Cincom L20

L20 (2M12)

Instruction Manual

(Original Manual)

CITIZEN MACHINERY MIYANO CO., LTD.

Remarks

- For safety operation, read thoroughly <2. Safety Precautions> to fully understand the safety precautions before starting work.
- 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 our company.
- 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.
-
-
- This machine is a product controlled under the Foreign Exchange and Foreign Trade Act of Japan and is subject to restriction in export. Therefore, you should take necessary procedures in accordance with the above Law to export this machine. Contact our company before exporting this machine.
- The company names and product names shown in this manual are trademarks or registered trademarks of the companies.
- The CE marking put on the machine indicates that the main unit of the machine and the standard attachment units conform to the EC Directive. The optional attachment units and the units of special specification may not conform to the EC Directive. If you intend to relocate the machine to the country where compliance to EC Directive is required, consult with our company beforehand.

Preface

- This Instruction Manual contains the following information.

  The Operator's Manual covers general information on the machine and procedures for basic daily machine operations.

  The Programmer's Manual is used for machine programming.
  It covers information for Cincom programming in order to carry out various basic machining.

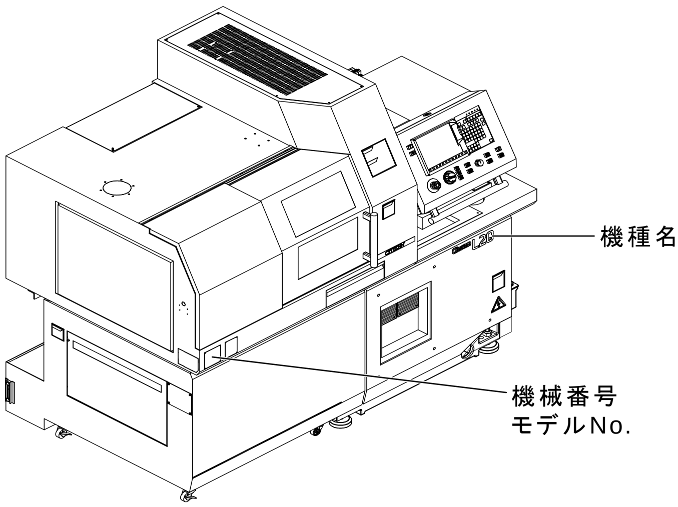
  The Maintenance Manual is used when checking, maintaining, and repairing the machine.
  It contains detailed information for locating problems in the machine, identifying and eliminating their cause, and otherwise maintaining the machine.
- The peripheral devices for this machine are shipped with the instruction manual for the device. Be sure read the instruction manual of the relevant device before using it.

Information to be attached to inquiries

When making inquiries, please confirm the following items as far as possible so that we can take quick action:

No.	Item	Example		Location
1	Machine type	L20		Aluminum nameplate on machine front panel, operation panel, instruction manual
2	Model number	2M12		Aluminum nameplate on machine front panel, cover of instruction manual
3	Machine number	QF0001		Aluminum nameplate on machine front panel
4	NC unit	CINCOM SYSTEM M70 LPC-V		<3. Machine Specifications> in the Operator's Manual, the operation panel, etc.
5	Delivery date	October, 2013		
6	Machine paint color	Gray two-tone color Your specified color		Appearance
7	Special specification	Dedicated transporter		
8	Page of the instruction manual that contains the information concerning your inquiry			Manual: Operator's Manual, Programmer's Manual, or Maintenance Manual
9	Software version	NCMAIN	BND-1010W000-**	On-screen display on the operation panel <23.6.3 Checking the version of software> in the Maintenance Manual.
		PLC	001-001	
		NC OS	BND-1000W022-**	
		HMIs	BND-1010W101-**	
		HMIs-Control	BND-1200W200-**	
		HMIu	001-001	
		MACRO	001-001	
		PARAMET	001-001	
10	H/W list	NC TYPE	MITSUBISHI CNC 70 LPC-V	
		MODEL NAME	FCA70LPC-2A	
		SERIAL NO	M7*********	
		UNIT NAME	FCU7-MU552-C01	

Before making inquiries about problems, please read <23. Troubleshooting> of the Maintenance Manual and take action if necessary.



Instruction Manual

0. Table of Contents

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

(Blank Page)

Product Code	C-L220E XII
Document Code	1E1-0001
Mfg. No.	L220E/0001~
Issue Date	2014.1

1. Outline

1.1 Outline of This Manual

Described below are the purposes of the manual:

- Provides information for operators who need to understand Cincom in order to operate the machine safely.
- Provides information for programmers who need to create a program to perform a variety of basic machining.
- Provides information for service and maintenance engineers who need to inspect, maintain, and repair the machine safely.

The manual contains information about how to handle, check, and maintain the machine while taking advantage of the best performance the machine can yield. Fully understand and learn operation and maintenance methods described in this manual for correct and safe operations in the best condition with any loss of performance.

The manual is composed of the following chapters:

Operator's Manual

Chapter 1 Outline
Explains the purpose and organization of this manual.

Chapter 2 Safety Precautions
Explains notes on safety in programming, setting up and adjusting, operating, and maintaining the machine.
This chapter also explains the provided safety devices and the operation procedures for emergencies.

Chapter 3 Machine Specifications
Explains the major parts of the machine and their functions.

Chapter 4 Operation Panel and Screens
Explains the operation panel, names and functions of switches, lamps, and keys, and various screens which are needed to operate the machine.

Chapter 5 Preparation for Operating the Machine
Explains power-on and power-off of the machine, menu keys, and page keys.

Chapter 6 Operation for Running the Machine
Explains the machining procedures in sequence of machine operation, and functions on each screen in detail.

Chapter 7 Mounting and Adjustment
Explains the methods of mounting and adjusting methods needed for daily operation.

Chapter 8 Operating Procedures
Explains the essential points of the selected operating methods frequently used in daily operation.

Programmer's Manual

Chapter 11 Fundamentals of Programming

Gives basic knowledge needed for programming.

Chapter 12 Command Codes

Explains in detail commands needed for programming.

Chapter 13 Sample Programming [Advanced]

Gives rather complex examples of programming and adds explanations.

Chapter 14 Sample Programming [Secondary Machining]

Gives examples of secondary machining programming and adds explanation.

Chapter 15 Sample Programming [Back Machining]

Gives examples of back machining programming and adds explanation.

Chapter 16 Programming Practices

Gives programming practices.

Chapter 17 Tooling

Lists various tooling systems available with this machine and gives information about the specifications and dimensions of the tools.

Maintenance Manual

Chapter 21 Machine Components

Explains the major parts of the machine and their functions.

Chapter 22 Scheduled Maintenance Checks

Explains regular maintenance and check items (e.g., daily, monthly, and semi-annually) and related information.

Chapter 23 Troubleshooting

Explains various types of machine troubles and alarms and how to respond to these events.

Chapter 24 Relocating the NC Machine

Explains reinstallation conditions and procedures for transportation when the installed machine should be relocated.

Operator's Manual (Separate Volume)

Parts List

Gives the detailed figures of machine parts and parts list.

Wiring Diagram

Gives wiring diagrams and terminal numbers.

Ladder I/O List

Gives the operation sequence of the machine.

1.2 Audience

This manual is intended for:

- All users and operators who use the Cincom machine
- Programmers who who want to make products using the Cincom machine
- Service and maintenance engineers who inspect, maintain, and repair the Cincom machine

To handle the machine safely, be sure to read through the manual and get an adequate understanding.

The target readers should read this manual thoroughly and fully understand it, including the information described in *<2. Safety Precautions>*.

(Blank Page)

Product Code	C-L220E XII
Document Code	1E1-0100
Mfg. No.	L220E/0001 ~
Issue Date	2013.10

2. Safety Precautions

EC DECLARATION OF CONFORMITY
(for reference information)

The construction of the machine

Machine model ____________________

Serial number ____________________

has been developed, constructed and manufactured in compliance with the following EC directives under the full responsibility of Citizen Machinery Miyano Co., Ltd.

The following EC directives:

- **2006/42/EC** **Machinery Directive**
- **2004/108/EC** **EMC Directive (Electro magnetic compatibility)**

The following harmonized standards have been referred to:

- **EN ISO 12100-1**
- **EN ISO 12100-2**
- **EN954-1**
- **EN12415**
- **EN60204 -1**

To evaluate the results regarding electromagnetic compatibility, the following standards were referred to:

- **EN50370-1 (for emission)**
- **EN50370-2 (for immunity)**

The technical file is complete and available.

Signature ____

Date and Place of the declaration ____

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 your Cincom Service.

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 from 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.

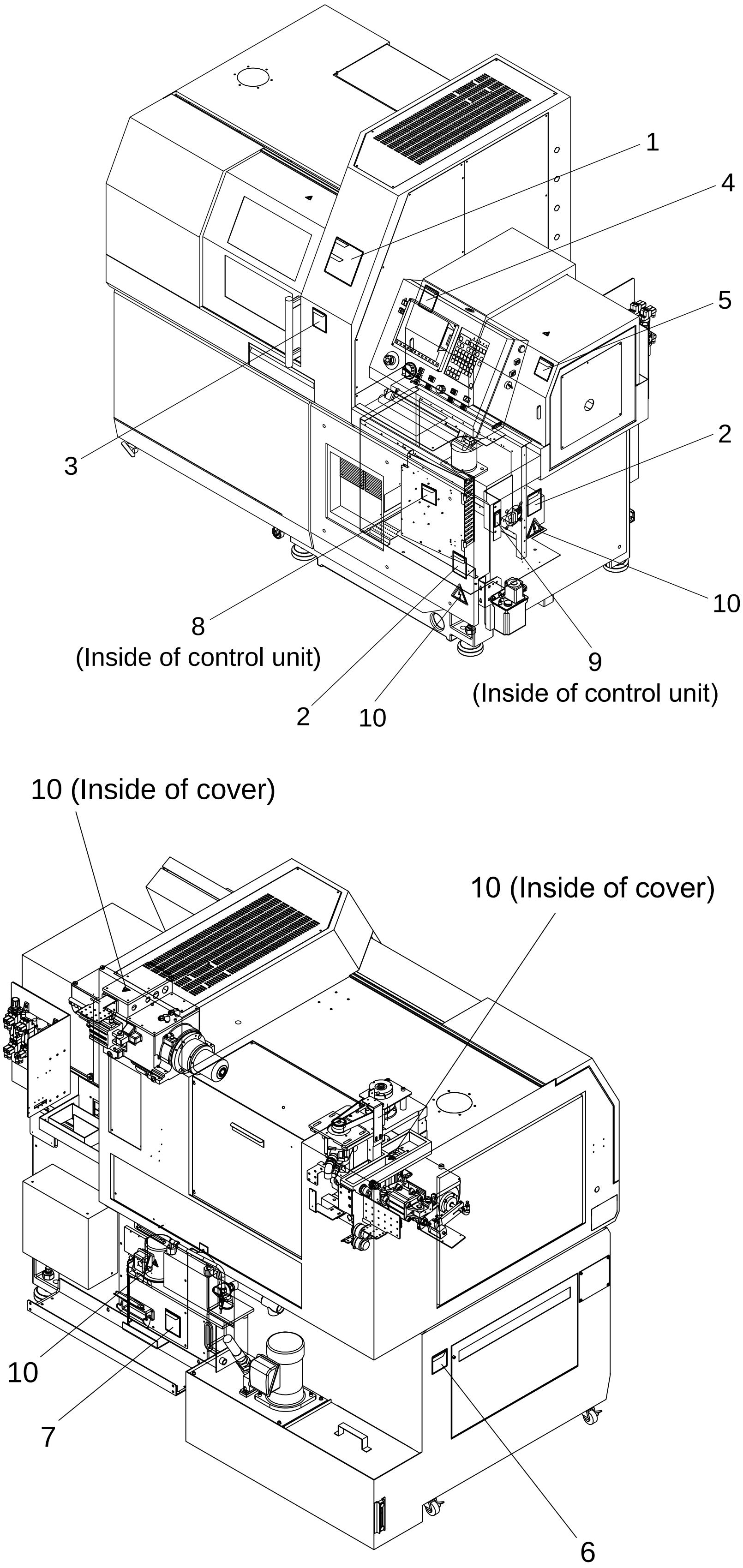


Figure 2.1-1 Locations on which the warning labels are put

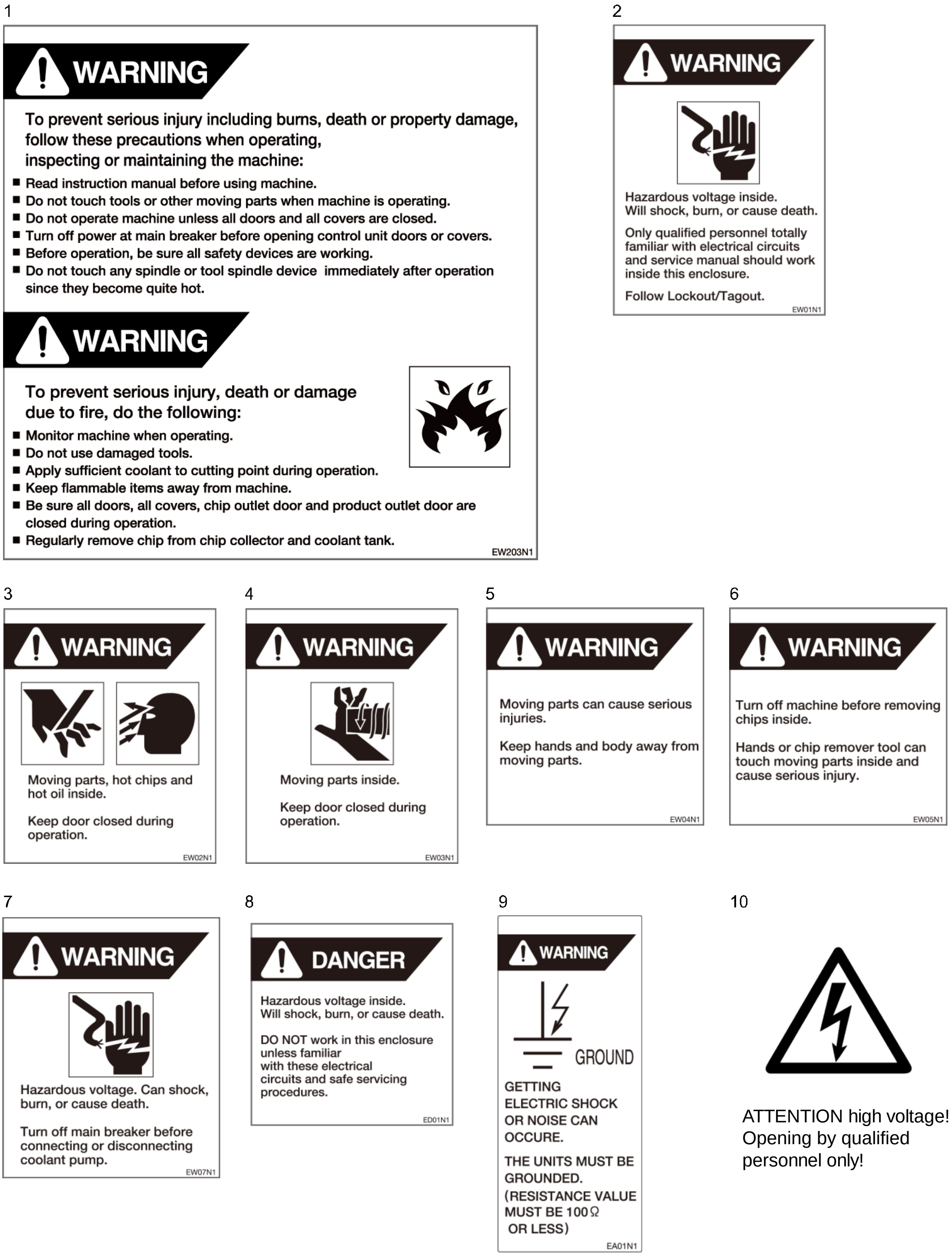


Figure 2.1-2 Contents of warning labels

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. Press this button anytime you feel dangerous while operating the machine. 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, and perform the following steps.

[Procedure]

1. Turn the locked emergency stop button clockwise to release the lock.
2. Press the Power OFF button [O] to turn the power off.
3. Press the Power ON button [I] to turn the power on.
4. 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.

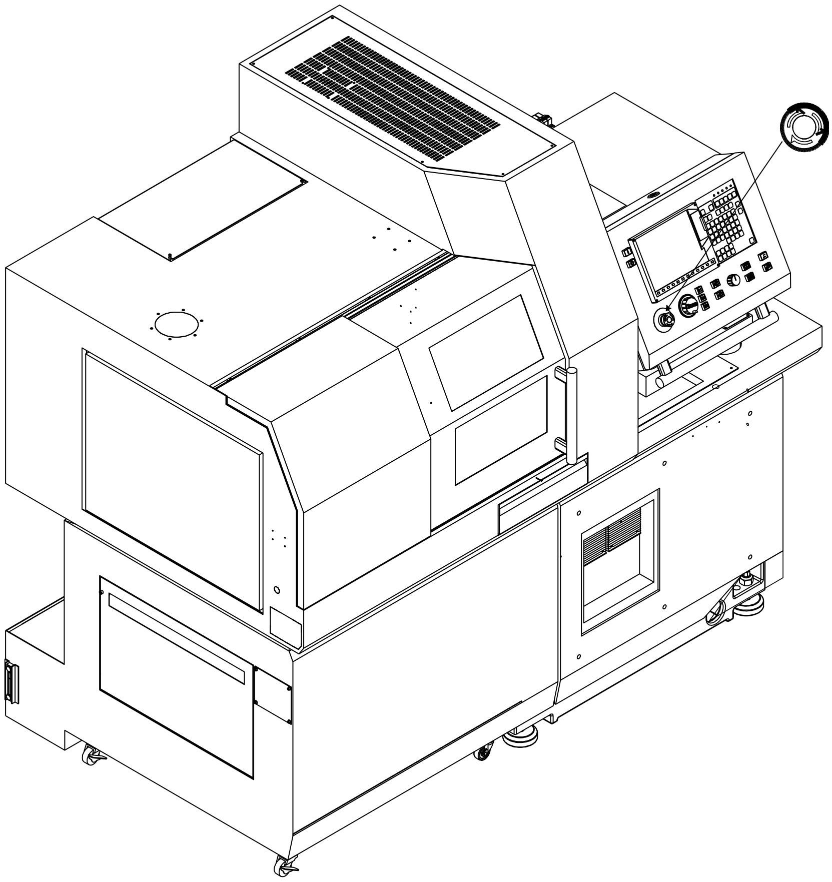


Figure 2.2-1 Location of the emergency stop button

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*> shows the locations on which safety devices are installed.

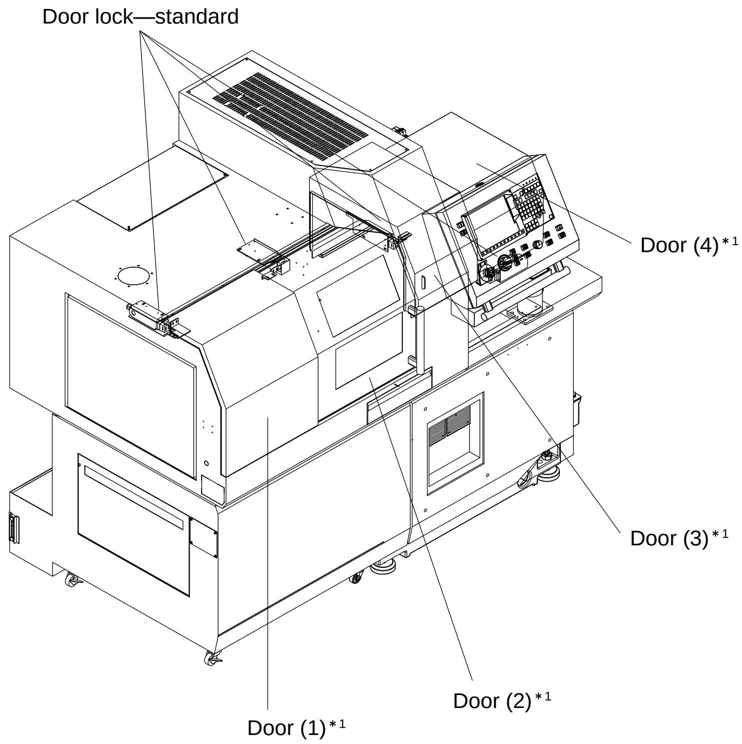


Figure 2.3-1 Locations of the safety devices (front view of the machine)

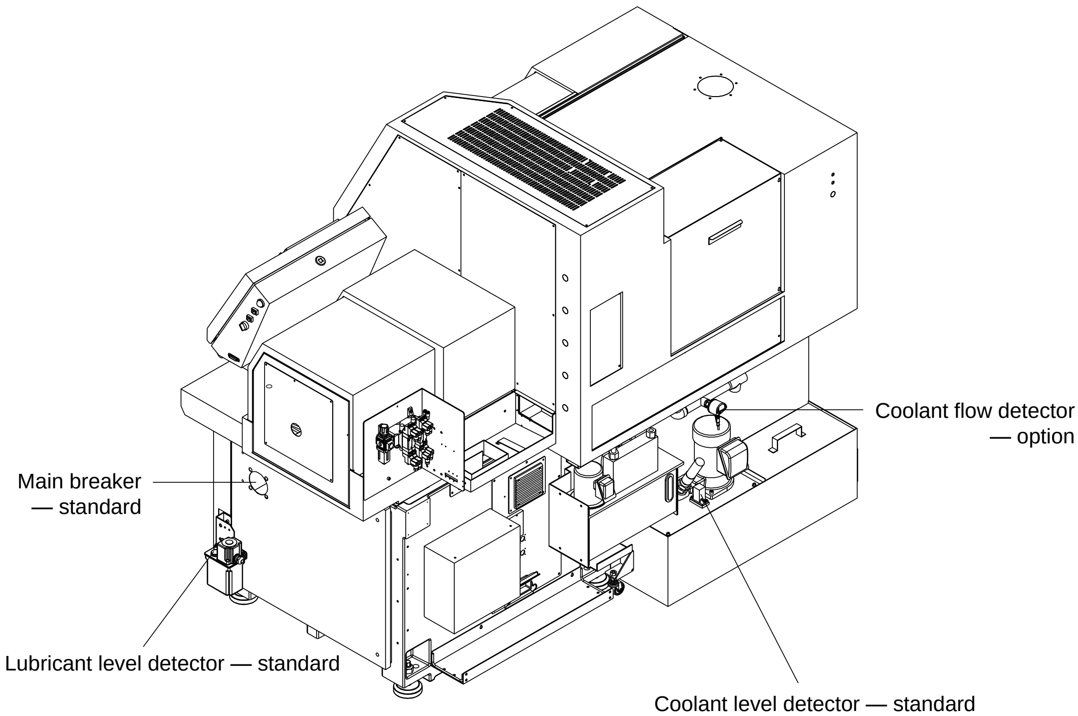


Figure 2.3-2 Locations of the safety devices (rear view of the machine)

The following are detailed descriptions of the safety devices:

Door locks — standard

Door locks prevent any person from opening the doors of cutting room (1) and main spindle side (2) during machine operation. See *<Figure 2.3-1>*. These doors will open only when the Door open button is pressed while the machine is stopped.

In the Handle Feed or Preparation mode, however, operations (excluding operations regarding the main spindle and the turret) 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 and turning coolant supply on or off can be performed whether the doors are open or closed, if the Setting switch O I is set to "|".

Main breaker — standard

This device automatically shuts itself off when it detects an over current of 40A or more, or an electric leakage of 30 mA or more.

Cut-off tool breakage detector — standard

A cut-off tool is very easily damaged. If you continue to operate the machine with a damaged cut-off tool, this might damage the machine or cause a fire. 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 alarm message "EX107 Tool bit breakage alarm. Cut off tool is broken." is displayed.

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 alarm message "EX203 Coolant oil alarm. Supply the oil." is displayed and 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 alarm message "EX202 Lubrication oil empty alarm. Supply the oil." is displayed and the operation of the machine is automatically stopped after completing one cycle to prevent machine damage.

Knock-out overload detector — standard

This device is used to detect an overload on the knock-out pipe.
If the workpiece which has been inserted too deep in the back spindle device at re-chucking interferes with the knock-out pipe, an alarm message "EX106 Knock-Out Overload. Knockout error sensor ON by overload." is displayed and the operation of the machine is automatically stopped to prevent a machine damage.

Coolant flow rate detector — option

This device is installed in the coolant supply path to observe the flow rate of the coolant.
When the coolant level gets lower than the setting value, the alarm message "EX105 Coolant discharge alarm. Discharge amount is decreased." is displayed and the operation of the machine is automatically stopped to prevent a machine damage.

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

Spindle speed change detection – standard

This function detects excessive changes in spindle speed to prevent machine damage. The alarm message "EX109 Main spindle speed fluctuation alarm." or "EX110 Back spindle speed fluctuation alarm." is displayed when it detects a change of more than 10% from the specified speed.

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 spindles, guide bushing, and tool posts of the machine. When the function determines the possibility of interference during machine operation, the NC alarm message "OP2□□ ○○ and △△ are interfered." is displayed and the operation of the machine is stopped to prevent machine damage.

If this alarm is displayed, the operator must correct the program.

2.4 Specifications for Safe Operation

The start, stop, and operation speeds of the machine are specified for safety as shown below.

	Door opened (The Door select switch key O I is set to "I")	Door closed
Automatic operation Programmed operation Program check MDI operation Chucking operation Cutting during cut-off machining Non-conformed material phase adjustment Knock-out advance/return	Activation disabled	Command-specified speed (Override enabled)
Preparation Return position Positioning point Start position Manual tool setting Gang tool post retraction Opposite tool post retraction Tool selection for cut-off machining Material setting Manual operation Handle feed	Operation at a speed of 2 m/min or less (Override enabled)	Operation at a speed of 10 m/min or less (Override enabled) In manual tool setting, operation speed is 2 m/min or less (Operation speed increases to 10 m/min when "Toolset Feed UP" on the Set SW screen is enabled.)
[Others] Opening/closing of chuck with the SP.CHUCK key	Operation at 100% speed	Operation at 100% speed

The machine has door locks standard with the splash guard and spindle cover sections.

[Note]

- If the Door select switch key O I is set in the "O" position, machine operation excluding handle feed is disabled while the door is open.
- With the machine equipped with Enable switch, press the Enable switch before performing Preparation operation while the Door Select switch key O I is set to "O" position.

2.5 General Precautions During Operation

Be sure to follow these general precautions for handling the machine.

DANGER

To prevent death or severe personal injury, always follow these safety practices:

- This Cincom is a machine that aimed to cut the workpiece (e.g., metal or resin) with the cutting tool in automatic operation mode. Do not use the machine in any other purpose than cutting.
- Do not operate the machine with any cover open.
- Do not put your fingers or hands into any moving part of the machine during machine operation. When the Start button LED on the operation panel lights or flashes, regard the machine status as Operating.
- Do not touch or stand close to any cutting tool or rotating part of the machine during machine operation.
- Do not touch any live electrical component of the electrical system. You could be electrocuted if you touch live electrical components.
  All the control unit covers and doors must be closed during machine operation. Shutdown the main breaker of the machine before removing the control unit covers and doors.
- Do not modify the machine and control circuit.
- Press the emergency stop button to stop all machine motions if an emergency state arises. To restore the machine after an emergency stop, make sure that the restoration of the machine is safe without causing danger, reset the pressed emergency stop button, shut off the power supply and the main breaker and then return them on immediately. After that return all machine moving parts to the respective return positions and remove all materials having been machined from the machine.

WARNING

Follow these safety practices while operating the machine. Failure to do so could result in death or serious personal injury.

- Never disable any safety devices while operating the machine during automatic operation.
- 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.
- When operating the machine, sufficiently understand the operation and visually confirm the operation switch to be used before actually pressing that switch.
- When machining a material combustible (flammable) during machining by cutting, operate the machine in a state in which the operator can always monitor the machining process.

CAUTION

Follow these safety practices. Failure to do so may result in minor/moderate personal injury and/or damage to the machine:

- The machine must be properly grounded. The ground must be electrically separated from power lines or the grounding wires of another machine that could be a source of massive electrical noise. See <*Chapter 24 Relocating the NC Machine*> in Maintenance Manual.
- Make sure that there is enough coolant in the machine and it is being supplied smoothly to all necessary parts.
- Check the tooling to see that it is securely clamped in place before starting the machine.
- Be sure to do the periodical checking described in the manual.
- 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.

2.6 Safety During Installing the Machine

To prevent accidents which could result in death or serious personal injury, be sure to observe the following precautions:

- 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.
- Be sure to shutdown the main breaker of the machine and the breaker for power supply to the machine on the plant side before connecting/disconnecting the power cable to/from the machine.
- Be sure to connect the ground line when connecting the power cable to the machine.

2.7 Before Starting 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.
- Fire prevention procedures (see *<2.12 Fire Prevention Practices>*).

Each time before you start the machine, do the following to prevent injury or damage:

- 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.
- 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.
- Inspect the area around the machine for spills or objects that could cause slipping, falling, or tripping.
- Make sure that no maintenance work is being performed on the machine.
- Check the inside and outside of the machine to see that it is free of all foreign objects (tools, workshop towels, etc.).
- Make sure there is enough oil in the lubricating oil tank and coolant tank.
- 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. NEVER 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.
- Check all screws of the tool holder, chuck, guide bushing and others to make sure they are not loose; tighten them if necessary.
- Make sure that the tools and the material are mounted firmly. Also make sure that the correct tools are installed for the machining program to be used and the offset has been specified properly (if necessary).
- Make sure that all doors are securely closed.

2.8 Safety During Setup

Disregarding this safety practice could cause severe personal injury.

- Never adjust the tools or measure the dimension of the material during machine operation. This can cause severe personal injury.

  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.
- Check whether the chucking force is sufficient for the material.
- Do not make the guide bushing too tight--it can cause burning or galling. Fix the guide bushing using the most suitable tightening torque.
- When you have used any jig or tool for adjusting the guide bushing or chuck, be sure to remove the jig or tool.
- Before attempting to move a part by pressing a button, be sure to visually confirm that part and button.

When setting up the machine, follow these safety practices to prevent injury or damage:

- Make sure that tools attached do not interfere with mechanical portions of the machine every time the machine is set up for new and exchanged programs or tools. If there is interference, this could cause machine damage and personal injury.
- When you check the tool movement, always know where you are in the machining program so that you know when and how the machine will move and can follow the necessary safety precautions for this movement.
- Verify the machining program and the actual setup. Check for unintended results, which you should always try to anticipate, such as machine damage and dangerous operating conditions.
- Because this machine has no way to check and correct user's machining programs, the machine will act exactly the way it is programmed.
- Verify the machining programs carefully. Failure to do so could cause machine damage and personal injury.
- Never 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.
- Use the proper tools and install them only after the machine is completely stopped.
- When any door is open during machine setup, do not select a tool, or do not operate the main and back spindles or the rotary tool driving device. Also, do not touch cutting tools on the tool posts during machine operation.

2.9 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.

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.
- Be aware of common operating problems and correct them immediately. Common problems include jammed chips, damaged tools, burned material, etc.
- Perform the following inspections on a regular basis. Monitoring intervals will depend on cutting condition, type of material, etc., and can only be determined by the operator through experience with the machine.
  - Clear the Machine of 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. Monitor the level and condition of coolant periodically. Failure to do so can result in damage to the tools and a possible fire hazard.
  - Inspect the Coolant
    Monitor the coolant condition and level periodically. 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.

2.10 Safety During Maintenance

Disregarding the following safety practices could cause machine damage and serious personal injury.

- Only qualified maintenance personnel should perform maintenance operations on the machine.
- The safety devices can be disengaged for maintenance operations using the operation panel. Before disengaging the safety devices, make sure you are familiar with the current program and where the machine is in the program. Make sure the machine has stopped before reaching into the machine to make any adjustments. Cincom recommends that the safety devices be engaged for all machine operations except during required maintenance.

  Before removing control unit covers and doors, turn off the work light, then turn off main circuit breaker of the machine.
- Always use the specified fuse. Installing a wrong type/size of fuse in the machine can cause machine damage and a possible fire.
- A blown fuse indicates an electrical problem that must be corrected.
- 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.
- Periodically clean the air filter. The cleaning interval depends on the operating environment. Operating the machine with a dirty air filter could damage the electrical components.
- Follow appropriate lockout/tagout procedures during maintenance.

2.11 Safety After Everyday Operation

- Turn off the work light, then turn off the main circuit breaker of the machine.
- Carefully remove any chips caught on the material and cutting tools.
- Apply rust prevention oil on all unprotected (unpainted) machine surfaces.

2.12 Fire Prevention Practices

■ 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
  - 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.
  - A coolant having too low flash temperature (100ºC or lower) is used 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 22 Periodical Check>* in the Maintenance 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 8 Operating Procedure>* in this 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 all doors on the cutting room, 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.

Equipment for fire prevention

Take appropriate measures for fire prevention.

- The machine sold in Japan is equipped with automatic fire extinguisher.
- If you operate the machine without the automatic fire extinguisher, take substitute fire prevention measures.

The automatic fire extinguisher is not a device that can suppress the fire to occur. It is a device to prevent spreading when a fire occurs. Even if an automatic fire extinguisher has been installed, fire might not be able to be extinguished depnding on the status of a fire. The most important thing is to prevent a fire from occurrence.

Be sure to operate, maintain, or inspect the automatic fire extinguisher according to Appendix *<Fire extinguisher manual>*.

WARNING

Observe the following to prevent fire from spreading.

- At the first time you operate the machine that has an automatic fire extinguisher, or before starting operation, maintenance work, and/or inspection of the machine, be sure to read thoroughly the Instruction Manual of Fire Extinguisher.
- If you operate the machine without the automatic fire extinguisher, be sure to take substitute fire prevention measures.
- The automatic fire extinguisher functions when the cutting room door, chip outlet, and product outlet are closed (in airtight state). Confirm that the cutting room door, chip outlet, and product outlet are closed during machine operation.
- Be sure to mount the shutter on the mist collector that closes automatically by interlocking with the automatic fire extinguisher. If the shutter is open, extinguishment will fail.

Also observe the following to prevent metal-caused fire from spreading.

- The combustible metal may ignite and burn by the cutting heat or others, if it is a thin tip, particle, or melting state. The provided CO_2 fire extinguisher cannot extinguish such as metal-caused fire. Be sure to use a fire extinguisher dedicated to metal-caused fire.
  Keep the following in mind when cutting the above workpiece:
  – Avoid unattended operation.
  – Prepare a fire extinguisher dedicated to metal fires near the machine.
  – Do not use water to fight such fires.

Listed below are typical combustible metals. Contact the manufacturer of the material for detailed information.

zirconium, magnesium, titanium, calcium, hafnium, lithium, plutonium, sodium, thorium, uranium, zinc, and potassium

2.13 Handling Emergency Situations

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

2.13.1 Machine operation at occurrence of an accident

If an accident (e.g., any part of an operator's body caught in the machine) occurs, immediately stop the machine and perform the following procedure:

When the Emergency Stop button is already pressed:

Release the Emergency Stop button by turning it clockwise, and press the Power off button $\boxed{\mathrm{O}}$. Then, turn OFF the main circuit breaker, then turn them ON again.After that, press the Power on button $\boxed{\mathrm{I}}$ to turn on the power, and move the control axis into a safety direction in manual operation mode. For the manual operation mode, see <6.7 Manual Operation> in the Operator's Manual.

When the Emergency Stop button is not pressed yet:

Move the control axis into a safety direction in manual operation mode. For the manual operation mode, see <6.7 Manual Operation> in the Operator's Manual.

2.13.2 Emergency situations requiring evacuation

When evacuation is required in the case of emergency situations, such as fire, earthquake, or lightning, perform these procedures with the top priority on human lives.

- If you have time before evacuating

  Stop the machine quickly and 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

  Press the emergency stop button to stop the machine immediately.

2.13.3 Power failure

When power failure occurs, turn off the main circuit breaker.

2.13.4 Resuming work

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

[Procedure]

1. Inspect the tool and machine carefully for damage and replace or repair if necessary.

   And confirm the machine can be operated safely and properly.
2. Cut off the material that was being machined when the emergency occurred.
3. Resume operation of the machine according to the Instruction Manual.

2.14 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.
- Pay attention to the cleanness of supply air. Contaminated air can cause damage to the machine.
  For the cleanness of supply air, see *<24.1 Selecting the Installation Site>* in the Maintenance Manual.

2.15 Notes on prevention of accidents caused by electromagnetic radiation and magnetism

WARNING

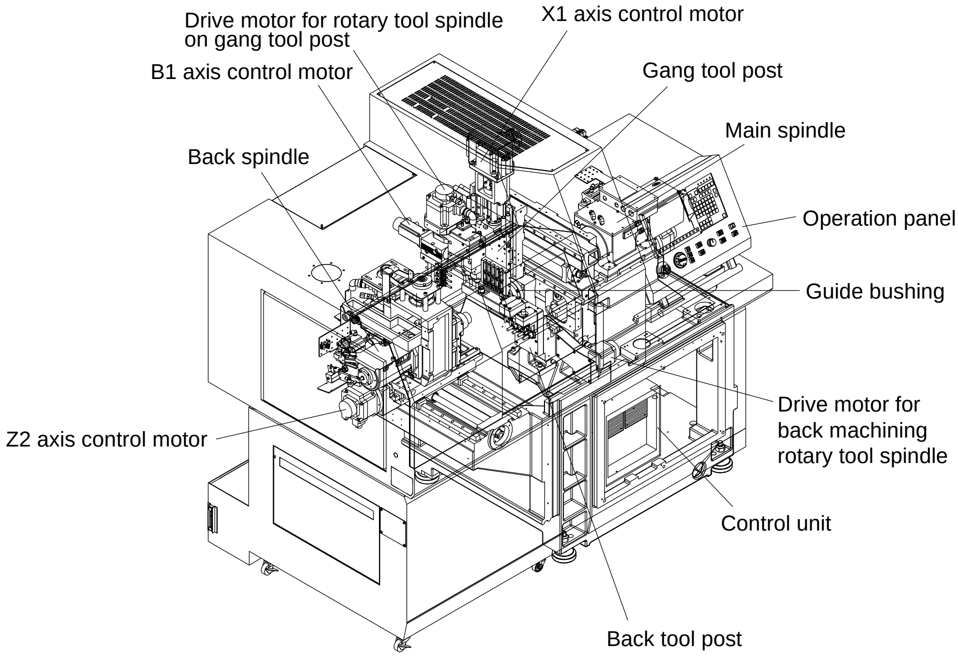
- This machine conforms to International Standard concerning the limiting value of the electromagnetic radiation. A person who uses an implanted medical device such as a pacemaker and other medical electronics device must consult with the medical institution or the manufacturer of such device for the influence caused by the electromagnetic radiation before operating this machine.
- In the machine that uses a linear motor to drive the feed axis, a strong magnet is used for the relevant locations. The magnetism is attenuated while the cover of the magnet is being closed. However, if the cover is removed for maintenance work or others, the magnet is exposed. Under such a condition, an electronic equipment and the precision instrument may be affected if they are placed in a area within 30 cm of the magnet. A person who uses an implanted medical device such as a pacemaker and other medical electronics device should not operate the machine in the state that the magnet is exposed. The fatal accident might be caused. If you feel an abnormality of medical equipment even if you are away from the magnet by 30 cm or more, immediately keep away from the magnet and consult your doctor.
- Do not bring a magnetic substance including the tool close to the magnet of a linear motor that is exposed. A strong magnetic force may cause a hazardous situation which will result in death or serious personal injury. Do not bring the clock, the cellular phone, the camera, personal computer, magnetic card, nor a magnetic storage media close to the magnet. Doing so may cause a damage or a failure of the equipment.

(Blank Page)

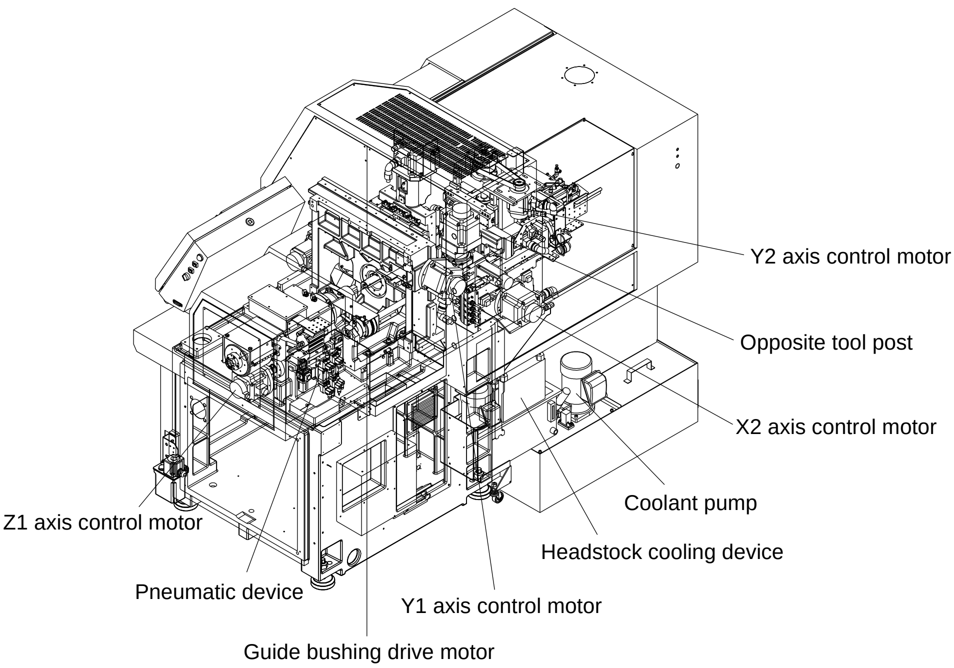
Product Code	C-L220E XII
Document Code	1E1-0200
Mfg. No.	L220E/0001 ~
Issue Date	2013.10

3. Machine Specifications

3.1 Machine Main Components

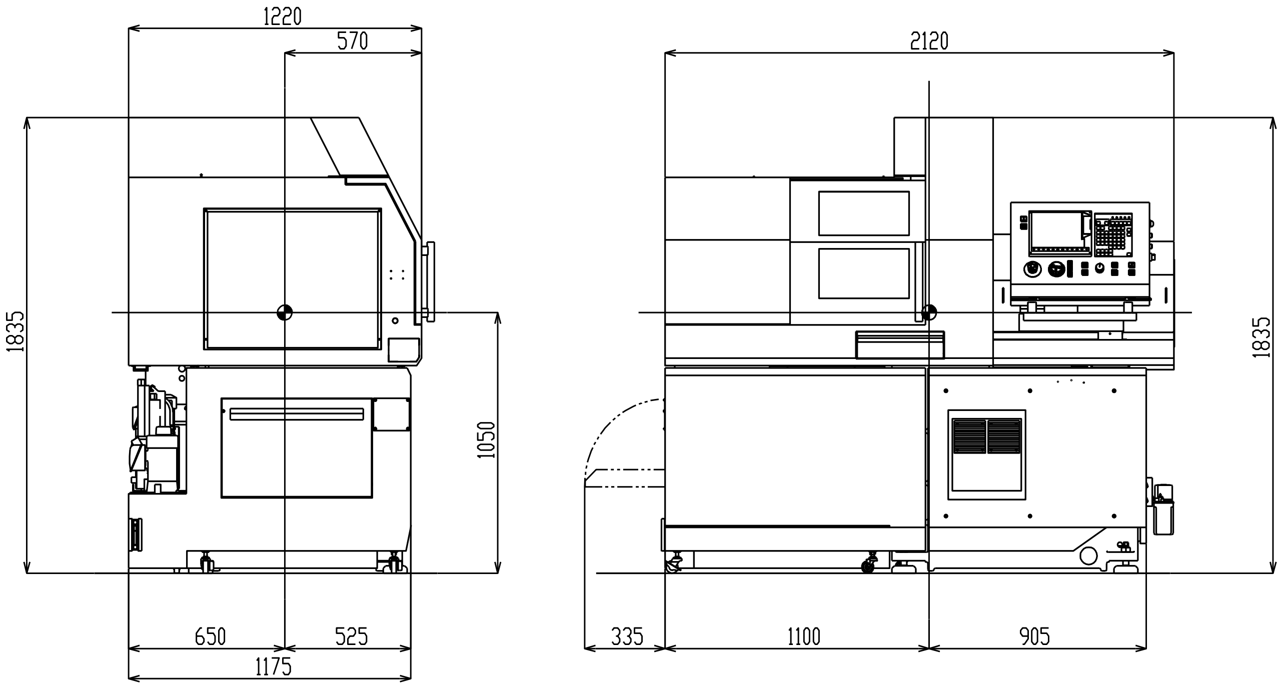


Front view of machine (L20 type XII)

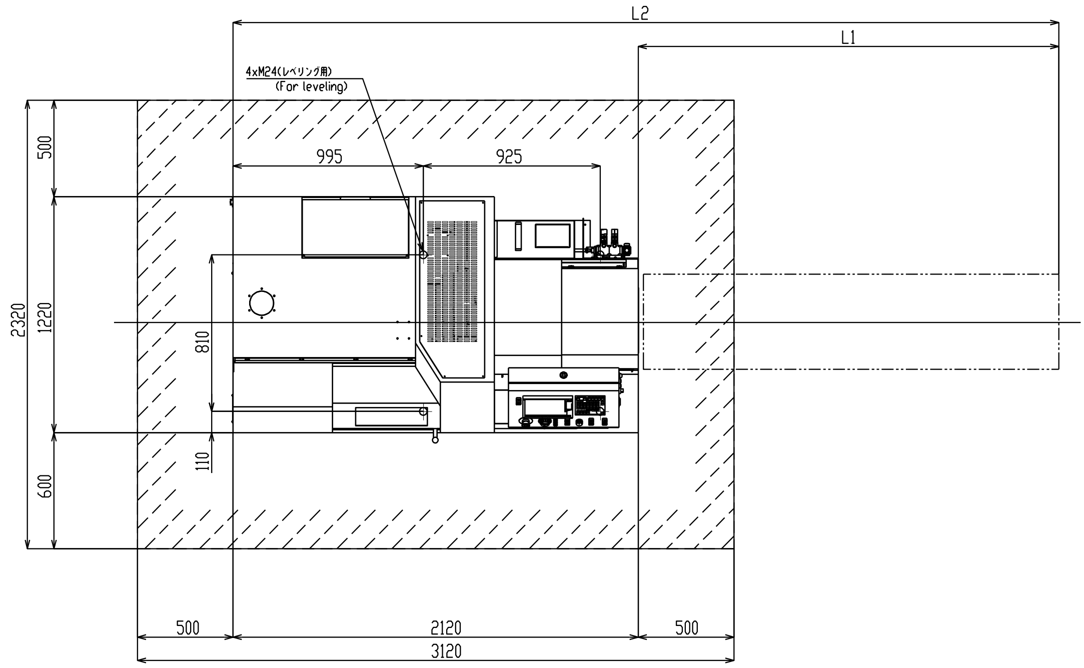


Rear view of machine (L20 type XII)

3.2 Outside Dimensions and Layout of Machine



Outside dimensions of the machine



Machine layout

Manufacturer	Type	L1	L2
IKURA SEIKI	OS20VC-4.0m	5020	7140
Alps Tool	ASR X-20ZC II-4.0m	4914	7034
	ASR X-26P-4.0m	5267	7387

3.3 Specifications

3.3.1 Standard machine specifications

No.	Item	L20 Type XII (L20E-2M12)	Remarks
1	Maximum machining diameter (Optimum machining diameter)	ø20 mm [ø0.79"] (ø6 mm to max. machining diameter) ⌀ 25 mm [⌀ 0.98"] (Option)	
2	Maximum machining length	GB:200 mm [7.87"]/1 chuck GBL:2.5D (Max. 50mm) [1.97"]/1 chuck	The maximum length of the products that can be collected by the workpiece separator is 80 mm. To machine products longer than this limit, use an optional long workpiece device. This device allows machining and collection of a product up to 600 mm long.
3	Maximum front drilling diameter	⌀10 mm [⌀0.39"]	Drilling larger holes than specified is possible, depending on cutting conditions and material.
4	Maximum front tapping diameter (tap and die)	M8 (Tap) M6 (Die)	The specification given on the left is the maximum tapping diameter for a cutting tap.
5	Spindle through-hole diameters	⌀26 mm [⌀0.95"]	The through-hole diameter of the chuck sleeve is ⌀21 mm (⌀ 25.3 mm [⌀0.99"]:Option). The maximum diameter of the finger of the bar loader must be smaller than above diameter.
6	Main spindle speed	200 to 10,000 min^{-1}	
7	Main spindle speed change steps	Stepless, S5-digit	
8	Main spindle indexing	1º	
9	Spindle C-axis	0.001º	Option
10	Maximum chucking diameter of back spindle	⌀20mm [⌀0.79"] ⌀ 25 mm [⌀ 0.98"] (Option)	
11	Maximum front protrusion length of back-machined workpiece	80mm [3.15"]	Optional long-workpiece unit can be used to collect up to 600 mm products.
12	Maximum external workpiece length of back spindle	30mm [1.18"]	Maximum length of the protrusion of workpiece from the end face of the back spindle chuck.
13	Back machining maximum drilling diameter	⌀8mm [⌀0.32"]	Drilling larger holes than specified is possible, depending on cutting conditions and material.
14	Back machining maximum tapping diameter	M6	The specification given on the left is the maximum tapping diameter for a cutting tap.
15	Back spindle speed	200 to 8,000 min^{-1}	
16	Back spindle speed change steps	Stepless, S4-digit	
17	Back spindle indexing	1º	Option
18	Back spindle C-axis	0.001º	Option

No.	Item	L20	Remarks
		Type XII (L20E-2M12)	
19	Rotary tool for cross machining Maximum drilling diameter	⌀8 mm [⌀0.32"]	Cross machining exceeding the specifications on the left is possible depending on the cutting conditions and material. The maximum tapping diameter on the left is the specification for the cutting tap. The B-axis tool rotates at a maximum of 8000 (6000) revolutions per minute. ( ): Rated spindle speed. The motor may be overloaded if the machining is continued with the spindle speed exceeding the rated value. In that case, reduce the spindle speed to prevent damage to the motor or gear.
	Maximum tapping diameter	M6	
	Main spindle speed	200 to 6,000 (4,500) min^{-1}	
	Main spindle speed change steps	Stepless, S5-digit	
20	Front machining rotary tool Maximum drilling diameter	⌀5 mm [⌀0.20"]	L20X: Option Drilling larger holes than specified is possible, depending on cutting conditions and material. The maximum tapping diameter on the left is the specification for the cutting tap. ( ): Rated spindle speed. The motor may be overloaded if the machining is continued with the spindle speed exceeding the rated value. In that case, reduce the spindle speed to prevent damage to the motor or gear.
	Maximum tapping diameter	M4	
	Main spindle speed	300 to 7,500 (6,000) min^{-1}	
	Main spindle speed change steps	Stepless, S5-digit	
21	Back machining rotary tool Maximum drilling diameter	⌀5 mm [⌀0.20"]	Drilling larger holes than specified is possible, depending on cutting conditions and material. The maximum tapping diameter on the left is the specification for the cutting tap. ( ): Rated spindle speed. The motor may be overloaded if the machining is continued with the spindle speed exceeding the rated value. In that case, reduce the spindle speed to prevent damage to the motor or gear.
	Maximum tapping diameter	M4	
	Main spindle speed	200 to 7,500 (6,000) min^{-1}	
	Main spindle speed change steps	Stepless, S5-digit	
22	Chucking, bushing types Spindle collet chuck Guide bushing Rego type chuck Back spindle collet chuck	 FC034-M WFG206-M ER11, AR11, ER16, AR16 FC034-M-K	
23	Number of tools to be mounted Lathing tool Gang tool drilling tool Front drilling tool Back drilling tool	30 5 11 6 8	These numbers indicate the number of tools that can be mounted when holders for tool sizes □12 and □13 are used. The holder for □13 is optional.

No.	Item	L20 Type XII (L20E-2M12)	Remarks
24	Tool size Tool bit	12 × 12 × 130 mm [0.47 × 0.47 × 5.12"]	Optional □3/8", and □13 tools can be used. However, each of these options needs a dedicated tool holder.
	Sleeve	⌀19.05 mm [⌀0.75"] ⌀25: dedicated sleeve	
25	Maximum shank diameter of tool mounted on rotary tool holder (Drill and end mill)	⌀7 mm [⌀0.28"] (ER11, AR11) ⌀10 mm [⌀0.39"] (ER16, AR16)	
26	Rapid federate X1 axis	32 m/min	
	Y1 axis	32 m/min	
	Z1 axis	32 m/min	
	X2 axis	32 m/min	
	Y2 axis	8 m/min	
	Z2 axis	32 m/min	
27	Least input increment X1 axis (diameter)	0.001 mm (0.0001 mm) [0.0001" (0.00001")]	( ): Option
	Y1 axis (diameter)	0.001 mm (0.0001 mm) [0.0001" (0.00001")]	
	Z1 axis	0.001 mm (0.0001 mm) [0.0001" (0.00001")]	
	X2 axis (diameter)	0.001 mm (0.0001 mm) [0.0001" (0.00001")]	
	Y2 axis (diameter)	0.001 mm (0.0001 mm) [0.0001" (0.00001")]	
	Z2 axis	0.001 mm (0.0001 mm) [0.0001" (0.00001")]	
28	Axis stroke X1 axis	126.5 mm [4.98"]	
	Y1 axis	342 mm [13.46"]	
	Z1 axis	205 mm [8.07"] (GB machine), 55 mm [2.17"] (GBL machine)	
	X2 axis	316 mm [12.44"]	
	Y2 axis	60 mm [2.36"]	
	Z2 axis	205 mm [8.07"]	
	B1 axis	135 deg (+90 to -45 deg)	
29	Bar length	2,500-4,000 mm [98.43-157.48"]	
30	Center height	1,050 mm [41.34"]	

No.	Item	L20 Type XII (L20E-2M12)	Remarks
31	Motors For spindle drive	2.2/3.7 kW (Continuous/15 min rating)	The control method is as follows: Built-in spindle motor
	Rotary tool for cross machining	1.0 kW	AC servo motor
	For back spindle drive	0.75/1.5 kW (Continuous/15 min rating)	Built-in spindle motor
	For back machining rotary tool	0.75 kW	AC servo motor
	For front machining rotary tool	0.75 kW	AC servo motor
	X1 axis	1.0 kW	AC servo motor
	Y1 axis	0.5 kW	AC servo motor
	Z1 axis	0.5 kW	AC servo motor
	X2 axis	0.5 kW	AC servo motor
	Y2 axis	0.75 kW	AC servo motor
	Z2 axis	0.5 kW	AC servo motor
	B1 axis	0.4 kW	AC servo motor
	For coolant oil	0.4 kW	
	For lubrication oil	0.003 kW	
32	Rated power consumption capacity	7.3 kVA	
33	Coolant tank capacity	165 liters	
34	Required floor space	1,220 mm [48.03"] (Depth) 2,120 mm [83.46"] (Width) 1,835 mm [72.24"] (Height)	
	Pneumatic device Pressure Consumption	0.5 MPa 53 (Power On), 59 (stationary), 210 (during air blow) Nℓ/min	Clean air should be supplied.
35	Noise level	75.2 dB[A]	Equivalent continuous sound level A at work place
36	Weight	2,400 kg [5.3 klb]	

3.3.2 Standard NC specifications

No.	Item	L20 Type XII (L20E-2M12)	Remarks
1	NC device	CINCOM SYSTEM M70LPC-VU	NC unit dedicated to CINCOM L20E / L20X series.
2	Display device	8.4 inch Color (LCD) Liquid Crystal Display	
3	Displayed language	English, Japanese, Chinese (traditional), German, French, Italian, Spanish, Russian, Polish, Czech (Portuguese)	( ): Option
4	Spindle	S1, S2, S3, (S4), S5	S1: Spindle S2: Back spindle S3: Rotary tool spindle on gang tool post (S4): Front machining rotary tool spindle S5: Back machining rotary tool spindle ( ): Option
5	Control axis (command axis)	X1, Y1, Z1, X2, Y2, Z2, (C1), (C2) , B1 axis	The machine is capable of simultaneous control of all axes. X1: Gang tool post Y1: Gang tool post Z1: Headstock X2: Back spindle headstock Y2: Back spindle headstock Z2: Back spindle headstock (C1): Main spindle C axis (C2): Back spindle C axis ( ): Option
6	Control axis groups	2	Auxiliary axis control group is enabled automatically as required.
7	Input code	ISO	
8	Command input system	Incremental or Absolute	
9	Feed command system	Feed per revolution or feed per minute (G code conversion)	
10	Linear interpolation	G00 (Standard function)	
11	G00 feed rate command	G00,F (Standard function)	
12	Dwell	G04 (Standard function)	
13	Workpiece coordinate system command	G50 (Standard function)	
14	Machine coordinate system command	G53 (Standard function)	
15	Call sub-program	M98 (Standard function)	
16	Override function Rapid feed Cutting feed	A rotary switch is used to set the override ratio. Range: 1, 3, 5, 10, 20, ….. 100% (Standard function)	The setting range for cutting feed can be expanded to the maximum of 200% by switching the setting of the software setting switch.
17	Zero point return function	Manual zero point return system (Standard function)	This machine has an absolute position detection function. Therefore, you normally do not have to perform the manual zero point return.
18	Automatic operation function	(Standard function)	This function enables the selected program to run. Continuous, 1 cycle, or 1 block can be selected.

No.	Item	L20 Type XII (L20E-2M12)	Remarks
19	On-machine program check function	Manual pulse generator rotation system (Standard function)	When a program is checked using the machine, the execution speed of the program is proportional to the rotation speed of the manual pulse generator. The program can be run backward by rotating the pulse generator in the minus direction.
20	Manual feed function	Available for all the axes (Standard function)	This function enables the operator to move all the control axes by turning the handle.
21	Manual data input (MDI) function	(Standard function)	This function enables the MDI input and execution of programs.
22	Self-diagnostic function	(Standard function)	This function automatically detects machine alarms and displays alarm messages.
23	Machine status display	(Standard function)	This function displays the machine status.
24	Backup function	(Standard function)	This function saves NC data such as parameter settings in memory.
25	Memory protection function	(Standard function)	This function protects the programs, offset data and parameter data.
26	Operating time display	(Standard function)	This function displays the machine's operating time, 1-cycle time, and actual cutting time in 1-cycle operation.
27	Cumulative power-ON hours	(Standard function)	The total of NC unit power-on hours is displayed.
28	Product counter indication	Max. 8 digit (Standard function)	The machine enters the 1-cycle stop state when the product counter reaches the specified number of products.
29	Machine operation information display	(Standard function)	This function displays the machine operation status in graph. The information includes sums of running time and alarm stop time, and ratio of cutting/no-cutting time.
30	Cycle time check function	(Standard function)	This function checks the machine for abnormal state by managing cycle time. If the cycle time exceeds 30 minutes, the system stops the machine assuming the occurrence of an alarm.
31	Automatic back-light off function	(Standard function)	This function automatically turns off the back-light when none of the keys or switches on the operation panel is operated for 10 minutes.
32	Eco display	(Standard function)	This function displays power consumption, idling stop state, etc.
33	Door open sensor function	(Standard function)	This function detects the door open status.
34	Door lock function	(Standard function)	This function locks the door while the machine is operating.
35	Enable switch	(Option)	
36	Alarm history display function	(Standard function)	This function displays the past 100 alarms.
37	Manual start/stop spindle function	(Standard function)	This function allows starting and stopping of the main spindle using the spindle operation keys.

No.	Item	L20	Remarks
		Type XII (L20E-2M12)	
38	Preparation functions		The functions below are provided to support preparation for automatic operation.
	Automatic return to the retract point	(Standard function)	All the axes automatically return to their retract points (fixed point) in the defined order.
	Automatic return to the positioning point	(Standard function)	Each axis automatically returns to the positioning point (fixed point, or point specified in the machining data).
	Automatic return to the start point	(Standard function)	Each axis automatically returns to the start point of automatic operation according to the numeric value specified in the machining data that can be set for each workpiece.
	Automatic cut-off machining function	(Standard function)	This function automatically performs cut-off machining (short cut).
	In-machine tool set function	(Standard function)	This function supports tool setting.
	Phase adjustment function	(Standard function)	Executes phase adjustment between the spindle and GB, the spindle and the back spindle, and the spindle C axis and the back spindle C axis.To be used together with the spindle synchronization control function and the C axis function.
	Non-conformed material phase adjustment function	(Option)	This function executes phase adjustment between the non-conformed material and main spindle C axis.
	Guide bushing adjustment function	(Standard function)	This function supports clearance adjustment of guide bushing.
39	Automatic power-off function	(Standard function)	This function shuts down the main breaker to turn off the power when an alarm occurs during automatic continuous operation.
40	Three-dimensional interference check function	(Standard function)	This function monitors interference with machine components and stops the machine if there is a risk of interference.
41	Collision detection function	(Standard function)	Detects collision of the machine during rapid feed (G0) operation to minimize damage to the machine.
42	Tool offset pairs	40 (Standard function) 80 (Option)	
43	Simultaneous 4-spindle speed specification	(Standard function)	Up to four spindle speed commands can be specified simultaneously.
44	Simultaneous 4-M specification	(Standard function)	Up to four M commands can be specified simultaneously for each block. There is a limit to M codes that can be specified simultaneously.
45	Double axis move command output	(Standard function)	When a moving axis is moving or it has reached the specified point, a new move command can be issued for another axis to move.
46	Auxiliary command output during axis move command execution	(Standard function)	When a moving axis is moving or it has reached the specified point, an M, S, or T command can be issued.
47	End-point specification queuing	(Standard function)	When a specified axis reaches the specified point, another axis can be moved to the end point at the same time.

No.	Item	L20 Type XII (L20E-2M12)	Remarks
48	Control axis change function	(Standard function)	All the axis control groups are permitted to specify an arbitrary axis or execute an auxiliary command.
49	Control axes superimposition function	(Standard function)	Simultaneous inner-outer diameter machining and pick-off operation can be performed by superimposing the Z2 axis on the Z1 axis.
50	Queuing between axis control groups	(Standard function)	
51	Axis overlap function	(Standard function)	Circular operation when executing a T code in thread cutting canned cycle (G92)
52	Macros dedicated to the Cincom L20 series	(Standard function)	The L20E is equipped with dedicated macros (e.g., T code macros).
53	Background editing	(Standard function)	While a program is running, another program can be edited.
54	Simultaneous 2-line program editing	(Standard function)	The programs of two axis control groups can be simultaneously edited on a single screen.
55	Editing support functions		The functions below are provided to support program editing.
	Calculator function	(Standard function)	This function calculates values for various operations.
	Code list display Code insertion	(Standard function)	This function displays the lists of available M codes and G codes. The function also allows entry of G and M codes on the List screen.
	Coordinates calculation function	(Standard function)	Coordinate values can be obtained by simply entering specified parameters.
	Format check function	(Standard function)	This function checks T, M and G code numbers and arguments before executing a program.
	Alarm block display function	(Standard function)	This function displays the block where an alarm occurred during program execution.
56	Program storage capacity	(Standard function) Program storage capacity Equivalent to 40 m tape [About 16 KB]	Each tape length at left includes the size of machining data. Each program work area capacity indicates the maximum size of the program that is actually run. If a subprogram is used, the subprogram size must also be added. To increase the work area capacity, add a program work area as an option.
		(Option) Program storage capacity Equivalent to 80 m tape [About 32 KB]	
		(Option) Program storage capacity Equivalent to 160 m tape [About 64 KB]	
		(Option) Program storage capacity Equivalent to 320 m tape [About 128 KB]	
		(Option) Program storage capacity Equivalent to 600 m tape [About 240 KB]	

No.	Item	L20 Type XII (L20E-2M12)	Remarks
57	Input/output interface	Compact flash card slot (Standard component)	A compact flash card can be used.
		RS232C connector (Standard component)	Various input/output devices can be used.
		USB slot (Standard component)	This function allows input/output of machining program from/to USB flash drive.
58	Optional block skip	(Standard function)	This function enables or disables the block including a slash ("/") in the program.
59	Optional block skip 9 sets	(Option)	This function enables or disables the block including a slash ("/") in the program. 9 sets
60	Optional stop	(Standard function)	This function stops (pauses) automatic operation at a block specified by the M01 command.
61	Chamfering ON/OFF	(Standard function)	This function allows the setting for whether chamfering is executed or not by specifying an M code (M204/M205).
62	Exact stop check function	(Standard function)	This function allows the specification for exact stop by the G code (G09).
63	Error detect function	(Standard function)	This function allows the setting for whether error detect is executed or not by specifying the M code (M92/M93).
64	Front spindle speed fluctuation detection function	(Standard function)	This function stops the machine if it detects that the difference between the specified and actual spindle speeds exceeds the preset fluctuation ratio. The function is effectively used to prevent overload during operation.
65	Back spindle speed fluctuation detection function	(Standard function)	This function stops the machine if it detects that the difference between the specified and actual spindle speeds exceeds the preset fluctuation ratio. The function is effectively used to prevent overload during operation.
66	Spindle indexing function Main spindle	1° (Standard function)	This function indexes the spindle at 1° intervals and positions the spindle by the holding force of the spindle motor without using any mechanical lock.
	Back spindle	1° (Option)	
67	Spindle C-axis function Main spindle	0.001° (Option)	This function controls the profile 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	0.001° (Option)	
68	Constant surface speed control Main spindle	(Standard function)	This function automatically controls the spindle speed for the tool position so that the workpiece surface speed becomes constant during the cutting process.
	Back spindle	(Option)	
69	Chasing function Main spindle	(Standard function)	Enables mm/rev feeding and thread cutting with a tool.
	Back spindle	(Option)	

No.	Item	L20 Type XII (L20E-2M12)	Remarks
70	Simplified cut-off tool breakage detection	(Standard function)	This function gives a speed command to the front spindle when the back spindle has picked off a workpiece. The function checks if the back spindle rotates together with the first spindle and determines if the cut-off tool is broken.
71	Corner chamfering/rounding function	(Standard function)	This function simplifies the specification of corner chamfering and corner rounding by using the "C" and "R" commands.
72	Tool nose radius 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 and the tool offset in the memory.
73	Arc radius specification	(Standard function)	This function simplifies arc machining by using the "R" (radius) command.
74	Continuous threading	(Standard function)	Continuous thread cutting can be performed by executing thread cutting commands (G32) consecutively.
75	Thread cutting canned cycle	(Standard function)	Canned cycle prepared for thread cutting
76	Multi-thread screw thread cutting cycle	(Standard function)	Thread cutting start position can be specified by an angle (Q argument).
77	Variable lead thread cutting cycle	(Option)	Lead increase/decrease amount per turn of a thread can be specified.
78	Circular threading function	(Option)	This function enables circular thread cutting.
79	Front-spindle synchronous tapping function	(Option)	Synchronized tapping can be performed with the front spindle.
80	Back-spindle synchronous tapping function	(Option)	Synchronized tapping can be performed with the back spindle.
81	Tool spindle synchronous tapping function	(Option)	Synchronized tapping can be performed with the tool spindle.
82	High-speed synchronous tapping function	(Option)	Synchronized tapping can be performed at a high speed.
83	Synchronous tapping phase adjustment function	(Option)	This function adjusts phase in synchronous tapping operation.
84	Spindle synchronization control function	(Option)	This function synchronizes the front spindle with the back spindle. The function is useful for picking off a workpiece.
85	User macro	(Option)	User macros enable the use of macro programs.
86	User macro G code call	(Option)	This function allows registration of user macro G codes.
87	Multiple repetitive cycles for turning	(Standard function)	This function enables the use of several types of canned cycles.
88	Canned drilling cycle	(Option)	This function enables use of canned cycles such as deep hole drilling cycles and boring cycles.
89	Differential rotary tool function	(Option)	This function enables drilling and tapping by using the difference between two spindle speeds. The function helps to decrease the cycle time.
90	Milling interpolation function	(Option)	This function enables contouring control in the workpiece end face direction using the linear and rotary (C-axis) axes.
91	Sub-micron command	(Option)	This command specifies the least input increment with 0.0001 mm.

No.	Item	L20	Remarks
		Type XII (L20E-2M12)	
92	Tool life management I	(Option)	This function stops the machine when a tool has reached its life, and reports the tool number.
93	Tool life management II	(Option)	When a tool has reached its life, this function automatically selects a spare tool.
94	Helical interpolation function	(Option)	This function enables helical interpolation with a tool spindle device.
95	Inclined helical interpolation function	(Option)	This function enables inclined helical interpolation with a tool spindle device.
96	Coornidate rotation command function	(Standard)	This function enables easier programming for inclined rotary tool.
97	Polygon machining function	(Option)	This function enables polygon machining by syncronizing tool spindle device and spindle rotation.
98	Hobbing function	(Option)	This function enables hobbing the spur gear, helical gear by syncronizing tool spindle device and C axis rotation.
99	Quick program check function	(Standard function)	Without running the machine, this function checks for program errors while running the machining program at a high speed. The function also measures approximate cycle time.
100	Line angle command function	(Standard function)	This function automatically calculates the coordinates of the end position by specifying the angle of the line.
101	Geometric command function	(Option)	When it is difficult to obtain the intersection point of two lines in execution of consecutive linear interpolation commands, an angle can be specified instead of the intersection point by using this function.
102	B code I/F function	(Option)	
103	Run using external memory program	(Option)	This function allows the use of subprograms stored in the CF card set in the CF card slot.
104	Network Input/output function	(Option)	This function enables input/output of programs and other data via the network.
105	Remote control function	(Option)	This function allows you to monitor or manipulate the screen of the machine from the remote PC.
106	alkartshop	(Standard function)	You can get a software options through Internet.
107	Back machining program skip function	(Option)	This function skips machining program steps between M239 and N19999 if no workpiece is chucked by back spindle.
108	Thermal Displacement Correction Function	(Standard function)	
109	C axis command during spindle synchronization	(Standard function)	This function allows specification of C axis command during spindle synchronization mode.
110	B-axis control function	(Standard function)	

3.3.3 Attachment specifications

No.	Item	L20 Type XII (L20E-2M12)	Remarks
1	Front-spindle chucking device	A320N U920Z (FC034-M) (Standard component) A320N U930Z (FC071-M) (Option) (25 mm in diameter specification)	Collet chucking device installed on the front spindle.
	Chuck sleeve for non-conformed material	A320N U620Z (FC917-M) (FC918-M) (Option) A320N U630Z (FC922-M) (FC923-M) (Option) (25 mm in diameter specification)	Main spindle chuck sleeve for machining square bars and hexagonal bars.
2	Rotary guide bushing drive unit	(Standard component)	This device is used to synchronize the rotary guide bushing device with the spindle, and driven by a belt and spline shaft.
3	Stationary guide bushing device	A320N U120Z (WFG206-M) (Option)	This device supports the guide bushing in stationary state, so it is useful for the high-precision machining of relatively small-diameter materials.
	Synchronous rotary guide bushing device	A320N U220Z (WFG206-M) (Option) A320N U230Z (WFG521-M) (Option) (25 mm in diameter specification)	Equipped with the belt and the spline shaft, this device supports the guide bushing while the guide bushing and spindle are being rotated synchronously. A super-high-precision, high-speed ball bearing is used with the device for performing turning precisely.
4	Drive unit for rotary tool spindle	L220E U33B (Standard component)	This unit drives a tool spindle which is mounted on the X1 axis slide to perform drilling or key grooving on the outer circumference of a workpiece.
5	Back spindle device	L220E U41B (Standard component)	This device enables the machine to simultaneously perform front machining, and machining similar to front machining on the cut-off face (back) of a workpiece. The product knock-out device is also installed as a standard component.
6	Back-spindle chucking device	A320N U920Z (Standard component) A320N U930Z (FC071-M) (Option) (25 mm in diameter specification)	Collet chucking device installed on the back spindle.
	Back chuck sleeve for non-conformed material	A320N U620Z (FC917-M) (FC-918-M) (Option) A320N U630Z (FC922-M) (FC923-M) (Option) (25 mm in diameter specification)	Back spindle chuck sleeve for machining square bars and hexagonal bars.

No.	Item	L20 Type XII (L20E-2M12)	Remarks
7	Back knock-out device	L220E U51B (Standard component)	This device is driven by air cylinder. Air blower for cleaning inside the back spindle chuck
8	Front 6 spindle holder	L20E U125B 6 sleeves (Standard component)	Holder for drilling a hole in the end face. The sleeve diameter is ∅19.05 mm (∅0.75").
		L20E U126B 6 sleeves (Option)	Holder for drilling a deep hole in the end face (up to three deep holes can be drilled).
9	Front rotary tool drive unit	L220E U128B (Driving of 3 tools) 3 sleeves (Option)	Rotary tools mounted in he end-face direction, which drill an eccentric hole in the end face of a workpiece.
10	Back rotary tool drive unit	L220E U155B (Driving of 4 tools) 4 sleeves (Standard component)	Rotary tools for back machining, which drill an eccentric hole in the back side of a workpiece. A stationary sleeve having a diameter of ∅19.05 mm (∅0.75") can also be used by using an adapter.
11	Knock-out jig for through hole workpiece	A320N U550B (Option) A320N U552B (Option) (25 mm in diameter specification)	Device to prevent chips from going into the back spindle during machining of through-hole workpieces. Workpieces are unloaded onto the front of the back spindle.
12	Workpiece separator	L220E U31J (Standard component) Maximum product length: 80mm [3.15"]	This device collects the machined workpieces in the product receiver box.
	Workpiece separator (collection by back basket)	L220E U352J (Option) Maximum product length: 45mm [1.77"]	Collects workpieces that cannot be picked-off in the basket mounted on the back spindle.
13	Long workpiece device	A320N U421B L20E U4206B L220E U4212B(ø12), U4216B(ø16), U4220B(ø20) A320N U922B, U932B (Option)	Use this device to machine a long workpiece which turns out a product longer than 80 mm up to 600 mm (~ø20). The device is equipped with a support pipe and unloads workpieces from the left side of the machine.
	GBL-specific component	A320N U60C (Standard compnent)	Spindle cap used for the GBL-machine.
	Support pipe	A320N U10C A320N U100C (blank) U106C(ø6) to U123C(ø23) (Option)	With the GBL-machine, these pipes are mounted in the spindle, and they suppress runout of workpiece while it is rotating.
14	Workpiece conveyor	A320N U37J (Option)	This conveyor unloads workpieces (collected by the workpiece separator) from the left side of the machine. (With scar prevention)
15	Chip conveyor	A320N U90J (Option)	This conveyor sends out chips from the machine.
16	Cut-off tool breakage detector	A320N Y90Z (Standard component)	This detector checks if a workpiece is cut off normally after completion of cut-off machining. If a workpiece remains due to a cut-off tool breakage, the system stops automatic machine operation.

No.	Item	L20 Type XII (L20E-2M12)	Remarks
17	Patrol lamp indication	L220E U80Z (Option)	The patrol lamp of which indication is linked with the alarm display on the machine's operation panel can be mounted on the top of the machine.
	3-color patrol lamp	L220E U81Z (Option)	The 3-color (green, yellow, and red) patrol lamp can be mounted on the top of the machine. Green: The machine is in continuous operation mode. Yellow: The machine is in cycle-stop state. Red: An alarm has occurred.
18	Automatic fire extinguisher	L220E U99Z (Standard component only in Japan)	Provided in case of a fire.
19	Coolant supply unit Pump type Cartridge-type tank capacity	A320N U10R (Standard component) 0.4 kW Dipping type 165 liters	The coolant level detection function is provided as standard.
20	Air supply system for air seal function	L20E U71N (Standard component)	Pressurized Air Seal system for Guide bush and front/back Spindle.
21	Coolant flow rate detector	A320N U53R (Option)	This detector monitors the discharge from the coolant nozzle. The system stops automatic machine operation when the flow rate becomes lower than the preset value.
22	Medium-pressure coolant device (4 lines)	A220 U75R L220E U743R (Option)	This device provides medium pressure coolant to the back spindle and the guide bushing device.
23	Oil air lubrication unit	L220E U82R L220E U821R (Option)	This device supplies oil to the gears in the gang tool post rotary tool drive unit.
24	Lubricating oil supply unit Lubricating oil tank capacity Discharge	(Standard component) 0.8 liter 2.5 cc/60 min	The lubricating oil level detection function is provided as standard.
25	Door lock	A320N Y76Z (Standard component)	This device locks the machine doors while the machine is running, and unlocks them only while the machine is stopped.
26	Ground fault circuit interrupter	Rated current 40A (Standard component)	Rated sensitivity current: 30 mA

3.3.4 Environmental information

Basic information

No.	Item	L20 Type XII (L20E-2M12)	Remarks
1	Energy usage		
	Power supply voltage	200 VAC	
	Rated power consumption	7.3 kVA	
	Required pneumatic pressure	0.5 MPa	

Environmental performance information

No.	Item	L20X Type XII (L20E-2M12)	Remarks
1	Power consumption		
	Standby power *1	0.300kW	
	Power consumption with model workpiece *2	0.0113kWh/ cycle	
	Power consumption value above converted to a CO_2 value *3	5.4g/cycle	
2	Air consumption		
	Required air flow rate	53 (Power On), 59 (stationary), 210 (during air blow) Nℓ/min	
3	Lubricant consumption		
	At power ON	2.5 cc/60 min	
4	Noise level		
	Value measured based on JIS	75.2 dB[A]	
5	Environmental load reduction		
	RoHS Directive/ REACH regulations	Compliant	

Approach to environmental issues

No.	Item	L20X Type XII (L20E-2M12)	Remarks
1	Recycling Indication of the material names of plastic parts	Covered in the Parts List (Separate Volume) *4	
2	Environmental management	• We are ISO14001 accredited. • We pursue "Green Procurement", whereby we make our purchases while prioritizing goods and services that show consideration for the environment.	

*1 This is the standby power in the idle stop mode (a function that turns servomotor excitation off when it is not necessary, for example during program editing).

*2 This is the power consumption in program operation (when not cutting) for one of our standard test pieces, shown for the purpose of comparing the environmental performance with that of existing models.

*3 This is the value converted in accordance with the CHUBU Electric Power CO_2 emissions coefficient for 2009 as published by the Ministry of the Environment.

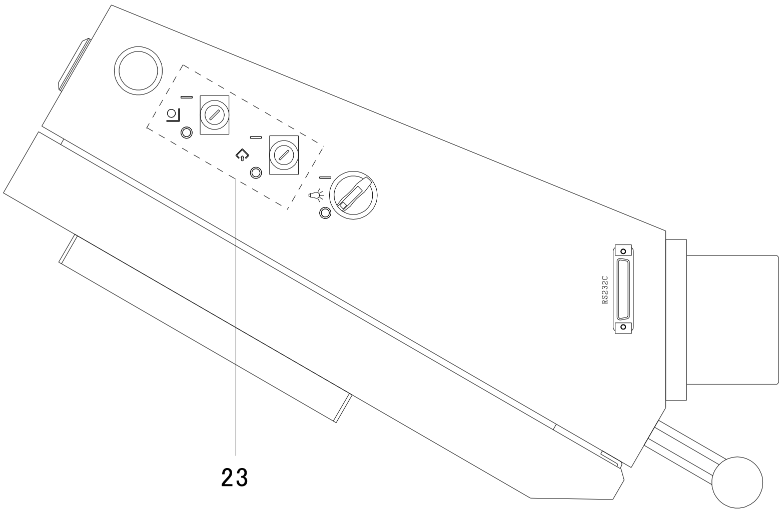
*4 If polyvinyl chloride (PVC) and fluoric resin are not processed correctly they can generate harmful gases. When recycling these materials, commission a contractor that is capable of processing them appropriately.

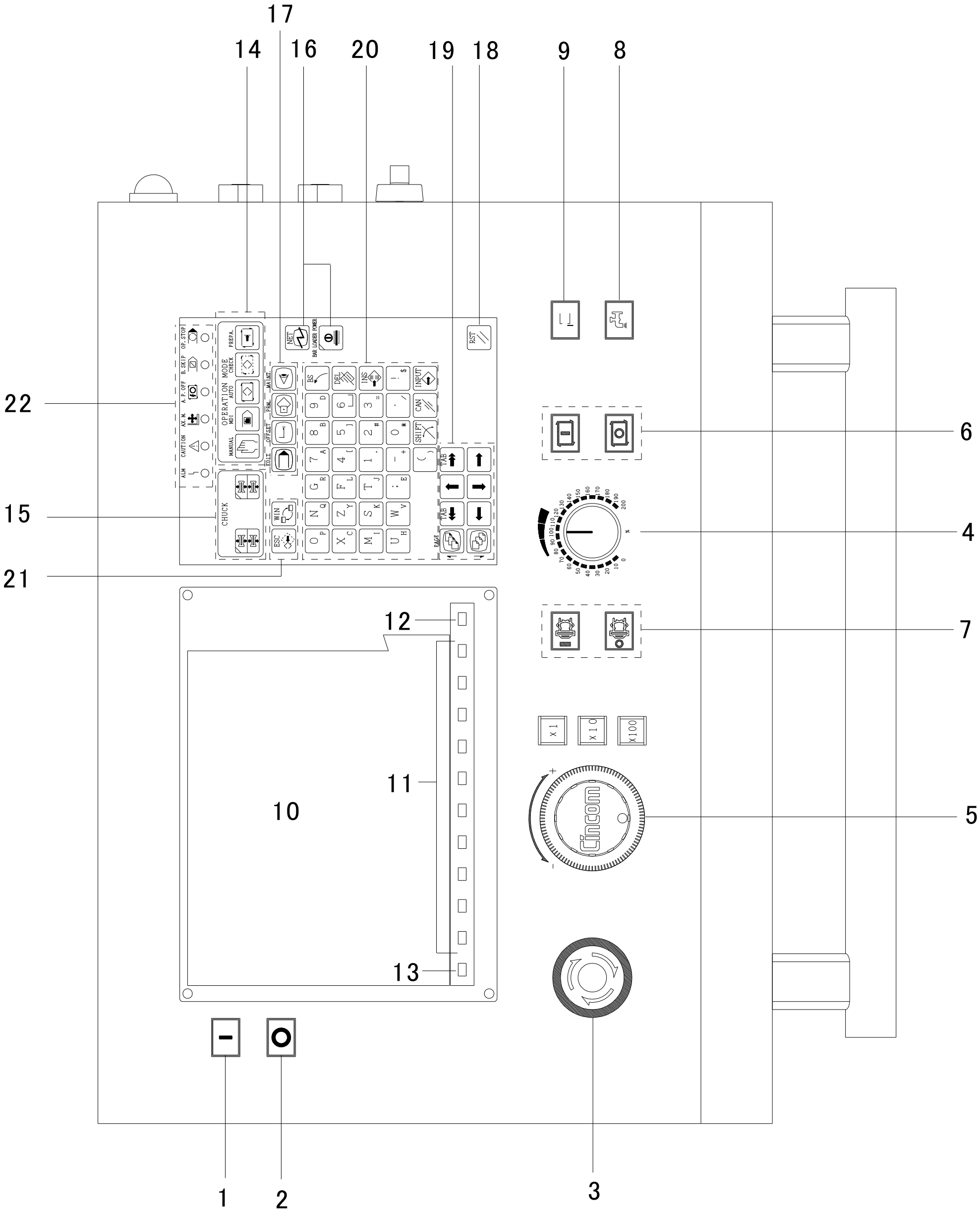
(Blank Page)

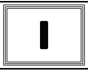
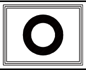
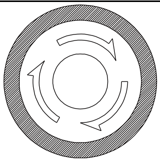
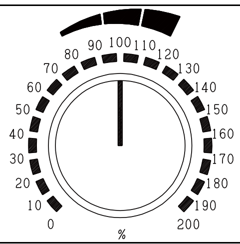
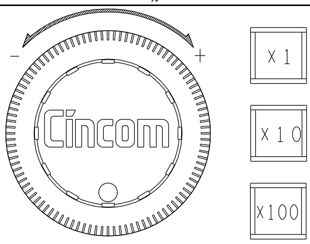
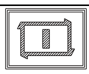
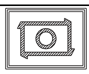
Product Code	C-L220E XII
Document Code	1E1-0300
Mfg. No.	L220E/00001
Issue Date	2013.10

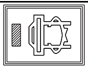
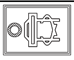
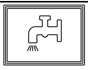
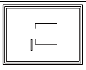
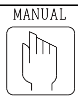
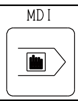
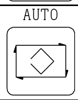
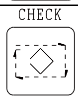
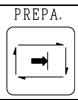
4. Operation Panel and Screen

4.1 Names and Functions of Operation Panel Components

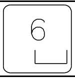
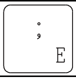
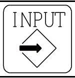
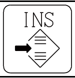
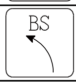
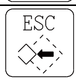
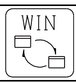




No.	Name	Symbol	Color of lamp	Function
1	Power ON button		–	Turns on the power of the NC.
2	Power OFF button		–	Turns off the power of the NC.
3	Emergency stop button		–	Firmly press this button in case of an emergency during machine operation to disconnect power from the machine and stop all machine motions. To reset, turn the button toward the direction shown by an arrow (clockwise). Then, turn OFF the NC power and the main circuit breaker, then turn them ON again.
4	Feed rate override		–	Changes feedrate at 0, 1, 3, 5 and 10 to 100% (10% increment in the range from 10 to 100%) of the feedrate specified in a program. The adjustable range can be expanded to 0 to 200% using the setting switch.
5	Handle magnification button		–	• In the manual operation mode, turning the handle clockwise or counterclockwise moves the X1, Y1, Z1, X2, and Z2 axes in the positive or negative direction, respectively. • In the program check mode, turning the handle clockwise or counterclockwise runs the program forward or backward, respectively. • In the Mechanism Adjustment mode, turning the handle clockwise or counterclockwise moves the X1, Y1, Z1, X2, and Z2 axes in the positive or negative direction, respectively. • The ×1, ×10, and ×100 multiplication factor keys select the handle feed speed in the manual operation mode or the program operation speed in the program check mode.
6 Automatic Operation	Start button		Green	Starts automatic operation. The lamp lights during automatic operation. The lamp blinks when the machine is being paused during preparation, waiting for cut-off operation or the doors are being locked.
	Hold button		Red	Pauses the automatic operation. The lamp lights if the automatic operation is paused.

