signs of X axis. Specify the relevant G code to change the sign of the value as the infeed amount of a front/back drill in the diametrical direction.

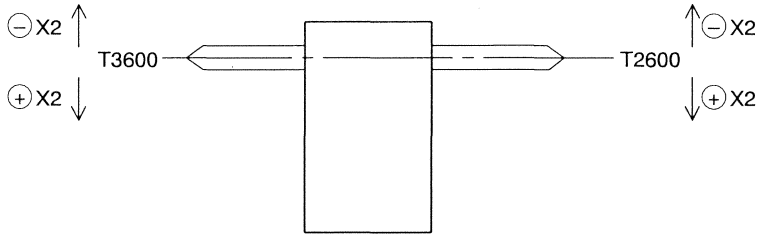
Command format

G68	X axis mirror image ON
G69	X axis mirror image OFF

In the case of X2 axis
X2 direction when the power is turned on or G69 is executed:



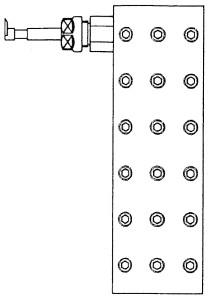
X2 direction when G68 is executed:



Program sample

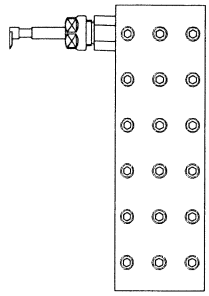
When a tool is mounted at T3600 with the tool nose upward:

$1	$2	$3
	: T3600 G00 X13.0 Z-0.5 T36 G00 X8.0 G01 Z5.0 F□ X13.0 F□ G00 X13.0 Z-0.5 T00 T3300 :	The tool performs infeed into the plus direction in the diametrical direction. Thus, G68 and G69 are not specified.



When a tool is mounted at T3600 with the tool nose downward:

$1	$2	$3
	: T3600 G68 ·····	G68 is executed to perform infeed into the plus direction in the diametrical direction.
	G50 U□ ·····	G50 (coordinate system shift command) is executed after G68 if necessary.
	G00 X13.0 Z-0.5 T36 ·····	The tool nose radius compensation function is enabled after execution of G68 if necessary.
	G00 X8.0 G01 Z5.0 F□ X13.0 F□ G00 X13.0 Z-0.5 T00 ·····	Execute the T00 command together with the X, Z axes operation command before execution of G69.
	(or G00 Z-0.5 T00 G00 X13.0 (U0))	
	G50 U□ ·····	G50 (coordinate system shift command) is executed before execution of G69.
	G69 ·····	After cutting is completed, G69 must be executed before tool selection by which the infeed direction in the diametrical direction is returned to the original.
	T3300 :	



Notes

- G68 or G69 command enables for X2 axis (the opposite tool post) or X3 axis (the back headstock).
- Do not start tool selection while G68 X axis mirror image ON is enabled. Specify G68 X axis mirror image OFF, and then start tool selection.
- While G68 X axis mirror image ON is enabled, the work coordinate of X axis is displayed with the sign opposite to the sign of the actually specified position. For example, when X axis moves to the position +A, the work coordinate is displayed as "–A."
- G68 must be specified while X-Z plane mode (G18) is being selected. Switching the plane mode must be executed after executing the G68 command.
- Do not specify G68 or G69 command while in milling mode (G12.1).

CAUTION

Finish tool selection before specifying G68 X axis mirror image ON.
If you want to enable tool nose radius compensation, do not specify the last 2 digits of a command expressed with T followed by a 4-digit number, but specify a command expressed with T followed by a 2-digit number. Conversely, if you want to cancel tool nose radius compensation, specify G68 X axis mirror image OFF after the T00 command. At this time, execute the T00 command together with the X-axis operation command. An incorrect specification order causes the work position of X axis to become invalid. Failure to do so may cause interference.

5.7 Special G Codes 1 and 2

The commands of G codes 1 and 2 are specific to this machine. They enable machining that extends over multiple axis control groups.
You can simplify programs (e.g., decrease of operating time) and improve productivity by using the commands effectively.

Command format

G□□□

5.7.1 List of special G codes 1

G code	Function	Reference
G226	G610 (front/back simultaneous machining) cancel	Basic Manual 5.7.3
G231	Pick-off cancel	Basic Manual 5.7.4
G600	Machining pattern cancel, Free machining in three axis control groups	Basic Manual 5.3.3
G610	$1 single machining, Front/back simultaneous machining	Basic Manual 5.3.4
G620	Outer diameter/inner diameter simultaneous machining	Basic Manual 5.3.5
G630	Front/back parallel machining	Basic Manual 5.3.6
G640	3-line simultaneous machining	Basic Manual 5.3.7
G650	Pick-off, Center-support	Basic Manual 5.3.8
G660	Front/back simultaneous machining	Basic Manual 5.3.9
G670	Sequential machining in a single axis control group	Basic Manual 5.3.10
G999	Last program execution	Basic Manual 5.7.5

Axis control groups

Specify the above G commands, except G226 and G231, for all the axis control groups $1, $2, and $3. The program automatically enters $1, $2, and $3 in the queuing state in order to synchronize them with each other, and then it proceeds to the next operation.

Notes

- You cannot use special G codes 1 with M, S, or T codes in the same block.
- You cannot execute special G codes 1 (G226 excluded) in the MDI mode.

5.7.2 List of special G codes 2

G code	Function	Reference
G113	Spindle synchronization control cancel	Basic Manual 5.7.6
G813	Spindle synchronization control cancel	Basic Manual 5.7.6
G814	Spindle synchronization control	Basic Manual 5.7.6
G899	Non-conform material phase adjustment [$1 only]	Basic Manual 5.7.7

Axis control groups

Specify the G commands for only axis control group $1, $2, or $3 that needs them.

Notes

- You cannot use special G codes 2 with M, S, or T codes in the same block.
- You cannot execute special G codes 2 (G899 excluded) in the MDI mode.
- G codes G113 and G813 has the same function.

5.7.3 Superimposition cancel, G610 (front/back simultaneous machining) cancel (G226), Z2-axis mirror image cancel command (Type VIII, IX)

The G226 command cancels the superimposition of X3 axis (superimposed axis) on X2 axis (reference axis). (Type IX)
The G226 command cancels the superimposition of Z2 axis (superimposed axis) on Z1 axis (reference axis). (Type VIII, IX)
The G226 command cancels the superimposition of Z3 axis (superimposed axis) on Z2 axis (reference axis). (Type VIII, IX)
The G226 command cancels the superimposition of Z3 axis (superimposed axis) on Z1 axis (reference axis). The command can also cancel front/back simultaneous machining specified by the machining pattern command G610.
The G226 command cancels the mirror image of the Z2 axis. (Type VIII, IX)

Command format

```
G226 X3 Z2,3 S0 M0
```

Arguments

- X3: Specify this argument to cancel the superimposition of X3 axis (superimposed axis) on X2 axis (reference axis). (Type IX)
- Z2: Specify this argument to cancel the superimposition of Z2 axis (superimposed axis) on Z1 axis (reference axis). (Type VIII, IX)
- Z3: Specify this argument to cancel the superimposition of Z3 axis (superimposed axis) on Z2 (reference axis) and the superimposition of Z3 axis (superimposed axis) on Z1 (reference axis). (Type VIII, IX)
- S0: Specify this argument to cancel front/back simultaneous machining with the tools of tool numbers in the T20's and T30's specified by G610.
- M0: Specify this argument to cancel the mirror image of the Z2 axis. (Type VIII, IX)

Program sample

Superimposition cancel

```
$1
:
T3000 A1 E10.0 S1000 Z1 ·····Z1-Z3 superimposition
:
G226 Z3 ·····Z1-Z3 superimposition cancel
:
```

G610 (front/back simultaneous machining) cancel

$1	$2	$3
: G610 T2100 : !2L1 T2200 ····· $1 single machining :	 G610 S1···· Front/back simultaneous machining T3100 : G226 S0 ···· Front/back simultaneous machining cancel !1L1	G610

Notes

- Superimposition can be automatically canceled by switching the machining pattern or executing the M2 command instead of using G226.
- G226 S0 is required to cancel G610 (front/back simultaneous machining) only when front/back simultaneous machining is switched to $1 single machining. Front/back simultaneous machining is also automatically canceled when front/back simultaneous machining by G610 is switched to the machining pattern by another machining pattern command or when the M2 command is executed.
- Do not specify X3, Z2 and Z3 argument in the same block. Specify these argument to the respectively separate block. (Following)

  G226 Z2
  G226 X3

5.7.4 Pick-off cancel (G231)

During cut-off machining in pick-off mode, this command retracts the back spindle at an arbitrary position of X1 axis being in the cut-off process.

Command format

G231 X[ ] F[ ] A[ ] Z[ ]

Axis control groups

Specify this command for $1.

Argument

- X[ ]: Specify the X1 axis work coordinate as the end position in the cut-off process.
- F[ ]: Specify the X1 axis feed rate in the cut-off process.
- A[ ]: Specify the X1 axis position for start-position queuing. When the X1 axis reaches the work coordinate specified by the A argument during cut-off machining, the back headstock (Z3 axis) start moving backward. For the A argument, specify a numeric value less than 0. However, you can 0 or a greater numeric value when 0 is not specified for Tubing Bar Stock I.D. in the machining data. An alarm is issued unless the A argument is specified.
- Z[ ]: Specify a machine position as the retract position of the back spindle (Z3 axis). If the Z argument is not specified, the axis moves to the machine zero point.

Program sample

$1	$2	$3
G600	G600	G600
:		:
G99 M03 S1=1500 M24 S2=1500		:
T0100		:
G00 X13.0 Z30.0 T01		:
G650 ····· Pick-off	G650 ····· Pick-off	G650 ····· Pick-off
	G00 Z–2.0	
	G98 G01 Z10.0 F1000	
	G4 U0.5	
	M15	
!3L1	!1L1	
G231 X–3.0 F0.03 A–1.0 ····· Pick-off cancel		
(When the X1 axis reaches the work coordinate –1.0, the back headstock moves to the retract position.)		
:	:	:

Notes

- The superimposition of the Z1 and Z3 axes is canceled when the G231 command is executed.
- The G231 does not permit arc cut-off machining.

5.7.5 Last program execution (G999)

This command must be specified in the last portion (end process) of each axis control group ($1, $2, $3) program that includes the last program.
In general, the last program is executed to perform back machining for workpieces with which front machining is completed. The back machining is performed in the last cycle while the machine is in the stopped state (e.g., 1-cycle stop or product counting by the counter).
Specify the G999 command for each axis control group to automatically enter the axis control groups in the queuing state. The last program between G999 and N999 is executed in the 1-cycle or 1-block operation mode.
Be sure to specify the N999 command at the end of the last program contents of each axis control group.
To finish program creation, specify three commands following N999 at the end of the end process. The commands must be specified in the sequence of M02, M99 and then %.

Command format

```
G999
```

Axis control groups

Specify this command for all the axis control groups $1, $2, and $3.

Program sample

$1	$2	$3
G610	G610	G610
M56		
G999 ····· Last program	G999 ····· Last program	G999 ····· Last program
: } Contents of the last program	: } Contents of the last program	: } Contents of the last program
N999	N999	N999
M02	M02	M02
M99	M99	M99
%	%	%

Notes

- Cancel the coordinate system shift command and compensation command before executing the G999 command.
- Specify the machining pattern command to execute the last program after the G999 command.
- See <Section 7.5.2 Last part function>.
- The cut-off tool on the gang tool post has been moved to the cut-off end position when the last program is executed. Consequently, if a tool T3100 to T3600 is selected, the tool mounted at T2100 to T2600 may interfere with the tool on the gang tool post. To avoid this problem, move away the gang tool post before executing the last program.
  The following shows a program sample:

$1	$2	$3
:	:	:
G999	G999	G999
G630	G630	G630
M6	M23 S2=[ ] G99	
G4 U0.2		
G0 W-1.0 ····· The Z1 axis is moved away.		
M151 ····· The gang tool post is moved away.		
!2L3	!1L3	
M3 S1=[ ] G99		
	T3600	
	:	
	T3500	
	:	
	M147	
	M25	
	M33	
!2L4	!1L4	
T0100 ····· Cut-off tool selection.		
G0 X17.0 W1.5		
G1 X-3.0 F[ ] ···· The end face of the material is cut off.		
M5		
M7		
G4 U0.2		
G0 W-0.5 ····· The Z1 axis is returned to the original position.		
N999	N999	N999
M2	M2	M2
M99	M99	M99
%	%	%

- To execute the last program using G610 (front/back simultaneous machining), specify dummy tool numbers in the T20's for $1.
  The following shows a program sample:

$1	$2	$3
:	:	:
G999	G999	G999
G630	G630	G630
M6 — Processing to move away the gang tool post	M23 S2=[ ] G99	
G4 U0.2		
G0 W-1.0		
M151		
M3 S1=[ ] G99		
G610	G610 S1	G610
T2600 ····· Selection of a dummy tool number	T3600	
	:	
T2500 ····· Selection of a dummy tool number	T3500	
	:	
	M147	
	M25	
	M33	
!2L4	!1L4	
G630	G630	G630
T0100		
G0 X17.0 W1.5		
G1 X-3.0 F[ ] — Processing to cut off the end face of the workpiece		
M5		
M7		
G4 U0.2		
G0 W-0.5		
N999	N999	N999
M2	M2	M2
M99	M99	M99
%	%	%

5.7.6 Spindle synchronization control (G114.1, G814, and G113) — optional

These commands control the speed and phase of a spindle in synchronous with another spindle.
Use a spindle synchronization control command when you need to synchronize two spindles with each other. For example, use the command in the following cases: the back spindle picks off the workpiece that is now chucked by the main spindle, a workpiece is machined chucked by both the main spindle and the back spindle, and a polygonal workpiece is pick off by adjusting the phases of the two spindles.
The G814 command defines the main spindle (S1) as the reference spindle and the back spindle (S2) as the synchronized spindle, and synchronizes the two spindles with each other.
The G114.1 command specifies a reference spindle and synchronized spindle, and synchronizes the two spindles with each other.
The G113 command cancels the synchronization of two spindles that was specified by a spindle synchronization command.

Command format

G814 R□	Spindle synchronization
G114.1 H□ D±□ R□	Spindle synchronization
G813	Spindle synchronization cancel
G113	Spindle synchronization cancel

Argument

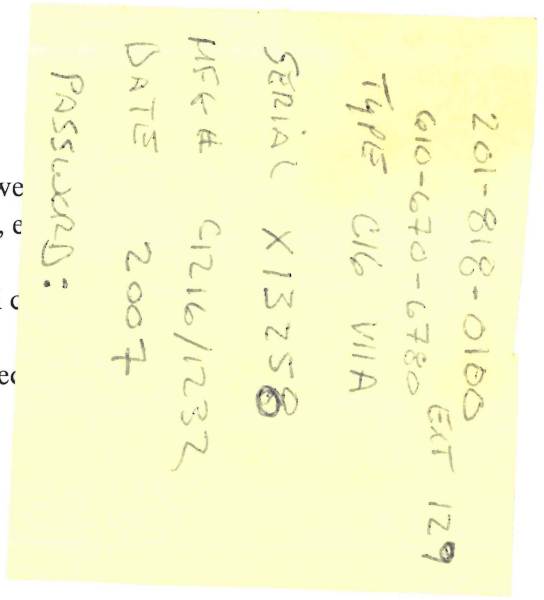
- R□: Specify the amount (angle) of the phase shift between synchronous spindles. The amount of the phase shift is automatically calculated by the non-conform material shift adjustment command (G899), and the calculation result is added to the stored value. The R argument is valid for a clockwise shift viewed from the spindle. If the R argument is not specified, the non-conform material shift adjustment command (G899) automatically calculates the amount of shift, and the stored value is used.
  Setting range: 0 to 359.999°
- H□: Specify the reference spindle and the direction in which the spindle rotates. When 1 is specified, the main spindle is selected.
- D±□: Specify the spindle to be synchronized and the rotating direction toward the spindle to be referenced. When – 2 is specified, the back spindle rotates backward.

Program sample

$1	$2	$3
: G99 M03 S1=1500 M24 S2=1500 G814 Spindle synchronization control T0100 G00 X13.0 Z30.0 T01 G650	G650 G00 Z-20.0 G98 G01 Z10.0 F1000 G4 U0.5 M77 ····· The state to wait for completion of spindle synchronization control is placed. M15 !1L1 :	G650
!2L1 G1 X-3.0 F[ ] G113 ····· Spindle synchronization cancel :		

Notes

- The spindle rotating under spindle synchronization control stops when the machine enters the emergency stop state. The spindle synchronization control mode is canceled at the same time.
- The speed clamp during spindle synchronization follows the reference spindle speed clamp value or the synchronized spindle speed clamp value whichever is smaller.
- Indexing for the reference spindle or the synchronized spindle is not permitted in the spindle synchronization control mode. To index either spindle, cancel the spindle synchronization control mode.
- Higher speeds at chucking tend to introduce large variations at synchronization. To get the back spindle to pick-off a workpiece under spindle synchronization control, set the spindle speed to 2500 min^{-1} or less (Speed-regulated area).
- The R argument specifies an amount (angle) of the phase shift between synchronous spindles. The synchronized spindle is shifted from the reference spindle by the phase shift amount specified by the R argument and rotates synchronizing with the reference spindle.
  Unless the R argument is specified in the G814 command, the value stored by executing the non-conform material phase adjustment command (G899) in the MDI mode is used as the amount of shift.
  Unless the R argument is specified in the G114.1 command, the phase is not shifted.
- The spindle synchronization control is canceled by pressing the RST key when all spindles stop (0 min^{-1}, indexing excluded).
- G813 and G113 have the same function.
- The state to wait for completion of spindle synchronization control is placed by executing M77 after G814 or G114.1.



5.7.7 Non-conform material phase adjustment (G899)

A phase shift occurs between the spindle and guide bushing and betwe
spindle when machining non-conform materials (hexagonal material, e
the values of phase shifts in the memory of the NC unit.
Be sure to specify this command when mounting the profile-material c
The command is for adjustment and is not written in the program.
Specify M26 (adjustment of the guide bushing phase) before workpie
in the material replacement program.

Command format

```
G899  R1  K1
```

Axis control groups

Specify this command for $1. You can execute the command in only the MDI mode.

Argument

- R1: Specify this argument to store the values of phase shifts (back spindle included) in the memory of the NC unit. The back spindle rotates when the G899 command is executed.
- K1: This argument does not permit the back chuck to close.

Procedure

Adjusting the phase between the main spindle and guide bushing

1. Put a material (non-conform material) through the main spindle and guide bushing.
2. Execute G899 in the MDI mode. The main spindle chucks the material.

Adjusting the phase between the main spindle and back spindle

1. Put a material (non-conform material) through the main spindle, guide bushing, and back spindle.
2. Execute G899R1 in the MDI mode. The main spindle and back spindle chuck the material.

Notes

- Specify the R1 argument whenever using the K1 argument. They must be specified in the format of G899R1K1.
- Specify G814 (spindle synchronization control) before getting the main spindle and the back spindle to chuck a workpiece. Consequently, the phase shift stored by G899 is automatically corrected.

5.8 M Functions (M Codes)

5.8.1 M code list

M code	Function	Remarks	Reference
M00	Program Stop	Automatic operation is stopped.	
M01	Optional Stop	Automatic operation is stopped when the switch is turned ON.	
M02*	Cycle Stop	1-cycle stop point	Instruction Manual 5.6
M03	Main Spindle Forward Rotation	The main spindle starts rotating forward.	Basic Manual 5.4
M04	Main Spindle Reverse Rotation	The main spindle starts rotating backward.	Basic Manual 5.4
M05	Main Spindle Stop	The main spindle stops rotating.	Basic Manual 5.4
M06*	Main Spindle Chuck Close	The main spindle chuck closes.	
M07*	Main Spindle Chuck Open	The main spindle chuck opens.	
M08	Enable Bar Stock Exchange Program	The material replacement program starts.	Instruction Manual 5.17.1
M09	Bar Stock Exchange Program Complete	The material replacement program terminates.	Instruction Manual 5.17.1
M10*	Knock-Out Advance	The knock-out axis of the back spindle moves forward.	Basic Manual 5.11.6
M11*	Knock-Out Retract	The knock-out axis of the back spindle moves backward.	Basic Manual 5.11.6
M15*	Back Spindle Chuck Close	The back spindle chuck closes.	
M16*	Back Spindle Chuck Open	The back spindle chuck opens.	
M18*	Enable Main Spindle C-Axis	Sequential operation of spindle C axis ON (optional)	Instruction Manual 5.21.3
M20	Main Spindle Index And C-Axis Release	Spindle servo OFF	Instruction Manual 5.21.1
M23	Back Spindle Forward Rotation	The back spindle starts rotating forward.	Basic Manual 5.4
M24	Back Spindle Reverse Rotation	The back spindle starts rotating backward.	Basic Manual 5.4
M25	Back Spindle Stop	The back spindle stops rotating.	Basic Manual 5.4
M26	Execute Guide Bushing Phasing	The phases of the spindle and guide bushing are aligned.	Basic Manual 5.7.7
M28*	Main Spindle Index	Sequential operation of main spindle indexing The 15° interval and 1° interval are optional.	Instruction Manual 5.21.1
M31	Workpiece Conveyor On	The workpiece conveyor is turned ON. (Optional)	
M32*	Front Basket Positioning		Basic Manual 5.11.4
M33*	Pick-off And Product Separation		Basic Manual 5.11.5
M34*	Sequential operation of workpiece unloading	(Optional)	
M35*	Product Unloading Movement Command	The workpiece unloading device is moved to unload products from the machine. (Optional)	
M36	Rotary Guide Bushing Torque Lower Limit L Selection	The guide bushing torque is set to 25%.	
M37	Rotary Guide Bushing Torque Limit Off	The guide bushing torque is set to 100%.	
M38	Bar Loader Torque Off, Synchronization Control On	Synchronization of the Z1 axis and bar loader ON (optional)	
M39	Bar Loader Torque On, Synchronization Control Off	Synchronization of the Z1 axis and bar loader OFF (optional)	
M43	Tip detection flag reset	The tip detection flag is reset. (optional)	
M46	Bar Loader Torque 0	(Optional)	
M48*	Enable Back Spindle C-Axis	Sequential operation of back spindle C axis ON (optional)	Basic Manual 5.10.2
M50	Back Spindle Pick-Off Failure Detection	Rotate the spindle at a low speed after cut-off machining, and enter M50.	Application Manual 8.14
M51*	Tool Breakage Detection On	A cut-off tool break is detected. (optional)	Application Manual 8.15
M52	Coolant On	Coolant starts discharging.	
M53	Coolant Off	Coolant stops discharging.	

M code	Function	Remarks	Reference
M54	Bar Loader Torque Off	The bar loader feed is stopped. (Optional)	Instruction Manual 5.17.2
M55*	Bar Loader Start	Workpiece replacement is started. (Optional)	Instruction Manual 5.17.2
M56	Products Count	Products are counted.	Instruction Manual 5.6
M57	Cycle Stop (Sub Program)	1-cycle stop point (for subprograms)	
M58	Gang Tool Spindle Forward Rotation	The tool spindle of the gang tool post starts rotating forward.	Basic Manual 5.4
M59	Gang Tool Spindle Reverse Rotation	The tool spindle of the gang tool post starts rotating backward.	Basic Manual 5.4
M60	Gang Tool Spindle Stop	The tool spindle of the gang tool post stops rotating.	Basic Manual 5.4
M61 to M65	External M Code	Use the codes to add optional functions.	
M68	External M code and relay ON		
M69	External M code and relay OFF		
M72	Back Spindle Air Blow On	The back spindle air blower is turned ON. If a long workpiece separator is mounted, the long workpiece separator's hand closes.	
M73	Back Spindle Air Blow Off	The back spindle air blower is turned OFF. If a long workpiece separator is mounted, the long workpiece separator's hand opens.	
M74	Back Face Tool Spindle Forward Rotation	(Optional)	Basic Manual 5.4
M75	Back Face Tool Spindle Reverse Rotation	(Optional)	Basic Manual 5.4
M76	Back Face Tool Spindle Stop	(Optional)	Basic Manual 5.4
M77	Waiting For Completion Of Spindle Synchronization	The queuing state is placed until the main and back spindles are synchronized.	
M78*	Back Spindle Indexing	Sequential operation of back spindle indexing (optional)	Basic Manual 5.10.1
M79	Back Spindle Indexing Cancel. Back Spindle C-Axis Cancel	Back spindle drive OFF (optional)	Basic Manual 5.10.1
M80	Front Face Tool Spindle Forward Rotation	(Optional)	Basic Manual 5.4
M81	Front Face Tool Spindle Reverse Rotation	(Optional)	Basic Manual 5.4
M82	Front Face Tool Spindle Stop	(Optional)	Basic Manual 5.4
M84	Cutting start interlock enabled		Basic Manual 5.8.2
M85	Cutting start interlock disabled		Basic Manual 5.8.2
M86	Back Spindle Speed Fluctuation Detection On	The back spindle speed change detection function is enabled.	
M87	Back Spindle Speed Fluctuation Detection Off	The back spindle speed change detection function is disabled.	
M88	Back Spindle Torque Lower Limit L Selection		
M89	Back Spindle Torque Upper Limit H Selection		
M90	Back Spindle Torque Limit Off		
M92	Error Detect On	Error detection ON (Use M92 when edge precision is required.)	
M93	Error Detect Off	Error detection OFF	
M94	Master Measure Start	(Optional)	
M95	Work Measure Start	(Optional)	
M96	Main Spindle Speed Fluctuation Detection On	The main spindle speed change detection function is enabled.	Instruction Manual 5.18
M97	Main Spindle Speed Fluctuation Detection Off	The main spindle speed change detection function is disabled.	Instruction Manual 5.18
M98	Sub Program	Subprograms are called.	Instruction Manual 5.16.1
M99	Return To Main Program	Be sure to specify M99 at the end of the main program and subprograms.	Instruction Manual 5.16.2
M100	Grip Bar Stock	The bar loader clamps the material. (optional)	
M101	Release Bar Stock	The bar loader releases the material. (optional)	
M102	Release Bar Stock Quickly	The bar loader releases the material quickly. (optional)	
M103	Release Bar Stock, Slide Channel Forward	The bar loader releases the material, and the rail moves forward. (optional)	

M code	Function	Remarks	Reference
M104	Slide Channel Forward	The bar loader rail moves forward. (optional)	
M105	Slide Channel Back	The bar loader rail moves backward. (optional)	
M106	Stabilizer Close	The bar loader support closes. (optional)	
M107	Stabilizer Open	The bar loader support opens. (optional)	
M108	Separator advancement		
M109	Separator retraction		
M110	Workpiece separator's hand air blower ON	Air blower ON	
M111	Workpiece separator's hand air blower OFF	Air blower OFF	
M112	Input G/B Phase Data	In general, the phase is automatically controlled by G899.	
M113*	Series of Operation of Long Workpiece Device	(Optional)	Application Manual 8.12
M114	F/B Drill Holder Interference Check Invalid		
M116	Work Hand Close	(Optional)	
M117	Work Hand Open	(Optional)	
M118	Interference Check Disabled		Application Manual 8.16
M119	Interference Check Enabled		Application Manual 8.16
M120	Back Spindle Phase Memory	In general, the phase is automatically controlled by G899.	
M121	Measuring Jaw Close	(Optional)	
M122	Measuring Jaw Open	(Optional)	
M123	Automatic Measure Sensor Centering	(Optional)	
M124	Centering is Completed	(Optional)	
M125	NC Reset & Rewind	Used in macro (not used in ordinary operation)	
M126	Non-Conditional Optional Stop	Used in macro (not used in ordinary operation)	
M127	Request For Recalculation	Used in macro (not used in ordinary operation)	
M140*	Opposite tool post advance (Z2 axis)	Type Ⅷ	
M141*	Opposite tool post retraction (Z2 axis)	Type Ⅷ	
M142	Main spindle C axis holdup ON		
M143	Main spindle C axis holdup OFF		
M144	Back spindle C axis holdup ON		
M145	Back spindle C axis holdup OFF		
M147*	Opposite tool post retraction (X2 axis)		Basic Manual 5.8.4
M151*	Gang tool post retraction		Basic Manual 5.8.3
M155*	Sequential operation of bar stock exchange		Basic Manual 5.8.6
M160	S1 cutting start interlock ON		Basic Manual 5.8.2
M161	S1 cutting start interlock OFF		Basic Manual 5.8.2
M162	S2 cutting start interlock ON		Basic Manual 5.8.2
M163	S2 cutting start interlock OFF		Basic Manual 5.8.2
M164	S3 cutting start interlock ON		Basic Manual 5.8.2
M165	S3 cutting start interlock OFF		Basic Manual 5.8.2
M166	S4 cutting start interlock ON		Basic Manual 5.8.2
M167	S4 cutting start interlock OFF		Basic Manual 5.8.2
M171*	Back spindle moves backward.		Basic Manual 5.8.5
M180*	T code duplication enabled	If the alarm "EX528" is issued, specify this M code before selecting the T code.	
M181*	T code duplication enabled	If the alarm "EX528" is issued, specify this M code after selecting the T code.	

Notes

- The M codes with an asterisk (*) marked uses macros.
- Execute a macro-used M code independently. Do not execute the M code in the same block as for another code.
- The above M codes include options. For the options, see Chapter <3 Machine Specifications>.

5.8.2 Cutting start interlock enabled/disabled (M84, M85, and M160-M167)

The cutting start interlocks are automatically turned ON/OFF when the machining pattern is changed. In general, you do not have to turn ON/OFF the cutting start interlocks by using the M codes. Use the cutting start interlocks mainly to perform machining by executing G600 (free machining in three axis control groups). All the cutting start interlocks are OFF when the power is turned on.

Command format

M84	Cutting start interlock enabled (The cutting start interlock of a specified axis control group is set back to original.)
M85	Cutting start interlock disabled (All the cutting start interlocks of a specified axis control group are disabled.)
M160	S1 cutting start interlock enabled (Main spindle)
M161	S1 cutting start interlock disabled (Main spindle)
M162	S2 cutting start interlock enabled (Back spindle)
M163	S2 cutting start interlock disabled (Back spindle)
M164	S3 cutting start interlock enabled (Gang tool spindle)
M165	S3 cutting start interlock disabled (Gang tool spindle)
M166	S4 cutting start interlock enabled (Front/back face machining tool spindle)
M167	S4 cutting start interlock disabled (Front/back face machining tool spindle)

Relationship between the machining patterns and cutting start interlocks:

	S1 (Main spindle)			S2 (Back spindle)			S3 (Gang tool spindle)			S4 (Front/back face machining tool spindle)		
	$1	$2	$3	$1	$2	$3	$1	$2	$3	$1	$2	$3
G600	–	–	–	–	–	–	–	–	–	–	–	–
G610	√	–	–	–	√	–	–	–	–	√	–	–
G620	√	√	–	–	–	–	–	–	–	–	√	–
G630	√	–	–	–	√	–	√	–	–	–	√	–
G640	√	√	–	–	–	√	–	–	–	–	√	√
G650	√	–	–	–	√	√	√	–	–	–	√	√
G660	√	–	–	—	√	–	√	√	–	–	–	–
G670	√	–	–	√	–	–	√	–	–	√	–	–

√: Cutting start interlock enabled

5.8.3 Gang tool post retraction (M151)

The M151 command moves the gang tool post to the retract position. The gang tool post is moved to the safe position so that the opposite tool post (X2 axis) can freely operate with its full stroke. Specify the command mainly to perform front machining with a tool T2100-T2600 and back machining with a tool T3100-T3600.

Command format

```
M151
```

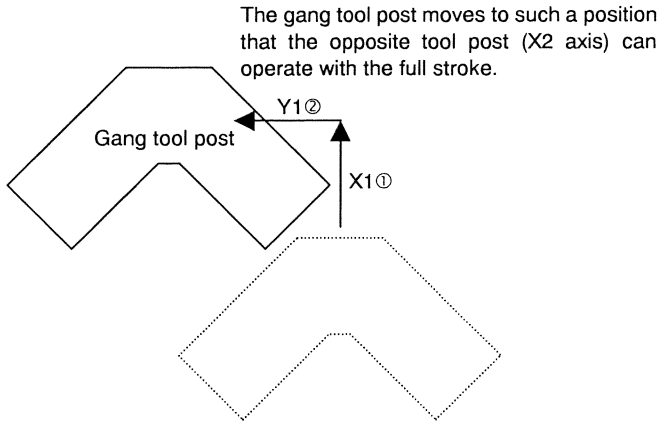
Axis control groups

Specify the M151 command for an axis control group in which X1 and Y1 axes are used. (Axis control group after selection of a gang tool)

Program sample

```
                    $1
G600
T0200
  :
M151 ····················· Gang tool post retraction
M162 ·············· S2 cutting start interlock enabled
M23 S2=
T3100
  :
```

Operation sample



Note

An alarm is issued if the M151 command is specified for an axis control group in which X1 and Y1 are not used.

5.8.4 Opposite tool post retraction (M147): Type VII, VIII, IX

The M147 command moves the opposite tool post (X2 axis) to the machine zero point. Execute the command after completion of machining with tools T2100-T2600 or T3100-T3600 on the opposite tool post.

Command format

```
M147
```

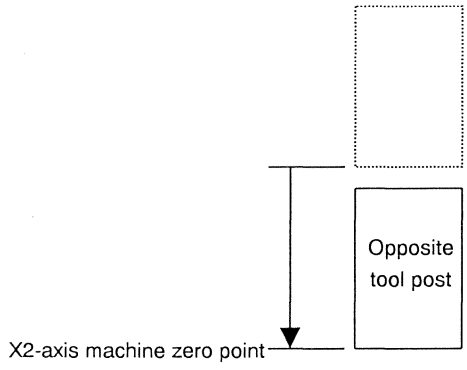
Axis control groups

Specify this command for $1, $2 or $3

Program sample

$1	
G610	
T2100	
:	
M147	The opposite tool post retracts.
G630	
T0200	
:	

Operation sample



5.8.5 Back headstock retraction (M171)

The M171 command moves the back headstock (Z3 axis) to the machine zero point.

Command format

```
M171
```

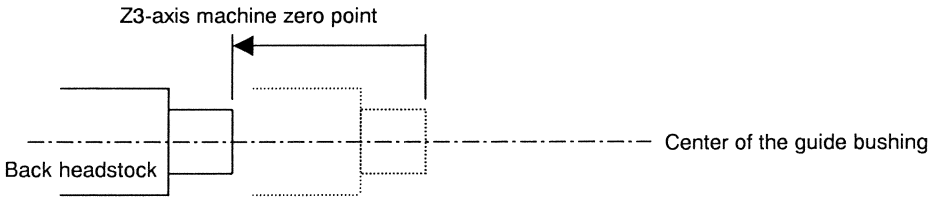
Axis control groups

Specify this command for $1, $2 or $3

Program sample

$1
G50 Z0
M33 ····· Sequential operation of product collection by the basket
:
T0100
G00 X17.0 Z30.0 T01
M32 X1.0 ····· The basket (back headstock) advances.
G01 X–3.0 F[ ]
M171 ····· The back headstock retracts.
:

Operation sample



5.8.6 Sequential operation for bar stock replacement (M155)

If the machine is equipped with an automatic magazine bar loader, you can use the M155 command to remove a bar remnant and supply bars.

Command format

M155 U[ ] A[ ] X[ ] Z[ ] K1 W[ ] F[ ] S[ ] R1 M1 T[ ] B[ ]

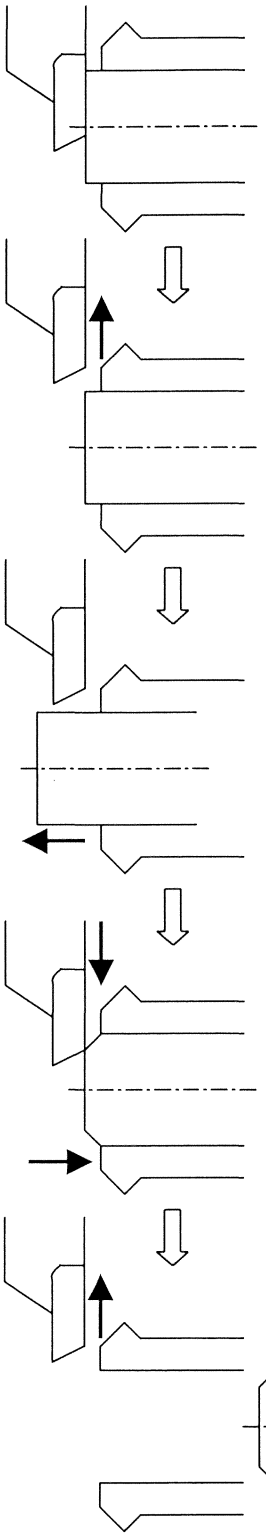
Arguments

- U[ ]: Specify the positioning point for deburring a bar remnant. The default is Tool Positioning Point in the machining data.
- A[ ]: Specify the feed rate at which the axis moves to the positioning point for deburring the bar remnant. The default is F0.1.
- X[ ]: Specify the distance by which the X axis moves for deburring the bar remnant. Deburring is not performed unless the X argument is specified.
- Z[ ]: Specify the distance by which the Z axis moves for deburring the bar remnant. Deburring is not performed unless the Z argument is specified.
- K1: Specify this argument not to stop coolant before drawing the bar remnant. The coolant stops if the K1 argument is not specified.
- W[ ]: Specify the distance by which the Z axis moves when drawing the bar remnant and when supplying bars. The default is 30 mm.
- F[ ]: Specify the feed rate at which the Z axis moves when drawing the bar remnant and when supplying bars. The default is F0.5.
- S[ ]: Specify the speed at which the main spindle rotates when drawing the bar remnant and when supplying bars. The default is 1000 min^{-1}.
- R1: Specify this argument not to stop the main spindle after drawing the bar remnant. The main spindle stops if the R1 argument is not specified.
- M1: When using non-conform materials, specify this argument to adjust the guide bushing phase before workpiece insertion. The guide bushing phase is not adjusted unless the M1 argument is specified.
- T[ ]: Specify the dwell time from when the chuck closes after workpiece replacement. The default is 2 seconds.
- B[ ]: Specify the dwell time between the coolant oil ON and the cut-off machining . The default is 0 second. (The B argument is supported in macro Ver 004-001 and later.)

Operation sample

(1) The tool moves to the position determined by (Bar Stock O.D. in the machining data + value specified by the U argument) at the feed rate specified by the A argument.
(2) The Z axis advances the distance specified by the Z argument.
(3) The machine executes the chamfering function for the bar remnant at the Cut-Off Feed in the machining data, depending on the values of the X and Z arguments.
(4) The coolant is turned OFF. It is not turned OFF if the K1 argument is specified.
(5) The main spindle rotates at the speed specified by the S argument.
(6) The tool moves to the position determined by (Bar Stock O.D. in the machining data + value specified by the U argument). At the same time, the bar remnant is drawn by the amount specified by the W argument, from the guide bushing at the feed rate specified by the F argument.
(7) The main spindle stops. It does not stop if the R1 argument is specified.
(8) The feed torque of the automatic magazine bar loader is turned off.
(9) The main spindle chuck opens.
(10) Material replacement is carried out.
(11) The main spindle chuck closes.
(12) Dwell time specified by the T argument
(13) The main spindle rotates at the speed specified by the S argument.
(14) If the M1 argument is specified, the guide bushing phase is adjusted.
(15) A material is inserted by the amount specified by the W argument, at the feed rate specified by the F argument.
(16) The coolant is turned ON.
(17) The main spindle rotates at the Cut-Off Speed in the machining data.
(18) The machine performs cut-off machining until reaching the Cut-Off End (DIA) (in the machining data) at the Cut-Off Feed in the machining data.

Overview of deburring (chamfering) of a bar remnant



The tool moves to the position determined by (Bar Stock O.D. in the machining data + value specified by the U argument).

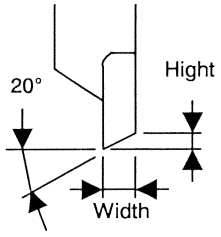
The Z axis advances the distance specified by the Z argument.

The machine executes the chamfering function for the bar remnant, depending on the values of the X and Z arguments.

The tool moves to the position determined by (Bar Stock O.D. in the machining data + value specified by the U argument). At the same time, the bar remnant is drawn by the amount specified by the W argument.

Example:
The following table shows the values of the X argument (distance by which the X axis moves for deburring) and Z argument (distance by which the Z axis moves for deburring) if C0.5 is to be set for the deburring (chamfering) of the bar remnant:
* Specify 1.0 for the U argument (positioning point for deburring) in this case.

Width	Hight	X argument	Z argument
2.0mm	0.728mm	3.4	1.7
3.0mm	1.092mm	4.2	2.1



Program sample

$1	$2	$3
: T0100 G00 X17.0 Z30.0 T01 G650	G650 G00 Z-1.0 G98 G01 Z10.0 F3000 G4 U0.5 M15	G650
!2L1 G99 G1 X-3.0 F0.03 M8 M8 /M155 X3.4 Z1.7 ···· Sequential operation for workpiece replacement M9 M5 M7 G00 X-3.0 Z0 T00 :	!1L1	

5.9 Exclamation (!) Symbol (Queuing Program)

To synchronize multiple axis control groups with each other, specify this function to place an axis control group's program in the queuing state while the operation of another axis control group's program is in progress.

Command format

!□ !□ L□ ..... Multi-axis control group program queuing

Argument

- !□: Specify the axis control group number (1 to 3) of an axis control group which enters the queuing state while the operation of another axis control group is in progress.
- L□: Queuing ID number (0 to 8999)

Program sample

Multi-axis control group program queuing pattern

$1	$2	$3
:	:	:
:	!1L1	:
!2L1 ←	:	:
:	:	:
:	!3L2 ←→	!2L2
:	:	:
:	:	:
:	!1!3L3 ←→	!1!2L3
!2!3L3 ←	:	:
:	:	:
:	:	:
:	:	:
!2!3L4 ←→	!1!3L4 ←→	!1!2L4
M56		
G999	G999	G999
N999	N999	N999
M02	M02	M02
M99	M99	M99
%	%	%

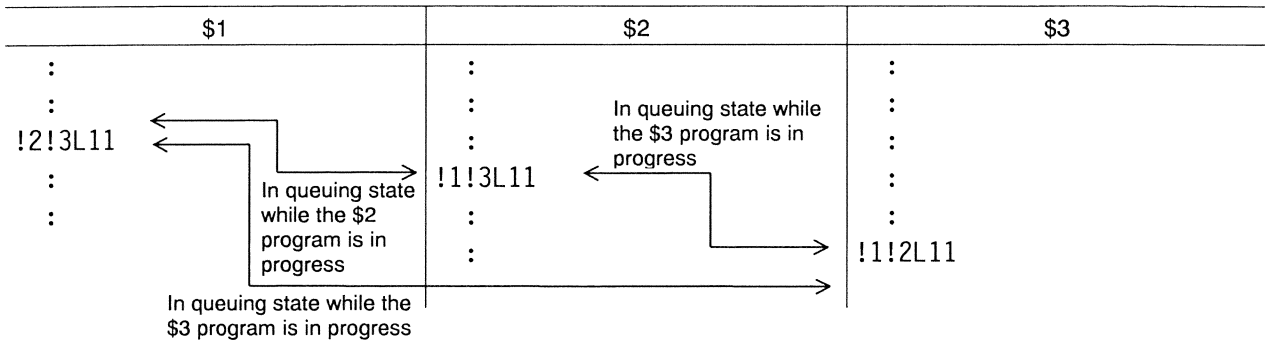
Notes

If a wrong queuing number is specified, the operation is unable to proceed because the program remains in the queuing state.

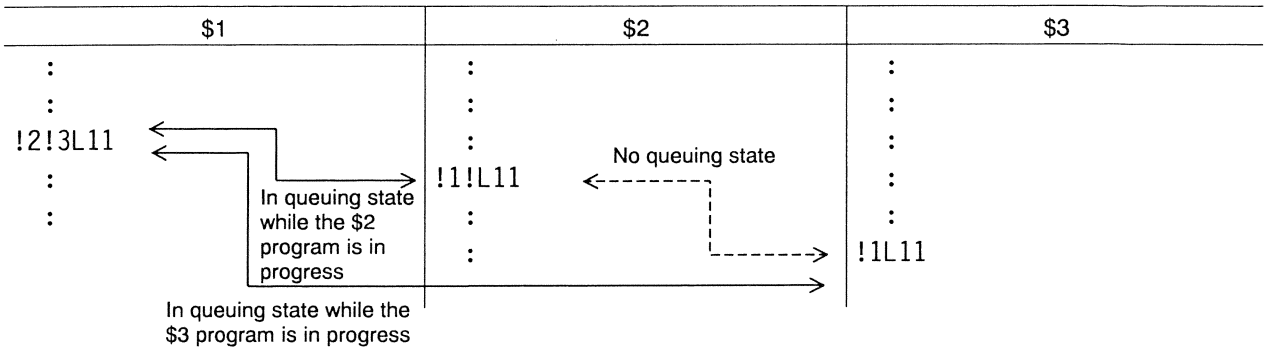
- To place the programs of all the three axis control groups in the queuing state, specify the two other axis control group numbers as well as the target axis control group number. If specification of another axis control group number and specification of two other axis control group numbers coexist using the same identification number, the program of an axis control group is unable to enter the queuing state. In this case, the program execution order is not as specified.

Program sample

a. Correct queuing pattern



b. Queuing pattern including both specification of another axis control group number and specification of two other axis control group numbers



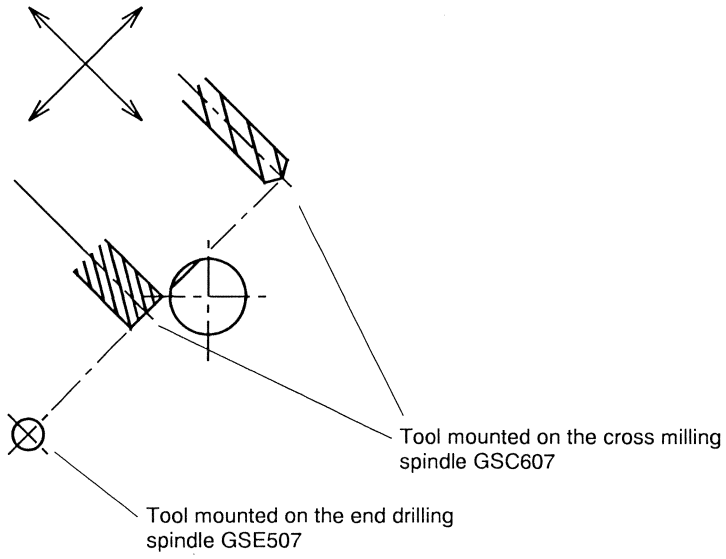
In the program sample b, !2!3 is specified for $1. Thus, the $1 program enters the queuing state while the $2 and $3 programs are in progress. However, since only !1 is specified for $2 and $3, the $2 and $3 programs do not enter the queuing state. If the $2 or $3 program enters the queuing state while the $1 program is in progress, the next block is executed. As a result, time for executing the program contents indicated by the identification number L11 is different between $2 and $3 depending on the program contents preceding L11, and the program execution order is not maintained as specified. This program sample may be useful for reducing operation time, but it is not recommended because it may also cause trouble such as interference when the program contents following L11 or the cutting condition are changed.

5.10 Secondary Machining Programs

5.10.1 Positional relation between spindles for secondary machining

Gang tool post

The following figure shows the spindle positions of the gang tool post for secondary machining when viewed from the end face of the material:



Note

Use the synchronous tapping function or floating chuck to perform tapping with a gang tool.

5.10.2 Transition between turning and secondary machining

Transition from turning to secondary machining

To switch from turning to secondary machining, specify the commands in the format shown below.

Program sample

- Main spindle indexing
  M05 G98 M58 S3=□□□□
  M28 S0

- Main spindle C axis indexing
  M05 G98 M58 S3=□□□□
  M18 C0

Functions of the words:

M05 .... Main spindle stop
G98 .... Millimeters per minute feed (mm/min)
M58 .... Rotate the tool spindle

	Gang tool post
Tool spindle forward rotation	M58
Tool spindle reverse rotation	M59
Tool spindle stop	M60

S3=□□□□ .... Speed command

S3	Tool spindle of the gang tool post

M28 S0 .... Return to zero point for main spindle indexing

When performing secondary machining consecutively, you do not have to enter the above commands for each secondary machining. Once the commands are entered at the beginning, they are enabled until the overall secondary machining is completed.

Note

The command "return to zero point for main spindle indexing" is used to maintain the same indexing position of a workpiece during secondary machining.
If a workpiece is machined in the sequence of turning, secondary machining, turning, and then secondary machining, this command enables the workpiece to always stop at the same position (the angle of rotation).

Transition from secondary machining to turning

To switch from secondary machining to turning, specify the commands in the format shown below.

Program sample

```
M60
M20
M03 S1=□□□□ G99
```

Functions of the words:

M60	.... Stop the tool spindle
M20	.... Main spindle index and C-axis release
G99	.... Millimeters per revolution feed (mm/rev)
S1=□□□□	.... Main spindle rotation
M03	.... Main spindle forward rotation

5.10.3 Back spindle indexing (M78 and M79) — optional

The back spindle can be indexed at 1-degree intervals. It is indexed in the direction in which the spindle rotates forward.

Command format

M78 S[ ]	Back spindle indexing command
M79	Back spindle indexing cancel command

Axis control groups

With the machining patterns G600, G610, and G630, specify the M78 command for $2.
Alternatively, specify the command for the axis control group after execution of a command T3100 to T3600 (Type VII, VIII), T5400 to T5700 (Type VI).

Argument

- S[ ]: Specify the indexing angle. The value must be specified in the range 0 to 359 with an absolute command.

Program sample

M25	.... The back spindle stops.
M78 S0	.... Back spindle indexing command
	Machining program
M79	.... Back spindle indexing cancel command
M23 S2=[ ]	.... The back spindle rotates forward.

Notes

If the back spindle is rotating, stop the back spindle temporarily to index it with the M78 S0 command. To return to the turning process, execute the M79 command to cancel back spindle indexing, then execute the M23 S2=[ ] command to rotate the back spindle.

5.10.4 Back spindle C axis indexing (M48) — optional

The back spindle C axis can indexed at 0.001-degree intervals.
The spindle C axis is turned ON when this command is executed. Zero point C0 is indexed, and then the coordinate system is set with the position as C0.
Use a plus (+) sign to express indexing in the direction in which the back spindle rotates forward, and define the indexing as "forward indexing". Use a minus (–) sign to express indexing in the direction in which the back spindle rotates backward, and define the indexing as "backward indexing". When viewed from the end face of the workpiece, the counterclockwise rotation indicates "forward indexing" and the clockwise rotation indicates "backward indexing".
A plus (+) sign is omitted.

Command format

M48 C□

Axis control groups

For the machining patterns G600, G610, and G630, specify the M48 command for $2.
Alternatively, specify the command for the axis control group after execution of a command T3100 to T3600 (Type VII, VIII), T5400 to T5700 (Type VI).

Argument

C□: Specify an indexing angle in the range 0 to 359.999°.

Note

When the M48 C□ command is specified, ordinary C-axis commands are enabled.
Example:

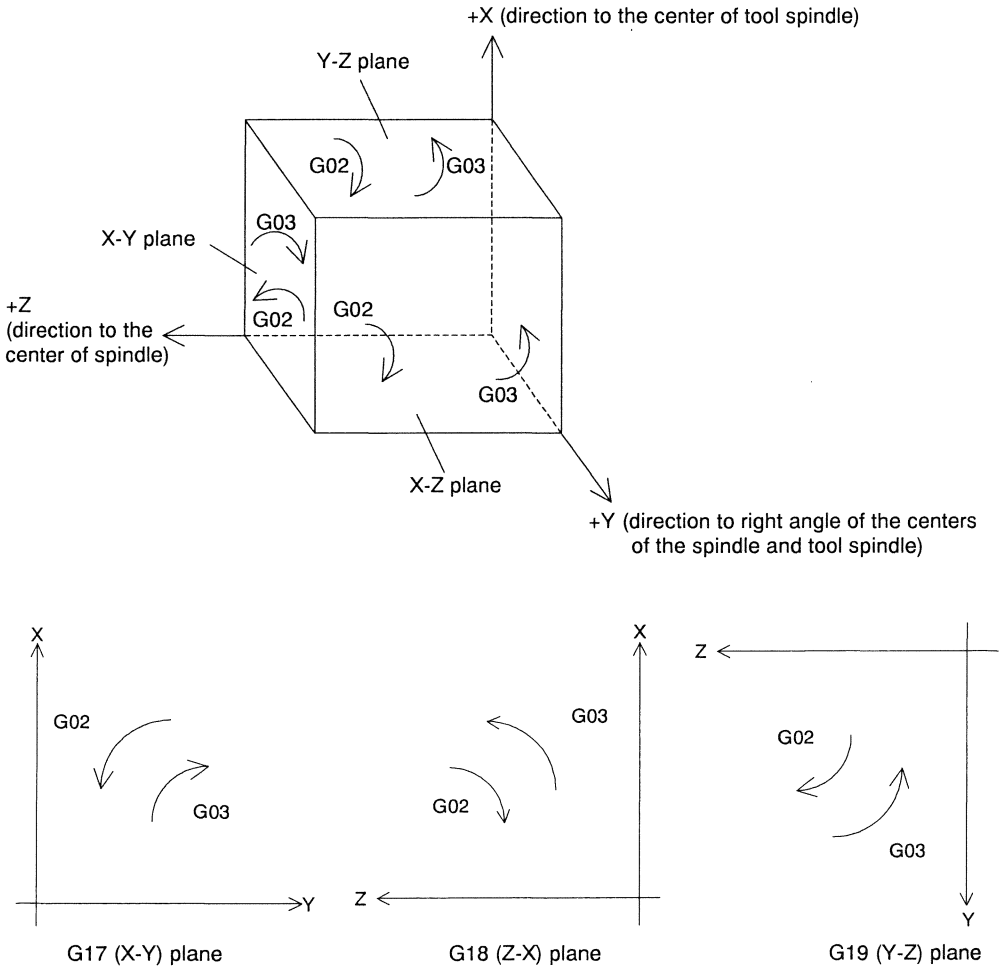
G50 C□	(H□)	H is an incremental command.
G00 C□	(H□)	
G01 C□	(H□) F□	

5.10.5 Circular interpolation in a secondary machining process

The circular interpolation feed is enabled on the following three planes in the secondary machining process:

X-Y plane: G17
Z-X plane: G18 · · · · Plane mode selected for ordinary turning process
Y-Z plane: G19

Generally, the Z-X plane (G18) is selected at power-on or reset. This plane is used for circular interpolation (G02 or G03) in the turning process. To use the circular interpolation on a different plane in the secondary machining process, execute the plane selection command G17 or G19.
Be sure to reselect the Z-X plane (G18) to return to the turning process.



Command format

```
G17
G18
G19
```

Axis control groups

Specify the commands for axis control groups in which circular interpolation is used.

5.10.6 Program sample including secondary machining processes

The following is a program sample in which machining is carried out in the sequence of outer diameter machining (turning), outer diameter drilling (secondary machining), slitting (secondary machining), and then cut-off machining (turning).

Program sample

$1	$2	$3
G50 Z0		
G600(G630)	G600(G630)	G600(G630)
G99 M06		
G00 X21.0 Z-0.5		
M03 S1=□□□□		
T□□□□		
N01		
G00 X□□□□ Z□□□□		
:		
Outer diameter machining		
N02		
:		
N03		
T1□□□		
G00 X□□□□ Z□□□□		
M05 G98 M58 S3=□□□□ ····· Command to switch to secondary machining process		
M28 S0		
:		
T1□□□		
N04		
:		
M60 ····· Command to switch to turning process		
M20		
M03 S1=□□□□ G99		
N05		
T0□□□		
G00 X□□□□ Z□□□□		
:		
M05		
M07		
G00 X-3.0 Z0 T00		
M56		
G999	G999	G999
N999	N999	N999
M02	M02	M02
M99	M99	M99
%	%	%

5.10.7 Program sample including main spindle indexing

Program sample

$1	$2	$3
G50 Z0 M06 G00 X□□□ Z-0.5 M03 S1=□□□ G99 T□□□ : G600(G630) T□□□ G00 X□□□ Z□□□ : M05 G98 ···· Spindle indexing for switching to secondary machining process, and zero point (0°) positioning. M58 S3=□□□ M28 S0 2-face width end milling G50 W-□□ G00 Y□□□ Z□□□ G01 Y□□□ F□□□ M28 S180.0 ····· Positioning at 180° from zero point G01 Y□□□ F□□□ M28 S90.0 ····· Positioning at 90° from zero point G50 W□□ T□□□ Key grooving G50 W-□□ G00 X□□□ Z□□□ G01 Z□□□ F100 G01 X□□□ F□□□ G50 W□□	G600(G630)	G600(G630)

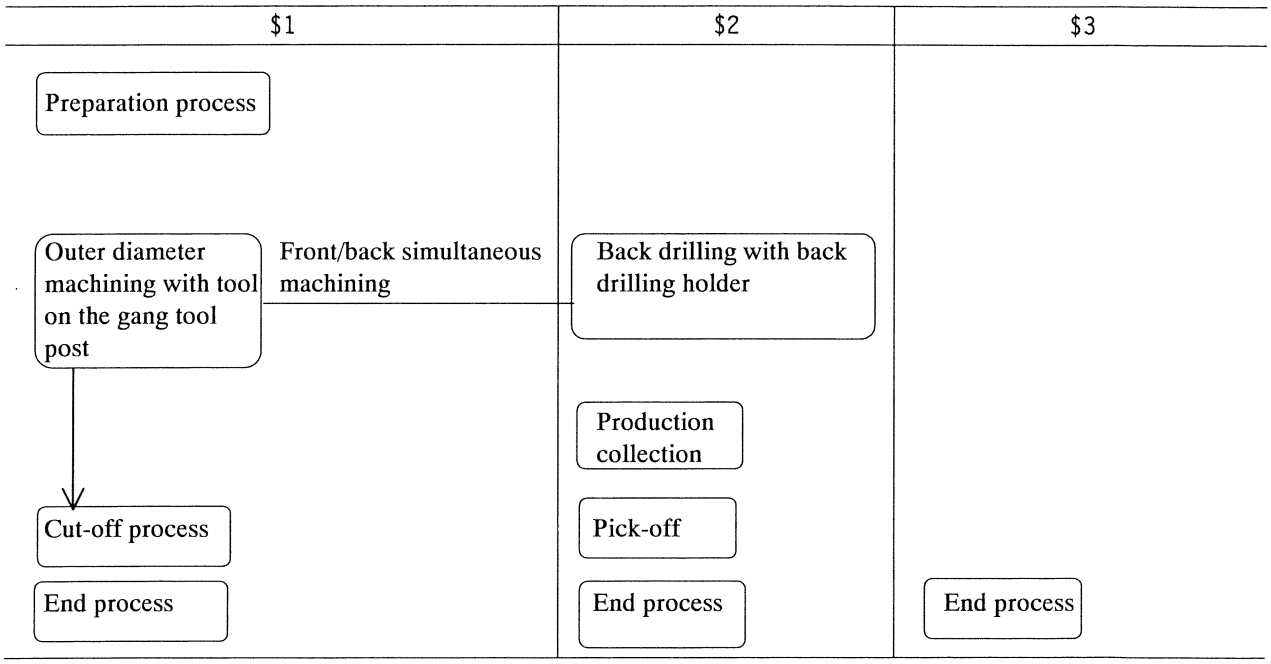
$1	$2	$3
M60 ····· Command to switch to turning process M20 M03 S1=[ ] G99 T[ ] : M05 M07 G00 X-3.0 Z0 T00 M56 G999 N999 M02 M99 %	G999 N999 M02 M99 %	G999 N999 M02 M99 %

5.11 Back Machining Programs

5.11.1 Back machining process

This section explains workpiece pick-off by the back spindle, machining with the back drilling holder.

Program sample

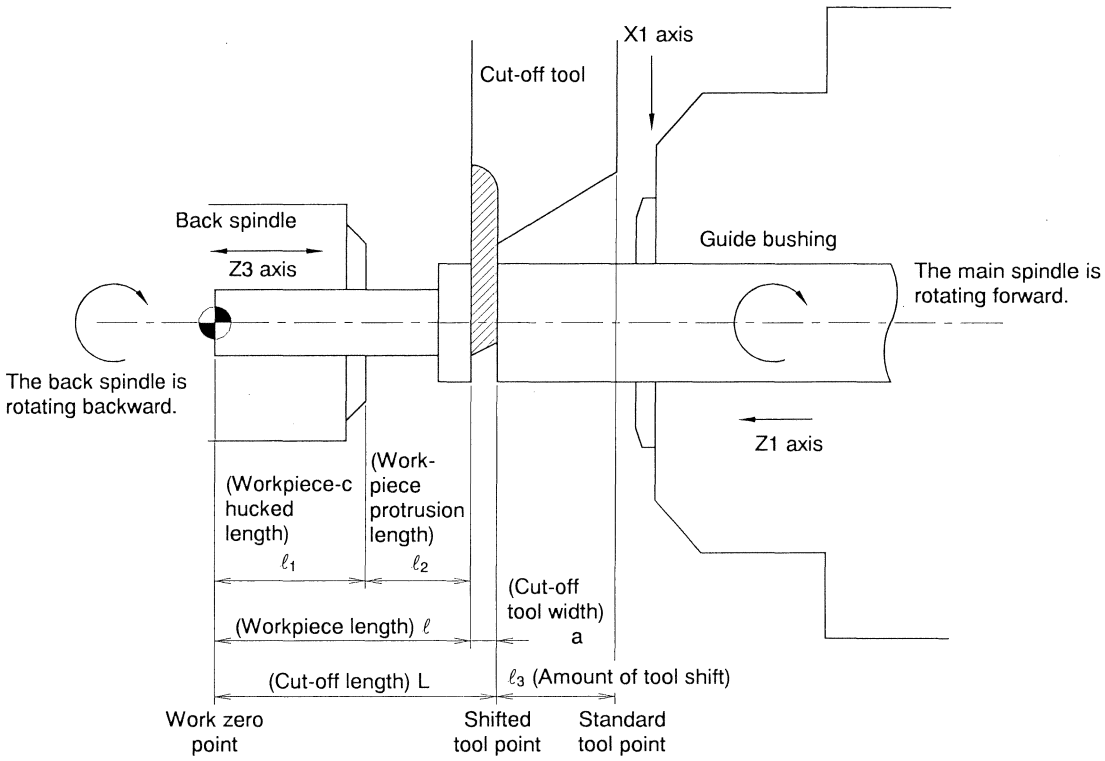


Notes

- When the program completes, the machine stops with the workpiece chucked to the back spindle.
- The program runs two cycles to complete the machining of a workpiece.
- To complete the machining of a workpiece in a cycle, include the back machining program in the last program execution (G999). The machine performs the back machining after the cut-off process to complete the workpiece in a cycle.
  Select the menu key [Last PRT] to execute the last program.

5.11.2 Pick-off and end process

The back spindle picks-off workpieces in the cut-off process.
Then, the back spindle moves to the return position, and the end process starts.



Program sample

$1	$2	$3
G630	G630	G630
:		:
:		:
G99 M03 S1=1500 M24 S2=1500		:
T0100		:
G00 X17.0 Z30.0 T[ ]		
G650 ····· Pick-off	G650 ····· Pick-off	G650 ····· Pick-off
	M72	
	G00 Z–2.0	
	G98 G01 Z10.0 F1000	
	G04 U0.5	
	M15 ····· The back spindle chuck closes.	
	M73	
!2L1	!1L1	
G01 X–1.0 F0.03 ···· Cut-off machining		
G630	G630	G630
X–3.0	:	:
:	:	:
M5		
M25		
:		

1. The back spindle rotates in the same direction and at the same speed as for the main spindle.

2. The cut-off tool is selected.

3. The tool moves to the position specified for cut-off machining in the longitudinal direction. (Z1 axis)

4. The pick-off command (G650) is executed. (Command for the superimposition of the end face of the workpiece (zero point) and the end face of the back spindle)

5. The back spindle moves to a position close to the end face of the workpiece.

6. The feed-per-minute command is executed to move the back spindle to the workpiece chuck position. (Z3 axis)

7. The back spindle chuck closes.

8. The queuing state is placed.

9. Cut-off machining starts.

10. The back spindle moves to the retract position.

11. The end process starts.

Notes

- Specify the machining pattern command G650 for all the axis control groups. This machining pattern automatically enters the queuing state. Cancel the coordinate system shift command and compensation command before executing the machining pattern command.
- Execute the pick-off command (G650). To move the back spindle to the position determined by the front work coordinates of the Z1 axis, superimpose the Z3 axis on the Z1 axis and specify the axis move command for $2.
- Position the Z1 axis at cut-off position before specifying the G650 command.
- To cut off a workpiece chucked by the back spindle, execute the M24 command (backward rotation) for the back spindle if the M03 command (forward rotation) has been executed for the main spindle. Conversely, execute the M23 command (forward rotation) for the back spindle if the M04 command (backward rotation) has been executed for the main spindle.

5.11.3 Back drilling tools (T3100 to T3600): Type VII, VIII, IX

Back drilling tools (T3100 to T3600) are used for the G600, G610, G630, G640 and G670 machining pattern.

With G630 back machining:

Program sample

$1	$2	$3
G630 ···· Front/back parallel machining	G630 ···· Front/back parallel machining	G630
T0200	T3600	
G00 X17.0 Z–0.5 T02	G00 Z–0.5	
G00 X15.0	G01 Z5.0 F[ ] T36	
G01 Z10.0 F[ ]	G00 Z–0.5 T00	
X17.0	T3500	
T0300	:	
:	:	
:		

With G610 back machining:

Program sample

$1	$2	$3
G610 ···· Front/back simultaneous machining	G610 S1 ···· Front/back simultaneous machining	G610
T2100	T3100	
G00 Z–0.5	G00 Z–0.5	
G01 Z5.0 F[ ] T21	G01 Z3.0 F[ ] T31	
G00 Z–0.5 T00	G00 Z–0.5 T00	
T2200	T3200	
:	:	

Notes

- Specify the machining pattern commands G610 and G630 for all the axis control groups. These machining patterns automatically enter the queuing state. Cancel the coordinate system shift command before executing the machining pattern commands.
- Each axis control group should hold the feed-per-rotation command and feed-per-minute command as modal functions.
- G610 and G630 are modal G codes that are enabled until another machining pattern command is executed.

5.11.4 Back drilling tools (T5400 to T5700): Type VI

Back drilling tools (T5400 to T5700) are used for the G600, G660, and G670 machining pattern.

With G600 back machining:

Program sample

$1	$2	$3
G600	G600 G44 ··· Back spindle feed-per-rotation ON M162 ····· S2 cutting start interlock enabled T5400 G00 Z-0.5 G01 Z5.0 F[ ] T34 G00 Z-0.5 T00 T5500 : :	G630

With G660 back machining:

Program sample

$1	$2	$3
G660 ···· Front/back simultaneous machining T1400 G00 Z-0.5 G01 Z5.0 F[ ] T14 G00 Z-0.5 T00 T1500 : :	G660 ····· Front/back simultaneous machining T5400 G00 Z-0.5 G01 Z5.0 F[ ] T34 G00 Z-0.5 T00 T5500 : :	G660

Notes

- When using G600 as a back machining, it is possible that a machining program create in $1.
- Specify the machining pattern commands G600 and G660 for all the axis control groups. These machining patterns automatically enter the queuing state. Cancel the coordinate system shift command before executing the machining pattern commands.
- Each axis control group should hold the feed-per-rotation command and feed-per-minute command as modal functions.
- G600 and G660 are modal G codes that are enabled until another machining pattern command is executed.

5.11.5 Basket advancement (M32)

Use this command to move the workpiece receiver mounted on the back spindle to the product collection position.

Command format

M32 X[ ] W0 Z[ ]

Axis control groups

Specify this command for $1, $2 or $3.

Argument

- X[ ]: Specify the X1 axis position for end-position queuing. When the X1 axis reaches the specified work coordinate during cut-off machining, positioning of the workpiece receiver of the back spindle is completed (product collection position). The end-position queuing is not performed unless X argument is specified.

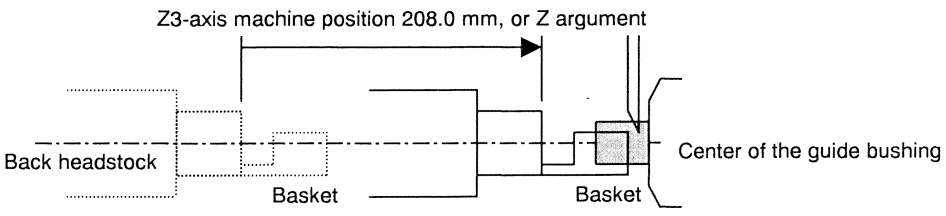
- W0: Specify this argument not to move the basket (back spindle Z3 axis) to the machine zero point. The basket moves to the machine zero point and advances if the W0 argument is not specified.

- Z[ ]: Specify the position to which the basket advances (back spindle Z3 axis) when cutting off a workpiece. The default is the machine coordinate 208.0 mm.

Operation

(1) An alarm is issued if the X2 axis is not at the machine zero point.
(2) The basket (back spindle Z3 axis) moves to the machine zero point. It does not move if the W0 argument is specified.
(3) The machine indexes the basket.
(4) The basket is positioned when the work position of the X1 axis has reached the value of the X argument. The basket starts advancing if the X argument is not specified.

Operation sample



If the X argument is specified, the basket is positioned when the work position of the X1 axis has reached the value of the X argument.

Notes

- Cut-off tool can be mounted at the position T01 or T04.
  Product receiver basket interferes in the tool when it mores forward the condition that cut-off tool is mounted at the other position.
- If the override is changed or operation is performed in the handle feed mode when the M32 command is executed, the end position specified by the X argument may not match the operation.
- See the program sample in Section <5.11.6 Sequential operation for product collection by the pick-off function>.
- Be sure to move the opposite tool post (X2 axis) to the machine zero point by executing the M147 command (see Section <5.8.4 Opposite tool post retraction>) before the M32 command. An alarm is issued if the X2 axis is not at the machine zero point.

5.11.6 Sequential operation for product collection by the pick-off function (M33)

The M33 command enables sequential operation for product collection. There are two methods to perform sequential operation for product collection: by using the pick-off function and by using the basket. If "Standard" is set for Back Spindle in the machining data, the pick-off function is used. If "Basket" is set, the basket is used. Operation is different between product collection by the pick-up function and product collection by the basket. Make sure of the setting of Back Spindle in the machining data before starting the program.

Command format

M33 Z[ ] T[ ] S[ ] R1 D1
M108 Workpiece separator advancement
M109 Workpiece separator retraction
You can also collect products by using the M108 and M109 commands instead of the M33 command. (sequential operation for product collection by using the pick-off function.) See program sample.

Axis control groups

Specify this command for $1, $2 or $3.

Argument

- Z[ ]: Specify the product collection position of the back spindle (Z3 axis) for the workpiece separator. If the Z argument is not specified, the product collection position of the back spindle is as shown below. Specify the Z argument in accordance with workpiece length as possible.

  Product collection by using the pick-off function
  The product collection position of the back spindle (Z3 axis) is determined by the value specified for Machining Length in the machining data.

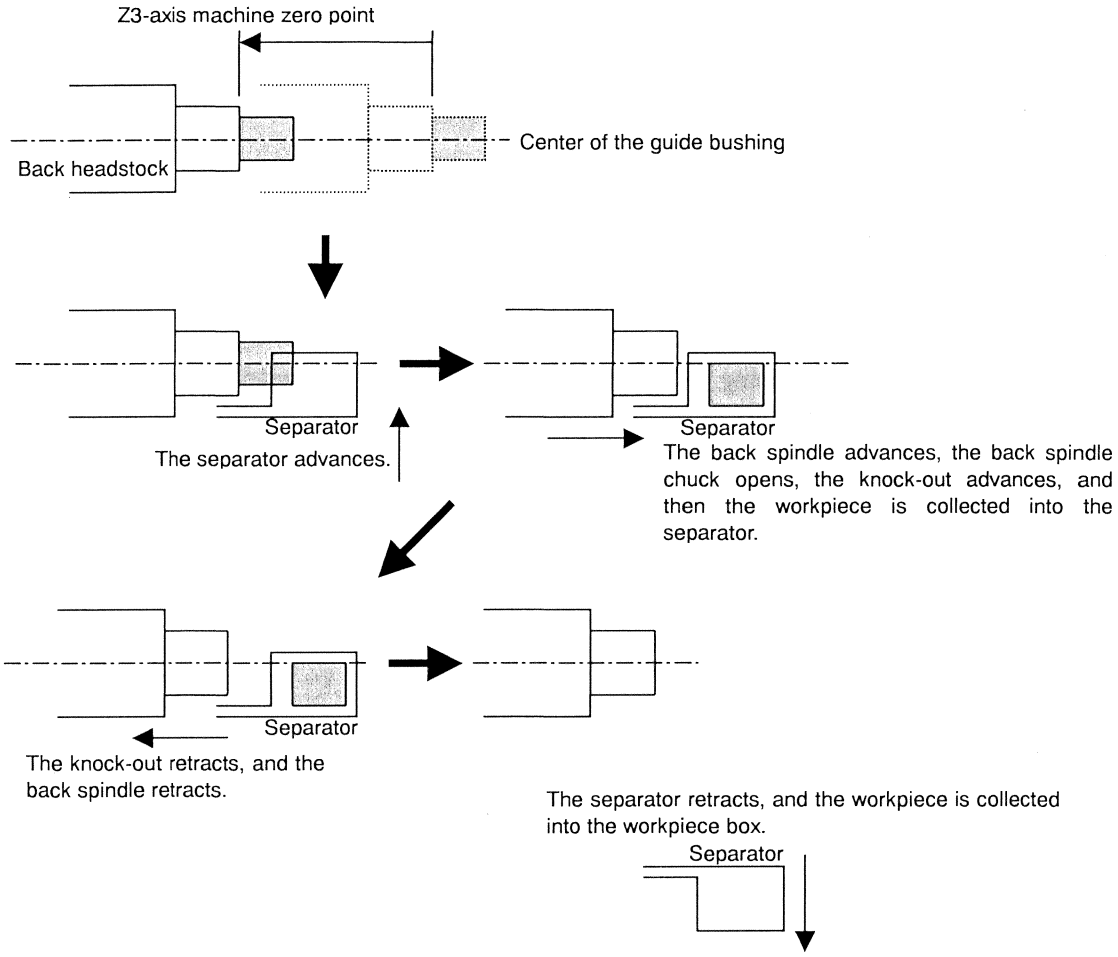
Value specified for Machining Length	Product collection position of the back spindle (Z3 axis)
Specified value < 60.0 mm	35.0 mm
60.0 mm ≤ specified value ≤ 90.0 mm	95.0mm – Machining length
Specified value > 90.0 mm	5.0 mm

  Product collection by using the basket
  Back spindle (Z3 axis) machine coordinate: 45.0 mm

- T[ ]: If using the pick-off function for product collection, specify the time while the air blower is to blow out chips from the back spindle. If using the basket for product collection, specify the time while the back spindle rotates at a low speed when a product is transferred from the basket to the workpiece separator. The default is 2 seconds.
- S[ ]: If using the basket for product collection, specify the speed of the back spindle when a product is transferred from the basket to the workpiece separator. The default is 30 min^{-1}.
- R1: Specify this argument to skip the operation to blow out chips from the back spindle during product collection by the pick-off function.
- D1: This argument determines when to execute the blocks following M33. If the D1 argument is not specified, the system executes the blocks following M33 when all the M33 operations are completed. If the D1 argument is specified, the system executes the blocks following M33, in parallel to the M33 operations. In this case, the blocks following the M33 are executed before the workpiece separator completes retraction. Thus, be very careful of back spindle (Z3 axis) operation.

Operation sample

When collecting workpiece from the back spindle:



Product collection by using the pick-off function

(1) The back spindle (Z3 axis) moves to the machine zero point.
(2) The workpiece separator advances.
(3) The back spindle (Z3 axis) moves to the product collection position. (The product collection position can be changed by the Z argument.)
(4) The back spindle chuck opens.
(5) The knock-out advances. (Products are transferred from the back spindle into the workpiece separator.)
(6) The knock-out retracts.
(7) The back spindle (Z3 axis) moves to the machine zero point.
(8) The workpiece separator retracts. (Products are transferred from the workpiece separator into the product chute.)
(9) The knock-out advances.
(10) The air blower is turned ON.
(11) Dwell time (The dwell time can be changed by the T argument.)
(12) The air blower is turned OFF.
(13) The knock-out retracts.

} The operation to blow out chips from the back spindle can be skipped by specifying the R1 argument.

Product collection by using the basket

(1) If the machine is equipped with front/back face machining tool spindle U121B, an alarm is issued unless the X2 axis is at the machine zero point.
(2) The basket (back spindle Z3 axis) moves to the machine zero point.
(3) The workpiece separator advances.
(4) The basket (back spindle Z3 axis) moves to the product collection position. (The product collection position can be changed by the Z argument.)
(5) The basket (back spindle) rotates at a low speed. (Products are transferred from the basket into the workpiece separator.)
(6) Dwell time (The dwell time can be changed by the T argument.)
(7) The basket (back spindle) stops.
(8) The basket (back spindle Z3 axis) moves to the machine zero point.
(9) The workpiece separator retracts. (Products are transferred from the workpiece separator into the product chute.)

Program sample

Product collection by using the pick-off function

$1	$2	$3
G630 ····· Front/back parallel machining	G630 ····· Front/back parallel machining	G630
T0200	T3600	
:	:	
	M147 ····· The opposite tool post retracts.	
	M25	
	M33 Z20.0 R1 ···· Sequential operation for product collection by using the pick-off function	
!2L1	!1L1	
G00 M03 S1=[ ] M24 S2=[ ]		
T0100		
G00 X17.0 Z30.0 T01		
G650 ····· Pick-off	G650 ····· Pick-off	G650
:	:	

When not using M33 for product collection by the pick-off function:

$1	$2	$3
G630	G630 ····· Front/back parallel machining	G630
T0200	T3600	
:	:	
	M147	
	M25	
	G53 Z0 ····· The back spindle retracts.	
	M108 ····· The workpiece separator advances.	
	G00 W20.0 ···· The back spindle is positioned.	
	M16 ····· The back spindle chuck opens.	
	M10 ····· Knock-out forward	
	M72 ····· Air blower ON	
	G4 U0.5	
	M73 ····· Air blower OFF	
	M11 ····· Knock-out backward	
	G53 Z0 ····· The back spindle retracts.	
	M109 ····· The workpiece separator retracts.	
!2L1	!1L1	
G00 M03 S1=[ ] M24 S2=[ ]		
T0100		
G00 X17.0 Z30.0 T01		
G650		
:		

Product collection by using the basket

$1	$2	$3
G50 Z0		
M33 D1 ····· Sequential operation for product collection by using the basket		
(Since the D1 argument is specified, the M33 operations are performed in parallel to the M06 operation and the subsequent processing.)		
M06		
G99 M03 S1=□		
:		
T0100 ····· The cut-off tool is selected.		
G00 X17.0 Z30.0 T01		
M32 X1.0 W0 ····· The basket advances.		
(The basket completes advancement when the X1 axis reaches the position 1.0 during cut-off machining.)		
G01 X-3.0 F□		
M171 ····· The basket retracts.		
:		

Notes

- Select the basket for product collection when workpieces do not require back machining. Workpieces not requiring back machining can also be collected when the pick-off function is selected for product collection. Cut-off tool can be mounted at the position T01 or T04.
- If the machine is equipped with U121B (front/back face machining tool spindle) during product collection by using the pick-off function, move the opposite tool post (X2 axis) to the machine coordinate 260.0 mm or further, and then execute the M33 command.
  If the machine is equipped with U121B (front/back face machining tool spindle) during product collection by using the basket, move the opposite tool post (X2 axis) to the machine zero point, and then execute the M33 command.
- When the door lock is mounted, the door lock is not released by one cycle atop after M33 is specified.
  Press the RST // key to release the door lock.

⚠ CAUTION

If the D1 argument (parallel operation) is specified in the M33 command, the blocks following M33 are executed in parallel to the M33 operations. In this case, be careful of interference.

5.11.7 Forward/backward movement of the product knock-out device (M10 and M11)

Use the M10 command to move the knock-out of the back spindle forward, and use the M11 command to move it backward.

Command format

M10 Z[ ] F[ ] T[ ]	Command to move the product knock-out device forward
M11 Z[ ] F[ ]	Command to move the product knock-out device backward

Argument

- Z[ ]: Specify the position to which the knock-out axis moves. If the Z argument is not specified, the axis moves to the forward end position (C16: 120.0 mm) when the M10 command (forward knock-out) is executed. The axis returns to the position 0 mm when the M11 command (backward knock-out) is executed.

- F[ ]: Specify the feed rate at which the knock-out axis moves. If the F argument is not specified, the axis moves at the rapid feed rate of 15000 (mm/min).

- T[ ]: This argument specifies the torque to be applied to the knock-out axis. Specify a ratio (%) to the maximum torque. A value in the range 0 to 200% can be specified. If the T argument is not specified, 200% is assumed. The T argument is valid only with the M10 command.

Notes

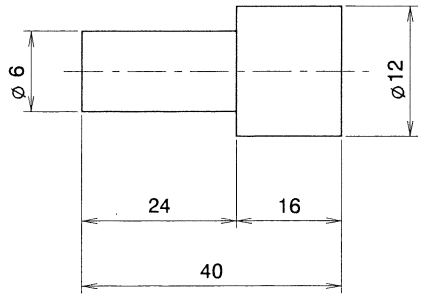
- The arguments are valid only with the motor type (option).
- The knock-out axis does not move if the back spindle chuck is closed or the back spindle is rotating. An alarm is issued in this case.

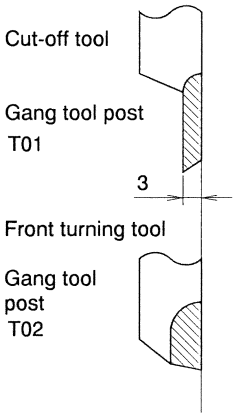
5.12 Program Samples

5.12.1 Program sample 1

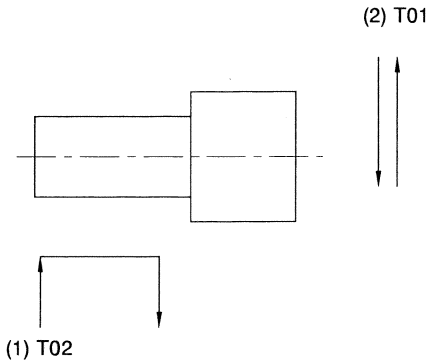
Machining drawing

Material to be machined SUM24L (free-cutting steel)
Material diameter ϕ 12.0





Machining layout



Note

The above is a program sample when the basket is mounted on the back spindle. Set "Basket" for Back Spindle in the machining data.

O0001

	$1
G50 Z0 ·····	Coordinate system setting
M33 D1 ·····	Sequential operation for product collection by using the basket (The blocks following M06 are executed in parallel to operations for product collection.)
M06 ·····	The chuck closes.
G00 X13.0 Z–0.5 ·····	The tool is moved away from the material. (Figure 2)
M03 S1=4000 G99 ·····	The main spindle rotates forward at the speed of 4,000 min^{-1}. Rotation per minute (mm/min^{-1})
T0200 ·····	Selection of T0200
G00 X6.0 Z–0.5 T02 ····	The tool is positioned ϕ6 at rapid feed rate. (Figure 3) Selection of compensation number 2
G01 Z24.0 F0.04 ·····	The tool machines the workpiece 24 mm in the longitudinal direction, with the diameter of ϕ6 mm. (Figure 4)
X13.0 ·····	The tool machines to the material outer diameter +1 mm. (Figure 5)
T0100 ·····	Selection of T0100
G00 X13.0 Z43.0 T01 ···	The tool is positioned for the cut-off machining at rapid feed rate. Selection of compensation number 1
M32 X1.0 ·····	The basket advances. (The basket completes advancement when the X1 axis reaches the position 1.0 during cut-off machining in the next block.)
G01 X–3.0 F0.03 ·····	The tool cuts off the workpiece.
M171 ·····	The basket retracts.
M05 ·····	The main spindle stops.
M07 ·····	The chuck opens.
G00 X–3.0 Z0 T00 ·····	The tool returns to the start position. The compensation command is canceled.
M56 ·····	Product count
M02 ·····	1-cycle stop
M99 ·····	Return to the main program
% ·····	Stop code

Calculation of spindle speed and feed rate

$$N = \frac{V}{\pi \times D} \times 1{,}000 = \frac{200}{3.14 \times 12} \times 1{,}000 = 5305 \approx 5000\ (min^{-1})$$

N: Speed (min^{-1})
V: Cutting speed 200 m/min.....Selected from the cutting condition table
D: Material diameter ϕ 12 mm
π: Circular constant (≈ 3.14)
Fx 0.03 mm/rev.....Selected from the cutting condition table
Fz 0.08 mm/rev.....Selected from the cutting condition table

Operation sample

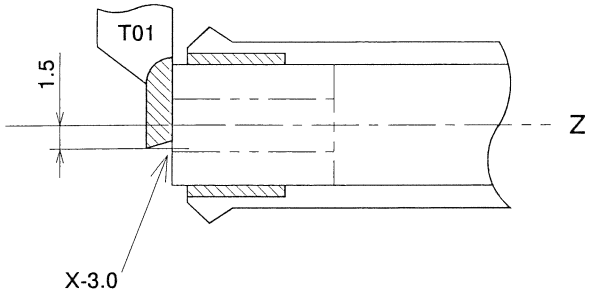


Figure 1

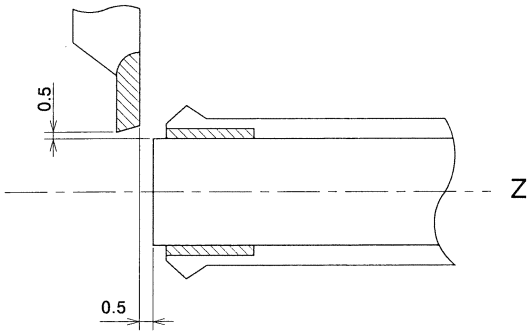


Figure 2

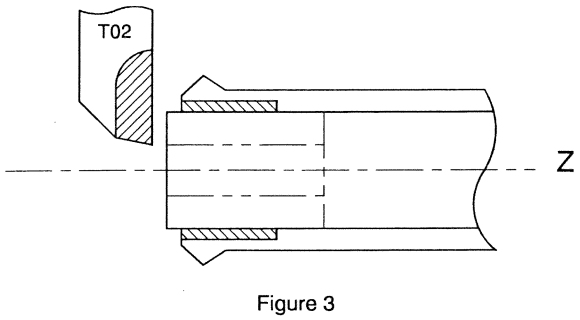


Figure 3

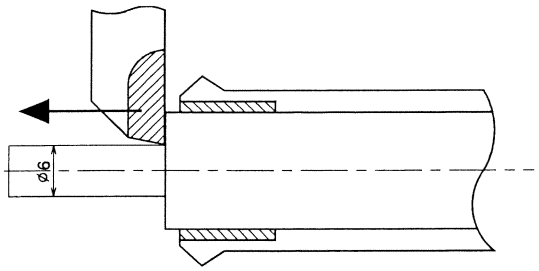


Figure 4

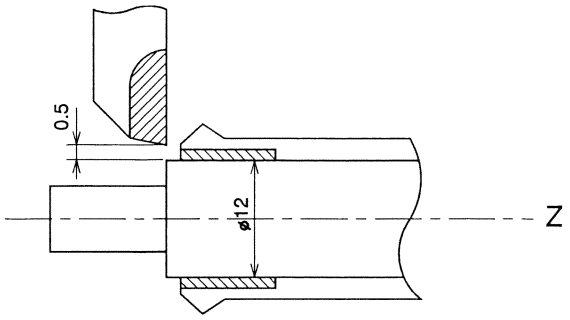


Figure 5

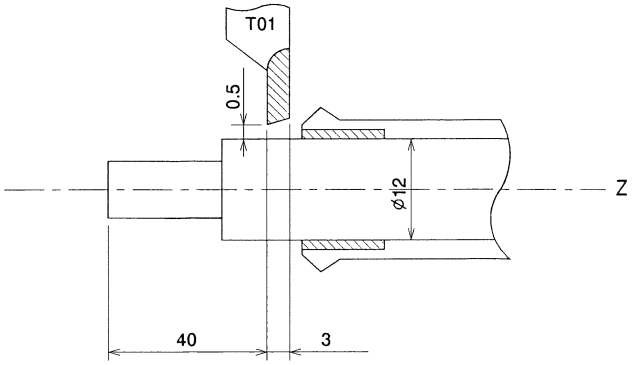


Figure 6

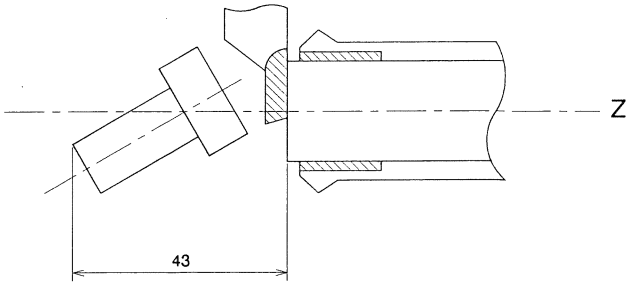


Figure 7

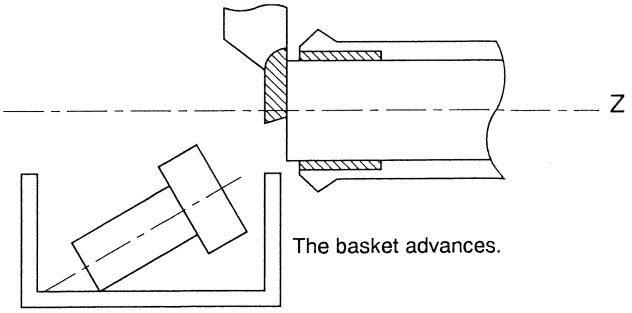
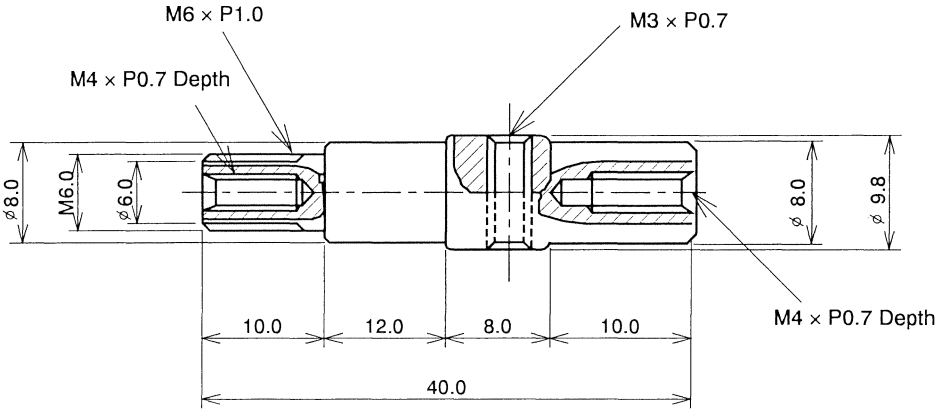


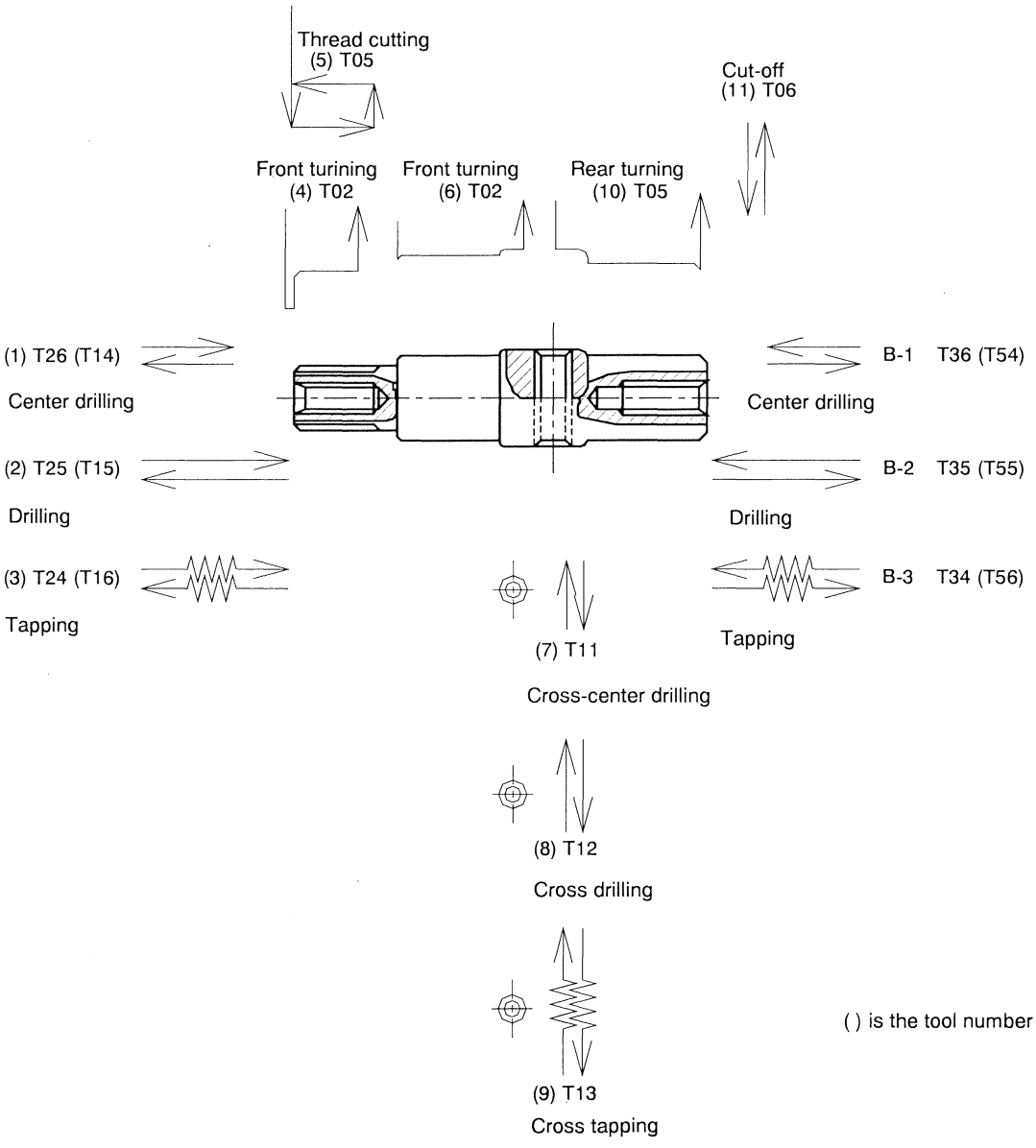
Figure 8

5.12.2 Program sample 2

Machining drawing

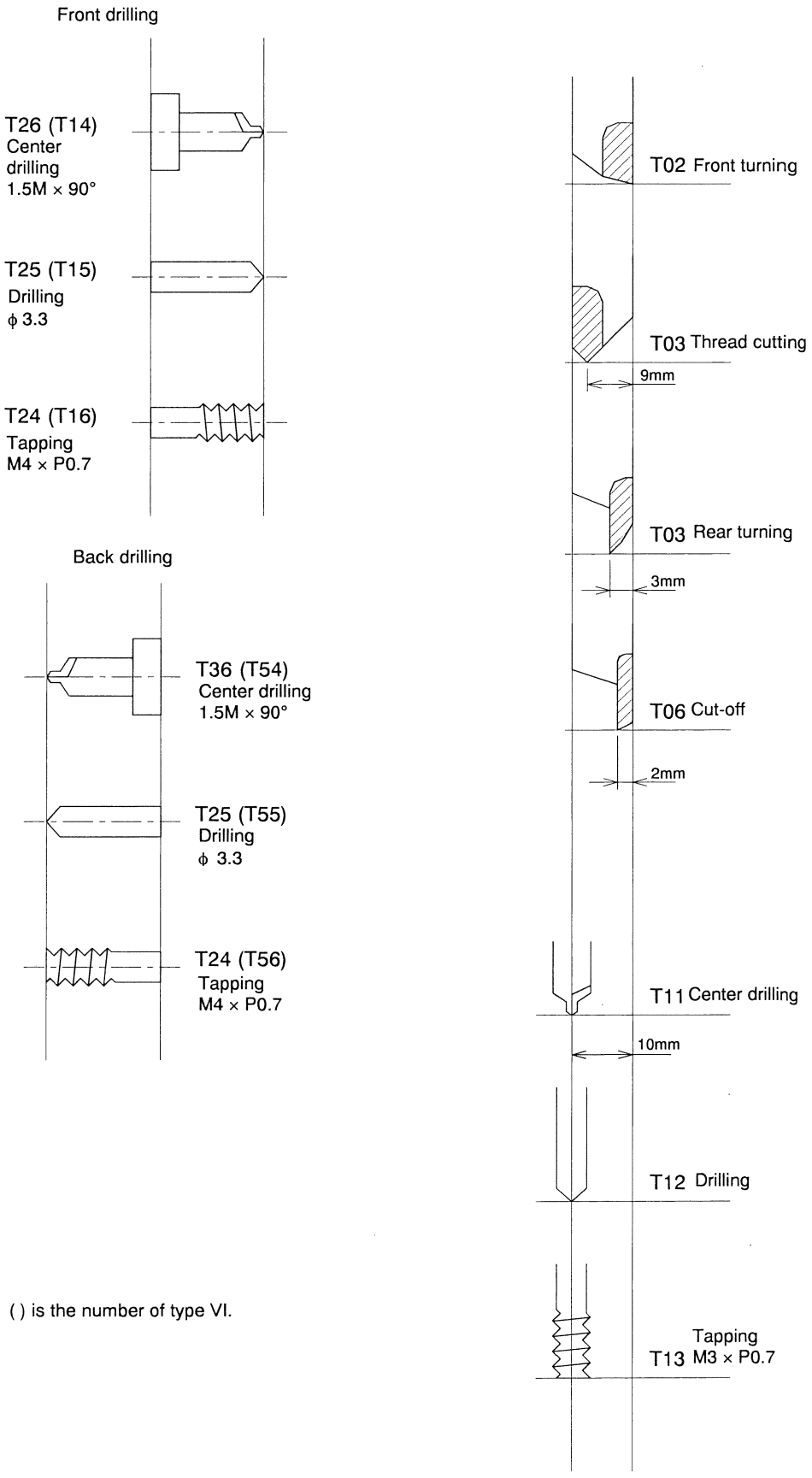


Machining layout



( ) is the tool number of type VI.

Tool Layout



( ) is the number of type VI.

Program sample

O0500 *( ) is the program of type VI.

$1

```
G50 Z-0.1 ..... Coordinate system setting
M06 ..... The chuck closes.
G99 M20 ..... Millimeters per revolution feed
G113 ..... Spindle synchronization cancel
M9 ..... End of the material replacement program
G0 X11.0 Z-1.0 ..... The tool is moved away from the material outer diameter. At the
                     same time, it is moved – 1 mm away from the end face of the material.

M3 S1=2500 ..... The main spindle rotates forward at the speed of 2500 min-1.
G610 (G660) ..... Front/back simultaneous machining
N226 T2600 (T1400) ..... Tool selection. The tool for center drilling is positioned to the center
(CENTER 1.5 × 90)    of the workpiece.
G0 Z-1.0 ..... The tool is moved to the position –1 mm from the end face of the
                     workpiece
G1 Z3.0 F0.1 T26 ..... Center drilling, 3 mm infeed into the Z direction, and selection of
                     compensation No. 26
G0 Z-1.0 T0 ..... The tool for center drilling is moved –1 mm away from the end face of
                     the workpiece. The compensation command is canceled.

M3 S1=3000 ..... The main spindle rotates at the speed of 3000 min-1.
N225 T2500 (T1500) (DRILL 3.3) ..... Tool selection. The drill is positioned to the center of the workpiece.
G0 Z-1.0 ..... The tool is moved to the position –1 mm from the end face of the
                     workpiece.
G1 Z10.0 F0.05 T25 ..... Drilling, 10 mm infeed into the Z direction, and selection of
                     compensation No. 25
G0 Z-1.0 T0 ..... The drill is moved –1 mm away from the end face of the workpiece.
                     The compensation command is canceled.
```

When the face drilling cycle is used:

```
M3 S1=3000
N225 T2500 (T1500) (DRILL 3.3)
G0 Z-1.0
G79 Z20.0 R0 I6.0 K3.0 A1.0 Q0 J500 F0.05 T25
                     ..... Face drilling cycle Z20.0 (drilling end point), R0 (increment from the
                     current value to the drilling start point), I6.0 (first infeed), K3.0
                     (second and subsequent infeed), A1.0 (distance required for safely
                     stopping the second and subsequent drilling), Q0 (dwell time at
                     infeed), and J500 (dwell time at return: decimal point invalid), F
                     (feed)
G0 Z-1.0 T0
G80

M3 S1=500 ..... The main spindle rotates at the speed of 500 min-1.
N324 T2400 (T1600) ..... Tool selection. The tap is positioned to the center of the workpiece.
(TAP M4 × P0.7)
G32 Z7.0 F0.56 T24 ..... Infeed by the tap (feed = pitch × 0.8), 7 mm into the Z direction, and
                     selection of compensation No. 24
Z-1.0 F0.7 M4 ..... The tap is returned. (Feed rate 100 %. The main spindle rotates
                     backward.)
G4 U0.5 ..... Dwell time (0.5 second)
G0 Z-2.0 T0 ..... The tap is moved –2 mm away from the end face of the workpiece.
G4 U0.5 ..... Dwell time (0.5 second)
```

Program sample

O0500

$2		$3
G610 S1 (G660)	····· Front/back simultaneous machining S1 (simultaneous machining with the tools of tool numbers in the $ 1 T20's and $2 T30's)	G610 (G660)
M98 H201	····· To the subprogram	

$1

When using the synchronized face tapping cycle:

```
M3 S1=500
N324 T2400 (T1600) (TAP M4 × P0.7)
G0 Z–2.0
G84 Z7.0 F0.7 D1 S500, R1 T24 ····· Synchronized face tapping cycle (G84). D1 (The main spindle rotates forward.)
G80 ····· The synchronized face tapping mode is canceled.
G0 Z–2.0 T0

!2L3
M3 S1=3000 ····· The main spindle rotates forward at the speed of 3000 min-1.
G630 (G600) ····· Front/back parallel machining
N402 T200 (FRONT T–1) ····· Tool selection, and front turning
G0 X11.0 Z–1.0 T2 ····· The tool is moved to the specified position (material outer diameter + 1 mm, and –1 mm from the end face of the material).
G1 X2.0 F0.1 ····· Moved to the position for obtaining ϕ2 at the cutting feed rate
Z0 ····· The tool comes in contact with the end face of the workpiece.
X3.98 F0.05 ····· Cutting continues until ϕ3.98 is obtained.
X5.89 Z1.0 F0.03 ····· Chamfering (C1)
Z10.0 F0.05 ····· The tool machines the workpiece 10 mm in the longitudinal direction.
X9.6 ····· Moved to a position immediately before the chamfering position.
X11.0 W0.7 F0.04 ····· Chamfering (The tool is moved away from the material outer diameter.)

M3 S1=1000 ····· The main spindle rotates at the speed of 1000 min-1.
N503 T300 (THREAD M6 × P1.0) ····· Tool selection, and threading
G50 W–9.0 ····· Coordinate system shift ON
G0 X11.0 Z–2.0 T3 ····· Rapid feed positioning
G1 X8.0 F0.5 ····· The tool is moved to the initial point.
G92 X5.79 Z8.5 F1.0 ····· Thread cutting cycle command, and first infeed
X5.5 ····· Second infeed
X5.06 ····· Third infeed
X4.8 ····· Fourth infeed
X4.7 ····· Fifth infeed
X4.7 ····· Zero cut (The same diameter as for the fifth infeed is specified.)
G0 X10.0 ····· Thread cutting cycle cancel
G50 W9.0 ····· Coordinate system shift OFF

M3 S1=3000 ····· The main spindle rotates at the speed of 3000 min-1.
N702 T200 (FRONT T–2) ····· Tool selection, and front turning
G0 X11.0 Z9.8 T2 ····· Rapid feed positioning
G1 X7.0 F0.1 ····· The tool is moved to the infeed start position.
X8.0 Z10.3 F0.03 ····· Chamfering (C0.3)
Z22.0 F0.05 ····· Machining up to 22.00 in the longitudinal direction
X9.0 F0.03 ····· Moved to the position for obtaining ϕ9
G2 X9.8 W0.4 R0.4 F0.02 ····· Chamfering by the G02 command (circular interpolation)
G1 Z31.0 F0.03 ····· Machining up to Z31.0 mm in the longitudinal direction
X11.0 W0.6 F0.05 ····· Chamfering (C1)
```

$2		$3
G630 (G600)	····· Front/back parallel machining	G630 (G600)
M98 H202	····· To the subprogram	
!1L1	····· Queuing	

	$1
M5	····· The main spindle stops.
M58 S3=2500 G98	····· The tool spindle rotates forward. Millimeters per minute feed
M28 S0	····· The main spindle is indexed.
N611 T1100	····· Tool selection, and cross-center drilling
G50 W–10.0	····· Coordinate system shift ON
G0 X11.0 Z26.0 T11	····· Rapid feed positioning
G1 X5.0 F100	····· Infeed up to X5.0 in the diametrical direction
G0 X11.0	····· The tool for cross-center drilling is moved away from the material outer diameter.
M28 S180	····· The main spindle is indexed at 180°.
G1 X5.0	····· Infeed up to X5.0 in the diametrical direction
G0 X11.0	····· The tool for cross-center drilling is moved away from the material outer diameter.
G50 W10.0	····· Coordinate system shift OFF
S3=3000	····· The tool spindle rotates at the speed of 3000 min^{-1}.
N712 T1200	····· Tool selection, and cross drilling
G50 W–10.0	····· Coordinate system shift ON
G0 X11.0 Z26.0 T12	····· Rapid feed positioning
G1 X0 F100	····· Infeed up to X0 in the diametrical direction
G0 X11.0	····· The tool for cross drilling is moved away from the material outer diameter.
X1.0	····· The tool is moved to the position for the second infeed in the cross drilling process.
G1 X–6.0	····· Infeed up to X–6.0 in the diametrical direction
G0 X11.0	····· The tool for cross drilling is moved away from the material outer diameter.
X–5.0	····· The tool is moved to the position for the third infeed in the cross drilling process.
G1 X–13.0	····· A through hole is made in the workpiece.
G0 X11.0	····· The tool for cross drilling is moved away from the material outer diameter.
G50 W10.0	····· Coordinate system shift OFF
M58 S3=500	····· The main spindle rotates at the speed of 500 min^{-1}.
N1013 T1300 (C.TAP. M3 × 0.7)	····· Tool selection, and cross tapping
G50 W–10.0	····· Coordinate system shift ON
G0 X12.0 Z26.0 T13	····· Rapid feed positioning
G88 X–12.0 F0.7 D3 S500, R1	····· Synchronized cross tapping mode ON
G80	····· Synchronized cross tapping mode OFF
G0 X11.0	····· The tap is moved away from the material outer diameter.
G50 W10.0	····· Coordinate system shift OFF
M20	····· Main spindle index release
G99 M60	····· The tool spindle stops. Millimeters per revolution feed
M3 S1=3000	····· The main spindle rotates forward at the speed of 3000 min^{-1}.
N1105 T500 (REAR.T)	····· Tool selection, and back turning
G50 W–3.0	····· Coordinate system shift ON
G0 X11.0 Z29.0 T5	····· Rapid feed positioning
G3 X9.8 W0.6 R0.6 F0.03	····· The tool approaches the outer diameter during circular interpolation.
G2 X9.0 W0.4 R0.4 F0.02	····· Chamfering by the G2 command (circular interpolation)

$2	$3

$1	
G1 X8.0	····· Moved to the position for obtaining ϕ8
Z39.7 F0.05	····· Machining up to 39.7 mm in the longitudinal direction
X7.4 Z40.0 F0.02	····· Chamfering (C0.3)
X11.0F0.2	····· The tool is moved away from the material.
G50W3.0	····· Coordinate system shift OFF
!2L1	····· Queuing
M3 S1=2500 M24 S2=2500	····· The main and back spindles rotate forward at the speed of 2500 min^{-1}
G114.1 H1 D–2	····· Spindle synchronization
N1206 T600	····· Tool selection and cut-off.
G0 X11.0 Z42.0 T6	····· Rapid feed positioning
G650	····· Pick-off
!2L2	····· Queuing
G1 X–1.0 F0.03	····· Cut-off machining
G113	····· Spindle synchronization cancel
G630 (G600)	····· Front/back parallel machining
G1 X–3.0 F0.05	····· Moved to the value as the cut-off end point
M5 M25	····· The main and back spindles stop.
M7	····· The main spindle chuck opens.
M8	} Material replacement program enabled
M8	
/M98 P8000	····· Material replacement
M9	····· End of the material replacement program
G0 X–3.0 Z–0.1 T0	····· The tool returns to the start position. The compensation command is canceled.
M56	····· Product count
G999	····· Last program execution command
M6	····· The main spindle chuck closes
G0 X11.0 W–1.0	····· The tool is moved away from the material on the main spindle side. (–1 mm from the end face of the workpiece)
!2L3	····· Queuing
G630 (G600)	····· Front/back parallel machining
M3 S1=2000	····· The main spindle rotates forward.
G99 G0 W1.1	····· The workpiece advances 1.1 mm.
G1 X–3.0 F0.03	····· Cut-off machining
M5	····· The main spindle stops.
M7	····· The chuck opens.
G0 X–3.0 Z–0.1 T0	····· The tool returns to the start point.

$2		$3
G650	····· Pick-off	G650
G01 Z–1.0 M72	····· The back spindle positions Z–1.0. Air blower ON	
G98 G1 Z20.0 F2000	····· Moved to the workpiece chuck position	
G4 U0.2	····· Dwell time (0.2 second)	
M77	····· State waiting for completion of spindle synchronization	
M15	····· The back spindle chuck closes.	
M73	····· Air blower OFF	
G4 U0.2	····· Dwell time	
!1L2	····· Queuing	
G630 (G600)	····· Front/back parallel machining	G630 (G600)
G999	····· Last program execution command	G999
!1L3	····· Queuing	
M98 H201	····· To the subprogram.	
		G630 (G600)
G630 (G600)	····· Front/back parallel machining	
M98 H202	····· To the subprogram.	