No.	Name	Symbol	Color of lamp	Function
7 Spindle Stop	Spindle start button		Green	Allows the main, back and tool spindles, temporarily stopped by pressing the all spindle stop key during automatic operation or program check, to start at the previously set speed. (This operation is permitted only while the machine is in the hold state or block stop state with the door closed.) While the spindle is rotating the lamp in the key lights. The lamp blinks in the spindle re-start enabled state.
	All spindles stop button		Red	Regardless of whether the manual operation mode or automatic operation mode is selected, this key stops the main spindle, back spindle, and tool spindle by the manual operation.
8 Manual Operation	Coolant button		Orange	Turns on/off the coolant during manual and automatic operations. The lamp lights when the coolant is discharged.
9 Manual Operation	Door Open button		Orange	While the door is locked, pressing this button unlocks the door for 10 seconds, and the button lamp blinks. While the door is locked, the button lamp is turned off. While the door is open, the button lamp turns on.
10	Liquid crystal display (LCD)	–	–	Displays the menus, coordinates, programs, offset data, alarm, and diagnostics.
11	Menu Keys		–	Select desired screens or operations from the menu at the bottom of the LCD screen. Software keys located just below the menu.
12	Menu Up/Down selection key (Right)	▶	–	Switches between the upper and lower menus. Pressing this key with the upper menu enabled enables the lower menu instead. Pressing the key with the lower menu enabled enables the upper one instead.
13	Menu Up/Down selection key (Left)	◀	–	
14 Operation Modes	Manual key		–	The lamp lights in manual operation mode. In this mode, you can manually feed the X1, Y1, Z1, X2, and Z2 axes using the handle or manually operate the dedicated bar loader (CAV).
	MDI key		–	The lamp lights in MDI mode. Create a program by manual data input and execute the command.
	Auto key		–	The lamp lights in automatic operation mode in which the program loaded in memory can be executed.
	Program Check key		–	The lamp lights in program check mode in which the program can be checked while actually operating the machine.
	Preparation key		–	The lamp lights in the preparation mode. You can perform the preparations for operations such as the tool set and the start position operation.

No.	Name	Symbol	Color of lamp	Function
15 Manual Operation	Back Spindle chuck key		Orange	Opens or closes the back chuck. The lamp on this key lights when the back spindle chuck is closed. Press this key once to close the back spindle chuck. Press this key again to open the chuck. The key works only in manual operation or preparation mode.
	Main Spindle chuck key		Orange	Opens or closes the main spindle chuck. The lamp on this key lights when the main spindle chuck is closed. Press this key once to close the main spindle chuck. Press the key again to open the chuck. The key works only in manual operation or preparation mode.
16 Bar Loader	Power key	BAR LOADER POWER	Orange	Enables activation of the dedicated bar loader (CAV). The bar loader can be activated when the lamp on this key is on.
17 Screen Operation Functions	Edit key	EDIT	–	Executes program editing including creation, save, search, insertion, and deletion, etc. Also press this key to enter machining data.
	Offset key	OFFSET	–	Sets the offset data.
	Parameter key	PRM.	–	Sets the parameters.
	Diagnosis key	MAINT.	–	Displays diagnostics data, alarm history, and software version numbers.
	Alkartnet key	NET	–	Not used
18	Reset key	RST	–	Clears an NC alarm or machine alarm status.
19	Previous page key		–	When there is multi-page data on the screen, press the Previous page key to scroll back one page.
	Next Page key		–	When there is multi-page data on the screen, press the Next page key to scroll forward one page.
	Up arrow key		–	Moves the cursor up, line by line. If you hold down a cursor key, the cursor will continue to move.
	Down arrow key		–	Moves the cursor down, line by line. If you hold down a cursor key, the cursor will continue to move.
	Right arrow key		–	Moves the cursor right, line by line. If you hold down a cursor key, the cursor will continue to move.
	Left arrow key		–	Moves the cursor left, line by line. If you hold down a cursor key, the cursor will continue to move.
	Right tab key	TAB	–	Moves the cursor to the beginning of the next word in the program, or to the right setting field in parenthesis.
	Left tab key	TAB	–	Moves the cursor to the beginning of the previous word in the program, or to the left setting field in parenthesis.

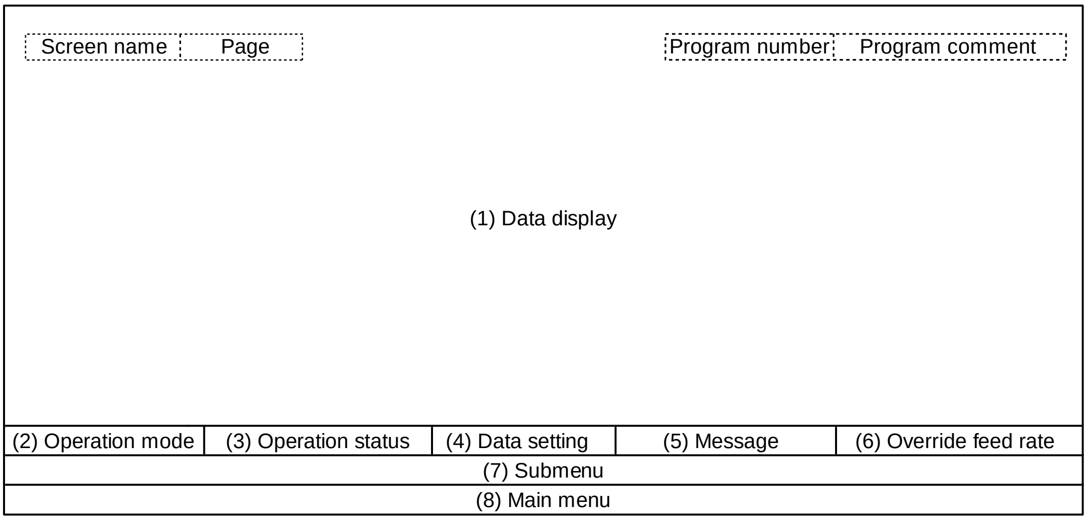
No.	Name	Symbol	Color of lamp	Function
20	Alphanumeric keys		–	Press the alphanumeric keys to enter letters, numbers, and special characters.
	Space key		–	Enters a blank space equivalent to one character.
	End of block key		–	Enters the end of the block symbol required at the end of each block (program line).
	Shift key		–	Press this key before pressing an alphanumeric key to enter the letter, symbol, or special character printed on the lower portion of the keycaps.
	Line deletion key		–	Deletes a specific block of data in a program (program line) or a setting data.
	Input key		–	Enters a editing program or a setting data in memory.
	Insert key		–	Toggles data input to the program between an insert mode and an overwrite mode.
	Delete key		–	Deletes next data pointed by cursor in a program or setting data.
	Backspace key		–	Deletes previous data pointed by cursor in a program or setting data.
21	Escape key		–	Cancels setting, reverting to the original state of the current setting. Pressing the ESC key with a window displayed on the screen closes that window. Pressing the key with two or more windows on the screen closes the topmost one.
	Window select key		–	Selects active window in multi-windows.

No.	Name	Symbol	Color of lamp	Function
22	Alarm lamp	ALM	Red	When a machine alarm occurs, this lamp will either blink or remain lit. If this lamp remains lit, turn off the machine once, remove the cause of the trouble, and then turn on the machine. If this lamp is blinking, remove the cause of the trouble and press the Reset key to clear the alarm status. If the lamp still remains blinking, turn off the power and shut down the main circuit breaker, remove the cause of the alarm, then turn it on again.
	Caution lamp	CAUTION	Orange	This lamp blinks when an error occurs. Remove the cause of the trouble to recover the machine. The lamp blinks in zero point return mode.
	Axis motion lamp	AX. M.	Green	This lamp lights while the axes are being moved. The lamp blinks during program reverse operation in program check mode.
	Automatic power off lamp	A. P. OFF	Orange	This function is enabled when the lamp lights. It disconnects power from the machine in the event of an alarm during continuous cycle operation.
	Skip lamp	B. SKIP	Orange	The block skip function is enabled when the lamp lights. The function disregards the commands specified in a block which is preceded by a slash (/) code.
	Optional stop lamp	OP. STOP	Orange	This function is enabled when the lamp lights. It stops automatic operation after executing a block using the M01 command in the machining program. When the program is being paused, the lamp on the Start key blinks. Pressing the Start key again restarts automatic operation.
23	Setting switch key	O I	–	If door is opened with this switch key set to "I" position, machine operation is disabled. If door is opened with this switch key set to "O" position, each axis moves at 2 m/min or slower, and opening/closing of chuck and charge/discharge of coolant are also available.
	Program edit switch key	O I	–	Set this key to "I" to turn ON the program protection. While the protection is ON, you can not edit or write a program. When this switch is set to "O", you can edit or write program.
	Enable switch	—	–	To operate the machine while the door is open, press this switch key, then press and hold Start button.

4.2 Fields and Functions of Liquid Crystal Display (LCD)

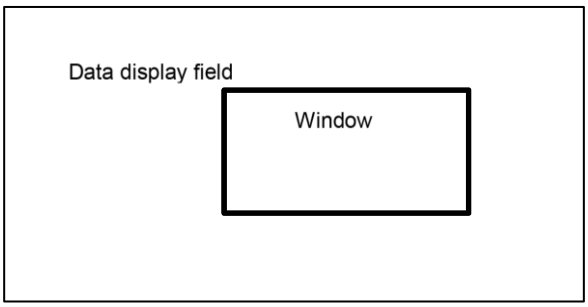
The LCD has eight fields or sections.

(1) Data display
(2) Operation mode display field
(3) Operation status field
(4) Data setting field
(5) Message display field
(6) Override feed rate display field
(7) Submenu display field
(8) Main menu display field



(1) Data display

The data display field displays data either in the entire field or with a window appearing for additional data display above the field.



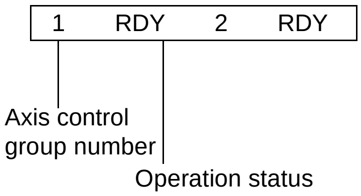
(2) Operation mode display field

This field displays the currently selected operation mode.
Operation mode display

Symbol	Description
MEM	Memory operation mode
HDL	Handle mode
MDI	MDI mode
AST	Zero return mode

(3) Operation status field

Preparation and operation statuses for each axis control group number are displayed.



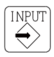
Operation status

Symbol	Description
EMG	Emergency stop
RST	Resetting the NC
RDY	Completed preparation
AUT	During automatic operation
SYN	Queuing for synchronization
CRS	Queuing for axis cross
BST	Stopping
HLD	Holding

Operation status display color and machine status

Display color	Machine status
Green	Normal
Red	Dangerous

(4) Data setting field

Set the data in the data setting filed.
After entering the data in this field, press the Input key 

and the data is set in the data display field.

(5) Message display field

This field displays error messages, machine operating status messages, operation guide messages, etc.

(6) Override feed rate display field

This field displays the override value for the feed rate, rapid feed rate or dwell.

[Note]

On some screens, this field is not available.

(7) Submenu display field

This field provides a submenu for selecting screens and operations.
The submenu is usually dimmed. Pressing the menu Up/Down selection key ▶ ◀ makes the submenu to appear in blue, indicating that it can be selected.
When the submenu is displayed in blue, pressing the menu key corresponding to a menu selects that menu.

Menu 1	Menu 2	Menu 3	Menu 4	Menu 5	Menu 6	Menu 7	Menu 8	Menu 9	Menu 10

(8) Main menu display field

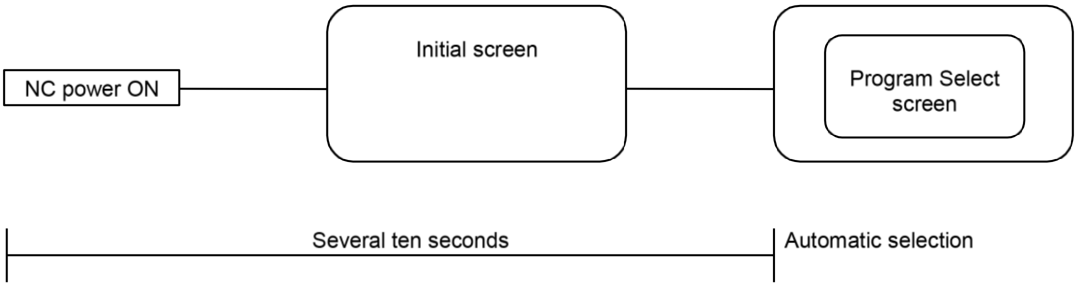
This field provides a main menu for selecting screens and operations.
The menu is usually displayed in blue.
When the menu is displayed in blue, pressing the menu key corresponding to a menu selects that menu.
This menu is dimmed and cannot be selected when the submenu is displayed in blue.
Pressing the menu Up/Down selection key ▶ ◀ then makes the menu to appear in blue, indicating that it can be selected.

Menu 1	Menu 2	Menu 3	Menu 4	Menu 5	Menu 6	Menu 7	Menu 8	Menu 9	Menu 10

4.3 Screen Transition

4.3.1 Screen transition when the machine is turned on

Turn the NC power on after turning the main circuit breaker on, and the initial screen is displayed. In several tens seconds the program selection screen in the Preparation mode is automatically displayed. Do not attempt to touch any key until the program selection screen is displayed.



[Note]

If none of the keys or switches on the operation panel are pressed for 10 minutes, the backlight of the LCD is automatically turned off for protection of the display. You can also turn on the backlight again by pressing either the operation mode selection keys screen operation keys, alphanumeric keys and arrow keys.

Product Code	C-L220E XII
Document Code	1E1-0400
Mfg. No.	L220E/0001~
Issue Date	2013.10

5. Preparation for Operating the Machine

5.1 Turning On/Off the Power

Turning On/Off the Power

1. Turn on the main circuit breaker.
2. Reset the emergency stop button if it is being pressed.
3. Press the Power ON button I on the operation panel.

   The safety relay is reset and the initial screen appears. In several tens seconds, the program selection screen in the Preparation mode appears.

[Note]

If the emergency stop button is pressed or if relays are melting and adhere together, the safety relays cannot be reset. If this happens, the alarm "EX052 Emergency button alarm. Emergency button is pressed." and "S01 0010 SERVO ALARM" will appear. Turn off the machine power and turn it on again.

To turn off the machine

1. Press the Power OFF button O on the operation panel.

   The screen is blank.
2. Turn the machine light off.
3. Turn off the main circuit breaker.

Emergency stop

1. Press the Emergency stop button.

Resetting emergency stop

1. After checking safety, turn the emergency stop button clockwise.
2. Press the Power OFF button and turn off the power.
3. Turn off the main circuit breaker.
4. Turn on the main circuit breaker.
5. Press the Power ON button and turn on the power.

5.2 Menu Keys

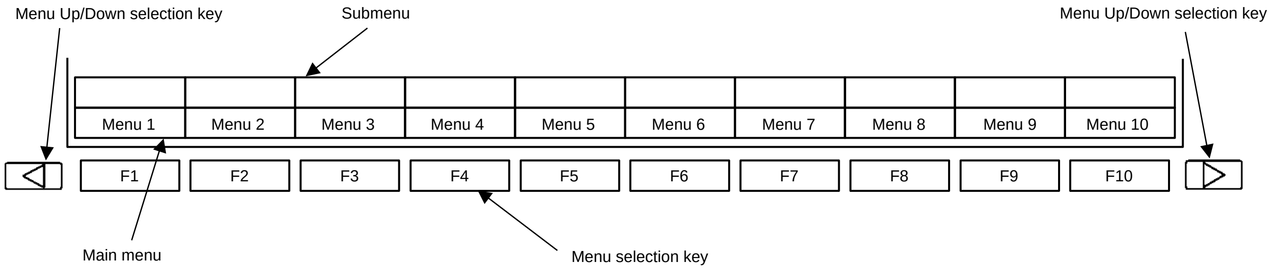
For efficiency in selecting the desired screen, screens with similar functions or related to a single operation are grouped together.

Press the key on the operation panel corresponding to the desired screen group, and then use the menu keys to select the desired screen from the displayed menu.

There are two on-screen menus: the main menu at the bottom of the screen and the submenu above the main menu.

5.2.1 Menu key operation

(Main menu displayed)



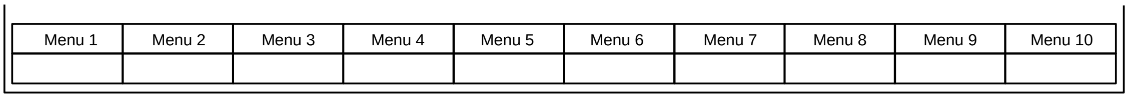
The two fields at the bottom of the screen are the menu display area. The lower menu is the main menu; the upper one the submenu.

Usually, the main menu is displayed in blue, allowing its menu selection keys to be selected.

The menu Up/Down selection keys are provided on the right and left sides of the menus. Pressing either menu Up/Down selection key makes the submenu be displayed in blue, allowing its menu selection keys to be selected.

The menu Up/Down selection key toggles menu selection between the main menu and submenu.

(Submenu displayed with menu Up/Down selection key pressed)



To select a screen, an operation, or another screen group, press the menu key (under the screen) corresponding to the desired menu right above it.

In the remainder of this manual, [Menu 1] to [Menu n] represent the menu selection keys for selecting menus 1 to n, respectively.

Each menu selection key is displayed like a button. It looks depressed when selected; it looks raised when not selected.

There are three types of menus. They are the screen selection menu which is used to change the screen, the operation selection menu which is used for editing and machine operation, and the screen groups selection menu-which is used to move to other screen groups.

5.2.2 Scrolling through menu pages

The screen can display only ten menus at a time.

Some screens, however, have more than ten menus. These screens require a menu selection menu [Menu SEL] to scroll through the menu pages. Press the corresponding menu key to scroll through the menu pages.

Menu 1	Menu 2	Menu 3	Menu 4	Menu 5	Menu 6	Menu 7	Menu 8	Menu 9	Menu SEL

Press the menu key (under the screen) corresponding to [Menu SEL]. The menu page is scrolled.

(Next menu page)

Menu 10	Menu 11	Menu 12	Menu 13	Menu 14	Menu 15	Menu 16	Menu 17	Menu 18	Menu SEL

Press the menu key corresponding to [Menu SEL]. If there is no menu page that follows, the menu display field returns to the previous menu page.

(Previous menu page)

Menu 1	Menu 2	Menu 3	Menu 4	Menu 5	Menu 6	Menu 7	Menu 8	Menu 9	Menu SEL

5.2.3 Selecting screen

Screens are classified into some groups to select desired screen among a hundred or more screens. The procedure to select and display desired screen is as follows.

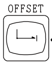
[Procedure]

1. Press the corresponding <screen operation function key> and <operation mode key> corresponding to the desired screen to select the screen group.

The last selected screen in the group appears.

2. Find the name of the desired screen from the main menu and the submenu and press the corresponding menu key to display the desired screen.

Example

1. Press the Offset key 

.

The screen in the same screen group is displayed.

Last selected screen (Tool data screen)

2. Press the menu key [Offset].

Desired Offset screen appears.

[Note]

The TOOL LIFE and SPARE TOOLS screens are available as options.

The tool life and spare tool screens are displayed only when the tool life management I option is purchased and the tool life management function is enabled on the CONTROL PARAMETERS screen.

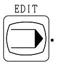
5.2.4 Selecting an operation

Select the screen to perform desired operation in the same manner as explained in <5.2.3 Selecting screen>.

Select the desired operation from the main menu or the submenu displayed (on the bottom of the screen) and then press the menu key corresponding to the desired operation.

Example

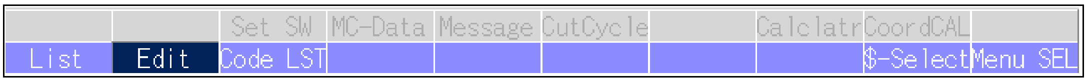
Example: Switching between the normal display and the simultaneous display on the edit screen.

1. Press the Edit key .

   The last screen selected appears.

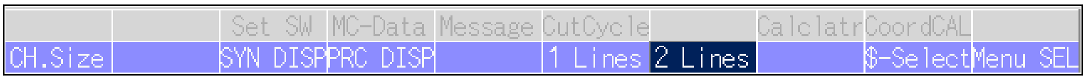
2. Press the menu key [Edit].

   The Edit screen appears.



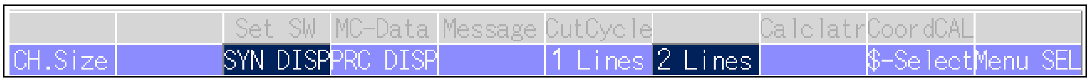
3. Press the menu key [Menu SEL].

   The menu display is changed.



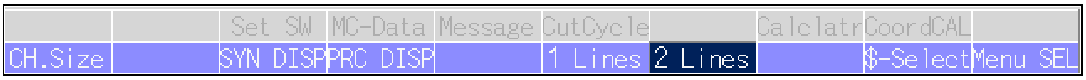
4. Press the menu key [SYN DISP].

   The screen changes to synchronous display.



5. Press the menu key [SYN DISP].

   The synchronous changes to normal display.



5.2.5 Selecting another menu group

Using the menu keys [Correct] and [Operat.] allows you to move from one screen group to another and then return to the previous screen.

The [Correct] is used to reinforce the editing function under the On-Machine Program Check mode. This screen group can be accessed from the On-Machine Check screen.

Example

Example: Use the procedure below to move to the Edit screen and then to return to the Program check screen

1. Press the menu key [Correct].

Op.Stop	POS Data	Set SW	MC-Data	Message		T-PATT	Offset	Counter	
Skip1		Act. Cut	HISPCHCK	Handle	1 Cycle	1 Block	Last PRT	Correct	Menu SEL

The On-Machine Check (Edit) screen appears. The menu keys change as shown below.

	POS Data	Set SW	MC-Data	Message	CutCycle	T-PATT	Calclatr	CoordCAL	
Find	Replace	Code LST	Range	Cut	Copy	Paste	$-Select	Operat.	

2. Press the menu key [Operat.].

	POS Data	Set SW	MC-Data	Message	CutCycle	T-PATT	Calclatr	CoordCAL	
Find	Replace	Code LST	Range	Cut	Copy	Paste	$-Select	Operat.	

The On-Machine Check screen appears. The menu keys change as shown below.

Op.Stop	POS Data	Set SW	MC-Data	Message		T-PATT	Offset	Counter	
Skip1		Act. Cut	HISPCHCK	Handle	1 Cycle	1 Block	Last PRT	Correct	Menu SEL

[Note]

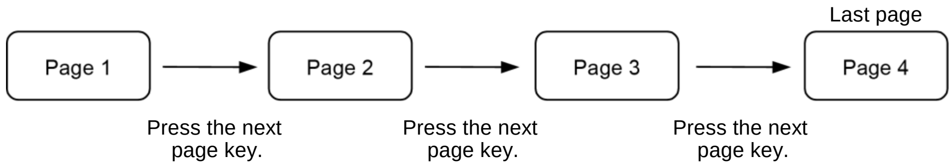
You can move to the Edit screen only while the program is running. Otherwise, the operating status will not change even if the menu key [Correct] is pressed.

5.3 Page Keys

If a screen has multiple pages, use the Page key or to move from page to page.

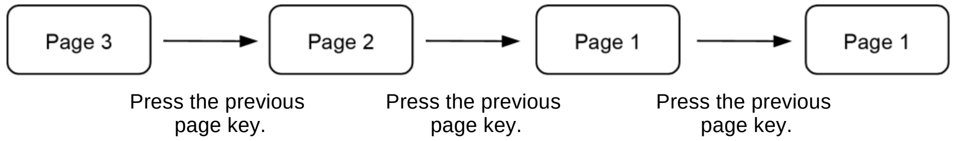
Next page key

Press the key to skip and display the next page of the screen. Nothing happens when you press this key at the last page of the screen.

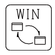


Previous page key

Press the key to skip and display the previous page of the screen. Nothing happens when you press this key at the first page of the screen.



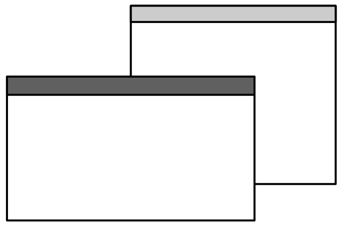
5.4 WIN Key

When you have multiple window screens displayed, pressing the Window select key 

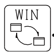
moves the focus from one window screen to another.

Moving the focus between window screens

This window screen has the focus.

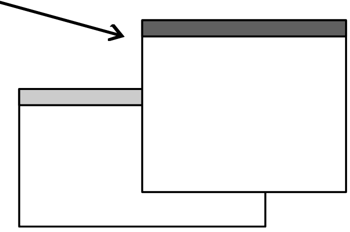


[Procedure]

1. Press the Window select key 

.

The focus moves to the another window.



5.5 Data Setting

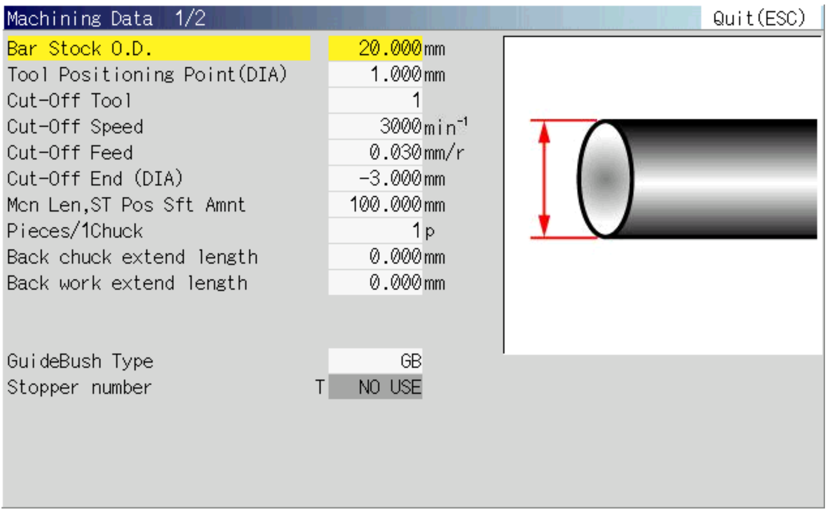
5.5.1 Data setting operation

When the screen where data input is possible is displayed, the data setting fields are enabled. The data setting fields are common to all screens. Move the cursor to the data setting field of the desired item using the arrow keys or the tab keys and then enter the data using the alphanumeric keys. Press the Input key and the entered data is accepted as the input to the field. To cancel the entered data, press the Escape key, an arrow key or a tab key before pressing the Input key.

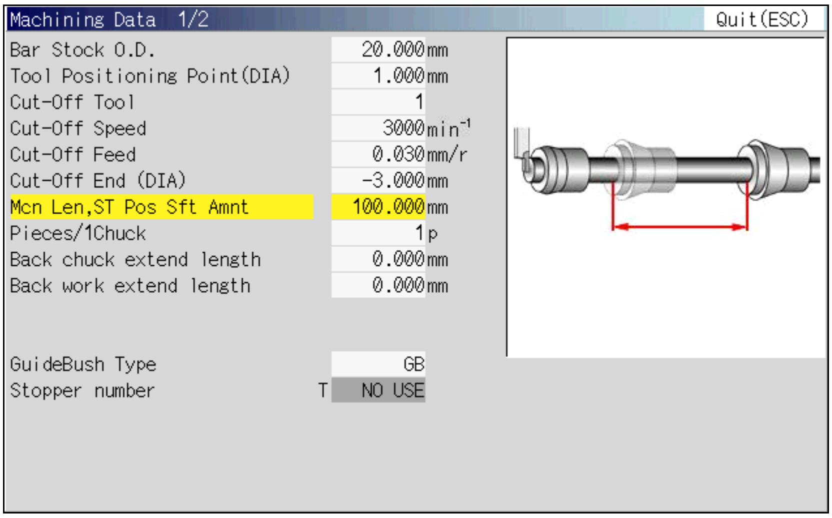
[Procedure]

Example: To change the Machining Length in the Machining Data to 15 mm

1. Select the Machining Data screen.

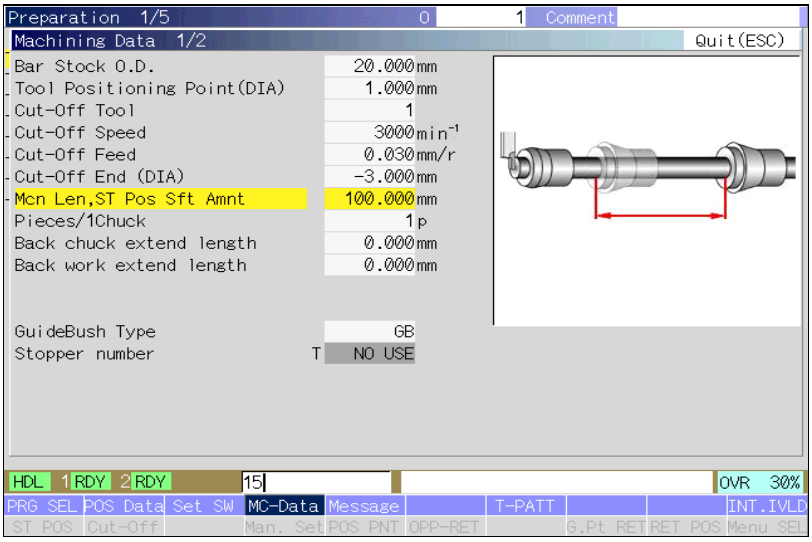


2. Press the arrow key to move the cursor to the setting field in which the machining length is set.



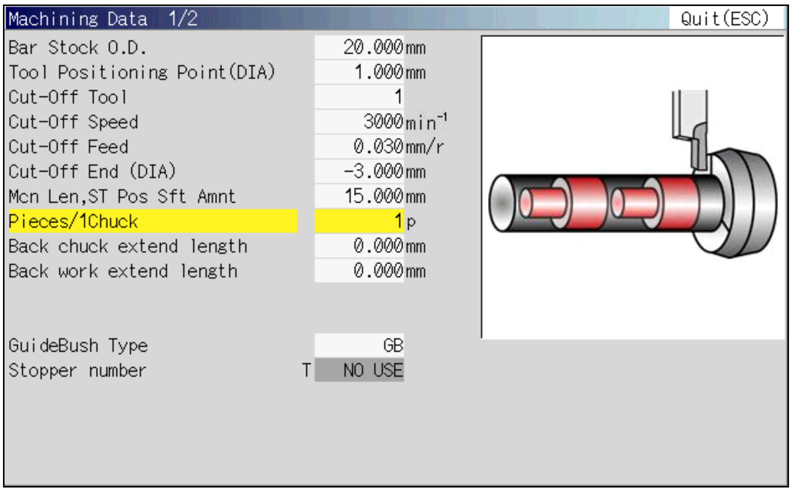
3. Press the number keys 1 and 5.

"15" is displayed.



4. Press the Input key INPUT.

The data is now set.
The cursor automatically moves to the next setting field.



5.5.2 Data in the data setting fields

The data input in the data setting fields can be edited using the procedure below.

Moving the cursor

[Procedure]

Press the arrow key ⬅ once to move the cursor key one character to the left.

1.234|
↓
1.23|4
↓
1.2|34

Press the arrow key ➡ once to move the cursor key one character to the right.

1.2|34
↓
1.23|4
↓
1.234|

Inserting characters

Character(s) can be inserted in front of the cursor.

[Procedure]

1. Move the cursor to the position at which data is to be inserted.

2. Press the alphanumeric key (5).

A "5" is inserted at the desired position and the next character can be entered.

3. Press the Input key INPUT.

The data is now set.

1.2|34
↓
1.25|34

Deleting characters

[Procedure]

a. Deleting the character at the cursor position

1. Move the cursor to the position at which data is to be deleted.
2. Press the Delete key DEL.

   One character is deleted and the space is left or right justified.

1.2|34
↓
1.2|4

b. Deleting the character to the left of the cursor position

1. Move the cursor to the character position that follows the character you want to delete.
2. Press the Backspace key BS.

   One character is deleted and the space is left or right justified.

1.2|34
↓
1.34

(Blank Page)

Product Code	C-L220E XII
Document Code	1E1-0500
Mfg. No.	L220E/0001 ~
Issue Date	2013.10

6. Operation for Running the Machine

The start, stop, and operation speeds of the machine are specified for safety as shown below.

	Door opened (The Setting switch key is set to "I")	Door closed
[Automatic operation] [Program check] Programmed operation [MDI operation] Knock-out advance/return [Preparation] Chucking operation Cutting during cut-off machining Non-conformed material phase adjustment	Activation disabled	Command-specified speed (Override enabled)
[Preparation] Return position Positioning point Start position Manual tool setting Gang tool return Opposite return Tool selection for cut-off machining Material setting Adjustment work [Manual operation] Handle feed	Operation at a speed of 2 m/min or less (Override enabled)	Operation at a speed of 10 m/min or less (Override enabled) In manual tool setting, operation speed is 2 m/min or less (Operation speed increases to 10 m/min when "Toolset Feed UP" on the Set SW screen is enabled.)
[Miscellaneous] Chuck open/close operation with chuck open/close key	Operation at 100% speed	Operation at 100% speed

The machine has door locks on the splash guard and spindle cover sections.

[Note]

- If the Setting switch key is set in the "O" position, machine operation is disabled while the door is open.
- With the machine equipped with Enable switch, press the Enable switch before performing Preparation operation while the Setting switch key is set to "I" position.

6.1 Selecting a Program

Selecting a program means the procedure to select a program to be executed from the stored programs. When you select a program for execution, the machining data associated with the selected program is also called up automatically.

Manual tool setting is to adjust the tool bit position of each tool in the holder mounted on the machine to each workpiece coordinate zero point. The mounted holder is set as tool post of machining data items for the program to be run. Since the tool layout pattern for the program is set automatically according to the holder name, there is no numeric data item to be set (except for a free tool layout pattern). Be sure to perform tool setting after mounting a holder or replacing the tool.

For the automatic operation, operate in accordance with the selected program and the machining data associated with the selected program.

For the MDI operation and the Preparation operation, operate in accordance with the machining data associated with the currently selected program.

6.1.1 Selecting an execution program

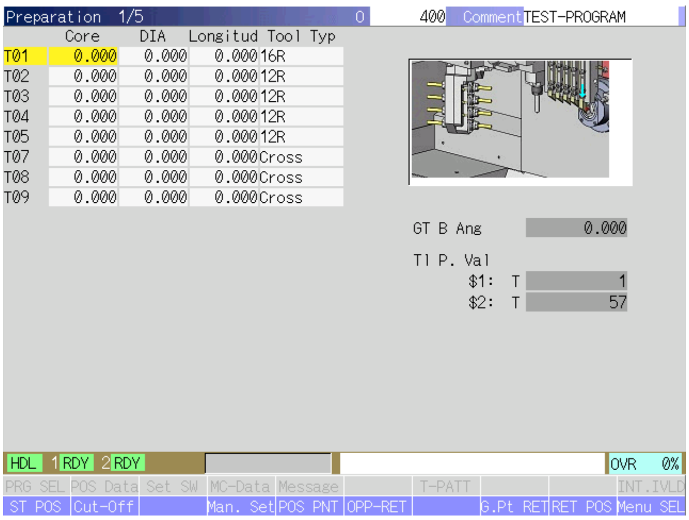
To select a stored program for execution.

[Procedure]

Selecting a program by directly entering the program number

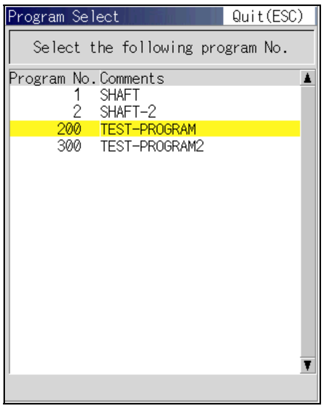
1. Press the Preparation key PREPA. .

The Preparation screen appears.



2. Press the Menu Up/Down selection key, then press the menu key [PRG SEL].

The Program Select screen appears.



3. Enter the desired program number in the data setting field.

4. Press the Input key INPUT.

The program is selected.

[Procedure]

Selecting a program by positioning the cursor in the program number list using the Page keys and up/down cursor keys

1. Press the Preparation key PREPA..

2. Press the Menu Up/Down selection key, then press the menu key [PRG SEL].

The Program Select screen appears.

3. Use the Page keys and Arrow keys to move the cursor in the program number list to the program number you want to select.

4. Press the Input key INPUT.

The program is selected.

[Note]

When a program is selected, the machining data associated with the retrieved program is also retrieved automatically. After the program is selected, be sure to check the machining data.

6.1.2 Setting the machining data

Machining data is needed for the preparation for the machine operation and automatic operation. Machining data is handled in the same way as a machining program and stored in the memory.

The machining data for execution other than the one currently used for operation can be created and edited as a background operation.

The machining data displayed on the Machining Data screen is currently effective and used for the preparation and automatic operation.

The machining data will not be changed by moving to other screens or turning off the power supply and turning it on again.

This section explains the setting of the Execution Machining Data that has been retrieved from the memory together with the execution program by program selection operation.

6.1.3 Required fields for machining data

- Bar Stock O.D.

  Outside diameter of material machined by currently selected program.

- Tool Positioning Point (DIA) Outside diameter of material +

  Use the menu key [POS PNT] to move the front machining tool selected (at the cursor position) on the Preparation screen to the position of (material outer diameter + tool positioning point). Move the tool in the same way even when you have specified a front machining tool for programmed operation.

  This value is the diameter converted from the distance in millimeters from the material to the positioning point at which the tool stops.

- Cut-Off Tool

  Number of tool used as cut-off tool, when cut-off machining is performed during preparation. When using the GB machine, the tool is also selected after start position operation.

- Cut-Off Speed

  The spindle speed when the tip of material is cut. The spindle rotates at the speed specified in "Cut-off" in Preparation menu and Bar Stock Exchange Program M108. When cut-off machining or face turning is performed during preparation, the selected tool (with the cursor positioned on the Preparation screen) stops once at the positioning point. When cut-off machining is restarted, the spindle starts rotating at this speed. The spindle stops after cut-off machining.

- Cut-Off Feed (in mm per revolution)

  The feed rate when the tip of material is cut. The cut-off tool moves at the feed rate specified in "Cut-off" in Preparation menu and Bar Stock Exchange Program M108. When cut-off machining or face turning is performed during preparation, the selected tool (with the cursor positioned on the Preparation screen) stops once at the positioning point. When cut-off machining is restarted and the spindle reaches the speed of "Cut-Off Speed" above, the tool operates at this cutting feed rate.

- Cut-Off End (DIA)

  The X axis cut-off end position specified in ."Cut-off" in Preparation menu and Bar Stock Exchange Program M108. End position of cut-off machining when cut-off machining is performed or face turning is performed during preparation. The tool specified in "Cut-Off Tool Tool Number" is positioned at this point when you move the axes to the machining start position during preparation.
  If moving axis to the center of the material is not required (e.g., pipe material), the positive value can be specified. In such a case, the value must be preceded by plug (+) sign. For example, enter as "+1.0".

- Machining Length, Start point shift amount

  The entered values are different when using the GB model and when using the GBL model + stopper enabled specifications.

  ＜GB machine＞

  Usually, enter the Z1 axis stroke required for machining one workpiece.
  (Product length + Cut-off tool width + Face-cut allowance + α (Margin))

<In GB, when stopper specifications are available>

Enter the value <(Maximum move distance required for machining the workpiece) − (Overall length of workpiece)>.

Right-handed cut-off tool:	Enter the value (Cut-off chip width) + (Shift amount of rear turning tool). (If tool secondary machining tool is used, enter the shift amount of that tool.)
Left-handed cut-off tool:	Enter the larger value either of Cut-off tool shank width, or shift amount of secondary machining tool.

- Pieces/1Chuck

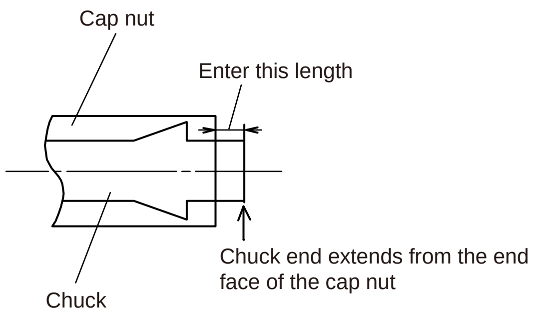
  Number of pieces machined per chuck. Normally, enter 1.

[Note]

The value "Machining Length × Pieces/1Chuck" exceeds the stroke (GB :205mm, GBL :55mm) of Z1, can not be set.

- Back chuck extend length

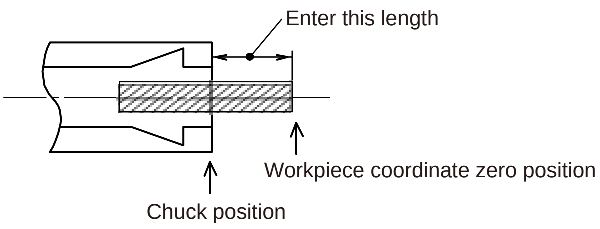
  Enter the length the chuck protrudes from the cap nut when mounted. In other words, enter the long-neck amount when a long-neck chuck is used instead of a standard chuck. If "With Stabilizer" is specified for Back Spindle of Machining data, the Back Chuck Extend Length is not displayed.



- Back work extend length

  Distance from back spindle work coordinate zero point to back spindle chuck position (back spindle cap nut end face) for machining the workpiece chucked on the back spindle.

  (Back work extend length) = (Overall workpiece length) – (Chuck position specified in program)



[Note]

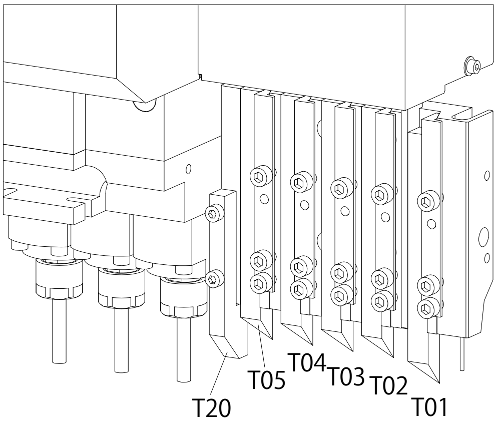
For the workpiece coordinate zero point for back machining, the back long-neck chuck protrusion amount and the back workpiece extend length are included.

- Guide bushing type

  Specify the type of guide bushing of the selected program by selecting one in the list.

- Stopper number (In GBL, when stopper specifications are available)

  Enter the stopper number. If the stopper is not used, enter "0"
  Stopper numbers are: T01 to T05,T20.



- Gang tool post

  Specify the name of the holder to be mounted on the gang tool post. Check the type of the holder mounted on the machine (the type marked on the holder) and select the holder name from the list.

- Gang tool post (rotary tools)

  Specify the name of the holder to be mounted on the gang tool post (T08～T10, T11～T13). Check the type of the holder mounted on the machine (the type marked on the holder) and select the holder name from the list.

- Front Drill post

  Specify the name of the holder mounted in the opposite post by selecting one in the list. Usually, select "1P Spindle Holder".

- Back Drill post

  Specify the name of the holder mounted in the back tool post by selecting one in the list. Usually, select "4P Spindle Holder".

- Back Spindle

  Set the type of the back spindle attached to the back spindle cap. Select "Standard" when the chuck is used.

  Select "With Basket" to attach a workpiece receiver box or "With Support" to attach an end bushing as a support for long workpiece machining.

[Note]

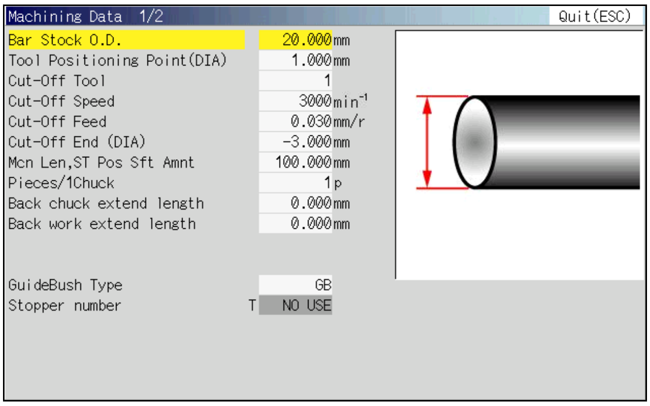
Once machining data has been set, new programs created later reflect the machining data.

6.1.4 Checking machining data

To check the machining data which is needed for machining use the following.

[Procedure]

1. Press the Preparation key PREPA., Automatic Operation key AUTO, Program Check key CHECK, MDI key MDI, Edit key EDIT, or Offset key OFFSET.
   The pressed key lamp lights or only the screen changes accordingly.
2. Press the Menu Up/Down selection key ▶, then press the menu key [MC-Data].
   The Machining Data screen appears.

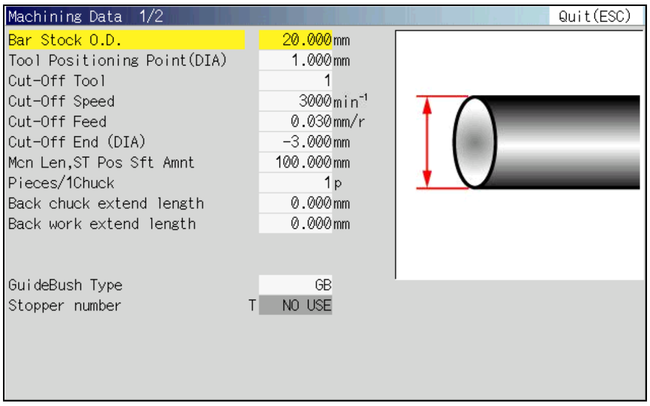


6.1.5 Entering, updating and storing the machining data

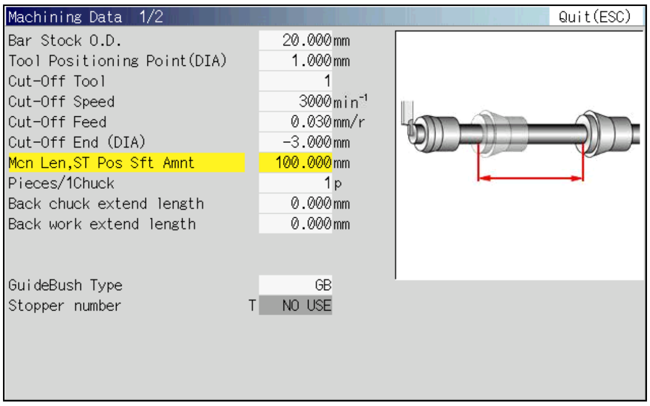
To enter, update and store the Machining Data use the following procedure.

[Procedure]

1. Set the Program protection switch key to "I". The program is released from the protected state.
2. Press the Preparation key, Automatic Operation key, Program Check key, MDI key, Edit key, or Offset key. The pressed key lamp lights or only the screen changes accordingly.
3. Press the Menu Up/Down selection key, then press the menu key [MC-Data].The Machining Data screen appears.



4. Use the Page key or the Arrow keys to select the item you want to specify or change.
5. Enter the desired value using numeric keys. For the tool post, find the selection number of the desired holder in the list and enter it to the setting field.



6. Press the Input key.

   The Machining Data screen is updated to reflect the data just entered.
   The machine will be operated in accordance with the new machining data you have set.

[Note]

While the Program protection switch key is set to "O", you cannot edit the program or machining data.

6.1.6 Verifying the tool layout pattern

To verify the tool layout pattern of the gang tool post, use the following procedure.

When a T code is specified in the program or when a tool changing on the Preparation screen, the machine operates based on this data.

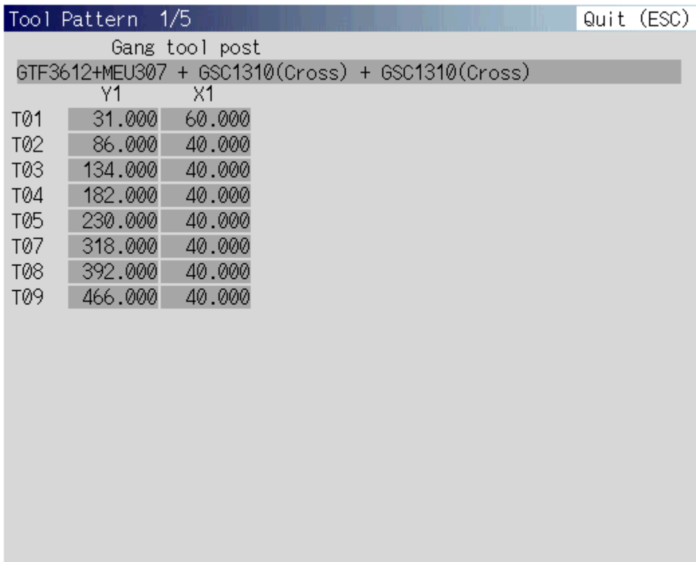
[Procedure]

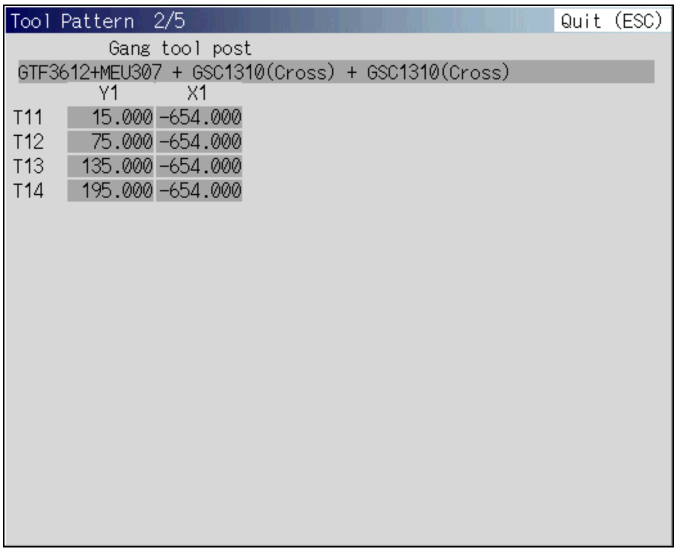
1. Press the Preparation key, Automatic Operation key, Program Check key, MDI key, or Offset key.

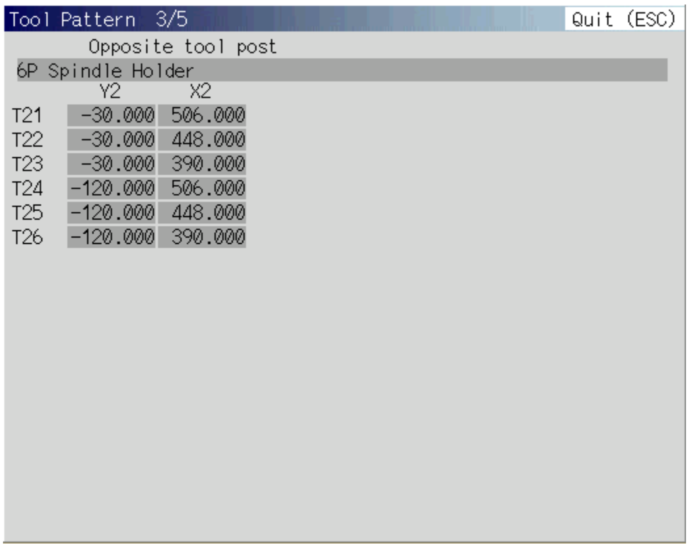
The pressed key lamp lights or only the screen changes accordingly.

2. Press the Up/Down selection key ▶, then press the menu key [T-PATT].

The Tool Pattern screen appears.





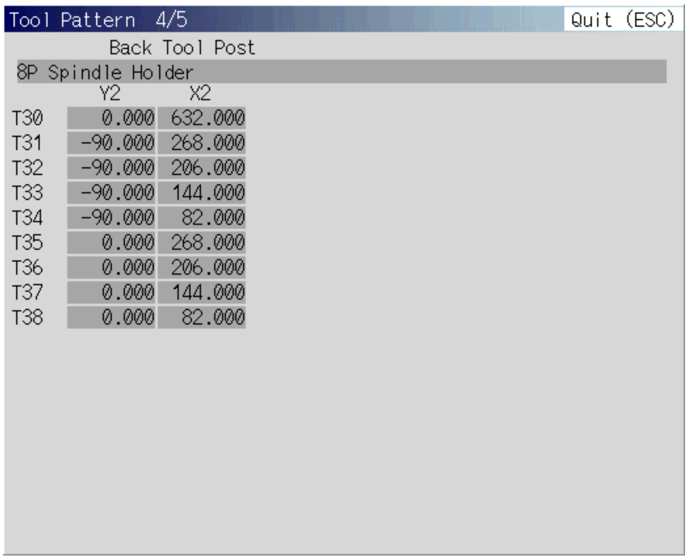


Tool Pattern 3/5 Quit (ESC)

Opposite tool post

6P Spindle Holder

	Y2	X2
T21	-30.000	506.000
T22	-30.000	448.000
T23	-30.000	390.000
T24	-120.000	506.000
T25	-120.000	448.000
T26	-120.000	390.000

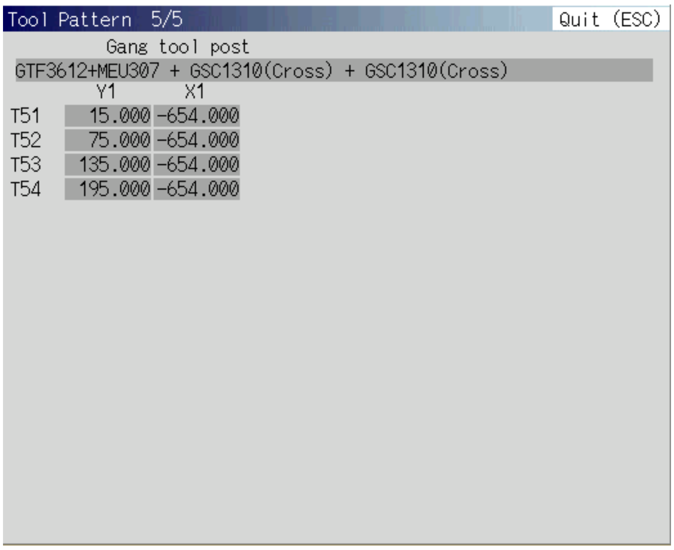


Tool Pattern 4/5 Quit (ESC)

Back Tool Post

8P Spindle Holder

	Y2	X2
T30	0.000	632.000
T31	-90.000	268.000
T32	-90.000	206.000
T33	-90.000	144.000
T34	-90.000	82.000
T35	0.000	268.000
T36	0.000	206.000
T37	0.000	144.000
T38	0.000	82.000



Tool Pattern 5/5 Quit (ESC)

Gang tool post

GTF3612+MEU307 + GSC1310(Cross) + GSC1310(Cross)

	Y1	X1
T51	15.000	-654.000
T52	75.000	-654.000
T53	135.000	-654.000
T54	195.000	-654.000

6.1.7 Setting the free tool layout pattern

To set the free tool layout pattern, use the following procedure.

When a T code is specified in the program or when a tool changing on the Preparation screen, with the free tool layout pattern specified as machining data, the machine operates based on this data.

Free tool layout pattern can be set separately for gang tool posts, front drilling posts and back drilling posts.

This section describes the operating procedure for setting the free tool layout pattern only for the gang tool posts. The free tool layout pattern for front drilling posts and back drilling posts can be set in the same manner.

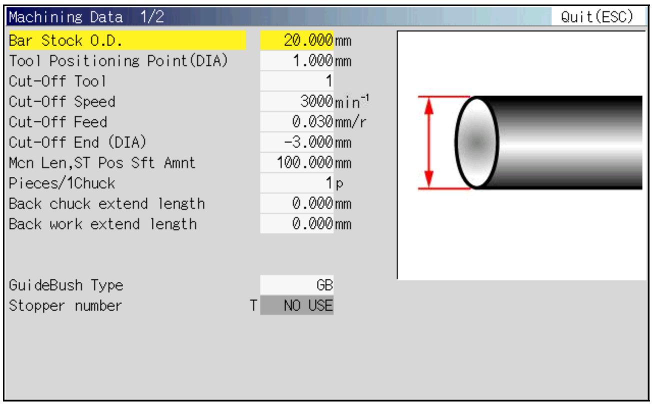
[Procedure]

1. Press the Preparation key PREPA., Automatic Operation key AUTO, Program Check key CHECK, MDI key MDI, or Offset key OFFSET.

The pressed key lamp lights or only the screen changes accordingly.

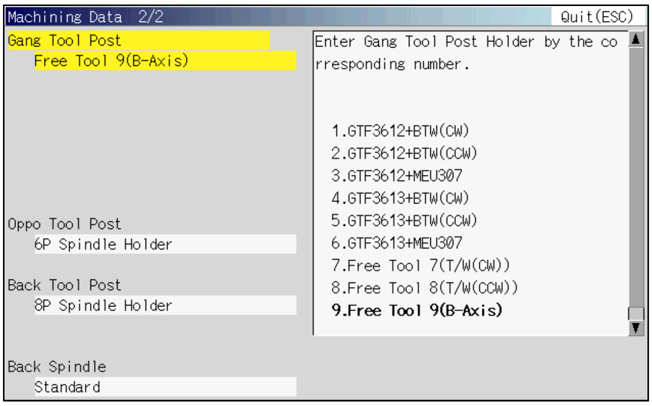
2. Press the Menu Up/Down selection key ▶, then press the menu key [MC-Data].

The Machining Data screen appears.

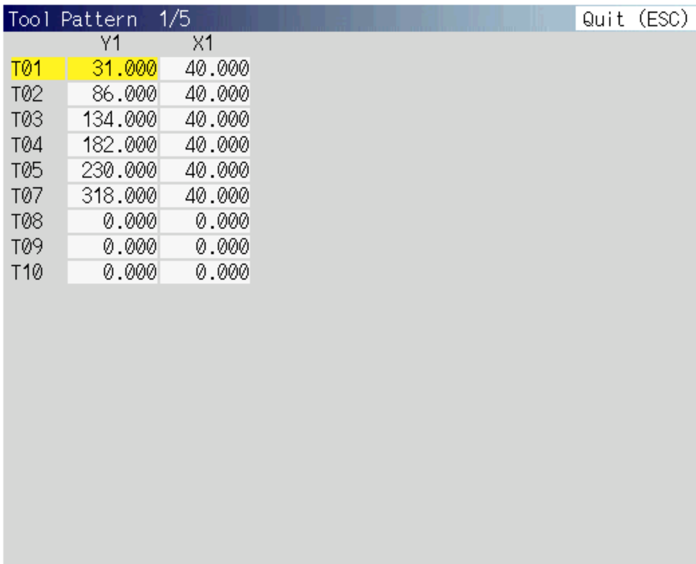


3. Make sure that the gang tool post is "Free Tool".

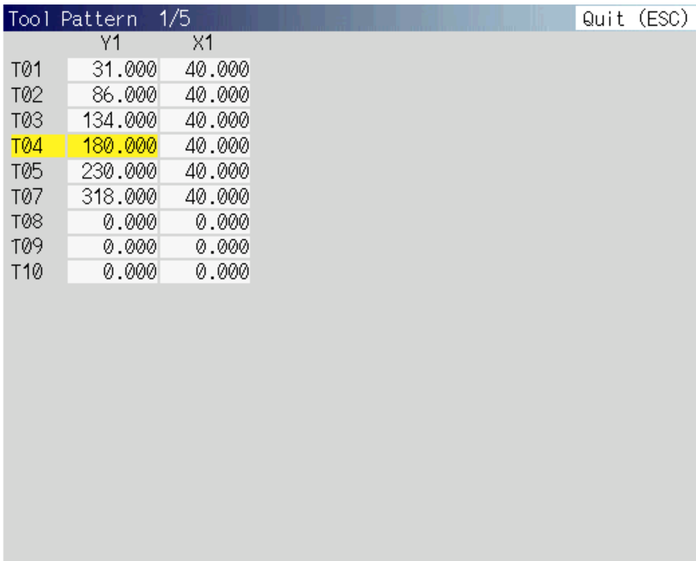
If not, set it to "Free Tool". See *<6.1.5 Entering, updating and storing the machining data>*.



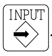
4. Press the menu key [T-PATT]. The Tool Pattern screen appears.



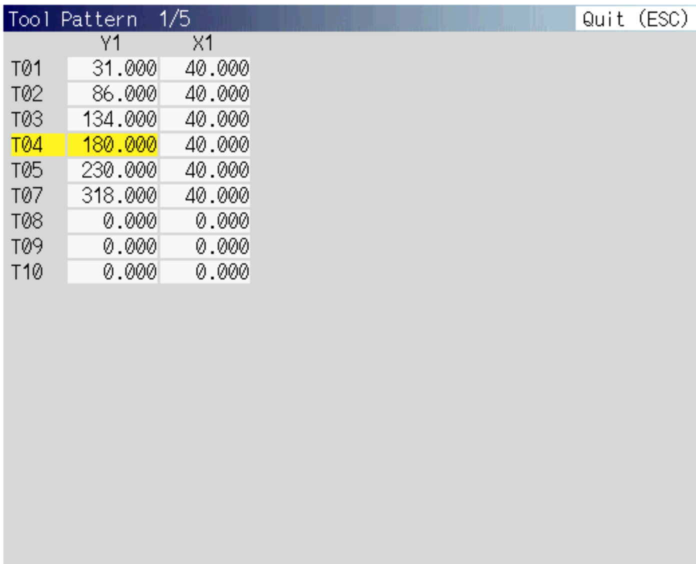
5. Use the Tab keys TAB TAB, page keys , or the Arrow keys to select the item you want to enter or change.



6. Use the numeric keys to enter the data to replace or add in the setting field of the Tool Pattern screen.

7. Press the Input key 

. The new data you have just entered is set.



	Y1	X1
T01	31.000	40.000
T02	86.000	40.000
T03	134.000	40.000
T04	180.000	40.000
T05	230.000	40.000
T07	318.000	40.000
T08	0.000	0.000
T09	0.000	0.000
T10	0.000	0.000

6.2 Preparation

After selecting a program you need to prepare the machine for operation before you can start machining a product. The Preparation functions involve tool setting, cut-off machining and moving of the axes to start positions before automatic operation after tool setup.

CAUTION

In the preparation mode if the door is open, the machine operates only while you are holding down the Start button (except during cut-off machining and phase adjustment). Releasing the Start button stops the machine and causes the Start button lamp to blink (remains on for two seconds and off for one second). Pressing the Start button with the lamp blinking restarts the machine. The Start button lamp goes out upon completion of the current operation. Make sure that the operation has been completed with the Start button lamp off, then proceed to the operation to be performed next.

If the machine is equipped with Enable switch and you want to operate the machine while the door is open, press the Enable switch and the Start button concurrently. The machine starts operation. If either of Enable switch or Start button is released during operation, the machine halts. Pressing the Enable switch and the Start button concurrently resumes operation.

To operate the machine while the door is closed, press the Start button once. The door is locked and the Start button lamp blinks (on for 0.3 seconds and off for 0.3 seconds). Press the Start button again while the lamp is blinking, and the machine starts operating. In this case, it is not necessary to hold down the Start button. The Start button lamp goes out upon completion of the operation.

Described below are operation procedures when the Setting switch is set to "O".

If you are going to work inside the machine, press the Door open button to unlock the door.

The door cannot be unlocked unless the following conditions are met.

All spindles have stopped.

All servo axes have stopped.

The machine is not operating.

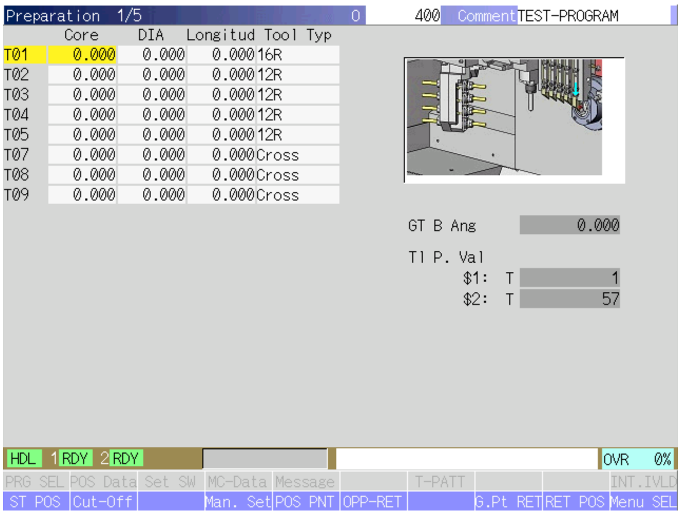
The coolant is not discharged:

The Door open button is pressed.

[Procedure]

1. Press the Preparation key PREPA..

The Preparation key lamp lights and the Preparation screen appears.



WARNING

The moving parts of the machine may move suddenly during some Preparation functions. Keep your hands and body away from the machining area during Preparation.When preparing to operate using new and existing machining programs, or executing a new machining program, make sure that the tools mounted on the machine do not interfere with a part of the machine. If a tool interferes with another part, this may damage the machine or may result in personal injury. When you check the tool operations, find the part of the machining program corresponding to tool motion and act safely by following the corresponding safety precautions. For other precautions, see *<2. Safety Precautions>*.

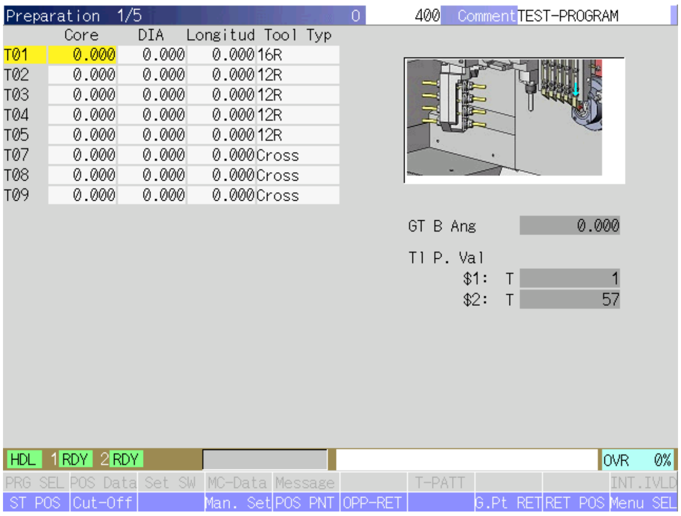
6.2.1 Return position

The axes move to the return position. That is, the X1 axis moves in the plus (upward) direction, the Z1 axis in the minus (rightward) direction, and the Z2 axis to the machine coordinate zero point Z2 0.0 (safe position in the left).

[Procedure]

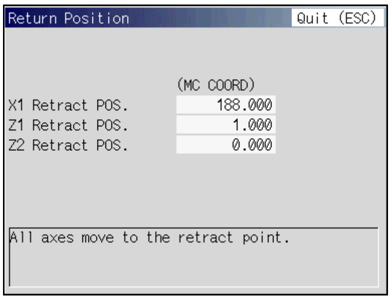
1. Press the Preparation key PREPA. .

The Preparation key lamp lights and the Preparation screen appears.



2. Press the menu key [RET POS].

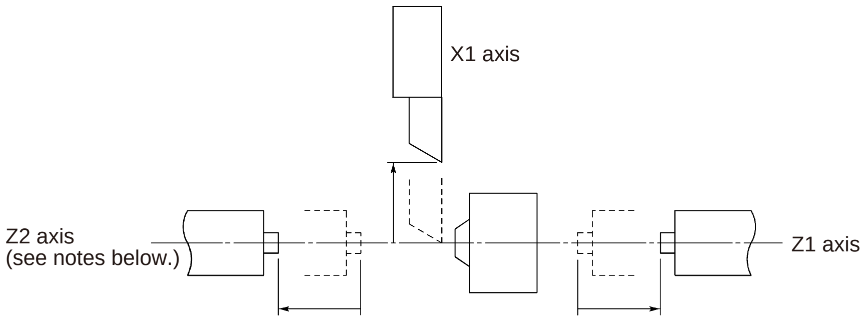
The Return Position screen appears.



The screen is for the machine equipped with guide bushing device. In GBL, the Z1 axis returns position is not displayed.

3. Make sure that all doors are closed, and press the Start button .
The Start button lamp blinks on and off, and the doors will lock.

4. Press the Start button .
The Start button lamp lights, and the X1, Z1, and Z2 axes return to their respective return position.



The X1, Z1, and Z2 axes return to the same positions as the zero points for them to return, respectively.

[Note]

- To enable operation while the door is open, press the Door open button to unlock the door, and set the Setting switch key to the "I" position. If this switch key is set to the "O" position, machine operation is prohibited while the door is open.
- The unit will only operate with the door open while the Start button is pressed.

  The axes speed at this time is limited to 2 m/min or less.
- Releasing the Start button stops axis feed in the hold state and the Start button lamp blinks (on for two seconds and off for one second). To restart the operation, press the Start button.
- If no operation is performed within 15 seconds after making menu selections, menu selections will automatically be canceled.
  Menu selections will also be canceled if a reset is performed during or after operations. To restart operations, be sure to again make selections with the menus.
- With GBL-machine, Return position operation does not move the Z1 axis.

6.2.2 Positioning point

The specified tool (the front machining tool with the cursor positioned) is moved to the positioning point.

> **CAUTION**
>
> Described below are operation procedures when the Setting switch is set to "O".
>
> If you are going to work inside the machine, press the Door open button to unlock the door.
>
> The door cannot be unlocked unless the following conditions are met.
>
> All spindles have stopped.
>
> All servo axes have stopped.
>
> The machine is not operating.
>
> The coolant is not discharged:
>
> The Door open button is pressed.

6.2.2.1 Positioning point

- Tool, cross machining tool

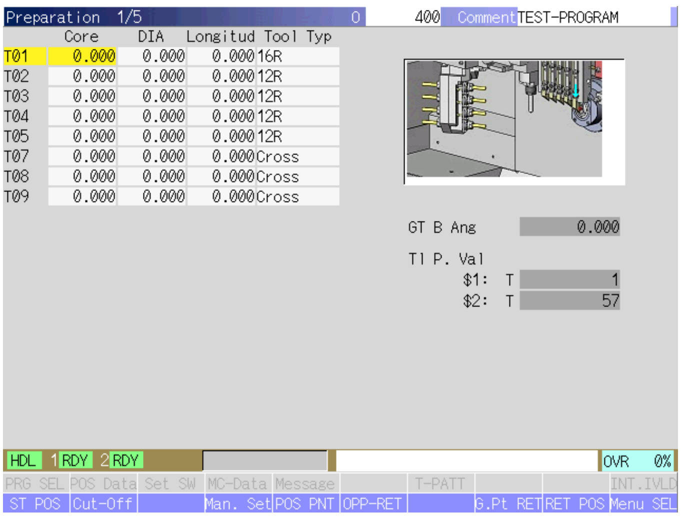
  (Material outer diameter) + (Positioning point) + (Diameter data set on the Preparation screen)
- The hole of the end face

  The center of the material

[Procedure]

1. Press the Preparation key PREPA..

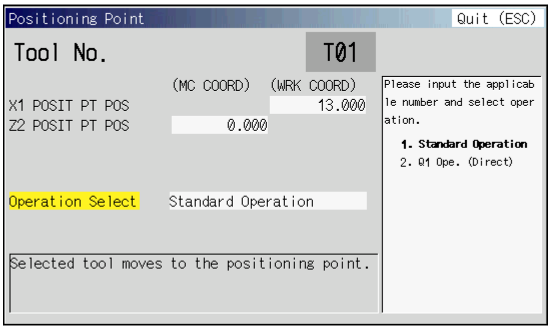
The Preparation key lamp lights and the Preparation screen appears.



2. Use the Arrow keys ⬆ ⬇ to select the desired tool number.

3. Press the menu key [POS PNT].

The Positioning Point screen appears.

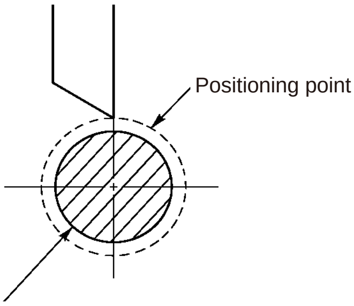


4. Select "Q1 Ope." as necessary.
After the Q1 Operation is selected, the specified tool directly moves to the queuing position from the current position. This motion is mainly used after the program auto run is terminated. When using the Q1 motion, be sure to unload the workpiece from the machine.

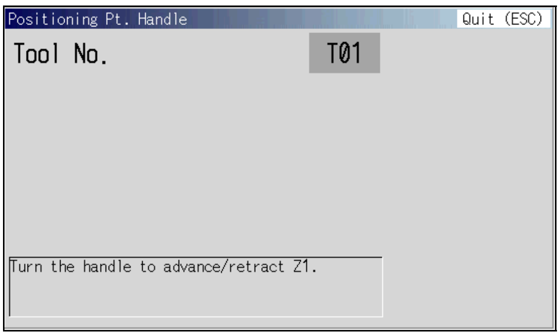
5. Make sure that all doors are closed, and press the Start button.
The Start button lamp blinks on and off, and the doors will lock.

6. Press the Start button.

The Start button lamp lights. The Z2 axis moves to return point and the specified tool moves to its positioning point.



7. When operations are completed, the Start button lamp turns off and the screen changes to the Positioning Point Handle screen.



8. Press one of the Handle Magnification button x1, x10, or x100.
The larger the number the faster the machine will move.

9. Move the Z1 axis by handle.

10. Press the Escape key ESC to finish the operation.

[Note]

- To enable operation while the door is open, press the Door open button to unlock the door, and set the Setting switch key to the "I" position. If this switch key is set to the "O" position, machine operation is prohibited while the door is open.
- The unit will only operate with the door open while the Start button is pressed.

  The axes speed at this time is limited to 2 m/min or less.
- Releasing the Start button stops axis feed in the hold state and the Start button lamp blinks (on for two seconds and off for one second). To restart the operation, press the Start button.
- If no operation is performed within 15 seconds after making menu selections, menu selections will automatically be canceled.
  Menu selections will also be canceled if a reset is performed during or after operations. To restart operations, be sure to again make selections with the menus.
- The menu key [POS PNT] is disabled if no program is selected.

6.2.3 Start position

Use the following procedure to move the axes into position for carrying out automatic operation. The Z1 axis will move for the machining length from the foremost position and X1 axis will move to the cut-off ending position. The Z2 axis moves to the return point.

[Note]

The cut-off ending position complies with the Cut-Off End (DIA) value set in the Machining Data .
The default Cut-Off End (DIA) value is –3.0 mm. The machining length indicates the data set in Machining Data.

CAUTION

Described below are operation procedures when the Setting switch is set to "O".

If you are going to work inside the machine, press the Door open button to unlock the door.

The door cannot be unlocked unless the following conditions are met.

All spindles have stopped.

All servo axes have stopped.

The machine is not operating.

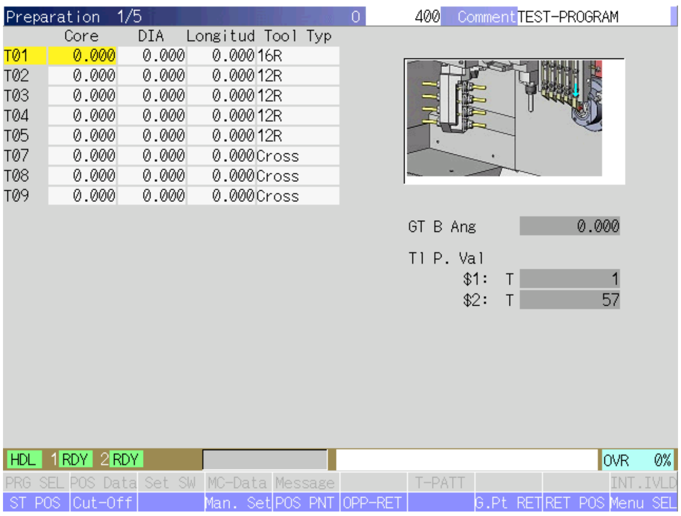
The coolant is not discharged:

The Door open button is pressed.

[Procedure]

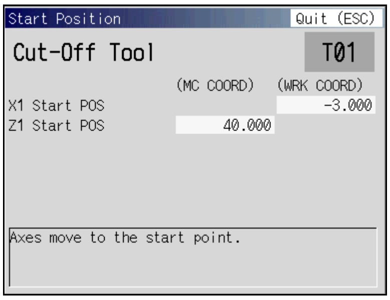
1. Press the Preparation key PREPA..

The Preparation key lamp lights and the Preparation screen appears.



2. Press the menu key [ST POS].

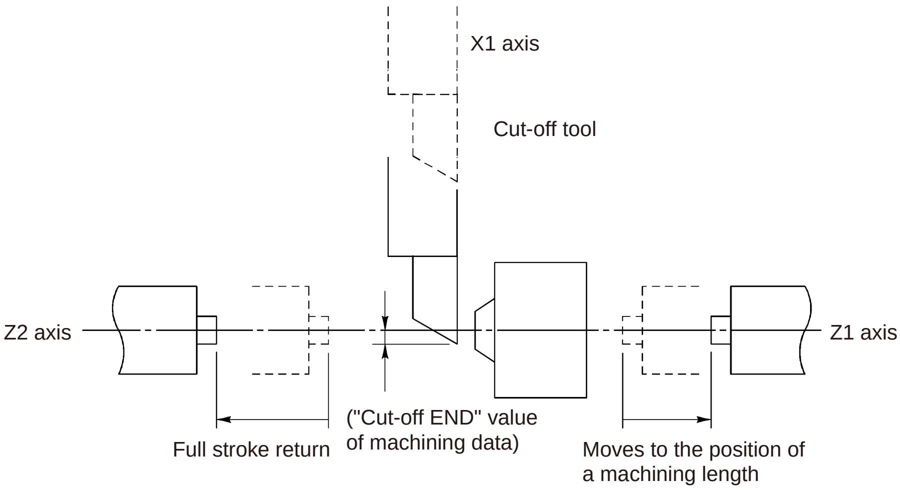
The Start Position screen appears.



3. Make sure that all doors are closed, and press the Start button.
The Start button lamp blinks on and off, and the doors will lock.

4. Press the Start button.
The Start button lamp lights and the start position operation is started. During machine operation the message "Moving to start position..." appears. When operation is completed, the Start button lamp turns off and the message changes to "Moving to start position completed."

5. When operations are completed, menu selection will be canceled.



[Note]

- The machine selects the tool specified in machining data "Cut-off Tool Tool Number" for start position operation, regarding it as the cut-off machining tool.
- If you specify the machining start position when already the cut-off tool has been selected and the X1 and Y1 axes have moved to the area within (±0.1 mm from the respective machining start position (completion position), the X1 axis will not move in the vertical direction (the Q1 motion of T code macro is performed).
- If you instruct the machine to perform start position operation with the main spindle chuck closed, the instruction results in an alarm of "EX501: Spindle chuck closed. Open spindle chuck". Open the main spindle chuck before retrying start position operation.
- You do not need to move the axes to the return position before moving them to the machining start position.
- To enable operation while the door is open, press the Door open button to unlock the door, and set the Setting switch key to the "I" position. If this switch key is set to the "O" position, machine operation is prohibited while the door is open.
- The unit will only operate with the door open while the Start button is pressed.

  The axes speed at this time is limited to 2 m/min or less.
- Releasing the Start button stops axis feed in the hold state and the Start button lamp blinks (on for two seconds and off for one second). To restart the operation, press the Start button.
- If no operation is performed within 15 seconds after making menu selections, menu selections will automatically be canceled.

  Menu selections will also be canceled if a reset is performed during or after operations. To restart operations, be sure to again make selections with the menus.
- The menu key [ST POS] is disabled if no program is selected.
- After the idle stop, be sure to perform Start position operation on Preparation screen before starting operation.
- With the GBL-machine, "Stopper" is selected for Start position operation instead of "Cut-off Tool".

WARNING

Moving the axes to the start position involves movement of the headstock. Keep away from any moving part of the machine to prevent injur. Be sure to remove the material before starting to move the axes to the machining start positions.
If the headstock moves forward while the material is inserted into headstock, the material and tool might cause interference.

6.2.4 Gang tool post retraction

Only the gang tool post moves to the retraction end position (upward direction).
(Movement to the machine coordinate X1 253.0)

CAUTION

Described below are operation procedures when the Setting switch is set to "O".

If you are going to work inside the machine, press the Door open button to unlock the door.

The door cannot be unlocked unless the following conditions are met.

All spindles have stopped.

All servo axes have stopped.

The machine is not operating.

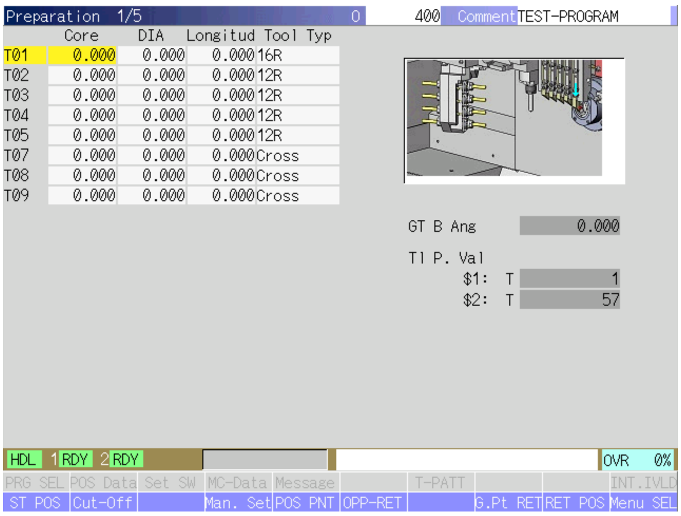
The coolant is not discharged:

The Door open button is pressed.

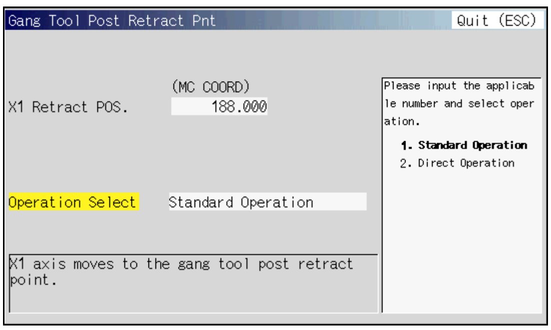
[Procedure]

1. Press the Preparation key PREPA..

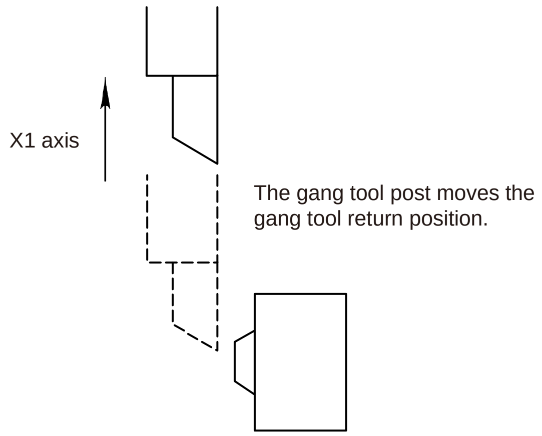
The Preparation key lamp lights and the Preparation screen appears.



2. Press the menu key [G.Pt RET].
The Gang Tool Post Retract Pnt screen appears.



3. Make sure that all doors are closed, and press the Start button.
   The Start button lamp blinks on and off, and the doors will lock.
4. Press the Start button. The Start button lamp lights and the gang tool post moves to the return position.
5. When operations are completed, menu selection will be canceled.



[Note]

- To enable operation while the door is open, press the Door open button to unlock the door, and set the Setting switch key to the "I" position. If this switch key is set to the "O" position, machine operation is prohibited while the door is open.
- The unit will only operate with the door open while the Start button is pressed.

  The axes speed at this time is limited to 2 m/min or less.
- Releasing the Start button stops axis feed in the hold state and the Start button lamp blinks (on for two seconds and off for one second). To restart the operation, press the Start button.
- If no operation is performed within 15 seconds after making menu selections, menu selections will automatically be canceled.
  Menu selections will also be canceled if a reset is performed during or after operations. To restart operations, be sure to again make selections with the menus.

6.2.5 Opposite tool post retraction

Only the opposite tool post Z2 axis moves to the retraction end position. (Movement to the machine coordinate Z2 0.0)

CAUTION

Described below are operation procedures when the Setting switch is set to "O".

If you are going to work inside the machine, press the Door open button to unlock the door.

The door cannot be unlocked unless the following conditions are met.

All spindles have stopped.

All servo axes have stopped.

The machine is not operating.

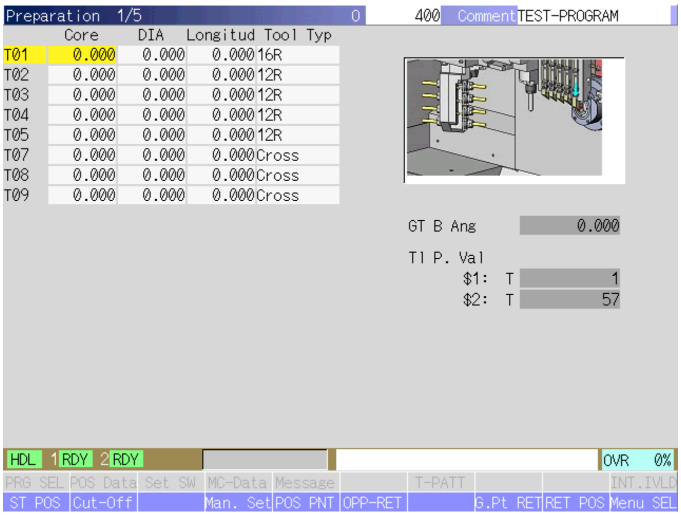
The coolant is not discharged:

The Door open button is pressed.

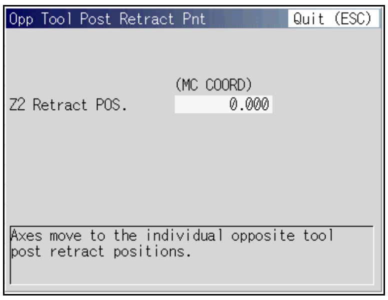
[Procedure]

1. Press the Preparation key PREPA..

The Preparation key lamp lights and the Preparation screen appears.



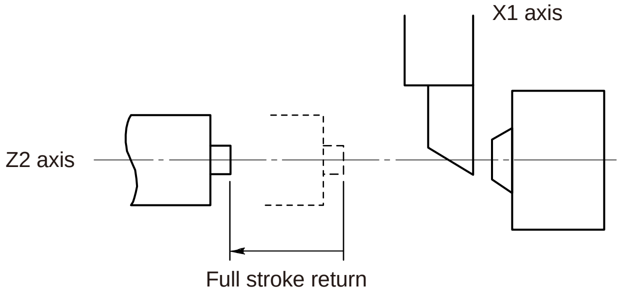
2. Press the menu key [OPP-RET].
The opposite tool post retract point screen appears.



3. Make sure that all doors are closed, and press the Start button.
The Start button lamp blinks on and off, and the doors will lock.

4. Press the Start button [Start button icon].
The Start button lamp lights and the machine moves to the opposite tool post return position.

5. When operations are completed, menu selection will be canceled.



The opposite tool post Z2 axis moves to the machine coordinate Z2 0.0 position.

[Note]

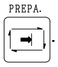
- To enable operation while the door is open, press the Door open button to unlock the door, and set the Setting switch key to the "I" position. If this switch key is set to the "O" position, machine operation is prohibited while the door is open.
- The unit will only operate with the door open while the Start button is pressed.

  The axes speed at this time is limited to 2 m/min or less.
- Releasing the Start button stops axis feed in the hold state and the Start button lamp blinks (on for two seconds and off for one second). To restart the operation, press the Start button.
- If no operation is performed within 15 seconds after making menu selections, menu selections will automatically be canceled.Menu selections will also be canceled if a reset is performed during or after operations. To restart operations, be sure to again make selections with the menus.

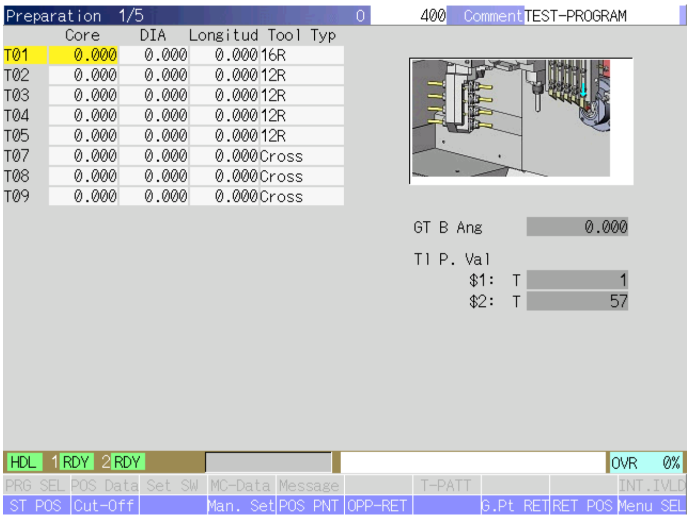
6.2.6 Chuck thrust

The chuck thrust screen shows a graph of the chucking force adjustment nut angle that is referenced during chuck thrust adjustments.

[Procedure]

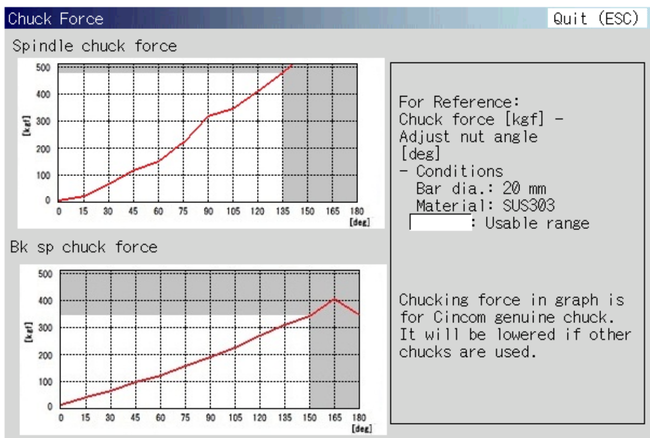
1. Press the Preparation key .

The Preparation key lamp lights and the Preparation screen appears.



2. Press the menu key [Chuck thrust]

Chuck thrust screen appears.



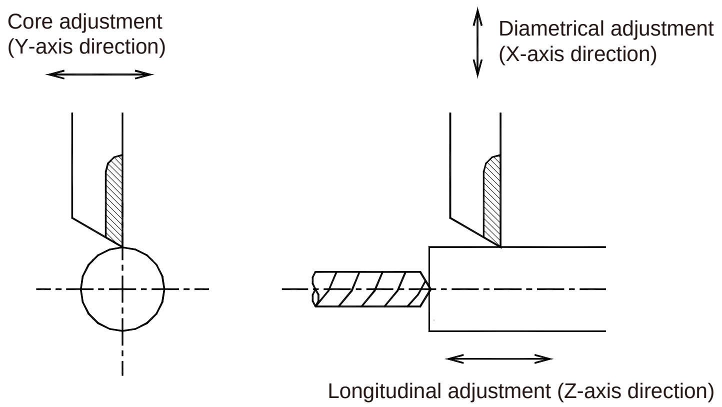
6.2.7 Setting tools

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.

If the program is changed in program selection or the tool holder set in the machining data is changed, you need to change the tool set data (DIA., Core, or Longitud) as needed.

The machine uses the in-machine tool setting function (except for special tools). The software for supporting the in-machine tool setting is provided.

CAUTION

Pull out the material carefully. If the tool interferes with the material, this could damage the tool and cause personal injury.

Described below are operation procedures when the Setting switch is set to "**O**".

If you are going to work inside the machine, press the Door open button to unlock the door.

The door cannot be unlocked unless the following conditions are met.

All spindles have stopped.

All servo axes have stopped.

The machine is not operating.

The coolant is not discharged:

The Door open button is pressed.

6.2.7.1 Tool setting procedure

To set the tools in the machine, use one of the two procedures given below.

[Procedure]

When the tool is attached right adjacent to the tool to be set

Steps enclosed by dashed lines can be omitted if it is not required.

1. Perform core adjustment of the tool.
   See *<6.2.7.4 Core (outer circumference)>*.

2. Perform longitude adjustment of the cut-off tool.
   See *<6.2.7.3 Longitude (gang tool)>*.

3. Perform longitude adjustments of tools other than the cut-off tool.
   See *<6.2.7.3 Longitude (gang tool)>*.

4. Attach the tool.
   See *<6.2.7.7 Diameter (gang tool / cross rotary tool)>*.

5. Perform end surface cutting using each tool.
   See *<6.2.5 Opposite tool post retraction>*.

6. Confirm the nib amount and directly enter the core height compensation amount.

7. Perform diameter adjustment of the tool again.
   See *<6.2.7.7 Diameter (gang tool / cross rotary tool)>*.

8. Attach the rotary tool.
   See *<6.2.7.7 Diameter (gang tool / cross rotary tool)>*.

9. Perform core adjustment of the rotary tool.
   See *<6.2.7.5 Core (core down)>*.

10. Attach the front drilling tool.
    See *<6.2.7.8 Longitude (core) (drill)>*.

11. Perform center adjustment of the front drilling tool.
    See *<6.2.7.10 Centering the tool (drill) (boring)>*, Procedure 1.

12. Perform the one cycle machining operation.

13. Attach the back machining drilling tool.
    See *<6.2.7.9 Longitude (back drill)>*.

14. Perform center adjustment of the back machining drilling tool.
    See *<6.2.7.10 Centering the tool (drill) (boring)>*, Procedure 1.

[Procedure]

When the tool is not attached right adjacent to the tool to be set

Steps enclosed by dashed lines can be omitted if it is not required.

1. Perform longitude adjustment of the cut-off tool.
   See <*6.2.7.3 Longitude (gang tool)*>.

2. Perform longitude adjustments of tools other than the cut-off tool.
   See <*6.2.7.3 Longitude (gang tool)*>.

3. Attach the tool.
   See <*6.2.7.7 Diameter (gang tool / cross rotary tool)*>.

4. Perform core adjustment of the tool.
   See <*6.2.7.5 Core (core down)*>.

5. Remove the tool once and after attaching it again, perform diametric adjustment.
   See <*6.2.7.7 Diameter (gang tool / cross rotary tool)*>.

6. Perform end surface cutting using each tool.
   See <*6.2.5 Opposite tool post retraction*>.

7. Confirm the nib amount and directly enter the core height compensation amount.

8. Perform diameter adjustment of the tool again.
   See <*6.2.7.7 Diameter (gang tool / cross rotary tool)*>.

9. Attach the rotary tool.
   See <*6.2.7.7 Diameter (gang tool / cross rotary tool)*>.

10. Perform core adjustment of the rotary tool.
    See <*6.2.7.5 Core (core down)*>.

11. Attach the front drilling tool.
    See <*6.2.7.8 Longitude (core) (drill)*>.

12. Perform center adjustment of the front drilling tool.
    See <*6.2.7.10 Centering the tool (drill) (boring)*>.

13. Perform the one cycle machining operation.

14. Attach the back machining drilling tool.
    See <*6.2.7.9 Longitude (back drill)*>.

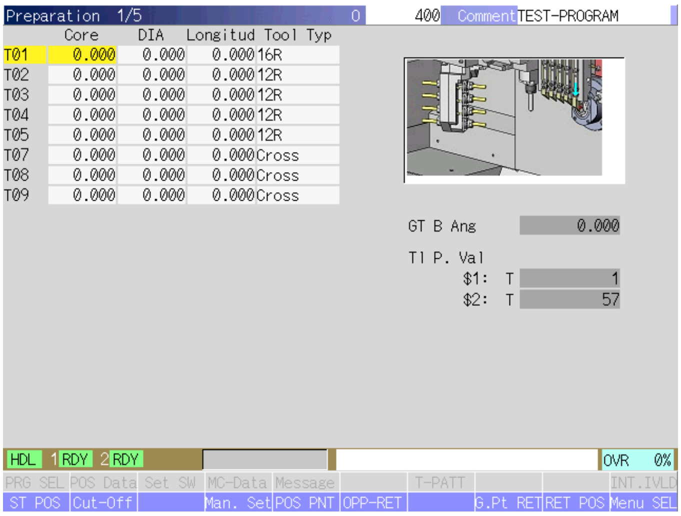
15. Perform center adjustment of the back machining drilling tool.
    See <*6.2.7.10 Centering the tool (drill) (boring)*>.

6.2.7.2 Selecting the tool to be set

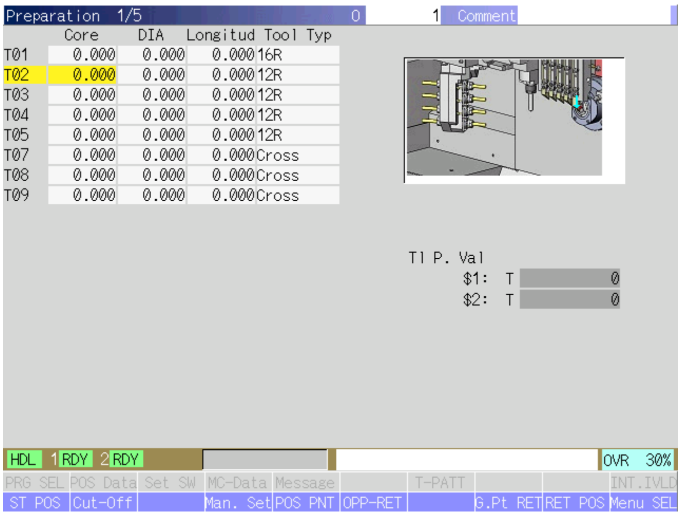
[Procedure]

1. Press the Preparation key PREPA..

The Preparation key lamp lights and the Preparation screen appears.

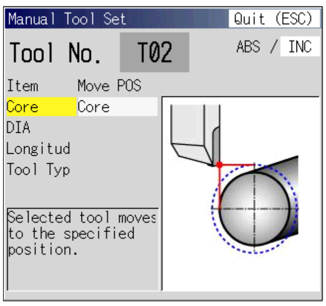


2. Use the Arrow keys ⬆ ⬇ or Tab keys TAB ⬅ TAB ➡ to move the cursor to the tool to be set.

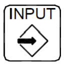


3. Press the menu key [Man. Set].

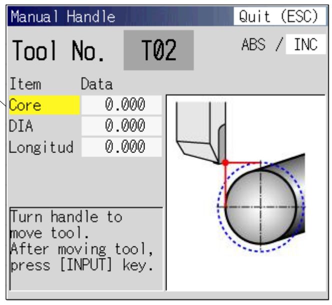
The Manual Tool Set screen appears.



4. After machine operation the Manual Handle screen appears.

With the cursor in this position press the 

key to enter the handle movement amount.



6.2.7.3 Longitude (gang tool)

Align the machining surface positions of other tools based on the machining surface of the cut-off tool. Adjust the tool position so that the machining surface of a tool matches with the end surface of the material.

This function shifts the program reference point and tool cutting point in the longitudinal direction. In normal case, use the longitudinal direction coordinate system shift command in the machining program. (See <*8.5 Coordinate System Shift*> in the Instruction Manual (Optional).)

[Procedure]

1. Press the Preparation key PREPA..

The Preparation key lamp lights and the Preparation screen appears.

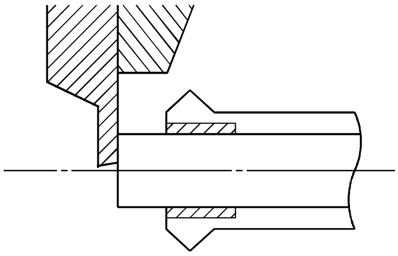
2. Turn off the bar loader torque and open the chuck.

(Turn off a dedicated bar loader. On other bar loaders, turn off torque.)

3. Perform start position operation.

See <*6.2.3 Start position*>.

4. Align the end surface of the material with the end surface of the cut-off tool.

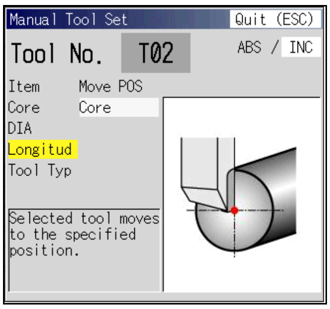


5. Close the chuck.

6. Use the Page keys or the Arrow keys ⬆ ⬇ or the Tab keys TAB ⬅ TAB ➡ to move the cursor to the Longitud of the desired tool.

7. Press the menu key [Man. Set].

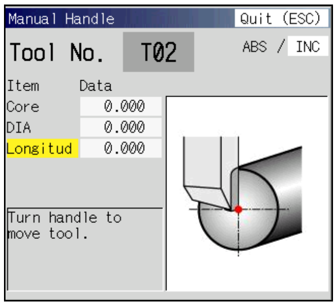
The Manual Tool Set screen appears. Confirm that the cursor is at the longitude. If the cursor is not at the Longitud, use the Arrow keys ⬆ ⬇ to move the cursor to the Longitud.



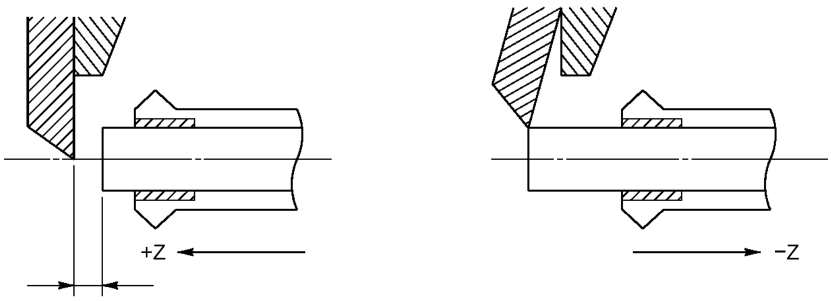
8. Make sure that all doors are closed, and press the Start button.
The Start button lamp blinks on and off, and the doors will lock.

9. Press the Start button.
The Start button lamp lights and the Z2, Z1, X1, and Y1 axes will move.

10. When operations are completed, the Start button lamp turns off and the screen changes to the Manual Handle screen.



11. Apply the tool to the tool holder and move the material (Z1 axis) back and forth in handle feed to perform longitudinal adjustment so that the end face of the tool is aligned with the end face of the material.



12. Press the Input key.

13. Open the chuck and return the material.

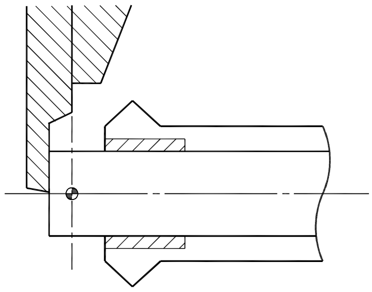
[Note]

- To enable operation while the door is open, press the Door open button to unlock the door, and set the Setting switch key to the "I" position. If this switch key is set to the "O" position, machine operation is prohibited while the door is open.
- The unit will only operate with the door open while the Start button is pressed.

  The axes speed at this time is limited to 2 m/min or less.
- Releasing the Start button stops axis feed in the hold state and the Start button lamp blinks (on for two seconds and off for one second). To restart the operation, press the Start button.
- If no operation is performed within 15 seconds after making menu selections, menu selections will automatically be canceled.

  Menu selections will also be canceled if a reset is performed during or after operations. To restart operations, be sure to again make selections with the menus.
- When performing adjustment for core (Y), diameter (X) and center (X) using the handle, selection of multiplication ratio $\times 100$ is not possible.
- If the cut-off tool is shifted to the longitudinal direction, this function is unavailable.

Example:
When the left-handed cut-off tool is used.



- The menu key [Man. Set] is disabled if no program is selected.

6.2.7.4 Core (outer circumference)

Adjust the core height of the gang tool. Move the tool so that the machining tool upper end (rake face) is aligned with the circumference of a material. The X1 axis is moved to the POSITIONING PT. preset in the machining data.

[Procedure]

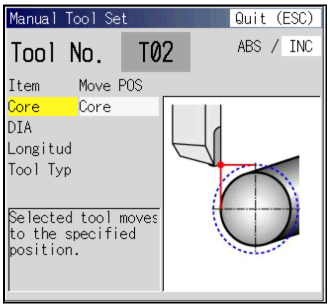
1. Press the Preparation key PREPA..

The Preparation key lamp lights and the Preparation screen appears.

2. Use the Arrow keys or the Tab keys TAB TAB to move the cursor to the core of the tool to be set.

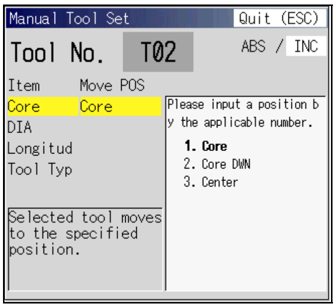
3. Press the menu key [Man. Set].

The Manual Tool Set screen appears. Confirm that the cursor is at the core. If the cursor is not at the core, use the Arrow keys to move the cursor to the Core.



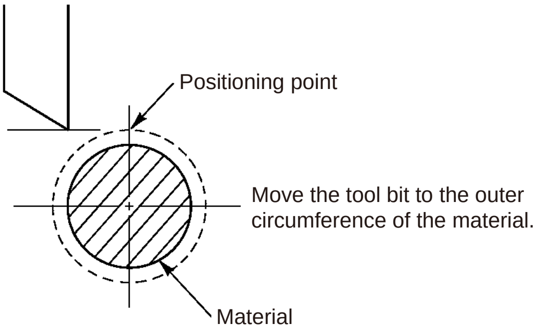
4. Press the Arrow key . Two cursors appear and the list showing the move positions is displayed. Enter the selection number of "Core" and press the Input key INPUT.

The Move POS setting is recorded as a default, so once "Core " is set it is valid for the next tool setting.

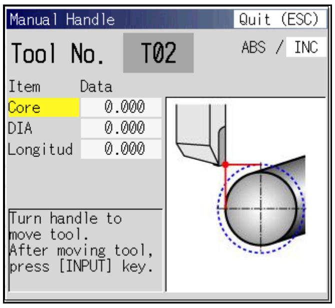


5. Make sure that all doors are closed, and press the Start button .
The Start button lamp blinks on and off, and the doors will lock.

6. Press the Start button.
The Start button lamp lights and the tool moves to the circumference of the material.
X1 axis is moved to the positioning point.

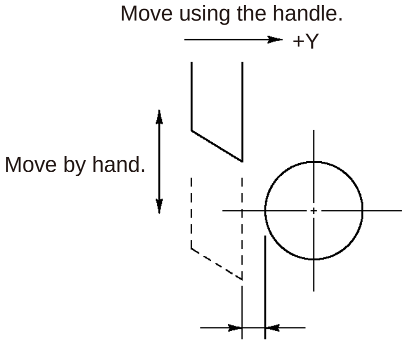


7. When operations are completed, the Start button lamp turns off and the screen changes to the Manual Handle screen.

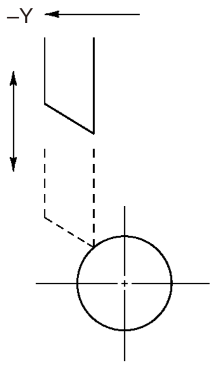


8. Press the Door open button to unlock the door, and open the door.

9. Pull out the material and move the tool up and down by hand to check whether the tool tip is aligned with the circumference of the material.



When the tool bit is separated from the material.



When the tool bit is touching the material.

10. Feed the Y1 axis by handle so that the tool tip is aligned with the circumference of the material.

11. Press the Input key.

12. Press the Escape key to finish the operation.

[Note]

- To enable operation while the door is open, press the Door open button to unlock the door, and set the Setting switch key to the "I" position. If this switch key is set to the "O" position, machine operation is prohibited while the door is open.
- The unit will only operate with the door open while the Start button is pressed.

  The axes speed at this time is limited to 2 m/min or less.
- Releasing the Start button stops axis feed in the hold state and the Start button lamp blinks (on for two seconds and off for one second). To restart the operation, press the Start button.
- If no operation is performed within 15 seconds after making menu selections, menu selections will automatically be canceled.

  Menu selections will also be canceled if a reset is performed during or after operations. To restart operations, be sure to again make selections with the menus.
- When performing adjustment for core (Y), diameter (X) and center (X) using the handle, selection of multiplication ratio ×100 is not possible.
- The menu key [Man. Set] is disabled if no program is selected.

6.2.7.5 Core (core down)

Adjust the core height of the gang tool and rotary tool drill. Move the tool or the rotary tool so that the tool rake face or the tip of rotary tool drill is aligned with the circumference of a material.

[Procedure]

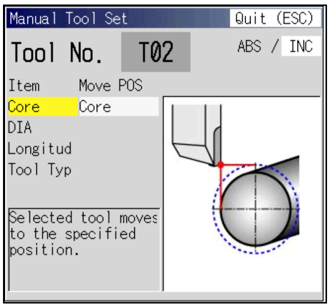
1. Press the Preparation key PREPA.

   The Preparation key lamp lights and the Preparation screen appears.

2. Use the Arrow keys ⬆ ⬇ or the Tab keys TAB ⬅ TAB ➡ to move the cursor to the core of the tool to be set.

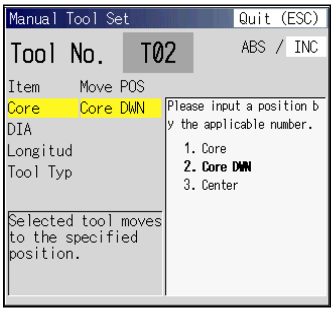
3. Press the menu key [Man. Set].

   The Manual Tool Set screen appears. Confirm that the cursor is at the core. If the cursor is not at the core, use the Arrow keys ⬆ ⬇ to move the cursor to the Core.



4. Press the Arrow key ⬆. Two cursors appear and the list showing the move positions is displayed. Enter the selection number of "Core DWN" and press the Input key INPUT.

   The Move POS setting is recorded as a default, so once Core DWN is set it is valid for the next tool setting.

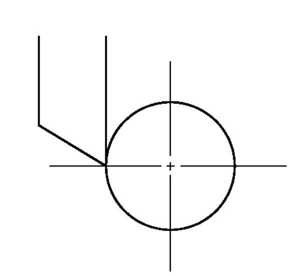


5. Make sure that the material is housed in the guide bushing.

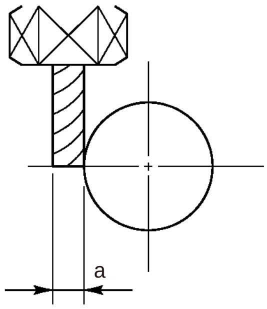
6. Make sure that all doors are closed, and press the Start button.
   The Start button lamp blinks on and off, and the doors will lock.

7. Press the Start button .

The Start button lamp lights and the tool moves to the circumference position of the material. The X1 axis is moved to X=0.

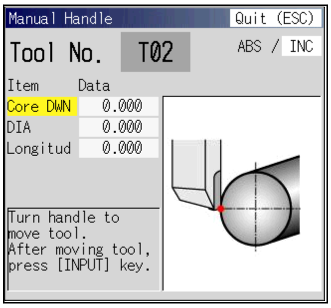


For the gang tool



For the rotary tool

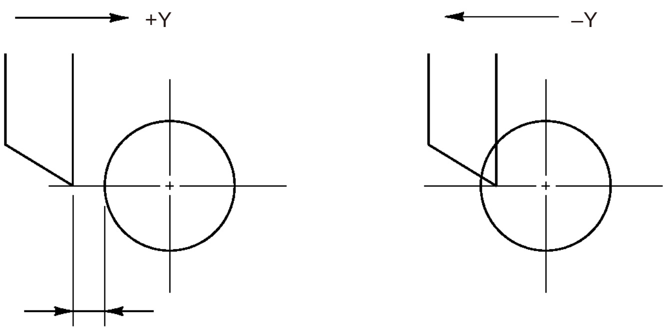
8. When operations are completed, the Start button lamp turns off and the screen changes to the Manual Handle screen.



9. Press the Door open button to unlock the door, and open the door.

10. Pull out the material and check whether the tool tip is aligned with the circumference of the material.

11. Adjust the core height in the Y1-axis direction by handle so that the tool tip is aligned with the circumference of the material.



12. Press the Input key .

If you set a rotary tool, perform the adjustment by manually entering tool diameter (a) to be added. After making this adjustment, revert the diameter value.

13. Press the Escape key to finish the operation.

CAUTION

Pull out the material carefully. If the tool interferes with the material, this could damage the tool and cause personal injury.
If you perform this function while the material is extending out of the guide bushing and the tool is attached next to the tool to be set, the tool may interfere with the material and could result in damage of the tool.

[Note]

- The X1 axis moves in the direction in which the selected tool cuts the material.
- The Y1 axis moves in the direction which intersects the cutting direction.
- To enable operation while the door is open, press the Door open button to unlock the door, and set the Setting switch key to the "I" position. If this switch key is set to the "O" position, machine operation is prohibited while the door is open.
- The unit will only operate with the door open while the Start button is pressed.

  The axes speed at this time is limited to 2 m/min or less.
- Releasing the Start button stops axis feed in the hold state and the Start button lamp blinks (on for two seconds and off for one second). To restart the operation, press the Start button.
- If no operation is performed within 15 seconds after making menu selections, menu selections will automatically be canceled.

  Menu selections will also be canceled if a reset is performed during or after operations. To restart operations, be sure to again make selections with the menus.
- When performing adjustment for core (Y), diameter (X) and center (X) using the handle, selection of multiplication ratio ×100 is not possible.
- The menu key [Man. Set] is disabled if no program is selected.

6.2.7.6 Core (center)

Adjust the core height of the gang tool. Move the tool so that the tool rake face is aligned with the center of the material.

[Procedure]

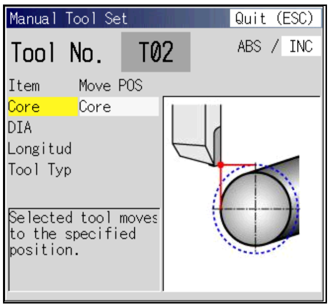
1. Press the Preparation key PREPA..

The Preparation key lamp lights and the Preparation screen appears.

2. Use the Arrow keys ⬆ ⬇ or the Tab keys TAB ⬅ TAB ➡ to move the cursor to the core of the tool to be set.

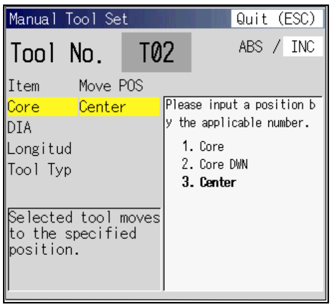
3. Press the menu key [Man. Set].

The Manual Tool Set screen appears. Confirm that the cursor is at the core. If the cursor is not at the core, use the Arrow keys ⬆ ⬇ to move the cursor to the Core.



4. Press the Arrow key ⬆ Two cursors appear and the list showing the move positions is displayed. Enter the selection number of "Center" and press the Input key INPUT.

The Move POS setting is saved as a default. Once you select "Center" when setting a tool, therefore, the setting applies also to the tool you set next.



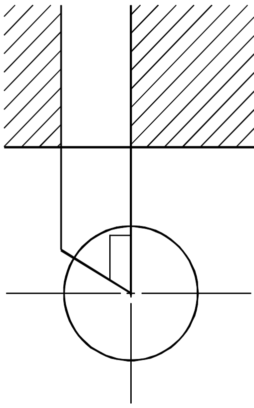
5. Make sure that the material is housed in the guide bushing.

6. Make sure that all doors are closed, and press the Start button.
The Start button lamp blinks on and off, and the doors will lock.

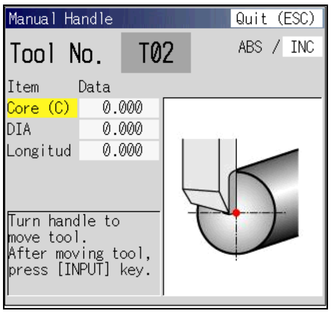
7. Press the Start button.

The Start button lamp lights and the tool moves to the center position of the material.

The X1 and Y1 axes are moved to X0, Y0.

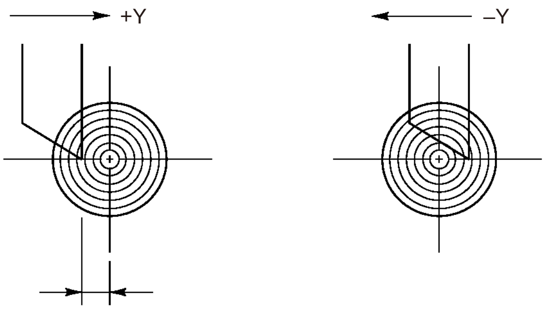


8. When operations are completed, the Start button lamp turns off and the screen changes to the Manual Handle screen.



9. Temporarily attach the tool to the tool holder and confirm that the tool tip is aligned with the material cut-off mark.

10. Move the Y1 axis in handle feed to make adjustment so that the tip of the tool bit is aligned with the center of the cut-off mark at the end of the material.



11. Press the Input key.

12. Press the Escape key to finish the operation.

[Note]

- The X1 axis moves in the direction in which the selected tool cuts the material.
- The Y1 axis moves in the direction which intersects the cutting direction.
- To enable operation while the door is open, press the Door open button to unlock the door, and set the Setting switch key to the "I" position. If this switch key is set to the "O" position, machine operation is prohibited while the door is open.
- The unit will only operate with the door open while the Start button is pressed.

  The axes speed at this time is limited to 2 m/min or less.
- Releasing the Start button stops axis feed in the hold state and the Start button lamp blinks (on for two seconds and off for one second). To restart the operation, press the Start button.
- If no operation is performed within 15 seconds after making menu selections, menu selections will automatically be canceled.

  Menu selections will also be canceled if a reset is performed during or after operations. To restart operations, be sure to again make selections with the menus.
- When performing adjustment for core (Y), diameter (X) and center (X) using the handle, selection of multiplication ratio $\times$100 is not possible.
- The menu key [Man. Set] is disabled if no program is selected.

6.2.7.7 Diameter (gang tool / cross rotary tool)

Position the tool or the cross rotary tool so that the tip of the tool or the cross rotary tool is aligned with the outer diameter of the material.

[Procedure]

To set the tool or cross rotary tool to the tool holder:

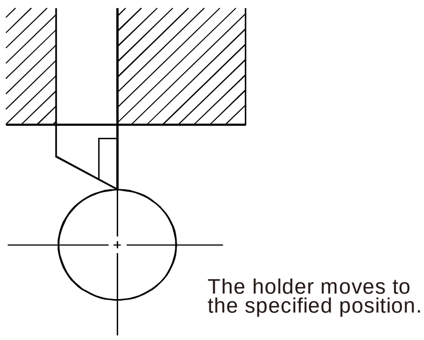
1. Press the Preparation key PREPA..

   The Preparation key lamp lights and the Preparation screen appears.

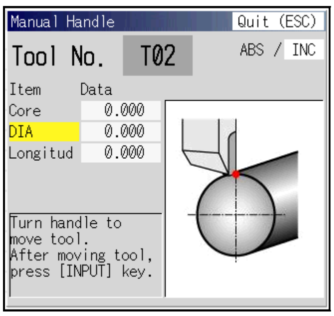
2. Use the Arrow keys or the Tab keys TAB TAB to move the cursor to the DIA of the tool to be set.
3. Make sure that the material is housed in the guide bushing.
4. Press the menu key [Man. Set].

   The Manual Tool Set screen appears. Confirm that the cursor is at the DIA. If the cursor is not at the DIA, use the Arrow keys to move the cursor to the DIA.

5. Make sure that all doors are closed, and press the Start button .
   The Start button lamp blinks on and off, and the doors will lock.
6. Press the Start button .
   The Start button lamp lights and the selected tool (gang tool post) moves to the position of the material outer diameter.

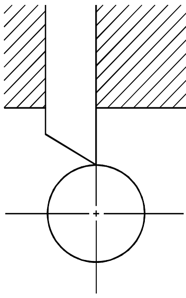


7. When operations are completed, the Start button lamp turns off and the screen changes to the Manual Handle screen.



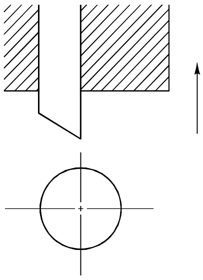
8. Press the Door open button to unlock the door, and open the door.

9. Pull out the material and bring the tool tip or cross rotary tool tip into light contact with the outer diameter of the material and temporarily secure the tool or the cross rotary tool.

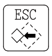


10. Move the X1 axis in the plus (+) direction by handle to relieve the tool or the cross rotary tool from the material. Or perform positioning point return operation.

See *<6.2.2 Positioning point>*



11. Fix the tool or cross rotary tool in the tool holder firmly.

12. Press the Escape key 

to finish the operation.

[Procedure]

To make fine adjustment for the fixed tool or cross rotary tool in the diametric direction

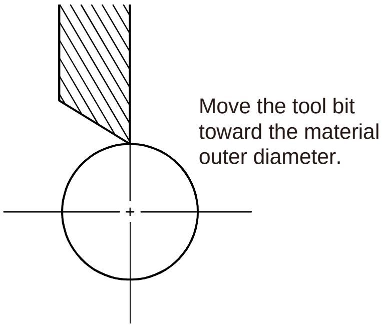
1. Press the Preparation key PREPA..

   The Preparation key lamp lights and the Preparation screen appears.

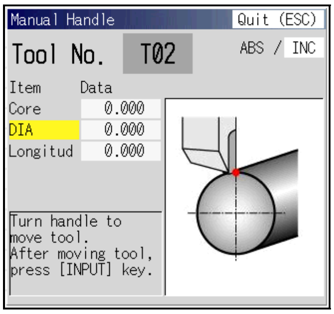
2. Use the Arrow keys ⬆ ⬇ or the Tab keys TAB ⬅ TAB ➡ to move the cursor to the DIA of the tool to be set.
3. Make sure that the material is housed in the guide bushing.
4. Press the menu key [Man. Set].

   The Manual Tool Set screen appears. Confirm that the cursor is at the DIA. If the cursor is not at the DIA, use the Arrow keys ⬆ ⬇ to move the cursor to the DIA.

5. Make sure that all doors are closed, and press the Start button.
   The Start button lamp blinks on and off, and the doors will lock.
6. Press the Start button.
   The Start button lamp lights and the tool moves to the outer diameter of the machining material.

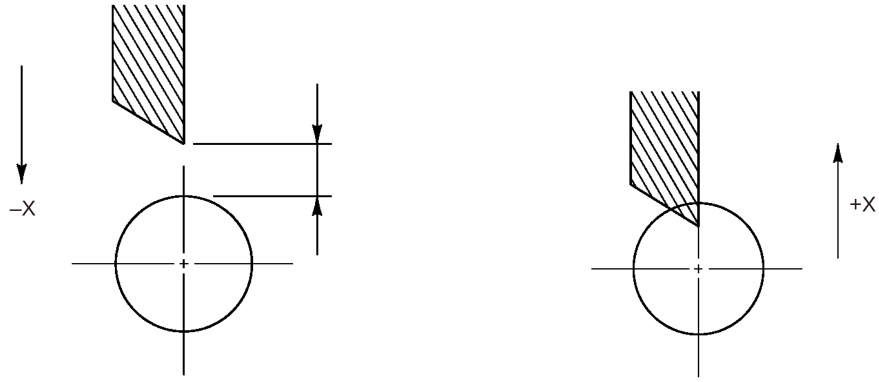


7. When operations are completed, the Start button lamp turns off and the screen changes to the Manual Handle screen.



8. Press the Door open button to unlock the door, and open the door.
9. Pull out the material.

10. Move the X1 axis by handle to align the tool tip or the cross rotary tool tip with the outer diameter of the material.



11. Press the Input key .

12. Press the Escape key to finish the operation.

[Procedure]

To set the tool or cross rotary tool to the tool holder with larger protrusion amount than normal
(When performing deep hole drilling for through hole machining using a cross rotary tool)

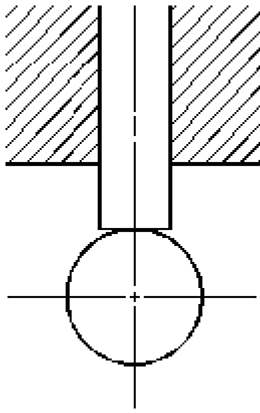
1. Press the Preparation key PREPA. [⇥].

   The Preparation key lamp lights and the Preparation screen appears.

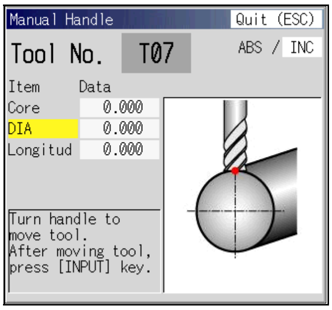
2. Use the Arrow keys [⬆][⬇] or the Tab keys [TAB ⇐][TAB ⇒] to move the cursor to the DIA of the tool to be set.
3. Make sure that the material is housed in the guide bushing.
4. Press the menu key [Man. Set].

   The Manual Tool Set screen appears. Confirm that the cursor is at the DIA. If the cursor is not at the DIA, use the Arrow keys [⬆][⬇] to move the cursor to the DIA.

5. Make sure that all doors are closed, and press the Start button [Start].
   The Start button lamp blinks on and off, and the doors will lock.
6. Press the Start button [Start].
   The Start button lamp lights and the tool moves to the outer diameter of the machining material.

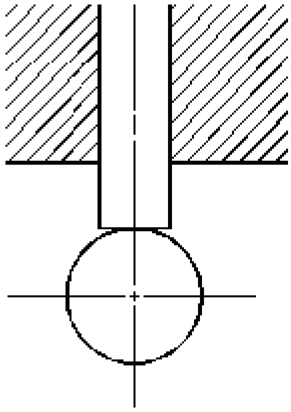


7. When operations are completed, the Start button [Start] lamp turns off and the screen changes to the Manual Handle screen.



8. Press the Door open button [Door] to unlock the door, and open the door.
9. Pull out the material.

10. Move the X1 axis in the plus (+) X direction by handle to provide protrusion amount necessary for machining and temporarily fix the tool or the cross rotary tool in the tool holder in the state the tool tip or the cross rotary tool tip is in contact with the outer diameter of the material.



11. Press the Input key INPUT.

12. Move the X1 axis in the plus (+) direction by handle to relieve the tool or the cross rotary tool from the material.

13. Fix the tool or cross rotary tool in the tool holder firmly.

14. Press the Escape key ESC to finish the operation.

[Note]

- To enable operation while the door is open, press the Door open button to unlock the door, and set the Setting switch key to the "|" position. If this switch key is set to the "O" position, machine operation is prohibited while the door is open.
- The unit will only operate with the door open while the Start button is pressed.

  The axes speed at this time is limited to 2 m/min or less.
- Releasing the Start button stops axis feed in the hold state and the Start button lamp blinks (on for two seconds and off for one second). To restart the operation, press the Start button.
- If no operation is performed within 15 seconds after making menu selections, menu selections will automatically be canceled.

  Menu selections will also be canceled if a reset is performed during or after operations. To restart operations, be sure to again make selections with the menus.
- When performing adjustment for core (Y), diameter (X) and center (X) using the handle, selection of multiplication ratio ×100 is not possible.
- The menu key [Man. Set] is disabled if no program is selected.

6.2.7.8 Longitude (core) (drill)

This function operates the machine so that the drill, to be mounted in the drilling tool holder on the gang tool post (T08 to T10, T11 to T17) or in the drilling tool holder on the opposite tool post (T21 to T23) , is mounted in the tool holder or the spindle with the specified overhang amount.

[Procedure]

To set the right-handed cut-off tool (T1****):

1. Press the Preparation key PREPA.

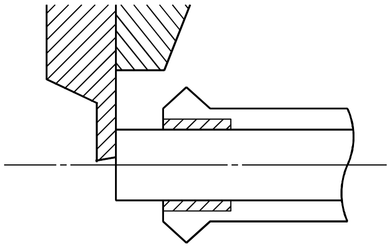
The Preparation key lamp lights and the Preparation screen appears.

2. Turn off the bar loader torque and open the chuck.

3. Perform cut-off operation on the end face of the material. (End the operation with the chuck closed.)

See *<6.2.8 Cut-off machining>*.

4. Align the material with the tip of the cut-off tool. Perform cut-off and the operation ends in the state the material and the cut-off tool are in contact with each other.



5. Move the cut-off tool to the positioning point.

6. Use the Arrow keys or the Tab keys TAB TAB to move the cursor to the core of the tool to be set.

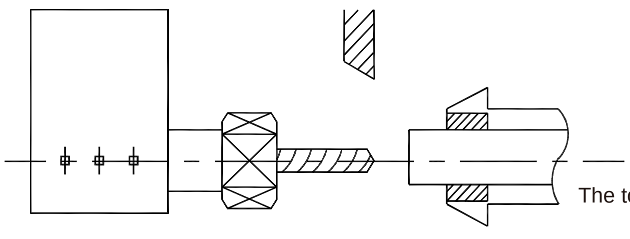
7. Attach the tool to the sleeve and insert the sleeve into the tool post deeply.

8. Press the menu key [Man. Set].

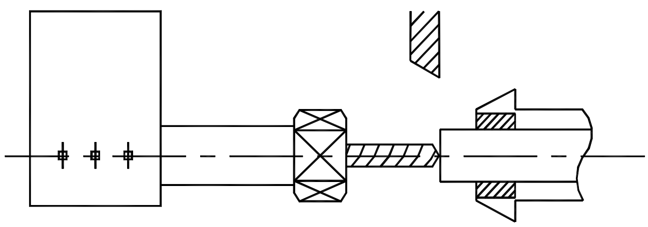
The Manual Tool Set screen appears. Confirm that the cursor is at the core. If the cursor is not at the core, use the Arrow keys to move the cursor to the Core.

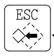
9. Make sure that all doors are closed, and press the Start button.
The Start button lamp blinks on and off, and the doors will lock.

10. Press the Start button.
The Start button lamp lights. The Z2 axis returns to return position, and the selected tool is positioned.



11. Bring the tool tip into contact with the end face of the material and temporarily tighten the sleeve.



12. Press the Escape key 

.

The Manual Tool Set screen closes.

13. Move the material away from the tool, then tighten up the sleeve firmly.

[Procedure]

To set the left-handed cut-off tool (T1****):

1. Press the Preparation key PREPA..

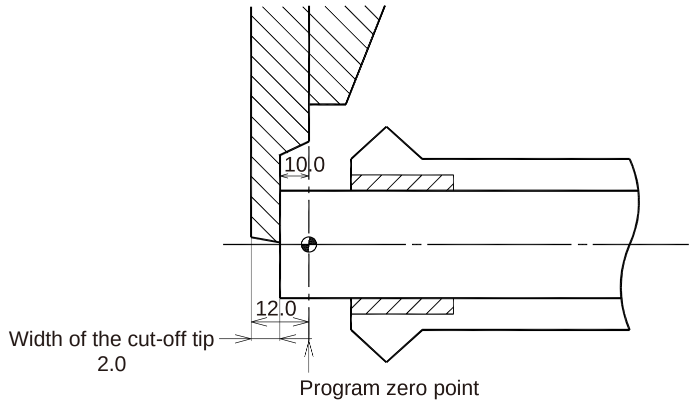
The Preparation key lamp lights and the Preparation screen appears.

2. Turn off the bar loader torque and open the chuck.

3. Perform cut-off operation on the end face of the material. (End the operation with the chuck closed.)

See <*6.2.8 Cut-off machining*>.

4. Assume that the tool shank width of the cut-off tool is 12.0 mm and the width of the cut-off tip is 2.0 mm. When the material is cut off with this cut-off tool, the end face of the material extends 10.0 mm from the program zero point.



5. Move the cut-off tool to the positioning position.

6. Press the MDI screen key MDI.

7. Enter "G0 W–10.0" in $1 program and, after closing the door, execute this program.

[Note]

The value of "G0 W–10.0" varies according to the tool shank width and tip width of the cut-off tool.

8. Press the Preparation key PREPA..

The Preparation key lamp lights and the Preparation screen appears.

9. Use the Arrow keys ⬆ ⬇ or the Tab keys TAB ⬅ TAB ➡ to move the cursor to the core of the tool to be set.

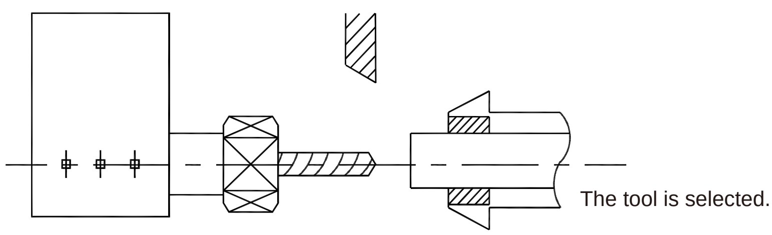
10. Attach the tool to the sleeve and insert the sleeve into the tool post deeply.

11. Press the menu key [Man. Set].

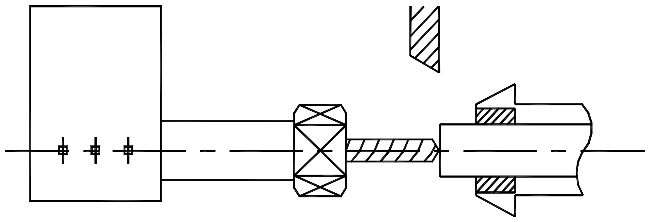
The Manual Tool Set screen appears. Confirm that the cursor is at the core. If the cursor is not at the core, use the Arrow keys ⬆ ⬇ to move the cursor to the Core.

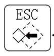
12. Make sure that all doors are closed, and press the Start button.
The Start button lamp blinks on and off, and the doors will lock.

13. Press the Start button.
The Start button lamp lights. The Z2 axis returns to the stroke end, the Z1 axis returns 1 mm, and after the selected tool is positioned, the Z1 axis advances 1 mm.



14. Bring the tool tip into contact with the end face of the material and temporarily tighten the sleeve.



15. Press the Escape key 

.

The Manual Tool Set screen closes.

16. Move the material away from the tool, then tighten up the sleeve firmly.

[Procedure]

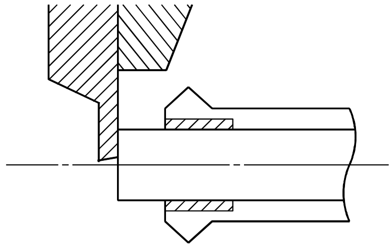
To set the right-handed cut-off tool (T2****):

1. Press the Preparation key PREPA.
The Preparation key lamp lights and the Preparation screen appears.

2. Turn off the bar loader torque and open the chuck.

3. Perform cut-off operation on the end face of the material. (End the operation with the chuck closed.)
See *<6.2.8 Cut-off machining>*.

4. Align the material with the tip of the cut-off tool. Perform cut-off and the operation ends in the state the material and the cut-off tool are in contact with each other.



5. Move the cut-off tool to the positioning point.

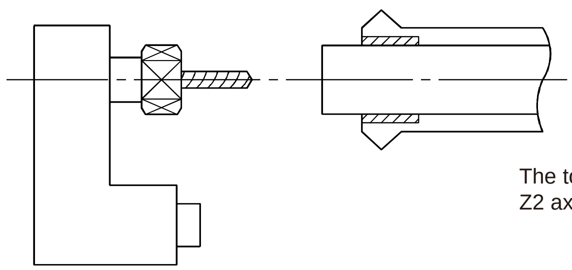
6. Use the Arrow keys or the Tab keys TAB TAB to move the cursor to the Longitud of the tool to be set.

7. Attach the tool to the sleeve and insert the sleeve into the tool post deeply.

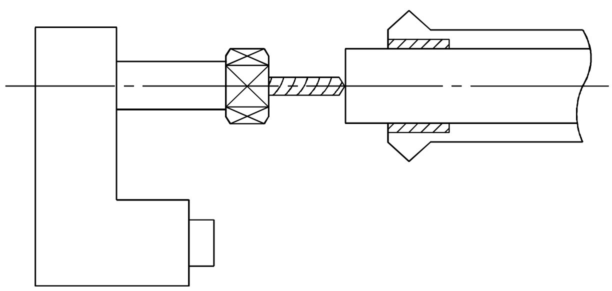
8. Press the menu key [Man. Set].
The Manual Tool Set screen appears. Confirm that the cursor is at the Longitud. If the cursor is not at the Longitud, use the Arrow keys to move the cursor to the Longitud.

9. Make sure that all doors are closed, and press the Start button.
The Start button lamp blinks on and off, and the doors will lock.

10. Press the Start button.
The Start button lamp lights and the gang tool post moves to the return position.
The selected tool is positioned in X2-axis direction, the Z2 axis moves forward to the set position.



11. Bring the tool tip into contact with the end face of the material and temporarily tighten the sleeve.



12. Press the menu key [OPP-RET], and press the Start button.
The Start button lamp blinks on and off, and the doors will lock.

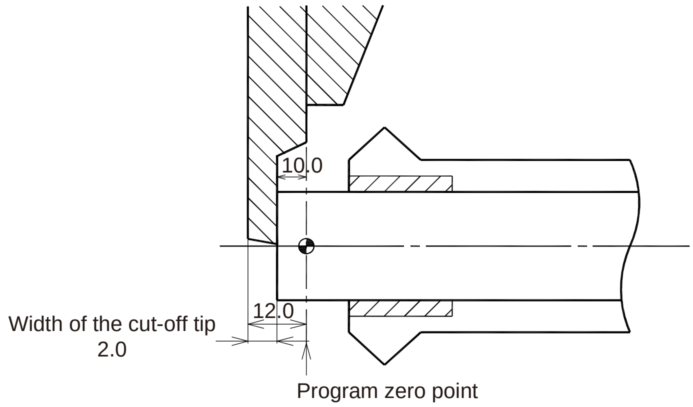
13. Press the Start button .
The Start button lamp lights and the opposite tool post returns to the Z2 axis zero point.

14. Tighten up the sleeve firmly.

[Procedure]

To set the left-handed cut-off tool (T2****):

1. Press the Preparation key PREPA. .
   The Preparation key lamp lights and the Preparation screen appears.
2. Turn off the bar loader torque and open the chuck.
3. Perform cut-off operation on the end face of the material. (End the operation with the chuck closed.)
   See *<6.2.8 Cut-off machining>*.
4. Assume that the tool shank width of the cut-off tool is 12.0 mm and the width of the cut-off tip is 2.0 mm. When the material is cut off with this cut-off tool, the end face of the material extends 10.0 mm from the program zero point.

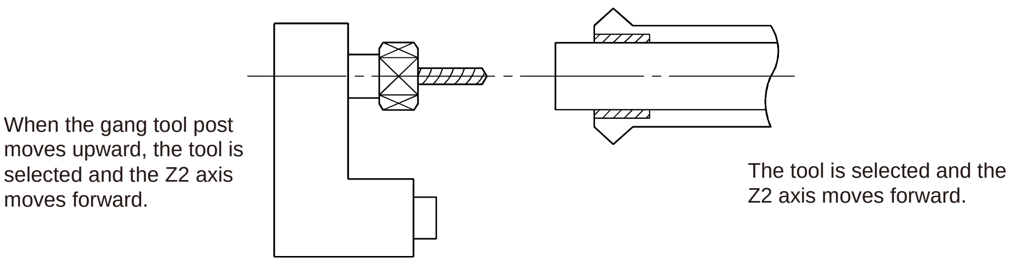


5. Move the cut-off tool to the positioning position.
6. Press the MDI screen key MDI .
7. Enter "G0 W–10.0" in $1 program and, after closing the door, execute this program.

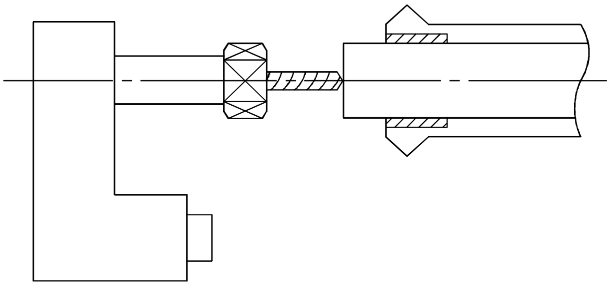
[Note]

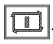
The value of "G0 W–10.0" varies according to the tool shank width and tip width of the cut-off tool.

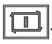
8. Press the Preparation key PREPA. .
   The Preparation key lamp lights and the Preparation screen appears.
9. Use the Arrow keys ⬆ ⬇ or the Tab keys TAB ⬅ TAB ➡ to move the cursor to the Longitud of the tool to be set.
10. Attach the tool to the sleeve and insert the sleeve into the tool post deeply.
11. Press the menu key [Man. Set].
    The Manual Tool Set screen appears. Confirm that the cursor is at the Longitud. If the cursor is not at the Longitud, use the Arrow keys ⬆ ⬇ to move the cursor to the Longitud.
12. Make sure that all doors are closed, and press the Start button .
    The Start button lamp blinks on and off, and the doors will lock.
13. Press the Start button .
    The Start button lamp goes on, and the gang tool post moves to the retract point. The selected tool is positioned in X2-axis direction, the Z2 axis moves forward to the set position.



14. Bring the tool tip into contact with the end face of the material and temporarily tighten the sleeve.

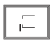
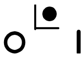
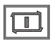


15. Press the menu key [OPP-RET], and press the Start button .
The Start button lamp blinks on and off, and the doors will lock.

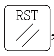
16. Press the Start button .
The Start button lamp lights and the opposite tool post returns to the Z2 axis zero point.

17. Tighten up the sleeve firmly.

[Note]

- To enable operation while the door is open, press the Door open button  to unlock the door, and set the Setting switch key  to the "|" position. If this switch key is set to the "O" position, machine operation is prohibited while the door is open.
- The unit will only operate with the door open while the Start button  is pressed.

  The axes speed at this time is limited to 2 m/min or less.
- Releasing the Start button stops axis feed in the hold state and the Start button lamp blinks (on for two seconds and off for one second). To restart the operation, press the Start button.
- If no operation is performed within 15 seconds after making menu selections, menu selections will automatically be canceled.

  Menu selections will also be canceled if a reset is performed during or after operations. To restart operations, be sure to again make selections with the menus.
- When performing adjustment for core (Y), diameter (X) and center (X) using the handle, selection of multiplication ratio ×100 is not possible.
- If the operation is interrupted by pressing the Reset key , the material (Z1 axis) may remain the position retracted 1.0 mm. Set the material to the proper position again.
- The menu key [Man. Set] is disabled if no program is selected.

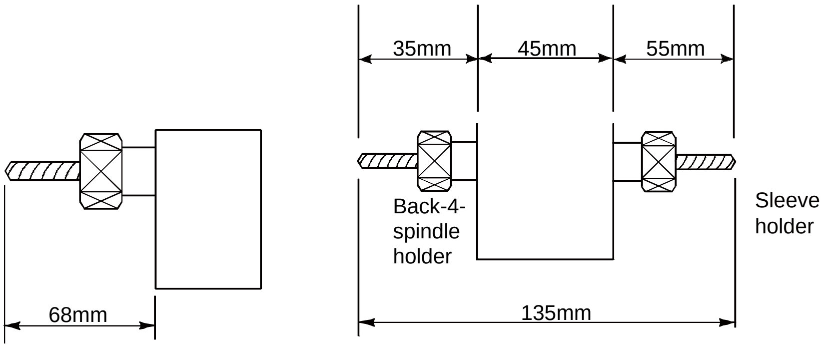
6.2.7.9 Longitude (back drill)

This function sets the drill to protrude the specified length. The length of the back drilling tool is given below.

For information on protruding length of tooling other than those below, refer to *<Chapter 17 Tooling>*.

Protruding drill length

Back drilling tool (T3***)   Front/back drilling tool (T1***, T5***)

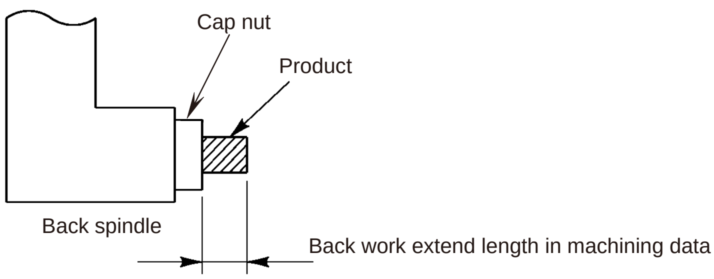


[Procedure]

Mounting a back drilling tool (T3***)

1. Chuck a product (material) on the back spindle.

Usually, chuck that product which has been front-machined by one-cycle operation and its overall length is corrected. Make sure that the product is protruding from the end of the back spindle (chucked at the right position).



2. Measure the length of the protrusion of the product from the back chuck and check whether the measured length matches the value entered in machining data "Back Spindle Chuck POS."

(For machining data, see *<6.1.3 Required fields for machining data>* and *<6.1.4 Checking machining data>*.)

3. Press the Preparation key PREPA..

The Preparation key lamp lights and the Preparation screen appears.

4. Use the Arrow keys ⬆ ⬇ or the Tab keys TAB ⬅ TAB ➡ to move the cursor to Longitud for the tool you want to set.

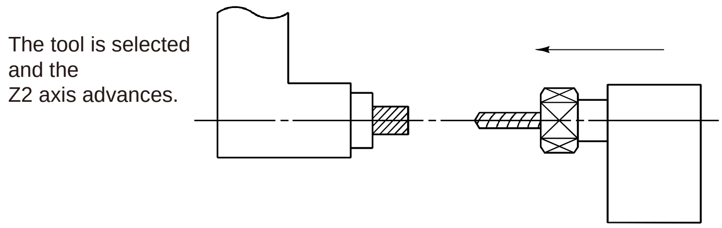
5. Put the sleeve deep into the tool post.

6. Press the menu key [Man. Set].

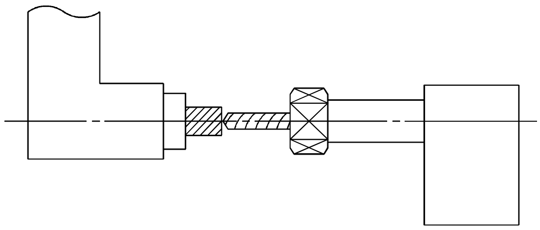
The Manual Tool Set screen appears. Confirm that the cursor is at the Longitud. If the cursor is not at the Longitud, use the Arrow keys ⬆ ⬇ to move the cursor to the Longitud.

7. Make sure that all doors are closed, and press the Start button.
The Start button lamp blinks on and off, and the doors will lock.

8. Press the Start button .
The Start button lamp lights, the back spindle (Z2 axis) moves return position to select the desired tool and then advances to the standard set position.



9. Bring the tool tip into contact with the end face of the material and temporarily tighten the sleeve.



10. Press the menu key [OPP-RET], and press the Start button .
The Start button lamp blinks on and off, and the doors will lock.

11. Press the Start button .
The Start button lamp lights and the opposite tool post returns to the Z2 axis zero point.

12. Tighten up the sleeve firmly.

[Note]

- To enable operation while the door is open, press the Door open button to unlock the door, and set the Setting switch key to the "|" position. If this switch key is set to the "O" position, machine operation is prohibited while the door is open.
- The unit will only operate with the door open while the Start button is pressed.

  The axes speed at this time is limited to 2 m/min or less.
- Releasing the Start button stops axis feed in the hold state and the Start button lamp blinks (on for two seconds and off for one second). To restart the operation, press the Start button.
- If no operation is performed within 15 seconds after making menu selections, menu selections will automatically be canceled.

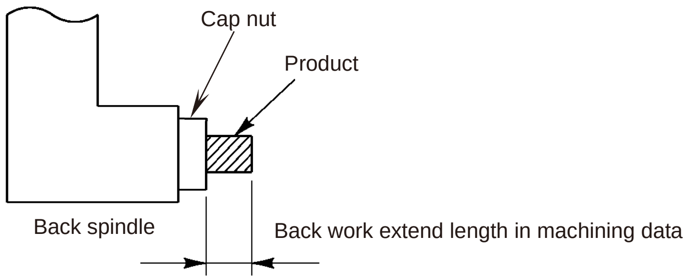
  Menu selections will also be canceled if a reset is performed during or after operations. To restart operations, be sure to again make selections with the menus.
- When performing adjustment for core (Y), diameter (X) and center (X) using the handle, selection of multiplication ratio ×100 is not possible.
- The tool may cause interference during back machining if the actual length of the protrusion from the back spindle is different from the chucking position entered in machining data.
- The menu key [Man. Set] is disabled if no program is selected.
- When GDF1207 is used together with U128B, set the protrusion length of tool on GDF1207 at back spindle side at the position 30 mm or shorter from the end face of the holder.

[Procedure]

To mount a drill in the back gang tool post (T51 to T58)

1. Chuck a product (material) on the back spindle.

   Usually, chuck that product which has been front-machined by one-cycle operation and its overall length is corrected.



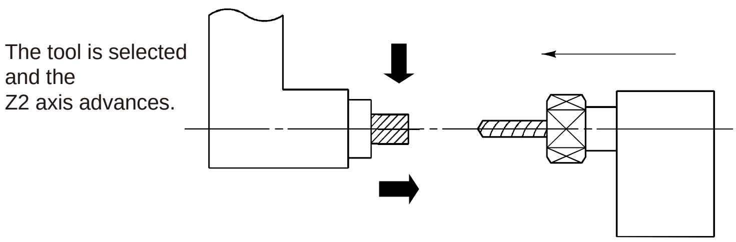
2. Measure the length of the protrusion of the product from the back chuck and check whether the measured length matches the value entered in machining data "Back Spindle Chuck POS."

   (For machining data, see <*6.1.3 Required fields for machining data*> and <*6.1.4 Checking machining data*>.)

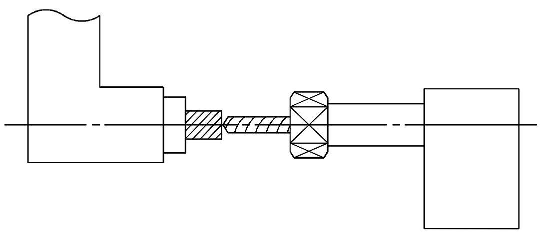
3. Press the Preparation key PREPA. .

   The Preparation key lamp lights and the Preparation screen appears.

4. Use the Page keys , Arrow keys , or the Tab keys TAB TAB to move the cursor to the Longitud of the desired tool.
5. Put the sleeve deep into the tool post.
6. Press the menu key [Man. Set].
   The Manual Tool Set screen appears. Confirm that the cursor is at the Longitud. If the cursor is not at the Longitud, use the Arrow keys to move the cursor to the Longitud.
7. Make sure that all doors are closed, and press the Start button .
   The Start button lamp blinks on and off, and the doors will lock.
8. Press the Start button .
   The Start button lamp lights. The Z2 axis returns to the stroke end, the X2 axis moves to the spindle center line position, and after the selected tool is positioned, the Z2 axis advances to the standard setting position.



9. Bring the tool tip into contact with the end face of the material and temporarily tighten the sleeve.



10. Press the menu key [OPP-RET], and press the Start button .
    The Start button lamp blinks on and off, and the doors will lock.

11. Press the Start button.
The Start button lamp lights and the opposite tool post returns to the Z2 axis zero point.

12. Tighten up the sleeve firmly.

[Note]

- To enable operation while the door is open, press the Door open button to unlock the door, and set the Setting switch key to the "I" position. If this switch key is set to the "O" position, machine operation is prohibited while the door is open.
- The unit will only operate with the door open while the Start button is pressed.

  The axes speed at this time is limited to 2 m/min or less.
- Releasing the Start button stops axis feed in the hold state and the Start button lamp blinks (on for two seconds and off for one second). To restart the operation, press the Start button.
- If no operation is performed within 15 seconds after making menu selections, menu selections will automatically be canceled.

  Menu selections will also be canceled if a reset is performed during or after operations. To restart operations, be sure to again make selections with the menus.
- When performing adjustment for core (Y), diameter (X) and center (X) using the handle, selection of multiplication ratio ×100 is not possible.
- The tool may cause interference during back machining if the actual length of the protrusion from the back spindle is different from the chucking position entered in machining data.
- The menu key [Man. Set] is disabled if no program is selected.

[Procedure]

Mounting a slitting cutter (GSS**30) for back machining (T3400)

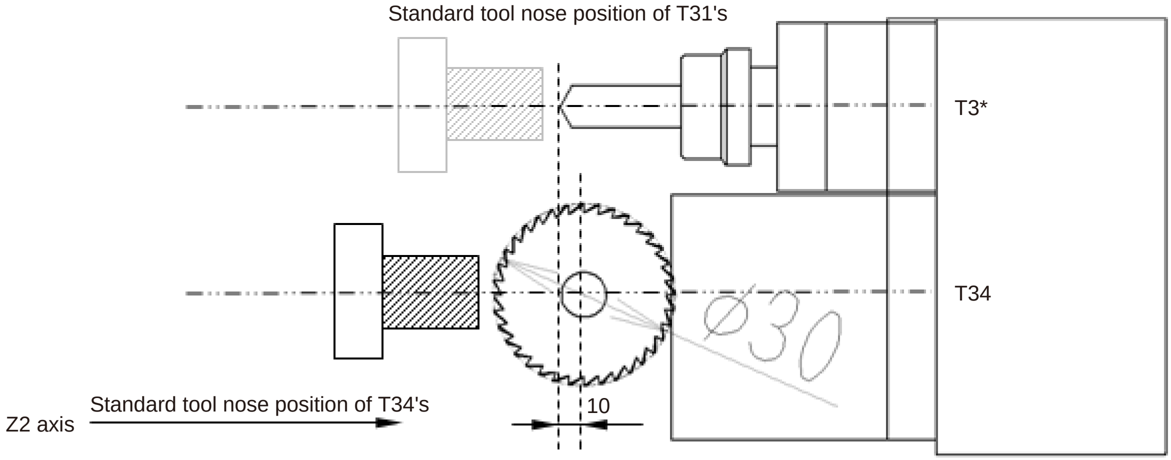
The back slitting cutter (GSS**30) is protruded from the standard tool nose position of T31's.
Calculate the protrusion length of the slitting cutter in longitudinal direction, and enter the value in tool set data.

To mount the slitting cutter (GSS**30), specify the value calculated by the following formula for "Longitud" field on Manual Tool Set screen.

CAUTION

Before mounting a slitting cutter, be sure to specify a value for setting tool in longitudinal direction according to the following formula. If the value is not specified correctly, the workpiece on back spindle may interfere with the slitting cutter.

Tool set position for longitudinal direction ＝ － { (Diameter of slitting cutter) / 2 }



1. Press the Preparation key PREPA..
   The Preparation key lamp lights and the Preparation screen appears.
2. Use the Arrow keys or the Tab keys to move the cursor to Longitud for the tool (T34) on which the slitting cutter is mounted.
3. Press the menu key [Man. Set].
   The Manual Tool Set screen appears. Confirm that the cursor is at the Longitud.
4. Enter the value obtained from the formula by using the sheet key, then press the Input key INPUT. On Preparation screen, make sure that the value shown in "Longitud (longitudinal setting)" field of T34 (on which the slitting cutter is mounted) is correct.

6.2.7.10 Centering the tool (drill) (boring)

Center the front drilling or back drilling tool.

[Note]

Since the machine is adjusted to center tools when assembled, perform this operation only when necessary.

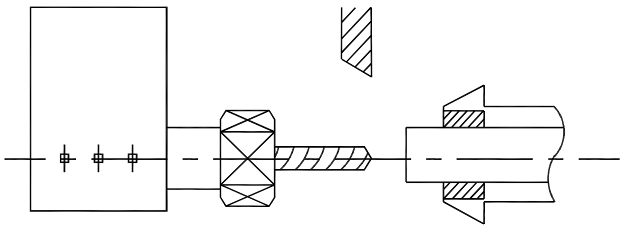
[Procedure]

To align the center of the drill in the front gang tool post (T11 to T18) with the center of material

1. Press the Preparation key PREPA.

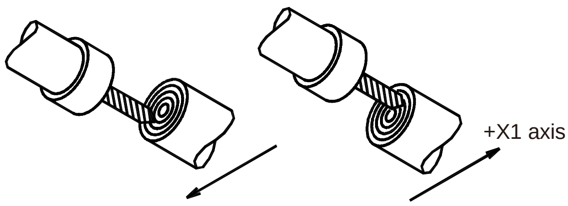
   The Preparation key lamp lights and the Preparation screen appears.
2. Use the Arrow keys ⬆ ⬇ or the Tab keys TAB ⬅ TAB ➡ to move the cursor to the Core or DIA for the tool you want to set.
3. Put the sleeve deep into the tool post.
4. Press the menu key [Man. Set].

   The Manual Tool Set screen appears. Make sure that the Core or DIA setting is highlighted with the cursor. If the cursor is not at the Core or DIA setting, use the Arrow keys ⬆ ⬇ to move the cursor to the Core or DIA setting.
5. Make sure that all doors are closed, and press the Start button.
   The Start button lamp blinks on and off, and the doors will lock.
6. Press the Start button.
   The Start button lamp lights. The material (Z1 axis) returns 1.0 mm, and after the selected tool is positioned, the material (Z1 axis) advances 1.0 mm.



The tool is selected.

7. Press the Door open button to unlock the door, and open the door.
8. Pull the material without letting its end face touch the tool.
9. Move the X1 or Y1 axis in handle feed so that the tip of the tool bit is aligned with the center of the cut-off mark at the end of the material.



10. Press the Input key INPUT.
11. Press the Escape key ESC.
    The Manual Tool Set screen closes.
12. Move the material away from the tool.

[Note]

- To enable operation while the door is open, press the Door open button to unlock the door, and set the Setting switch key to the "|" position. If this switch key is set to the "O" position, machine operation is prohibited while the door is open.
- The unit will only operate with the door open while the Start button is pressed.

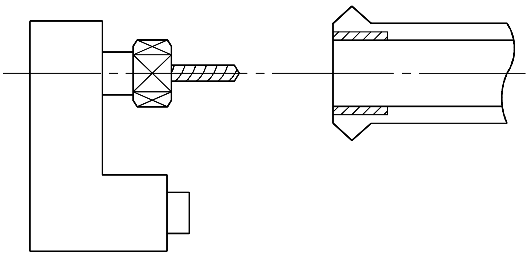
  The axes speed at this time is limited to 2 m/min or less.
- Releasing the Start button stops axis feed in the hold state and the Start button lamp blinks (on for two seconds and off for one second). To restart the operation, press the Start button.
- If no operation is performed within 15 seconds after making menu selections, menu selections will automatically be canceled.

  Menu selections will also be canceled if a reset is performed during or after operations. To restart operations, be sure to again make selections with the menus.
- When performing adjustment for core (Y), diameter (X) and center (X) using the handle, selection of multiplication ratio ×100 is not possible.
- The menu key [Man. Set] is disabled if no program is selected.

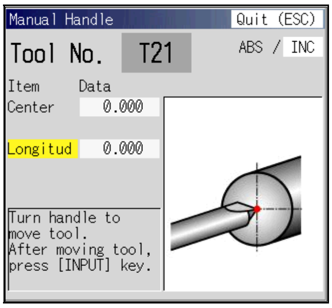
[Procedure]

To align the center of the front drill (T2***)

1. Press the Preparation key [PREPA.].
   The Preparation key lamp lights and the Preparation screen appears.
2. Use the Arrow keys [↑][↓] or the Tab keys [TAB ←][TAB →] to move the cursor to the center setting for the tool you want to set.
3. Put the sleeve deep into the tool post.
4. Press the menu key [Man. Set].
   The Manual Tool Set dialog box appears. Make sure that the center setting is highlighted with the cursor. If the cursor is not at the center setting, use the Arrow keys [↑][↓] to move the cursor to the center setting.
5. Make sure that all doors are closed, and press the Start button [ ].
   The Start button lamp blinks on and off, and the doors will lock.
6. Press the Start button [ ].
   The Start button lamp lights and the gang tool post moves to the return position.
   The selected tool is positioned in X2-axis direction, the Z2 axis moves forward to the set position.

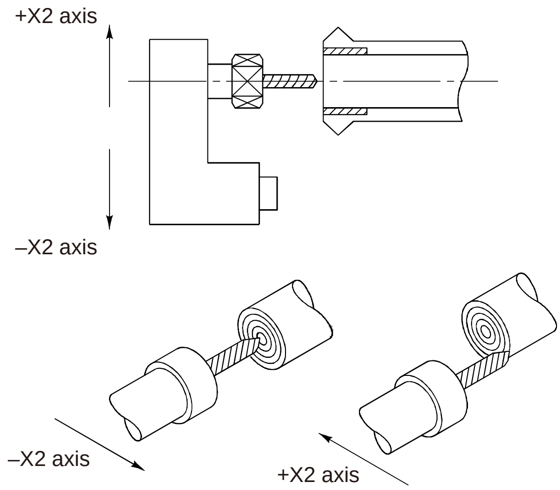


7. Use the Arrow keys [↑][↓] to move the cursor to the Longitud.
8. Use the handle to advance the Z2 axis 175.0 mm.



9. Move the cursor to Center by pressing the Arrow keys [↑][↓].
10. Press the Door open button [ ] to unlock the door, and open the door.
11. Pull the material without letting its end face touch the tool.

12. Move the X2 axis in handle feed so that the tip of the tool bit is aligned with the center of the cut-off mark at the end of the material.



13. Press the Input key .

14. Press the menu key [OPP-RET], make sure that all doors are closed, and press the Start button .
The Start button lamp blinks on and off, and the doors will lock.

15. Press the Start button .The Start button lamp lights and the opposite tool post returns to the Z2 axis zero point.

[Note]

- To enable operation while the door is open, press the Door open button to unlock the door, and set the Setting switch key O I to the "I" position. If this switch key is set to the "O" position, machine operation is prohibited while the door is open.
- The unit will only operate with the door open while the Start button is pressed.

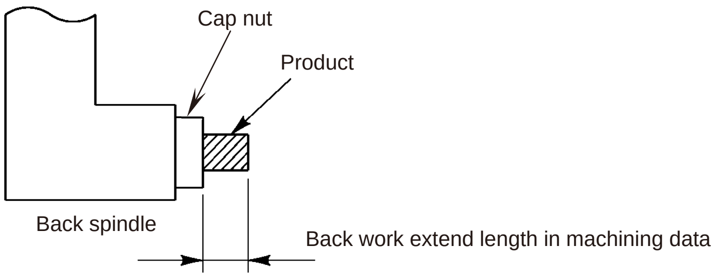
  The axes speed at this time is limited to 2 m/min or less.
- Releasing the Start button stops axis feed in the hold state and the Start button lamp blinks (on for two seconds and off for one second). To restart the operation, press the Start button.
- If no operation is performed within 15 seconds after making menu selections, menu selections will automatically be canceled.

  Menu selections will also be canceled if a reset is performed during or after operations. To restart operations, be sure to again make selections with the menus.
- When performing adjustment for core (Y), diameter (X) and center (X) using the handle, selection of multiplication ratio ×100 is not possible.
- The menu key [Man. Set] is disabled if no program is selected.

[Procedure]

To align the center of the back drill (T3***)

1. Chuck a product (material) on the back spindle.



2. Press the Preparation key [PREPA.].

The Preparation key lamp lights and the Preparation screen appears.

3. Use the Arrow keys [↑][↓] or the Tab keys [TAB ⇐][TAB ⇒] to move the cursor to the center setting for the tool you want to set.

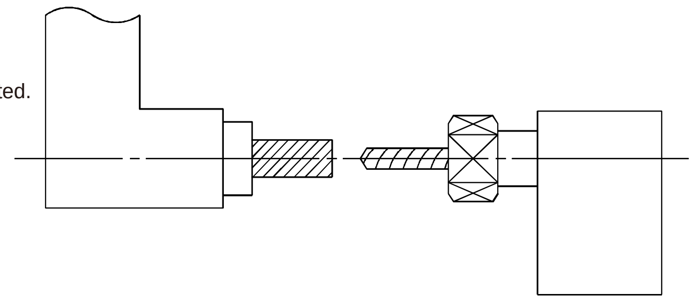
4. Press the menu key [Man. Set].

The Manual Tool Set dialog box appears. Make sure that the center setting is highlighted with the cursor. If the cursor is not at the center setting, use the Arrow keys [↑][↓] to move the cursor to the center setting.

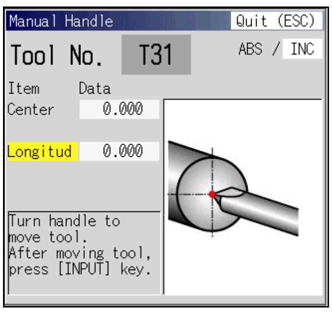
5. Make sure that all doors are closed, and press the Start button [I].
The Start button lamp blinks on and off, and the doors will lock.

6. Press the Start button [I].
The Start button lamp lights and the back spindle is selected for being centering.

The tool is selected.

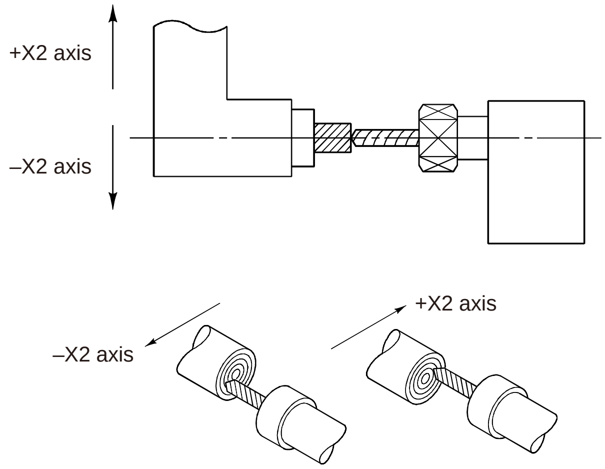


7. Move the cursor to Longitud by pressing the Arrow keys [↑][↓] and advance the Z2 axis by handle.



8. Move the cursor to Center by pressing the Arrow keys [↑][↓].

9. Move the X2 axis in handle feed so that the tip of the tool bit is aligned with the center of the cut-off mark at the end of the material.



10. Press the Input key [INPUT].

11. Press the menu key [OPP-RET], make sure that all doors are closed, and press the Start button.
The Start button lamp blinks on and off, and the doors will lock.

12. Press the Start button. The Start button lamp lights and the opposite tool post returns to the Z2 axis zero point.

[Note]

- To enable operation while the door is open, press the Door open button to unlock the door, and set the Setting switch key to the "I" position. If this switch key is set to the "O" position, machine operation is prohibited while the door is open.
- The unit will only operate with the door open while the Start button is pressed.

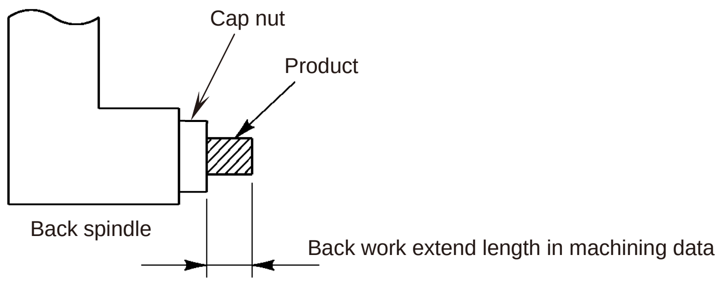
  The axes speed at this time is limited to 2 m/min or less.
- Releasing the Start button stops axis feed in the hold state and the Start button lamp blinks (on for two seconds and off for one second). To restart the operation, press the Start button.
- If no operation is performed within 15 seconds after making menu selections, menu selections will automatically be canceled.