	$1
N999	····· Last program queuing
M2	····· 1-cycle stop
M99	····· Return to the main program

$2		$3
N999	····· Last program queuing	N999
M2	····· 1-cycle stop	M2
M99	····· Return to the main program.	M99
N201 (BACK. SUBPRO)	····· Subprogram sequence	
G99 M23 S2=2000	····· The back spindle rotates forward. Millimeters per revolution feed	
T3600 (T5400) Q1	····· Tool selection, and back center drilling Q1 (no move to the retraction position)	
G0 Z–1.0	····· Rapid feed positioning	
G1 Z3.1 F0.1 T36	····· Infeed up to 3.01 mm in the back longitudinal direction	
G0 Z–1.0 T0	····· Retraction to the position –1 mm from the end face of the workpiece	
T3500 (T5500) Q1	····· Tool selection, and back drilling. Q1 (no move to the retraction position)	
G0 Z–1.0	····· Rapid feed positioning	
G1 Z10.0 F0.05 T35	····· Infeed up to 10 mm in the back longitudinal direction	
G0 Z–1.0 T0	····· Retraction to the position –1 mm from the end face of the workpiece	
When using the face drilling cycle:		
T3500 (T5500) Q1	····· Tool selection, and back drilling. Q1 (no move to the retraction position)	
G0 Z–1.0	····· Rapid feed positioning	
G79 Z20.0 R0 I6.0 K3.0 A1.0	····· Face drilling cycle Z20.0 (drilling end point).	
Q0 J500 F0.05 T35	R0 (increment from the current value to the drilling start point)	
	I6.0 (first infeed)	
	K3.0 (second and subsequent infeed)	
	A1.0 (distance required for safely stopping the second and subsequent drilling)	
	Q0 (dwell time at infeed: decimal point invalid)	
	J500 (dwell time at return: decimal point invalid)	
	F (feed)	
G80		
G0 Z–1.0 T0		
M23 S2=500	····· The back spindle rotates at the speed of 500 min^{-1}.	
T3400 (T5600) Q1	····· Tool selection, and back tapping Q1 (no move to the retraction position)	
G0 Z–2.0	····· Rapid feed positioning	
G32 Z7.0 F0.56 T34	····· Tapping (feed pitch × 0.8)	
Z–2.0 F0.8 M24	····· The tap is returned. (Feed rate 100 %).	
G0 Z–2.0 T0	····· The tap is moved –2 mm away from the end face of the workpiece.	
When using the synchronized face tapping cycle:		
M23 S2=500	····· The back spindle rotates at the speed of 500 min^{-1}.	
T3400 (T5600) Q1	····· Tool selection, and back tapping Q1 (no move to the retraction position)	
G0 Z–2.0	····· Rapid feed positioning	

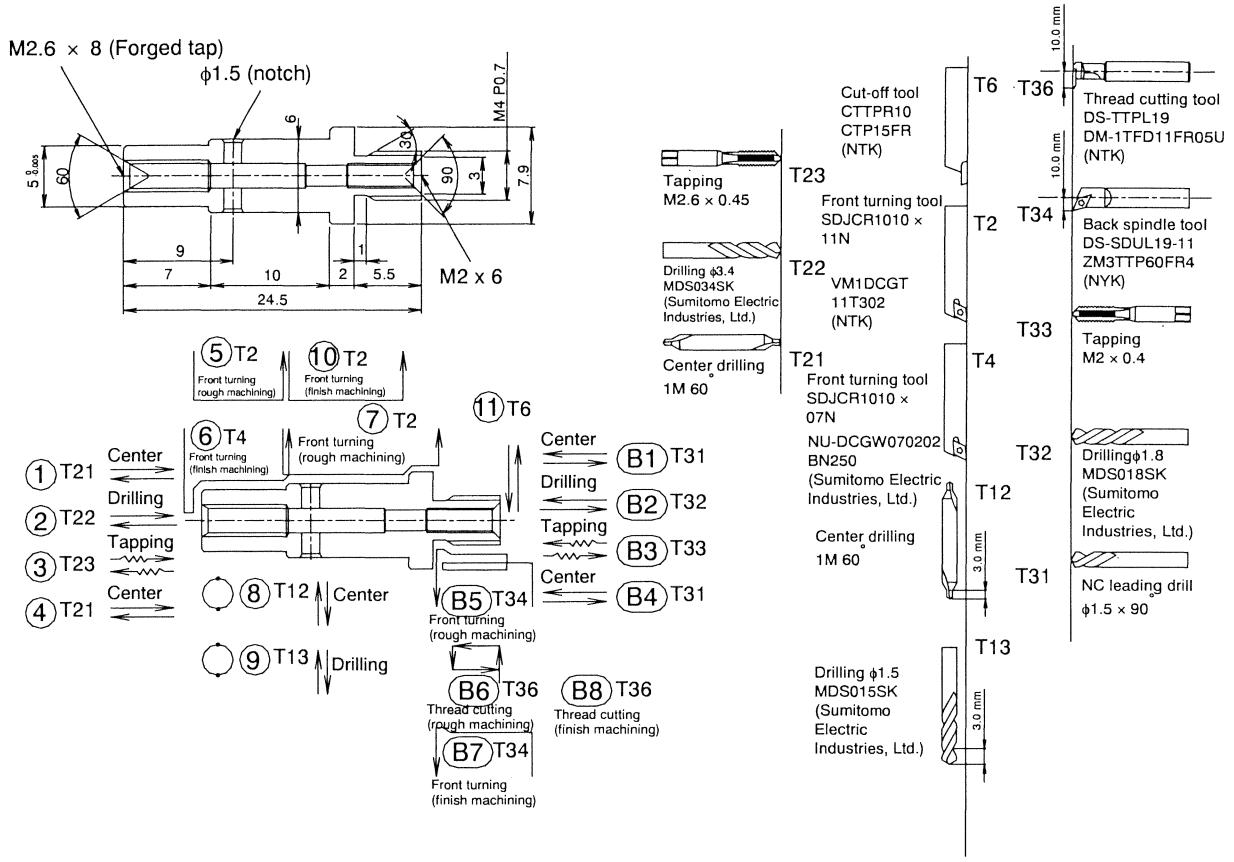
$1

$2	$3
G84 Z7.0 F0.7 D2 S500, R1 T34 ····· Synchronized face tapping (G84).	
G80 ····· The synchronized face tapping mode is canceled.	
G0 Z–2.0 T0	
!1L3	
M147 (Type VII, VIII) ····· The opposite tool post retracts.	
M25 ····· The back spindle stops.	
M99	
N202	
M33 ····· Sequential operation for product collection by using the pick-off function	
M99	

5.12.3 Program sample 3: Type VII, VIII, IX

Machining drawing and layout

Material diameter	ϕ6
Material to be machined	SUS303



Program sample

O107

$1	$2	$3
G50 Z7.9		
G113		
M6		
M9 M20		
G0 X9.0 Z–1.0		
M51 ···· Cut-off tool breakage detection ON		
M3 S1=3000 G99		
G610 ···· Front/back simultaneous machining	G610 S1 ····· Front/back simultaneous machining	G610
	M98H1 ····· To the subprogram	
N126 T2100 (CENTER=1.0 X60)		
···· Center drilling		
G0 Z–1.0		
G1 Z3.0 F0.03 T21		
G0 Z–1.0 T0		
S1=4000		
N225 T2200 (DRILL=2.4)		
···· Drilling		
G0 Z–1.0		
G1 Z7.0 F0.04 T22		
G0 Z–1.0		
Z6.5		
G1 Z10.0		
G0 Z–1.0 T0		
S1=500		
M97 ····· Spindle speed change detection OFF		
N324 T2300 (TAP=M2.6 X0.45)		
···· Tapping		
Z–2.0		
G84 Z7.0 F0.45 D1 S500, R1 T23		
G80		
G0 Z–2.0 T0		
M96 ···· Spindle speed change detection ON		

$1	$2	$3
S1=4000 M3		
N126 T2100 (CENTER=1.0 X60) ···· Chip removal G0 Z-1.0 G0 Z2.8 G1 Z3.7 F0.03 T21 G0 Z-1.0 T0		
G630 ···· Front/back parallel machining	G630 U0 ····· Front/back parallel machining M98H2 ····· To the subprogram !1L2	G630 W0
S1=5000 M3 G99		
N52 (NTK.STJCR1010 X11N) T200 ···· Front turning (rough machining) G0 X9.0 Z-1.0 T2 G1 X5.1 F0.2 G1 Z7.0 F0.02 X7.6 X8.0 W0.2 X9.0 W1.0 S1=7000		
N64 T400 ···· Front turning (finish machining) G0 X9.0 Z-1.0 T4 G1 X2.0 F0.2 Z0 F0.05 X3.4 F0.03 G2 X5.0 Z0.7 R0.7 F0.02 G1 Z7.0 X5.6 X6.6 Z7.5 X9.0 F0.3 S1=5000		

$1	$2	$3
N72 T200 ···· Front turning (rough machining) G0 X9.0 Z6.5 T2 G1 X6.0 F0.2 Z12.0 F0.02 X7.3 X7.9 Z12.3 Z14.5 X9.0 F0.3 M5 M58 S3=3000 G98 } Commands to move to the secondary machining process M18 C0 N512 (CROSS.CENTER) T1200 ···· Cross center machining G50 W-10.0 G0 X9.0 Z9.5 T12 G1 X7.0 F500 X3.00 F100 G4 U0.1 G0 X7.0 C180.0 G1 X3.00 F100 G4 U0.1 G0 X9.0 G50 W10.0 N613 (CROSS.DRILL) T1300 ···· Cross drilling G50 W-10.0 G0 X9.0 Z9.5 T13 G1 X0 F100 U0.1 X-3.0 U0.1 X-7.0 G0 X9.0 G50 W10.0 M20 M60 } Commands to move to the turning process G99 Z1=7000 M3		

$1	$2	$3
N72 T200 ···· Front turning (finish machining)		
G0 X9.0 Z6.5 T2		
G1 X6.0 F0.5		
Z12.0 F0.2		
X9.0 F0.5		
G0 Z-1.0		
G0 U0 W0 T0		
!2L2		
G99 M3 S1=5000 S2=5000 M24		
G114.1 H1 D-2 ··· Spindle synchronization		
N76 T600		
G0 X9.0 Z29.1 T6		
G650 ···· Pick-off	G650 ····· Pick-off	G650 W0
	G0 Z-1.0 M72	
	G98 G1 Z11.0 F2000	
	M77	
	M15	
	M73	
!2L3	!1L3	
G231 X-3.0 F0.02 A-1.0		
···· Cut-off machining Pick-off cancel		
G113 ···· Spindle synchronization cancel		
M5 M25		
M7		
M8		
M8		
/M98 P1000		
M9		
G0 Z-3.0 Z7.9 T0		
M56		
G999	G999	G999
N999	N999	N999
M2	M2	M2
M99	M99	M99

$1	$2	$3
	N1 M23 S2=3500 G99 T3100 W90.0 Q1 ··· Back center machining G0 Z-1.0 G1 Z3.0 F0.03 T31 G0 Z-1.0 T0 S2=4000 N2235 T3200 Q1 (DRILL=2.4) ····· Back drilling G0 Z-1.0 G1 Z2.5 F0.2 T32 G1 Z7.0 F0.04 G0 Z-1.0 Z6.5 G1 Z12.0 G0 Z-2.0 T0 S2=500 M87 ····· Back spindle speed change detection OFF N4231 (TAP=M2 X0.4) T3300 Q1 ····· Back tapping G0 Z-2.0 G84 Z6.0 F0.4 D2 S500, R1 T33 G80 G0 Z-2.0 T0 M86 ····· Back spindle speed change detection ON S2=4000 M23 T3100 Q1 ····· Chip removal G0 Z-1.0 Z2.8 G1 Z3.7 F0.03 T31 G0 Z-1.0 T0 M99 N2 (B.T.DS-SDUL19-11) M23 S2=5000 G99 T3400 Q1 ····· Front turning (rough machining) G0 X9.0 Z-1.0 T34 X5.0 G1 Z4.9 F0.03 X5.5 F02	

$1	$2	$3
	G0 Z-1.0 G1 X1.0 F0.2 Z0 F0.1 X2.53 F0.03 X3.93 Z0.7 Z2.9 X3.0 Z4.0 Z5.0 X8.9 W0.9 X7.2 W-0.9 X3.0 W-0.1 G0 X9.0 G0 Z-2.0 S2=1200 T3600 Q1 ····· Back threading (rough machining) S2=1200 G99 G0 X9.0 Z-1.5 T36 X6.0 G92 X4.63 Z4.0 F0.7 X4.3 X4.15 X4.0 X3.93 X3.93 G0 X9.0 Z-2.0 M23 S2=7000 T3400 Q1 ····· Front turning (finish machining) G0 X9.0 Z-1.0 T34 X5.5 Z0 G1 X1.0 F0.05 X2.53 F0.05 X3.93 Z0.7 Z2.9 X3.0 Z4.0 G0 X9.0 Z-2.0 S2=1200 G4 U1.0	

$1	$2	$3
	T3600 Q1 ····· Back threading (finish machining) G0 X9.0 Z–1.5 T36 X6.0 G92 X3.93 Z4.0 F0.7 G0 X9.0 G0 U0 W0 T0 M147 ···· The opposite tool post retracts. M25 ···· The back spindle stops. G53 Z0 ···· The back spindle retracts. M108 ···· The workpiece separator advances. G0 W17.0 ···· The back spindle advances. M16 ···· The back chuck opens. M10 ···· Knock-out forward M11 ···· Knock-out backward M109 ···· The workpiece separator retracts. } Production collection M99 (G53 Z0 works similar to M33 Z17.0 R1.)	

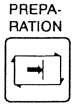
Product Code C – C1216VI VII VIII IX – 770

Document Code 5E1 – 0502

5E2 – 0502

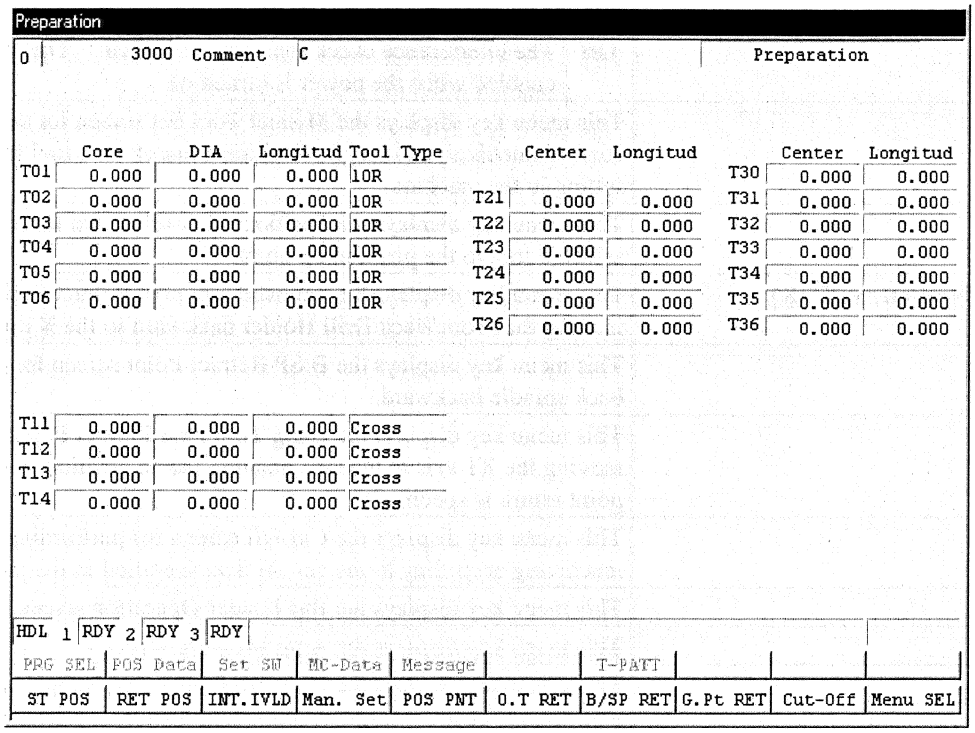
6. Screen Functions

6.1 Preparation Screen

Press 

to display the Preparation screen.

This screen supports preparation for machining workpieces. The menu keys of various operations are on the screen. You can operate the machine by selecting the desired menu key.



(This screen is for type VII, VIII, IX.)

Display items

Type VII, VIII, IX

Item		Explanation
O		Program number of the currently selected execution program
T01 to T08 T11 to T17 T21 to T28 T30 to T38	Core DIA Longitud Tool Type Center Longitud	Tool set data (values for error adjustment)

Type VI

Item		Explanation
O		Program number of the currently selected execution program
T01 to T08 T11 to T17 T51 to T57 T30	Core DIA Longitud Tool Type Center	Tool set data (values for error adjustment)

Main menu keys

Key name	Function	
ST POS	This menu key displays the Start Position screen for moving each axis to the position for the starting automatic operation.	
RET POS	This menu key displays the Return Position screen for moving each axis to the safe position (same position as when zero point return is specified).	
INT.IVLD	On	The interference check function is disabled.
	Off	The interference check function is enabled. The function is enabled when the power is turned on.
Man.Set	This menu key displays the Manual Tool Set screen for supporting the core, diametrical, and longitudinal adjustments of a tool for the tool setting in the machine.	
POS PNT	This menu key displays the Positioning Point screen for moving a selected tool to the positioning point.	
Oppo Ret (Type VII, VIII, IX)	This menu key displays the opposite tool post Retract Point screen for moving the Front/Back Drill Holder backward to the X direction.	
B/SP RET	This menu key displays the B/SP Retract Point screen for moving the back spindle backward.	
G.Pt RET	This menu key displays the Gang Tool Post Retract Point screen for moving the X1 axis to the safe position (same position as when zero point return is specified).	
Cut-Off	This menu key displays the Cut-Off screen for performing cut-off machining according to the cut-off data specified in the machining data.	
B/Loader	This menu key displays the Bar Loader Operation screen. (optional)	
AT MEAS.	This menu key displays the Auto Measure screen.	
Unload	This menu key displays the Unloading screen for supporting adjustments of the workpiece unloading device. (optional)	
Auto Set	This menu key displays the Manual Tool Set screen for automatically adjusting the core, diameter, and longitudinal of a tool for the tool setting in the machine (optional).	
GB ADJ	This menu key displays the Guide Bushing Adjustment screen for adjusting a clearance with the guide bushing.	
CHCK ADJ	This menu key displays the Chuck Adjustment screen for adjusting the chucking force.	

Submenu keys

Key name	Function
PRG SEL	This menu key displays the Program Select screen for selecting the machining program to be executed in the preparation mode.
POS Data	This menu key displays the machine values, workpiece coordinates, and remaining move distances of all the axis control groups.
Set SW	This menu key displays the Set SW screen for setting various control signals for NC unit operation to on/off.
MC-Data	This menu key displays the Machining Data screen.
Message	This menu key displays the Message screen.
T-PATT	This menu key displays the Tool Pattern screen corresponding to the tool holder.

Notes

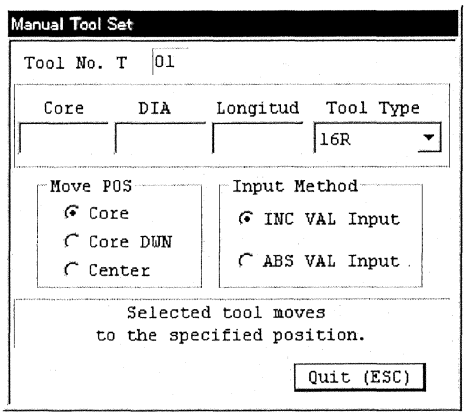
- Positioning position = (Bar Stock O.D. specified in the machining data) + (Tool Positioning Point specified in the machining data) + (DIA value used in the preparation mode)
- The Bar Loader Operation screen, Auto Measure screen, Work Unloader screen, or Auto Set screen is displayed only when the optional device is mounted. Otherwise, the menu key is not displayed.

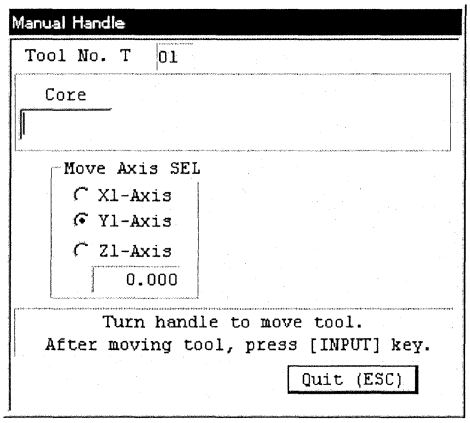
6.2 Manual Tool Set Screen

Press PREPA-RATION and then [Man.Set] to display the Manual Tool Set screen.

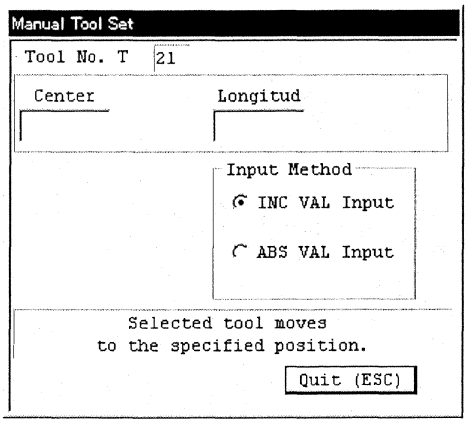
This screen supports the core, diametrical, and longitudinal adjustments of a tool for the tool setting in the machine.

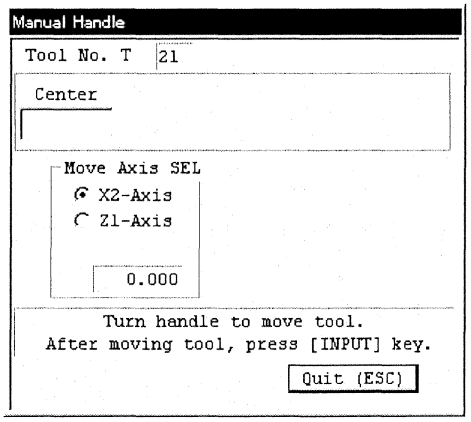
- Screens when selecting any of tools described below:
  T01 to T08, T11 to T17, T51 to T57 (Type VI)





- Screens when selecting any of tools described below:
  T21 to T28, T31 to T38





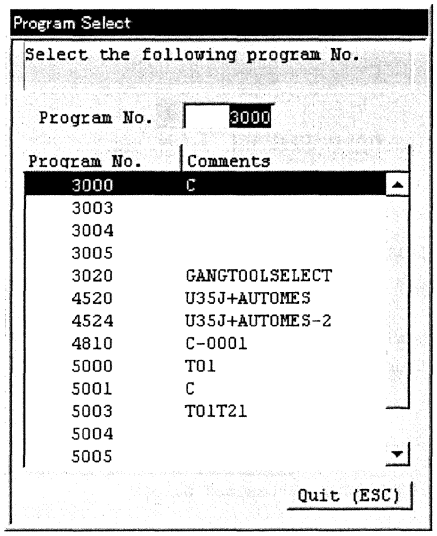
Display items

Item	Explanation
Core	Value for the error adjustment of the core height of the tool nose
DIA	Value for the error adjustment of the tool nose in the diametrical direction
Longitud	Value for the error adjustment of the tool nose in the longitudinal direction
Center	Value for the error adjustment of the core height of the tool nose
Tool Type	Tool shank size, and right-handed or left-handed tool

6.3 Program Select Screen

Press PREPA-RATION, ▷, and [PRG. SEL] in this order to display the Program Select screen.

This screen selects the machining program to be operated in the automatic operation mode or the program check mode. The corresponding machining data is also selected at the same time.



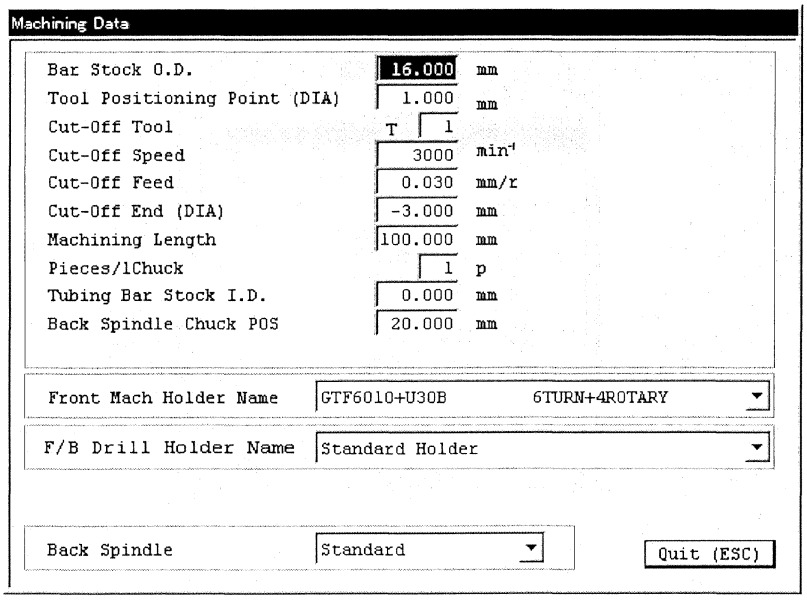
Display items

Item	Explanation
Program No. (upper)	Enter the program number of the program you want to execute.
Program No. (lower)	List of the program numbers of currently registered programs
Comments	Comments on currently registered programs

6.4 Machining Data Screen

Press PREPA-RATION, ▷, and [MC-Data] in this order to display the Machining Data screen.

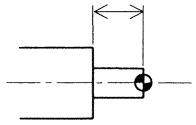
A collection of data necessary for machining and preparation for operation is called machining data. The Machining Data screen displays the machining data. The machining data is processed in the same manner as for the machining programs and stored in the memory.



(This screen is for type VII, VIII, IX.)

Display items

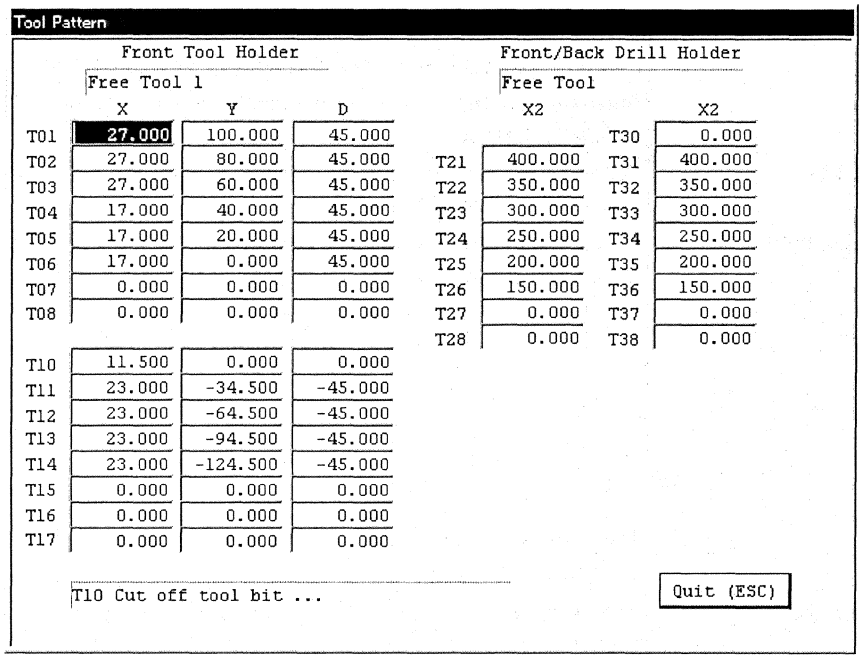
Item	Explanation
Bar Stock O.D.	Value of the outer diameter of material that are machined with the currently selected program
Tool Positioning Point (DIA)	Value calculated with the diameter that determines where to position the tool (how many mm are left) before it reaches the material outer diameter in the following cases: The front machining tool selected with the cursor on the Preparation screen is moved to the position (material outer diameter + tool positioning point) by selecting the menu key [POS PNT]. The front machining tool is specified for program operation.
Cut-Off Tool	Tool number of the tool used for cut-off machining for preparation. This tool is also selected when the start position operation is performed.
Cut-Off Speed	When cut-off machining is performed for preparation or when end-face turning is performed, the tool (selected with the cursor on the Preparation screen) stops at the positioning point. When the cut-off machining is restarted, the spindle starts rotating at this speed. The spindle stops when the cut-off machining is completed.

Item	Explanation
Cut-Off Feed (Specification of feed per rotation (mm/rev))	When cut-off machining is performed for preparation or when end-face turning is performed, the tool (selected with the cursor on the Preparation screen) stops at the positioning point. When the cut-off machining is restarted and the spindle reaches the speed specified for item 4, the tool works at this cutting feed rate.
Cut-Off End (DIA)	End position of cut-off machining for preparation or end position of end-face turning. When the start position operation is performed for preparation, the tool specified for item 3 is also moved to this end position.
Machining Length	Generally, enter the Z1 axis stroke necessary for machining a workpiece. (Workpiece length + cut-off tool width + end face finish allowance + α (extra))
Pieces/1Chuck	Enter the number of workpieces machined with a chuck. 1 should be generally entered.
Tubing Bar Stock I.D	When a pipe material is to be machined, enter the value of the pipe inner diameter. For solid pipes, enter 0.
Back Spindle Chuck POS	When machining a workpiece chucked by the back spindle chuck, enter the protrusion length from the workpiece coordinate zero point (for back machining) to the chuck position (front end of the spindle cap). Back Spindle Chuck POS  Back Spindle Chuck POS = (total length of the workpiece) - (chuck position specified in the program)
Front Mach Holder Name	Select the name of the holder mounted on the gang tool post. Check the model of the holder mounted on the machine, and select the name from the list. The holder model is inscribed on the holder.
F/B Drill Holder Name (Type VII, VIII, IX)	Select the name of the holder mounted on the opposite tool post. Select the name from the list. "Standard Holder" should be generally selected.
Back Spindle	Select the type of the back spindle mounted on the back spindle cap. Specify as "Standard" for collection by pick-off operation, and as "Basket" for collection using a product receiver basket. Select "With Support" when mounting a bushing for supporting the tip of a long workpiece.

6.5 Tool Pattern Screen

Press PREPA-RATION, ▷, and [T-PATT] in this order to display the Tool Pattern screen.

This screen displays the machine position data of each tool number that corresponds to the tool holder. As for a free tool layout pattern, use the screen to set the machine position data.



(This screen is for type VII, VIII, IX.)

Display items

Item	Explanation
Front Tool Holder	Name of front tool holder specified by the machining data.
X Y D T01 to T08 T10 to T17 T51 to T57 (Type VI)	For the tool position X, Y, specify the distance from the machine zero point to the tool nose with a radius value. For X, enter the X component in the virtual coordinate system. For Y, enter the Y component in the virtual coordinate system. For the tool position D, enter the angle of the virtual coordinate system to the motor shaft.
Front/Back Drill Holder (Type VII, VIII, IX)	Name of front/back drill holder specified by the machining data.
X2 T21 to T28 T31 to T38	Data X2, which is the distance from the machine zero point of X2 axis to the tool selection position, is expressed with the value of the diameter.

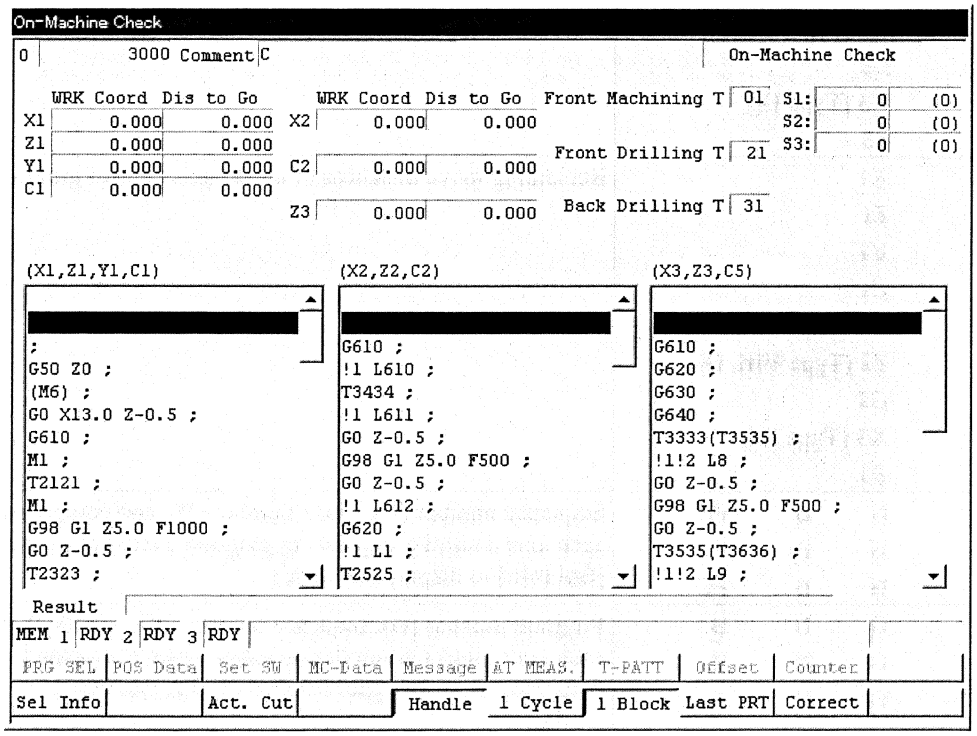
Note

As for a tool layout pattern, the values are determined according to the type of holder specified in the machining data.
As for a free tool layout pattern selected for special tooling, you can set arbitrary values.

6.6 On-Machine Check Screen

Press PROGRAM CHECK to display the On-Machine Check screen.

This screen operates the machine while program checking is in progress in the program check mode. You can stop the operation to edit the program being executed when required, and restart the operation later.



(This screen is for type VII.)

Display items

Item				Explanation
O				Program number of the execution program in the program check mode
WRK Coord	X1 Z1 Y1 C1 X2 (Type VII, VIII, IX) Z2 (Type VIII, IX) C2 X3 (Type IX) Z3			Work coordinates of all the axis control groups.
Dis to Go	X1 Z1 Y1 C1 X2 (Type VII, VIII, IX) Z2 (Type VIII, IX) C2 X3 (Type IX) Z3			Remaining move distances of all the axis control groups.
$1 $2 $3	N N N	B B B	FC FC FC	Sequence number (N), block number (B), and composite feed rate for each axis control group during program execution (Press the menu key [Sel Info] to display this item.)
$1 Sub $2 Sub $3 Sub	O O O	N N N	B B B	Program number (O), sequence number (N), and block number (B) for each axis control group during subprogram execution by the program (Press the menu key [Sel Info] to display this item.)
S1: S2: S3:				Specified speed of the main spindle (left) and actual speed (right) Specified speed of the back spindle (left) and actual speed (right) Specified speed of the tool spindle of the gang tool post (left) and actual speed (right)
Front machining Front drilling Back drilling	T T (Type VII, VIII, IX) T (Type VII, VIII, IX)			Tool number of the selected tool
Program display area				The block being executed is reversed on line 2. Stop the block and press the menu key [Correct] to edit the machining program from line 3 to the end of the program.

Note: The C1 and C2 axes are optional.

Main menu keys

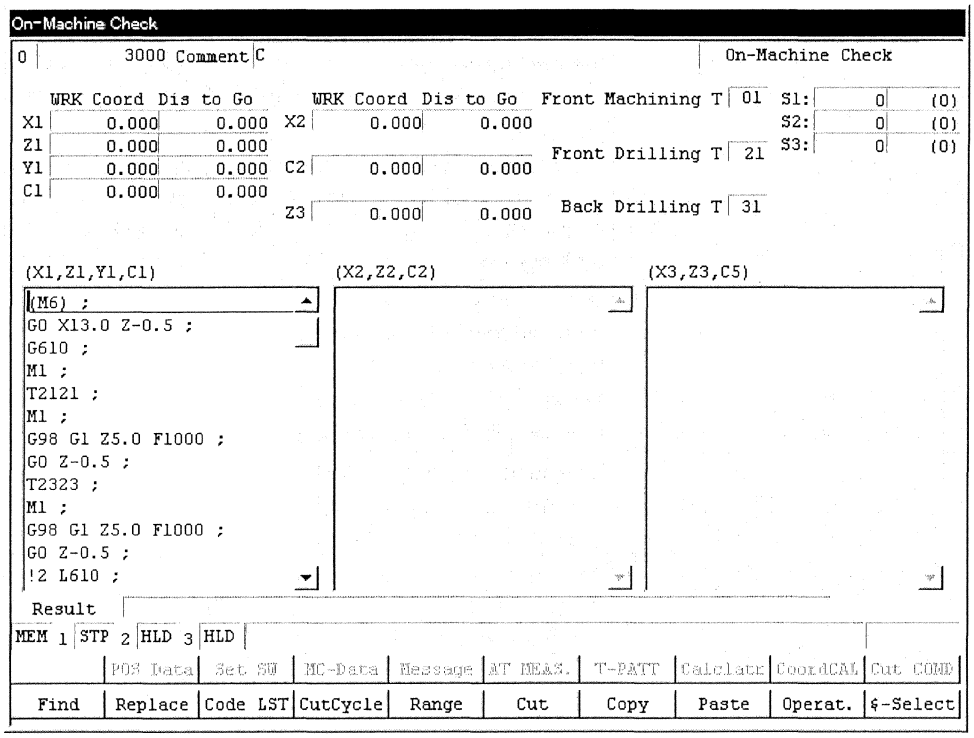
Key name	Function
Act. Cut	This menu key limits execution of thread cutting with the handle. The forward handle operation is suspended when the previous thread cutting block completes (block stop). Press the Start key to carry out the normal continuous operation between the thread cutting block and the next positioning block. When the program proceeds to the next positioning block for thread cutting, the handle is enabled. The reverse handle operation is suspended when the reverse movement to the second next positioning block for thread cutting (the end position of the next thread cutting block in the forward operation) completes. You can still turn the handle, but the reverse operation is no longer performed.
Handle	This menu key executes the program with the manual pulse handle. You can adjust the program execution speed to the rate at which you turn the handle. No matter how fast you turn the handle, the rate is limited to the feed rate specified in the program.
1 Cycle	This menu key automatically executes the program for one cycle. To execute 1-cycle operation by turning the handle, press the menu key [Handle] with [1 Cycle].
1 Block	This menu key automatically executes the program for one block. To execute 1-block operation by turning the handle, press the menu key [Handle] with [1 Block].
Last PRT	This menu key executes the last program.
Correct	This menu key activates the program correction (edit) mode.
Sel Info	This menu key switches the displayed information to sequence number (N), block number (B). Pressing the key again returns to the original display items.

Submenu keys

Key name	Function
PRG SEL	This menu key displays the Program Select screen for selecting the machining program to be executed in the on-machine program check mode.
POS Data	This menu key displays the machine coordinates, work coordinates, and remaining move distances of all the axis control groups.
Set SW	This menu key displays the Set SW screen for setting various control signals for NC unit operation to on/off.
MC-Data	This menu key displays the Machining Data screen.
Message	This menu key selects and displays the Message screen.
T-PATT	This menu key displays the Tool Pattern screen corresponding to the tool holder.
Offset	This menu key displays the Offset2 screen.
Counter	This menu key display the Counter screen

6.7 On-Machine Check (Edit) Screen

To display the On-Machine Check (Edit) screen, press the menu key [Correct] while the program operation is in the stop state or reset state in the on-machine program check mode.



(This screen is for type VII.)

Display items

See <Section 6.6 On-Machine Check Screen>.

Main menu keys

Key name	Function
Find	This menu key searches the program for a specified character string.
Replace	This menu key replaces a specified character string with a new character string in the program.
Code LST	This menu key displays a list of M or G codes.
CutCycle	This menu key inserts a prepared canned cycle program into an existing machining program.
Range	This menu key specifies the character string to be copied or cut.
Cut	This menu key cuts a character string specified by [Range]. The specified character string disappears.
Copy	This menu key copies a character string specified by [Range]. The specified character string remains as is.
Paste	This menu key inserts a cut or copied character string to the cursor position.
Operat.	This menu key terminates editing in the on-machine program check mode, and returns to the On-Machine Check screen.
$-Select	This menu key selects an axis control group.

Submenu keys

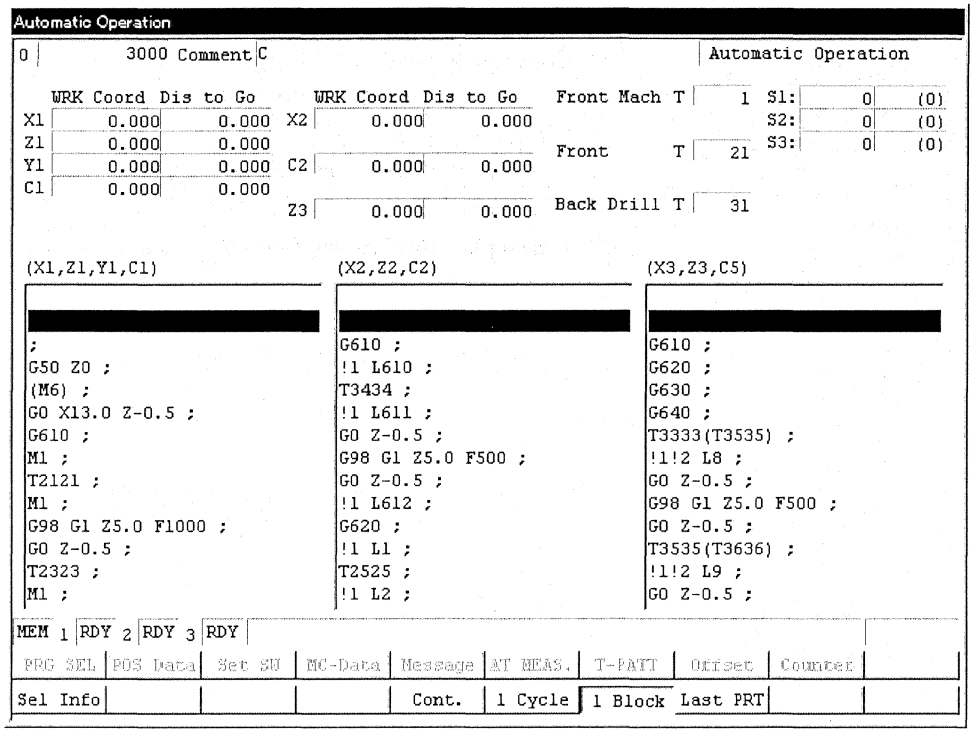
Key name	Function
POS Data	This menu key displays the machine coordinates, work coordinates, and remaining move distances of specified axes of all the axis control groups.
Set SW	This menu key displays the Set SW screen for setting various control signals for NC unit operation to on/off.
MC-Data	This menu key displays the Machining Data screen.
Message	This menu key selects and displays the Message screen.
T-PATT	This menu key displays the Tool Pattern screen corresponding to the tool holder.
Calclatr	This menu key displays the Calculator screen.
CoordCAL	This menu key displays the Coordinate Calculation screen.
Cut COND	This menu key displays the Cutting Conditions screen.

6.8 Automatic Operation Screen

Press 

to display the Automatic Operation screen.

This screen operates the machine in the automatic operation mode. Select the menu key [Cont.], [1 Cycle], or [1 Block] for automatic operation.



(This screen is for type VII.)

Display items

Item				Explanation
O				Program number of the program being executed in the automatic operation mode
WRK Coord		X1 Z1 Y1 C1 X2 (Type VII, VIII, IX) Z2 (Type VIII, IX) C2 X3 (Type IX) Z3		Work coordinates of all the axis control groups.
Dis to Go		X1 Z1 Y1 C1 X2 (Type VII, VIII, IX) Z2 (Type VIII, IX) C2 X3 (Type IX) Z3		Remaining move distances of all the axis control groups.
$1 $2 $3	N N N	B B B	FC FC FC	Sequence number (N), block number (B), and composite feed rate for each axis control group during program execution (Press the menu key [Sel Info] to display this item.)
$1 Sub $2 Sub $3 Sub	O O O	N N N	B B B	Program number (O), sequence number (N), and block number (B) for each axis control group during subprogram execution by the program (Press the menu key [Sel Info] to display this item.)
S1: S2: S3:				Specified speed of the main spindle (left) and actual speed (right) Specified speed of the back spindle (left) and actual speed (right) Specified speed of the tool spindle of the gang tool post (left) and actual speed (right)
Front Mach Front Back Drill		T T (Type VII, VIII, IX) T (Type VII, VIII, IX)		Tool number of the selected tool
Program display area				The program being executed is reversed on line 2.

Note: The C1 and C2 axes are optional.

Main menu keys

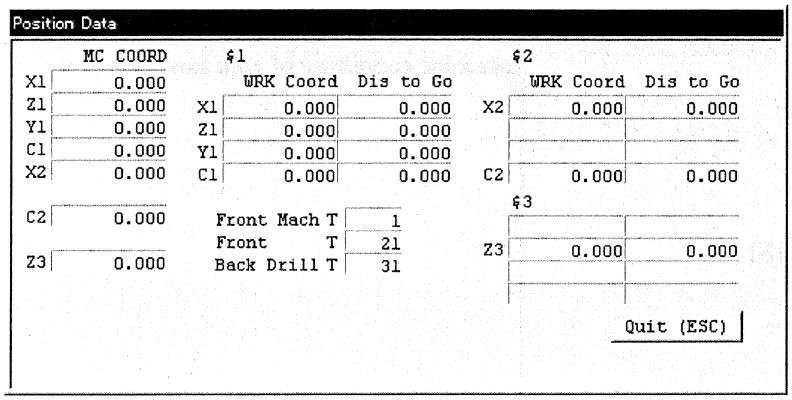
Key name	Function
Cont.	This menu key automatically executes the program continuously.
1 Cycle	This menu key automatically executes the program for one cycle.
1 Block	This menu key automatically executes the program for one block.
Last PRT	This menu key executes the last program.
Sel Info	This menu key switches the displayed information to sequence number (N), block number (B). Pressing the key again returns to the original display items.

Submenu keys

Key name	Function
PRG SEL	This menu key displays the Program Select screen for selecting the machining program to be executed in the automatic operation mode.
POS Data	This menu key displays the machine coordinates, work coordinates, and remaining move distances of specified axes of all the axis control groups.
Set SW	This menu key displays the Set SW screen for setting various control signals for NC unit operation to on/off.
MC-Data	This menu key displays the Machining Data screen.
Message	This menu key selects and displays the Message screen.
T-PATT	This menu key displays the Tool Pattern screen corresponding to the tool holder.
Offset	This menu key displays the Offset2 screen.
Counter	This menu key displays the Counter screen
B/L Set	This menu key displays the Bar Loader Setting screen. (optional)

6.9 Position Data Screen

Press PREPARATION, ▷, and [POS Data] in this order to display the Position Data screen. (This is one of methods to display the Position Data screen.)
This screen displays the position data of all axes.



(This screen is for type VII.)

Display items

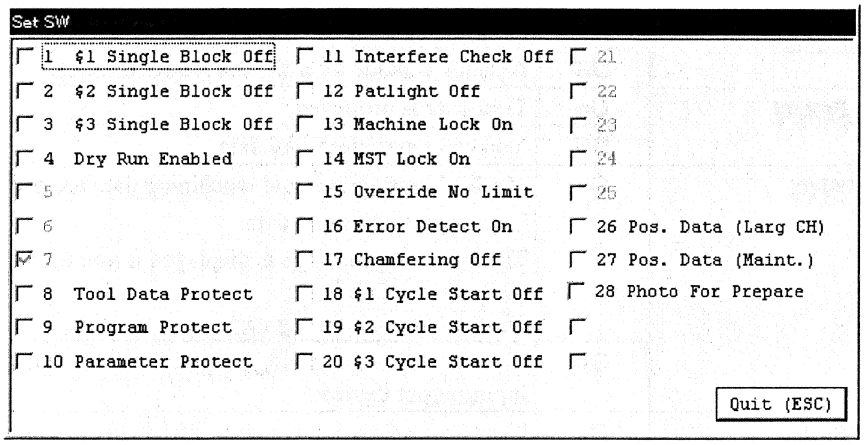
Item		Explanation
$1		Axis control groups
$2		
$3		
Front machining	T	Command values of T codes
Front drilling	T (Type VII, VIII, IX)	
Back drilling	T (Type VII, VIII, IX)	
MC COORD		Machine coordinate of each axis
X1		
Z1		
Y1		
C1		
X2 (Type VII, VIII, IX)		
Z2 (Type VIII, IX)		
C2		
X3 (Type IX)		
Z3		
WRK Coord		Work coordinates of all the axis control groups.
X1		
Z1		
Y1		
C1		
X2 (Type VII, VIII, IX)		
Z2 (Type VIII, IX)		
C2		
X3 (Type IX)		
Z3		
Dis to Go		Remaining move distances of all the axis control groups.
X1		
Z1		
Y1		
C1		
X2 (Type VII, VIII, IX)		
Z2 (Type VIII, IX)		
C2		
X3 (Type IX)		
Z3		

Note: The C1 and C2 axes are optional.

6.10 Set SW Screen

Press PREPA-RATION, ▷, and [Set SW] in this order to display the Set SW screen. (This is one of method to display the Set SW screen.)

This screen sets various control signals for NC unit operation to on/off.



Display items

Item	Setting	Explanation
1 $1 Single Block Off 2 $2 Single Block Off	On	Single block operation is disabled. Set the single block operation switch for each axis control group.
3 $3 Single Block Off	Off	Single block operation is enabled for each axis control group.
4 Dry Run Enabled	On	Dry run is enabled.
	Off	Dry run is disabled.
7 Loader Disconnect	On	No check is made on if the bar loader is on/off.
	Off	A check is made on if the bar loader is on/off.
8 Tool Data Protect	On	Tool data is protected.
	Off	You can enter/erase tool data.
9 Program Protect	On	Machining programs and machining data are protected. You cannot edit/input data. The following message is displayed if you attempt to input data while this switch is on: "Cannot set data during memory protection."
	Off	You can edit a machining program and enter data from an input/output device.
10 Parameter Protect	On	User parameters and common variables are protected. A user parameter is the generic name of control parameters, axis parameters, and preparation parameters.
	Off	You can enter/change user parameters and command variables.
11 Interfere Check Off	On	The interference check function is disabled.
	Off	The interference check function is enabled. The function is enabled when the power is turned on.
12 Patlight Off	On	The patrol lamp is not turned on when an alarm occurs.
	Off	The patrol lamp is turned on when an alarm occurs
13 Machine Lock On	On	The axis move command is disabled.
	Off	The axis move command is enabled.
14 MST Lock On	On	The functions of M, S, T, and B codes are locked.
	Off	The functions of M, S, T, and B codes are enabled.
15 Override No Limit	On	The upper limit of feed rate override is set to 200%. However, override over 100% is permitted for only the cutting feed rate. Even when you set 100 to 200%, the override is always limited to 100% for the rapid feed rate.
	Off	Even when you set the feed rate override dial to 100% or more, the override is limited to 100%.
16 Error Detect On	On	Errors are detected.
	Off	Errors are not detected.

Item	Setting	Explanation
17 Chamfering Off	On	Chamfering is disabled in the thread cutting cycle.
	Off	Chamfering is enabled in the thread cutting cycle.
18 $1 Cycle Start Off	On	The cycle start of a selected axis control group is disabled.
19 $2 Cycle Start Off 20 $3 Cycle Start Off	Off	The cycle start of a selected axis control group is enabled.
21 Auto Measure On	On	The automatic measure compensation device (optional) is enabled.
	Off	The automatic measure compensation device (optional) is disabled
22 Work hand ID chucking	On	Select this setting to chuck the inner diameter of a workpiece (by opening the workpiece hand). (optional)
	Off	Select this setting to chuck the outer diameter of a workpiece (by closing the workpiece hand). (optional)
26 Pos. Data (Larg CH)	On	Characters are displayed enlarged on the positional information screen.
	Off	Characters are displayed in standard size on the positional information screen.
27 Pos. Data (Maint.)	On	The positional information screen is displayed in maintenance mode.
	Off	The positional information screen is displayed in standard mode.

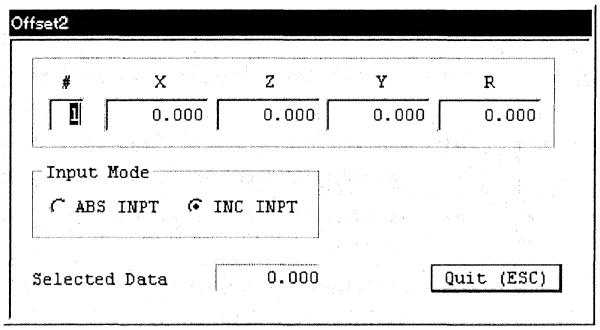
Note

- "7 Loader Disconnect" is not displayed when no bar loader is selected on the Machine Structure screen.
- "9 Program Protect" is not displayed for the EC specification.

6.11 Offset2 Screen

Press AUTO, ▷, and [Offset] in this order to display the Offset2 screen.

This screen sets values for tool wear compensation.



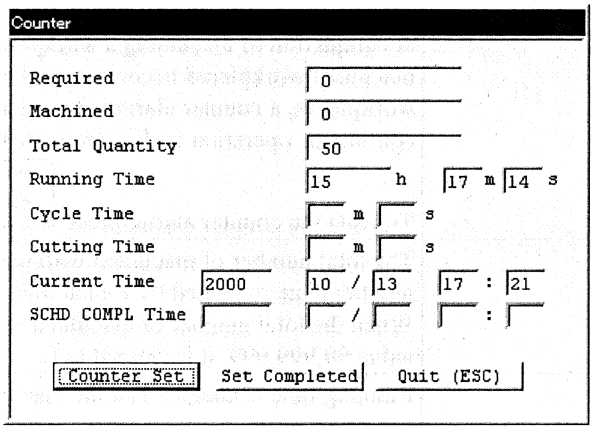
Display items

Item	Explanation
#	Enter the offset number.
X	Enter the value for tool wear compensation in the direction of X axis.
Z	Enter the value for tool wear compensation in the direction of Z axis.
Y	Enter the value for tool wear compensation in the direction of Y axis.
R	Enter the value for tool wear compensation nose radius.
Input Mode	Select the input mode for setting offset data. • ABS INPT An input value is used as the setting value. • INC INPT An input value is added to the preset value.

6.12 Counter Screen

Press AUTO, ▷, and [Counter] in this order to display the Counter screen.

The counter screen informs the progress of machining. If the required number of workpieces is set and automatic operation is executed continuously, the number of machined workpieces is incremented by one each time a workpiece is machined. When the number of machined workpieces becomes equal to the required number of workpieces, a counter alarm is issued and the machine stops.



Display items

Item	Explanation
Required	Set the number of workpieces to be machined. When the number of machined workpieces becomes equal to the required number of workpieces, a counter alarm is issued and the machine stops continuous operation in the automatic operation mode Up to 99,999,999 can be set as the required number of workpieces. If 0 is set, the counter does not count machined workpieces.
Machined	The number of machined workpieces is set/displayed. The preset counter is incremented by 1 each time the M56 command is executed at completion of machining a workpiece. When the number of machined workpieces becomes equal to the required number of workpieces, a counter alarm is issued and the machine stops continuous operation in the automatic operation mode. To reset the counter alarm, press  to clear the counter to 0.
Total Quantity	The total number of machined workpieces is displayed. The total number is incremented by 1 each time the M56 command is executed. When the total number of machined workpieces reaches the maximum value 99,999,999, it is cleared to 0.
Running Time	Running time is added when the machine is started in the automatic operation mode.
Cycle Time	One-cycle machining time in the automatic operation mode is measured and displayed. The time is measured each time 1-cycle machining is performed in the automatic operation, and the result is displayed at the end of each cycle. Up to 59 minutes 59 seconds can be measured. When the measured time reaches the maximum time, it is resumed to 0 minute 0 second, and time is measured and added again.
Cutting Time	Execution time of a command for actual machining during 1-cycle machining in the automatic operation mode is measured and displayed. When the cutting command is issued to either one of the three axis control group, all the actual machining duration is added up and indicated as the total execution time. The Cutting Time does not include the time required for tool selection or rapid feed positioning. Up to 59 minutes 59 seconds can be measured. When the measured time reaches the maximum time, it is resumed to 0 minute 0 second, and time is measured and added again.
Current Time	Current date and time
SCHD COMPL Time	Date and time scheduled when the number of machined workpieces becomes equal to the required number of workpieces
Counter Set	This button enables counter setting.
Set Completed	This button terminates counter setting.

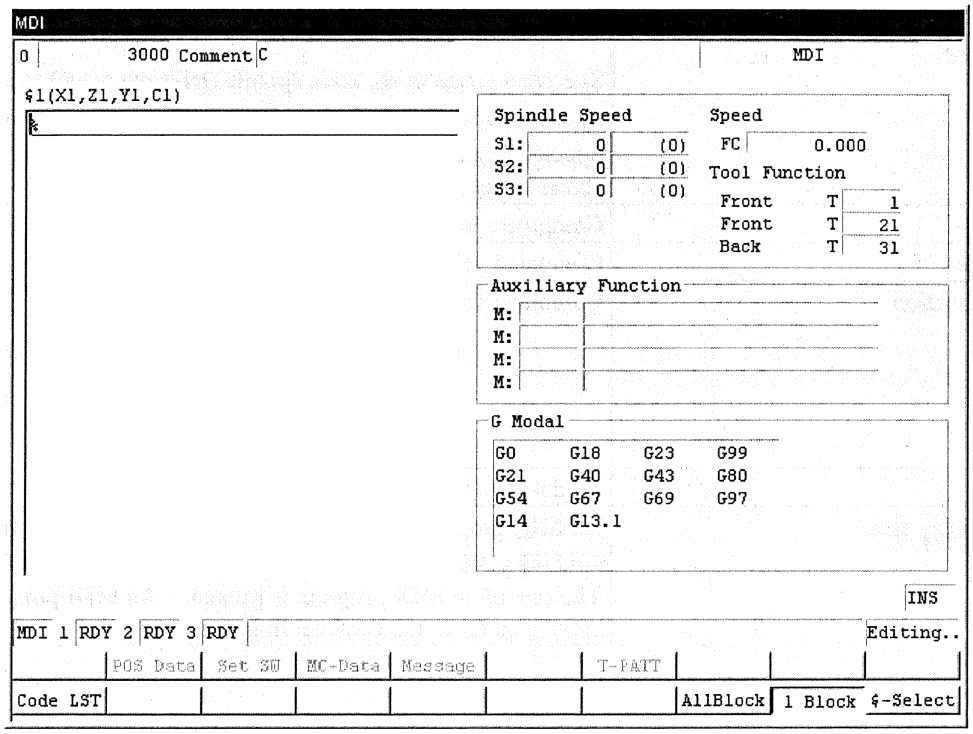
Note: The items Cycle Time, Cutting Time, and SCHD COMPL Time are only displayed. You cannot set these items.

6.13 MDI Screen

Press 

to display the MDI screen.

This screen creates and executes a program for MDI operation.



(This screen is for type VII.)

Display items

Item	Explanation
O	Program number of the currently selected execution program
$1	Currently selected axis control group
(X1, Z1, Y1, C1)	Axes of the currently selected axis control group Note: The C1 axis is optional.
Spindle Speed S1: S2: S3:	 Specified speed of the main spindle (left) and actual speed (right) Specified speed of the back spindle (left) and actual speed (right) Specified speed of the tool spindle of the gang tool post (left) and actual speed (right)
Speed	Composite feed rate or dwell time
Tool Function	Command value of the tool function
Auxiliary Function M: M: M: M:	Command value of the auxiliary function
G Modal	Modal status of the current G command
Program display area	An MDI program can be created for 3 axis control groups each, but only the program of the displayed axis control group is executed. The size of an MDI program is limited. An MDI program can be created as far as the contents fit into a screen.

Main menu keys

Key name	Function
AllBlock	This menu key continuously executes the first through final blocks of the MDI program entered in the MDI input area.
1 Block	This menu key executes the blocks of the MDI program (entered in the MDI input area) one by one, and the operation is stopped each time a block is executed.
$-Select	This menu key selects an axis control group.
Code LST	This menu key displays a list of M or G codes.

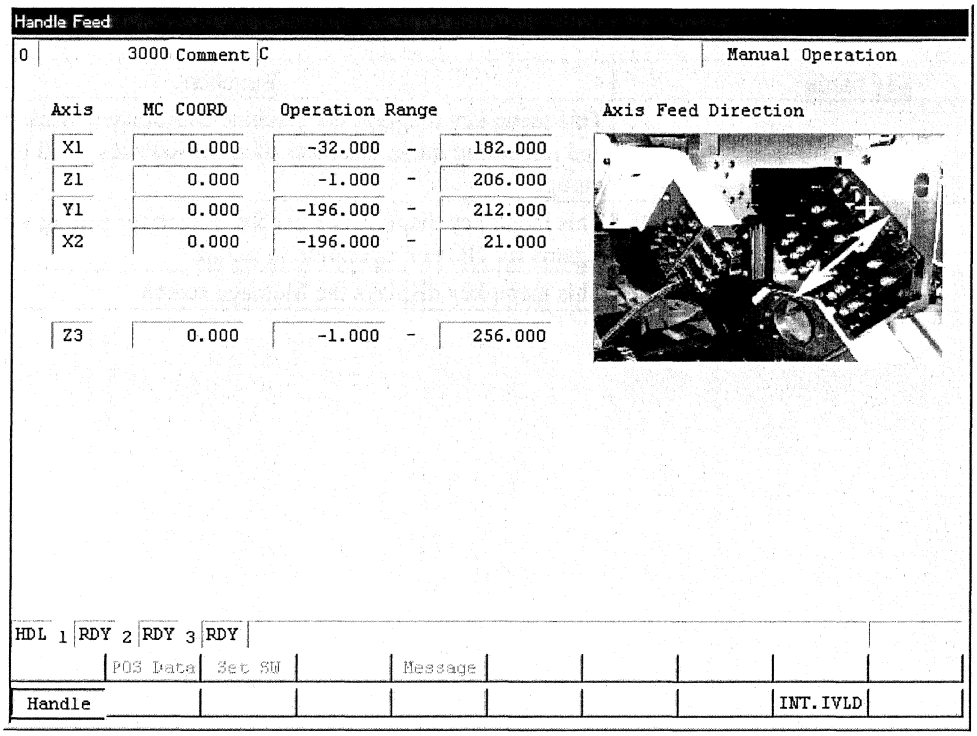
Submenu keys

Key name	Function
POS Data	This menu key displays the machine coordinates, work coordinates, and remaining move distances of specified axes of all the axis control groups.
Set SW	This menu key displays the Set SW screen for setting various control signals for NC unit operation to on/off.
MC-Data	This menu key displays the Machining Data screen.
Message	This menu key displays the Message screen.
T-PATT	This menu key displays the Tool Pattern screen corresponding to the tool holder.

6.14 Handle Feed Screen

Press MANUAL  to display the Handle Feed screen.

This screen moves each axis with the handle in the manual operation mode.



(This screen is for type VII.)

Display items

Item	Explanation
O	Program number of the currently selected execution program
Axis X1 Z1 Y1 X2 (Type VII, VIII, IX) Z2 (Type VIII, IX) X3 (Type IX) Z3	Names of the currently selected axes
MC COORD	Current machine coordinate of each axis
Operation Range	The range in which each axis can operate is displayed with machine coordinates. An attempt to operate an axis outside the range results in an alarm.
Axis Feed Direction	Direction in which the currently selected axis moves

Main menu keys

Key name	Function
Handle	This menu key displays the Handle Feed screen.
Int-Off	This menu key disable the interference check function.

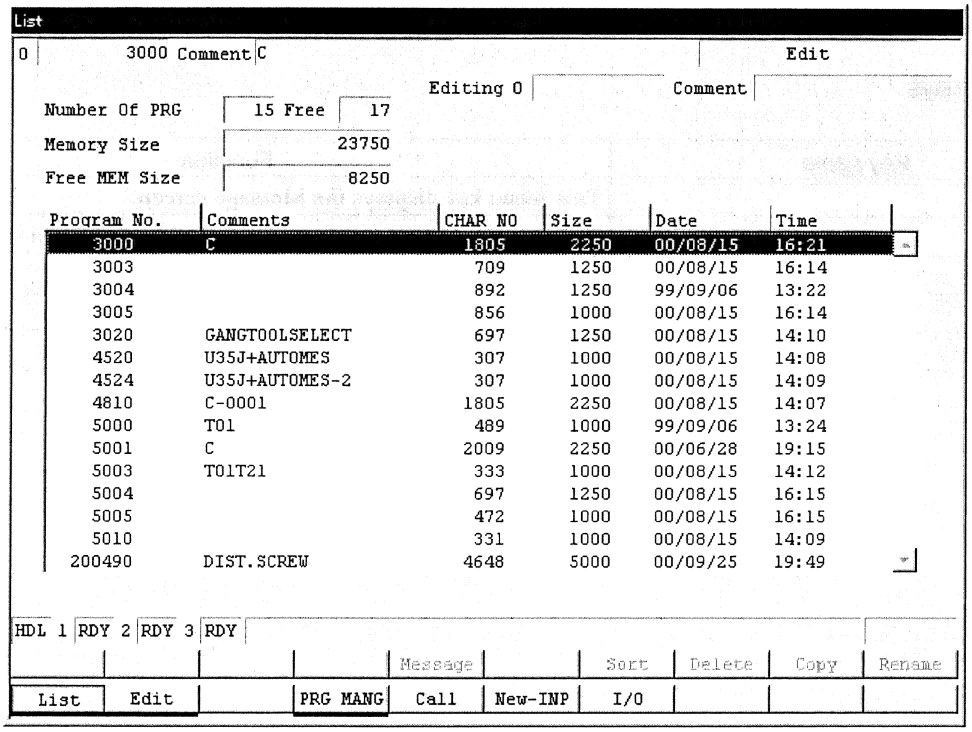
Submenu keys

Key name	Function
POS Data	This menu key displays the machine coordinates, work coordinates, and remaining move distances of specified axes of all the axis control groups.
Set SW	This menu key displays the Set SW screen for setting various control signals for NC unit operation to on/off.
Message	This menu key displays the Message screen.

6.15 List Screen

Press EDIT and then [List] to display the List screen.

This screen displays the list of machining programs registered in the memory.
Use the screen to create a new machining program, call a machining program, or input/output a machining program.



Display items

Item	Explanation
O	Program number of the currently selected execution program
Comment	Comment statement assigned to the currently selected execution program
Editing O	Program number of the program selected by the edit function
Comment	Comment statement assigned to the program selected by the edit function
Number Of PRG	Number of currently registered machining programs
Free	Number of machining programs that can be still registered
Memory Size	Memory size used with all the currently registered machining programs
Free MEM Size	Memory size available for registering more machining programs
Program No.	Program numbers of the machining programs registered in the memory
Comment	Comments on machining programs
CHAR NO	Number of characters in each machining program
Size	Size of each machining program
Date	Date when each machining program was created/corrected
Time	Time when each machining program was created/corrected

Main menu keys

Key name	Function
List	This menu key displays the List screen.
Edit	This menu key displays the Edit screen.
PRG MANG	This menu key displays the Program Management screen.
Call	This menu key displays the Calling Up Program screen.
New-INP	This menu key displays the Create New Program screen
I/O	This menu key displays the Input/Output screen.

Submenu keys

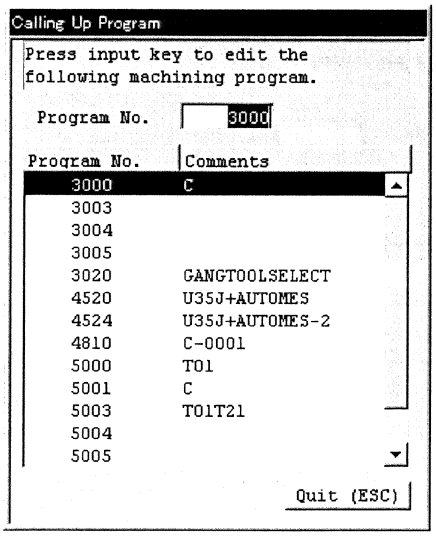
Key name	Function
Message	This menu key displays the Message screen.
Sort	This menu key displays the List Display Arrangement screen.
Delete	This menu key displays the Program Delete screen.
Copy	This menu key displays the Program Copy screen.
Rename	This menu key displays the Program Rename screen.

6.15.1 Calling Up Program screen

Press 

, [List], and [Call] in this order to display the Calling Up Program screen.

This screen calls the machining program and machining data you want to edit.



Display items

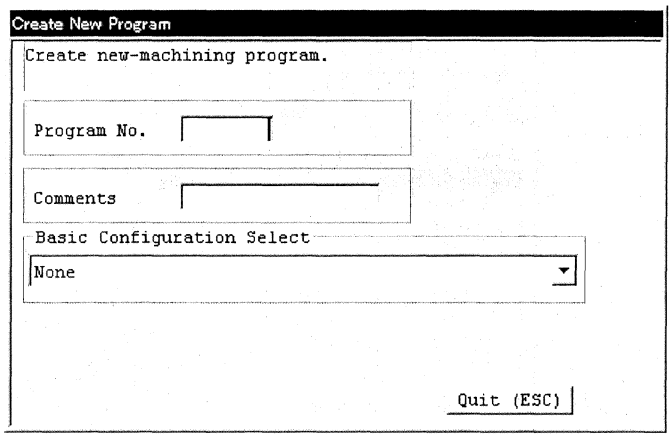
Item	Explanation
Program No. (upper)	Enter the program number of the program you want to edit.
Program No. (lower)	List of the program numbers of currently registered programs
Comments	Comments on currently registered programs

6.15.2 Create New Program screen

Press 

, [List], and [New-INP] in this order to display the Create New Program screen.

This screen creates a new program.



Display items

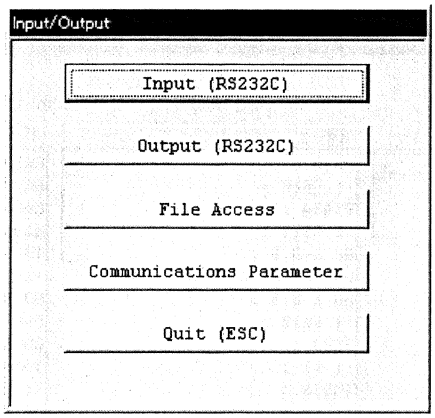
Item	Explanation
Program No.	Enter the program number of a new machining program you are going to create.
Comments	Enter comments on the functions, specifications, or usage of the machining program. You can enter comments with up to 18 characters such as alphanumeric characters, and symbol characters. (You can omit comments.)
Basic Configuration Select	Basic program structures are provided in several patterns. Set a basic structure to create a new program. The provided pattern is automatically inserted.

6.15.3 Input/Output screen

Press 

, [List], and [I/O] in this order to display the Input/Output screen.

This screen inputs/outputs a machining program.



Display items

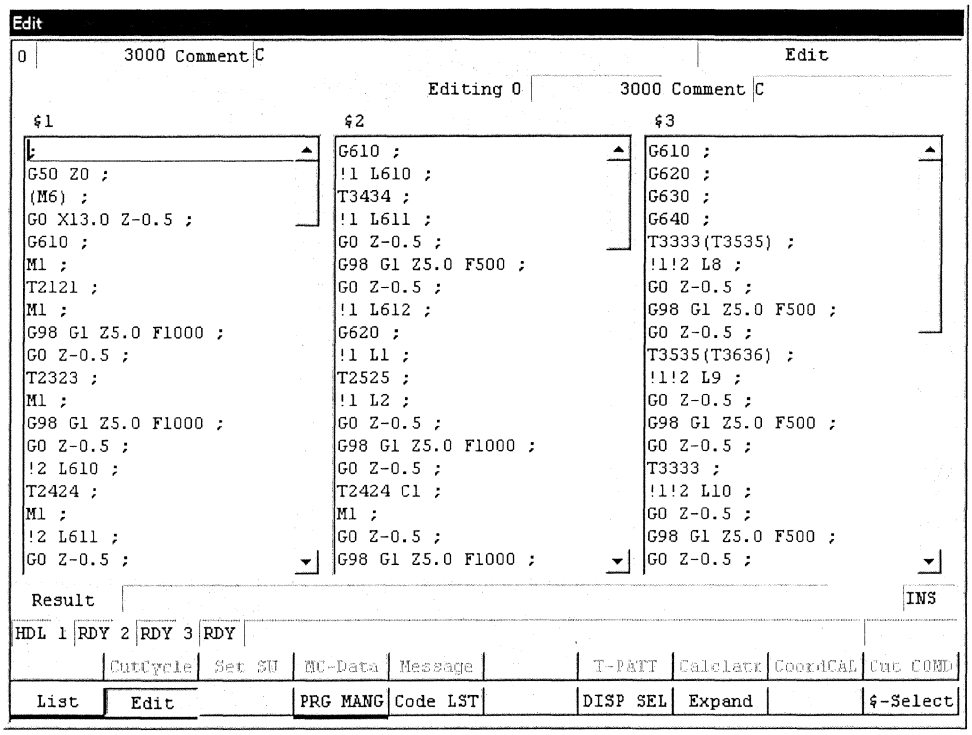
Item	Explanation
Input (RS232C)	Input a machining program, tool offset data, and parameter through RS232C from the external input/output device (IC card reader/writer).
Output (RS232C)	Output a machining program, tool offset data, and parameter to the external input/output device (IC card reader/writer) through RS232C.
File Access	Input or output a machining program, tool offset data, and parameter to floppy disk drive, PCMCIA, HDD, or front IC card slot.
Communications Parameter	Set communication parameters.

6.16 Edit Screen

Press 

to display the Edit screen.

This screen edits a machining program.



Display items

Item	Explanation
O	Program number of the currently selected execution program
Comment	Comment statement assigned to the currently selected execution program
Editing O	Program number of the program selected by the edit function
Comment	Comment statement assigned to the program selected by the edit function
Result	Area for displaying the results of calculation with the calculator, etc.

Main menu keys

Key name	Function
List	This menu key displays the List screen.
Edit	This menu key displays the Edit screen.
PRG MANG	This menu key displays the Program Management screen.
Code LST	This menu key displays the Code List screen.
DISP SEL	This menu key displays the Display Selection menu.
Expand	This menu key displays the Expanded Edit menu.
$-Select	This menu key selects an axis control group for editing.

Submenu keys

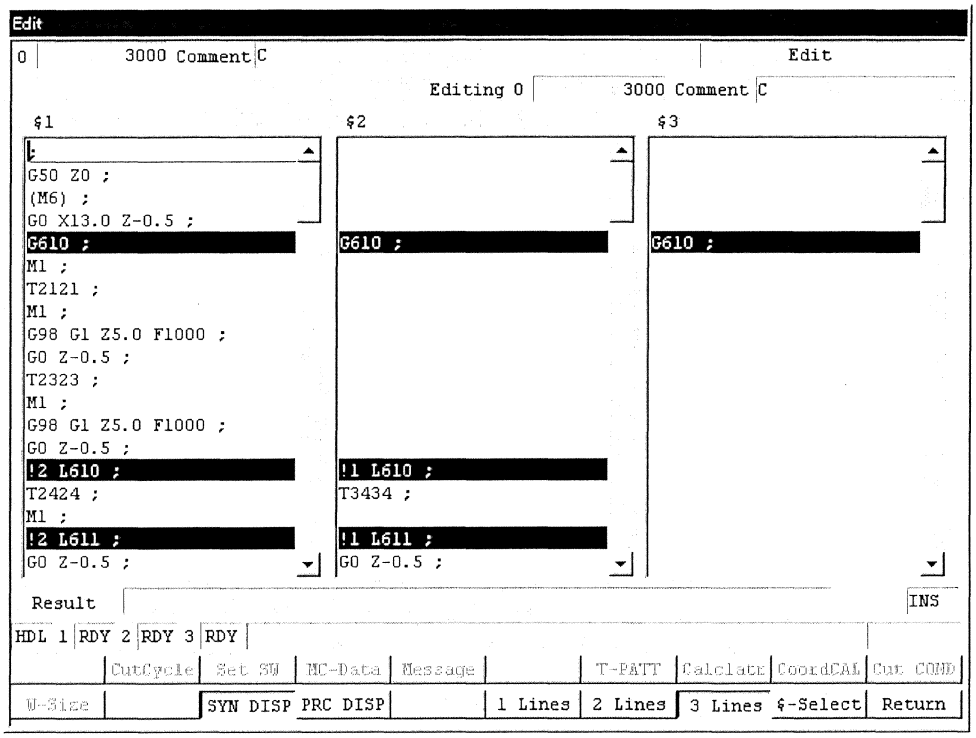
Key name	Function
CutCycle	This menu key displays the Cutting Canned Cycle Insertion screen.
Set SW	This menu key displays the Set SW screen for setting various control signals for NC unit operation to on/off.
MC-Data	This menu key displays the Machining Data screen.
Message	This menu key displays the Message screen.
T-PATT	This menu key displays the Tool Pattern screen corresponding to the tool holder.
Calclatr	This menu key displays the Calculator screen.
CoordCAL	This menu key displays the Coordinate Calculation screen.
Cut COND	This menu key displays the Cutting Conditions screen.

6.16.1 Display Selection menu

Press 

and then [DISP SEL] to display the Display Selection menu.

These menu keys change the display format of the machining program being edited.



Display items

See <Section 6.16 Edit Screen>.

Main menu keys

Key name	Function
W-Size	Pressing this button enlarges characters written in the machining program when they are displayed. Pressing it again sets them back into standard size.
SYN DISP	This menu key aligns lines with the queuing command (! symbol) in the same raw throughout the axis control groups.
PRC DISP	This menu key displays only the T codes and queuing command (! symbol) in the program.
1 Lines	This menu key displays the programs of axis control groups 1.
2 Lines	This menu key displays the programs of axis control groups 1 and 2.
3 Lines	This menu key displays the programs of axis control groups 1, 2, and 3.
$-Select	This menu key selects an axis control group for editing.
Return	This menu key terminates the Display Selection menu and returns to the Edit screen.

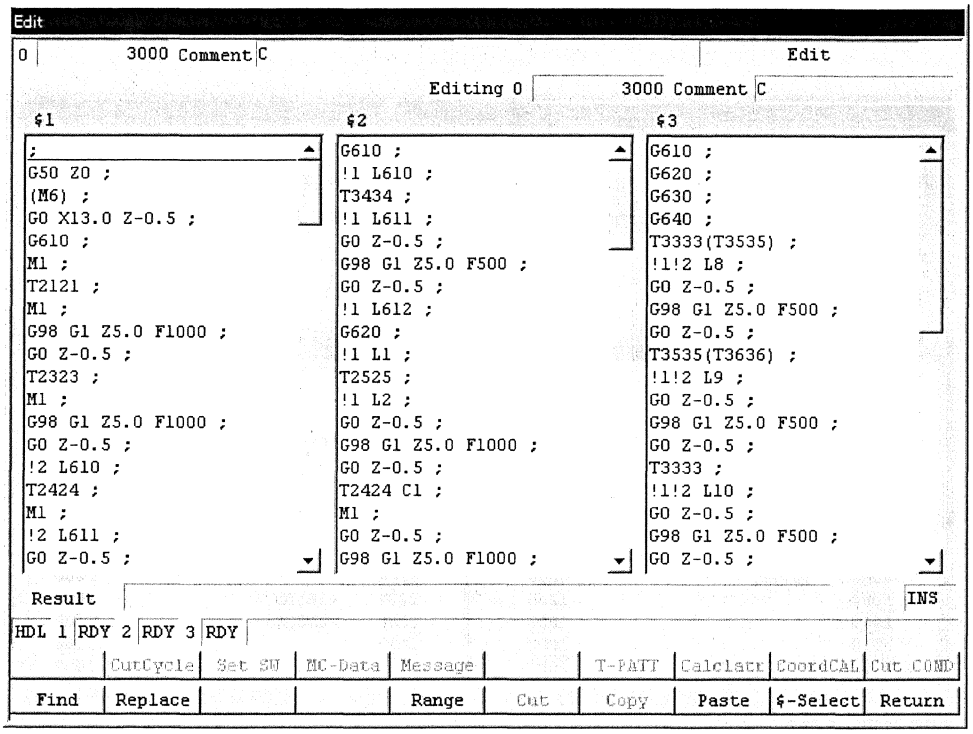
Submenu keys

See <Section 6.16 Edit Screen>.

6.16.2 Expanded Edit menu

Press 

and then [Expand] to display the Expanded Edit menu.



Display items

See <Section 6.16 Edit Screen>.

Main menu keys

Key name	Function
Find	This menu key searches the program for a specified character string.
Replace	This menu key replaces a specified character string with a new character string in the program.
Range	This menu key specifies the character string to be copied or cut.
Cut	This menu key cuts a character string specified by [Range]. The specified character string disappears.
Copy	This menu key copies a character string specified by [Range]. The specified character string remains as is.
Paste	This menu key inserts a cut or copied character string to the cursor position.
$-Select	This menu key selects an axis control group for editing.
Return	This menu key terminates the Expanded Edit menu and returns to the Edit screen.

Submenu keys

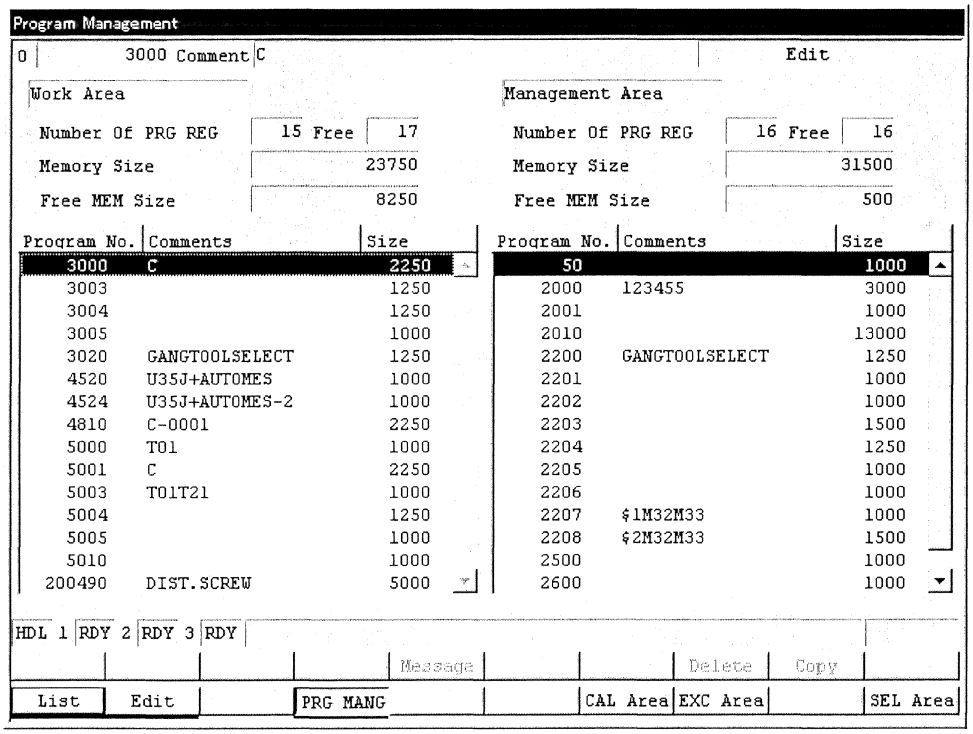
See <Section 6.16 Edit Screen>.

6.17 Program Management Screen

Press 

and then [PRG MANG] to display the Program Management screen.

This screen displays the list of all the areas for program management.
Use the screen to call an area or exchange areas.



Display items

Item	Explanation
O	Program number of the currently selected execution program
Comment	Comment statement assigned to the currently selected execution program
Work Area Management Area	Area name. The name of the currently selected area is reversed in yellow.
Number Of PRG REG	Number of currently registered machining programs
Free	Number of machining programs that can be still registered
Memory Size	Memory size used with all the currently registered machining programs
Free MEM Size	Memory size available for registering more machining programs
Program No.	Program numbers of the machining programs registered in the memory
Comment	Comments on machining programs
Size	Size of each machining program

Main menu keys

Key name	Function
List	This menu key displays the List screen.
Edit	This menu key displays the Edit screen.
PRG MANG	This menu key displays the Program Management screen.
CAL Area	This menu key calls and displays the area with which you want to work.
EXC Area	This menu key exchanges the work area contents with the management area contents.
SEL Area	This menu key selects either of the two displayed area for which processing is performed.

Submenu keys

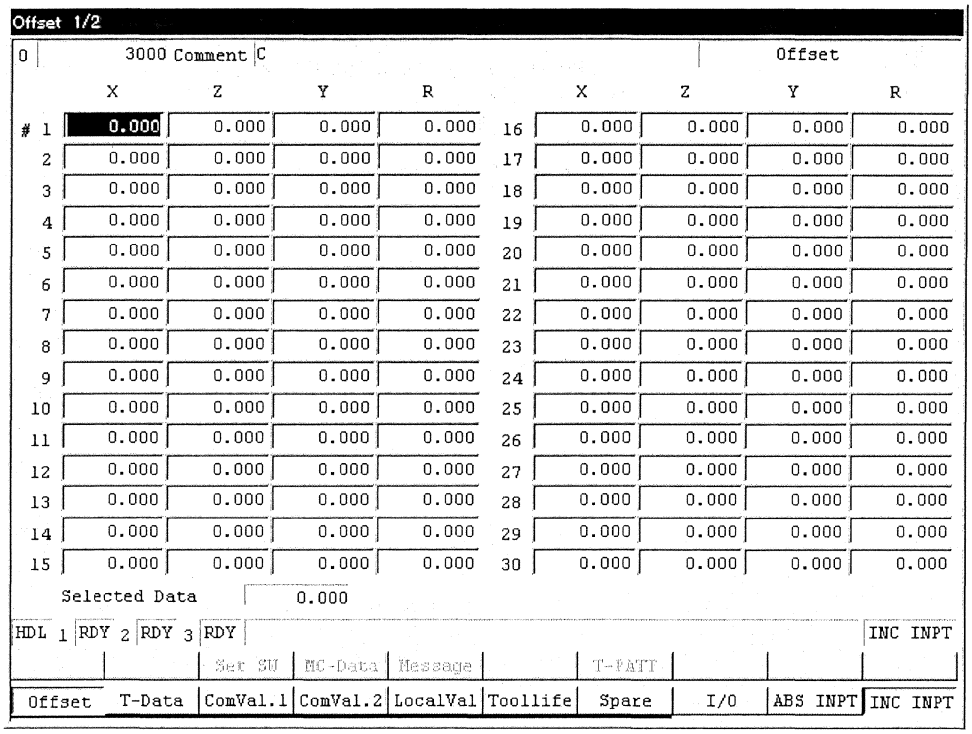
Key name	Function
Message	This menu key displays the Message screen.
Delete	This menu key deletes the program from the selected area.
Copy	This menu key copies programs between the displayed two areas.

6.18 Offset Screen

Press 

to display the Offset screen.

This screen sets values for tool wear compensation.



Display items

Item	Explanation
O	Program number of the currently selected execution program
#	Offset number
X	Value for tool wear compensation in the direction of X axis
Z	Value for tool wear compensation in the direction of Z axis
Y	Value for tool wear compensation in the direction of Y axis
R	Value for tool wear compensation of nose R

Main menu keys

Key name	Function
Offset	This menu key displays the Offset screen.
T-Data	This menu key displays the Tool Data screen.
ComVal.1	This menu key displays the Common Variable 1 screen.
ComVal.2	This menu key displays the Common Variable 2 screen.
LocalVal	This menu key displays the Local Variable screen.
Toollife	This menu key displays the Tool Life Data screen.
Spare	This menu key displays the Spare Tool screen.
I/O	This menu key displays the Input/Output screen.
ABS INPT	This menu key selects the absolute input method for offset data.
INC INPT	This menu key selects the incremental input method for offset data.

Submenu keys

Key name	Function
Set SW	This menu key displays the Set SW screen for setting various control signals for NC unit operation to on/off.
MC-Data	This menu key displays the Machining Data screen.
Message	This menu key selects and displays the Message screen.
T-PATT	This menu key displays the Tool Pattern screen corresponding to the tool holder.

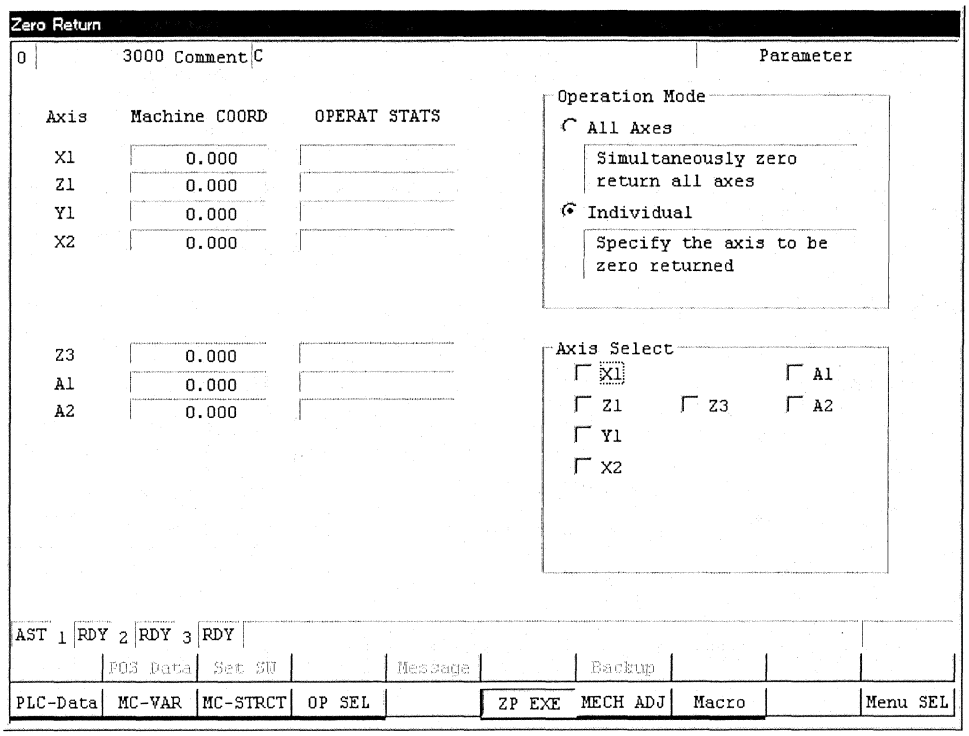
Note

The Tool Life Data screen and Spare Tool screen are optional. The screens are available only when the following requirements are cleared: Management of Tool life (optional) has been purchased or it is set on the Selection of using option screen, and "11 Management of Tool life" is checked on the Control Parameter screen.

6.19 Zero Return Screen

Press PRM. and then [ZP EXE] to display the Zero Return screen.

This screen returns each axis to the machine zero point.



(This screen is for type VII.)

Display items

Item	Explanation
O	Program number of the currently selected execution program
Axis X1 Z1 Y1 X2 (Type VII, VIII, IX) Z2 (Type VIII, IX) X3 (Type IX) Z3 A1 A2 (A3) (A7)	Axes which return to zero point The axis that is selected to return to zero point is reversed.
Machine COORD	Current machine coordinate of each axis
OPERAT STATS	The message "Executing Zero Return" is displayed for axes that are returning to zero point. The message "Completed Zero Return" is displayed for axes that have reached zero point.
Operation Mode All Axes Individual	 All the axes are selected to return to zero point. An axis, which is selected in the Axis Select field, returns to zero point.
Axis Select X1 X3 (Type IX) A1 Z1 Z2 (Type VIII, IX) A2 Y1 Z3 (A3) X2 (Type VII, VIII, IX) (A7)	When selecting Individual, select an axis that returns to zero point. You can select more than one axis.

Note: The A3 and A7 axes are optional.

Main menu keys

Key name	Function
PLC-Data	This menu key displays the PLC Data screen.
MC-VAR	This menu key displays the Machine Variable screen.
MC-STRCT	This menu key displays the Machine Structure screen.
OP SEL	This menu key displays the Selection of using option screen.
ZP EXE	This menu key displays the Zero Return screen.
MECH ADJ	This menu key displays the Mechanism Adjustment screen.
Macro	This menu key displays the Macro File screen.

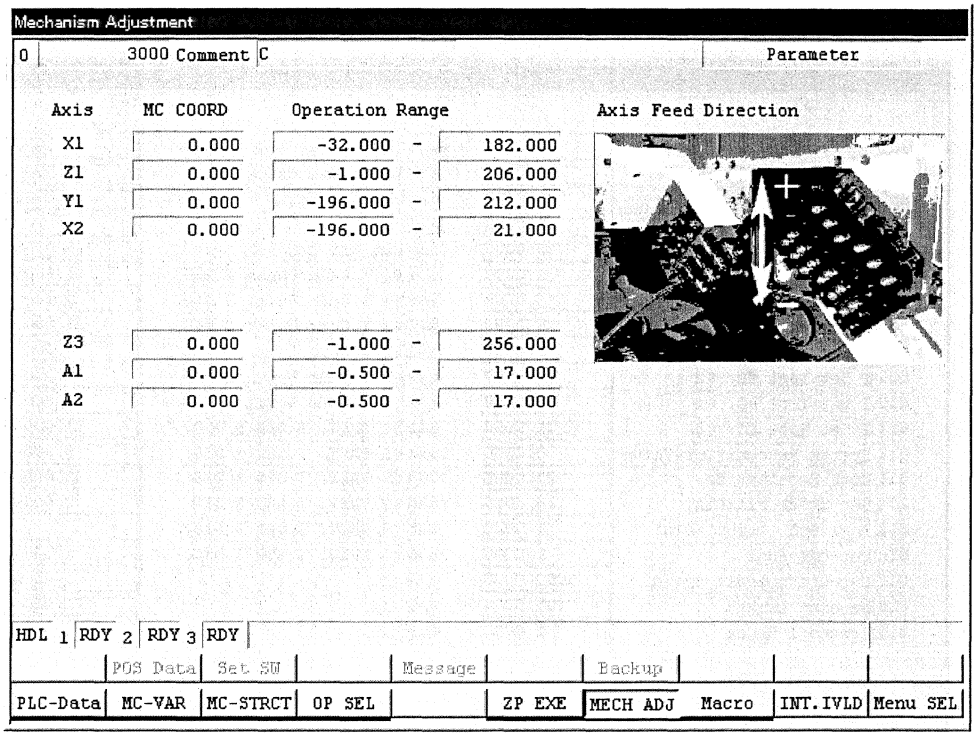
Submenu keys

Key name	Function
POS Data	This menu key displays the machine coordinates, work coordinates, and remaining move distances of specified axes of all the axis control groups.
Set SW	This menu key displays the Set SW screen for setting various control signals for NC unit operation to on/off.
Message	This menu key displays the Message screen.
Backup	This menu key displays the Backup screen.

6.20 Mechanism Adjustment Screen

Press PRM. and then [MECH ADJ] to display the Mechanism Adjustment screen.

In the machine coordinate system, this screen moves each axis with the handle manually.



(This screen is for type VII.)

Display items

Item	Explanation
O	Program number of the currently selected execution program
Axis X1 Z1 Y1 X2 (Type VII, VIII, IX) Z2 (Type VIII, IX) X3 (Type IX) Z3 A1 A2 (A3) (A7)	The name of the currently selected axis is reversed.
MC COORD	Current machine coordinate of each axis
Operation Range	The range in which each axis can operate is displayed with machine coordinates. An attempt to operate an axis outside the range results in an alarm.
Axis Feed Direction	Direction in which the currently selected axis moves

Notes

The A3 and A7 axes are optional.

6.21 Machine Variable Screen

Press 

and then [MC-VAR] to display the Machine Variable screen.

This screen displays variables that are mainly used with the machine manufacturer's macros.

Machine Variable 1/23

O 3000 Comment C

		Parameter			Parameter
#800			#820	Tubing Bar I.D. (DIA)	0.0000
801	Bar Stock DIA Limit	18.0000	821	B/Spindle Chuck POS.	20.0000
802	Mach. Pattern Level 1	600.0000	822	Cut-Off Feed	0.0300
803	Mach. Pattern Level 2	0.0000	823		
804	Mach. Pattern Level 3	0.0000	824	Cut-Off End POS	-3.0000
805	???	0.0000	825	$1 T Code Upper 2 DIG	1.0000
806	Mac. Pat. Z Axis Pos.	-5.0000	826	$1 T Code Lower 2 DIG	0.0000
807	Stored X1 Tool Set Dia.	0.0000	827	$2 T Code Upper 2 DIG	21.0000
808	Selected Last Tool No.	1.0000	828	$2 T Code Lower 2 DIG	0.0000
809	T Set Tool NO. $2	21.0000	829	$3 T Code Upper 2 DIG	31.0000
810	T Set Tool NO. $1	2.0000	830	$3 T Code Lower 2 DIG	0.0000
811	T Set Tool NO. $3	31.0000	831	$1 PREV. T UPP 2 DIG.	1.0000
812	System Recognition Value	3.0000	832	$1 PREV. T LOW 2 DIG.	0.0000
813	Work Receiver No.	20.0000	833	$2 PREV. T UPP 2 DIG.	31.0000
814	Bar Stock DIA(DIA)	16.0000	834	$2 PREV. T LOW 2 DIG.	0.0000
815	Tool POS. Point (DIA)	1.0000	835	$3 PREV. T UPP 2 DIG.	
816	Cut-Off Tool	1.0000	836	$3 PREV. T LOW 2 DIG.	1.0000
817	Cut-Off Spindle Speed	3000.0000	837	???	1.0000
818	Machine Length	100.0000	838	???	21.0000
819	Pieces/1 Chuck	1.0000	839	???	31.0000

HDL 1 | RDY 2 | RDY 3 | RDY

POS Data | Ser SW | Message | Backup

PLC-Data | MC-VAR | MC-STRCT | OP SEL | ZP EXE | MECH ADJ | Macro | I/O | Menu SEL

Display items

Item	Explanation
O	Program number of the currently selected execution program
#	Variable numbers
(Name)	Variable names
(Numeric value)	Values of variables

6.22 Machine Structure Screen

Press 

and then [MC-STRCT] to display the Machine Structure screen.

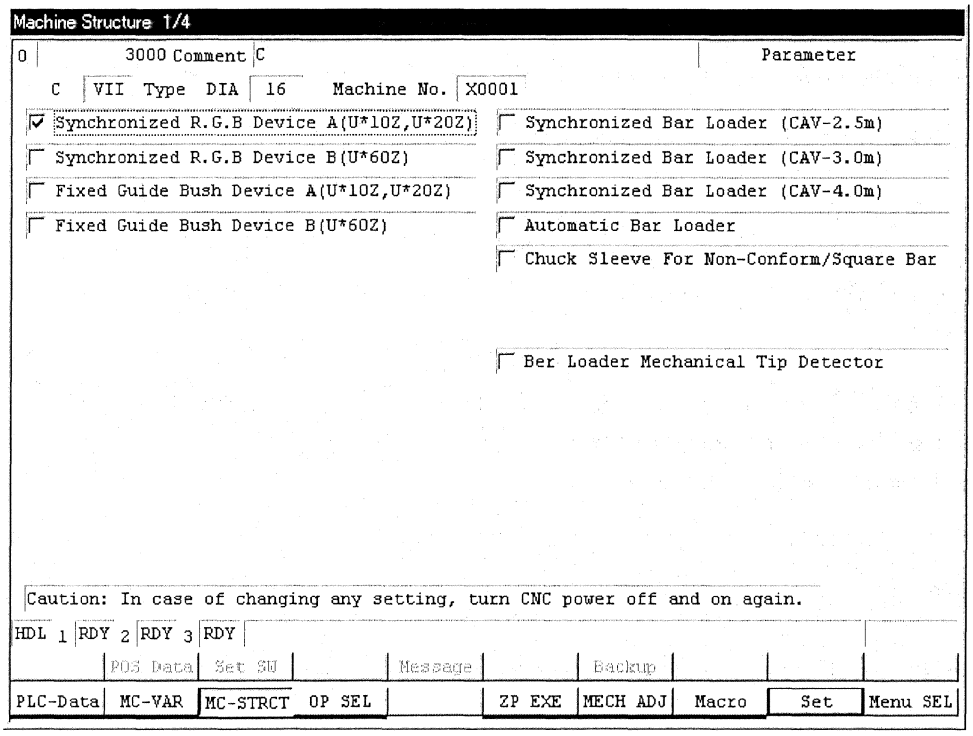
The Machine Structure screen consists of four pages.

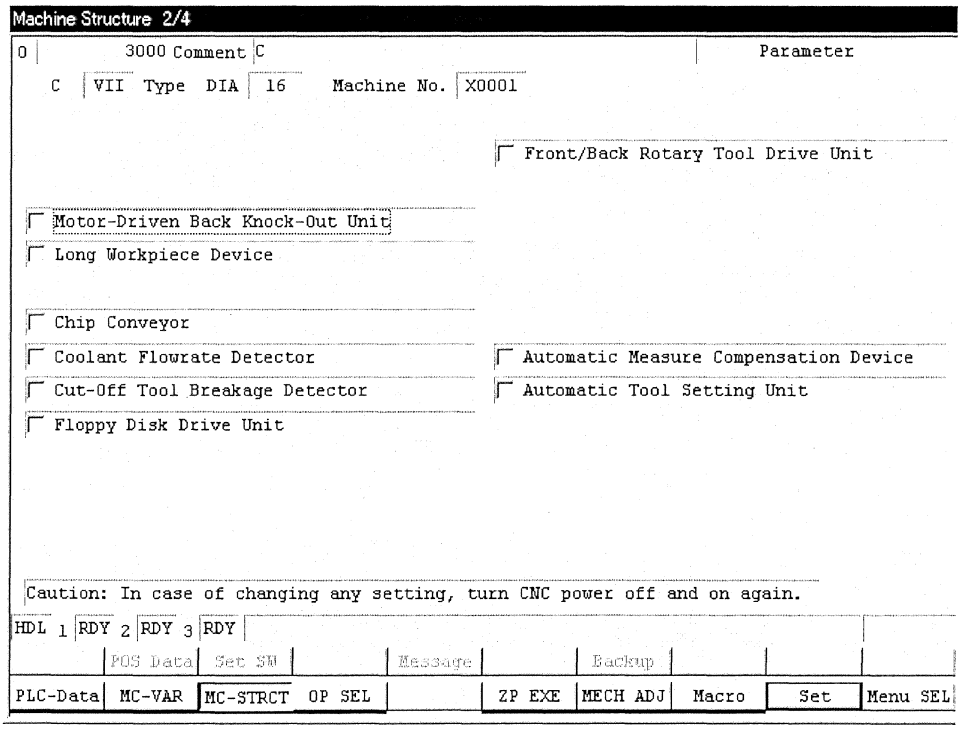
Page 1, 2: Machine Structure screen 1/2 is provided for checking if optional devices are mounted. When mounting a device listed on this screen, check the device. When dismounting it, uncheck the device. Press the menu key [Set] to check the optional devices listed on this screen.

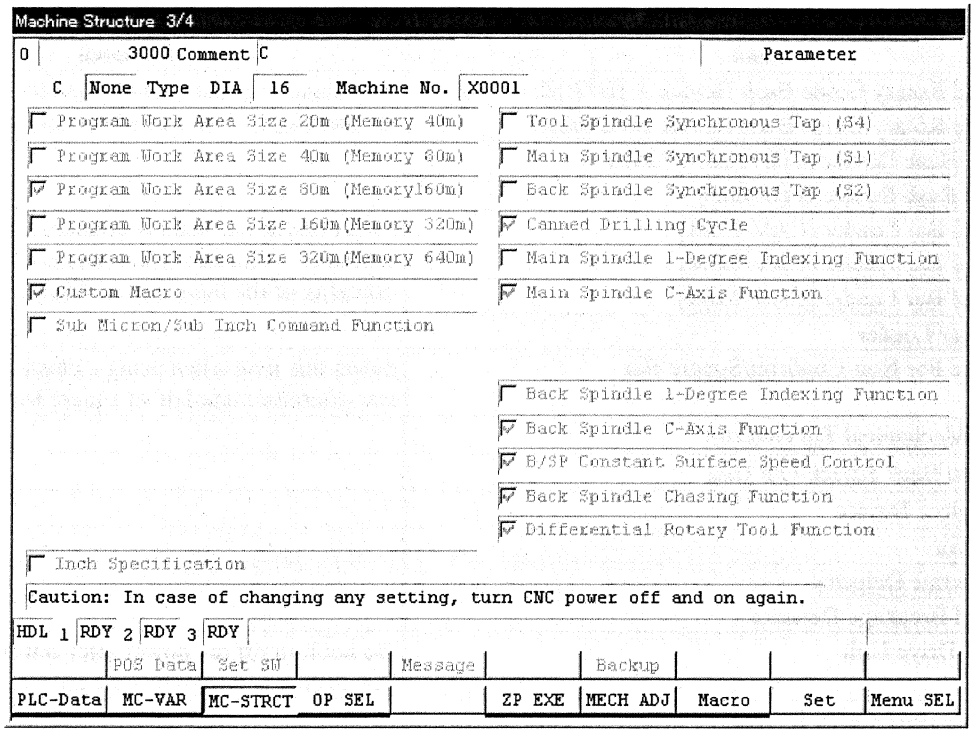
Page 3, 4: Machine Structure screen 2/2 is provided for checking if optional software is mounted. The screen lists the optional software mounted in the machine. You cannot check any optional software listed on this screen.

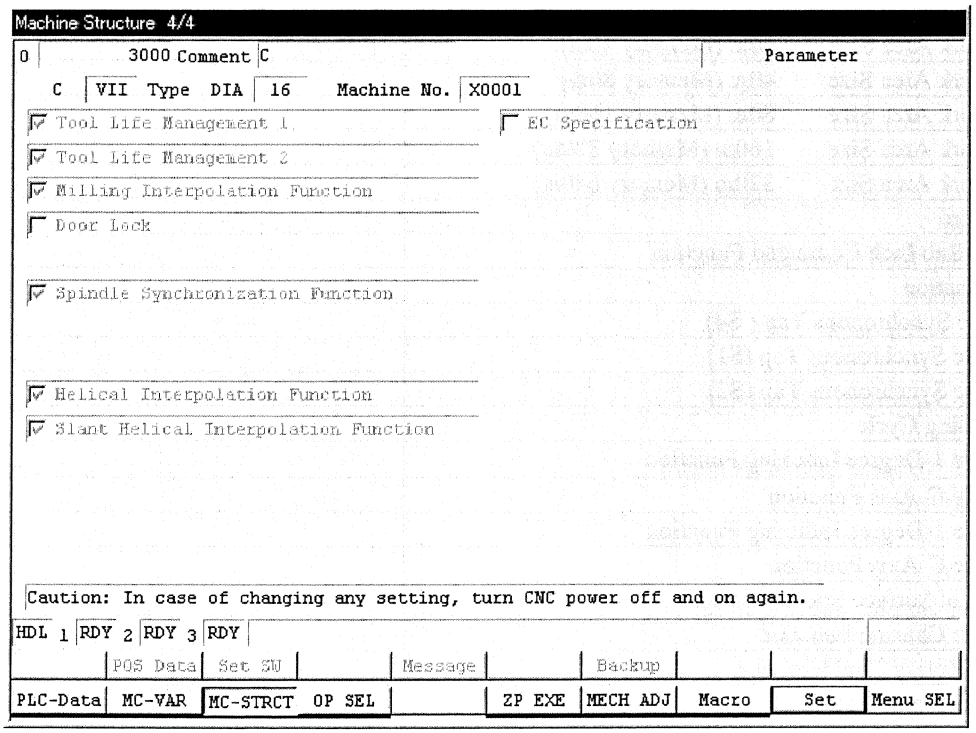
 CAUTION

Do not operate the machine while the settings on this screen do not match the actual configuration of the machine. Doing so may cause an alarm or machine damage. Set the items on this screen correctly.


Machine Structure 1/4
0
3000 Comment C
Parameter
C
VII Type DIA 16
Machine No. X0001
Synchronized R.G.B Device A(U*10Z,U*20Z)
Synchronized R.G.B Device B(U*60Z)
Fixed Guide Bush Device A(U*10Z,U*20Z)
Fixed Guide Bush Device B(U*60Z)
Synchronized Bar Loader (CAV-2.5m)
Synchronized Bar Loader (CAV-3.0m)
Synchronized Bar Loader (CAV-4.0m)
Automatic Bar Loader
Chuck Sleeve For Non-Conform/Square Bar
Ber Loader Mechanical Tip Detector
Caution: In case of changing any setting, turn CNC power off and on again.
HDL 1 RDY 2 RDY 3 RDY
POS Data Set SW Message Backup
PLC-Data MC-VAR MC-STRCT OP SEL ZP EXE MECH ADJ Macro Set Menu SEL


Machine Structure 2/4
0
3000 Comment C
Parameter
C
VII Type DIA 16
Machine No. X0001
Front/Back Rotary Tool Drive Unit
Motor-Driven Back Knock-Out Unit
Long Workpiece Device
Chip Conveyor
Coolant Flowrate Detector
Cut-Off Tool Breakage Detector
Floppy Disk Drive Unit
Automatic Measure Compensation Device
Automatic Tool Setting Unit
Caution: In case of changing any setting, turn CNC power off and on again.
HDL 1 RDY 2 RDY 3 RDY
POS Data Set SW Message Backup
PLC-Data MC-VAR MC-STRCT OP SEL ZP EXE MECH ADJ Macro Set Menu SEL


Machine Structure 3/4
0
3000 Comment C
Parameter
C None Type DIA 16 Machine No. X0001
Program Work Area Size 20m (Memory 40m)
Program Work Area Size 40m (Memory 80m)
Program Work Area Size 80m (Memory160m)
Program Work Area Size 160m(Memory 320m)
Program Work Area Size 320m(Memory 640m)
Custom Macro
Sub Micron/Sub Inch Command Function
Inch Specification
Tool Spindle Synchronous Tap (S4)
Main Spindle Synchronous Tap (S1)
Back Spindle Synchronous Tap (S2)
Canned Drilling Cycle
Main Spindle 1-Degree Indexing Function
Main Spindle C-Axis Function
Back Spindle 1-Degree Indexing Function
Back Spindle C-Axis Function
B/SP Constant Surface Speed Control
Back Spindle Chasing Function
Differential Rotary Tool Function
Caution: In case of changing any setting, turn CNC power off and on again.
HDL 1 RDY 2 RDY 3 RDY
POS Data Set SW Message Backup
PLC-Data MC-VAR MC-STRCT OP SEL ZP EXE MECH ADJ Macro Set Menu SEL


Machine Structure 4/4
0
3000 Comment C
Parameter
C VII Type DIA 16 Machine No. X0001
Tool Life Management 1
Tool Life Management 2
Milling Interpolation Function
Door Lock
Spindle Synchronization Function
Helical Interpolation Function
Slant Helical Interpolation Function
EC Specification
Caution: In case of changing any setting, turn CNC power off and on again.
HDL 1 RDY 2 RDY 3 RDY
POS Data Set SW Message Backup
PLC-Data MC-VAR MC-STRCT OP SEL ZP EXE MECH ADJ Macro Set Menu SEL

Display items

Item	Remarks
Synchronized Rotary Guide Bush Device A (U*10Z, U*20Z)	Select one of guide bushing devices mounted on the machine. You can select only one.
Synchronized Rotary Guide Bush Device B (U*60Z)	
Fixed Guide Bush Device A (U*10Z, U*20Z)	
Fixed Guide Bush Device B (U*60Z)	
Synchronized Bar Loader (CAV-2.5M)	Select one of bar loaders connected with the machine. You can select only one. No checking of the items means "Single bar feeder".
Synchronized Bar Loader (CAV-3.0M)	
Synchronized Bar Loader (CAV-4.0M)	
Automatic Bar Loader	
Chuck Sleeve For Non-Conform/Square Bar	Select this item when using a chuck sleeve for non-conform materials or square bars.
Bar Loader Mechanical Tip Detector	
Motor-Driven Back Knock-Out Unit	
Long Workpiece Device	
Chip Conveyor	
Coolant Flowrate Detector	
Cut-Off Tool Breakage Detector	
Floppy Disk Drive Unit	Do not turn off the power after setting this item. The setting is reset when the power is turned off.
Automatic Measure Comp Device	
Automatic Tool Setting Unit	
Front/Back Rotary Tool Drive Unit	Type VII, VIII, IX
Program Work Area Size 20m (Memory 40m)	
Program Work Area Size 40m (Memory 80m)	
Program Work Area Size 80m (Memory 160m)	
Program Work Area Size 160m (Memory 320m)	
Program Work Area Size 320m (Memory 640m)	
Custom Macro	
Sub Micron/Sub Inch Command Function	
Inch Specification	
Tool Spindle Synchronous Tap (S4)	
Main Spindle Synchronous Tap (S1)	
Back Spindle Synchronous Tap (S2)	
Canned Drilling Cycle	
Main Spindle 1-Degree Indexing Function	
Main Spindle C-Axis Function	
Back Spindle 1-Degree Indexing Function	
Back Spindle C-Axis Function	
B/SP Constant Surface Speed Control	
Back Spindle Chasing Function	
Differential Rotary Tool Function	
Tool Life Management I	
Tool Life Management II	
Milling Interpolation Function	
Door Lock	
Spindle Synchronization Function	
Helical Interporation Function	
Slant Helical Interporation Function	
EC Specification	

Note

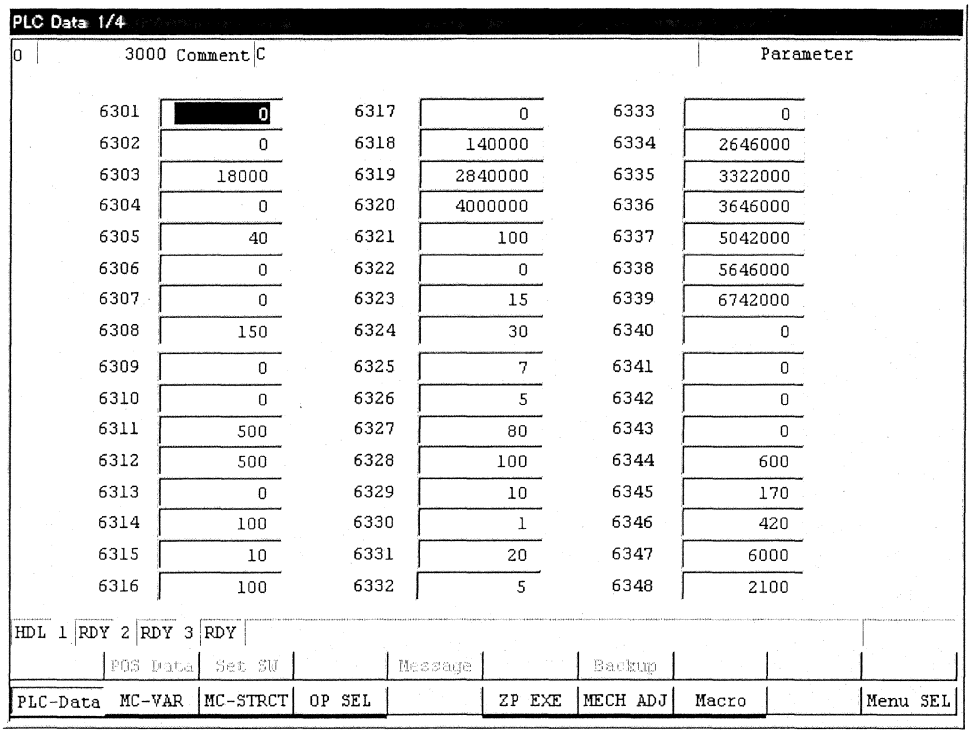
Turn off the power after changing the settings. The new settings become effective when the power is turned on next time.

6.23 PLC Data Screen

Press 

and then [PLC-Data] to display the PLC Data screen.

This screen lists constants used with PLC.



Display items

Item	Explanation
O	Program number of the currently selected execution program
(Number)	Constant numbers
(Numeric value)	Values of constants

6.24 Bit Select Screen

Press PAGE  or  on the PLC Data screen and select page 4/4 to display the Bit Select screen. This screen lists bit parameters used with PLC.

Bit Select 4/4

O 3000 Comment C Parameter

	76543210		76543210		76543210		76543210		76543210		76543210
6401	00010000	6417	00000000	6433	00000000	6449	00000011	6465	00000000	6481	00000000
6402	00000011	6418	00110000	6434	00000000	6450	00000000	6466	00000000	6482	00000000
6403	01000000	6419	10000001	6435	00000000	6451	00000000	6467	00000000	6483	00000000
6404	00000010	6420	00000000	6436	00000000	6452	00000000	6468	00000000	6484	00000000
6405	00000000	6421	00000011	6437	00000000	6453	00000000	6469	00000000	6485	00000000
6406	10000000	6422	00000000	6438	00000000	6454	00000000	6470	00000000	6486	00000000
6407	10000100	6423	00000000	6439	00000000	6455	00000000	6471	00000000	6487	00000000
6408	00000100	6424	00000000	6440	00000000	6456	00000000	6472	00000000	6488	00000000
6409	11101100	6425	00000000	6441	00000000	6457	00000000	6473	00000000	6489	00000000
6410	01000011	6426	00000100	6442	00000000	6458	00000000	6474	00000000	6490	00000000
6411	10000001	6427	00101110	6443	00000000	6459	00000000	6475	00000000	6491	00000000
6412	00110000	6428	00100100	6444	00000000	6460	00000000	6476	00000000	6492	00000000
6413	00010110	6429	00000101	6445	00000000	6461	00000000	6477	00000000	6493	00000000
6414	00000000	6430	00000000	6446	00000000	6462	00000000	6478	00000000	6494	00000000
6415	00000000	6431	00000000	6447	00000000	6463	00000000	6479	00000110	6495	00000000
6416	00000000	6432	00000000	6448	00000000	6464	00000000	6480	00000000	6496	00000000

HDL 1 | RDY 2 | RDY 3 | RDY

POS Data | Set SW | Message | Backup

PLC-Data | MC-VAR | MC-STRCT | OP SEL | ZP EXE | MECH ADJ | Macro | Menu SEL

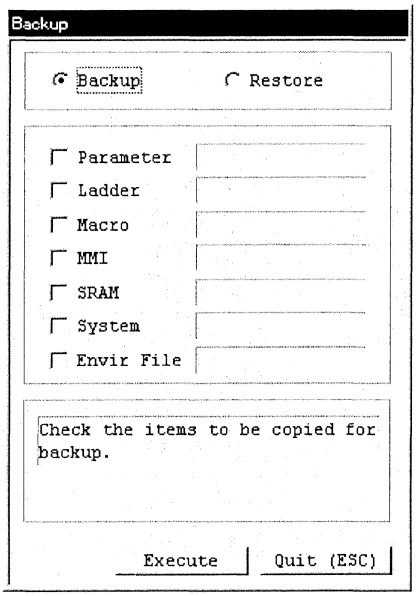
Display items

Item	Explanation
O	Program number of the currently selected execution program
(Number)	Bit parameter numbers
76543210	Values (0 or 1) of bit parameters

6.25 Backup Screen

Press PRM.  ,  , and [Backup] in this order to display the Backup screen.

This screen backs up various types of software and parameters.