  Menu selections will also be canceled if a reset is performed during or after operations. To restart operations, be sure to again make selections with the menus.
- When performing adjustment for core (Y), diameter (X) and center (X) using the handle, selection of multiplication ratio ×100 is not possible.
- The tool may cause interference during back machining if the actual length of the protrusion from the back spindle is different from the value entered in the back spindle work length of machining data.
- The tool may cause interference during back machining if the actual length of the protrusion from the back spindle is different from the back chuck extend length entered in machining data.
- The menu key [Man. Set] is disabled if no program is selected.

[Procedure]

To align the center of the drill in the back gang tool post (T5***)

1. Chuck a product (material) on the back spindle.



2. Press the Preparation key .

The Preparation key lamp lights and the Preparation screen appears.

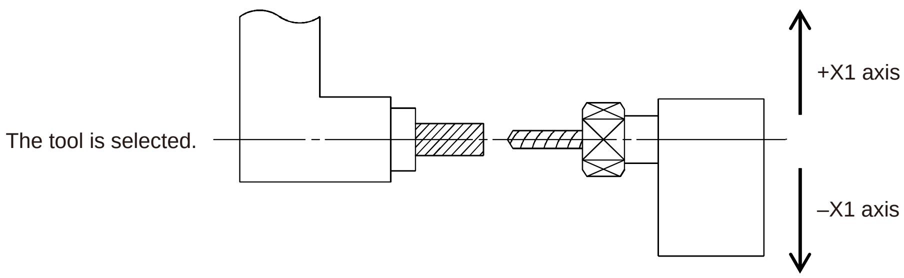
3. Use the Arrow keys or the Tab keys to move the cursor to the Core or DIA for the tool you want to set.

4. Press the menu key [Man. Set].

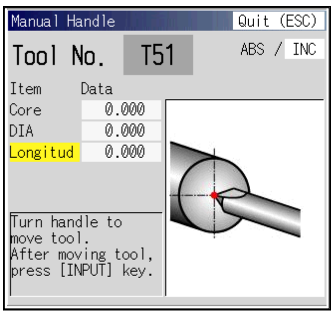
The Manual Tool Set screen appears. Make sure that the Core or DIA setting is highlighted with the cursor. If the cursor is not at the Core or DIA setting, use the Arrow keys to move the cursor to the Core or DIA setting.

5. Make sure that all doors are closed, and press the Start button .
The Start button lamp blinks on and off, and the doors will lock.

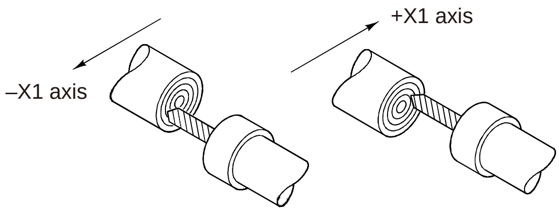
6. Press the Start button .
The Start button lamp lights, the Z2 axis retracts full stroke, the selected tool is positioned and then the back spindle is selected for being centering.



7. Move the cursor to Longitud by pressing the Arrow keys and advance the Z2 axis by handle.



8. Move the X1 or Y1 axis in handle feed so that the tip of the tool bit is aligned with the center of the cut-off mark at the end of the material.



9. ess the Input key [INPUT].

10. Press the menu key [OPP-RET], make sure that all doors are closed,
and press the Start button.
The Start button lamp blinks on and off, and the doors will lock.

11. Press the Start button.
The Start button lamp lights and the opposite tool post returns to the Z2 axis zero point.

[Note]

- To enable operation while the door is open, press the Door open button to unlock the door, and set the Setting switch key to the "I" position. If this switch key is set to the "O" position, machine operation is prohibited while the door is open.
- The unit will only operate with the door open while the Start button is pressed.

  The axes speed at this time is limited to 2 m/min or less.
- Releasing the Start button stops axis feed in the hold state and the Start button lamp blinks (on for two seconds and off for one second). To restart the operation, press the Start button.
- If no operation is performed within 15 seconds after making menu selections, menu selections will automatically be canceled.

  Menu selections will also be canceled if a reset is performed during or after operations. To restart operations, be sure to again make selections with the menus.
- When performing adjustment for core (Y), diameter (X) and center (X) using the handle, selection of multiplication ratio ×100 is not possible.
- The tool may cause interference during back machining if the actual length of the protrusion from the back spindle is different from the value entered in the back spindle work length of machining data.
- The tool may cause interference during back machining if the actual length of the protrusion from the back spindle is different from the back chuck extend length entered in machining data.
- The menu key [Man. Set] is disabled if no program is selected.

6.2.7.11 Tool Name

Specify the tool name.

[Note]

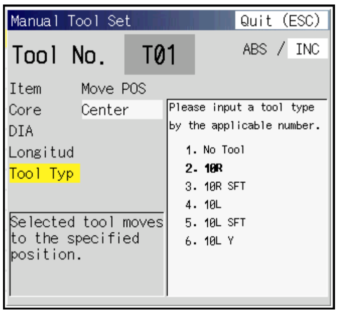
The standard tool name is selected at the program selection.Perform steps below to change the tool name.

[Procedure]

1. Press the Preparation key PREPA..
The Preparation key lamp lights and the Preparation screen appears.

2. Use the Arrow keys or the Tab keys TAB TAB to move the cursor to the tool to be set.

3. Press the menu key [Man. Set].
The Manual Tool Set dialog box appears. Make sure that the tool name setting is highlighted with the cursor. If the cursor is not at the tool name setting, use the Arrow keys to move the cursor to the tool name setting.



4. Enter the number assigned to the tool you want to select in the tool list, then press the Input key INPUT.
You can find the tool you want to select by scrolling the list with the Page key.

Example of the tool name

Name	Description
12R	12 sq. tool for forward rotation
12R SFT	12 sq. tool for right-shifted forward rotation
12L	12 sq. tool for reverse rotation
12L SFT	12 sq. tool for left-shifted forward rotation
12 LY	12 sq. tool for Y-axis cutting tool
10R	10 sq. tool for forward rotation
10R SFT	10 sq. tool for right-shifted forward rotation
10L	10 sq. tool for reverse rotation
10L SFT	10 sq. tool for left-shifted forward rotation
10 LY	10 sq. tool for Y-axis cutting tool
13R	13 sq. tool for forward rotation
13R SFT	13 sq. tool for right-shifted forward rotation
13L	13 sq. tool for reverse rotation
13L SFT	13 sq. tool for left-shifted forward rotation
13 LY	13 sq. tool for Y-axis cutting tool
Cross	Cross machining rotary tool
Face	End-face drilling tool, end-face rotary tool
Back	Back end-face drilling tool, back end-face rotary tool
Clined	Adjustable angle rotary tool
No Tool	No tool is mounted

6.2.8 Cut-off machining

Perform cut-off machining or end face turning based on the data set as machining data.

CAUTION

Described below are operation procedures when the Setting switch is set to "O".

If you are going to work inside the machine, press the Door open button to unlock the door.

The door cannot be unlocked unless the following conditions are met.

All spindles have stopped.

All servo axes have stopped.

The machine is not operating.

The coolant is not discharged:

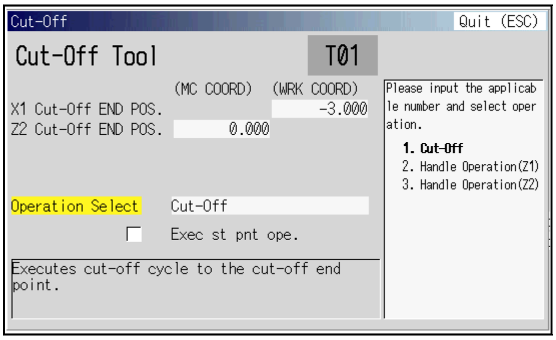
The Door open button is pressed.

[Procedure]

1. Press the Preparation key PREPA..

   The Preparation key lamp lights and the Preparation screen appears.

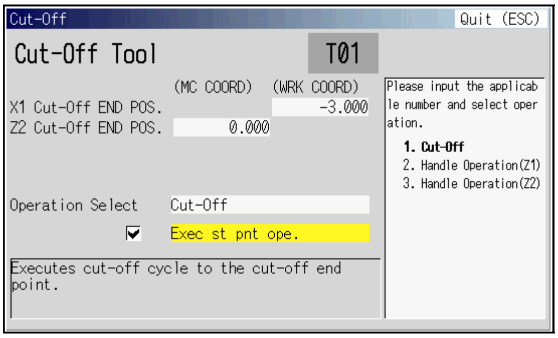
2. Use the Arrow keys ⬆ ⬇ to position the cursor at the tool to be used for cut-off machining.
3. Feed the material so that it projects for the length to be cut-off or the length for end face cutting.
4. Press the menu key [Cut-Off].The Cut-Off screen appears.



5. Make sure that all doors are closed, and press the Start button.
   The Start button lamp blinks on and off, and the doors will lock.

[Note]

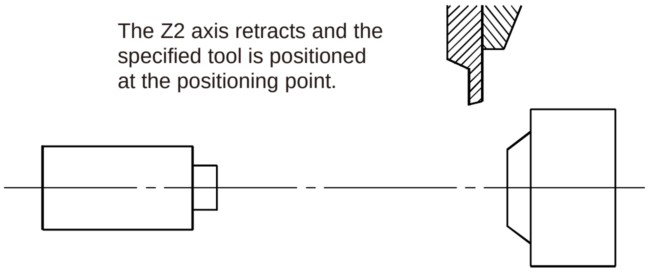
On the Cut-Off window, if the check mark is entered for the "Exec st pnt ope." item below Operation Select, the chuck automatically opens and the Z1 moves to the start position after the completion of cut-off operation.



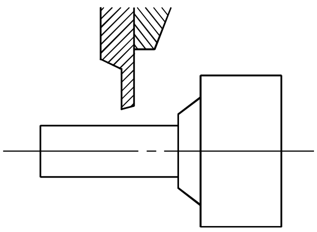
6. Press the Start button.

   The Start button lamp lights. The Z2 axis returns and the tool is positioned at the positioning point. (The operation can be executed while the door is open up to the completion of positioning at the position point.)

   After the completion of operation, the Start button lamp blinks (on for 0.5 seconds and off for 0.5 seconds).



7. Press the Door open button to unlock the door, and open the door.
8. Protrude only that portion of the material which is to be subject to cut-off machining or end face turning.
   You can feed the Z1 axis by handle to protrude the material by selecting "Handle Operation (Z1)" in the Cut-Off screen.



9. Close all the doors.
10. Make sure that all doors are closed, and press the Coolant button.
11. Press the Start button.
    The Start button lamp blinks on and off, and the doors will lock.
12. Press the menu key [Cut-Off] and the Start button at the same time.

    The Start button lamp lights, the chuck is closed, and cut-off machining starts. The message "Executing cut-off" is displayed during the operation. Upon completion of the operation, the message disappears and the Start button lamp turns off. (The spindle stops rotation but the chuck remains closed.)

    If the check mark is set for the "Exec st pnt ope." item, the Z1 axis moves to the start position after the chuck is opened.

[Note]

- To open the door, set the Setting switch key to the "|" position, and press the Door open button. If this key is set to the "O" position, machine operation is not allowed while the door is open.
- The unit will only operate with the door open while the Start button is pressed.

  The axes speed at this time is limited to 2 m/min or less.
- Releasing the Start button stops axis feed in the hold state and the Start button lamp blinks (on for two seconds and off for one second). To restart the operation, press the Start button.
- If no operation is performed within 15 seconds after making menu selections, menu selections will automatically be canceled.

  Menu selections will also be canceled if a reset is performed during or after operations. To restart operations, be sure to again make selections with the menus.
- Cut-off machining is disabled while the door is open. Lock the door before starting cut-off machining.
- The door lock cannot be released unless the following conditions are met.
  All spindles have stopped.
  All servo axes have stopped.
  The machine is not operating.
  The coolant is not discharged.
- The menu key [Cut-Off] is disabled if no program is selected.
- If the check mark is set for the "Exec st pnt ope." item, the chuck automatically opens after the completion of cut-off operation and the Z1 moves to the start position. Accordingly, the end face of the material may move away from the cut-off tool if the gap adjustment for the guide bushing is loose. Before turning on the bar loader torque, make sure to check the position of the material end face.

6.2.9 Non-conformed material phase adjustment

Adjust the phase between the main spindle and the back spindle to machine non-conformed materials. When the chuck for non-conformed material is used, make sure to adjust the phase.

CAUTION

Described below are operation procedures when the Setting switch is set to "O".

If you are going to work inside the machine, press the Door open button to unlock the door.

The door cannot be unlocked unless the following conditions are met.

All spindles have stopped.

All servo axes have stopped.

The machine is not operating.

The coolant is not discharged:

The Door open button is pressed.

CAUTION

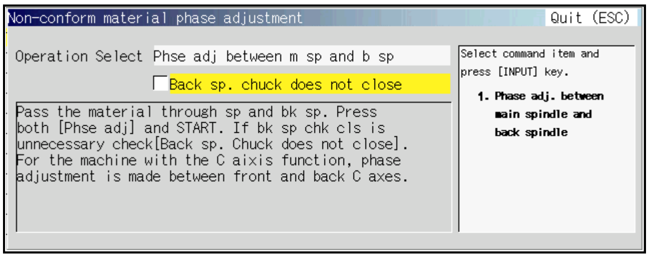
To adjust the non-conform material phase, you must first adjust the non-conformed material.

See *<6.2.10 Adjustment>*

[Procedure]

Adjust the phase between the main spindle and back spindle.
(If C axis function is enabled, adjust the phase between the front C axis and the back C axis so that the 0° of front C axis matches the 0° of back C axis.

1. Put a material (non-conformed material) through the main spindle and back spindle.
2. Press the menu key [Phse adj].
   The Non-conform material phase adjustment screen appears.
3. In Operation Select field, select "Phse adj between m sp and b sp". Press the numeric key "1" and then Input key INPUT.
   When "Phse adj between m sp and b sp" is selected for Operation Select field, the cursor moves to "Back sp. chuck does not close".



4. If you do not need to close the back spindle chuck during phase adjustment, press the Input key INPUT.
   A check mark is placed in the "Back sp. chuck does not close" check box.
5. Make sure that all doors are closed, and press the Start button.
   The Start button lamp blinks on and off, and the doors will lock.
6. Press the menu key [Phse adj] and the Start button at the same time.
   The Start button lamp lights, and phase adjustment between the main spindle and back spindle is executed. During adjustment, the message "Executing non-conformed material phase adjustment" is displayed on the screen. Upon completion of adjustment, the message "Completion of non-conformed material phase adjustment" appears on the screen.

[Note]

If you take no action for 15 seconds after pressing the menu key [Phse adj], the Non-conform material phase adjustment dialog box is closed automatically.

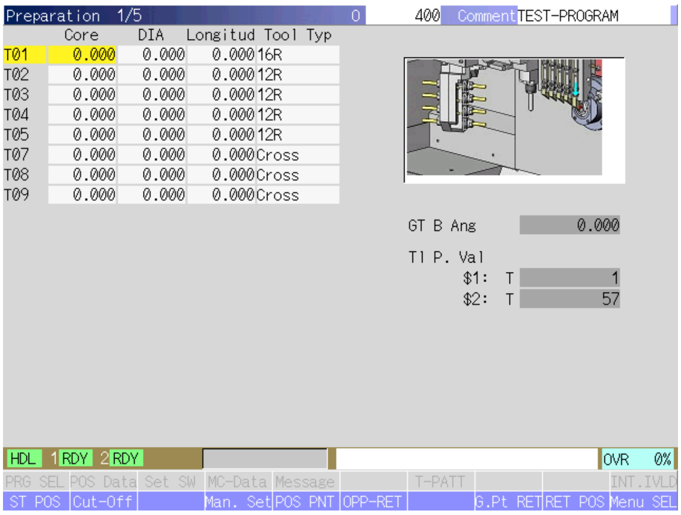
6.2.10 Adjustment

Move the tool holder to the position so that the guide bushing can be replaced smoothly.

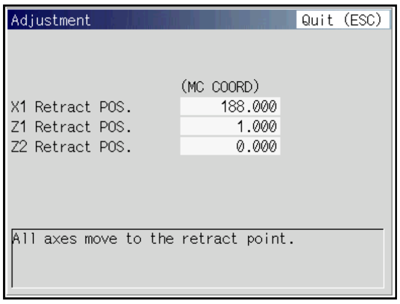
This section explains guide bushing replacement on the screen.
See *<Section 7.2 Mounting a Guide Bushing Device>* and *<Section 7.3 Replacing and Adjusting Guide Bushing>* when you actually replace or adjust the guide bushing.

[Procedure]

1. Press the Preparation key PREPA. .
The Preparation key lamp lights and the Preparation screen appears.



2. Press the menu key [ADJUST].
The adjustment screen is displayed.

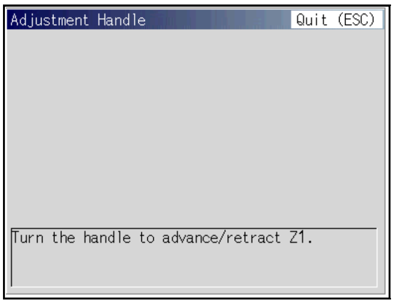


3. Make sure that all doors are closed, and press the Start button .
The Start button lamp blinks on and off, and the doors will lock.

4. Press the Start button .
The Start button lamp lights, and the X1, Z1, and Z2 axes return to their respective return position.

> **WARNING**
>
> The machine may move suddenly while the Start button lamp goes on.
> Be sure to keep hands away from the machine while the lamp is going on.

5. When operations are completed, the Start button lamp turns off and the screen changes to the Adjustment Handle screen.



6. Press one of the Handle Magnification button X1, X10, or X100.
The larger the number the faster the machine will move.

7. Return the Z1 axis in handle feed and perform replace or adjust the chuck and the guide bushing.

8. Press the Escape key ESC to finish the operation.

[Note]

- To enable operation while the door is open, press the Door open button to unlock the door, and set the Setting switch key to the "I" position. If this switch key is set to the "O" position, machine operation is prohibited while the door is open.
- The unit will only operate with the door open while the Start button is pressed.

  The axes speed at this time is limited to 2 m/min or less.

  Releasing the Start button stops axis feed in the hold state and the Start button lamp blinks (on for two seconds and off for one second). To restart the operation, press the Start button.
- If no operation is performed within 15 seconds after making menu selections, menu selections will automatically be canceled.

  Menu selections will also be canceled if a reset is performed during or after operations. To restart operations, be sure to again make selections with the menus.
- With the GBL-machine, performing adjustment does not move the Z1 axis.

6.2.11 Adjustment of non-conformed material

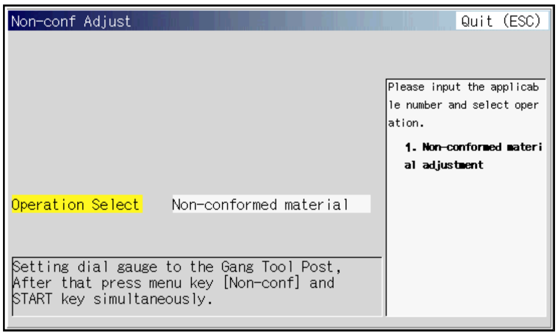
Adjust the zero point of non-conformed material by handle operation from outside of the machine.

CAUTION

To adjust the non-conform material phase, you must first adjust the non-conformed material.

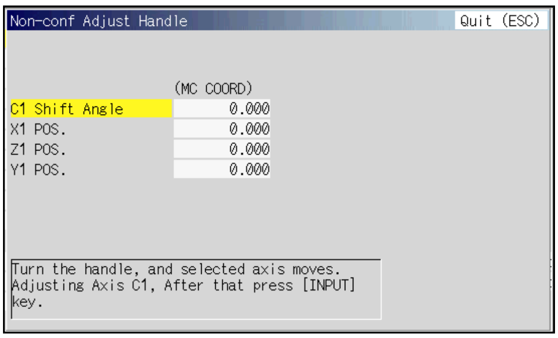
[Procedure]

1. Insert the non-conformed material into the main spinle, and close the chuck.
2. Press the Preparation key PREPA.
   The Preparation key lamp lights and the Preparation screen appears.
3. Press the menu key [Non-conf].
   The Non-conf Adjust screen appears.
4. Select "Non-conformed material adjustment" for "Operation Select" field. Enter the numeric key "1" and press the Input key INPUT.



5. Close all doors.
6. Make sure that all doors are closed, and press the Start button.
   The Start button lamp blinks on and off, and the doors will lock.
7. Press the menu key [Non-conf] and the Start button simultaneously.
   The Start button lamp lights, the main chuck closes, and the main spindle enters into the C-axis mode.

   Note) The main chuck closes if it is open at the execution of adjustment. You need to confirm the position of material before starting adjustment.
8. The Non-conf Adjust Handle screen appears.



9. Press the Door open button to unlock the door.
10. Set the dial gauge to the gang tool post.
11. Select "C1 Shift Angle" on the Non-conf Adjust Handle screen. When you turn the handle, the non-conformed material rotates.

12. When you select "X1 POS." and "Y1 POS." on the Non-conf Adjust Handle screen and turn the handle, the dial gauge mounted on the gang tool post moves. Measure the run-out of the material, and define the zero point of material surface.
Moving the cursor to "Z1 POS." and turning the handle can move the material forward or backward.

13. When the material surface is adjusted to the zero point, move the cursor to "C1 Shift Angle" and press the Input INPUT key.

14. When finished, press the Escape key ESC to close the window.

[Note]

If no operation is performed within 15 seconds after making menu selections, menu selections will automatically be canceled.
Menu selections will also be canceled if a reset is performed during or after operations. To restart operations, be sure to again make selections with the menus.

6.2.12 Exciting Guide Bushing

Excite the guide bushing to secure it.

Adjustment of clearance between the guide bushing and the material can be performed without using the adjustable pin wrench.

[Procedure]

1. Retract the Z1 axis by handle feed.
2. Press the Preparation key PREPA.
   The Preparation key lamp lights and the Preparation screen appears.
3. Press the menu key [GB Excite].
4. Close all the doors.
5. Make sure that all doors are closed, and press the Start button.
   The Start button lamp blinks on and off, and the doors will lock.
6. Press the Start button.
   The guide bushing is excited.
   During exciting, a message "Exciting GB" is displayed.

[Note]

- To enable operation while the door is open, press the Door open button to unlock the door, and set the Setting switch key to the "I" position. If this switch key is set to the "O" position, machine operation is prohibited while the door is open.
- Exciting GB is disabled while the door is open. Lock the door before starting exciting GB.
- The door lock cannot be released unless the following conditions are met.
  All spindles have stopped.
  All servo axes have stopped.
  The machine is not operating.
  The coolant is not discharged.
- To cancel excitation, move to any other screen, select any other operation menu, or press the menu key [GB Excite] again.
- Exciting GB cannot be canceled by pressing the Reset key.
- A caution screen is displayed while the message "Exciting GB" is displayed.

6.3 On-machine Program Check

Actually operate the machine in the forwards and backward directions to check for machine interference, and if any problems are found the program can be corrected on the spot. Switching between 1 cycle operation, 1 block operation, and handle operation can be done during operation. Backward running is also possible during the handle feed mode.

CAUTION

If you want to perform actual thread cutting or tapping in on-machine program check mode, be sure to press the menu key [Act. Cut] before starting operation. Pressing the menu key [Act. Cut] during operation does not enable the function. The accurate shape will not be displayed if [Act. Cut] is disabled.

6.3.1 Program check for 1-cycle operation

The program is executed for one cycle in the automatic operation mode.

[Procedure]

1. Press the Preparation key PREPA..

   The Preparation key lamp lights and the Preparation screen appears.
2. Select the program to be executed.

   See <*6.1.1 Selecting an execution program*>.
3. Press the menu key [ST POS]. See <*6.2.3 Start position*>.
4. Make sure that all doors are closed, and press the Start button.
   The Start button lamp blinks on and off, and the doors will lock.
5. Press the Start button.
   The Start button lamp lights and the start position operation is started. During machine operation the message "Moving to start position" appears. When operation is completed the Start button lamp turns off and the message changes to "Moving to start position completed.".
6. Press the Program Check key CHECK.

   The Program Check key lamp lgihts and the On-Machine Check screen appears.
7. Press the menu key [1 Cycle]. Check that the menu key [Handle] is not pressed.

   If [Handle] is pressed, press the menu key [Handle] to cancel its selection.
8. Make sure that all doors are closed, and press the Start button.
   The Start button lamp blinks on and off, and the doors will lock.
9. Press the Start button.
   The Start button lamp lights and the program will be executed.

[Note]

- Reverse operations can be performed only when the menu key [Handle] is being pressed.
- The "1 Block" operation cannot be performed in the reverse operation.
- Switching between [1 Cycle], [1 Block], and [Handle] can be done during program execution.
- To open the door, set the Setting switch key to the "I" position, and press the Door open button. If this key is set to the "O" position, the door cannot be opened.

  To make readjustments, press the Reset key RST and start over from the operations ready start point.
- The door lock cannot be released unless the following conditions are met.

  All spindles have stopped.
  All servo axes have stopped.
  The machine is not operating.
  The coolant is not discharged.

6.3.2 Program check for 1-block operation

Press the Start button to have the program execute for 1 block and then stop.

[Procedure]

1. Press the Preparation key PREPA.. The Preparation key lamp lights and the Preparation screen appears.
2. Select the program to be executed. See *<6.1.1 Selecting an execution program>*.
3. Press the menu key [ST POS]. See *<6.2.3 Start position>*.
4. Make sure that all doors are closed, and press the Start button.
   The Start button lamp blinks on and off, and the doors will lock.
5. Press the Start button.
   The Start button lamp lights and the start position operation is started. During machine operation the message "Moving to start position" appears. When operation is completed the Start button lamp turns off and the message changes to "Moving to start position completed.".
6. Press the Program Check key CHECK.

   The Program Check key lamp lgihts and the On-Machine Check screen appears.
7. Press the menu key [1 Block]. Check that the menu key [Handle] has not been pressed.

   If [Handle] is pressed, press the menu key [Handle] to cancel its selection.
8. Make sure that all doors are closed, and press the Start button.
   The Start button lamp blinks on and off, and the doors will lock.
9. Press the Start button.
   The Start button lamp lights. After one block of the program is executed, the lamp blinks on and off.

[Note]

- Reverse operations can be performed only when the menu key [Handle] is being pressed.
- The "1 Block" operation cannot be performed in the reverse operation.
- Switching between [1 Cycle], [1 Block], and [Handle] can be done during program execution.
- To open the door, set the Setting switch key to the "I" position, and press the Door open button. If this key is set to the "O" position, the door cannot be opened.

  To make readjustments, press the Reset key RST and start over from the operations ready start point.
- The door lock cannot be released unless the following conditions are met.

  All spindles have stopped.
  All servo axes have stopped.
  The machine is not operating.
  The coolant is not discharged.

6.3.3 On-machine program check by handle feed

The operation can be automatically switched from forward action to backward running by turning the handle in either the forward or backward direction. This can be done to check for interference, etc.

[Procedure]

1-cycle handle operation

1. Press the Preparation key PREPA..

The Preparation key lamp lights and the Preparation screen appears.

2. Select the program to be executed.

See *<6.1.1 Selecting an execution program>*.

3. Press the menu key [ST POS]. See *<6.2.3 Start position>*.

4. Make sure that all doors are closed, and press the Start button .
The Start button lamp blinks on and off, and the doors will lock.

5. Press the Start button .
The Start button lamp lights and the start position operation is started. During machine operation the message "Moving to start position" appears. When operation is completed, the Start button lamp turns off and the message changes to "Moving to start position completed.".

6. Press the Program Check key CHECK.

The Program Check key lamp lgihts and the On-Machine Check screen appears.

7. Press the menu key [1 Cycle]. Check that the menu key [Handle] has been pressed.

If [Handle] is not pressed, press the menu key [Handle] to select it.

8. Press one of the Handle Magnification button X1, X10, or X100.

The larger the number the faster the machine will move.

9. Make sure that all doors are closed, and press the Start button .
The Start button lamp blinks on and off, and the doors will lock.

10. Press the Start button .
The Start button lamp lights.

11. Turn the handle clockwise, and the program is executed in proportion to the amount of rotation of the handle. (The programmed feed rate is not exceeded.) To check machine operation carefully, turn the handle slowly or set the Handle Magnification button to a lower value.

12. Stop the handle.

The program will stop. The Start button lamp remains lit.

13. Turn the handle counterclockwise. The program is executed in the backward direction in proportion to the amount the handle is turned.

[Procedure]

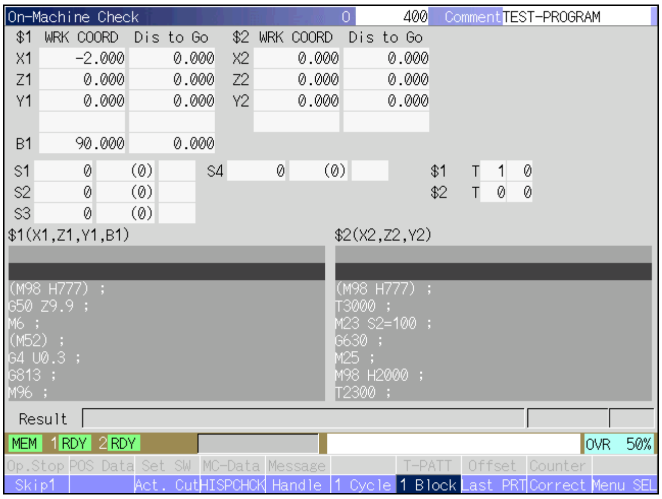
1-block handle operation

1. Press the Preparation key PREPA. The Preparation key lamp lights and the Preparation screen appears.
2. Select the program to be executed. See *<6.1.1 Selecting an execution program>*.
3. Press the menu key [ST POS]. See *<6.2.3 Start position>*.
4. Make sure that all doors are closed, and press the Start button .
   The Start button lamp blinks on and off, and the doors will lock.
5. Press the Start button .
   The Start button lamp lights and the start position operation is started. During machine operation the message "Moving to start position" appears. When operation is completed, the Start button lamp turns off and the message changes to "Moving to start position completed.".
6. Press the Program Check key CHECK.

   The Program Check key lamp lgihts and the On-Machine Check screen appears.
7. Press the menu key [1 Block]. Check that the menu key [Handle] has been pressed.

   If [Handle] is not pressed, press the menu key [Handle] to select it.
8. Press one of Handle Magnification button x1, x10, or x100.

   The larger the number the faster the machine will move.
9. Make sure that all doors are closed, and press the Start button .
   The Start button lamp blinks on and off, and the doors will lock.
10. Press the Start button . The Start button lamp lights.
11. Turn the handle clockwise, and the program is executed in proportion to the amount of rotation of the handle. (The programmed feed rate is not exceeded.) To check machine operation carefully, turn the handle slowly or set the Handle Magnification button to a lower value.
12. Stop the handle. The program will stop. The Start button lamp remains lit.
13. Turn the handle counterclockwise. The program is executed in the backward direction in proportion to the amount the handle is turned.
14. Turn the handle clockwise. The program is executed in the forward direction in proportion to the amount the handle is turned. When the operation is finished, the Start button lamp turns off.



WARNING

While executing the on-machine program check by pressing the menu key [Handle], the machine does not move unless the handle is turned. However, never put your hands in the machine during on-machine program check operation since the machine can suddenly move if the handle is turned by mistake and you could suffer serious injury.
During on-machine program check, although feed rate of axes is determined according to the rate of handle turning, the spindles rotate at the speed specified in the program, so the cutting conditions differ from those assumed in programming. Therefore, if a workpiece is actually cut, pay sufficient care to the feed rate. Cutting under inappropriate conditions could cause the material or the tool to be broken.

[Note]

- Backward running is inhibited for the M code, T code, and G code macros.
- Backward running can be specified for M codes and S codes other than the above, but will not be executed.
- You can use the on-machine program check function in the handle operation or automatic operation. When performing actual thread cutting or tapping, press the menu key [Act. Cut] before starting operation. Pressing the menu key [Act. Cut] during operation does not enable the function. With the menu key [Act. Cut] being pressed, the machine stops just before the tread cutting or tapping command. Press the Start button again. Override function is disabled while executing thread cutting or tapping (100%).
- Chamfering of the thread cutting is not performed in the on-machine program check function.
- Cycle start remains disabled with any door left open.
- To open the door, set the Setting switch key to the "I" position, and press the Door open button. If this key is set to the "O" position, the door cannot be opened.

  To make readjustments, press the Reset key RST and start over from the operations ready start point.
- The door lock cannot be released unless the following conditions are met.

  All spindles have stopped.
  All servo axes have stopped.
  The machine is not operating.
  The coolant is not discharged.

6.3.4 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 <*6.3.1 Program check for 1-cycle operation*>.

Execute the on-machine program check function in 1-cycle operation.

2. Press the menu key [1 Block].

When a block is executed, the program stops, and the Start button lamp turns off.
If one of the programs of two axis control groups enters the "block stop" state, the programs of the other axis control groups enter the "hold" state during execution.
"BST" is displayed as the operation status of the axis control group in the block stop state.
"HLD" is displayed for the other axis control group in the hold state.
The cursor of that axis control group is reversed just as when its program is being executed.
Make sure that the program of the axis control group you want to edit is in the block stop status.

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 the Start button 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 key also change.)

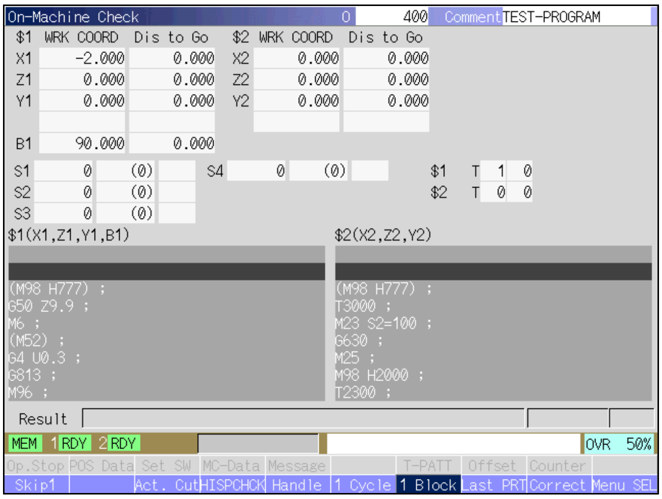
5. Edit the program.

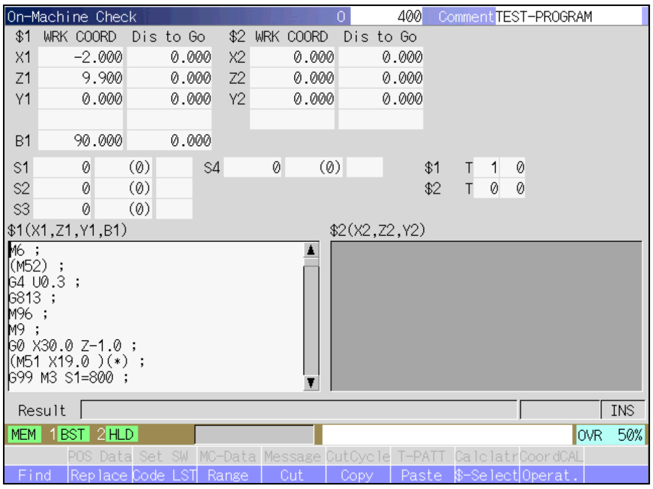
6. Press the menu key [Operat.] at completion of editing.
The On-Machine Check screen appears. (The menu keys also change.)

7. Make sure that all doors are closed, and press the Start button .
The Start button lamp blinks on and off, and the doors will lock.

8. Press the Start button [ ].

The Start button lamp lights and the program execution restarts.





[Note]

Program can be edited only for the axis control group stopped in the block stop state. To edit the program, the memory area must have at least 500 characters of free area. Remember that, however, the program in external memory cannot be edited.
(Editing is possible only for the axis control group for which "BST" is displayed at the lower left area of the screen.)

6.3.5 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 *<6.3.3 On-machine program check by handle feed>*.

   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 two axis control groups enters the "block stop" state, the programs of the other axis control group enters 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 the Start button 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.

   If the On-Machine Check (Edit) screen is not displayed, run the program backward several blocks to edit the program.
7. Edit the program.
8. Press the menu key [Operat.] at completion of editing.
   The On-Machine Check screen appears.
9. Make sure that all doors are closed, and press the Start button .
   The Start button lamp blinks on and off, and the doors will lock.
10. Press the Start button . Turn the handle clockwise.

    The Start button lamp lights and the program execution is restarted.

[Note]

- 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 either of the machining programs of two axis control groups reaches the last of the blocks with which backward running is permitted, the program of the other axis control group can no longer run backward.
- The Axis Motion 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 Motion lamp turns off.

DANGER

Do not put your hand into the machine while the Axis Motion 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 two 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.
- When you edit a block of the program after running it backward, there are cases the program must be run backward at least five blocks, from the backward running start block, to enable editing.
- Cycle start remains disabled with any door left open.

CAUTION

Even in the block with which backward running is inhibited, the program can run backward for the axis control group to which the backward running is permitted. In the case, an axis move forward and another axis move backward, and these axes may interfere each other. Therefore, the operation for avoiding an interference may be required.

- To open the door, set the Setting switch key to the "I" position, and press the Door open button . If this key is set to the "O" position, the door cannot be opened.

  To make readjustments, press the Reset key and start over from the operations ready start point.
- The door lock cannot be released unless the following conditions are met.

  All spindles have stopped.
  All servo axes have stopped.
  The machine is not operating.
  The coolant is not discharged.

6.3.6 High-speed program check

This function is used to check a program at a high speed for detecting programming errors and other problems. If the machine coordinates of the X1, Z1 and Y1 axes differ between the program start and end positions, the function displays an alarm message.

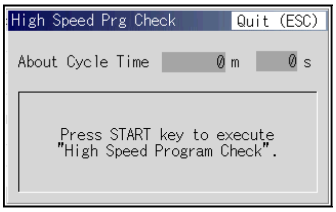
[Procedure]

1. Press the Preparation key PREPA..

   The Preparation key lamp lights and the Preparation screen appears.
2. Select the program to be executed.

   See *<6.1.1 Selecting an execution program>*.
3. Press the menu key [ST POS]. See *<6.2.3 Start position>*.
4. Make sure that all doors are closed, and press the Start button.
   The Start button lamp blinks on and off, and the doors will lock.
5. Press the Start button.
   The Start button lamp lights and the start position operation is started. During machine operation the message "Moving to start position" appears. When operation is completed the Start button lamp turns off and the message changes to "Moving to start position completed.".
6. Press the Program Check key CHECK.

   The Program Check key lamp lgihts and the On-Machine Check screen appears.
7. Press the menu key [HISPCHCK]. The High Speed Prg Check screen appears.



8. Make sure that all doors are closed, and press the Start button.
   The Start button lamp blinks on and off, and the doors will lock.
9. Press the Start button. The Start button lamp lights and the program is executed.

[Note]

- Use the 1-cycle time (rough estimate) as an approximate time.
- The high-speed program check operation is executed about five times faster than the actual machining time.
- Be sure to perform start position operation in preparation mode before high-speed program check operation. If you have cancelled high-speed program check operation due to an alarm, you must also perform start position operation.

6.3.7 Block skip

The block skip function can be used during program execution to ignore blocks which include a slash ("/") at the beginning of the block.

[Procedure]

1. Press the menu key [Skip 1].

The menu key [Skip 1] enters the selected state and the block skip function is turned on. At the same time, the Skip lamp 

on the operation panel lights.

6.3.8 Optional stop

The optional stop function suspends execution of the program of the control axis group for which the "M1" block is executed. When the execution of the program is suspended in the optional block stop state, the Start button lamp blinks on and off.

Pressing the Start button restarts the automatic operation.

[Procedure]

1. Press the menu key [Op.Stop].
The menu key [Op.Stop] enters the selected state and the optional stop function is turned on. At the same time, the Optional stop lamp 

on the operation panel lights.

6.3.9 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 at an appropriate position.

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]

When machining is performed using the two axis control groups, setting the override more than 100% causes the feed rate to exceed the maximum rapid feed and the speed is clamped. The speed clamp may cause the machining timing to shift, and the product may not be manufactured as it should be with the machining program being used. Therefore, the feed rate override is normally effective up to 100%. If 100% to 200% is specified, 100% will be set as the override.

If the machining involve the possible speed clamp, you can make the feed rate override effective up to 200% by setting Turn on the Override No Limit switch (Item 15) from the Set SW screen (See *<6.8.6 Setting switches>*). When the Override No Limit setting is set to ON, you can set the override of the cutting speed to up to 200%. The rapid feed commanded between 100% and 200% override will be assumed as 100%.

6.3.10 Disconnect Bar Loader (BL DIS)

If Disconnect Bar Loader function is enabled, the power status of the bar loader will not be checked.

[Procedure]

1. Press the menu key [BL DIS].

The menu key [BL DIS] enters the selected state and the Disconnect Bar Loader function is turned on.

6.3.11 Coolant Flow Rate Detector

On the right bottom corner of the Screen, the status of coolant flow rate sensor (Enabled or Disabled) is displayed.

[Note]

The procedures to perform Block skip, Optional stop, Override, Load/unload bar feeder, and Coolant flow rate detection device on Automatic Operation screen are same as on the On-machine Program Check screen.

6.4 Automatic Operation

Automatic operation is used to execute a selected program to perform machining operation.
There are three functions available in automatic mode, they are continuous automatic operation, one cycle automatic operation and one block automatic operation.

[Procedure]

1. Press the menu key [Cont.], [1 Cycle], or [1 Block] on the Automatic Operation screen.
   The automatic operation mode corresponding to the key pressed is selected.
   Pressing [Last PRT] executes the last program.

[Note]

You can switch the automatic mode ([Cont.], [1Cycle] or [1 Block]) during program execution. For example, you can change the mode to the one cycle automatic operation during execution of continuous automatic operation and stop the operation after completing one cycle operation.

6.4.1 Continuous automatic operation

The continuous automatic operation executes the program repeatedly and continuously.

[Procedure]

1. Press the Preparation key PREPA..
   The Preparation key lamp lights and the Preparation screen appears.
2. Select the program to be executed.
   See *<6.1.1 Selecting an execution program>*.
3. Press the menu key [ST POS]. See *<6.2.3 Start position>*.
4. Make sure that all doors are closed, and press the Start button .
   The Start button lamp blinks on and off, and the doors will lock.
5. Press the Start button .
   The Start button lamp lights and the start position operation is started. During machine operation the message "Moving to start position" appears. When operation is completed the Start button lamp turns off and the message changes to "Moving to start position completed.".
6. Press the Automatic Operation key AUTO.
   The Automatic Operation key lamp lights and the Automatic Operation screen appears.
7. Press the menu key [Cont.].
8. Make sure that all doors are closed, and press the Start button .
   The Start button lamp blinks on and off, and the doors will lock.
9. Press the Start button .
   The Start button lamp lights and the program is executed.

[Note]

- When a program is called, the machining data associated with the called program is also automatically called up. After the program is selected, be sure to check the machining data.
- To open the door, set the Setting switch key to the "⫴" position, and press the Door open button . If this key is set to the "◎" position, the door cannot be opened.
  To make readjustments, press the Reset key RST and start over from the operations ready start point.
- The door lock cannot be released unless the following conditions are met.
  All spindles have stopped.
  All servo axes have stopped.
  The machine is not operating.
  The coolant is not discharged.

6.4.2 One cycle automatic operation

The one cycle automatic operation executes the program to the end of one cycle.

[Procedure]

1. Press the Preparation key PREPA.. The Preparation key lamp lights and the Preparation screen appears.
2. Select the program to be executed. See *<6.1.1 Selecting an execution program>*.
3. Press the menu key [ST POS]. See *<6.2.3 Start position>*.
4. Make sure that all doors are closed, and press the Start button.
   The Start button lamp blinks on and off, and the doors will lock.
5. Press the Start button.
   The Start button lamp lights and the start position operation is started. During machine operation the message "Moving to start position" appears. When operation is completed the Start button lamp turns off and the message changes to "Moving to start position completed.".
6. Press the Automatic Operation key AUTO.
   The Automatic Operation key lamp lights and the Automatic Operation screen appears.
7. Press the menu key [1 Cycle].
8. Make sure that all doors are closed, and press the Start button. The Start button lamp blinks on and off, and the doors will lock.
9. Press the Start button. The Start button lamp lights and the program is executed.

[Note]

- When a program is called, the machining data associated with the called program is also automatically called up. After the program is selected, be sure to check the machining data.
- Cycle start remains disabled with any door left open.
- Pressing the menu key [Last PRT] in the one-cycle operation mode executes the last program (back machining) to complete one cycle.
- To open the door, set the Setting switch key to the "" position, and press the Door open button. If this key is set to the "◎" position, the door cannot be opened.

  To make readjustments, press the Reset key RST and start over from the operations ready start point.
- The door lock cannot be released unless the following conditions are met.

  All spindles have stopped, and the machine is not executing indexing or C axis command.
  All servo axes have stopped.
  The machine is not operating.
  The coolant is not discharged.

6.4.3 One block automatic operation

The one block automatic operation executes only one block of the execution program.

[Procedure]

1. Press the Preparation key PREPA.

   The Preparation key lamp lights and the Preparation screen appears.
2. Select the program to be executed.

   See <*6.1.1 Selecting an execution program*>.
3. Press the menu key [ST POS]. See <*6.2.3 Start position*>.
4. Make sure that all doors are closed, and press the Start button.
   The Start button lamp blinks on and off, and the doors will lock.
5. Press the Start button.
   The Start button lamp lights and the start position operation is started. During machine operation the message "Moving to start position" appears. When operation is completed the Start button lamp turns off and the message changes to "Moving to start position completed.".
6. Press the Automatic Operation key AUTO.

   The Automatic Operation key lamp lights and the Automatic Operation screen appears.
7. Press the menu key [1 Block].
8. Make sure that all doors are closed, and press the Start button.
   The Start button lamp blinks on and off, and the doors will lock.
9. Press the Start button.

   The Start button lamp lights. After one block of the program is executed, the lamp turns off.
10. Press the Start button.

    The Start button lamp lights, and the next block or the program is executed.

[Note]

- When a program is called, the machining data associated with the called program is also automatically called up. After the program is selected, be sure to check the machining data.
- Cycle start remains disabled with any door left open.
- One block operation may not be executed depending on the G code used. (See <*Programmer's Manual*>.)
- To open the door, set the Setting switch key to the "|" position, and press the Door open button. If this key is set to the "◎" position, the door cannot be opened.

  To make readjustments, press the Reset key RST and start over from the operations ready start point.
- The door lock cannot be released unless the following conditions are met.

  All spindles have stopped.
  All servo axes have stopped.
  The machine is not operating.
  The coolant is not discharged.

6.4.4 Hold and restart the automatic operation

You can suspend the program being executed by pressing the Hold button . The program will be resumed by pressing the Start button .

[Procedure]

To suspend the program

1. Press the Hold button .

   The Start button lamp blinks on and off, the door lock is released the Hold button lamp lights, and the machine stops.

To restart the machine after a hold

2. Make sure that all doors are closed, and press the Start button .
   The Start button lamp blinks on and off, and the doors will lock.
3. Press the Start button .

   The Hold button lamp turns off, the Start button lamp lights, and the machine resumes execution of the program.

> **WARNING**
>
> Do not put your hand or any other part of your body into the machine during automatic operation. Even when automatic operation is being paused, the spindle remains rotating, involving a potential risk of causing a fatal or severe injury.

[Note]

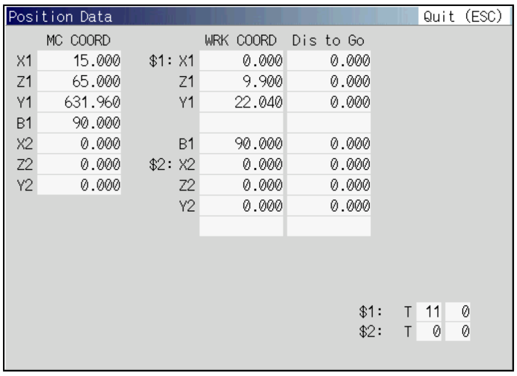
- For procedures to perform Block skip, Optional stop, Override, Load/unload bar feeder, and Coolant flow rate detection device, see "On-machine Program Check".
- To open the door, set the Setting switch key to the "|" position, and press the Door open button . If this key is set to the "O" position, the door cannot be opened.

  To make readjustments, press the Reset key RST and start over from the operations ready start point.
- The door lock cannot be released unless the following conditions are met.

  All spindles have stopped.
  All servo axes have stopped.
  The machine is not operating.
  The coolant is not discharged.

6.5 Position Data Screen

The Position Data screen shows the position information of command specified axes.



[Procedure]

1. Press the Menu Up/Down selection key [▶].

The submenus are enabled.

2. Press the menu key [POS Data].

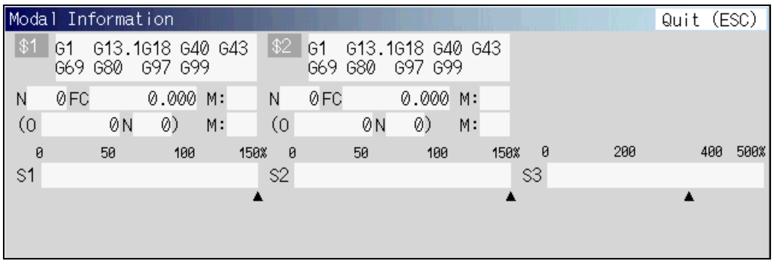
The Position Data screen appears.

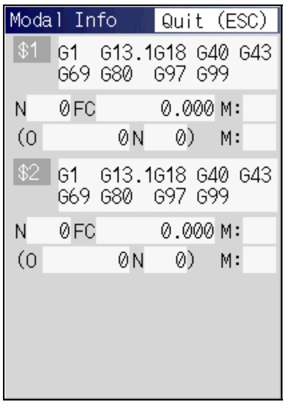
Item	Description
\$1 \$2	Indicates an axis control group.
MC COORD	Machine coordinate value of each axis.
WRK COORD	Work coordinate value of each axis.
Dis to Go	Move distance of each axis required for next command execution.
\$1 T \$2 T	Shows a T-code command value and offset number.

6.6 Modal Information Screen

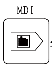
This screen displays the G modal information.

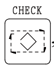
The Modal Information screen is accessed from the Automatic Operation, On-Machine Check, and MDI screen.

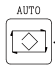




[Procedure]

1. Press the MDI key 

, Program Check key 

, or Automatic Operation key 

.

The pressed key lamp lights and the screen changes accordingly.

2. Press the menu key [Menu SEL] until the menu key [Modal] is displayed.

The Modal Information screen appears.

Item	Description
$1 $2	Indicates an axis control group.
G	Displays the G modal every axis control group.
N	Displays the program sequence number every axis control group.
FC	Displays the feed rate every axis control group.
M	Displays up to two G modals every axis control group.
(O, N)	Displays the program and sequence numbers of the subprogram every axis control group.
Load value	Indicates the load value for the spindle and tool spindle with the meter. ▲ represents the maximum value.

6.7 Manual Operation

Manual operation is used for individually operating each axis of the machine as required. There are two functions to move the machine by handle or MDI in manual mode.

6.7.1 Handle feed

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, Z2).

CAUTION

Described below are operation procedures when the Setting switch is set to "◎".

If you are going to work inside the machine, press the Door open button to unlock the door.

The door cannot be unlocked unless the following conditions are met.

All spindles have stopped.

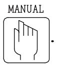
All servo axes have stopped.

The machine is not operating.

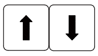
The coolant is not discharged:

The Door open button is pressed.

[Procedure]

1. Press the Manual key .

The Manual key lamp lights and the Handle Feed screen is displayed.

2. Use the Arrow keys  to select the control axis you want to feed.

The name portion of the selected control axis will be displayed in yellow.

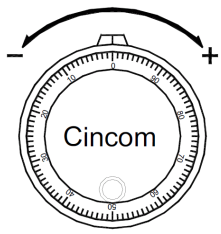
3. Press one of the Handle Magnification button [x1], [x10], or [x100].

The Handle Magnification button lamp goes on.
The larger the number the faster the machine will move.

4. Turn the handle.

The feed direction is determine by the direction the handle is turned.

If a feed instruction that exceeds the stored stroke limit is issued, the overrun alarm "M01 Operation error 007□" will be generated.
See *<6.8.2 Recovering from overrun status>*.



The axis is moved by turning the handle.

[Note]

To confirm the position (coordinate) of the workpiece that is moving, select the menu key [POS Data] to open the sub-window.

6.7.2 Manual data input (MDI)

You can create or execute a program in MDI operation mode in the same way as a usual program.

The MDI operation has two execution patterns: "1 block" and "all blocks".

[Note]

Only an MDI program that fits into a screen can be created.

6.7.2.1 Selecting the axis control group for MDI operation

[Procedure]

1. Press the MDI key . The MDI key lamp lights and the MDI screen appears.
2. Press the menu key [$-Select] to display the program input screen for $1 or $2.

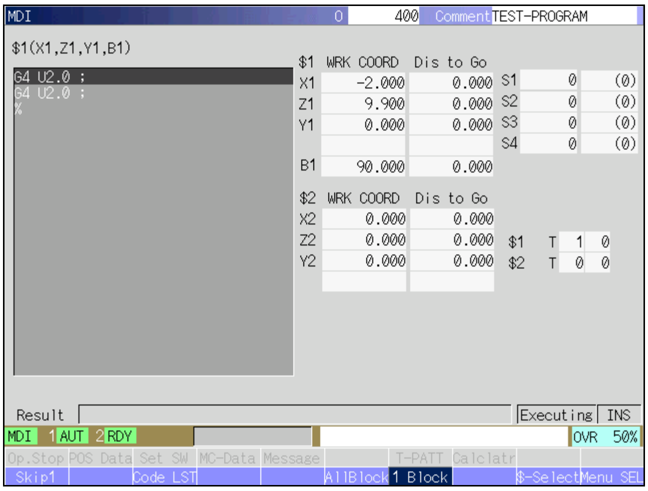
6.7.2.2 Executing MDI operation

1 Block

The program can be executed 1 block at a time.

[Procedure]

1. Press the MDI key The MDI key lamp lights and the MDI screen appears.
2. Press the menu key [$-Select] to display the program input screen for $1 or $2.
3. Enter a program and press the Input key . The message "Set COMP" is displayed.
4. Press the menu key [1 Block].
5. Make sure that all doors are closed, and press the Start button .
   The Start button lamp blinks on and off, and the doors will lock.
6. Press the Start button .
   The Start button lamp lights and the one block of the specified program is executed. The execution cursor will move to that block. While the program is executing the message "Executing" is displayed.
   After the one block is executed, The Start button lamp turns off.
   If you are going to open the door, press the Door open button to unlock the door.
   If door lock release conditions are not met, door locks will not be released.



7. Make sure that all doors are closed, and press the Start button .
   The Start button lamp blinks on and off, and the doors will lock.

8. Press the Start button.
The Start button lamp lights and the next one block is executed. The execution cursor will move to that block.
While the program is executing the message " Executing" is displayed.
After the one block is executed, The Start button lamp turns off.
If you are going to open the door, press the Door open button to unlock the door.
If door lock release conditions are not met, door locks will not be released.

[Note]

- 1 block operation cannot be performed for T code, M code, or G code macros.
  The Start button must be pressed several times until the execution of 1 block is completed.
- To open the door, set the Setting switch key to the "|" position, and press the Door open button. If this key is set to the "◎" position, the door cannot be opened.

  To make readjustments, press the Reset key RST and start over from the operations ready start point.
- The door lock cannot be released unless the following conditions are met.

  All spindles have stopped.
  All servo axes have stopped.
  The machine is not operating.
  The coolant is not discharged.

All blocks

The programs of all the blocks can be executed at a time.

[Procedure]

1. Press the MDI key .

   The MDI key lamp lights and the MDI screen appears.

2. Press the menu key [$-Select] to display the program input screen for $1 or $2.
3. Enter the program.

   The message "Editing ... " appears.

4. Press the Input key .

   The message "SET COMP" appears.

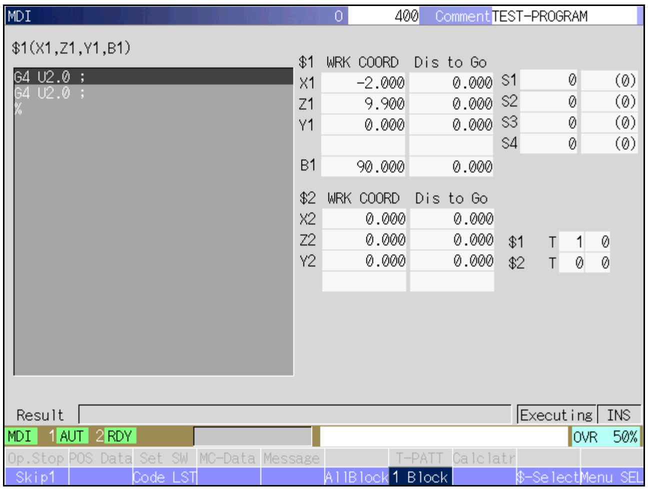
5. Press the menu key [All Block].
6. Make sure that all doors are closed, and press the Start button .
   The Start button lamp blinks on and off, and the doors will lock.
7. Press the Start button .

   The Start button lamp lights and the MDI program for all blocks are executed. The message "Executing" appears during program execution.

   After all blocks are executed, The Start button lamp turns off.

   If you are going to open the door, press the Door open button to unlock the door.

   If door lock release conditions are not met, door locks will not be released.



[Note]

- Cycle start remains disabled with any door left open.
- To open the door, set the Setting switch key to the "|" position, and press the Door open button . If this key is set to the "◎" position, the door cannot be opened.

  To make readjustments, press the Reset key and start over from the operations ready start point.

- The door lock cannot be released unless the following conditions are met.

  All spindles have stopped.
  All servo axes have stopped.
  The machine is not operating.
  The coolant is not discharged.

6.7.2.3 Editing MDI program

You can edit MDI programs as described below.

Any change you made to the MDI program is confirmed by pressing the Input key INPUT. If you switch to another screen before pressing the Input key, the edit processing is canceled.

(1) Insert mode and overwrite mode switching

[Procedure]

1. Press the Insert key INS.

Each time the Insert key is pressed, the mode is switched as insert → overwrite → insert. The mode is also switched to the insert mode if the Input key INPUT is pressed in the overwrite mode.

When a switch is made the corresponding mode "INS" or "OVR" message is displayed on the lower right of the screen.

(2) Entering a character

[Procedure]

1. Use the Arrow keys ↑ ↓ ← → or the Tab keys TAB ← TAB → to move the cursor to the position where you want to enter a character.

2. Enter a character using the alphanumeric keys.

Example: To type "C" after "AB":

1. Move the cursor to the right side of "B".

AB|

2. Press the C key to type "C".

ABC|

(3) Replacing a character

[Procedure]

1. Use the Arrow keys ↑ ↓ ← → or the Tab keys TAB ← TAB → to move the cursor over the character to be edited.

2. Enter a character using the alphanumeric keys.

The original character will be overwritten.
Example: To change "ABC" to "ABD":

1. Move the cursor before "C".

AB|C

2. Press the Insert key INS.
   (Make sure that the status display field at the lower right corner of the screen indicates "OVR".)

Press the D key to type "D".

ABD

(4) Inserting character

[Procedure]

1. Use the Arrow keys [↑] [↓] [←] [→] or the Tab keys [TAB ←] [TAB →] to move the cursor to the position where you want to insert a character.

2. Insert additional character using the alphanumeric key.

The characters will be inserted.

Example: To type in "D" between "B" and "C" of "ABC":

1. Move the cursor between "B" and "C".

   AB|C

2. Press the Insert key [INS].

   (Make sure that the status display field at the lower right corner of the screen indicates "INS".)

3. Press the [D] key to type "D".

   ABD|C

(5) Deleting a character

- Deleting a character after the cursor position

[Procedure]

1. Use the Arrow keys [↑] [↓] [←] [→] or the Tab keys [TAB ←] [TAB →] to move the cursor to the left side of character to be deleted.

2. Press the Delete key [DEL].

The character after the cursor position will be deleted.

Example: To delete "B" in the "ABC":

1. Move the cursor before "B".

   A|BC

2. Press the Delete key [DEL].

   A|C

- Deleting the character in front of the cursor

[Procedure]

1. Use the Arrow keys [↑] [↓] [←] [→] or the Tab keys [TAB ←] [TAB →] to move the cursor to the right side of character to be delete.

2. Press the Backspace key [BS].

The character in front of the cursor will be deleted.

Example: To delete "B" from a previously entered "ABC":

1. Move the cursor after "B".

   AB|C

2. Press the Backspace key [BS].

   A|C

(6) Deleting a line

[Procedure]

1. Move the cursor to the block to be deleted using the Arrow keys [↑][↓][←][→] and the Tab keys [TAB ←][TAB →].

2. Press the Line deletion key [CAN].

The character string in the block located by the cursor is deleted.

Example: To delete "DEF;" in the "ABC;DEF;":

1. Move the cursor to the block to be deleted.

```
ABC;
|DEF;
```

2. Press the Line deletion key [CAN].

```
ABC;
|
```

[Note]

If the block where the cursor is positioned does not fit in the display screen, deletion is not possible.

(7) Cleaning a screen

[Procedure]

1. When the screen to be deleted is displayed, press the Shift key [SHIFT] and the Line deletion key [CAN].

All characters on the screen will be cleared.

Example: Delete a screen displaying multiple program lines.

```
|ABC;
DEF;
GHI;
```

1. Press the Shift key [SHIFT] and the Line deletion key [CAN].

```
|
```

[Note]

The block that does not fit in the display screen cannot be deleted.

6.8 Maintenance

6.8.1 Zero return

This section provides instructions for the zero return of each axis. Normally, there is no need to perform the zero return. The zero return must be performed only when the zero return alarm is issued.

[Procedure]

Zero return for the specified axis only

1. Press the Parameter key .
2. Press the menu key [Menu SEL] until the menu key [ZP EXE] is displayed.
3. Press the menu key [ZP EXE].

   The Zero Return screen appears.
4. Press the menu key [Indvdal].
5. Use the Arrow keys ⬆ ⬇ to move the cursor to the axis for which zero return will be performed.

   Press the Input key .

   The axis with check mark is selected.
   Pressing the Input key again to cancel the selection.

   Several axes can be selected at one time.
6. Make sure that all doors are closed, and press the Start button .
   The Start button lamp blinks on and off, and the doors will lock.
7. Press the Start button .
   The Start button lamp lights and the zero return operations for the selected axes are performed.

   The message "Executing Zero Returning" appears in the operation status field for the axis during zero return, and the message "Completed Zero Return" appears for axes for which zero return is completed.
8. The Start button lamp goes off at the completion of zero return of the selected axis.

[Procedure]

Zero return for all axes

1. Press the Parameter key .
2. Press the menu key [Menu SEL] until the menu key [ZP EXE] is displayed.
3. Press the menu key [ZP EXE].

   The Zero Return screen appears.
4. Press the menu key [All axes].
5. Make sure that all doors are closed, and press the Start button .
   The Start button lamp blinks on and off, and the doors will lock.
6. Press the Start button .

   The Start button lamp lights and the zero return operations for all axes are performed.

   The message "Executing Zero Returning" appears in the operation status field for the axis during zero return, and the message "Completed Zero Return" appears for axes for which zero return is completed.

[Note]

- Normally, you need not manually return any axis to the zero point during routine operations. Manually return an axis to the zero point only when required either after the zero point return request message is displayed or after the motor is replaced.
- When zero return is executed for all axes, the Z1, X1 and Z2 axes return to the zero point first. The Y1 axis starts zero return after the completion of X1 axis zero return and the X2 axis starts zero return after the completion of zero return of the Z2 axis,

6.8.2 Recovering from overrun status

If you specify a position which exceeds the stored stroke limit (permitted operating range) of each axis, the machine will display the overrun alarm message. If this happens, use the following procedure to clear the overrun alarm.

[Procedure]

1. Any overrun alarm will flash the Caution lamp (yellow) and automatically displays a Message screen.

Confirm the overrun axis in the message.
If the Message screen is not displayed, press the Menu Up/Down selection key ▶ to enable the submenu.

Press the menu key [Message] to display the Message window.

2. Press the Manual key MANUAL.

The Handle Feed screen appears.

3. Use the Arrow keys ⬆ ⬇ to select the control axis that is over running.

The axis name field of the selected control axis will be reverse displayed in yellow.

4. Press one of the Handle Magnification button X1, X10, or X100.

The Handle Magnification button lamp goes on.
The larger the number, the machine operates faster.

5. For the control axis that is over running, turn the handle in the overrun cancel direction.

When the overrun is canceled the yellow light (alarm) will turn off.

[Note]

- If operations with the door open are to be allowed, press the Door open button to unlock the door, and set the Setting switch key in the "|" position. If doors are opened while set to disallowed, "O", the servos will turn off and an alarm will be generated.
- Handle feed can be performed using the menu key [MECH ADJ] instead of Handle key MANUAL.

  Press the Parameter key PRM. and then press the menu key [MECH ADJ].

  The Mechanism Adjustment screen appears. The operation method thereafter is the same as that for the Handle Feed screen.
- The Message screen displays messages like those below when the overrun alarm occurs.

Overrun axis		Message			
$1	X1 axis	M01 S/W STROKE END AXIS OPERATION ERROR	0007	X1	$1
	Z1 axis	M01 S/W STROKE END AXIS OPERATION ERROR	0007	Z1	$1
	Y1 axis	M01 S/W STROKE END AXIS OPERATION ERROR	0007	Y1	$1
$2	X2 axis	M01 S/W STROKE END AXIS OPERATION ERROR	0007	X2	$2
	Z2 axis	M01 S/W STROKE END AXIS OPERATION ERROR	0007	Z2	$2
	Y2 axis	M01 S/W STROKE END AXIS OPERATION ERROR	0007	Y2	$2

Note) The Y2 axis is available on L220X only.

- Stored stroke limit values (machine value)

(mm)

Overrun axis		–Limit	+Limit
$1	X1 axis	–1.0	127.5
	Z1 axis	–0.5	GB 205.5 GBL 301.5
	Y1 axis	–0.5	302.5
$2	X2 axis	–1.0	317.0
	Z2 axis	–1.0	206.0
	Y2 axis	–1.0	61.0

Note 1) The numerical values are the radius value displays for all axes.
Note 2) The Y2 axis is available on L220X only.

6.8.3 Counter set

The Counter screen is used for calculating the running time of a program executed in automatic operation mode and for counting the number of finished workpieces.

[Procedure]

1. Press the Automatic Operation key AUTO.

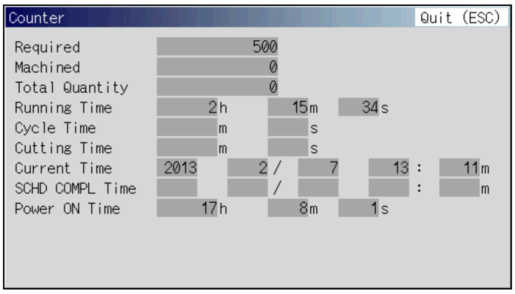
   The Automatic Operation screen appears.

2. Press the Menu Up/Down selection key ▶.

   The sub-menu is enabled.

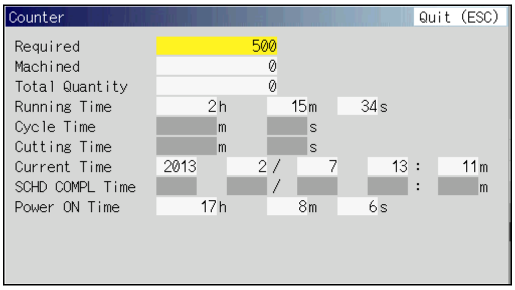
3. Press the menu key [Counter].

   The Counter screen appears.



4. Press the menu key [Cnt Set].

   The Counter screen changes to a status that setting is enabled. (Fields that can be input become white.)



5. Use the Arrow keys ⬆ ⬇ or the Tab keys TAB ⬅ TAB ➡ to move the cursor to the desired field.
6. Press the alphanumeric keys. Enter the data.
7. Press the Input key INPUT. The entry data will be confirmed.
8. Press the menu key [Set Cmpl] to determine the entry data.

[Note]

- Required

  When the number of workpieces already "Machined" matches the "Required" number of workpieces, an alarm is raised to stop the machine. Press the Reset key  to reset the alarm. If automatic power off function has been turned on, the machine stops operation and the power supply is turned off.

  The "Required" number of workpieces can be set up to 99,999,999.
  If you set this item to "0", the counter does not count "Machined" workpieces.
  If you want to change the value of "Required", clear the scheduled completion time (SCHD COMPL Time)

- Machined

  The value of this item is incremented by one for each instance of M56 issued after machining of one workpiece.

  When the value reaches the "Required" number of workpieces, an alarm is raised to stop the machine. Pressing the Reset key  clears only this item to "0".

  The "Machined" number of workpieces can be set up to 99,999,999. If you want to change the value of "Machined", clear the scheduled completion time (SCHD COMPL Time).

- Total Quantity

  This item indicates the total number of workpieces machined. The value is incremented in response to M56.
  The value is incremented to a maximum of 99,999,999 and returns to "0".

- Running Time

  The automatic operation run time is added.
  To clear counter data, set the counter you want to clear to zero.

- Current Time

  If you want to change the value of "Current Time", clear the scheduled completion time (SCHD COMPL Time)

- Power ON Time

  Only "0" can be set.

- "Cycle Time", "Cutting Time" and "SCHD COMPL Time" can not be set. They are only displayed.
- The menu keys "Cnt Set" and "Set Cmpl" appear as the upper menu.
  When "Cnt Set" is selected, "Set Cmpl" is displayed. When "Set Cmpl" is selected, "Cnt Set" is displayed.

6.8.4 Setting and displaying the tool offset

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.

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 the Page key ↑ and/or ↓ to display the data of the desired number.

6.8.4.1 Setting the tool wear data

[Procedure]

Add (increment) the offset amount to the preset value

1. Press the Offset key OFFSET.

The previously selected screen appears.

2. Press the menu key [Offset].

The Offset screen appears.

3. If the "INC" is not displayed on the upper right corner of the screen, press the menu key [ABS/INC].

The "INC" is reversed.

4. Use the Arrow keys ⬆ ⬇ or the Tab keys TAB ⬅ TAB ➡ to move the cursor to the input field in which you want to enter data.

5. Press the alphanumeric key.

Enter the data.

6. Press the Input key INPUT.

The data is added to the original data and stored.

[Procedure]

Input an absolute value

1. Press the Offset key OFFSET.

The previously selected screen appears.

2. Press the menu key [Offset].

The Offset screen appears.

3. If the "ABS" is not displayed on the upper right corner of the screen, press the menu key [ABS/INC].

The "ABS" is reversed.

4. Use the Arrow keys ⬆ ⬇ or the Tab keys TAB ⬅ TAB ➡ to move the cursor to the input field in which you want to enter data.

5. Press the alphanumeric key.

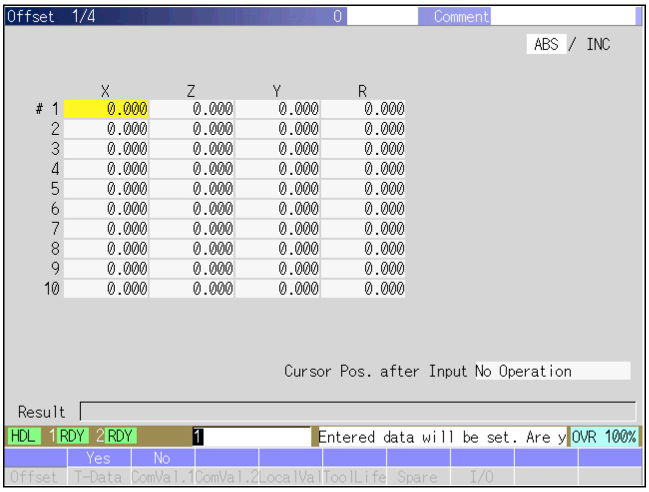
Enter the data.

6. Press the Input key INPUT.

The message "Entered data will be set. Are you sure?" is displayed together with the menu keys [Yes] and [No].

7. Press the menu key [Yes].

The data is overwritten.



6.8.4.2 Setting the tool data

[Procedure]

Add (increment) the offset amount to the preset value

1. Press the Offset key .

   The previously selected screen appears.

2. Press the menu key [T-Data].

   The Tool Data screen appears.

3. If the "INC" is not displayed on the upper right corner of the screen, press the menu key [ABS/INC].

   The "INC" is reversed.

4. Use the Arrow keys or the Tab keys to move the cursor to the input field in which you want to enter data.

5. Press the alphanumeric key.

   Enter the data.

6. Press the Input key .

   The entered value is added.

[Procedure]

Input an absolute value

1. Press the Offset key [OFFSET].

   The previously selected screen appears.
2. Press the menu key [T-Data].

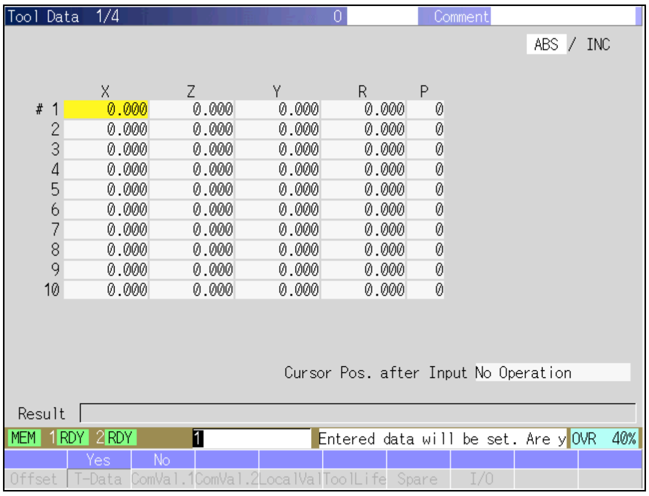
   The Tool Data screen appears.
3. If the "ABS" is not displayed on the upper right corner of the screen, press the menu key [ABS/INC].

   The "ABS" is reversed.
4. Use the Arrow keys [↑][↓] or the Tab keys [TAB ⇐][TAB ⇒] to move the cursor to the input field in which you want to enter data.
5. Press the alphanumeric key.

   Enter the data.
6. Press the Input key [INPUT].

   The message "Entered data will be set. Are you sure?" is displayed together with the menu keys [Yes] and [No].
7. Press the menu key [Yes].

   The data is overwritten.



6.8.4.3 Setting the tool nose radius

If the tool nose is rounded, there will be a slight difference between the programmed shape and actual cutting shape. You can perform an automatic correction of the tool nose radius by setting the tool nose radius correction.

[Procedure]

Set the data in the R and P fields using the same procedure as setting the offset and tool data.
See *<6.8.4.1 Setting the tool wear data>* and *<6.8.4.2 Setting the tool data>*.

6.8.4.4 Clearing all screen data

Use the following procedure to clear all data displayed on a screen at a time.

[Procedure]

1. Display the screen for which all of the displayed data is to be cleared.
2. Press the Shift key SHIFT and the Line deletion key CAN.

   The message "Is it eliminated? Yes/No" is displayed together with the menu keys [Yes] and [No].
3. Press the menu key [Yes].

   All the data is cleared to 0.

[Note]

- The all clear function is enabled with only a screen. Executing this function does not affect data in the other pages.
- When the menu key [UNDO] is displayed, operation can be returned by one step to the previous operation.

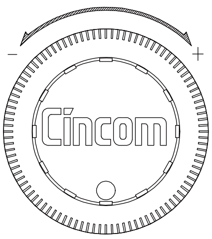
6.8.5 Mechanism adjustment

You can move a selected control axis to an arbitrary position on machine coordinate system by turning the handle (manual pulse generator).

[Procedure]

1. Press the Parameter key PRM. .
2. Press the menu key [Menu SEL] until the menu key [MECH ADJ] is displayed.
3. Press the menu key [MECH ADJ]. The Mechanism Adjustment screen appears.

4. Use the arrow keys ⬆ ⬇ to select an axis you want to feed.
   The name of the selected control axis is displayed in yellow.
5. Press one of the Handle Magnification button x 1, x 10, or x100.
   The Handle Magnification button lamp goes on.
   The larger the number the faster the machine will move.
6. Turn the handle.
   The feed direction is determine by the direction the handle is turned.
   If a feed instruction that exceeds the stored stroke limit is issued, the overrun alarm "M01 Operation error 0007□" will be generated.
   See <*6.8.2 Recovering from overrun status*>.



[Note]

To confirm the position (coordinate) of the workpiece that is moving, select the menu key [POS Data] to open the sub-window.

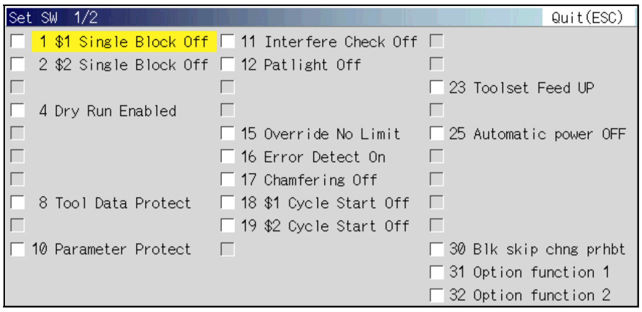
6.8.6 Setting switches

Switches are provided with various control signals for NC operation. You can set the switches to on or off.

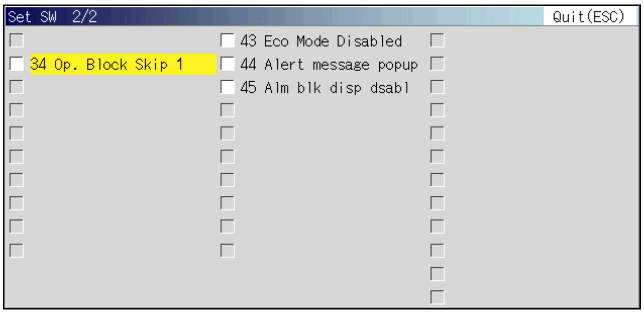
[Procedure]

1. Press the Menu Up/Down selection key ▶.
   The submenus are enabled.
2. Press the menu key [Set SW].
   The Set SW screen appears.
3. Move the cursor to the item (switch) to be set using the Arrow keys ⬆ ⬇, the Tab keys TAB ⬅ TAB ➡, and/or the Page keys.
4. Press the Input key INPUT.

   If the selected switch is ON, the switch is set to OFF and the check mark is cleared.
   If the selected switch is OFF, the switch is set to ON and the check mark is set.



Setting switch 1



Setting switch 2

Explanation

The details of each PLC switch is summarized below.

No.	Item	Setting	Explanation	Remarks
1 2	$1 Single Block Off $2 Single Block Off	ON	Disables single block operation for each axis control group.	
		OFF	Enables single block operation for each axis control group. Set the single block operation switch for each axis control group.	
4	Dry Run Enabled	ON	Enables dry run.	
		OFF	Disables dry run.	
7	Loader Disconnect	ON	The power supply of the bar loader is not checked. (option)	Displayed only when the bar loader is connected.
		OFF	The power supply of the bar loader is checked. (option)	
8	Tool Data Protect	ON	Protects the tool data. Tool data cannot be edited or input/output. While this function is ON, the following message is displayed. "Tool data is protected."	
		OFF	Tool data can be edited or input/output.	
10	Parameter Protect	ON	Protects the user parameters and common variable data from accidental modification or erasure. These data cannot be edited, input or otuput. While this function is ON, the following message is displayed. "Parameter are protected." The user parameter includes CONTROL PARAMETER, AXIS PARAMETER, and SETUP DATA.	
		OFF	Allows input, output, and modification of user parameters and common variable data.	
11	Interfere Check Off	ON	Disables the interference check.	
		OFF	Enables the interference check. Enables the check when the machine is turned on.	
12	Patlight Off	ON	Disables the beacon lamp from lighting even when an alarm is raised.	
		OFF	Enables the beacon lamp to light when an alarm is raised.	
13	Cycle Time ALM. OFF	ON	Disables the cycle time alarm.	Displayed when External Memory Program Operation is enabled.
		OFF	Enables the cycle time alarm.	
15	Override No Limit	ON	Allows the feed rate to be set between 0 and 200%. (The rapid feed rate override is 100% when the rapid feed rate override switch is set at 100% to 200%.)	
		OFF	The feed rate override is 100% when the feed rate override switch is set at more than 100%.	
16	Error Detect On	ON	Enables error detection.	
		OFF	Disables error detection.	
17	Chamfering Off	ON	Disables chamfering in the thread cutting cycle.	
		OFF	Enables chamfering in the thread cutting cycle.	
18 19	$1 Cycle Start Off $2 Cycle Start Off	ON	Disables cycle start of selected axis control group.	
		OFF	Enables cycle start of selected axis control group.	

No.	Item	Setting	Explanation	Remarks
23	Tool set Feed UP	ON	While the door is closed, axis feed rate in tool setting is set at 10 m/min.	
		OFF	While the door is closed, axis feed rate in tool setting is set at 2 m/min.	
25	Automatic power off	ON	Enables the automatic power off function. Automatically turns off the power if an alarm occurs during machine operation in continuous cycles.	Turns to OFF at power-on.
		OFF	Disables the automatic power off function.	
28	Photo For Prepare	ON	Displays the holder picture on the Preparation screen.	
		OFF	Does not display the holder picture on the Preparation screen.	
30	Blk skip chng prhbt	ON	Prohibits turning on/off operation of the menu key [Skip 1].	
		OFF	Enables turning on/off operation of the menu key [Skip 1].	
31	Optional Function 1	ON	Customer-specific function	
32	Optional Function 2	OFF	Customer-specific function	
33	Remote Access Permission	ON	Permits a remote access from PC.	Displayed only when Remote Control function is enabled.
		OFF	Prohibits a remote access from PC.	
34 35 36 37 38 39 40 41 42	Optional Block Skip 1 Optional Block Skip 2 Optional Block Skip 3 Optional Block Skip 4 Optional Block Skip 5 Optional Block Skip 6 Optional Block Skip 7 Optional Block Skip 8 Optional Block Skip 9	ON	Specify whether to skip the block which includes a slash ("/") at the beginning of the block. The number following the slash (/) corresponds to Block Skip 1 to 9. If the number is omitted, the block is regarded as "/1" (Block Skip 1). If this switch is set to ON, the block that contains slash (/) at the beginning of the block is ignored during program execution. Block Skip 1 is linked with the [Skip1] menu on Automatic Operation, On-machine Program Check, and MDI screens.	
		OFF	When this item is set to OFF, the block which includes a slash ("/") at the beginning of the block is not ignored and executed in normal way.	
43	Eco Mode Disabled	OFF	If this item is set to ON, the machine enters into power-saving status when changed to Edit mode (except during program running mode).	
		ON	If this item is set to OFF, the status remains unchanged even when changed to Edit mode.	
44	Alert Message Popup	ON	A message window pops up every time when an alert is issued.	
		OFF	A message window does not pop up (except for certain alerts).	
45	Alm blk disp dsabl	ON	Disables the reverse display of alarm block.	
		OFF	Enables the reverse display of alarm block.	
47	Z1 Axis Stroke Limit Cancel	ON	Cancels the stroke limit of Z1 axis.	Displayed only on the GBL machine. When enabled, switching to GB machine is unavailable. Turn to disabled state at power-on.
		OFF	Limits the Z1 axis stroke.	

6.9 Creating and Editing the Program

This machine achieves various types of machining with five axes (X1, Y1, Z1, X2, and Z2) which are divided into two 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 two axis control groups, and consider the programs created for the axis control groups to be one program when performing various operations for the programs.

The Edit screen for program displays the programs of two axis control groups individually or simultaneously.

You can also create a program for one axis control group only.

WARNING

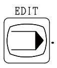
The machine components may move suddenly during some preparation tasks. Keep your hands and body away from the operating area during the preparation.
When preparing to operate using new and existing machining programs, or executing a new machining program, make sure that the tools mounted on the machine do not interfere with a part of the machine. If a tool interferes with another part, this may damage the machine or may result in personal injury.
When you check the tool operations, find the part of the machining program corresponding to tool motion and obey the safety precaution items corresponding to that movement.
For other precautions, see <*Chapter 2 Safety Precautions*>.

6.9.1 Program list

Use the following procedure to display the program numbers, comments, character number, program size, and creation date of a registered program.

Creating a new program and inputting/outputting a program are possible using the List screen.

[Procedure]

1. Press the Edit key 

.

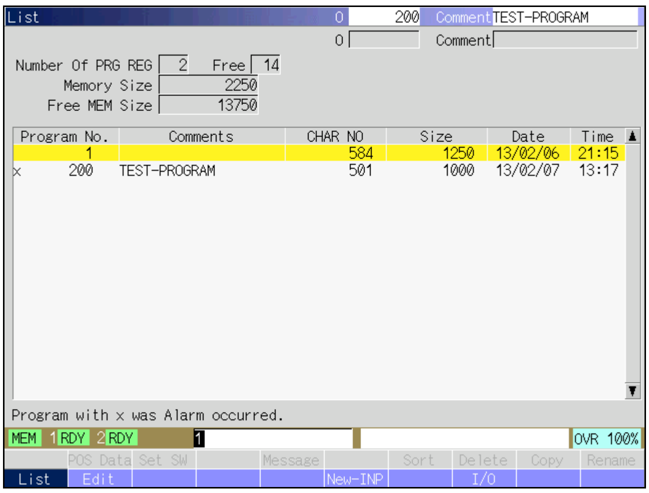
The previously selected screen appears.

2. Press the menu key [List].

The List screen appears.

[Note]

Programs are initially listed in the ascending order of program numbers.

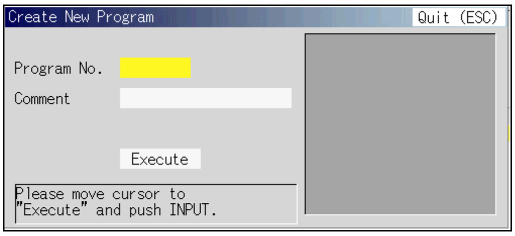


6.9.2 Creating a new program

Use the following procedure to create a new program.

[Procedure]

1. Set the Program edit switch key to "I".
   The program is released from the protected state.
2. Press the Edit key .
   The previously selected screen appears.
3. 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.
4. Press the menu key [New-INP].
   The Create New Program screen appears.
5. Press the Numeric keys to enter the new program number and then press the Input key .
   (The program number can have up to eight digits.)
6. Use the Arrow keys to move the cursor to "Comment", enter the comment and then press the Input key . The comment can be omitted.
7. Use the Arrow keys to move the cursor to "Execute" and then press the Input key . The program number is stored and the Edit screen for new program appears.



[Note]

- The program number can be of up to 8 digits. However, do not assign the numbers between 9000 and 9999. These numbers are reserved by the manufacturer for customized programs.
- While the Program edit switch key is set to "O", you cannot edit the program or machining data.

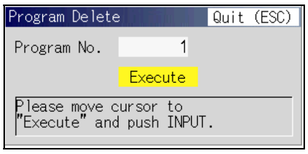
6.9.3 Editing individual programs

6.9.3.1 Deleting a program

Use the following procedure to delete a single program.

[Procedure]

1. Set the Program edit switch key to "I".
   The program is released from the protected state.
2. Press the Edit key .
   The previously selected screen appears.
3. Press the menu key [List].
   The List screen appears.
4. Use the Arrow keys or the Page key to move the cursor to the program to be deleted.
5. Press the Menu Up/Down selection key to enable the submenu key.
6. Press menu key [Delete].
   The Program Delete screen appears.
   The last selected program number is displayed, if you want to delete a different program, use the Arrow keys to move the cursor to "Program No.", enter the number of the program to be deleted and then press the Input key .
7. Use the Arrow keys to move the cursor to "Decision" and then press the Input key .



8. The menu keys [Yes] and [No] are displayed. Press the menu key [Yes].
   The specified program is deleted.

[Note]

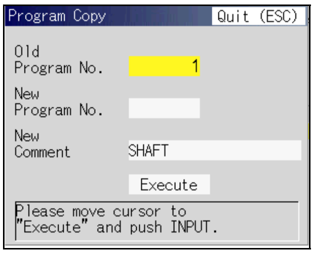
While the Program edit switch key is set to "O", you cannot edit the program or machining data.

6.9.3.2 Copying a program

Use the following procedure to copy an existing program with a different number.

[Procedure]

1. Set the Program edit switch key to "I".
   The program is released from the protected state.
2. Press the Edit key.
   The previously selected screen appears.
3. Press the menu key [List].
   The List screen appears.
4. Use the Arrow keys or the Page key to move the cursor to the program to be copied.
5. Press the Menu Up/Down selection key to enable the submenu key.
6. Press the menu key [Copy].
   The Program Copy screen appears.
7. Use the Arrow keys to move the cursor to "New Program No.", enter the new program number for copied program and then press the Input key.
8. Use the Arrow keys to move the cursor to "New Comment", enter the new comment for copied program and then press the Input key.
   This comment can be omitted.
9. Use the Arrow keys to move the cursor to "Decision" and then press the Input key.
   The specified program is copied.



[Note]

While the Program edit switch key is set to "O", you cannot edit the program or machining data.

6.9.3.3 Changing a program number

Use the following procedure to change a program number that has been stored.

[Procedure]

1. Set the Program edit switch key to "I".
The program is released from the protected state.

2. Press the Edit key.
The previously selected screen appears.

3. Press the menu key [List].
The List screen appears.

4. Use the Arrow keys or the Page key to move the cursor to the program to be changed.

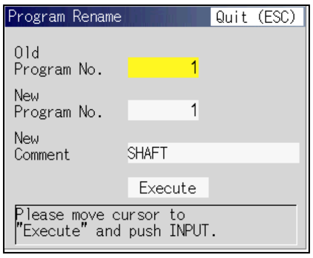
5. Press the Menu Up/Down selection key to enable the submenu key.

6. Press the menu key [Rename].
The Program Rename screen appears.

7. Use the Arrow keys to move the cursor to "New Program No.", enter the new program number and then press the Input key. When changing the comment only, enter the same program number as before.

8. Use the Arrow keys to move the cursor to "New Comment", enter the new comment and then press the Input key.
This comment can be omitted.

9. Use the Arrow keys to move the cursor to "Decision" and then press the Input key. The program number is changed to the specified number.



[Note]

While the Program edit switch key is set to "O", you cannot edit the program or machining data.

6.9.3.4 Sorting program list

This section explains the procedure for setting the order in which programs are listed.

[Procedure]

1. Press the Edit key.

   The previously selected screen appears.

2. Press the menu key [List].

   The List screen appears.

3. Press the Menu Up/Down selection key to enable the submenu key.

4. Press the menu key [Sort].

   The List display Arrangement screen appears.

5. Use the Arrow keys to move the cursor to "Sort Item". Enter a number corresponding to "Program No." or "Date".

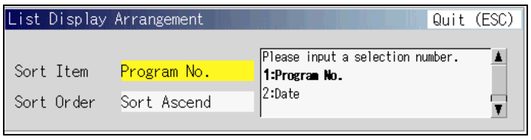
6. Press the Input key.

   The programs are sorted in the specified order.

7. Use the Arrow keys to move the cursor to "Sort Order". Enter a number corresponding to "Sort Ascend" or "Sort Descend".

8. Press the Input key.

   The multiple programs are sorted in the specified order.



9. When you have finished, press the Escape key to close the dialog box.

6.9.4 Editing a program

6.9.4.1 Selecting an axis control group for editing a program

Take the following procedure to select the target axis control group when editing a program.

[Note]

Up to 247 characters per block can be entered. (A space and end of block are counted as a character.) Up to 2,000 blocks can be entered per axis control group.
The allowable size for entire program is 96,000 characters or less.
To edit the program, the memory area must have at least 500 characters of free area.

A comment enclosed by "(" and ")" is regarded as one block.

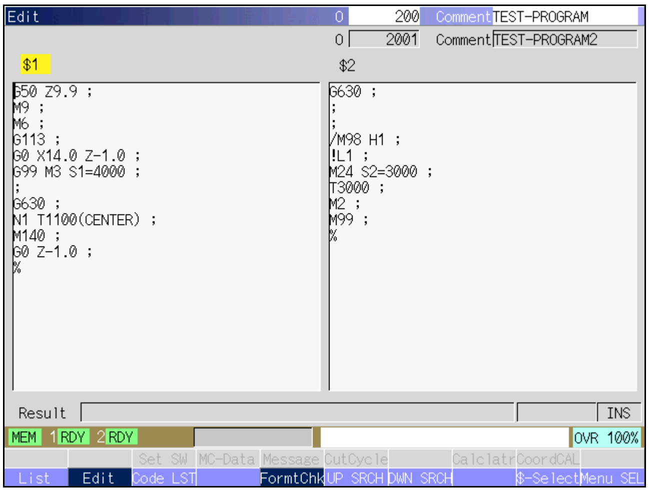
[Procedure]

To display two control axis groups simultaneously

1. Select the Edit screen.

   The programs of two axis control groups are usually displayed at the same time.

2. Press the menu key [$-Select].
   The cursor moves to $1, $2, then back to $1 each time [$-Select] is pressed.



[Procedure]

To display single control axis group

1. Select the Edit screen.

   The programs of two axis control groups are usually displayed at the same time.

2. Press the menu key [Menu SEL].
3. Press the menu key [1 Lines].

   Only the program for control axis group 1 is displayed.

6.9.4.2 General editing operations

You can edit programs using the following procedure:

[Procedure]

See *<6.7.2.3 Editing MDI program>*.

[Note]

- While the Program edit switch key is set to "O", you cannot edit the program or machining data.
- Press the Input key INPUT to save the effect of editing.

(1) Insert mode and overwrite mode switching

[Procedure]

1. Press the Insert key INS.

Each time the Insert key is pressed, the mode is switched as insert → overwrite → insert. The mode is also switched to the insert mode if the Input key INPUT is pressed in the overwrite mode.

The message "INS" or "OVR" is displayed on the lower right of the screen.

(2) Entering a character

[Procedure]

1. Use the Arrow keys or the Tab keys TAB TAB to move the cursor to the position where you want to enter a character.

2. Press the Insert key INS. The display on the lower right of the screen will change to "INS".

3. Enter a character using the alphanumeric keys.

Example: To type "C" after "AB":

1. Move the cursor to the right side of "B".

   AB|

2. Press the Insert key INS.
   (Make sure that the status display field at the lower right corner of the screen indicates "INS".)
3. Press the C key to type "C".

   ABC|

(3) Replacing a character

[Procedure]

1. Use the Arrow keys [↑][↓][←][→] or the Tab keys [TAB ←][TAB →] to move the cursor over the character to be edited.

2. Press the Insert key [INS]. The display on the lower right of the screen will change to "OVR".

3. Enter a character using the alphanumeric keys.

The original character will be overwritten.

Example: To change "ABC" to "ABD":

1. Move the cursor before "C".

   AB|C

2. Press the Insert key [INS].
   (Make sure that the status display field at the lower right corner of the screen indicates the "OVR".)
3. Press the [D] key to type "D".

   ABD

(4) Inserting character

[Procedure]

1. Use the Arrow keys [↑][↓][←][→] or the Tab keys [TAB ←][TAB →] to move the cursor to the position where you want to insert a character.

2. Insert additional character using the alphanumeric key.

The characters will be inserted.

Example: To type in "D" between "B" and "C" of "ABC":

1. Move the cursor between "B" and "C".

   AB|C

2. Press the Insert key [INS].
   (Make sure that the status display field at the lower right corner of the screen indicates "INS".)
3. Press the [D] key to type "D".

   ABD|C

(5) Deleting a character

- Deleting a character after the cursor position

[Procedure]

1. Use the Arrow keys ↑ ↓ ← → or the Tab keys TAB ← TAB → to move the cursor to the left side of character to be deleted.

2. Press the Delete key DEL.

The character after the cursor position will be deleted.

Example: To delete "B" in the "ABC":

1. Move the cursor before "B".

   A|BC

2. Press the Delete key DEL.

   A|C

- Deleting the character in front of the cursor

[Procedure]

1. Use the Arrow keys ↑ ↓ ← → or the Tab keys TAB ← TAB → to move the cursor to the right side of character to be delete.

2. Press the Backspace key BS.

The character in front of the cursor will be deleted.

Example: To delete "B" from a previously entered "ABC":

1. Move the cursor after "B".

   AB|C

2. Press the Backspace key BS.

   A|C

(6) Deleting a line

[Procedure]

1. Move the cursor to the block to be deleted using the Arrow keys [↑][↓][←][→] and the Tab keys [TAB ←][TAB →].

2. Press the Line deletion key [CAN].

The character string in the block located by the cursor is deleted.

Example: To delete "DEF;" in the "ABC;DEF;":

1. Move the cursor to the block to be deleted.

```
ABC;
|DEF;
```

2. Press the Line deletion key [CAN].

```
ABC;
|
```

[Note]

If the block where the cursor is positioned does not fit in the display screen, deletion is not possible.

(7) Cleaning a screen

[Procedure]

1. When the screen to be deleted is displayed, press the Shift key [SHIFT] and the Line deletion key [CAN].

All blocks on the screen will be cleared.

Example: Delete a screen displaying multiple program blocks.

```
|ABC;
DEF;
GHI;
```

1. Press the Shift key [SHIFT] and the Line deletion key [CAN].

```
|
```

[Note]

The block that does not fit in the display screen cannot be deleted.

6.9.4.3 Undo

Pressing the menu key [Undo] returns the program to the state just before the Input key was pressed.

[Procedure]

1. Once the Input key is pressed while the Edit screen is displayed, [Undo] appears in the upper menu..
2. Press the menu key [Undo].
3. The program state just before the Input key was pressed is displayed.

   The menu key [Undo] no longer appears.
   If you edit the program and press the Input key again, the menu key [Undo] reappears.

6.9.4.4 Find

Take the following procedure to search for a character string.

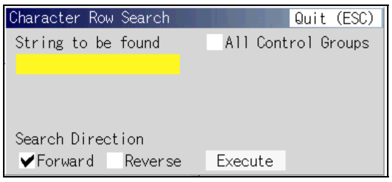
[Procedure]

1. Press the Edit key EDIT.

   The previously selected screen appears.
2. Press the menu key [Edit].

   The Edit screen appears.
3. Press the menu key [Menu SEL] twice.
4. Press the menu key [Find].

   The Character Row Search screen appears.
5. Enter the character string (you want to find) into the "String to be found" text box, then press the Input key INPUT.
6. Use the Arrow keys ⬆ ⬇ to move the cursor to "Reverse" or "Forward", then press the Input key INPUT to decide the search direction.
7. To search the programs of both control groups for the character string, use the Arrow keys ⬆ ⬇ or the Tab keys TAB ⬅ TAB ➡ o move the cursor to "All Control Groups", then press the Input key INPUT.
8. Use the Arrow keys ⬆ ⬇ to move the cursor to "Execute", then pressing the Input key INPUT starts searching for the specified character string.
9. When you have finished, press the Escape key ESC to close the dialog box.



[Note]

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.
All Control Groups	The find function searches the entire program of axis control groups 1 and 2

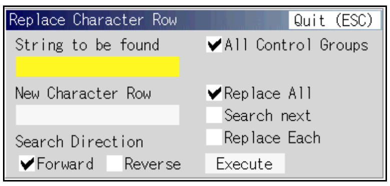
6.9.4.5 Replace

Take the following procedure to replace a character string.

[Procedure]

Replace all of the character strings in the program

1. Set the Program edit switch key to "I".
   The program is released from the protected state.
2. Press the Edit key.
   The previously selected screen appears.
3. Press the menu key [Edit].
   The Edit screen appears.
4. Press the menu key [Menu SEL] twice.
5. Press the menu key [Replace].
   The Replace Character Row screen appears.
6. Enter the character string to be replaced into the "String to be found" text box, then press the Input key.
7. Use the Arrow keys to move the cursor to the "New Character Row" text box, enter a new character string, then press the Input key.
8. Use the Arrow keys to move the cursor to "Reverse" or "Forward", then press the Input key to decide the search direction.
9. To replace character strings for all axis control groups, move the cursor to "All Control Groups" using the Arrow keys and the Tab keys and enter the check mark for this item by pressing the Input key.
10. Use the Arrow keys or the Tab keys to move the cursor to "Replace All", then press the Input key.
11. Use the Arrow keys to move the cursor to "Execute", then press the Input key.
12. The character string is replaced for the program of the axis control group currently edited or for the programs of all of the displayed axis control groups.
13. When you have finished, press the Escape key to close the dialog box.



[Procedure]

Replace a part of character strings in the program

1. Set the Program edit switch key to "I". The program is released from the protected state.

2. Press the Edit key. The previously selected screen appears.

3. Press the menu key [Edit]. The Edit screen appears.

4. Press the menu key [Menu SEL] twice.

5. Press the menu key [Replace]. The Replace Character Row screen appears.

6. Enter the character string to be replaced into the "String to be found" text box, then press the Input key.

7. Use the Arrow keys to move the cursor to the "New Character Row" text box, enter a new character string, then press the Input key.

8. Use the Arrow keys to move the cursor to "Reverse" or "Forward", then press the Input key to decide the search direction.

9. Use the Arrow keys or the Tab keys to move the cursor to "Replace Each", then press the Input key.

10. Use the Arrow keys to move the cursor to "Execute", then press the Input key.

Searching for the specified character string starts.

To skip replacing the found string, use Arrow key to move the cursor to "Search next" and press the Input key, then select "Execute" and press the Input key.

To replace the found string, move the cursor to "Execute", and press the Input key.

11. Repeat step 9 to continue find and replace operation.

12. When you have finished, press the Escape key to close the dialog box.

[Note]

- Replace function is disabled during editing.
- Replace options

Replace All	Replaces all character strings in the program of the control axis group being edited.
Search next	Start searching from the cursor position.
Replace Each	Start replacing from the cursor position.
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.
All Control Groups	The find function searches the entire program of axis control groups 1 and 2.

- While the Program edit switch key is set to "O", you cannot edit the program or machining data.

6.9.4.6 Copy

The copy function copies the specified blocks in the program.

[Procedure]

1. Set the Program edit switch key to "I".
   The program is released from the protected state.
2. Press the Edit key.
   The previously selected screen appears.
3. Press the menu key [Edit].
   The Edit screen appears.
4. Press the menu key [Menu SEL] twice.
5. Use the Arrow keys ⬆ ⬇ to move the cursor in the line containing the character string you wish to copy.
6. Press the menu key [Range].
   The block at the cursor position is highlighted (reverse video).
7. Use the Arrow keys ⬆ ⬇ to move the cursor to the last block of the blocks to be copied.
   All blocks to be copied are highlighted.
8. Press the menu key [Copy].
   Reverse display is canceled and the specified character strings are stored.
9. Use the Arrow keys ⬆ ⬇ to move the cursor to the start of the character string where you wish to insert the selected blocks.
10. Press the menu key [Paste].
    The character string copied in step 8 is inserted at the current cursor position. (The status display "Editing... " appears at the lower right corner of the screen.)
11. Press the Input key INPUT.
    The status display "Editing... " at the lower corner of the screen disappears, indicating that the pasted character string has been accepted.

[Note]

- If you switch to another screen before pressing the Input key INPUT while pasting the copied character string, the copy processing is canceled, and the program is restored to the original status.
- Since the copied program is retained, it can be inserted at several places.
- While the status display "Editing... " is displayed at the lower right corner of the screen, the menu key [Range] is not active. In this case, press the Input key INPUT to confirm the entered data.
- While the Program edit switch key is set to "O", you cannot edit the program or machining data.

6.9.4.7 Move

The move function moves the specified blocks in a program to the desired position.

[Procedure]

1. Set the Program edit switch key to "I".
   The program is released from the protected state.
2. Press the Edit key.
   The previously selected screen appears.
3. Press the menu key [Edit].
   The Edit screen appears.
4. Press the menu key [Menu SEL] twice.
5. Use the Arrow keys ↑ ↓ to move the cursor to the first block of the blocks to be moved.
6. Press the menu key [Range].
   The block at the cursor position is highlighted (reverse video).
7. Use the Arrow keys ↑ ↓ to move the cursor to the last block of the blocks to be moved.
   All blocks to be moved are highlighted.
8. Press the menu key [Cut]. (The message "Editing... " appears at the lower right corner of the screen.)
   The specified character strings are cut and stored.
9. Use the Arrow keys ↑ ↓ to move the cursor to the destination location.
10. Press the menu key [Paste].
    Character string stored in step 8 is inserted where the cursor is located.
11. Press the Input key.
    The status display "Editing... " at the lower corner of the screen disappears, indicating that the pasted character string has been accepted.

[Note]

- If you switch to another screen before pressing the Input key while pasting the move (cut) character string, the move processing is canceled, and the program is restored to the original status.
- Since the cut program is retained, it can be inserted at several places.
- While the status display "Editing... " is displayed at the lower right corner of the screen, the menu key [Range] is not active. In this case, press the Input key to confirm the entered data.
- While the Program edit switch key is set to "O", you cannot edit the program or machining data.

6.9.4.8 Cutting cycle

Take the following procedure to enter setup items which will automatically create a cutting cycle program with multiple blocks.

[Procedure]

Inserting G92 thread cutting canned cycle

1. Set the Program edit switch key to "I".
   The program is released from the protected state.
2. Press the Edit key.
   The previously selected screen appears.
3. Press the menu key [Edit].
   The Edit screen appears.
4. Move the cursor onto the program you want to insert a cutting cycle by using Arrow keys ⬆ ⬇ and menu key [$-Select].
5. Press the Menu Up/Down selection key ▶ to enable the submenu key.
6. Press the menu key [CutCycle].
   The Cycle Selection screen appears.
7. Use the Arrow keys ⬆ ⬇ to select "G92 Thread Cutting Canned Cycle", then press the Input key.
   The G92 Thread Cutting Canned Cycle screen appears.
8. Use the Arrow keys ⬆ ⬇ to move the cursor to "Material". Enter the number for the material, then press the Input key. The selected material is set. The cursor moves to the next item.
9. Use the Arrow keys ⬆ ⬇ to move the cursor to "Lead(L) ". Enter the number for the thread, then press the Input key. The selected thread lead is set. The cursor moves to the next item.
10. Use the Arrow keys ⬆ ⬇ to move the cursor to "Tool shift(a)". Press the Numeric keys to enter the amount of tool shift, then press the Input key.
    The entered new value is set. The cursor moves to the next item.
11. Use the Arrow keys ⬆ ⬇ to move the cursor to "Thread Diameter(M)". Press the Numeric keys to enter the thread diameter, then press the Input key.
    The entered new value is set. The cursor moves to the next item.
12. Use the Arrow keys ⬆ ⬇ to move the cursor to "Effective Length(Za)". Press the Numeric keys to enter the effective length of thread, then press the Input key. The entered new value is set. The cursor moves to the next item.
13. Use the Arrow keys ⬆ ⬇ to move the cursor to "Cycle INS", then press the Input key.
    The screen closes and the cutting cycle program is inserted into the axis control group being edited.
    (The message "Editing... " is displayed at the lower right of the screen.)
14. Press the Input key.
    The status display "Editing... " at the lower corner of the screen disappears, indicating that the pasted character string has been accepted.

[Note]

- If you switch to another screen before pressing the Input key to confirm the inserted cutting cycle program, the cutting cycle insertion processing is canceled, and the program is restored to the original status.
- Only the G92 thread cutting canned cycle for outer diameter can be inserted.
- While the Program edit switch key is set to "O", you cannot edit the program or machining data.

6.9.4.9 Calculator

Take the following procedure for calculation with the displayed calculator.

[Procedure]

1. Press the Edit key EDIT.

   The previously selected screen appears.

2. Press the menu key [Edit].

   The Edit screen appears.

3. Press the Menu Up/Down selection key ▶ to enable the submenu key.

4. Press the menu key [Calclatr].

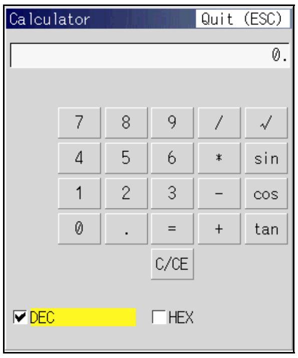
   The Calculator screen appears.

5. Press the Alphanumeric keys to enter the formula.

6. Press the Shift key SHIFT and the Equal key 3 =. Or press the Input key INPUT.

   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 the Line deletion key CAN to erase the previous calculation result.

8. Press the Escape key ESC to close the Calculator screen at completion of operation.

[Procedure]

How to use the √ sin cos and tan

1. Execute the steps 1 to 4 of the Procedure 1.
For example, to calculate "sin30", enter "30".

2. Move the yellow cursor on the Calculator screen to sin by pressing the Arrow keys ⬆ ⬇.

3. Press the Shift key SHIFT and the Space key 6.
The data is calculated. The calculation result for "sin30" is displayed.
To calculate the subsequent data, press the Line deletion key CAN to erase the previous calculation result.
Press the Escape key ESC to close the Calculator screen at completion of operation.

[Note]

The above explains decimal calculation. To do hexadecimal calculation, press the Arrow keys ⬅ ➡ 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 Arrow keys ⬅ ➡ to select "HEX" after the operation in step 6.

6.9.4.10 Coordinate calculation

Take the following procedure to automatically calculate defined pattern coordinates.

[Procedure]

1. Press the Edit key [EDIT].

   The previously selected screen appears.
2. Press the menu key [Edit].

   The Edit screen appears.
3. Press the Menu Up/Down selection key ▶ to enable the submenu key.
4. Press the menu key [CoordCAL].

   The Coordinate Calculation screen appears.
5. Press the Arrow keys ⬆ ⬇ and/or Tab keys [TAB ⇐] [TAB ⇒].

   Move the cursor to the pattern (drawing) you want to use.
6. Press the Input key [INPUT].

   The selected pattern screen appears.
7. Use the Tab keys [TAB ⇐] [TAB ⇒] to move the cursor to "Rotate 90-DEG".

   Keep pressing the Input key [INPUT] until the desired drawing appears.
8. Use the Tab keys [TAB ⇐] [TAB ⇒] to move the cursor to the data input field.
9. Press the Alphanumeric keys to enter the values for calculation, then press the Input key [INPUT].
10. Use the Tab keys [TAB ⇐] [TAB ⇒] to move the cursor to "Calculation", then press the Input key [INPUT]. The calculation result is displayed.

    It is also displayed in the calculation result display area on the Edit screen.
11. Press the Escape key [ESC] twice to close the Coordinate Calculation screen at completion of operation.

6.9.4.11 Insert code

This function is used to select a code (M, G, Special G, or T code) and insert it into the program.

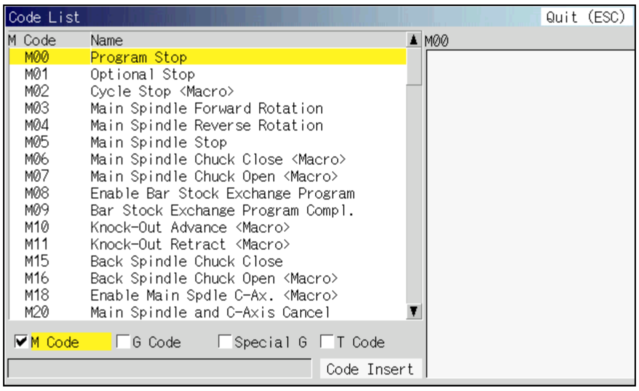
[Procedure]

1. Press the Edit key EDIT.
The previously selected screen appears.

2. Press the menu key [Edit].
The Edit screen appears.

3. Move the cursor onto the program you want to insert a code by using Arrow keys and menu key [$-Select].

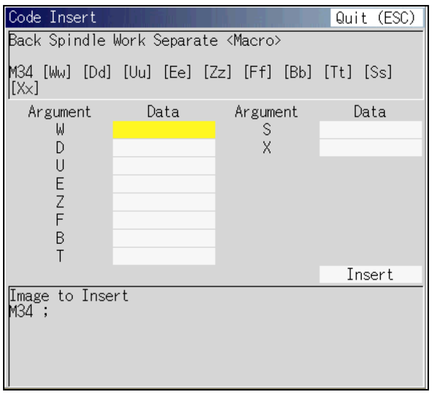
4. Press the menu key [Code LST].
The Code List screen appears.



5. Press the Tab key TAB TAB to move the cursor horizontally to the desired code type, and press the Input key INPUT to determine it.

6. Press the Arrow keys to move the cursor vertically to select a code you want to insert.

7. Press the Tab key TAB TAB to move the cursor to "Code Insert" field, and press the Input key INPUT.
The Code Insert screen appears.



8. Use the Arrow keys and Tab keys TAB TAB to move the cursor to an argument field you want to specify.

9. Enter an argument with the alphanumeric key, and press the Input key INPUT.
The code to be inserted is displayed in the "Image to Insert" field.

10. When you have specified the argument, move the cursor to "Insert" field by using the Arrow key and press the Input key.

The specified code is inserted into the program currently being edited. (The message "Editing..." is displayed at the lower right of the screen.)

11. Press the Input key.

The message "Editing... " is cleared and the change you have made on the program is determined.

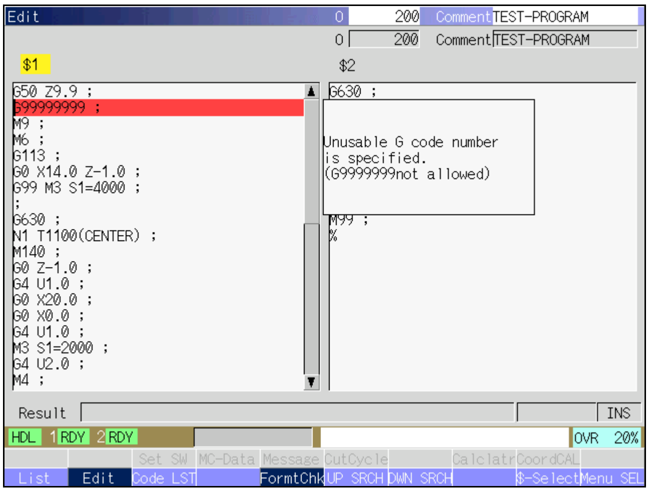
[Note]

If you switch to another screen before pressing the Input key after the code is inserted into the program, code insertion processing is canceled and the program is restored to the original status.

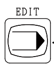
Optional codes are not displayed in Code List screen.

6.9.4.12 Format check

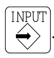
On the Edit screen, you can check if the syntax error is contained in your machining program. If a syntax error is found, the block containing a syntax error is reversed (red). Move the cursor to the error block, the contents of an error is displayed in the box adjacent to the error block.



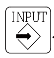
[Procedure]

1. Press the Edit key .
The previously selected screen appears.

2. Press the menu key [Edit].
The Edit screen appears.

3. Edit the machining program, and press the Input key .
The format check is performed. The block that contains a syntax error is reversed in red, and the menu keys [UP SRCH] and [DWN SRCH] appear.

4. Pressing the menu key [UP SRCH] or [DWN SRCH] moves the cursor to the error block one by one.

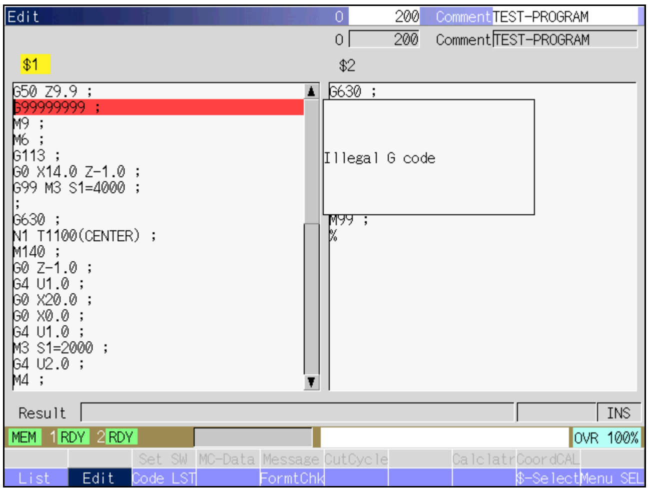
5. Modify the machining program according to the error contents, and press the Input key .
The format check is performed again. If no syntax error is found, the reverse display is canceled. (While editing the machining program, the reverse display is canceled and the menu keys [UP SRCH] and [DWN SRCH] disappear.)

[Note]

The Format Check function is enabled at power on. If you press the menu key [FormtChk] to deselect the menu, the Format Check function is disabled (reverse display of error block is canceled).

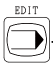
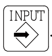
6.9.4.13 Display of alarm block

If an alarm occurs while the machining program is running, the error block is reversed in red on the Edit screen. You can easily identify the block that contains an alarm.



[Procedure]

When an alarm occurs in machining program while running the automatic operation or on-machine program check. (The program stops at an error block and an alarm display appears on the screen.)

1. Press the Reset key  to reset the alarm.
2. Press the Edit key .
   The previously selected screen appears.
3. Press the menu key [List].
   The List screen appears.
4. Call the machining program that was running.
   The Edit screen appears and the block that contains an alarm is reversed in red.
5. Modify the error in the error block, and press the Input key .
   While editing the machining program, the reverse display is canceled and the window showing the error contents also disappears.

[Note]

- Put a check mark on "45 Alarm block OFF" on the Set SW screen to disable the reverse display of alarm block.
- The error information of the alarm block is retained even the machine is powered off by the automatic shutoff function.
- If "x" appears in the Program No. field of the List screen, it indicates that the program contains an alarm block information.
- When the subprogram is called and an alarm occurs in the subprogram, the error block information is contained in subprogram. (The call origin program does not have the error block information.)
- The reverse display is canceled when editing a block other than the error block.

Alarm messages displayed when syntax error occurs

No.	Code	Alarm condition	Alarm message (24 characters by 7 lines)
	T code		
1	T0100's	Specified argument is not Q, H, X, Z, Y, K, E, R, A, nor M.	Invalid argument was specified. Valid arguments are: Q, H, X, Z, Y, K, E, R, A, M (*** argument not allowed)
		Other than 1 is specified for Q argument.	Allowable value for Q argument is 1.
		Other than 2 is specified for K argument.	Allowable value for K argument is 2.
		The value specified for E argument is out of the range between 0 and less than 360.	Allowable range for E argument is between 0 and less than 360.
		Other than 1 or 2 is specified for R argument.	Allowable value for R argument is 1 or 2.
		Other than 1 is specified for M argument.	Allowable value for M argument is 1.
2	T1100's	Specified argument is not Q, X, Z, Y, E, R nor M.	Invalid argument was specified. Valid arguments are: H, X, Z, Y, E, R, M (*** argument not allowed)
		Other than 1 is specified for Q argument.	Allowable value for Q argument is 1.
		The value specified for E argument is out of the range between 0 and less than 360.	Allowable range for E argument is between 0 and less than 360.
		Other than 1 or 2 is specified for R argument.	Allowable value for R argument is 1 or 2.
		Other than 1 is specified for M argument.	Allowable value for M argument is 1.
3	T2100's	Specified argument is not Q, W, X, Z, nor S.	Invalid argument was specified. Valid arguments are: Q, W, X, Z, S (*** argument not allowed)
		Specified argument is not Q, W, X, Z, Y, S nor S in $2.	Invalid argument was specified. Valid arguments are: Q, W, X, Z, Y, S (*** argument not allowed)
		Other than 1 is specified for Q argument.	Allowable value for Q argument is 1.
		The value specified for E argument is out of the range between 0 and less than 360.	Allowable range for E argument is between 0 and less than 360.
4	T3000	Specified argument is not W, Q, nor E.	Invalid argument was specified. Valid arguments are: Q, W, E (*** argument not allowed)
		Other than 1 is specified for Q argument.	Allowable value for Q argument is 1.
		The value specified for E argument is out of the range between 0 and less than 360.	Allowable range for E argument is between 0 and less than 360.
5	T3100's	Specified argument is not Z, W, X, Q, nor E.	Invalid argument was specified. Valid arguments are: Z, W, X, Q, E (*** argument not allowed)
		Other than 1 or 3 is specified for Q argument.	Allowable value for Q argument is 1 or 3.
		The value specified for E argument is out of the range between 0 and less than 360.	Allowable range for E argument is between 0 and less than 360.

No.	Code	Alarm condition	Alarm message (24 characters by 7 lines)
6	T5100 and higher	Specified argument is not X, Y, Z, W, Q, E, nor R.	Invalid argument was specified. Valid arguments are: X, Y, Z, W, Q, E, R (*** argument not allowed)
		Other than 1 or 3 is specified for Q argument.	Allowable value for Q argument is 1 or 3.
		The value specified for E argument is out of the range between 0 and less than 360.	Allowable range for E argument is between 0 and less than 360.
		Other than 1 is specified for R argument.	Allowable value for R argument is 1.
	G code		
1	G0	Specified argument is not X, U, Z, W, Y, V, C, H, ",F", F, ", nor I".	Invalid argument was specified. Valid arguments are: X, U, Z, W, Y, V, C, H, ",F", F, ",I" (*** argument not allowed)
2	G1	Specified argument is not X, U, Z, W, Y, V, C, H, F, A, ",C", ",R", nor I.	Invalid argument was specified. Valid arguments are: X, U, Z, W, Y, V, C, H, F, A, ",C", ",R", I (*** argument not allowed)
3	G2	Specified argument is not X, U, Z, W, Y, V, C, H, I, J, K, R, F, ",C, ",R, P, nor ",I".	Invalid argument was specified. Valid arguments are: X, U, Z, W, Y, V, C, H, I, J, K, R, F, ",C", ",R", P, ",I" (*** argument not allowed)
4	G3	Specified argument is not X, U, Z, W, Y, V, C, H, I, J, K, R, F, ",C, ",R, P, nor ",I".	Invalid argument was specified. Valid arguments are: X, U, Z, W, Y, V, C, H, I, J, K, R, F, ",C", ",R", P, ",I" (*** argument not allowed)
5	G4	Specified argument is not U, P, nor X.	Invalid argument was specified. Valid arguments are: U, P, X (*** argument not allowed)
6	G12.1	Specified argument is not D, nor E=.	Invalid argument was specified. Valid arguments are: D, "E=" (*** argument not allowed)
		Other than 0 or 1 is specified for D argument.	Allowable value for D argument is 0 or 1.
7	G16	Specified argument is not C.	Invalid argument was specified. Valid argument is: C (*** argument not allowed)
8	G32	Specified argument is not X, U, Z, W, F, E, Q, nor A.	Invalid argument was specified. Valid arguments are: X, U, Z, W, F, E, Q, A (*** argument not allowed)
		A value out of range between 0 and 360 is specified for Q argument.	Allowable range for Q argument is between 0 and 360.
9	G34	Specified argument is not X, U, Z, W, F, E, K, nor A.	Invalid argument was specified. Valid arguments are: X, U, Z, W, F, E, K, A (*** argument not allowed)
10	G35	Specified argument is not X, U, Z, W, I, K, R, F, E, nor Q.	Invalid argument was specified. Valid arguments are: X, U, Z, W, I, K, R, F, E, Q (*** argument not allowed)
		A value out of range between 0 and 360 is specified for Q argument.	Allowable range for Q argument is between 0 and 360.
11	G36	Specified argument is not X, U, Z, W, I, K, R, F, E, nor Q.	Invalid argument was specified. Valid arguments are: X, U, Z, W, I, K, R, F, E, Q (*** argument not allowed)
		A value out of range between 0 and 360 is specified for Q argument.	Allowable range for Q argument is between 0 and 360.

No.	Code	Alarm condition	Alarm message (24 characters by 7 lines)
12	G40	Specified argument is not X, U, Z, W, Y, V, C, H, F, I, J, nor K.	Invalid argument was specified. Valid arguments are: X, U, Z, W, Y, V, C, H, F, I, J, K (*** argument not allowed)
13	G41	Specified argument is not X, U, Z, W, Y, V, C, H, F, I, J, nor K.	Invalid argument was specified. Valid arguments are: X, U, Z, W, Y, V, C, H, F, I, J, K (*** argument not allowed)
14	G42	Specified argument is not X, U, Z, W, Y, V, C, H, F, I, J, nor K.	Invalid argument was specified. Valid arguments are: X, U, Z, W, Y, V, C, H, F, I, J, K (*** argument not allowed)
15	G50	Specified argument is not X, U, Z, W, Y, V, C, H, S nor Q.	Invalid argument was specified. Valid arguments are: X, U, Z, W, Y, V, C, H, S, Q (*** argument not allowed)
16	G53	Specified argument is not X, U, Z, W, Y, V, C, H, ",F", nor ",I".	Invalid argument was specified. Valid arguments are: X, U, Z, W, Y, V, C, H, ",F", ",I" (*** argument not allowed)
17	G65	Invalid argument is specified.	Invalid argument was specified. (*** argument not allowed)
		P argument is not specified.	Specify the P argument.
18	G66	Invalid argument is specified.	Invalid argument was specified. (*** argument not allowed)
		P argument is not specified.	Specify the P argument.
19	G66.1	Invalid argument is specified.	Invalid argument was specified. (*** argument not allowed)
		P argument is not specified.	Specify the P argument.
20	G70	Specified argument is not A, P, Q, nor ",I".	Invalid argument was specified. Valid arguments are: A, P, Q, ",I" (*** argument not allowed)
21	G71	Specified argument is not U, R, A, P, Q, W, F, S, T, nor ",I".	Invalid argument was specified. Valid arguments are: U, R, A, P, Q, W, F, S, T, ",I" (*** argument not allowed)
22	G72	Specified argument is not U, R, A, P, Q, W, F, S, T, nor ",I".	Invalid argument was specified. Valid arguments are: U, R, A, P, Q, W, F, S, T, ",I" (*** argument not allowed)
23	G73	Specified argument is not U, R, A, P, Q, W, F, S, T, nor ",I".	Invalid argument was specified. Valid arguments are: U, R, A, P, Q, W, F, S, T, ",I" (*** argument not allowed)
24	G74	Specified argument is not R, X, U, Z, W, P, Q, F, nor ",I".	Invalid argument was specified. Valid arguments are: R, X, U, Z, W, P, Q, F, ",I" (*** argument not allowed)
25	G75	Specified argument is not R, X, U, Z, W, P, Q, F, nor ",I".	Invalid argument was specified. Valid arguments are: R, X, U, Z, W, P, Q, F, ",I" (*** argument not allowed)
26	G76	Specified argument is not R, X, U, Z, W, P, Q, F, nor ",I".	Invalid argument was specified. Valid arguments are: R, X, U, Z, W, P, Q, F, ",I" (*** argument not allowed)
27	G76.1	Specified argument is not R, X, U, Z, W, P, Q, F, nor ",I".	Invalid argument was specified. Valid arguments are: R, X, U, Z, W, P, Q, F, ",I" (*** argument not allowed)
28	G76.2	Specified argument is not R, X, U, Z, W, P, Q, F, A, nor ",I".	Invalid argument was specified. Valid arguments are: R, X, U, Z, W, P, Q, F, A, ",I" (*** argument not allowed)
29	G79	Specified argument is not X, U, Z, W, R, I, K, A, Q, J, F,nor ",F".	Invalid argument was specified. Valid arguments are: X, U, Z, W, R, I, K, A, Q, J, F, ",F" (*** argument not allowed)

No.	Code	Alarm condition	Alarm message (24 characters by 7 lines)
30	G83	Specified argument is not X, U, Z, W, C, H, R, Q, P, F, K, M, ",F", nor ",I".	Invalid argument was specified. Valid arguments are: X, U, Z, W, C, H, R, Q, P, F, K, M, ",F", ",I" (*** argument not allowed)
31	G84	Specified argument is not Z, W, R, F, D, S, Q, ",R", C, E, H, K, M, P, X, U, ",F", nor ",I".	Invalid argument was specified. Valid arguments are: Z, W, R, F, D, S, Q, ", R", C, E, H, K, M, P, X, U, ",F", ",I" (*** argument not allowed)
		Other than ±1, ±2, ±3, ±4, or ±5 is specified for D argument.	Allowable value for D argument is ±1, ±2, ±3, ±4, or ±5.
		A value out of range between 100 and 500 is specified for Q argument.	Allowable range for Q argument is between 100 and 500.
		Other than 1 or 2 is specified for ,R argument.	Allowable value for ",R" argument is 1 or 2.
32	G85	Specified argument is not X, U, Z, W, C, H, R, P, F, K, M, ",F", nor ",I".	Invalid argument was specified. Valid arguments are: X, U, Z, W, C, H, R, P, F, K, M, ",F", ",I" (*** argument not allowed)
33	G87	Specified argument is not X, U, Z, W, C, H, R, Q, P, F, K, M, ",F", nor ",I".	Invalid argument was specified. Valid arguments are: X, U, Z, W, C, H, R, Q, P, F, K, M, ",F", ",I" (*** argument not allowed)
34	G88	Specified argument is not X, U, R, F, D, S, Q, ",R", C, E, H, K, M, P, Z, W, ",F", nor ",I".	Invalid argument was specified. Valid arguments are: X, U, R, F, D, S, Q, ",R", C, E, H, K, M, P, Z, W, ",F", ",I" (*** argument not allowed)
		Other than ±3, ±4, or ±5 is specified for D argument.	Allowable value for D argument is ±3, ±4, or ±5.
		A value out of range between 100 and 500 is specified for Q argument.	Allowable range for Q argument is between 100 and 500.
		Other than 1 or 2 is specified for ",R" argument.	Allowable value for ",R" argument is 1 or 2.
35	G89	Specified argument is not X, U, Z, W, C, H, R, P, F, K, M, ",F", nor ",I".	Invalid argument was specified. Valid arguments are: X, U, Z, W, C, H, R, P, F, K, M, ",F", ",I" (*** argument not allowed)
36	G90	Specified argument is not X, U, Z, W, R, F, nor ",I".	Invalid argument was specified. Valid arguments are: X, U, Z, W, R, F, ",I" (*** argument not allowed)
37	G92	Specified argument is not X, U, Z, W, R, F, E, Q, nor ",I".	Invalid argument was specified. Valid arguments are: X, U, Z, W, R, F, E, Q, ",I" (*** argument not allowed)
		A value out of range between 0 and 360 is specified for Q argument.	Allowable range for Q argument is between 0 and 360.
38	G96	Specified argument is not S nor P.	Invalid argument was specified. Valid arguments are: S, P (*** argument not allowed)
39	G97	–	–
40	G113	Specified argument is not H nor D.	Invalid argument was specified. Valid arguments are: H, D (*** argument not allowed)
		Other than 1, 2, 3, or 5 is specified for H argument.	Allowable value for H argument is 1, 2, 3, or 5.

No.	Code	Alarm condition	Alarm message (24 characters by 7 lines)
41	G114.1	Specified argument is not H, D, R, nor A.	Invalid argument was specified. Valid arguments are: H, D, R, A (*** argument not allowed)
		H argument is not specified.	Specify the H argument. Allowable value for H argument is 1 or 2.
		D argument is not specified.	Specify the D argument. Allowable value for D argument is ±1 or ±2.
		Other than 1 or 2 is specified for H argument.	Allowable value for H argument is 1 or 2.
		Other than ±1 or ±2 is specified for D argument.	Allowable value for D argument is ±1 or ±2.
		A value out of range between 0 and 360 is specified for Q argument.	Allowable range for R argument is between 0 and less than 360.
42	G114.2	Specified argument is not H, D, E, L, R, nor A.	Invalid argument was specified. Valid arguments are: H, D, E, L, R, A (*** argument not allowed)
		H argument is not specified.	Specify the H argument. Allowable value for H argument is 3, 4, or 5.
		D argument is not specified.	Specify the D argument. Allowable value for D argument is ±1 or ±2.
		Other than 3, 4, or 5 is specified for H argument.	Allowable value for H argument is 3, 4, or 5.
		Other than 1 or ±2 is specified for D argument.	Allowable value for D argument is ±1 or ±2.
		A value out of range between 1 and 999 is specified for E argument.	Allowable range for E argument is between 1 and 999.
		A value out of range between 1 and 999 is specified for L argument.	Allowable range for L argument is between 1 and 999.
		A value out of range between 0 and 360 is specified for Q argument.	Allowable range for R argument is between 0 and less than 360.
43	G114.3	Specified argument is not H, D, E, L, R, P, Q, nor A.	Invalid argument was specified. Valid arguments are: H, D, E, L, R, P, Q, A (*** argument not allowed)
		H argument is not specified.	Specify the H argument. Allowable value for H argument is 3, 4, or 5.
		D argument is not specified.	Specify the D argument. Allowable value for D argument is ±9.
		Other than 3, 4, or 5 is specified for H argument.	Allowable value for H argument is 3, 4, or 5.
		Other than ±9 is specified for D argument.	Allowable value for D argument is ±9.
		A value out of range between 1 and 999 is specified for E argument.	Allowable range for E argument is between 1 and 999.
		A value out of range between 1 and 999 is specified for L argument.	Allowable range for L argument is between 1 and 999.
		In millimeter/inch specification: A value out of range between -89000 and 89000 is specified for P argument. In sub-micron/sub-inch specification: A value out of range between -890000 and 890000 is specified for P argument.	In millimeter/inch specification: Allowable range for P argument is between -89000 and 89000. In sub-micron/sub-inch specification: Allowable range for P argument is between -89000 and 890000.

No.	Code	Alarm condition	Alarm message (24 characters by 7 lines)
43	G114.3 (Cont'd)	In millimeter specification: A value out of range between 100 and 25000 is specified for Q argument. In sub-micron/inch specification: A value out of range between 1000 and 250000 is specified for Q argument. In sub-inch specification: A value out of range between 10000 and 250000 is specified for Q argument.	In millimeter specification: Allowable range for Q argument is between 100 and 25000. In sub-micron/inch specification: Allowable range for Q argument is between 100 and 250000. In sub-inch specification: Allowable range for Q argument is between 10000 and 250000.
		A value out of range between 0 and 360 is specified for R argument.	Allowable range for R argument is between 0 and less than 360.
44	G115	Specified argument is not X, Z, nor C.	Invalid argument was specified. Valid arguments are: X, Z, C (*** argument not allowed)
45	G116	Specified argument is not X, Z, nor C.	Invalid argument was specified. Valid arguments are: X, Z, C (*** argument not allowed)
46	G117	Specified argument is not X, Z, nor C.	Invalid argument was specified. Valid arguments are: X, Z, C (*** argument not allowed)
47	G140	–	–
48	G149	Specified argument is not Q, X, Z, nor C.	Invalid argument was specified. Valid arguments are: Q, X, Z, C (*** argument not allowed)
49	G156	–	–
50	G164	Specified argument is not H nor D.	Invalid argument was specified. Valid arguments are: H, D (*** argument not allowed)
		H argument is not specified.	Specify the H argument.
		D argument is not specified.	Specify the D argument.
		Other than 1 or 2 is specified for H argument.	Allowable value for H argument is 1 or 2.
		Other than ±3, ±4, or ±5 is specified for D argument.	Allowable value for D argument is ±3, ±4, or ±5.
51	G173	Specified argument is not X, Z, Y, U, W, V, nor D.	Invalid argument was specified. Valid arguments are: X, Z, Y, U, W, V, D (*** argument not allowed)
52	G200's (except G231)	Other than G231 is specified for G200's.	Invalid G code is specified. (G*** command not allowed)
53	G231	G231 is specified in $2.	Specify G231 in $1.
		Specified argument in $1 is not $1 で X, F, A, nor Z.	Invalid argument was specified. Valid arguments are: X, F, A, Z (*** argument not allowed)
		A argument is not specified in $1.	Specify the A argument.
54	G300's	G300's command is specified.	Invalid G code is specified. (G*** command not allowed)
55	G400's	G400's command is specified.	Invalid G code is specified. (G*** command not allowed)
56	G500's	G500's command is specified.	Invalid G code is specified. (G*** command not allowed)

No.	Code	Alarm condition	Alarm message (24 characters by 7 lines)
57	G600's (except G600, G610, G620 G621, G622, G630 G650, G660)	Other than G600, G610, G620, G621, G622, G630, G650, or G66 is specified for G600's.	Invalid G code is specified. (G*** command not allowed)
58	G600	Other than W argument is specified in $2.	Invalid argument was specified. Valid argument is: W (*** argument not allowed)
		A value other than 0 is specified for W argument in $2.	Allowable value for W argument is 0.
59	G610	Other than W argument is specified in $2.	Invalid argument was specified. Valid argument is: W (*** argument not allowed)
		A value other than 0 is specified for W argument in $2.	Allowable value for W argument is 0.
60	G620	Other than Z argument is specified in $2.	Invalid argument was specified. Valid argument is: Z (*** argument not allowed)
61	G621	Argument was specified.	Invalid argument was specified. (*** argument not allowed)
62	G622	Argument was specified.	Invalid argument was specified. (*** argument not allowed)
63	G630	Argument was specified.	Invalid argument was specified. (*** argument not allowed)
64	G650	Specified argument in $2 is not Z, U, nor W.	Invalid argument was specified. Valid arguments are: Z, U, W (*** argument not allowed)
		A value other than 0 is specified for U argument in $2.	Allowable value for U argument is 0.
		A value other than 0 is specified for V argument in $2.	Allowable value for V argument is 0.
		A value other than 0 is specified for W argument in $2.	Allowable value for W argument is 0.
65	G660	Other than W or X argument is specified in $2.	Invalid argument was specified. Valid arguments are: W, X (*** argument not allowed)
		A value other than 0 is specified for W argument in $2.	Allowable value for W argument is 0.
		A value other than 2 is specified for X argument in $2.	Allowable value for X argument is 2.
66	G700's	G700's command is specified.	Invalid G code is specified. (G*** command not allowed)
67	G800's (except G813, G814, G899)	Other than G813, G814, or G899 is specified for G800's.	Invalid G code is specified. (G*** command not allowed)
68	G814	A value out of range between 0 and 360 is specified for R argument.	Allowable range for R argument is between 0 and less than 360.
69	G900's (except G950, G951, G999)	Other than G950, G951, or G999 is specified for G900's.	Invalid G code is specified. (G*** command not allowed)

No.	Code	Alarm condition	Alarm message (24 characters by 7 lines)
70	G900	Other than $1 or $2 is specified.	Specify the G900 in $1 or $2.
		Specified argument is not X, Z, Y nor B.	Invalid argument was specified. Valid arguments are: X, Z, Y, B (*** argument not allowed)
		X argument is not specified.	Specify the X argument.
		Z argument is not specified.	Specify the Z argument.
		A value out of range between -91.0 and 91.0 is specified for B argument.	Allowable range for L argument is between -91.0 and 91.0.
71	G901	Specified argument is not B.	Invalid argument was specified. Valid arguments is B (*** argument not allowed)
		A value out of range between -91.0 and 91.0 is specified for B argument.	Allowable range for L argument is between -91.0 and 91.0.
72	G902	Specified argument is not B nor D.	Invalid argument was specified. Valid arguments is B, D (*** argument not allowed)
		A value out of range between -91.0 and 91.0 is specified for B argument.	Allowable range for L argument is between -91.0 and 91.0.
		Other than 1 is specified for D argument.	Allowable value for D argument is 1.
73	G950	Specified argument is not X, Z, Y, nor B.	Invalid argument was specified. Valid arguments are: X, Z, Y, B (*** argument not allowed)
		A value out of range between -91.0 and 91.0 is specified for B argument.	Allowable range for B argument is between -91.0 and less than 91.0.
74	G1200 to G1399 are user macro numbers.	–	–
75	G1400 or higher	Greater than G1400 is specified.	Invalid G code is specified. (G*** command not allowed)

No.	Code	Alarm condition	Alarm message (24 characters by 7 lines)
	M code		
1	M12 to M14, M17, M19, M21, M22, M26, M27, M29, M30, M32, M35 to M37, M40 to M42, M44, M45, M49, M57 to M60, M66 to M71, M74 to M76, M90, M91, M109 to M121, M125 to M139, M142 to M150, M152 to M159, M170 to M179, M183 to M189, M192 to M203, M206 to M209, M213 to M237, M240 to M259, M270 to M319, M321 to M349, M352 to M359, M362 to M429, M438 to M451, M454 to M779, M781 to M799, M801, M803 or higher	The M codes described left was specified.	Invalid M code is specified. (M*** command not allowed)
2	M3	–	–
3	M4	–	–
4	M5 to M9	Argument was specified.	Invalid argument was specified. (*** argument not allowed)
5	M10	Specified argument is not Z, F nor T.	Invalid argument was specified. Valid arguments are: Z, T, F (*** argument not allowed)
		A value out of range between 0 and 100 is specified for T argument.	Allowable range for T argument is between 0 and 100.
6	M11	Specified argument is not Z nor F.	Invalid argument was specified. Valid arguments are: Z, F (*** argument not allowed)
7	M15, M16	Argument was specified.	Invalid argument was specified. (*** argument not allowed)
8	M18	Other than C argument is specified.	Invalid argument was specified. Valid argument is: C (*** argument not allowed)
		A value out of range between 0 and 360 is specified for C argument.	Allowable range for C argument is between 0 and less than 360.
9	M20	Argument was specified.	Invalid argument was specified. (*** argument not allowed)
10	M23	–	–
11	M24	–	–
12	M25	Argument was specified.	Invalid argument was specified. (*** argument not allowed)

No.	Code	Alarm condition	Alarm message (24 characters by 7 lines)
13	M28	Specified argument is not S.	Invalid argument was specified. Valid argument is: S (*** argument not allowed)
		A value out of range between 0 and 359 is specified for S argument.	Allowable range for S argument is between 0 and 359.
14	M31	Argument was specified.	Invalid argument was specified. (*** argument not allowed)
15	M33	Specified argument is not W, D, U, nor E.	Invalid argument was specified. Valid arguments are: W, D, U, E (*** argument not allowed)
16	M34	Specified argument is not X, W, D, U, E, Z, F, T, B, S, nor K1.	Invalid argument was specified. Valid arguments are: X, W, D, U, E, Z, F, T, B, S, K1. (*** argument not allowed)
		Other than 1 is specified for K argument.	Allowable value for K argument is 1.
17	M48	Specified argument is not C.	Invalid argument was specified. Valid argument is: C (*** argument not allowed)
		A value out of range between 0 and 360 is specified for C argument.	Allowable range for C argument is between 0 and less than 360.
18	M50	Argument was specified.	Invalid argument was specified. (*** argument not allowed)
19	M51	Specified argument is not X, W, nor F.	Invalid argument was specified. Valid arguments are: X, W, F (*** argument not allowed)
20	M52 to M56 M61 to M65 M72, M73, M77	Argument was specified.	Invalid argument was specified. (*** argument not allowed)
21	M78	Specified argument is not S.	Invalid argument was specified. Valid argument is: S (*** argument not allowed)
		A value out of range between 0 and 359 is specified for S argument.	Allowable range for S argument is between 0 and 359.
22	M79	Argument was specified.	Invalid argument was specified. (*** argument not allowed)
23	M80	–	–
24	M81	–	–
25	M82	Argument was specified.	Invalid argument was specified. (*** argument not allowed)
26	M83	–	–
27	M84	–	–
28	M85 to M89 M92 to M97	Argument was specified.	Invalid argument was specified. (*** argument not allowed)
29	M98	Specified argument is not P, H, L, nor ",D".	Invalid argument was specified. Valid arguments are: P, H, L, ",D" (*** argument not allowed)
		A value out of range between 0 and 99999999 is specified for P argument.	Allowable range for P argument is between 0 and 99999999.
		A value out of range between 0 and 99999 is specified for H argument.	Allowable range for H argument is between 0 and 99999.
		A value out of range between 0 and 9999 is specified for L argument.	Allowable range for L argument is between 0 and 9999.
		Other than 2 is specified for ,D argument.	Allowable value for ",D" argument is 2.

No.	Code	Alarm condition	Alarm message (24 characters by 7 lines)
30	M99	Specified argument is not P nor L.	Invalid argument was specified. Valid arguments are: P, L (*** argument not allowed)
31	M100 to M107	Argument was specified.	Invalid argument was specified. (*** argument not allowed)
32	M108	Specified argument is not U, C, B, D, W, F, S, R, A, K, nor T.	Invalid argument was specified. Valid arguments are: U, C, B, D, W, F, S, R, A, K, T (*** argument not allowed)
		Other than 1 is specified for R argument.	Allowable value for R argument is 1.
		Other than 1 is specified for K argument.	Allowable value for K argument is 1.
33	M122 to M124	Argument was specified.	Invalid argument was specified. (*** argument not allowed)
34	M140	Specified argument is not W nor Z.	Invalid argument was specified. Valid arguments are: W, Z (*** argument not allowed)
35	M141	Specified argument is not W nor Z.	Invalid argument was specified. Valid arguments are: W, Z (*** argument not allowed)
36	M151 M160 to M169	Argument was specified.	Invalid argument was specified. (*** argument not allowed)
37	M180	–	–
38	M181	–	–
39	M182 M190,M191 M204,M205	Argument was specified.	Invalid argument was specified. (*** argument not allowed)
40	M211	Specified argument is not Y.	Invalid argument was specified. Valid argument is: Y (*** argument not allowed)
		Other than 1 or 2 is specified for Y argument.	Allowable value for Y argument is 1 or 2.
41	M212	Specified argument is not Y.	Invalid argument was specified. Valid argument is: Y (*** argument not allowed)
		Other than 1 or 2 is specified for Y argument.	Allowable value for Y argument is 1 or 2.
42	M238	Specified argument is not A.	Invalid argument was specified. Valid argument is: A (*** argument not allowed)
		A value out of range between 1 and 9 is specified for A argument.	Allowable range for A argument is between 1 and 9.
43	M239	Specified argument is not Z, F, T, nor K.	Invalid argument was specified.Valid arguments are: Z, F, T, nor K.
		A value out of range between 0 and 100 is specified for S argument.	Allowable range for S argument is between 0 and 100.
		Other than 1 or 2 is specified for K argument.	Allowable value for K argument is 1 or 2.
44	M260 to M269	Argument was specified.	Invalid argument was specified. (*** argument not allowed)
45	M320	Specified argument is not X, W, U, V, Z, R, nor F.	Invalid argument was specified. Valid arguments are: X, W, U, V, Z, R, F (*** argument not allowed)

No.	Code	Alarm condition	Alarm message (24 characters by 7 lines)
46	M350	Specified argument is not X, Z, nor Y.	Invalid argument was specified. Valid arguments are: X, Z, Y (*** argument not allowed)
		A value out of range between 1 and 32000 is specified for X argument.	Allowable range for X argument is between 1 and 32000.
		A value out of range between 1 and 32000 is specified for Z argument.	Allowable range for Z argument is between 1 and 32000.
		A value out of range between 1 and 32000 is specified for Y argument.	Allowable range for Y argument is between 1 and 32000.
47	M351	Argument was specified.	Invalid argument was specified. (*** argument not allowed)
48	M360	Specified argument is not X, Z, nor Y.	Invalid argument was specified. Valid arguments are: X, Z, Y (*** argument not allowed)
		A value out of range between 1 and 6 is specified for X argument.	Allowable range for X argument is between 1 and 6.
		A value out of range between 1 and 6 is specified for Z argument.	Allowable range for Z argument is between 1 and 6.
		A value out of range between 1 and 6 is specified for Y argument.	Allowable range for Y argument is between 1 and 6.
49	M361 M430 to M437 M452, M453, M780	Argument was specified.	Invalid argument was specified. (*** argument not allowed)
	S code		
1	S number	Other than S1, S2, S3, S5, or S9 is specified.	Invalid S number is specified. (S* command not allowed)
		A value over 10000 is specified for S1 argument.	Allowable range for "S1" argument is between 0 and 10000.
		A value over 8000 is specified for S2 argument.	Allowable range for "S2" argument is between 0 and 8000.
		A value over 5000 is specified for S3 argument.	Allowable range for "S3" argument is between 0 and 5000.
		A value over 7500 is specified for S4 argument.	Allowable range for "S2" argument is between 0 and 7500.
		A value over 5000 is specified for S5 argument.	Allowable range for "S5" argument is between 0 and 5000.
	B code		
1	MB number	A value out of range between 1 and 255 is specified for MB number.	Allowable range for MB number is between 1 and 255.
	N number		
1	N number	A value out of range between 1 and 99999 is specified for N number.	Allowable range for N number is between 1 and 99999.
	Others		
1	Parentheses in comment	Number of parentheses unmatch.	Number of open parentheses "(" do not match the close parentheses ")".
		The specified parentheses are six fold or more.	Specify the parentheses five folds or less.
2	Variable number	The specified variable number of out of range.	The specified variable number of out of range.
3	Arithmetic expression	Illegal arithmetic expression.	The specified arithmetic expression is illegal.
4	No. of characters	Over 250 characters are specified for single block.	Allowable number of characters per block is 250. Use alphanumeric and symbol characters.

6.9.5 Creating and editing machining data

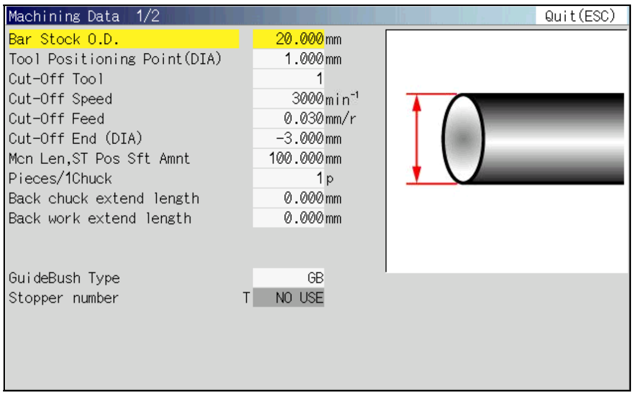
This section explains the procedure for creating and editing the machining data of a program.

You cannot edit the machining data of the program that is being operated.

[Procedure]

Creating and editing machining data

1. Set the Program edit switch key to "|". The program is released from the protected state.
2. Press the Edit key . The previously selected screen appears.
3. Press the menu key [List]. The List screen appears.
4. Call the program you want to edit. The Edit screen appears.
5. Press the Menu Up/Down selection key to enable the submenu key.
6. Press the menu key [MC-Data].
   The Machining Data screen appears.
7. Use the Page key or the Arrow keys to move the cursor to the item for which you want to enter a value or change the current value.
8. Press the alphanumeric keys to enter the numeric value.
9. Press the Input key .
10. The entered new value is set. The cursor moves to the next item. If you press the Arrow keys before the Input key , the entered value is set back to the original value.



[Note]

Entering the value in the field "Cut-Off End (DIA)" with no + or – sign causes an input error to occur.

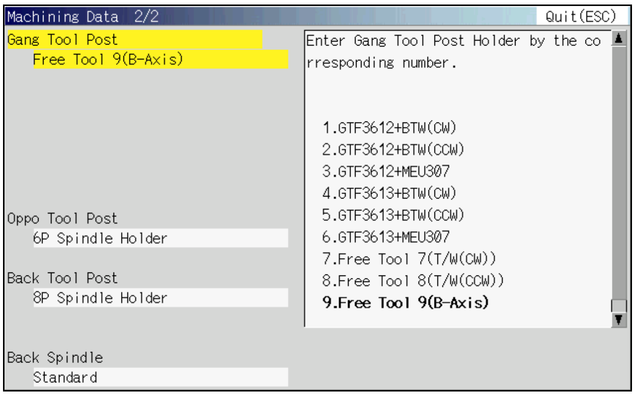
[Procedure]

Setting the holder name

1. Set the Program edit switch key to "O".
   The program is released from the protected state.
2. Press the Edit key.
   The previously selected screen appears.
3. Press the menu key [List].
   The List screen appears.
4. Call the program you want to edit. The Edit screen appears.
5. Press the menu Up/Down selection key to enable the submenu key.
6. Press the menu key [MC-Data].
   The Machining Data screen appears.
7. Use the Page key or the Arrow keys to move the cursor to the Tool Post field for which you want to change a tool post. List of available holders is displayed.

[Note]

Use the Page key to view the guidance.



8. Press the Numeric key to select the number.
9. Press the Input key.

   The selected holder name is set. The cursor moves to the next item. If you press the Arrow keys before the Input key, the entered value is set back to the original value.

[Note]

While the Program edit switch key is set to "O", you cannot edit the program or machining data.

6.9.6 Data input and output

The machining program, offset, and parameters can be input and output by connecting various input devices to the machine.

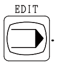
6.9.6.1 Setting the input/output device and communication parameter

The Citizen recommended input/output unit are as follows.

- CF (Compact flash) card drive (front slot)
- USB flash drive (front slot)

When using a different input/output unit, refer to "Input/output Parameters" in the NC unit instruction manual and set the RS-232C conditions.

[Procedure]

1. Press the Edit key 

.

The previously selected screen appears.

2. Press the menu key [List].

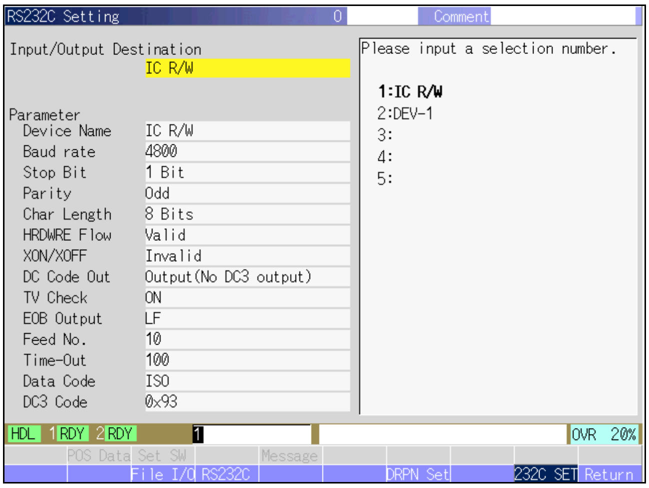
The List screen appears.

3. Press the menu key [I/O].

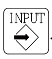
The last screen selected appears.

4. Press the menu key [232C SET].

The RS232C Setting screen appears.



5. With the cursor in "Input/Output Destination", enter the number for the unit to be selected.

6. Press the Input key 

.

The unit is selected and the information of parameter is displayed.

7. Set the communication parameters of the selected unit.

6.9.6.2 Connecting input/output unit

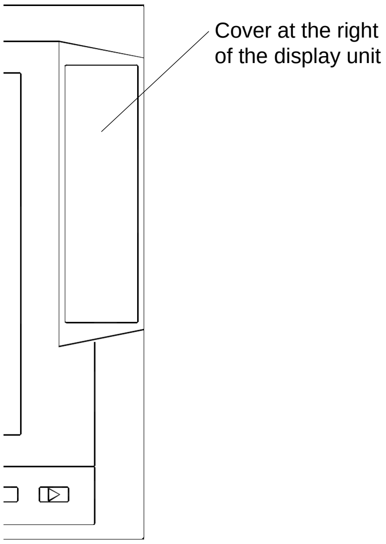
DANGER

Be sure to turn off the machine main power before installing or removing the input/output device. There is the danger of electric shock if this work is done while the power is on.

[Procedure]

Installation of CF card

1. Open the cover at the right of the display unit.
2. Insert the CF card into the CF card slot with its surface leftward. Pay attention to the orientation of the card.



Removal of CF card

1. Confirm that no data is being input or output, and pull out the CF card from the CF card slot.
2. Certainly close the cover at the right of the display unit.

The following memory cards commercially available are already checked to operate normaly.

Memory cards

- Compact flash cards

  Recommended CF cards

  CITIZEN MACHINERY MIYANO

Type	Capacity
M416U810T	512 MB
M416U811T	1 GB
M416U812T	2GB

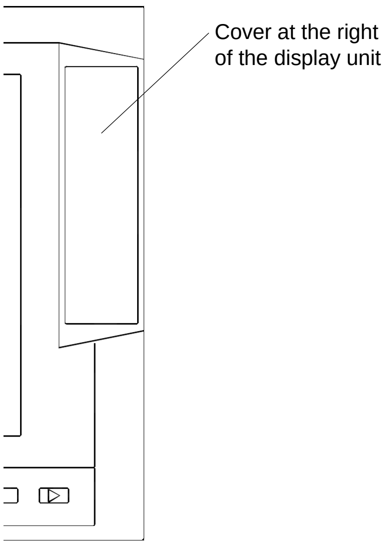
Notes on Memory Cards

- To prevent any error from occurring, turn off the power of the system before removeing or insreting a memory card.
- To prevent memory data from being erased, neither pull out a memory card and nor turn off the power during data accessing.
- We cannot secure damaged and erased data. Accordingly, back up important data to other storage as a precaution.

[Procedure]

Installation of USB flash drive

1. Open the cover at the right of the display unit.
2. Insert the USB flash drive into USB port. Pay attention to the orientation of the USB flash drive.



Removal of USB flash drive

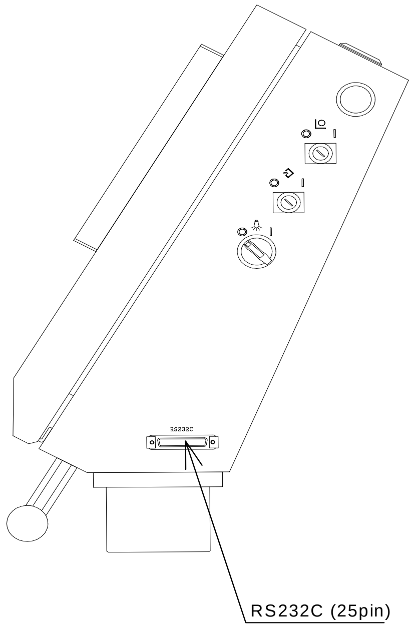
1. Confirm that no data is being input or output, and pull out the USB flash drive from the USB port.
2. Certainly close the cover at the right of the display unit.

[Procedure]

Connecting the RS232C communication device

[Note]

A 25-pin RS232C connector is provided. Connect the RS232C communication device using the cable with the 25-pin RS232C connector.



6.9.6.3 Inputting the machining program from external devices

You can load the machining program from the input/output device using the following procedure.

[Procedure]

Input from the CF card

1. Set the Program edit switch key O I to "I".
   The program is released from the protected state.
2. Press the Edit key.
   The previously selected screen appears.
3. Press the menu key [List].
   The List screen appears.
4. Press the menu key [I/O].
   The last screen selected appears.
5. Press the menu key [File I/O].
   The File Access screen appears.
6. Press the menu key [Input].
7. Use the Arrow key to move the cursor to I/O Contents field. Enter the number assigned to "Machining program", then press the Input key.
   The "Machining program" is selected, and the cursor moves to Folder in "External" field.
8. Enter the number assigned to "CF Card 1", then press the Input key.
   The "External" folder is determined, and the cursor moves to Folder in "External" field.
9. Enter the number for the target directory, then press the Input key.
   The "External" directory is selected.
10. Use the Arrow key to move the cursor to "program" in "External" field. Enter the number for the target file, then press the Input key.
    The "External" file is selected.
11. Press the Input key.
    Data input starts.

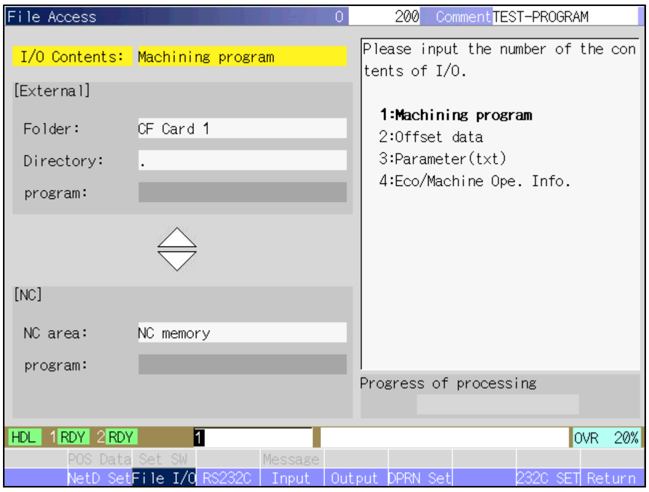
[Procedure]

Input from the USB flash drive

1. Set the Program edit switch key to "I".
   The program is released from the protected state.
2. Press the Edit key.
   The previously selected screen appears.
3. Press the menu key [List].
   The List screen appears.
4. Press the menu key [I/O].
   The last screen selected appears.
5. Press the menu key [File I/O].
   The File Access screen appears.
6. Press the menu key [Input].
7. Use the Arrow key to move the cursor to I/O Contents field. Enter the number assigned to "Machining program", then press the Input key.
   The "Machining program" is selected, and the cursor moves to Folder in "External" field.
8. Enter the number assigned to "CF Card 2", then press the Input key.
   The "External" folder is determined, and the cursor moves to Folder in "External" field.
9. Enter the number for the target directory, then press the Input key.
   The "External" directory is selected.
10. Use the Arrow key to move the cursor to "program" in "External" field. Enter the number for the target file, then press the Input key.
    The "External" program is selected.
11. Press the Input key.
    Data input starts.

[Note]

- An error message is displayed if 100 items are exceeded for any of Folder, Directory and Program.
- A folder name, directory name and program name can contain up to 62 characters. If several directories are structured in hierarchy, the limit of the number of characters of total directory names is 62 characters. Characters specified over 62 characters cannot be displayed.
- A space character or a 2-byte character is not allowed for directory name.



[Procedure]

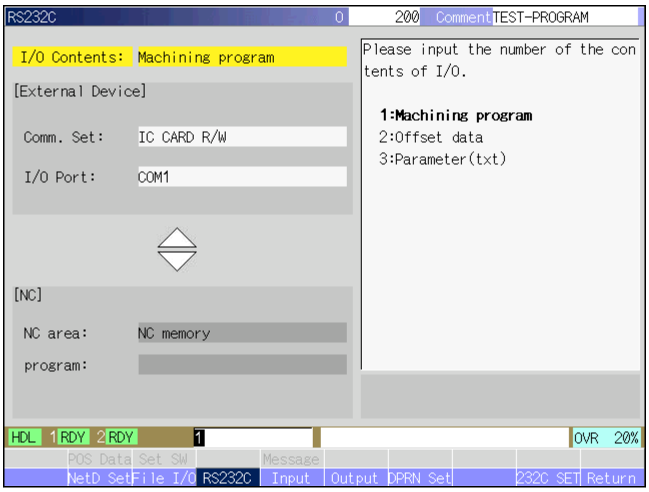
Input from the RS232C communication device

The explanation hereafter assumes the RS232C communication device has already been installed.

1. Power on the RS232C communication device to make it ready to communicate.
2. Set the Program edit switch key to "I".
   The program is released from the protected state.
3. Press the Edit key.
   The previously selected screen appears.
4. Press the menu key [List].
   The List screen appears.
5. Press the menu key [I/O].
   The last screen selected appears.
6. Press the menu key [RS232C].
   The RS232C screen appears.
7. Press the menu key [Input].
8. Use the Arrow key to move the cursor to I/O Contents field. Enter the number assigned to "Machining program", then press the Input key.
   The "Machining program" is selected, and the cursor moves to Communication Parameter in "External Device" field.
9. Enter the number for the target parameter, then press the Input key.
   The parameter for communication is determined, and the cursor moves to Input/Output Port in "External Device" field.
10. Enter the number for the I/O Port, then press the Input key.
    The input/output port is determined.
11. Press the Input key.
    The machine is ready to receive data.
12. Output data from the RS232C communication device.
13. A message "Input Completed" will be displayed when all the data is received by the machine side.

[Note]

- For details on the operation method, see the *<RS232C communication device Instruction Manual>*.
- Confirm that the communication protocol of the machine side is identical to that of the RS232C communication device.



6.9.6.4 Inputting the tool offset data from external devices

You can load the tool offset data from the input/output device using the following procedure.

[Procedure]

Input from the CF card

1. Press the Offset key OFFSET.

   The previously selected screen appears.
2. Press the menu key [I/O].

   The previously selected screen appears.
3. Press the menu key [File I/O].

   The File Access screen appears.
4. Press the menu key [Input].
5. Use the Arrow key to move the cursor to I/O Contents field. Enter the number assigned to "Offset data", then press the Input key INPUT.

   The "Offset data" is selected, and the cursor moves to Folder in "External" field.
6. Enter the number assigned to "CF Card 1", then press the Input key INPUT.

   The "External" folder is selected, and the cursor moves to Directory in "External" field.
7. Enter the number for the target directory, then press the Input key INPUT.

   The "External" directory is selected.
8. Use the Arrow key to move the cursor to "file" in "External" field. Enter the number for the target file, then press the Input key INPUT.

   The "External" file is selected.
9. Press the Input key INPUT.

   Data input starts.

[Note]

- An error message is displayed if 100 items are exceeded for any of Folder, Directory and Program.
- A folder name, directory name and program name can contain up to 62 characters. If several directories are structured in hierarchy, the limit of the number of characters of total directory names is 62 characters. Characters specified over 62 characters cannot be displayed.
- A space character or a 2-byte character is not allowed for directory name.

[Procedure]

Input from the USB flash drive

1. Press the Offset key OFFSET.

   The previously selected screen appears.

2. Press the menu key [I/O].

   The previously selected screen appears.

3. Press the menu key [File I/O].

   The File Access screen appears.

4. Press the menu key [Input].

5. Use the Arrow key to move the cursor to I/O Contents field. Enter the number assigned to "Offset data", then press the Input key INPUT.

   The "Offset data" is selected, and the cursor moves to Folder in "External" field.

6. Enter the number assigned to "CF Card 2", then press the Input key INPUT.

   The "External" folder is selected, and the cursor moves to Directory in "External" field.

7. Enter the number for the target directory, then press the Input key INPUT.

   The "External" directory is selected.

8. Use the Arrow key to move the cursor to "file" in "External" field. Enter the number for the target file, then press the Input key INPUT.

   The "External" file is selected.

9. Press the Input key INPUT.

   Data input starts.

[Note]

- An error message is displayed if 100 items are exceeded for any of Folder, Directory and Program.
- A folder name, directory name and program name can contain up to 62 characters. If several directories are structured in hierarchy, the limit of the number of characters of total directory names is 62 characters. Characters specified over 62 characters cannot be displayed.
- A space character or a 2-byte character is not allowed for directory name.

[Procedure]

Input from the RS232C communication device

1. Power on the RS232C communication device to make it ready to communicate.
2. Press the Offset key OFFSET.

   The previously selected screen appears.
3. Press the menu key [I/O].

   The previously selected screen appears.
4. Press the menu key [RS232C].

   The RS232C screen appears.
5. Press the menu key [Input].
6. Use the Arrow key to move the cursor to I/O Contents field. Enter the number assigned to "Offset data", then press the Input key INPUT.

   The "Offset data" is selected, and the cursor moves to Communication Parameter in "External Device" field.
7. Enter the number for the target parameter, then press the Input key INPUT.

   The parameter for communication is selected, and the cursor moves to Input/Output Port in "External Device" field.
8. Enter the number for the "I/O Port", then press the Input key INPUT.

   The input/output port is selected.
9. Press the Input key INPUT.

   The machine is ready to receive data.
10. Output data from the RS232C communication device.
11. A message "Input Completed" will be displayed when all the data is received by the machine side.

[Note]

- For details on the operation method, see the *<RS232C communication device Instruction Manual>*.
- Confirm that the communication protocol of the machine side is identical to that of the RS232C communication device.

6.9.6.5 Inputting the parameter (including machine-specific data)

You can load the tool parameter (including machine-specific data) from the input/output device using the following procedure.

[Procedure]

Input from the CF card

1. Press the Parameter key PRM.

   The previously selected screen appears.

2. Press the menu key [Menu SEL] until the menu key [I/O] is displayed.

   Press the menu key [I/O].

   The previously selected screen appears.

3. Press the menu key [File I/O].

   The File Access screen appears.

4. Press the menu key [Input].

5. Use the Arrow key to move the cursor to I/O Contents field. Enter the number assigned to "Parameter (txt)", then press the Input key INPUT.

   The "Parameter (txt)" is selected, and the cursor moves to Folder in "External" field.

6. Enter the number assigned to "CF Card 1", then press the Input key INPUT.

   The "External" folder is selected, and the cursor moves to Directory in "External" field.

7. Enter the number for the target directory, then press the Input key INPUT.

   The "External" directory is selected.

8. Use the Arrow key to move the cursor to "program" in "External" field. Enter the number for the target file, then press the Input key INPUT.

   The "External" file is selected.

9. Press the Input key INPUT.

   Data input starts.

[Note]

- An error message is displayed if 100 items are exceeded for any of Folder, Directory and Program.
- A folder name, directory name and program name can contain up to 62 characters. If several directories are structured in hierarchy, the limit of the number of characters of total directory names is 62 characters. Characters specified over 62 characters cannot be displayed.
- A space character or a 2-byte character is not allowed for directory name.

[Procedure]

Input from the USB flash drive

1. Press the Parameter key .

   The previously selected screen appears.
2. Press the menu key [Menu SEL] until the menu key [I/O] is displayed.

   Press the menu key [I/O].

   The previously selected screen appears.
3. Press the menu key [File I/O].

   The File Access screen appears.
4. Press the menu key [Input].
5. Use the Arrow key to move the cursor to I/O Contents field. Enter the number assigned to "Parameter (txt)", then press the Input key .

   The "Parameter (txt)" is selected, and the cursor moves to Folder in "External" field.
6. Enter the number assigned to "CF Card 2", then press the Input key .