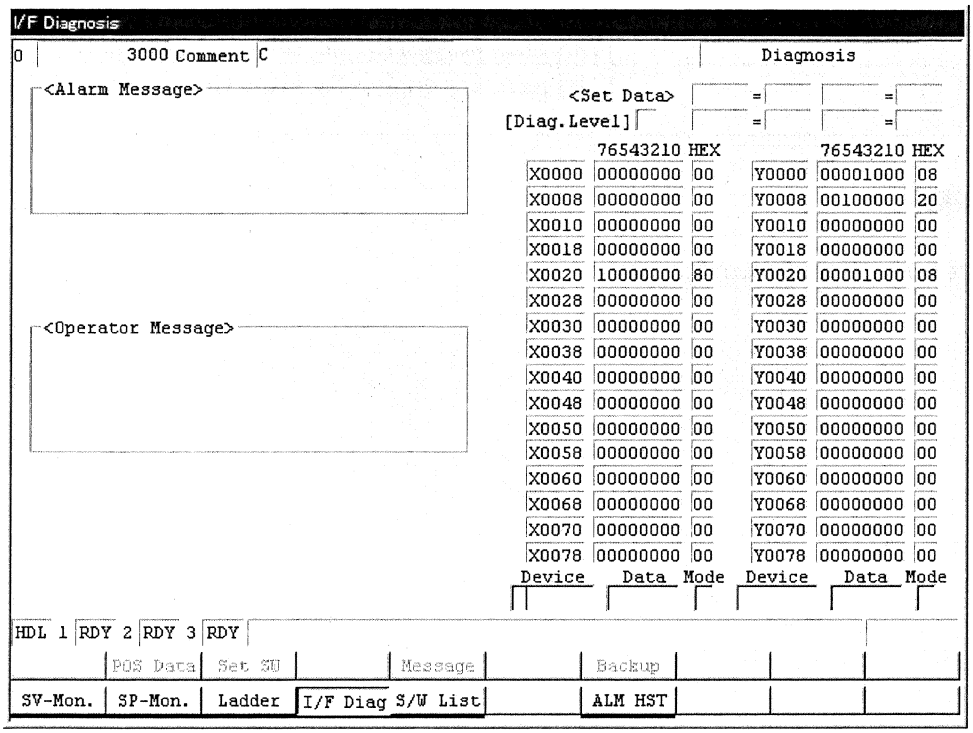
Display items

Item	Explanation
Backup	The memory contents of the NC unit are backed up into the hard disk.
Restore	The backup contents in the hard disk are restored to the NC unit memory.
Parameter	Parameter data in the NC unit memory
Ladder	Ladder program in the NC unit memory
Macro	Macro program in the NC unit memory
MMI	MMI program in the NC unit memory
SRAM	SRAM data (e.g., machining programs, common variables, etc.) in the NC unit memory. The parameter data is included.
System	System program in the NC unit memory
Envir File	Environmental file in the NC unit memory
Execute	Performs a backup or restore operation.

6.26 I/F Diagnosis Screen

Press MAINT. and then [I/F Diag] to display the I/F Diagnosis screen.

This screen displays various input/output signals for PLC control.



Display items

Item	Explanation
O	Program number of the currently selected execution program
(Device number)	Device numbers (input XOOO, output YOOO, timer TOO) of devices used with PLC
76543210	The data corresponding to each device number is expressed with bits.
HEX	Data is displayed in hexadecimal.
Device	Set the device number.
Data	Set the data corresponding to the set device number.
Mode	Set the mode for forcibly setting data: 1: One shot 2: Modal

Main menu keys

Key name	Function
SV-Mon.	This menu key displays the Servo Monitor screen.
SP-Mon.	This menu key displays the Spindle Monitor screen.
Ladder	This menu key displays the M6MONITOR screen.
I/F Diag	This menu key displays the I/F Diagnosis screen.
S/W List	This menu key displays the Software List screen.
ALM HST	This menu key displays the Alarm History and the Key History on the screen.

Submenu keys

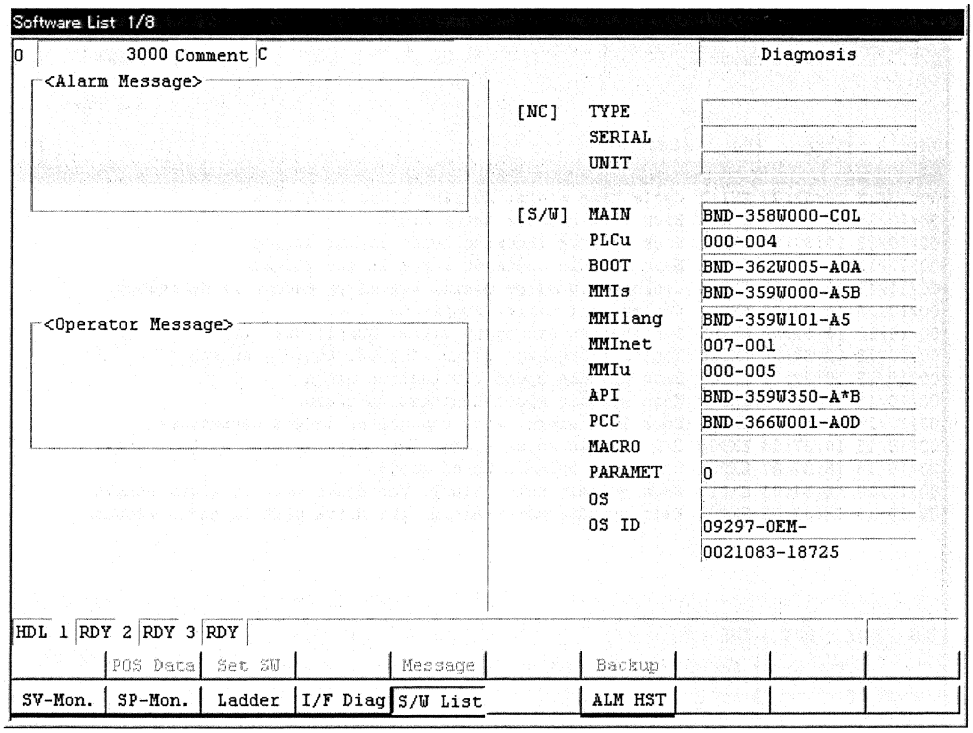
See <Section 6.19 Zero Return Screen>.

6.27 Software List Screen

Press 

and then [S/W List] to display the Software List screen.

This screen displays software versions.



Display items

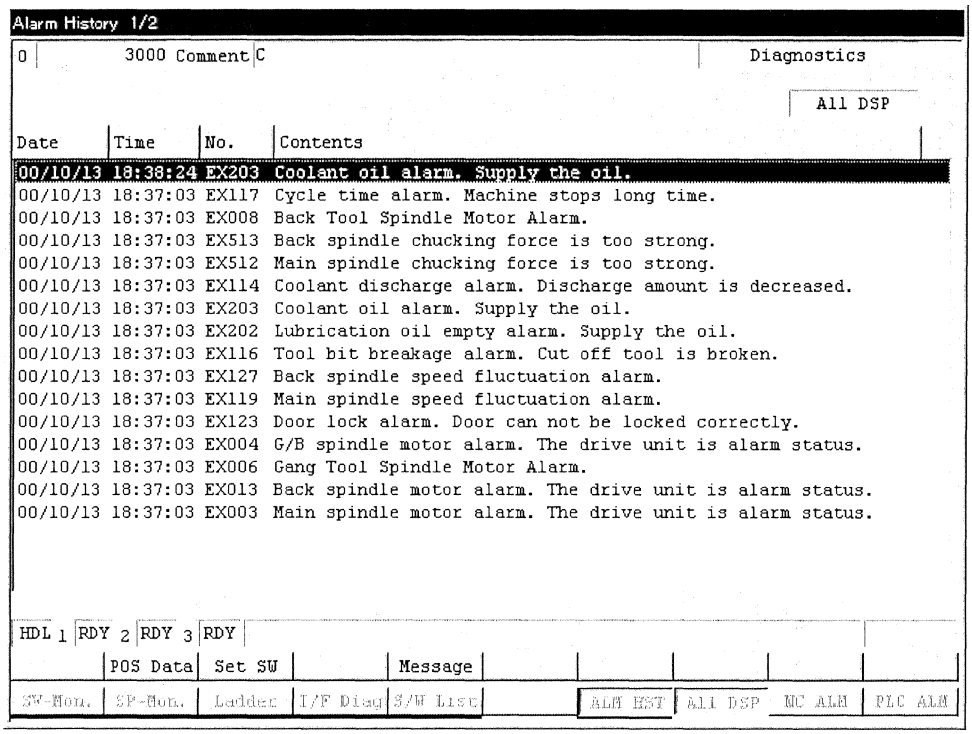
Item	Explanation
TYPE	Software versions
SERIAL	
MAIN	
PLC u	
BOOT	
MMI s	
MMI larg	
MMI net	
MMI u	
API	
PCC	
MACRO	
PARAMET	
OS	
OS ID	

6.28 Alarm History Screen

Press 

and then [ALM HST] to display the Alarm History screen.

The information of alarms in the machine is stored. This screen displays the alarm history.



Display items

Item	Explanation
Date	Date when alarms occurred
Time	Time when alarms occurred
No.	Alarm numbers
Contents	Explanation of alarms

Main menu keys

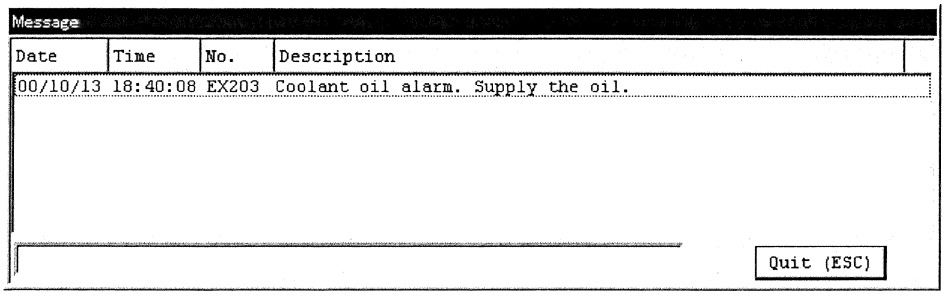
Key name	Function
All DSP	This menu key displays all alarms.
NC ALM	This menu key displays NC alarms only.
PLC ALM	This menu key displays machine alarms only.

Note

The information of up to 100 alarms can be stored and displayed on the Alarm History screen. When the number of alarms exceeds 100, the information of alarms is automatically deleted in the order in which the alarms occurred.

6.29 Message Screen

Press PREPARATION, ▷, and [Message] in this order to display the Message screen. (This is one of methods to display the Message screen.)
This screen displays messages such as alarm messages.



Display items

Item	Explanation
Date	Date when alarms/errors occurred
Time	Time when alarms/errors occurred
No.	Error/alarm numbers
Description	Explanation of alarms/errors

Product Code C – C 1 2 1 6 VI VII VIII IX – 7 7 0

Document Code 5 E 1 – 0 6 0 2

5 E 2 – 0 6 0 2

7. Operation

7.1 Power-On/Power-Off, Operation Flow

7.1.1 Turning on the power

Procedure

1. Turn on the main breaker of the machine.
   The power lamp on the right side of the operation panel lights.

2. Press the Power ON switch [I] on the left of the LCD screen.
   The Initial screen appears. Subsequently, the Program Select screen and the Preparation screen appear a few minutes later.

7.1.2 Turning off the power

Procedure

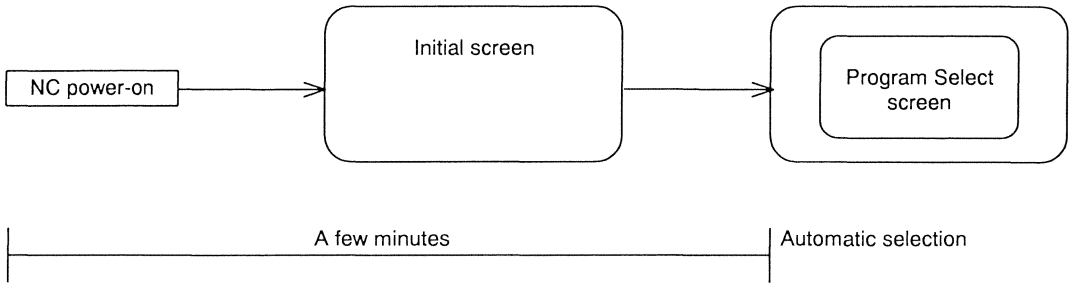
1. Press the Power OFF switch [O] on the left of the LCD screen.
   Messages appear on the screen.
   Software termination processing is automatically performed, and the screen is erased.

2. Turn off the main breaker of the machine.

> ⚠ CAUTION
>
> Confirm that the messages on the LCD screen have been erased and that the lamps on the operation panel are off before turning off the main breaker of the machine.
> Do not turn off the main breaker of the machine before the messages on the LCD are erased or the lamps on the operation panel go off. Doing so may cause the NC unit to malfunction.
> Also, confirm that the messages on the LCD screen have been erased before turning on the power again. The NC may malfunction if the Power ON switch is pressed during the software termination processing (before erasure of the messages on the LCD screen).

7.1.3 Screen transition at power-on

Turn on the main breaker of the machine to turn on the NC unit. The Initial screen appears a few minutes later. Subsequently, the Program Select screen of the preparation mode automatically appears tens of seconds later. Key operation is invalid before the automatic selection of the Program Select screen.



Notes

- If the operation panel is not used for ten minutes or more during operation, the LCD back light turns off automatically for protecting the LCD screen. To turn on the back light again, press any key such as an OPERATION MODE key, Alphanumeric key, or Arrow key on the operation panel.
- The system does not start up while the HEATER lamp lights. (The initial screen does not appear.) The heater warms the hard disk until the temperature becomes high enough for operation. The lamp turns off when the hard disk becomes ready for operation, then the system starts up. The lamp turns off a few minutes after it turns on.
- The screen may fail to appear normally. However, this is not an error. For details, see <Section 11.16 Troubleshooting at Power-on and Screen Operation>.

CAUTION

Never change the operating system (OS) screen properties. The OS screen saver function is used to turn off the LCD back light. Changing the OS screen properties shortens the life of the back light.

7.1.4 Stopping operation in emergency

Procedure

1. Press the Emergency Stop button.

7.1.5 Resetting the emergency stop state

Procedure

1. Turn the Emergency Stop button clockwise.

7.1.6 Procedure from power-on to continuous operation

This section explains the procedure for continuously machining workpieces from power-on to continuous operation.

	Procedure	Reference
1	Empty the chip tank if it contains chips.	<Section 10.1.4 Removing chips>
2	Set the main breaker to ON.	<Section 7.1.1 Turning on the power>
3	Press the Power ON switch on the operation panel. The screen appears after a short time. (For machines of the EC specifications, press the Power OFF switch before the Power ON switch to activate the safety circuit.)	<Section 7.1.1 Turning on the power>
4	Select the program for the workpiece to be machined.	<Section 7.2.1 Selecting a program to be executed>
5	To prevent interference with the tool, move the material back to the end face of the spindle.	
6	Open the spindle chuck.	
7	Close the safety cover.	
8	Execute Start Position in the Preparation mode. Each axis moves to the start position for the program.	<Section 7.3.1 Start position, return position, opposite tool post retract position, back spindle retract position and gang tool post return position>
9	Slowly push the material against the cut-off tool by hand.	
10	Operate the bar loader to apply torque to the push rod.	
11	Select the Automatic Operation mode.	<Section 7.5.1 Automatic operation>
12	Select 1 Cycle.	<Section 7.5.1 Automatic operation>
13	Supply coolant. Confirm that the coolant discharge is enough and that the discharge position is correct.	
14	Press the Start key. The machine performs 1-cycle machining in the Automatic Operation mode. (For machines of the EC specifications, press the Start key and lock the door first, then perform 1-cycle machining.)	<Section 7.5.1 Automatic operation>
15	Measure the workpiece diameter and the longitudinal dimension, and set offset values and change the program to correct the product dimensions.	<Section 7.10 Setting Tool Offset Data>
16	Select Continuous Cycle in the Automatic Operation mode.	<Section 7.5.1 Automatic operation>
17	Press the Start key. The machine performs continuous machining in the Automatic Operation mode. (For machines of the EC specifications, press the Start key and lock the door first, then perform continuous cycle machining.)	<Section 7.5.1 Automatic operation>

Check the workpiece dimensions periodically during automatic operation, and set offset value to correct the dimensions if necessary. Also, periodically check the chip tank, the tool, and the coolant discharge. Empty the chip tank, clean the tool if chips are caught in it, and check the coolant level if necessary.

7.1.7 Procedure from continuous operation to power-off

This section explains the procedure from automatic continuous operation to power-off.

	Procedure	Reference
1	Select 1 Cycle in the Automatic Operation mode.	<Section 7.5.1 Automatic operation>
2	Wait until the Start key lamp goes off.	
3	Stop coolant.	
(4)	(Stop the spindle.)	
5	Press the Power OFF switch on the operation panel. The screen display disappears after a short time.	<Section 7.1.2 Turning off the power>
6	Set the main breaker to OFF.	<Section 7.1.2 Turning off the power>

7.1.8 Re-arrangement procedure

This section roughly explains the procedure for re-arrangement required when the next workpieces are different from the previously machined workpieces.

	Procedure	Reference
1	On the Edit screen, enter the program and machining data for the workpieces to be machined next. If an already created program is saved in a tape or another medium, load the program in the machine.	<Section 7.7 Creating and Editing Programs> <Section 7.9 Data Input/Output>
2	Dismount the tools used to machine the previous workpieces. If you use the same tools, you do not have to dismount them. When you set a tool in a machine equipped with the back machining function, be sure to dismount the back machining tool to prevent interference.	
3	Replace/adjust the chuck and the guide bushing. If the machine is equipped with the back machining function, replace the knock-out jig, and replace/adjust the back spindle chuck.	<Section 7.3.3 Adjusting chucking force> <Section 7.3.7 Adjusting a guide bushing clearance> <Section 9.1 Mounting/Adjusting Chucks > <Section 9.2 Mounting/Adjusting a Guide Bushing Device > <Section 9.3 Re-arranging the Back Spindle >
4	Select the program for the workpieces to be machined in the Preparation mode. The machining data is automatically selected at the time. Confirm that the selected machining data is correct.	<Section 7.2 Program Selection>
5	Execute Start Position in the Preparation mode. Each axis moves to the start position specified in the program.	<Section 7.3.1 Start position, return position, opposite tool post retract position, back spindle retract position and gang tool post return position>
6	Set offset values on the Offset screen. Generally, clear all the offset setting values of the dismounted tool to 0.000. If the tool nose radius compensation function is used, set the tool nose data.	<Section 7.10 Setting Tool Offset Data>
7	In the Program Check mode or Automatic Operation mode, run the machine with no material inserted (dry run) to check if the machine operates normally. If an alarm is issued or a program error is detected, correct the program. The Program Check function is useful to correct the program.	<Section 7.4 On-machine Program Check Function> <Section 7.5 Automatic Operation>

	Procedure	Reference
8	Set the gang and front face tools. Use the Preparation mode to set the tools. Set the back face tools in step 14.	<Section 7.3.11 Setting tools > <Section 9.4 Mounting/Adjusting Tools>
9	In the Program Check mode or Automatic Operation mode, run the machine with no load to check if the tools move as expected. Correct the program if necessary.	<Section 7.4 On-machine Program Check Function> <Section 7.5 Automatic Operation>
10	Select 1 Cycle in the Program Check mode or Automatic Operation mode to machine a workpiece actually. Since no offset values have been set at this stage, the back spindle and the workpiece may interfere with each other. To prevent the interference, perform cut-off machining without the workpiece being chucked by the back spindle. However, if the back spindle is designed to chuck the outer diameter of material (blank), get the back spindle to chuck the workpiece.	<Section 7.4 On-machine Program Check Function> <Section 7.5 Automatic Operation>
11	Measure the workpiece diameter and the longitudinal dimension, and set offset values or change the program to correct the product dimensions.	<Section 7.10 Setting Tool Offset Data>
12	When the back spindle did not chuck the workpiece in Step 10, get the back spindle to chuck it after the front machining dimension is corrected in step 11, and change the program to perform cut-off machining.	<Section 7.7 Creating and Editing Programs>
13	Select 1 Cycle in the Automatic Operation mode to machine the workpiece.	<Section 7.5.1 Automatic operation>
14	Set the back face tools.	<Section 7.3.11 Setting tools>
15	Select 1 Cycle in the Automatic Operation mode to machine the workpiece.	<Section 7.5.1 Automatic operation>
16	Measure and correct the dimensions (including the back machining dimension) of all the workpieces.	<Section 7.10 Setting Tool Offset Data>
17	Check the product count set in the product counter and set a new value.	<Section 7.5.7 Setting counter>
18	Select Continuous Cycle in the Automatic Operation mode to continuously machine workpieces.	<Section 7.5.1 Automatic operation>

7.2 Program Selection

Program selection is to choose a program to be executed from registered programs. When a program is selected, the corresponding machining data is chosen at the same time.

7.2.1 Selecting a program to be executed

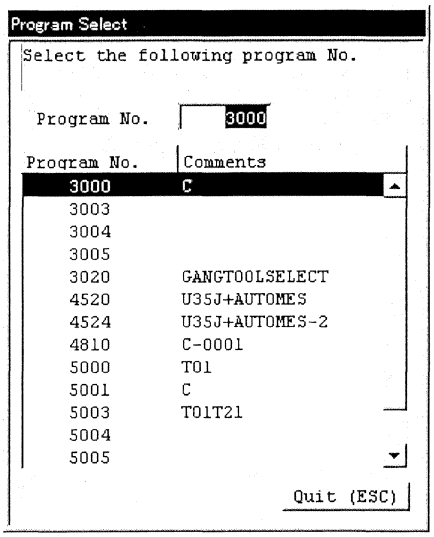
Select the program you want to execute.

Procedure

You can select the program on the Automatic Operation screen, Preparation screen, or On-Machine Check Program Check screen.

1. Press AUTO, PREPA-RATION, or PROGRAM CHECK.

2. Press ▷ (Menu Up/Down Selection key), then press the menu key [PRG SEL].
   The Program Select screen appears.



3. Press PAGE ↑, PAGE ↓, ↑, and/or ↓ to move the cursor to the program (you want to select) in the Program No. field. Alternatively, enter the program number (you want to select) in the input area.

4. Press INPUT.
   The program is selected.

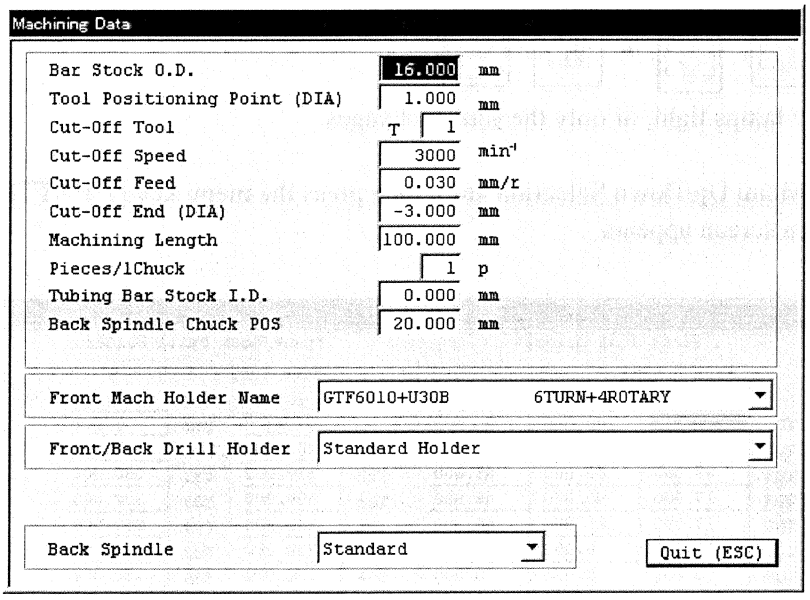
7.2.2 Entering, changing, or registering machining data

Machining data is necessary for operating the machine in the automatic operation mode and preparation mode. It is processed in the almost same manner as for machining programs and stored in the memory. Data other than the actual Machining Data that is being used in operation can be created/edited in the background.
Machining data displayed on the Machining Data screen is valid. The displayed values are used in the automatic operation mode and preparation mode.
The values do not change when the screen is switched or when the power is turned off and then turned on.
This section explains the settings for the machining data called with the machining program from the memory during program selection.

Procedure

1. Press PREPA-RATION, AUTO, PROGRAM CHECK, MDI or EDIT, OFFSET.
   The pressed key lamps light, or only the screen changes.

2. Press ▷ (Menu Up/Down Selection key), then press the menu key [MC-Data].
   The Machining Data screen appears.



(This screen is for type VII, VIII, IX.)

3. Press GRP to select the group for which you want to enter or change machining data.

4. Press ↑ and/or ↓ to select the item for which you want to enter a value or change the current value.

5. Press the numeric keys to enter or change the numeric value. In the holder name areas, press ↑ and/or ↓ to select one of the registered holders.

6. Press INPUT.
   The data is rewritten.
   The machine operates according to the data.

Note

For details on the machining data, See <Section 6.4 Machining Data Screen>.

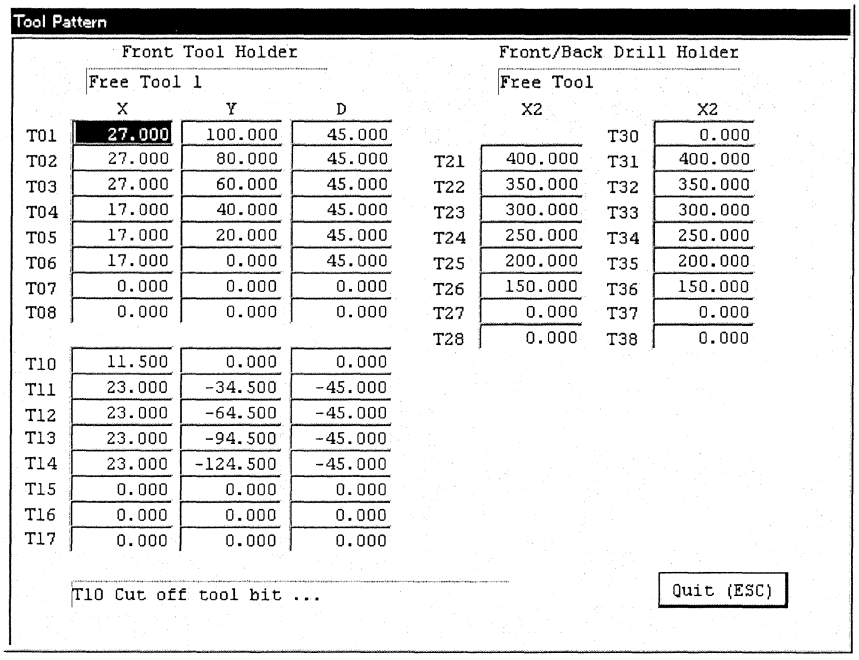
7.2.3 Confirming a tool layout pattern

This section explains the procedure for confirming the tool layout pattern of a front tool post.
The machine operates according to the tool layout pattern when a T code is specified in the program or when a tool is called on the Preparation screen.

Procedure

1. Press PREPA-RATION, AUTO, PROGRAM CHECK, or MDI, OFFSET.
   The pressed key lamps light, or only the screen changes.

2. Press ▷ (Menu Up/Down Selection key), then press the menu key [T-PATT].
   The Tool Pattern screen appears.



(This screen is for type VII, VIII, IX.)

7.2.4 Confirming/setting a free tool layout pattern

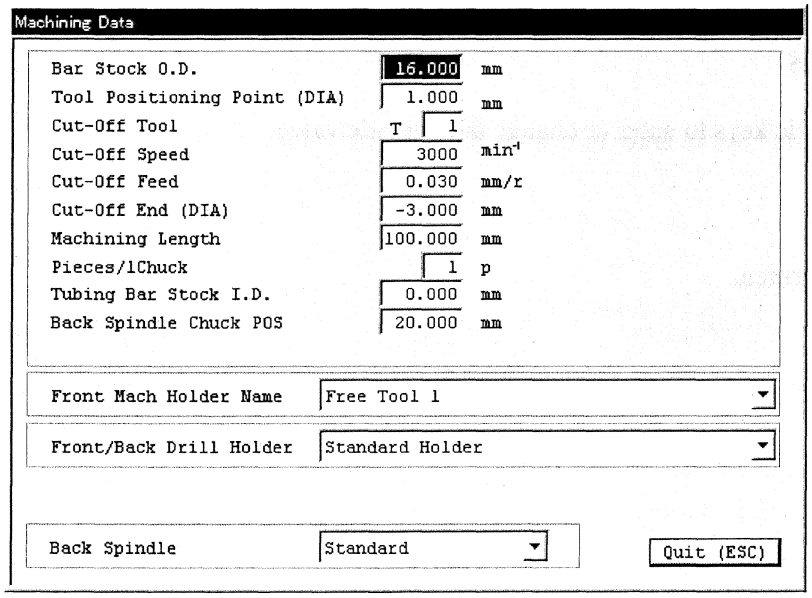
This section explains the procedure for setting a free tool layout pattern.
A free tool layout pattern is set for the front machining tool holder or back drilling tool holder in the machining data. The machine operates according to the free tool layout pattern when a T code is specified in the program or when a tool is called on the Preparation screen.
You can set a free tool layout pattern for the front tool holder and front/back drill holder each.
The following explains the method of setting a free tool layout pattern for only the front tool holder.
The same method applies to the front/back drill holder.

Procedure

1. Press AUTO, PREPA-RATION, PROGRAM CHECK, MDI, or OFFSET.
   The pressed key lamps light, or only the screen changes.

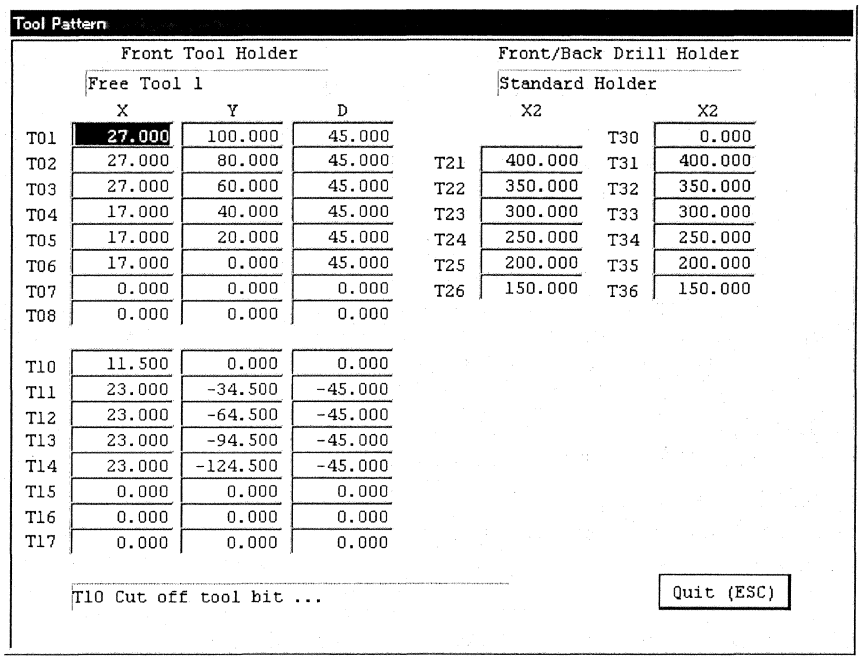
2. Press ▷ (Menu Up/Down Selection key), then press the menu key [MC-Data].
   The Machining Data screen appears.

3. Confirm that "Free Tool 1" is selected in the Front Mach Holder Name box.
   Select "Free Tool 1" if not selected. See <Section 7.2.2 Entering, changing, or registering machining data>.



(This screen is for type VII, VIII, IX.)

4. Press [T-PATT].
   The Tool Pattern screen appears.



(This screen is for type VII, VIII, IX.)

5. Press TAB ⬅, TAB ➡, or ⬆ ⬇ to select the item for which you want to enter a value or change the current value.

6. Press the numeric keys to enter or change the numeric value.

7. Press INPUT ➡.
   The data is rewritten.

7.3 Preparation

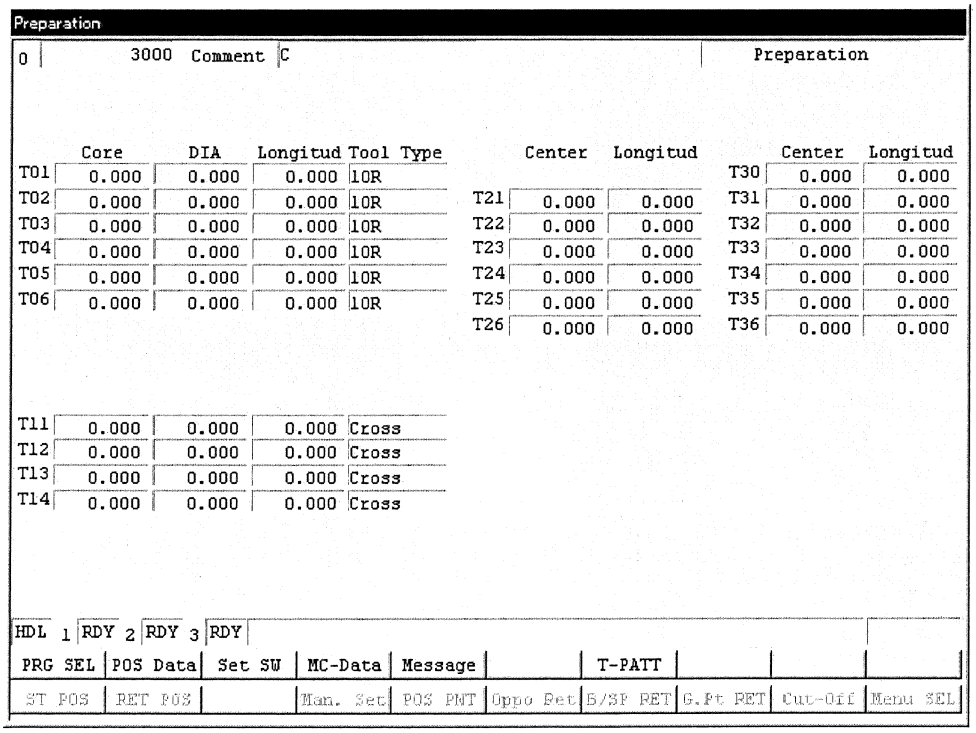
After program selection, a preparation operation is necessary for machining workpieces. The preparation operation includes operations such as setting tools inside the machine, moving axes to the positions for automatic operation (after completion of tool setting), and cut-off machining for the end faces of materials.

Notes

- The machine operates in the preparation operation only while START  is held down. (Cut-off machining excluded)
- When the START key is released, the machine stops, and the START key lamp flashes. The machine restarts operating if the START key is pressed again in the state. The START key lamp turns off at completion of operation.
- Confirm that the operation is completed and the START key lamp has turned off, then proceed to the next operation.

Procedure

1. Press PREPA-RATION  . The PREPARATION key lamp lights, and the Preparation screen appears.



(This screen is for type VII, VIII, IX.)

WARNING

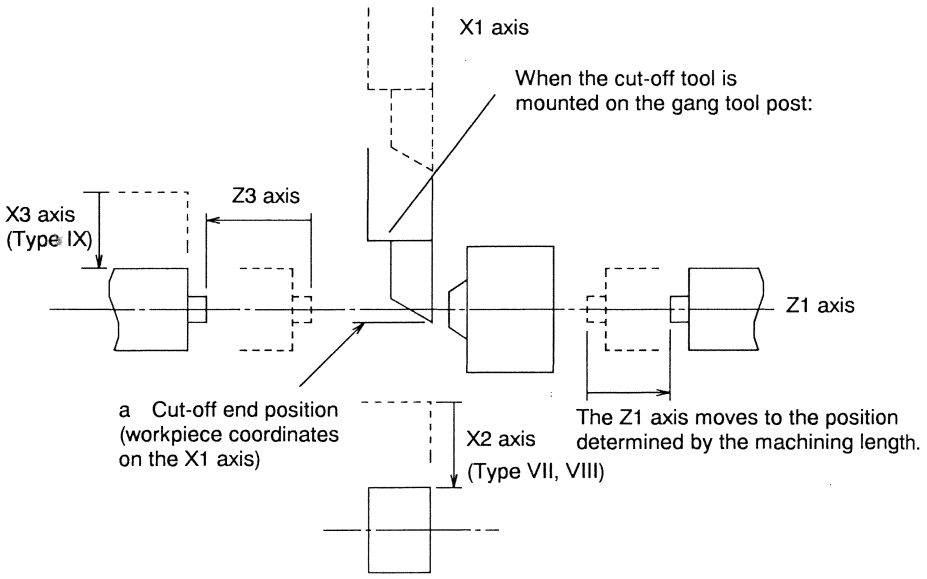
- Never get close to any mobile section of the machine during preparation for operation. Doing so could result in serious personal injury if the mobile sections of the machine suddenly start operating during execution of the preparation function.
- Confirm that a mounted tool does not interfere with any part of the machine whenever preparing a new program or existing program for operation or whenever executing a new program. Interference could result in machine damage or personal injury.
- When confirming tool operation, make sure of the part of the machining program that is being executed, and observe the safety precautions of the operation.
- For the other precautions, see <Chapter 2 Safety Precautions.>

7.3.1 Start position, Return position, Opposite tool post retract position (Type VII, VIII, IX), Back spindle retract position and Gang tool post return position

Start position

Each axis moves to the position for starting automatic preparation.
The Z1 axis moves backward by the Machining Length (specified in the machining data) from the forward end position. The tool specified for "Cut-off Tool" in the machining data is selected as the cut-off tool. The X1 axis moves to the Cut-Off End position specified in the machining data. Z3 axis moves to the machine zero point. X2 axis moves to the machine zero point. Return the axis to the queuing position if the knock-out device or the workpiece separator is at the forward position. When the long workpiece device (optional) is mounted, return the axis to the queuing position.

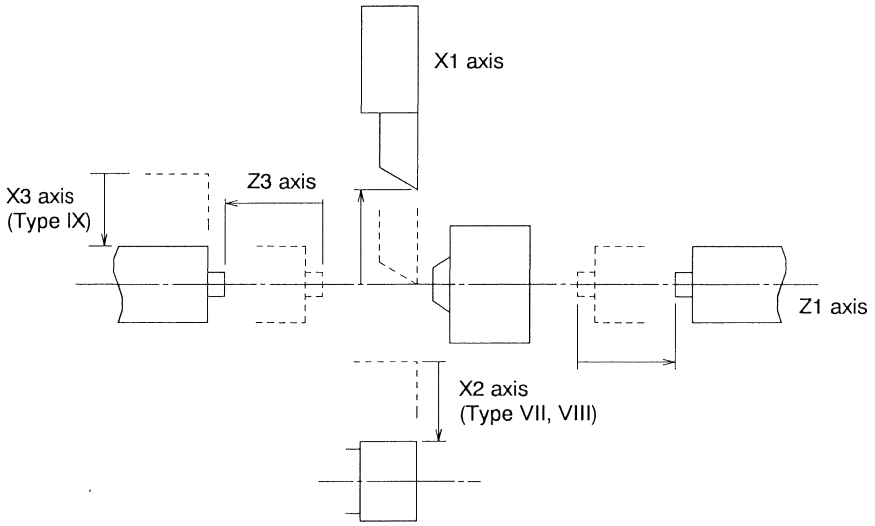
The X1 axis does not move when the start position is specified in the following conditions: a cut-off tool has been selected, and the X1 and Y1 axes are at a position in the range ± 0.1 mm with the start position (completion position) as the center.
If the start position is specified while the main spindle chuck is closed, the alarm message "EX501 Spindle chuck closed Open spindle chuck" is displayed. Open the main spindle chuck and then execute the operation.



	C12/C16
a Cut-off end point (standard value)	– 3.0

Return position

The X1, X2, Z1, Z3, (A3) axes move to the safe positions.
Return the axis to the queuing position if the knock-out device or workpiece separator is at the forward position.
When the long workpiece device is mounted, return the axis to the queuing position.



Opposite tool post return position (Type VII, VIII, IX)

The Z1 axis retracts 1.0 mm.
The X2 axis moves to the same position as the zero point return.
The Z1 axis advances 1.0 mm.

Back spindle return position

The Z3 axis moves to the same position as the zero point return.

Gang tool post return position

The X1 axis moves to the same position as the zero point return.

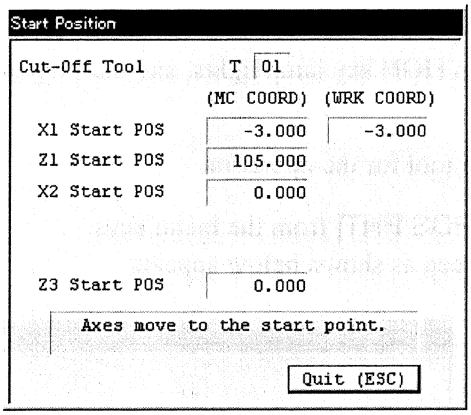
> **WARNING**
>
> Do not get close to the mobile sections of the machine. The headstock also moves during execution. Doing so could result in personal injury.

> **CAUTION**
>
> Execute these operations after extracting the material.
> If the headstock moves forward while a material is inserted into the main spindle, the material and tool interfere with each other. Interference may cause machine damage.

Procedure

1. Press PREPARATION. The PREPARATION key lamp lights, and the preparation screen appears.

2. Select the desired operation [ST POS], [RET POS], [Oppo RET] (Type VII, VIII, IX), [B/SP RET] or [G.Pt RET] from the menu keys.
   The operation explanation screen as shown below appears.



(This screen is for type VII.)

3. Press START. The START key lamp lights, and each axis moves to the specified position.
   The machine operates only while the START key is held down.

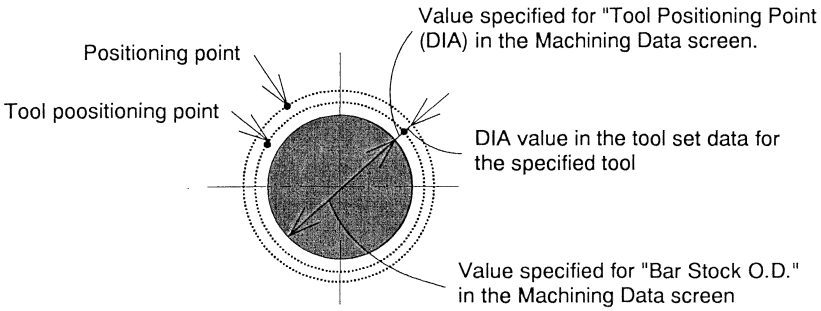
Notes

- If the start position or retract position is specified after the power is turned on, the separator advances or retracts only once.
- The machine operates at the speed of 2 m/min or less if a door is open.
- The operation explanation screen automatically closes if the subsequent operation is not performed for 15 seconds after an operation item is selected with the menu key.

7.3.2 Positioning point

Positioning point

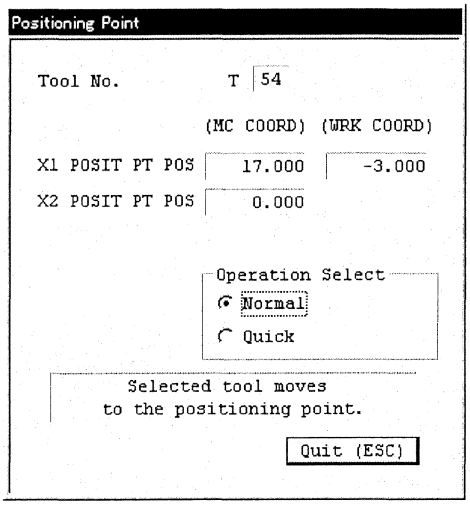
A selected gang tool moves to the positioning point (Bar Stock O.D. specified in the machining data + Tool Positioning Point specified in the machining data + DIA value of tool set data specified in the Preparation screen).



> **WARNING**
>
> Do not get close to the mobile sections of the machine. Doing so could result in personal injury.

Procedure

1. Press PREPARATION. The PREPARATION key lamp lights, and the Preparation screen appears.
2. Press Arrow keys to select the tool for the operation.
3. Select the desired operation [POS PNT] from the menu keys.
   The operation explanation screen as shown below appears.



(This screen is for type VII.)

4. Select "Quick" as needed using 

and 

keys. When "Quick" is selected, the specified tool moves from the current position directly to the positioning point. This function is mainly used after an automatic operation of program stopped halfway. In this case, make sure to remove the workpiece before executing this function.

5. Press 

. The START key lamp lights, and each axis moves to the specified position.
The machine operates only while the START key is held down.

Notes

- The machine operates at the speed of 2 m/min or less if a door is open.
- The operation explanation screen automatically closes if the subsequent operation is not performed for 15 seconds after an operation item is selected with the menu key.
- When executing [POS PNT], the axes X1 and Y1 of the tool being selected moves to the positioning point, and then the axes X1 and Y1 of the specified tool moves to the positioning point. If you want to move the specified tool to the positioning point, first select "Quick" on "Operation Select" section of Positioning Point screen, and execute [POS PNT]. When "Quick" operation complete, the state returns to "Normal" (initial state).

7.3.3 Adjusting chucking force

Two methods are available for adjusting chucking force: automatic adjustment and conventional manual adjustment. There are two operation of automatic adjustment: chucking force setting (chucking force is checked and stored in the memory of the NC unit), and automatically obtaining the chucking force stored by the chucking force setting. For details, see the following sections. This section outlines the adjustment and machine operation.

> ⚠ CAUTION
>
> - Confirm that the chucking force is suitable for machine operation. Incorrect chucking force may cause machine damage.
>   The chucking force obtained by automatic adjustment is only reference value. Be sure to check the chucking force manually before operating the machine in the automatic operation mode.
> - Be sure to remove all the jigs at completion of adjustment. Machine operation with a jig mounted may cause machine damage.

Chucking force setting

Judging from his/her experience, the operator enters a value 1 to 10 as chucking force data into the machine. The machine automatically adjusts the chucking force with the data.
Then, the operator checks manually if the automatically adjusted chucking force is suitable. If it is not suitable, the operator enters another numeric value as chucking force data with which the machine readjusts the chucking force. Repeat this operation until the chucking force becomes suitable. Finally, press [Memory 1], [Memory 2], or [Memory 3] on the screen to store the chucking force.

Automatic adjustment of chucking force

Select [Memory 1], [Memory 2], or [Memory 3] that corresponds to the chucking force stored by the chucking force setting. Then, the machine automatically operates and obtains the chucking force.

Manual adjustment of chucking force

This method is conventional. For the adjustment method, see <Section 9.1.2 Checking the chucking force>.

7.3.4 Chucking force setting

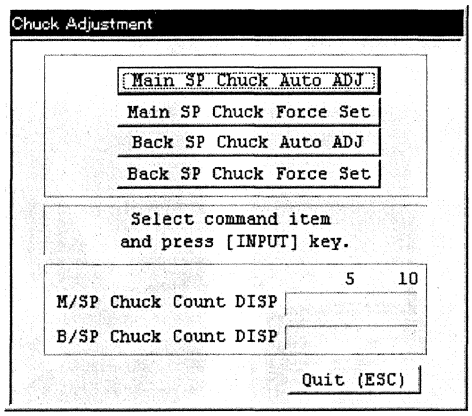
The machine automatically adjusts chucking force, and the operator checks manually if the automatically obtained chucking force is suitable until the chucking force becomes suitable. Finally, the operator stores the suitable chucking force in the memory of the machine.
You can set the chucking force of the main spindle and back spindle.

Procedure

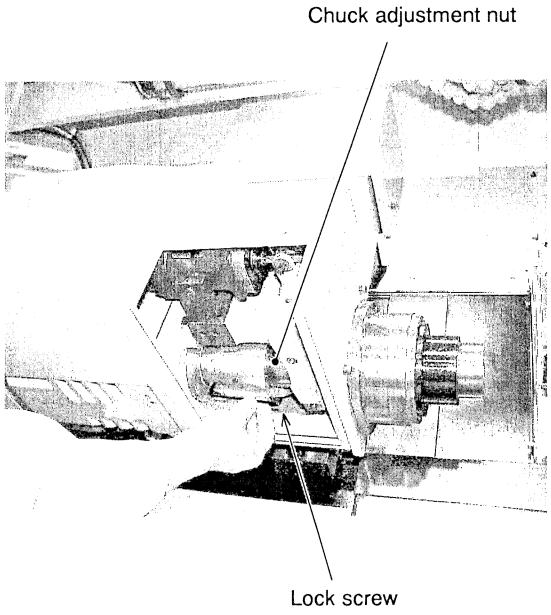
1. Press PREPA-RATION. The PREPARATION key lamp lights, and the Preparation screen appears.

2. Perform the return position operation. See <Section 7.3.1 Start position, return position, opposite tool post retract position (Type VII, VIII, IX), back spindle retract position and gang tool post return position>.
   Each axis moves to the return position.

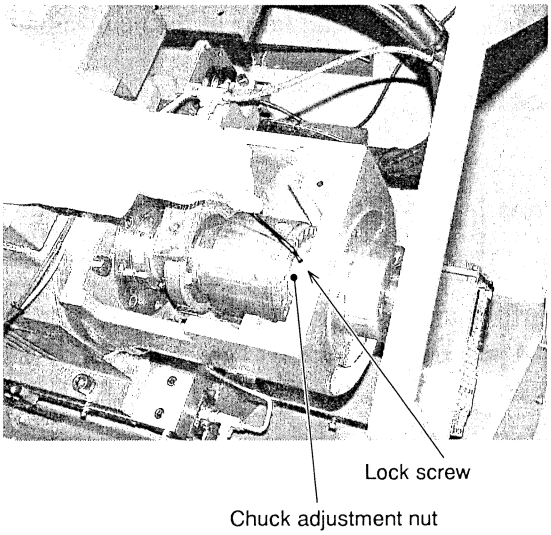
3. Press the menu key [Menu SEL] on the Preparation screen, select the second main menu, and press the menu key [CHCK ADJ].
   The Chuck Adjustment screen appears.



4. Loosen the lock screw of the chuck adjustment nut of the spindle you want to adjust, then confirm that you can turn the chuck adjustment nut smoothly by hand.

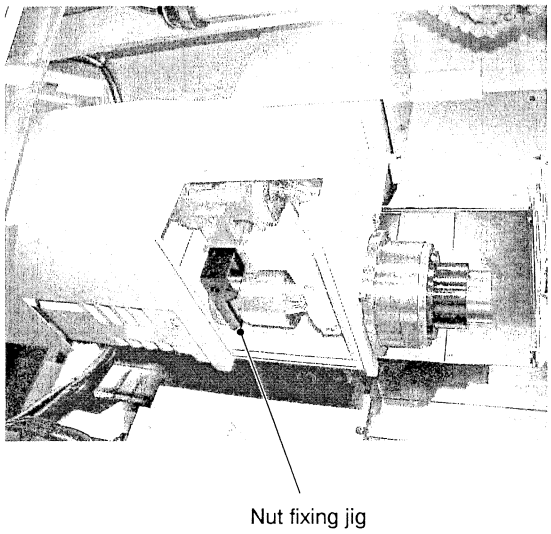


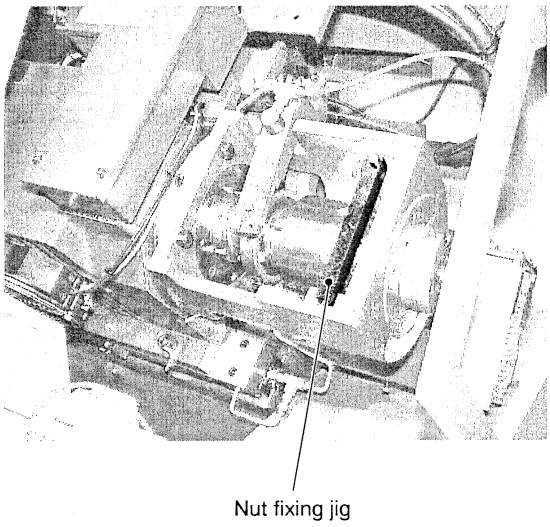
Back spindle



Main spindle

5. Mount the nut fixing jig on the spindle you want to adjust.

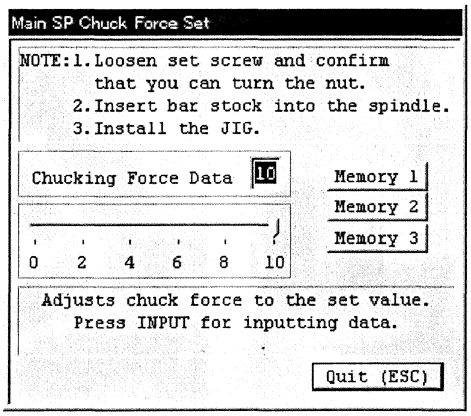




6. When setting the chucking force of the main spindle, insert a material into the main spindle. When setting the chucking force of the back spindle, insert a workpiece into the back spindle.

7. Press ↑ and/or ↓ to select [Main SP Chuck Force Set] or [Back SP Chuck Force Set], then press INPUT.

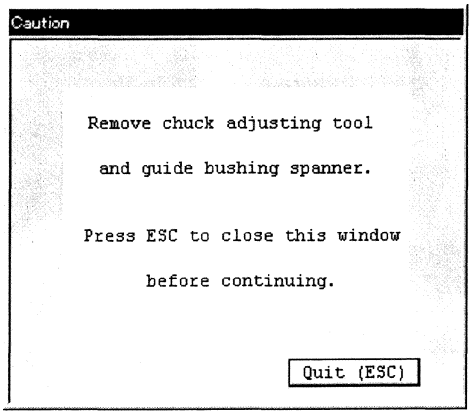
   The SP Chuck Force Set screen as shown below appears.



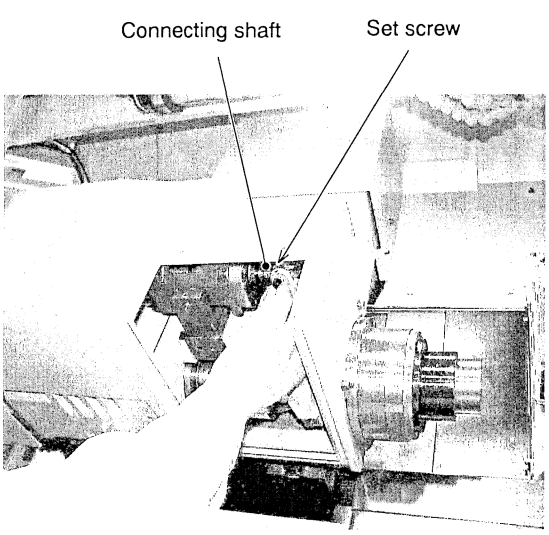
8. Press GRP to select the adjustment bar, then press ← and/or → to set it to the chucking force data you want to set. Alternatively, you can select the numeric value area by GRP key and enter the numeric value as chucking force data directly into the area.

9. Press START. The START key lamp lights, and the machine automatically starts adjusting the chucking force.

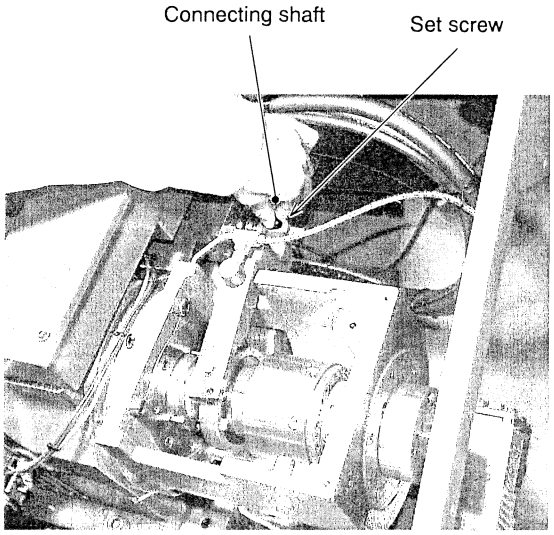
   The machine operates only while the START key is held down. While the machine is operating, the message "Executing OOOO chuck force automatic adjustment..." is displayed. The Caution screen shown below appears at completion of adjustment, and the START key lamp turns off.



10. Loosen the set screw to remove the connecting shaft of the servo motor for opening/closing the chuck. Beforehand the chuck opens or closes if necessary.

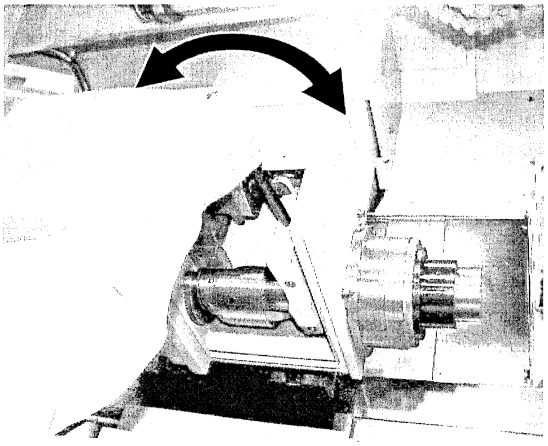


Back spindle

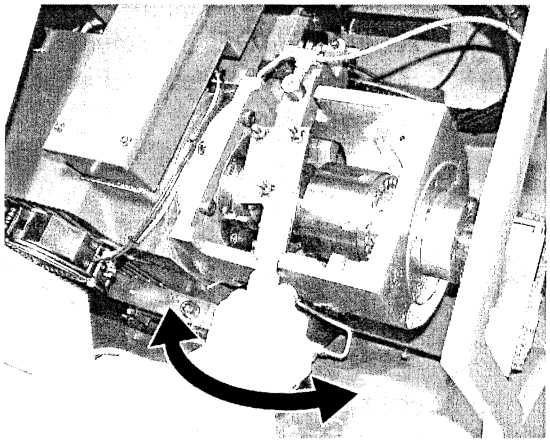


Main spindle

11. Check the chucking force with the dedicated tool and decide if it is suitable by your experience. If the chucking force is suitable, proceed to the next step. If readjustment is necessary, return the connecting shaft (of the servo motor for opening/closing the chuck) to the original position, press ESC to erase the Caution screen, press the menu key [CHCK ADJ] to redisplay the Chuck Adjustment screen, then restart the adjustment from step 7.



Back spindle



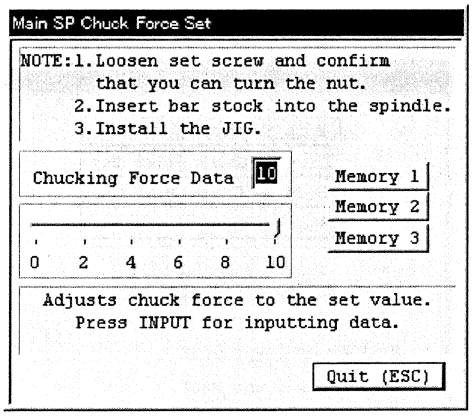
Main spindle

12. Return the connecting shaft (of the servo motor for opening/closing the chuck) to the original position, and tighten the set screw. Remove the nut fixing jig, tighten the lock screw, press ESC to erase the Caution screen, then press the menu key [CHCK ADJ] to redisplay the Chuck Adjustment screen.

13. Press ↑ and/or ↓ to select [Main SP Chuck Force Set] or [Back SP Chuck Force Set], then press INPUT.

The SP Chuck Force Set screen as shown below appears.

14. Press GRP to select the storage number field on the screen, press ↑ and/or ↓ to select [Memory 1], [Memory 2], or [Memory 3], then press INPUT.



15. Press ESC twice.

The SP Chuck Force Set screen disappears.

16. Remove the material or workpiece from the spindle.

Notes

- You cannot switch the screen or start automatic operation while the Caution screen is displayed.
- The screen automatically returns to the previous Chuck Adjustment screen if the subsequent operation is not performed for 15 seconds after the Main SP Chuck Force Set screen or Back SP Chuck force Set screen appears.

7.3.5 Automatic adjustment of chucking force

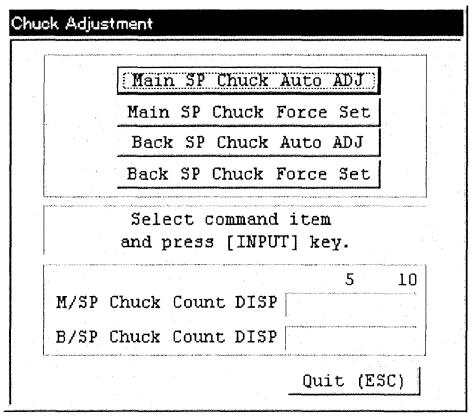
The machine automatically adjusts the chuck until it obtains the stored chucking force.

Procedure

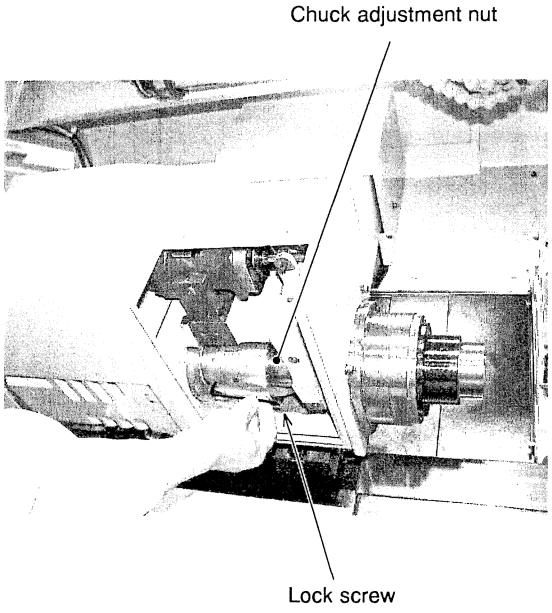
1. Press PREPARATION. The PREPARATION key lamp lights, and the Preparation screen appears.

2. Perform the return position operation. See <Section 7.3.1 Start position, return position, opposite tool post retract position (Type VII, VIII, IX), back spindle retract position and gang tool post return position>.
   Each axis moves to the return position.

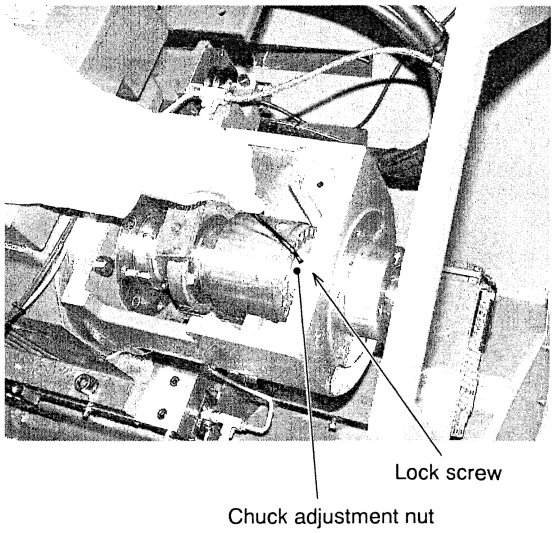
3. Press the menu key [CHCK ADJ].
   The Chuck Adjustment screen appears.



4.Loosen the lock screw of the chuck adjustment nut of the spindle you want to adjust, then confirm that you can turn the chuck adjustment nut smoothly by hand.

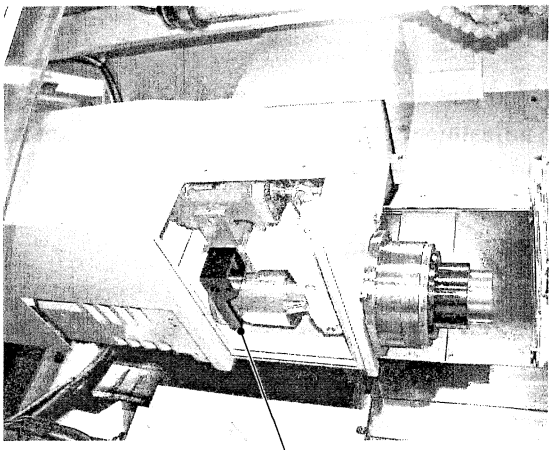


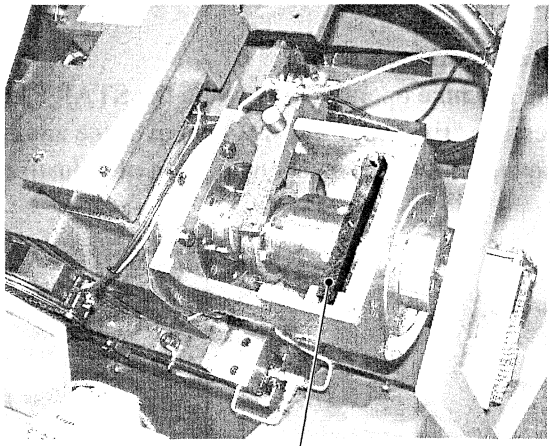
Back spindle



Main spindle

5. Mount the nut fixing jig on the spindle you want to adjust.

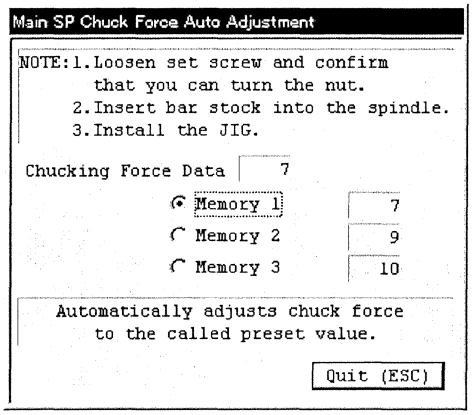




6. When setting the chucking force of the main spindle, insert a material into the main spindle. When setting the chucking force of the back spindle, insert a workpiece into the back spindle.

7. Press ↑ and/or ↓ to select [Main SP Chuck Force ADJ] or [Back SP Chuck Force ADJ], then press INPUT.

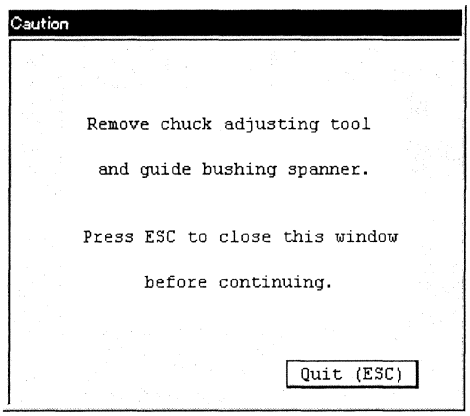
   The SP Chuck Force Auto Adjustment screen as shown below appears.



8. Press GRP to select the storage number field on the screen, press ↑ and/or ↓ to select [Memory 1], [Memory 2], or [Memory 3], then press INPUT.

9. Press START. The START key lamp lights, and the machine automatically starts adjusting the chucking force.

   The machine operates only while the START key is held down. While the machine is operating, the message "Executing OOOO chuck force automatic adjustment..." is displayed. The Caution screen shown below appears at completion of adjustment, and the START key lamp turns off.



10.Remove the nut fixing jig, tighten the lock screw, then press 

to erase the Caution screen.

11.Remove the material or workpiece from the spindle.

Notes

- The chucking force obtained by automatic adjustment is only reference value. Be sure to check the chucking force at completion of adjustment.
- You cannot switch the screen or start automatic operation while the Caution screen is displayed.
- The screen automatically returns to the previous Chuck Adjustment screen if the subsequent operation is not performed for 15 seconds after the Main SP Chuck Force Auto Adjustment screen or Back SP Chuck Force Auto Adjustment screen appears.

7.3.6 Manual adjustment of chucking force

See <Section 9.1.2 Checking the chucking force>.

7.3.7 Adjusting a guide bushing clearance

The following two types of guide bushing devices are available:

- Rotary guide bushing device
- Fixed guide bushing device

Two methods are available for adjusting a clearance with the guide bushing of each type: automatic adjustment and conventional manual adjustment. There are two operation of automatic adjustment: clearance setting (a clearance with the rotary guide bushing is obtained and stored in the memory of the NC unit), and automatically obtaining the clearance stored by the clearance setting. The operation vary slightly depending on the type of guide bushing. For details, see the following sections. This section outlines the adjustment and machine operation.
A clearance with a fixed guide bushing can be adjusted only manually.

CAUTION

- Confirm that the clearance is suitable for machine operation. Incorrect clearance may cause machine damage.
  The clearance obtained by automatic adjustment is only reference value. Be sure to check the clearance before operating the machine in the automatic operation mode.
- Be sure to remove the guide bushing adjustment wrench at completion of adjustment. Machine operation with the guide bushing adjustment wrench mounted may cause machine damage.

Clearance setting of a rotary guide bushing

The operator enters a numeric value as clearance data. The machine automatically adjusts the clearance with the data. Then, the operator checks manually if the automatically adjusted clearance is suitable. If it is not suitable, the operator enters another numeric value as clearance data with which the machine readjusts the clearance. Repeat this operation until the clearance becomes suitable. Finally, press [Memory 1], [Memory 2], or [Memory 3] on the screen to store the guide bushing clearance.

Automatic adjustment of a rotary guide bushing clearance

Select [Memory 1], [Memory 2], or [Memory 3] that corresponds to the clearance stored by the clearance setting. Then, the machine automatically operates and obtains the clearance.

Note

Be sure to set "Bar Stock O.D." in machining data before starting automatic adjustment of a clearance with a guide bushing. The machine adjusts the clearance on the basis of this data.

Manual adjustment of a guide bushing clearance

This method is conventional. For the adjustment method, see <Chapter 9 Mounting, Adjustment, and Replacement>. Select a guide bushing device on the Machine Structure screen.

7.3.8 Clearance setting of a rotary guide bushing

The machine automatically adjusts a clearance with the guide bushing, and the operator checks manually if the automatically obtained clearance is suitable until the clearance becomes suitable. Finally, the operator stores the suitable clearance in the memory of the machine.

Note the following points in using the automatic adjustment function of a guide bushing clearance:

- Clearance data is a numeric value for obtaining a reference value by automatic adjustment. Check if the clearance is suitable by moving the material back and forth by hand, then decide a numeric value as clearance data.
- The numeric value as clearance data is defined with the Citizen guide bushing as reference. The standard value may vary if you use a non-Citizen product.

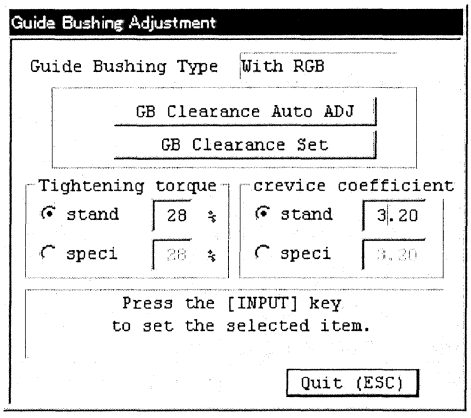
> ⚠ CAUTION
>
> Confirm that the guide bushing clearance is suitable for machine operation. Incorrect clearance may cause machine damage.
> This function is provided to simply support clearance adjustment. The purpose of the function is not to automatically determine a rotary guide bushing clearance.
> Check if the clearance automatically adjusted by this function is suitable by moving the material back and forth by hand, then start the machining process if the clearance is suitable.

Procedure

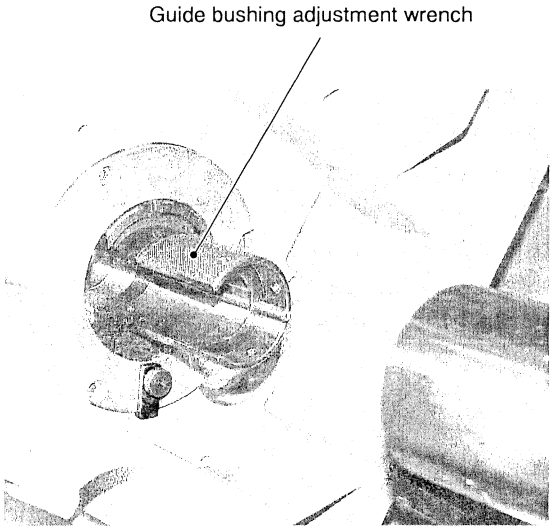
1. Press PREPA-RATION. The PREPARATION key lamp lights, and the Preparation screen appears.

2. Perform the return position operation. See <Section 7.3.1 Start position, return position, opposite tool post retract position (Type VII, VIII, IX), back spindle retract position and gang tool post return position>.

3. Press the menu key [GB ADJ].
   The Guide Bushing Adjustment screen appears.

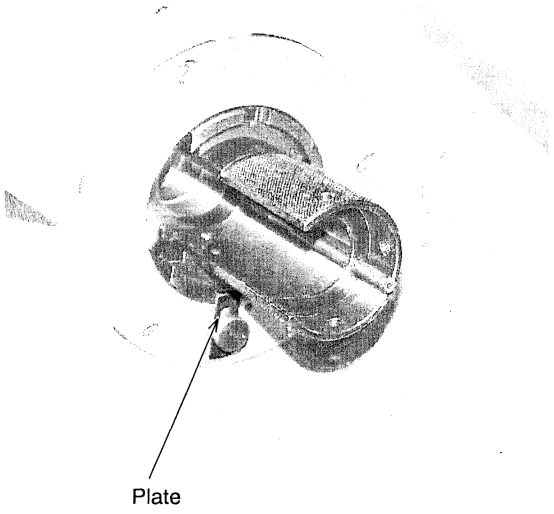


4. Loosen the lock screw with the guide bushing adjustment wrench, then confirm that you can turn the guide bushing adjustment nut smoothly by hand.

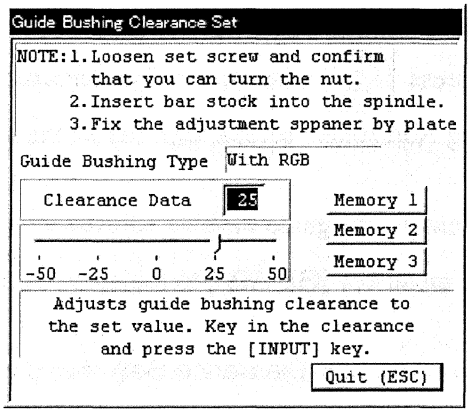


5. Insert a material into the main spindle and guide bushing.

6. Slide the plate provided with the flange to secure the guide bushing adjustment wrench.

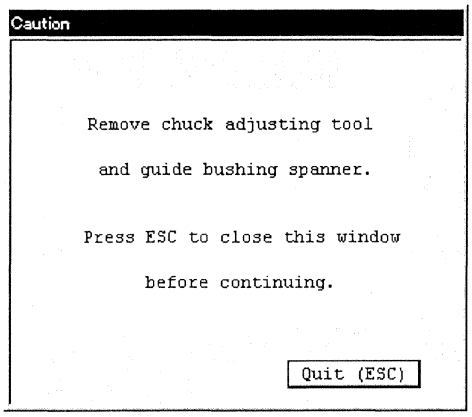


7. Press [↑] and/or [↓] to select [GB Clearance Set], then press [INPUT]. The Guide Bushing Clearance Set screen appears.



8. Press [GRP] to select the adjustment bar, then press [←] and/or [→] to set it to the clearance value you want to obtain. Alternatively, you can select the numeric value area by GRP key and enter the numeric value directly into the area.

9. Press START. The START key lamp lights, and the machine automatically starts adjusting the guide bushing clearance. The machine operates only while the START key is held down. While the machine is operating, the message "Executing guide bushing clearance setting..." is displayed. The Caution screen shown below appears at completion of adjustment, and the START key lamp turns off.

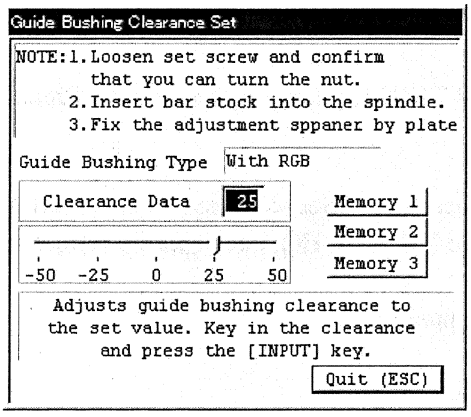


10. Slide the plate provided with the flange back to normal to free the guide bushing adjustment wrench, then check if the clearance is suitable by moving the material back and forth by hand.
If the clearance is suitable, proceed to the next step.
If readjustment is necessary, press ESC to erase the Caution screen, press the menu key [GB ADJ] to redisplay the Guide Bushing Adjustment screen, then restart the adjustment from step 6.

11. Tighten the lock screw, then remove the guide bushing adjustment wrench. Press ESC to erase the Caution screen, then press the menu key [GB ADJ] to redisplay the Guide Bushing Adjustment screen.

12. Press ↑ and/or ↓ to select [GB Clearance Set], then press INPUT.
The Guide Bushing Clearance Set screen appears.

13.Press GRP to select the storage number field on the screen, press ↑ and/or ↓ to select [Memory 1], [Memory 2], or [Memory 3], then press INPUT.



14.Press ESC twice.

The Guide Bushing Clearance Set screen disappears.

15.Remove the material from the guide bushing.

Notes

- You cannot switch the screen or start automatic operation while the Caution screen is displayed.
- The screen automatically returns to the previous Guide Bushing Adjustment screen if the subsequent operation is not performed for 15 seconds after the Guide Bushing Clearance Set screen appears.

7.3.9 Automatic adjustment of a rotary guide bushing clearance

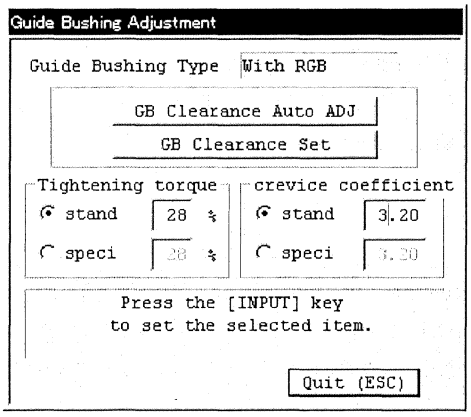
The machine automatically adjusts the guide bushing clearance until it obtains the stored clearance.

Procedure

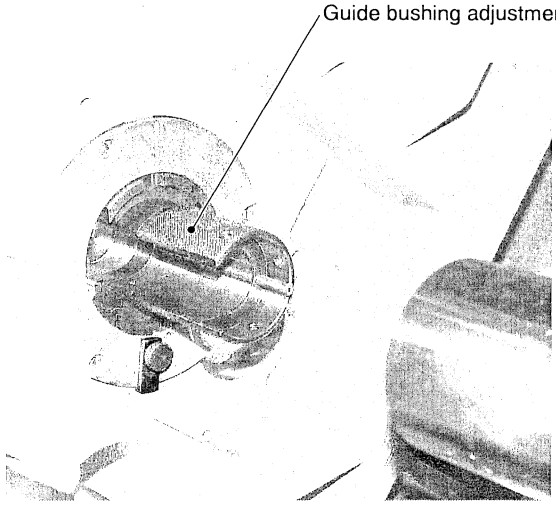
1. Press PREPA-RATION. The PREPARATION key lamp lights, and the Preparation screen appears.

2. Perform the return position operation. See <Section 7.3.1 Start position, return position, opposite tool post retract position (Type VII, VIII, IX), back spindle retract position and gang tool post return position>.
   Each axis moves to the return position.

3. Press the menu key [GB ADJ].
   The Guide Bushing Adjustment screen appears.

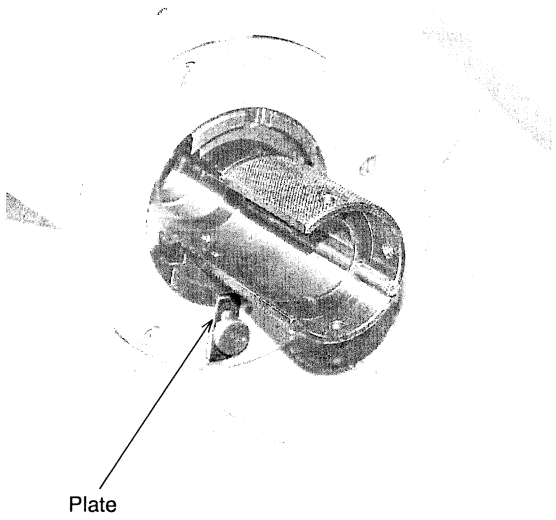


4. Loosen the lock screw with the guide bushing adjustment wrench, then confirm that you can turn the guide bushing adjustment nut smoothly by hand.

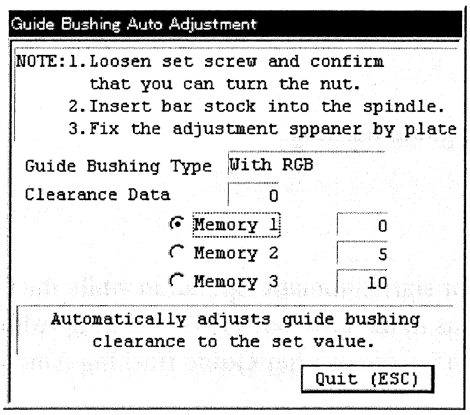


5. Insert a material into the main spindle and guide bushing.

6. Slide the plate provided with the flange to secure the guide bushing adjustment wrench.

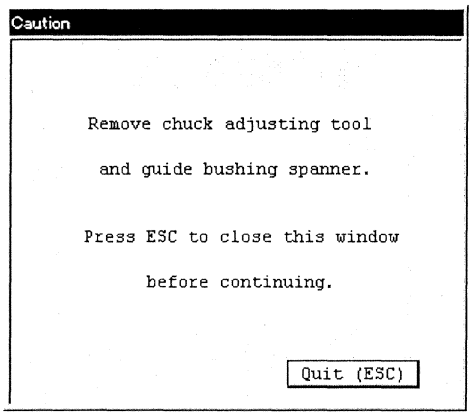


7. Press ↑ and/or ↓ to select [GB Clearance Auto ADJ], then press INPUT.
   The Guide Bushing Auto Adjustment screen appears.



8. Press ↑ and/or ↓ to select [Memory 1], [Memory 2], or [Memory 3] on the screen, then press INPUT.

9. Press START. The START key lamp lights, and the machine automatically starts adjusting the guide bushing clearance. The machine operates only while the START key is held down. While the machine is operating, the message "Executing guide bushing clearance automatic adjustment..." is displayed. The Caution screen shown below appears at completion of adjustment, and the START key lamp turns off.



10. Slide the plate provided with the flange back to normal to free the guide bushing adjustment wrench, then check if the clearance is suitable by moving the material back and forth by hand. Tighten the lock screw, then remove the guide bushing adjustment wrench. Press ESC to erase the Caution screen.

11. Remove the material from the guide bushing.

Notes

- You cannot switch the screen or start automatic operation while the Caution screen is displayed.
- The screen automatically returns to the previous Guide Bushing Adjustment screen if the subsequent operation is not performed for 15 seconds after Guide Bushing Auto Adjustment screen appears.

7.3.10 Manual adjustment of a rotary guide bushing

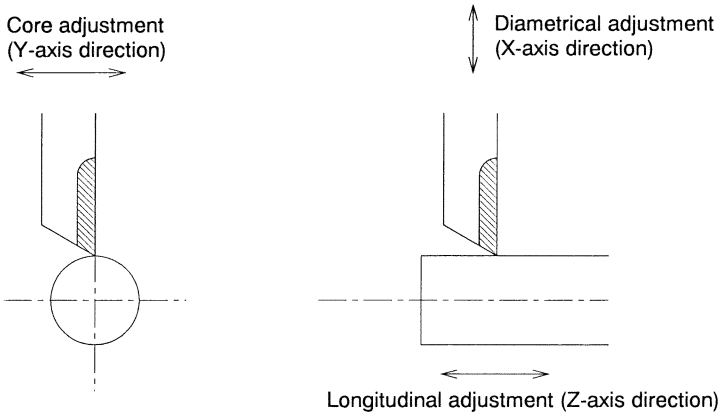
See <Chapter 9 Mounting, Adjustment, and Replacement>.

7.3.11 Setting tools

Overview of tool setting

A tool must be mounted so that the tool nose position (cutting point) is identical to the defined reference point. The purpose of tool setting is to set the tool nose position (cutting point) of each tool to the reference point.
If a tool nose position is not identical to the reference point, the dimensions of a machined workpiece become different from the dimensions specified in the program. Also, a material (workpiece) may interfere with the tool during operation.

Tool setting requires the following adjustment in mounting tools:
Diametrical adjustment: In the X-axis direction
Core adjustment: In the Y-axis direction
Longitudinal adjustment: In the Z-axis direction



Be sure to correctly adjust the core as well. Core adjustment does not directly influence product dimensions, but it influences the surface roughness and tool's life.

Tool setting methods

Tool on gang tool post		
Set the tool with the material as reference in the machine	Diametrical adjustment	Mount the tool in contact with the material outer diameter.
	Core and longitudinal adjustment	Core and longitudinal positions are determined by mounting the tool. Therefore store the shift amount (between the tool nose position and reference point) as tool set data in the memory of the machine. When the tool is selected, the machine automatically makes compensation for the shift amount. With a rotary tool, the machine is adjusted for setting the suitable core when the tool is selected. The shift amount in the longitudinal direction is fixed.
Front/back drilling tool		
Set the tool with the workpiece, chucked by the main/back spindle, as reference in the machine.	Longitudinal adjustment	Mount the tool in contact with the end face of the workpiece.
	Center	The machine is adjusted for aligning the tool center with the workpiece center when the tool is selected. With a boring tool, diametrical adjustment may be easy to achieve by storing the shift amount into the machine. For core adjustment, mount the tool onto the tool sleeve.

The following explains the methods of setting tools on the gang tool post and front/back drilling tool in the machine.

Tool setting procedure

The following outlines the tool setting flow.
The asterisks (*) indicate the work should be done if necessary.

1. Selecting a machining program
   Select the machining program to be executed, and be sure confirm that the machining data is correct.

2. Setting a cut-off tool
   2-1. Core adjustment
   2-2. Diametrical adjustment

3. Setting turning tools other than cut-off tools
   3-1. * Longitudinal adjustment
   3-2. Core adjustment
   3-3. Diametrical adjustment

4. Setting a rotary tool
   4-1. * Core adjustment
   4-2. Diametrical adjustment

5. Setting tools on the front drilling tool
   5-1. * Core adjustment
   5-2. Longitudinal adjustment

6. Program check

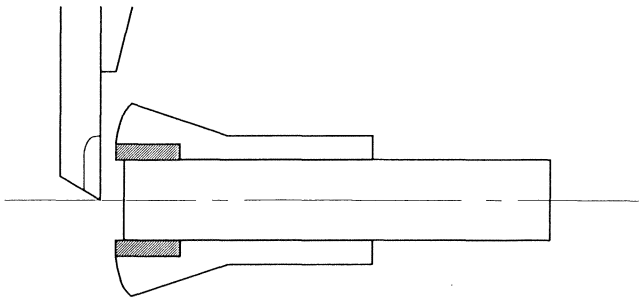
7. Actual machining check

8. Automatic 1-cycle operation
   Front machining is completed, and the workpiece is chucked by the back spindle.
   The total length of the workpiece must be within the tolerance.

9. Setting tools on the back drilling tool
   9-1. * Core adjustment
   9-2. Longitudinal adjustment

Adjusting tools on the gang tool post in the longitudinal direction

Adjust a tool on the gang tool post in the longitudinal direction.
Start the longitudinal tool setting function for tools on the gang tool post. The tool is moved to the position shown in the figure below.



This function sets a tool longitudinal adjustment by aligning the end face of the cut-off tool (as reference) with the end face of the material. Then, it adjusts the tool in the longitudinal direction by aligning the end face of the material with the end face of the tool to be set.

CAUTION

- If the tool set mode is selected, confirm that the end face of a material is cut off normally. If not, perform cut-off machining according to <7.3.13 Performing cut-off machining>.
- Slowly pull out the material from the guide bushing at completion of tool selection. Failure to do so may cause the tool to break or chip and will cause interference between the material and tool.
- Insert a material into the guide bushing before starting tool setting. Failure to do so may cause the tool breakage and will cause interference between the material and tool.

Procedure

1. Press PREPA-RATION. The PREPARATION key lamp lights, and the Preparation screen appears.

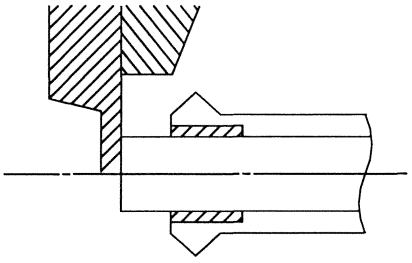
2. Align the end face of the cut-off tool with the end face of the material.

   2-1. Specify bar loader torque OFF, then press MAIN SP. CHUCK to open the chuck.

   With the dedicated bar loader, set POWER on the operation panel to OFF. With other bar loaders, operate bar loader torque OFF on the bar loader.
   When the main spindle chuck opens, insert a material into the guide bushing to prevent interference between the material and cut-off tool during the subsequent start position operation.

   2-2. Perform the start position operation. See <Section 7.3.1 Start position, return position, opposite tool post retract position (Type VII, VIII, IX), back spindle retract position and gang tool post return position>.

   2-3. Pull out the material until it comes into contact with the end face of the cut-off tool.



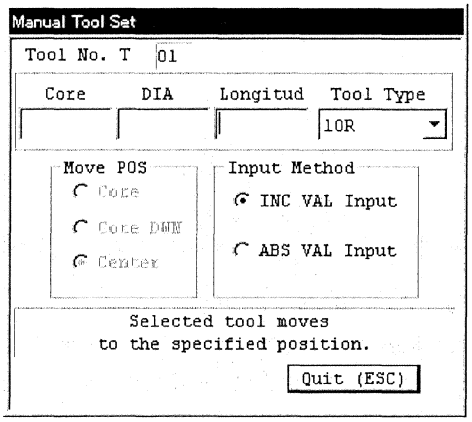
   2-4. Press the MAIN SP.CHUCK key to chuck the material.

3. Select the tool to be set.

   3-1. Remove the tool (to be set) from the tool holder.

   3-2. Press ↑ ↓ or TAB ← TAB → to move the cursor to "Longitud" of the tool you want to set.

3-3. Press the menu key [Man.Set]. The Manual Tool Set screen appears.
Confirm that the cursor is positioned to "Longitud". If the cursor is not there, press GRP or the TAB key to move the cursor to "Longitud".

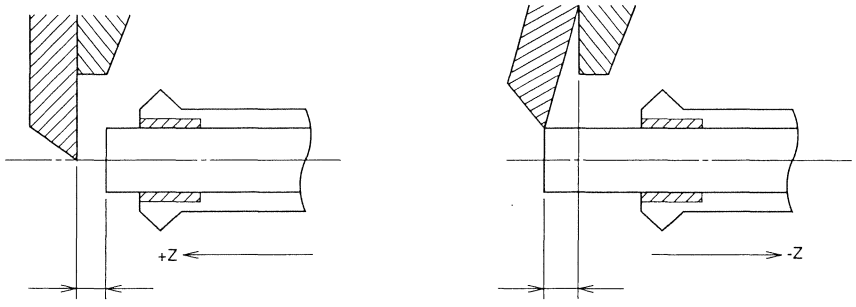


3-4. Press START to start tool setting. The START key lamp lights, and the X2, Z1, X1, and Y1 axes start moving. The machine operates only while the START key is held down.

4. Align the end face of the material with the end face of the tool to be set.

4-1. Push the tool into the tool holder by hand.

4-2. Make longitudinal adjustment. Turn the handle to move the material (Z1) back and forth until the end face of the tool aligns with the end face of the material.



5. Press INPUT to enter the adjusted value.

6. Press the MAIN SP.CHUCK key to open the chuck, then push the material back into the guide bushing.

7. Remove the tool from the tool holder. Secure the tool with the diametrical adjustment.

Longitudinal position of the sleeve holder for the gang tool

Procedure

1. Press PREPARATION. The PREPARATION key lamp lights, and the Preparation screen appears.

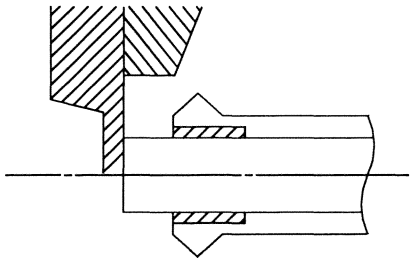
2. Align the end face of the cut-off tool with the end face of the material.

   2-1. Specify bar loader torque OFF, then press MAIN SP. CHUCK to open the chuck.

   With the dedicated bar loader, set POWER on the operation panel to OFF. With other bar loaders, operate bar loader torque OFF on the bar loader.
   When the main spindle chuck opens, insert a material into the guide bushing to prevent interference between the material and cut-off tool during the subsequent start position operation.

   2-2. Perform the start position operation. See <Section 7.3.1 Start position, return position, opposite tool post retract position (Type VII, VIII, IX), back spindle retract position and gang tool post return position>.

   2-3. Pull out the material until it comes into contact with the end face of the cut-off tool.



   2-4. Press the MAIN SP.CHUCK key to chuck the material.

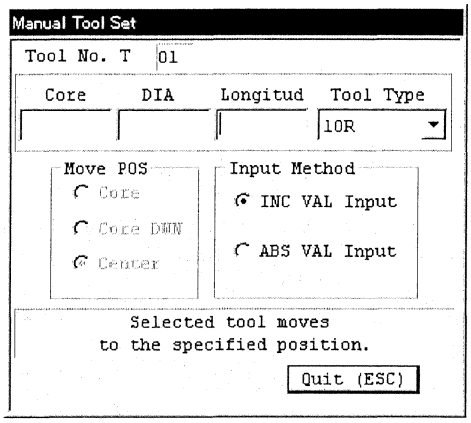
3. Select the tool to be set.

   3-1. Confirm the tool number T01, and perform queuing-point operation. See <7.3.2 Positioning point>.

   3-2. Confirm that the tool mounted in the sleeve holder does not interfere with the workpiece.

   3-3. Press ↑ ↓ or TAB ← TAB → to move the cursor to "Longitud" of the tool you want to set.

3-4. Press the menu key [Man.Set]. The Manual Tool Set screen appears.
Confirm that the cursor is positioned to "Longitud". If the cursor is not there, press GRP or the TAB key to move the cursor to "Longitud".

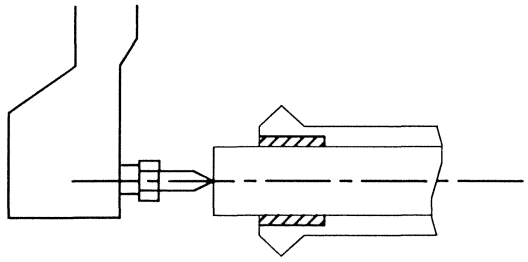


3-5. Press START to start tool setting. The START key lamp lights, and the X2, Z1, X1, and Y1 axes start moving. The machine operates only while the START key is held down.

4. Align the end face of the material with the end face of the tool to be set.

4-1. Bring the tip of the tool into contact with the end face of the material, then temporarily tighten the hexagonal socket set screw of the sleeve.

4-2. With a tool such as a boring end mill, adjust the core height to the cut-off tool mark.



5. To set a tool of another sleeve holder, move the material away by operating the handle (minus direction of Z1), and repeat steps 3 and 4.

6. Press the MAIN SP.CHUCK key to open the chuck, then push the material back into the guide bushing.

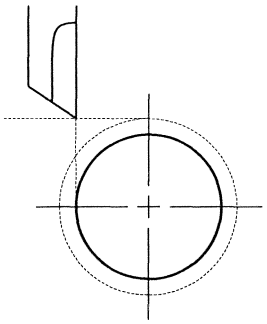
7. Firmly tighten the hexagonal socket set screw of the sleeve with Allen wrench.

Notes

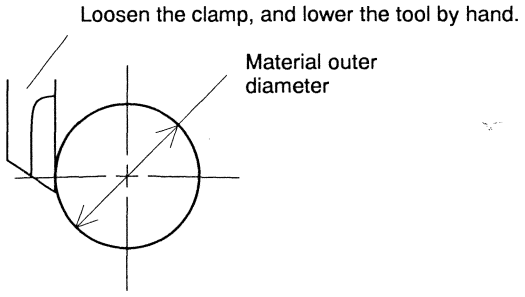
- The machine operates at the speed of 2 m/min or less if a door is open.
- The Manual Tool Set screen automatically closes if the subsequent operation is not performed for 15 seconds after the menu key [Man.Set] is selected.
- The X100 key for adjustment with the handle cannot be used in the above adjustment.

Core (Core)

Use the tool core (Core) setting function to adjust the core of a tool on the gang tool post.
Start the tool core (Core) setting function. The tool is moved to the position shown in the figure below.



This function requires the operator to align the outer circumference of the material with the top face (rake face) of the tool. After that, the machine automatically sets the core adjustment value on the basis of "Bar Stock O.D." specified in the machining data.



Procedure

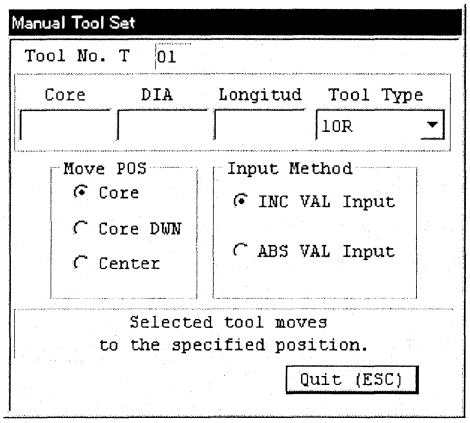
1. Press PREPA-RATION. The PREPARATION key lamp lights, and the Preparation screen appears.

2. Select the tool to be set.

   2-1. Remove the tool (to be set) from the tool holder.

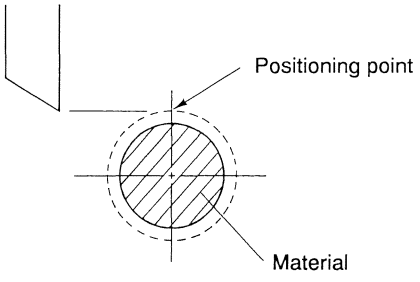
   2-2. Press ↑ ↓ or TAB ← TAB → to move the cursor to "Core" of the tool you want to set.

   2-3. Press the menu key [Man.Set]. The Manual Tool Set screen appears.
   Confirm that the cursor is positioned to "Core". If the cursor is not there, press GRP or the TAB key to move the cursor to "Core".

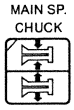


   2-4. Press the GRP key to select the "Move POS" area, press the Up/Down Arrow key to select "Core", then press INPUT.

   2-5. Confirm that the material does not protrude from the guide bushing, then press START to start tool setting. The START key lamp lights. The X1 axis moves to the positioning point, and the Y1 axis moves to the position on the outer circumference of the material. The machine operates only while the START key is held down.

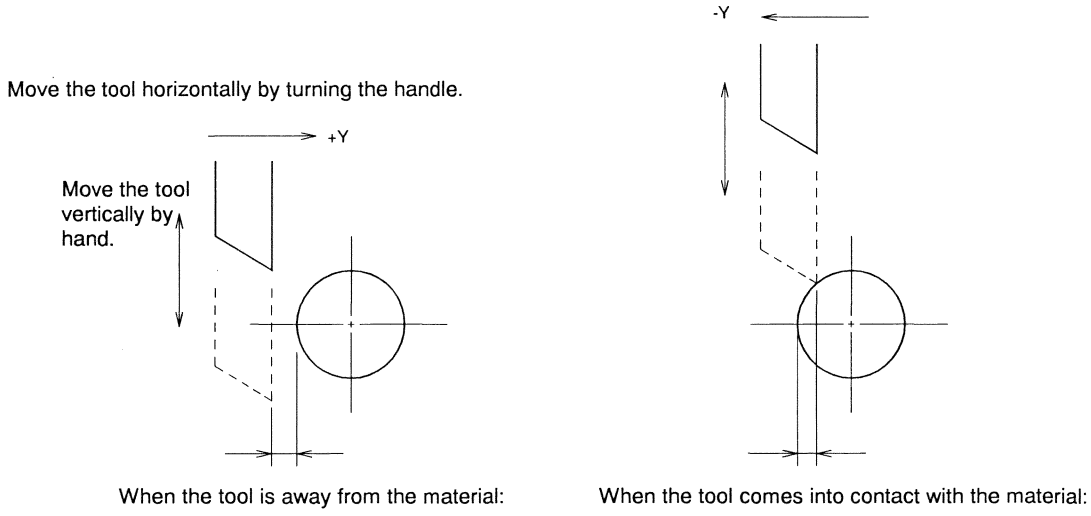


3. Adjust the tool position until the top face of the tool aligns with the outer circumference of the material.

   3-1. Pull out the material, then press 

   to chuck the material.

   3-2. Make core adjustment. Move the tool vertically while pushing it into the tool holder by hand, and turn the handle to move the tool (Y1) horizontally until the top face (rake face) of the tool aligns with the outer circumference of the material.



4. Press 

to enter the adjusted value. The machine calculates and sets the adjusted value from the current Y1 axis coordinate and "Bar Stock O.D." specified in the machining data.

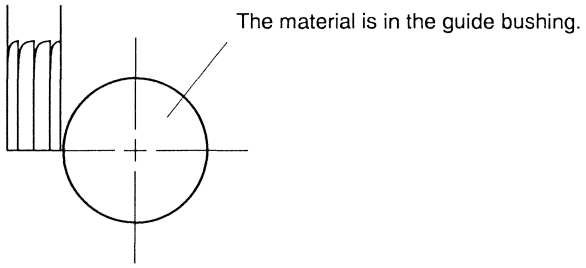
5. Press the MAIN SP.CHUCK key to open the chuck, then push the material back into the guide bushing.

Notes

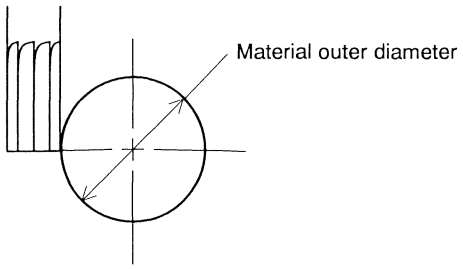
- The machine operates at the speed of 2 m/min or less if a door is open.
- The Manual Tool Set screen automatically closes if the subsequent operation is not performed for 15 seconds after the menu key [Man.Set] is selected.
- The X100 key for adjustment with the handle cannot be used in the above adjustment.

Core (Core DWN)

Use the tool core (Core DWN) setting function to adjust the core of a rotary tool and the core of a gang tool that does not interfere with the material.
Start the tool core (Core DWN) setting function. The tool is moved to the position shown in the figure below.



This function requires the operator to align the outer circumference of the material with the top face of the tool. After that, the machine automatically sets the tool core adjustment value on the basis of "Bar Stock O.D." specified in the machining data.



CAUTION

- Slowly pull out the material from the guide bushing at completion of tool selection. Failure to do so may cause interference between the material and tool, and the tool may break or chip.
- Push a material into the guide bushing before starting tool setting. Failure to do so may cause interference between the material and tool, and the tool may be damaged.

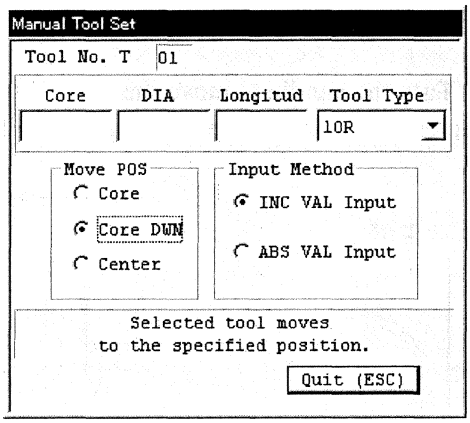
Procedure

1. Press PREPARATION. The PREPARATION key lamp lights, and the Preparation screen appears.

2. Select the tool to be set.

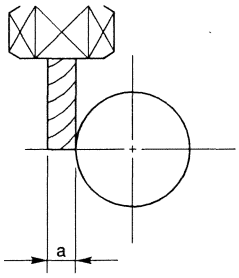
   2-1. Press ↑ ↓ or TAB ← TAB → to move the cursor to "Core" of the tool you want to set.

   2-2. Press the menu key [Man.Set]. The Manual Tool Set screen appears.
   Confirm that the cursor is positioned to "Core". If the cursor is not there, press GRP or the TAB key to move the cursor to "Core".

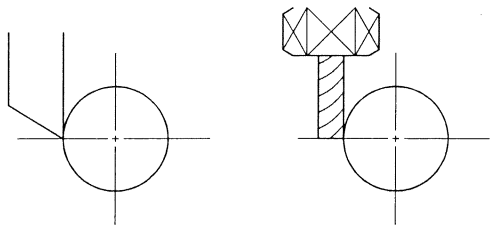


   2-3. Press the GRP key to select the "Move POS" area, press the Up/Down Arrow key to select "Core DWN", then press INPUT.

   2-4. When a rotary tool is set, the center of the tool moves to the position on the outer circumference of the material. In this case, add manually the half value of the tool diameter (a/2) to the core data before starting core adjustment. Set the added value back to the original at completion of core adjustment.



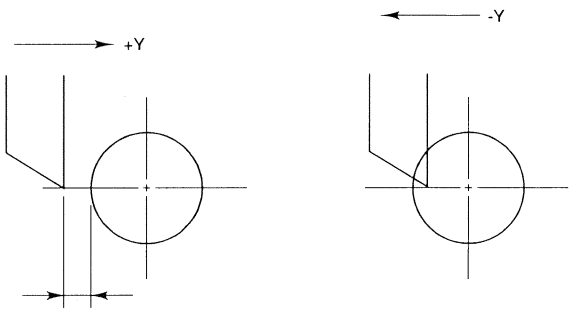
2-5. Confirm that the material does not protrude from the guide bushing, then press START to start tool setting. The START key lamp lights. The X1 axis moves to the position X=0, and the Y1 axis moves to the position on the outer circumference of the material. The machine operates only while the START key is held down.



With a turning tool: With a rotary tool:

3. Adjust the tool position until the top face of the tool aligns with the outer circumference of the material.

3-1. Make core adjustment. Turn the handle to move the tool (Y1) horizontally until the top face of the tool aligns with the outer diameter of the material while pull out and in the material.



4. Press INPUT to enter the adjusted value. The machine calculates and sets the adjusted value from the current Y1 axis coordinate and "Bar Stock O.D." specified in the machining data.

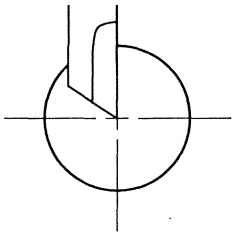
5. Press the MAIN SP.CHUCK key to open the chuck, then push the material back into the guide bushing.

Notes

- The machine operates at the speed of 2 m/min or less if a door is open.
- The Manual Tool Set screen automatically closes if the subsequent operation is not performed for 15 seconds after the menu key [Man.Set] is selected.
- The X100 key for adjustment with the handle cannot be used in the above adjustment.

Core (center)

Use the tool core (center) setting function to adjust the core of a turning tool.
Start the tool core (center) setting function. The tool is moved to the position shown in the figure below.



This function sets a tool core by aligning the top face (rake face) of the tool with the center of the material. A cut-off tip left on the cut-off face of a material is used for the alignment.

> **CAUTION**
>
> - Slowly pull out the material from the guide bushing at completion of tool selection. Failure to do so may cause the tool to break or chip and will cause interference between the material and tool.
> - Push a material into the guide bushing before starting tool setting. Failure to do so may cause the tool breakage and will cause interference between the material and tool.

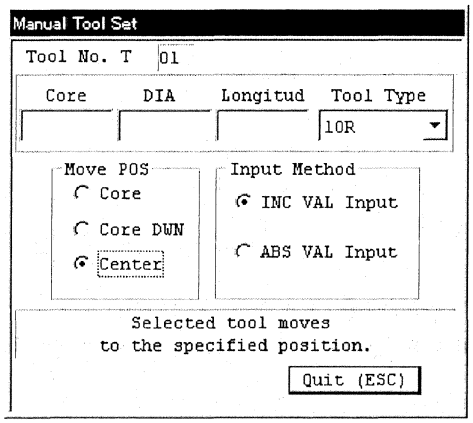
Procedure

1. Press PREPA-RATION. The PREPARATION key lamp lights, and the Preparation screen appears.

2. Select the tool to be set.

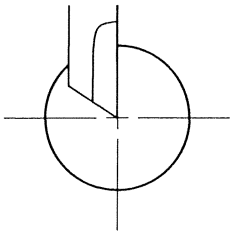
   2-1. Press ↑ ↓ or TAB ← TAB → to move the cursor to "Core" of the tool you want to set.

2-2. Press the menu key [Man.Set]. The Manual Tool set screen appears.
Confirm that the cursor is positioned to "Core". If the cursor is not there, press GRP or the TAB key to move the cursor to "Core".

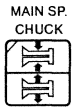


2-3. Press the GRP key to select the "Move POS" area, press the Up/Down Arrow key to select "Center ", then press INPUT.

2-4. Confirm that the material does not protrude from the guide bushing, then press START to start tool setting. The START key lamp lights. The X1 axis moves to the position X=0, and the Y1 axis moves to the position Y=0. The machine operates only while the START key is held down.

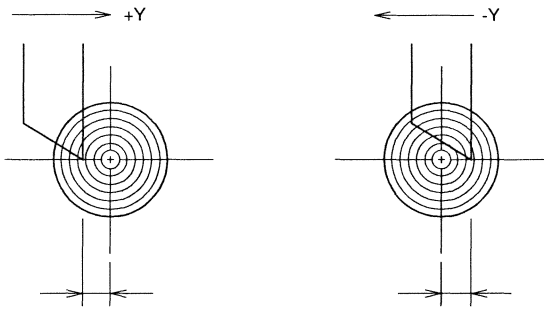


3. Adjust the tool position until the top face of the tool aligns with the center of the material.

   3-1. Pull out the material, then press 

   to chuck the material.

   3-2. Make core adjustment. Turn the handle to move the tool (Y1) horizontally until the top face of the tool aligns with the center of the material. A cut-off tip left on the cut-off face of the material is used for the alignment.



4. Press 

to enter the adjusted value. The machine sets the current Y1 axis coordinate.

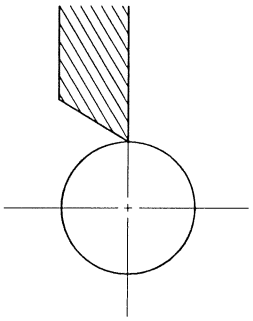
5. Press the MAIN SP.CHUCK key to open the chuck, then push the material back into the guide bushing.

Notes

- The machine operates at the speed of 2 m/min or less if a door is open.
- The Manual Tool Set screen automatically closes if the subsequent operation is not performed for 15 seconds after the menu key [Man.Set] is selected.
- The X100 key for adjustment with the handle cannot be used in the above adjustment.

Diameter

Use the diametrical tool setting function to adjust the diameters of a turning tool and rotary tool.
Start the diametrical tool setting function. The tool is moved to the position shown in the figure below.



This function positions the tip (top face) of a tool or rotary tool to the point where it comes in contact with the outer diameter of the material.
Three methods are available for tool diametrical adjustment.

- Adjusting the diameters of mounted tools
  This method adjusts the diameters of a turning tool and rotary tool that are already mounted.

- Mounting tools while adjusting the diameters
  This method adjusts the diameter when mounting a new turning tool or rotary tool on the holder. The method may fail to set tools precisely. In this case, readjust the tools by the above method "adjusting the diameters of mounted tools".

- Mounting a tool with the diameter offset
  Use this method when performing deep hole machining (through-hole) with a rotary tool. The method mounts a rotary tool or turning tool while protrusion it more than usual.

CAUTION

- Slowly pull out the material from the guide bushing at completion of tool selection. Failure to do so may cause interference between the material and tool, and the tool may break or chip.
- Push a material into the guide bushing before starting tool setting. Failure to do so may cause interference between the material and tool, and the tool may be damaged.

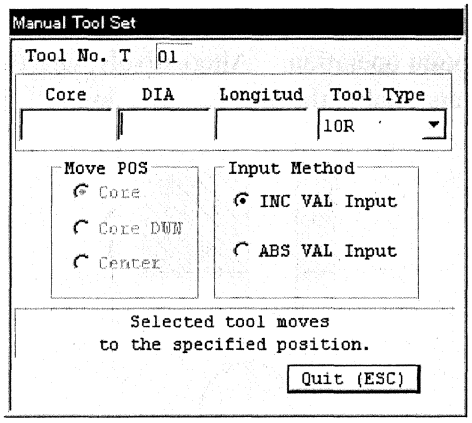
Procedure

1. Press PREPA-RATION. The PREPARATION key lamp lights, and the Preparation screen appears.

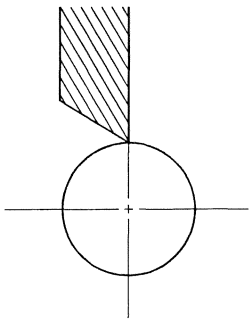
2. Select the tool to be set.

   2-1. Press ↑ ↓ or TAB ← TAB → to move the cursor to "DIA" of the tool you want to set.

   2-2. Press the menu key [Man.Set]. The Manual Tool Set screen appears.
   Confirm that the cursor is positioned to "DIA". If the cursor is not there, press GRP or the TAB key to move the cursor to "DIA".

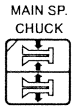


   2-3. Confirm that the material does not protrude from the guide bushing, then press START to start tool setting. The START key lamp lights. The X1 axis moves to the position on the outer diameter of the material, and the Y1 axis moves to the position Y=0. The machine operates only while the START key is held down.



3. Adjust the tool the position until the top face of the tool aligns with the outer diameter of the material, then enter the setting value.

- When mounting tools while adjusting the diameters:

3-1. Pull out the material, then press 

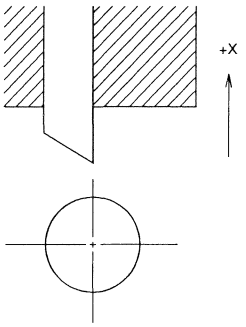
to chuck the material.

3-2. Softly bring the tip of the turning tool or the tip (face) of the rotary tool into contact with the outer diameter of the material, then secure it temporarily.

3-3. Press 

to enter the adjusted value. The machine calculates and sets the current X1 axis coordinate on the basis of "Bar Stock O.D." specified in the machining data.

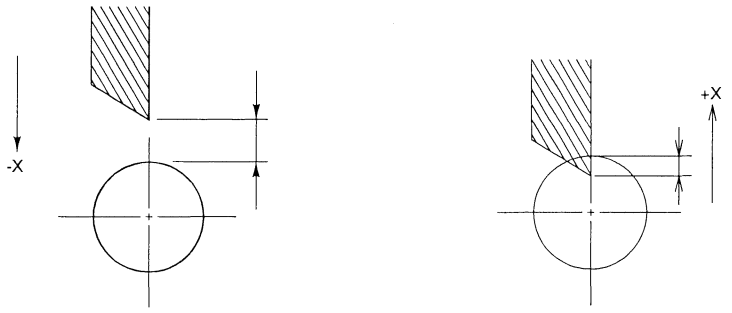
3-4. Perform the positioning point operation. Alternatively, turn the handle to vertically move the tool (X1) away from the material in the plus (+) direction. See <Section 7.3.2 Positioning point>.



3-5. Firmly secure the tool. Be careful not to move the tool in securing it.

- When adjusting the diameters of mounted tools:

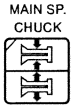
3-1. Make diametrical adjustment. Turn the handle to vertically move the tool (X1) until the outer diameter of the material aligns with the tip (face) of the tool while pull out and in the material.



3-2. Press 

to enter the adjusted value. The machine calculates and sets the adjusted value from the current X1 axis coordinate and "Bar Stock O.D." specified in the machining data.

- When mounting a tool with the diameter offset:

3-1. Pull out the material, then press 

to chuck the material.

3-2. Turn the handle to move X1 axis to the position where its protrusion is enough for machining in the plus (+) direction. Softly bring the tip of the turning tool or the tip (face) of the rotary tool into contact with the outer circumference of the material, then secure the tool temporarily.

3-3. Press 

to enter the adjusted value.

3-4. Perform the positioning point operation. Alternatively, turn the handle to vertically move the tool (X1) away from the material in the plus (+) direction. See <Section 7.3.2 Positioning point>.

3-5. Firmly secure the tool. Be careful not to move the tool in securing it.

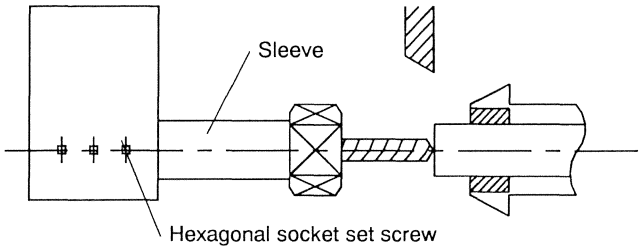
4. Press the MAIN SP.CHUCK key to open the chuck, then push the material back into the guide bushing.

Notes

- The method "when mounting tools while adjusting the diameters" may fail to set tools precisely. In this case, readjust the tools by the method "When adjusting the diameters of mounted tools".
- The machine operates at the speed of 2 m/min or less if a door is open.
- The Manual Tool Set screen automatically closes if the subsequent operation is not performed for 15 seconds after the menu key [Man.Set] is selected.
- The X100 key for adjustment with the handle cannot be used in the above adjustment.

Front drilling tool in the longitudinal direction

The following example shows to adjust the front drilling tool in the longitudinal direction.
Start the longitudinal tool setting function for tools on the front drilling tool. The machine moves the tool to the standard setting position shown in the figure below.



This function sets the tool in the longitudinal direction by aligning the tool tip of a tool on the opposite tool post with the end face of the material chucked by the main spindle.

> ⚠ CAUTION
>
> • Move the sleeve of the front drilling tool to the backward end position before starting tool setting. Failure to do so may cause interference between the workpiece and tool, and the tool may be damaged.

Procedure

1. Set a material into the main spindle.

   1.1 Turn off the bar loader torque and open the chuck. (For the dedicated bar loader, set the POWER key on the operation panel to OFF. For any other bar loader, turn off the torque on the bar loader.)

   1-2. Perform start position operation. <Section 7.3.1 Start position, return position, opposite tool post retract position (Type VII, VIII, IX), back spindle retract position and gang tool post return position >

   1-3. Align the end face of the material with the end face of the cut-off tool.

   1-4. Close the chuck.

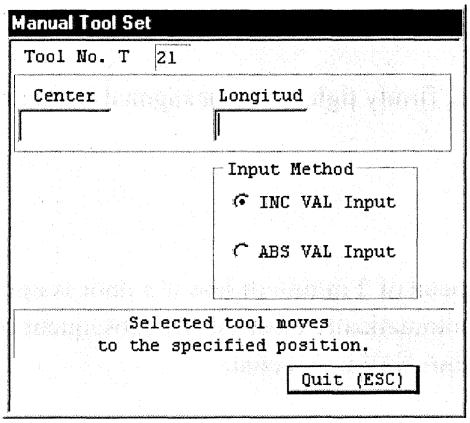
2. Press PREPARATION. The PREPARATION key lamp lights, and the Preparation screen appears.

3. Select the tool to be set.

3-1. Loosen the hexagonal socket set screw securing the sleeve of the front drilling tool on which the tool to be set is mounted. (Use Allen wrench to loosen the set screw.) Move the sleeve to the backward end position (shortest). Leave the hexagonal socket set screw loose.

3-2. Press   or   to move the cursor to "Longitud" of the tool you want to set.

3-3. Press the menu key [Man.Set]. The Manual Tool Set screen appears.
Confirm that the cursor is positioned to "Longitud". If the cursor is not there, press  or the TAB key to move the cursor to "Longitud".

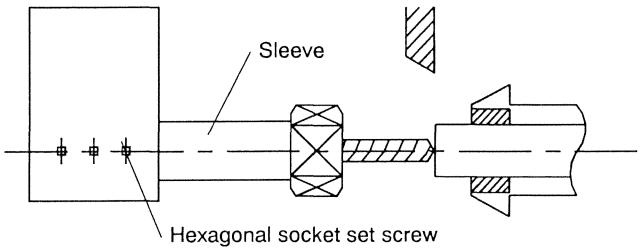


(This screen is for type VII, VIII, IX.)

3-4. Press  to start tool setting. The START key lamp lights. The X2 axis (X1, Y1 axes: type VI) moves to the selected tool position. The machine operates only while the START key is held down.

4. Adjust the tool position until the tip of the tool aligns with the end face of the material.

   4-1. Bring the tip of the tool into contact with the end face of the material, then temporarily tighten the hexagonal socket set screw of the sleeve.



   4-2. Press ESC to complete the adjustment.

   Repeat the operations in steps 3 and 4 to adjust the positions of all the front drilling tools, then secure the sleeve temporarily.

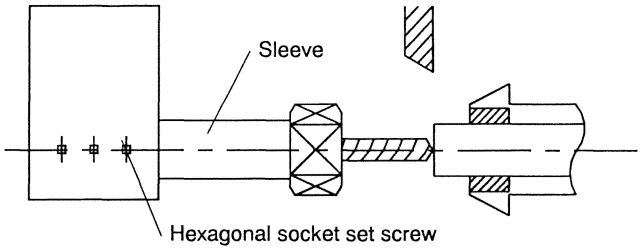
5. After retracting the workpiece, firmly tighten the hexagonal socket set screw of the sleeve with Allen wrench.

Notes

- The machine operates at the speed of 2 m/min or less if a door is open.
- The Manual Tool Set screen automatically closes if the subsequent operation is not performed for 15 seconds after the menu key [Man.Set] is selected.

Adjusting the centers of tools on the front drilling tool

Adjust the center of a drilling tool on the front drilling tool. The center position has been adjusted during the manufacturing process of this machine. Use the tool center setting function to confirm the center position or to make fine adjustment.
Start the tool center setting function for tools on the front drilling tool. The machine moves the tool to the position shown in the figure below.



This function sets the center of a drilling tool by aligning the center of tool with the center of the material. The fretting on the end face of the material is used for the center adjustment.

> **CAUTION**
>
> Be sure to complete the longitudinal tool setting before starting tool setting for the center of a tool on the front drilling tool. Failure to do so may cause interference between the workpiece and tool, and the tool may be damaged.

Procedure

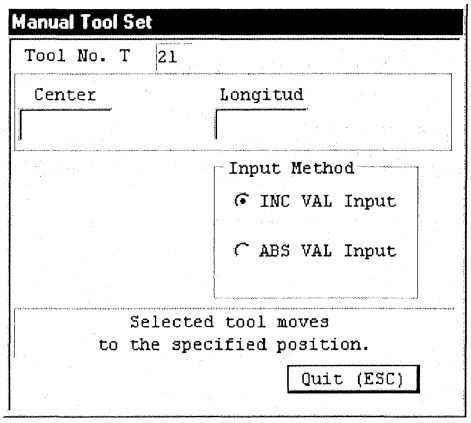
1. Press PREPARATION. The PREPARATION key lamp lights, and the Preparation screen appears.

2. Select the tool to be set.

2-1 Confirm that the longitudinal adjustment of the tool is completed.

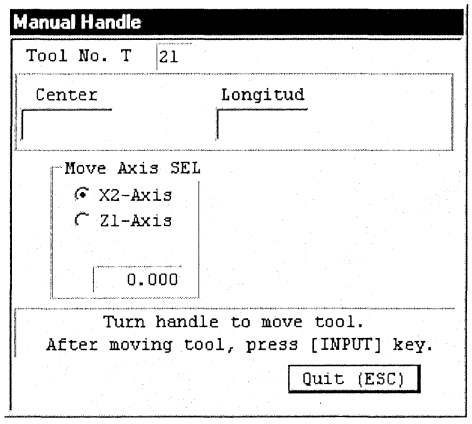
2-2. Press ↑ ↓ or TAB ← TAB → to move the cursor to "Center" ("Core" or "DIA": type VI) of the tool you want to set.

2-3. Press the menu key [Man.Set]. The Manual Tool Set screen appears. Confirm that the cursor is positioned to "Center" ("Core" or "DIA": type VI). If the cursor is not there, press GRP or the TAB key to move the cursor to "Center" ("Core" or "DIA": type VI).



(This screen is for type VII, VIII, IX.)

2-4. Press START to start tool setting. The START key lamp lights. The X2 axis (X1, Y1 axes: type VI) moves to the selected tool position. The machine operates only while the START key is held down.

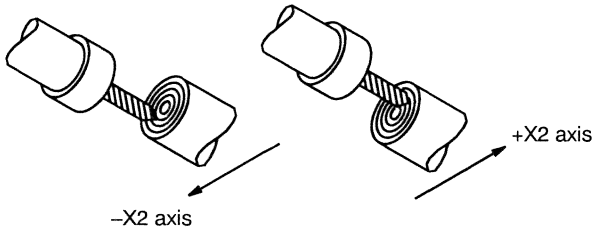


(This screen is for type VII, VIII, IX.)

3. Adjust the tool position until the center of the tool aligns with the center of the material.

   3-1. Pull the material without letting its end face touch the tool.

   3-2. Turn the handle to move the X2 axis until the center of the tool (tip of the tool nose) aligns with the center of fretting on the end face of the workpiece.



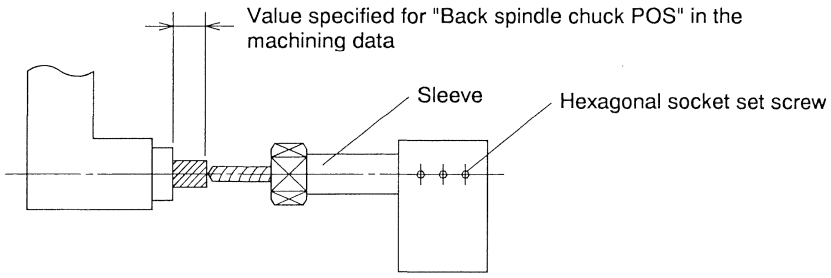
   3-3. Press the INPUT key to enter the adjusted value. The machine sets the current X2 axis (X, Y axes: type VI) coordinate.

Notes

- The machine operates at the speed of 2 m/min or less if a door is open.
- The Manual Tool Set screen automatically closes if the subsequent operation is not performed for 15 seconds after the menu key [Man.Set] is selected.
- The x100 key for adjustment with the handle cannot be used in the above adjustment.

Back drilling tool in the longitudinal direction

The following example shows to adjust the back drilling tool in the longitudinal direction.
Start the longitudinal tool setting function for tools on the back drilling tool. The machine moves the tool to the standard setting position shown in the figure below.



This function sets the tool in the longitudinal direction by aligning the tool tip of a tool on the opposite tool post with the end face of the workpiece chucked by the back spindle.

> ⚠ CAUTION
>
> - Move the sleeve of the back drilling tool to the backward end position before starting tool setting. Failure to do so may cause interference between the workpiece and tool, and the tool may be damaged.
> - Confirm that a correct value is set for "back spindle chuck POS" in the machining data, then start the tool setting function. Failure to do so may cause interference between the workpiece and tool, and the tool may be damaged.

Procedure

1. Set a workpiece into the back spindle.

   1.1 Stop the machine with [1 Cycle] in the automatic operation mode while the workpiece with which front machining is completed is chucked by the back spindle.

   1-2. Measure the workpiece protrusion length from the back spindle chuck, then compare the measured value with the value specified for "back spindle chuck POS" in the machining data. If the values are not identical, correct the program or machining data.

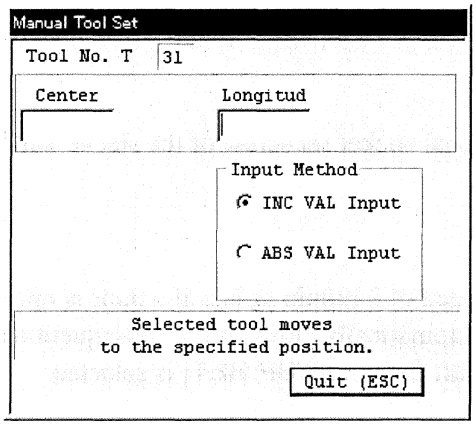
2. Press PREPA-RATION. The PREPARATION key lamp lights, and the Preparation screen appears.

3. Select the tool to be set.

3-1. Loosen the hexagonal socket set screw securing the sleeve of the front/back drilling tool on which the tool to be set is mounted. (Use Allen wrench to loosen the set screw.) Move the sleeve to the backward end position (shortest). Leave the hexagonal socket set screw loose.

3-2. Press ↑ ↓ or TAB ← TAB → to move the cursor to "Longitud" of the tool you want to set.

3-3. Press the menu key [Man.Set]. The Manual Tool Set screen appears. Confirm that the cursor is positioned to "Longitud". If the cursor is not there, press GRP or the TAB key to move the cursor to "Longitud".

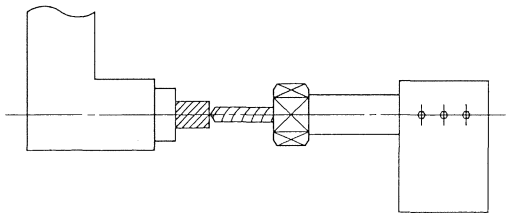


(This screen is for type VII, VIII, IX.)

3-4. Press START to start tool setting. The START key lamp lights. The Z3 axis moves to the return position, the X2 axis (X1, Y1 axes: type VI) moves to the selected tool position, and then the Z3 axis advances to the standard setting position. The machine operates only while the START key is held down.

4. Adjust the tool position until the tip of the tool aligns with the end face of the workpiece.

   4-1. Bring the tip of the tool into contact with the end face of the workpiece, then temporarily tighten the hexagonal socket set screw of the sleeve.



   4-2. Press 

to complete the adjustment.

   4-3. Press the menu key [B/SP RET], then press the START key. The START key lamp lights, and the back spindle moves back to Z3 axis zero point. The machine operates only while the START key is held down.

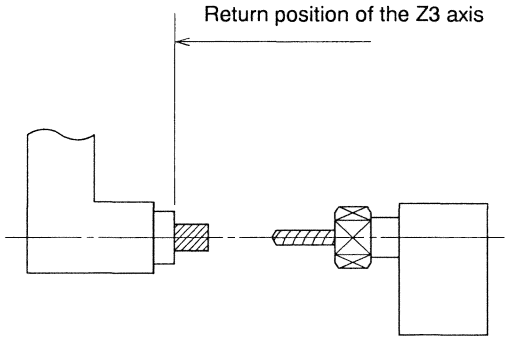
   4-4. Firmly tighten the hexagonal socket set screw of the sleeve with Allen wrench.

Notes

- The machine operates at the speed of 2 m/min or less if a door is open.
- The Manual Tool Set screen automatically closes if the subsequent operation is not performed for 15 seconds after the menu key [Man.Set] or [B/SP RET] is selected.

Adjusting the centers of tools on the back drilling tool

Adjust the center of a drilling tool on the back drilling tool. The center position has been adjusted during the manufacturing process of this machine. Use the tool center setting function to confirm the center position or to make fine adjustment.
Start the tool center setting function for tools on the back drilling tool. The machine moves the tool to the position shown in the figure below.



This function sets the center of a drilling tool by aligning the center of tool with the center of the workpiece. The fretting on the end face of the workpiece is used for the center adjustment.

> ⚠ CAUTION
>
> Be sure to complete the longitudinal tool setting before starting tool setting for the center of a tool on the back drilling tool. Failure to do so may cause interference between the workpiece and tool, and the tool may be damaged.

Procedure

1. Set a workpiece into the back spindle.

   1-1. Stop the machine with [1 Cycle] in the automatic operation mode while the workpiece with which front machining is completed is chucked by the back spindle.

   1-2. Measure the workpiece protrusion length from the back spindle chuck, then compare the measured value with the value specified for "back spindle chuck POS" in the machining data. If the values are not identical, correct the program or machining data.

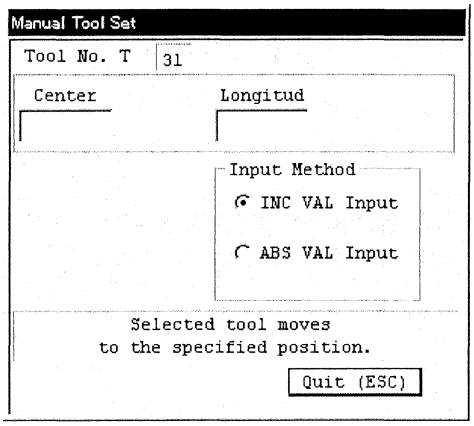
2. Press PREPA-RATION. The PREPARATION key lamp lights, and the Preparation screen appears.

3. Select the tool to be set.

3-1 Confirm that the longitudinal adjustment of the tool is completed.

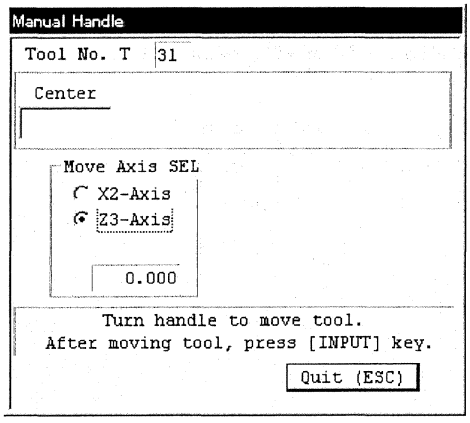
3-2. Press ↑ ↓ or TAB ← TAB → to move the cursor to "Center" ("Core" or "DIA": type VI) of the tool you want to set.

3-3. Press the menu key [Man.Set]. The Manual Tool Set screen appears.
Confirm that the cursor is positioned to "Center" ("Core" or "DIA": type VI). If the cursor is not there, press GRP or the TAB key to move the cursor to "Center" ("Core" or "DIA": type VI).



(This screen is for type VII, VIII, IX.)

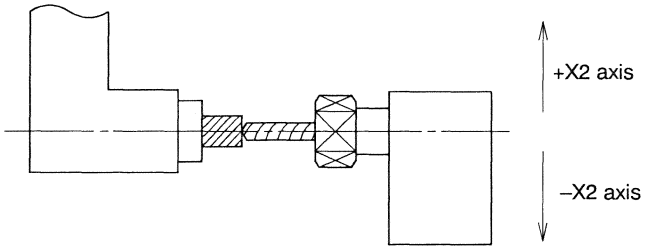
3-4. Press START I to start tool setting. The START key lamp lights. The Z3 axis moves to the return position, and the X2 axis (X1, Y1 axes: type VI) moves to the selected tool position. The Z3 axis remains at the return position. The machine operates only while the START key is held down.



(This screen is for type VII, VIII, IX.)

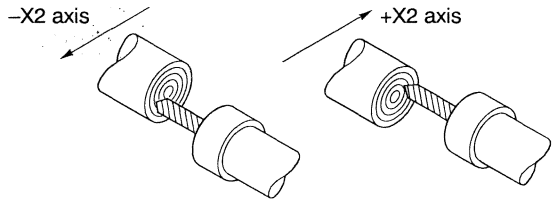
4. Adjust the tool position until the center of the tool aligns with the center of the workpiece.

   4-1. Press GRP and ↑ ↓ to move the cursor to the "Z3-Axis" in the "Move Axis SEL" area, press INPUT, then turn the handle to advance the Z3 axis to the position where the tip of the tool comes into contact with the end face of the workpiece.



   4-2. Press the GRP key and Up/Down Arrow key to move the cursor to the "X2 axis" (X, Y axes: type VI) in the "Move Axis SEL" area, then press the INPUT key.

   4-3. Turn the handle to move the X2 axis (X, Y axes: type VI) until the center of the tool (tip of the tool nose) aligns with the center of fretting on the end face of the workpiece.



   4-4. Press the INPUT key to enter the adjusted value. The machine sets the current X2 axis (X, Y axes: type VI) coordinate.

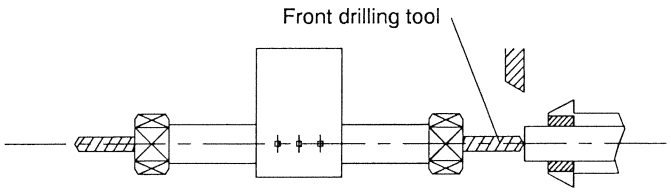
5. Press the menu key [B/SP RET], then press the START key to move the back spindle backward.

Notes

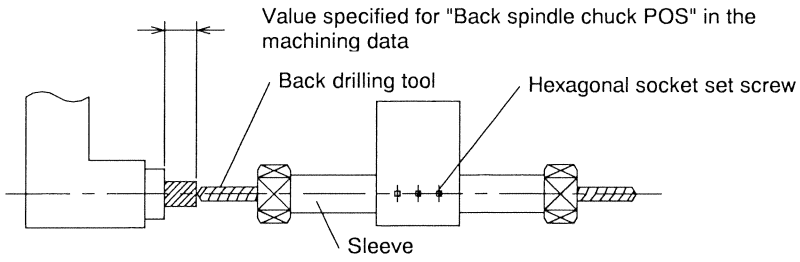
- The machine operates at the speed of 2 m/min or less if a door is open.
- The Manual Tool Set screen automatically closes if the subsequent operation is not performed for 15 seconds after the menu key [Man.Set] or [B/SP RET] is selected.
- The X100 key for adjustment with the handle cannot be used in the above adjustment.

Front/Back both end drilling tool in the longitudinal direction

The following example shows to adjust the front/back both end drilling tool in the longitudinal direction. First adjust the front drilling tool in the longitudinal direction, and then adjust the back drilling tool in the longitudinal direction.



Adjust fornt drilling tool



Adjust back drilling tool

This function sets the tool in the longitudinal direction by aligning the tool tip of a tool on the opposite tool post with the end face of the material chucked by the main spindle, and the end face of the workpiece chucked by the back spindle.

> ⚠ CAUTION
>
> - Insert the back drilling tool into the sleeve before setting the tool. Failure to do so may cause interference between the workpiece and tool, and the tool may be damaged.
> - Confirm that a correct value is set for "back spindle chuck POS" in the machining data, then start the tool setting function. Failure to do so may cause interference between the workpiece and tool, and the tool may be damaged.

Procedure

1. Adjust the front drilling tool in the longitudinal direction.
   Set a material into the main spindle.

   1.1 Turn off the bar loader torque and open the chuck. (For the dedicated bar loader, set the 

   key on the operation panel to OFF. For any other bar loader, turn off the torque on the bar loader.)

   1-2. Perform start position operation. <Section 7.3.1 Start position, return position, opposite tool post retract position (Type VII, VIII, IX), back spindle retract position and gang tool post return position >

1-3. Align the end face of the material with the end face of the cut-off tool.

1-4. Close the chuck.

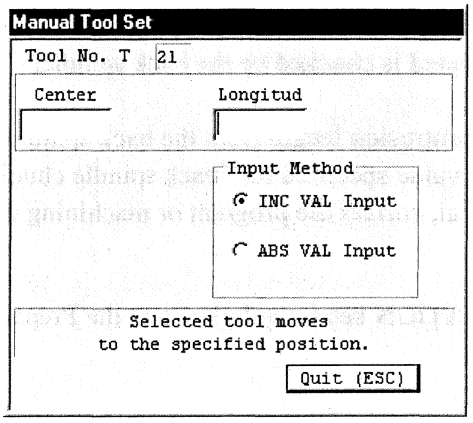
2. Press PREPA-RATION. The PREPARATION key lamp lights, and the Preparation screen appears.

3. Select the tool to be set.

3-1. Loosen the hexagonal socket set screw securing the sleeve of the front/back both end drilling tool on which the tool to be set is mounted. (Use Allen wrench to loosen the set screw.) Move the sleeve to the backward end position (shortest). Leave the hexagonal socket set screw loose. Be sure to insert the back drilling tool into the sleeve beforehand.

3-2. Press ↑ ↓ or TAB TAB to move the cursor to "Longitud" of the tool you want to set.

3-3. Press the menu key [Man.Set]. The Manual Tool Set screen appears.
Confirm that the cursor is positioned to "Longitud". If the cursor is not there, press GRP or the TAB key to move the cursor to "Longitud".

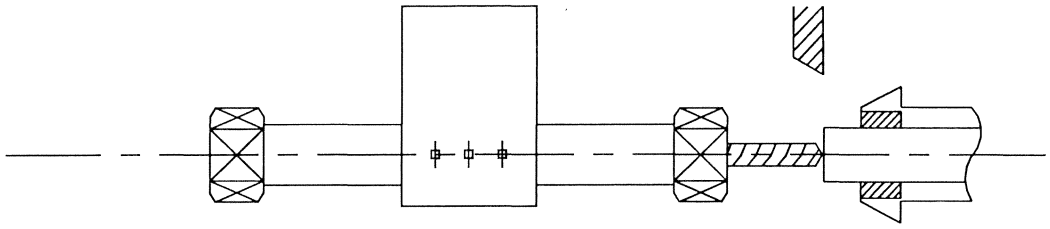


(This screen is for type VII, VIII, IX.)

3-4. Press START to start tool setting. The START key lamp lights. The X2 axis (X1, Y1 axes: type VI) moves to the selected tool position. The machine operates only while the START key is held down.

4. Adjust the tool position until the tip of the tool aligns with the end face of the material.

   4-1. Bring the tip of the tool into contact with the end face of the material, then temporarily tighten the hexagonal socket set screw of the sleeve.



   4-2. Press ESC to complete the adjustment.

   Repeat the operations in steps 3 and 4 to adjust the positions of all the front drilling tools, then secure the sleeve temporarily.

5. After retracting the workpiece, firmly tighten the hexagonal socket set screw of the sleeve with Allen wrench.

6. Adjust the back drilling tool in the longitudinal direction.
   Set a workpiece into the back spindle.

   6.1 Stop the machine with [1 Cycle] in the automatic operation mode while the workpiece with which front machining is completed is chucked by the back spindle.

   6-2. Measure the workpiece protrusion length from the back spindle chuck, then compare the measured value with the value specified for "back spindle chuck POS" in the machining data. If the values are not identical, correct the program or machining data.

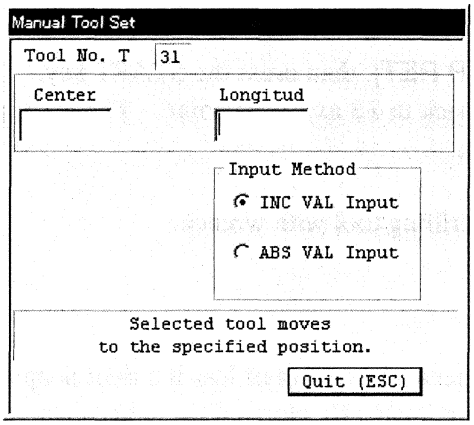
7. Press PREPARATION. The PREPARATION key lamp lights, and the Preparation screen appears.

8. Select the tool to be set.

8-1. Insert the back drilling tool into the sleeve for front/back both end drilling.

8-2. Press ↑ ↓ or TAB ← TAB → to move the cursor to "Longitud" of the tool you want to set.

8-3. Press the menu key [Man.Set]. The Manual Tool Set screen appears. Confirm that the cursor is positioned to "Longitud". If the cursor is not there, press GRP or the TAB key to move the cursor to "Longitud".

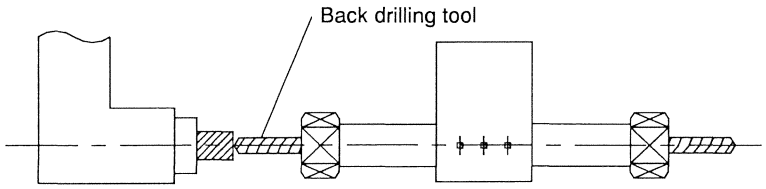


(This screen is for type VII, VIII, IX.)

8-4. Press START I to start tool setting. The START key lamp lights. The Z3 axis moves to the return position, the X2 axis (X1, Y1 axes: type VI) moves to the selected tool position, and then the Z3 axis advances to the standard setting position. The machine operates only while the START key is held down.

9. Adjust the tool position until the tip of the tool aligns with the end face of the workpiece.

   9-1. Bring the tip of the tool into contact with the end face of the workpiece, then temporarily tighten the back drilling tool.



   9-2. Press  to complete the adjustment.

   9-3. Press the menu key [B/SP RET], then press the START key. The START key lamp lights, and the back spindle moves back to Z3 axis zero point. The machine operates only while the START key is held down.

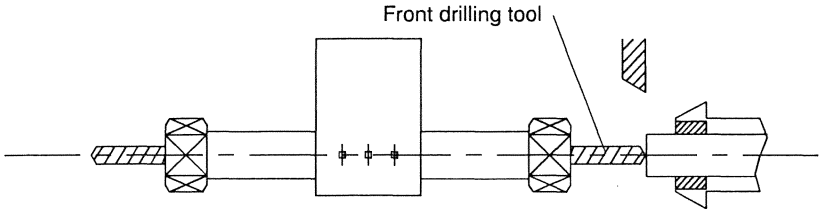
   9-4. Firmly tighten the back drilling tool with wrench.

Notes

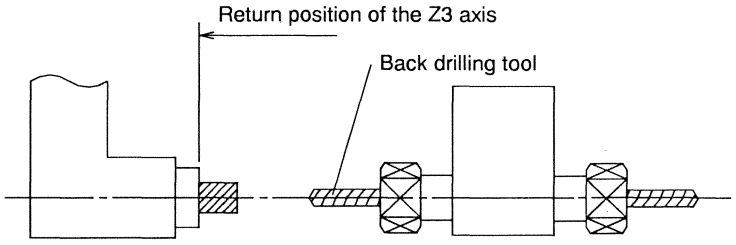
- The machine operates at the speed of 2 m/min or less if a door is open.
- The Manual Tool Set screen automatically closes if the subsequent operation is not performed for 15 seconds after the menu key [Man.Set] or [B/SP RET] is selected.

Adjusting the centers of tools on the front/back both end drilling tool

Adjust the center of a drilling tool on the front/back both end drilling tool. The center position has been adjusted during the manufacturing process of this machine. Use the tool center setting function to confirm the center position or to make fine adjustment.
First adjust the center position of the front drilling tool, and then adjust the center position of the back drilling tool.



Adjust fornt drilling tool



Adjust back drilling tool

This function sets the center of a drilling tool by aligning the center of tool with the center of the material or the workpiece. The fretting on the end face of the material or the workpiece is used for the center adjustment.

> ⚠ CAUTION
>
> Be sure to complete the longitudinal tool setting before starting tool setting for the center of a tool on the front/back both end drilling tool. Failure to do so may cause interference between the workpiece and tool, and the tool may be damaged.

Procedure

1. Adjust the center position of the front drilling tool.

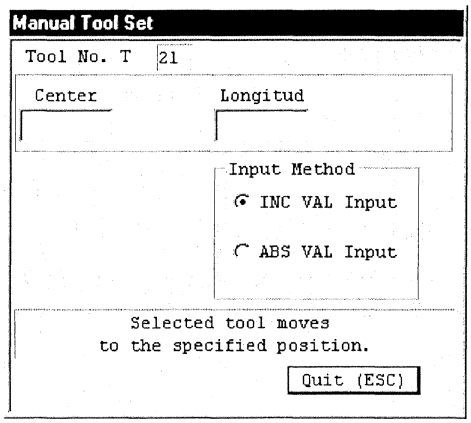
   Press PREPA-RATION. The PREPARATION key lamp lights, and the Preparation screen appears.

2. Select the tool to be set.

2-1 Confirm that the longitudinal adjustment of the tool is completed.

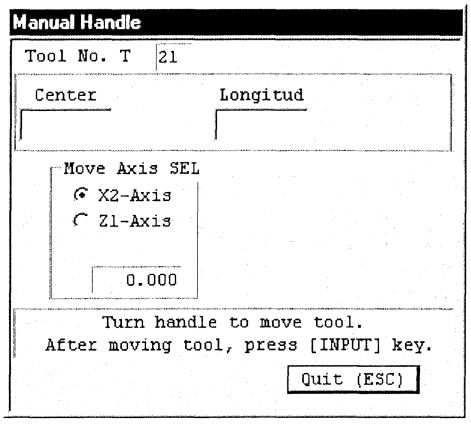
2-2. Press   or   to move the cursor to "Center" ("Core" or "DIA": type VI) of the tool you want to set.

2-3. Press the menu key [Man.Set]. The Manual Tool Set screen appears. Confirm that the cursor is positioned to "Center" ("Core" or "DIA": type VI). If the cursor is not there, press  or the TAB key to move the cursor to "Center" ("Core" or "DIA": type VI).



(This screen is for type VII, VIII, IX.)

2-4. Press  to start tool setting. The START key lamp lights. The X2 axis (X1, Y1 axes: type VI) moves to the selected tool position. The machine operates only while the START key is held down.

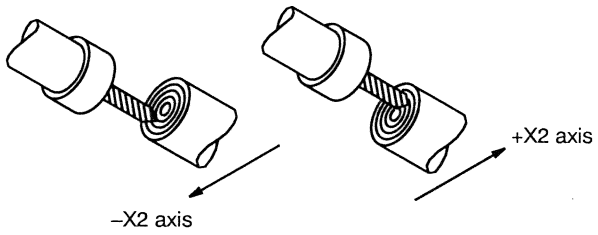


(This screen is for type VII, VIII, IX.)

3. Adjust the tool position until the center of the tool aligns with the center of the material.

   3-1. Pull the material without letting its end face touch the tool.

   3-2. Turn the handle to move the X2 axis (X, Y axes: type VI) until the center of the tool (tip of the tool nose) aligns with the center of fretting on the end face of the material.



   3-3. Press the INPUT key to enter the adjusted value. The machine sets the current X2 axis (X, Y axes: type VI) coordinate.

4. Adjust the center position of the back drilling tool.
   Set a workpiece into the back spindle.

   4-1. Stop the machine with [1 Cycle] in the automatic operation mode while the workpiece with which front machining is completed is chucked by the back spindle.

   4-2. Measure the workpiece protrusion length from the back spindle chuck, then compare the measured value with the value specified for "back spindle chuck POS" in the machining data. If the values are not identical, correct the program or machining data.

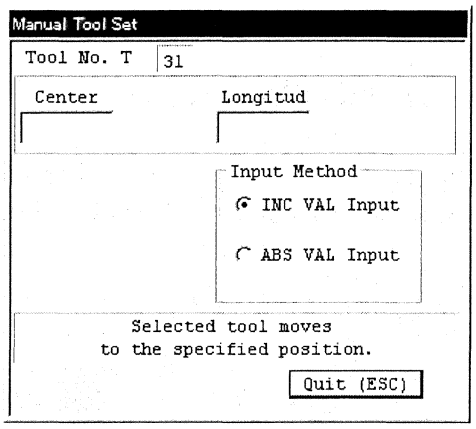
5. Press PREPA-RATION. The PREPARATION key lamp lights, and the Preparation screen appears.

6. Select the tool to be set.

   6-1 Confirm that the longitudinal adjustment of the tool is completed.

   6-2. Press ↑ ↓ or TAB ← TAB → to move the cursor to "Center" ("Core" or "DIA": type VI) of the tool you want to set.

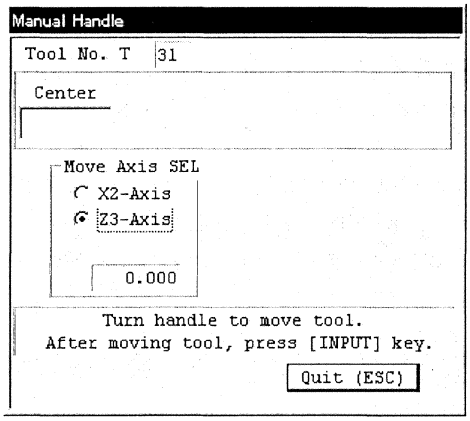
   6-3. Press the menu key [Man.Set]. The Manual Tool Set screen appears.
   Confirm that the cursor is positioned to "Center" ("Core" or "DIA": type VI). If the cursor is not there, press GRP or the TAB key to move the cursor to "Center" ("Core" or "DIA": type VI).



(This screen is for type VII, VIII, IX.)

6-4. Press 

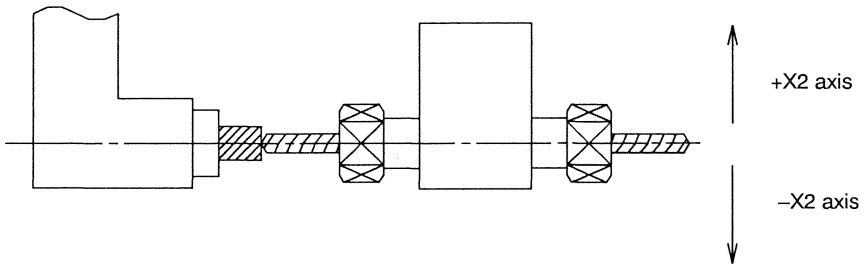
to start tool setting. The START key lamp lights. The Z3 axis moves to the return position, and the X2 axis (X1, Y1 axis: type VI) moves to the selected tool position. The Z3 axis remains at the return position. The machine operates only while the START key is held down.



(This screen is for type VII, VIII, IX.)

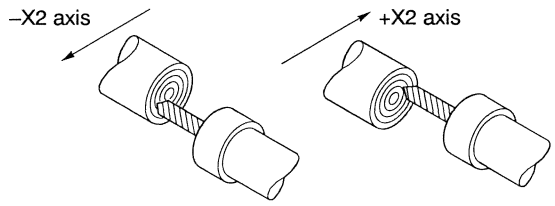
7. Adjust the tool position until the center of the tool aligns with the center of the workpiece.

   7-1. Press GRP and ↑ ↓ to move the cursor to the "Z3-Axis" in the "Move Axis SEL" area, press INPUT, then turn the handle to advance the Z3 axis to the position where the tip of the tool comes into contact with the end face of the workpiece.



   7-2. Press the GRP key and Up/Down Arrow key to move the cursor to the "X2 axis" (X, Y axes: type VI) in the "Move Axis SEL" area, then press the INPUT key.

   7-3. Turn the handle to move the X2 axis (X, Y axes: type VI) until the center of the tool (tip of the tool nose) aligns with the center of fretting on the end face of the workpiece.



   7-4. Press the INPUT key to enter the adjusted value. The machine sets the current X2 axis (X, Y axes: type VI) coordinate.

8. Press the menu key [B/SP RET], then press the START key to move the back spindle backward.

Notes

- The machine operates at the speed of 2 m/min or less if a door is open.
- The Manual Tool Set screen automatically closes if the subsequent operation is not performed for 15 seconds after the menu key [Man.Set] or [B/SP RET] is selected.
- The x100 key for adjustment with the handle cannot be used in the above adjustment.

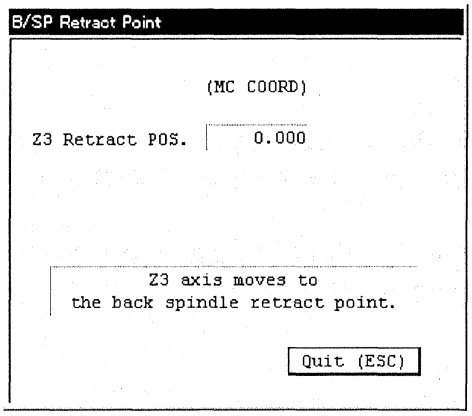
7.3.12 Moving the back spindle backward

Move the back spindle backward.

Procedure

1. Press PREPARATION  .
   The PREPARATION key lamp lights, and the Preparation screen appears.

2. Press the menu key [B/SP RET].
   The B/SP Retract Point screen appears.



3. Press START  .
   The START key lamp lights. The Z3 axis moves backward only while the START key is held down.

Notes

- The machine operates at the speed of 2 m/min or less if a door is open.
- The B/SP Retract Point screen automatically closes if the subsequent operation is not performed for 15 seconds after the menu key [B/SP RET] is selected.

7.3.13 Performing cut-off machining

> **WARNING**
>
> Do not put your hand into the machine during the cut-off process. Doing so could result in serious personal injury. When you open the door during the cut-off process, the feed function stops, but the main spindle continues rotating.

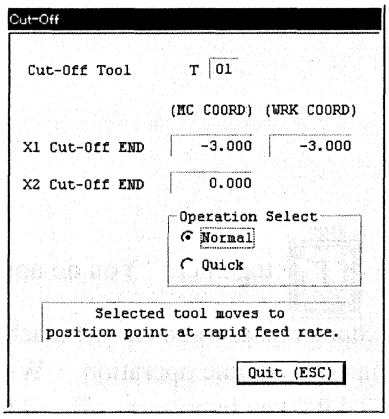
Use the values set in the machining data to perform cut-off machining or end-face turning.

Procedure

1. Press PREPARATION.
   The PREPARATION key lamp lights, and the Preparation screen appears.

2. Press the menu key [POS PNT] to select a tool for cut-off machining.
   See <Section 7.3.2 Positioning point>.

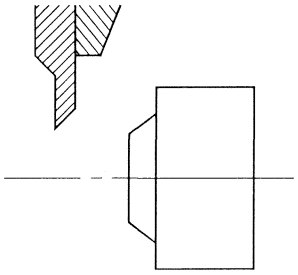
3. Press the menu key [Cut-Off].
   The Cut-Off screen appears.



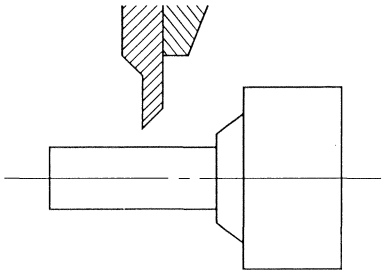
(This screen is for type VII.)

4. Confirm that the material does not protrude from the guide bushing, then press START.

   The START key lamp lights. The specified tool is moved to the positioning point only while the START key is held down.



5. Pull out the material by the cut-off length or the amount of end-face turning.



6. Press COOLANT.

7. Close all the doors.

8. Press the menu key [Cut-Off] and START together. You do not have to hold down the keys.

   The START key lamp lights, the chuck closes, then cut-off machining starts. The message "Executing cut-off" is displayed during machine operation. When cut-off machining is completed, the message disappears, and the START key lamp turns off. The main spindle stops with the chuck closed. Coolant is still discharging.

Note

- The machine does not start cut-off machining if a door is open.
  Note that when you open the door during the cut-off process, the feed function stops, but the spindle continues rotating. To restart the cut-off process, press the menu key [Cut-Off] and the START key together.
- When executing [CUT-OFF], the axes X1 and Y1 of the tool being selected moves to the positioning point, and then the axes X1 and Y1 of the specified tool moves to the positioning point. If you want to move the specified tool to the positioning point, first select "Quick" on "Operation Select" section of Positioning Point screen, and execute [CUT-OFF]. When "Quick" operation complete, the state returns to "Normal" (initial state).

7.4 On-machine Program Check Function

The on-machine program check function runs a program forward and backward on the machine to check if interference occurs in the machine.
You can correct a program error as soon as it is detected during execution of the on-machine program check function.
The on-machine program check function performs the following operations:

- 1-cycle operation
  The program is executed for one cycle in the automatic operation mode.

- 1-block operation
  The program is executed for one block and stopped each time the START key is pressed.

- 1-cycle handle operation
  The program is executed for one cycle in accordance with the amount that the handle turns.

- 1-block handle operation
  The program is executed for one block in accordance with the amount that the handle is turns.

You can switch between the above operations during execution of the on-machine program check function.
You can also perform the above handle operations by turning the handle backward.

7.4.1 Operating the on-machine program check function

WARNING

Do not put your hand into the machine during handle operation by the on-machine program check function. The machine does not operate in the handle operation unless you turn the handle. However, if the handle happens to turn, the machine operates, which could result in serious personal injury.

CAUTION

Be careful of the feed rate when machining a workpiece in actual machining.
The spindle rotates at the speed specified in the program, but the axis feed rate is determined by the speed that the handle turns. If the spindle speed does not match the axis feed rate, the workpiece or tool may be damaged because of the wrong machining conditions.

Procedure

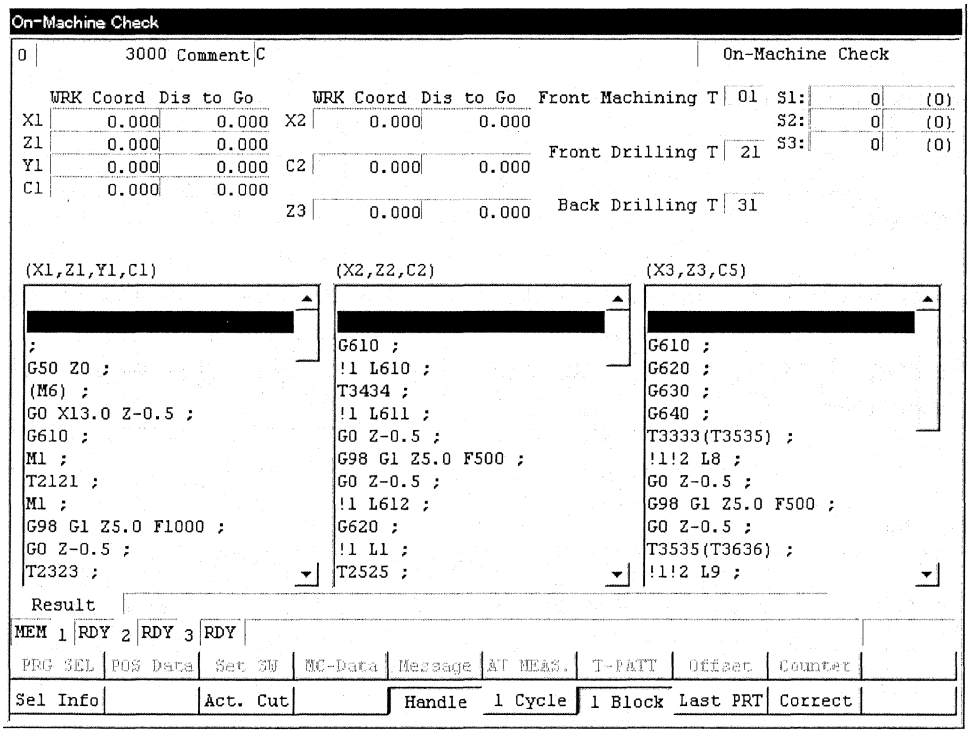
1. Press PREPARATION. The PREPARATION key lamp lights, and the Preparation screen appears.

2. Select the program you want to execute. See <Section 7.2.1 Selecting a program to be executed>.

3. Press the menu key [ST POS]. See <Section 7.3.1 Start position, return position, opposite tool post retract position (Type VII, VIII, IX), back spindle retract position and gang tool post return position>.

4. Press START. The START key lamp lights, and the machine performs the start position operation only while the START key is held down. The message "Moving to start position..." is displayed during operation. The START key lamp turns off at completion of operation, and the message changes to "Moving to start position completed".

5. Press PROGRAM CHECK. The PROGRAM CHECK key lamp lights, and the On-Machine Check screen appears.



(This screen is for type VII.)

6. Set operation with the menu keys.
   Select [1 Cycle] or [1 Block], then decide whether to perform handle operation with [Handle]. The [Handle] button is of toggle type. It is switched between ON and OFF each time it is pressed.

7. With handle operation, select a Handle Magnification key [x1], [x10], or [x100].
   The machine operates faster as a greater magnification is selected.

8. Press the START key. The START lamp key lights, and the program is executed.
   With handle operation, the machine operates when the handle is turned.

   - When the handle is turned clockwise: The program runs forward.
   - When the handle is turned counterclockwise: The program runs backward.

9. The START lamp key turns off at completion of operation.

Notes

- Some blocks cannot be run backward in the handle mode. An attempt to run the blocks backward results in an error. However, they can be run forward in the state.
  Blocks that cannot be run backward consist of M, T, or G code macros. Blocks of M and S codes other than the code macros can be run backward, but they are not executed.

- Handle operation is executed at the rate in proportion to the speed that the handle turns. The rate is limited to the feed rate specified in the program.

- You can use the on-machine program check function in the handle operation or automatic operation. The following steps can perform the thread cutting and tapping, however actual shape of the thread is not completed. You can make actual thread cutting and tapping only in the automatic operation.
  1. Stop the machine before the block of thread cutting.
  2. Press [Handle] once to cancel the handle operation, then select [1 Block].
  3. Press START. The block of thread cutting is executed with override 100%.
  4. Select [Handle] again, then select [1 Cycle].
  5. Press the START key, and turn the handle to operate the machine at the handle rotation.

- Chamfering of the thread cutting is not available in the on-machine program check function.

- You can use the on-machine program check function with machining programs that complete one cycle within 120 minutes when the override is 100%.

- The START key is invalid while a machine door is open.

7.4.2 Editing a program being executed

This edit function corrects a program being executed by the on-machine program check function in the automatic operation. Use the edit function if a program error is likely to cause interference.
For the sake of safety, do not machine a workpiece when using the edit function.

Procedure

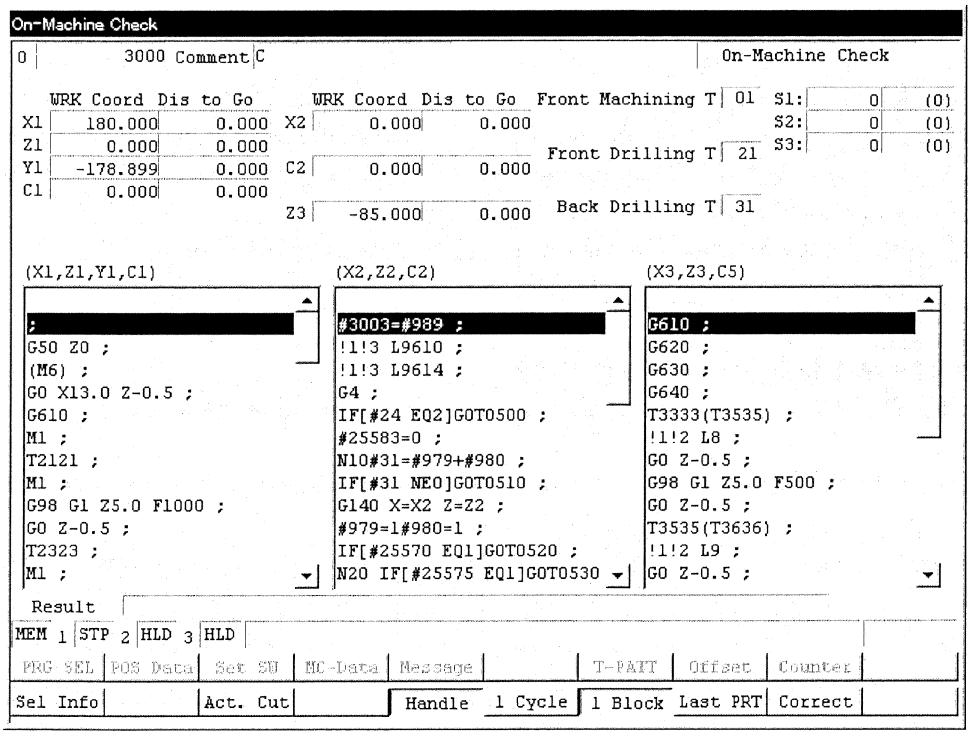
1. Perform operation with the on-machine program check function. See <Section 7.4.1 Operating the on-machine program check function>.
   Execute the on-machine program check function with [1 Cycle] selected.

2. Press the menu key [1 Block].

   When a block is executed, the program stops, and the 

key lamp turns off.

   If one of the programs of three axis control groups enters the "block stop" state, the programs of the other two axis control groups enter the "hold" state during execution.
   "STP" is displayed as the operation status of the axis control group in the block stop state. "HLD" is displayed for the two axis control groups in the hold state. The cursors of the two groups are reversed just as when their programs are being executed. Be sure to place the axis control group program (which you want to edit) into the block stop state.

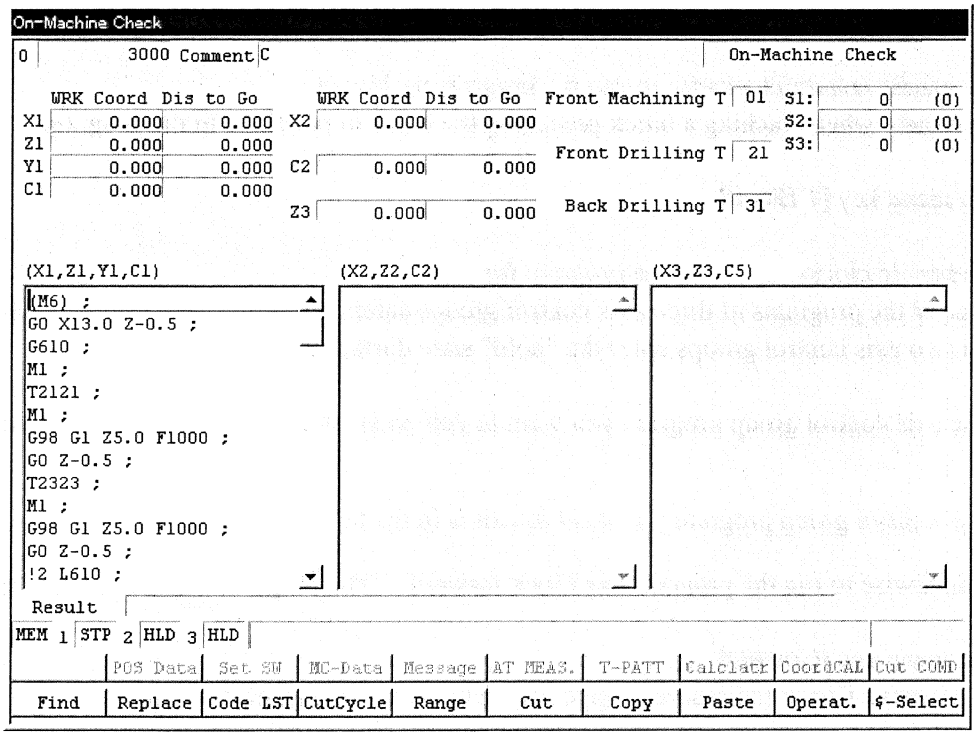


(This screen is for type VII.)

3. When the axis control group program you want to edit enters the block stop state, do the operation in step 4.

   If the axis control group program you want to edit is in the hold state, press START  again to run the program one block forward. The program enters the block stop state.

4. Press the menu key [Correct].
   The On-Machine Check (edit) screen appears. The menu keys change.



(This screen is for type VII.)

5. Edit the program. You can edit only the axis control group program for which "STP" (block stop state) is displayed at the lower left of the screen.

6. Press the menu key [Operat.] at completion of editing.
   The On-Machine Check screen appears.

7. Press START  . The START key lamp lights, and program execution is restarted.

7.4.3 Editing an executed program by backward running

Use this edit function when you want to change an executed block in a program. To edit the block, operate the machining program backward to a block preceding the block.

Procedure

1. Perform operation with the on-machine program check function. See <Section 7.4.1 Operating the on-machine program check function>.
   Execute the on-machine program check function with [Handle] and [1 Cycle] selected.

2. Turn the handle counterclockwise to run the program backward.
   Stop the handle when reaching a block preceding the block to be edited in the program.

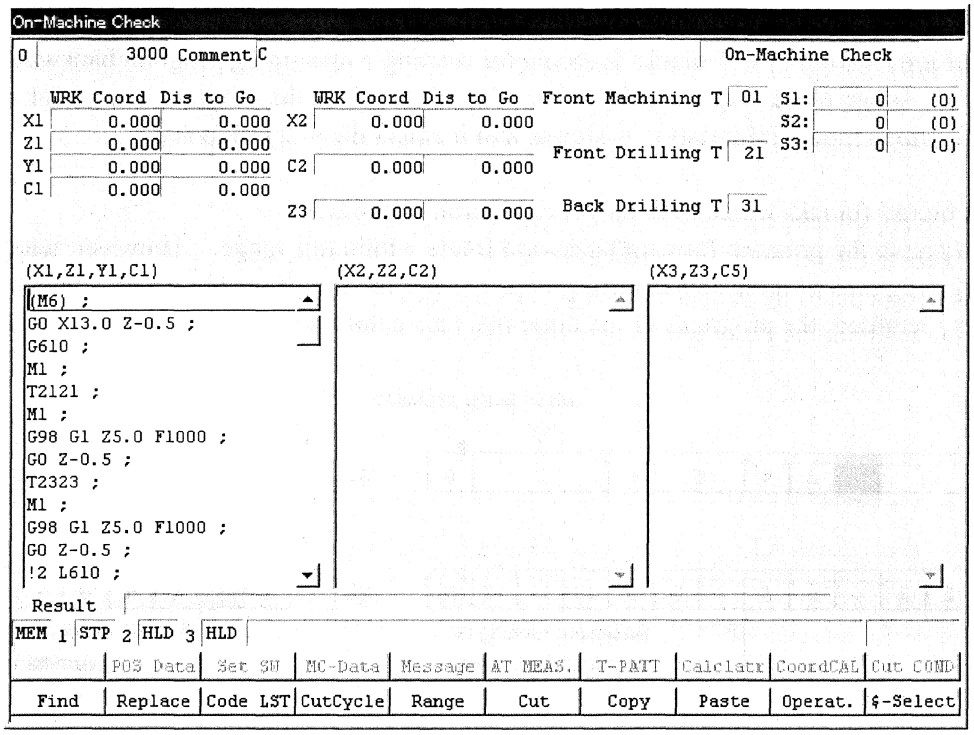
3. Press the menu key [1 Block].

4. Turn the handle clockwise to run the program forward.
   When one of the programs of three axis control groups enters the "block stop" state, the programs of the other two axis control groups enter the "hold" state during execution.

5. When the axis control group program you want to edit enters the block stop state, do the operation in step 6.
   If the axis control group program you want to edit is in the hold state, press 

   again, then turn the handle clockwise to run the program one block forward. The program enters the block stop state.

6. Press the menu key [Correct].
   The On-Machine Check (edit) screen appears. The menu keys change.



(This screen is for type VII.)

7. Edit the program. You can edit only the axis control group program for which "STP" (block stop state) is displayed at the lower left of the screen.

8. Press the menu key [Operat.] at completion of editing.
   The On-Machine Check screen appears.

9. Press 

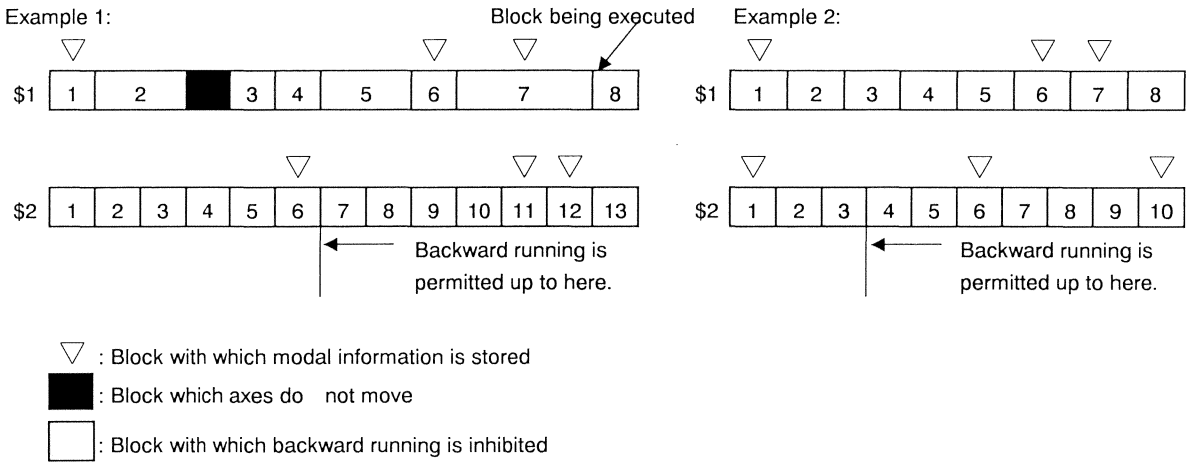
. Turn the handle clockwise.

   The START key lamp lights, and program execution is restarted.

Notes

- The modal information of five blocks is stored for running a machining program backward (backward execution). When block stop in backward running is specified, the program is run backward to the block with which modal information is stored, and it enters the block stop state.

- Up to ten blocks (blocks which axes move) can be run backward.
  You can execute the program forward/backward freely within this range. However, when one of the machining programs of three axis control groups reaches the last of the blocks with which backward running is permitted, the programs of the other two axis control groups can no longer run backward.



- When there is a block with which backward running is inhibited, storing of modal information starts every five blocks from the block.

- The AXIS.M. lamp flashes during backward running. When the program reaches the last of the blocks with which backward running is permitted, backward running is stopped, and the AXIS.M. lamp turns off.

> ⚠ WARNING
>
> Do not put your hand into the machine while the AXIS.M. lamp is flashing. Doing so could result in serious personal injury. The machine appears to be in the stopped state, but it is in the operation mode for backward running.

- When you run a program backward to edit it or operate 1-block operation, the three axis control groups may become asynchronous with each other. To maintain the synchronous status, make the final program check by running the program forward or executing automatic operation.

- The START key is invalid while a machine door is open.

7.5 Automatic Operation

7.5.1 Automatic operation

Continuous cycle operation, 1-cycle operation, and 1-block operation are provided as functions for executing programs selected for machining.
When the automatic operation mode is active, you can switch between the continuous operation, 1-cycle operation, and 1-block operation even during program execution. For example, when executing a program in the continuous cycle mode, you can switch to the 1-cycle mode and stop the operation at completion of 1-cycle execution.

Continuous cycle operation

Automatic operation is executed in the continuous cycle mode.
The machine stops when the number of machined workpieces counted by the counter becomes equal to the required number of workpieces.
See <Section 7.5.7 Setting counter>.

One cycle operation

Automatic operation is executed in the 1-cycle mode.

One block operation

When 

is pressed, automatic operation is stopped after 1-block execution. When the START key is pressed again, another block is executed.

Procedure

1. Press PREPARATION  . The PREPARATION key lamp lights, and the Preparation screen appears.

2. Select a program you want to execute. See <Section 7.2.1 Selecting a program to be executed>.

3. Press the menu [ST POS]. See <Section 7.3.1 Start position, return position, opposite tool post retract position (Type VII, VIII, IX), back spindle retract position and gang tool post return position>.

4. Press START  . The START key lamp lights, and the machine performs the start position operation only while the START key is held down. The START key lamp turns off at completion of operation.

5. Press AUTO  . The AUTO key lamp lights, and the Automatic Operation screen appears.

6. Select the menu key [Cont.], [1 Cycle], or [1 Block].

7. Press START  . The START key lamp lights, and the program is executed.
   If 1-block operation is specified, the next block is executed when the START key is pressed again at completion of operation.

Notes

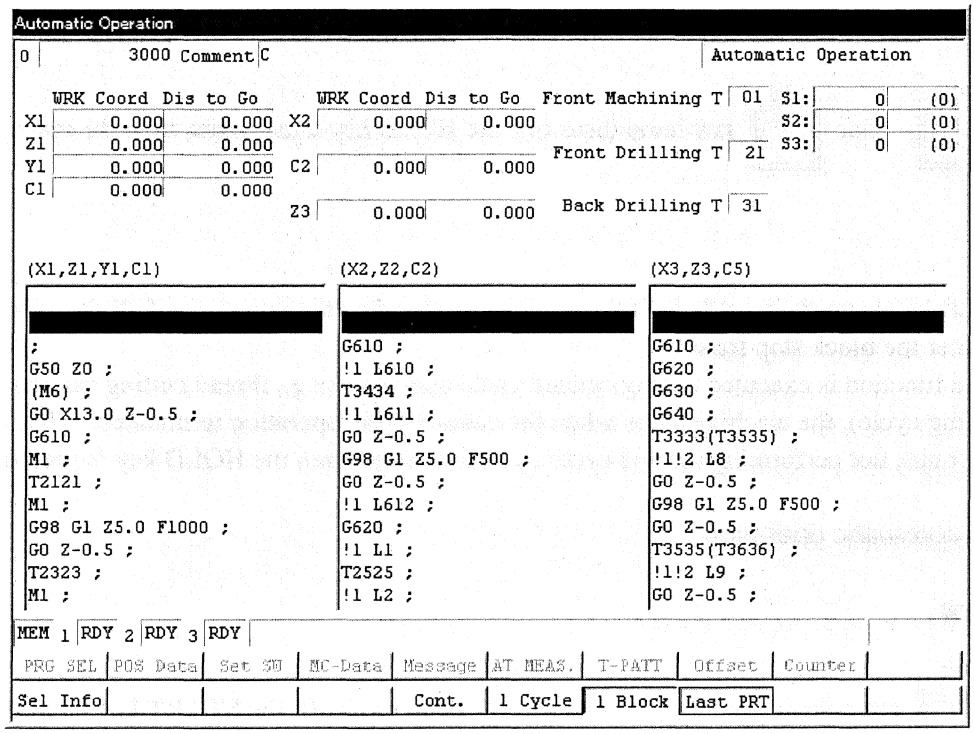
- When a program is selected, the corresponding machining data is selected at the same time. Check if the machining data is correct after selecting the program.
- The START key is invalid while a machine door is open.
- 1-block operation may be disabled depending on the G code.

7.5.2 Last part function

When [Last PRT] is selected, the last part function executes the last program to terminate operation. The last program machines the workpiece chucked by the back spindle.
As a result, the operation can terminate with no workpiece left in the machine.
If 1-cycle operation is selected, the last part function executes the last program for the workpiece. If continuous cycle operation is selected, the last part function executes the last program for the last of the required number of workpieces set to the counter.

Procedure

1. Press the menu key [Last PRT] on the Automatic Operation screen.



(This screen is for type VII.)

Note

See <Section 5.7.5 Last program execution (G999)>.

7.5.3 Holding and restarting automatic operation

Use the hold function to hold the current program. You can restart the program.

> ⚠ WARNING
>
> Do not put your hand into the machine during automatic operation. Doing so could result in serious personal injury. Even while the automatic operation in the hold state, the spindle is rotating and canned cycle operation is in progress.

Holding automatic operation

Procedure

1. Press 

. The 

key lamp turns off, the HOLD key lamp lights, then the machine stops.

Notes

- The HOLD key lamp does not light if the machine stops when the programs of all the axis control groups enter the block stop state.
- If the hold function is executed during canned cycle operation (e.g., thread cutting canned cycle or deep hole drilling cycle), the machine stops when the canned cycle operation terminates. The other axis control groups, not performing canned cycle operation, stop when the HOLD key is pressed.

Restarting automatic operation

Procedure

1. Press 

after closing the door. The 

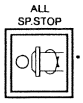
lamp turns off, the START key lamp lights, then program execution is restarted.

7.5.4 Stopping and restarting spindles during automatic operation

Use the stop function to stop the current program and spindles. You can restart the spindles.

> **WARNING**
>
> Do not put your hand into the machine during automatic operation. Doing so could result in serious personal injury. Even while the automatic operation is in the hold state, spindle indexing, C axis cancel, or canned cycle operation is in progress.

Stopping spindles

1. Press 

. The 

key lamp turns off, the 

key lamp lights, the machine stops, then all the spindles stop and the 

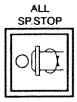
lamp flashes.

Notes

- The HOLD key lamp does not light if the machine stops when the programs of all the axis control groups enter the block stop state.
- If the stop function is executed during canned cycle operation (e.g., thread cutting canned cycle or deep hole drilling cycle), the machine and spindles stop when the canned cycle operation terminates. The other axis control groups, not performing canned cycle operation, stop when the SP.STOP key is pressed.
- The SP.START key lamp flashes when any of the spindles is stopped by pressing the SP.STOP key.
- Spindle indexing and C axis are not released.

Restarting spindles

1. While the 

lamp is flashing, close the door, then press the SP.START key. The SP.START key lamp lights, and the spindles (that were rotating immediately before 

was pressed) restart rotating.

Notes

- The SP.START key is enabled only during operation in the automatic operation mode or on-machine program check mode.
- The SP.START key is enabled only while the key lamp is flashing.
- If the SP.START key lamp is flashing, program execution can be restarted only after the spindles restart rotating.

7.5.5 Block skip function

The block skip function skips blocks with a slash (/) prefixed during program execution. To use the function, specify block skip ON.

Procedure

1. Press 

.

   The B.SKIP key lamp lights, and the block skip function is enabled (ON).

7.5.6 Override

The same override (percentage) is valid for the feed rate, rapid feed rate, and dwell time.

Procedure

1. Set the feed rate override dial to the graduation of a desired rate.
   100% · · · · · The machine operates at the feed rate specified by the program.
   0% · · · · · · · The machine does not operate.
   200% · · · · · The machine operates twice as fast as the feed rate specified by the program.

Note

If the feed rate override dial is set to 100% or more for machining that is performed by more than one axis control group, the maximum feed rate is exceeded while the axes are fed at the rapid feed rate. The speed clamp may be applied in this case. Consequently, machining cannot be carried out at the specified timing, and the machine fails to produce products as specified in the machining program. To prevent this problem, the feed rate override dial is enabled up to 100%. Even when you set the dial to a value 100 to 200%, the override is limited to 100%.
For machining free from the speed clamp, turn on "15 Override No Limit" switch on the Set SW screen. See <Section 6.10 Set SW screen> and <Section 7.5.8 Setting switches>. When the feed rate override dial is enabled up to 200%, override 200% is permitted for only the cutting feed rate. However, override over 100% is always limited to 100% for the rapid feed rate.

7.5.7 Setting counter

The Counter screen is provided for displaying the execution time of a program in the continuous automatic operation mode or for entering the number of workpieces to be machined.

- Required
  When the machined number of workpieces becomes equal to the required number of workpieces, an alarm is issued and the machine stops. Press RST to recover the machine. If A.P.OFF has been pressed, the machine stops and the power is turned off.
  Up to 99,999,999 can be set as the required number of workpieces.
  If "0" is set, the counter does not count machined parts.

- Machined
  The machined number of workpieces is incremented by one each time the M56 command is executed at completion of machining a workpiece.
  When the machined number of workpieces becomes equal to the required number of workpieces, an alarm is issued and the machine stops. Pressing the RST key clears only the machined number of workpieces to "0".

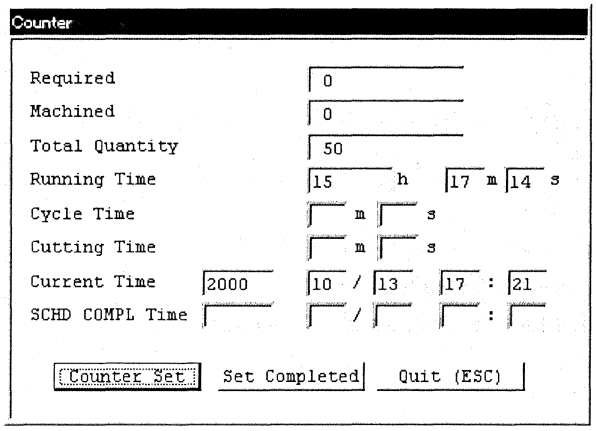
- Total Quantity
  The total quantity of machined workpieces is displayed. The total quantity is incremented by one each time the M56 command is executed.
  When the total quantity of machined workpieces reaches the maximum value 99,999,999, it is cleared to "0".

Procedure

1. Press AUTO.
   The Automatic Operation Screen appears.

2. Press ▷ (Menu Up/Down Selection key).
   The submenus become effective.

3. Press the menu key [Counter].
   The Counter screen appears.



4. The cursor is initially positioned to the [Counter Set] button.

5. Press INPUT.
   The input fields on the Counter screen turns white, which indicate that you can set values into the fields.

6. Press ↑ ↓ ← → TAB and/or TAB to move the cursor to the input field into which you want to enter a value.

7. Press the Numeric keys to enter the value.

8. Press INPUT to confirm the entered value.

9. Press GRP to move the cursor to the [Set Completed] button.

10. Press INPUT.
    Setting on the Counter screen is now completed.

Note

Press A.P.OFF to enable the automatic power off function.

If an alarm is generated during execution of a program in the continuous cycle, the power will be automatically turned off.

7.5.8 Setting switches

Switches are provided with various control signals for NC operation. You can set the switches to on or off.

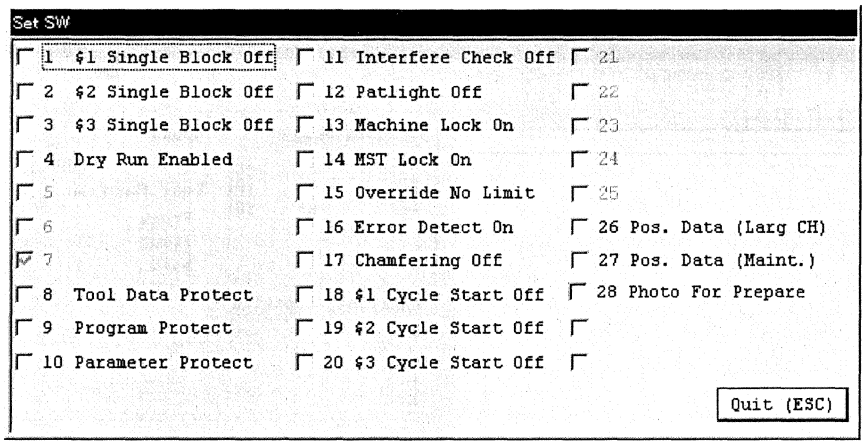
Procedure

1. Press [▷] (Menu Up/Down Selection key).
   The submenus become effective.

2. Press the menu key [Set SW].
   The Set SW screen appears.

3. Press [↑] [↓] [TAB ←] and/or [TAB →] to move the cursor to the item you want to set.

4. Press [INPUT →◇].
   If the selected item is on, it is set to off (unchecked).
   If the selected item is off, it is set to on (checked).



7.5.9 MDI operation

MDI stands for Manual Data Input. Words (commands) are usually expressed as programs before they are given to the machine. However, you can input words directly into the machine. This input method is called MDI operation.
The MDI operation has two execution patterns "1 block" and "all blocks".

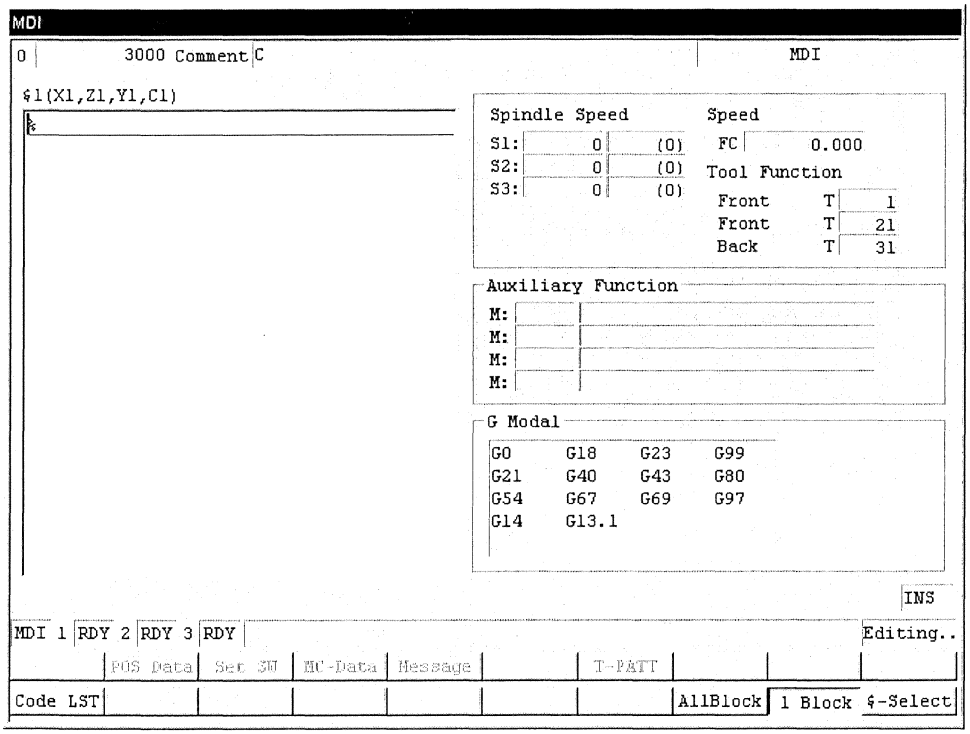
- 1 block: Words can be executed for each block.
- All blocks: The words of all the blocks can be executed at a time.

Procedure

1. Press MDI. The MDI key lamp lights and the MDI screen appears.

2. Press the menu key [$-Select] to select the axis control group for MDI operation.
   Axis control groups are switched $2, $3, then back to $1 each time [$-Select] is pressed.

3. Enter the program (word), then press INPUT. The message "Set COMP" is displayed.



4. Press the menu key [1 Block] or [AllBlock].

5. Press 

. The START key lamp lights and the entered program (word) is executed.

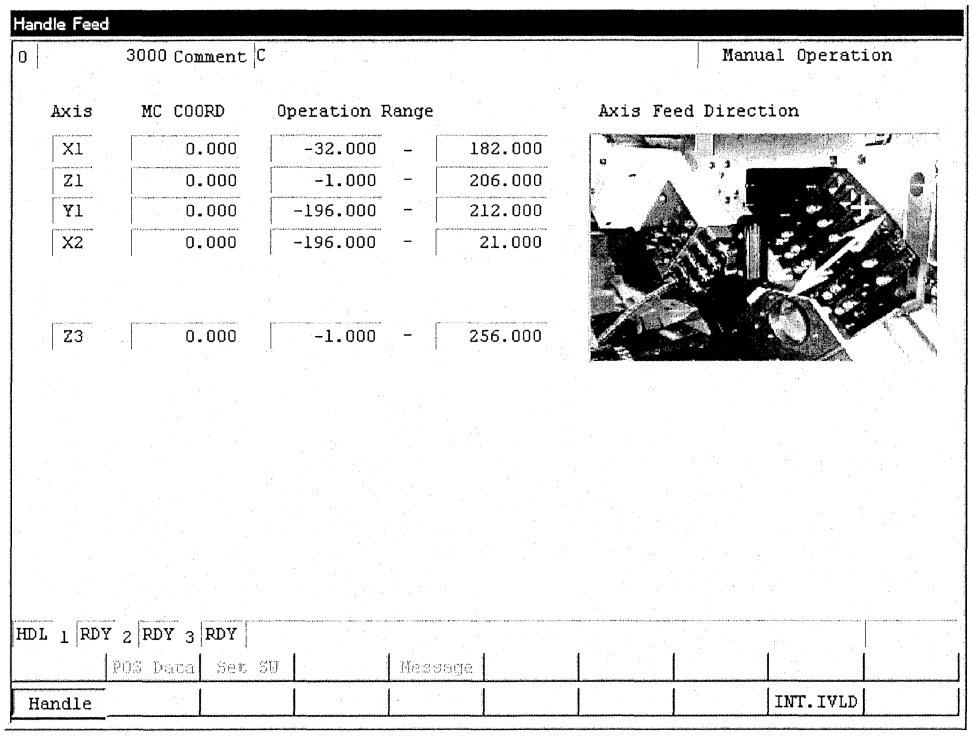
   The message "Executing..." is displayed during program execution.
   The START key lamp turns off at completion of program execution.
   If [1 Block] is selected, the next block is executed when the START key is pressed again.

Notes

- When selecting [1 Block], you have to press the START key for T codes, M code macros, and G code macros several times before completion of 1-block execution.
- The START key is invalid while a machine door is open.
- The MDI function is enabled for operation with only the selected axis control group. It is unable to do operation extending over multiple axis control groups.

7.6 Manual Operation

The handle (manual pulse generator) is provided for manual operation. You can move a selected control axis to an arbitrary position at an arbitrary feed rate by turning the handle.
The selected control axis can be moved into three directions: the direction in which the selected tool cuts into the workpiece (X1, X2), the direction of the core height (Y1), and the longitudinal direction (Z1, Z3).
The Handle Feed screen displays the stroke range and the move direction on the right of each control axis. Manual operation can also be performed on the Mechanism Adjustment screen.



(This screen is for type VII.)

Procedure

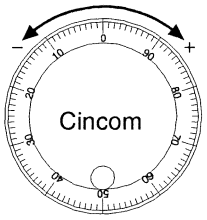
1. Press MANUAL [ ]. The MANUAL key lamp lights, and the previously selected screen appears.

2. If another screen appears, press the menu key [Handle].
   The Handle Feed screen appears.

3. Press [↑] and/or [↓] to select the control axis you want to operate.
   The name of the selected control axis is reversed in yellow.

4. Press a Handle Magnification key [x1], [x10], or [x100].
   The Handle Magnification key lamp lights.
   The machine operates faster as a greater magnification is selected.

5. Turn the handle.
   The direction for moving the axis is determined by the direction into which you turn the handle.
   Overturning the handle to a value exceeding the OT (stored stroke limit) will display the message "M01 Operation Error 0007" to indicate an overrun alarm.
   See <Section 7.11.2 Resetting an overrun alarm>.



[INT.IVLD] menu key

If "M04 COLLISION AREA" alarm is issued, pressing this menu key temporarily disables the checking of interference. Meanwhile, you can move away the axis from the interference area by operating the handle.
(This menu key operates linked with " 11 Interfere Check Off" on the Set SW screen.)

7.7 Creating and Editing Programs

This machine achieves various types of machining with five axes (X1, Y1, Z1, X2, and Z3), which are divided into three axis control groups. To perform various types of machining, create a program for each axis control group, and execute the programs at the same time.
Assign a program number to the overall program of three axis control groups, and consider the programs created for the axis control groups to be one program when performing various operations for the programs (e.g., saving, transferring, deleting, and copying data).
The Edit screen for program displays the programs of two axis control groups or three axis control groups at the same time.

7.7.1 Displaying a program list

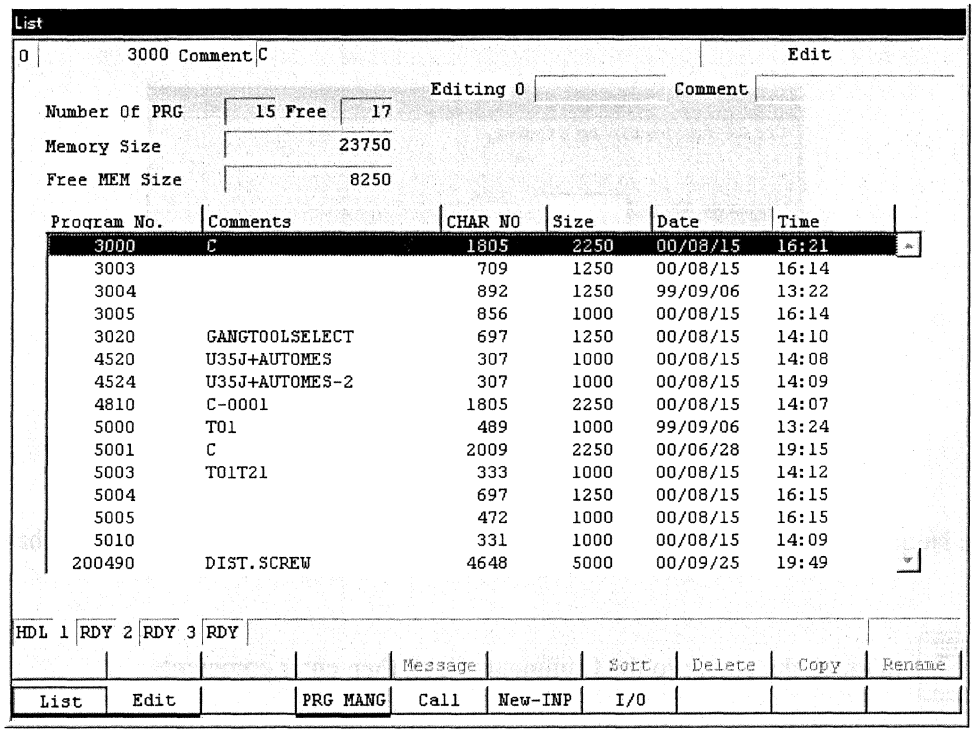
The List screen displays the program numbers, comments, sizes, and creation dates/time of registered programs.
Use the List screen to create a new program, call a program, or input/output a program.

Procedure

1. Press 

.
   The previously selected screen appears.

2. Press the menu key [List].
   The List screen appears.



Note

Programs are initially listed in the ascending order of program numbers.
Each program number must be unique. Take a look at the list, and assign a non-existing program number to the program to be newly created.

7.7.2 Creating a new program

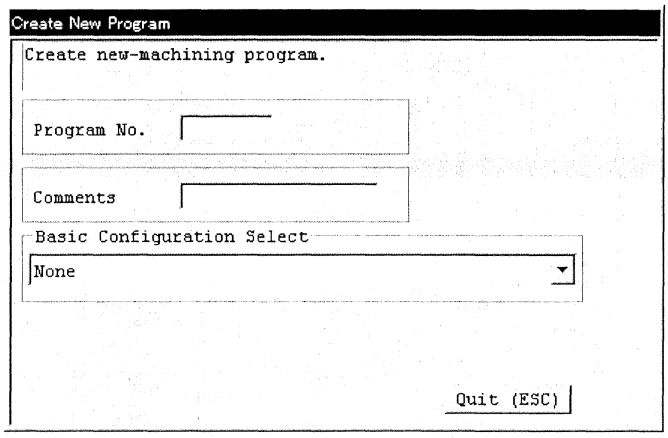
This section explains the procedure for creating a new program.

Procedure

1. Press EDIT.
   The previously selected screen appears.

2. Press the menu key [List].
   The List screen appears. Take a look at the list, and choose a non-existing program number as a new program number.

3. Press the menu key [New-INP].
   The Create New Program screen appears.



4. Press the Numeric keys to enter the new program number. The program number can have up to eight digits.

5. Press GRP to move the cursor to the Comments field, then enter comments.
   You can omit comments.

6. Press GRP to move the cursor to the Basic Configuration Select field, then select a basic Configuration.

7. Press INPUT.
   The new program number is registered, and the Edit screen for the newly registered program appears.

Note

A program number can have up to eight digits. Note that you cannot use the following numbers because they are used with the custom programs.

Nos. 8000 to 8999: For user
Nos. 9000 to 9999: For machine manufacturer

7.7.3 Editing a program

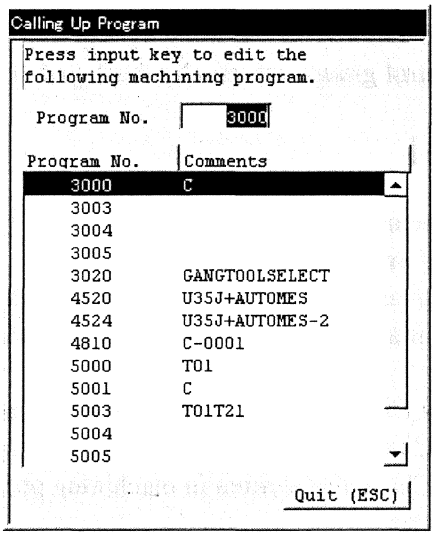
This section explains the procedure for calling a program to the edit area.

Procedure

1. Press EDIT.
   The previously selected screen appears.

2. Press the menu key [List].
   The List screen appears. Make sure of the program number of the program you want to edit.

3. Press PAGE, , , and/or to select the program number. Alternatively, press the menu key [Call] to display the Calling Up Program screen, then press the Numeric keys to enter the program number.



4. Press INPUT.
   The selected program is called. You can now edit the program.

Selecting an axis control group for editing a program

Take the following procedure to select the target axis control group when editing a program:

Procedure

1. Select the Edit screen. See <Section 7.7.3 Editing a program>.
   The programs of three axis control groups are usually displayed at the same time.

2. Press the menu key [$-Select].
   The cursor moves to $1, $2, $3, then back to $1 each time [$-Select] is pressed.

Selecting display for editing a program

Take the following procedure to select the target display axis control group and the display size when editing a program:

Procedure

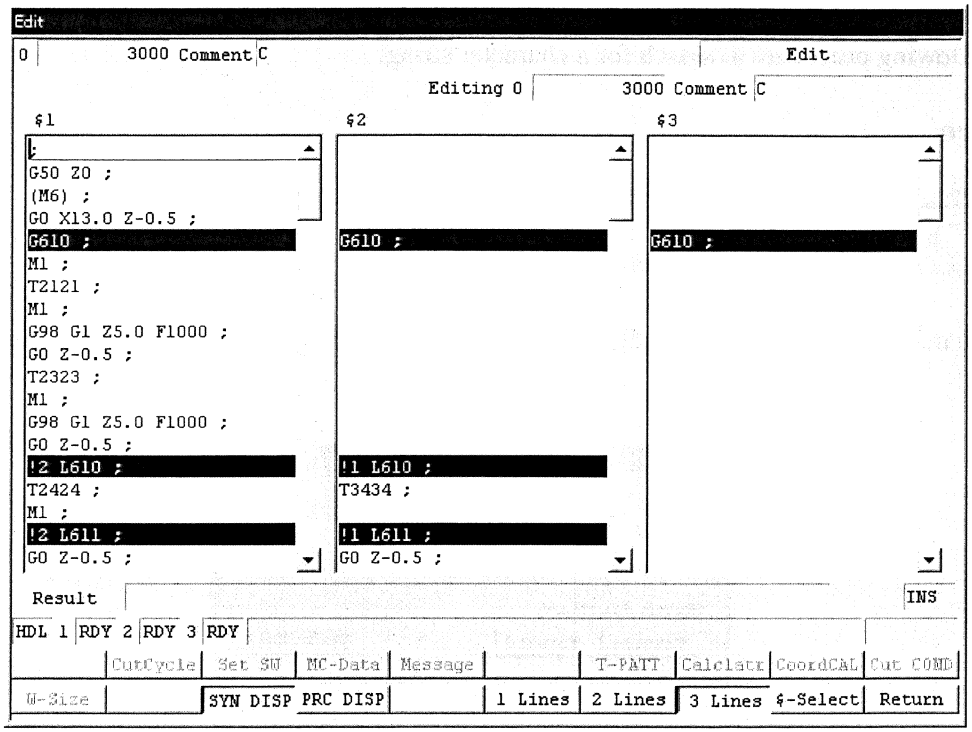
1. Select the Edit screen. See <Section 7.7.3 Editing a program>.
   The programs of three axis control groups are usually displayed at the same time.

2. Press the menu key [DISP SEL].

3. Select a display method with the menu key.
   - [1 Line] The program of axis control group 1 is displayed.
   - [2 Lines] The programs of axis control groups 1 and 2 are displayed at the same time.
   - [3 Lines] The programs of axis control groups 1, 2, and 3 are displayed at the same time.

4. Press the menu key [Character Size], and select the display character size.
   - [W-Size] ON: Characters written in machining programs are displayed enlarged.
   - [W-Size] OFF: Characters written in machining programs are displayed in standard size.

   It is not permitted to set [W-Size] to ON when [SYN DISP] or [PRC DISP] is ON. If [SYN DISP] or [PRC DISP] is set to ON while [W-Size] is ON, [W-Size] is automatically set to OFF and characters are displayed in standard size.


Edit
0 3000 Comment C
Edit
Editing 0 3000 Comment C
$1
$2
$3
G50 Z0 ;
(M6) ;
G0 X13.0 Z-0.5 ;
G610 ;
M1 ;
T2121 ;
M1 ;
G98 G1 Z5.0 F1000 ;
G0 Z-0.5 ;
T2323 ;
M1 ;
G98 G1 Z5.0 F1000 ;
G0 Z-0.5 ;
!2 L610 ;
T2424 ;
M1 ;
!2 L611 ;
G0 Z-0.5 ;
G610 ;
!1 L610 ;
T3434 ;
!1 L611 ;
G0 Z-0.5 ;
G610 ;
Result
INS
HDL 1 RDY 2 RDY 3 RDY
CutCycle Set SW MC-Data Message T-PATT Calclatr CoordCAL Cut COND
W-Size SYN DISP PRC DISP 1 Lines 2 Lines 3 Lines $-Select Return

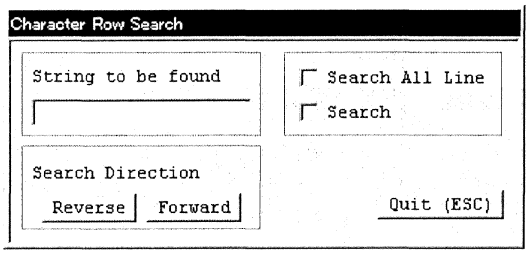
Find

Take the following procedure to search for a character string:

Procedure

1. Press EDIT. The previously selected screen appears.

2. Press the menu keys [Edit], [Expand], and [Find] in this order. The Character Row Search screen appears.



3. Enter the character string (you want to find) into the "String to be found" text box.

4. Press INPUT. The cursor moves to [Forward] in the Search Direction field.

5. Press ← and/or → to select the search direction, then press INPUT. The find function is executed.

   To search the programs of all the axis control groups for the character string, press GRP to move the cursor, press ↑ ↓ to select "Search All Line", then press INPUT.

6. Press ESC to close the Character Row Search screen at completion of operation.

Notes

- You cannot use the find function with the program you are editing. Use the find function with programs confirmed by pressing 

.

- Search options

  Reverse: The find function searches backward from the current cursor position, toward the beginning of the program.

  Forward : The find function searches forward from the current cursor position, toward the end of the program.

  Search All Line: The find function searches the entire program of axis control groups 1, 2, and 3.

  Search: The find function starts the first search from the cursor position.

- Press the menu key [Return] to terminate the Expanded Edit menu.

Replace

Take the following procedure to replace a character string. There are two replacement methods.

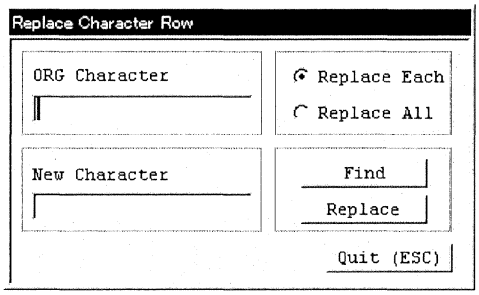
- Replace each:
  The replace function replaces found character strings one by one when instructed by the operator.

- Replace all:
  The replace function replaces all character strings found by the find function at a time.

Procedure

1. Press EDIT. The previously selected screen appears.

2. Press the menu keys [Edit], [Expand], and [Replace] in this order. The Replace Character Row screen appears.



3. Enter the character string to be replaced into the "ORG Character" text box.

4. Press GRP to move the cursor to the "New Character" text box, then enter a new character string.

5. Press GRP to move the cursor, press ↑ ↓ to select the replacement method "Replace Each" or "Replace All", then press INPUT.

6. To search the program downward from the cursor position, select [Find] and press INPUT.
   To replace the character string selected by the cursor, select [Replace] and press INPUT.
   To replace all the character strings found in the program of the axis control group program that is being edited, select [Execute] and press INPUT. The [Execute] button is displayed when "Replace All" is selected.

7. Press ESC to close the Replace Character Row screen at completion of operation.

Move and copy

Take the following procedure to move/copy specified character strings to another position in a program.

Procedure

1. Press 

. The previously selected screen appears.

2. Press the menu key [Edit], and then [Expand]. The Extended Edit menu appears.

3. Press  and/or  to move the cursor to the beginning of the character strings you want to move or copy.

4. Press the menu key [Range].
   The line on which the cursor exists is reversed.

5. Press  and/or  to move the cursor to the end of the character strings you want to move or copy.
   All the character strings in the selected range is reversed.

6. To move the character strings, press [Cut]. To copy the character strings, press [Copy]. The message "Editing..." is displayed at the lower right of the screen, and the specified character strings are saved.

7. Press  and/or  to move the cursor to the beginning of the character string to which you want to insert the saved character strings.

8. Press the menu key [Paste].
   The characters strings saved in step 6 are moved (inserted) or copied.

9. Press 

. The message "Editing..." disappears, and the moved or copied character strings are confirmed. If you switch to another screen before pressing 

, the copy processing is canceled, and the program is restored to the original status.

   Once you specify and save character strings to move or copy, you can copy them to different positions consecutively by repeating the operations in steps 7 to 9.

Cutting cycle

Taking the following procedure to enter setup items which will automatically create a cutting cycle program with multiple blocks.

Procedure

Example: Inserting G92 thread cutting canned cycle

1. Press EDIT. The previously selected screen appears.
2. Press the menu key [Edit]. The Edit screen appears.
3. Move the cursor to the position to which you want to insert a cutting cycle program.
4. Press ▷ (Menu Up/Down Selection key). The submenus become effective.
5. Press the menu key [CutCycle].
   The Cycle Selection screen appears.
6. Press ↑ and/or ↓ to select "G92 Thread Cutting Canned Cycle", then press INPUT.
   The G92 Thread Cutting Canned Cycle screen appears.
7. Press the Up/Down Arrow key to select the material, then press INPUT.
8. Press the Up/Down Arrow key to select the thread lead, then press INPUT.
9. Press the Numeric keys to enter the amount of tool shift, then press INPUT.
10. Press the Numeric keys to enter the thread diameter, then press INPUT.
11. Press the Numeric keys to enter the effective length of thread, then press INPUT.

12. Move the cursor to [Cycle INS] button, then press 

.

The message "Editing..." is displayed at the lower right of the screen, and the cutting cycle program is inserted to the position at the cursor.

13. Press 

. The message "Editing..." disappears, and the cutting cycle program is confirmed.

Note

If you switch to another screen before pressing 

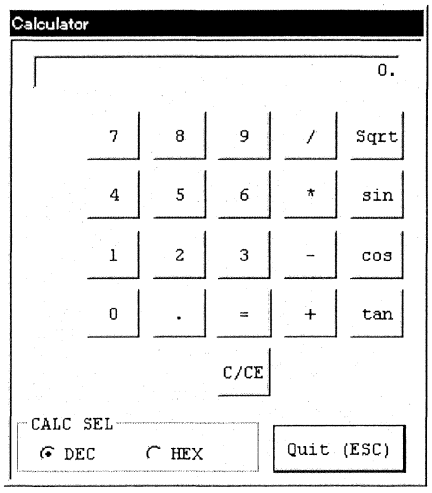
to confirm the inserted cutting cycle program, the cutting cycle insertion processing is canceled, and the program is restored to the original status.

Calculator

Take the following procedure for calculation with the displayed calculator:

Procedure

1. Press EDIT. The previously selected screen appears.
2. Press the menu key [Edit]. The Edit screen appears.
3. Press ▷ (Menu Up/Down Selection key). The submenus become effective.
4. Press the menu key [Calclatr]. The Calculator screen appears.



5. Press the Alphanumeric keys to enter the formula.
6. Press [=].
   The data is calculated.
   The calculation result is displayed in the display area of the Calculator screen. It is also displayed in the calculation result display area on the Edit screen.
7. To calculate the subsequent data, press CAN to erase the previous calculation result.
8. Press ESC to close the Calculator screen at completion of operation.

Note

The above explains decimal calculation. To do hexadecimal calculation, press → to select "HEX" in the CALC SEL field after the operation in step 4. To convert calculation results from decimal to hexadecimal value, press the Right Arrow key to select "HEX" after the operation in step 6.

Coordinate calculation

Take the following procedure to automatically calculate defined pattern coordinates:

Procedure

1. Press EDIT. The previously selected screen appears.
2. Press the menu key [Edit]. The Edit screen appears.
3. Press ▷ (Menu Up/Down Selection key). The submenus become effective.
4. Press the menu key [CoordCAL]. The Coordinate Calculation screen appears.
5. Press ↑, ↓, TAB ←, and/or TAB → to move the cursor to the pattern (drawing) you want to use.
6. Press INPUT.
   The selected pattern screen appears.
   "Right Triangle Calculation" is selected as the default.
7. Keep pressing INPUT until the desired drawing appears.
8. Press GRP to move the cursor to the data input field.
9. Press the Alphanumeric keys to enter the values for calculation.
10. Press GRP to move the cursor to "Calculation", then press INPUT.
    The calculation result is displayed.
    It is also displayed in the calculation result display area on the Edit screen.
11. Press INPUT twice to close the Coordinate Calculation screen at completion of operation.

Cutting condition calculation

Take the following procedure for cutting condition calculation on the basis of given data such as the material of workpiece. The calculation results are displayed as the reference values.

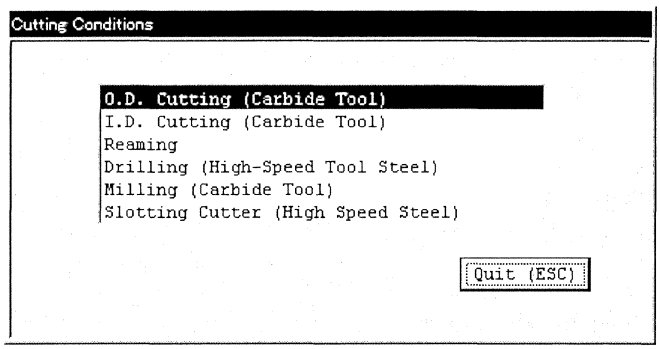
Procedure

1. Press EDIT. The previously selected screen appears.

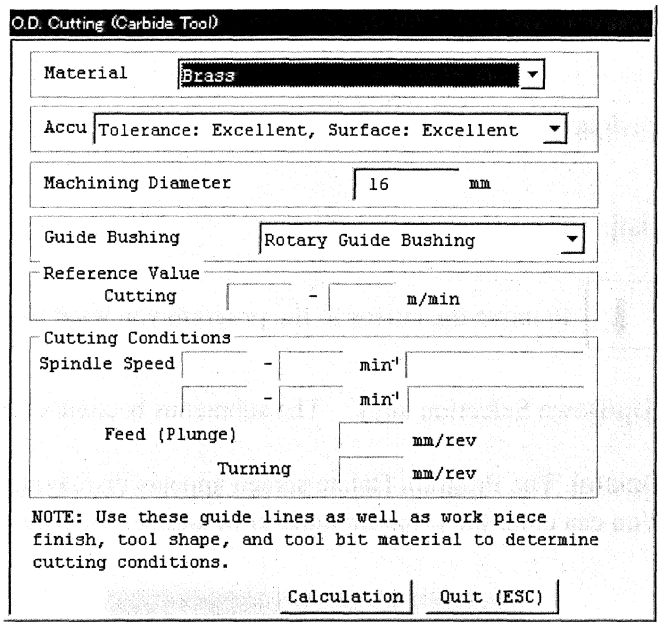
2. Press the menu key [Edit]. The Edit screen appears.

3. Press ▷ (Menu Up/Down Selection key). The submenus become effective.

4. Press the menu key [Cut COND]. The Cutting Conditions screen appears.



5. Press ↑ and/or ↓ to select the item, then press INPUT. The selected item screen appears.



6. Press ↑ ↓ and INPUT to set the required parameters.

7. If the cursor is not positioned to the [Calculation] button, press GRP to move the cursor to [Calculation], then press INPUT. The cutting condition calculation result is displayed as the reference value.

8. Press ESC twice to close the Cutting Conditions screen at completion of operation.

7.7.4 Deleting a program

This section explains the procedure for deleting a program.

Procedure

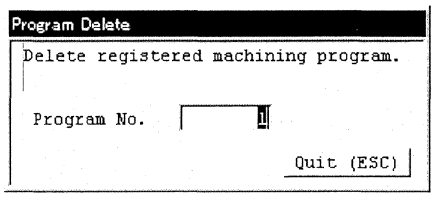
1. Press EDIT. The previously selected screen appears.

2. Press the menu key [List]. The List screen appears.

3. Press ↑ and/or ↓ to move the cursor to the program you want to delete.

4. Press ▷ (Menu Up/Down Selection key). The submenus become effective.

5. Press the menu key [Delete]. The Program Delete screen appears displaying the program number selected in Step 3. You can enter the program number of another program you want to delete.



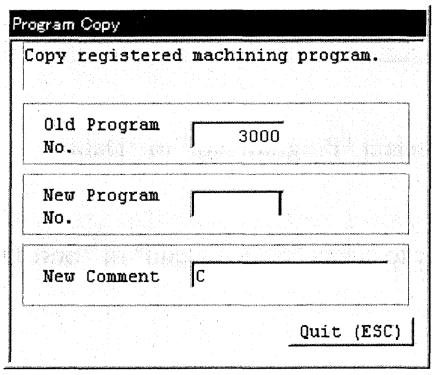
6. Press INPUT.
   The specified program is deleted.

7.7.5 Copying a program and changing a program number

This section explains the procedure for copying an existing program with a new number. When a program number is changed, the source program is deleted.

Procedure

1. Press EDIT. The previously selected screen appears.
2. Press the menu key [List]. The List screen appears.
3. Press ↑ and/or ↓ to move the cursor to the source program you want to copy or rename.
4. Press ▷ (Menu Up/Down Selection key). The submenus become effective.
5. To copy the program, press the menu key [Copy]. To change the program number, press the menu key [Rename].
   The screen as shown below appears.



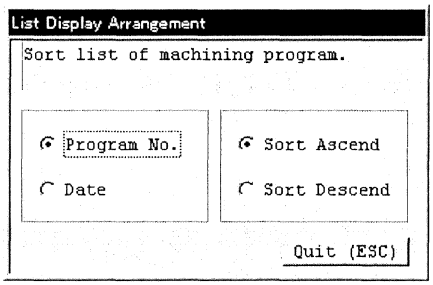
6. Enter the copy destination program number or the new program number.
7. Press GRP to move the cursor to the new comment field, then enter comments.
   You can omit comments.
   Do not include any parentheses in comments.
8. Press INPUT.
   The specified program is copied, or the program number is changed.

7.7.6 Sorting programs

This section explains the procedure for setting the order in which programs are listed.

Procedure

1. Press EDIT. The previously selected screen appears.
2. Press the menu key [List]. The List screen appears.
3. Press ▷ (Menu Up/Down Selection key). The submenus become effective.
4. Press the menu key [Sort]. The List Display Arrangement screen appears.



5. Press ↑ and/or ↓ to select "Program No." or "Date".
6. Press GRP to move the cursor to select "Sort Ascend" or "Sort Descend".
7. Press INPUT.
   The programs are sorted in the specified order.
8. Press ESC to close the List Display Arrangement screen at completion of operation.

7.7.7 Creating and editing machining data

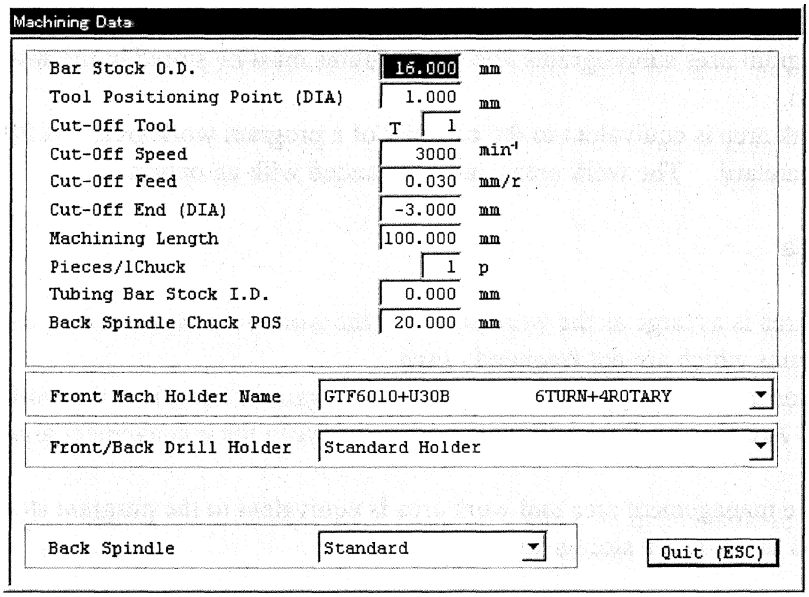
This section explains the procedure for creating and editing the machining data of a program.
You cannot edit the machining data of the program that is being operated.

Procedure

1. Call the program you want to edit. See <Section 7.7.3 Editing a program>.
   The Edit screen appears.

2. Press [▷] (Menu Up/Down Selection key). The submenus become effective.

3. Press the menu key [MC-Data].
   The Machining Data screen appears.



(This screen is for type VII, VIII, IX.)

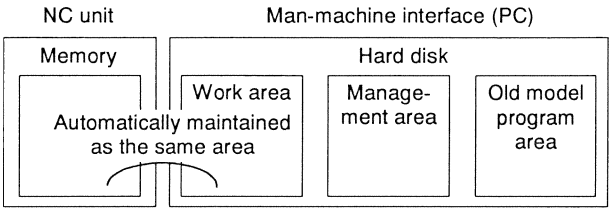
4. Press [↑] and/or [↓] to move the cursor to the item for which you want to enter a value or change the current value.

5. Press the Numeric keys to enter or change the numeric value.

6. Press [INPUT ⇨].
   The entered new value is set. The cursor moves to the next item. If you press the Up/Down Arrow key before the INPUT key, the entered value is set back to the original value.

7.8 Program Management

This machine stores programs in the areas as shown below.



Work area

The work area stores the programs that are actually operated. Only the programs in this work area can be selected on the Program Select screen.
The List screen displays the programs in the work area.
If a machining program uses subprograms, the subprograms must be stored in the work area as well as the machining program.
The size of the work area is equivalent to the capacity of a program work area. A 20 m tape is used with the work area as standard. The work area can be expanded with an option.

Management area

The management area is as large as the work area. If the work area becomes full, use the management area to store programs which are not frequently used.
To operate programs in the management area, you have to transfer them into the work area.
You can use [EXC Area] to exchange all the programs between the management area and work area at a time.
The total size of the management area and work area is equivalent to the program storage capacity. A 40 m tape is used with the program storage as standard.

Old model program area

The old model program area stores old Cincom model programs. Use the input/output screen to store old model programs into this area.
To operate programs in the old model program area, you have to copy the desired program into the work area. When the program is copied into the work area, only the machining data is automatically converted into the format for this machine. Since the program conforms to the old model specifications, it cannot be operated as is. The program needs to be modified for this machine.

Note

You cannot use [EXC Area] to exchange all the programs between the old model program area and work area at a time.
Copy the programs from the old model program area into the work area one by one.

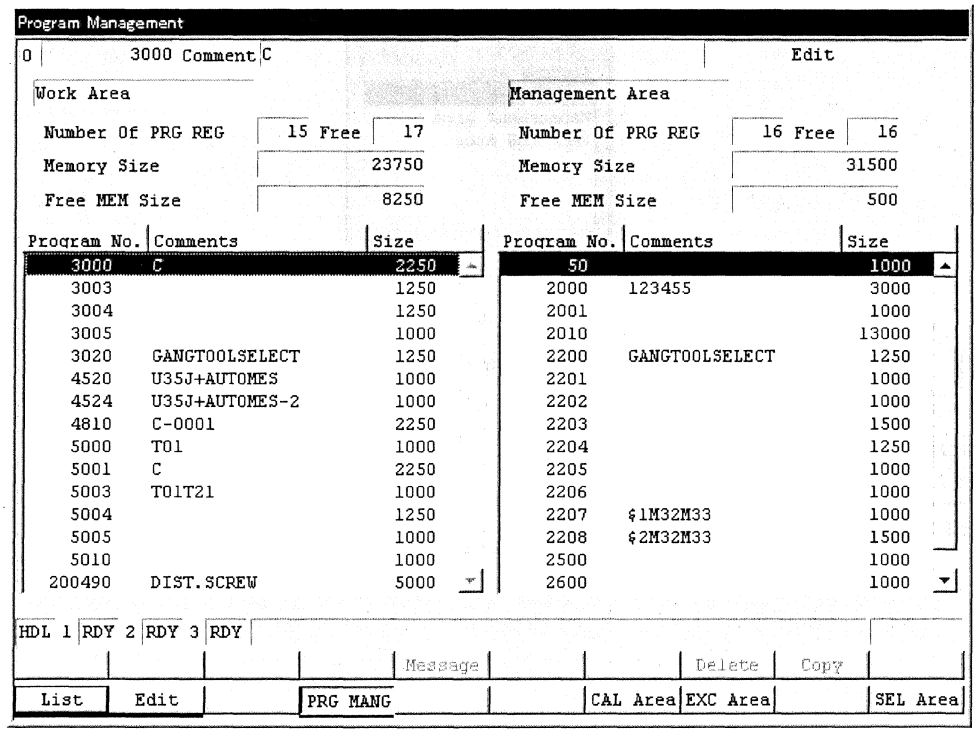
7.8.1 Selecting an area

This section explains the procedure for selecting either of two displayed areas with which processing is performed.
The currently selected area name is reversed in yellow. You can delete/copy data from/into the currently selected area.

Procedure

1. Press EDIT.
   The previously selected screen appears.

2. Press the menu key [PRG MANG].
   The Program Management screen appears.



3. Press the menu key [SEL Area].
   Move the (yellow) cursor to select either of two program areas displayed on the screen.

7.8.2 Calling an area

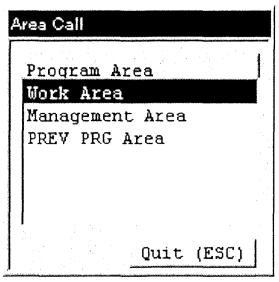
This section explains the procedure for calling and displaying one of areas with which you want to work.

Procedure

1. Press EDIT.
   The previously selected screen appears.

2. Press the menu key [PRG MANG].
   The Program Management screen appears.

3. Press the menu key [CAL Area].
   The Area Call window appears.



4. Press ↑ and/or ↓ to select the area you want to call.

5. Press INPUT.
   The called area is displayed in the selected area.

7.8.3 Exchanging areas

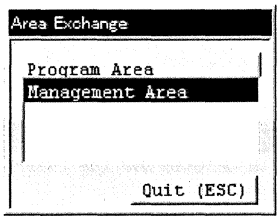
This section explains the procedure for exchanging the contents between the work area and management area.

Procedure

1. Press EDIT.
   The previously selected screen appears.

2. Press the menu key [PRG MANG].
   The Program Management screen appears.

3. Press the menu key [EXC Area].
   The Area Exchange window appears.



4. Press INPUT.
   The message "Please wait..." is displayed at the bottom of the screen. After that, the contents in the work area and management area are exchanged.

7.8.4 Deleting a program from an area

This section explains the procedure for deleting a program from a selected area.

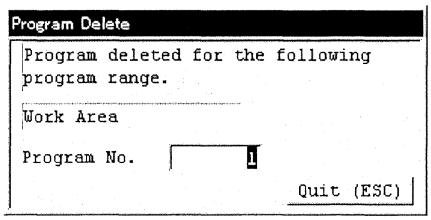
Procedure

1. Press EDIT.
   The previously selected screen appears.

2. Press the menu key [PRG MANG].
   The Program Management screen appears.

3. Press ↑ and/or ↓ to select the program you want to delete.

4. Press ▷ (Menu Up/Down Selection key). The submenus become effective.
5. Press the menu key [Delete].
   The Program Delete window appears.



6. Press INPUT.
   The selected program is deleted.

Note

The work area is equivalent to a program work area. This area stores the programs that are actually operated. Deletion of a program in the work area means that delete information such as a program (e.g., a subprogram of the currently selected program) necessary for operation.

7.8.5 Copying a program between areas

This section explains the procedure for copying a program between two displayed areas. The program is copied from the selected area into the non-selected area.

Procedure

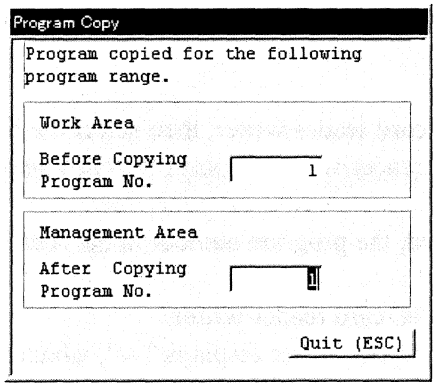
1. Press EDIT.
   The previously selected screen appears.