   The "External" folder is selected, and the cursor moves to Directory in "External" field.
7. Enter the number for the target directory, then press the Input key .

   The "External" directory is selected.
8. Use the Arrow key to move the cursor to "file" in "External" field. Enter the number for the target file, then press the Input key .

   The "External" file is selected.
9. Press the Input key .

   Data input starts.

[Note]

- An error message is displayed if 100 items are exceeded for any of Folder, Directory and Program.
- A folder name, directory name and program name can contain up to 62 characters. If several directories are structured in hierarchy, the limit of the number of characters of total directory names is 62 characters. Characters specified over 62 characters cannot be displayed.
- A space character or a 2-byte character is not allowed for directory name.

[Procedure]

Input from the RS232C communication device

1. Power on the RS232C communication device to make it ready to communicate.
2. Press the Parameter key PRM..

   The previously selected screen appears.
3. Press the menu key [Menu SEL] until the menu key [I/O] is displayed.
   Press the menu key [I/O].

   The previously selected screen appears.
4. Press the menu key [RS232C].

   The RS232C screen appears.
5. Press the menu key [Input].
6. Use the Arrow key to move the cursor to I/O Contents field. Enter the number assigned to "Parameter(txt)", then press the Input key INPUT.

   The "Parameter(txt)" is selected, and the cursor moves to Communication Parameter in "External Device" field.
7. Enter the number for the target parameter, then press the Input key INPUT.

   The parameter for communication is selected, and the cursor moves to Input/Output Port in "External Device" field.
8. Enter the number for the I/O Port, then press the Input key INPUT.

   The input/output port is selected.
9. Press the Input key INPUT.
   The machine is ready to receive data.
10. Output data from the RS232C communication device.
11. A message "Input Completed" will be displayed when all the data is received by the machine side.

[Note]

- For details on the operation method, see the *<RS232C communication device Instruction Manual>*.
- Confirm that the communication protocol of the machine side is identical to that of the RS232C communication device.

6.9.6.6 Outputting the machining program to external devices

You can send the machining program to the input/output device using the following procedure.

[Procedure]

Output to the CF card

1. Set the Program edit switch key to "I".
   The program is released from the protected state.
2. Press the Edit key.
   The previously selected screen appears.
3. Press the menu key [List].
   The List screen appears.
4. Press the menu key [I/O].
   The previously selected screen appears.
5. Press the menu key [File I/O].
   The File Access screen appears.
6. Press the menu key [Output].
7. Use the Arrow key to move the cursor to I/O Contents field. Enter the number assigned to "Machining program", then press the Input key.
   The "Machining program" is selected, and the cursor moves to Folder in "External" field.
8. Enter the number assigned to "CF Card 1", then press the Input key.
   The "External" folder is selected, and the cursor moves to Directory in "External" field.
9. Enter the number for the target directory, then press the Input key.
   The "External" directory is selected.
10. Use the Arrow key to move the cursor to I/O Contents field. Enter the number assigned to "NC", then press the Input key.
    The "NC" program is selected.
11. Press the Input key.
    Data output starts.

[Note]

- An error message is displayed if 100 items are exceeded for any of Folder, Directory and Program.
- A folder name, directory name and program name can contain up to 62 characters. If several directories are structured in hierarchy, the limit of the number of characters of total directory names is 62 characters. Characters specified over 62 characters cannot be displayed.
- A space character or a 2-byte character is not allowed for directory name.

[Procedure]

Output to the USB flash drive

1. Set the Program edit switch key to "I".
   The program is released from the protected state.
2. Press the Edit key.
   The previously selected screen appears.
3. Press the menu key [List].
   The List screen appears.
4. Press the menu key [I/O].
   The previously selected screen appears.
5. Press the menu key [File I/O].
   The File Access screen appears.
6. Press the menu key [Output].
7. Use the Arrow key to move the cursor to I/O Contents field. Enter the number assigned to "Machining program", then press the Input key.
   The "Machining program" is selected, and the cursor moves to Folder in "External" field.
8. Enter the number assigned to "CF Card 2", then press the Input key.
   The "External" folder is selected, and the cursor moves to Directory in "External" field.
9. Enter the number for the target directory, then press the Input key.
   The "External" directory is selected.
10. Use the Arrow key to move the cursor to I/O Contents field. Enter the number assigned to "NC", then press the Input key.
    The "NC" program is selected.
11. Press the Input key.
    Data output starts.

[Note]

- An error message is displayed if 100 items are exceeded for any of Folder, Directory and Program.
- A folder name, directory name and program name can contain up to 62 characters. If several directories are structured in hierarchy, the limit of the number of characters of total directory names is 62 characters. Characters specified over 62 characters cannot be displayed.
- A space character or a 2-byte character is not allowed for directory name.

[Procedure]

Output to the RS232C communication device

1. Power on the RS232C communication device to make it ready to communicate.
2. Set the Program edit switch key to "I".
   The program is released from the protected state.
3. Press the Edit key .
   The previously selected screen appears.
4. Press the menu key [List].
   The List screen appears.
5. Press the menu key [I/O].
   The previously selected screen appears.
6. Press the menu key [RS232C].
   The RS232C screen appears.
7. Press the menu key [Output].
8. Use the Arrow key to move the cursor to I/O Contents field. Enter the number assigned to "Machining program", then press the Input key .
   The "Machining program" is selected, and the cursor moves to Communication Parameter in "External Device" field.
9. Enter the number for the target parameter, then press the Input key .
   The parameter for communication is selected, and the cursor moves to Input/Output Port in "External Device" field.
10. Enter the number for the I/O Port, then press the Input key .
    The input/output port is selected.
11. Use the Arrow key to move the cursor to I/O Contents field. Enter the number assigned to "NC", then press the Input key .
    The "NC" is selected.
12. Press the Input key .
    The machine is ready to send data.
13. Data input from the RS232C communication device starts.
14. A message "Output Completed" will be displayed when all the data is received by the machine side.

[Note]

- For details on the operation method, see the *<RS232C communication device Instruction Manual>*.
- Confirm that the communication protocol of the machine side is identical to that of the RS232C communication device.

6.9.6.7 Outputting the tool offset data to external devices

You can output the tool offset data to the input/output device using the following procedure.

[Procedure]

Output to the CF card

1. Press the Offset key .

   The previously selected screen appears.
2. Press the menu key [I/O].

   The previously selected screen appears.
3. Press the menu key [File I/O].

   The File Access screen appears.
4. Press the menu key [Output].
5. Use the Arrow key to move the cursor to I/O Contents field. Enter the number assigned to "Offset data", then press the Input key .

   The "Offset data" is selected, and the cursor moves to Folder in "External" field.
6. Enter the number assigned to "CF Card 1", then press the Input key .

   The "External" folder is selected, and the cursor moves to Directory in "External" field.
7. Enter the number for the target directory, then press the Input key .

   The "External" directory is selected.
8. Use the Arrow key to move the cursor to I/O Contents field. Enter the number assigned to "NC" file, then press the Input key .

   The "NC" file is selected.
9. Press the Input key .

   Data output starts.

[Note]

- An error message is displayed if 100 items are exceeded for any of Folder, Directory and Program.
- A folder name, directory name and program name can contain up to 62 characters. If several directories are structured in hierarchy, the limit of the number of characters of total directory names is 62 characters. Characters specified over 62 characters cannot be displayed.
- A space character or a 2-byte character is not allowed for directory name.

[Procedure]

Output to the USB flash drive

1. Press the Offset key .

   The previously selected screen appears.
2. Press the menu key [I/O].

   The previously selected screen appears.
3. Press the menu key [File I/O].

   The File Access screen appears.
4. Press the menu key [Output].
5. Use the Arrow key to move the cursor to I/O Contents field. Enter the number assigned to "Offset data", then press the Input key .

   The "Offset data" is selected, and the cursor moves to Folder in "External" field.
6. Enter the number assigned to "CF Card 2", then press the Input key .

   The "External" folder is selected, and the cursor moves to Directory in "External" field.
7. Enter the number for the target directory, then press the Input key .

   The "External" directory is selected.
8. Use the Arrow key to move the cursor to I/O Contents field. Enter the number assigned to "NC" file, then press the Input key .

   The "NC" file is selected.
9. Press the Input key .

   Data output starts.

[Note]

- An error message is displayed if 100 items are exceeded for any of Folder, Directory and Program.
- A folder name, directory name and program name can contain up to 62 characters. If several directories are structured in hierarchy, the limit of the number of characters of total directory names is 62 characters. Characters specified over 62 characters cannot be displayed.
- A space character or a 2-byte character is not allowed for directory name.

[Procedure]

Output to the RS232C communication device

1. Power on the RS232C communication device to make it ready to communicate.
2. Press the Offset key .

   The previously selected screen appears.
3. Press the menu key [I/O].

   The previously selected screen appears.
4. Press the menu key [RS232C].

   The RS232C screen appears.
5. Press the menu key [Output].
6. Use the Arrow key to move the cursor to I/O Contents field. Enter the number assigned to "Offset data", then press the Input key .

   The "Offset data" is selected, and the cursor moves to Communication Parameter in "External Device" field.
7. Enter the number for the target parameter, then press the Input key .

   The parameter for communication is selected, and the cursor moves to Input/Output Port in "External Device" field.
8. Enter the number for the I/O Port, then press the Input key .

   The input/output port is selected.
9. Use the Arrow key to move the cursor to I/O Contents field. Enter the number assigned to "NC file", then press the Input key .

   The "NC file" is selected.
10. Press the Input key .

    The machine is ready to send data.
11. Data input from the RS232C communication device starts.
12. A message "Output Completed" will be displayed when all the data is received by the machine side.

[Note]

- For details on the operation method, see the *<RS232C communication device Instruction Manual>*.
- Confirm that the communication protocol of the machine side is identical to that of the RS232C communication device.

6.9.6.8 Outputting the parameter (including machine-specific data)

You can send the tool parameter (including machine-specific data) to the input/output device using the following procedure.

[Procedure]

Output to the CF card

1. Press the Parameter key .

   The previously selected screen appears.

2. Press the menu key [Menu SEL] until the menu key [I/O] is displayed.
   Press the menu key [I/O].

   The previously selected screen appears.

3. Press the menu key [File I/O].

   The File Access screen appears.

4. Press the menu key [Output].

5. Use the Arrow key to move the cursor to I/O Contents field. Enter the number assigned to "Parameter(txt)", then press the Input key .

   The "Parameter(txt)" is selected, and the cursor moves to Folder in "External" field.

6. Enter the number assigned to "CF Card 1", then press the Input key .

   The "External" folder is selected, and the cursor moves to Directory in "External" field.

7. Enter the number for the target directory, then press the Input key .

   The "External" directory is selected.

8. Use the Arrow key to move the cursor to I/O Contents field. Enter the number assigned to "NC file", then press the Input key .

   The "NC file" is selected.

9. Press the Input key .

   Data output starts.

[Note]

- An error message is displayed if 100 items are exceeded for any of Folder, Directory and Program.
- A folder name, directory name and program name can contain up to 62 characters. If several directories are structured in hierarchy, the limit of the number of characters of total directory names is 62 characters. Characters specified over 62 characters cannot be displayed.
- A space character or a 2-byte character is not allowed for directory name.

[Procedure]

Output to the USB flash drive

1. Press the Parameter key PRM.

   The previously selected screen appears.

2. Press the menu key [Menu SEL] until the menu key [I/O] is displayed.
   Press the menu key [I/O].

   The previously selected screen appears.

3. Press the menu key [File I/O].

   The File Access screen appears.

4. Press the menu key [Output].

5. Use the Arrow key to move the cursor to I/O Contents field. Enter the number assigned to "Parameter(txt)", then press the Input key INPUT.

   The "Parameter(txt)" is selected, and the cursor moves to Folder in "External" field.

6. Enter the number assigned to "CF Card 2", then press the Input key INPUT.

   The "External" folder is selected, and the cursor moves to Directory in "External" field.

7. Enter the number for the target directory, then press the Input key INPUT.

   The "External" directory is selected.

8. Use the Arrow key to move the cursor to I/O Contents field. Enter the number assigned to "NC file", then press the Input key INPUT.

   The "NC file" is selected.

9. Press the Input key INPUT.

   Data output starts.

[Note]

- An error message is displayed if 100 items are exceeded for any of Folder, Directory and Program.
- A folder name, directory name and program name can contain up to 62 characters. If several directories are structured in hierarchy, the limit of the number of characters of total directory names is 62 characters. Characters specified over 62 characters cannot be displayed.
- A space character or a 2-byte character is not allowed for directory name.

[Procedure]

Output to the RS232C communication device

1. Power on the RS232C communication device to make it ready to communicate.
2. Press the Parameter key .

   The previously selected screen appears.
3. Press the menu key [Menu SEL] until the menu key [I/O] is displayed.
   Press the menu key [I/O].

   The previously selected screen appears.
4. Press the menu key [RS232C].

   The RS232C screen appears.
5. Press the menu key [Output].
6. Use the Arrow key to move the cursor to I/O Contents field. Enter the number assigned to "Parameter(txt)", then press the Input key .

   The "Parameter(txt)" is selected, and the cursor moves to Communication Parameter in "External Device" field.
7. Enter the number for the target parameter, then press the Input key .

   The parameter for communication is selected, and the cursor moves to Input/Output Port in "External Device" field.
8. Enter the number for the I/O Port, then press the Input key .

   The input/output port is selected.
9. Use the Arrow key to move the cursor to I/O Contents field. Enter the number assigned to "NC file", then press the Input key .

   The "NC file" is selected.
10. Press the Input key .
    The machine is ready to send data.
11. Data input from the RS232C communication device starts.
12. A message "Output Completed" will be displayed when all the data is received by the machine side.

[Note]

- For details on the operation method, see the *<RS232C communication device Instruction Manual>*.
- Confirm that the communication protocol of the machine side is identical to that of the RS232C communication device.

6.9.6.9 Output of Eco / Machine Operation Information

Use the following procedures to output the Eco / Machine Operation Information data file from the machine to external I/O device.

[Note]

- Eco / Machine Operation Information cannot be output to the machine.
- Eco / Machine Operation Information data file cannot output through the RS232C interface.

[Procedure]

Output to CF card

1. Press the Edit key EDIT. The previously selected screen appears.
2. Press the menu key [List]. The List screen appears.
3. Press the menu key [I/O]. The previously selected screen appears.
4. Press the menu key [File I/O]. The File Access screen appears.
5. Press the menu key [Output].
6. Use the Arrow key to move the cursor to I/O Contents field. Enter the number assigned to "Eco / Machine Operation Information", then press the Input key INPUT.
   The "Eco / Machine Operation Information" is selected, and the cursor moves to Folder in "External" field.
7. Enter the number assigned to "CF Card 1", then press the Input key INPUT.
   The "External" folder is selected, and the cursor moves to Directory in "External" field.
8. Enter the number for the target directory, then press the Input key INPUT.
   The "External" directory is selected.
9. Use the Arrow key to move the cursor to "NC file" field. Enter the number assigned to "NC", then press the Input key INPUT. The "NC" program is selected.
10. Press the Input key INPUT. Data output starts.

[Procedure]

Output to USB flash drive

1. Press the Edit key . The previously selected screen appears.
2. Press the menu key [List]. The List screen appears.
3. Press the menu key [I/O]. The previously selected screen appears.
4. Press the menu key [File I/O]. The File Access screen appears.
5. Press the menu key [Output].
6. Use the Arrow key to move the cursor to I/O Contents field. Enter the number assigned to "Eco / Machine Operation Information", then press the Input key .
   The "Eco / Machine Operation Information" is selected, and the cursor moves to Folder in "External" field.
7. Enter the number assigned to "CF Card 2", then press the Input key .
   The "External" folder is selected, and the cursor moves to Directory in "External" field.
8. Enter the number for the target directory, then press the Input key .
   The "External" directory is selected.
9. Use the Arrow key to move the cursor to "NC file" field. Enter the number assigned to "NC", then press the Input key . The "NC" program is selected.
10. Press the Input key . Data output starts.

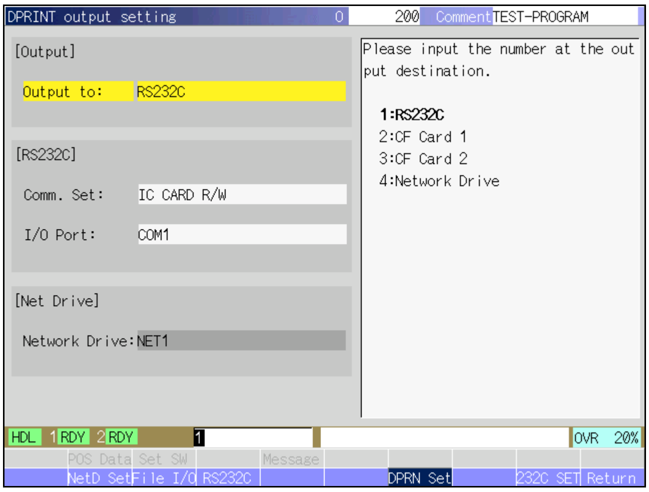
[Note]

- When outputting the Power Consumption data, three files (YYCns.csv, MMCns.csv, and DDCns.csv) are output at the same time. Each file cannot be output individually.
- When outputting the Electric Power Regeneration data, three files (YYRgn.csv, MMRgn.csv, and DDRgn.csv) are output at the same time. Each file cannot be output individually.
- When outputting the Reduction Power data, three files (YYEco.csv, MMEco.csv, and DDEco.csv) are output at the same time. Each file cannot be output individually.
- When outputting the Machine Operation Information data, four files (YYEchOpe.csv, MMEchOpe.csv, DDEchOpe.csv, and DDTimOpe.csv) are output at the same time. Each file cannot be output individually.
- When outputting the Cutting Time data, three files (YYCut.csv, MMCut.csv, and DDCut.csv) are output at the same time. Each file cannot be output individually.
- See *<8.18 csv File Format of Eco and Machine Operation Information>* for the data format of each file.
- The last output date is displayed on each graph screen. See *<6.19 Eco>* or *<6.20 Machine Operation Information>* for details of graph indication.
- A folder name and directory name can contain up to 62 characters. If several directories are structured in hierarchy, the limit of the number of characters of total directory names is 62 characters. Characters specified over 62 characters cannot be displayed.
- A space character or a 2-byte character is not allowed for directory name.

6.9.6.10 DPRINT output setting

Use this menu to provide setting for external device when transferring data which is output at the program execution using the DPRINT function.

When the output destination is "CF Card 1" or "Network Drive", the data is transferred with the file name "PRNT.PRN".



[Procedure]

Output to the RS232C communication device

1. Press the Edit key EDIT.

   The previously selected screen appears.
2. Press the menu key [List].
   The List screen appears.
3. Press the menu key [I/O].
   The previously selected screen appears.
4. Press the menu key [DRPN Set].
   The DPRINT output setting screen appears.
   The cursor appears on the output destination.
5. Enter the number assigned to "RS232C", and press the Input key INPUT.The "RS232C" is selected, and the cursor moves to Communication Parameter in "RS232C" field.
6. Enter the number assigned to the communication parameter to be used for output, and press the Input key INPUT.The parameter for communication is determined, and the cursor moves to Input/Output Port in "RS232C" field.
7. Enter the destination port number, and press the Input key INPUT.

[Procedure]

Output to the CF card

1. Press the Edit key . The previously selected screen appears.
2. Press the menu key [List]. The List screen appears.
3. Press the menu key [I/O]. The previously selected screen appears.
4. Press the menu key [DRPN Set].
   The DPRINT output setting screen appears.
   The cursor appears on the output destination.
5. Enter the number assigned to "CF Card 1", and press the Input key .

[Procedure]

Output to the USB flash drive

1. Press the Edit key . The previously selected screen appears.
2. Press the menu key [List]. The List screen appears.
3. Press the menu key [I/O]. The previously selected screen appears.
4. Press the menu key [DRPN Set].
   The DPRINT output setting screen appears.
   The cursor appears on the output destination.
5. Enter the number assigned to "CF Card 2", and press the Input key .

[Procedure]

Output to the Network Drive (Option)

1. Press the Edit key .
   The previously selected screen appears.
2. Press the menu key [List].
   The List screen appears.
3. Press the menu key [I/O].
   The previously selected screen appears.
4. Press the menu key [DRPN Set].
   The DPRINT output setting screen appears.
   The cursor appears to the output destination.
5. Enter the number assigned to "Network Drive", and press the Input key .
   The "Network Drive" is selected, and the cursor moves to Network drive in "Network Drive" field.
6. Enter the network drive number to send data, and press the Input key .

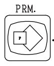
[Note]

- A custom macro option is required to use the DPRINT function.
- For details on the operation method, see the *<RS232C communication device Instruction Manual>*.
- Confirm that the communication protocol of the machine side is identical to that of the RS232C communication device.
- If you specify the "CF Card 1" as output destination, insert the CF card into the card slot before executing the DPRINT function. Do not remove the CF card while the DPRINT function is running.
- If you specify the "CF Card 2" as output destination, insert the USB flash drive into the USB port before executing the DPRINT function. Do not remove the USB flash drive while the DPRINT function is running.
- The maximum file size that can be output to CF card or USB flash drive is 1.4 MB. If the file size exceeds 1.4 MB, an alarm message "MEMORY CAPACITY OVER" is displayed.
- The maximum file size that can be output to network drive is 500 KB. If the file size exceeds 500 KB, an alarm message "MEMORY CAPACITY OVER" is displayed.

6.10 Machine Variable Screen

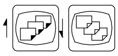
This screen displays variables that are mainly used with the machine manufacturer's macros.

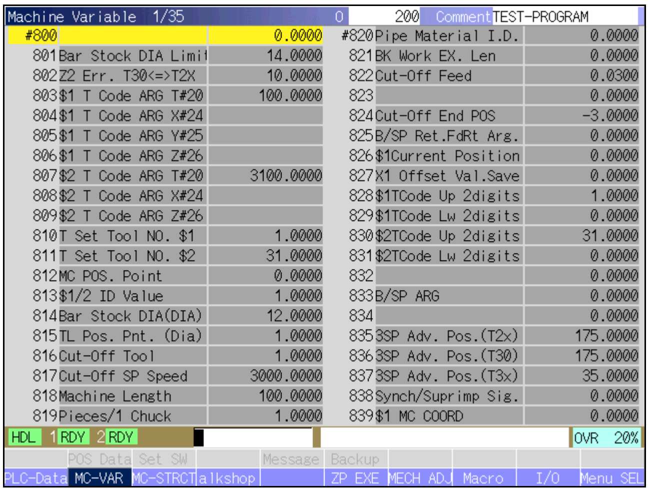
[Procedure]

1. Press the Parameter key .

The previously selected screen appears.

2. Press the menu key [MC-VAR].
The Machine Variable screen appears.

3. Press the Page keys  until the desired number is displayed.



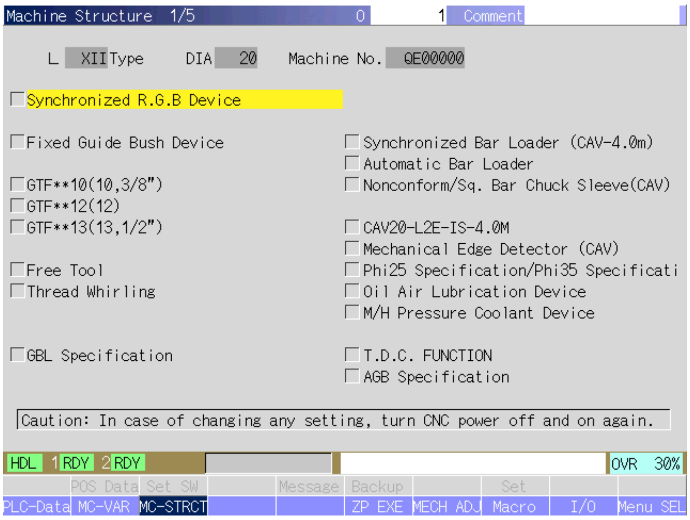
6.11 Machine Structure

Use this screen to provide settings for the specification and optional functions of the machine.

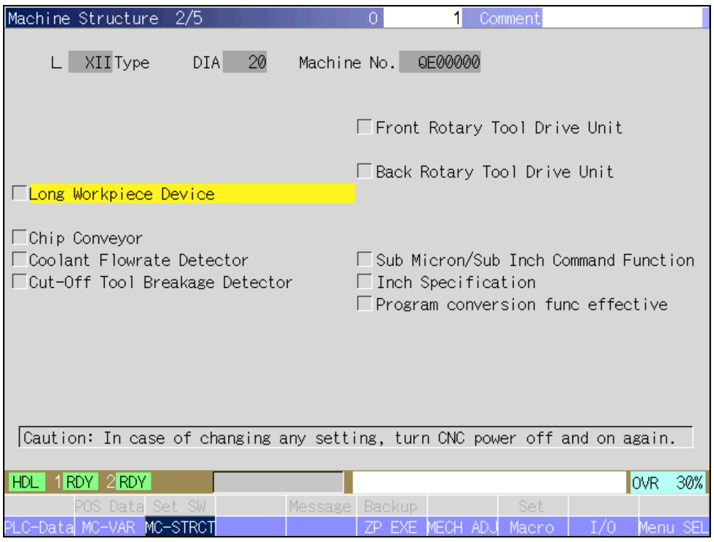
The Machine Structure screen consists of five pages.

First and second pages: These pages are used to check mounted optional devices. When you have mounted a device listed on these pages, check the device. If you have removed it, uncheck it. Pressing the menu key [Set] allows you to check devices on the page.

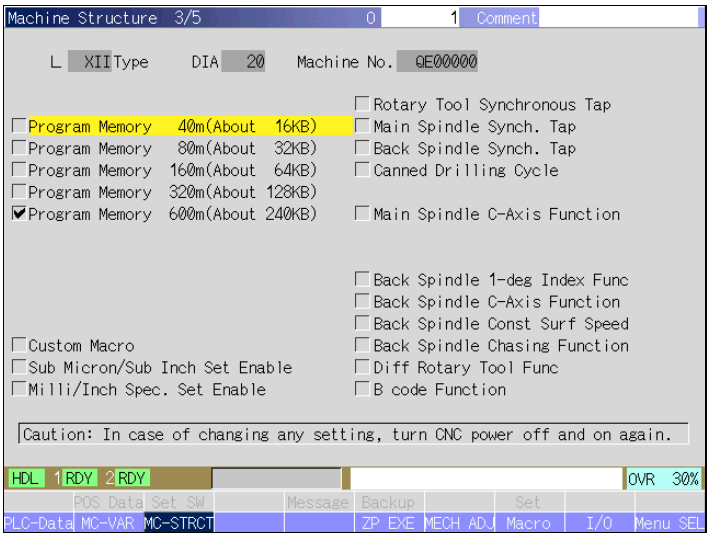
Third, fourth, and fifth pages: These pages are used to view installed software options. You can confirm the software options installed on the machine; you cannot check items on the page.



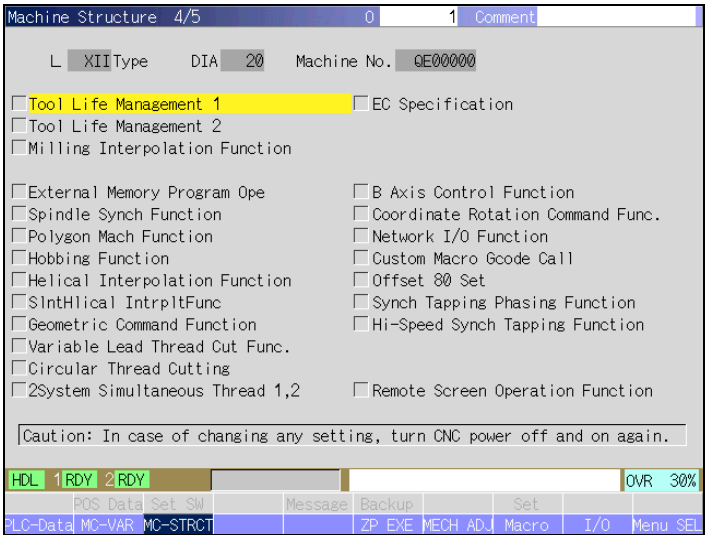
Machine Structure screen 1



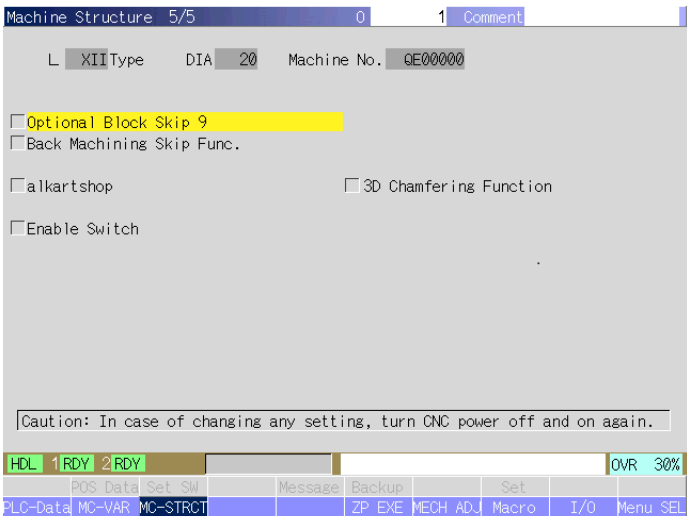
Machine Structure screen 2



Machine Structure screen 3



Machine Structure screen 4



Machine Structure screen 5

Display items

Item	Remarks
Synchronized R.G.B Device	Be sure to select only one among "Synchronized R.G.B Device", "Fixed Guide Bush Device", and "GBL Specification".
Fixed Guide Bush Device	
GTF**10 (□10, □3/8")	Only one of the two items can be selected.
GTF**12(□12)	
GTF**13(□13, □1/2")	
Free Tool	
Thread Whirling	
"Start point shift amount" in machining data	Displayed if stopper specifications are available.
GBL Specification	Be sure to select only one among "Synchronized R.G.B Device", "Fixed Guide Bush Device", and "GBL Specification".
Sttoper Specification	Displayed when "GBL Specification" is selected.
Synchronous Bar Loader (CAV-4.0M)	Only one of the five items can be selected. If none is selected, "Automatic Bar Loader" is selected.
Automatic Bar Loader	
Chuck Sleeve for Non-Conformed Materials (CAV)	Select to use chuck sleeve for non-conform/square bar
CAV20LE-IS-4.0M	The synchronous bar loader (CAV-4.0M) is also selected.
Mechanical edge detector (CAV)	
φ25 Specification/φ35 Specification	
Oil Air Lubrication Unit	
Medium-pressure / High-pressure Coolant device	
Enable Thermal Displacement Correction Function	
AGB Specification	Cannot be switched when a GBL specification is enabled Only one of synchronous R.G.B. system, fixed G.B. system or AGB specification can be selected.
Long Workpiece Device	
Chip Conveyor	
Coolant Flowrate Detector	
Cut-Off Tool Breakage Detector	
Front Rotary Tool on Opposite Tool Post	
Back Rotary Tool Drive Unit	
Sub Micron/Sub Inch Command Function	Displayed if sub micron/sub inch command is enabled.
Inch Specification	Displayed if mm/inch specification setting enabled is enabled.
Program Conversion Function Enabled	
ProgWork Area 40m (App. 16 KB)	Only one of the five items can be selected.
ProgWork Area 80m (App. 32 KB)	
ProgWork Area 160m (App. 64 KB)	
ProgWork Area 320m (App. 128 KB)	
ProgWork Area 600m (App. 240 KB)	
MCC Package	
Custom Macro	
Sub Micron/Sub Inch Command Setting Enabled	
MM/Inch Specification Setting Enabled	
Rotary Tool Synchronous Tap	
Main Spindle Synch. Tap	
Back Spindle Synch. Tap	
Canned Drilling Cycle	

Item	Remarks
Main Spindle C-Axis Function	
Back Spindle 1-deg Index Func	
Back Spindle C-Axis Func	
Back Spindle Constant Surface Speed	
Back Spindle Chasing Function	
Diff Rotary Tool Func	
B code Function	
Tool Life Management 1	
Tool Life Management 2	
Milling Interpolation Function	
Door Lock	
External Memory Program Ope	
Spindle Synchronization Function	
Polygon Machining Function	
Hobbing Function	
Helical Interpolation Function	
Slant Helical Interpolation Function	
Geometric Command Function	
Variable Lead Thread Cut Func.	
Circular Thread Cutting	
2 System Simultaneous Thread 1,2	
EC Specification	
Coordinate Rotation Command Func.	
Network I/O Function	
Custom Macro G code Call	
Offset 80 Set	
Synchronous Tapping Phase Adjustment Function	
High Speed Synchronous Tapping Function	
Remote Screen Operation Function	
Nine Optional Block Skip 9	
Back Machining Programs Skip Function	
alkart shop	
Enable switch	

[Procedure]

1. Press the Parameter key PRM..
   The previously selected screen appears.
2. Press the menu key [Menu SEL] until the menu key [MC-STRCT] is displayed.
3. Press the menu key [MC-STRCT].
   The Machine Structure screen appears.
4. Use the arrow keys ⬆⬇, Tab keys TAB⬅ TAB➡, and Page keys to move the cursor to the item you want to change setting.
5. Press the Menu Up/Down Selection key ▶ and press the menu key [Set].
6. Press the Input key INPUT to determine the setting you have made.

[Note]

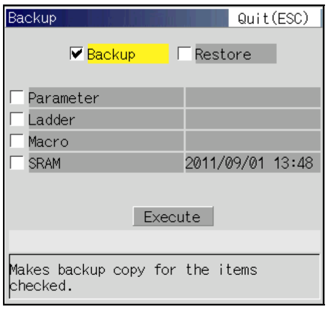
When you have changed settings, turn the power off once, then turn it on back for the changes you made to take effect.

CAUTION

If you operate the machine with the settings (checks) on the machine structure screen not matching the machine structure of actually mounted devices, an alarm may be generated or the machine may be damaged. Make correct settings (checks) on the screen.

6.12 Backup Screen

Use this screen to backup various types of software and parameters.



[Procedure]

To backup data

1. Press the Parameter key .
   The previously selected screen appears.
2. Press the Menu Up/Down selection key to switch the menu, and press the menu key [Backup].
   The Backup screen appears.
3. Use the arrow keys to move the cursor to "Backup".
4. Press the Input key to put a checkmark on "Backup".
5. Use the arrow keys to move the cursor to the item you want to backup the data.
6. Press the Input key to put a checkmark on the item you want to backup the data.
   If several items are to be backed up, repeat steps 5 and 6.
7. Use the arrow keys to move the cursor to "Execute".
8. Press the Input key to execute backup operation.

[Procedure]

To restore the data

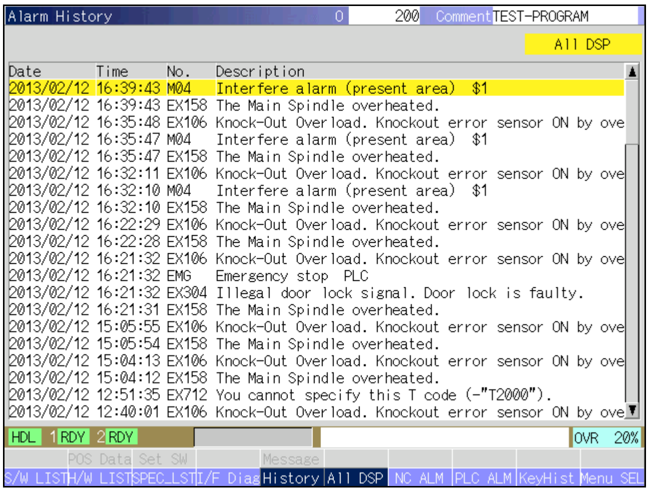
1. Press the Parameter key PRM.. The previously selected screen appears.
2. Press the Menu Up/Down selection key ▶ to switch the menu, and press the menu key [Backup]. The Backup screen appears.
3. Use the arrow keys ⬆ ⬇ to move the cursor to "Restore".
4. Press the Input key INPUT to put a checkmark on "Restore".
5. Use the arrow keys ⬆ ⬇ to move the cursor to the item you want to restore the data.
6. Press the Input key INPUT to put a checkmark on the item you want to restore the data. If several items are to be restored, repeat steps 5 and 6.
7. Use the arrow keys ⬆ ⬇ to move the cursor to "Execute".
8. Press the Input key INPUT to execute restore operation.

[Note]

Restoration is unavailable for the item which has not been backed up.

6.13 Alarm History

The alarms generated on the machine are stored and listed on the Alarm History screen.



[Procedure]

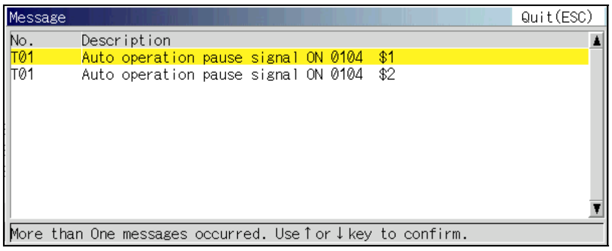
1. Press the Diagnosis key (MAINT.).
   The previously selected screen appears.
2. Press the menu key [Menu SEL] until the menu key [History] is displayed.
3. Press the menu key [History].
   The Alarm History screen appears.
4. Use the arrow keys ⬆ ⬇ or Page keys to browse the alarm history.

[Note]

- Pressing the menu key [NC ALM] displays the NC alarm only.
- Pressing the menu key [PLC ALM] displays the PLC alarm only.
- Pressing the menu key [KeyHist] displays the Key History screen.
- The information of up to 100 alarms can be stored and displayed on the Alarm History screen. When the number of alarms has reached 100, they are deleted automatically in the order in which they were recorded.

6.14 Message Screen

Use this screen to display messages such as alarm messages.



[Procedure]

1. Press the Menu Up/Down selection key ▶ to switch the menu, and press the menu key [Message].
2. Use the arrow keys ⬆ ⬇ and Page keys to browse messages.

6.15 Tool Life Management (Option)

The tool life management feature accumulates the tool usage time and count commanded by programs and monitors the usage status of the tools. Up to 99 tools (tool 1 - tool 99) can be subject to the tool life management.

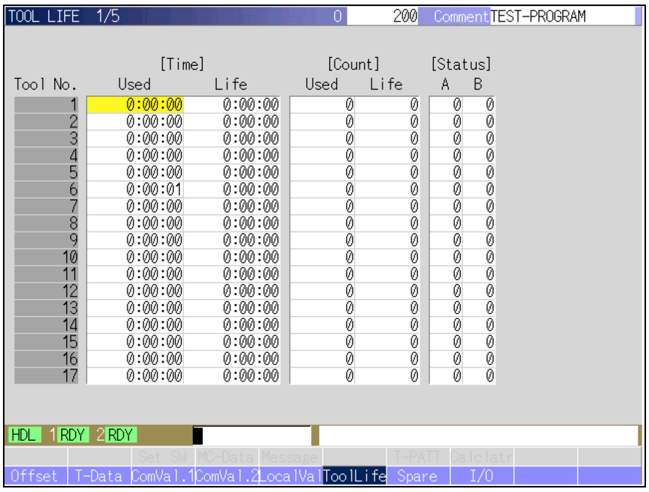
Tool life management I

- Management based on tool usage time

  The cutting time after a tool selection command (such as G01, G02 and G03) is added to the tool usage time corresponding to the commanded tool number. If the tool usage time exceeds its life in the tool selection command, a single cycle is halted and the alarm occurs.

- Management based on tool usage count

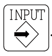
  The cutting command after a tool selection command (such as G01, G02 and G03) causes the tool usage count corresponding to the commended tool number to be incremended. If the tool usage count exceeds its life in a tool selection command, a single cycle is halted and the alarm occurs.



Display item		Explanation	Setting range
Tool No.		If the tool usage time reaches to its life time or the tool usage count exceeds its life count, the corresponding tool number is highlighted resersely.	–
Time	Used	Accumulated tool usage time Accumulated during cutting time.	0:0:0 - 4999:59:0 (Time: min.:sec.)
	Life	Tool life time Set the maximum tool usage time.	0:0:0 - 4999:59:0
Count	Used	Accumulated tool usage count Incremented during curring after tool selection.	0 - 99999 (Count)
	Life	Tool life count Set the maximum tool usage count.	0 - 99999
Status	A	Indicates the tool life management status. 0: Not used 1: Used 2: Life over	0 – 2
	B	Not used	–

[Note]

- The tool usage count is not incremented without the change of the tool number.
- The time management in seconds cannot be set.
- Set the tool usage time and life as follows:

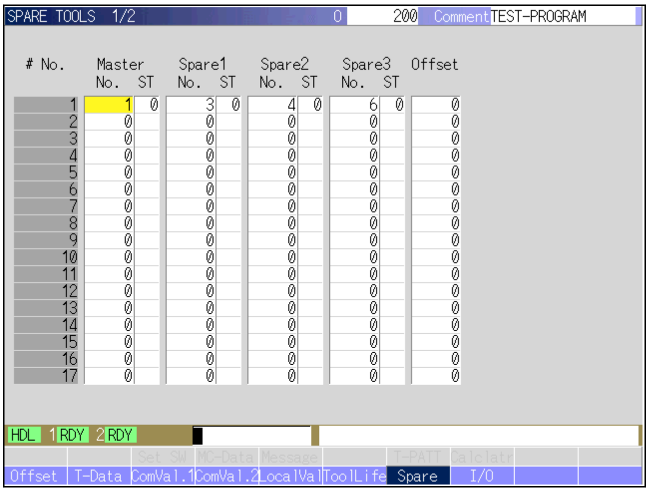
  For example, to set the time to 100 hours and 30 minutes, point the cursor to the item to be set and enter 100/30 in the setting field and press the Input key 
.

- After the life comes to cause alarm stop, clear the tool usage time or count and return the status to "0".
- If the life time or count is set to "0", the tool usage time is accumulated or the tool usage count is incremented but the alarm stop does not occur.
- Use shift to increment the tool number for tools that are called by specifying the R argument using the T code.

T code	Tool number	T code	Tool number	T code	Tool number
T0700	07	T0700 R1	62	T0700 R2	82
T0800	08	T0800 R1	63	T0800 R2	83
T0900	09	T0900 R1	64	T0900 R2	84
T1000	10	T1000 R1	65	T1000 R2	85
T1100	11	T1100 R1	66	T1100 R2	86
T1200	12	T1200 R1	67	T1200 R2	87
T1300	13	T1300 R1	68	T1300 R2	88
T1400	14	T1400 R1	69	T1400 R2	89
T1500	14	T1500 R1	70	T1500 R2	90
T5100	51	T5100 R1	71	T5100 R2	91
T5200	52	T5200 R1	72	T5200 R2	92
T5300	53	T5300 R1	73	T5300 R2	93
T5400	54	T5400 R1	74	T5400 R2	94
T5500	55	T5500 R1	75	T5500 R2	95
T5600	56	T5600 R1	76	T5600 R2	96
T5700	57	T5700 R1	77	T5700 R2	97
T5800	58	T5800 R1	78	T5800 R2	98
T5900	59	T5900 R1	79	T5900 R2	99

6.16 Spare Tool Life Management (Option)

In this management, the life (usage time and count) of each tool is managed and, if the life comes, a spare tool set for the tool is automatically selected to continue the machining. Up to 20 sets of master tools can be set. Up to three spare tools can be set for each master tool.



Display item		Explanation	Remarks
# No.		Number of master tools for which tool numbers can be registered.	
Master	No.	Master tool number. Set a master tool number.	
	ST	Indicates the life management status of a master tool.	0: Not used 1: Used 2: Life over
Spare 1	No.	Spare tool number. Set the first spare tool number.	
	ST	Indicates the life management status of spare tool 1.	0: Not used 1: Used 2: Life over
Spare 2	No.	Spare tool number. Set the second spare tool number.	
	ST	Indicates the life management status of spare tool 2.	0: Not used 1: Used 2: Life over
Spare 3	No.	Spare tool number. Set the third spare tool number.	
	ST	Indicates the life management status of spare tool 3.	0: Not used 1: Used 2: Life over
Offset		Offset value. For example, if the offset number is set to 3 and the offset number of a master tool number is T05, the offset number of spare tool 1 is T (5 + 3), or T8 and that of spare tool 2 is T (5 + 3 + 3), or T11.	

[Note]

To allow the tools of process pattern G660 (front/back simultaneous machining) to be replaced with spare tools, the tools on the front side (No. T1*) and rear side (No. T5*) should reach to their lives in the same cycle.

6.17 Running Program in External Memory

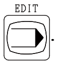
The external memory operation means to call and execute the program stored in the CF card. This is controlled by NC memory.

See the Programmer's Manual for calling a program.

To execute the program, store it in the CF card prepared for execution. The program in the size up to 1 GB can be executed.

To browse the program in the CF card, use the following procedure.

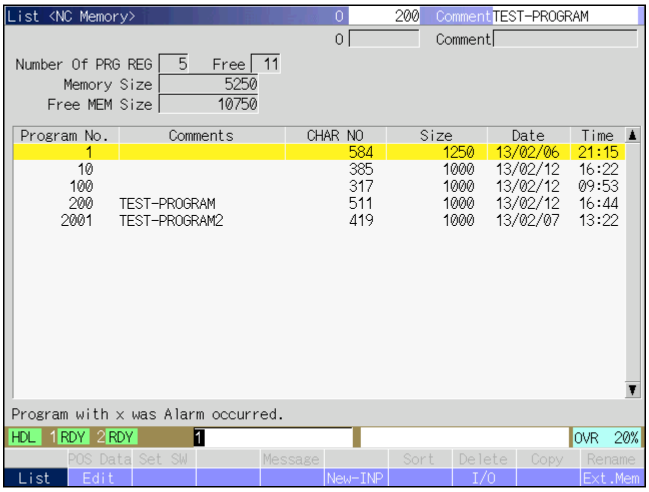
[Procedure]

1. Press the Edit key .

The previously selected screen appears.

2. Press the menu key [List].

The programs in the NC memory are listed.



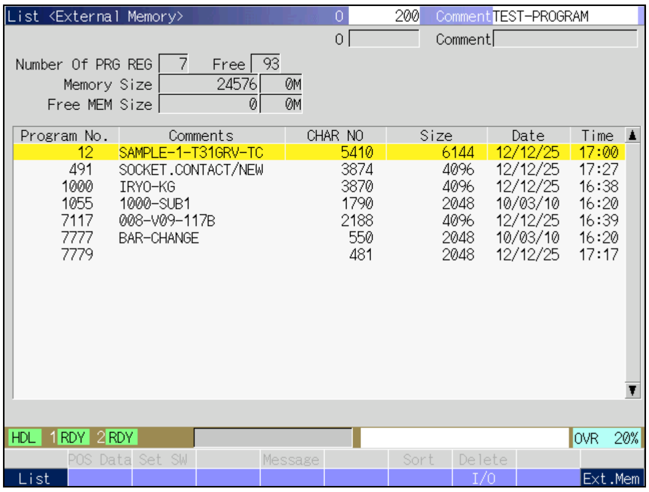
3. Set the CF card that contains the program to be executed.

4. Press the menu key [Ext.Mem].

[Note]

If the CF card is not set, the List <NC Memory> screen will not appear.

5. The list of programs stored in CF card are displayed.



[Note]

- Only the programs stored in the root directory of the CF card can be browsed or run.
- The program stored in CF card cannot be edited on the machine. Use the external device such as PC to edit the program.
- The program stored in CF card is regarded as a subprogram. In actual operation, the machining data of the main program is used.
- If cycle time is longer than 30 minutes due to large program size, enter the check mark for the setting switch of "13 Cycle Time ALM. OFF". For setting switches, see *<6.8.6 Setting switches>*. By checking the setting switch "13 Cycle Time ALM. OFF", operation stop mode at the occurrence of an alarm is changed from 1 cycle stop to immediate stop. This change applies to the following alarms.

Alarm No.	Symptom
EX202	Lubrication oil empty alarm.
EX203	Coolant oil alarm. Supply the oil.

- Do not remove the CF card while the program is running by the external memory device.
- If the CF card is removed while the program is running, an alarm "P463 Device not installed" will occur when the NC unit reads the program from the external memory device.
- If the external memory program operation is performed in automatic operation mode or on-machine program check mode, the program is read from the CF card. The screen display may not be refreshed while the program is being read.

6.18 Workpiece Conveyor (Option)

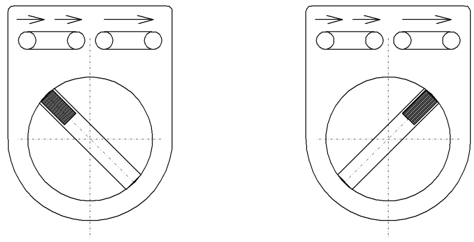
The workpiece conveyor unloads the workpiece out of the machine. The conveyor unloads workpieces one by one. Using the workpiece conveyor protects the workpiece from being damaged due to an interference with other workpieces.

Operation Modes

Two operation modes, continuous and intermittent, are available and the desired mode can be selected with the switch provided at the workpiece exit side of the conveyor. Turning the switch to the left selects the intermittent operation mode and turning it to the right selects the continuous operation mode.
In the continuous operation mode, the conveyor keeps running when the main breaker is ON. In the intermittent mode, the conveyor runs for a preset length of time according to the specification of an M code (M31 or M34). For details of M code specification, refer to the Programmer’s Manual.

Selection of Intermittent Operation Mode

Selection of Continuous Operation Mode



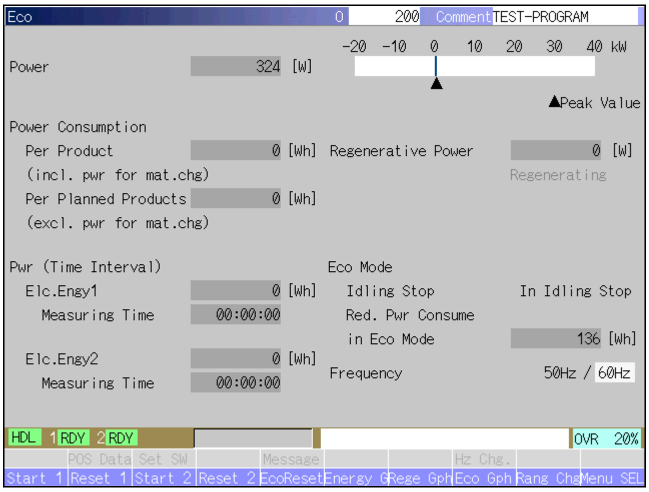
Workpiece Conveyor Operation Mode Selection Switch

CAUTION

When taking out products from the workpiece conveyor, unload them outside the machine by operating the workpiece conveyor in the continuous operation mode. Putting your hands in the workpiece conveyor could result in personal injury.

6.19 Eco Screen

Use this screen to see the power status of your machine for energy-saving purpose. The screen shows the current Power Consumption, Maximum Power Consumption, Total Power Consumption, and Regenerative status.



[Procedure]

1. Press the Diagnosis key .
   The previously selected screen appears.
2. Press the menu key [Menu SEL] until the menu key [Eco] is displayed.
3. Press the menu key [Eco].
   The Eco screen appears.

6.19.1 Power

The current power consumption is displayed in the numerical value and the graph. When regenerative energy is large, it becomes a minus indication, and it is displayed on a minus side in the graph because the regenerative energy is included in Power Consumption. In addition, the peak value after the power-on is displayed by ▲ mark.

6.19.2 Power Consumption

Power Consumption per product and per planned products are displayed.

After machining one material (after exchanging materials), the Power Consumption per product is calculated and displayed. The displayed Power Consumption contains electric power required for the material exchange .

The Power Consumption per planned products is calculated based on the electric power used in the second cycle of continuous operation. However, the electric power required for material exchange is not included in the displayed Power Consumption.

6.19.3 Regenerative energy

The current regenerative energy is displayed by positive numeric value.

Regenerative energy: The electric power generated when the main axis motor or the feed axis motor is decelerated.

6.19.4 Pwr (Time Interval)

The electric energy can be measured at an arbitrary time interval. During measurement, the message "Measuring" blinks, and the electric energy and the measurement time are updated.

[Procedure]

Elc.Engy1

1. Press the menu key [Start 1].

   The measurement starts and the message "Measuring" blinks. At this time, the menu key [Start 1] changes to [Stop 1].
2. Press the menu key [Stop 1].

   The measurement stops. At this time, the menu key [Stop 1] changes to [Restart 1]. Press the menu key [Restart 1] to continue measurement.
3. Press the menu key [Reset 1].

   The measured electric energy and the measuring time is cleared to 0.

The Elc.Engy2 can be measured in similar way.

6.19.5 Power regeneration

If any one of the main spindle, guide bushing, back spindle, or rotary tool is decelerated while the machine is running in automatic operation, on-machine program check, or MDI mode, the power regeneration status is displayed. "Regenerating" is displayed on green background.

6.19.6 Eco Mode

While the Eco Mode is enabled, status of Idling stop and the Reduced Power Consumption in Eco mode are displayed. When the Eco mode is disabled, they are displayed by thin characters.
To enable the Eco mode, remove the checkmark from "Eco Mode Disabled" on Set SW screen. (By the default, the Eco mode is enabled.)
During idling stop, the message "In Idling Stop" blinks on the green background.

Idling stop

- When the mode screen (e.g., Preparation, On-machine Check, Automatic Operation, MDI, Manual Operation), Zero Return screen, Mechanism Adjustment screen is not displayed and automatic power-on is disabled, excitation of the feed axis motor is turned off and the machine enters into the idling stop status.
  For example, in the default setting, the machine enters the idling stop status 30 minutes after transiting to the Edit screen except during automatic operation. On the Edit screen and List screen, the message "In Idling Stop" is displayed.
  Use PLC-Data #18123 to change the time until the machine enters idling stop status.# Since the 18123 value is set in 100 ms units, enter a #18123 = 6000 to have the machine enter idling stop status after 10 minutes.
  For details regarding PLC-Data settings, refer to *<23.6.1 PLC constant setting >*.
- The machine enters into idling stop status when no operation is performed on operation panel for five minutes.

[Note]

When exiting from Idling Stop status, perform ST POS (start position operation) on the Preparation screen before restarting machine operation.

6.19.7 Frequency

The frequency currently used is displayed. Selection of other frequency is also available.

[Procedure]

1. Press the Menu Up/Down selection key [▶].

   The submenus are enabled.
2. Press the menu key [Hz Chg.].

   The cursor is positioned at the selected Hz.
3. Use the Arrow keys [⬅][➡] to move the cursor to the desired Hz.
4. Press the menu key [Hz Chg.].

   The frequency is changed.

6.19.8 Eco Reset

Clears the value of Power Consumption per one product and per planned products.

[Procedure]

1. Press the menu key [EcoReset].

   The message "Are you sure to clear the value? Yes/No" and the menu keys [Yes] [No] are displayed.
2. Press the menu key [Yes].

   All the data is cleared.

6.19.9 Range change of displayed electric energy (W/kW)

You can change the range for the Power and Power Consumption displayed on the screen.

[Procedure]

1. Press the menu key [Ran Chg].

   The cursor is displayed on [W] or [kW] of Power Consumption range.
2. Use the Arrow keys [⬆][⬇] to Power Consumption you want to change.
3. Press the Menu Up/Down selection key [▶].

   The submenus are enabled.
4. Press the menu key [W/kW].

   The range is changed as [W]⇔[kW] or [Wh] ⇔[kWh].
5. Menu Up/Down selection key [▶].
6. Press the menu key [Rang Chg]. The cursor disappears.

[Note]

- Power Consumption

  The Power Consumption displays a rough value.

  Power Consumption of the peripheral devices are measured. Measurement is not performed on customer-specific device and the general-purpose bar feeder .

  Power Consumption of the devices other than driving device such as the main spindle motors and the feed axis motors is the mean values measured beforehand.
- When the operating program is changed, the amount of Power Consumption per one product and per planned products are cleared to 0.
- If the machine clock is changed to the next day or the subsequent days, the normal operation starts at the next day of the changed time. If the machine clock is changed to the last day and previous days, the normal operation will not start until the next day of the day before changed.

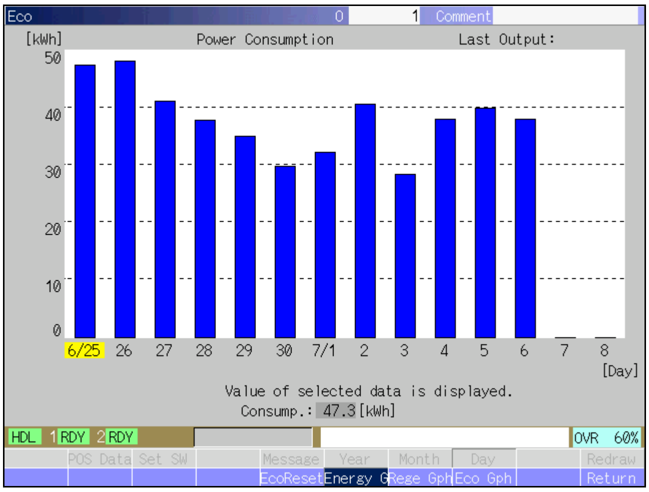
6.19.10 Graph display of electric energy

Power Consumption, Regenerative Energy, Reduction Power are displayed in graph format.

[Procedure]

Power Consumption

1. Press the Diagnosis key MAINT..
   The previously selected screen appears.
2. Press the menu key [Menu SEL] until the menu key [Eco] is displayed.
3. Press the menu key [Eco].
   The Eco screen appears.
4. Press the menu key [Energy G].
   Electric Energy is displayed in graph.
5. Press the Tab keys TAB ⇐ TAB ⇒ to select a graph you want to view the detailed information. The selected data is displayed in display area.



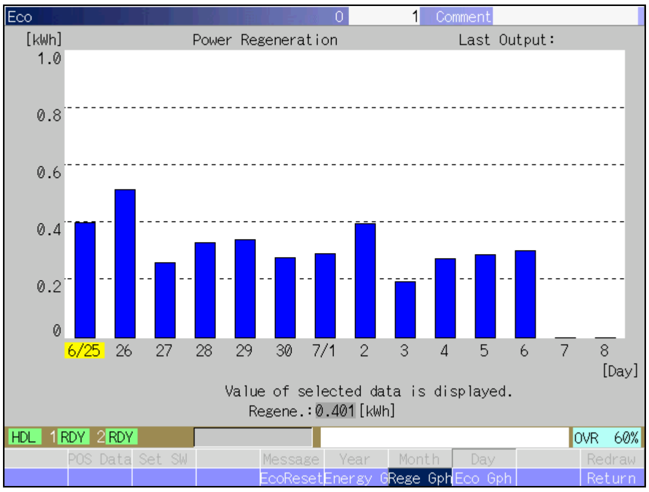
[Note]

- Pressing the menu key [Year], [Month], or [Day] displays the graph for each year, month, or day respectively.
- Up to 14 columns is displayed on graph.
- Pressing the menu key [Redraw] displays the latest data.
- The graph data displayed on screen can be output as the data in csv file format by using the File Access screen. See <*6.9.6.9 Output of Eco / Machine Operation Information*> for more information.
- The date you output the data last is displayed in "Last Output".
- If the machine clock is changed to the next day or the subsequent days, the normal operation starts at the next day of the changed time. If the machine clock is changed to the last day and previous days, the normal operation will not start until the next day of the day before changed.

[Procedure]

Regenerative energy

1. Press the Diagnosis key .
   The previously selected screen appears.
2. Press the menu key [Menu SEL] until the menu key [Eco] is displayed.
3. Press the menu key [Eco].
   The Eco screen appears.
4. Press the menu key [Rege Gph].
   Regenerative energy is diplayed in graph.
5. Press the Tab keys to select a graph you want to view the detailed information. The selected data is displayed in display area.



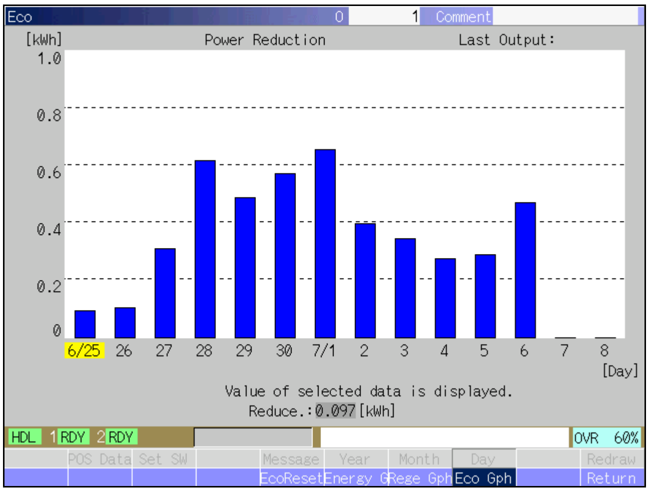
[Note]

- Pressing the menu key [Year], [Month], or [Day] displays the graph for each year, month, or day respectively.
- Up to 14 columns is displayed on graph.
- Pressing the menu key [Redraw] displays the latest data.
- The graph data displayed on screen can be output as the data in csv file format by using the File Access screen. See *<6.9.6.9 Output of Eco / Machine Operation Information>* for more information.
- The date you output the data last is displayed in "Last Output".
- If the machine clock is changed to the next day or the subsequent days, the normal operation starts at the next day of the changed time. If the machine clock is changed to the last day and previous days, the normal operation will not start until the next day of the day before changed.

[Procedure]

Reduced Power

1. Press the Diagnosis key MAINT..
   The previously selected screen appears.
2. Press the menu key [Menu SEL] until the menu key [Eco] is displayed.
3. Press the menu key [Eco].
   The Eco screen appears.
4. Press the menu key [Eco Gph].
   Reduced Power is displayed in graph.
5. Press the Tab keys TAB ⬅ TAB ➡ to select a graph you want to view the detailed information. The selected data is displayed in display area.



[Note]

- Pressing the menu key [Year], [Month], or [Day] displays the graph for each year, month, or day respectively.
- Up to 14 columns is displayed on graph.
- Pressing the menu key [Redraw] displays the latest data.
- The graph data displayed on screen can be output as the data in csv file format by using the File Access screen. See *<6.9.6.9 Output of Eco / Machine Operation Information>* for more information.
- The date you output the data last is displayed in "Last Output".
- If the machine clock is changed to the next day or the subsequent days, the normal operation starts at the next day of the changed time. If the machine clock is changed to the last day and previous days, the normal operation will not start until the next day of the day before changed.

6.20 Machine Operation Information Screen

6.20.1 Machine Operation Status

Classify the customer's Machine Operation Information into Automatic operation time, Alarm stop time, Setup time, No operation time, and Power off time. The information is displayed in graph by each item or time series.

Items of Machine Operation Status screen

Item	Description
Auto	Time measured when the continuous operation is performed in automatic operation mode.
Setup	Time measured when the setup is performed.
ALM Stop	Time measured from the occurrence of alarm to reset of alarm.
No Ope	Time measured when no operation is performed on operation panel.
PowerOFF	Time measured from power-off to next power-on of operation panel.

[Note]

- If switching to one-cycle operation during continuous operation in automatic operation mode, the time to end of one-cycle operation is summed up as automatic operation time.
- If switching to one-block operation during continuous operation in automatic operation mode, the time to switch point is summed up as automatic operation time.
- If switching to continuous operation during one-cycle operation in automatic operation mode, the time used for one-cycle operation is summed up as automatic operation time.
- If switching to continuous operation during one-block operation in automatic operation mode, the time after switch point is summed up as automatic operation time.
- One-cycle/one-block operation in automatic operation mode, running time on On-machine program check screen, and running time in MDI mode are summed up as setuptime.
- The time used for startup is summed up as setup time.
- The alarm stop due to counter full error is summed up as no operation time.
- When a cycle time over alarm is generated, the fixed time is added as alarm stop time and summed up. The fixed time depends on the setting.
- The change of the mode over 300 times per day is not recorded.
- If the machine clock is changed to the next day or the subsequent days, the normal operation starts at the next day of the changed time. If the machine clock is changed to the last day and previous days, the normal operation will not start until the next day of the day before changed.

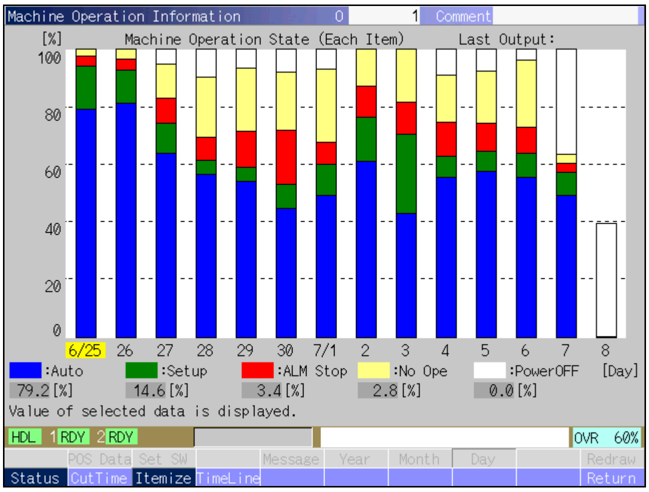
6.20.1.1 Operation information by item

When you press the menu key [Itemize], the five items are displayed in column chart.

[Procedure]

Machine Operation Information - Each Item

1. Press the Diagnosis key MAINT..
   The previously selected screen appears.
2. Press the menu key [Menu SEL] until the menu key [Status] is displayed.
3. Press the menu key [Status].
   The Machine Operation Information screen appears.
4. Press the menu key [Itemize].
   Itemized Machine Operation Information is displayed in graph.
5. Press the Tab keys TAB TAB to select a graph you want to view the detailed information. The selected data is displayed in display area.



[Note]

- Pressing the menu key [Year], [Month], or [Day] displays the graph for each year, month, or day respectively.
- Up to 14 columns is displayed on graph.
- "100%" indicates one year for Year data, one month for Month data, and 24 hours for Day data, respectively.
- Pressing the menu key [Redraw] displays the latest data.
- The graph data displayed on screen can be output as the data in csv file format by using the File Access screen. See *<6.9.6.9 Output of Eco / Machine Operation Information>* for more information.
- The date you output the data last is displayed in "Last Output".
- If the machine clock is changed to the next day or the subsequent days, the normal operation starts at the next day of the changed time. If the machine clock is changed to the last day and previous days, the normal operation will not start until the next day of the day before changed.

6.20.1.2 Machine Operation Information - Time Line

Five items are displayed by the time series in graphs by pressing the menu key[TimeLine].

[Procedure]

Machine Operation Information - Time Line

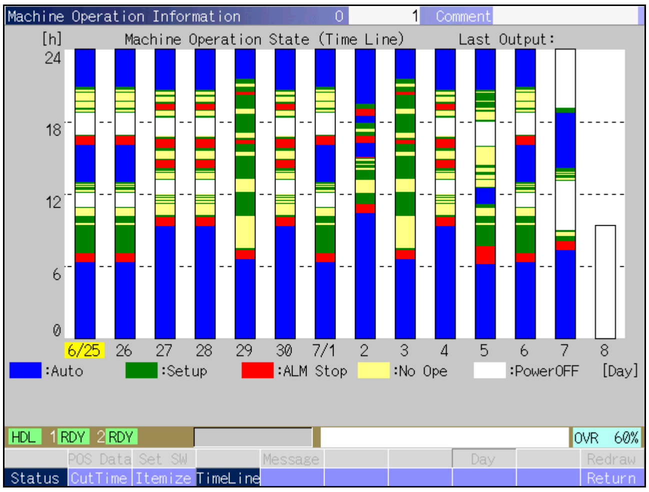
1. Press the Diagnosis key .

The previously selected screen appears.

2. Press the menu key [Menu SEL] until the menu key [Status] is displayed.

3. Press the menu key [Status].
The Machine Operation Information screen appears.

4. Press the menu key [TimeLine].
The graph of the time series of the machine operation information is displayed.



[Note]

- Only the daily data is displayed. The data display of month or year is not available.
- Up to 14 columns is displayed on graph.
- 24 hours are displayed as an elapsed time of a day in the time line display, and one scale represents about 5 minutes.
- Pressing the menu key [Redraw] displays the latest data.
- The graph data displayed on screen can be output as the data in csv file format by using the File Access screen. See <*6.9.6.9 Output of Eco / Machine Operation Information*> for more information.
- The date you output the data last is displayed in "Last Output".
- If the machine clock is changed to the next day or the subsequent days, the normal operation starts at the next day of the changed time. If the machine clock is changed to the last day and previous days, the normal operation will not start until the next day of the day before changed.

6.20.2 Ratio of cut time / non-cut time

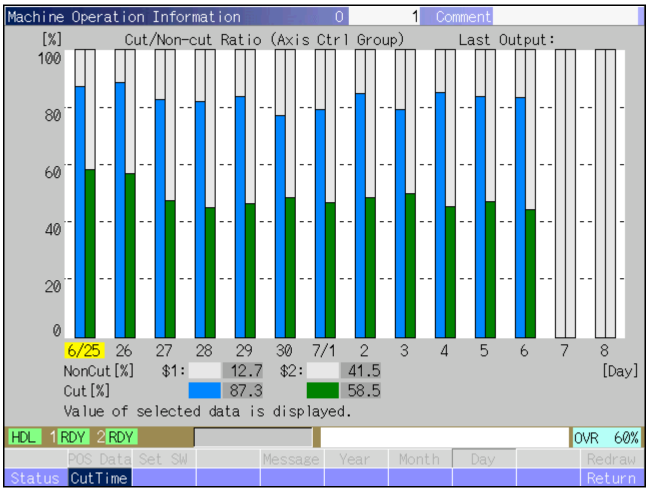
The ratio of cut time/non-cut time for each axis control group during continuous operation performed in the automatic operation mode is displayed in the graph.

Ratio of cut time / non-cut time

Item	Description
Cut time	Shows the cutting time in automatic operation.
Non-cut time	Shows the non-cutting time in automatic operation.

[Procedure]

1. Press the Diagnosis key MAINT. The previously selected screen appears.
2. Press the menu key [Menu SEL] until the menu key [Status] is displayed.
3. Press the menu key [Status]. The Machine Operation Information screen appears.
4. Press the menu key [CutTime]. The graph of the cutting time/non-cutting time ratio is displayed.
5. Press the Tab keys TAB TAB to select a graph you want to view the detailed information. The selected data is displayed in display area.

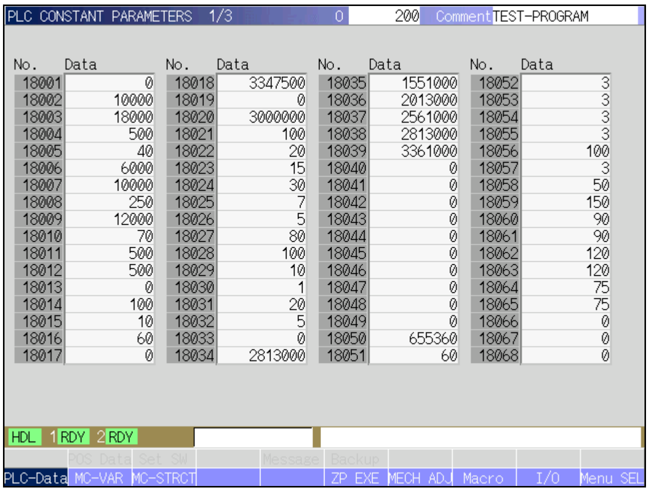


[Note]

- Pressing the menu key [Year], [Month], or [Day] displays the graph for each year, month, or day respectively.
- Up to 14 columns is displayed on graph.
- 100% represents:
  Year data: automatic operation time for one year
  Month data: automatic operation time for one moth
  Day data: automatic operation time for one day
- Pressing the menu key [Redraw] displays the latest data.
- The graph data displayed on screen can be output as the data in csv file format by using the File Access screen. See *<6.9.6.9 Output of Eco / Machine Operation Information>* for more information.
- The date you output the data last is displayed in "Last Output".
- If the machine clock is changed to the next day or the subsequent days, the normal operation starts at the next day of the changed time. If the machine clock is changed to the last day and previous days, the normal operation will not start until the next day of the day before changed.

6.21 PLC Data Screen

This screen lists constants used with PLC.



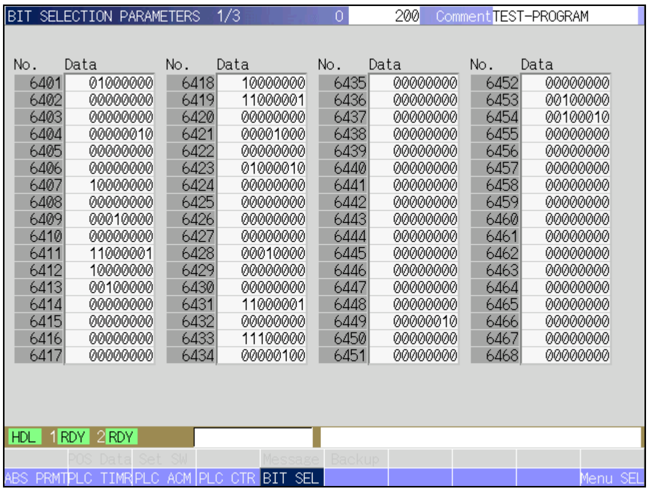
[Procedure]

1. Press the Parameter key PRM.
   The previously selected screen appears.
2. Press the menu key [Menu SEL] until the menu key [PLC-Data] is displayed.
3. Press the menu key [PLC-Data].
   The PLC CONSTANT PARAMETERS screen appears.
4. Use the Page keys to browse the parameters.

Item	Description
No.	Constant numbers
Data	Values of constants

6.22 Bit Select Screen

This screen lists bit parameters used with PLC.



[Procedure]

1. Press the Parameter key PRM..
The previously selected screen appears.

2. Press the menu key [Menu SEL] until the menu key [BIT SEL] is displayed.

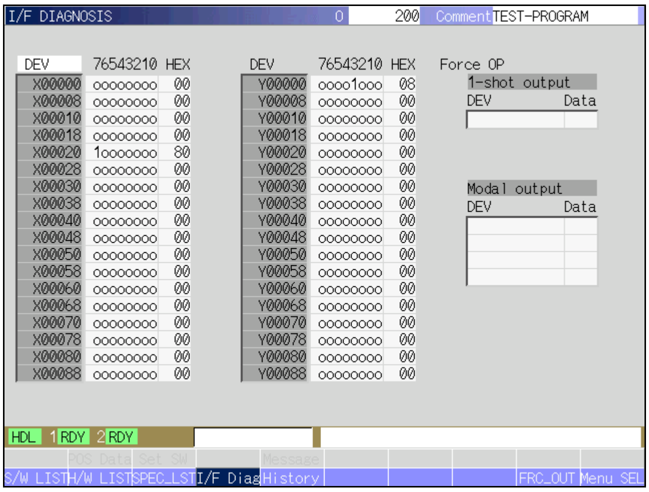
3. Press the menu key [BIT SEL].
The BIT SELECTION PARAMETERS screen appears.

4. Use the Page keys ↑ ↓ to browse the parameters.

Item	Description
No.	Bit parameter numbers
Data	Values (0 or 1) of bit parameters

6.23 I/F Diagnosis Screen

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



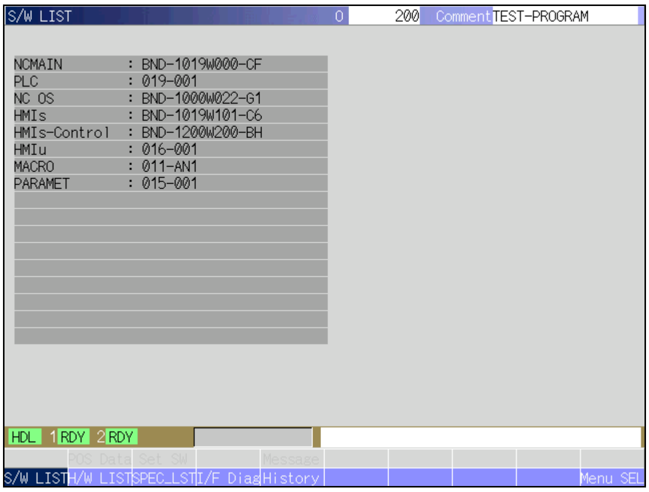
[Procedure]

1. Press the Diagnosis key.
   The previously selected screen appears.
2. Press the menu key [Menu SEL] until the menu key [I/F Diag] is displayed.
3. Press the menu key [I/F Diag].
   The I/F DIAGNOSIS screen appears.
4. Use the Tab keys to select a table to be displayed. The "DEV" column of the selected table is reversed in white.
5. Enter the number you want to browse and press the Input key, the screen jumps to the specified signal. Or, you can use the Page keys to browse them.

Item	Description
DEV	Device numbers used with PLC
76543210	The data corresponding to each device number is expressed with bits.
HEX	Data is displayed in hexadecimal.

6.24 Software List Screen

This screen displays software versions.



[Procedure]

1. Press the Diagnosis key .
   The previously selected screen appears.
2. Press the menu key [Menu SEL] until the menu key [S/W LIST] is displayed.
3. Press the menu key [S/W LIST].
   The S/W LIST screen appears.

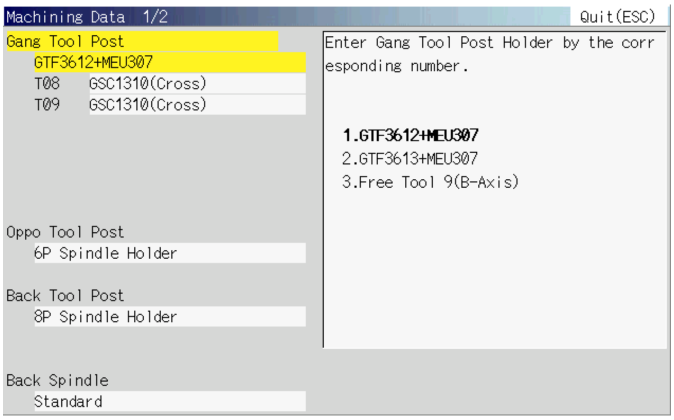
Item	Description
NCMAIN	Shows software versions.
PLC	
NC OS	
HMIs	
HMIs-Control	
HMIu	
MACRO	
PARAMET	

6.25 B-axis Operation

B-axis operation requires a previous setting

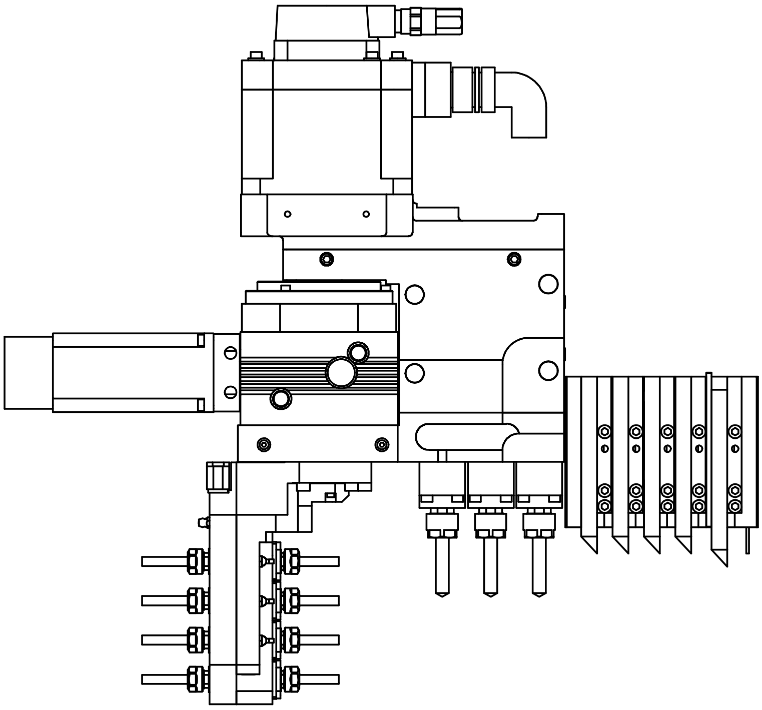
6.25.1 Machining data setting

When a holder mounted on the B-axis unit is selected, specify the holder with the B-axis (***+MEU207) for gang tool post in machining data.



6.25.2 Tool setting for B-axis tools

Adjust the shift amount and center position of the B-axis. It can be adjusted in cross direction and end-face direction. Set the tool names in the Preparation screen to distinguish between cross direction and end-face direction. A tool name like B-axis cross machining indicates cross direction, while names like B-axis end-face drilling indicates end-face direction.Normally, tools should be set in cross direction.

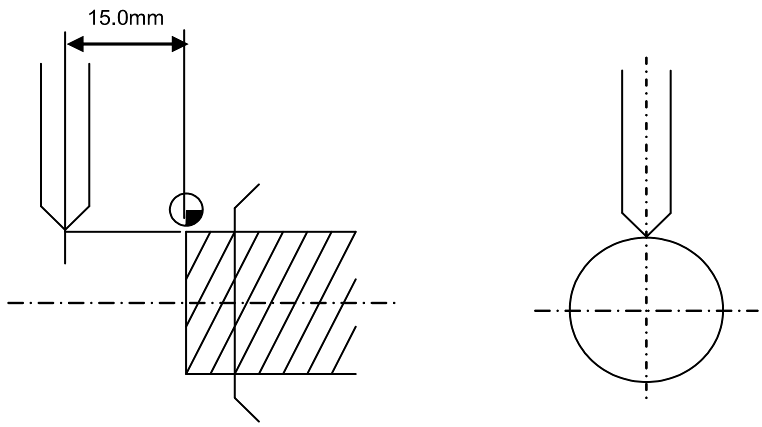


6.25.3 Shift amount adjustment

■ Adjusting shift amount for a B-axis tool in cross direction

Place the cursor on the diameter and select [Man. Set] to turn the B-axis tool to cross position (a B-axis angle of 90 degrees) and call it to the diameter position.Then adjust it in diameter direction.

Diameter position



[Procedure]

1. Press the Preparation key PREPA..

   The Preparation key lamp lights and the Preparation screen appears.

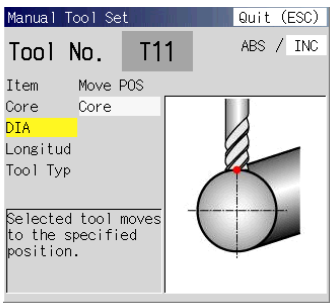
2. Specify the tool name as B-axis cross machining.

   See <*6.2.7.11 Tool Name*>.

3. Use the arrow keys and tab key to place the cursor on the diameter of the tool (T1100 to T1400) you want to set.
4. Make sure that the material does not protrude from the guide bushing.
5. Press the menu key [Man. Set].

   The Manual Tool Set screen appears.

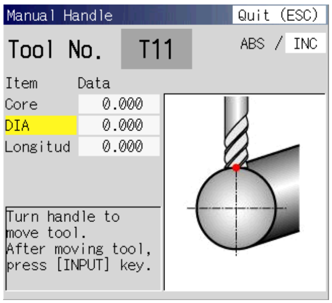
   Make sure that the cursor is placed on the diameter. If the cursor is not on the diameter, use the arrow keys ⬆ ⬇ to move the cursor to the diameter.



6. Make sure that all doors are closed, and press the Start button.
   The Start button lamp blinks on and off, and the doors will lock.
7. Press the Start button.

   The Start button lights and the selected tool rotates to the cross position (a B-axis angle of 90 degrees) and the tool tip moves to a point on the circumference of the material.

8. When operations are completed, the Start button lamp turns off and the screen changes to the Manual Handle screen.



9. Press the Door open button to unlock the door, and open the door.

10. Pull out the material.

11. Make sure that the cursor is on the diameter and handle feed the X-axis in the + X direction to obtain the necessary protrusion amount for machining and temporarily stop the tool when it comes into contact with the outer diameter of the material.

12. Press the Input key.

The diameter value is set in the Preparation screen.

13. Handle feed the Y-axis in the + direction to move the rotary tool away from the material.

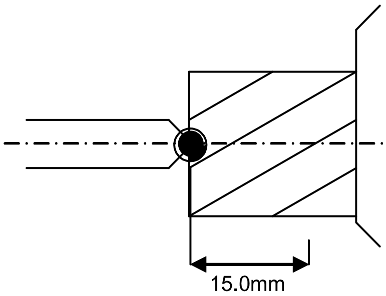
14. Firmly secure the rotary tool.

15. Press the Escape key.

■ Adjusting shift amount for a B-axis tool in end-face direction (T1100 to T1400)

Place the cursor at the longitude and select [Man. Set] to turn the B-axis tool to end-face position (a B-axis angle of 0 degrees) and call it to the longitudinal position G/B center).Then adjust it in longitudinal direction.

Longitudinal position



[Procedure]

1. Press the Preparation key PREPA..

   The Preparation key lamp lights and the Preparation screen appears.