2. Press the menu key [PRG MANG].
   The Program Management screen appears.

3. Press ↑ and/or ↓ to select the program you want to copy.

4. Press ▷ (Menu Up/Down Selection key). The submenus become effective.

5. Press the menu key [Copy].
   The program copy window appears.



6. Press INPUT.
   The selected program is copied.

7.9 Data Input/Output

This section explains the procedures for transferring data (e.g., machining programs, tool offset data, and parameters) between this machine and the following input/output devices:

- IC card reader/writer(Option)
- Floppy disk drive(Option)
- PCMCIA card drive (Standard component)
- Front IC card slot (Standard component)

7.9.1 Transferring data between the machine and IC card reader/writer

This section explains data transfer on the assumption that the IC card reader/writer is already mounted. For the mounting, see <Chapter 7 Attachment> of the Cincom M12 Application Manual.

Procedure

1. Turn on the IC card reader/writer, then set the IC card.

2. Press the SEL switch of the IC card reader/writer.
   To input data into the machine from the IC card reader/writer, select the READ mode and press the SET switch.
   To output data from the machine to the IC card reader/writer, select the WRITE mode and press the SET switch.

3. Press the SEL switch of the IC card reader/writer, then select the file number (IC card storage area).
   The LCD screen of the IC card reader/writer displays the file number while the SEL switch is held down.
   When the SEL switch is released, the program number in the area is displayed in the READ mode.

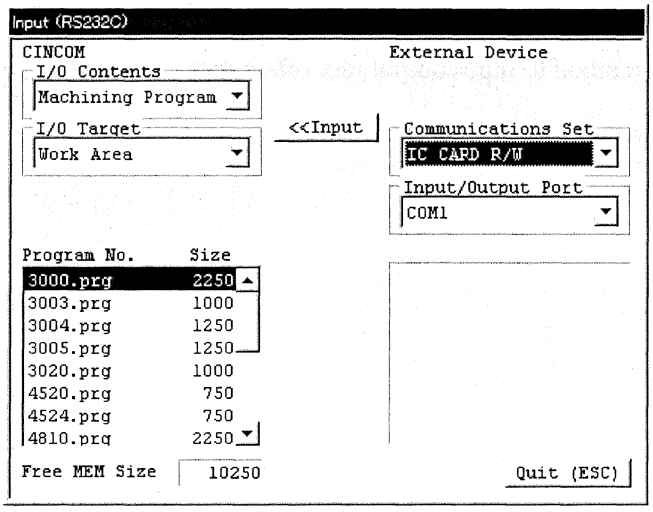
4. Press the START switch of the IC card reader/writer.
   The LCD screen of the IC card reader/writer displays "---", which indicates the transfer wait state.

5. Press EDIT. The previously selected screen appears.

6. Press the menu keys [List], and then [I/O].

7. To input data into the machine, use the Arrow keys to select [Input (RS232C)] and press INPUT.

   To output data from the machine, use the Arrow keys to select [Output (RS232C)] and press INPUT.
   The screen as shown below appears.



   "CINCOM" at the upper left of the screen indicates the machine. "External Device" indicates the IC card reader/writer.

8. Select the data you want to input or output.
   Use the Arrow keys to select "Machining Program", "Offset", or "Parameter" in the I/O Contents field, then press INPUT.
   If you have selected "Machining Program" in the I/O Contents field, select the storage area in the I/O Target field.
   When you want to output data from the machine to the IC card reader/writer, select the target program in the Program No. field, then press INPUT.

9. Use the Arrow keys to select "IC CARD R/W" as the input/output destination in the Communications Set field, then press INPUT.
   Select "COM1" in the Input/Output Port field, then press INPUT.

10. Select the [<<Input] or [Output>>] button, then press 

.

Data transfer starts. The LCD screen of the IC card reader/writer displays "END" at completion of the transfer.

Notes

- For data input, you can use the SEL switch of the IC card reader/writer to select only a file containing data.
- If you want to cancel processing or do not understand the operation method during operation, eject the IC card.
- For details on the operation method, see the <IC Card Reader/Writer Instruction Manual>.
- There is an alternative method to input/output tool offset data. Press 

, then press the [I/O] button on the Offset screen.

7.9.2 Transferring data between the machine and floppy disk drive

Procedure

1. Press PRM. . The previously selected screen appears.

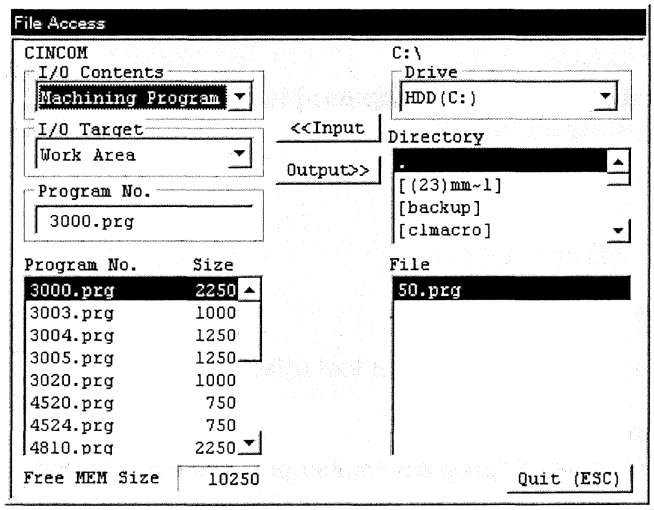
2. Press the menu keys [MC-STRCT], and then [Set].

3. Press ↑ and/or ↓ to select "Floppy Disk Drive Unit", then press INPUT.
   The Floppy Disk Drive Unit item is checked. Do not turn off the power at this time. If you turn off the power, the Floppy Disk Drive Unit item is automatically unchecked.

4. Press EDIT. The previously selected screen appears.

5. Press the menu keys [List], and then [I/O].

6. Use the Arrow keys to select [File Access] and press INPUT.
   The screen as shown below appears.



"CINCOM" at the upper left of the screen indicates the machine. The Drive field at the upper right indicates the currently select drive.

7. Insert a floppy disk (1.44MB format) into the floppy disk drive.

8. Select the data you want to input or output.
   Use the Arrow keys to select "Machining Program", "Offset", or "Parameter" in the I/O Contents field.
   If you have selected "Machining Program" in the I/O Contents field, select the storage area in the I/O Target field.

   - When you input data into the machine from the floppy disk drive:
     Use the Arrow keys to select "FDD (A:)" in the Drive field, then press 

     .
     Select the directory in the Directory field, then press 

     . The data in the selected directory is displayed in the File field.
     Press 

     to move the cursor to the File field, use the Arrow keys to select the data you want to input.

   - When you output data from the machine to the floppy disk drive:
     If you have selected "Machining Program" in the I/O Contents field, select the target program in the Program No. field, then press 

     .
     Use the Arrow keys to select "FDD (A:)" in the Drive field, then press 

     .
     Select the directory in the Directory field, then press 

     .

9. Press 

   to select the [<<Input] or [Output>>] button, then press 

   .
   Data transfer starts.

Notes

- Use a floppy disk in the 1.44MB format.
- There is an alternative method to input/output tool offset data. Press 

  , then press the [I/O] button on the Offset screen.
- A machining program is renamed "program-number.prg" when it is output.
- To input a machining program, rename the program "program-number.prg".

7.9.3 Transferring data between the machine and PCMCIA card drive

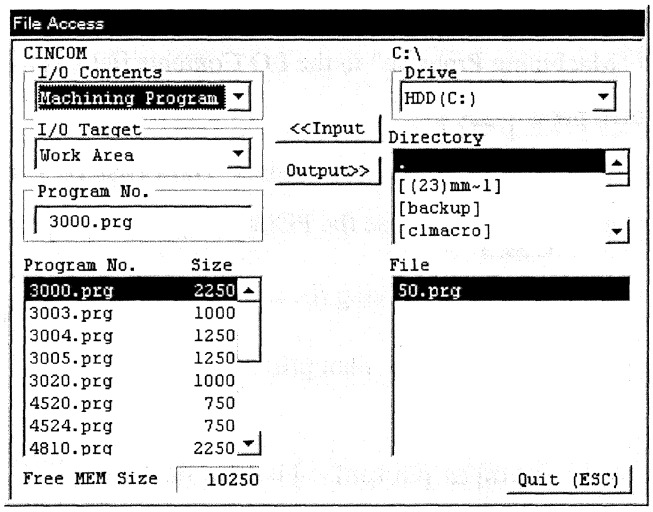
This section explains data transfer on the assumption that the PCMCIA card drive is already mounted.

Procedure

1. Press EDIT. The previously selected screen appears.

2. Press the menu keys [List], and then [I/O].

3. Use the Arrow keys to select [File Access] and press INPUT.
   The screen as shown below appears.



"CINCOM" at the upper left of the screen indicates the machine. The Drive field at the upper right indicates the currently select drive.

4. Select the data you want to input or output.
   Use the Arrow keys to select "Machining Program", "Offset", or "Parameter" in the I/O Contents field. If you have selected "Machining Program" in the I/O Contents field, select the storage area in the I/O Target field.

   - When you input data into the machine from the PCMCIA card drive:
     Use the Arrow keys to select "HDD (E:)" in the Drive field, then press INPUT.
     Select the directory in the Directory field, then press INPUT. The data in the selected directory is displayed in the File field.
     Press GRP to move the cursor to the File field, use the Arrow keys to select the data you want to input, then press INPUT.

   - When you output data from the machine to the PCMCIA card drive:
     If you have selected "Machining Program" in the I/O Contents field, select the target program in the Program No. field, then press INPUT.
     To use the PCMCIA slot in the operation panel, select "Hard Disk (E:)" in the Drive field by using the arrow keys, then press INPUT. To use the PCMCIA slot in the operation panel front, select "Removable Disk" in the Drive field by using the arrow keys, then press INPUT.
     Select the directory in the Directory field, then press INPUT.

5. Press GRP to select the [<<Input] or [Output>>] button, then press INPUT.
   Data transfer starts.

Note

There is an alternative method to input/output tool offset data. Press OFFSET, then press the [I/O] button on the Offset screen.

7.9.4 Mounting the PCMCIA card

> **WARNING**
>
> Turn off the main breaker of the machine before opening the operation panel. Failure to do so could result in electrocution or serious personal injury.

When using the PCMCIA slot in the operation panel:

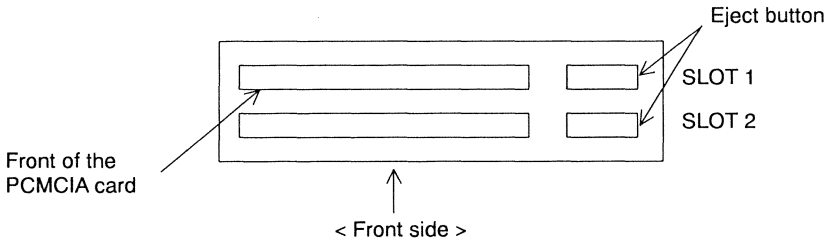
Procedure

1. Turn off the main breaker of the machine according to the procedure explained in <Section 7.1.2 Turning off the power>.

2. Unlock the operation panel with the key handle provided as a standard attachment tool.

3. Open the operation panel.

4. Insert the PCMCIA card into the PCMCIA card slot at the upper part of the NC unit. As a rule, use SLOT1. Push the card until the eject button on the right of the slot rises. Make sure of the direction of the PCMCIA card.



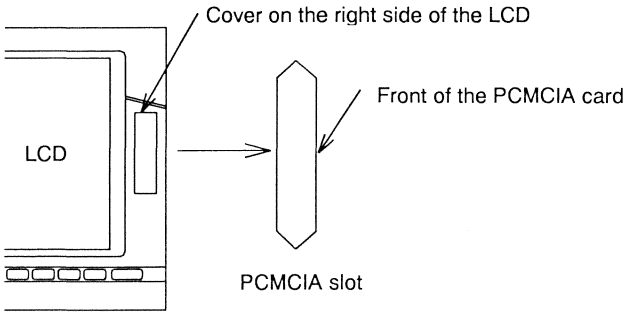
5. Close the operation panel, and lock the operation panel with the key handle.

When using the PCMCIA slot in the operation panel front:

Procedure

1. Open the cover on the right side of the LCD.

2. Insert the PCMCIA card into the PCMCIA card slot.
   Make sure of the direction of the PCMCIA card.



Note

We recommend using PCMCIA card which memory was made by Sun Disk Company. We don't assure PCMCIA card except for the above one.

CAUTION

Do not leave the PCMCIA card in the PCMCIA slot in the operation panel front for a long time. (Do not open the cover on the right side of the LCD for a long time.) Doing so may cause coolant to damage the NC unit.

7.9.5 Dismounting the PCMCIA card

> ⚠ WARNING
>
> Turn off the main breaker of the machine before opening the operation panel. Failure to do so could result in electrocution or serious personal injury.

When using the PCMCIA slot in the operation panel:

Procedure

1. Turn off the main breaker of the machine according to the procedure explained in <Section 7.1.2 Turning off the power>.
2. Unlock the operation panel with the key handle provided as a standard attachment tool.
3. Open the operation panel.
4. Press the eject button on the right of the PCMCIA card slot.
5. Remove the PCMCIA card from the PCMCIA card slot.
6. Close the operation panel, and lock the operation panel with the key handle.

When using the PCMCIA slot in the operation panel front:

Procedure

1. Remove the PCMCIA card from the PCMCIA card slot.
2. Firmly close the cover on the right side of the LCD.

Note

We recommend using PCMCIA card which memory was made by Sun Disk Company. We don't assure PCMCIA card except for the above one.

> ⚠ CAUTION
>
> Do not leave the PCMCIA card in the PCMCIA slot in the operation panel front for a long time. (Do not open the cover on the right side of the LCD for a long time.) Doing so may cause coolant to damage the NC unit.

7.10 Setting Tool Offset Data

This section explains the procedure for setting tool offset data. When a tool offset number is specified by the tool command (T command), the machine makes compensation for the total of tool nose wear and tool length. For each tool to be used, the tool nose wear is set in the offset data, and the tool length is set in the tool data.
If the tool nose is round, an error occurs between the workpiece shape specified in the program and an actually machined workpiece. To avoid the error, set tool nose radius compensation. The machine automatically makes compensation for the error.
Both the Offset screen and the Tool Data screen consist of more than one page. If the screen does not display the data number of the data you want to set or confirm, press 

and/of  to display the data of the desired number.

7.10.1 Setting data

There are two data input methods.

- INC INPT: An input value is added to the preset value.
- ABS INPT: An input value is used as the setting value.

Procedure

1. Press 

.
   The previously selected screen appears.

2. Press the menu key [Offset] or [T-Data] to select the data to set.

3. Press the menu key [INC INPT] or [ABS INPT] to select the input method.
   The message "INC INPT" or "ABS INPT" is displayed at the lower right of the screen.

4. Press  ,  , 

and/or 

to move the cursor to the input field in which you want to enter data.

5. Press the Numeric keys to enter the data.

6. Press 

. The data is set.

7.10.2 Clearing all screen data

This section explains the procedure for clearing all the data on the screen at a time.

Procedure

1. Press the menu key [Offset] or [T-Data] to select the screen having data you want to clear.
2. Press 

and then 

in this order.
   All the data disappear from the fields on the screen.
3. Press 

.
   All the data is cleared to 0.

Notes

- The all clear function is enabled with only a screen. Executing this function does not affect data in the other pages.
- If you have erased all displayed data by accident, you can cancel to erase only before pressing 

to execute the all clear function. Display another page, then return to the original page. The page redisplays the data that you have erased.

7.11 Maintenance

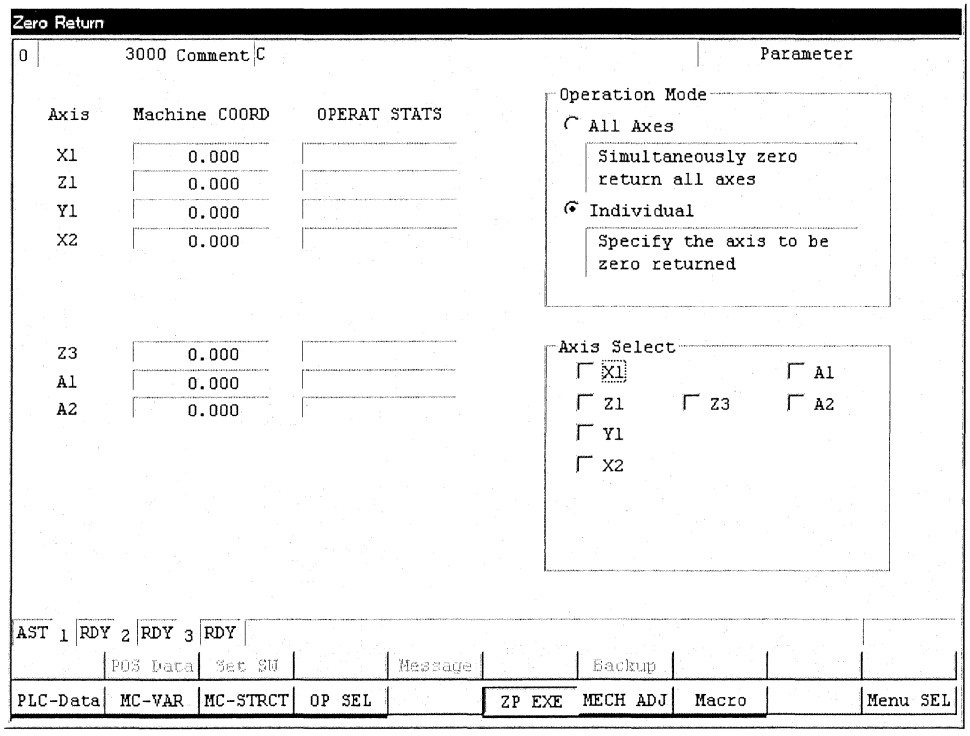
7.11.1 Returning axes to zero point

This section explains the procedure for returning each axis to the machine zero point. The zero point return function is required only when an alarm is issued requesting axes to return to zero point. You do not have to execute this function in general operation.
There are two operation modes for returning axes to zero point.

- All axes: The machine returns all axes to zero point.
- Individual: The machine returns only specified axes to zero point.

Procedure

1. Press PRM.  .
2. Repeatedly press the menu key [Menu SEL] until the menu key [ZP EXE] appears.
3. Press the menu key [ZP EXE].
   The Zero Return screen appears.



(This screen is for type VII.)

4. Press GRP ↑ ↓ to select the operation mode "All Axes" or "Individual", then press INPUT.
   If you have selected "Individual", press the GRP and Arrow keys to move the cursor to the axis you want to return to zero point, then press INPUT. When you want to cancel the axis selection, press INPUT again. You can select more than one axis at a time.

5. Press START. The START key lamp lights. The machine returns axes to zero point in the specified operation mode.
   For the axis that is returning to zero point, the message "Executing Zero Return" is displayed in the OPERAT STATS field. The message "Completed Zero Return" is displayed for the axis that has returned to the zero point.

Notes

- You do not have to execute the zero point return function during general everyday operation. Execute the function only when required (e.g., an alarm is issued requesting axes to return to zero point, or motor replacement is necessary).
- When the operation mode "All axes" is selected, the machine moves the axes to zero point as follows:

X1 axis returns to zero point → A1 axis returns to zero point → A2 axis returns to zero point

→ A7 axis returns to zero point → Z1 axis returns to zero point → Y1 axis returns to zero point
→ Z3 axis returns to zero point → X2 axis returns to zero point → Z2 axis returns to zero point
→ X3 axis returns to zero point → Z2 axis returns to zero point
→ A3 axis returns to zero point → Z2 axis returns to zero point

- A3 axis and A7 axis are optional.
- Type VI does not have X2, Z2 and X3 axes.
- Type VII does not have Z2 and X3 axes.

7.11.2 Resetting an overrun alarm

An overrun alarm is issued when an axis receives a command to move exceeding the limit. This section explains the procedure for resetting an overrun alarm.

Procedure

1. Any overrun alarm will flash the CAUTION lamp (yellow) at the top right of the operation panel and automatically displays a Message screen.
   Confirm the overrun axis in the message.
   If the Message screen does not appear, press ▷ (Menu Up/Down Selection key). The submenus become effective.
   Press the menu key [Message] to display the Message screen.

2. Press MANUAL.
   The Handle Feed screen appears.
   If the Handle Feed screen does not appear, press the menu key [Handle] to display the screen.

3. Press ↑ and/or ↓ to select the overrun axis.
   The name of the selected axis is reversed in yellow.

4. Press a Handle Magnification key x1, x10, or x100.
   The Handle Magnification key lamp lights.
   The machine operates faster as a greater magnification is selected.

5. For the overrun axis, turn the handle in the direction for freeing the axis from the overrun state.
   The CAUTION lamp (yellow) turns off when the overrun state is reset.

Notes

- There is an alternative method to enable handle feed. Use [MECH ADJ] in the Parameter screen instead of MANUAL.
  Press PRM., then press the menu key [MECH ADJ].
  The Mechanical Adjustment screen appears. The subsequent steps are the same as for the Handle Feed screen.
- For A axes, you can use only the handle feed specified on the Mechanical Adjustment screen.
- The Message screen displays the messages as shown below.

Overrun axis		Message			
$1	X1 axis	M01 Operation Error	0007	X1	$1
	Z1 axis	M01 Operation Error	0007	Z1	$1
	Y1 axis	M01 Operation Error	0007	Y1	$1
$2	X2 axis (Type VII, VIII)	M01 Operation Error	0007	X2	$2
	Z2 axis (Type VIII)	M01 Operation Error	0007	Z2	$2
$3	X3 axis (Type IX)	M01 Operation Error	0007	X3	$3
	Z3 axis	M01 Operation Error	0007	Z3	$3
$4	A1 axis	M01 Operation Error	0007	A1	$4
	A2 axis	M01 Operation Error	0007	A2	$4
	A3 axis	M01 Operation Error	0007	A3	$4
$5	A7 axis	M01 Operation Error	0007	A7	$5

- The A3 axis and A7 axis are optional.

- The following table lists the OT (stored stroke limit):

Overrun axis		C12		C16	
		Limit in - direction	Limit in + direction	Limit in - direction	Limit in + direction
$1	X1 axis	– 16.000	91.000	– 16.000	91.000
	Z1 axis	– 1.000	206.0/226.0	– 1.000	206.0/216.0
	Y1 axis	– 98.000	106.000	– 98.000	106.000
$2	X2 axis (Type VII, VIII)	– 0.500	200.500	– 0.500	200.500
	Z2 axis (Type VIII)	– 1.000	76.000	– 1.000	76.000
$3	X3 axis (Type IX)	– 59.000	8.200	– 59.000	8.200
	Z3 axis	– 1.000	256.000	– 1.000	256.000
$4	A1 axis	– 0.500	17.000	– 0.500	17.000
	A2 axis	– 0.500	17.000	– 0.500	17.000
	A3 axis	– 0.500	120.500	– 0.500	120.500
$5	A7 axis	– 2.000	4400.000	– 2.000	4400.000

Notes

- The above numeric values are expressed with the radius for all the axes.

- Z1 axis limit in the plus direction:

	C12	C16
When a synchronous rotary guide bushing is used:	206.000	206.000
When a fixed guide bushing is used:	226.000	216.000

Product Code C – C 1 2 1 6 VI VII VIII IX – 7 7 0

Document Code 5 E 1 – 0 7 0 3

8. TOOLING

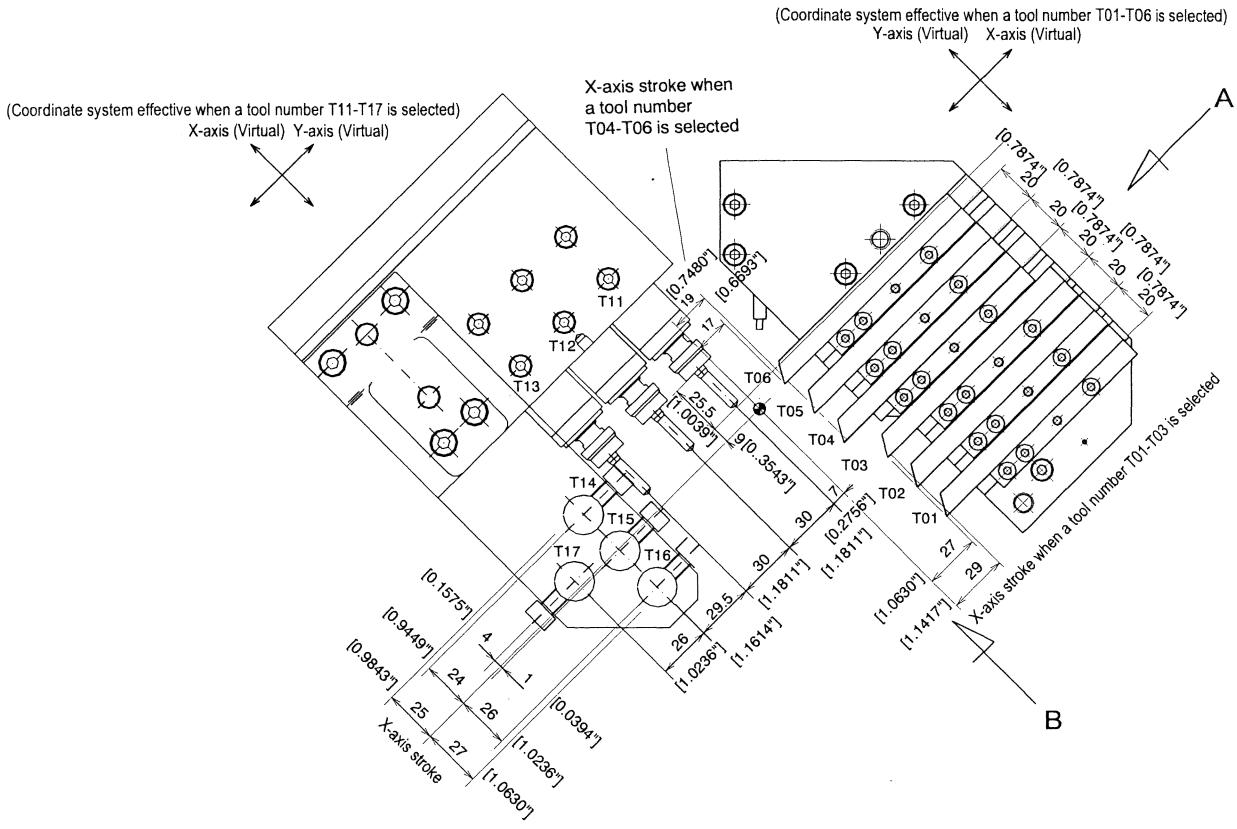
Code No.	C-C1216VI VII VIII IX-770 5E1-0802 5E2-0802	MFG No.	C1216/0359~	Issue Date	2001.12

Name		Specification	Model	Page
■ Rotary Tool				
[Cross Milling Spindle]				
Rego type chuck(~ø7)	Example of mounting the U30B	[ER11][AR11]	GSC607	8-37
Rego type chuck(~ø7)	Example of mounting the U31B : Type VI	[ER11][AR11]	GSC607	8-38
[End-drilling Spindles]				
Rego type chuck(~ø7)	Example of mounting the U30B	[ER11][AR11]	GSE507	8-39
Rego type chuck(~ø7, Short type)	Example of mounting the U30B	[ER11][AR11]	GSE707	8-40
Rego type chuck(~ø7, Angle-adjustable type)	Example of mounting the U30B	[ER11][AR11]	GSA207	8-41
Rego type chuck (~ø7, Short type ,Angle-adjustable type)	Example of mounting the U30B	[ER11][AR11]	GSA307	8-42
Rego type chuck(~ø7)	Example of mounting the U31B : Type VI	[ER11][AR11]	GSE507	8-43
Rego type chuck(~ø7, Short type)	Example of mounting the U31B : Type VI	[ER11][AR11]	GSE707	8-44
Rego type chuck(~ø7, Angle-adjustable type)	Example of mounting the U31B : Type VI	[ER11][AR11]	GSA207	8-45
Rego type chuck (~ø7, Short type ,Angle-adjustable type)	Example of mounting the U31B : Type VI	[ER11][AR11]	GSA307	8-46
Rego type chuck(~ø7)	U121B	[ER11][AR11]	GSE807	8-47
[Slitting Spindle]				
Slitting Spindle		[ø50 xø12.7 / ø15.875]	GSS150	8-48
Slitting Spindle		[ø50xø13 / ø16]	GSS250	8-49
■ U120B/GDF905/U121B Sleeve				
[Boring · Drill]				
Rego type chuck (~ø7, Short type)	Shank dia.ø19.05	[ER11][AR11]	BDS507	8-50
Rego type chuck (~ø10, Short type)	Shank dia.ø19.05	[ER16][AR16]	BDS508	8-51
Rego type chuck (~ø7, Semi-short type)	Shank dia.ø19.05	[ER11][AR11]	BDS607	8-52
Rego type chuck (~ø10, Semi-short type)	Shank dia.ø19.05	[ER16][AR16]	BDS610	8-53
Rego type chuck (~ø7, Short type)	Shank dia.ø19.05	[ER11][AR11]	BDS707	8-54
Rego type chuck (~ø7)	Shank dia.ø19.05	[ER11][AR11]	VDS506	8-55
Rego type chuck (~ø10)	Shank dia.ø19.05	[ER16][AR16]	VDS110	8-56
Rego type chuck (~ø7, Long type)	Shank dia.ø19.05	[ER11][AR11]	LDS107	8-57
Rego type chuck (~ø10, Long type)	Shank dia.ø19.05	[ER16][AR16]	LDS110	8-58
Rego type chuck (~ø7, Both end short type)	Shank dia.ø19.05	[ER11][AR11]	HDS5406	8-59
Rego type chuck (~ø7, Both end)	Shank dia.ø19.05	[ER11][AR11]	HDS5506	8-60
Rego type chuck (~ø7, Both end semi long)	Shank dia.ø19.05	[ER11][AR11]	HDS5706	8-61
Rego type chuck (~ø7, Both end long)	Shank dia.ø19.05	[ER11][AR11]	HDS5806	8-62
[Tap · Die]				
Rego type chuck (~ø7, Short type)	Shank dia.ø19.05	[ER11][AR11] Die ø16,ø20	BNS407	8-63
Rego type chuck (~ø7, Short type)	Shank dia.ø19.05	[ER11][AR11] Die ø5/8",ø13/16"	BNS507	8-64
Rego type chuck (~ø7)	Shank dia.ø19.05	[ER11][AR11] Die ø16,ø20	VNS406	8-65
Rego type chuck (~ø7)	Shank dia.ø19.05	[ER11][AR11] Die ø5/8",ø13/16"	VNS506	8-66

Name	Specification			Model	Page
■ GDF901 Sleeve					
[Boring · Drill]					
Rego type chuck (~ø7, Short type)	Shank dia.ø19.05	[ER11][AR11]		BDS507	8-67
Rego type chuck (~ø7, Semi-short type)	Shank dia.ø19.05	[ER11][AR11]		BDS607	8-68
Rego type chuck (~ø7, Short type)	Shank dia.ø19.05	[ER11][AR11]		BDS707	8-69
Rego type chuck (~ø7)	Shank dia.ø19.05	[ER11][AR11]		VDS506	8-70
[Tap]					
Rego type chuck (~ø7, Short type)	Shank dia.ø19.05	[ER11][AR11]		BNS407	8-71
Rego type chuck (~ø7, Short type)	Shank dia.ø19.05	[ER11][AR11]		BNS507	8-72
■ GDF907 Sleeve					
[Boring · Drill]					
Rego type chuck (~ø7, Short type)	Shank dia.ø19.05	[ER11][AR11]		BDS807	8-73
■ ADAPTER					
Dedicated adapter	Outside dia.ø33		Shank dia.ø19.05	SAU319	8-74

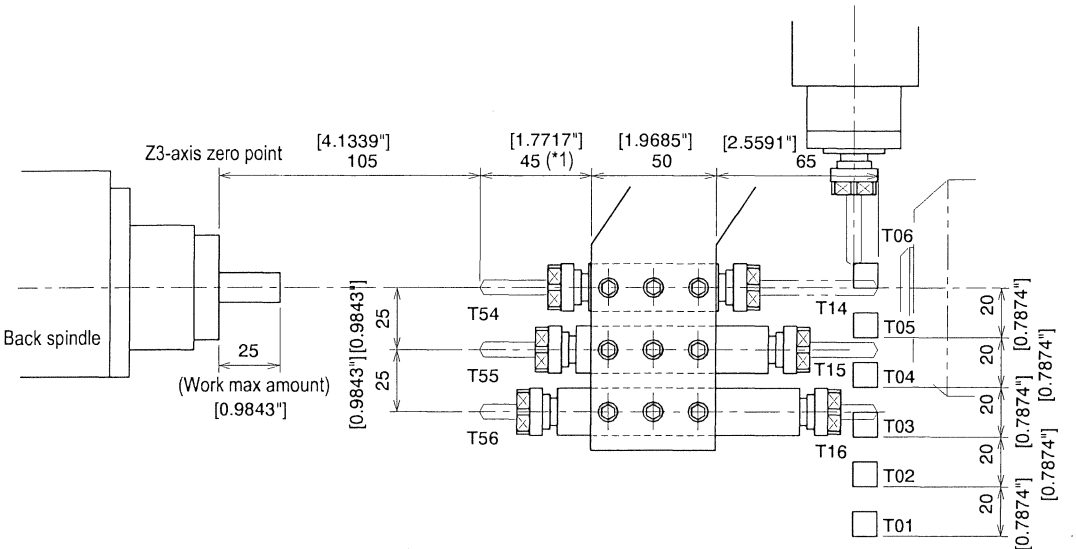
<Blank Page>

Type VI Standard Tool Layout

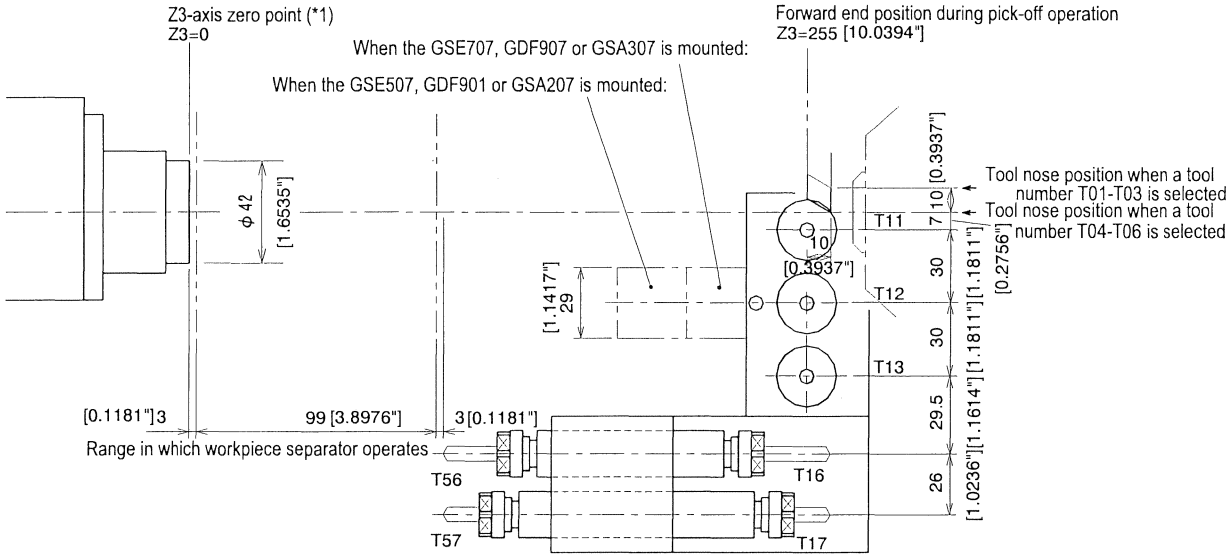


* The X and Y axes switch to each other in accordance with the selected tool number.

- When viewed from A:

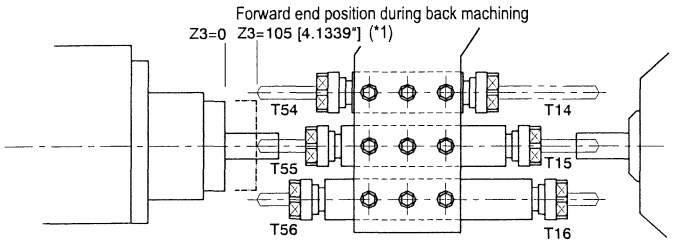


- When viewed from B:



*1 The workpiece separator is able to advance when it is at the position Z3=0. Be sure to limit the protrusion of the back machining tool to 45 mm [1.7717"] shown in the above figure.

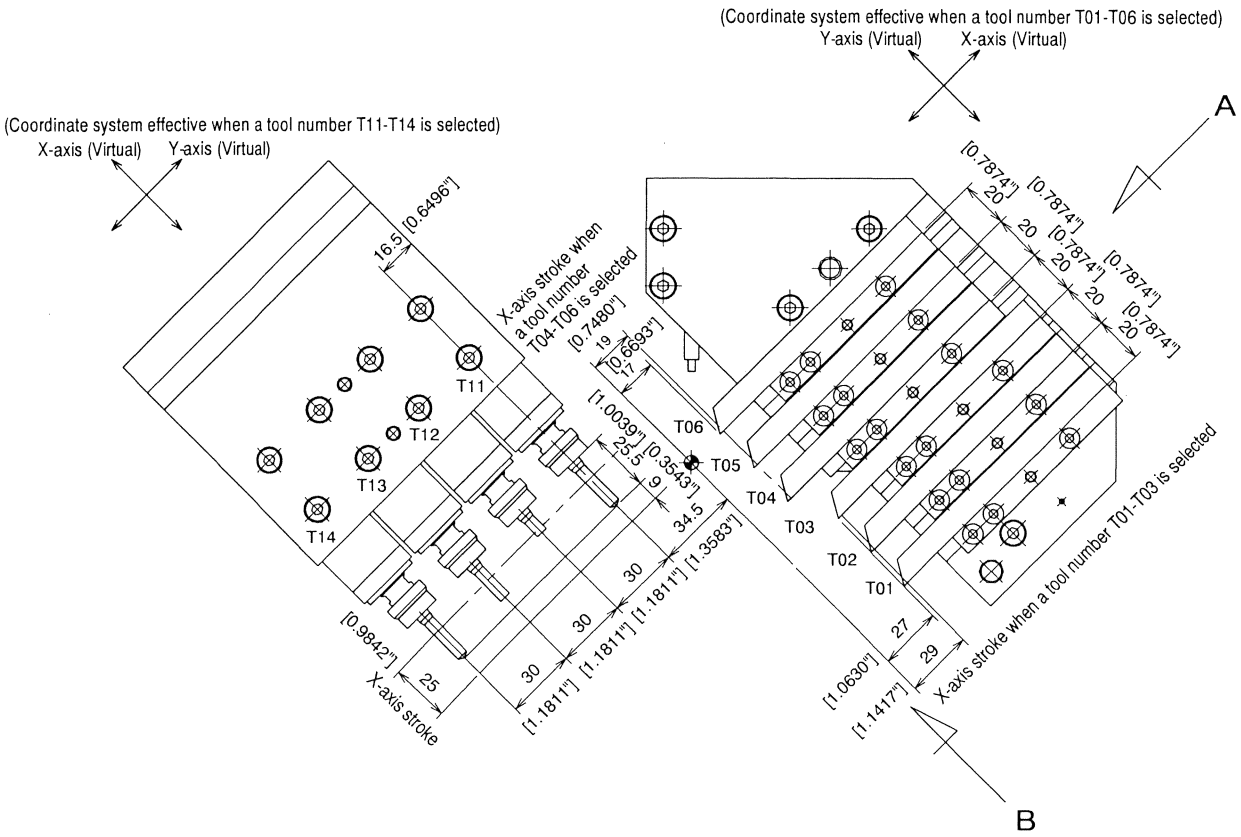
<Machining sample: Front drilling and back drilling at the same time>



Use of a two-way sleeve enables the machine to perform front drilling and back drilling at the same time or simultaneous tapping.

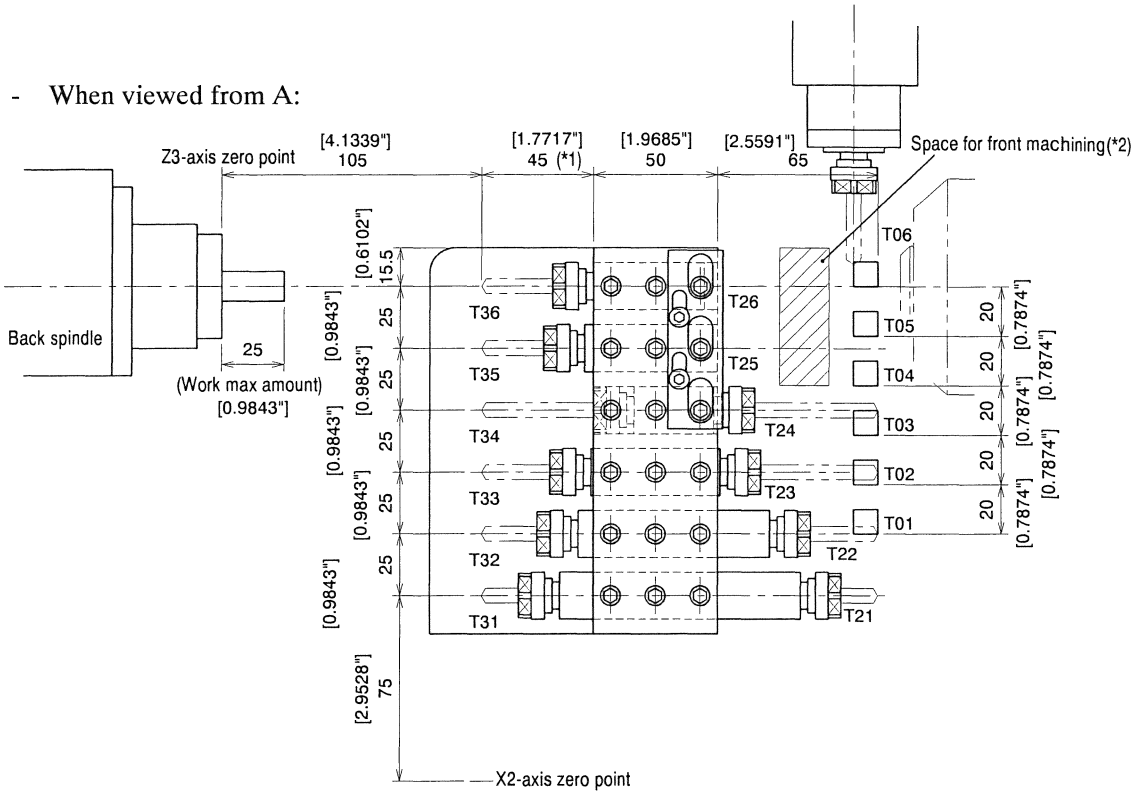
*1 The top end of the back spindle may interfere with the adjacent tool depending on the size of the tool. To avoid the interference, specify a value smaller than 105 [4.1339"] for the forward end position of the Z3 axis.

Type VII Standard Tool Layout

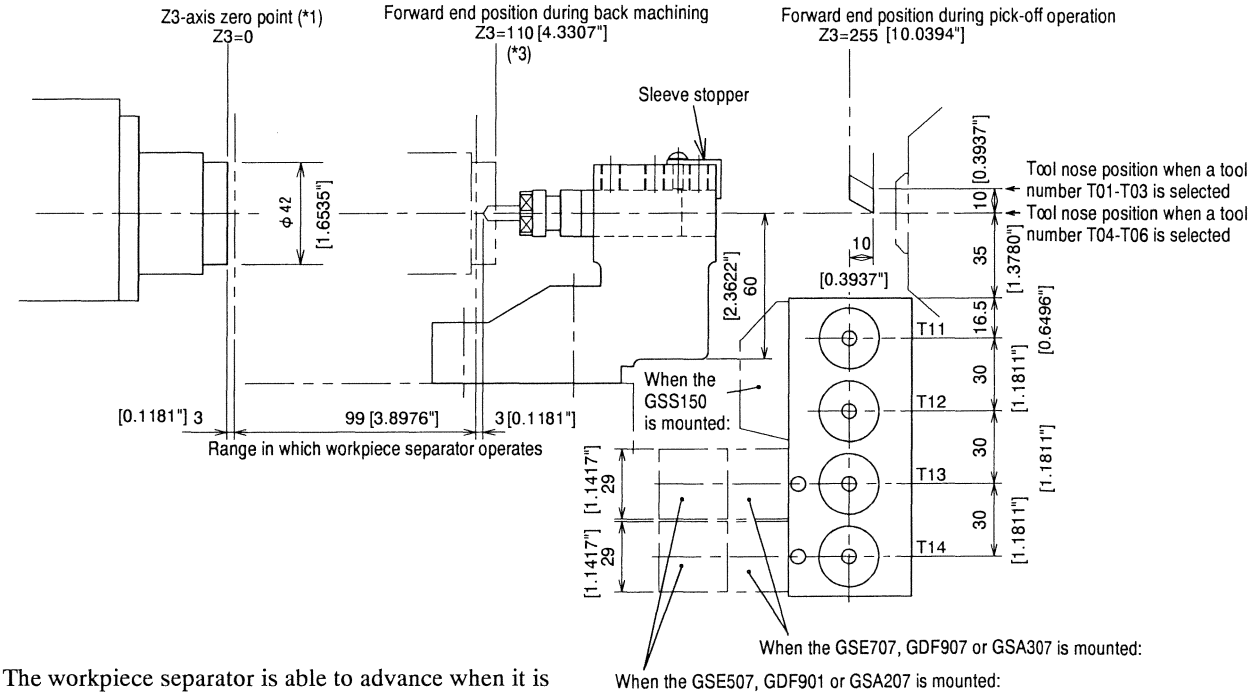


* The X and Y axes switch to each other in accordance with the selected tool number.

- When viewed from A:



- When viewed from B:

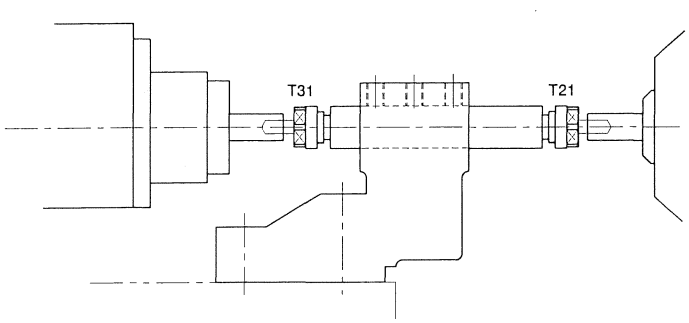


*1 The workpiece separator is able to advance when it is at the position Z3=0. Be sure to limit the protrusion of the back machining tool to 45 mm [1.7717"] shown in the above figure.

*2 This space is necessary for performing front turning and back drilling at the same time. To allocate the space, the "stopper" is mounted as standard.If a sleeve is mounted at the positions T25 and T26 in the above figure,the machine is unable to perform front turning and back drilling at the same time.

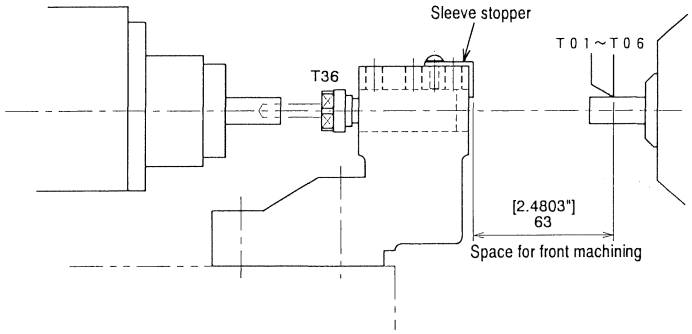
*3 The top end of the back spindle may interfere with the adjacent tool depending on the size of the tool. To avoid the interference, specify a value smaller than 105 [4.1339"] for the forward end position of the Z3 axis.

<Machining sample: Front drilling and back drilling at the same time>

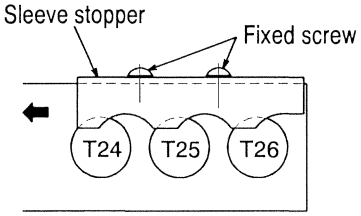


Use of a two-way sleeve enables the machine to perform front drilling and back drilling at the same time or simultaneous tapping.

<Machining sample: Front turning and back drilling at the same time>

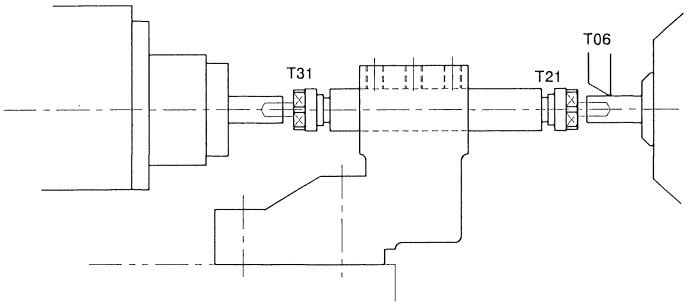


Allocation of the space for front machining enables the machine to perform front turning and back drilling at the same time.



To mount sleeves at the positions T24-T26 sequentially, tighten the screws securing the "stopper" and shift the stopper into the direction indicated by the arrow.

<Machining sample: Front turning, front drilling, and back drilling at the same time>



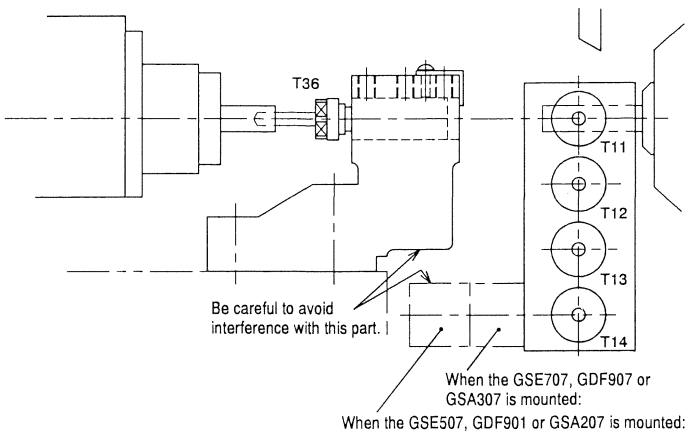
The following table shows combinations of tools acceptable and unacceptable for simultaneously performing front turning and front drilling. A circle is marked for acceptable combinations of tools.

Front drill \ Front turning tool	T06	T05	T04	T03	T02	T01
T26	○×	○×	○×	○×	○×	○○
T25	○×	×	×	×	×	×
T24	○×	×	×	×	×	×
T23	○×	×	×	×	×	×
T22	○×	×	×	×	×	×
T21	○○	×	×	×	×	×

(Upper case) Front drill Tool size: Up to φ7[0.2756"]

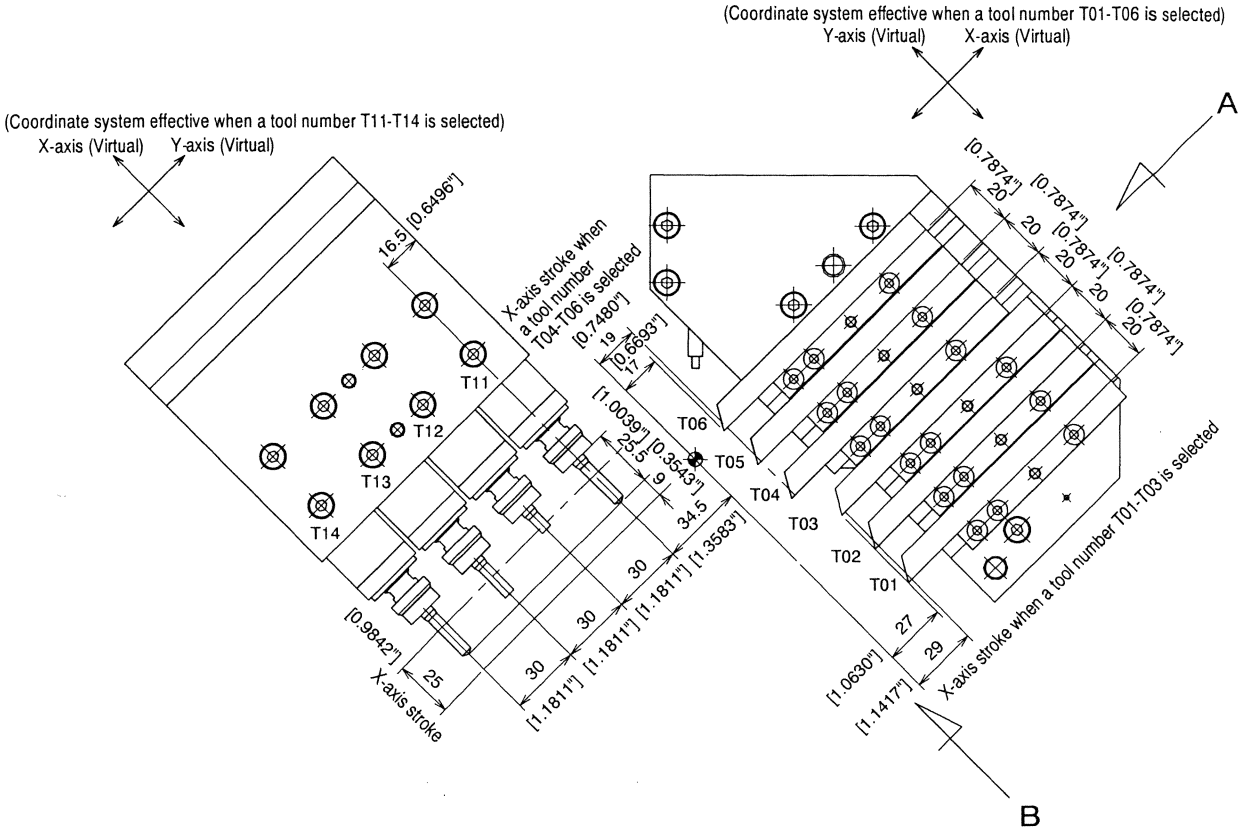
(Lower case) Front drill Tool size: Over φ7[0.2756"]

<Machining sample: Front cross drilling and back drilling at the same time>



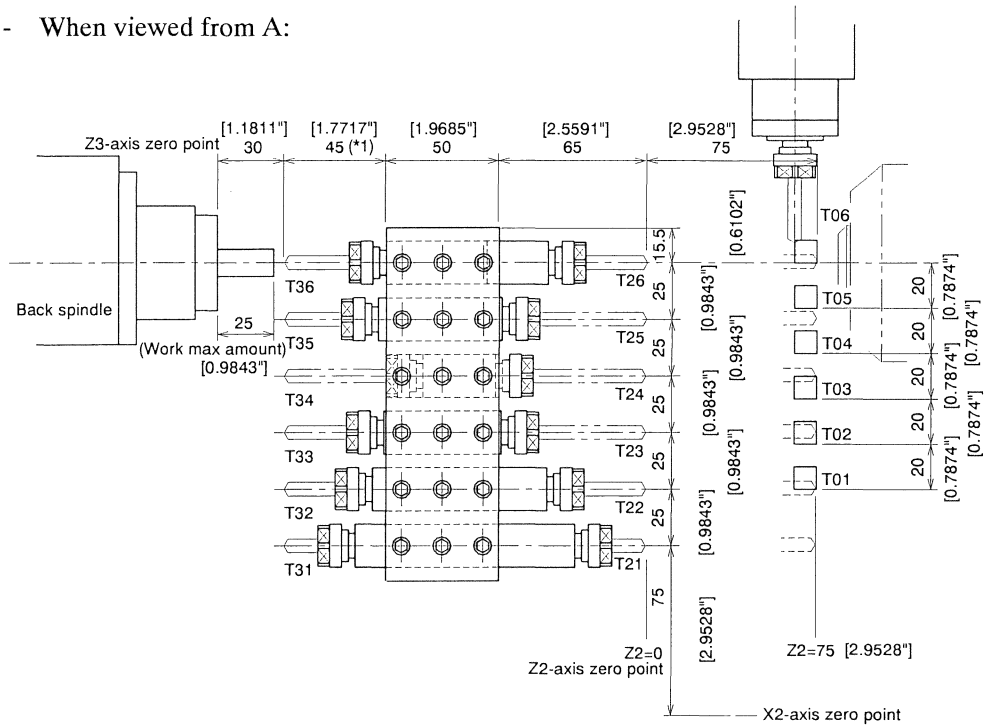
With the GSE507 or GDF901 mounted: If front cross drilling and back drilling are to be performed at the same time, the machine is able to perform front cross drilling (end milling) by only using T11. In this case, only one GSE507 or GDF901 can be mounted at the position T14. (Mounting of two of them is not permitted.)
Without the GSE507 or GDF901 mounted: The machine is able to perform simultaneous cross drilling (end milling) by using T11-T14. (Use of T14 is not permitted for end milling.)

Type VIII Standard Tool Layout

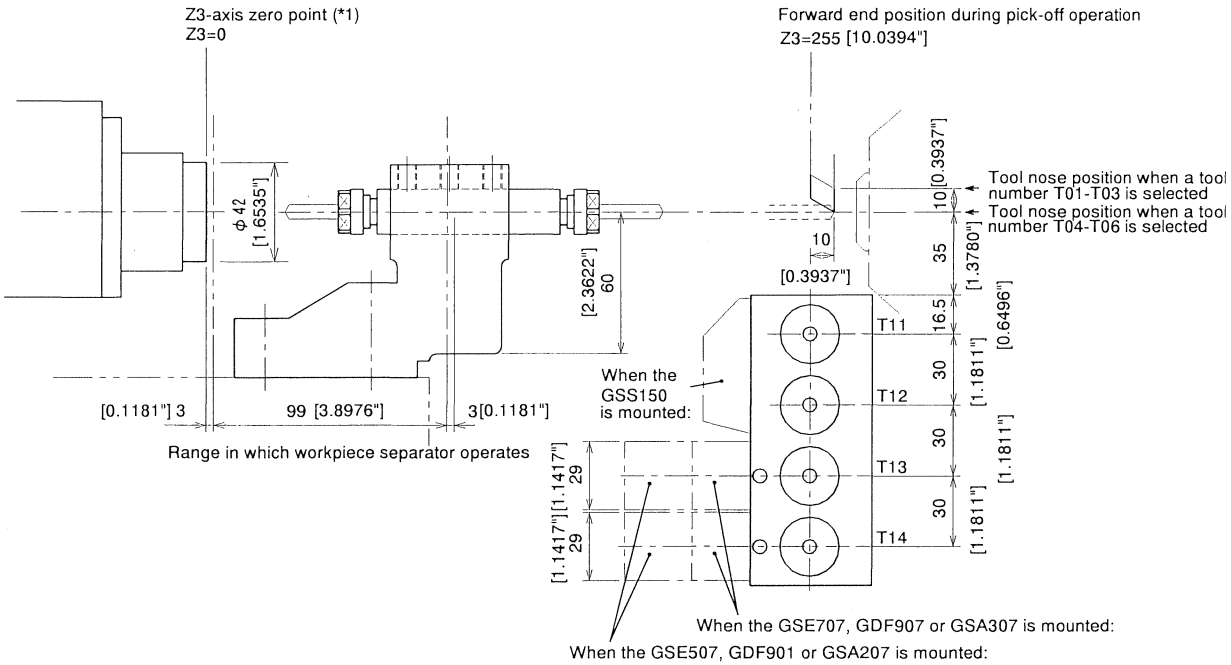


* The X and Y axes switch to each other in accordance with the selected tool number.

- When viewed from A:

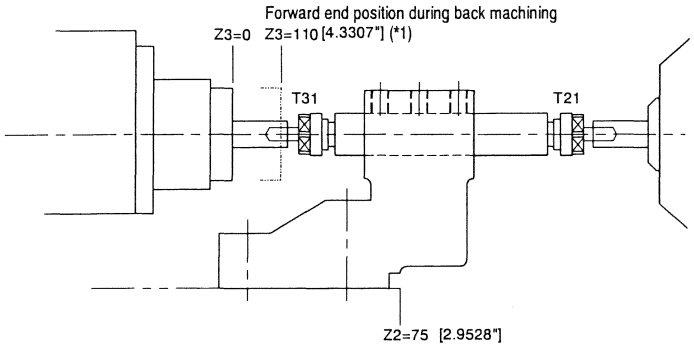


- When viewed from B:



*1 The workpiece separator is able to advance when it is at the position Z3=0, Z2=75.0 [2.9528"] or Z3=0, X2=0. Be sure to limit the protrusion of the back machining tool to 45 mm [1.7717"] shown in the above figure.

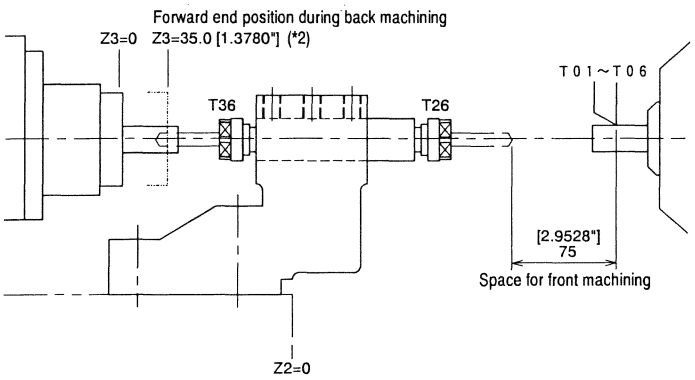
<Machining sample: Front drilling and back drilling at the same time>



Use of a two-way sleeve enables the machine to perform front drilling and back drilling at the same time or simultaneous tapping.

*1 The top end of the back spindle may interfere with the adjacent tool depending on the size of the tool. To avoid the interference, specify a value smaller than 105 [4.1339"] for the forward end position of the Z3 axis.

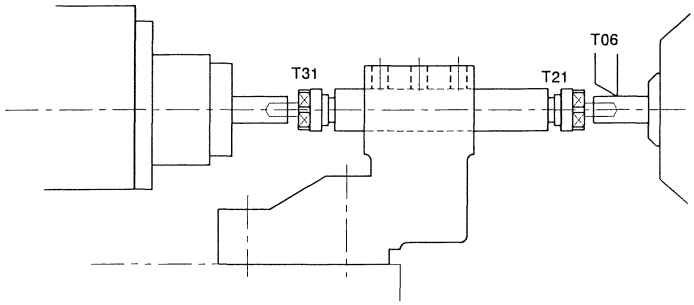
<Machining sample: Front turning and back drilling at the same time>



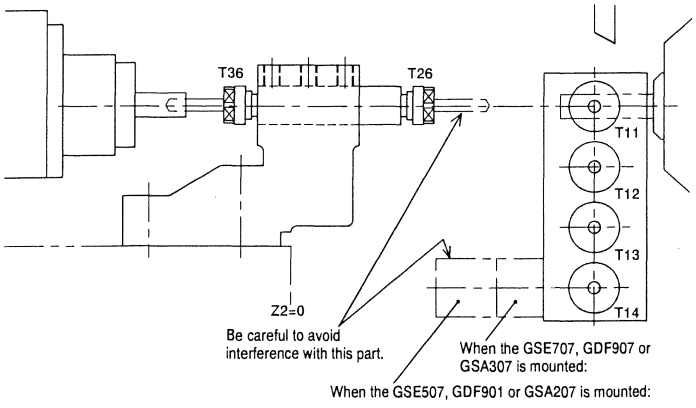
Allocation of the space for front machining (Z2=0) enables the machine to perform front turning and back drilling at the same time.

*2 The top end of the back spindle may interfere with the adjacent tool depending on the size of the tool. To avoid the interference, specify a value smaller than 30 [1.1811"] for the forward end position of the Z3 axis.

<Machining sample: Front turning, front drilling,



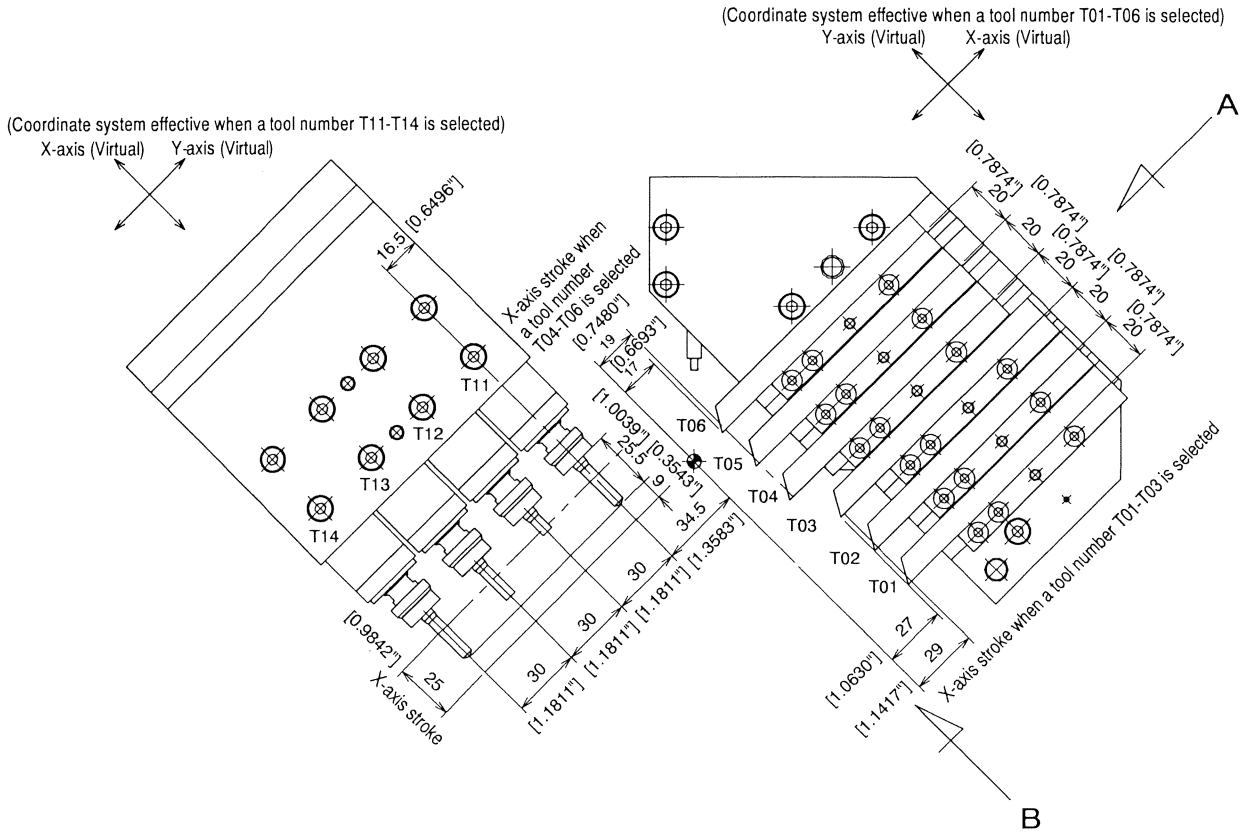
<Machining sample: Front cross drilling and back drilling at the same time>



Allocation of the space for front machining (Z2=0) enables the machine to perform front cross drilling (end milling) and back drilling at the same time. (Use of T14 in not permitted for end milling.)

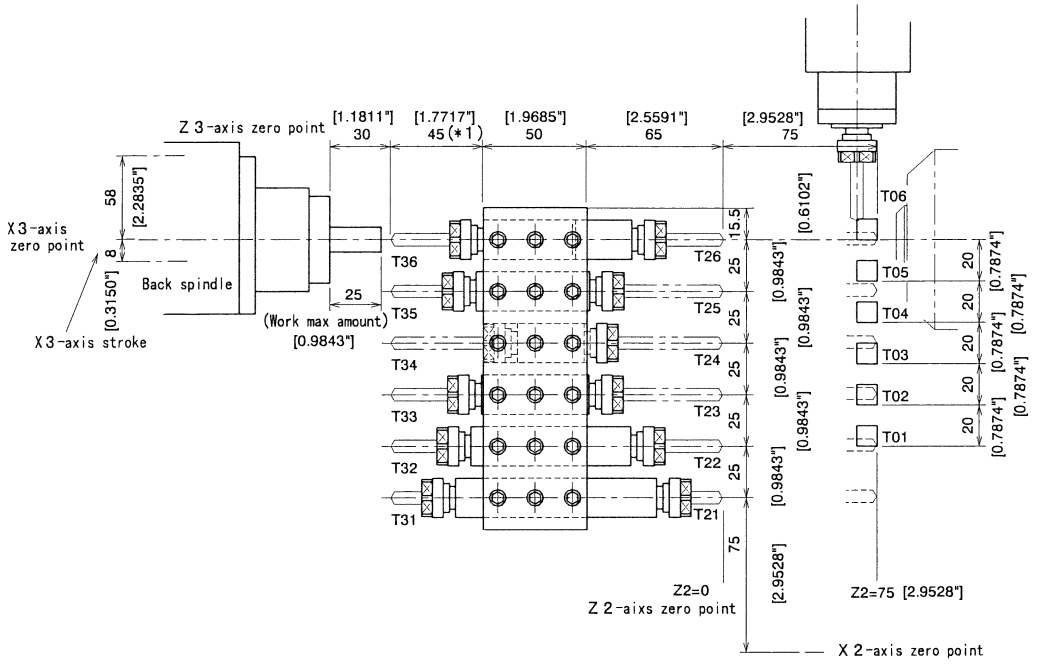
Be careful to avoid interference of T2*** and GSE507,GDF901, GSA207.

Type IX Standard Tool Layout

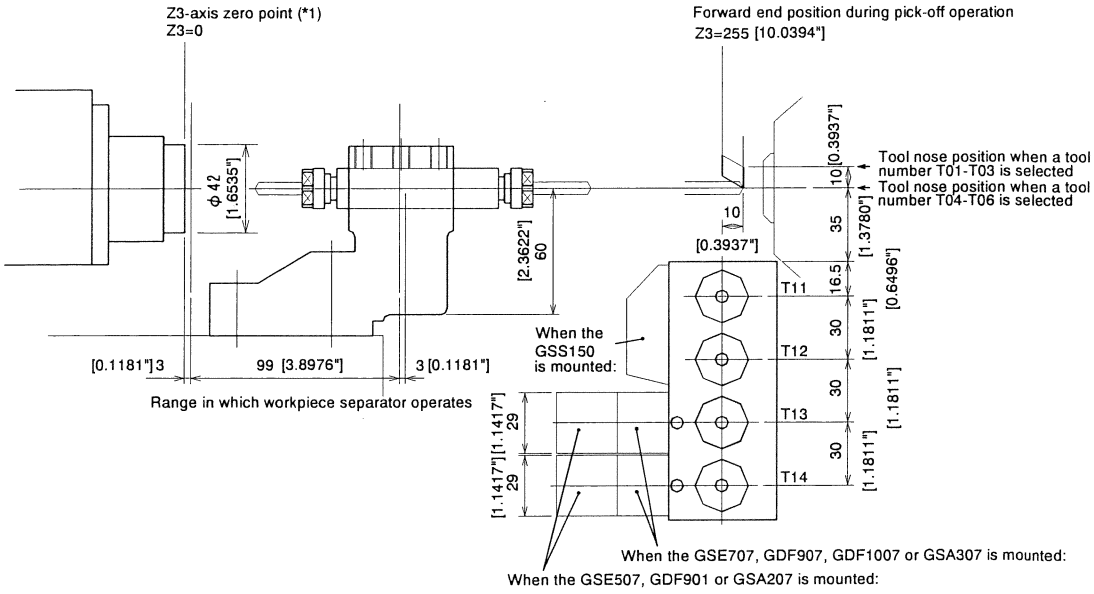


* The X and Y axes switch to each other in accordance with the selected tool number.

- When viewed from A:

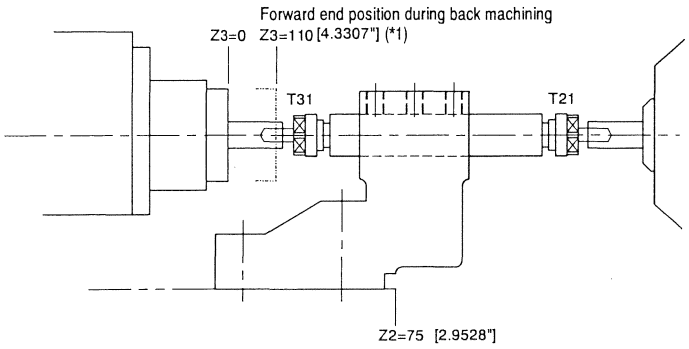


- When viewed from B:



*1 The workpiece separator is able to advance when it is at the position X3=0, Z3=0, Z2=75.0 [2.9528"] or X3=0, Z3=0, X2=0. Be sure to limit the protrusion of the back machining tool to 45 mm [1.7717"] shown in the above figure.

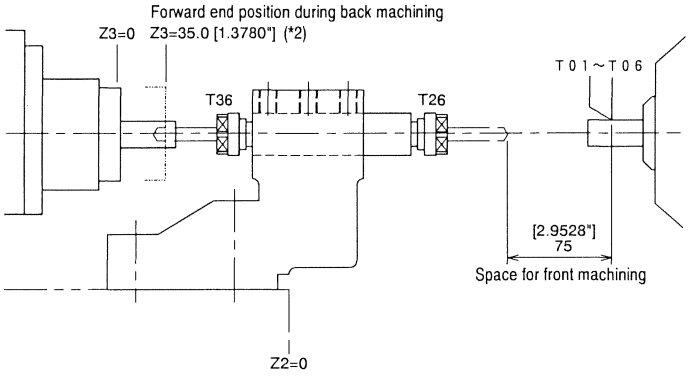
<Machining sample: Front drilling and back drilling at the same time>



Use of a two-way sleeve enables the machine to perform front drilling and back drilling at the same time or simultaneous tapping.

*1 The top end of the back spindle may interfere with the adjacent tool depending on the size of the tool. To avoid the interference, specify a value smaller than 105 [4.1339"] for the forward end position of the Z3 axis.

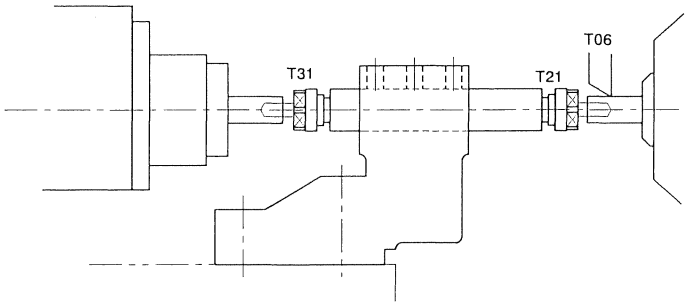
<Machining sample: Front turning and back drilling at the same time>



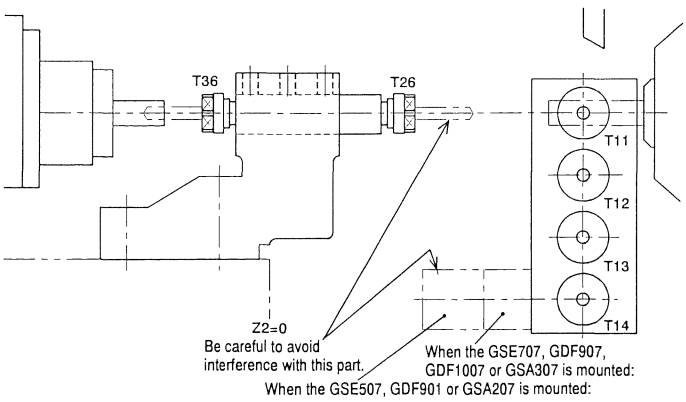
Allocation of the space for front machining (Z2=0) enables the machine to perform front turning and back drilling at the same time.

*2 The top end of the back spindle may interfere with the adjacent tool depending on the size of the tool. To avoid the interference, specify a value smaller than 30 [1.1811"] for the forward end position of the Z3 axis.

<Machining sample: Front turning, front drilling,

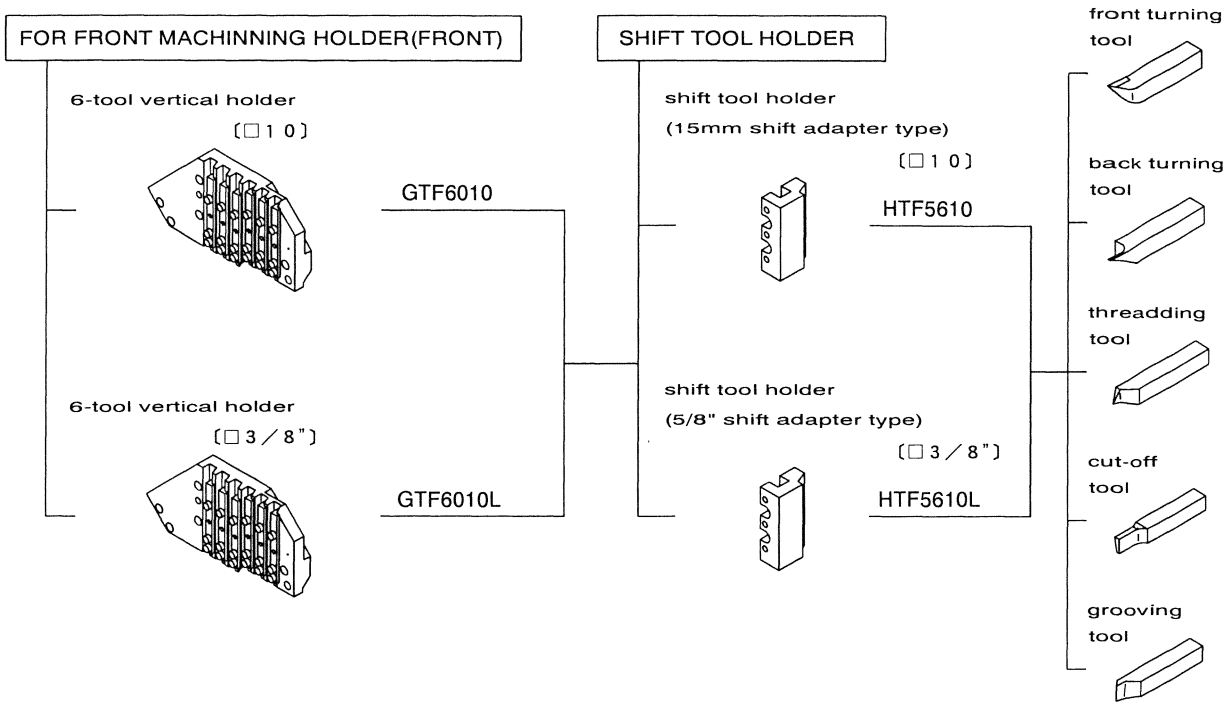


<Machining sample: Front cross drilling and back drilling at the same time>



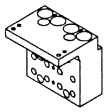
Allocation of the space for front machining (Z2=0) enables the machine to perform front cross drilling (end milling) and back drilling at the same time. (Use of T14 in not permitted for end milling.)

Be careful to avoid interference of T2*** and GSE507,GDF901, GSA207.


FOR FRONT MACHINNING HOLDER(FRONT)
SHIFT TOOL HOLDER
6-tool vertical holder
〔□10〕
GTF6010
6-tool vertical holder
〔□3／8"〕
GTF6010L
shift tool holder
(15mm shift adapter type)
〔□10〕
HTF5610
shift tool holder
(5/8" shift adapter type)
〔□3／8"〕
HTF5610L
front turning tool
back turning tool
threadding tool
cut-off tool
grooving tool

FOR FRONT MACHINNING HOLDER(BACK)

4-tool rotary tool holder



U30B

ROTARY TOOL

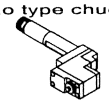
cross-milling spindle

Rego type chuck 〔ER11/AR11〕

GSC607

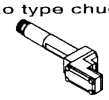
end-drilling spindle

Rego type chuck 〔ER11/AR11〕



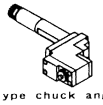
GSE507

Rego type chuck (short) 〔ER11/AR11〕



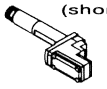
GSE707

Rego type chuck angle-adjustable type(±6°) 〔ER11/AR11〕



GSA207

Rego type chuck angle-adjustable type(±10°) (short) 〔ER11/AR11〕



GSA307

center drill

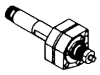
drill

tap

end milling tool

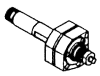
SLITTING SPINDLE

1/2〔φ50×φ12.7/φ15.875〕



GSS150

1/2〔φ50×φ13/φ16〕



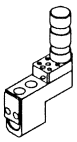
GSS250

slitting cutter

Ⓐ

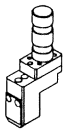
VERTICAL SLEEVE HOLDER

end-drilling 1-tool holder (φ19.05)



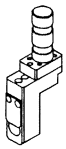
GDF901

end-drilling 1-tool holder (φ19.05) (short)



GDF907

end-drilling 2-tool holder (φ19.05) (short)



GDF1007

SHIFT TOOL HOLDER

shift tool holder (5mm shift, quill-type) 〔□10, □3/8"〕

GTF6110

VERTICAL SLEEVE

boring . drill

boring . drill sleeve (short) 〔ER11/AR11〕

BDS507

boring . drill sleeve (semi-short) 〔ER11/AR11〕

BDS607

boring . drill sleeve 〔ER11/AR11〕

VDS506

boring . drill sleeve (short) 〔ER11/AR11〕

BDS707

tap

tap sleeve (short) 〔ER11/AR11〕

BNS407

tap sleeve (short) 〔ER11/AR11〕

BNS507

center drill

drill

tap

end milling tool

tap

VERTICAL SLEEVE

boring . drill

boring . drill sleeve (short) 〔ER11/AR11〕

BDS807

center drill

drill

tap

end milling tool

front turning tool

back turning tool

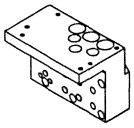
threadding tool

cut-off tool

grooving tool

FOR FRONT MACHINNING HOLDER(BACK):Type VI

3-tool rotary tool holder



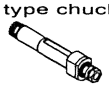
U31B

ROTARY TOOL

cross-milling spindle

Rego type chuck (ER11/AR11)

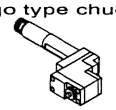
GSC607



end-drilling spindle

Rego type chuck (ER11/AR11)

GSE507



Rego type chuck (short)

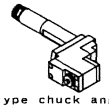
(ER11/AR11)

GSE707

Rego type chuck angle-adjustable type(±6°)

(ER11/AR11)

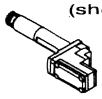
GSA207



Rego type chuck angle-adjustable type(±10°)

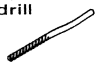
(short) (ER11/AR11)

GSA307



center drill

drill

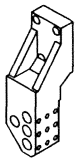


tap

end milling tool

front page (A) reference

end-drilling 4-tool holder (φ19.05)



GDF905

VERTICAL SLEEVE

boring . drill

boring . drill sleeve

(short) (ER11/AR11)

BDS507

boring . drill sleeve

(semi-short) (ER11/AR11)

BDS607

boring . drill sleeve

(ER11/AR11)

VDS506

boring . drill sleeve

(short) (ER16/AR16)

BDS508

boring . drill sleeve

(semi-short) (ER16/AR16)

BDS610

boring . drill sleeve

(ER16/AR16)

VDS110

boring . drill sleeve

(short) (ER11/AR11)

BDS707

boring . drill both end

boring . drill sleeve

(short) (ER11/AR11)

HDS5406

boring . drill sleeve

(ER11/AR11)

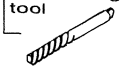
HDS5506

center drill

drill

tap

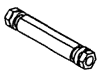
end milling tool



boring . drill sleeve

(semi-long) (ER11/AR11)

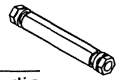
HDS5706



boring . drill sleeve

(long) (ER11/AR11)

HDS5806



center drill

drill

tap

end milling tool

tap . die

tap.die sleeve

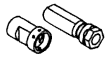
(short) (ER11/AR11)

BNS407

tap.die sleeve

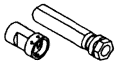
(short) (ER11/AR11)

BNS507



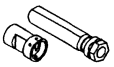
tap.die sleeve (ER11/AR11)

VNS406



tap.die sleeve (ER11/AR11)

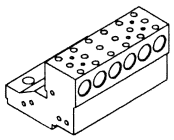
VNS506



tap

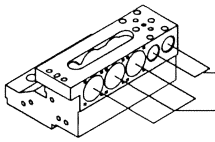
FRONT / BACK DRILLING HOLDER

front / back 6-drill holder



U120B

front / back rotary tool holder



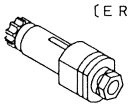
U121B

U121B

ROTARY TOOL

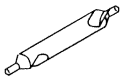
end-drilling spindle

Rego type chuck
〔ER11/AR11〕



GSE807

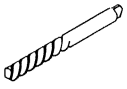
center drill



drill

tap

end milling tool



DEDICATED ADAPTER

dedicated adapter
(φ33−φ19.05)

SAU319

SLEEVE

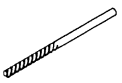
boring . drill

boring . drill sleeve
(short) 〔ER11/AR11〕

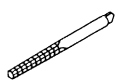
BDS707

center drill

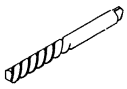
drill



tap



end milling tool



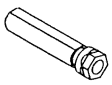
SLEEVE

boring . drill

boring . drill sleeve
(short) 〔ER11/AR11〕
BDS507

boring . drill sleeve
(semi-short) 〔ER11/AR11〕
BDS607

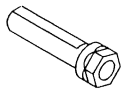
boring . drill sleeve
〔ER11/AR11〕
VDS506



boring . drill sleeve
(short) 〔ER16/AR16〕
BDS508

boring . drill sleeve
(semi-short) 〔ER16/AR16〕
BDS610

boring . drill sleeve
〔ER16/AR16〕
VDS110

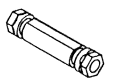


boring . drill sleeve
(short) 〔ER11/AR11〕
BDS707

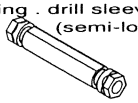
boring . drill both end

boring . drill sleeve
(short) 〔ER11/AR11〕
HDS5406

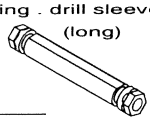
boring . drill sleeve
〔ER11/AR11〕
HDS5506



boring . drill sleeve
(semi-long) 〔ER11/AR11〕
HDS5706

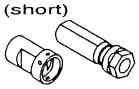


boring . drill sleeve
(long) 〔ER11/AR11〕
HDS5806

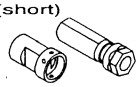


tap . die

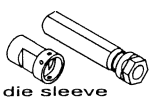
tap . die sleeve
(short) 〔ER11/AR11〕
BNS407



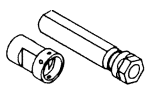
tap . die sleeve
(short) 〔ER11/AR11〕
BNS507



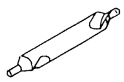
tap . die sleeve
〔ER11/AR11〕
VNS406



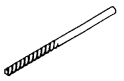
tap . die sleeve
〔ER11/AR11〕
VNS506



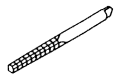
center drill



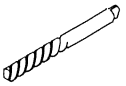
drill



tap

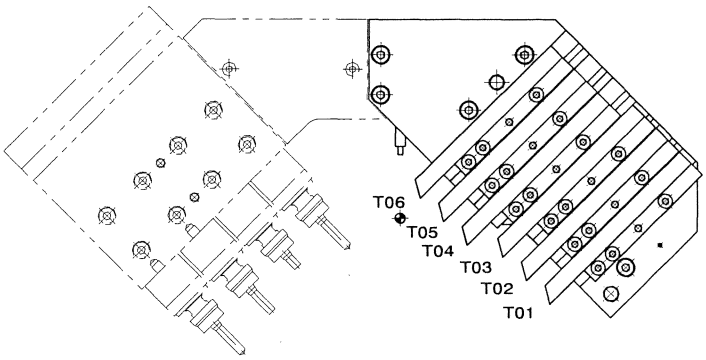


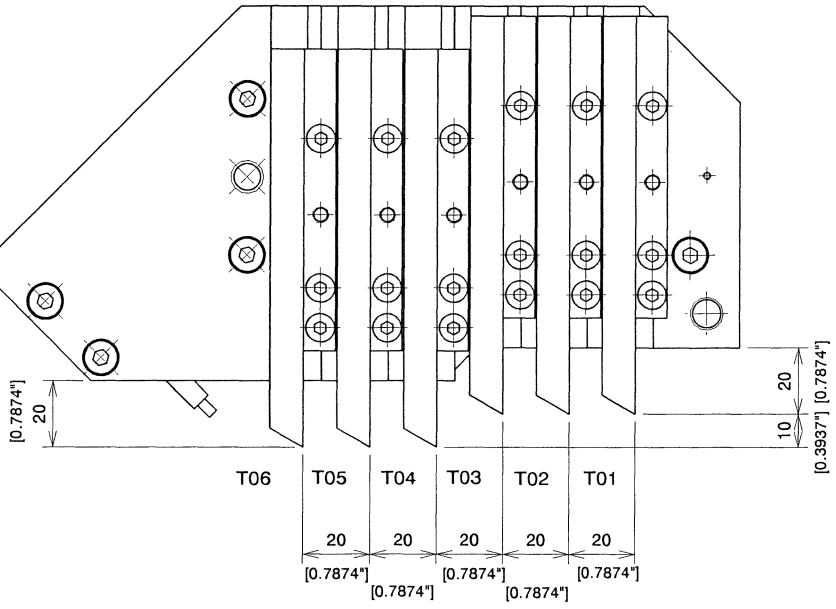
end milling tool



tap

GTF6010 6-Tool Vertical Holder



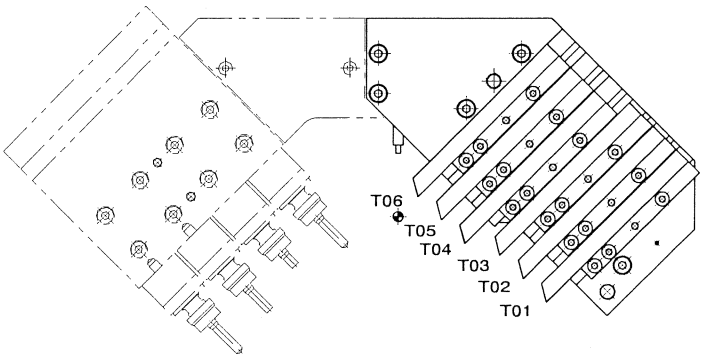


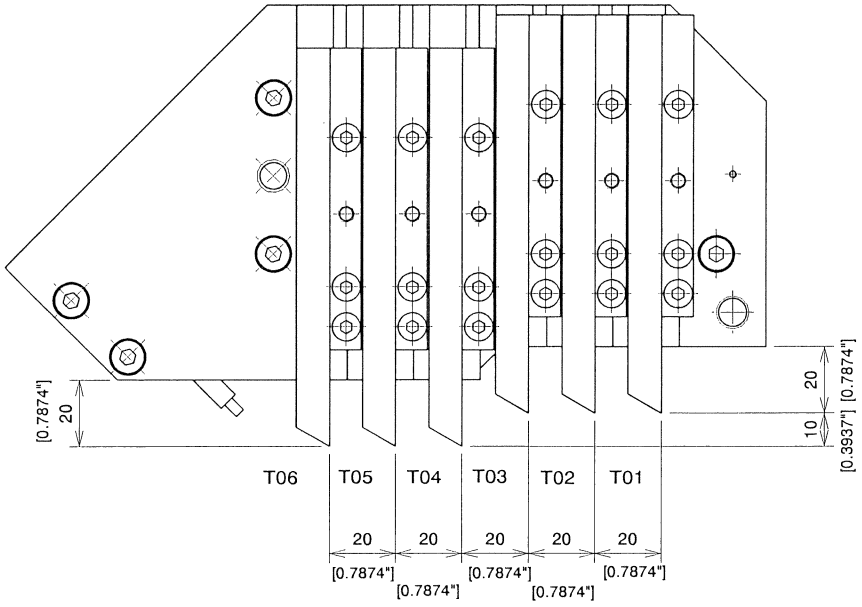
GTF6010 is provided for outer diameter cutting.

NOTICE - Mount the cut-off tool at the position (T06).

Tool holder name	GTF6010
Usage	All types of tools
Tool size	10 x 10 x 120mm
Shift tool holder	HTF5610

GTF6010L 6-Tool Vertical Holder



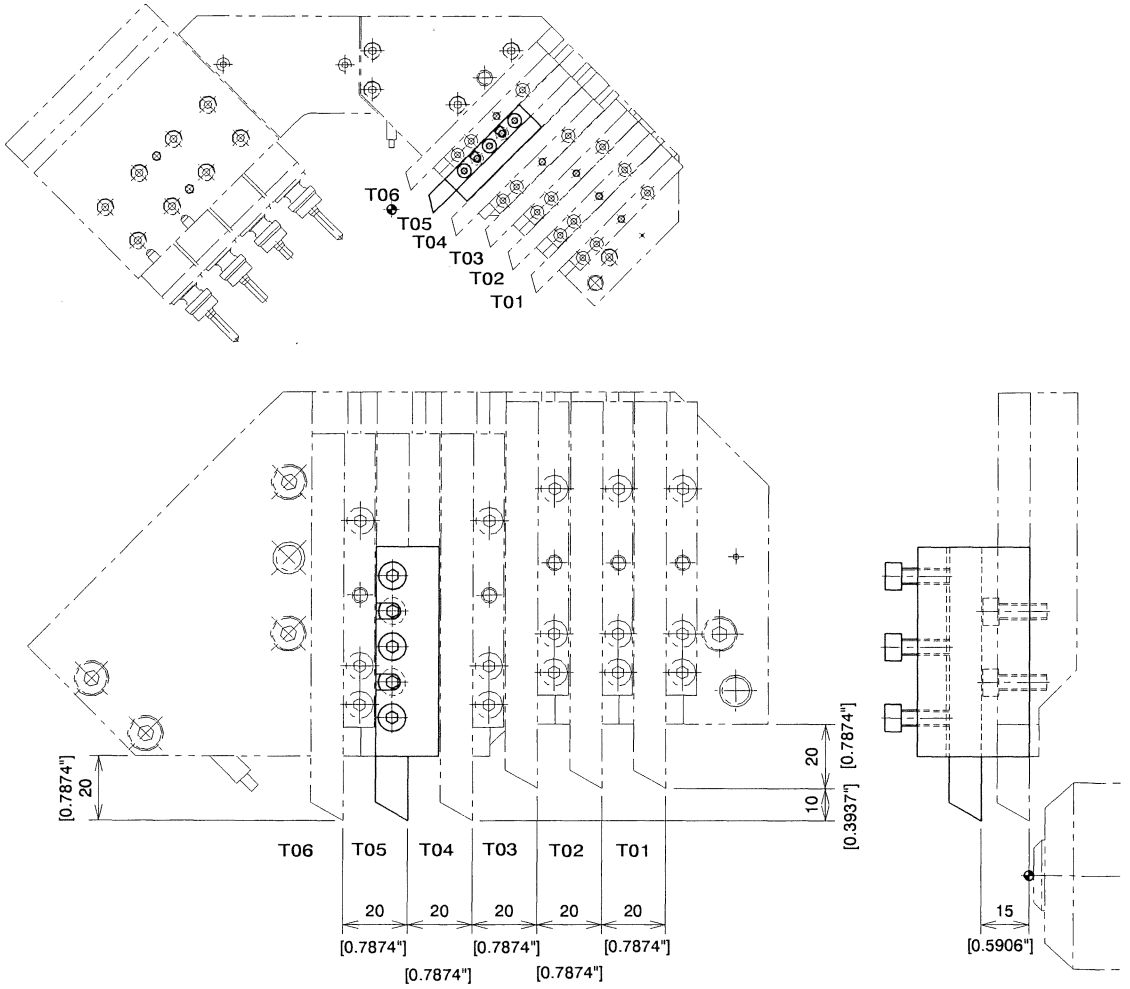


GTF6010L is provided for outer diameter cutting.

NOTICE - Mount the cut-off tool at the position (T06).

Tool holder name	GTF6010L
Usage	All types of tools
Tool size	3/8" x 3/8" x 4-3/4"
Shift tool holder	HTF5610L

HTF5610 Shift Tool Holder(Adapter type)

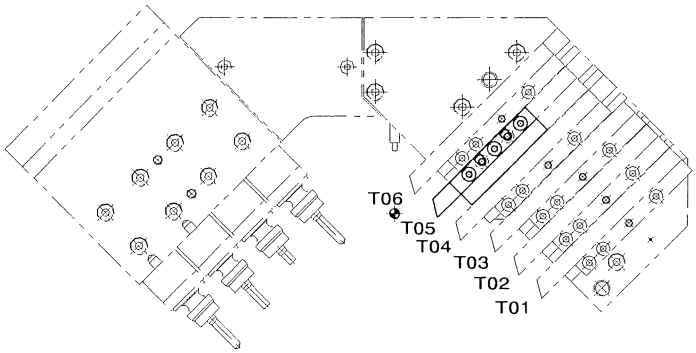


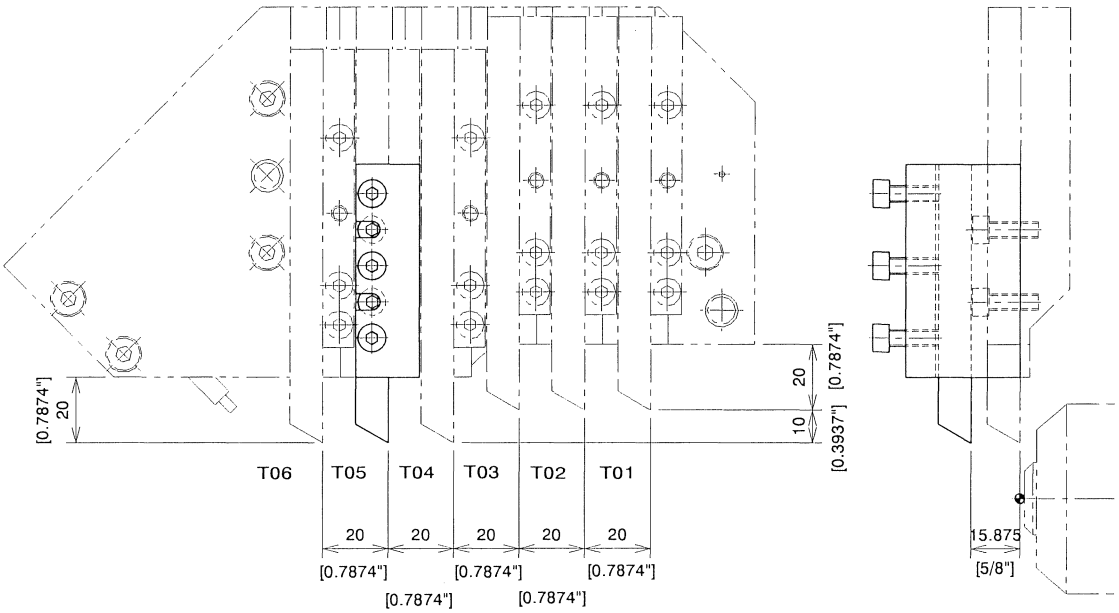
This tool holder is provided for outer cutting. A tool can be mounted shifted 15 mm from the standard mounting position in the longitudinal direction.
Mount this tool holder at a non-T03 tool mounting position when using it with the vertical holder GTF6010.

NOTICE - If this tool holder is mounted, be careful for the tool holder not to interfere with the back spindle or the opposite sleeve.

Tool holder name	HTF5610
Usage	Threading tool
Tool size	10 x 10 x 120mm (15mmShift)

HTF5610L Shift Tool Holder(Adapter type)





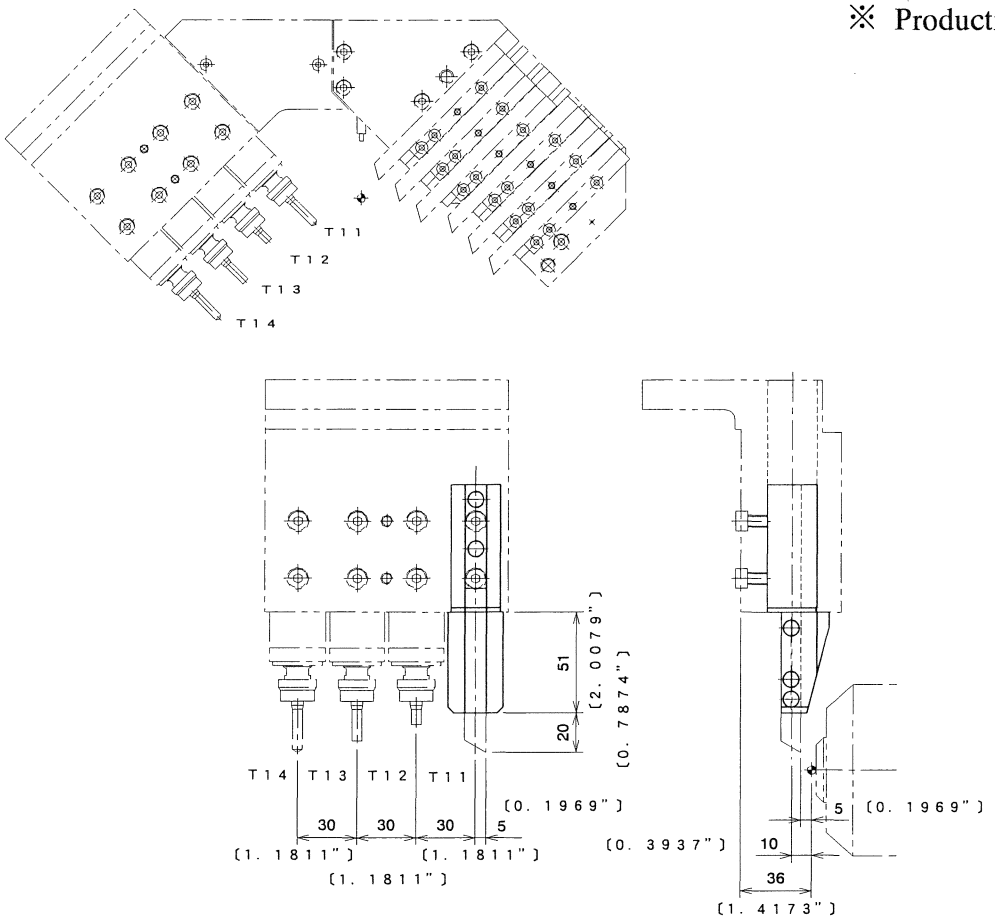
This tool holder is provided for outer cutting. A tool can be mounted shifted 15.875(5/8")mm from the standard mounting position in the longitudinal direction.
Mount this tool holder at a non-T03 tool mounting position when using it with the vertical holder GTF6010L.

NOTICE - If this tool holder is mounted, be careful for the tool holder not to interfere with the back spindle or the opposite sleeve.

Tool holder name	HTF5610L
Usage	Threading tool
Tool size	3/8" x 3/8" x 4-3/4" (5/8" Shift)

GTF6110 Shift Tool Holder(Quill type)

※ Production by order



This tool holder is provided for outer cutting. A tool can be mounted shifted 5(0.1969")mm from the standard mounting position in the longitudinal direction.
Mount this tool holder at a cross milling spindle mounting position when using it with the rotary tool holder (U30B,U31B).

NOTICE - Choose "Free tool layout pattern 1" when you use this holder.
Establish the XY data of the tool which mounted this holder as follows.

In the case of U30B

	X	Y
T11	9.0 (23.0)	-29.5 (-34.5)
T12	9.0 (23.0)	-59.5 (-64.5)
T13	9.0 (23.0)	-89.5 (-94.5)
T14	9.0 (23.0)	-119.5(-124.5)

In the case of U31B(type VI)

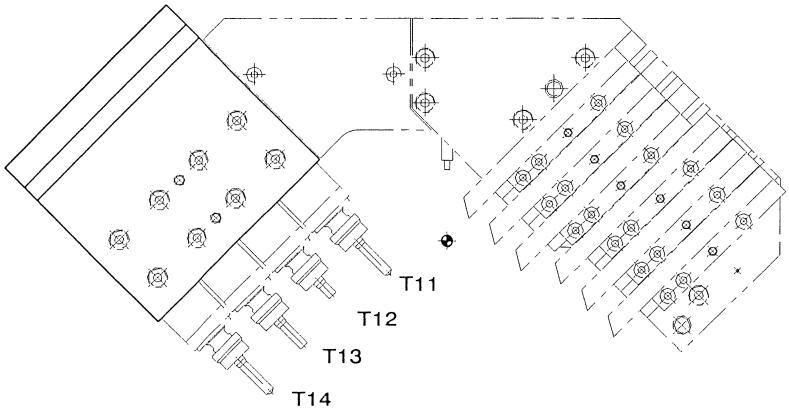
	X	Y
T11	9.0 (23.0)	-2.0 (-7.0)
T12	9.0 (23.0)	-32.0 (-37.0)
T13	9.0 (23.0)	-62.0 (-67.0)

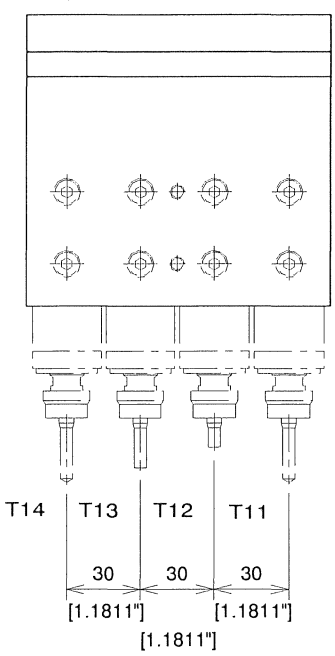
( ) is standard setup value.

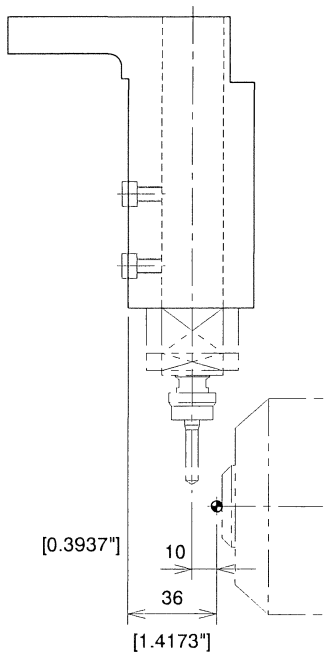
- When the tool of this holder is fixed, you must remove the tool of the tront.

Tool holder name	GTF6110
Usage	All types of tools
Tool size	10x10x80mm(5mm Shift) 3/8"x3/8"x3"(0.1969" Shift)

U30B 4-Tool Rotary Tool Holder(4 Rotary Tools)





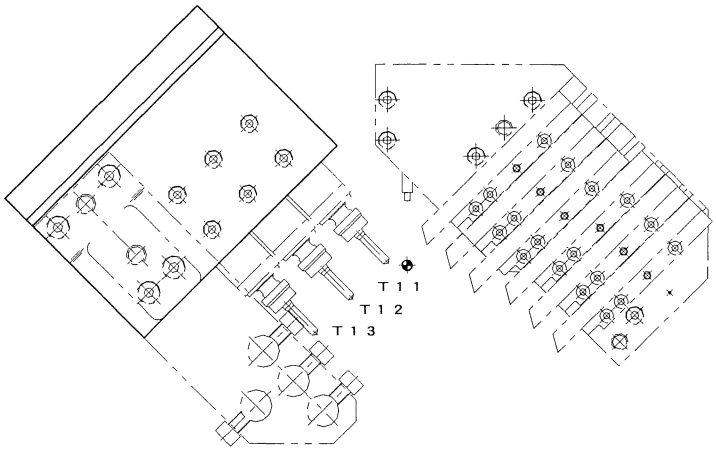


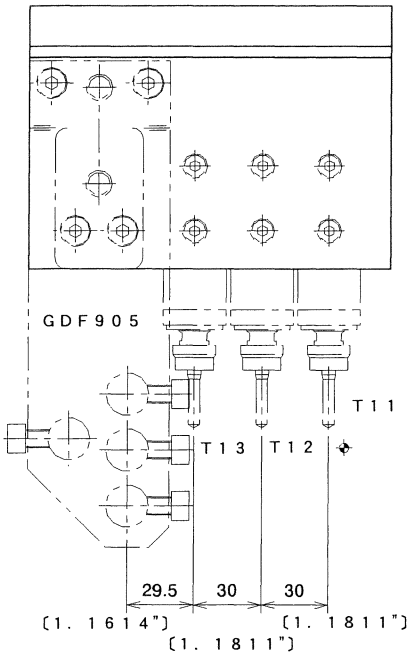
Four rotary tools can be mounted at positions shifted 10 mm.

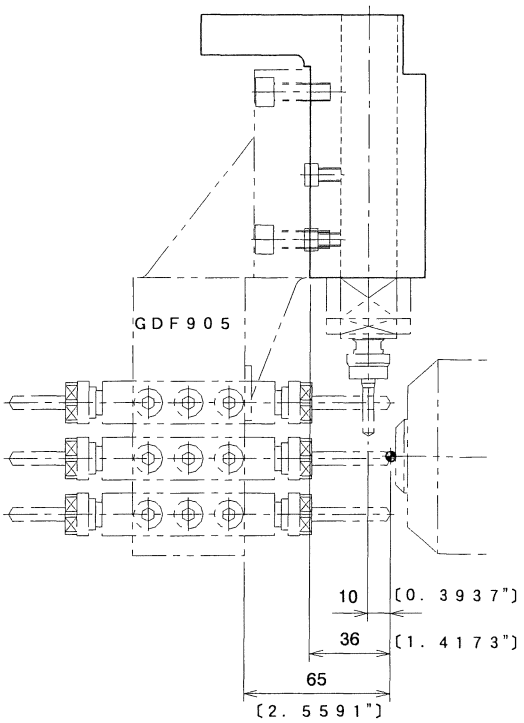
NOTICE - If end milling is performed with the leftmost tool (T14), stroke restrictions are imposed. As a result, the machine is unable to perform D-cut machining. Use a tool T11-T13 to perform end milling.

Tool holder name (attachment name)	U30B
Rotary tool	GSC607,GSE507,GSE707,GSA207,GSA307
End-drilling sleeve holder	GDF901,GDF907
Shift tool holder	GTF6110

U31B 3-Tool Rotary Tool Holder(3 Rotary Tools : Type VI)





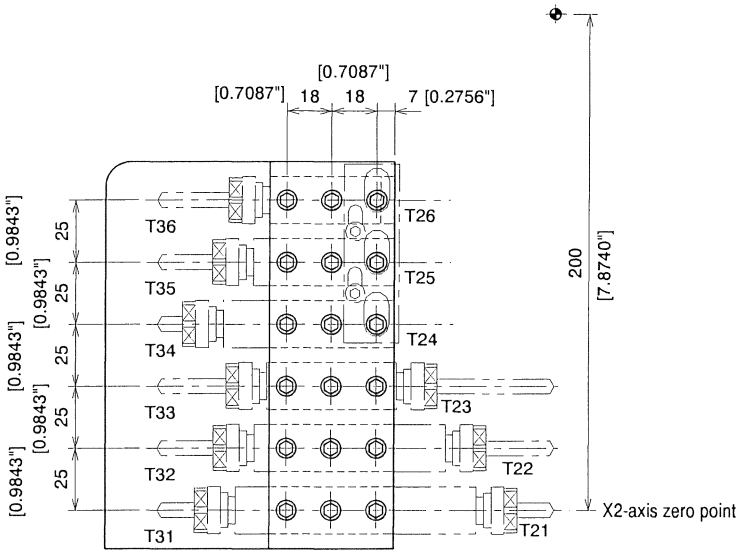


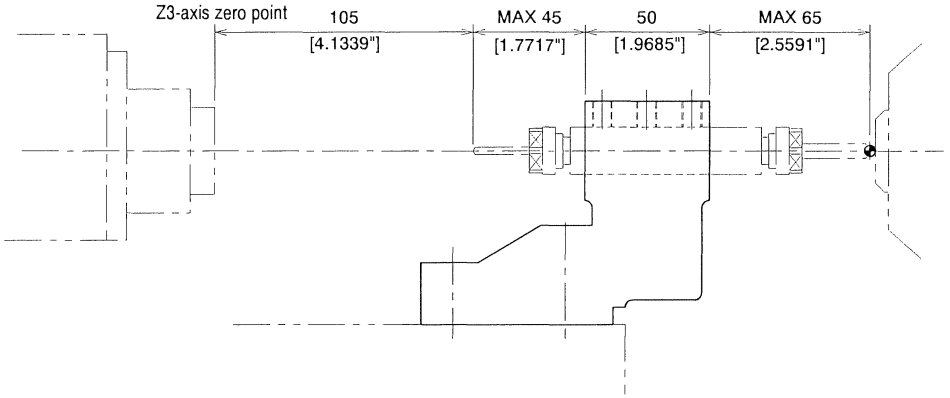
Three rotary tools can be mounted at positions shifted 10 mm.

NOTICE - If end milling is performed with the leftmost tool (T13), be careful for GDF905 not to interfere with the work piece.

Tool holder name (attachment name)	U31B
Rotary tool	GSC607,GSE507,GSE707,GSA207,GSA307
End-drilling sleeve holder	GDF901,GDF907
Shift tool holder	GTF6110

U120B Front/Back 6-Drill Holder



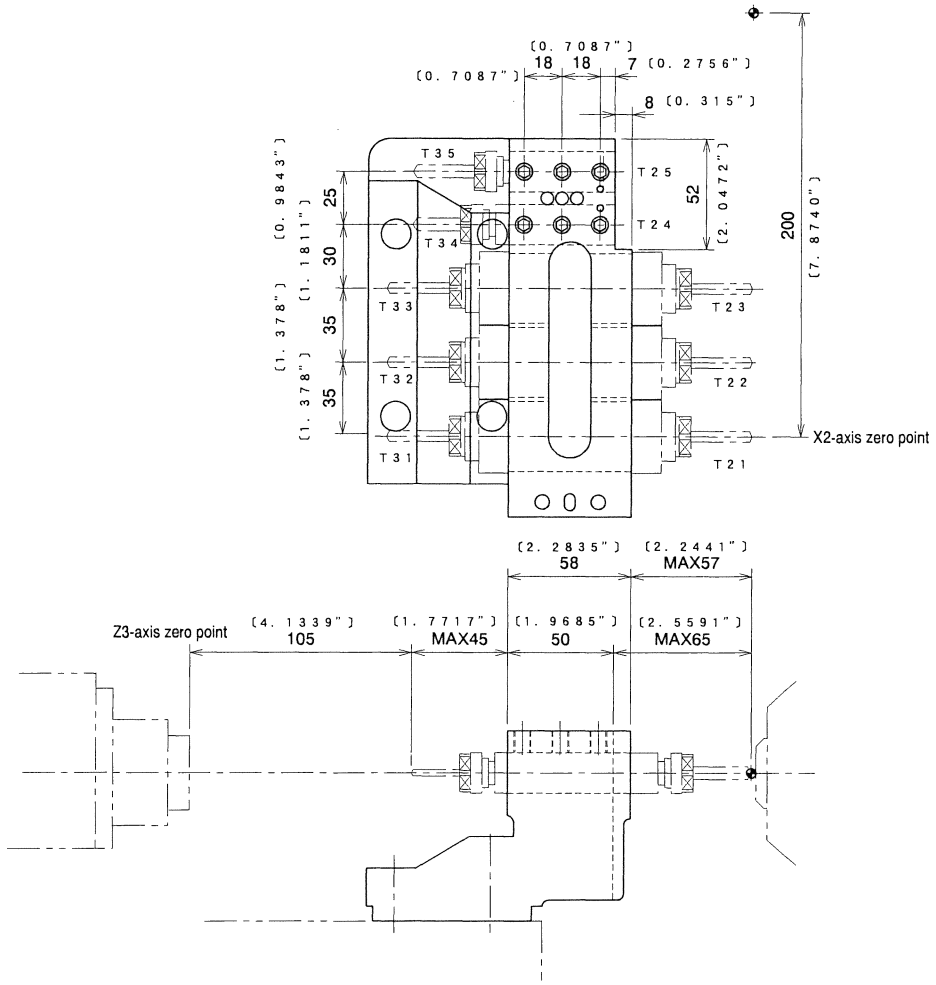


Six front drill sleeves and back drill sleeves can be mounted.
Use of two-way sleeves enables mounting of six front drills and six back drills at maximum.

NOTICE - Do not mount the sleeve for ER16 and the sleeve for AR16 next to each other. A tool cannot also be mounted on the opposite side of the station at which the sleeve for ER16 or AR16 is mounted. (There is no two-way sleeve for ER16 or AR16.)

Tool holder name (attachment name)	U120B
Usage	BDS507,BDS508,BDS607,BDS610,BDS707 VDS506,VDS110,LDS107,LDS110 HDS5406,HDS5506,HDS5706,HDS5806 BNS407,BNS507,VNS406,VNS506
Mounting hole diameter of sleeve	ø19.05mm(ø3/4")

U121B Front/Back Rotary Tool Holder

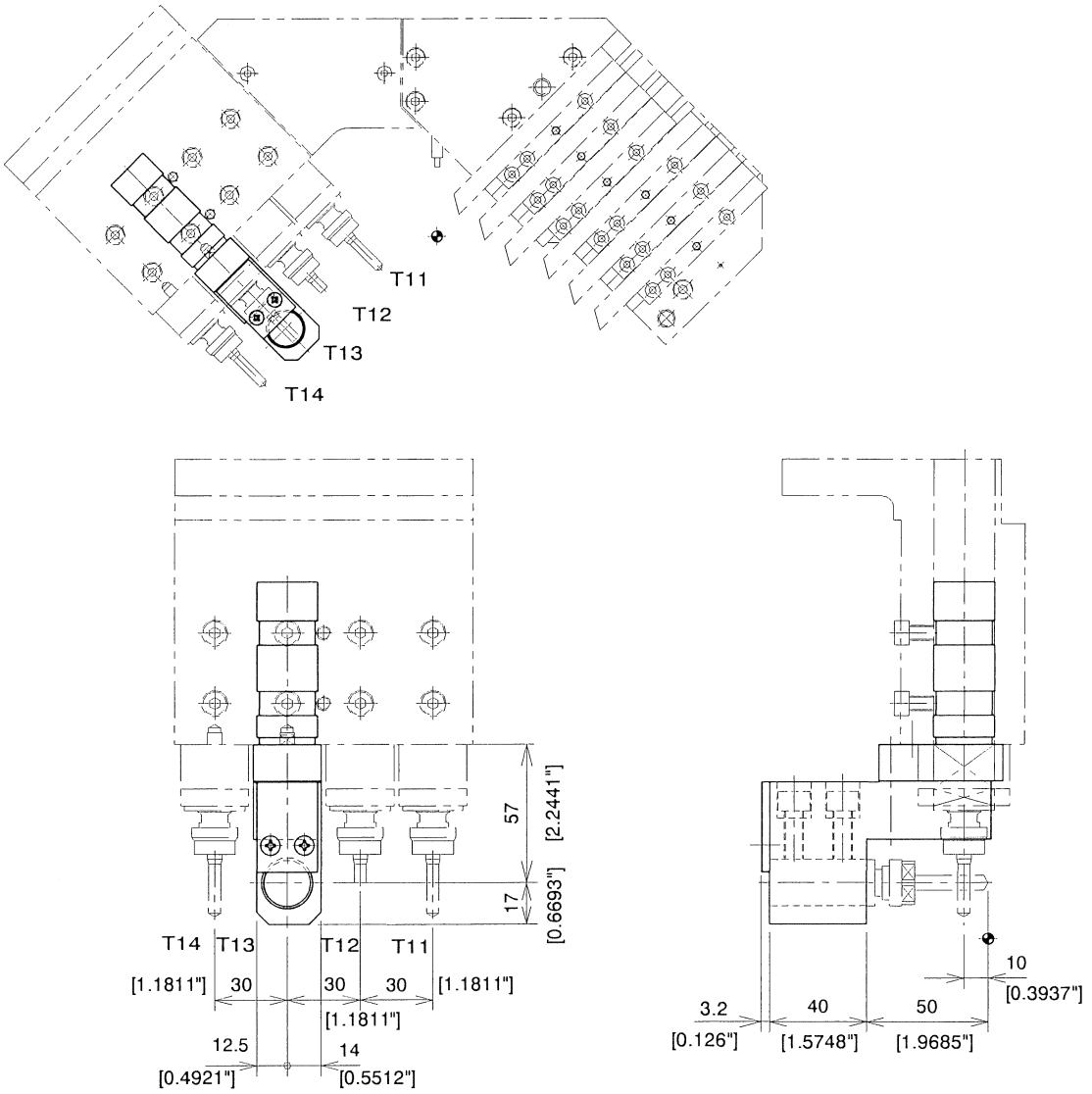


Rotary tool (GSE807) can be mounted at the position T21,T22,T23,T31,T32 or T33. Dedicated adapter provided to mount a fixed-type sleeve (BDS707) when rotary tools are not used. Fixed-type sleeve can be mounted at the position T24,T25,T34 or T35.

NOTICE - Do not mount the sleeve for ER16 and the sleeve for AR16 next to each other. A tool cannot also be mounted on the opposite side of the station at which the sleeve for ER16 or AR16 is mounted. (There is no two-way sleeve for ER16 or AR16.)

Tool holder name (attachment name)	U121B	
Usage	‹T21,T22,T23, T31,T32,T33› BDS707	<T24,T25,T34,T35> BDS507,BDS508,BDS607,BDS610, BDS707 VDS506,VDS110,LDS107,LDS110 HDS5406,HDS5506,HDS5706,HDS5806 BNS407,BNS507,VNS406,VNS506
Rotary tool	‹T21,T22,T23, T31,T32,T33› GSE807	<T24,T25,T34,T35> ———
Dedicated adapter	‹T21,T22,T23, T31,T32,T33› SAU319	<T24,T25,T34,T35> ———
Mounting hole diameter of sleeve	ø19.05mm(ø3/4")	

GDF901 End-Drilling 1-Tool Holder(Example Of Mounting The U30B)



This sleeve holder is provided for end-face drilling. Mount the sleeve holder on the U30B (holder for 4 rotary tools).

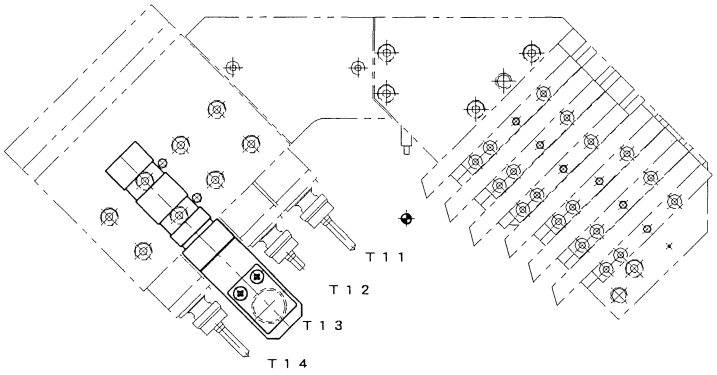
Command format

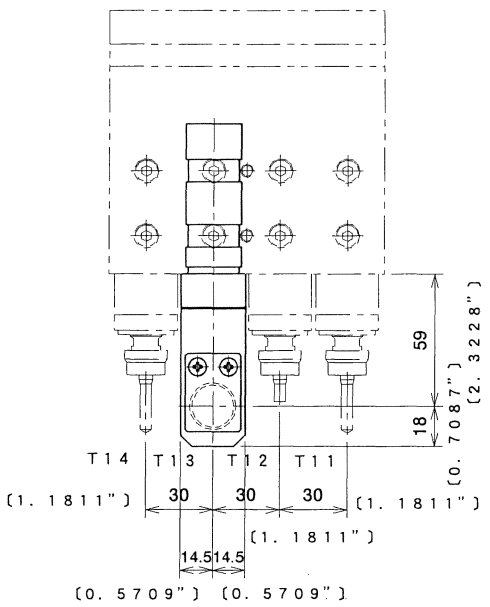
T1300 x 0 ;

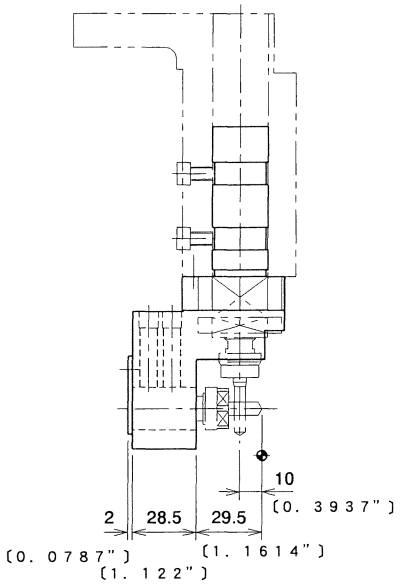
NOTICE
- If performing milling with a rotary tool mounted next to this holder, be careful for the holder not to interfere with the workpiece.
- If this holder is mounted, be careful for the holder not to interfere with the back spindle or the opposite sleeve.
- This holder can be mounted at the position T13 or T14. If using only one holder, mount it at the position T14.

Tool holder name	GDF901
Usage	BDS507,BDS607,BDS707,VDS506 BNS407,BNS507
Mounting hole diameter of sleeve	ø19.05mm(ø3/4")

GDF907 End-Drilling 1-Tool Holder
(Short Type : Example Of Mounting The U30B)







This sleeve holder is provided for end-face drilling. Mount the sleeve holder on the U30B (holder for 4 rotary tools).

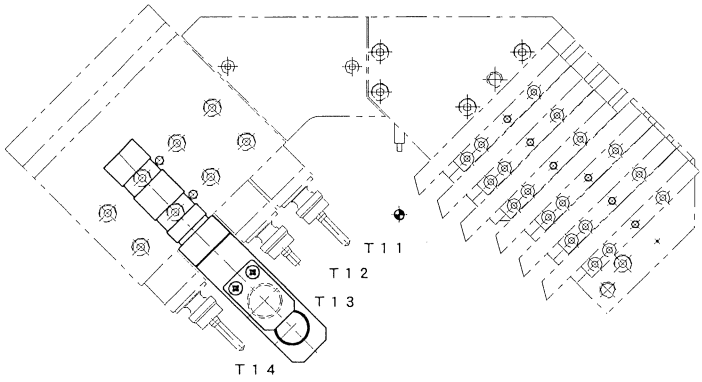
Command format

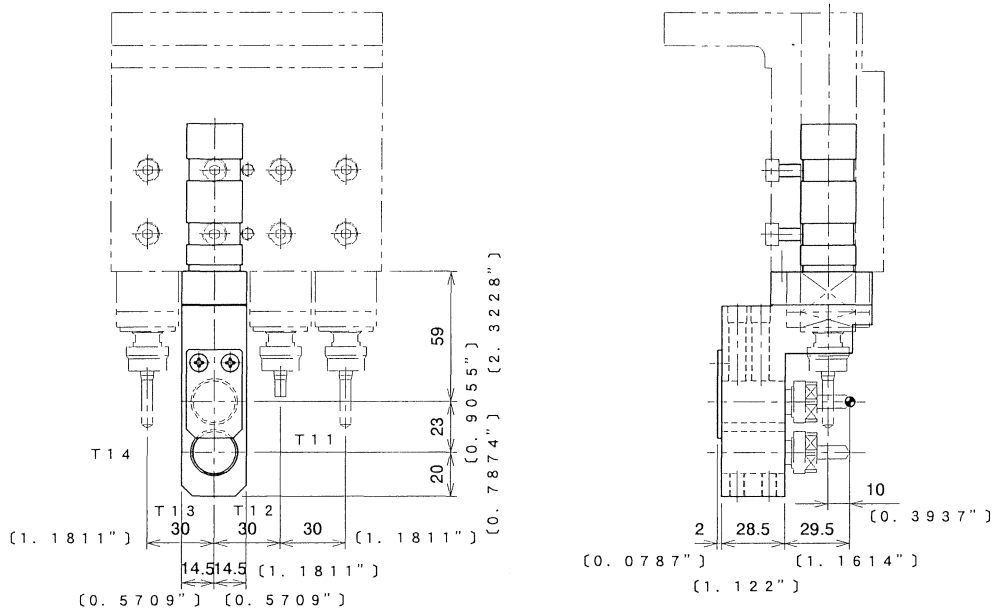
T1300 x 4.0 ;

NOTICE
- If performing milling with a rotary tool mounted next to this holder, be careful for the holder not to interfere with the workpiece.
- If this holder is mounted, be careful for the holder not to interfere with the back spindle or the opposite sleeve.
- This holder can be mounted at the position T13 or T14. If using only one holder, mount it at the position T14.

Tool holder name	GDF907
Usage	BDS807
Mounting hole diameter of sleeve	ø19.05mm(ø3/4")

GDF1007 End-Drilling 2-Tool Holder
(Short Type : Example Of Mounting The U30B)





This sleeve holder is provided for end-face drilling. Mount the sleeve holder on the U30B (holder for 4 rotary tools).

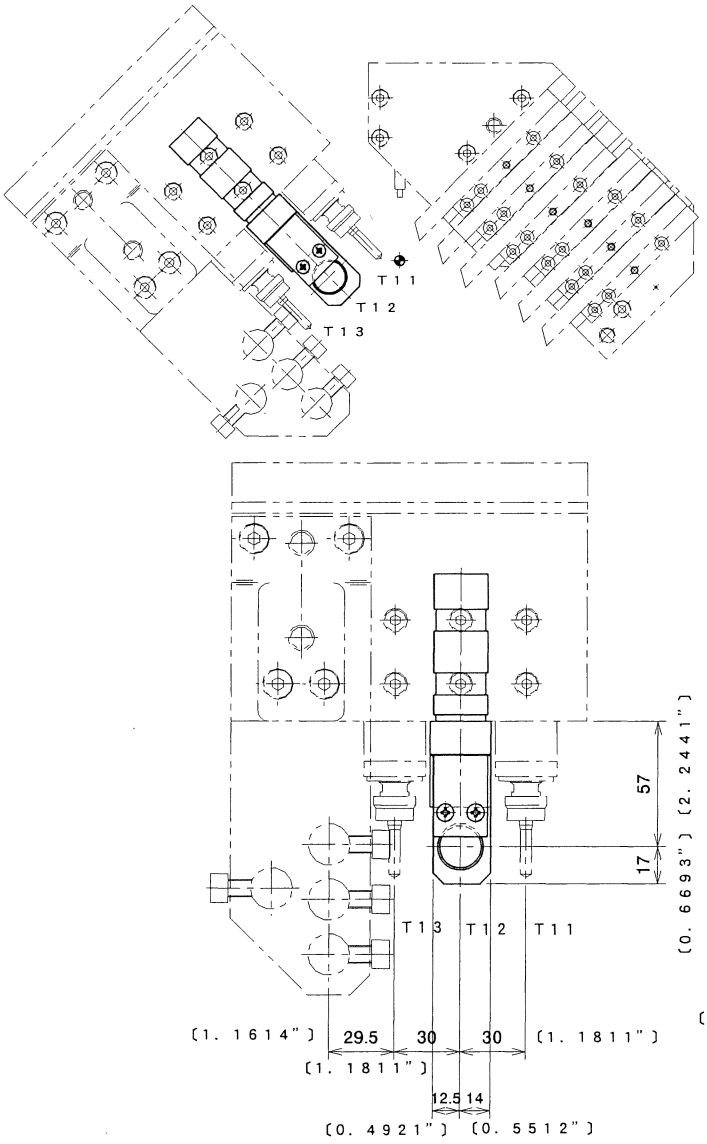
Command format

T1300 X 4.0 ; Upper hole.
T1300 X 50.0 ; Lower hole.

NOTICE
- If performing milling with a rotary tool mounted next to this holder, be careful for the holder not to interfere with the workpiece.
- If this holder is mounted, be careful for the holder not to interfere with the back spindle or the opposite sleeve.
- This holder can be mounted at the position T13 or T14. If using only one holder, mount it at the position T14.

Tool holder name	GDF1007
Usage	BDS807
Mounting hole diameter of sleeve	ø19.05mm(ø3/4")

GDF901 End-Drilling 1-Tool Holder
(Example Of Mounting The U31B : Type VI)



This sleeve holder is provided for end-face drilling. Mount the sleeve holder on the U31B (holder for 3 rotary tools).

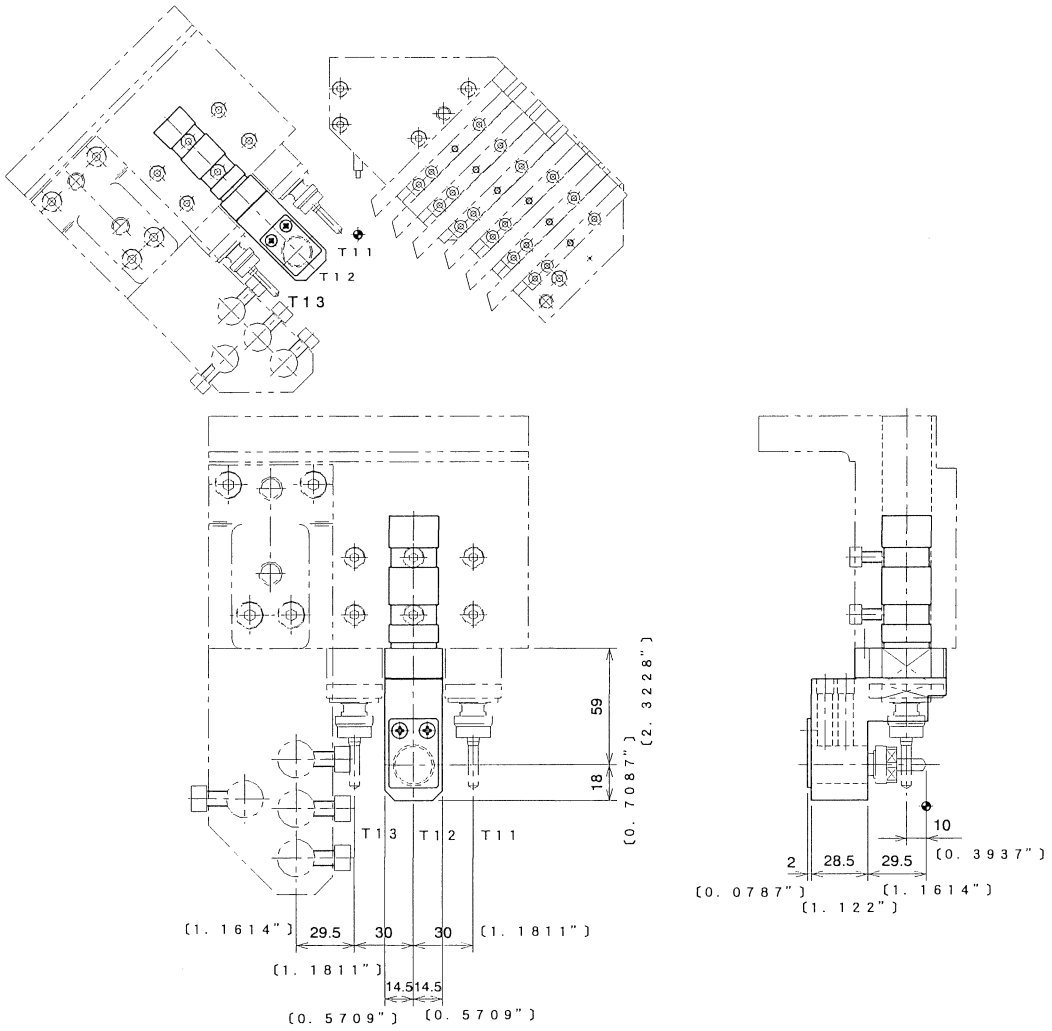
Command format

T1200 x 0 ;

NOTICE
- If performing milling with a rotary tool mounted next to this holder, be careful for the holder not to interfere with the workpiece.
- If this holder is mounted, be careful for the holder not to interfere with the back spindle.
- This holder can be mounted at the position T12.

Tool holder name	GDF901
Usage	BDS507,BDS607,BDS707,VDS506 BNS407,BNS507
Mounting hole diameter of sleeve	ø19.05mm(ø3/4")

GDF907 End-Drilling 1-Tool Holder
(Short Type : Example Of Mounting The U31B : Type VI)



This sleeve holder is provided for end-face drilling. Mount the sleeve holder on the U31B (holder for 3 rotary tools).

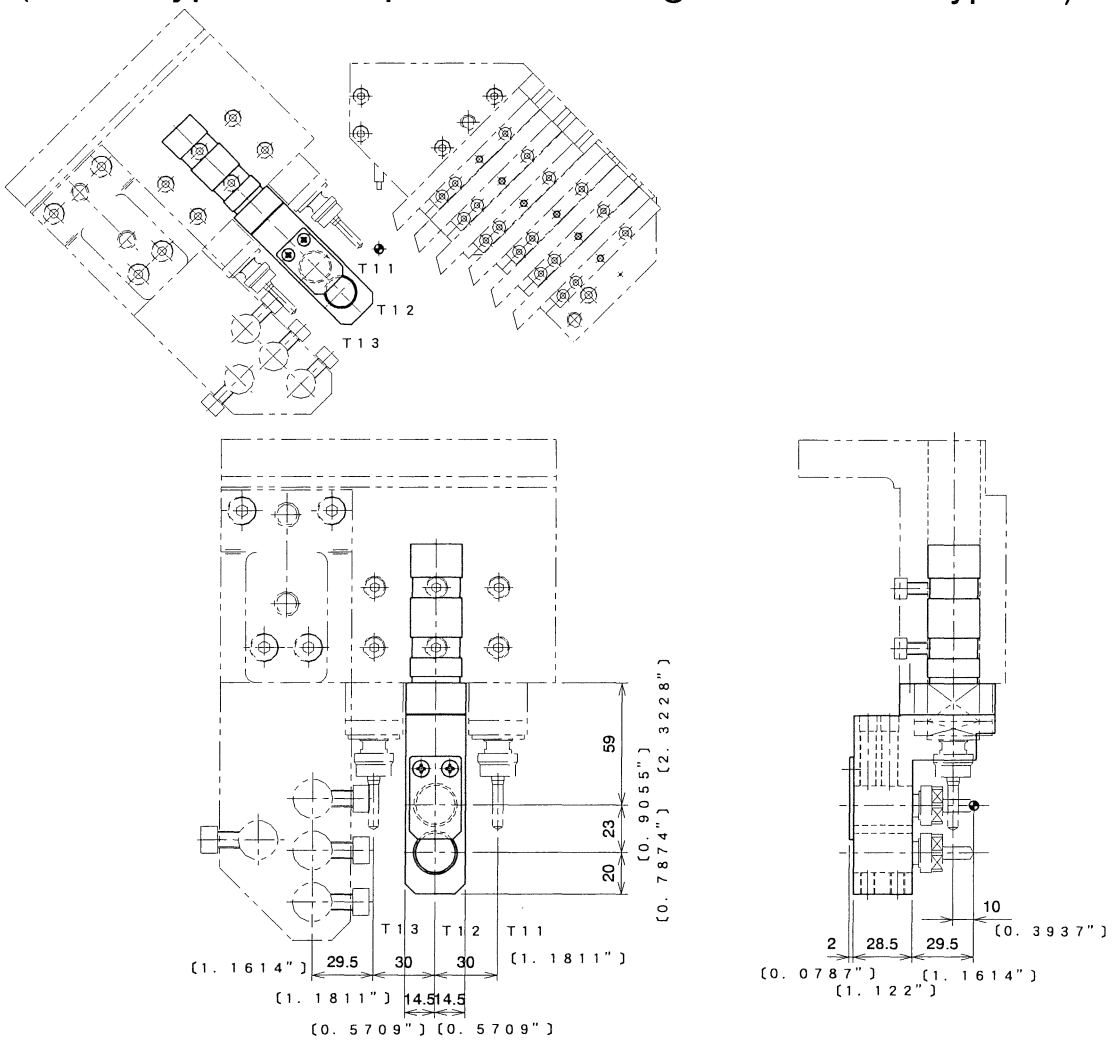
Command format

T1200 x 4.0 ;

NOTICE
- If performing milling with a rotary tool mounted next to this holder, be careful for the holder not to interfere with the workpiece.
- If this holder is mounted, be careful for the holder not to interfere with the back spindle.
- This holder can be mounted at the position T12.

Tool holder name	GDF907
Usage	BDS807
Mounting hole diameter of sleeve	ø19.05mm(ø3/4")

GDF1007 End-Drilling 2-Tool Holder
(Short Type : Example Of Mounting The U31B : Type VI)



This sleeve holder is provided for end-face drilling. Mount the sleeve holder on the U31B (holder for 3 rotary tools).

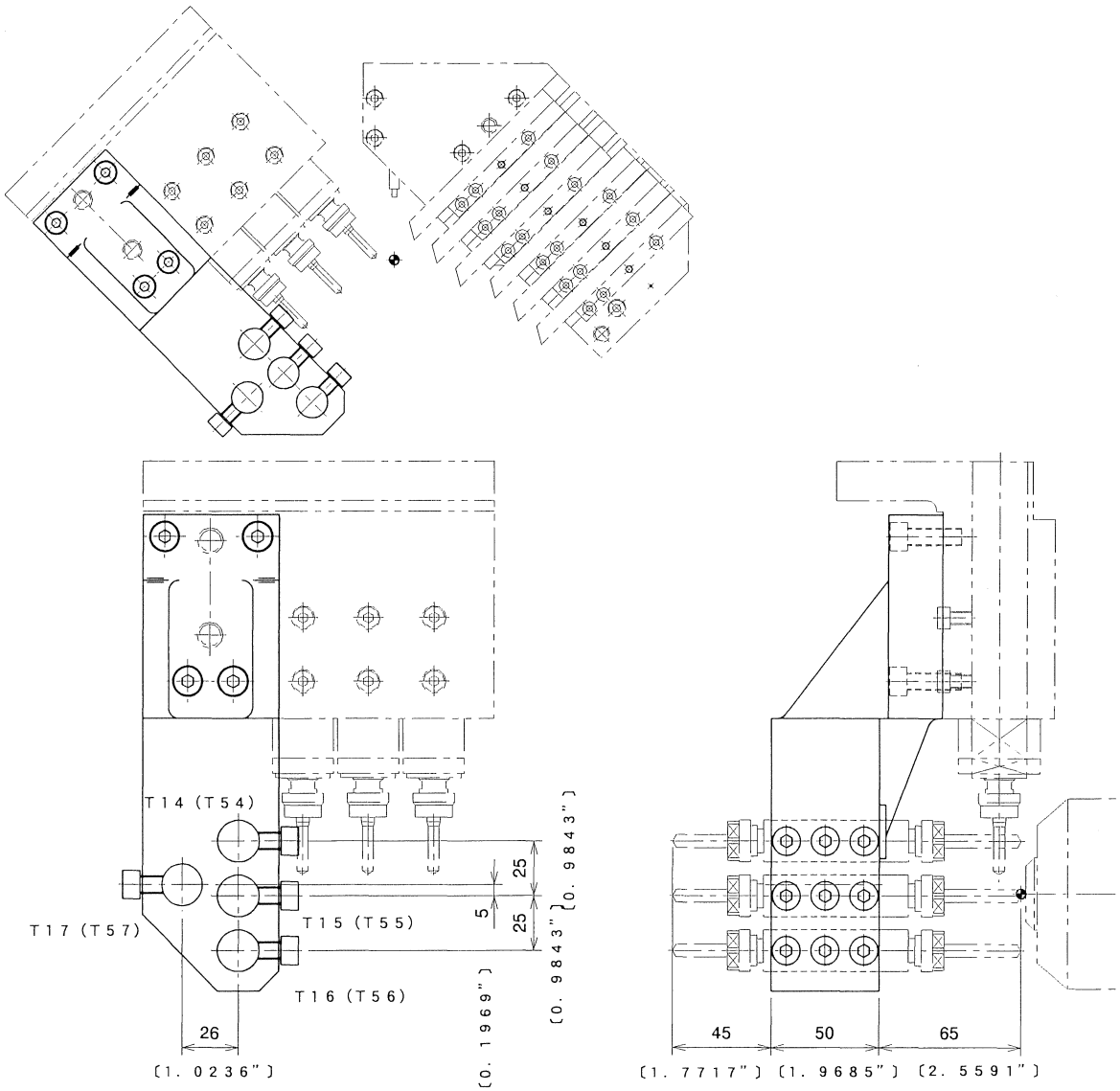
Command format

T1200 X 4.0 ; Upper hole.
T1200 X 50.0 ; Lower hole.

NOTICE
- If performing milling with a rotary tool mounted next to this holder, be careful for the holder not to interfere with the workpiece.
- If this holder is mounted, be careful for the holder not to interfere with the back spindle.
- This holder can be mounted at the position T12.
- Be careful for the holder not to interfere with the material when T13 or GDF905 is selected.

Tool holder name	GDF1007
Usage	BDS807
Mounting hole diameter of sleeve	ø19.05mm(ø3/4")

GDF905 End-Drilling 4-Tool Holder (Type VI)

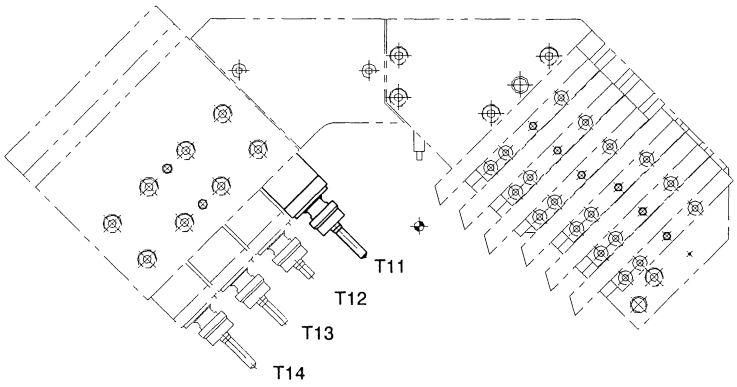


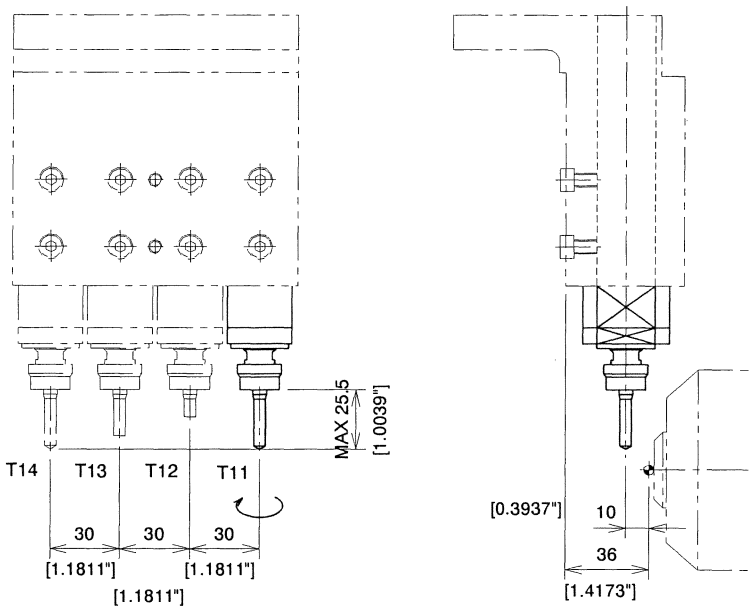
This sleeve holder is provided for end-face drilling. Mount the sleeve holder on the U31B (holder for 3 rotary tools). This sleeve holder is only for type VI.

NOTICE
- If performing milling with a rotary tool mounted next to this holder, be careful for the holder not to interfere with the workpiece.
- Be careful for the holder not to interfere with the back spindle.

Tool holder name	GDF905
Usage	BDS507,BDS508,BDS607,BDS610,BDS707 VDS506,VDS110,LDS107,LDS110 HDS5406,HDS5506,HDS5706,HDS5806 BNS407,BNS507,VNS406,VNS506
Mounting hole diameter of sleeve	ø19.05mm(ø3/4")

GSC607 Cross Milling Spindle (Example Of Mounting The U30B)



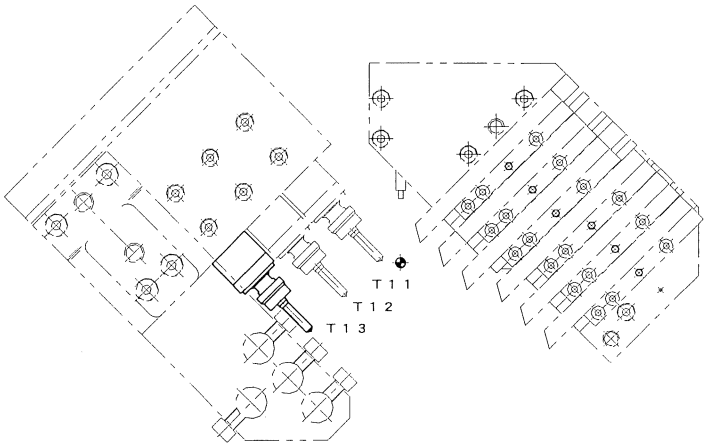


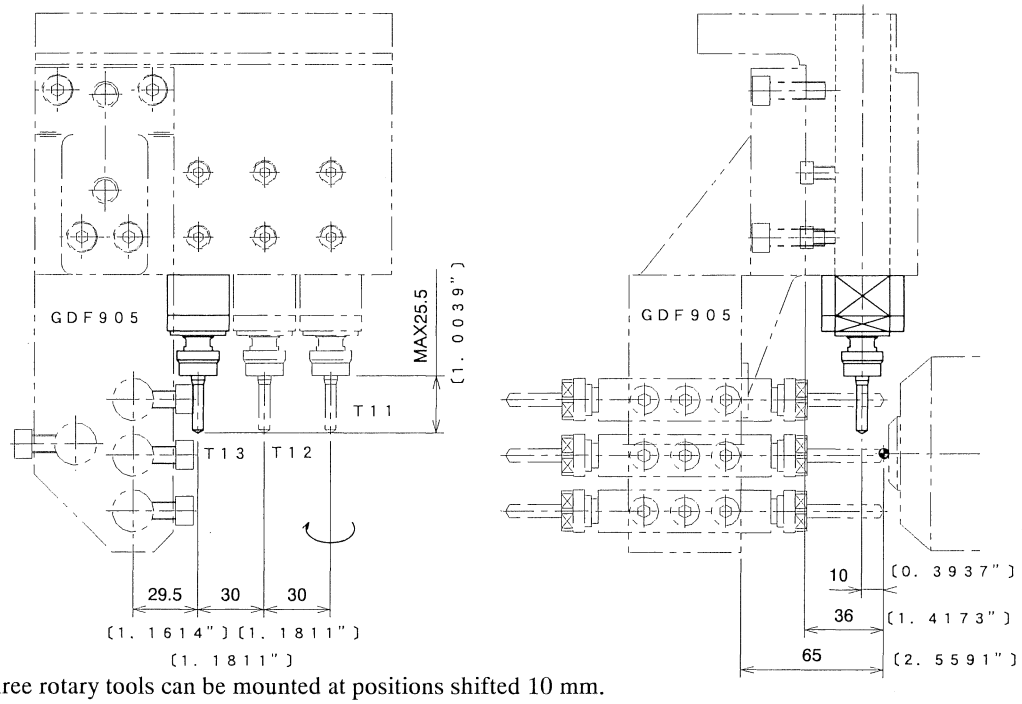
Four rotary tools can be mounted at positions shifted 10 mm.

NOTICE - If end milling is performed with the leftmost tool (T14), stroke restrictions are imposed. As a result, the machine is unable to perform D-cut machining. Use a tool T11-T13 to perform end milling.

Tool holder name	GSC607
Max.chuck dia.	ø7mm[ø0.2756"]
Spindle speed	200 ~ 8,000min^{-1}
Chuck model	ER11, AR11
Spindle rotation direction	Direction shown by the arrow in the figure above for a forward rotation command

GSC607 Cross Milling Spindle (Example Of Mounting The U31B : Type VI)



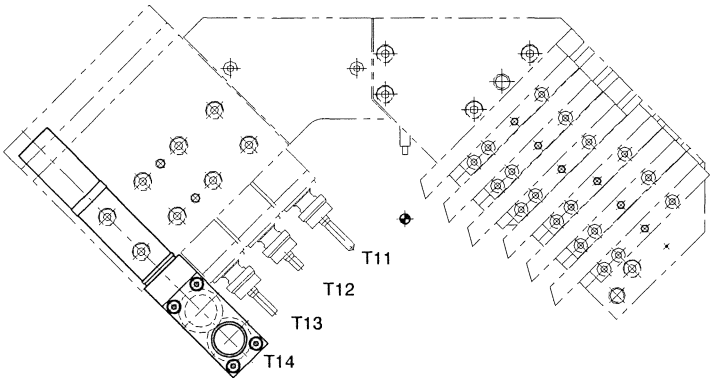


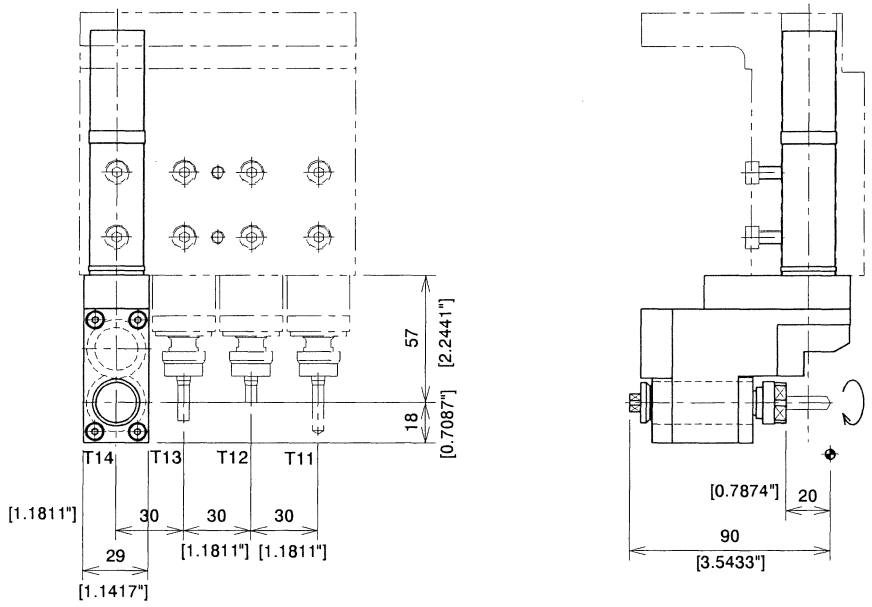
Three rotary tools can be mounted at positions shifted 10 mm.

NOTICE - If end milling is performed with the leftmost tool (T13), be careful for GDF905 not to intertere with the workpiece.

Tool holder name	GSC607
Max.chuck dia.	ø7mm[ø0.2756"]
Spindle speed	200 ~ 8,000min^{-1}
Chuck model	ER11, AR11
Spindle rotation direction	Direction shown by the arrow in the figure above for a forward rotation command

GSE507 End-drilling Spindles (Example Of Mounting The U30B)





This holder is provided for end-face drilling. Mount the sleeve holder on the U30B (holder for 4 rotary tools).

Command format

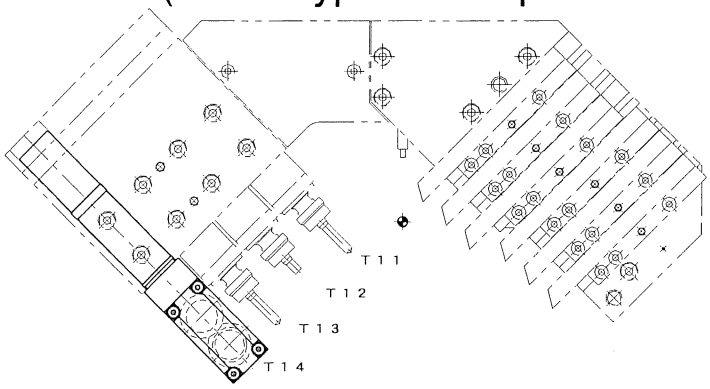
T1400 x 0 ;

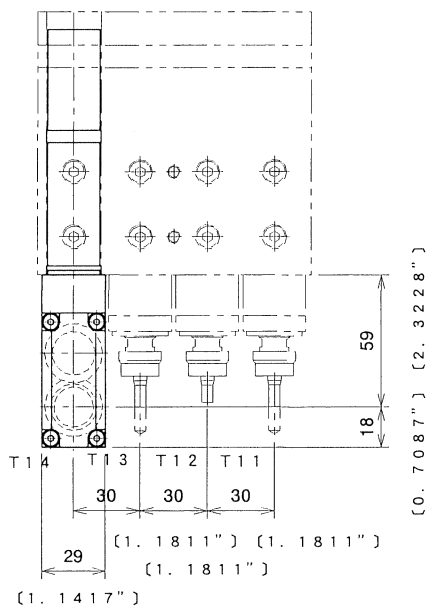
NOTICE
- If performing milling with a rotary tool mounted next to this holder, be careful for the holder not to interfere with the workpiece.
- If this holder is mounted, be careful for the holder not to interfere with the back spindle or the opposite sleeve.
- This holder can be mounted at the position T13 or T14. If using only one holder, mount it at the position T14.

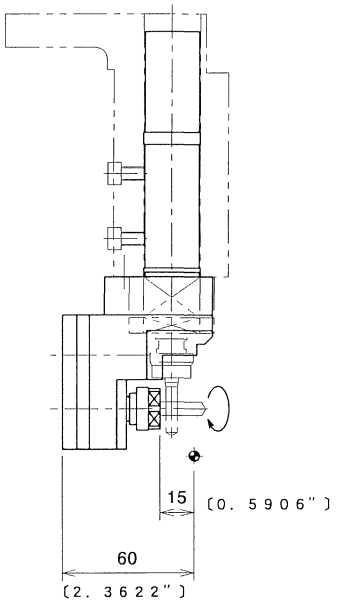
Tool holder name	GSE507
Max.chuck dia.	ø7mm[ø0.2756"]
Spindle speed	200 ~ 8,000min^{-1}
Chuck model	ER11 , AR11
Spindle rotation direction	Direction shown by the arrow in the figure above for a forward rotation command

GSE707 End-drilling Spindles
(Short Type : Example Of Mounting The U30B)

※ Production by order







This holder is provided for end-face drilling. Mount the sleeve holder on the U30B (holder for 4 rotary tools).

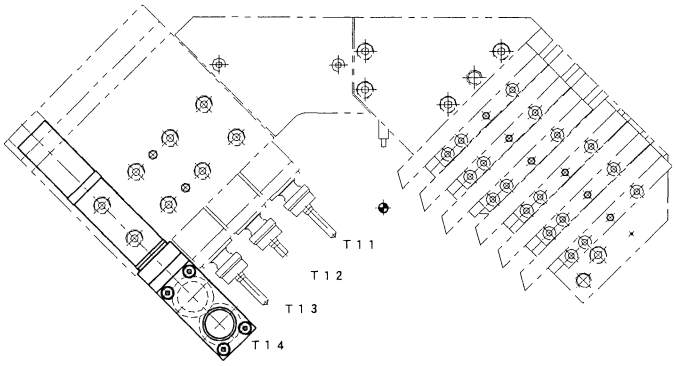
Command format

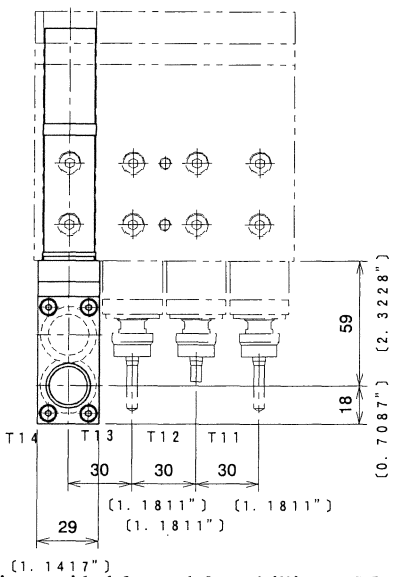
T1400 x 4.0 ;

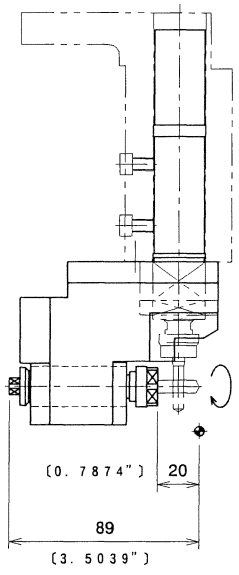
NOTICE
- If performing milling with a rotary tool mounted next to this holder, be careful for the holder not to interfere with the workpiece.
- If this holder is mounted, be careful for the holder not to interfere with the back spindle or the opposite sleeve.
- This holder can be mounted at the position T13 or T14. If using only one holder, mount it at the position T14.

Tool holder name	GSE707
Max.chuck dia.	ø7mm[ø0.2756"]
Spindle speed	200 ~ 8,000min^{-1}
Chuck model	ER11 , AR11
Spindle rotation direction	Direction shown by the arrow in the figure above for a forward rotation command

GSA207 End-drilling Spindles
(Angle-adjustable Type : Example Of Mounting The U30B)







This holder is provided for end-face drilling. Mount the sleeve holder on the U30B (holder for 4 rotary tools).

Command format

T1400 x 4.0 ;

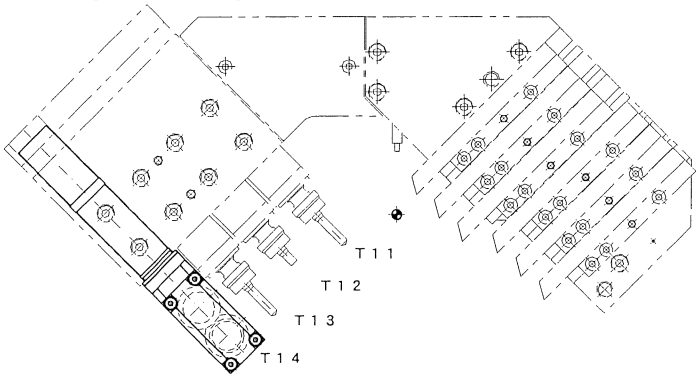
NOTICE
- If performing milling with a rotary tool mounted next to this holder, be careful for the holder not to interfere with the workpiece.
- If this holder is mounted, be careful for the holder not to interfere with the back spindle or the opposite sleeve.
- This holder can be mounted at the position T13 or T14. If using only one holder, mount it at the position T14.

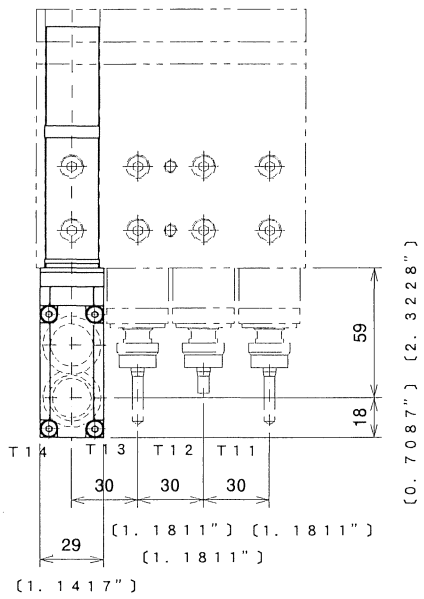
Tool holder name	GSA207
Max.chuck dia.	ø7mm[ø0.2756"]
Spindle speed	200 ~ 8,000min^{-1}
Chuck model	ER11 , AR11
Spindle rotation direction	Direction shown by the arrow in the figure above for a forward rotation command
Angle adjustment	± 6º

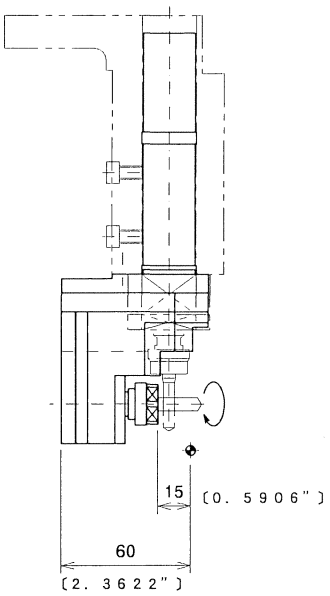
GSA307 End-drilling Spindles

(Short Type : Angle-adjustable Type : Example Of Mounting The U30B)

※ Production by order







This holder is provided for end-face drilling. Mount the sleeve holder on the U30B (holder for 4 rotary tools).

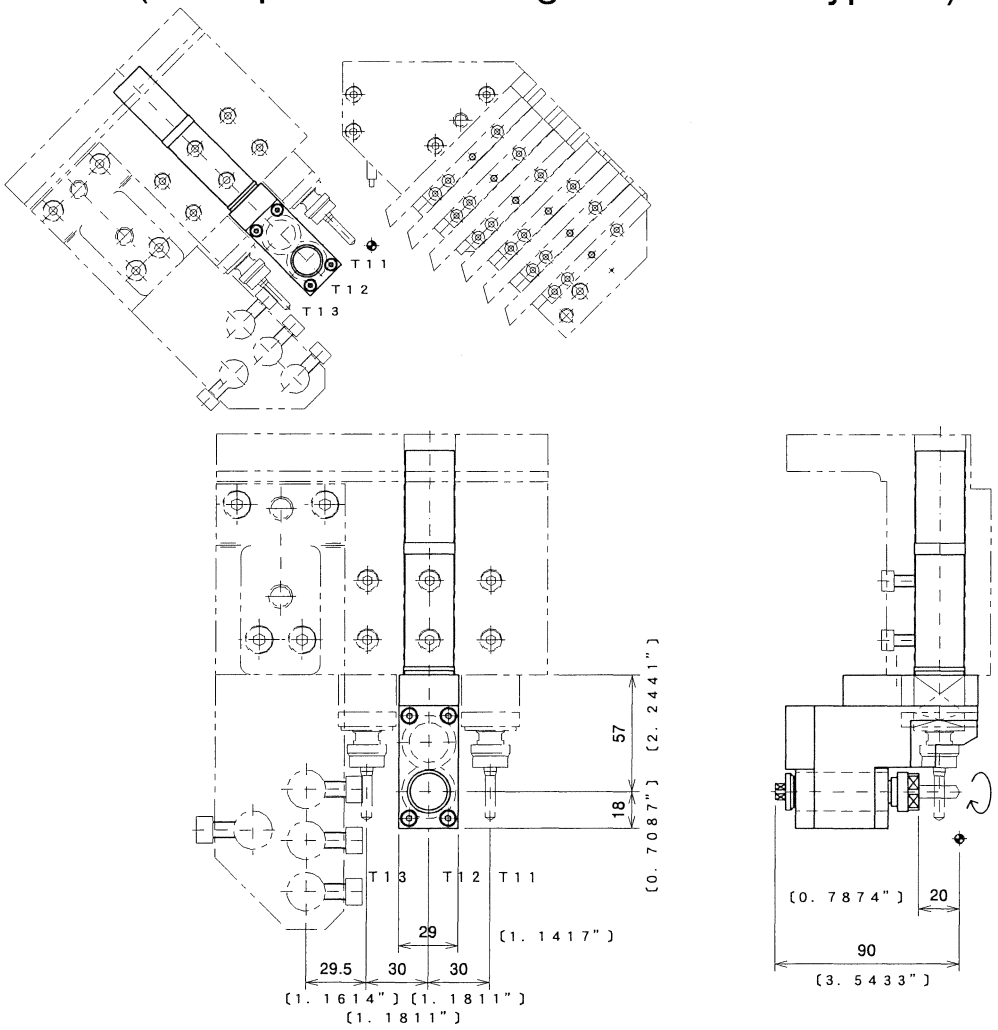
Command format

T1400 x 4.0 ;

NOTICE
- If performing milling with a rotary tool mounted next to this holder, be careful for the holder not to interfere with the workpiece.
- If this holder is mounted, be careful for the holder not to interfere with the back spindle or the opposite sleeve.
- This holder can be mounted at the position T13 or T14. If using only one holder, mount it at the position T14.

Tool holder name	GSA307
Max.chuck dia.	ø7mm[ø0.2756"]
Spindle speed	200 ~ 8,000min^{-1}
Chuck model	ER11 , AR11
Spindle rotation direction	Direction shown by the arrow in the figure above for a forward rotation command
Angle adjustment	± 10º

GSE507 End-drilling Spindles
(Example Of Mounting The U31B : Type VI)



This holder is provided for end-face drilling. Mount the sleeve holder on the U31B (holder for 3 rotary tools).

Command format

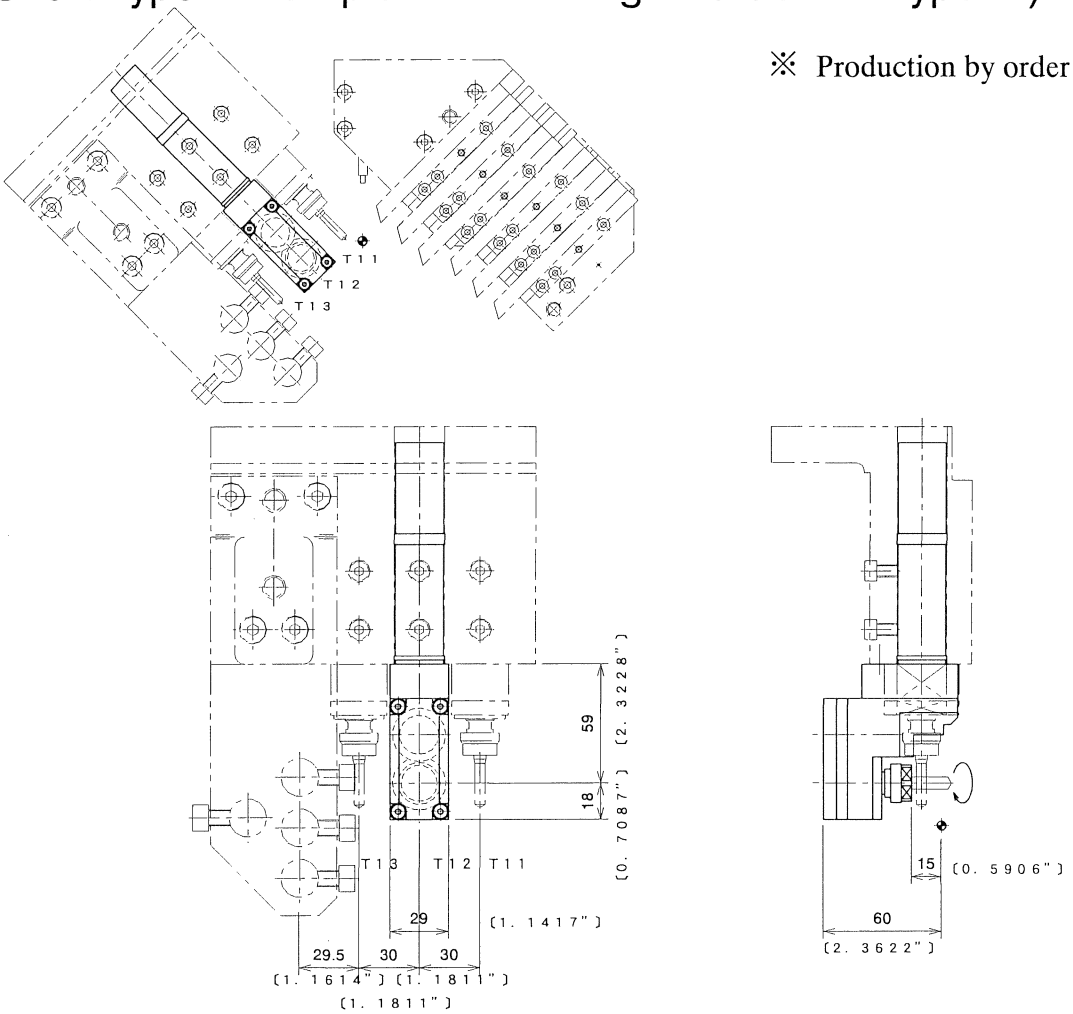
T1200 x 0 ;

NOTICE
- If performing milling with a rotary tool mounted next to this holder, be careful for the holder not to interfere with the workpiece.
- If this holder is mounted, be careful for the holder not to interfere with the back spindle.
- This holder can be mounted at the position T12.

Tool holder name	GSE507
Max.chuck dia.	ø7mm[ø0.2756"]
Spindle speed	200 ~ 8,000min^{-1}
Chuck model	ER11 , AR11
Spindle rotation direction	Direction shown by the arrow in the figure above for a forward rotation command

GSE707 End-drilling Spindles
(Short Type : Example Of Mounting The U31B : Type VI)

※ Production by order



This holder is provided for end-face drilling. Mount the sleeve holder on the U31B (holder for 3 rotary tools).

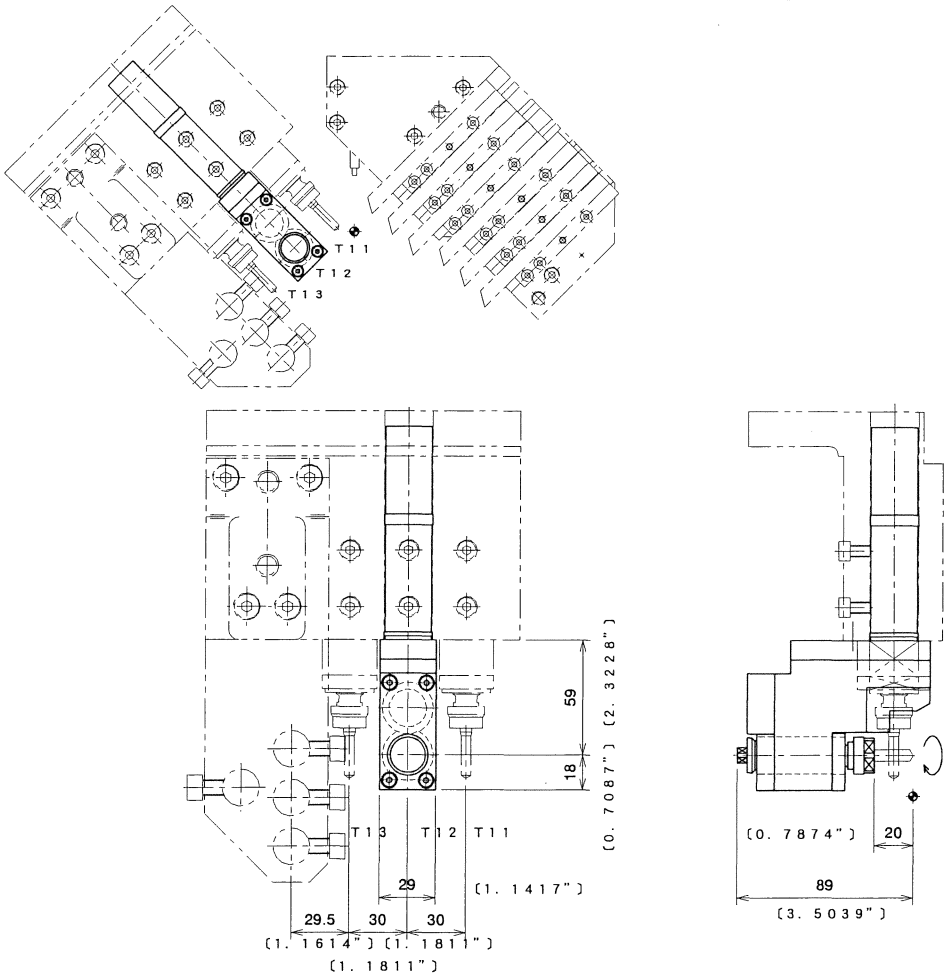
Command format

T1200 x 4.0 ;

NOTICE
- If performing milling with a rotary tool mounted next to this holder, be careful for the holder not to interfere with the workpiece.
- If this holder is mounted, be careful for the holder not to interfere with the back spindle.
- This holder can be mounted at the position T12.

Tool holder name	GSE707
Max.chuck dia.	ø7mm[ø0.2756"]
Spindle speed	200 ~ 8,000min^{-1}
Chuck model	ER11 , AR11
Spindle rotation direction	Direction shown by the arrow in the figure above for a forward rotation command

GSA207 End-drilling Spindles
(Angle-adjustable Type : Example Of Mounting The U31B : Type VI)



This holder is provided for end-face drilling. Mount the sleeve holder on the U31B (holder for 3 rotary tools).

Command format

T1200 x 4.0 ;

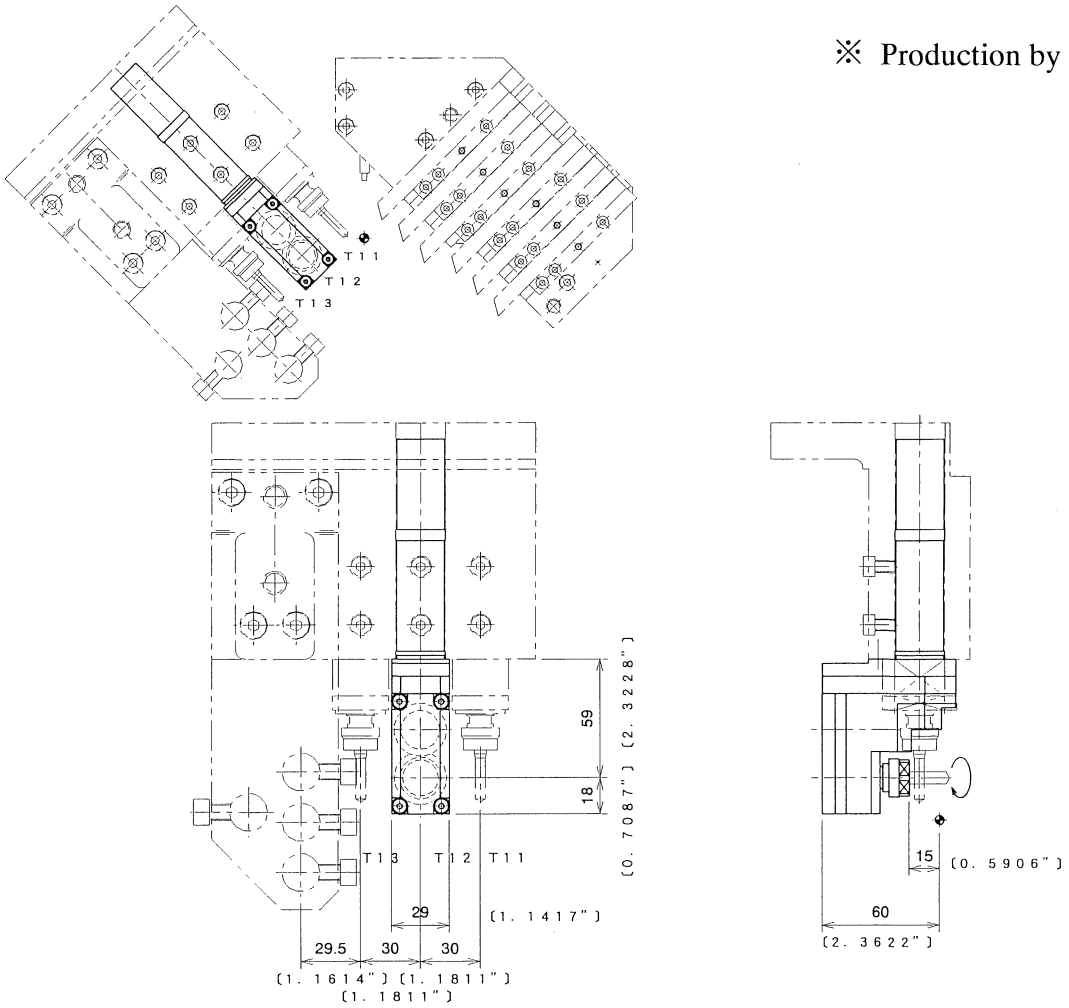
NOTICE
- If performing milling with a rotary tool mounted next to this holder, be careful for the holder not to interfere with the workpiece.
- If this holder is mounted, be careful for the holder not to interfere with the back spindle.
- This holder can be mounted at the position T12.

Tool holder name	GSA207
Max.chuck dia.	ø7mm[ø0.2756"]
Spindle speed	200 ~ 8,000min^{-1}
Chuck model	ER11 , AR11
Spindle rotation direction	Direction shown by the arrow in the figure above for a forward rotation command
Angle adjustment	± 6º

GSA307 End-drilling Spindles

(Short Type : Angle-adjustable Type : Example Of Mounting The U31B : Type VI)

※ Production by order



This holder is provided for end-face drilling. Mount the sleeve holder on the U31B (holder for 3 rotary tools).

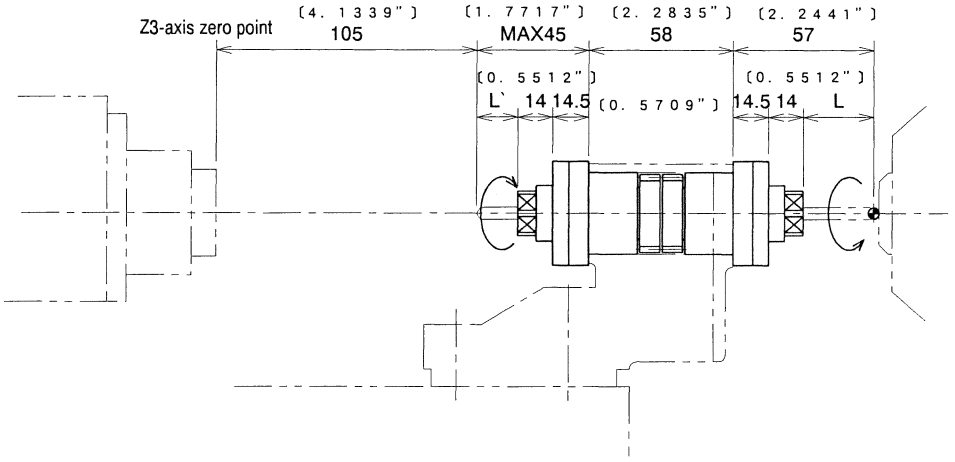
Command format

T1200 x 4.0 ;

NOTICE
- If performing milling with a rotary tool mounted next to this holder, be careful for the holder not to interfere with the workpiece.
- If this holder is mounted, be careful for the holder not to interfere with the back spindle.
- This holder can be mounted at the position T12.

Tool holder name	GSA307
Max.chuck dia.	ø7mm[ø0.2756"]
Spindle speed	200 ~ 8,000min^{-1}
Chuck model	ER11 , AR11
Spindle rotation direction	Direction shown by the arrow in the figure above for a forward rotation command
Angle adjustment	± 10º

GSE807 End-drilling Spindles

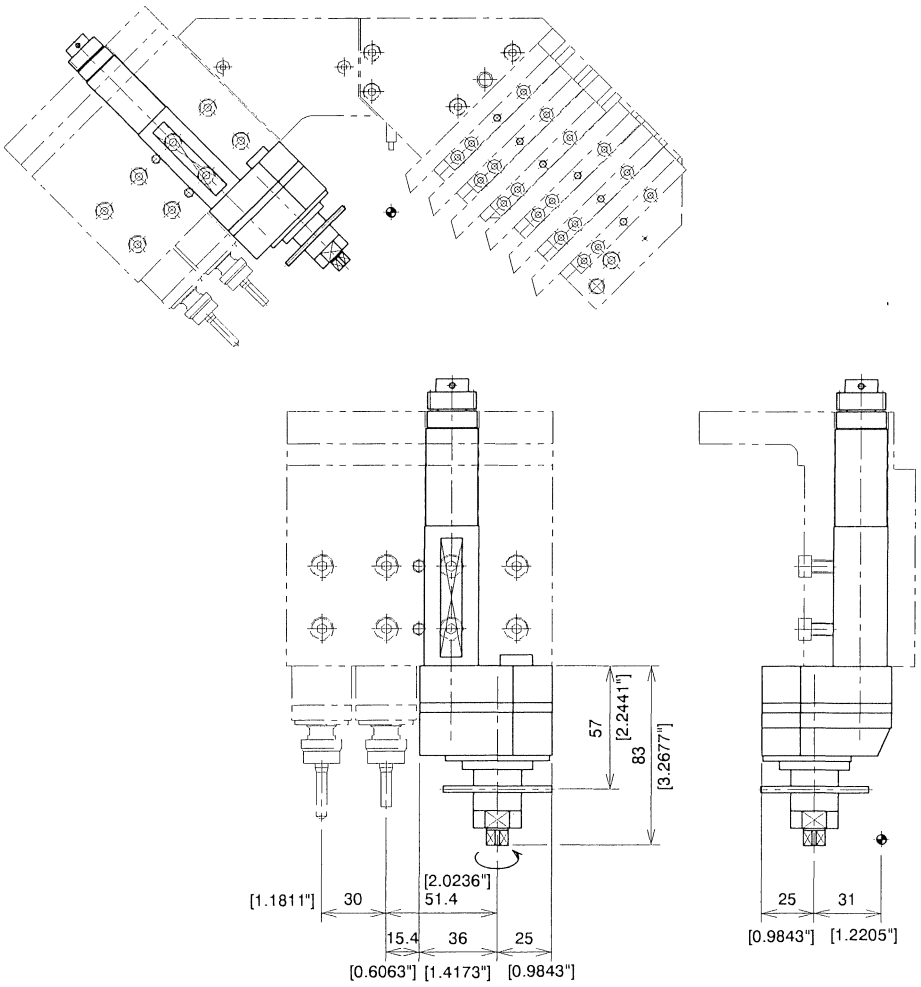


This holder is provided for end-tace drilling. Mount the holder on the U121B (Front / back rotary tool holder).

NOTICE
- If this holder is mounted, be careful for the holder not to interfere with the back spindle or the front machining tool holder (vertical holder).
- This holder can be mounted at the position T21,T22,T23,T31,T32 or T33..

Tool holder name	GSE807
Max.chuck dia.	ø7mm(ø0.2756")
Spindle speed	200 ~ 5,000min^{-1}
Chuck model	ER11 , AR11
Sleeve Holder	U121B
Spindle rotation direction	Direction shown by the arrow in the figure above for a forward rotation command

GSS150 Slitting Spindle

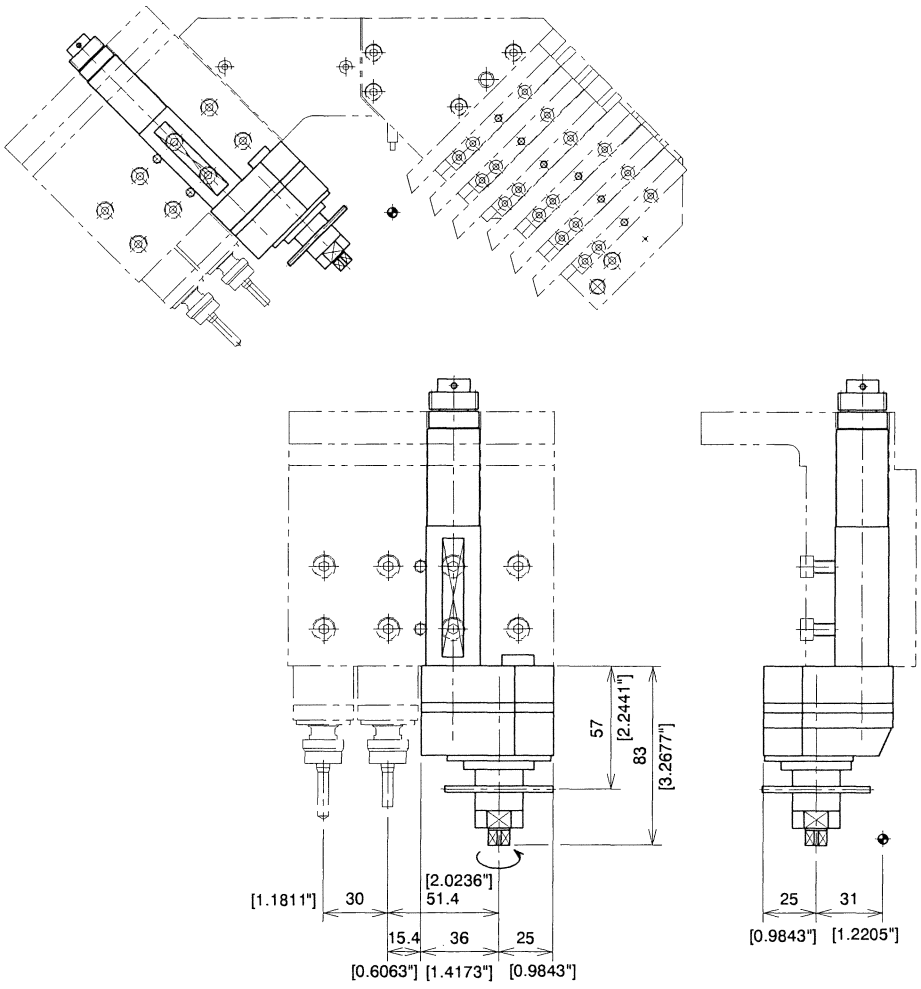


This holder is provided for end-face slitting. Mount the sleeve holder on the vertical holder.

NOTICE
- If performing milling with a rotary tool mounted next to this holder, be careful for the holder not to interfere with the workpiece.
- If this holder is mounted, be careful for the holder not to interfere with the back spindle or the opposite sleeve.
- Spacers are prepared for t0.5, 1.0, 3.0, it is possible to adjust the double cutter each 0.5mm.

Tool holder name	GSS150
Outer diameter of slitting cutter	ø45(1.7717") , ø50mm(1.9685")
Inner diameter of slitting cutter	ø12.7mm(ø1/2") , ø15.875mm(ø5/8")
Maximum thickness of slitting cutter	6(0.2362")
Spindle speed	100 ~ 4,000min^{-1} Redaction 1/2
Spindle rotation direction	Direction shown by the arrow in the figure above for a forward rotation command

GSS250 Slitting Spindle

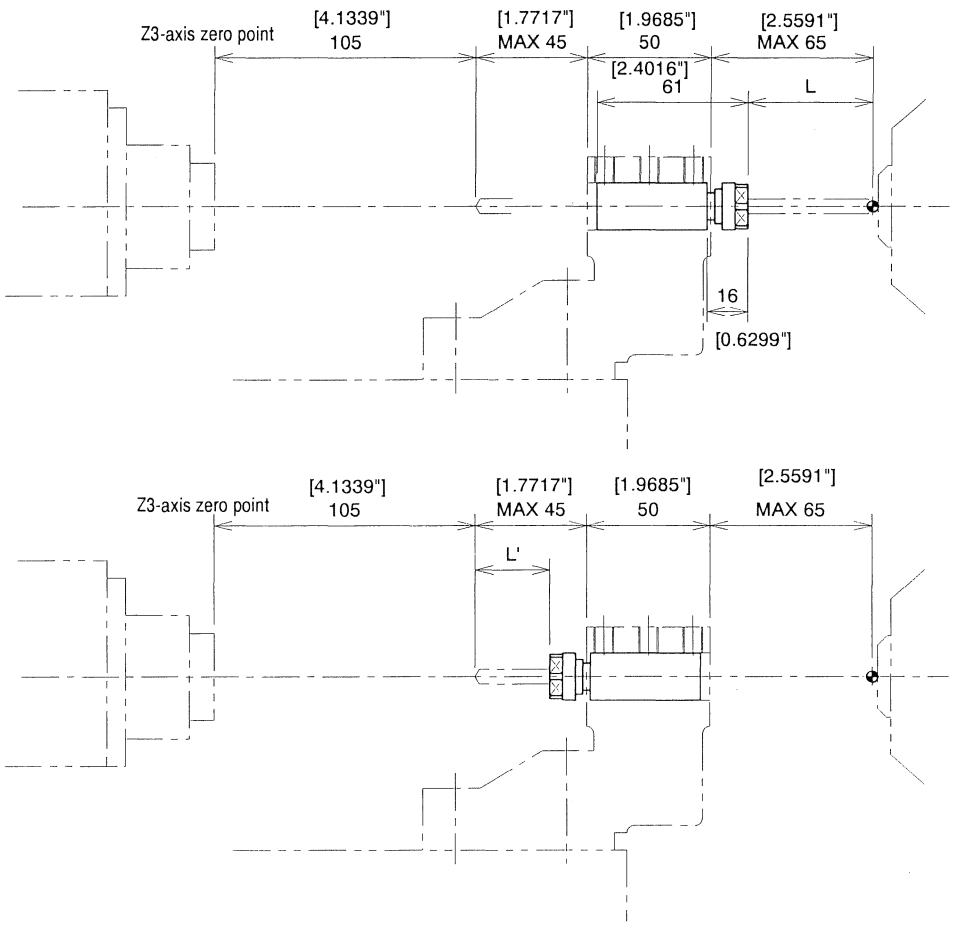


This holder is provided for end-face slitting. Mount the sleeve holder on the vertical holder.

NOTICE
- If performing milling with a rotary tool mounted next to this holder, be careful for the holder not to interfere with the workpiece.
- If this holder is mounted, be careful for the holder not to interfere with the back spindle or the opposite sleeve.
- Spacers are prepared for t0.5, 1.0, 3.0, it is possible to adjust the double cutter each 0.5mm.

Tool holder name	GSS250
Outer diameter of slitting cutter	ø45(1.7717") , ø50mm(1.9685")
Inner diameter of slitting cutter	ø13, ø16 mm (ø0.5118",ø0.6299")
Maximum thickness of slitting cutter	6(0.2362")
Spindle speed	100 ~ 4,000min^{-1} Redaction 1/2
Spindle rotation direction	Direction shown by the arrow in the figure above for a forward rotation command

BDS507 Boring · Drill Sleeve(~ø7、Short type)

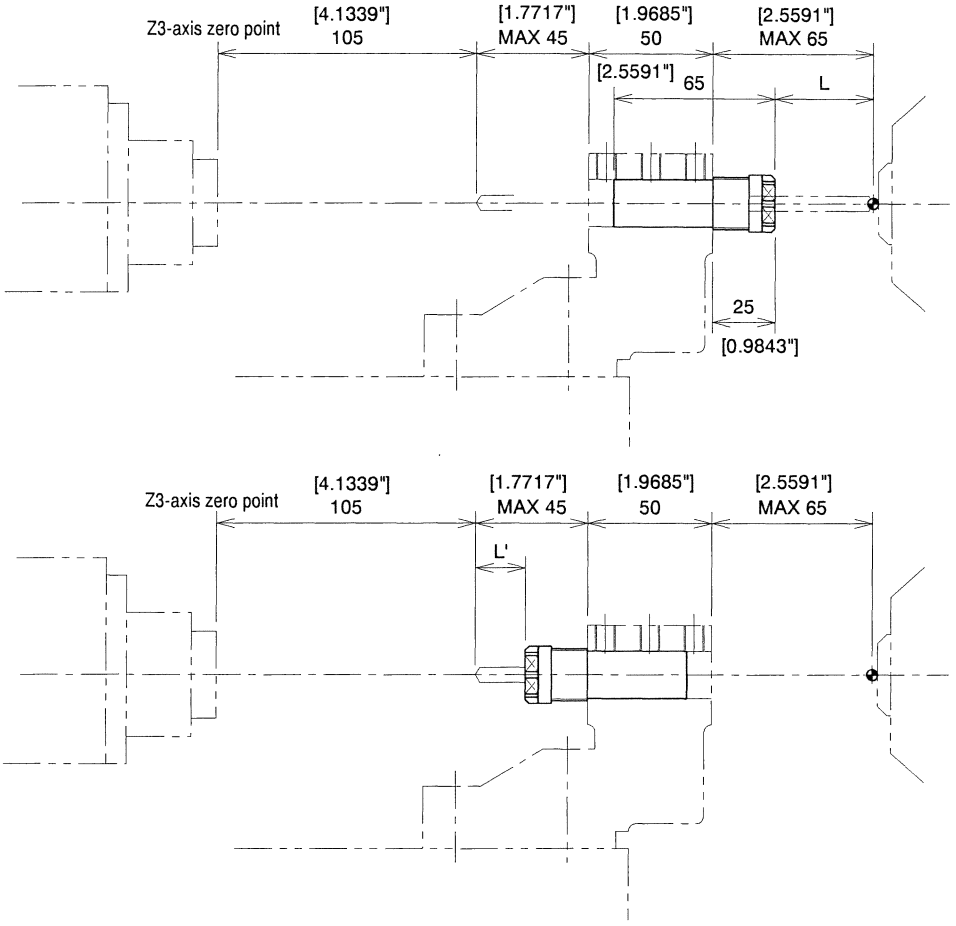


This sleeve is provided for a drill or gimlet, chucked by Rego chuck, to perform front machining and back machining. The dimensions L and L' can be changed by adjusting the protrusion of the sleeve or tool.

NOTICE - If this sleeve is mounted, be careful for the sleeve not to interfere with the back spindle or the front machining tool holder (vertical holder).

Sleeve	BDS507
Max.chuck dia.	ø7mm(ø0.2756")
Chuck model	ER11 , AR11
Sleeve outer dia.	ø19.05mm(ø3/4")
Sleeve Holder	U120B,GDF905,U121B(T24,T25)

BDS508 Boring · Drill Sleeve(~ø10、Short type)

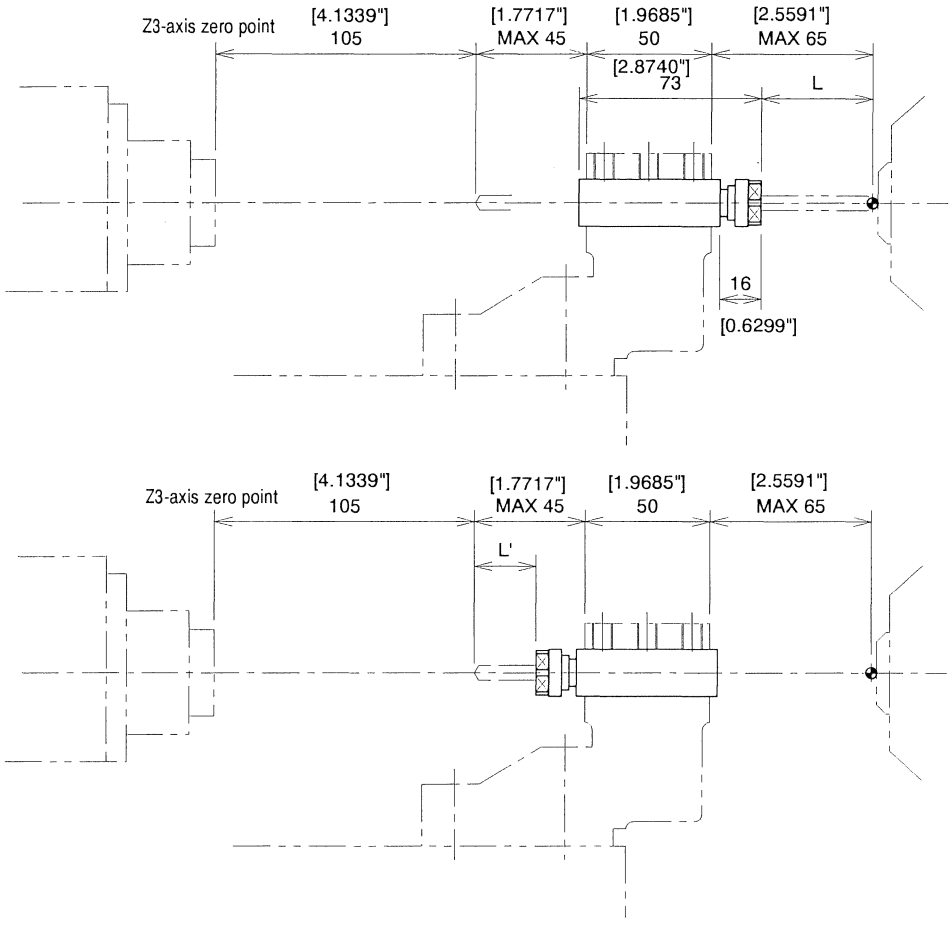


This sleeve is provided for a drill or gimlet, chucked by Rego chuck, to perform front machining and back machining. The dimensions L and L' can be changed by adjusting the protrusion of the sleeve or tool.

NOTICE - If this sleeve is mounted, be careful for the sleeve not to interfere with the back spindle or the front machining tool holder (vertical holder). Do not mount the sleeve for ER16 and the sleeve for AR16 next to each other.

Sleeve	BDS508
Max.chuck dia.	ø10mm(ø0.3937")
Chuck model	ER16 , AR16
Sleeve outer dia.	ø19.05mm(ø3/4")
Sleeve Holder	U120B,GDF905,U121B(T25)

BDS607 Boring · Drill Sleeve(~ø7、Semi-Short type)

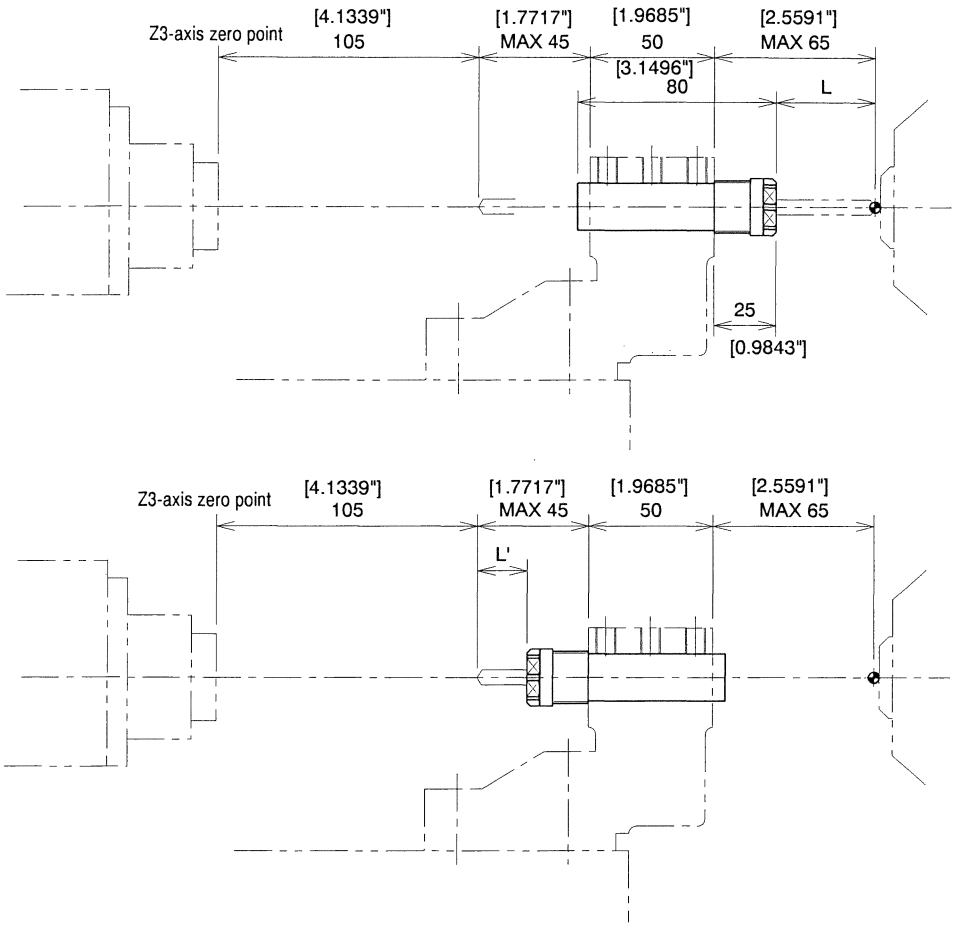


This sleeve is provided for a drill or gimlet, chucked by Rego chuck, to perform front machining and back machining. The dimensions L and L' can be changed by adjusting the protrusion of the sleeve or tool.

NOTICE - If this sleeve is mounted, be careful for the sleeve not to interfere with the back spindle or the front machining tool holder (vertical holder).

Sleeve	BDS607
Max.chuck dia.	ø7mm(ø0.2756")
Chuck model	ER11 , AR11
Sleeve outer dia.	ø19.05mm(ø3/4")
Sleeve Holder	U120B,GDF905,U121B(T24,T25)

BDS610 Boring · Drill Sleeve(~ø10、Semi-Short type)

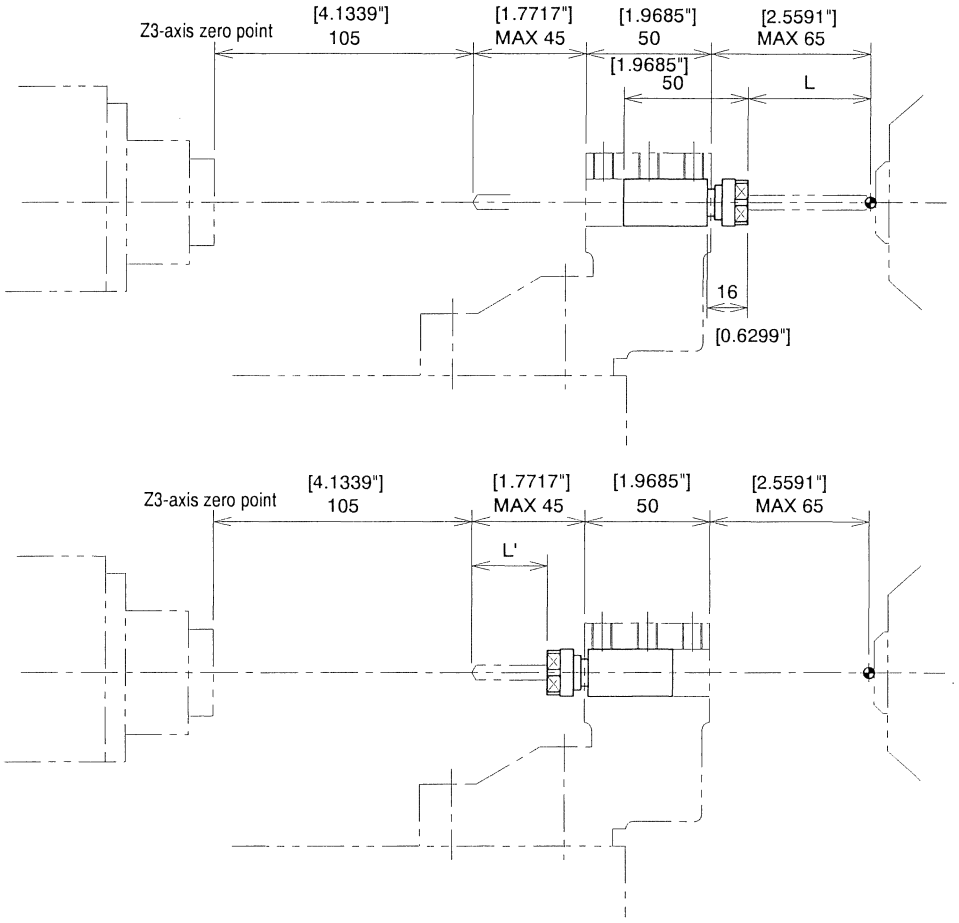


This sleeve is provided for a drill or gimlet, chucked by Rego chuck, to perform front machining and back machining. The dimensions L and L' can be changed by adjusting the protrusion of the sleeve or tool.

NOTICE - If this sleeve is mounted, be careful for the sleeve not to interfere with the back spindle or the front machining tool holder (vertical holder). Do not mount the sleeve for ER16 and the sleeve for AR16 next to each other.

Sleeve	BDS610
Max.chuck dia.	ø10mm(ø0.3937")
Chuck model	ER16 , AR16
Sleeve outer dia.	ø19.05mm(ø3/4")
Sleeve Holder	U120B,GDF905,U121B(T25)

BDS707 Boring · Drill Sleeve(~ø7、Short type)

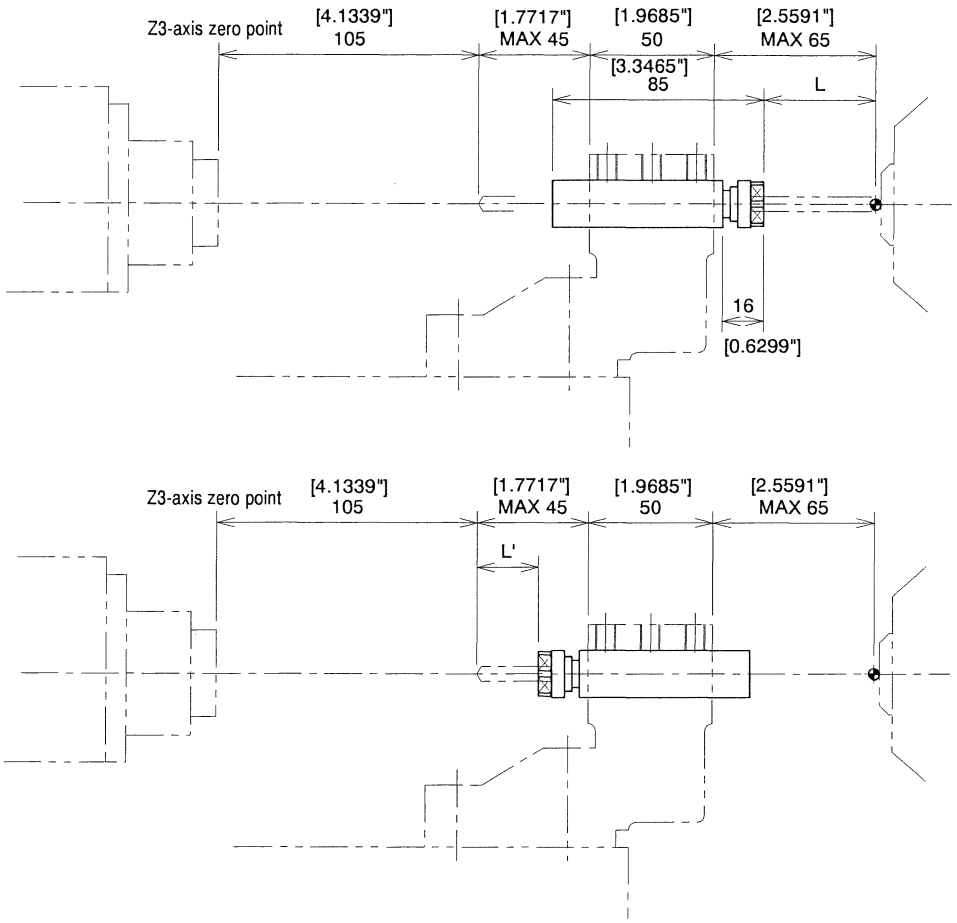


This sleeve is provided for a drill or gimlet, chucked by Rego chuck, to perform front machining and back machining. The dimensions L and L' can be changed by adjusting the protrusion of the sleeve or tool.

NOTICE - If this sleeve is mounted, be careful for the sleeve not to interfere with the back spindle or the front machining tool holder (vertical holder).

Sleeve	BDS707
Max.chuck dia.	ø7mm(ø0.2756")
Chuck model	ER11 , AR11
Sleeve outer dia.	ø19.05mm(ø3/4")
Sleeve Holder	U120B,GDF905,U121B(T24,T25)

VDS506 Boring · Drill Sleeve(~ø7)

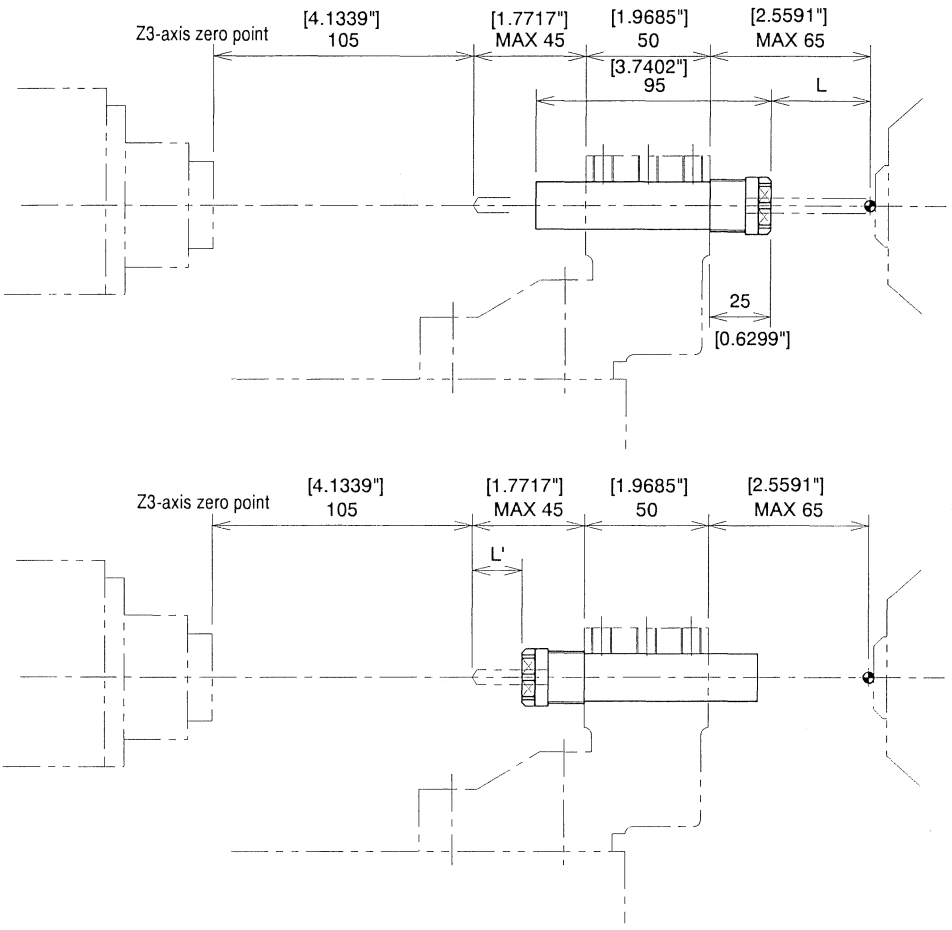


This sleeve is provided for a drill or gimlet, chucked by Rego chuck, to perform front machining and back machining. The dimensions L and L' can be changed by adjusting the protrusion of the sleeve or tool.

NOTICE - If this sleeve is mounted, be careful for the sleeve not to interfere with the back spindle or the front machining tool holder (vertical holder).

Sleeve	VDS506
Max.chuck dia.	ø7mm(ø0.2756")
Chuck model	ER11 , AR11
Sleeve outer dia.	ø19.05mm(ø3/4")
Sleeve Holder	U120B,GDF905,U121B(T24,T25)

VDS110 Boring · Drill Sleeve(~ø10)

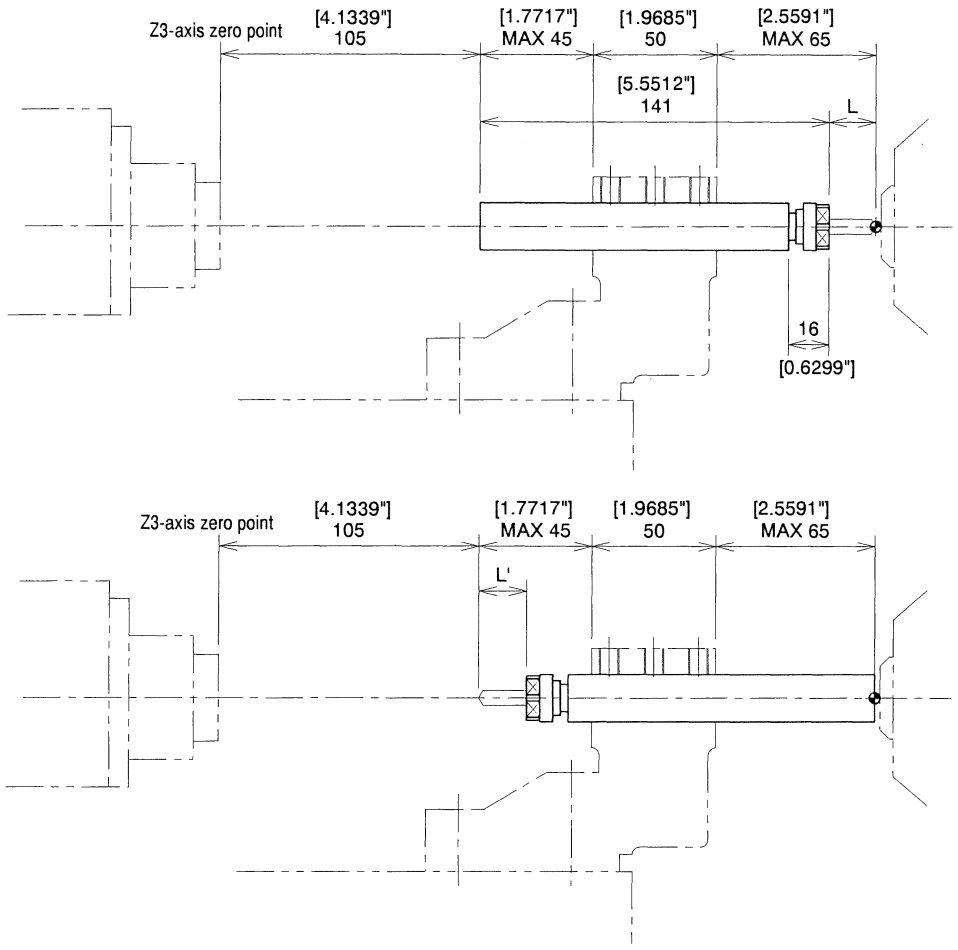


This sleeve is provided for a drill or gimlet, chucked by Rego chuck, to perform front machining and back machining. The dimensions L and L' can be changed by adjusting the protrusion of the sleeve or tool.

NOTICE - If this sleeve is mounted, be careful for the sleeve not to interfere with the back spindle or the front machining tool holder (vertical holder). Do not mount the sleeve for ER16 and the sleeve for AR16 next to each other.

Sleeve	VDS110
Max.chuck dia.	ø10mm(ø0.3937")
Chuck model	ER16 , AR16
Sleeve outer dia.	ø19.05mm(ø3/4")
Sleeve Holder	U120B,GDF905,U121B(T25)

LDS107 Boring · Drill Sleeve(~ø7、Long type)

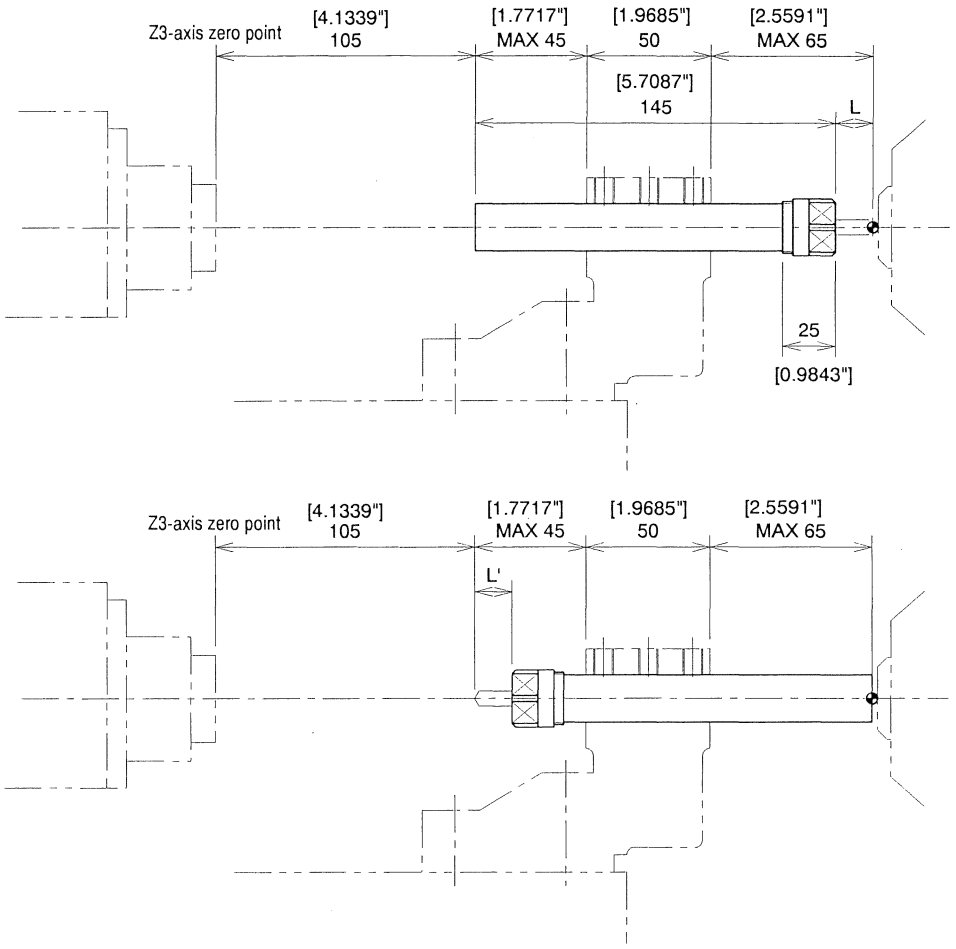


This sleeve is provided for a drill or gimlet, chucked by Rego chuck, to perform front machining and back machining. The dimensions L and L' can be changed by adjusting the protrusion of the sleeve or tool.

NOTICE - If this sleeve is mounted, be careful for the sleeve not to interfere with the back spindle or the front machining tool holder (vertical holder).

Sleeve	LDS107
Max.chuck dia.	ø7mm(ø0.2756")
Chuck model	ER11 , AR11
Sleeve outer dia.	ø19.05mm(ø3/4")
Sleeve Holder	U120B,GDF905,U121B(T24,T25)

LDS110 Boring · Drill Sleeve(~ø10、Long type)

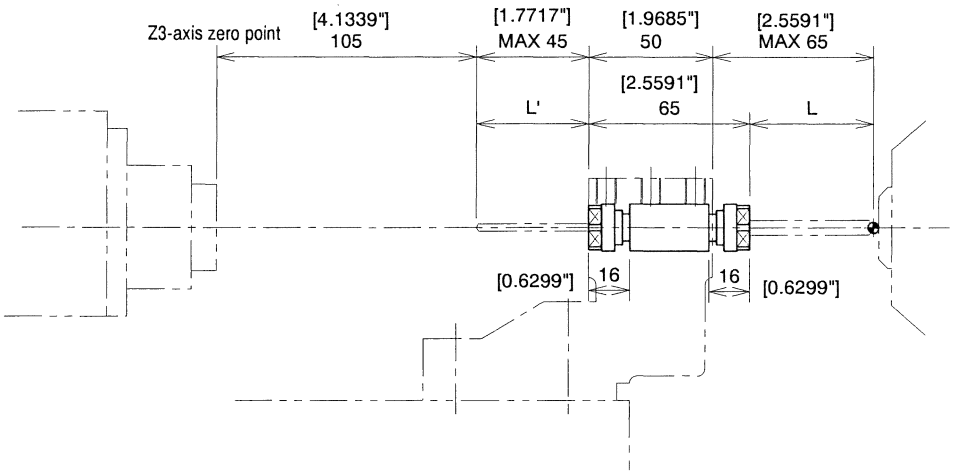


This sleeve is provided for a drill or gimlet, chucked by Rego chuck, to perform front machining and back machining. The dimensions L and L' can be changed by adjusting the protrusion of the sleeve or tool.

NOTICE - If this sleeve is mounted, be careful for the sleeve not to interfere with the back spindle or the front machining tool holder (vertical holder). Do not mount the sleeve for ER16 and the sleeve for AR16 next to each other.

Sleeve	LDS110
Max.chuck dia.	ø10mm(ø0.3937")
Chuck model	ER16 , AR16
Sleeve outer dia.	ø19.05mm(ø3/4")
Sleeve Holder	U120B,GDF905,U121B(T25)

HDS5406 Boring · Drill Sleeve(~ø7、Both end Short type)

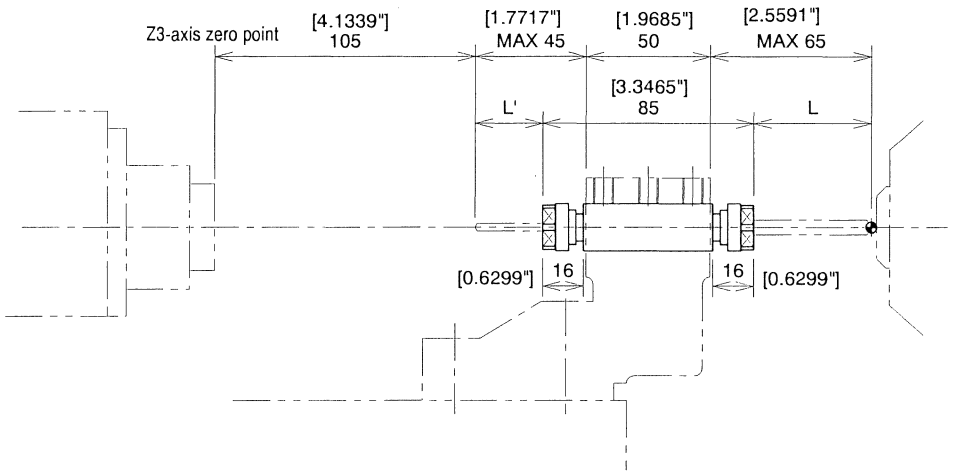


This sleeve is provided for a drill or gimlet, chucked by Rego chuck, to perform front machining and back machining. The dimensions L and L' can be changed by adjusting the protrusion of the sleeve or tool.

NOTICE - If this sleeve is mounted, be careful for the sleeve not to interfere with the back spindle or the front machining tool holder (vertical holder).

Sleeve	HDS5406
Max.chuck dia.	ø7mm(ø0.2756")
Chuck model	ER11 , AR11
Sleeve outer dia.	ø19.05mm(ø3/4")
Sleeve Holder	U120B,GDF905,U121B(T24,T25)

HDS5506 Boring · Drill Sleeve(~ø7、Both end)

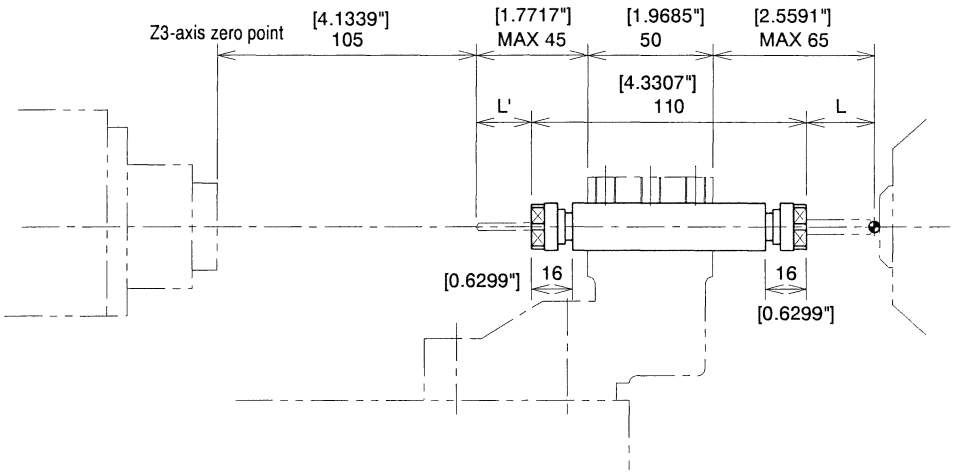


This sleeve is provided for a drill or gimlet, chucked by Rego chuck, to perform front machining and back machining. The dimensions L and L' can be changed by adjusting the protrusion of the sleeve or tool.

NOTICE - If this sleeve is mounted, be careful for the sleeve not to interfere with the back spindle or the front machining tool holder (vertical holder).

Sleeve	HDS5506
Max.chuck dia.	ø7mm(ø0.2756")
Chuck model	ER11 , AR11
Sleeve outer dia.	ø19.05mm(ø3/4")
Sleeve Holder	U120B,GDF905,U121B(T24,T25)

HDS5706 Boring · Drill Sleeve(~ø7、Both end Semi-long)

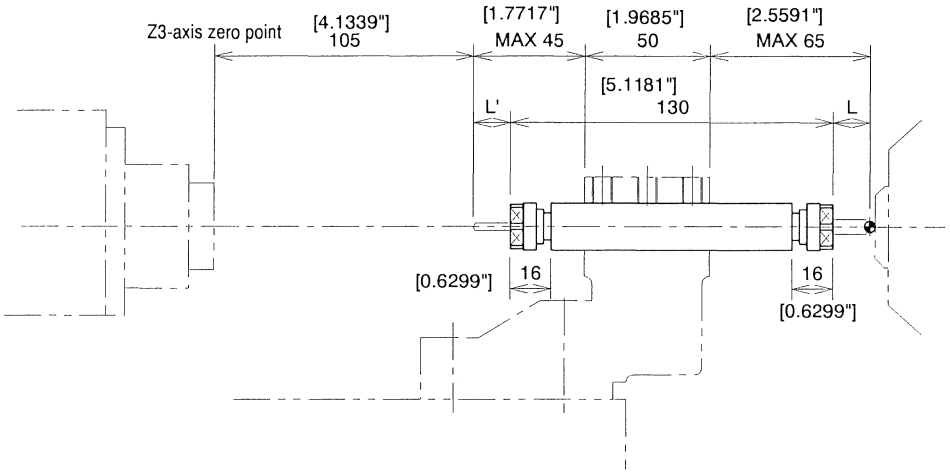


This sleeve is provided for a drill or gimlet, chucked by Rego chuck, to perform front machining and back machining. The dimensions L and L' can be changed by adjusting the protrusion of the sleeve or tool.

NOTICE - If this sleeve is mounted, be careful for the sleeve not to interfere with the back spindle or the front machining tool holder (vertical holder).