2. Specify the tool name as B-axis end-face drilling.

   See <*6.2.7.11 Tool Name*>.

3. Turn off bar loader torque before opening the chuck.

   (Turn off a dedicated bar loader BAR LOADER POWER . On other bar loaders, turn off torque.)

4. Cut-off machining is performed at the end-face of the material. (The chuck remains closed when the operation ends.)

   See <*6.2.8 Cut-off machining*>.

5. The material is aligned with the end-face of the cut-off tool. Cut-off machining ends when the material and the cut-off tool are brought into contact with each other.

6. The cut-off tool moves to the positioning point.

7. A left-handed cut-off tool ... Moves to the next position.

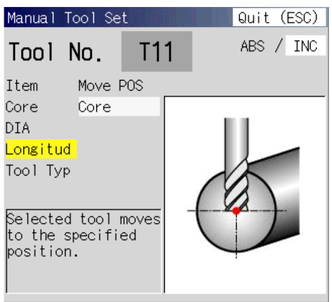
   A right-handed tool... Moves the material end-face (Z1-axis) to the program zero point.

8. Use the Arrow keys ⬆ ⬇ or the Tab keys TAB ⬅ TAB ➡ to move the cursor to the Longitud of the desired tool (T1100 to T1400).

9. Mount a tool in the rotary tool. Insert it deep enough that it does not interfere with the material.

10. Press the menu key [Man. Set].

    The Manual Tool Set screen appears. Make sure that the cursor is placed at the longitude. If the cursor is not on the longitude, use the arrow keys ⬆ ⬇ to move it to the longitude.



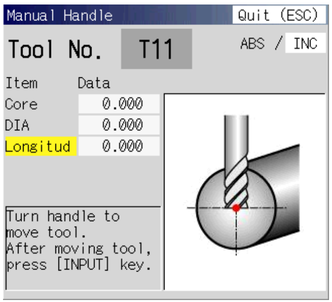
11. Enter numerics so that the longitudinal value becomes 15.0 mm and press the Input key INPUT. (This is done because the program zero point of the B-axis tool is 15.0 mm away from the guide bushing.)

12. Make sure that all doors are closed, and press the Start button.
The Start button lamp blinks on and off, and the doors will lock.

13. Press the Start button.

The Start button lamp lights, the material (Z1-axis) retracts 1 mm, the selected tool rotates to the end-face position (a B-axis angle of 0 degrees) and when it has moved to the center of the guide bushing, the material (Z1-axis) moves forward 15.0 mm (longitude value) and then one more mm.

14. When operations are completed, the Start button lamp turns off and the screen changes to the Manual Handle screen.



15. Make sure that cursor is at the longitude and handle feed the material to the position you want to place it.

16. Press the Input key.

The longitudinal value is set in the Preparation screen.

17. Press the Door open button to unlock the door, and open the door.

18. Bring the tip of the tool into contact with the material and temporarily stop the sleeve.

19. Press the Escape key to finish the operation.

20. Release the material and firmly secure the sleeve.

■ Precautions for adjusting shift amount for a B-axis tool in end-face direction

When making adjustments in end-face direction and placing the tip of the tool at the program zero point as shown in Figure 1, the tool will be 15.0 mm away from the center or B-axis rotation.In this situation, use the customary longitudinal value of 0 for tool setting.

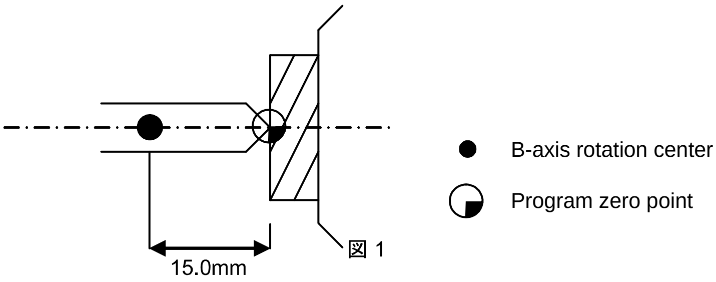


図 1

As shown in Figure 2, when tools are placed so that they protrude only slightly beyond the program zero point, be sure to use the coordinate system shift without programming to set the shift amount at the longitudinal value of the tool set.

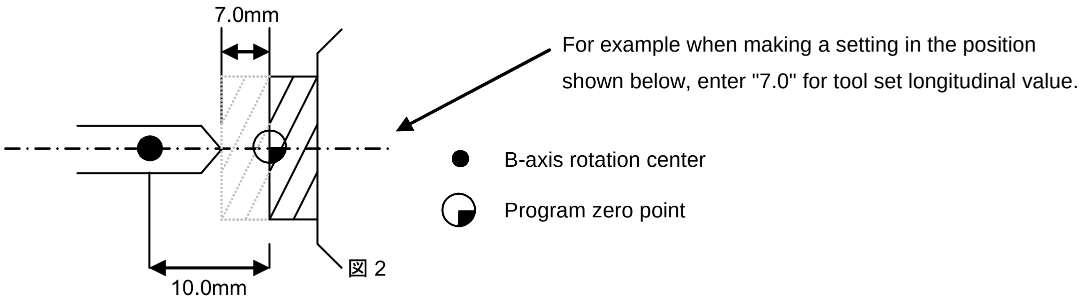


図 2

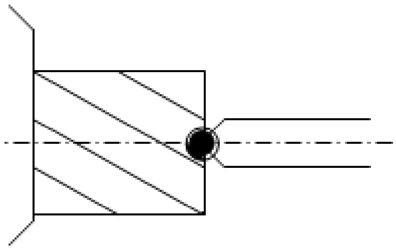
[Note]

- To enable operation with doors open, press DOOR OPEN, the Door open button, to open the door lock and set the setting switch to the [I] position. When this key is set to [O], the machine cannot be operated with doors open.
- When the doors are open, the unit will operate only when the Start button is held down.
  Then axis speed is limited to 2m/min.
- Releasing the Start button stops axis movement and the Start button lamp flashes (on: 2 sec, off: 1 sec).Press the Start button to resume operation.
- Menu selections are automatically canceled if no operation is performed within 15 seconds.
- A reset performed during or after operation will also cancel menu selections.To resume operation, make new menu selections.
- The speed of a handle feed operation is limited to 1.8 m/min.
- Multiplication ratio x 100 is not available during core, diameter and center adjustments using the handle.
- The menu key [Man. Set] are not available when the program is not selected.

■ Adjusting shift amount for a B-axis tool in end-face direction (T5100 to T5400)

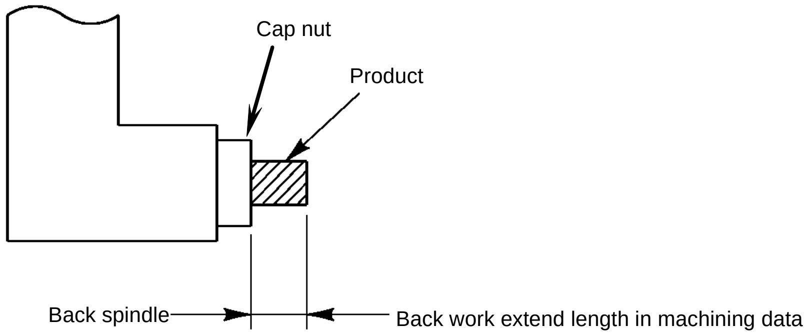
Place the cursor at the longitude and select [Man. Set] to turn the B-axis tool to end-face position (a B-axis angle of 0 degrees) and call it to the longitudinal position G/B center).Then adjust it in longitudinal direction.

Longitudinal position



[Procedure]

1. Chuck a product (material) on the back spindle.
   Usually, chuck that product which has been front-machined by one-cycle operation and its overall length is corrected.



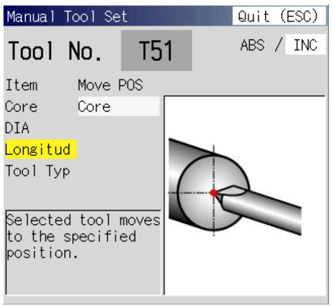
2. Measure the length of the protrusion of the product from the back chuck and check whether the measured length matches the value entered in machining data "Back Spindle Chuck POS."
   (For machining data, see *< 6.1.3 Required fields for machining data>* and *<6.1.4 Checking machining data>*.)
3. Press the Preparation key PREPA..

   The Preparation key lamp lights and the Preparation screen appears.
4. Use the Arrow keys ⬆⬇ or the Tab keys TAB⬅ TAB➡ to move the cursor to the Longitud of the desired tool (T5100 to T5400)
5. Specify the tool name as B-axis end-face drilling.

   See *<6.2.7.11 Tool Name>*.
6. Insert the tool deeply.

7. Press the menu key [Man. Set].

The Manual Tool Set screen appears. Make sure that the cursor is placed at the longitude. If the cursor is not on the longitude, use the arrow keys ⬆ ⬇ to move it to the longitude.

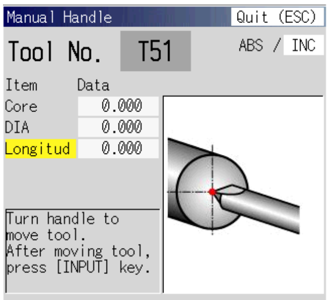


8. Make sure that all doors are closed, and press the Start button.
The Start button lamp blinks on and off, and the doors will lock.

9. Press the Start button.

The Start button lamp lights, the Z2-axis moves to the return position, the X2-axis moves to the spindle center line position and after positioning the selected tool, the Z2-axis advances to the standard setting position.

10. When operations are completed, the Start button lamp turns off and the screen changes to the Manual Handle screen.



11. Press the Door open button to unlock the door, and open the door.

12. Bring the tip of the tool into contact with the material and temporarily stop it.

13. Press the menu key [OPP-RET], make sure that all doors are closed, and press the Start button.
The Start button lamp blinks on and off, and the doors will lock.

14. Press the Start button.
The Start button lamp lights and the opposite tool post returns to the Z2 axis zero point.

15. Firmly secure the tool.

[Note]

- To enable operation with doors open, press DOOR OPEN [ ], the Door open button, to open the door lock and set the setting switch to the [I] position. When this key is set to [O], the machine cannot be operated with doors open.
- When the doors are open, the unit will operate only when the Start button [ ] is held down.
  Then axis speed is limited to 2m/min.
- Releasing the Start button stops axis movement and the Start button lamp flashes (on: 2 sec, off: 1 sec).Press the Start button to resume operation.
- Menu selections are automatically canceled if no operation is performed within 15 seconds.
- A reset performed during or after operation will also cancel menu selections. To resume operation, make new menu selections.
- The speed of a handle feed operation is limited to 1.8 m/min.
- Multiplication ratio x 100 is not available during core, diameter and center adjustments using the handle.
- The menu key [Man. Set] are not available when the program is not selected.

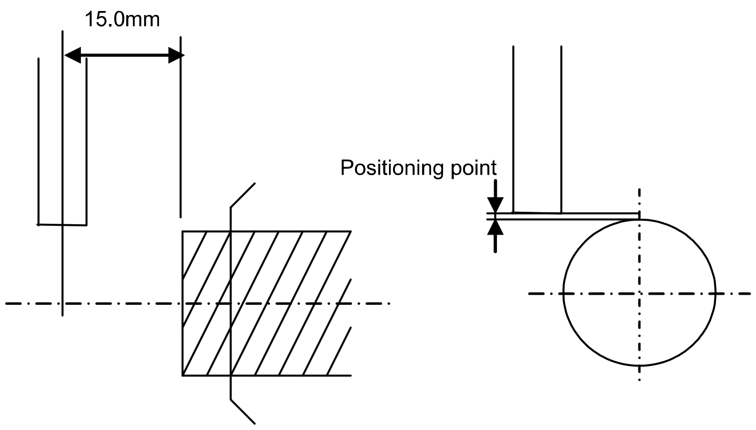
6.25.4 Center (core) position adjustments

Align the center (core) of a B-axis tool.

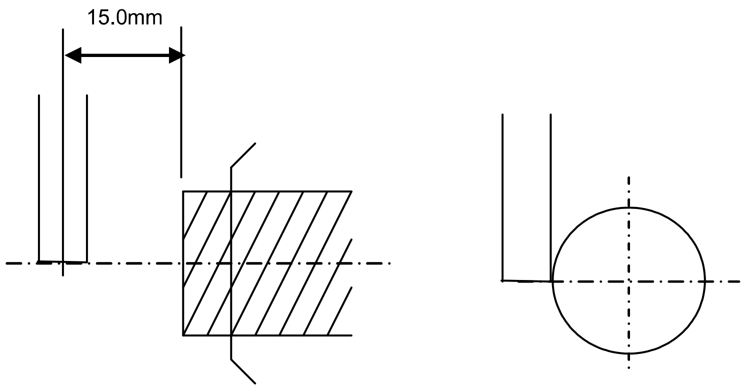
■ Adjusting center position (core) of a B-axis tool in cross direction

Place the cursor on the core and select [Man. Set] to turn the B-axis tool to cross position (a B-axis angle of 90 degrees) and call it to the core (outer circumference, core down) position.Then adjust it in core direction.

Core (outer circumference) position



Core down position



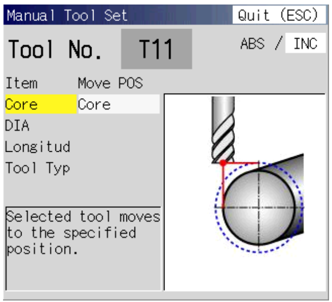
[Procedure]

1. Press the Preparation key PREPA..

   The Preparation key lamp lights and the Preparation screen appears.
2. Specify the tool name as B-axis cross machining.

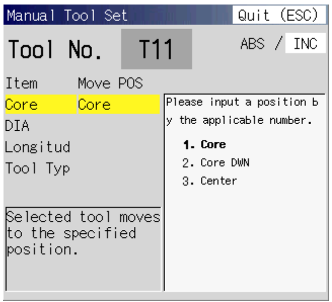
   See <*6.2.7.11 Tool Name*>.
3. Use the Arrow keys ⬆ ⬇ or the Tab keys TAB ⬅ TAB ➡ to move the cursor to the Core of the desired tool (T1100 to T1400).
4. Press the menu key [Man. Set].

   The Manual Tool Set screen appears. Make sure that the cursor is placed on the core. If the cursor is not on the core, use the arrow keys ⬆ ⬇ to move the cursor to the core.



5. Press the arrow key ⬆.

   A moving position list is displayed.



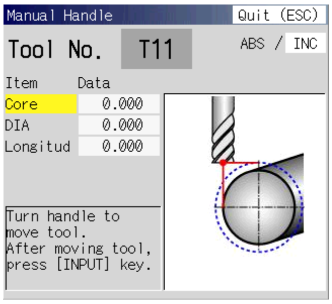
6. Enter the selected number for Core or Core DWN from the moving position list and press the Input key INPUT.

   A moving position selection is stored as a default value. For that reason, once "Core" or "Core DWN" has been selected, it will be available at the next tool setting.
7. Make sure that the material does not protrude from the guide bushing.
8. Make sure that all doors are closed, and press the Start button.

   The Start button lamp blinks on and off, and the doors will lock.
9. Press the Start button.

   The Start button lights and the selected tool rotates to the cross position (a B-axis angle of 90 degrees) and the tool tip moves to a point on the circumference of the material.

10. When operations are completed, the Start button lamp turns off and the screen changes to the Manual Handle screen.



11. Press the Door open button to unlock the door, and open the door.

12. Pull out the material and check that the top of the tool is aligned with the outer circumference of the material.

13. Make sure that the Core or Core DWN is selected, then use handle feed to align the top of the tool with the outer circumference of the material.

14. Consider the tool diameter in calculating the handle feed amount (handle feed amount + tool diameter) when you manually enter the total amount.

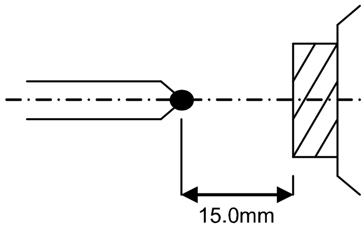
The core value is set in the Preparation screen

15. Press the Escape key ESC to finish the operation.

■ Adjusting center position (core) of a B-axis tool in end-face direction (T1100 to T1400)

Place the cursor at the core or diameter and select [Man. Set] to turn the B-axis tool to end-face position (a B-axis angle of 0 degrees) and call it to the core/diameter position (G/B center). Then adjust it in core/diameter direction.

Core/diameter position



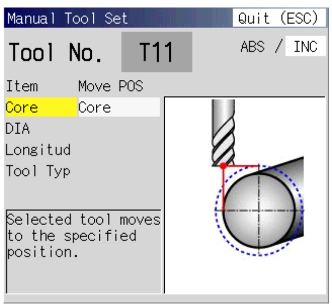
[Procedure]

1. Press the Preparation key PREPA. The Preparation key lamp lights and the Preparation screen appears.
2. Specify the tool name as B-axis cross machining. See *<6.2.7.11 Tool Name>*.
3. Turn off bar loader torque before opening the chuck.

   (Turn off a dedicated bar loader BAR LOADER POWER . On other bar loaders, turn off torque.)
4. Cut-off machining is performed at the end-face of the material. (The chuck remains closed when the operation ends.)

   See *<6.2.8 Cut-off machining>*.
5. The material is aligned with the end-face of the cut-off tool. Cut-off machining ends when the material and the cut-off tool are brought into contact with each other.
6. The cut-off tool moves to the positioning point.
7. Use the arrow keys ⬆ ⬇ and tab key TAB ⬅ TAB ➡ to place the cursor on the core or the diameter of the tool (T1100 to T1400) you want to set.
8. Press the Door open button to unlock the door, and open the door.
9. Mount the tool in the sleeve and insert the sleeve deep into the tool post.
10. Press the menu key [Man. Set].

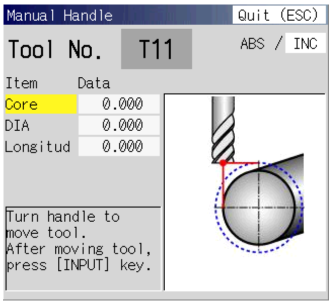
    The Manual Tool Set screen appears. Make sure that the cursor is placed on the core or diameter. If the cursor is not on the core or diameter, use the arrow keys ⬆ ⬇ to move the cursor to the core or the diameter.



11. Make sure that all doors are closed, and press the Start button.
    The Start button lamp blinks on and off, and the doors will lock.
12. Press the Start button.

    The Start button lamp lights, the material (Z1-axis) retracts 1 mm, the selected tool rotates to the end-face position (a B-axis angle of 0 degrees) and when it has moved to the center of the guide bushing, the material (Z1-axis) moves forward 1 mm.

13. When operation is completed, the Start button lamp goes off and the Manual Handle screen appears.



14. Press the Door open button to unlock the door, and open the door.

15. Pull out the end-face of the material to a position where it is not in contact with the tool.

16. Select the core or diameter and use handle feed to position and adjust the front end and tip of the tool to the center of the circumference of the material.

17. Press the Input key .

The core and diameter value is set in the Preparation screen.

18. Press the Escape key to finish the operation.

19. Move the material away from the tool.

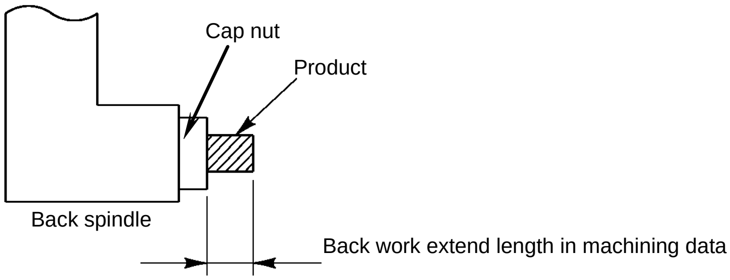
[Note]

- To enable operation with doors open, press DOOR OPEN , the Door open button, to open the door lock and set the setting switch to the [I] position. When this key is set to [O], the machine cannot be operated with doors open.
- When the doors are open, the unit will operate only when the Start button is held down.
  Then axis speed is limited to 2m/min.
- Releasing the Start button stops axis movement and the Start button lamp flashes (on: 2 sec, off: 1 sec).Press the Start button to resume operation.
- Menu selections are automatically canceled if no operation is performed within 15 seconds.
- A reset performed during or after operation will also cancel menu selections. To resume operation, make new menu selections.
- The speed of a handle feed operation is limited to 1.8 m/min.
- Multiplication ratio x 100 is not available during core, diameter and center adjustments using the handle.
- The menu key [Man. Set] are not available when the program is not selected.

■ Adjusting center position (core) of a B-axis tool in end-face direction (T5100 to T5400)

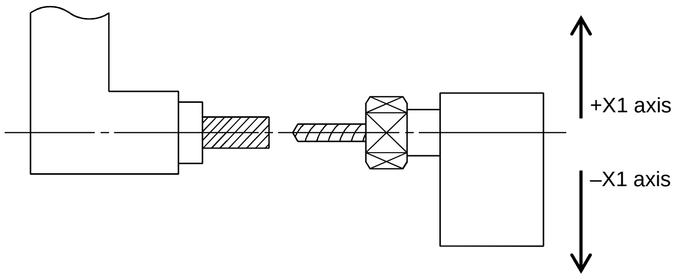
[Procedure]

1. Chuck a product (material) on the back spindle.

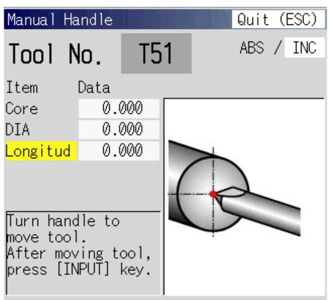


2. Press the Preparation key [PREPA.]. The Preparation key lamp lights and the Preparation screen appears.
3. Use the Arrow keys [↑][↓] or the Tab keys [TAB ←][TAB →] to move the cursor to the Core or DIA for the tool you want to set.
4. Press the menu key [Man. Set].
   The Manual Tool Set screen appears. Make sure that the Core or diameter setting is highlighted with the cursor. If the cursor is not at the Core or diameter setting, use the Arrow keys [↑][↓] to move the cursor to the Core or diameter.
5. Make sure that all doors are closed, and press the Start button [Start].
   The Start button lamp blinks on and off, and the doors will lock.
6. Press the Start button [Start].
   The Start button lamp lights, the Z2 axis retracts full stroke, the selected tool is positioned and then the back spindle is selected for being centering.

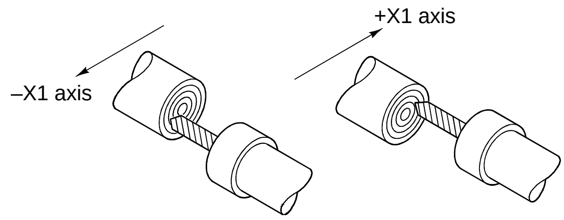
The tool is selected.



7. Move the cursor to Longitud by pressing the Arrow keys [↑][↓] and advance the Z2 axis by handle.



8. Move the X1 or Y1 axis in handle feed so that the tip of the tool bit is aligned with the center of the cut-off mark at the end of the material.



9. Press the Input key .

10. Press the menu key [OPP-RET], make sure that all doors are closed, and press the Start button .
The Start button lamp blinks on and off, and the doors will lock.

11. Press the Start button .
The Start button lamp lights and the opposite tool post returns to the Z2 axis zero point.

[Note]

- To enable operation with doors open, press DOOR OPEN , the Door open button, to open the door lock and set the setting switch to the [] position.When this key is set to [], the machine cannot be operated with doors open.
- When the doors are open, the unit will operate only when the Start button is held down.

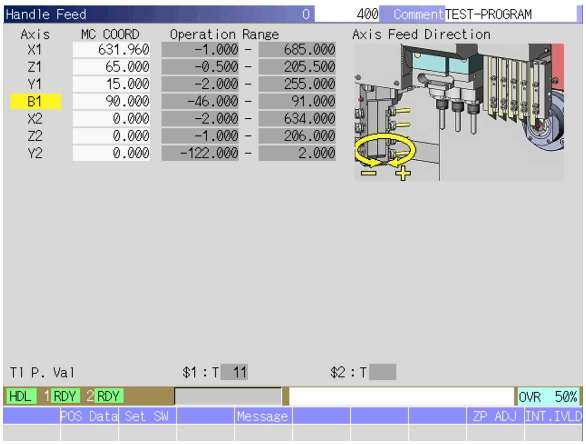
  Then axis speed is limited to 2m/min.
- Releasing the Start button stops axis movement and the Start button lamp flashes (on: 2 sec, off: 1 sec).Press the Start button to resume operation.
- Menu selections are automatically canceled if no operation is performed within 15 seconds.
- A reset performed during or after operation will also cancel menu selections. To resume operation, make new menu selections.
- The speed of a handle feed operation is limited to 1.8 m/min.
- Multiplication ratio x 100 is not available during core, diameter and center adjustments using the handle.
- The menu key [Man. Set] are not available when the program is not selected.

6.25.5 Handle feed

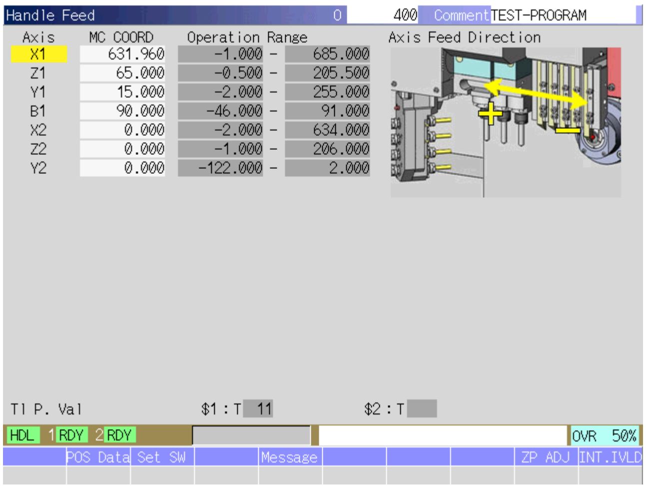
■ B-axis front machining (T11's)

If for some reason, operation stops during front machining using B-axis tools, and the program operation is reset, use the Handle Feed screen to move the Z-axis in minus direction and retract the B-axis tools from the material.

When T1100 - T1400 is selected



To retract a B-axis tool in B-axis angle direction



■ B-axis back machining (T51's)

If for some reason, operation stops during front machining using B-axis tools, and the program operation is reset, instead of using the Handle Feed screen in B-axis front machining (T11's) like in the example above, use MDI program operation to remove the tool from the workpiece. For details, refer to the "Programmer's Manual."

[Procedure]

1. Make sure that the selected tool is T1100 - T1400.

   If the selected tool is something other than T1100 - T1400, it will not be possible to move in the B-axis angle direction.

2. Press the Manual key MANUAL.

   The Manual key lamp goes on and the Handle Feed screen appears.

3. Deselect the menu key (motor axis) (no menu key reversal).

4. Use the arrow keys ⬆ ⬇ to select Z1.

   Z1 is highlighted yellow.

5. Press any of the Magnification buttons x1, x10, or x100.

   The Magnification button lamp lights.

   Higher numbers indicate faster machine speed.

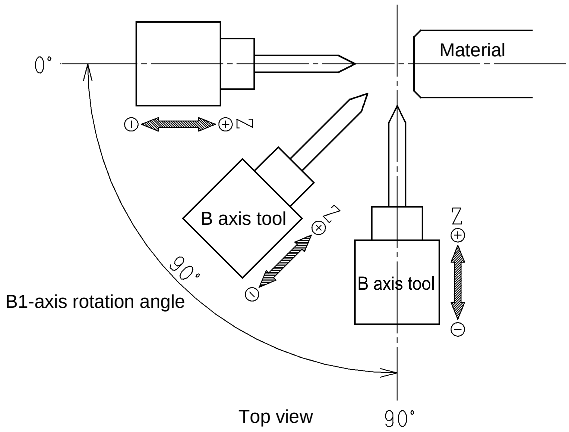
6. Turn the handle in minus direction.

   The tool is retracted from the material.

   When a movement command that causes movement to exceed stored stroke limit is issued, an overrun alarm indicating that "M01 S/W stroke limit axis detected" is generated.

   Refer to *<6.8.2 Recovering from overrun status>* for information on how to clear an overrun alarm.

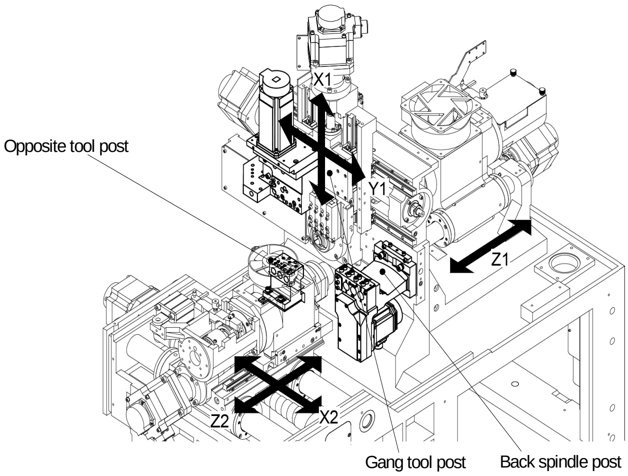
Direction of movement during Z1 handle feed



■ To move the B-axis in motor axis direction

What is motor axis direction?

Motor axis direction indicates the direction that the feed axis motor is installed in.See the figure below.



[Procedure]

1. Press the Manual key MANUAL.

   The Manual key lamp goes on and the Handle Feed screen appears.

2. Deselect the menu key [motor axis] (no menu key reversal).

3. Use the arrow keys ⬆ ⬇ to select the control axis you want to feed.

   The name of the selected control axis is highlighted yellow.

4. Press any of the Magnification buttons X1, X10 or X100.

   The Magnification button lamp lights.

   Higher numbers indicate faster machine speed.

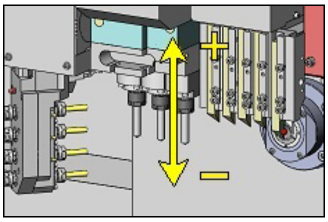
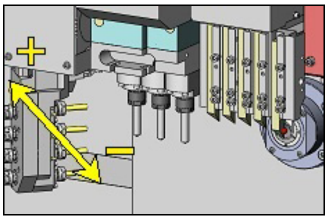
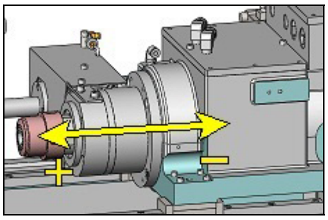
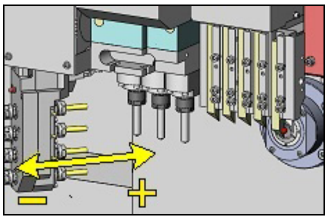
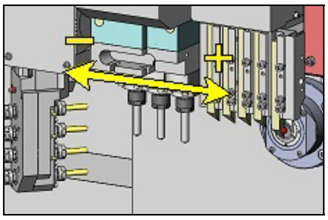
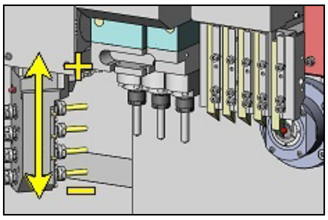
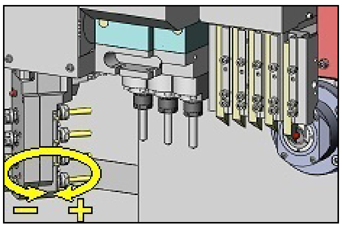
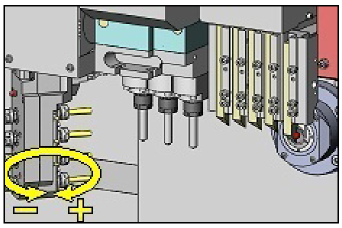
5. Turn the handle.

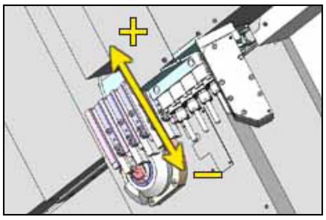
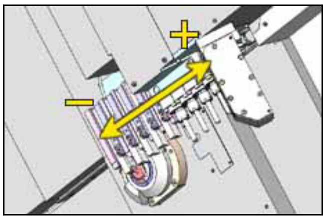
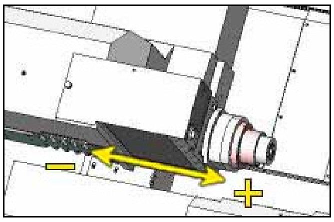
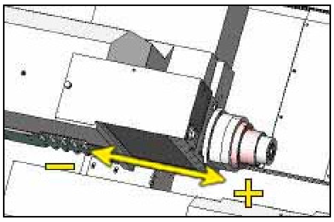
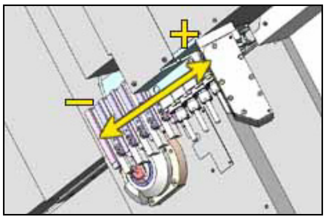
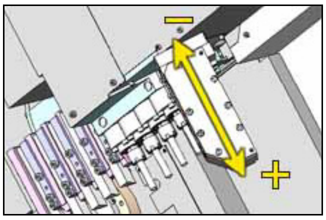
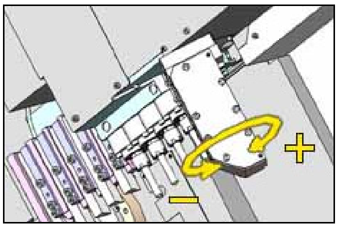
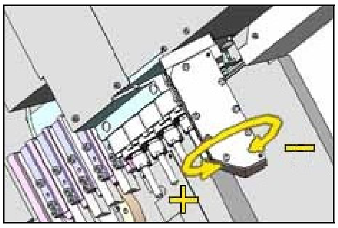
   The feed direction is determined by the direction the handle is turned.

   When a movement command that causes movement that exceeds stored stroke limit is issued, an overrun alarm indicating that "M01 S/W stroke limit axis detected" is generated.
   Refer to *<6.8.2 Recovering from overrun status>* for information on how to clear an overrun alarm.

Axis feed direction

Depending on what menu key [motor axis] is selected and the selected tool number, the direction of each axis is switched depending on whether the X1, Z1, Y1, Z2 or B1-axis is selected.

	Pattern 1	Pattern 2
Selected axis	○T0100 - T1000 tool (tool not related to the B-axis) is being selected when menu key [motor axis] is selected or not selected	○T11000 - T1400 B-axis tool is being selected when the menu key [motor axis] is not selected
X1	 Gang tool post moves in diameter direction.	 The B-axis tool moves in vertical B-axis angle direction (The Z1 and Y1 motor axes moves using interpolation).
Z1	 The headstock moves.	 The B-axis tool moves in B-axis angle direction (The Z1 and Y1 motor axes moves using interpolation).
Y1	 Gang tool post moves in core direction.	 The B-axis tool moves in core direction.
B1	 It moves in the same direction as handle feed.	 It moves in the same direction as handle feed.

	Pattern 1	Pattern 2
Selected axis	○T0100 - T1000 tool (tool not related to the B-axis) is being selected when menu key [motor axis] is selected or not selected	○T5100 - T5400 B-axis tool is being selected when the menu key [motor axis] is not selected
X1	 Gang tool post moves in diameter direction.	 Gang tool post moves in diameter direction.
Z2	 The back headstock moves.	 The back headstock moves.
Y1	 Gang tool post moves in core direction.	 The B-axis tool moves in core direction.
B1	 It moves in the same direction as handle feed.	 It moves in the same direction as handle feed.

[Note]

- Tool movement in B-axis rotation angle using handle feed along the Z1 and Z2 axes cannot be performed in the Mechanism Adjustment screen. The direction of movement in the Mechanism Adjustment screen is always in motor axis direction.
- The speed of a handle feed operation is limited to 1.8 m/min or less.

6.25.6 Zero return

Return the B-axis to zero.

[Procedure]

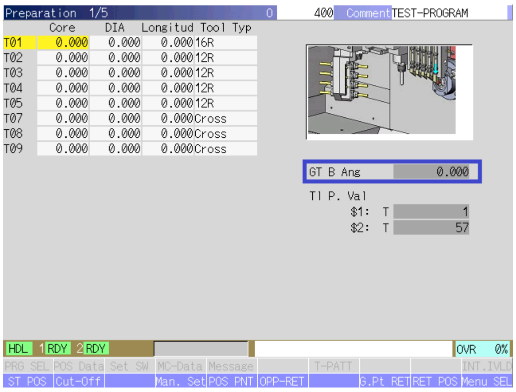
1. Press the Parameter key PRM..
2. Press the menu key [ZP EXE].
   The Zero Return screen appears.

3. Use the arrow keys to move the cursor to B1 and press the Input key INPUT.
   B1-axis is selected.
4. Make sure that all doors are closed, and press the Start button.
   The Start button lamp blinks on and off, and the doors will lock.
5. Press the Start button.
   The Start button lamp lights and the B1-axis starts to return to zero.
6. When zero return is completed, the Start button lamp goes off, Completed Zero Return is displayed and the B-axis tool stops at cross direction (B-axis 90 degrees) position.
7. The operation panel and the breaker are turned off.
8. Turn on the breaker and operation panel.

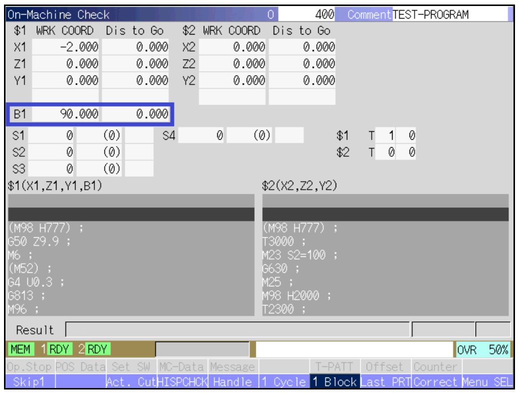
6.25.7 B-axis coordinate display

The B-axis coordinates appear on the Preparation screen, On-Machine Check screen, Automatic Operation screen, MDI screen, Handle Feed screen and the Position Data Screen.

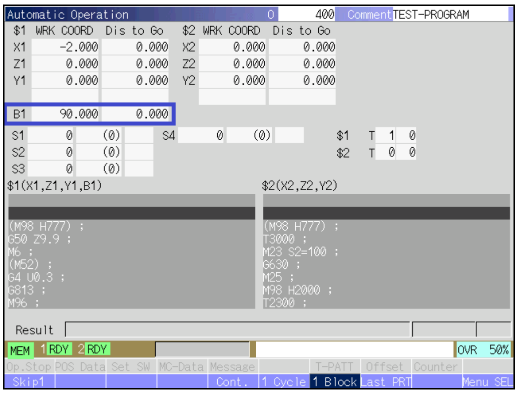
Preparation screen



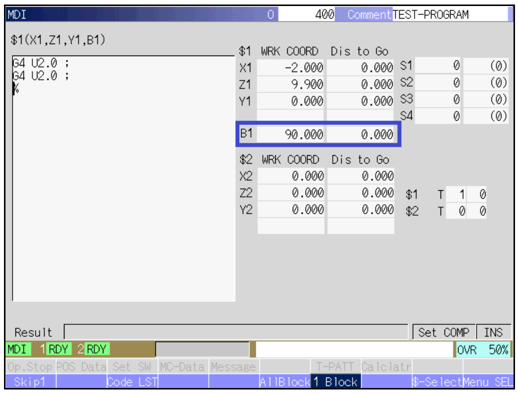
On-Machine Check screen



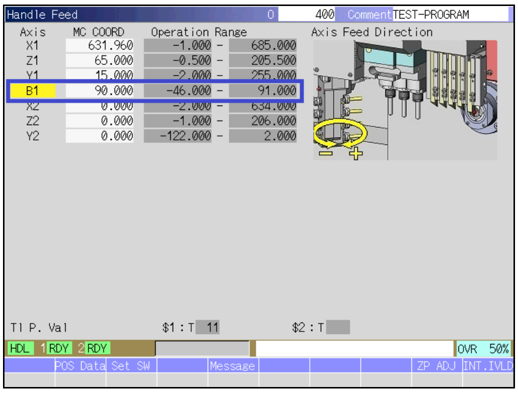
Automatic Operation screen



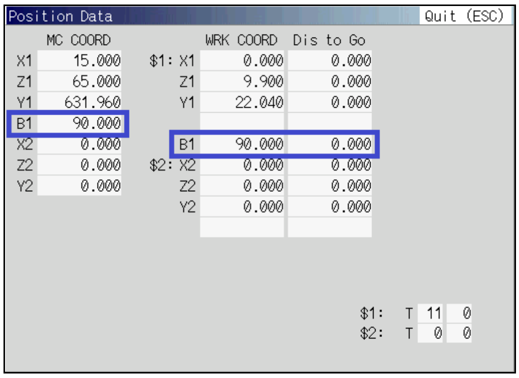
MDI screen



Handle Feed screen



Position Data Screen



(Blank Page)

Product Code	C-L220E XII
Document Code	1E1-0601
Mfg. No.	L220E/00001
Issue Date	2014.1

7. Mounting and Adjusting Chuck

7.1 Mounting and Adjusting Chuck

CAUTION

Described below are operation procedures when the Setting switch is set to the "I".
If you are going to open the door, press the Door open button to unlock the door.

The door cannot be unlocked unless the following conditions are met.

All spindles have stopped.

All servo axes have stopped.

The machine is not operating.

The coolant is not discharged:

The Door open button is pressed.

7.1.1 Mounting and replacing spindle chuck

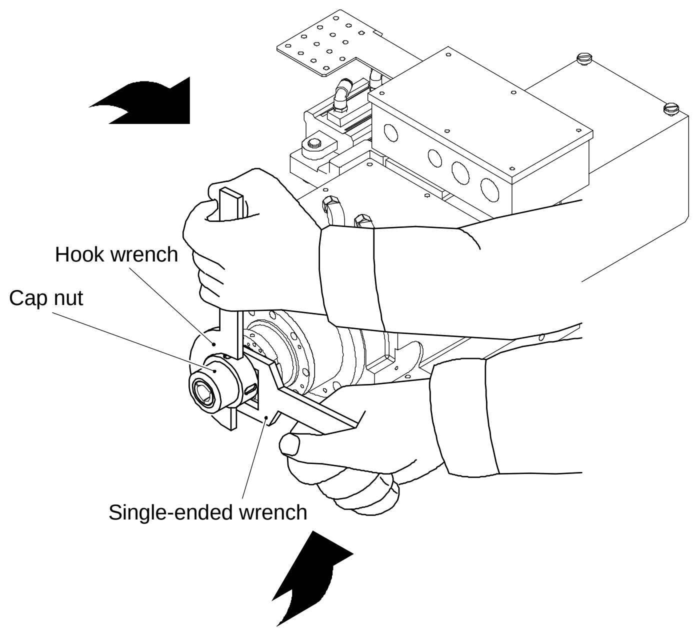
[Procedure]

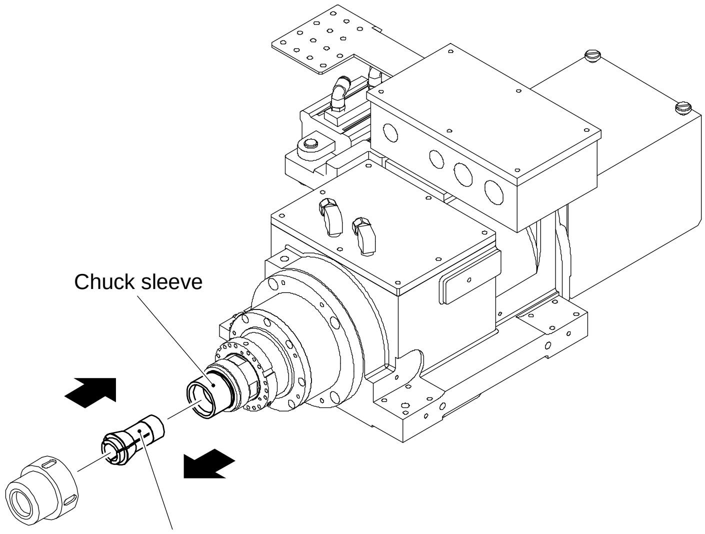
1. Use the Operation mode switch button on the operation panel to select Preparation .

   The Main spindle chuck key lamp goes off.
2. Place the hook wrench (provided as a standard tool) on the cap nut, and fit a single-ended wrench onto the two-side width (50 mm wide) of the spindle to loosen the cap nut (right-hand screw).
3. Turn the cap nut to remove the cap nut from the spindle.
4. Mount a new chuck on the chuck sleeve inserted into the spindle.

   Before mounting it, check the square spring is in the chuck sleeve.

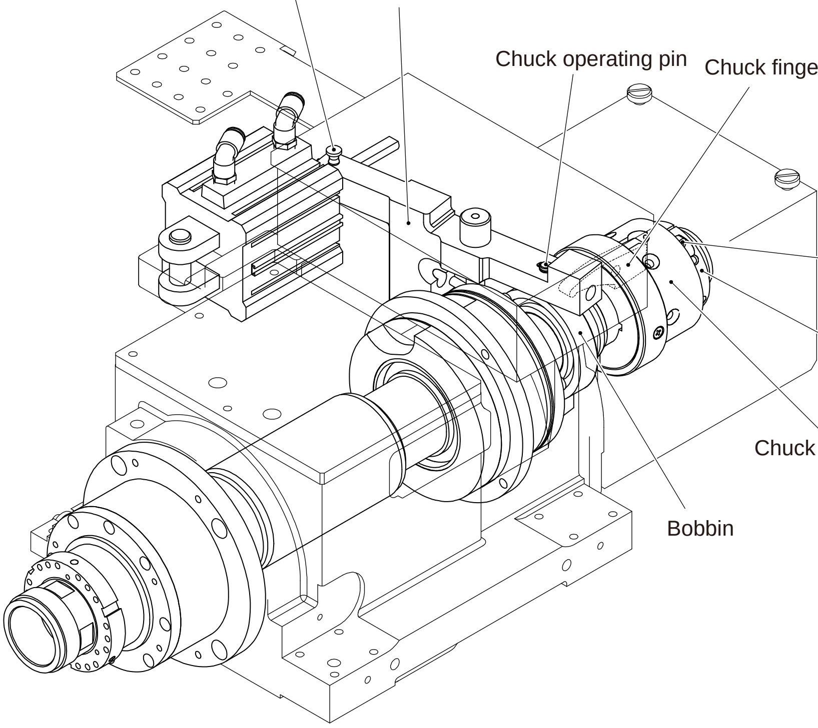
   If an old chuck is on the chuck sleeve, hold it and pull it out to this side, and mount a new one.
5. Screw the cap nut into the spindle, and firmly tighten it with the hook wrench and single-ended wrench.



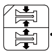


7.1.2 Adjusting spindle chucking force

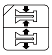
> **WARNING**
>
> Be sure to confirm that the spindles and machine have stopped before attempting to adjust the chucking force.
> The work during machine operation could result in serious personal injury.



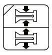
[Procedure]

1. Use the Operation mode switch button on the operation panel to select Preparation .

   The Preparation screen appears.

2. Press the Main spindle chuck button  on the operation panel to open the chuck. Then, insert the material into it.

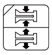
3. Loosen the lock screw of chuck adjustment nut located behind the chuck finger holder, and turn and loosen the chuck adjustment nut counterclockwise.

4. Press the Main spindle chuck button  on the operation panel to close the chuck.

   Check the Main spindle lamp goes on.

5. With the chuck adjustment nut being fully loose and chuck closed, slowly tighten the chuck adjustment nut until the chuck gets in touch with the material and starts to chuck it.

   Define the point as the nut angle zero point.

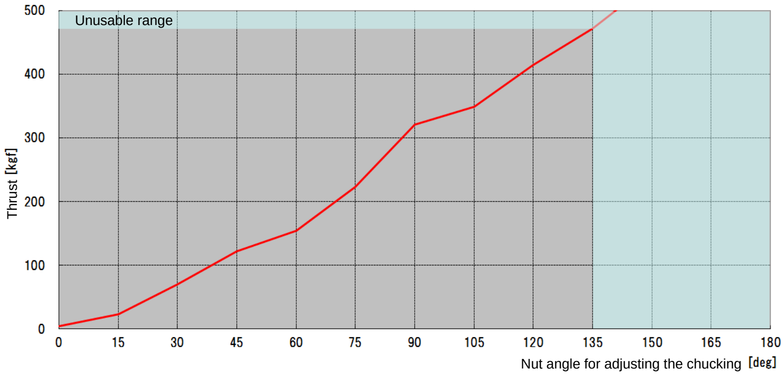
6. Press the Main spindle chuck button  on the operation panel to open the chuck.

   The Main spindle chuck button lamp goes off.

7. Pull out the connecting shaft that secures the chucking lever and the chuck open/close cylinder shaft.

   The connecting shaft is secured by a ball plunger.

8. From the nut angle zero point defined in Step 5, tighten the chuck adjustment nut by the specified angle according to the guideline shown in the figure below.



Relationship between adjustment nut rotating angle of main spindle and chuck thrust

The screw hole interval on the chuck finger holder side is 60°.

The figure shown above indicates the data when a Citizen authorized chuck is used. If a third party chuck is used, the force may not be equivalent to the value shown in the figure above.

9. Firmly insert the tommy rod, which is a standard accessory for the machine, in the tommy hole on the chucking lever and use it to move the chucking lever right and left to check chucking force.

   If chucking force is too large or too small, turn the chuck adjustment nut to adjust it.

10. When obtaining a suitable chucking force, set the chucking lever to place the spindle chuck in open state (the chuck finger is away from the bobbin).

11. Insert the connecting shaft in the chucking lever and the chuck open/close cylinder shaft until it clicks into place.

12. Tighten the lock screw of the chuck adjustment nut to prevent loosening. Then, remove the material.

CAUTION

Avoid empty chuck (chucking without bar material). Otherwise, malfunctioning or damage to the collet chuck may occur.

In the machining where higher cutting force is generated, set strong chucking force. If the chucking force is weak, the material may slip.
Note, however, do not set the value beyond the adaptable range shown in the graph on previous page. Doing so may cause damage to the machine.

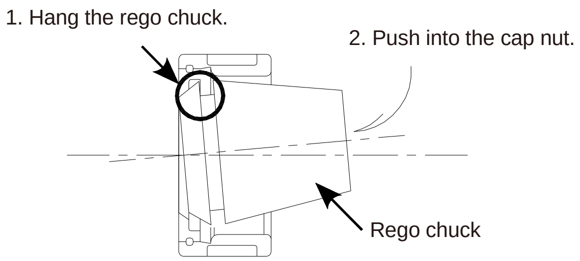
When chucking the thin material of which diameter is about 1 to 2 mm or pipe material, excess chucking force may deform the material. In such the case, readjust the chucking force.

7.1.3 Mounting rego chuck

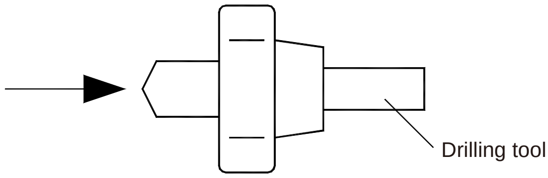
[Procedure]

1. Mount the rego chuck to the cap nut.

Wash the rego chuck cleanly and use the one free from scratches.

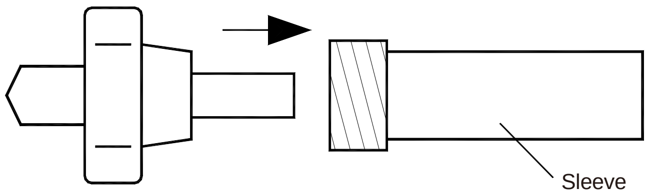


2. Insert a drilling tool into the rego chuck.



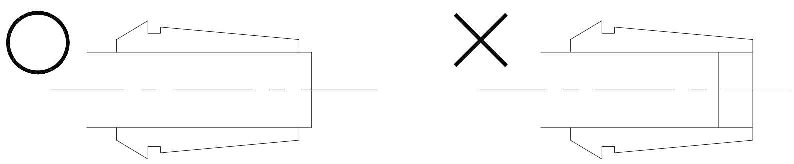
3. Screw the cap nut to which the tool was set into the sleeve thread.

Wash the sleeve inner tapered portion cleanly and check it is free from scratches.



4. Fit a dedicated wrench onto the two-side width of the sleeve and tighten the cap nut firmly.

> **CAUTION**
>
> Be sure to mount the rego chuck to the cap nut before inserting the drilling tool.
> The Inserted drilling tool length must always be longer than the overall length of the rego chuck.
> Otherwise, the tool and/or rego chuck may be damaged.

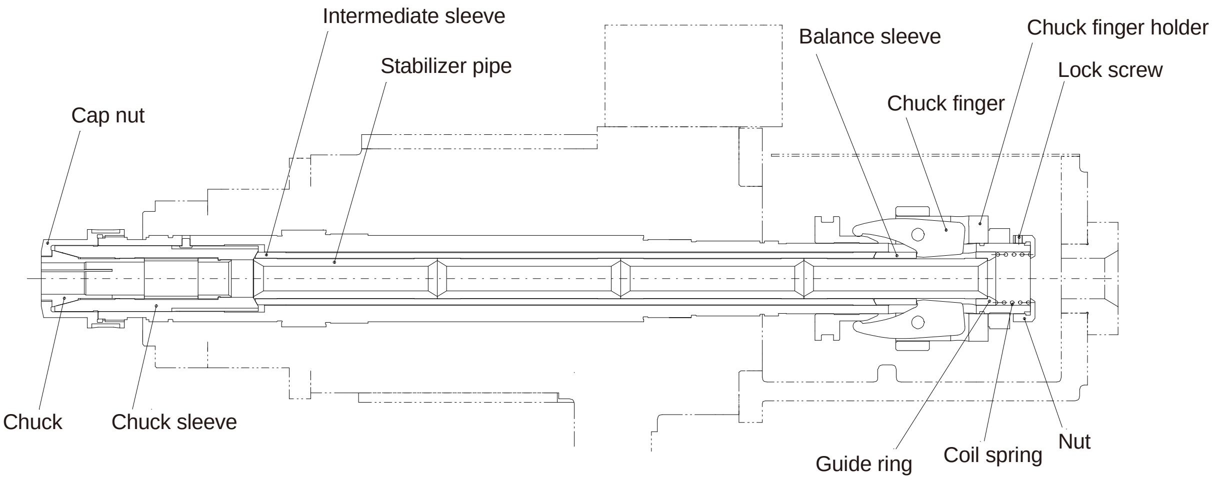


7.1.4 Mounting and replacing the spindle stabilizer pipe

(U10C, U100C to U123C)

In bar machining, the vibration due to the rotation of the workpiece greatly influences the machining accuracy.

To suppress the vibration due to rotation of the workpiece, use the stabilizer ring and stabilizer pipe of which outside diameter approximately +0.3 to 1.0 mm.



[Procedure]

1. Remove the main spindle chucking device (chuck and chuck sleeve) according to the procedures described in *<7.1.1 Mounting and replacing spindle chuck>*.

2. Loosen the lock nut at the rear of the spindle.
Insert a single-ended wrench (width: 50 mm) into the front end of the spindle, remove the nut, and then take off the coil spring and guide ring.

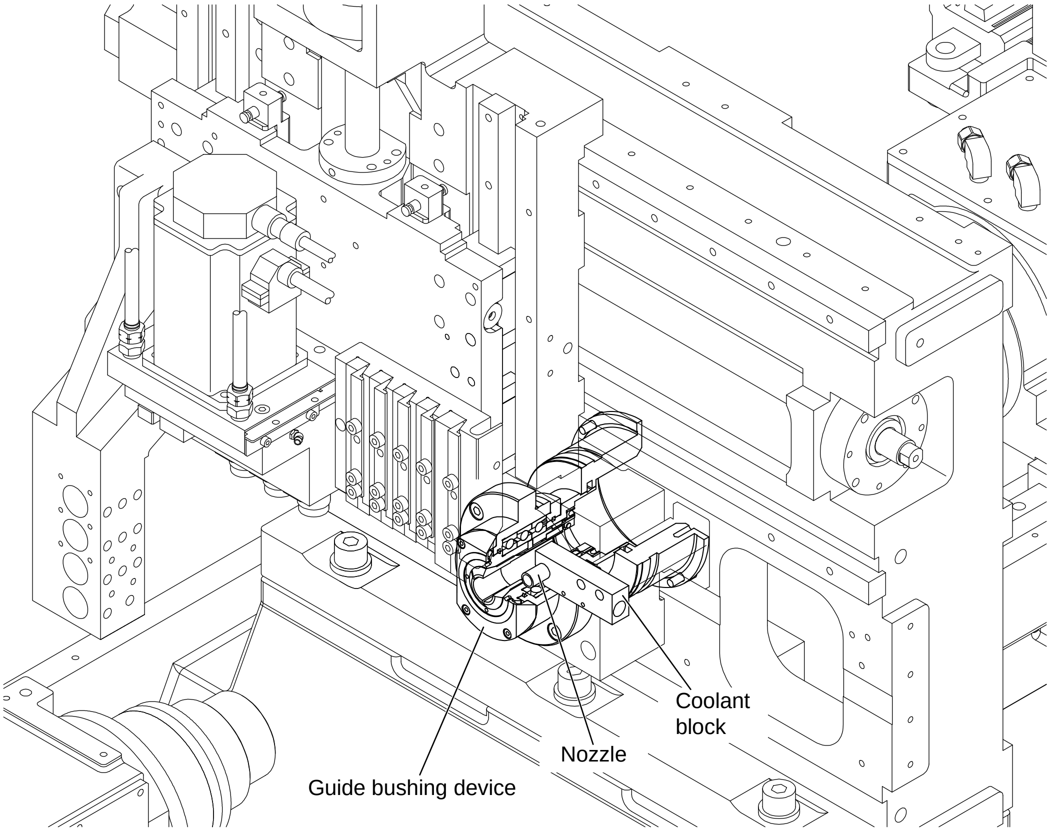
3. Insert a rod of suitable length from the rear end of the spindle, and push the four stabilizer pipes inserted in the spindle toward the front of the spindle.

4. Insert the four new stabilizer pipes one by one into the spindle from the front end of the spindle, and then mount the spindle chucking device.

5. Insert the guide ring and coil spring removed in Step 2 from the rear end of spindle, and tighten the lock nut.
Then, secure the lock screw of the nut.

7.2 Mounting/Adjusting a Guide Bushing Device

Two types of guide bushing devices are available: synchronous rotary guide bushing devices and fixed guide bushing devices (adjustable type) . Confirm the type of the mounted guide bushing device on the Machine structure screen. See <*Section 6.12 Machine Structure Screen*>.



7.2.1 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.

[Procedure]

1. Remove the three connecting block bolts.
2. Loosen the fixing bolt of guide bushing drive device and tension bolt to loosen enough the tension of belt.

   : Do not unfasten the retensioning belt. This bolt is used as reference for applying tension to the belt.
3. Remove the coolant block, the flange cover of the guide bushing device and the guide bushing device.
   Hold the guide bushing device by the flange unit and gently pull it out toward the front.

   When you remove the guide bushing device, the air outflows from the air pipe hole and the sound is heard. But it is not a failure.
4. Insert the new guide bushing device into the mounting hole while hanging belt, and secure it with fixing bolt. At this time, pay attention so that the belt is not crushed by the guide bushing device.
5. Install the coolant block.
6. Tighten the tension bolt until the bracket of guide bushing drive device touches the retensioning bolt, then secure the guide bushing device temporarily with two fixing bolts.

   The belt tension is suitable at this position.

7. Make sure that the tension of belts on the timing pulleys at the spindle side and the guide bushing device side are same. Turn the spindle clockwise and counterclockwise by hand to make sure that the guide bushing driving device rotates along with the spindle rotation.
Then, firmly tighten the bolts.
8. Secure the connection block to the plate.
9. Make sure that the spline nut moves along with the spindle rotation (parallelism between the spindle and the spline shaft) by moving the Z1 axis forward and backward.
10. Make sure that the guide bushing drive device does not interfere with the spline nut at the forward end position of Z1 axis.
11. Mount the flange cover of guide bushing device from the spindle room side.

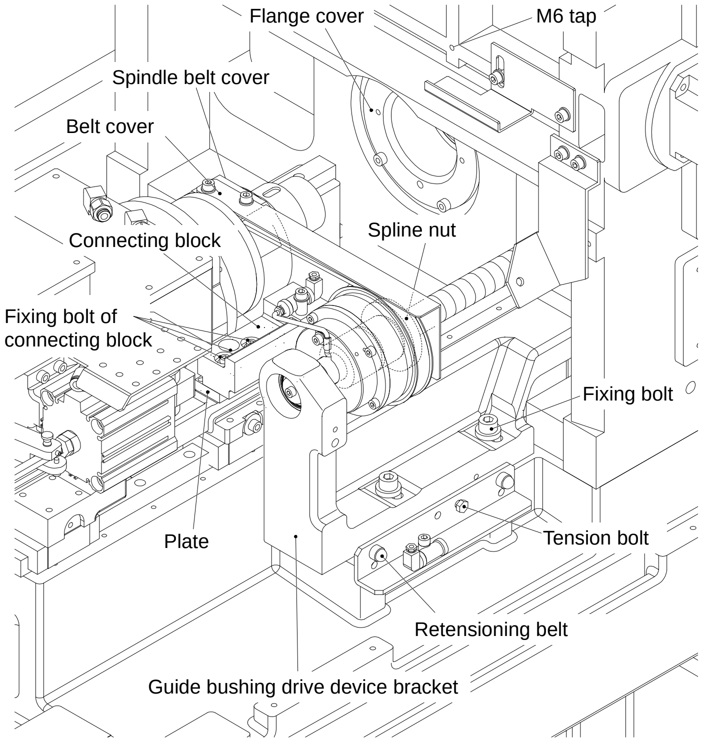


Figure1 Guide bushing drive device

7.2.2 Replacing fixed type guide bushing device

The fixed type guide bushing device is an optional device.

To use the fixed type guide bushing device, you need to remove the belt from the counter shaft. See *<7.2.3 Replacing synchronous rotary guide bushing device with fixed guide bushing device>* for the procedure.

> **CAUTION**
>
> Be sure to remove the belt at the guide bushing side when using the fixed type guide bushing device.
> The belt will skid when the spindle rotates, and it may cause a damage to the machine.

[Procedure]

1. Remove the coolant block, the flange cover of the guide bushing device and the guide bushing device.
   Hold the guide bushing device by the flange unit and gently pull it out toward the front.

   When you remove the guide bushing device, the air outflows from the air pipe hole and the sound is heard. But it is not a failure.
2. Insert the new guide bushing device into the mounting hole while hanging belt, and secure it with fixing bolt.
3. Install the coolant block.
4. Mount the flange cover of guide bushing device from the spindle room side.

7.2.3 Replacing synchronous rotary guide bushing device with fixed guide bushing device

[Procedure]

1. Move the Z1 axis to backward end position.
See <*6.2.1 Return position*>.

2. Remove the three connecting block bolts.

3. Remove the spindle belt cover of GB specification.

4. Loosen the fixing bolt of guide bushing drive device bracket and tension bolt to loosen enough the tension of belt.

: Do not unfasten the retensioning belt. This bolt is used as reference for applying tension to the belt.

5. Remove the coolant block and flange cover of guide bushing device, and pull out the guide bushing device toward the front carefully while holding the flange.

When you remove the guide bushing device, the air outflows from the air pipe hole and the sound is heard. But it is not a failure.

6. Open belt post cover at the cutting room side, and remove the belt for driving the guide bushing device. Close the belt cover after removing the belt.

7. Insert the new guide bushing device of fixed type into the mounting hole, and secure it with fixing bolt.

8. Install the coolant block.

9. Mount the flange cover of guide bushing device from the spindle room side.

10. Remove the belt from the spindle pulley and move the connecting block and spline nut assembly to the right.
Move the connection block upwards and secure it to the block for connecting the connection block as shown in the figure 2.
At this time, place the timing belt on the belt hook.

11. Loosen the two fixing bolts of the belt hook and secure it in a location where the belt turns with a minimum of slack.

12. Tighten the tension bolt until the bracket of guide bushing drive device contacts with the retensioning bolt, then secure the guide bushing device with four fixing bolts.

13. Mount the spindle belt cover of GB specification.

> **CAUTION**
>
> Be sure to pull the bracket of guide bushing drive device to the operator's side and fix it.
>
> Otherwise, the guide bushing drive device interferes with headstock when Z1 axis moves forward.

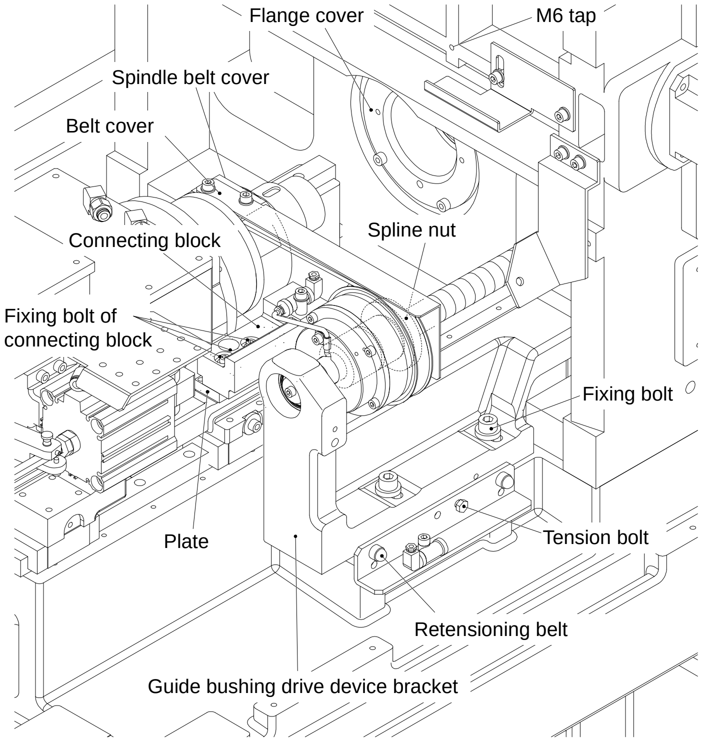


Figure 1 Rotary Guide Bushing

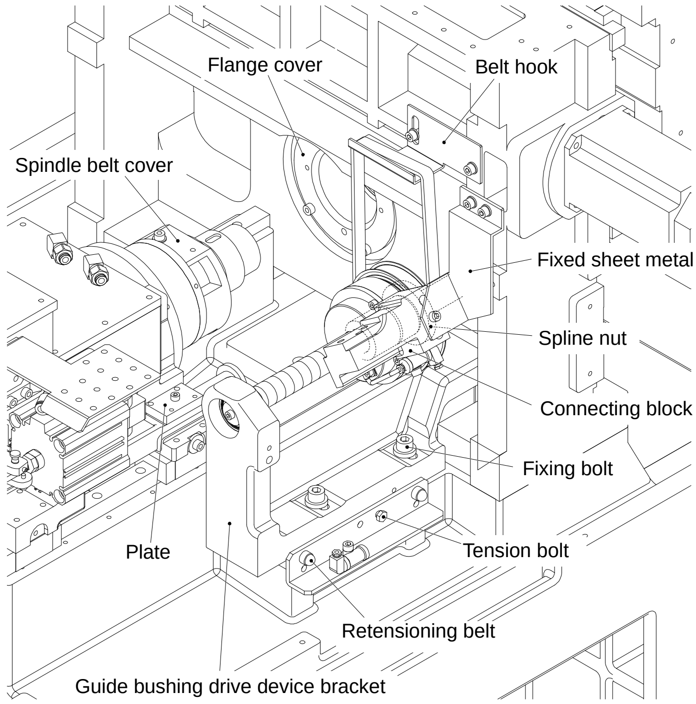


Figure 2 Fixed Guide Bushing

7.2.4 Replacing fixed guide bushing device with synchronous rotary guide bushing device

[Procedure]

1. Move the Z1 axis to backward end position.

   See *<6.2.1 Return position>*.

2. Remove the spindle belt cover of GB specification.
3. Remove the bolts that secure the plate for securing the connection block and the connection block.
4. Fully loosen the fixing bolt and tension bolt of guide bushing drive device bracket, and move the drive device to the spindle side.

   : Do not unfasten the retensioning belt. This bolt is used as reference for applying tension to the belt.

5. Pull out the connection block and spline nut assembly to the left side and place the belt on the spindle pulley.
6. Remove the coolant block, the flange cover of the guide bushing device and the guide bushing device.
   Hold the guide bushing device by the flange unit and gently pull it out toward the front.

   When you remove the guide bushing device, the air outflows from the air pipe hole and the sound is heard. But it is not a failure.

7. Open the belt cover of the cutting room and place the belt inside the guide bushing base.
8. Insert the replacing guide bushing device of rotary type into the mounting hole while hanging the belt onto the pulley, and firmly secure it with fixing bolt. At this time, pay attention so that the belt is not crushed by the guide bushing device.
9. Install the coolant block.
10. Tighten the tension bolt until the bracket of guide bushing drive device comes into contact with the retensioning bolt and stops. Then secure the guide bushing device temporarily with two fixing bolts.

    The belt tension is suitable at this position.

11. Make sure that the tension of belts on the timing pulleys at the spindle side and the guide bushing device side are same. Turn the spindle clockwise and counterclockwise by hand to make sure that the guide bushing driving device rotates along with the spindle rotation. Then, firmly tighten the bolts.
12. Install the spindle belt cover.
13. Secure the connection block to the plate.
14. Make sure that the spline nut moves along with the spindle rotation (parallelism between the spindle and the spline shaft) by moving the Z1 axis forward and backward.
15. Make sure that the guide bushing drive device does not interfere with the spline nut at the forward end position of Z1 axis.
16. Mount the flange cover of synchronous rotary guide bushing device from the spindle room side.
17. Close the tool post cover at the front of cutting room.

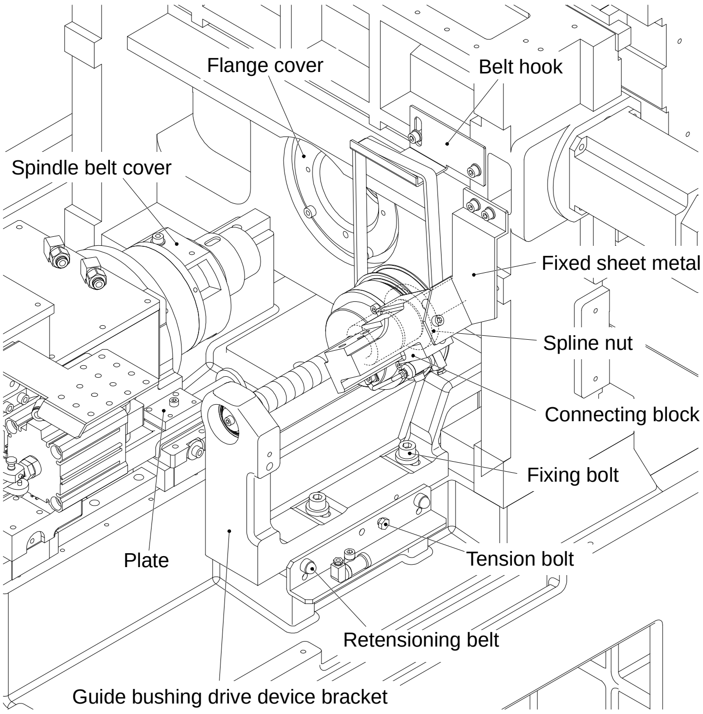


Figure 1 Fixed Guide Bushing

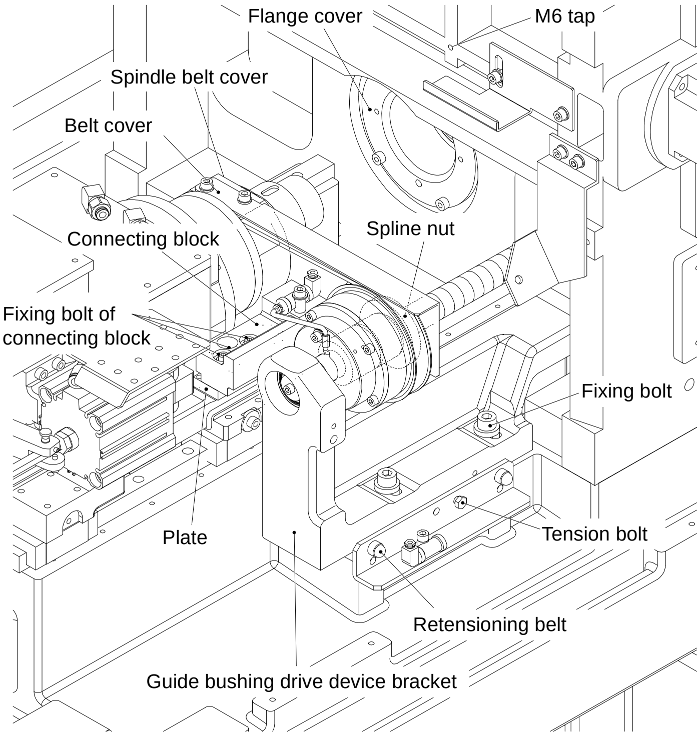


Figure 2 Rotary Guide Bushing

7.3 Replacing and Adjusting Guide Bushing

There are two types of guide bushing devices: synchronized rotary guide bushing and fixed guide bushing devices. Be sure to refer to the machine configuration screen to check the type of the guide bushing device installed on your machine. See *<6.12 Machine Structure Screen>*.

7.3.1 Replacing and adjusting the guide bushing (synchronized rotary type)

[Procedure]

1. Insert the guide bushing adjustment wrench (provided as a standard tool) into the tommy hole of the drawbar, which is mounted at the rear of the guide bushing device, and loosen the drawbar clamp screw with a hexagonal rod wrench.

   The rod wrench should be attached to the guide bushing adjustment wrench before the work.

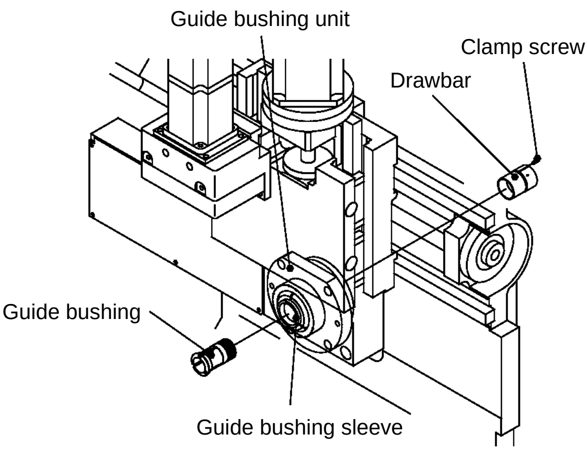


Figure 1 Guide bushing

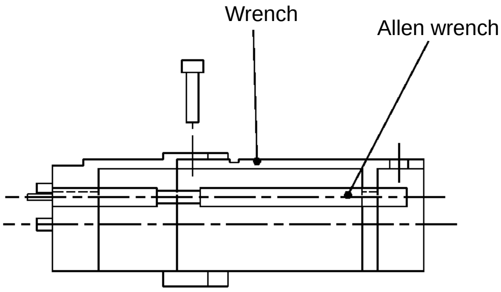


Figure 2 Wrench used for drawbar

2. Turn the guide bushing adjustment wrench counterclockwise to loosen the drawbar. To prevent the guide bushing device from turning, insert the Y wrench (provided as standard tool) into the tommy hole of the guide bushing sleeve.
3. Pull out the guide bushing toward the front of the device.
4. Insert a new guide bushing into the guide bushing sleeve.
   Align the key in the sleeve with the key groove on the outer circumference of the guide bushing.
5. Screw the drawbar into the guide bushing rear and turn the drawbar clockwise for tightening.
6. Using the guide bushing adjustment wrench, tighten the drawbar. Insert a bar material into the guide bushing and adjust the clearance between guide bushing and bar material while moving the bar material to the axis direction.
7. With the guide bushing adjustment wrench inserted into the drawbar, tighten the clamp screw of the drawbar with a hexagonal rod wrench.
   Be sure to cut-off or remove the bar material.

WARNING

After the work, be sure to remove the used tool from the guide bushing.

If the machine is operated without removing the tool, the machine may seriously be damaged.

7.3.2 Replacing and adjusting the guide bushing (fixed adjustable type)

[Procedure]

1. Insert the wrench for adjusting the guide bushing (provided as a standard accessory tool) into the tommy hole in the draw bar at the rear of the guide bushing device, and loosen the clamp screw of the draw bar with an Allen wrench.
Fit the Allen wrench into the guide bushing adjustment wrench in advance.

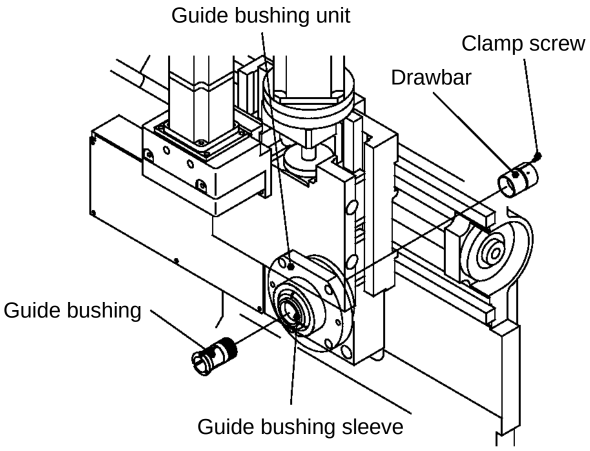


Figure 1 Guide bushing

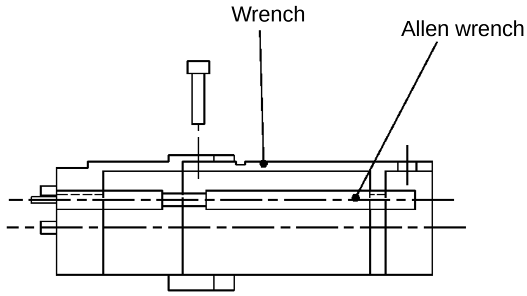


Figure 2 Wrench used for drawbar

2. Turn the wrench for adjusting the guide bushing counterclockwise to loosen the draw bar.

3. Pull the guide bushing from the machine to the front of the unit to remove it.

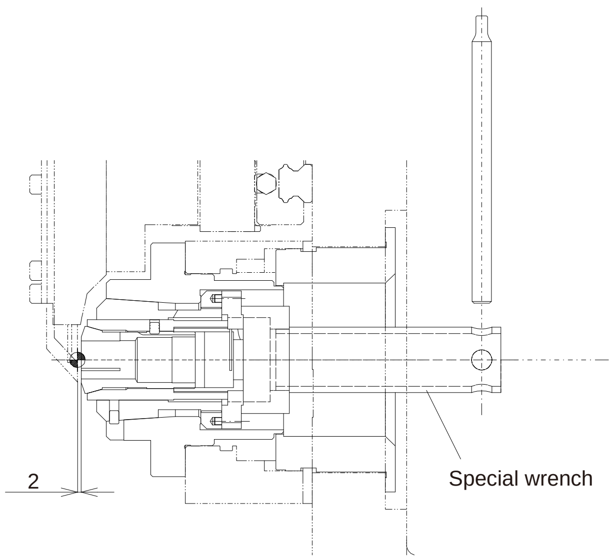
4. Put the new guide bushing into the guide bushing sleeve, aligning the groove outer circumference in the guide bushing with the key in the sleeve.

5. Screw the drawbar back on the rear of the guide bushing. Rotate the nut clockwise to tighten it.

6. Use the guide bushing adjustment wrench to tighten the drawbar. Lead the bar material into the guide bushing and move it in the direction of the shaft to adjust the clearance of the guide bushing with the bar material.

7. With the guide bushing adjustment wrench inserted in the tommy hole in the drawbar, turn the Allen wrench to tighten the clamp screw of the drawbar.

8. Insert the pin of special wrench into the tommy hole of large-diameter nut located at the rear side of guide bushing device (at spindle side), and rotate it counterclockwise to loosen the device.



9. Adjust the guide bushing sleeve position by moving the sleeve in the axial direction, while checking the clearance between the tool and the opening of the guide bushing.

10. Firmly tighten the large-diameter nut with the special wrench (an accessory tool provided with the machine). Either cut off or remove the material.

> **WARNING**
>
> Be sure to remove all tools used for adjustment from the guide bushing device.
> Operating the machine with any of the tools left on the guide bushing device may severely damage the machine.

7.4 Switching over between GB-specification and GBL-specification

7.4.1 Switching from GB-specification (synchronous rotary guide bushing) to GBL-specification

[Procedure]

1. Move the Z1 axis to backward end position.
   See *<6.2.1 Return position>*.
2. Remove the three connecting block.
3. Remove the spindle belt cover of GB specification.
4. Loosen the fixing bolt of guide bushing drive device and tension bolt to loosen enough the tension of belt.
   : Do not unfasten the retensioning belt. This bolt is used as reference for applying tension to the belt.
5. Remove the coolant block, the flange cover of the guide bushing device and the guide bushing device.
   Hold the guide bushing device by the flange unit and gently pull it out toward the front.
6. When you remove the guide bushing device, the air outflows from the air pipe hole. Block the hole with the hexagon socket set screw (M6 x 10).
7. Open the tool post cover at the cutting room side, and remove the belt for driving the guide bushing device.
   Close the tool post cover after removing the belt.
8. Install the coolant block.
9. Remove the belt from the spindle pulley and move the connecting block and spline nut assembly to the right.
   Move the connection block upwards and secure it to the block for connecting the connection block as shown in the figure 2.
   At this time, place the timing belt on the belt hook.
10. Loosen the two fixing bolts of the belt hook and secure it in a location where the belt turns with a minimum of slack.
11. Tighten the tension bolt until the bracket of guide bushing drive device contacts with the retensioning bolt, then secure the guide bushing device with two fixing bolts.
12. Install the spindle cap with the one without guide bushing specifications.
13. Press the Parameter key PRM..
    The previously selected screen appears.
14. Press the menu key [Menu SEL] until the menu key [MC-STRCT] is displayed.
15. Press the menu key [MC-STRCT].
    The Machine Structure screen appears.
16. Use the Arrow keys ⬆⬇, Tab keys TAB TAB, or Page key to move the cursor to "GBL Specification".
17. Press the Menu selection key ▶ to make an upper menu active, and press the menu key [Set].
18. Press the Input key INPUT to determine the selection.
19. Turn off the NC power, and then on.

[Note]

If you change setting, turn off the NC power. The new setting is made valid at next power-on.

CAUTION

Be sure to pull the bracket of guide bushing drive device to the operator's side and fix it.

Otherwise, the guide bushing drive device interferes with headstock when Z1 axis moves forward.