Sleeve	HDS5706
Max.chuck dia.	ø7mm(ø0.2756")
Chuck model	ER11 , AR11
Sleeve outer dia.	ø19.05mm(ø3/4")
Sleeve Holder	U120B,GDF905,U121B(T24,T25)

HDS5806　Boring · Drill Sleeve(~ø7、Both end Long)

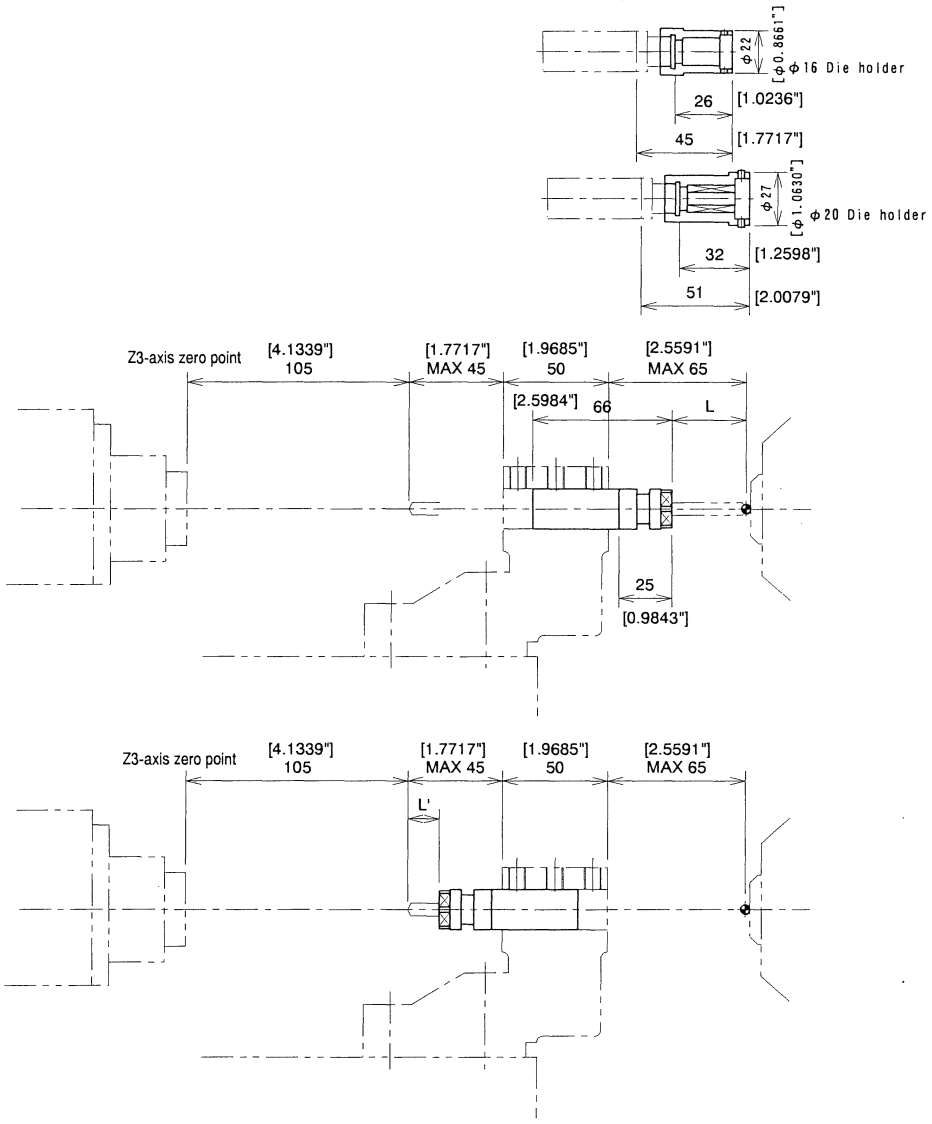


This sleeve is provided for a drill or gimlet, chucked by Rego chuck, to perform front machining and back machining.　The dimensions L and L' can be changed by adjusting the protrusion of the sleeve or tool.

NOTICE　- If this sleeve is mounted, be careful for the sleeve not to interfere with the back spindle or the front machining tool holder (vertical holder).

Sleeve	HDS5806
Max.chuck dia.	ø7mm(ø0.2756")
Chuck model	ER11 , AR11
Sleeve outer dia.	ø19.05mm(ø3/4")
Sleeve Holder	U120B,GDF905,U121B(T24,T25)

BNS407　Tap · Die Sleeve(~ø7、Short type)
(Die ø16、ø 20)

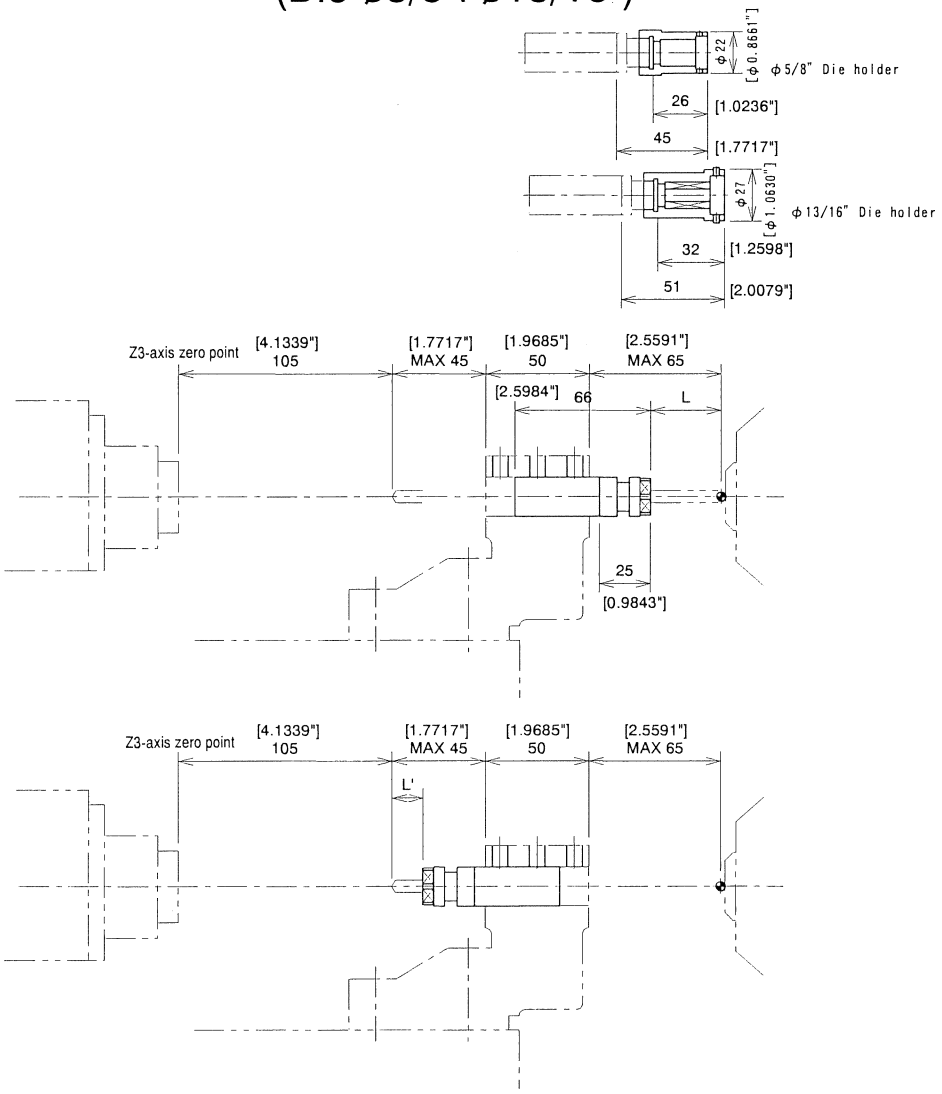


This sleeve is provided for a tap or die, chucked by Rego chuck, to perform front machining and back machining. The dimensions L and L' can be changed by adjusting the protrusion of the sleeve or tool.

NOTICE　- If this sleeve is mounted, be careful for the sleeve not to interfere with the back spindle or the front machining tool holder (vertical holder). If using this sleeve for back machining, mounting of a die is not permitted. Mounting of two die sleeves next to each other is not permitted either. When performing front machining, be careful for the front machining tool not to interfere with the die.

Sleeve	BNS407
Max.chuck dia.	ø7mm(ø0.2756")
Chuck model	ER11 , AR11
Sleeve outer dia.	ø19.05mm(ø3/4")
Sleeve Holder	U120B,GDF905,U121B(T24,T25)

BNS507　Tap · Die Sleeve(~ø7、Short type)
(Die ø5/8"、ø13/16")

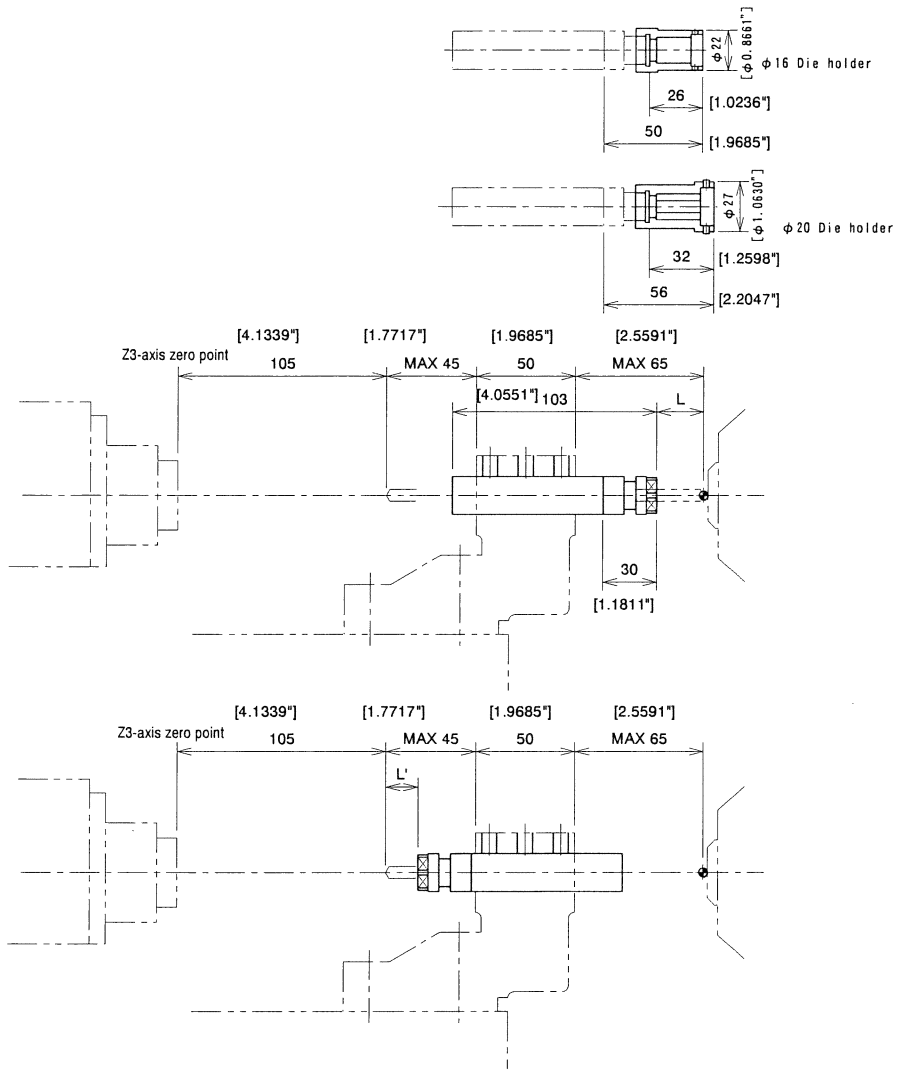


This sleeve is provided for a tap or die, chucked by Rego chuck, to perform front machining and back machining. The dimensions L and L' can be changed by adjusting the protrusion of the sleeve or tool.

NOTICE - If this sleeve is mounted, be careful for the sleeve not to interfere with the back spindle or the front machining tool holder (vertical holder). If using this sleeve for back machining, mounting of a die is not permitted. Mounting of two die sleeves next to each other is not permitted either. When performing front machining, be careful for the front machining tool not to interfere with the die.

Sleeve	BNS507
Max.chuck dia.	ø7mm(ø0.2756")
Chuck model	ER11 , AR11
Sleeve outer dia.	ø19.05mm(ø3/4")
Sleeve Holder	U120B,GDF905,U121B(T24,T25)

VNS406 Tap · Die Sleeve(~ø7)
(Die ø16、ø20)

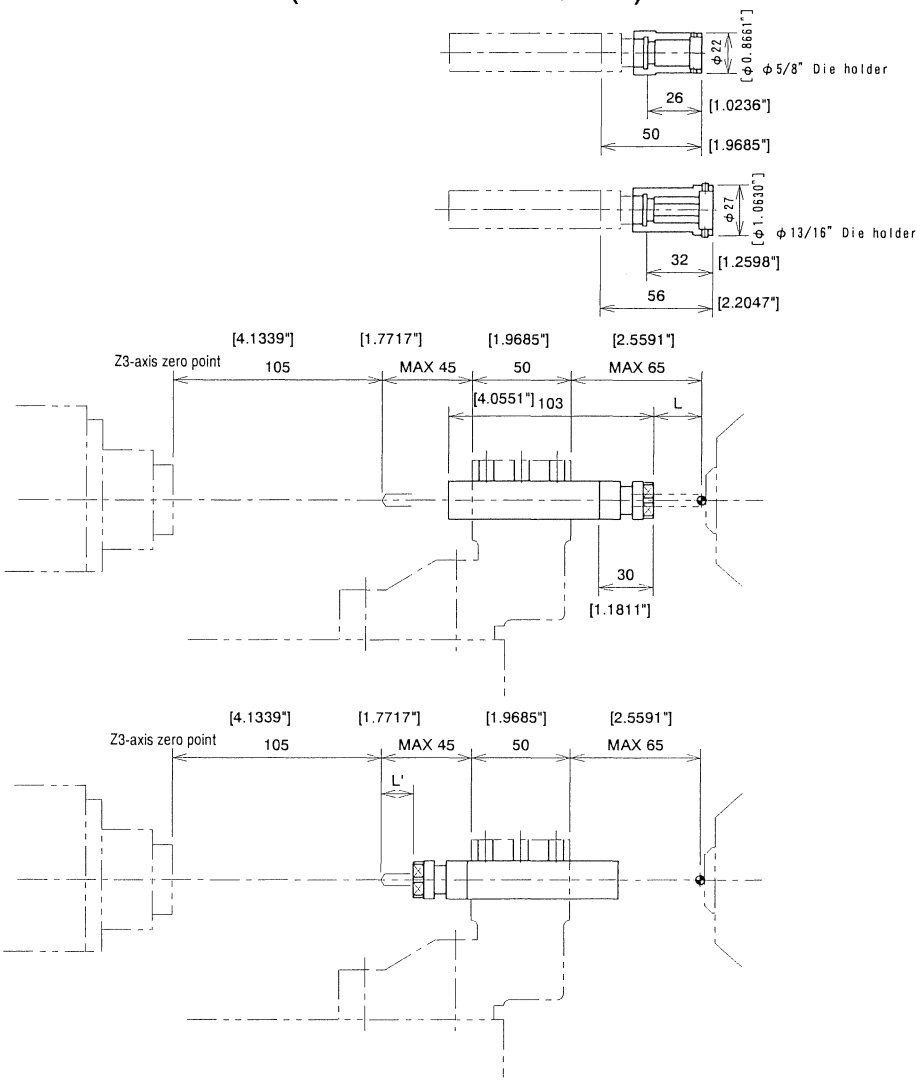


This sleeve is provided for a tap or die, chucked by Rego chuck, to perform front machining and back machining. The dimensions L and L' can be changed by adjusting the protrusion of the sleeve or tool.

NOTICE - If this sleeve is mounted, be careful for the sleeve not to interfere with the back spindle or the front machining tool holder (vertical holder). If using this sleeve for back machining, mounting of a die is not permitted. Mounting of two die sleeves next to each other is not permitted either. When performing front machining, be careful for the front machining tool not to interfere with the die.

Sleeve	VNS406
Max.chuck dia.	ø7mm(ø0.2756")
Chuck model	ER11 , AR11
Sleeve outer dia.	ø19.05mm(ø3/4")
Sleeve Holder	U120B,GDF905,U121B(T24,T25)

VNS506 Tap · Die Sleeve(~ø7)
(Die ø5/8"、ø13/16")

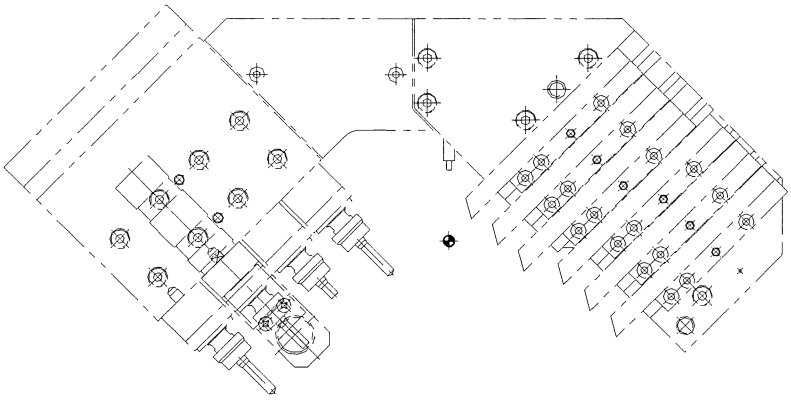


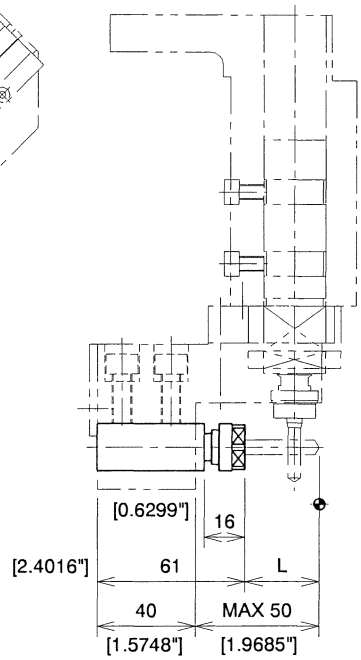
This sleeve is provided for a tap or die, chucked by Rego chuck, to perform front machining and back machining. The dimensions L and L' can be changed by adjusting the protrusion of the sleeve or tool.

NOTICE - If this sleeve is mounted, be careful for the sleeve not to interfere with the back spindle or the front machining tool holder (vertical holder). If using this sleeve for back machining, mounting of a die is not permitted. Mounting of two die sleeves next to each other is not permitted either. When performing front machining, be careful for the front machining tool not to interfere with the die.

Sleeve	VNS506
Max.chuck dia.	ø7mm(ø0.2756")
Chuck model	ER11 , AR11
Sleeve outer dia.	ø19.05mm(ø3/4")
Sleeve Holder	U120B,GDF905,U121B(T24,T25)

BDS507 Boring · Drill Sleeve(~ø7、Short type)



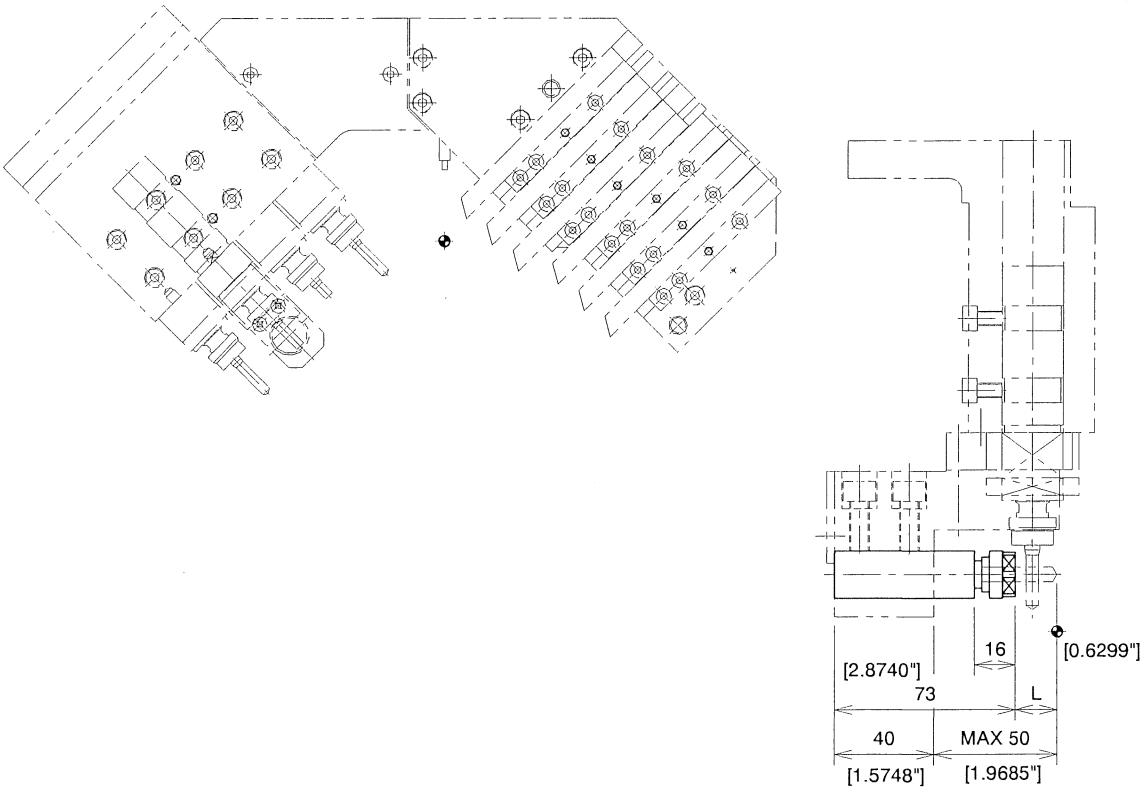


This sleeve is provided for a drill or gimlet, chucked by Rego chuck, to perform front machining and back machining. The dimensions L can be changed by adjusting the protrusion of the sleeve or tool.

NOTICE - If this sleeve is mounted, be careful for the sleeve not to interfere with the back spindle or the opposite sleeve.

Sleeve	BDS507
Max.chuck dia.	ø7mm(ø0.2756")
Chuck model	ER11 , AR11
Sleeve outer dia.	ø19.05mm(ø3/4")
Sleeve Holder	GDF901

BDS607　Boring · Drill Sleeve(~ø7、Semi-Short type)

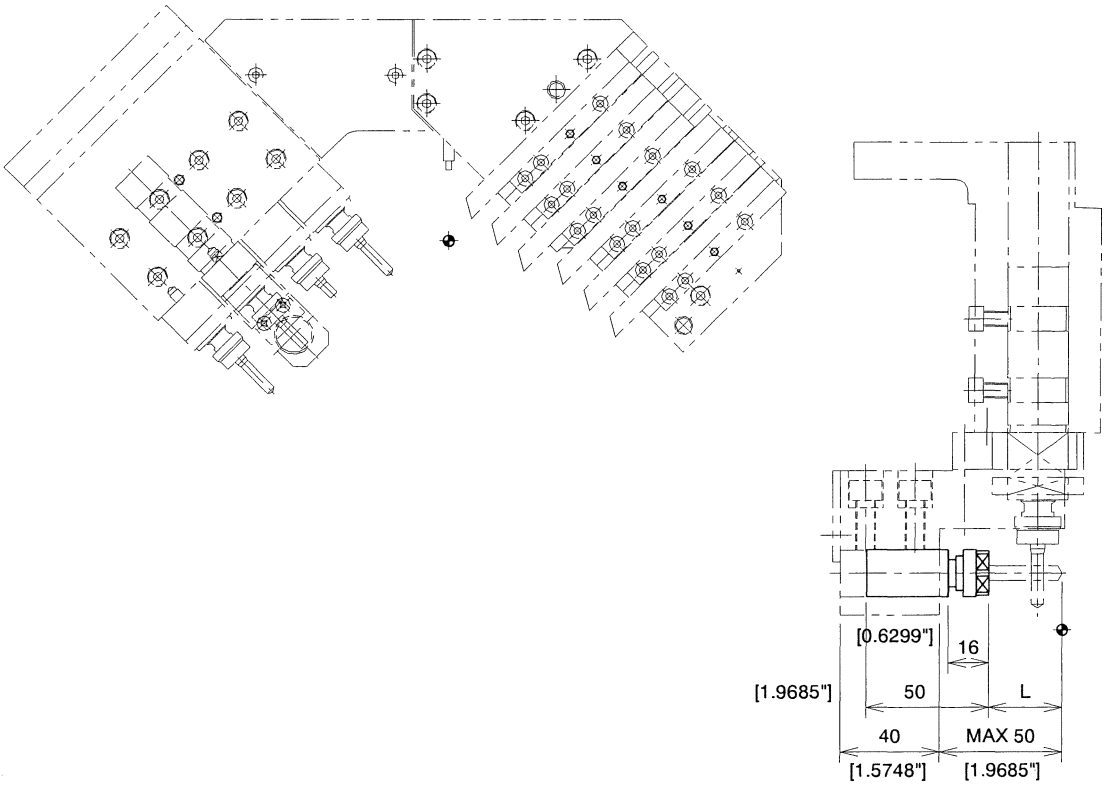


This sleeve is provided for a drill or gimlet, chucked by Rego chuck, to perform front machining and back machining.　The dimensions L can be changed by adjusting the protrusion of the sleeve or tool.

NOTICE　- If this sleeve is mounted, be careful for the sleeve not to interfere with the back spindle or the opposite sleeve.

Sleeve	BDS607
Max.chuck dia.	ø7mm(ø0.2756")
Chuck model	ER11 , AR11
Sleeve outer dia.	ø19.05mm(ø3/4")
Sleeve Holder	GDF901

BDS707　Boring · Drill Sleeve(~ø7、Short type)

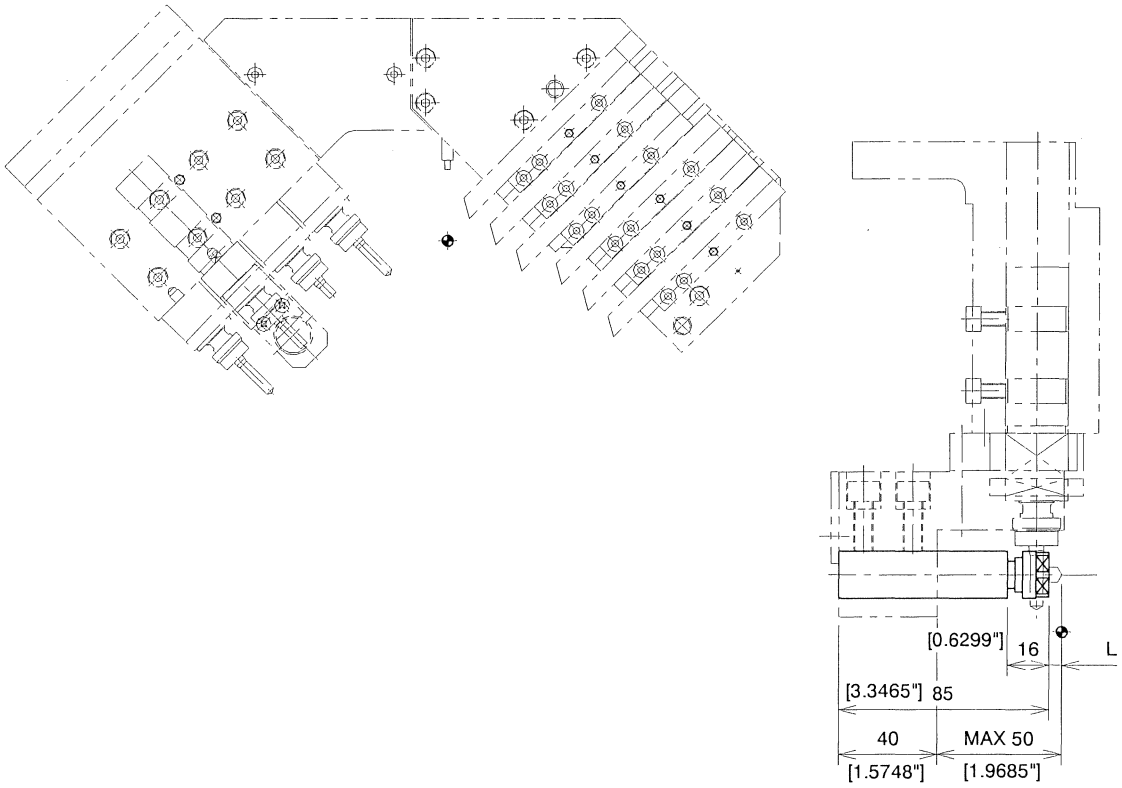


This sleeve is provided for a drill or gimlet, chucked by Rego chuck, to perform front machining and back machining.　The dimensions L can be changed by adjusting the protrusion of the sleeve or tool.

NOTICE　- If this sleeve is mounted, be careful for the sleeve not to interfere with the back spindle or the opposite sleeve.

Sleeve	BDS707
Max.chuck dia.	ø7mm(ø0.2756")
Chuck model	ER11 , AR11
Sleeve outer dia.	ø19.05mm(ø3/4")
Sleeve Holder	GDF901

VDS506 Boring · Drill Sleeve(~ø7)

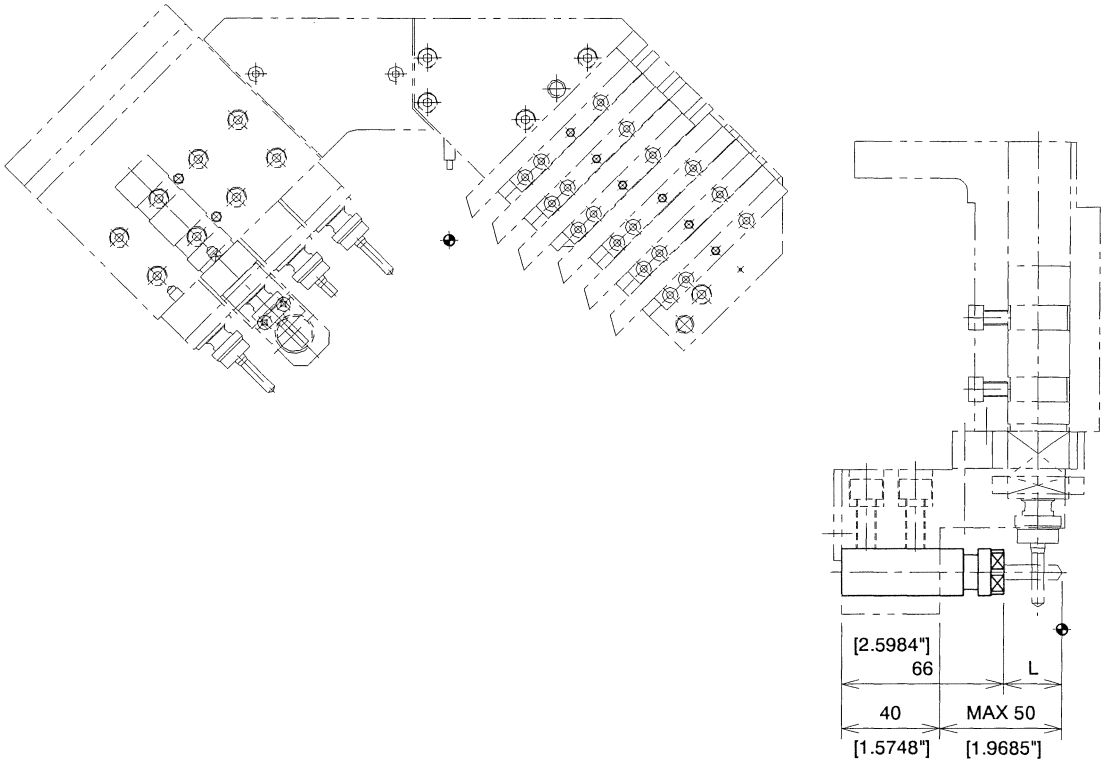


This sleeve is provided for a drill or gimlet, chucked by Rego chuck, to perform front machining and back machining. The dimensions L can be changed by adjusting the protrusion of the sleeve or tool.

NOTICE - If this sleeve is mounted, be careful for the sleeve not to interfere with the back spindle or the opposite sleeve.

Sleeve	VDS506
Max.chuck dia.	ø7mm(ø0.2756")
Chuck model	ER11 , AR11
Sleeve outer dia.	ø19.05mm(ø3/4")
Sleeve Holder	GDF901

BNS407　Tap Sleeve(~ø7、Short type)

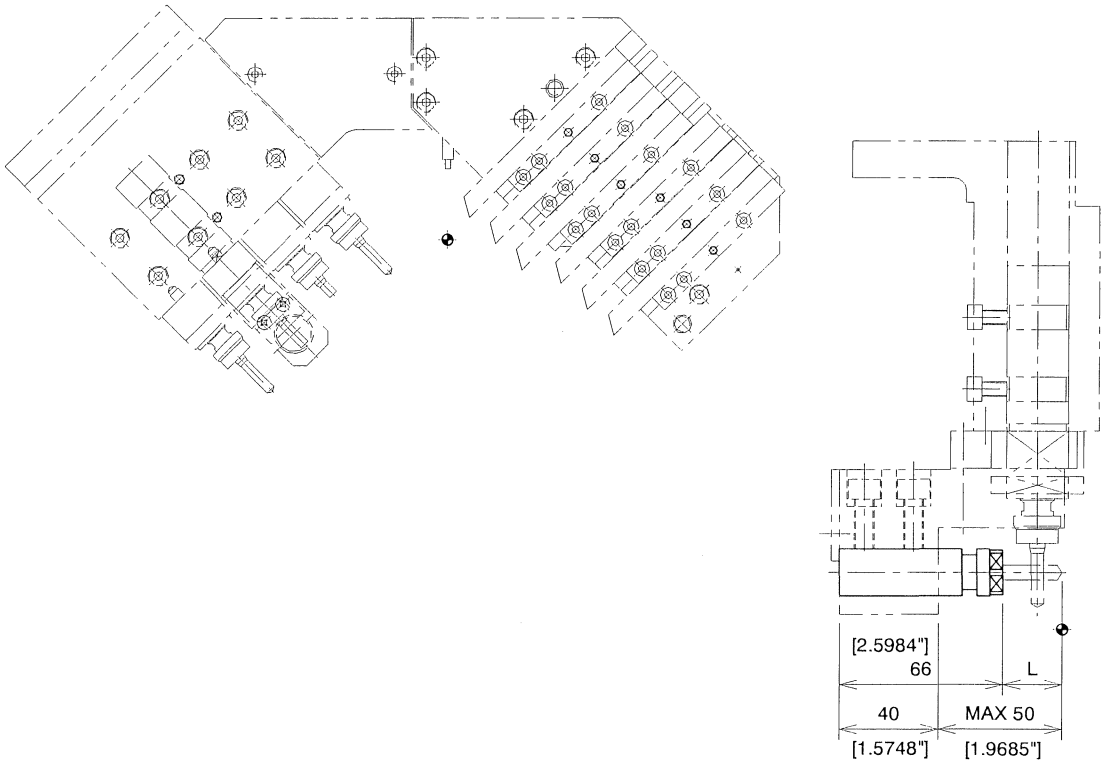


This sleeve is provided for a drill or gimlet, chucked by Rego chuck, to perform front machining and back machining.　The dimensions L can be changed by adjusting the protrusion of the sleeve or tool.

NOTICE　- If this sleeve is mounted, be careful for the sleeve not to interfere with the back spindle or the opposite sleeve.

Sleeve	BNS407
Max.chuck dia.	ø7mm(ø0.2756")
Chuck model	ER11 , AR11
Sleeve outer dia.	ø19.05mm(ø3/4")
Sleeve Holder	GDF901

BNS507　Tap Sleeve(~ø7、Short type)

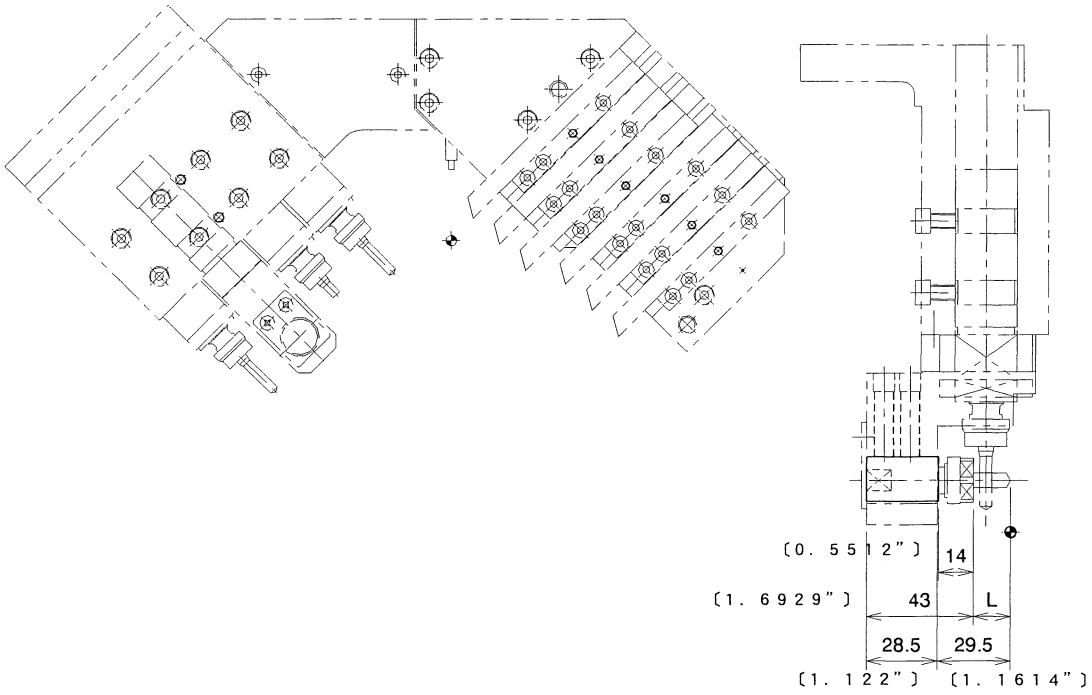


This sleeve is provided for a drill or gimlet, chucked by Rego chuck, to perform front machining and back machining.　The dimensions L can be changed by adjusting the protrusion of the sleeve or tool.

NOTICE　- If this sleeve is mounted, be careful for the sleeve not to interfere with the back spindle or the opposite sleeve.

Sleeve	BNS507
Max.chuck dia.	ø7mm(ø0.2756")
Chuck model	ER11 , AR11
Sleeve outer dia.	ø19.05mm(ø3/4")
Sleeve Holder	GDF901

BDS807　Boring · Drill Sleeve(~ø7、Short type)

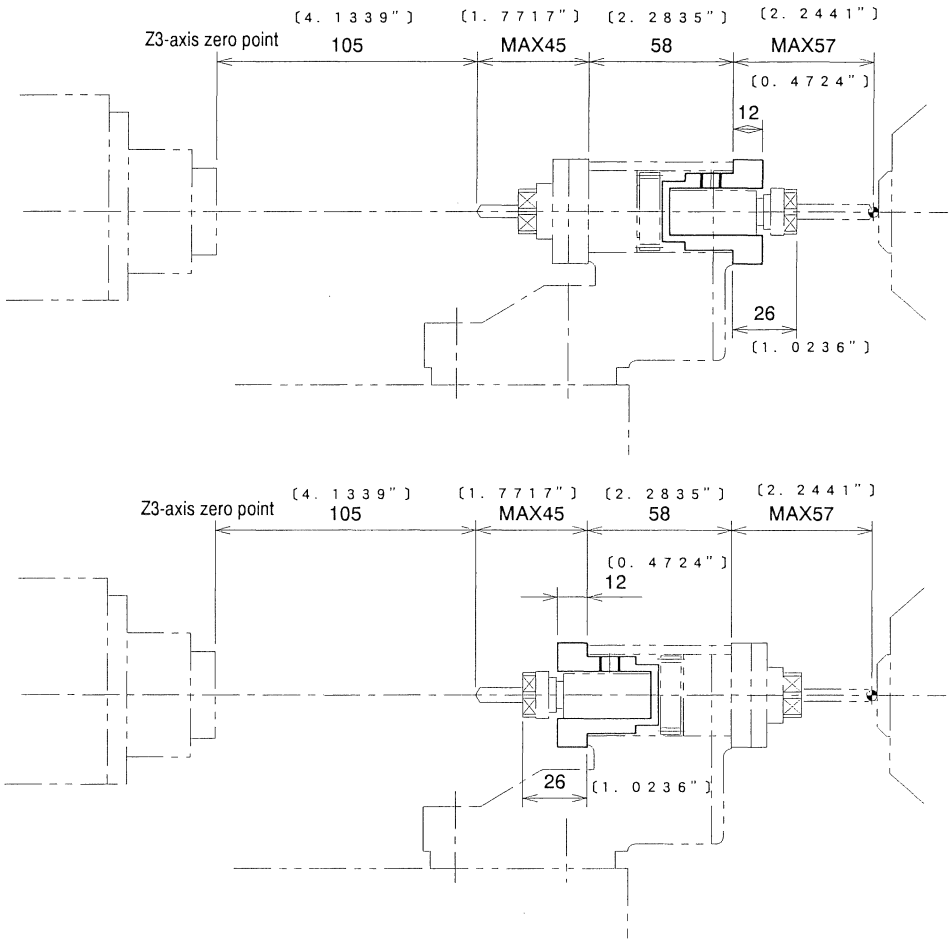


This sleeve is provided for a drill or gimlet, chucked by Rego chuck, to perform front machining and back machining.　The dimensions L can be changed by adjusting the protrusion of the sleeve or tool.

NOTICE　- If this sleeve is mounted, be careful for the sleeve not to interfere with the back spindle or the opposite sleeve.

Sleeve	BDS807
Max.chuck dia.	ø7mm(ø0.2756")
Chuck model	ER11 , AR11
Sleeve outer dia.	ø19.05mm(ø3/4")
Sleeve Holder	GDF907,GDF1007

SAU319 Dedicated Adapter



This adapter is necessary when a fixed-type sleeve (BDS707) is mounted instead of rotary tool (GSE807). Mount the adapter on the U121B (Front / back rotary tool holder).

NOTICE
- If this sleeve is mounted, be careful for the sleeve not to interfere with the back spindle or the front machining tool holder (vertical holder).
- This sleeve can be mounted at the position T21, T22, T23, T31, T32 or T33.
- Positioning accuracy of this holder is just for reference.

Tool holder name	SAU319
Usage	BDS707
Mounting hole diameter of sleeve	ø19.05mm(ø3/4")
Sleeve Holder	U121B (T21,T22,T23,T31,T32,T33)

<Blank Page>

Product code C – C 1 2 1 6 VI VII VIII IX / / / – 7 7 0

Document code 5 E 1 – 0 8 0 2

5 E 2 – 0 8 0 2

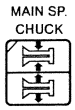
9. Mounting, Adjustment, and Replacement

(Blank page)

9.1 Mounting/Adjusting Chucks

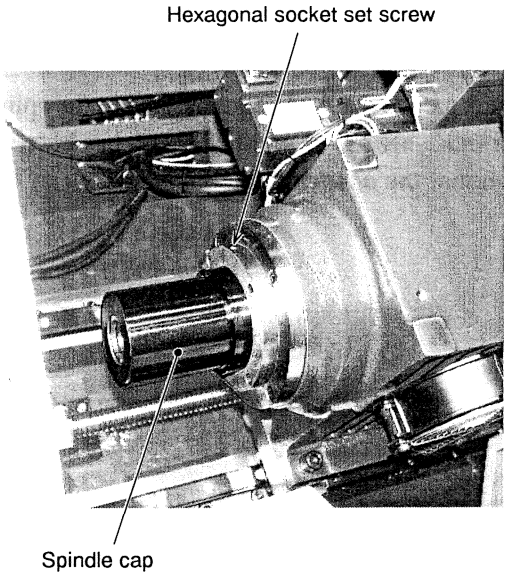
9.1.1 Mounting/replacing the spindle chuck

> **WARNING**
>
> Be sure to confirm that the spindles and machine have stopped before attempting to replace the chuck.
> The work during machine operation could result in serious personal injury.

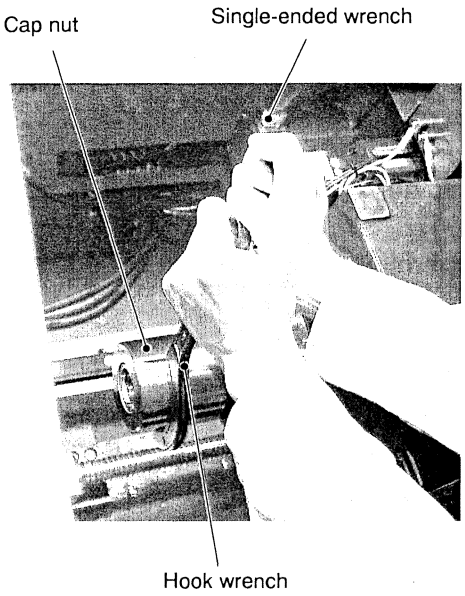
1. Press 

on the operation panel to open the main spindle chuck.
   Note: The MAIN SP. CHUCK key on the operation panel is enabled in the preparation mode or manual operation mode.

2. Loosen the hexagonal socket set screw, then pull out the spindle cap.

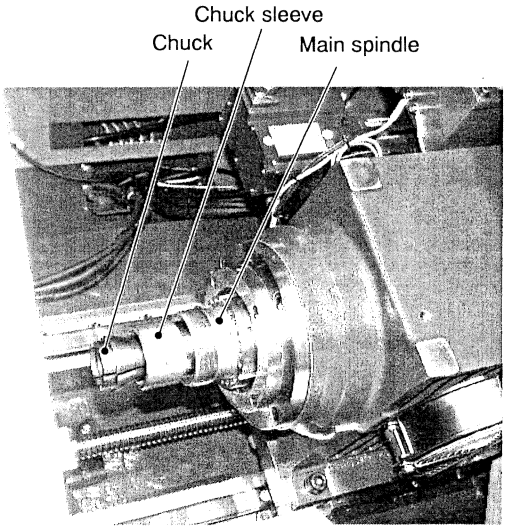


3. Hold the cap nut with the hook wrench (an accessory tool provided with the machine), insert a single-ended wrench into the 2-face width (36 in width) section, then loosen the cap nut. The cap nut is right-handed.



4. Turn the cap nut to remove it from the spindle.

5. Mount the chuck on the chuck sleeve that has been inserted in the spindle. At this time, confirm that the square spring is in the chuck sleeve. If another chuck has been mounted, hold the chuck and pull it out toward the front, then mount the new chuck.



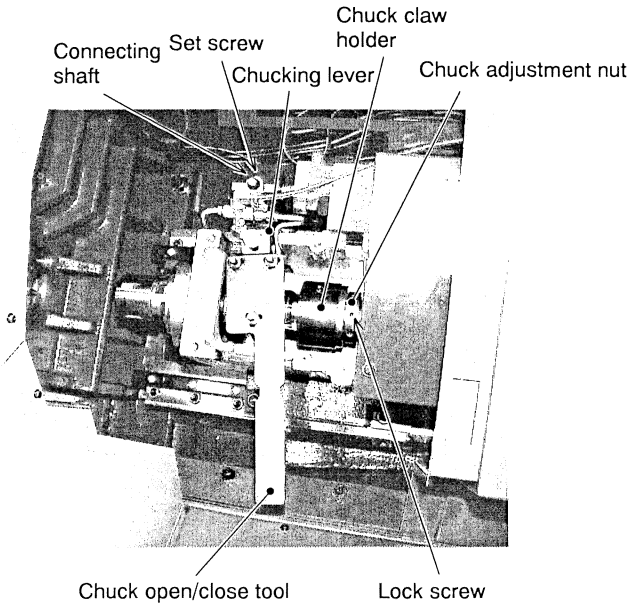
6. Screw the cap nut into the spindle, then firmly tighten the cap nut with the hook wrench and single-ended wrench.

7. Finally, secure the spindle cap with the set screw. Be sure to mount the spindle cap so that the oval drain hole faces down.

> ⚠ CAUTION
>
> Be sure to mount the spindle cap so that the oval drain hole faces down. Failure to do so causes oil to flow into the bearing, which may damage the bearing.

9.1.2 Checking the chucking force

> **WARNING**
>
> Be sure to confirm that the spindles and machine have stopped before attempting to check the chucking force.
> The work during machine operation could result in serious personal injury.



1. Press MAIN SP. CHUCK on the operation panel to open the main spindle chuck, loosen the screw of the top cover of the headstock, then remove the top cover.

2. Loosen the lock screw of the chuck adjustment nut at the rear of the chuck claw holder.

3. Turn the chuck adjustment nut to move the chuck claw holder in the axial direction. To increase the chucking force, turn the chuck adjustment nut toward the other side. To decrease the chucking force, turn the chuck adjustment nut toward the front.

4. Insert a material into the chuck.

> **CAUTION**
>
> Do not close the chuck without inserting any material.
> Doing so may cause the machine to malfunction or may damage the collect chuck.

5. Press the MAIN SP. CHUCK key to open and close the chuck with the material at a suitable position, then confirm the chucking force in the M/SP Chuck Count DISP field. To display the M/SP Chuck Count DISP field, press 

, and then the menu key [CHCK ADJ].

6. Firmly tighten the lock screw of the chuck adjustment nut at completion of adjustment, then pull out the material.

Note

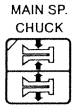
The chuck counter displays a reference value. Control the chucking force with the angle at which you tighten the chuck adjustment nut. Check the chucking force with the chuck open/close tool.

Checking the chucking force with the chuck open/close tool

> **WARNING**
>
> Be sure to confirm that the spindles and machine have stopped before attempting to check the chucking force.
> The work during machine operation could result in serious personal injury.

Take the following procedure to check the chucking force manually, using the chuck open/close tool (an accessory tool provided with the machine):

1. To roughly adjust the chucking force, take steps 1 to 4 of the procedure in <Section 9.1.2 Checking the chucking force>.

2. Press 

on the operation panel to open the chuck.
   Note: MAIN SP. CHUCK key on the operation panel is enabled in the preparation mode or manual operation mode.

3. Loosen the set screw at the tip of the chucking lever, then lift off the connecting shaft that secures the chucking lever and the ball screw shaft for opening/closing chucks.

4. Using the chuck open/close tool (an accessory tool provided with the machine), check the chucking force while moving the chucking lever to right and left. Turn the chuck adjustment nut with the tommy rod until obtaining suitable chucking force.

5. Set the chucking lever to open the chuck (turn the chuck open/close tool to the left).

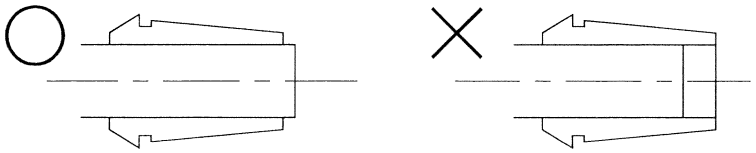
6. Insert the connecting shaft into the chucking lever and the ball screw shaft for opening/closing chucks, then secure the connecting shaft with the set screw.

7. Finally, tighten the lock screw of the chuck adjustment nut, then pull out the material.

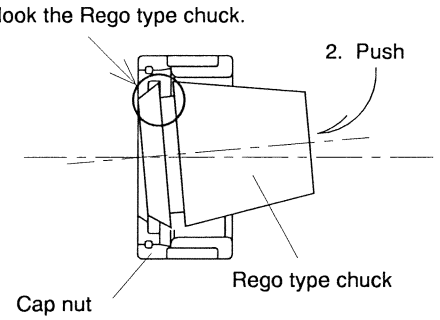
9.1.3 Mounting the Rego type chuck

CAUTION

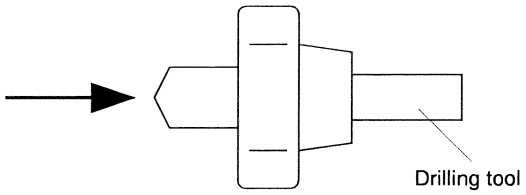
Be sure to insert a drilling tool into the Rego type chuck after mounting the Rego type chuck on the cap nut.
Insert the drilling tool until it goes through the Rego type chuck. Failure to do so may damage the tool or Rego type chuck.



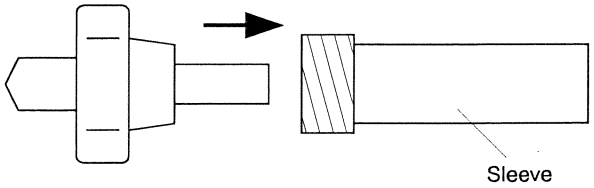
1. Mount the Rego type chuck on the cap nut.
   Confirm that the Rego type chuck is clean and free from any scratch. Wash the chuck if necessary.



2. Insert the drilling tool into the Rego type chuck.



3. Screw the cap nut on which the tool is set into the sleeve screw.
   Confirm that the inside tapered part of the sleeve is clean and free from any scratch. Wash the inside of the sleeve if necessary.



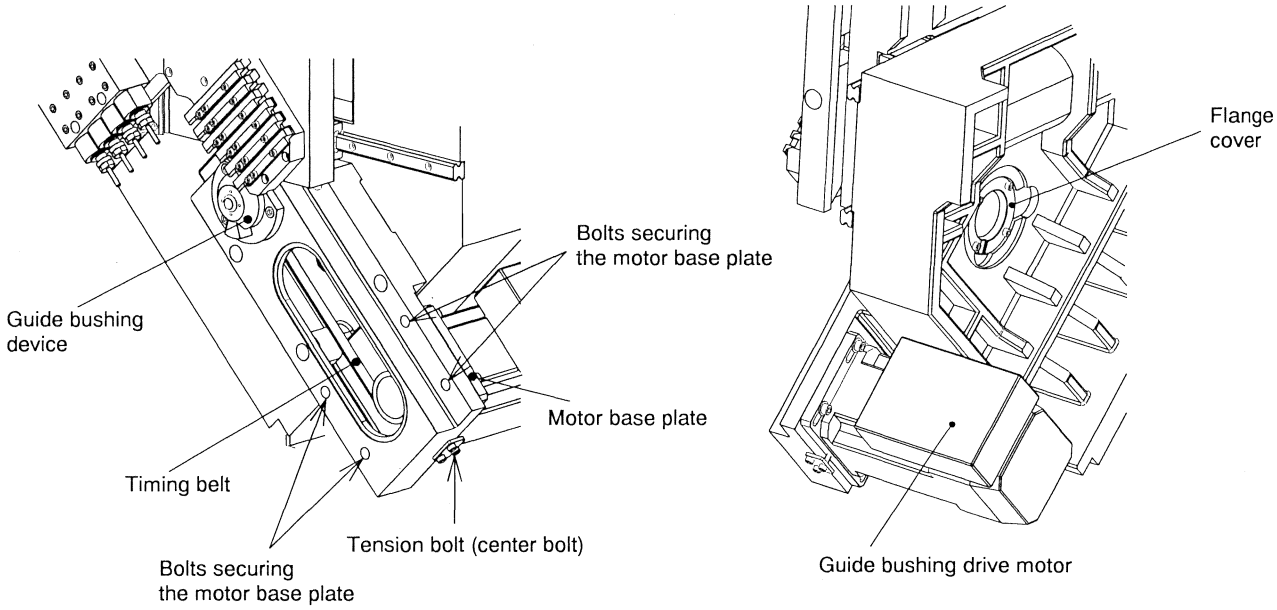
4. Hook the wrench onto the 2-face width section of the sleeve and hook the dedicated wrench onto the cap nut, then firmly tighten the cap nut. Both 2 wrenches are accessory tools provided with the machine.

9.2 Mounting/Adjusting a Guide Bushing Device

Two types of guide bushing devices are available: synchronous rotary guide bushing devices and fixed guide bushing devices. Confirm the type of the mounted guide bushing device, and check the guide bushing device on the Machine Structure screen. See <Section 6.22 Machine Structure Screen>.

9.2.1 Replacing a synchronous rotary guide bushing device

> ⚠ WARNING
>
> Be sure to confirm that the spindles and machine have stopped before attempting to replace the guide bushing device.
> The work during machine operation could result in serious personal injury.



1. Remove two covers of the timing belt in the coolant room and the guide bushing drive motor at the rear of the operation panel.

2. Loosen the bolts securing the base plate of the guide bushing drive motor. Loosen the tension bolt of motor base plate, then shift the motor base plate to loosen the timing belt.
   Note: Do not loosen the stopper bolt. This bolt is used as reference for applying tension to the belt.

3. Remove the set bolts from the flange cover at the rear of the guide bushing, then pull out the flange cover toward the rear.

4. Remove the set bolts from the flange of the guide bushing device.

5. Hold the flange, then slowly pull out the guide bushing device toward the front.

6. Insert the replacement guide bushing device into the mounting hole in the guide bushing device, then firmly tighten the set bolts.

7. Put the timing belts on the pulley of the guide bushing device.

8. Confirm that the timing belt is on the motor pulley normally.

9. Tighten the tension bolt until the motor base plate comes into contact with the stopper bolt and stops. The belt tension is suitable at this position.

10. Firmly tighten the set bolts of the motor base plate.

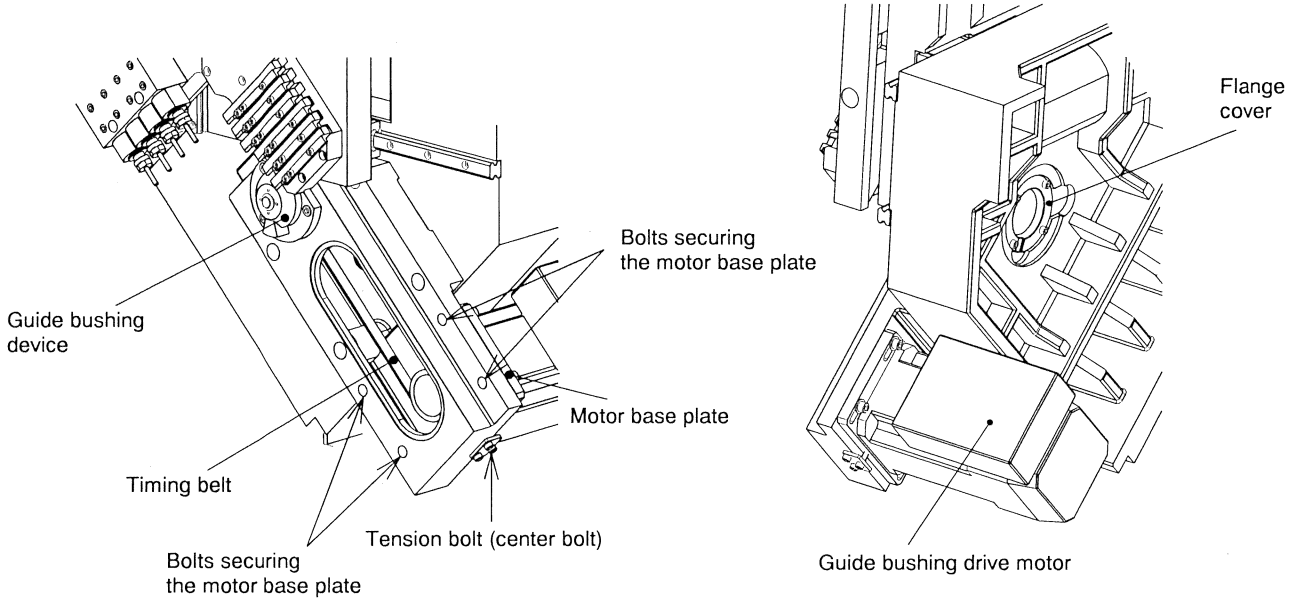
11. Mount the flange cover so that the drain hole faces down, then secure the flange cover with the set bolts.

> ⚠ CAUTION
>
> Be sure to mount the flange cover so that the drain hole faces down. Failure to do so causes oil to flow into the bearing, which may damage the bearing.

12. Mount two covers removed in Step 1.

9.2.2 Replacing a fixed guide bushing device

> **WARNING**
>
> Be sure to confirm that the spindles and machine have stopped before attempting to replace the guide bushing device.
> The work during machine operation could result in serious personal injury.



When replacing the synchronous rotary guide bushing device with a fixed guide bushing device

1. Remove two covers of the timing belt in the coolant room and the guide bushing drive motor at the rear of the operation panel.

2. Loosen the bolts securing the base plate of the guide bushing drive motor. Loosen the tension bolt of motor base plate, then shift the motor base plate to loosen the timing belt.
   Note: Do not loosen the stopper bolt. This bolt is used as reference for applying tension to the belt.

3. Remove the set bolts from the flange cover at the rear of the guide bushing, then pull out the flange cover toward the rear.

4. Remove the set bolts from the flange of the guide bushing device.
5. Hold the flange, then slowly pull out the guide bushing device toward the front.
6. Remove the timing belt.
7. Firmly tighten the set bolts of the motor base plate.
8. Slightly tighten the tension bolt to prevent it from dropping.
9. Mount two covers removed in Step 1.
10. Mount the flange cover dedicated to the fixed guide bushing device, and secure the flange cover with the set bolts.
11. Insert the replacement guide bushing into the mounting hole in the guide bushing device, then firmly tighten the set bolts.

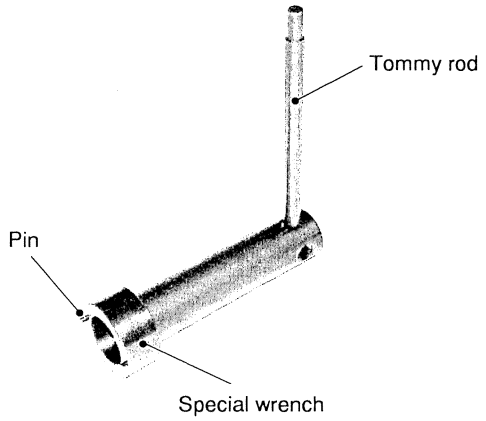
When replacing the fixed guide bushing device with another fixed guide bushing device

1. Remove the set bolts from the flange of the guide bushing device.
2. Hold the flange, then slowly pull out the guide bushing device toward the front.
3. Insert the replacement guide bushing into the mounting hole in the guide bushing device, then firmly tighten the set bolts.

9.2.3 Adjusting a fixed guide bushing device

> **WARNING**
>
> Be sure to confirm that the spindles and machine have stopped before attempting to replace the guide bushing device.
> The work during machine operation could result in serious personal injury.

1. Insert the pins of the special wrench (an accessory tool provided with the machine) into the tommy hole in the large-diameter nut at the rear (spindle side) of the guide bushing device, then turn the pin counterclockwise to loosen it.



2. Adjust the guide bushing sleeve position while moving the sleeve in the axial direction. Pay attention to the clearance between the tool and the opening of the guide bushing while doing so.

3. Firmly tighten the large-diameter nut with the special wrench (an accessory tool provided with the machine).

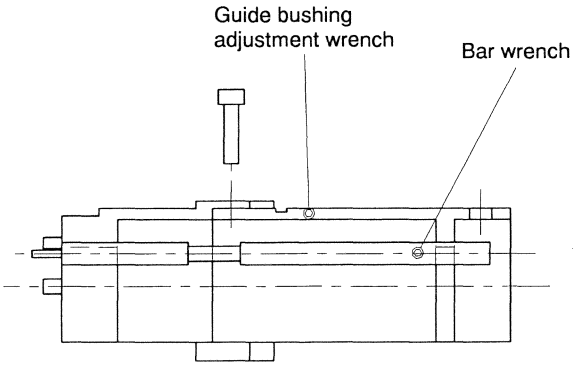
4. Remove all the tools you used.

> **CAUTION**
>
> Be sure to remove all the used tools at completion of adjustment. Machine operation with a tool mounted may cause serious machine damage.

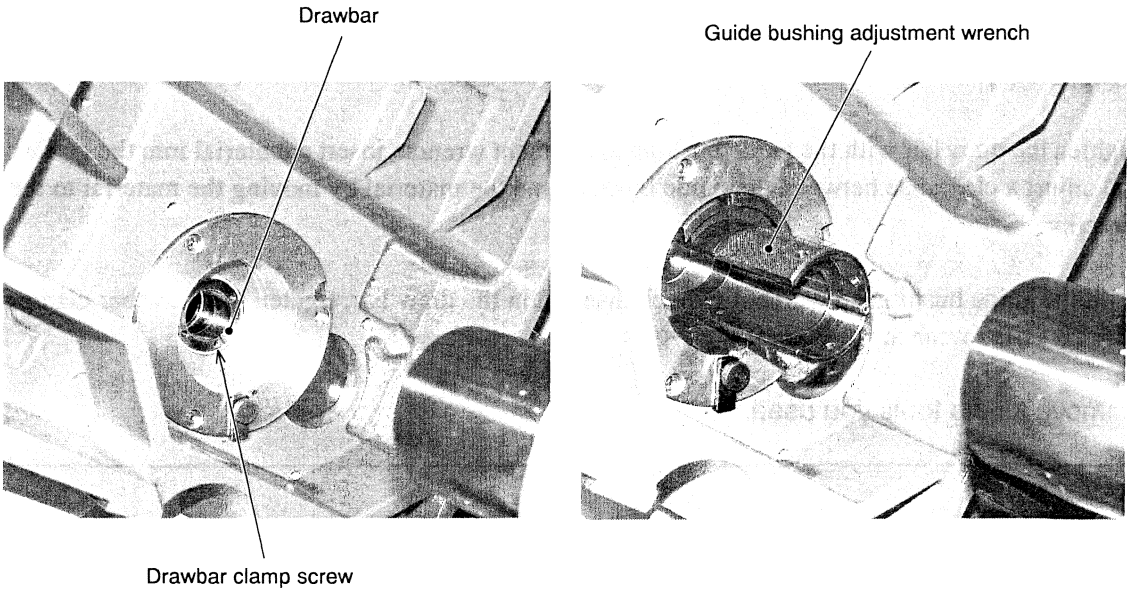
9.2.4 Replacing/adjusting the guide bushing

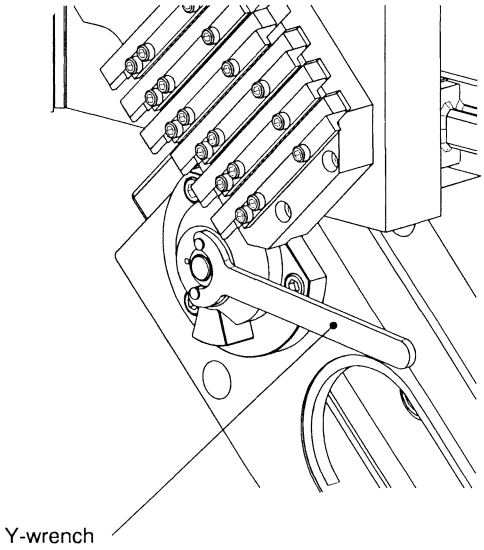
> **WARNING**
>
> Be sure to confirm that the spindles and machine have stopped before attempting to replace/adjust the guide bushing.
> The work during machine operation could result in serious personal injury.

1. Insert a bar wrench into the guide bushing adjustment wrench (an accessory tool provided with the machine).



2. Insert the guide bushing adjustment wrench (prepared in step 1) into the tommy holes in the drawbar at the rear of the guide bushing device, then loosen the drawbar clamp screw with the Allen wrench.





3. With a rotary guide bushing device, insert the provided Y-wrench into the tommy holes on the guide bushing sleeve to hold the guide bushing, then loosen the draw bar. With a fixed guide bushing device, turn the guide bushing adjustment wrench counterclockwise to loosen the draw bar.

4. Pull out the guide bushing toward the unit front.

5. Fit the key in the guide bushing sleeve into the key groove in the outer circumference of a new guide bushing, and insert the guide bushing into the sleeve.

6. Screw the draw bar into the end of the guide bushing, then turn the draw bar clockwise by hand to tighten it temporarily.

7. Tighten the draw bar with the guide bushing adjustment wrench, insert a material into the guide bushing and adjust a clearance between the guide bushing and the material by moving the material to the axis direction.

8. With the guide bushing adjustment wrench inserted in the draw bar, tighten the draw bar clamp screw with the Allen wrench.

9. Remove all the tools you used.

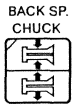
> ⚠ CAUTION
>
> Be sure to remove all the used tools at completion of adjustment/replacement. Machine operation with a tool mounted may cause serious machine damage.

10. Cut off or pull out the material.

9.3 Re-arranging the Back Spindle

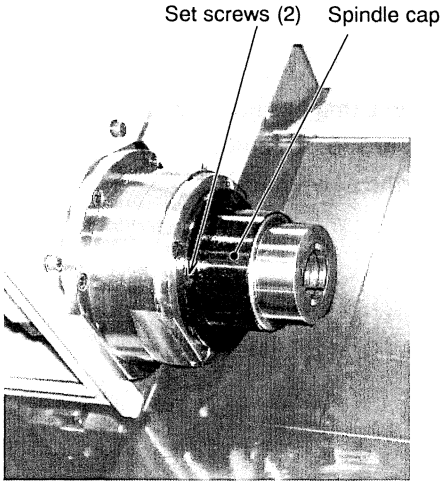
9.3.1 Mounting/replacing the back spindle chuck

> **WARNING**
>
> Be sure to confirm that the spindles and machine have stopped before attempting to replace the chuck.
> The work during machine operation could result in serious personal injury.

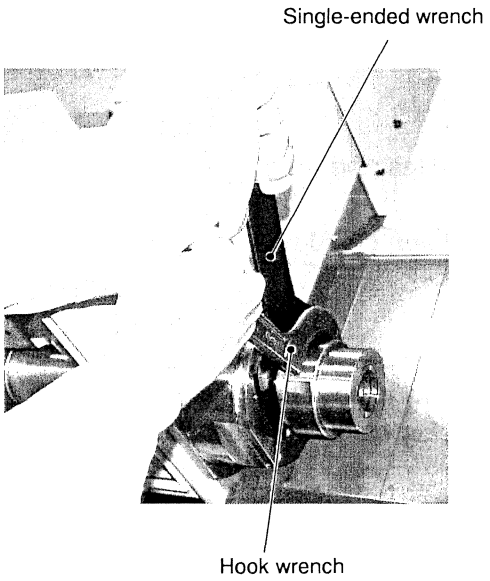
1. Press 

on the operation panel to open the back spindle chuck.
   Note: The BACK SP. CHUCK key on the operation panel is enabled in the preparation mode or manual operation mode.

2. Loosen the set screws, then pull out the spindle cap.

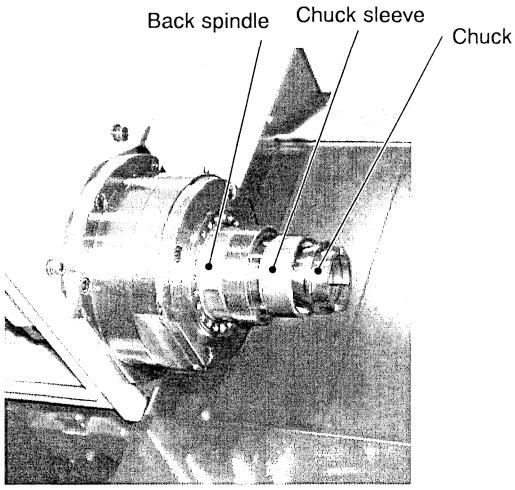


3. Hold the cap nut with the hook wrench (an accessory tool provided with the machine), insert a single-ended wrench into the 2-face width (36 in width) section, then loosen the cap nut. The cap nut is right-handed.



4. Turn the cap nut to remove it from the spindle.

5. Mount the chuck on the chuck sleeve that has been inserted in the spindle. At this time, confirm that the square spring is in the chuck sleeve. If another chuck has been mounted, hold the chuck and pull it out toward the front, then mount the new chuck.



6. Screw the cap nut into the spindle, then firmly tighten the cap nut with the hook wrench and single-ended wrench.

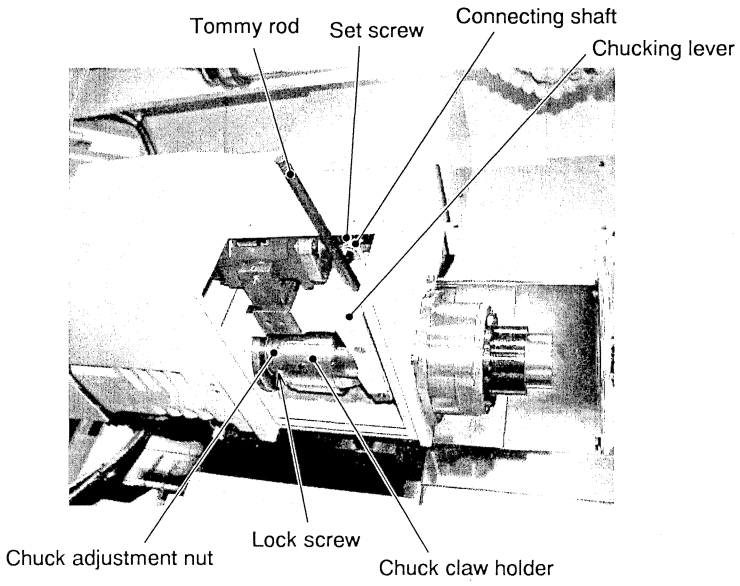
7. Finally, secure the spindle cap with the set screw. Be sure to mount the spindle cap so that the oval drain hole faces down.

CAUTION

Be sure to mount the spindle cap so that the oval drain hole faces down. Failure to do so causes oil to flow into the bearing, which may damage the bearing.

9.3.2 Checking the chucking force of the back spindle chuck

> **WARNING**
>
> Be sure to confirm that the spindles and machine have stopped before attempting to check the chucking force.
> The work during machine operation could result in serious personal injury.



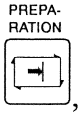
1. Press BACK SP. CHUCK on the operation panel to open the back spindle chuck, loosen the screw of the top cover of the back spindle device, then remove the top cover.

2. Loosen the lock screw of the chuck adjustment nut at the rear of the chuck claw holder.

3. Turn the chuck adjustment nut to move the chuck claw holder in the axial direction. To increase the chucking force, turn the chuck adjustment nut toward the front. To decrease the chucking force, turn the chuck adjustment nut toward the other side.

4. Insert a workpiece into the back spindle chuck.

> **CAUTION**
>
> Do not close the chuck without inserting any workpiece.
> Doing so may cause the machine to malfunction or may damage the collect chuck.

5. Press the BACK SP. CHUCK key to open and close the chuck with the workpiece at a suitable position, then confirm the chucking force in the back spindle chuck counter display area. To display the back spindle chuck counter, press 

, and then the menu key [CHCK ADJ].

6. Firmly tighten the lock screw of the chuck adjustment nut at completion of adjustment, then pull out the workpiece.

Note

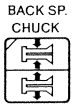
The chuck counter display area displays a reference value. Control the chucking force with the angle at which you tighten the adjustment nut. Check the chucking force with the chuck open/close tool when required.

Checking the chucking force of the back spindle chuck with the tommy rod

> **WARNING**
>
> Be sure to confirm that the spindles and machine have stopped before attempting to check the chucking force.
> The work during machine operation could result in serious personal injury.

Take the following procedure to check the chucking force manually, using the tommy rod (an accessory tool provided with the machine):

1. To roughly adjust the chucking force, take steps 1 to 4 of the procedure in <Section 9.3.2 Checking the chucking force of the back spindle chuck>.

2. Press 

on the operation panel to open the back spindle chuck.
   Note: The BACK SP. CHUCK key on the operation panel is enabled in the preparation mode or manual operation mode.

3. Loosen the set screw at the tip of the chucking lever, then pull out the connecting shaft (which secures the chucking lever and the ball screw shaft for opening/closing chucks) toward the front.

4. Insert the tommy rod (an accessory tool provided with the machine) into the hole in the upper part of the chucking lever, then check the chucking force while moving the chucking lever to right and left. Turn the chuck adjustment nut until obtaining suitable back spindle chucking force.

5. Set the chucking lever to open the back spindle chuck (turn the tommy rod to the left).

6. Insert the connecting shaft into the chucking lever and the ball screw shaft for opening/closing chucks, then secure the connecting shaft with the set screw.

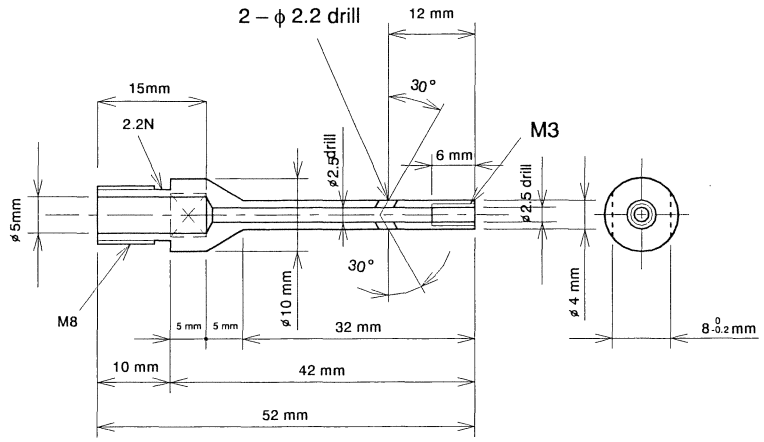
7. Finally, tighten the lock screw of the chuck adjustment nut, then pull out the workpiece.

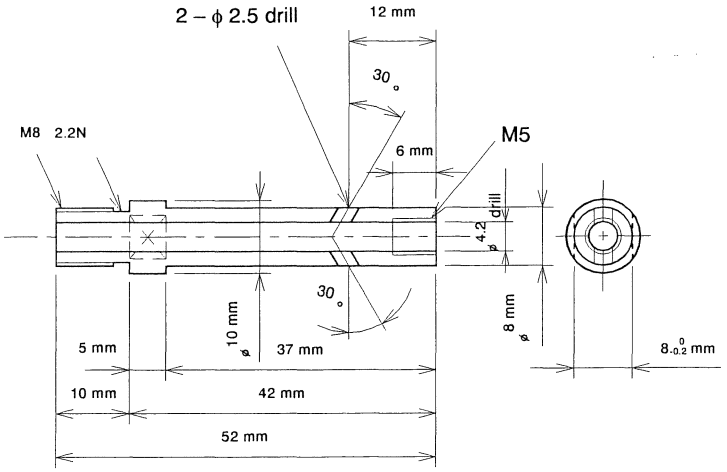
9.3.3 Replacing the knock-out jig

> **WARNING**
>
> Be sure to confirm that the spindles and machine have stopped before attempting to replace the knock-out jig.
> The work during machine operation could result in serious personal injury.

The knock-out jig needs to be replaced depending on the workpiece shape (e.g., chuck diameter or through-hole diameter).

The following shows the two types of jigs provided as accessories. If the jigs are unable to knock out the workpieces, it is necessary to manufacture a jig in accordance with the workpiece shape.





1. To dismount the back spindle chuck and the chuck sleeve, take in <Section 9.3.1 Mounting/replacing the back spindle chuck>.

2. Move the knock-out jig forward in the MDI mode (M10 command). See <Section 7.5.9 MDI operation>.

3. The knock-out jig extends from the front end of the back spindle. To replace the knock-out jig, hook the wrench onto the 2-face width section of the knock-out jig and the 2-face width section of the knock-out pipe at the rear of the back spindle.

4. Move the knock-out jig backward in the MDI mode (M11 command).

5. Mount the chuck sleeve and the chuck on the back spindle, then take step 6 and the subsequent steps of the procedure in <Section 9.3.1 Mounting/replacing the back spindle chuck> to mount the cap.

Note

An alarm is issued if the MDI command M10 or M11 is executed while the back spindle chuck is closed.

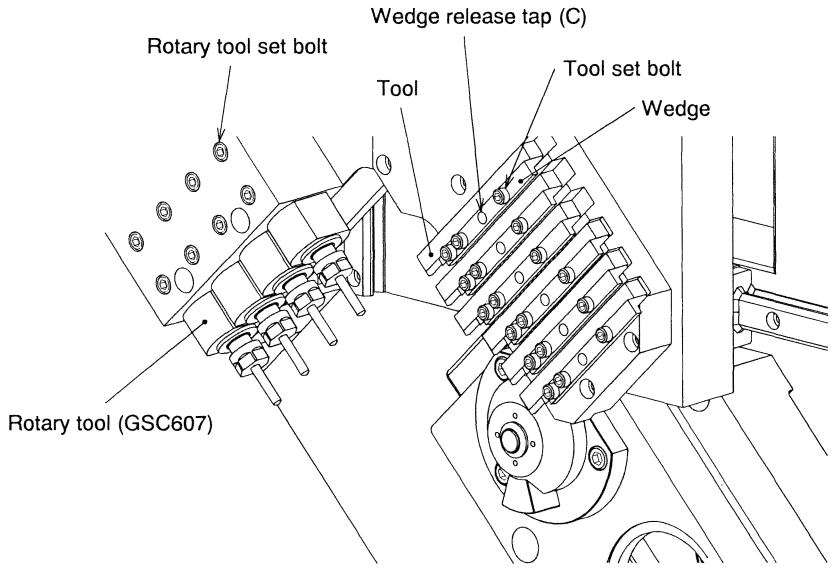
9.4 Mounting/Adjusting Tools

9.4.1 Mounting a tool on the holder of the gang tool post

> ⚠ WARNING
>
> Be sure to confirm that the spindles and machine have stopped before attempting to mount/dismount a tool.
> The work during machine operation could result in serious personal injury.

This section explains the mechanical mounting of a tool. For the tool setting, see <Section 7.3.11 Setting tools>.

1. Insert the tool into the holder, then evenly tighten the wedges with three tool set bolts. The tightening torque of the tool set bolts is 15N m (1.5 kg m), which is obtained by tightening them with the Allen wrench provided as an accessory.



2. Also tighten the set bolt to secure a wedge where no tool is mounted.

> ⚠ CAUTION
>
> Be sure to tighten the set bolts to secure the wedge of a groove in which a tool on the holder is not mounted. The extent of the wedge causes interference. The interference may cause serious machine damage.

9.4.2 Dismounting a tool

> **WARNING**
>
> Be sure to confirm that the spindles and machine have stopped before attempting to mount/dismount a tool.
> The work during machine operation could result in serious personal injury.

1. Loosen the tool set bolts, then dismount the tool. If the tool does not come out because the wedge clings tight to it, remove the tool set bolts, screw a bolt into the wedge release tap (service hole) at the center of the wedge, and dismount the tool. (See (C) in the figure in the previous section.)

9.4.3 Mounting a rotary tool (e.g., GSC607) on the holder of the gang tool post

This section explains the mechanical mounting of a tool. For the tool setting, see <Section 7.3.11 Setting tools>.

WARNING

Be sure to confirm that the spindles and machine have stopped before attempting to mount a tool. The work during machine operation could result in serious personal injury.

CAUTION

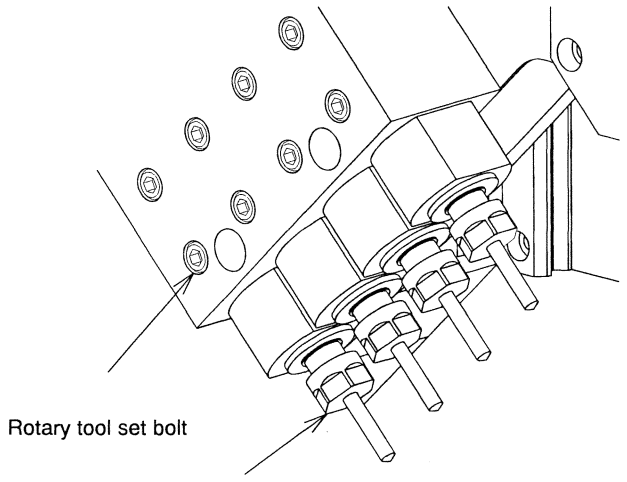
- Firmly tighten the set bolts of the tool. Failure to do so may cause the tool to come off during machine operation, which may cause serious personal injury.
- Be very careful not to hurt yourself with the cutting edge of a tool when mounting/dismounting the tool.

1. Insert the rotary tool into the hole in the holder so that the stationary face (D cut face) of the rotary tool fits to the rotary tool set bolts. At this time, insert the rotary tool until it comes into contact with the end face of the holder.

2. Evenly secure the rotary tool with the two set bolts.

3. Insert a tool (e.g., drill or end mill) into the rotary tool.

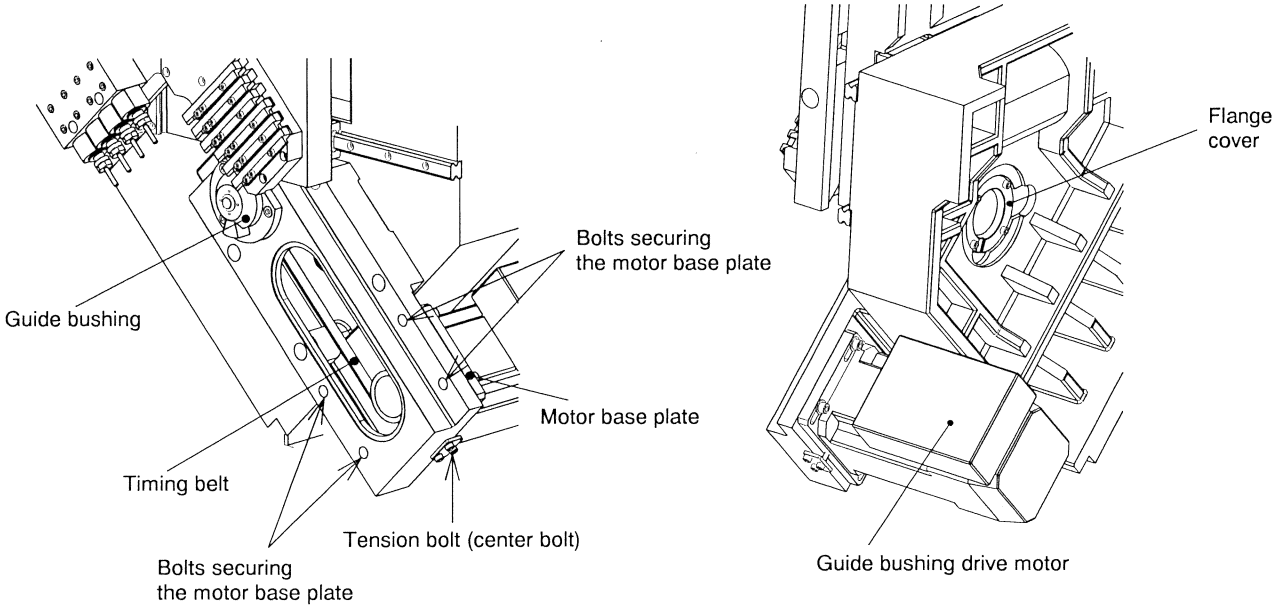
4. Firmly secure the tool by tightening the nut with the wrench (an accessory tool provided with the machine).



9.5 Replacing Belts

9.5.1 Replacing the timing belt for driving the synchronous rotary guide bushing

> ⚠ WARNING
>
> Be sure to confirm that the spindles and machine have stopped before attempting to replace the belt.
> The work during machine operation could result in serious personal injury.



1. Remove two covers of the timing belt in the coolant room and the guide bushing drive motor at the rear of the operation panel.

2. Loosen the bolts securing the base plate of the guide bushing drive motor. Loosen the tension bolt, then shift the motor base plate toward the rear to loosen the timing belt. To shift the motor base plate, push the guide bushing drive motor from the front side of the machine.
   Note: Do not loosen the stopper bolt. This bolt is used as reference for applying tension to the belt.

3. Remove the set bolts from the flange cover at the rear of the guide bushing, then pull out the flange cover toward the rear.

4. Remove the set bolts from the flange of the guide bushing device.

5. Hold the flange, then slowly pull out the guide bushing device toward the front until you can release the belt. At this time, you do not have to pull out the guide bushing device all the way.

6. Remove the old timing belt. Clean the pulley if dust resulting from belt wear and chips are stuck to it.

7. Put the new timing belt on the pulley of the guide bushing device and the motor pulley.

8. Push the guide bushing device to the original position, then firmly tighten the set bolt.

9. Confirm that the timing belt is on the pulleys normally.

10. Tighten the tension bolt to apply tension to the belt. You do not have to bring the motor base plate into contact with the stopper bolt. The belt has initial extension, so do not apply excess tension to the belt.

11. Firmly tighten the set bolts of the motor base plate. As test operation, rotate the spindle at the speed of 5,000 min^{-1} for about eight hours.

12. Stop the spindle, loosen the set bolts of the motor base plate again and tighten the tension bolt until the motor base plate comes into contact with the stopper bolt. The belt tension is suitable at this position.

13. Firmly tighten the set bolt of the motor base plate.

14. Mount the flange cover so that the drain hole faces down, then secure the flange cover with the set bolt.

> ⚠ CAUTION
>
> Be sure to mount the flange cover so that the drain hole faces down. Failure to do so causes oil to flow into the bearing, which may damage the bearing.

15. Mount two covers removed in Step 1.

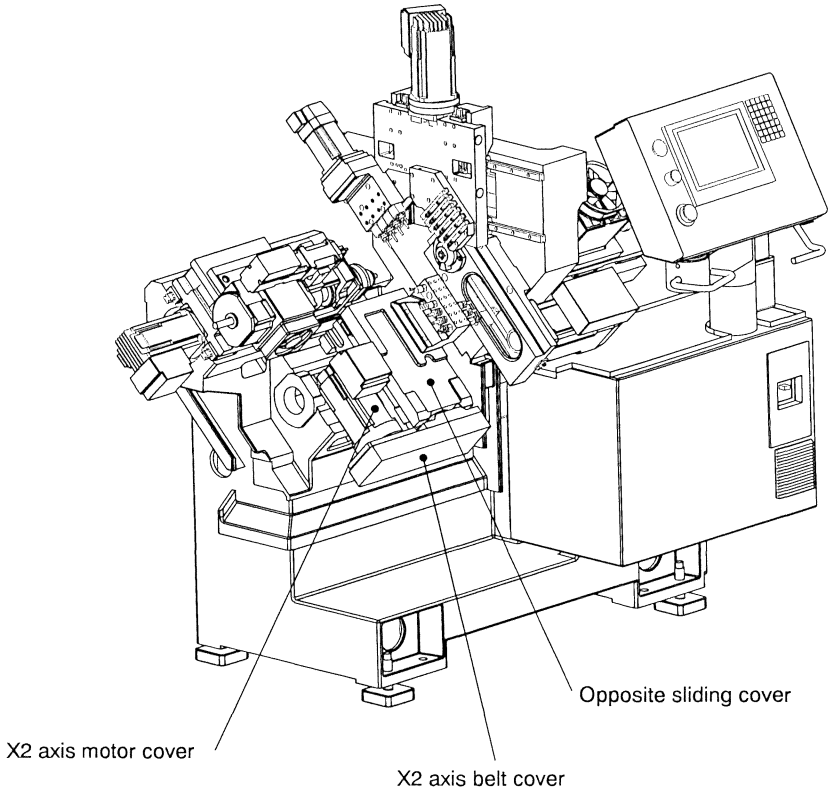
Note

If non-conform material machining is suspended because of belt replacement, execute Non-conform material phase adjustment (execute the G899 command) at completion of belt replacement. For the procedure, see <Section 5.7.7 Non-conform material phase adjustment (G899)>.

9.5.2 Replacing the timing belt for X2 axis feed (Type VII, VIII, IX)

> ⚠ WARNING
>
> Turn off the main breaker of the machine before attempting to replace the timing belt for X2 axis feed.
> Failure to do so causes the tool post to drop, which could result in serious personal injury.

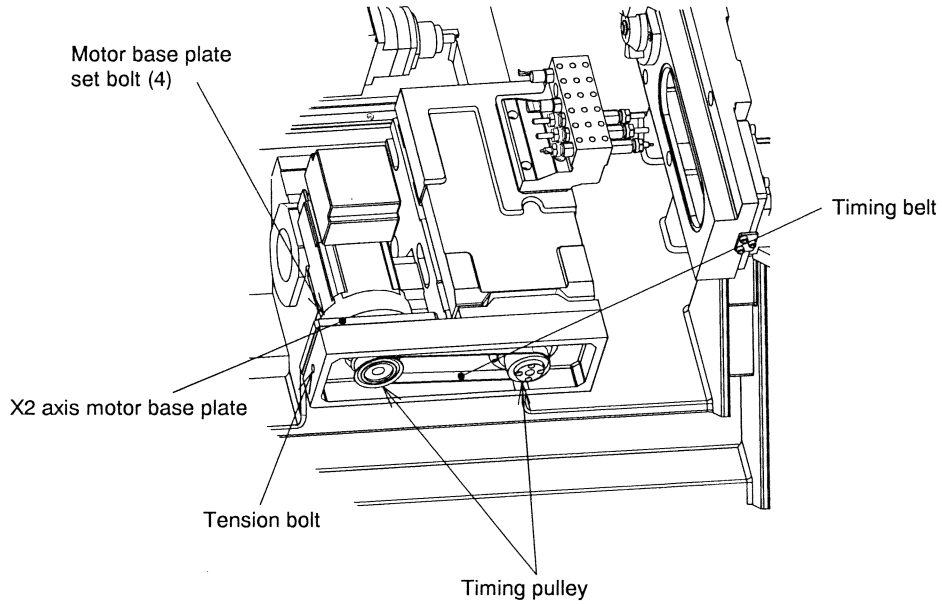
> ⚠ CAUTION
>
> Be sure to replace the timing belt for X2 axis feed not to move the timing pulley, then return the X2 axis to zero point. Failure to do so causes the damage of the machine.



1. Remove the coolant tank.

2. Remove the opposite sliding cover and X2 axis motor cover.

3. Remove X2 axis belt cover.



4. Turn off the main breaker of the machine according to the procedure in <Section 7.1.2 Turning off the power>.

5. Loosen the four bolts securing the base plate of the X2 axis motor. Loosen the tension bolt, then shift the motor base plate toward the right side of the machine to loosen the timing belt. To shift the motor base plate, push the X2 axis motor from the left side of the machine. Be sure not to move the timing pulley at this time.

6. Remove the old timing belt. Clean the pulley if dust resulting from belt wear and chips are stuck to it. Be sure not to move the timing pulley at this time.

7. Put a new timing belt on the ball screw pulley of X2 axis and on the motor pulley. Be sure not to move the timing pulley at this time.

8. Tighten the tension bolt to apply tension to the belt. Be careful not to apply excess tension. Apply tension enough to avoid deflection.

9. Firmly tighten the set bolts of the motor base plate.

10. Mount the cover removed in Step 3.

11. Mount two covers removed in Step 2.

12. Mount the coolant tank.

13. Be sure to return the X2 axis to zero point. See <Section 7.11.1 Returning axes to zero point>. Check that the X2 axis positions at the original point.

9.6 Replacing a Filter

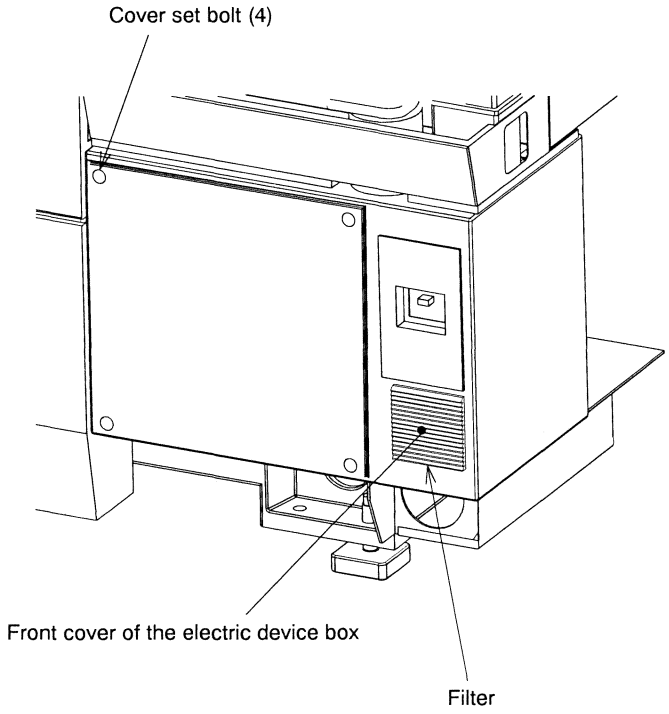
9.6.1 Replacing the fan filter of the electric device box

> ⚠ DANGER
>
> Be sure to turn off the main breaker of the machine before attempting to replace the fan filter. Failure to do so will result in death or severe personal injury from electric shock.

1. Turn off the main breaker of the machine according to the procedure in <Section 7.1.2 Turning off the power>.

2. Remove the cover at the front of the electric device box, then dismount the old filter.

3. Mount a new filter with the rough surface outside, then close the cover.



Product Code C – C 1 2 1 6 VI VII VIII IX – 7 7 0

Document Code 5 E 1 – 0 9 0 2

10. Scheduled Maintenance

(Blank page)

10.1 Scheduled Maintenance Checks

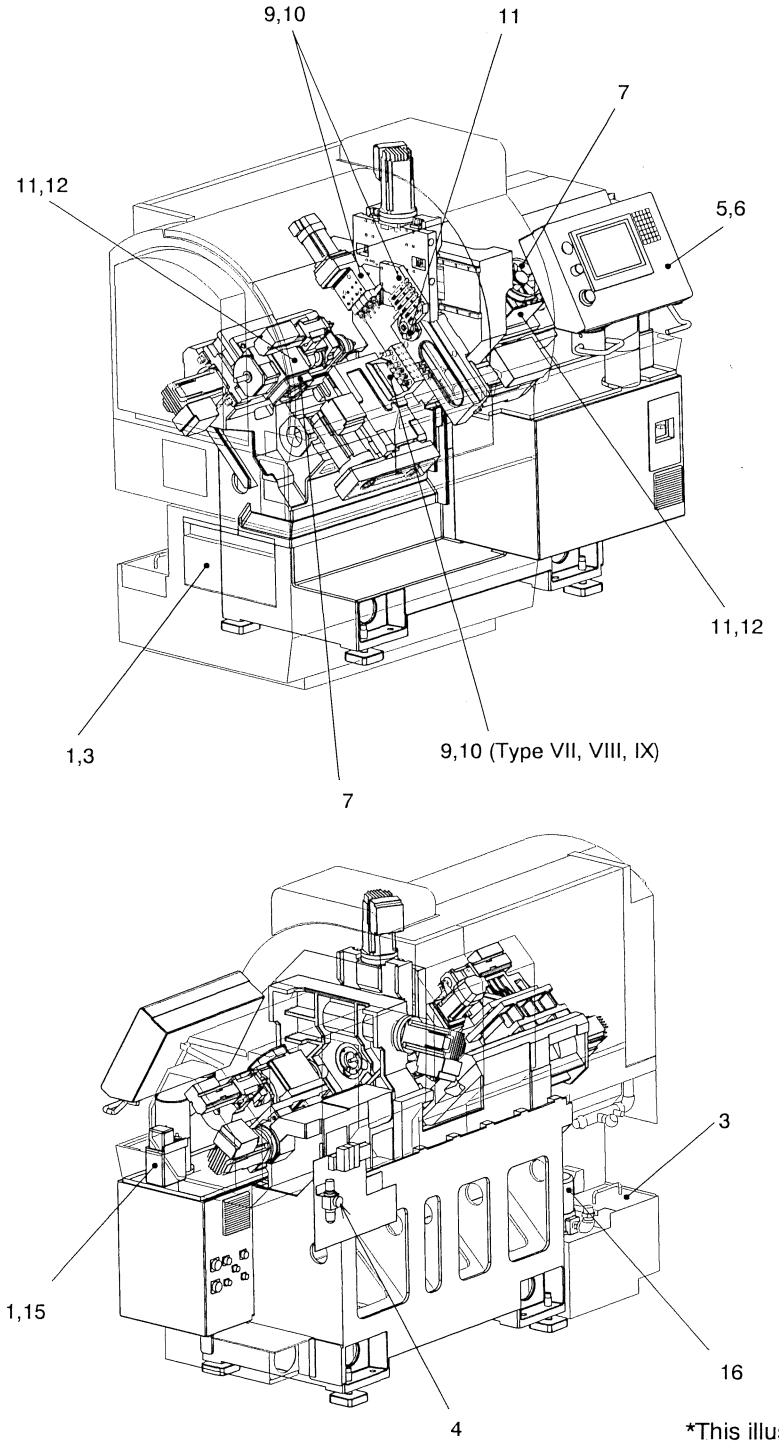
Periodic maintenance checks of the machine, regardless of the actual failures, are required for prediction or earlier detection of a problem. And it will minimize the loss to the production.
This section summarizes the daily, monthly, and biannual maintenance check items.

CAUTION

Periodic maintenance checks are mandatory. If neglected, serious damage to the machine may result.

10.1.1 Daily

Daily maintenance checks are simple inspections to be performed before and during operation of the machine. For the machine, perform the checks listed in the table below. Note also that you can detect some abnormal conditions of the machine, including those covered in the table, through your senses, for example, by checking a change in machine noise, temperature, surface conditions of machined workpieces, or in smell during operation. Therefore, be familiar with the normal operating conditions of the machine, and you can accordingly perform the potential but most effective daily inspection only by observing machine conditions during daily operation. Keep in mind that this approach is the most important part of machine maintenance.



*This illustration is for type VII.

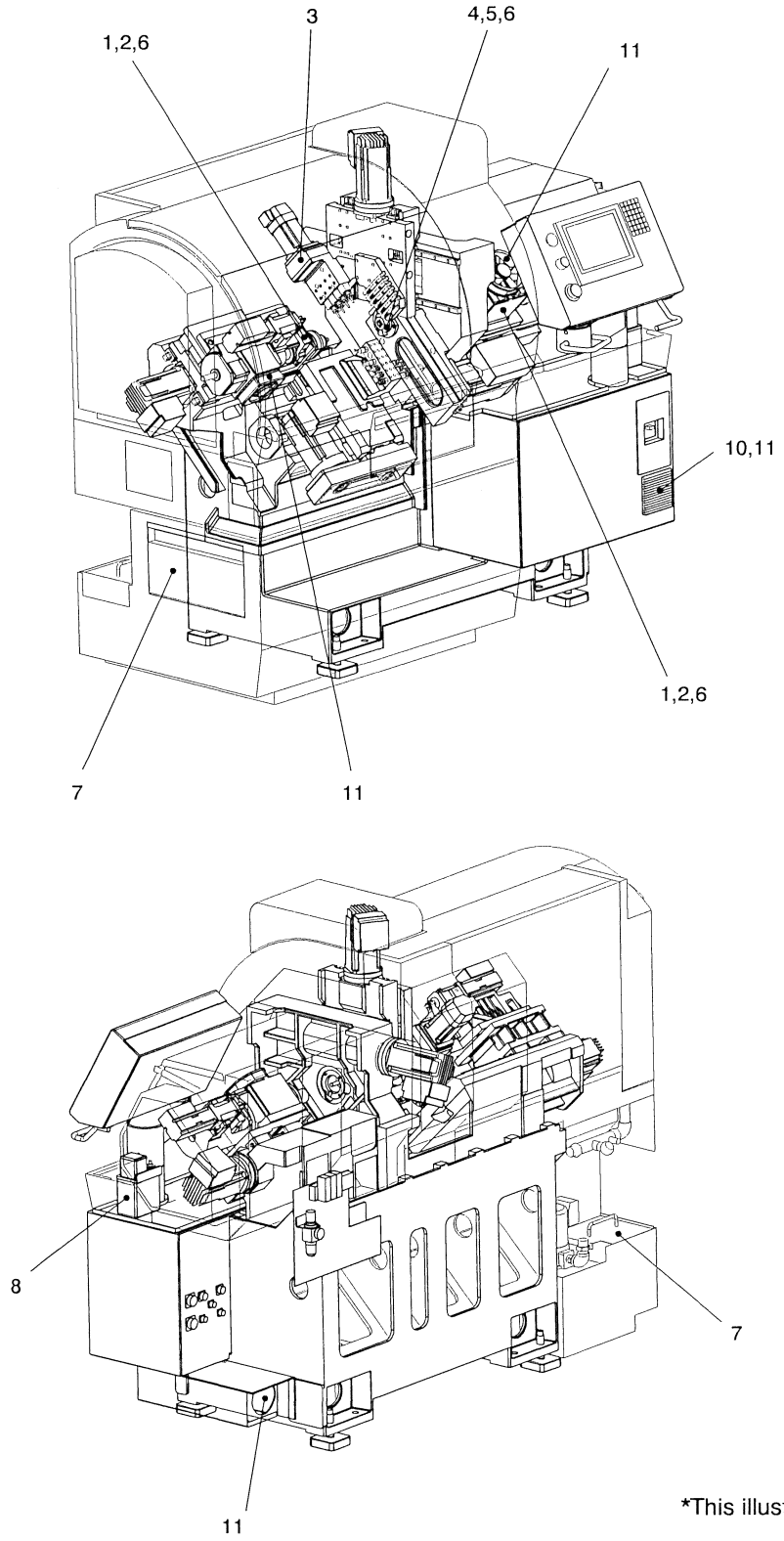
Daily Maintenance Check Points

Daily check items

Type	Check item	Check method or point
Before turning the power on	1. Coolant level, lubricating oil level. Air pressure.	Check the coolant level with the coolant gauge. Check the level of lubricating oil in the transparent tank to make sure that the tank contains at least one-fourth as much oil as the tank capacity. Check the air pressure is about 0.5MPa.
	2. Arrangement of tools and measuring instruments	Be careful in particular not to leave any tool or instrument on or near any moving part of the machine.
	3. Removal of chips	Remove chips from the oil pan as soon as possible. Take out the chip receiver box and remove accumulated chips. Be sure to remove chips according to <Section 10.1.4 Removing Chips>
	4. Has the air equipment been drained?	Check if water has accumulated in the drain tank of the filer regulator. If so, push the drain button at the lower part using a cloth.
After turning the power on	5. ALM (alarm) lamps on the operation panel	Make sure that the LCD displays no alarm and that no alarm lamp is lit on the machine operation panel.
	6. Lamp operation on the operation panel	Make sure that all push-button switches and lamps on the operation panel work normally.
	7. Rotation of the main spindle and back spindle cooling fans	Put your hand in front of the outlet to make sure that it is running normally.
	8. Operation of automatic fire extinguisher (if installed)	Check if the lamp on the operation panel of the automatic fire extinguisher lights indicating the normal state. Check that the Start button is not damaged. Open the fire extinguisher door, and check that the gas cylinder is not empty.
After preparation	9. Mounting conditions of tools	Check if any fixing screw is loose between the tool and the holder and between the holder and the tool post.
	10. Tool damage	Check for any damage to the cutting edge of each tool.
	11. Adjustment of guide bushing and chuck	Check the clearance between the material and guide bushing, chucking force of the main spindle and the back spindle.
	12. Lubrication of sliding parts of the bobbin	If the sliding parts of the bobbin are dry, apply grease to them.
During machine operation	13. Abnormal noise during operation	Check for abnormal noise from any rotary or sliding part of the machine.
	14. Unusual symptom	Make sure that the machine is operating as it should. Check for abnormal noise, temperature, machine or tool damage, smell.
	15. Lubricating oil pump	Check that the motor shaft in the transparent tank rotates so that the manual pump handle goes up to the top to naturally fall down once per 30 minutes.
	16. Coolant pump	Look into the window in the upper part of the pump to check that the cooling fan is turning in the direction of the arrow. Check also that the coolant nozzle is jetting coolant.

10.1.2 Monthly

The monthly maintenance check is required to find out problems and troubles that are not found out by daily routine checks. Do not omit the monthly maintenance check although it is a little complicated and time-consuming.



*This illustration is for type VII.

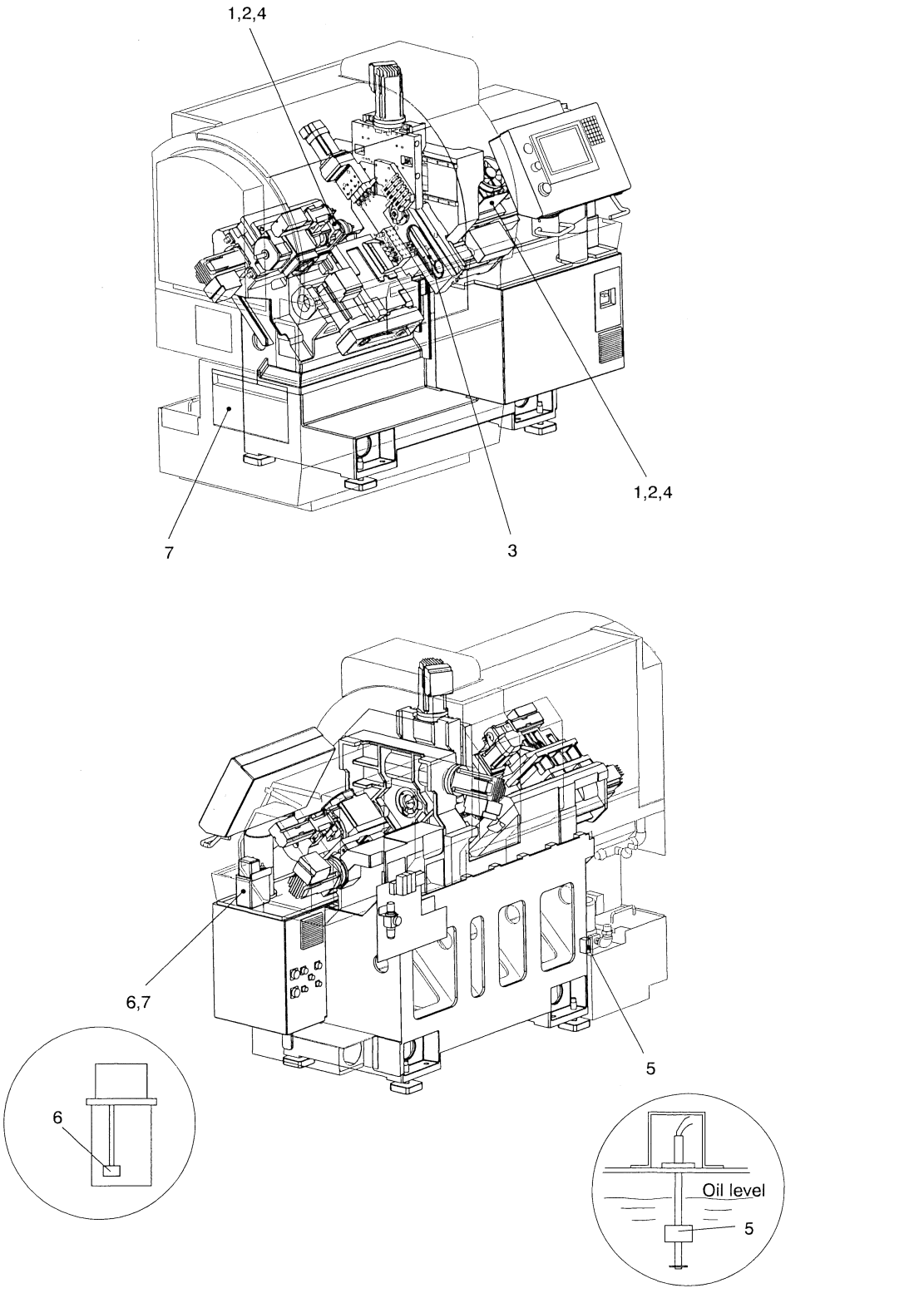
Monthly Maintenance Check Points

Monthly check items

Type	Check item	Check method	Criteria and action
Main spindle, back spindle	1. Overheating of the bearing	Stop the spindle after turning it at about 3000 min^{-1} for 30 minutes.	Touch the housing with your hand to make sure that it is not hot. Spindle operation is normal if the housing is warm.
	2. Wear of the chuck finger	Visually check the chucking claw.	If the chuck finger is remarkably worn out, replace the chucking claw.
Tool posts	3. Lubrication of the tool spindle and spur gear	Remove the tool spindle cover and check for lubrication visually and by touching.	Apply lubricating grease to the gear if it is dry.
Guide bushing device	4. Guide bushing inner sleeve hole	Check the inner sleeve hole visually and by touching.	Make sure there is no defect such as flaw or burr.
	5. Chips clogging the guide bushing front cap	Remove the guide bushing front cap.	Take out chips from the cap.
	6. Chuck and guide bushing	Remove the collet chuck and springs, then remove the chuck sleeve, intermediate sleeve, and balance sleeve.	Clean the spindle inside and the removed components. Also clean the guide bushing.
Coolant device	7. Check for chip accumulation in the areas around the coolant pump, on the outsides of the partitions in the oil pan, and in the coolant tank.	Remove the partition from the side on which the coolant pump is mounted and the partitions from the oil pan. See <Section 10.1.4 Removing chips.> Draw the coolant tank and check for chip accumulation in the tank.	Remove chip accumulation from the coolant tank, the areas around and under the coolant pump, and the corners of the oil pan.
Lubricating oil device	8. Pump unit discharge amount	Check the manual pump handle falling speed.	If the handle falls straight down faster, inspect the piping.
	9. Damage or oil leak in lubricating oil piping parts	Visually check the piping parts and pipes.	If oil leaks from a piping joint, replace the oil seal or tighten up the joint. Replace the pipe if it has been cracked or cut.
Filter	10. Dirt condition of filter	Remove the filter and check it visually.	Clean the filter if it is dirty. See <Section 10.1.5 Cleaning the fan filter for electric device box>.
Cooling device	11. Operation of cooling fans	Cooling fans are located on main spindle, back spindle, guide bushing motor, regenerative resistor mount section, air inlet on the rear of the electric device box, upper face of the NC unit in the operator panel, and on the amplifiers in the electric device box. Put your hand in front of the outlet to make sure that it is running normally.	Replace the fan if it does not work normally.
Wiring of motor	12. Connectors of the motor	Check looseness of the connector.	Retighten the connector when loosening.

10.1.3 Biannual

A biannual maintenance check is to determine the time for replacement of worn parts and parts which rarely become defective. These maintenance checks are essential. Perform following checks every six months in addition to monthly maintenance checks. If it is omitted, a fundamental serious trouble may occur.



*This illustration is for type VII.

Biannual Maintenance Check Points

Biannual check items

Type	Check item	Check method	Criteria and action
Main spindle, back spindle	1. Runout of the spindle straight hole	Apply the dial indicator to the opening of the hole used for receiving the spindle chuck sleeve and slowly turn the spindle once.	Spindle operation is normal if the pointer falls within the tolerance specified in the inspection table supplied with the machine (TIR0.005).
	2. Wear of the chuck bobbin	Visually check the outer circumference of the chuck bobbin.	The chuck bobbin is acceptable if it has not been worn out.
Guide bushing device	3. Slack and tension in the timing belt for driving the guide bushing	Remove the belt cover to visually check how much the belt has worn. Press the center part of the belt lightly with your finger for its tension.	Replace the belt if it has been peeled or cracked on the surface. Make sure that the belt is tensioned so that the belt and the pulley may be well engaged without slipping.
	4. Overheating, rattling noise or thrusting play of the bearing	Stop the spindle after turning it at about 3000 min^{-1} for 30 minutes. Push and pull the sleeve manually in the axial direction.	Touch the housing with your hand to check for abnormal heat. Check for abnormal noise during rotation. Check for any play.
Coolant device	5. Operation of the coolant tank float switch	Drain all of the coolant from the tank.	Check that the LCD panel displays a message that warns the empty of coolant.
Lubricating oil device	6. Operation of the pump unit float switch	Drain all of the lubricating oil from the pump unit tank.	Check that the LCD panel displays a message that warns the empty of lubricating oil.
Coolant tank and lubricating oil tank	7. Fine chips, iron chips, oil-cake residues in each tank	Check if chips or oil-cake residues remain at the bottom of the tank after draining oil.	Remove chips and oil-cake residues and charge new oil.

10.1.4 Removing chips

WARNING

Never discharge coolant for machining with fine chips accumulated at the bottom of the coolant tank.
Doing so may cause not only damage to the coolant pump but also a fire which involves serious personal injury or death at worst.

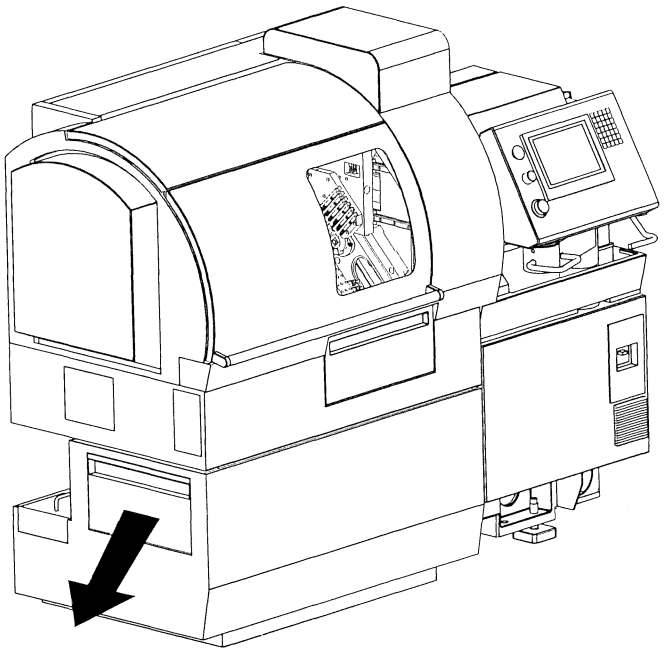
DANGER

- Be sure to turn off the main circuit breaker before removing the wiring connector of the coolant tank from the machine.
  Failure to do so will result in death or serious personal injury from electric shock.
- Take out chips with the attached chip remover rod.
  Failure to do so will result in serious personal injury from chips.

CAUTION

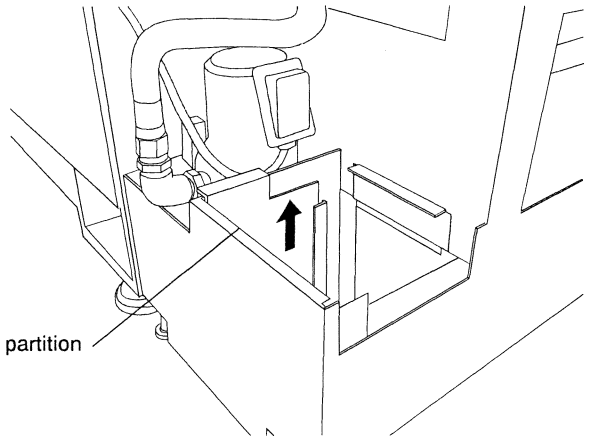
Be sure to disconnect the wiring connectors before separating the coolant tank from the machine. Otherwise, the coolant pump cable and the level detection cable may be damaged.

1. Remove larger chips. Open the chip outlet door provided on the lower part of the left side of the machine, take out chips with the attached chip remover rod, and close the door.

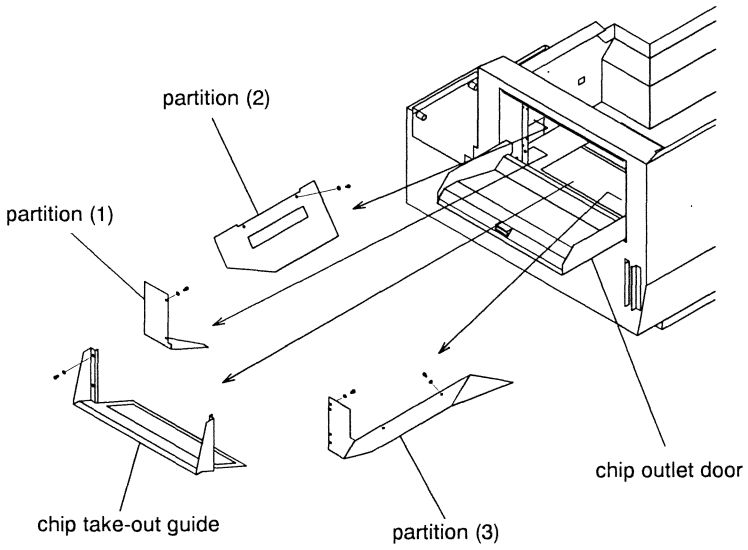


2. Remove fine chips. Remove the oil splash protection cover provided at the back of the left side of the machine, gently lift up the chip receiver box, throw away chips, then remount the chip receiver box and the cover.
   Sometimes check whether the filter in the chip receiver box is clogged. Clean the filter with a wire brush if it is clogged.

3. Remove accumulated chips from the areas around and under the coolant pump. Remove the oil cover, the chip receiver box, and the partition on the side on which the coolant pump is mounted. Remove chips around the suction opening of the pump, using a shovel, etc. Return the cover and chip receiver box to the original positions.



4. Remove chip accumulation from the outsides of the partitions in the oil pan. Open the "chip outlet door" and remove the chip take-out guide. Remove partition (1), partition (2), and then partition (3). After removing chip accumulation in the corners of the oil pan, return partitions (1) to (3) and then the chip take-out guide to the original positions. Check regularly if partitions (1) to (3) clog with chips, and clean them with a metal brush if necessary.



Notes

When the machine is used for a long time period, very fine chips may accumulate at the bottom of the coolant tank.
Keep the coolant tank clean by removing chip accumulation whenever necessary. Draw the coolant tank, remove the floorboard on which chips have accumulated, and remove the chip accumulation.

10.1.5 Cleaning the fan filter for electric device box

> ⚠ DANGER
>
> Be sure to turn off the main circuit breaker before removing the fan filter from the machine. Failure to do so will result in death or serious personal injury from electric shock.

1. Turn off the main circuit breaker according to procedure described in <7.1.2 Turning off the power>.

2. Remove the door located on the front of the electric device box, then remove the filter.

3. Blow off the dust adhered to the filter with the air gun. Or, wash the dust with volatile oil if the filter is very dirty.

4. Attach the filter so that its rough side faces outside, then mount the door as it was.

10.2 Lubrication

For trouble-free operation, the machine requires lubrication by additionally supplying or replacing appropriate oils on a regular basis.
This section describes lubrication required for the machine.

10.2.1 Lubrication list

For details of lubrications, see <Section 10.2.2 Notices for lubrications and oil change>.

Lubricating Position	Frequency	Volume	Lubrication type and name	Manufacturer
Slide lubricating oil pump	a	0.8 lit.	Mobil Vactra No. 2 or its equivalent	Mobil
Coolant tank	a	100 lit.	Unicut Terami DX30 or its equivalent	Nippon Oil Corporation
Bobbin sliding portion	a	Adequate	Mobilith SHC220 or its equivalent	Mobil
Bar feeder (U10J) rod	Weekly	Adequate	Mobil Vactra No. 2 or its equivalent	Mobil
Ball screw	c	Adequate	Multemp LRL-3 or its equivalent	Kyodo Yushi Co., Ltd.
Gang tool spindle spur gear	b	Adequate	Mobilith SHC220 or its equivalent	Mobil

Lubrication frequency:
a: Indicates that the components/parts should be lubricated during daily inspection if necessary.
b: Indicates that the components/parts should be lubricated during monthly inspection if necessary.
c: Indicates that the components/parts must always be lubricated during monthly inspection.

Notes

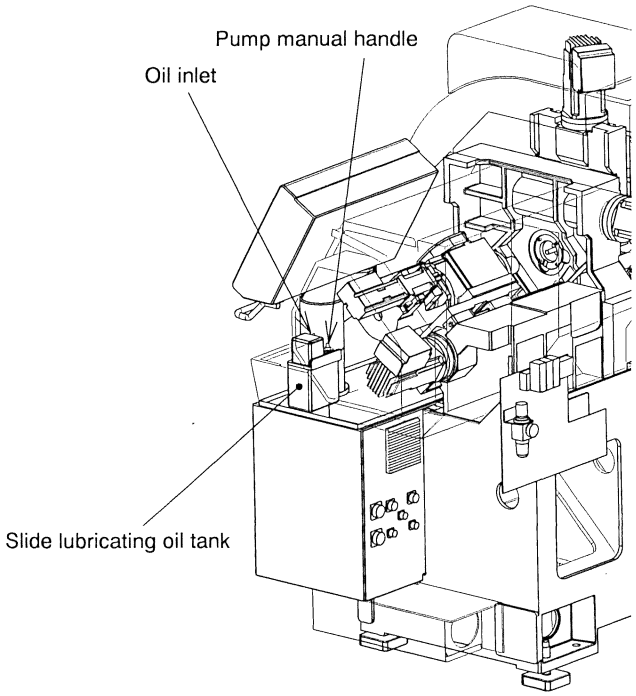
- The slide lubricating oil pump supplies the lubricating oil to the machine during the power is on. And the discharge amount of it is adjusted at 2 cc/30 min.
- The spindle bearing and the support bearing of each axis ball screw are lubricated with high quality, long life grease (Isoflex NBU15 manufactured by Cluber), requiring no additional lubrication except an appropriate amount of the grease used when the spindle bearing is replaced.

10.2.2 Notices for lubrications and oil change

Slide lubricating oil

> **CAUTION**
>
> Do not push the pump manual handle down forcibly.
> Doing so will cause damage to the lubricating pump gear.

- When the slide lubricating oil remaining in the tank has come down to Low Level, pour clean oil into the tank through the oil inlet port until the tank is at full capacity.
- Before starting the machine left off for an extended period of time, repeat pulling up and releasing the pump manual handle several times to supply lubricating oil to each part of the machine.
- If the pump manual handle falls faster than usual, a damage to the pipe is assumed. Check the pipe.



Lubricating Oil Pump Unit

Coolant

> **WARNING**
>
> Monitor the level and condition of the coolant in the machine every day. Low or ineffective coolant can result in damage to the tool and a possible fire.

- The machine tends to become short of coolant in a relatively short time because coolant is discharged along with chips. You should therefore watch the coolant oil level gauge periodically to add coolant as required.

Ball screw lubricating

> **WARNING**
>
> Be sure to turn off the main circuit breaker of the machine before cleaning and applying grease to the ball screw. If you do it with the power left on, your hand may be caught in the machine, resulting in a severe injury, when the machine malfunctions or runs out of control.

When applying grease to the ball screw for smooth operation, first position each axis by executing manual handle operation on [MECH ADJ] screen. Then, apply grease to the ball screw as described below after turning off the main circuit breaker of the machine according to <7.1.2 Turning off the power>. For operation procedure of [MECH ADJ] screen, see Chapter 6 for details.

Axis name	Greasing procedure
X1 axis	1. On [MECH ADJ] screen, move the Y1 axis to the position around 70.0. 2. On [MECH ADJ] screen, move the X1 axis to the position around 50.0. 3. Turn off the power. 4. Use a grease gun to force grease into the grease nipple at the front of the gang tool post.
Y1 axis	1. On [MECH ADJ] screen, raise the X1 axis to the full stroke position. 2. On [MECH ADJ] screen, move the Y1 axis to the position around 165.0. 3. Turn off the power. 4. Open the spindle cover. Use a grease gun to force grease into the grease nipple on the Y1-axis ball screw nut through the cast hole.
Z1 axis	1. Turn off the power. 2. Open the spindle cover. 3. Use a grease gun to force grease into the grease nipple at the front of the Z1-axis slide.
X2 axis (Type VII, VIII, IX)	1. On [MECH ADJ] screen, move the Z3 axis to the position around 0. 2. On [MECH ADJ] screen, move the X2 axis to the position around 400.0. 3. Turn off the power. 4. Use a grease gun to force grease into the grease nipple at the opposite slide seen through the hole on the left side of the cover. Supply grease after you remove chips around the grease nipple.
Z2 axis (Type VIII, IX)	1. On [MECH ADJ] screen, move the Z3 axis to the position around 0. 2. On [MECH ADJ] screen, move the X2 axis to the position around 0. 3. On [MECH ADJ] screen, move the Z2 axis to the position around 75.0. 4. Turn off the power. 5. Use a grease gun to force grease into the grease nipple at the opposite slide seen on the left side of the cover. Supply grease after you remove chips around the grease nipple.
X3 axis (Type IX)	1. On [MECH ADJ] screen, move the X2 axis to the position around 0. 2. On [MECH ADJ] screen, move the Z3 axis to the position around 230.0. 3. Turn off the power. 4. Remove the left side cover. Use a grease gun to force grease into the grease nipple at the upper part of the Z3-axis motor.
Z3 axis	1. On [MECH ADJ] screen, move the X2 axis to the position around 0. 2. On [MECH ADJ] screen, move the Z3 axis to the position around 230.0. 3. Turn off the power. 4. Remove the left side cover. Use a grease gun to force grease into the grease nipple at the upper part of the Z3-axis motor.
A1 axis A2 axis A3 axis (OP)	1. Turn off the power. 2. Open the spindle cover (for A1 axis), or back spindle cover (for A2 axis or A3 axis) 3. Apply grease to the ball screw.

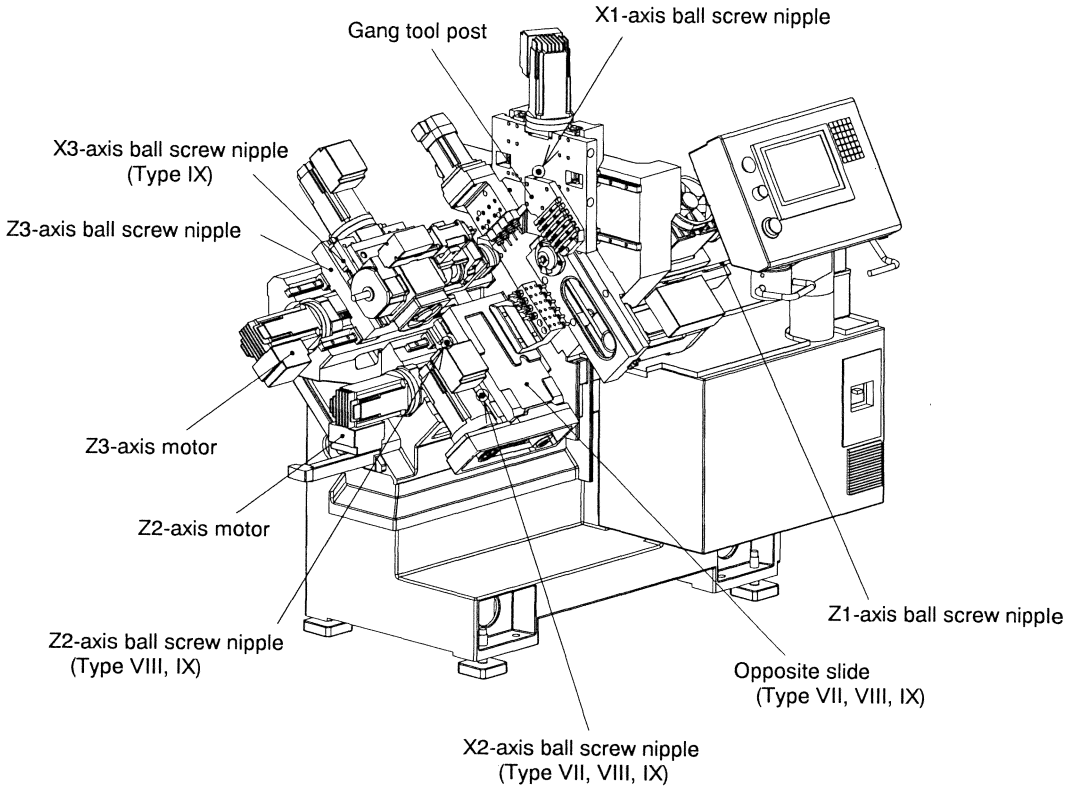
Notes

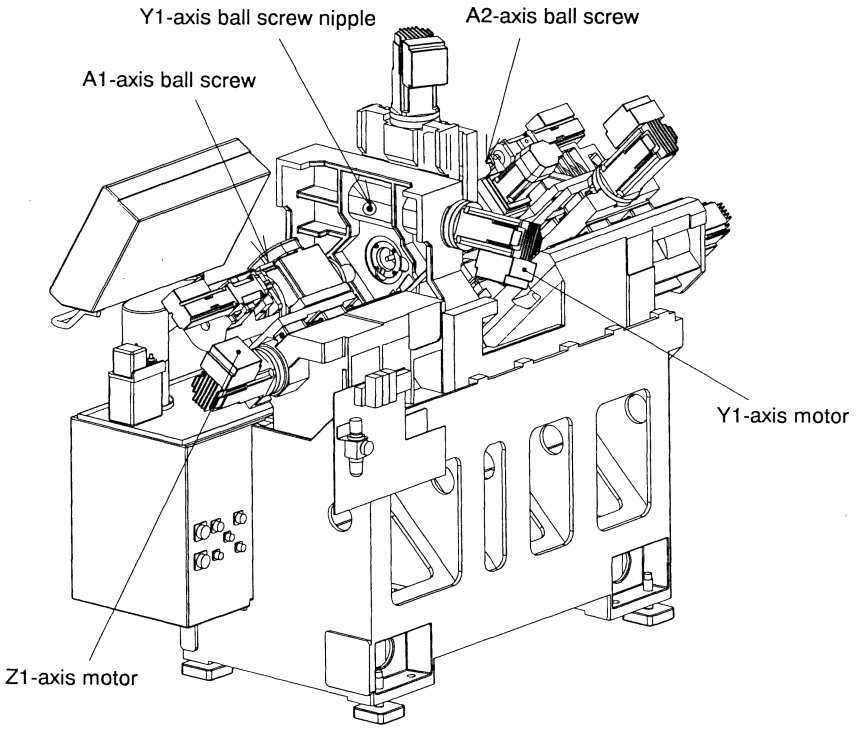
- Use the greasegun with flexible nozzle.

<Recommended grease guns>

Name	Model	Manufacturer	Remarks
Hand grease gun	KH-35	Yamada Corporation	
Micro-hose	SPK-3C	Yamada Corporation	Flexible nozzle for KH-35

- The status message "Ball screw needs lube. Reset on mech-adjustment screen." is displayed every 30 days. It is displayed regardless of presence/absence of lubrication. See <Section 11.12.7 Procedure if a ball screw lubrication alarm is issued >.





Axis Feed Mechanism

Gang tool post spindle spur gear lubricating

> ⚠ WARNING
>
> Make sure to turn off the main circuit breaker of the machine before applying grease to gang tool spindle spur gear.
> If you do it with the power left on, your hand may be caught in the machine, resulting in a severe injury, when the machine malfunctions or runs out of control.

1. Turn off the main circuit breaker according to procedure described in <7.1.2 Turning off the power>.

2. Remove the rotary tool for cross machining and apply grease to the teeth of the spur gear located at the top of tool spindle section.

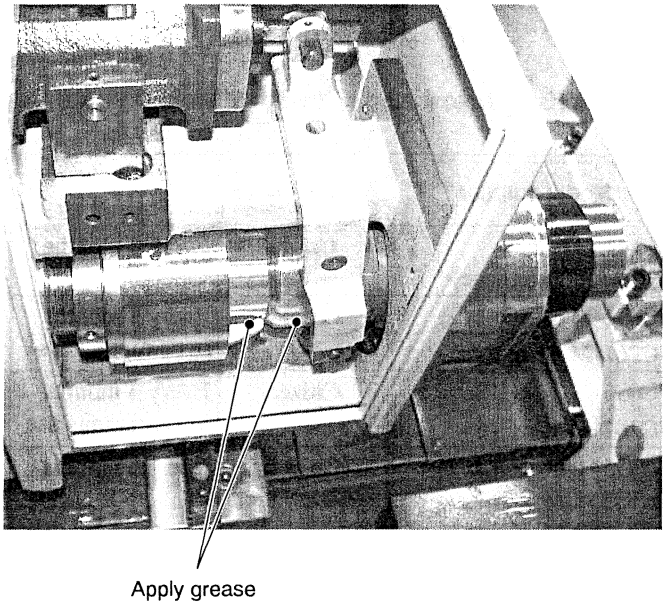
Bobbin sliding portion (back spindle section)

> ⚠ WARNING
>
> Make sure to turn off the main circuit breaker of the machine before applying grease to bobbin or spindle.
> If you do it with the power left on, your hand may be caught in the machine, resulting in a severe injury, when the machine malfunctions or runs out of control.

> ⚠ CAUTION
>
> Avoid "empty chucking" (chucking with no bar material).
> Doing so may cause the machine to malfunction or damage the collet chuck.

1. Turn off the main circuit breaker according to procedure described in <7.1.2 Turning off the power>.

2. Apply grease on the sliding portion of the back spindle bobbin.
   Apply grease on each location with the chuck both opened as well as closed.



Back Spindle Bobbin Sliding Portion

Bar feeder (U10J) rod

1. Pull the rod of the bar feeder toward you and apply lubricating oil to the outer periphery.

10.2.3 List of lubricants (reference)

Vendor	Slide lubricant
Mobil	Vactra Oil No.2
COSMO	COSMO Dinaway 68
Nippon Oil Corporation	Uniway D68
Esso Standard Oil	Phoebis K68
Shell	Tona Oil S68
Idemitsu	Daphne Super Multi 68
MITSUI Oil & Gas	MITSUI Slideway Oil E68
BP	BP Maccurat 68

10.3 Replacement of Worn Parts

Some of the components on this machine have a relatively short operating life. Check the condition of these parts periodically and replace as necessary. The table below lists the part name, model number, manufacturer, inspection frequency, and criteria for replacement.

10.3.1 Replacement parts list

For replacement method of parts listed below, see <Chapter 9 Mounting, Adjustment, and Replacement>.

Part Name	Model number	Manufacturer	Inspection Frequency	Criteria for Replacement
Timing Belt for Driving the Guide Bushing	750-U5GT-15 (Oil-proof, urethane)	Unitta	Every 6 months	Cracked or worn-out
Main Spindle Chuck	FC096-M <C12> FC261-M <C16>	Citizen	Every 3 months in continuous use	Worn-out, damaged chuck inside or abnormal abrasion
Back Spindle Chuck	FC096-M-K <C12> FC261-M-K <C16>	Citizen	Every 3 months in continuous use	Worn-out, damaged chuck inside or abnormal abrasion
Guide Bushing	WFG541-M <C12> WFG551-M <C12> WFG660-M <C16>	Citizen	Every 6 months	Cracked or worn-out
Rotary Guide Bushing Bearing	7906CDB/G8P4 <C12> 7907CDB/GNP4 <C16>	NTN	Every 6 months	Rattle; increasing runout; abnormal noise
X2 axis feed timing belt	624-8YU-15	Unitta	Every 6 months	Cracked or worn-out
Material Feeding Cord	ϕ4 nylon Cord	Citizen	–	Fracture
Fan filter for electric device box	SK3170100 (additional machining required)	Citizen (Rittal)	–	When clogged
Spindle cooling fan	5915PC-20W-B20-S05 (Wiring work required)	NMB	–	Abnormal noise or malfunction * Contact Cincom Service Office for replacement or purchasing the fan.
Back spindle cooling fan	S12D20-TW2G 120□ x 38t (Wiring work required)	Style Electronics		
Regenerative resistor cooling fan	S12D20-TW2G 120□ x 38t (Wiring work required)	Style Electronics		
Fans in electric box (2 piece)	S12D20-TW2G 120□ x 38t (Wiring work required)	Style Electronics		

(Blank page)

Product Code C – C 1 2 1 6 VI VII VIII IX – 7 7 0

Document Code 5 E 1 – 1 0 0 4

11. Troubleshooting

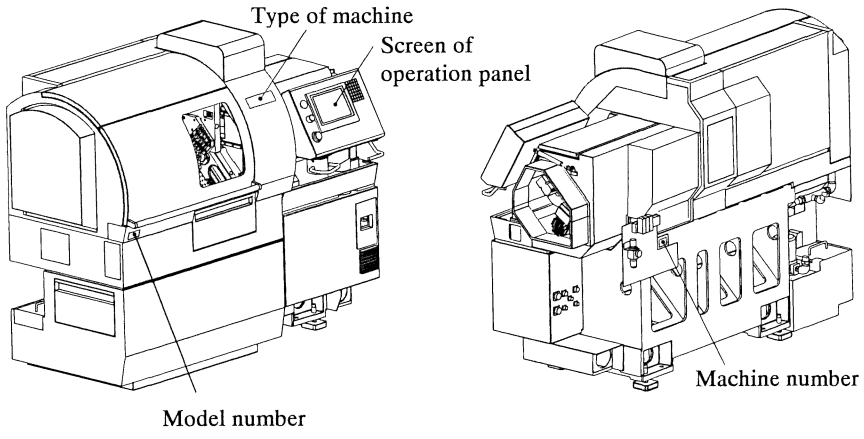
(Blank page)

11.1 Inquiry about Trouble

Request for inquiry

The table below lists the items we may need to solve trouble. Please use the table for correct and quick response to your inquiry. We would like you to complete as may items as possible and inform us of them upon inquiry.

No.	Item	Example	Where to check
1	Type of machine	C16	Aluminum nameplate on the rear of the machine, the operation screen, or the instruction manual
2	Model number	1M7	Red sticker on the front of the machine, on the back of the instruction manual
3	Machine number	X12345	Aluminum nameplate on the rear of the machine
4	NC unit	Cincom SYSTEM M6B	<Section 3.3.2 NC Specifications> of this manual, or the Software List screen
5	Delivery date	December 2000	
6	Machine coating color	Two-toned purple Your specified color	Appearance
7	Presence/absence of optional specifications	e.g., provision of dedicated workpiece unloading device	
8	Page of the instruction manual that contains the information concerning your inquiry		
9	Software version	TYPE FCA635LC-AC11 SERIAL M658546026Z MAIN BND-358W000-A*0 PLCu 001-000 BOOT BND-362W001-A*D MMIs BND-359W000-A*0 MMIu 001-000 API BND-660W300-A*A MACRO 001-000 PARAMET 001-000 OS 001-000	On the screen of the operation panel. See <Section 6.27 Software List Screen> of this manual



11.2 Identifying the Causes of Trouble

Trouble may occur even if the machine is well maintained (e.g., the defined periodical checking). If trouble occurs, the cause of the trouble must be eliminated as soon as possible. This section explains procedures for identifying the causes of relatively simple machine trouble and the recovery procedures.

11.2.1 First thing you have to do if trouble occurs

If trouble occurs, first thing you have to do is to eliminate the cause of the trouble in the earliest stages in order to prevent a serious accident resulting from the trouble. To do so, use your five senses to detect appearance abnormality (e.g., broken part), abnormal heat output, odor, or abnormal sound.
If you sense imminent danger, turn off the main breaker immediately.

11.2.2 Procedures for identifying the causes of trouble

Identifying the cause of trouble that is indicated by the alarm lamp
When the machine detects a failure, the alarm lamp on the operation panel lights or flashes. In this case, see <Section 11.3 Alarms>.

↓

Identifying the cause of trouble by executing the interface diagnosis
When an unprecedented failure occurs or when a failure which the machine fails to recognize as a failure occurs, an alarm is not issued. Consequently, the incident may affect machine operation. In this case, execute the interface diagnosis to check the input/output information in order to identify the cause of trouble. See <Section 11.13 Procedure If the Machine Fails to Start> and the subsequent sections. For the usage of interface diagnosis, see <Section 6.26 I/F Diagnosis Screen>.

↓

Identifying the cause of trouble in the mechanical system
When the input/output signals are normal, the mechanical system probably has trouble. In this case, check the mechanical system for mechanical transmission section trouble (e.g., loose/damaged screw, belt, or key) and mechanical operation section trouble (e.g., galling of slides or damaged bearing)

11.3 Alarms

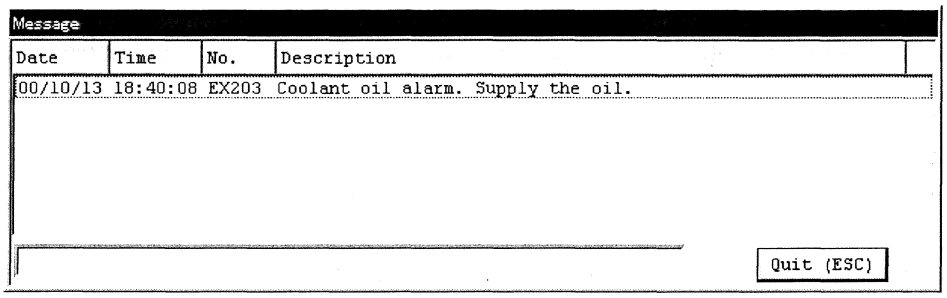
11.3.1 Error and alarm messages

If an error/alarm occurs, the CAUTION/ALM lamp on the operation panel lights or flashes, and the Message screen automatically appears. Read the description of the error/alarm shown in the message. The Message screen may not automatically appear depending on the type of the error/alarm. In this case, press the menu key [Message] to display the Message screen.

Procedure

1. Press ⬭ (Menu UP/Down Selection key). The submenu keys (in the upper row) become effective. This operation is not necessary if the submenu keys are already selected.

2. Press the menu key [Message].
   The Message screen appears.



11.3.2 Alarm lists

List of messages displayed when the alarm lamp flashes

Alarm no.	Message	Description	Procedure
M01	OPERATION ERROR Error number OOOO	An operator's operation error or machine trouble occurred during NC unit operation. Check the detailed information depending on the error number.	See the <Instruction manual> issued by MITSUBISHI ELECTRIC CORPORATION.
M02	ABSOLUTE POSITION RECOVERY Error number OOOO	The detection of the absolute position is not normal because the absolute position data has been erased or the detector malfunctions. Check the detailed information depending on the error number.	See the <Instruction manual> issued by MITSUBISHI ELECTRIC CORPORATION.
M03	COLLISION ALARM	The given command causes interference between two components.	See <Section 11.5 Recovery from Interference Check Alarms>.
M04	COLLISION AREA	Two components are in the interference area.	See <Section 11.5 Recovery from Interference Check Alarms>.

Alarm no.	Message and description	Procedure
OP001	Work counter is full. Auto operation stops end of cycle.	When the machine enters the cycle stop state, press the RST key.
OP002	Lubrication oil is empty. Auto operation stops end of cycle.	When the machine enters the cycle stop state, supply lubrication oil, then press the RST key.
OP003	Chip conveyor is overloaded.	See <Section 11.11 Recovery from External Device Alarms>.
OP004	Coolant oil is empty. Auto operation stops end of cycle.	When the machine enters the cycle stop state, supply coolant, then press the RST key.
OP005	External error. Auto operation stops end of cycle.	
OP006	Bar stock is empty. Auto operation stops end of cycle.	When the machine enters the cycle stop state, supply bars onto the shelf of the bar loader, then press the RST key.
OP007	Door is opened. Close the door.	Close the door. Automatic operation is disabled while the door is open.
OP008	Zero return mode. Other mode is disable.	To select another mode, press the MANUAL key, MDI key, or AUTO key.
OP009	Mechanical adjustment mode. Other mode is disable.	To select another mode, press the MANUAL key, MDI key, or AUTO key.
OP010	Interference check is released. Enable handle feed only.	"11 Interfere Check Off" is enabled on the Set SW screen. Disable this function for automatic operation.
OP011	Main and back spindle synchronous mode.	To cancel spindle synchronization, execute G113 in the program, or press the RST key while all spindles are in the stop state.
OP012	Bar loader is disconnected. Setting switch No.7 is ON.	To enable the bar loader ready signal, set the setting switch "7 Loader Disconnect" to OFF in the Set SW screen.
OP013	Override switch is 0%. Then machine can not be moved.	Set the FEED RATE OVERRIDE dial to a suitable value.
OP014	Tool life is reached. Auto operation stops end of cycle.	When the machine enters the cycle stop state, press the RST key.
OP015	Interference check is disable. Setting switch No.11 is ON.	To enable the interference check function, set the setting switch "11 Interfere Check Off" to OFF in the Set SW screen.
OP037	Door can not be locked. Please shut the door.	Close the door.
OP046	Spindle not restarted. Press SPINDLE START button.	The spindle is stopped. Press the SP.START key to restart the spindle rotating and then press the START key to run the program.

Alarm no.	Message and description	Procedure
OP050	Intermission of axial movement.	Axis move is paused because the START key is released during preparatory operation. Press the START key or close all the doors. (For the EC specification)
OP054	No virtual axis setting. Remove error/alarm, press reset.	Setting of virtual axis failed due to error or alarm occurrence. Remove the error or alarm cause, and press the RST key.
OP056	Loader not ready. Machine stops after this cycle.	The bar loader is not ready for operation. Stop the machine and check the state of the bar loader.
OP057	Dry run valid. Setting switch No. 4 is OFF.	Dry run is enabled. Check the switch setting.
OP058	Machine lock ON. Setting switch No. 13 is OFF.	Machine lock is enabled. Check the switch setting.
OP059	MST lock ON. Setting switch No. 14 is OFF.	Auxiliary function is being locked. Check the switch setting.
OP060	Material change command. Auto operation stops end of cycle.	M55 command is issued when single bar feeder is used.
OP063	Bar loader door is open. Close the door.	Close the bar loader door. (for dedicated magazine bar loader)
OP067	AT Meas-Comp.Device enabled. Setting switch No.21 is ON.	Setting switch No. 21 is turned on. Check the switch setting.
OP068	AT Meas-Comp.Device disabled. Setting switch No.21 is OFF.	Setting switch No. 21 is turned off. Check the switch setting.

Alarm no.	Message and description	Procedure
EX205	External alarm 2. Option device is in alarm status.	
EX208	Chip conveyor overload. Chip is jammed in the conveyor.	See <Section 11.11 Recovery from External Device Alarms>.
EX209	Software over travel alarm. Any of axes is over traveled.	See the NC message to identify the axis. Then, stop the axis feed and correct the program.
EX211	Tool life alarm. Specified tool's cutting time is over.	Replace the tool, then press the RST key.
EX212	Bar loader door open alarm. Close the door.	Close the bar loader door. (for dedicated magazine bar loader)
EX213	Material change command. Not allowed for single bar spec.	M55 command is prohibited for single bar feeder. Review the program.
EX300	Illegal spindle power off. Power turned off incorrectly.	The spindle was not turned off when the door was open. Turn off the main breaker of the machine, then eliminate the cause of the alarm. (For the EC specification)
EX301	Abnormal feedrate. Axis feedrate exceeded limit.	The axis feed rate exceeded the safety speed when the door was open. Turn off the main breaker of the machine, then eliminate the cause of the alarm. (For the EC specification)
EX302	Servo being turned off. Door opened though not permitted.	The door was opened while DOOR SW ○ was disabled, and the servo power switch was turned off. Close the door, or enable the key. (For the EC specification)
EX303	Cycle start disabled. Started although door is open.	An attempt was made to start the program while the door was open. Press the RST key to reset the alarm state, then restart operation from the beginning. (For the EC specification)
EX304	Illegal door lock signal. Door lock is faulty.	The door was locked before operation but unlocked during operation. Shut off the main breaker of the machine and remove the cause. (Fot the EC specification)
EX305	Safety feedrate exceeded. Exceeded in PH operation.	The handle feed rate exceeded the safety speed. Press the RST key to reset the alarm state. (For the EC specification)
EX400	Machine number not registered.	Call the Cincom Service Office.
EX401	Detection of the machine moving.	Call the Cincom Service Office.
EX402	Machine transfer detection alarm	Call the Cincom Service Office.
EX501	Spindle chuck is closed. Open spindle chuck.	Open the main spindle chuck, then perform the start position operation.
EX502	Cut-off tool number on the Machining Data is not set.	"Cut-Off Tool " in the machining data is not set. Review the machining data.
EX503	Cut-off tool number is not selected	"Cut-Off Tool " in the machining data is not set. Review the selected tool number.
EX504	Tool number is not set.	Set the tool number for tool selection.
EX505	Tool number is not selected.	Select the tool number.
EX506	Specified tool number is out of range.	The selected tool number is outside the designated range. Review the selected tool number.
EX507	Tool set function is possible only for gang tools.	Do not execute the "positioning point, Core, Core DWN, and DIA" functions for any tools other than tools on the gang tool post.

Alarm no.	Message and explanation	Procedure
EX512	Main spindle chucking force is too strong.	Adjust the main spindle chucking force.
EX513	Back spindle chucking force is too strong.	Adjust the back spindle chucking force.
EX520	Cancel tool nose R compensation (G40)	To cancel tool nose R compensation, execute G40 before the T code command.
EX521	Cancel constant surface speed control function. (G97)	To cancel constant surface speed control, execute G97 before the T code command.
EX522	You cannot specify this T code.	The specified T code is outside the designated range. Review the program.
EX524	You cannot specify a value less than "0" for "H" argument.	Specify a value greater than 0 for the H argument in the T code command for a gang tool.
EX526	Illegal argument is specified.	An invalid argument is specified in the T code command. Review the program.
EX528	Specified tool post axis is used by other control group.	The specified tool post axis is being used by other control group. Review the program.
EX529	Back spindle is in interference area	The back spindle is in interference area when front drilling tool T2000's is selected. Review the program.
EX530	Use same number for X in T2X00 and T3X00.	In front/back simultaneous drilling, specify the same number for third digit of front drilling tool 2000's and back drilling tool T3000's.
EX532	Opposite tool post is in interference area.	The opposite tool post is in interference area when T3000A1 is specified (with A1 argument). Review the program.
EX533	Unspecified argument in T3000A1	The unspecified argument is found in T3000A1 command. Review the program.
EX551	Argument A in G231 block is illegal.	The A argument is not specified in the G231 command, or a value greater than 0 is specified for the A argument whereas 0 is specified for "Tubing Bar Stock I.D." in the machining data. Review the program.
EX555	Designation of the machining pattern is not allowed.	Invalid machining pattern is specified. Review the program.

Alarm no.	Message and explanation	Procedure
EX562	Back spindle setting in machining data is illegal.	In M32 or M33 command, an illegal value is set in machining data for back spindle to collect the product. Review the machining data.
EX563	Return X2 to machine zero point before M32 or M33.	The X2 axis is in interference area when M32 or M33 command is specified. Return X2 axis to zero point using M147 (retract X2 axis of opposite tool post), then issue M32 or M33 command.
EX565	Move X2 to 260.00 or smaller pos (machine) before M33.	The X2 axis is in interference area when M33 command is issued on the machine with U121B (front/back face machining tool spindle) installed. Move the X2 axis away the machine coordinate 260.0 mm or shorter, then issue M33 command.
EX580	C axis option for the main spindle is not installed.	The optional C axis of the main spindle is not installed, but the M18 command is specified. Review the program.
EX581	C axis option for the back spindle is not installed.	The optional C axis of the back spindle is not installed, but the M48 command is specified. Review the program.
EX582	Indexing option for the back spindle is not installed.	The optional back spindle indexing device is not installed, but the M78 command is specified. Review the program.
EX583	Indexing angle value must be within 0 to 360.	A value outside the range 0 to 360 is specified as the indexing angle in the M18, M48, M28, or M78 command. Review the program.
EX584	Illegal indexing command specified	An invalid value is specified as an indexing angle in the M18, M48, M28, or M78 command. Review the program.
EX585	Argument S is not specified.	The S argument is not specified in the M28 or M78 command. Review the program.
EX590	Back spindle chuck closed.	Open the back spindle chuck, then execute the M10 command.
EX591	Back spindle rotating.	Stop the back spindle, then execute the M10 command.
EX592	Knock-out can not be advanced to correct position.	See <Section 11.10.3 Procedure if the message "EX592 Knock-out cannot be advanced to the correct position>.
EX594	"M113" cannot be specified.	M113 command is not available without the long workpiece device. Review the progarm.

Alarm no.	Message and explanation	Procedure
EX620	Remnant bar did not retract	Move the remnant backward.
EX621	Remnant bar was not removed	Pull out the remnant.
EX622	Bar stock empty	Supply bars onto the shelf.
EX623	Material is not in correct position.	Confirm the bar position, and execute Material Set.
EX640	Incorrect Automatic Measure Argument and Variable.	Incorrect arguments and variables are specified for the automatic measure compensation device. Review the arguments.
EX641	Work Roundness exceeds Setting Value.	The deviation from circularity is greater than the setting value. Eliminate the cause of the error, then press the RST key.
EX642	Compensation count exceeds + NG Value.	The amount of compensation exceeded the setting + NG value. Eliminate the cause of the error, then press the RST key.
EX643	Automatic Measure Sensor Cable not connected Amp.	The sensor cable was not connected with the amplifier when centering was attempted. Connect the sensor cable with the amplifier.
EX644	Sensor measurement position is incorrect. Check X.	Centering failed. Confirm the centering procedure, then retry the centering.
EX645	G411, G412 cannot be specified.	G411 and G412 commands are not available unless in the G610 or G630 mode. Review the program.
EX646	Compensation count exceeds – NG Value.	The amount of compensation exceeds the setting – NG Value. Eliminate the cause of the error, then press the RST key.
EX650	All tools are not mounted.	Automatic tool setting was executed for all tools on the gang tool post, but a tool has not been mounted or has been mounted at a wrong position. Check if the tool is mounted.
EX651	Tool is not mounted.	Automatic tool setting was executed for a selected tool number of the gang tool post, but the tool corresponding to the selected tool number has not been mounted or has been mounted at a wrong position. Check if the tool is mounted.
EX652	Tool not applicable for auto tool setting is selected.	Automatic tool setting was executed for a selected tool number of the gang tool post, but the selected tool number is invalid for automatic tool setting. Review the selected tool number.
EX653	Check the signal.	Make a continuity check on the tool to be automatically set before starting automatic tool setting.
EX654	Wiring for automatic tool setting is not complete.	Check wire connections.
EX655	Bar O.D. (set data) and measured O.D. differ excessively.	The values specified for "Bar Stock O.D." in the machining data and for the measuring diameter differ by more than ±0.5 mm. Enter the outer diameter size of the material for the measuring diameter.

List of NC alarm messages

Alarm no.	Message	Explanation	Procedure
S01	SERVO ALARM: PR Alarm number: OOOO	A servo alarm occurred. Check the detailed information depending on the error number.	See the <Instruction manual> issued by MITSUBISHI ELECTRIC CORPORATION.
S02	INITIAL PARAMETER ERROR Parameter number: OO	When the NC unit was turned on, it sent an invalid parameter to the servo amplifier.	See the <Instruction manual> issued by MITSUBISHI ELECTRIC CORPORATION.
S03	SERVO ERROR: NR Alarm number: OOOO	A servo alarm occurred. Check the detailed information depending on the error number.	See the <Instruction manual> issued by MITSUBISHI ELECTRIC CORPORATION.
S04	SERVO ERROR: AR Alarm number: OOOO	A servo alarm occurred. Check the detailed information depending on the error number.	See the <Instruction manual> issued by MITSUBISHI ELECTRIC CORPORATION.
S51	PARAMETER ERROR Parameter number: OOOO	An invalid servo parameter was set.	See the <Instruction manual> issued by MITSUBISHI ELECTRIC CORPORATION.
S52	SERVO WARNING Warning number: OOOO	A servo alarm occurred. Check the detailed information depending on the error number.	See the <Instruction manual> issued by MITSUBISHI ELECTRIC CORPORATION.
Y02	SYSTEM ALARM Error number: OOOO	The NC unit was turned on, then an error occurred during data transfer between the NC unit and servo amplifier. Check the detailed information depending on the error number.	See the <Instruction manual> issued by MITSUBISHI ELECTRIC CORPORATION.
Y03	AMP UNEQUIPPED	The servo amplifier has not been mounted.	See the <Instruction manual> issued by MITSUBISHI ELECTRIC CORPORATION.
Y05	INITIAL PARAMETER ERROR Error number: OOOO	A parameter necessary for turning on the NC unit is invalid. Check the detailed information depending on the error number.	See the <Instruction manual> issued by MITSUBISHI ELECTRIC CORPORATION.
Y06	mcp_no SETTING ERROR Error number: OOOO	When the NC unit is turned on, the NC servo interface (MCP) does not match the axis parameter. Check the detailed information depending on the error number.	See the <Instruction manual> issued by MITSUBISHI ELECTRIC CORPORATION.
Y07	AMPLIFIER POWER OFF Error number: OOOO	A servo alarm occurred. Check the detailed information depending on the error number.	See the <Instruction manual> issued by MITSUBISHI ELECTRIC CORPORATION.
Y51	PARAMETER ERROR Error number: OOOO	A parameter error occurred causing an alarm to be issued when a control axis moves. Check the detailed information depending on the error number.	See the <Instruction manual> issued by MITSUBISHI ELECTRIC CORPORATION.
ZOO	OOOOO	An NC system alarm occurred. Check the detailed information depending on the alarm number.	See the <Instruction manual> issued by MITSUBISHI ELECTRIC CORPORATION.

11.3.3 Stop code list

The stop codes indicate that the NC unit has stopped because some stop conditions were produced.

Alarm no.	Message	Explanation	Procedure
T01	CAN'T CYCLE ST Error number: OOOO	An attempt to execute automatic operation fails while the NC unit is in the stopped state. Check the detailed information depending on the error number.	See the <Instruction manual> issued by MITSUBISHI ELECTRIC CORPORATION.
T02	FEED HOLD Error number: OOOO	Some conditions were produced during automatic operation, and the automatic operation entered the hold state. Check the detailed information depending on the error number.	See the <Instruction manual> issued by MITSUBISHI ELECTRIC CORPORATION.
T03	BLOCK STOP Error number: OOOO	A block in the program was executed during automatic operation, then the machine entered the block stop state. Check the detailed information depending on the error number.	See the <Instruction manual> issued by MITSUBISHI ELECTRIC CORPORATION.

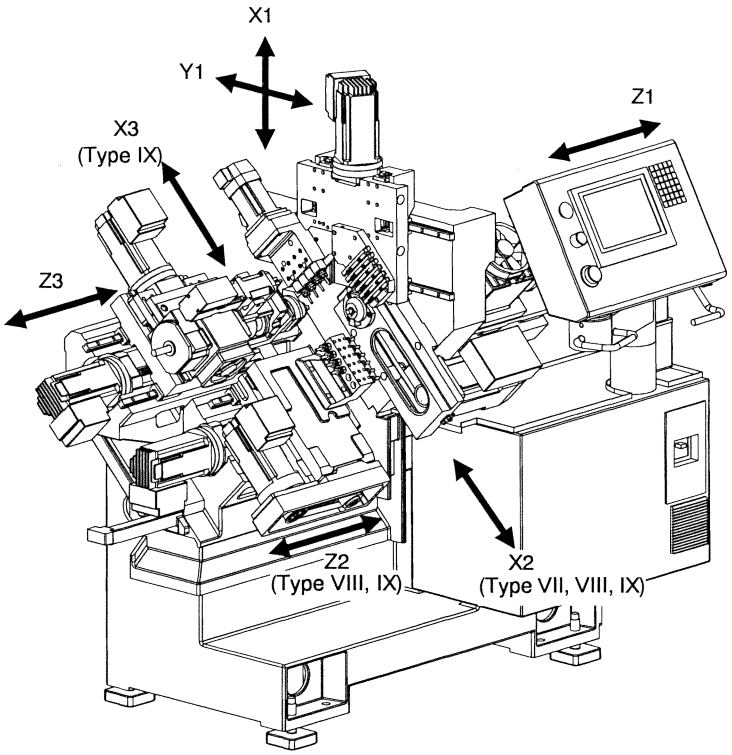
11.4 Troubleshooting

The subsequent sections explain procedures for troubleshooting. The procedures are described on the assumption of the causes of trouble. If the cause of trouble is unknown, call the Cincom Service Office.

11.5 Recovery from Interference Check Alarms

11.5.1 What is an interference check?

This machine makes a check on interference between components.



The interference check function roughly recognizes the shape of the machine by the machining data. The function checks the machine for interference on the basis of the recognized machine shape. By doing so, the function stops the machine before interference occurs.

Notes

- Set the machining data correctly.
- The NC unit recognizes the machine in relatively rough shape when using the interference check function.
  Consequently, interference may occur even when an interference check alarm is not issued, or an interference check alarm may be issued when interference is unlikely to occur.
  When using the interference check function, also make visual inspection to prevent interference.

11.5.2 Procedure if the message "M03 COLLISION ALARM" appears

Procedure

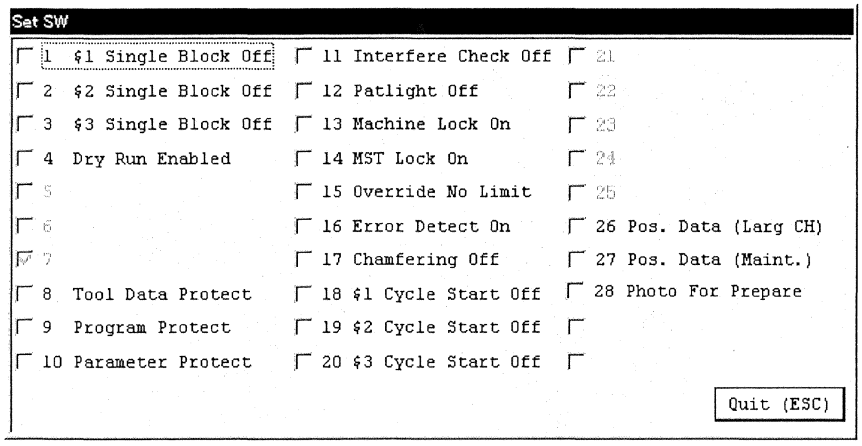
1. Press RST.
   The error display disappears.

11.5.3 Procedure if the message "M04 COLLISION AREA" appears

Procedure

1. Press ⌨ (Menu Up/Down Selection key). The submenu keys (in the upper row) become effective.
   This operation is not necessary if the submenu keys are already effective.

2. Press the menu key [Set SW].
   The Set SW screen appears.



3. Press ↑ and/or ↓ to move the cursor to the setting switch "11 Interface Check Off".

4. Press INPUT.
   "11 Interface Check Off" is checked.

5. Press 

.
The MANUAL key lamp lights.

6. Press the Up/Down Arrow key to select the axis you want to escape from interference, then turn the handle to move the axis.
The ALM lamp turns off.

7. Perform the operations in steps 1 to 4, then uncheck "Interface Check Off".

Notes

- You can switch the interference check function on and off by specifying the M119 command (interference check on) and the M118 command (interference check off).
- Regardless of whether the automatic operation mode or MDI mode is active, the interference check function is enabled when 

is pressed while the M118 (interference check off) command is specified. The RST key switches the interference check function to the enabled state when pressed for any other reasons (e.g., for resetting other alarms).
- A program error or operation error causes an interference check alarm. Create a program in such a manner that an interference check alarm will not occur.

11.6 Recovery from Coolant Pump Overload Alarm

11.6.1 Procedure if the message "EX051 Coolant pump overload alarm" appears

Procedure

> ⚠ WARNING
>
> Be sure to turn off the main breaker of the machine before starting the work.
> Failure to do so could result in electrocution or serious injury.

1. Turn off the main breaker of the machine according to the procedure in <Section 7.1.2 Turning off the power>, then open the door of the front electric device box to check the thermal relay.

   When the thermal relay is tripped:
   - The coolant motor is overloaded.
     Check the amount and condition of the coolant. Also check if the pump is clogged with chips and if an excess load is applied when the pump sucks coolant.
   - The value set for the thermal relay is not suitable.

   The above are two probable causes of the alarm. Eliminate the cause, then proceed to step 2.

   When the thermal relay is not tripped:
   - A probable cause of the alarm is the disconnection of the signal line. Eliminate the cause, then proceed to step 2.

2. Reset the thermal relay according to <Section 11.6.2 Procedure for resetting the thermal relay>, close the door of the front electric device box, then turn on the machine according to the procedure in <Section 7.1.1 Turning on the power>.
   The alarm lamp turns off.

Note

According to the reset bar state, determine whether the thermal relay is tripped. When the thermal relay is tripped, the reset bar springs out and the triangle mark (Δ) becomes visible.

11.6.2 Procedure for resetting the thermal relay

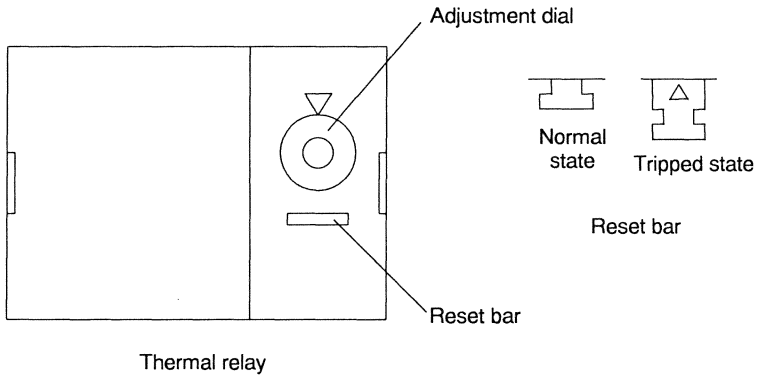
Procedure

> ⚠ DANGER
>
> Be sure to turn off the main breaker of the machine before starting the work.
> Failure to do so will result in death or serious personal injury from electric shock.

1. Turn off the main breaker of the machine according to the procedure in <Section 7.1.2 Turning off the power>, then open the door of the front electric device box to check the thermal relay.

2. When the thermal relay works, the reset bar springs out and the triangle mark (Δ) becomes visible. To reset the tripped state, push the reset bar.

3. Close the door of the front electric device box.

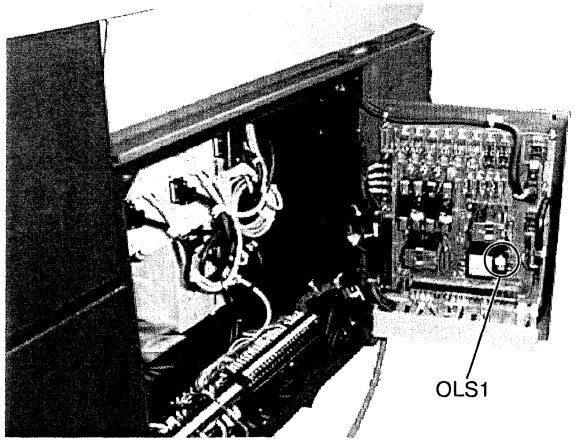


Thermal relay

	Thermal relay symbol	Setting current	Usage
C16	OLS1	1.4 A	For coolant pump motor

Note

- The thermal relay can be set up with the adjustment dial.
- If the same thermal relay works after operation is restarted, eliminate the main cause of the alarm.



11.7 Recovery from Overcurrent Alarm

11.7.1 Procedure if the message "EX001 200VAC over current alarm" appears

Procedure

> ⚠ DANGER
>
> Be sure to turn off the main breaker of the machine before starting the work.
> Failure to do so will result in death or serious personal injury from electric shock.

1. Turn off the main breaker of the machine according to the procedure in <Section 7.1.2 Turning off the power>, then open the door of the front electric device box to check the circuit protector CP1.

   When the circuit protector CP1 is OFF:
   - Check if the 200 V circuit is short-circuited.
   - Check if a device is connected causing a current over the specified current.
     If so, supply power to the device from another power supply source.

   The above are probable causes of the alarm. Eliminate the cause, then proceed to step 3.

   When the circuit protector CP1 is not OFF:
   - Proceed to step 2.

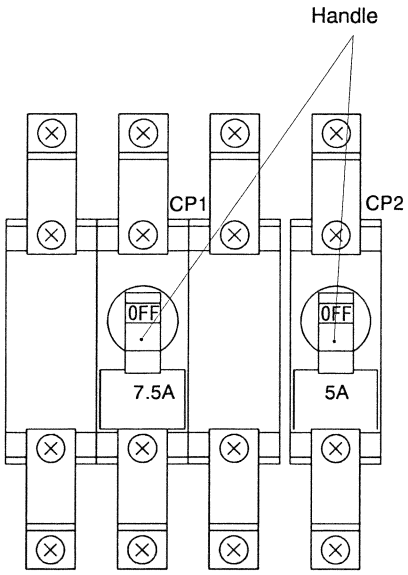
2. Check the circuit protector CP2.

   When the circuit protector CP2 is OFF:
   - Check if the fan motor circuit is short-circuited.
   - Check if the fan motor has stopped (locked).

   The above are probable causes of the alarm. Eliminate the cause, then proceed to step 3.

   When the circuit protector CP2 is not OFF:
   - A probable cause of the alarm is the disconnection of the signal line. Eliminate the cause, then proceed to step 3.

3. Set the circuit protector to ON, then close the door of the front electric device box, then turn on the machine according to the procedure in <Section 7.1.1 Turning on the power>.



Circuit protector

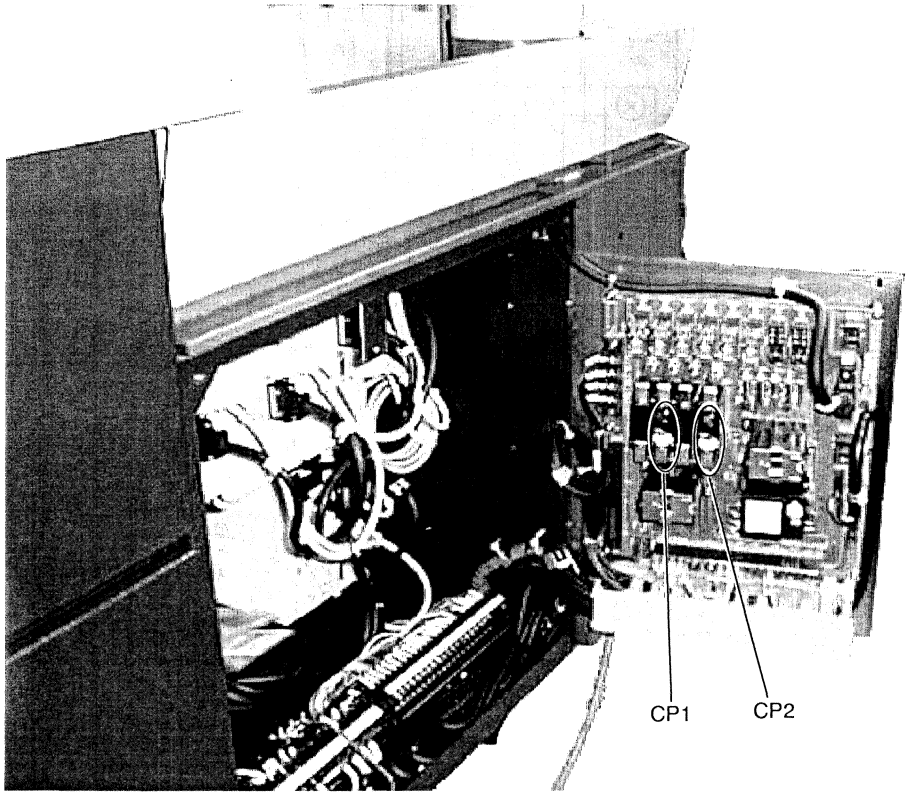
Circuit protectors

Symbol	Capacity	Usage
CP1	7.5 A	For lubrication oil pump, bar loader, coolant pump, chip conveyor, and other 200 VAC circuits
CP2	5 A	For fan motor

Note

If the same circuit protector or fuse works after operation is restarted, eliminate the main cause of the alarm.

Positions of circuit breakers/protectors



11.7.2 Procedure if 24 VDC power supply fails

Procedure

1. Turn off the main circuit breaker according to the procedure described in <Section 7.1.2 Turning off the power> and open the right side cover to check weather the fuse is burned out or not.

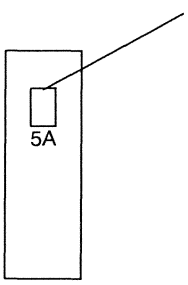
   The fuse is burned out:
   - Check if any of the DC power lines (24 VDC, 5 VDC or 0V line) is short-circuited.
   - Check if the total connected load exceeds the capacity of fuse. If so, connect another power supply to the machine to supply sufficient power.

   Then after removing the cause of the fault, proceed to step 2.

   The fuse is not burned out:
   - Any of the DC power lines (24 VDC, 5 VDC or 0V line) may possibly be disconnected. After removing the cause of the fault, proceed to step 3.

2. Remove the burned fuse and replace it with a new one properly. Then proceed to step 3.

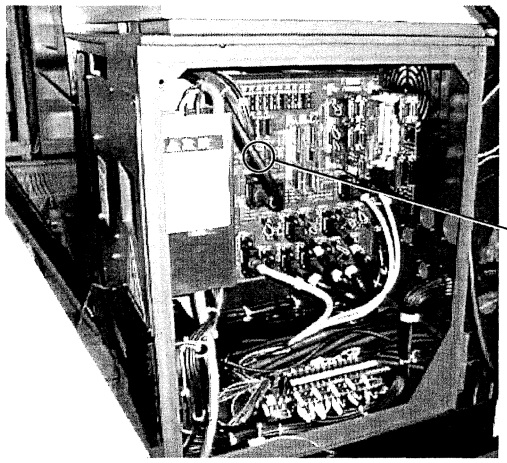
3. Close the right side cover and turn on the power of the machine according to the procedure described in <Section 7.1.1 Turning on the power>.



Window
If the fuse is burned out, you can view the spring (metal piece) through the window.

Note

If the same status recurs after restarted operation, the major cause must be removed.



11.8 Recovery from Main Spindle, Back Spindle and Tool Spindle Alarms

EX003 Main spindle motor alarm
EX013 Back spindle motor alarm
EX006 Gang Tool Post Spindle Motor Alarm
EX007 F/B TSP motor alarm. Alarm of tool spindle drive unit
EX008 Back tool spindle motor alarm

When any of the above alarms occurs, the machine stops and an NC alarm is issued. For the information and cause of the alarm, see the <Instruction manual> issued by MITSUBISHI ELECTRIC CORPORATION.

11.9 Recovery from Guide Bushing Spindle Motor Alarm

11.9.1 Procedure if the message "EX004 G/B spindle motor alarm" appears

Procedure

1. Check the main spindle chuck.

   When the chucking force is weak:
   - Adjust the chucking force.

   When the chucking force is suitable:
   - An NC alarm is issued with the above alarm. For the information and cause of the alarm, see the <Instruction manual> issued by MITSUBISHI ELECTRIC CORPORATION>.

11.10 Recovery from Spindle Alarms

11.10.1 Procedure if the message "EX119 Main spindle speed fluctuation alarm" appears

Procedure

1. Turn off the main breaker of the machine according to the procedure in <Section 7.1.2 Turning off the power>, then rotate the main spindle by hand.

   When the main spindle does not rotate or it is abnormally hard to rotate:
   - The main spindle probably has mechanical trouble. Call the Cincom Service Office.

   When the main spindle rotates normally:
   - Proceed to step 2.

2. Check if a chipped, broken, or worn tool (e.g., drill) causes a greater load than normal.

   When the tool (e.g., drill) is chipped, broken, or worn:
   - Replace or sharpen the defective tool.

   When the tool (e.g., drill) is normal:
   - Proceed to step 3.

3. Turn on the machine according to the procedure in <Section 7.1.1 Turning on the power>, verify that the machine is not in operation, and rotate the main spindle by hand. Then, check if the MDI screen displays 0 for the spindle speed S1.

> ⚠ WARNING
>
> Be sure to confirm that the machine has stopped before starting the work.
> The work during machine operation could result in serious personal injury.

When 0 is displayed:

- The encoder cable is disconnected or the connector is loose.
- Encoder trouble
- NC unit trouble

The above are three probable causes of the alarm.
When the encoder cable is firmly connected and cable disconnection is not detected by visual inspection, call the Cincom Service Office.

When the value changes:

- Proceed to step 4.

4. Pull out the material, then execute the main spindle speed fluctuation detection OFF command (M97) in the MDI mode. Specify M3 S1=1000 to rotate the main spindle at the speed of 1,000 min^{-1}, and check if the MDI screen displays a value within 1,000 ± 100 min^{-1} for the spindle speed S1.

When the main spindle speed is within the range of 1,000 ± 100 min^{-1}:

- The standard value is not set for the speed change detection level. Change the setting value on the SP-NC Parameter screen.
  Tolerances of speed change rate: ± 10% of the specified speed (alarm issued when the speed change is outside the range) Spindle parameter #3024<1> Main spindle

When the main spindle speed is outside the range of 1,000 ± 100 min^{-1} or a main spindle motor alarm is issued:

- Encoder trouble
- Speed adjustment failure
- Main spindle motor trouble
- The encoder cable is disconnected or the connector is loose.
- Main spindle servo amplifier trouble
- NC unit trouble

The above are six probable causes of the alarm.
Call the Cincom Service Office in the following cases:
The encoder cable is firmly connected and you do not find any cable disconnection during visual inspection.
The alarm lamp does not turn off after you take the recovery procedure for "EX003 Main spindle motor alarm" in <Section 11.8 Recovery from Main Spindle, Back Spindle and Tool Spindle Alarms>.

11.10.2 Procedure if the message "EX127 Back spindle speed fluctuation alarm" appears

Procedure

1. Turn off the main breaker of the machine according to the procedure in <Section 7.1.2 Turning off the power>, then rotate the back spindle by hand.

   When the back spindle does not rotate or it is abnormally hard to rotate:
   - The back spindle probably has mechanical trouble. Call the Cincom Service Office.

   When the back spindle rotates normally:
   - Proceed to step 2.

2. Check if a chipped, broken, or worn tool (e.g., drill) causes a greater load than normal.

   When the tool (e.g., drill) is chipped, broken, or worn:
   - Replace or sharpen the defective tool.

   When the tool (e.g., drill) is normal:
   - Proceed to step 3.

3. Turn on the machine according to the procedure in <Section 7.1.1 Turning on the power>, and verify that the machine is not in operation, and rotate the back spindle by hand. Then, check if the MDI screen displays 0 for the spindle speed S2.

> ⚠ WARNING
>
> Be sure to confirm that the machine has stopped before starting the work.
> The work during machine operation could result in serious personal injury.

   When 0 is displayed:
   - The encoder cable is disconnected or the connector is loose.
   - Encoder trouble
   - NC unit trouble

   The above are three probable causes of the alarm.
   When the encoder cable is firmly connected and cable disconnection is not detected by visual inspection, call the Cincom Service Office.

   When the value changes:
   - Proceed to step 4.

4. Pull out the workpiece, then execute the back spindle speed fluctuation detection OFF command (M95) in the MDI mode. Specify M23 S2=1000 to rotate the back spindle at the speed of 1,000 min^{-1}, and check if the MDI screen displays a value within 1,000 ± 100 min^{-1} for the back spindle speed S2.

   When the back spindle speed is within the range of 1,000 ± 100 min^{-1}:
   - The standard value is not set for the speed change detection level. Change the setting value on the SP-NC Parameter screen.
     Tolerances of speed change rate: ± 10% of the specified speed (alarm issued when the speed change is outside the range) Spindle parameter #3024<2> Back spindle

   When the back spindle speed is outside the range of 1,000 ± 100 min^{-1} or a back spindle motor alarm is issued:
   - Encoder trouble
   - Speed adjustment failure
   - Back spindle motor trouble
   - The encoder cable is disconnected or the connector is loose.
   - Back spindle servo amplifier trouble
   - NC unit trouble

   The above are six probable causes of the alarm.
   Call the Cincom Service Office in the following cases:
   The encoder cable is firmly connected and you do not find any cable disconnection during visual inspection.
   The alarm lamp does not turn off after you take the recovery procedure for "EX013 Back spindle motor alarm" in <Section 11.8 Recovery from Main Spindle, Back Spindle and Tool Spindle Alarms>.

11.10.3 Procedure if the message "EX592 Knock-out cannot be advanced to correct position" appears (With Motor Back Knock-out Device<Option>)

Procedure

1. Check if a workpiece is caught between the knock-out jig/rod and the back spindle.

   When a workpiece is caught:
   - Remove the workpiece.

   When no workpiece is caught:
   - The knock-out jig has a greater diameter than the chuck does.
   - The knock-out rod has a bend.

   The above are probable causes (the knock-out rod is caught before reaching the target position) of the alarm. If you do not find any trouble, call the Cincom Service Office.

11.10.4 Procedure if the message "EX115 Knock-out Overload" appears (with Air Back Knock-out Device <Standard>)

Procedure

1. Check if a workpiece is caught between the knock-out jig/rod and the back spindle.

   When a workpiece is caught:
   - Remove the workpiece.

   When no workpiece is caught:
   - The knock-out jig has a greater diameter than the chuck does.
   - The knock-out rod has a bend.

   It is assumed that the knock-out rod is clinched on halfway.
   If no abnormality is found in knock-out rod, the sensor to detect the knock-out error is assumed to be faulty.
   If you do not find any trouble, call the Cincom Service Office.

11.11 Recovery from External Device Alarms

11.11.1 Procedure if the message "EX133 Bar loader is not ready" appears

Procedure

With the dedicated magazine bar loader:

1. Check if the 

key lamp is on or off.

   When the lamp is off:
   - Press the POWER key on the operation panel.

   When the lamp is on:
   - The NC unit probably has trouble. Call the Cincom Service Office.

With another bar loader:

1. Check if the bar loader is on/off.

   When the power is OFF:
   - Turn on the power.

   When the power is ON:
   - Cable disconnection or wiring error
   - NC unit trouble
   - Bar loader trouble

   The above are three probable causes of the alarm. Check the bar loader according to <Maintenance Manual> of the bar loader, and check if the cable is connected normally. If you do not find any trouble, call the Cincom Service Office.

11.11.2 Procedure if the message "EX101 Bar loader alarm" appears

Procedure

1. Check the bar loader status.

   When an alarm has occurred:
   - Reset the alarm.

   When no alarm has occurred:
   - Off-powered
   - Wiring error or cable disconnection
   - NC unit trouble

   Any of the above three may have occurred. Check power supply and cable connections. If you do not find any trouble, call the Cincom Service Office.

Note

When an automatic bar loader (but not a dedicated magazine bar loader) is mounted, ask the bar loader manufacturer for alarm information.

11.11.3 Procedure if the message "EX201 Bar stock empty" appears

Procedure

1. Check if the bar loader has bars on the shelf.

   When there are no bars:
   - Supply bars onto the shelf.

   When there are bars:
   - Bar detection sensor trouble
   - Wiring error or cable disconnection

   The above are two probable causes of the alarm. Check if the cable is connected normally and if the sensor is normal. If you do not find any trouble, call the Cincom Service Office

Note

When an automatic bar loader (but not a dedicated magazine bar loader) is mounted, ask the bar loader manufacturer for alarm information.

11.11.4 Procedure if the message "EX208 Chip conveyor overload" appears

Procedure

1. Check if the chip conveyor is overloaded clogged with chips.

   When the chip conveyor is overloaded:
   - Eliminate the cause of the overload.

   When the chip conveyor is normal:
   - Wiring error or cable disconnection
   - Off-powered

   Either of the above two may have occurred. Refer to Chip Conveyor Operation Manual to inspect the chip conveyor. If you do not find any trouble, call the Cincom Service Office.

Note

Clean the chip conveyor periodically to prevent overload due to chips. Pull out the chip conveyor from the machine, and remove fine chips from the chip conveyor.

11.12 Recovery from Other Alarms

11.12.1 Procedure if the message "EX117 Cycle time alarm" appears

Procedure

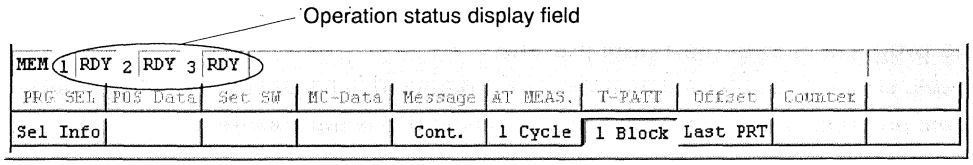
If a cycle time alarm occurs each time the program is executed:

1. Check if the program cycle time exceeds the setting time (standard setting time 30 minutes).

   When it exceeds the setting time:
   - Increase the setting time.
     Disable the cycle time check function and execute the program for 1 cycle, then measure the actual cycle time. To disable the cycle time check function, display the Bit Select screen, and set bit 1 (second bit from the right) of bit parameter number 6401 to "1". See <Section 6.24 Bit Select Screen>.
     Note: If the cycle time check function is disabled, a machine alarm is not issued when unprecedented trouble occurs. Be sure to enable the cycle time check function except when you measure cycle time to determine the setting time.
     Use PLC constant #3 (unit: 0.1 second) to set cycle time check function.
     The maximum setting time is 54 minutes, and the setting value is 32400.
     For PLC constants, see <Section 6.23 PLC Data Screen.>

   When it does not exceed the setting time:
   - Proceed to step 2.

2. Execute the program for one block, then confirm the operation status display at the point at which there is no response to the pressing of the START key. The START key lamp lights and remains on.
   See <Section 4.6 Fields on the Liquid Crystal Display (LCD) and their Functions> to check the list of the operation status display field.



When SYN (synchronization queuing state) is displayed for all axis control groups 1, 2, and 3:

- Queuing commands (!) of different queuing ID numbers are executed for axis control groups 1, 2, and 3, and they have stopped.
  Review the queuing ID numbers in the program.

When no symbol (during automatic operation) is displayed for one or two of axis control groups 1, 2, and 3 and SYN is displayed for the other(s), or when no symbol is displayed for all axis control groups 1, 2, and 3:

- A command with no completion signal issued exists in a block being executed in the axis control group with which no operation status symbol is displayed. If this block contains a word (command) that can be executed alone, place the word (command) into a separate block. Then, execute 1-block operation again, and proceed to step 3.

3. Confirm the command that caused the block to stop.

When the block has stopped with G1, G2, or G3 for a cutting block:

- A probable cause of the alarm is that the main/back spindle is not rotating but that the feed per rotation command (G99) is specified.
  If the rotation command is not specified, specify the rotation command before the cutting block. If the rotation command is specified, eliminate the cause of a rotation failure according to <Section 11.14.1 Procedure If the Main Spindle Fails to Rotate in Automatic Operation>, then reexecute the command.

When the block has stopped with an M code:

- A probable cause of the alarm is that an unusable M code is specified or that the completion conditions are not satisfied. Check if the specified M code is usable. Confirm the current position display and machine position, and check if the specified operation has been executed (e.g., the M6 command was executed, and the chuck closed normally, the chuck failed to close normally, or the chuck remains open). Then, take the suitable procedure.

When the block has stopped with another command:

- Call the Cincom Service Office.

When the program is executed until the end:

- A probable cause of the alarm is that the M02 command is not specified in the program.
  Check the program, and specify the M02 command if it is not specified.

When an alarm is issued sometimes when the program is executed:

- A probable cause of the alarm is a contact failure of the door switch. The door open signal is sometimes issued even if the door is closed. Adjust or replace the door switch.
- Another probable cause of the alarm is that the value set for the override is too low. Change the override setting value.

Notes

- The M codes for resetting the cycle time count are M1, M56, M57, and M2.
- Check cycle time in only the continuous automatic operation mode.

11.12.2 Procedure if the message "EX114 Coolant discharge alarm" appears

Procedure

1. Press 

to reset the alarm, then make adjustment according to <Section 11.12.3 Adjusting the coolant flow rate detector (U52R)>.

   When an alarm is issued as soon as the Coolant switch is pressed for flow rate adjustment in step 4 of the procedure in <Section 11.12.3 Adjusting the coolant flow rate detector (U52R)>:
   - A probable cause of the alarm is that the coolant is running short. Check the amount of coolant, and supply coolant if there is not enough coolant.
   - Another probable cause of the alarm is that filters in the coolant tank are clogged and the coolant level is below the suction opening of the pump. Clean the inside of the tank.

2. The flow rate sensor probably malfunctions. Turn the "potentiometer for setting value adjustment" counterclockwise until it stops.

   When the "Flow rate alarm LED" does not turn on:
   - A probable cause of the alarm is that the flow rate sensor malfunctions. Replace the sensor.

   When the "Flow rate alarm LED" turns on:
   - A probable cause of the alarm is an adjustment failure. Readjust the flow rate from the beginning.

3. Adjustment can be made according to the procedure.
   - A probable cause of the alarm is that filters in the coolant tank are clogged and it takes time until the coolant level reaches the suction opening of the pump. Clean the inside of the tank.
   - Another probable cause of the alarm is that the flow rate adjustment valve is tightened too firmly or that the "potentiometer for setting value adjustment" is not adjusted correctly. Correct the adjustment, and continue operation in the condition. If the same alarm recurs, call the Cincom Service Office.

Note

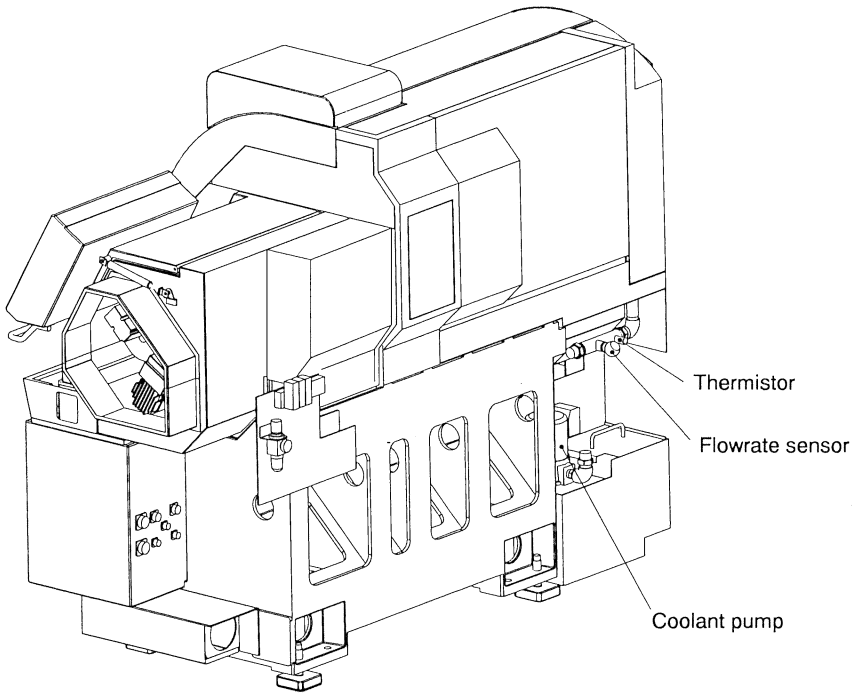
The coolant flow rate detector is optional.

11.12.3 Adjusting the coolant flow rate detector (U52R)

If the coolant flow rate drops for some reason, coolant fails to reach the cutting point, which may cause a fire. The coolant flow rate detector detects a drop of the coolant flow rate and stops machine operation automatically.

Notes

- The coolant flow rate varies with temperature change. To distinguish the flow rate change in the stationary condition from the abnormal state, set a flow rate for an alarm (potentiometer for setting value adjustment) to 2/3 or less of the normal flow rate.
  Also, determine the normal flow rate in the state in which the valve is fully open.
- The coolant flow rate detector does not work when the coolant temperature is lower than 15°C when the machine starts operating. When operating the machine early in the winter morning, you have to monitor the coolant flow rate because the coolant flow rate detector does not work until the coolant temperature reaches 15°C. Once the coolant temperature reaches 15°C, the detector continues working. The detector is not disabled even when the temperature lowers due to environmental change later.



Names and functions of the U52R components

The coolant flow rate detector consists of a flow rate sensor and a coolant temperature setter.

- Flow rate sensor

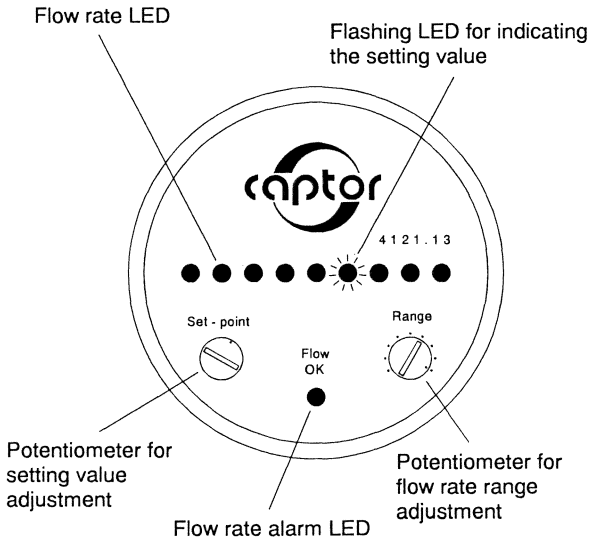
  - Flow rate LEDs
    The current flow rate is indicated relatively to the setting value with nine LEDs ON/OFF. When the sensor is on, the leftmost LED is always on even if the flow rate is "0".

  - Flashing LED for indicating the setting value
    The flashing LED indicates the setting value as a flow rate for an alarm.

  - Flow rate alarm LED
    This LED is on while the flow rate is normal. The LED turns off when the coolant flow rate becomes lower than the flow rate set for an alarm.

  - Potentiometer for setting value adjustment
    This potentiometer is used to set a flow rate for an alarm. Turning the potentiometer clockwise increases the setting of the flow rate for alarm.

  - Potentiometer for flow rate range adjustment
    This potentiometer is used to adjust the sensor sensitivity. Turning the potentiometer counterclockwise improves the sensor sensitivity.

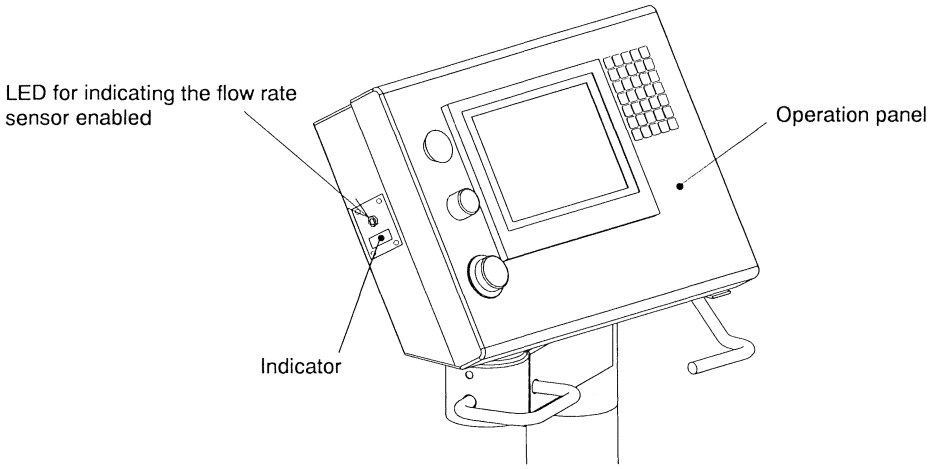


- Coolant temperature setter
  (consisting of a setter, an indicator, and a temperature sensing element of thermistor)

  - LED for indicating the flow rate sensor enabled
    This LED indicates the flow rate sensor function is enabled. The function is able to detect a flow rate alarm in the temperature zone (standard setting temperature of this machine: 15°C or higher) in which the function is enabled. The function is unable to detect a flow rate alarm in the temperature zone (LED off: lower than 15°C) in which the function is disabled.

  - Indicator
    The coolant temperature is always measured by the thermistor. The indicator indicates the measured temperature.

  - Setter (in the operation panel)
    The setter sets the temperature zone in which the flow rate sensor is enabled. (Standard setting temperature of this machine: 15°C)



Procedure for adjusting the coolant flow rate detector

Procedure

1. Turn off the main breaker of the machine according to the procedure in <Section 7.1.2 Turning off the power>.

2. Confirm that the dial of the coolant temperature setter indicates 15°C.
   Open the operation panel, and check the dial scale of the setter.

3. Turn on the power while coolant is stopped.
   All the "Flow rate LEDs" of the flow rate sensor turn on when the power is turned on. The eight LEDs turns off after several seconds while only the leftmost LED remains on. Then, start adjustment.

4. Discharge coolant until the coolant flow rate reaches the normal level (normal flow rate with the valve fully open).

5. Confirm that the indicator of the coolant temperature setter indicates 15°C or higher.

6. Slowly turn the "Potentiometer for flow rate range adjustment" until eight or nine "Flow rate LEDs" from the left turn on.

7. Turn the valve while checking the coolant flow rate by the amount of coolant applied to the cutting point, and stop tightening the valve at the flow rate you want to set for an alarm (2/3 or lower of the normal flow rate).

8. Turn the "Potentiometer for setting value adjustment" counterclockwise until it stops.

9. Slowly turn the "Potentiometer for setting value adjustment" clockwise until the "Flow rate alarm LED" turns off.

10. Set the coolant flow rate back to normal. At this time, confirm that the "Flow rate alarm LED" turns on. If the difference between the normal flow rate and the flow rate you want to set for an alarm is small, the "Flow rate alarm LED" does not turn on when the coolant flow rate is set back to normal. Consequently, an alarm may not be reset. In this case, turn off the power, and then turn it on. Set the flow rate for an alarm lower than the previously set flow rate, then take steps 8 to 10 again.

11. Slowly lower the coolant flow rate and confirm that an alarm is issued when it reaches the flow rate set for an alarm.

11.12.4 Procedure if the message "EX116 Tool bit breakage alarm" appears

Procedure

1. Check if a workpiece remains without being cut off.

   When the workpiece remains:
   - Replace the cut-off tool.

   When no workpiece remains:
   - Proceed to step 2.

2. Check if chips are caught in areas around the guide bushing.

   When chips are caught:
   - Remove the chips.

   When chips are not caught:
   - Sensor adjustment failure
   - Sensor trouble
   - Disconnection of the sensor cable

   The above are probable causes of the alarm. Check if the cable and sensor are normal. If you do not find any trouble, call the Cincom Service Office.

Note

The cut-off tool breakage detection function is optional.

11.12.5 Procedure if the message "EX202 Lubrication oil empty alarm" appears

Procedure

1. Check the amount of lubrication oil.

   When lubrication oil is running short:
   - Supply lubrication oil.

   When lubrication oil is not running short:
   - Sensor trouble
   - Cable disconnection

   The above are two probable causes of the alarm. Check if the cable and sensor are normal. If you do not find any trouble, call the Cincom Service Office.

11.12.6 Procedure if the message "EX203 Coolant oil alarm" appears

Procedure

1. Check the amount of coolant.

   When coolant is running short:
   - Supply coolant.

   When coolant is not running short:
   - Proceed to step 2.

2. Check the amount of the chips.

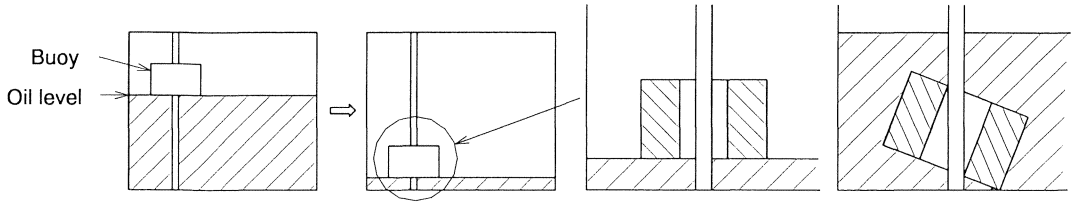
   When the chips are accumulated in the cutting area in the machine and coolant cannot be collected in the tank:
   - Remove chips.

   When the chips are not accumulated in the cutting area in the machine:
   - Sensor trouble
   - Cable disconnection

   The above are two probable causes of the alarm. Check if the cable and sensor are normal. If you do not find any trouble, call the Cincom Service Office.

Note

The sensor uses a buoy to detect the oil level. When the message "EX203 Coolant oil alarm" appears, the buoy lowers as the oil level lowers, and an alarm is issued a little before it reaches the lowest level.
When oil in the tank is used up and the buoy reaches the lowest level, the buoy becomes askew and is unable to float when oil is supplied.
In this case, an alarm is issued even if oil is supplied. Dismount the sensor from the tank, make the buoy level, and mount the sensor.



When there is enough coolant: When coolant is running short: When the sensor is hooked:

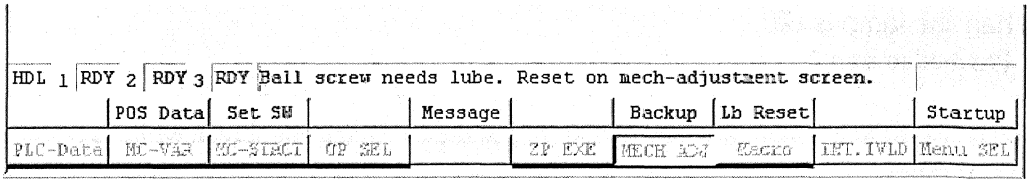
11.12.7 Procedure if a ball screw lubrication alarm is issued

The status message "Ball screw needs lube. Reset on mech-adjustment screen." is displayed every 30 days.

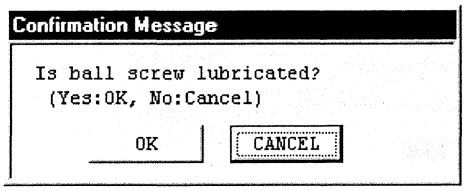
Procedure

1. Apply the specified grease to the ball screws. For details, see <chapter 10 Scheduled Maintenance >.

2. Press the menu key [Lb Reset] on the mechanical adjustment screen. For details, see <section 6.20 Mechanism Adjustment Screen > in the Operation Manual.



3. The confirmation message screen appears.



4. Select [OK] and press 

.

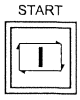
   The status message "Ball screw needs lube. Reset on mech-adjustment screen." disappears from the status message area.

Note

The status message is redisplayed 30 days after you clear it by pressing the menu key [Lb Reset] and selecting [OK]. It is displayed regardless of presence/absence of lubrication.

11.13 Procedure If the Machine Fails to Start

Procedure

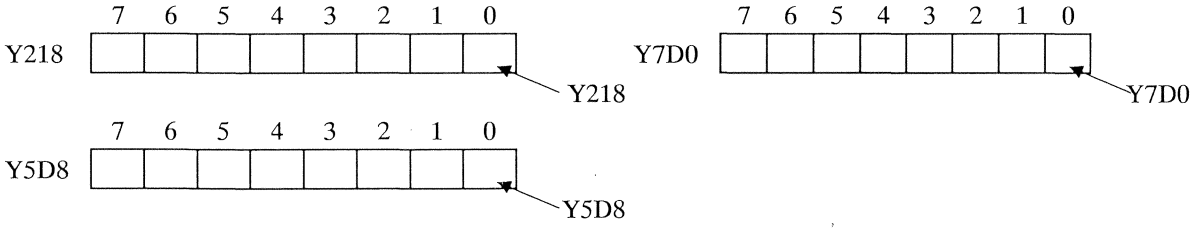
1. Check if the 

key lamp is on.

   When the lamp is on:
   - A probable cause of the failure is that the override is 0%.

   When the lamp is off:
   - Proceed to step 2.

2. Check Y218 ($1), Y5D8 ($2), and Y7D0 ($3) of the I/F Diagnosis screen while pressing the START key several times.



   When any of Y218, Y5D8, or Y7D0 changes to 1:
   - A probable cause of the failure is NC unit trouble. Call the Cincom Service Office.

   When Y218, Y5D8, and Y7D0 remain to be 0 unchanged:
   - Proceed to step 3.

3. Check if the MDI mode, automatic or on-machine program check operation mode is selected.

   When the mode is not selected:
   - Select a mode in which the machine can be executed automatic operation.

   When the mode is selected:
   - Proceed to step 4.

4. Check if "18 $1 Cycle Start Off", "19 $2 Cycle Start Off", and "20 $3 Cycle Start Off" are not checked in the Set SW screen.

   When the switches are checked:
   - Remove their check.

   When the switches are not checked:
   - Proceed to step 5.

5. Check if an alarm has occurred.

   When an alarm has occurred:
   - Reset the alarm.

   When no alarm has occurred:
   - A probable cause of the failure is START key trouble or cable disconnection. Call the Cincom Service Office.

Notes

- For the I/F Diagnosis screen, see <Section 6.26 I/F Diagnosis Screen>.
- For the Set SW screen, see <Section 6.10 Set SW Screen>.

11.14 Procedures If Automatic Operation Fails

11.14.1 Procedure If the Main Spindle Fails to Rotate in Automatic Operation

Procedure

1. Specify the main spindle rotation command (M03 S1=1000) on the MDI screen.

When the main spindle rotates:

- A probable cause of the failure is that a command for setting turning off the main spindle rotation conditions (e.g., C axis command) is specified before or after the main spindle rotation command in the program. Another probable cause is that the main spindle rotation command is not specified correctly (e.g., S1= OOOO not specified).

When the main spindle does not rotate:

- NC unit trouble
- Motor trouble
- Drive unit trouble
- Cable disconnection

The above are four probable causes of the failure. Check if the cable and sensor are normal. If you do not find any trouble, call the Cincom Service Office.

11.14.2 Procedure If the Back Spindle Fails to Rotate in Automatic Operation

Procedure

1. Specify the back spindle rotation command (M23 S2=1000) on the MDI screen.

   When the back spindle rotates:
   - A probable cause of the failure is that a command for turning off the back spindle rotation conditions (e.g., back spindle C axis command) is specified before or after the back spindle rotation command in the program. Another probable cause is that the back spindle rotation command is not specified correctly (e.g., S2= OOOO not specified).

   When the back spindle does not rotate as specified by the back spindle rotation command:
   - The back spindle chuck is open.
   - <Basket> is selected for back spindle on the [MC Data] screen.

   When the back spindle does not rotate:
   - NC unit trouble
   - Motor trouble
   - Drive unit trouble
   - Cable disconnection

   The above are four probable causes of the failure. Check if the cable and sensor are normal. If you do not find any trouble, call the Cincom Service Office.

11.14.3 Procedure If the Tool Spindle Fails to Rotate in Automatic Operation

Procedure

1. Specify the tool spindle rotation command (with gang tool post: M58 S3=1000, with front/back face machining tool: M80 S4=1000) on the MDI screen.

   When the tool spindle rotates:
   - A probable cause of the failure is that the tool spindle rotation command is not specified correctly (e.g., S3= OOOO not specified) in the program.

   When the tool spindle does not rotate:
   - NC unit trouble
   - Motor trouble
   - Drive unit trouble
   - Cable disconnection

   The above are four probable causes of the failure. Check if the cable and sensor are normal. If you do not find any trouble, call the Cincom Service Office.

11.15 Procedures for Alarms Related to the Guide Bushing

11.15.1 Fretting problem

Procedure

1. Reconfirm the cutting conditions.

   Check if the current spindle speed, feed rate, coolant, tool material, and grinding are suitable for the material to be machined.
   Check if the tool is free from any damage (e.g., built-up edge, or being worn, chipped, or broken) and if the tool has rigidity suitable for the material.

   When the cutting conditions are not suitable:
   - Reexamine the cutting conditions.

   When the cutting conditions are suitable:
   - Proceed to step 2.

2. Check if the fit or contact between the guide bushing and the material is suitable.

   When the fit/contact is not suitable:
   - Dismount the guide bushing and draw bar, clean them, mount them again, then adjust the clearance between the guide bushing and the material.
   - Lap the guide bushing along with the material to make its interior surface completely round.
   - Replace the guide bushing.

   When the fit/contact is suitable:
   - Proceed to step 3.

3. Check if the machine, the slides, and the rotary guide bushing device are normal. (No machine vibration or slide play)

   When abnormality is detected:
   - As for the rotary guide bushing, check if the timing belt for driving the guide bushing and the support bearing are normal. If the timing belt is worn, replace the belt. If the support bearing is abnormal, replace the bearing.
   - As for machine vibration or slide play, if it seems to affect the rotary guide bushing device, call the Cincom Service Office.

Note

For belt replacement, see <Section 9.5.1 Replacing the timing belt for driving the synchronous rotary guide bushing>.

11.15.2 Procedure if the dimension of a machining diameter changes largely

If diameter unevenness is within the specified tolerance in the diametrical or longitudinal direction:

Procedure

1. Make the same checks as for steps 1 to 3 in <11.15.1 Fretting problem>.

If diameter unevenness varies with the elapse of time:

Procedure

1. Check when the diameter dimension changes.

   When it changes largely at startup:
   - Make a check on the initial tool wear. If the tool is defective, replace it.
   - Carry out warming up, then restart machining.

   When it changes due to variation in temperature for many hours:
   - Check if the room temperature is kept constant. Improve the operating environment.
   - Check if turning on/off the air conditioning equipment affects the machine. Improve the operating environment.
   - Check if the temperature change in the morning and evening affects the machine. Improve the operating environment.

If diameter unevenness occurs in cases other than the above:

Procedure

1. Check the following probable causes other than the above:
   Clearance with all ball screws of the mechanical system, bearing fail, fail such as play or run-out in the spindle thrust or radial direction, or end-face swing
   Other mechanical fails, or drive unit fails

   - If any of the above is detected, call the Cincom Service Office.

11.15.3 Poor roundness

Procedure

1. Check the roundness of the material.
   If the material has poor roundness, machined workpieces are subjected to poor roundness.
   When non-conform materials are machined, the workpieces are apt to have poor roundness.

   When the material has good roundness:
   - Proceed to step 2.

2. Make the same checks as for steps 1 and 2 in <11.15.1 Fretting problem>.

   When the conditions are suitable:
   - Proceed to step 3.

3. When a synchronous rotary guide bushing device is used, dismount the guide bushing, and use an indicator to check if the tapered part at the opening is subjected to run-out or play.
   If the run-out is great, machined workpieces are subjected to poor roundness.

   When the conditions are not suitable:
   - Call the Cincom Service Office.

Note

Adjusting the clearance between the guide bushing and the material improves the roundness.

11.15.4 Procedure if abnormal sound is produced during rotation

Procedure

1. When a rotary guide bushing is used, check if the timing belt for driving the guide bushing and the support bearing are normal. If the timing belt is worn, replace the belt. If the support bearing is abnormal, replace the bearing.

Note

For belt replacement, see <Section 9.5.1 Replacing the timing belt for driving the synchronous rotary guide bushing>.

11.16 Troubleshooting at Power-on and Screen Operation

11.16.1 Troubleshooting at power-on

When the scandisk is displayed at power-on:

When the message " ... Press any key to run Scandisk on these drives..." appears at power-on, just wait. The scandisk is automatically executed in the MS-DOS mode, and the screen starts up.
If the scandisk is executed but the screen does not start up, call the Cincom Service Office.

When Windows95 enters "Safe Mode" at power-on:

Windows95 may start up in "Safe Mode" at power-on. Turn off the operation panel, and turn it on. Windows95 is generally switched into normal mode by doing so.
If Windows95 still starts up in "Safe Mode" after repetition of turning off and on the operation panel several times, call the Cincom Service Office.

When Windows95 fails to start up at power-on:

If Windows95 fails to start up at power-on (the lamps on the operation panel do not light at all), call the Cincom Service Office.

CAUTION

Do not give any shock to the operation panel and hard disk.
Failure to do so may damage the hard disk in the operation panel. The hard disk is easily affected by shock.

Note

If the hard disk is damaged by man (e.g., shock), we will repair or replace it at a cost.

11.16.2 Troubleshooting at normal screen operation

This machine is designed with the operation screens as personal computer applications. Therefore, a screen may be forcibly terminated or may enter the hang-up state. The symptom is not trouble. Turn off the NC power by the switch on the operation panel, and then turn it on. In general, correction can be made by reactivating the operation screen (restarting only the personal computer).

When an operation screen enters the hang-up state during automatic operation:

1. Take the following steps to forcibly operate the machine in 1-cycle mode, and restart only the personal computer when the 1-cycle operation is completed.

When an operation screen enters the hang-up state in cases other than the above:

1. Make sure that the machine is in stopped state, restart only the personal computer, and then follow the procedure below (starting with step 2).

The following explains the above procedures:

Restart the personal computer.

> **WARNING**
>
> In general, do not perform this operation. If it is inevitable to perform the operation, do not get close to the mobile sections of the machine. Failure to do so could result in serious personal injury. The screen disappears during restart processing for the personal computer, but the machine is operating.

> **CAUTION**
>
> Do not operate the machine because the NC unit is active during this operation.
> Doing so may cause interference. The interference may damage the machine.

Procedure

1. Press x1 and START together to forcibly execute 1-cycle operation.
   The machine stops for 1 cycle.
2. Make sure that HDD (indicator lamp) on the operation panel is neither on nor flashing.
3. Keep pressing x1 and x100 together on the operation panel for approximately five seconds (until the indication on the screen disappears).
4. The personal computer restarts up, and the screen appears.

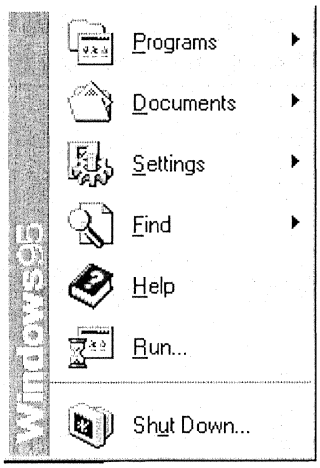
If the Cincom operation screen does not start up when the above operation is performed, diagnose the hard disk (execute the scan disk). You can diagnose the hard disk on condition that Windows95 has started up (the lamps on the operation panel are on and the personal computer responds).

Diagnosing the hard disk (executing the scandisk)

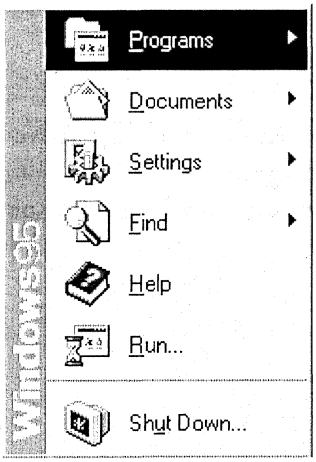
Procedure

The scandisk is a function that searches for and repairs disk errors.

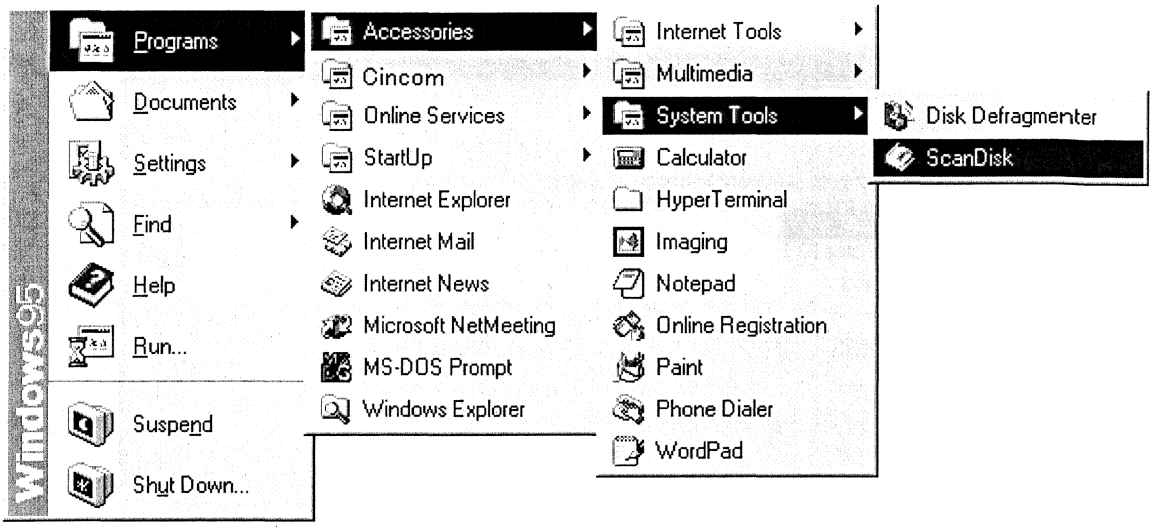
1. Press SHIFT and B in this order to terminate the operation screen.
2. Press SHIFT and A in this order.
   The Start menu appears.



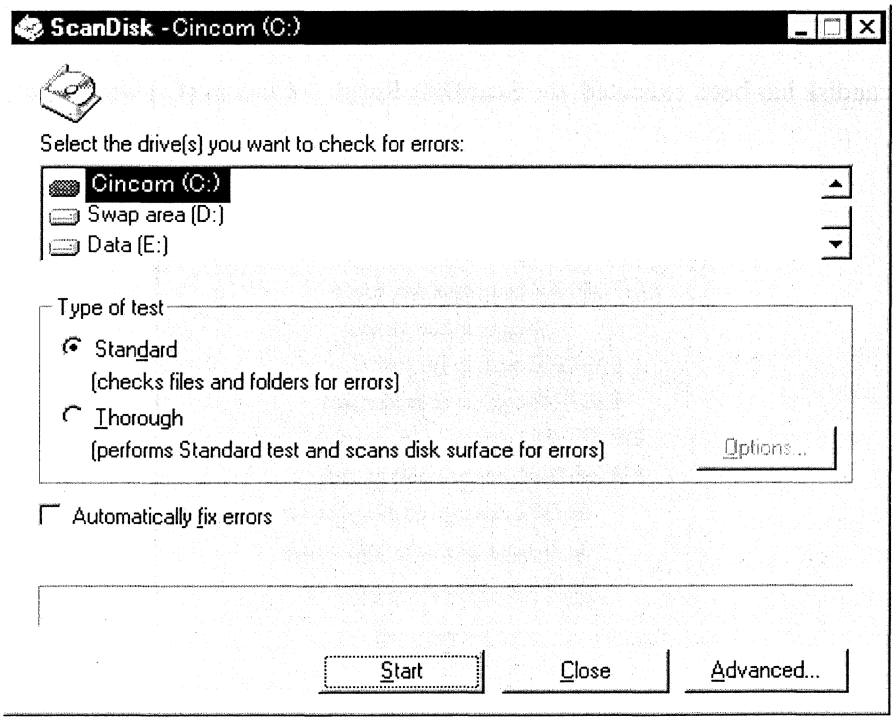
3. Press ↓ to select [Programs].



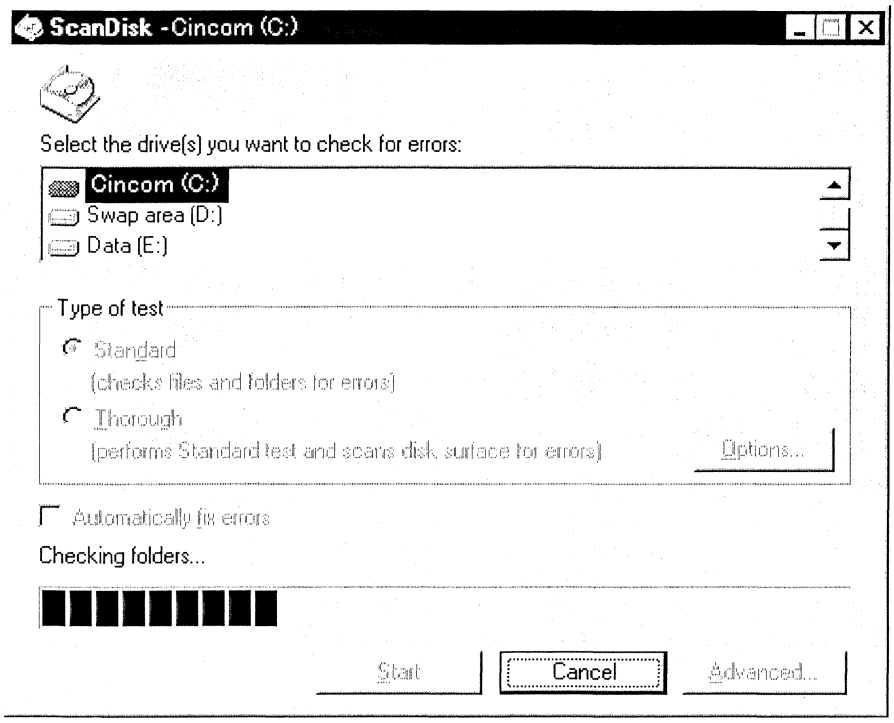
4. Press → and ↓ to select [Programs], [Accessories], [System Tools], and then [ScanDisk].



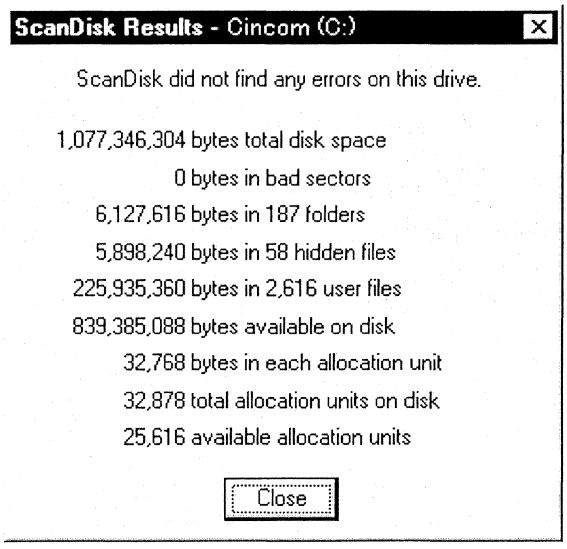
5. Press INPUT.
The ScanDisk – Cincom (C:) window appears.



6. Press INPUT.
   The scandisk is executed. (It takes some time to complete the execution.)



7. When the scandisk has been executed, the ScanDisk Result – Cincom (C:) window appears.



8. Press INPUT.
   The ScanDisk Results – C1 (C:) window closes.

9. Press 

.
The ScanDisk – Cincom (C:) window closes.

10. Press the Power OFF switch O of the operation panel. The messages appear on the screen. Wait until the messages disappear. Then, press the Power ON switch I to turn on the power again.

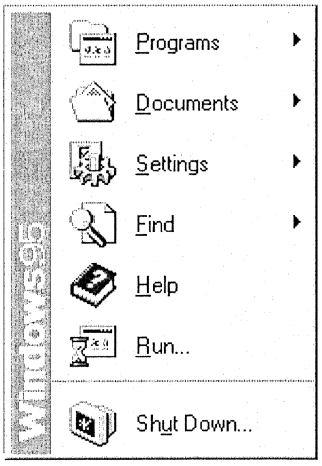
When the operation screen becomes slow to respond:

As the hard disk is used longer, the data in the hard disk is easier to be fragmented. This symptom does not cause any machine operation trouble, but the performance of the personal computer functions lower and the screen becomes slow to respond.

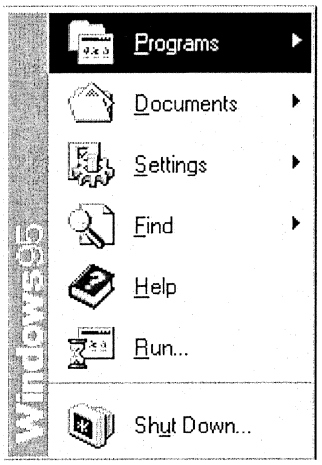
Defrag is a function to optimize the hard disk.

Procedure

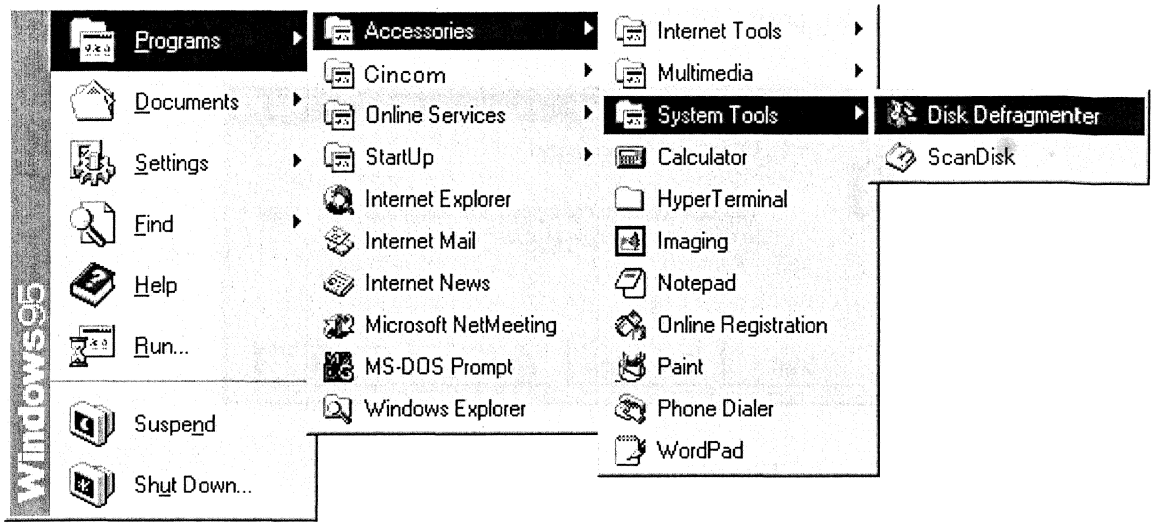
1. Press SHIFT and B in this order to terminate the operation screen.
2. Press SHIFT and A in this order.
   The Start menu appears.



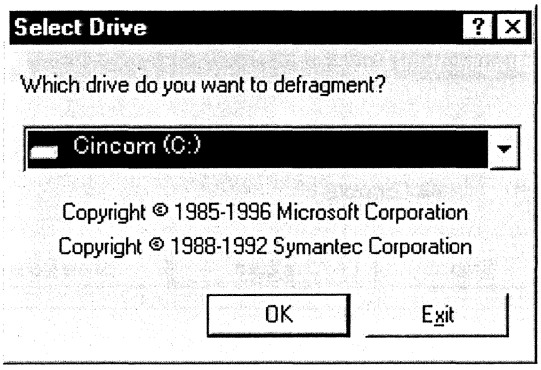
3. Press ↓ to select [Programs].



4. Press [→] and [↓] to select [Programs], [Accessories], [system Tools], and then [Disk Defragmenter].



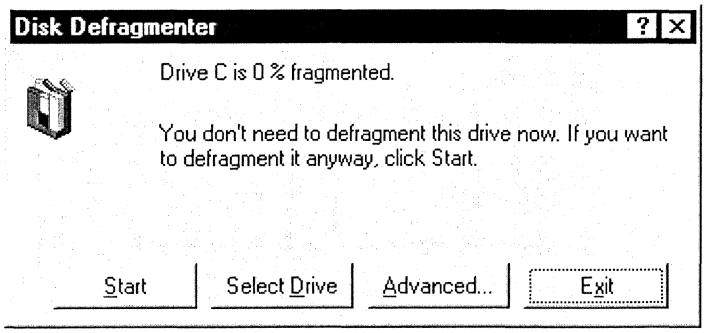
5. Press [INPUT →].
The Select Drive window appears.



6. Press [TAB →] to select the [OK] button.

7. Press INPUT.
   The Disk Defragmenter window appears.

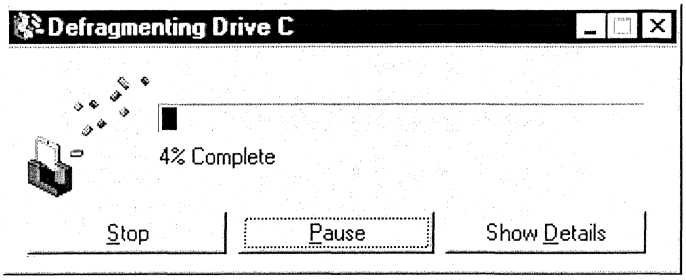
   To cancel disk defragmenter, press INPUT at this stage.



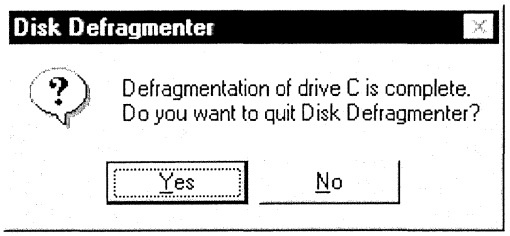
8. Press the TAB key to select the [Start] button.

9. Press INPUT.
   Defrag is executed.

   To temporarily stop disk defragmenter, press INPUT at this stage.



10. When Defrag has been executed, the Disk Defragmenter window below appears.



11. Press INPUT.
    The Disk Defragmenter window closes.

12. Press the Power OFF switch O of the operation panel. The messages appear on the screen. Wait until the messages disappear. Then, press the Power ON switch I to turn on the power again.

11.17 Other Operation Trouble

This machine uses Windows95 as the operating system (OS). An operation screen acts as an application under Windows95. All the settings of Windows95 environment are completed. Never change the settings, and do not install any general application either. Please refrain from inquiries about the setting change and the application installation.

> ⚠ CAUTION
>
> Making changes to the Windows 95 environment or certain files may disturb the machine from operating normally. Installing general applications may case the same type of failure. If you ask Citizen for recovery, you are charged for it in either case.

Product Code	C	–	C	1	2	1	6	VI	VII	VIII	IX				–	7	7	0
Document Code	5	E	1	–	1	1	0	4										