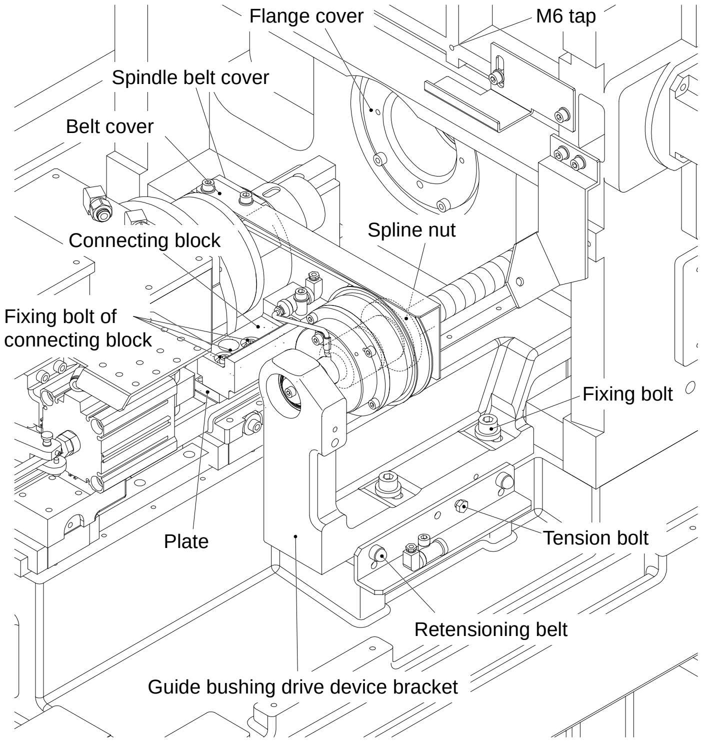


Figure 1 Rotary Guide Bushing

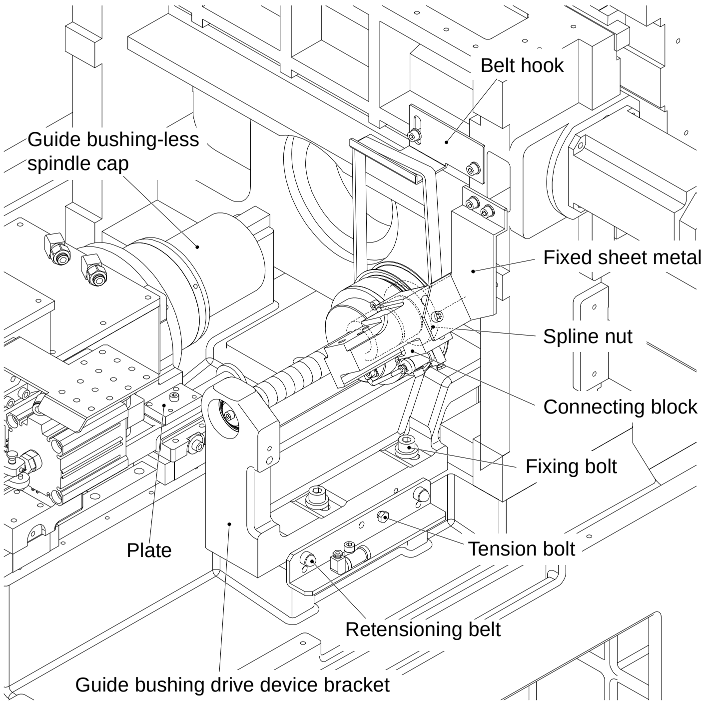


Figure 2 Guide bushing-less

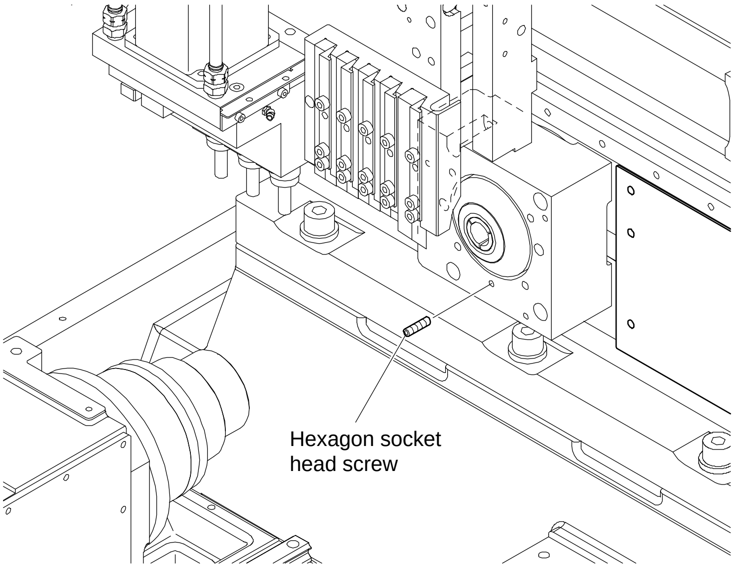


Figure 3 Guide bushing-less (Installing head screw)

7.4.2 Switching from GBL-specification to GB-specification (synchronous rotary guide bushing)

[Procedure]

1. Move the Z1 axis to backward end position.

See *<6.2.1 Return position>*.

2. Press the Parameter key .

The previously selected screen appears.

3. Press the menu key [Menu SEL] until the menu key [MC-STRCT] is displayed.

4. Press the menu key [MC-STRCT].
The Machine Structure screen appears.

5. Use the Arrow keys , Tab keys , or Page key to move the cursor to "Synchronized R.G.B Device".

6. Press the Menu selection key to make an upper menu active, and press the menu key [Set].

7. Press the Input key to determine the selection.

8. Turn off the NC power, and then on.

9. Remove the spindle cap of GBL-specification.

10. Remove the bolts that secure the plate for securing the connection block and the connection block.

11. Fully loosen the fixing bolt and tension bolt of guide bushing drive device bracket, and move the drive device to the spindle side.

: Do not unfasten the retensioning belt. This bolt is used as reference for applying tension to the belt.

12. Pull out the connection block and spline nut assembly from the left side and place the belt on the spindle pulley.

13. Open the belt cover of the cutting room and place the belt inside the guide bushing base.

14. Remove the socket set screws (M6 x 10) in the guide bushing mounting surface on the cutting room side.

When you remove the socket set screws, the air outflows from the air pipe hole. Pay attention not to drop the screw. You can store the socket set screws by screwing it to the M6 tap hole of belt hook left at the rear of tool post.

15. Remove the coolant block.

16. Insert the replacing guide bushing device of rotary type into the mounting hole while hanging the belt onto the pulley, and firmly secure it with fixing bolt.

Pay attention so that the belt is not crushed by the guide bushing device.

17. Install the coolant block.

18. Tighten the tension bolt until the bracket of guide bushing drive device comes into contact with the retensioning bolt and stops. Then secure the guide bushing device temporarily with two fixing bolts. The belt tension is suitable at this position.

19. Make sure that the tension of belts on the timing pulleys at the spindle side and the guide bushing device side are same. Turn the spindle clockwise and counterclockwise by hand to make sure that the guide bushing driving device rotates along with the spindle rotation. Then, firmly tighten the bolts.

20. Mount the spindle belt cover.

21. Secure the connection block to the plate.

22. Make sure that the spline nut moves along with the spindle rotation (parallelism between the spindle and the spline shaft) by moving the Z1 axis forward and backward.

23. Make sure that the guide bushing drive device does not interfere with the spline nut at the forward end position of Z1 axis.

24. Mount the flange cover of synchronous rotary guide bushing device from the spindle room side.

25. Close the belt cover at the front of cutting room.

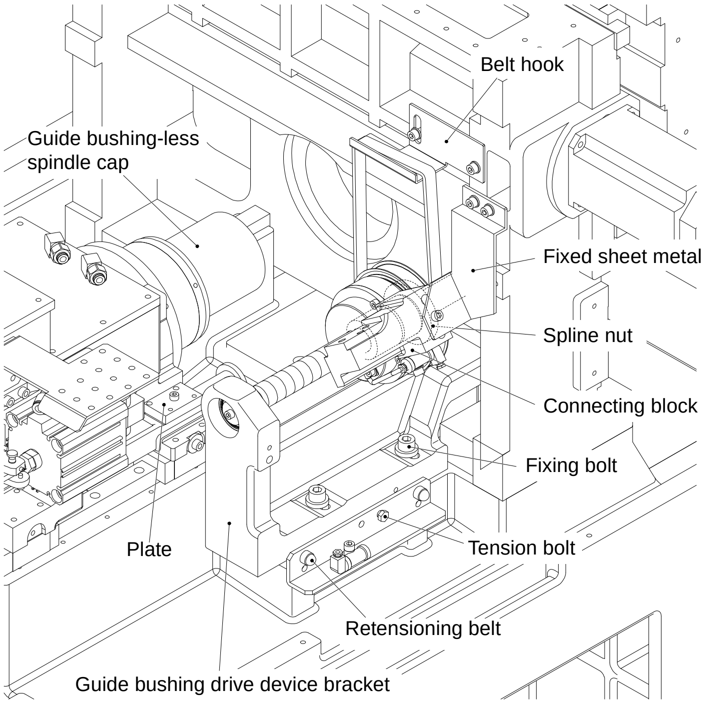


Figure 1 Guide Bushing-less

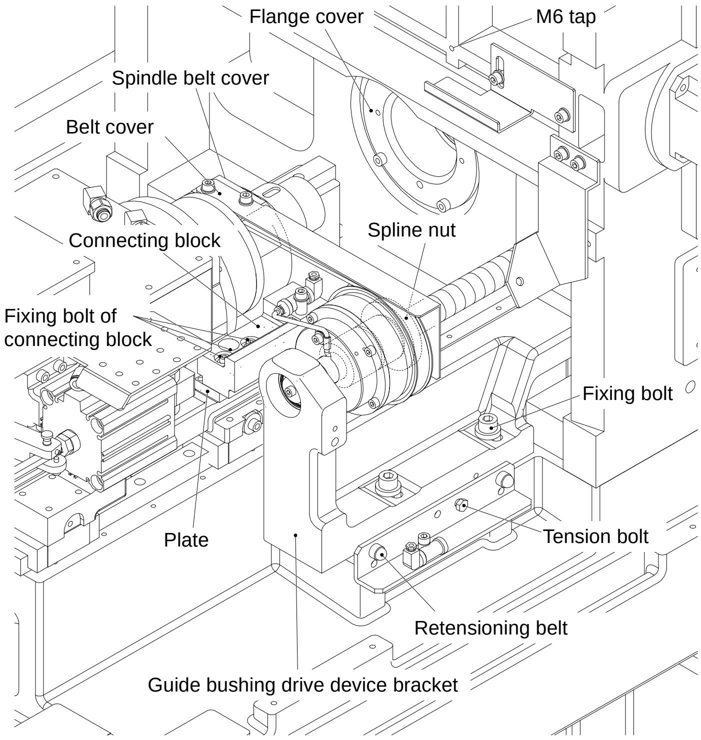


Figure 2 Rotary Guide Bushing

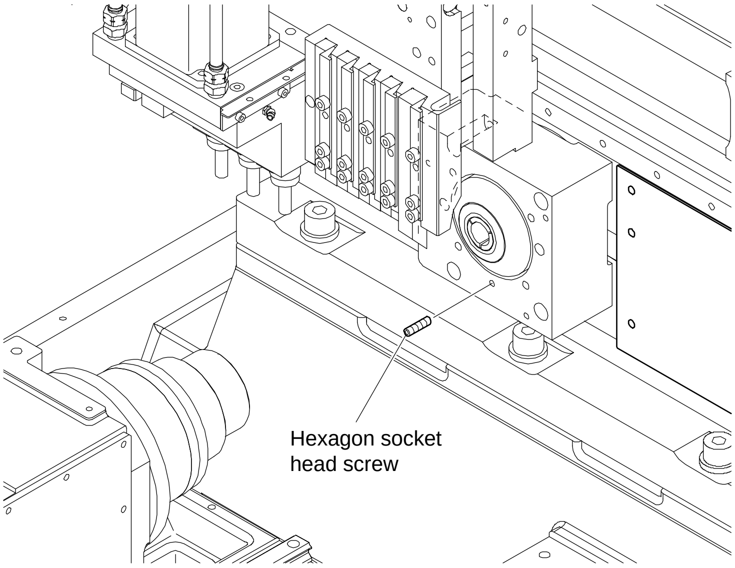


Figure 3 Fixed Guide Bushing (Removing head screw)

7.4.3 Switching from GB-specification (fixed guide bushing) to GBL-specification

[Procedure]

1. Move the Z1 axis to backward end position.

See *<6.2.1 Return position>*.

2. Remove the spindle belt cover for the guide bushing specifications.

3. Remove the coolant block beside the guide bushing, the flange cover of the guide bushing device and the guide bushing device.
Hold the guide bushing device by the flange unit and gently pull it out toward the front.

4. When you remove the guide bushing device, the air outflows from the air pipe hole. Block the hole with the hexagon socket set screw (M6 x 10).

5. Install the coolant block.

6. Press the Parameter key [PRM.].

The previously selected screen appears.

7. Press the menu key [Menu SEL] until the menu key [MC-STRCT] is displayed.

8. Press the menu key [MC-STRCT].
The Machine Structure screen appears.

9. Use the Arrow keys [↑] [↓], Tab keys [TAB ←] [TAB →], or Page key [↑] [↓] to move the cursor to "GBL Specification".

10. Press the Menu selection key [▶] to make an upper menu active, and press the menu key [Set].

11. Press the Input key [INPUT] to determine the selection.

12. Turn off the NC power, and then on.

[Note]

If you change setting, turn off the NC power. The new setting is made valid at next power-on.

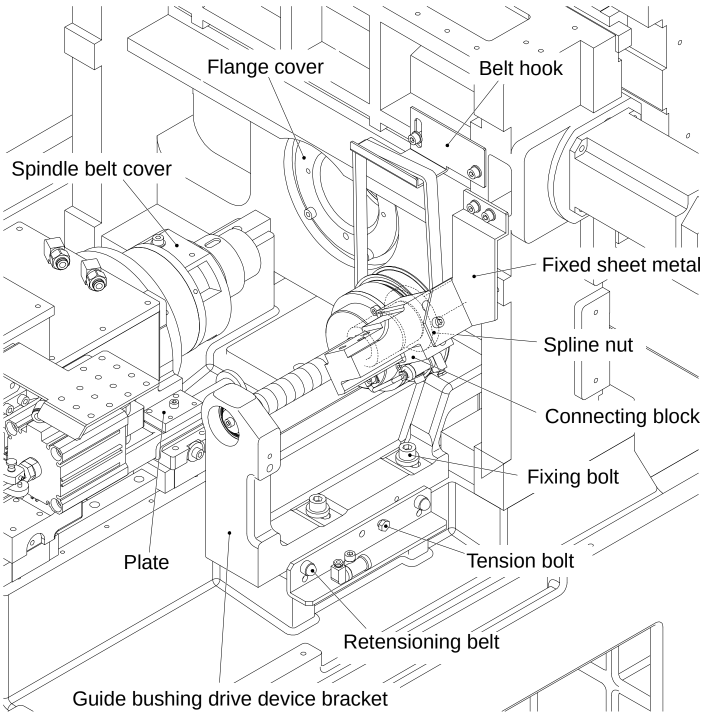


Figure 1 Fixed Guide Bushing

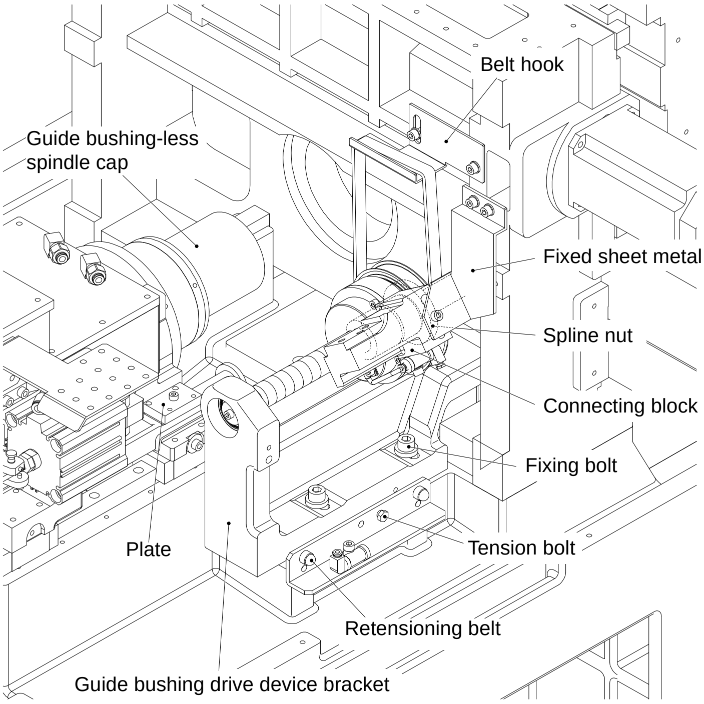


Figure 2 Guide Bushing-less

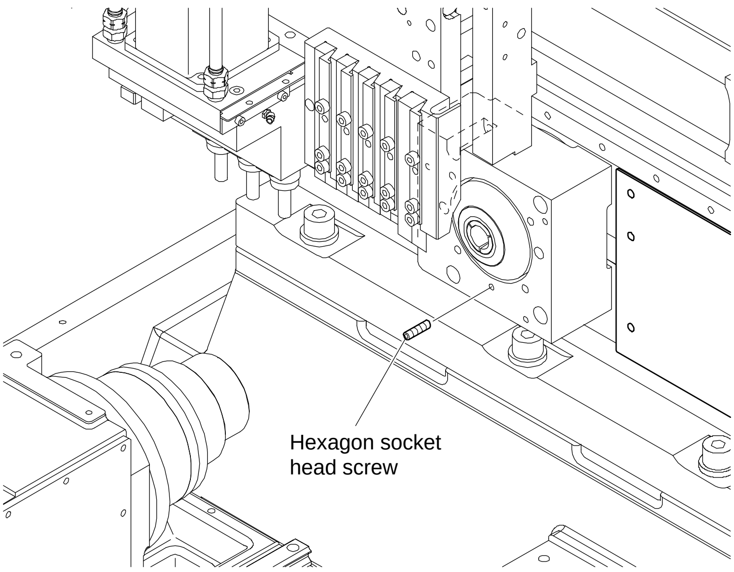


Figure 3 Guide Bushing-less (Installing head screw)

7.4.4 Switching from GBL-specification to GB-specification (fixed guide bushing)

[Procedure]

1. Move the Z1 axis to backward end position.

   See *<6.2.1 Return position>*.
2. Press the Parameter key PRM..

   The previously selected screen appears.
3. Press the menu key [Menu SEL] until the menu key [MC-STRCT] is displayed.
4. Press the menu key [MC-STRCT].
   The Machine Structure screen appears.
5. Use the Arrow keys ⬆ ⬇, Tab keys TAB ⬅ TAB ➡, or Page key to move the cursor to "Fixed Guide Bush Device".
6. Press the Menu selection key ▶ to make an upper menu active, and press the menu key [Set].
7. Press the Input key INPUT to determine the selection.
8. Turn off the NC power, and then on.
9. Remove the spindle cap of GBL-specification.
10. Remove the socket set screws (M6x10) in the guide bushing mounting surface on the cutting room side.

    When you remove the socket set screws device, the air outflows from the air pipe hole. Pay attention not to drop the screws. You can store the socket set screws by screwing it to the M6 tap hole of belt hook left at the rear of tool post.
11. Remove the coolant block.
12. Insert the replacing guide bushing device of fixed type into the mounting hole, and firmly secure it with fixing bolt.
13. Install the coolant block.
14. Mount the flange cover of the guide bushing device from the spindle room side.
15. Mount the spindle cap of GB specification and spindle belt cover to the spindle.

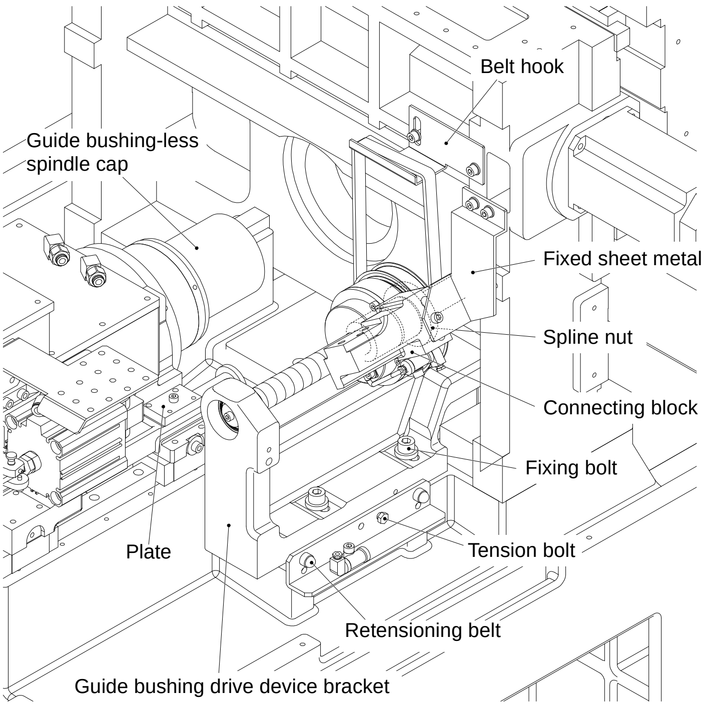


Figure 1 Guide Bushing-less

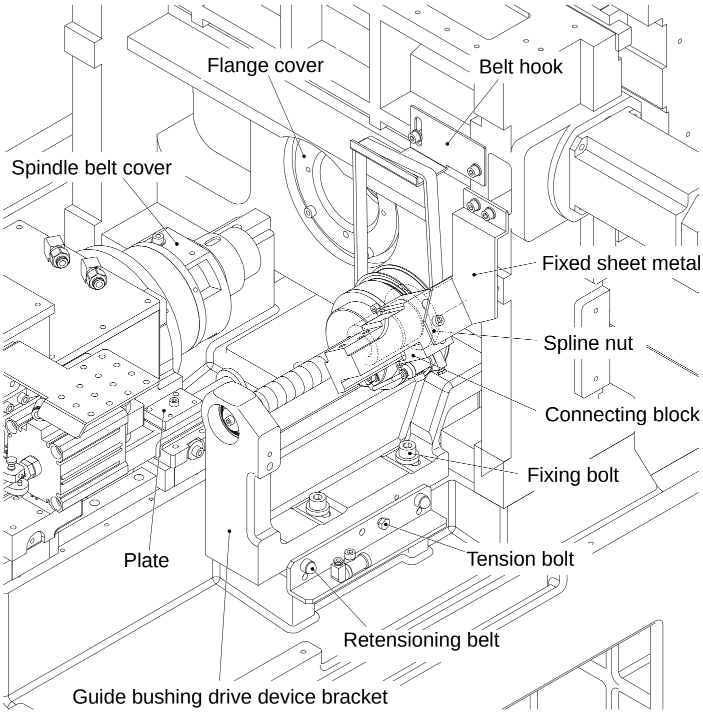


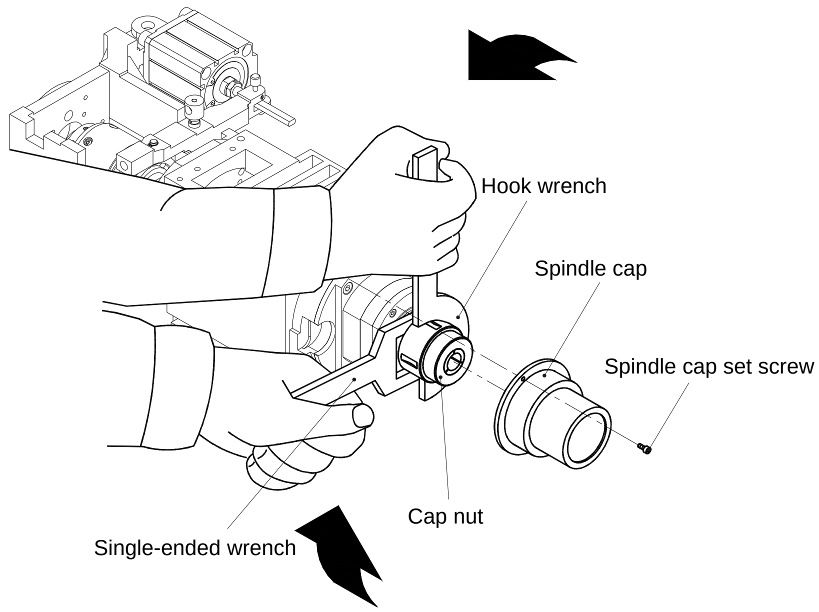
Figure 2 Fixed Guide Bushing

7.5 Rearrangement of Back Spindle Device

7.5.1 Mounting and adjusting chuck

[Procedure]

1. Loosen the spindle cap set screws and remove the spindle cap.



2. Place the hook wrench (provided as a standard tool) on the cap nut, fit a single-ended wrench onto the two-side width (width:50 mm) of the spindle, and loosen the cap nut.

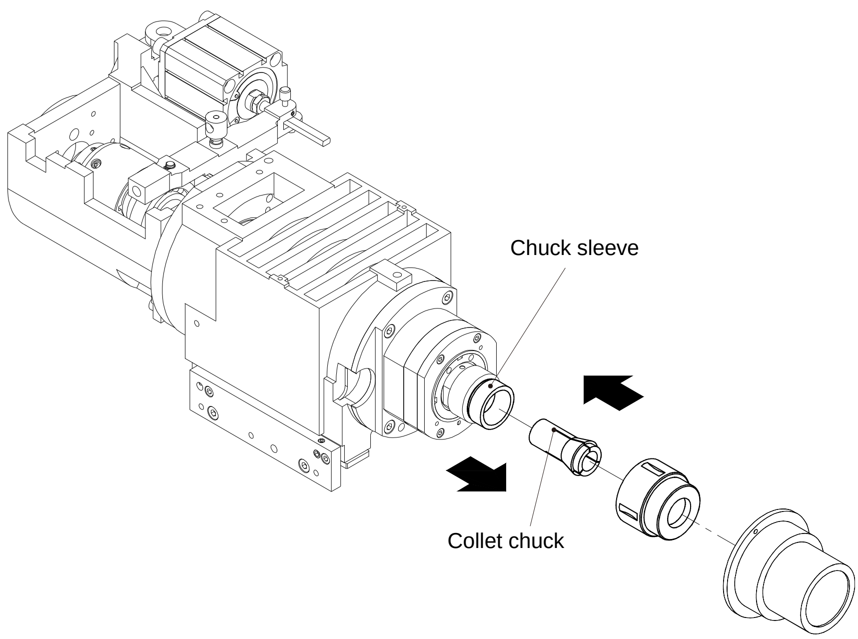
: The cap nut is right hand screw.

3. Turn the cap nut to remove it from the spindle.

4. Mount a new chuck on the chuck sleeve inserted into the spindle.

: Check that the square spring is in the chuck sleeve.

If an old chuck is on the chuck sleeve, hold it and pull it out to this side, and mount the new one.



5. Screw the cap nut into the spindle, and firmly tighten it with the hook wrench and single-ended wrench.

6. Finally, firmly secure the spindle cap with the set screws.

Mount so that the oil drain hole of the chuck faces downwards.

> **CAUTION**
>
> If the oil drain hole of the spindle cap does not face straight downward, oil enters into the bearing to cause possible damage. Be sure to mount the cap so that the oil drain hole faces straight downward.

7.5.2 Adjusting back spindle chucking force

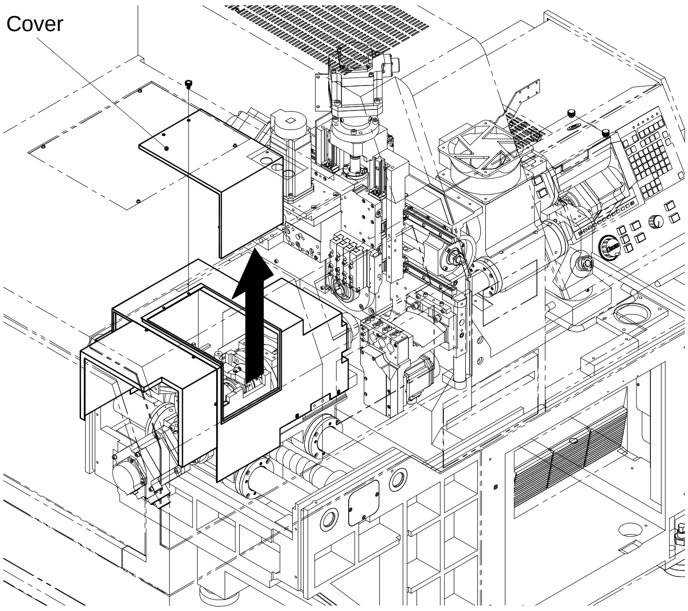
[Procedure]

1. Use the Operation mode switch key on the operation panel to select preparation PREPA..

The Preparation screen appears.

2. Remove the cover.

3. Press the Back spindle chuck key on the operation panel to open the chuck. Then, insert the material into the chuck.



4. Loosen the lock screw of chuck adjustment nut located behind the chuck finger holder, and turn and loosen the chuck adjustment nut counterclockwise.

5. Press the Back spindle chuck button on the operation panel to close the chuck.

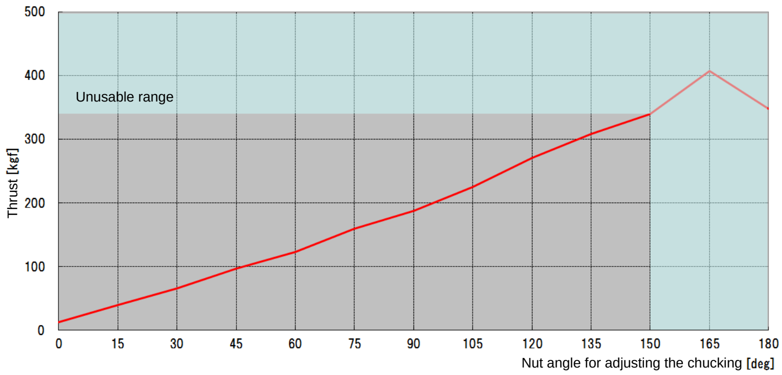
: Check the Back spindle chuck key lamp goes on.

6. With the adjusting nut sufficiently loose, close the chuck, tighten the adjusting nut so that the chuck comes into contact with the material and starts holding on to it.
With the chuck in this state, set nut angle to 0 degrees.

7. Press the back spindle chuck button to open the chuck.
The back spindle chuck button goes off.

8. Pull out the connecting shaft that secures the chucking lever and the chuck open/close cylinder shaft.
The connecting shaft is secured with a plunger.

9. Tighten the chuck adjustment nut from the 0 degree nut angle confirmed in step 6 to the required angle as shown in the figure.



Relationship between adjustment nut rotating angle of back spindle and chuck thrust

The screw hole interval on the chuck finger holder side is 60°.

The figure shown above indicates the data when a Citizen authorized chuck is used. If a third party chuck is used, the force may not be equivalent to the value shown in the figure above.

10. Firmly insert the tommy rod, which is a standard accessory for the machine, in the tommy hole on the chucking lever and use it to move the chucking lever right and left to check chucking force.
If chucking force is too large or too small, turn the chuck adjustment nut to adjust it.

11. When the appropriate chucking force is obtained, move the chucking lever to chuck open (when the chuck finger is not in contact with the bobbin).

12. Insert the connecting shaft in the chucking lever and the chuck open/close cylinder shaft until it clicks into place.

13. Tighten the lock screw of the chuck adjustment nut to prevent it from coming loose and losing grip of material.

CAUTION

Avoid empty chuck (chucking without bar material). Otherwise, malfunctioning or damage to the collet chuck may occur.

In the machining where the large cutting force is applied, set the chucking force rather strong. If the chucking force is insufficient, the material will be slipped.
Note, however, the chucking force exceeding the usable range shown in the graph above may cause a damage to the machine.

When chucking the thin material of which diameter is about 1 to 2 mm or pipe material, excess chucking force may deform the material. In such the case, readjust the chucking force.

7.6 Adjusting knock-out device

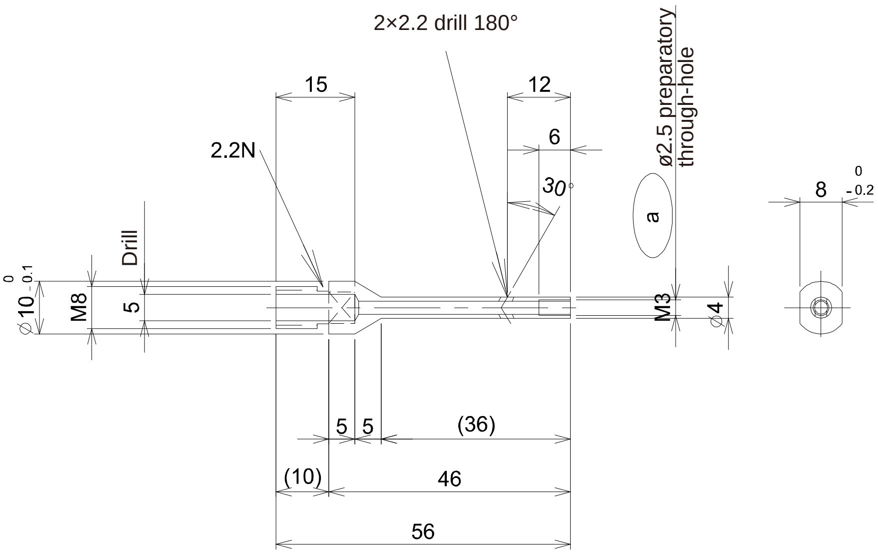
The knock-out device unloads the workpiece from the back spindle. The knock-out device is also used as an air blower in the back spindle device. The device is equipped with a safety unit so that an alarm occurs to stop it if the workpiece is not unloaded appropriately in spite of specification of workpiece unloading. A knock-out jig appropriate to the diameter of the workpiece must be used. Two knock-out jigs are provided as standard accessories for the device. If these knock-out jigs are not appropriate to the diameter of the machined workpiece, the customer must manufacture a proper knock-out jig.

7.6.1 Mounting and replacing knock-out jig

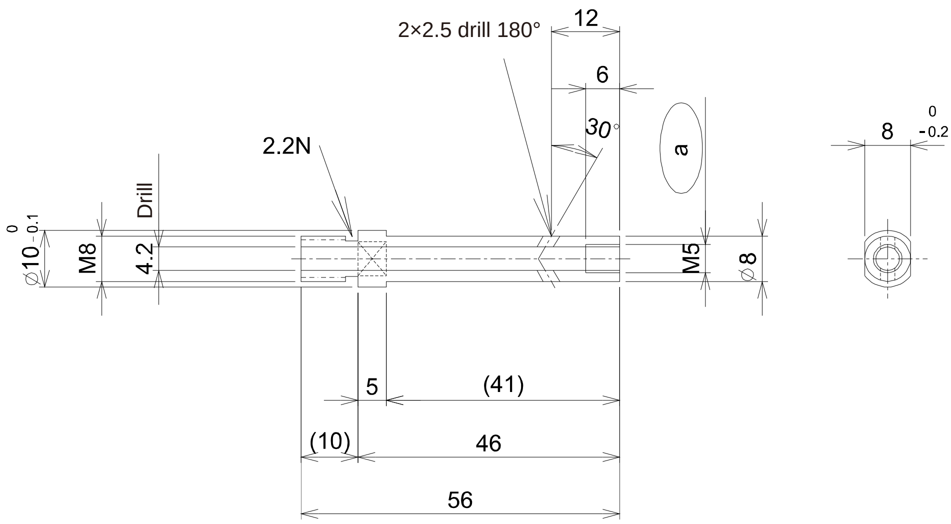
[Procedure]

1. Advance the knock-out device to allow you to see the knock-out jig through the front-end of the back spindle cap.
2. Stop the machine and remove the back spindle cap.
3. Remove the cap nut, chuck and chuck sleeve.
4. Loosen the bolt fixing the knock-out pipe, and pull the pipe toward front of the machine.
   The knock-out pipe and the knock-out jig extrude from the front end of the spindle.
5. Remove the knock-out jig by using the dedicated wrench, which is an accessory for the machine.

> **CAUTION**
>
> When replacing the knock-out jig, pay attention not to apply excess force to the knock-out pipe. Deformed knock-out pipe will fail to knock-out the workpiece correctly, and it causes a damage to the machine.



ø4 knock-out jig



ø8 knock-out jig

6. Mount a new knock-out jig appropriate to the workpiece diameter on the pipe.

7. Insert the chuck sleeve and the chuck and tighten the cap nut.

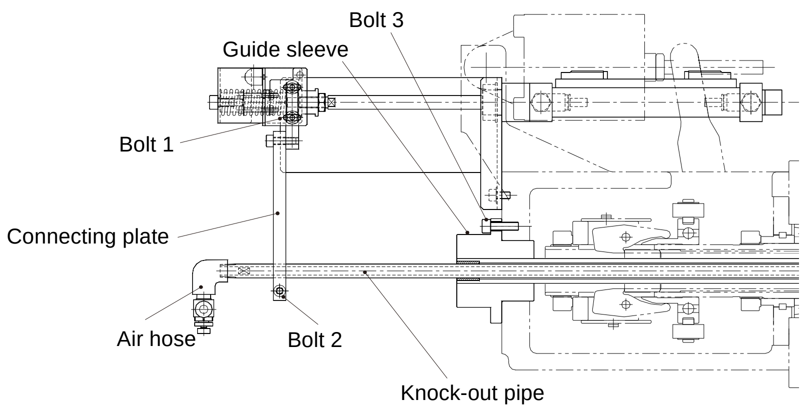
8. Adjust the length of the knock-out pipe so that the knock-out jig extends 4 mm beyond the front end of the cap nut and tighten the bolt.

See *<Section 7.6.2 Adjusting protrusion amount of knock-out jig>*.

9. Mount the back spindle cap.

10. Adjust the chucking force of back spindle.

See *<Section 7.5.2 Adjusting back spindle chucking force>*.



7.6.2 Adjusting protrusion amount of knock-out jig

Adjust the knock-out jig protrusion amount from the end face of the back cap nut.

[Procedure]

1. Stop the machine and remove back spindle cover.
2. Make the knock-out device advanced.
3. Loosen the bolt, and adjust the position of knock-out pipe appropriately.

   Adjust the knock-out jig protrusion amount from the front end of the cap. Set the protrusion amount to 4 mm.
4. Tighten the bolt to fix the knock-out pipe.
5. Advance and retract the knock-out device several times to make sure that the bolt is not loosened and the knock-out jig protrusion amount does not change.
6. Mount back spindle cover.

[Note]

When the "KNOCK-OUT OVERLOAD" alarm occurs after the adjustment, the sensor position must be readjusted. See *<Section 7.6.3 Resetting knock-out overload alarm>*.

7.6.3 Resetting knock-out overload alarm

If the knock-out operation fails, the workpiece may be left in the back spindle chuck. The sensor mounted on the knock-out device detects error status to prevent the machine from being damaged if the knock-out cylinder does not make normal stroke or the knock-out pipe is pushed backward by larger than the predetermined value.

Reset the "KNOCK OUT OVERLOAD" alarm in the following procedures.

[Procedure]

1. Remove the back spindle cap and the cap nut. Then, remove the workpiece from the chuck.
2. Make sure that the knock-out pipe and the knock-out jig are not damaged.

> **CAUTION**
>
> If the knock-out pipe is seriously damaged (e.g., bent or deformed), replace the knock-out pipe with new one. With the bent knock-out pipe will fail the knock-out operation, and causes a damage to the machine.

3. Move the knock-out device to the retract position.
4. Press the Reset key 

to cancel "KNOCK OUT OVERLOAD" alarm.

   If pressing the Reset key does not reset the alarm, the sensor must be adjusted again. If the alarm still appears after the adjustment of the sensor, see *<23.4.13 Restoration from Other Alarms>* in *<Chapter 23 Troubleshooting>* of the Maintenance Manual.
5. Mount the chuck, cap nut and the back spindle cap.
6. Make sure that the knock-out jig is protruding from the front-end of the cap nut by 4 mm.
7. Advance and retract the knock-out device several times to make sure that the alarm does not occur.
8. Adjust the chucking force of back spindle.

   See *<Section 7.5.2 Adjusting back spindle chucking force>*.

7.7 Rearrangement of through-hole workpiece

7.7.1 Mounting knock-out device for through-hole workpiece knock-out device

[Procedure]

1. Remove the knock-out jig from the knock-out pipe.

: You need not to remove the knock-out pipe.

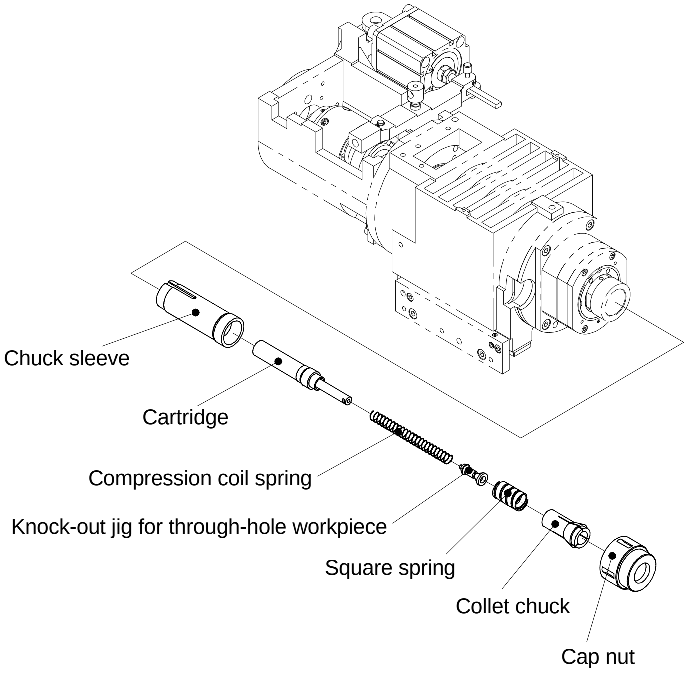
See *<Section 7.6.1 Mounting and replacing knock-out jig>*.

2. Pull out the collet chuck, chuck sleeve, square spring, and collar.

See *<Section 7.5.1 Mounting and adjusting chuck>*.

3. Insert the assembly of the cartridge, sleeve, compression coil spring and through-hole workpiece knock-out jig (to be referred to as a cartridge, hereafter) into the chuck sleeve.

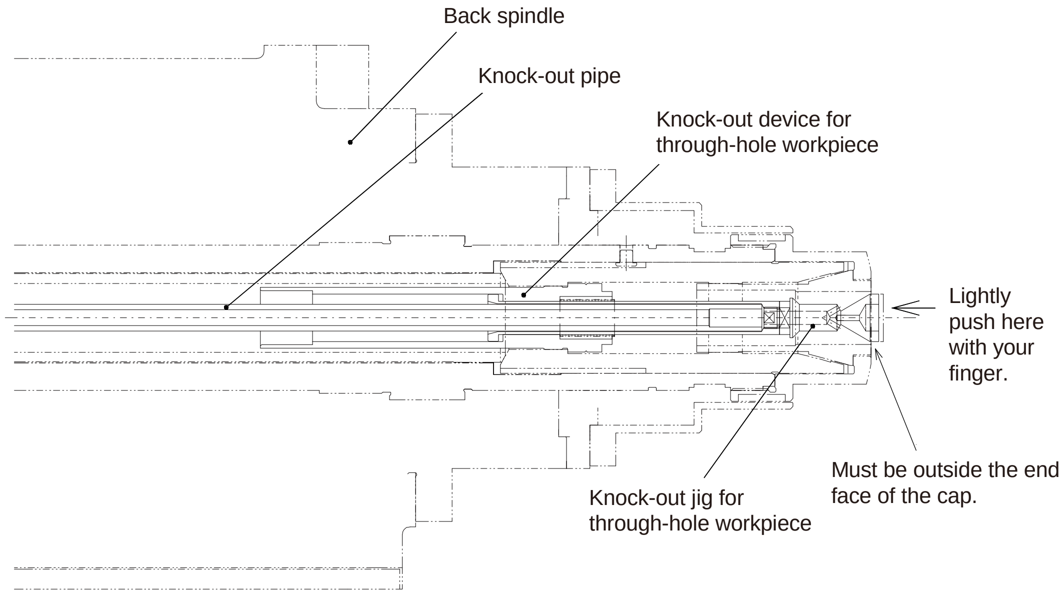
4. Next, insert the square spring, and collet chuck into the sleeve in this order.



5. Insert the chuck sleeve partially assembled in Step 3 to 4 into the back spindle, and mount the cap nut and spindle cap.

See *<Section 7.5.1 Mounting and adjusting chuck>*.

> **CAUTION**
>
> Do not use the standard knock-out jig to mount the through-hole workpiece knock-out device.
> The machine may be damaged by interference with the knock-out jig.



6. Move the knock-out device forward and backward to make sure that nothing engages with the knock-out pipe.

7. Advance the knock-out device, and lightly push the tip of the jig with your finger to make sure that the jig does not go any further from the cap end face.
If the jig goes into the cap, it is necessary to adjust the knock-out device.

See *<Section 7.6 Adjusting knock-out device>*.

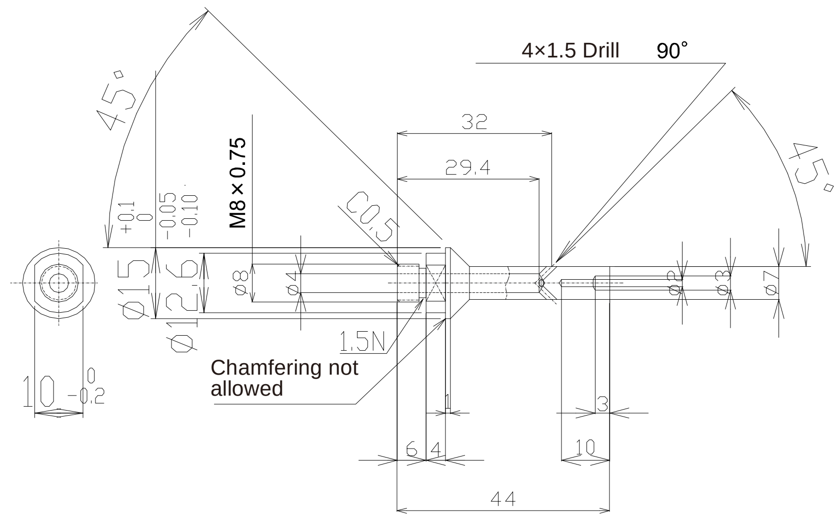
8. Adjust the chucking force of back spindle.

See *<Section 7.5.2 Adjusting back spindle chucking force>*.

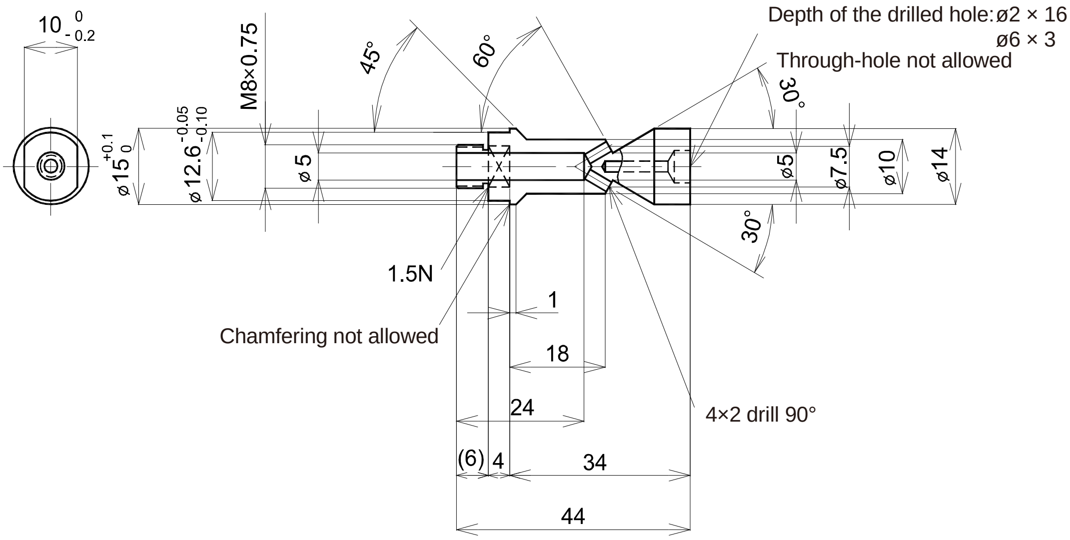
7.7.2 Replacing the knock-out jig for through-hole workpieces

You need to replace the knock-out jig depending on the workpiece profile (chuck diameter and through-hole diameter). The following four types of knock-out jigs for through-hole workpieces and a blank knock-out jig are available.

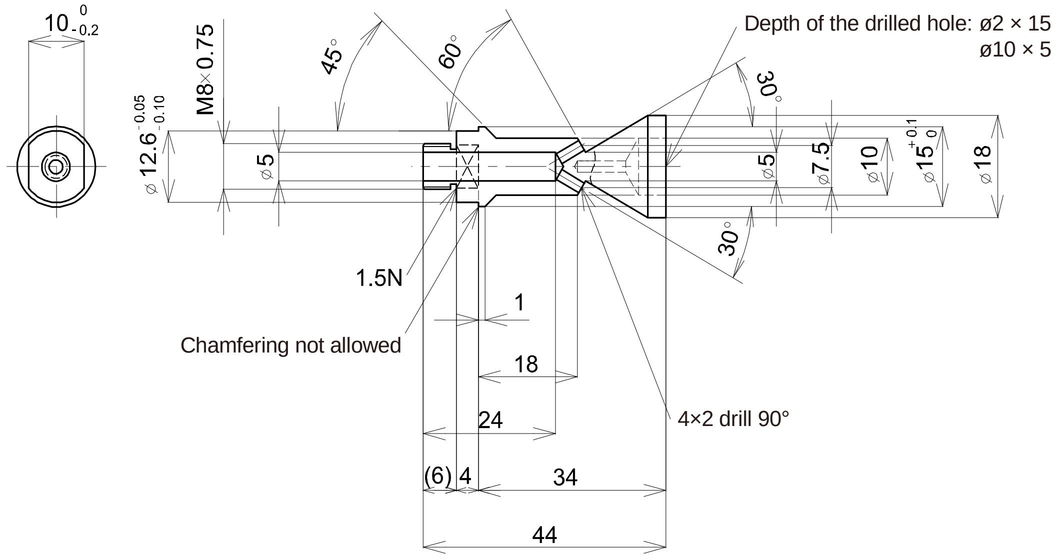
ø8 through-hole diameter knock-out jig (material: C3604B)



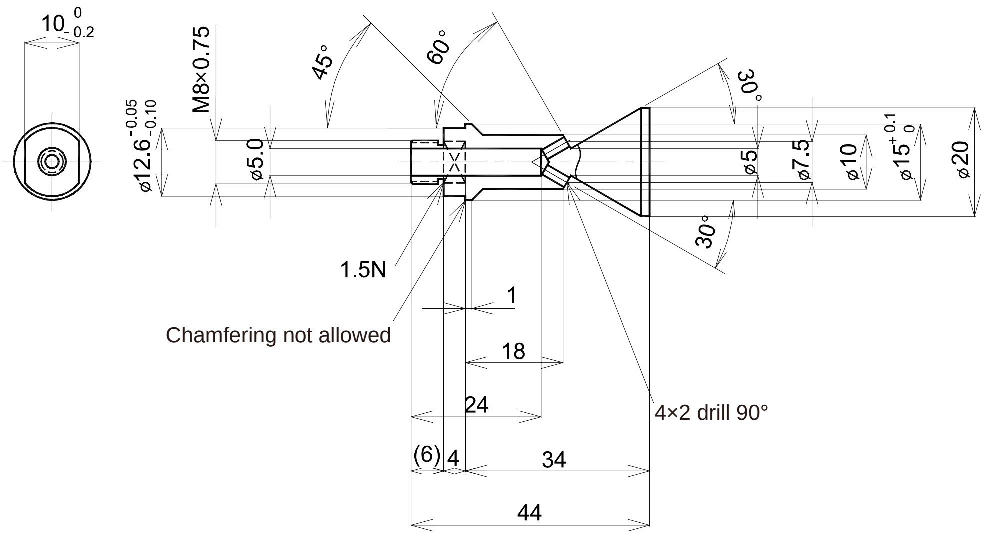
- ø16 through-hole diameter knock-out jig (material: C3604B)



- ø20 through-hole diameter knock-out jig (material: C3604B)



- Through-hole diameter knock-out jig for blank (material: C3604B)



[Procedure]

1. Remove the spindle cap, cap nut, and collet chuck.

   See *<Section 7.5.1 Mounting and adjusting chuck>*.

2. While holding the tip of knock-out jig for through-hole workpiece, pull out the square spring and the cartridge of knock-out device for through-hole workpiece toward you.
3. Replace the knock-out jig for through-hole workpiece at the tip of cartridge by using a wrench.
4. Insert the cartridge, square spring and collet chuck into the chuck sleeve in order. Then, mount the cap and spindle cap.

   See *<Section 7.5.1 Mounting and adjusting chuck>*.

5. Move the knock-out device forward and backward to make sure that nothing engages with the knock-out jig.
6. Advance the knock-out device. Lightly push the tip of the jig with your finger to make sure that the jig does not go any further from the cap end face.
   If the jig goes into the cap in, it is necessary to adjust the knock-out device.

   See *<Section 7.6 Adjusting knock-out device>*.

7. Adjust the chucking force of back spindle.

   See *<Section 7.5.2 Adjusting back spindle chucking force>*.

[Note]

- Two kinds of compression coil springs are provided with the machine. If the knock-out device does not work as expected, use the stronger one.
- If the knock-out device is used together with medium pressure coolant device, the same through-hole knock-out jig to be used.

7.8 Phase Adjustment between Spindle and Guide Bushing (Non-conformed Material)

To use a non-conformed material, the phase between the main spindle and the guide bushing device must be adjusted. Take the following procedures to adjust the phase. Before starting, provide a sufficient work space because the adjustment is to be performed from the rear of the machine.

[Procedure]

1. Mount the chuck for non-conformed material, and adjust the chucking force.
For mounting the chuck for non-conformed material, see *<Section 7.1.1 Mounting and replacing spindle chuck>* and *<Section 7.1.2 Adjusting spindle chucking force>*.

> **CAUTION**
>
> A pin is inserted into the chuck sleeve for non-conformed material. Be sure to align the chuck with a pin when tightening the main spindle cap nut. If the chuck and the pin are not aligned, the chuck may be damaged when tightening the main spindle cap nut.

2. Mount the guide bushing for non-conformed material.

See *<Section 7.3 Replacing and Adjusting Guide Bushing>*.

The clearance of guide bushing must be adjusted after the phases between the main spindle and the guide bushing device has been adjusted.

Subsequent operations are performed on the rear side of the machine.

3. Remove the rear cover.

4. Remove the belt cover.

You will see the timing pulley of the guide bushing driving device.

5. Loosen the four bolts securing the flange with the timing pulley. You do not need to remove all the bolts.

6. Rotate the timing pulley to align the phase of the main spindle with that of the guide bushing device so that non-conformed material can go into the guide bushing device.
For adjustment, adjust the phases between the flange and the pulley.
If the phases between the spindle and the guide bushing device cannot be aligned completely, you need to move the hanging position of the belt by one teeth.

See *<Section 7.2 Mounting/Adjusting a Guide Bushing Device>*.

7. Adjust the clearance between the guide bushing and the material.

See *<Section 7.3 Replacing and Adjusting Guide Bushing>*.

8. Fix the phases of the spindle and the guide bushing.
Firmly tighten the four bolts that have been loosened in Step 5.
Make sure that the belt is properly set to the timing pulley.

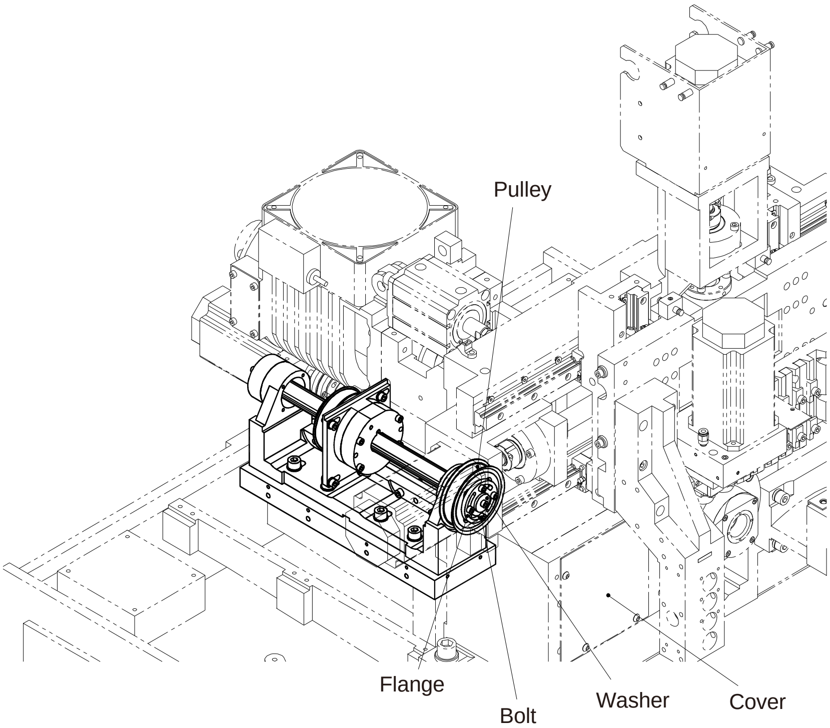
9. Slowly move the spindle forward and backward to make sure that the guide bushing device and the non-conformed material are properly adjusted.

10. Make sure that the non-conformed material can be inserted into guide bushing device repeately.

11. If the phases are not aligned, perform adjustment again.

12. Install the belt cover.

13. Mount the rear cover you have removed in Step 3.


Pulley
Flange
Bolt
Washer
Cover

7.9 Mounting and Adjusting Workpiece Separator

7.9.1 Workpiece collection mode

This machine provides the following two methods for collecting the workpiece.

- Pick-off collection mode (U31J) — (standard)

  Holds a workpiece with the back spindle ("pick-off") and puts it in a workpiece chute ("knock-out"). This method is normally used for collecting a workpiece which can be chucked by the back spindle.
  A product of upto 80 mm in workpiece length.

- Workpiece receiver box collection mode (U352J) — (optional)

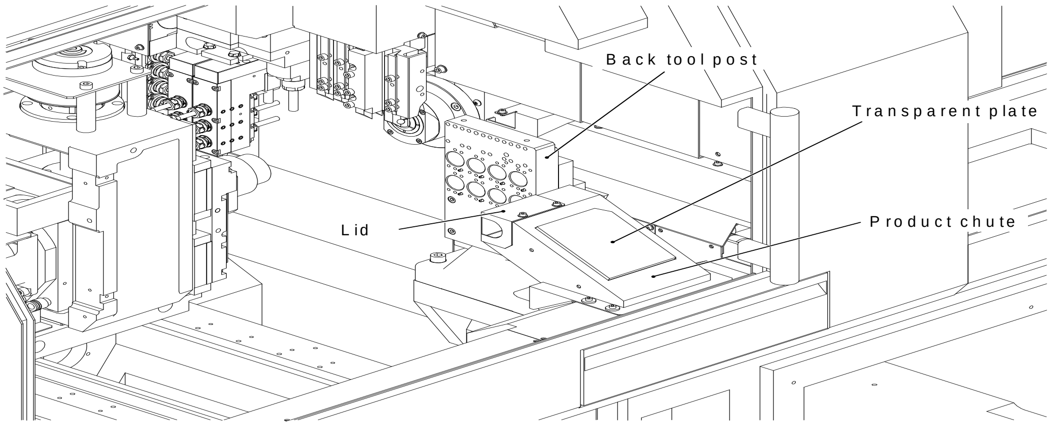
  This method is used for collecting a workpiece (a short workpiece or disc shape workpiece) which cannot be chucked by the back spindle. The back spindle chucking device is removed and replaced with the workpiece receiver box. The finished workpiece is cut off and collected in this workpiece receiver box.

[Note]

The workpiece receiver box (U352J) can collect a workpiece which is up to 45 mm [1.77"] in length. When this method is used, however, back machining is disabled.

7.9.2 Pick-off collection mode (U31J)

The back spindle approaches the fixed product chute, then holds and puts the workpiece in a workpiece chute. Place the lid on the product chute and move the lid into contact with the back tool post to position the tool. If the workpiece is too thin or small to collect, the advance position of the back spindle must be changed.



7.9.2.1 Installing the scar-preventing chute part (U305J option)

A POM (delrin) plate can be installed in the product chute to prevent severe scarring of workpieces when they are collected. Adding a suitable amount of coolant is extremely effective in reducing adhesion.

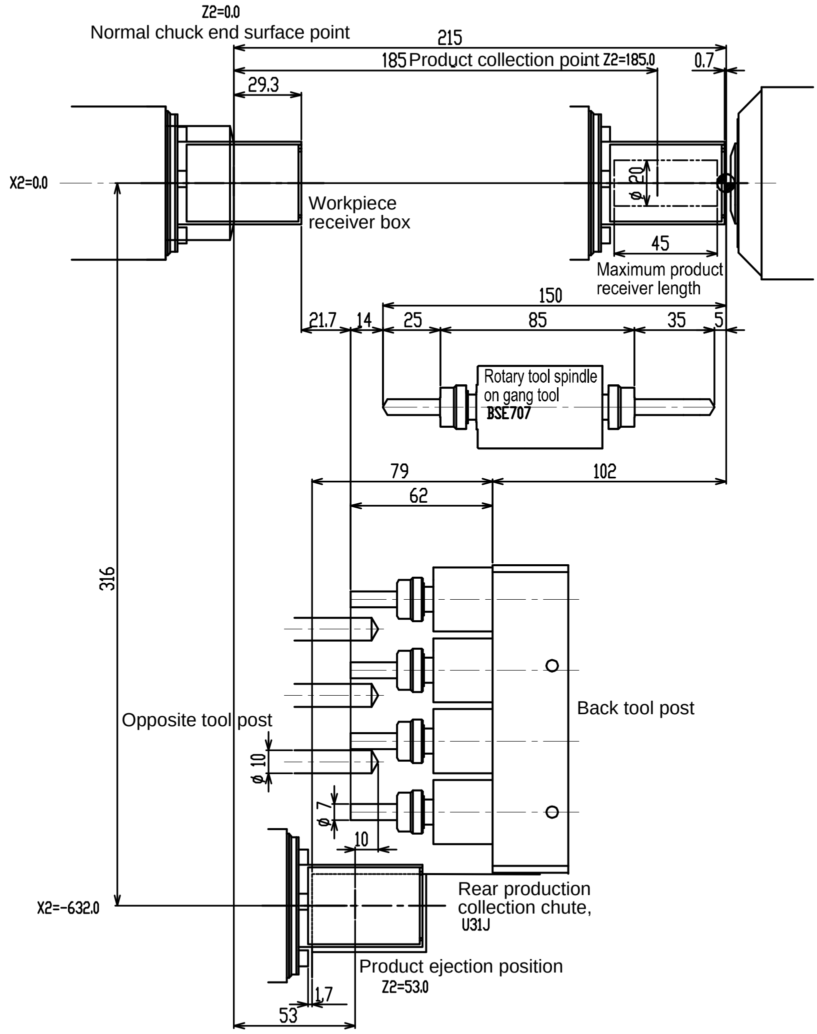
[Procedure]

1. Stop the machine operation.
2. Remove the transparent plate from the top of the product chute.

   : The plate is not fixed with bolts.
3. Remove the product chute.
4. Insert the bottom sheet in the product chute and secure it using bolts.
   In the same way, insert the side sheet in the product chute and secure it using button bolts.
5. Install the product chute.
6. Mount the transparent plate on the top of the product chute.
7. If the direction of the coolant nozzle has been changed, adjust the nozzle to the proper position.

7.9.3 Setup for retrieving workpieces from the workpiece receiver box (U352J)

Mount the workpiece receiver box at the tip of back spindle. The machined workpiece is collected in the receiver box, and ejected to the product chute. Remove the lid of the product chute (U31J) and use the machine without the chute.
The opposite tool post, the back tool post tool type and cutting tool length interfere with each other.
If this happens, remove the tool that causes interference.



⚠ CAUTION

Described below are operation procedures when the Setting switch is set to the "|".
If you are going to open the door, press the Door open button to unlock the door.

The door cannot be unlocked unless the following conditions are met.

All spindles have stopped.

All servo axes have stopped.

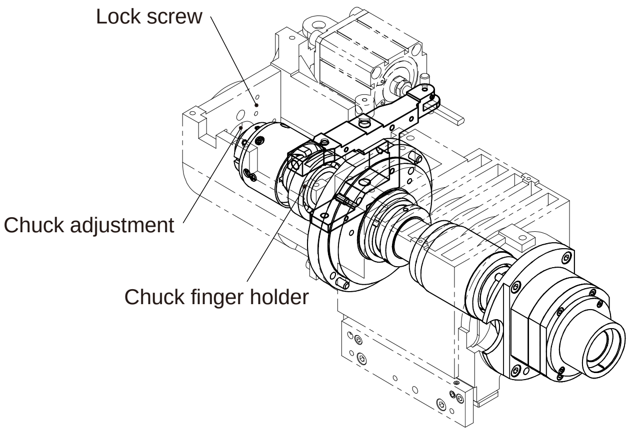
The machine is not operating.

The coolant is not discharged:

The Door open button is pressed.

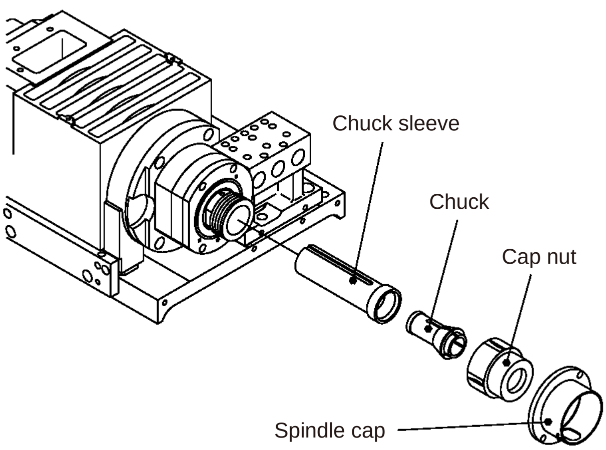
[Procedure]

1. Press the Back spindle chuck key to open the back spindle chuck.
2. Remove the back spindle cover from the back spindle device.
3. Loosen the lock screw of the chuck adjustment nut on the rear side of the chuck finger holder.
4. Loosen the chuck adjustment nut.
5. Move the chuck finger holder to the rear side of the spindle.



6. Remove the back spindle cap and cap nut.

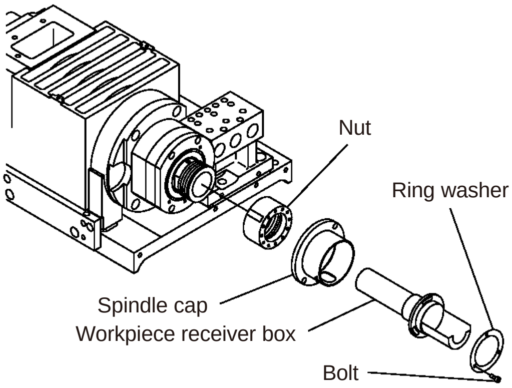
   See *<Section 7.5.1 Mounting and adjusting chuck>*.
7. Remove the chuck and the square spring.



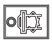
8. Remove the knock-out.

See *<Section 7.6.1 Mounting and replacing knock-out jig>*.

9. Screw the nut into the front end of the back spindle (mount the nut in a direction that allows the 12 taps on the end face of the nut to be seen). Use the hook wrench and dedicated wrench to tighten it firmly.



10. Specify M780 command in MDI mode to index the back spindle.

11. Press the All Spindle Stop button  or Reset key 

to cancel indexing of the back spindle.

12. Temporarily fix the workpiece receiver box and ring washer with bolt.

13. Mount the spindle cap, and secure the spindle cap with fixing screw.
Be sure to mount the spindle cap so that its drain cutout facing straight down.

> **CAUTION**
>
> Otherwise, oil will enter the bearings and damage them.
> Mount the spindle cap with its oil drain cutout facing straight down.

14. Insert the tommy rod into the chuck finger holder to hold the spindle. Then, tighten the bolts that has been temporarily fixed.

15. With the spindle chuck open state, turn the chuck adjustment nut toward you and lightly tighten it until it stops. Then, tighten the lock screw securely.

16. After mounting the receiver box, issue the M780 command again to confirm the normal operataion, since the back spindle is not energized.

17. Make sure that the device does not interfere with the guide bushing at cut-off machining.
Slowly advance the back spindle to X2=630.0, Z2=185.0 of the machine coordinate.

18. Make sure that the device does not interfere with the product chute.

Slowly advance the back spindle device until it reaches X2=0.0, Z2=53.0 (product ejection position) observing the interference with the tools for back machining.

7.10 Setting Up and Adjusting the Long Workpiece Device (Option)

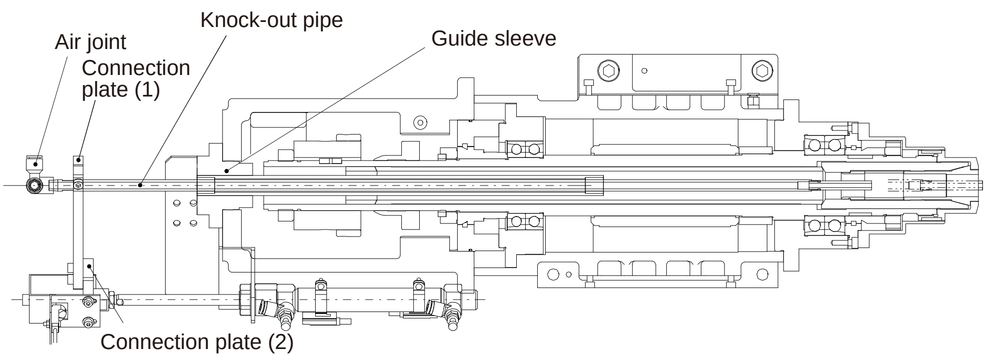
CAUTION

Qualified maintenance personnel should perform mounting and removing receiver shelf for long workpiece device or pipe.

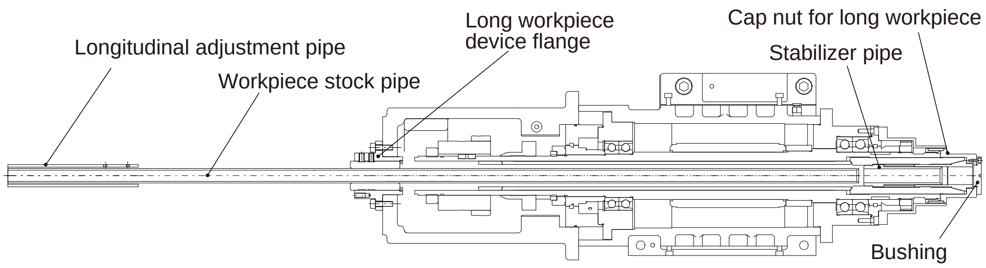
The device is used to unload products exceeding 100 mm [3.94"] in workpiece length. A product of up to 600 mm in workpiece length can be unloaded with this device. The product is transferred through the workpiece stock pipe mounted in the back spindle and unloaded to the workpiece receiver shelf mounted on the left side of the machine. Select the pipe (option) and bushing (option) appropriate to the diameter of the workpiece.

WARNING

If the specific receiver shelf for long workpiece is not mounted, the stock pipe extrudes from the machine. Running the machine with that state may cause a workpiece to run out from the pipe, resulting in a serious personal injury. Be sure to mount the workpiece receiver shelf specified by the manufacturer for your safety.

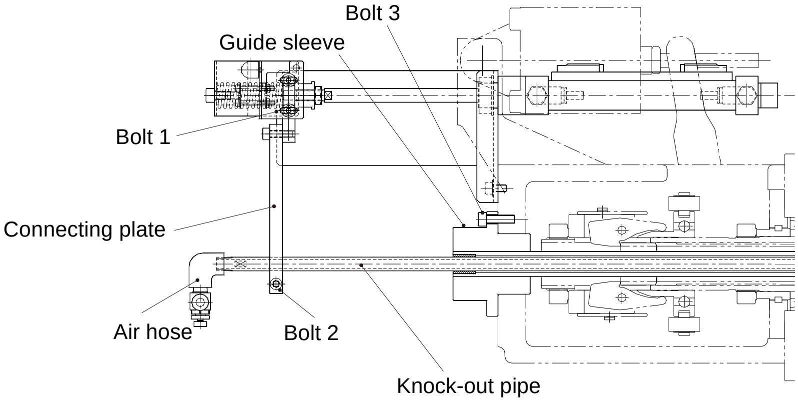


Standard Knock-out Pipe Type



Long Workpiece Device Type

7.10.1 Removing the standard knock-out pipe



Outline of Standard Knock-out Pipe

Mount the stock pipe for long workpiece device in the back spindle. Accordingly, the standard knock-out device must be removed. The work must be performed from the rear of the machine. Secure the sufficient work space in the rear of the machine.

[Procedure]

1. Remove left side cover to allow you to do jobs from the rear of the back spindle.
2. Remove the knock-out device. Remove the bolts to separate connection plate (1) from connection plate (2).
3. Pull out the knock-out pipe from the back spindle.
   First remove the air tube inserted to the back end of the knock-out pipe.
   Mount emboli on the removed air tube.
   Then fix the tube within the machine so that it does not disturb the machine operation.

   Cut the air supply to the machine before doing the above job, if necessary.
4. Remove the guide sleeve inserted to the back spindle.

7.10.2 Mounting stock pipe for long workpiece device

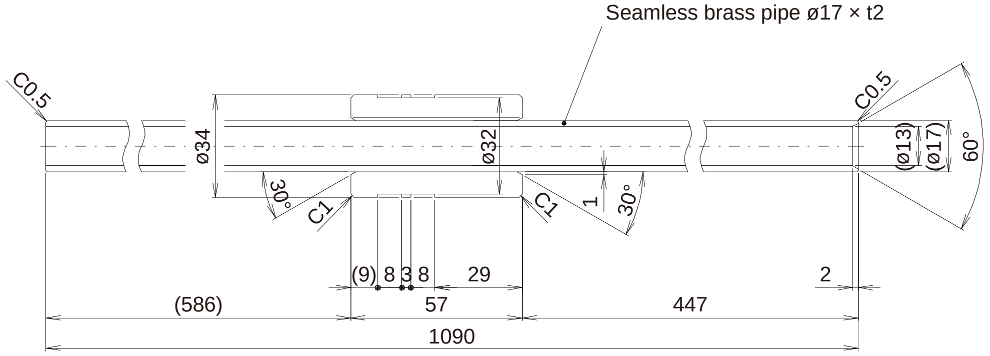
Three kinds of stock pipes and one longitudinal adjustment pipe, and blank bushings are provided with the machine.

[Procedure]

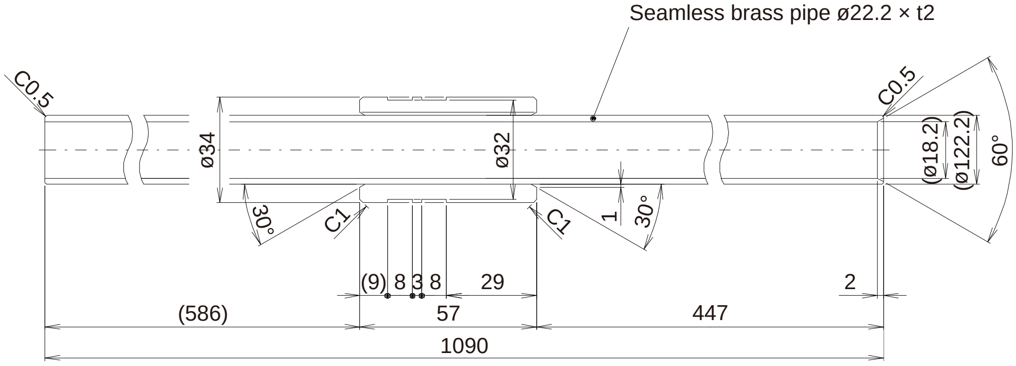
1. Install the long workpiece device flange on the back headstock.
2. Insert a workpiece stock pipe appropriate to the workpiece diameter and fix the pipe with bolts.
3. Mount a longitudinal adjustment pipe to be appropriate to the pipe diameter.
   Fix the longitudinal adjustment pipe to the workpiece stock pipe securely.
   For adjusting the longitudinal adjustment pipe.

   See *<7.10.5 Adjusting position of longitudinal adjustment pipe>*
4. Advance the air cylinder of the knock-out device to shorten the cylinder rod. This is intended to protect the knock-out from vibration during the machine operation.

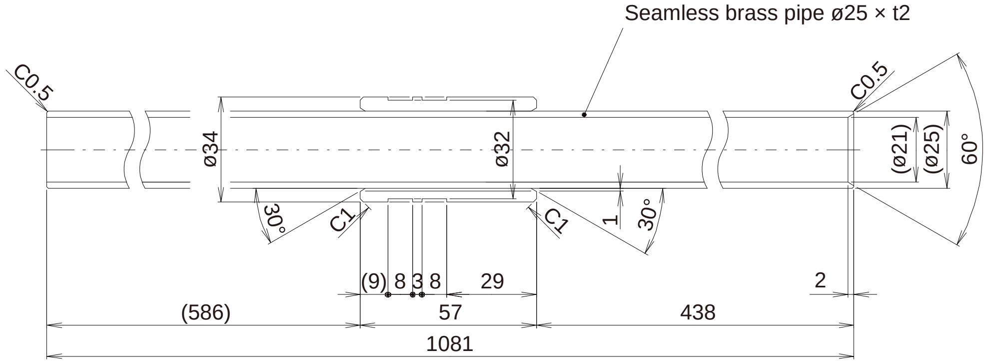
- ø12 workipiece stock pipe



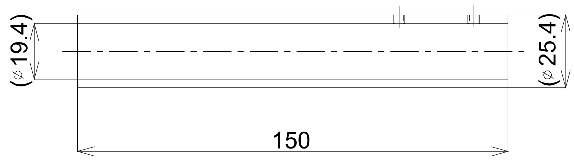
- ø16 workipiece stock pipe



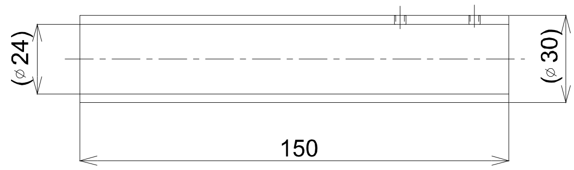
- ø20 workipiece stock pipe



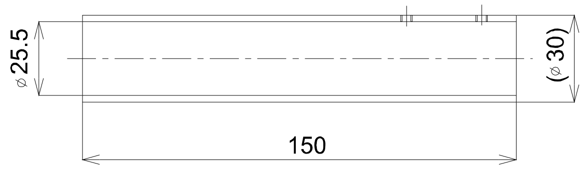
- ø12 longitudinal adjustment pipe



- ø16 longitudinal adjustment pipe



- ø20 longitudinal adjustment pipe



7.10.3 Mounting cap nut and collar for long workpiece device

Three types of collars and one kind of bushing are provided with the device.

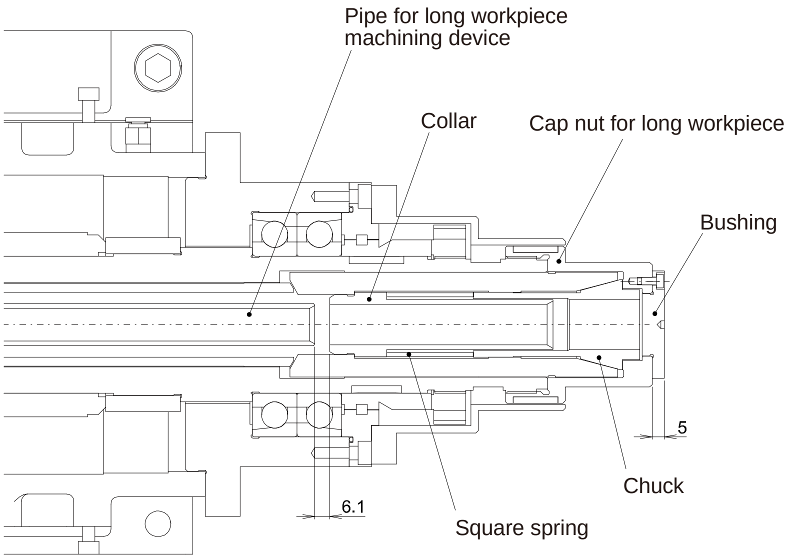
[Procedure]

1. Remove the chuck, square spring, and collar from the chuck sleeve of back spindle.

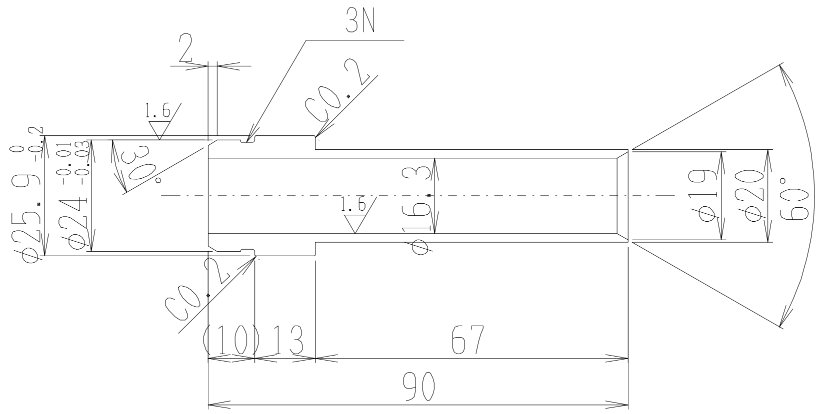
   See *<Section 7.5.1 Mounting and adjusting chuck>*.
2. Mount the cap having the diameter appropriate to the workpiece.

   Use long-body shape for small-diameter and short workpiece
3. Mount the cap nut for long workpiece device.
4. Mount the bushing on the end of the cap nut.
5. Adjust the chucking force again.

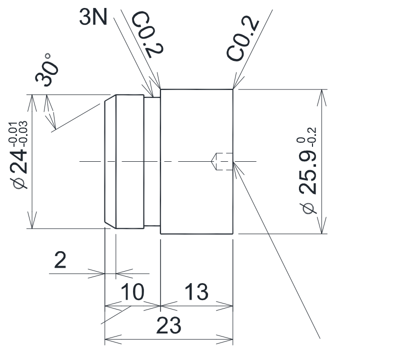
   See *<7.5.2 Adjusting back spindle chucking force>*.



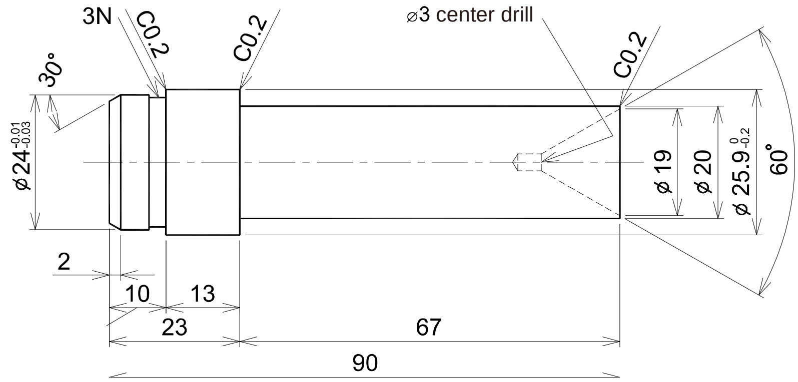
- ø16 collar



- Blank (Standard shape)



- Blank (Long-body shape)



[Note]

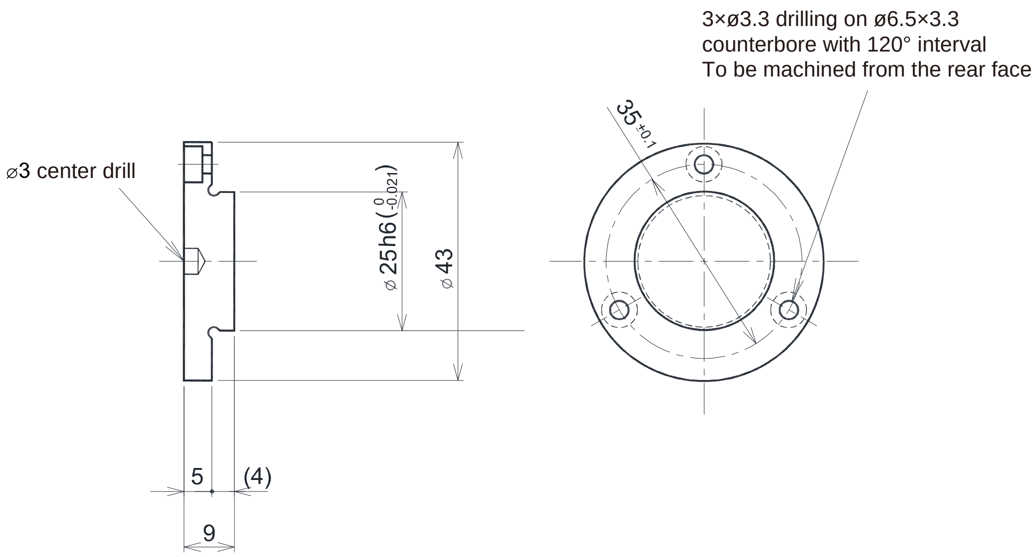
Collars, as shown above, are provided for the cap nut of the long workpiece device as a standard accessory.

If you need a collars in standard shape that has a nominal diameter of 20.3, use the collars provided with the chucking device.

Before using the blank collar, machine the hole that satisfies the conditions: inner diameter of blank collar (øD1) = D + 0.3 ± 0.1

D: Maximum workpiece diameter

- Bushing (Blank)



[Note]

Before using the blank bushing, machine the hole that satisfies the conditions: hole diameter of blank bushing(øD2) = D + 0.5 ± 0.1

- D: Maximum workpiece diameter

7.10.4 Mounting the receiver shelf for long workpiece device

> **CAUTION**
>
> The receiver shelf is very large and heavy component. At least two persons are required to handle the shelf to avoid personal injury.

[Procedure]

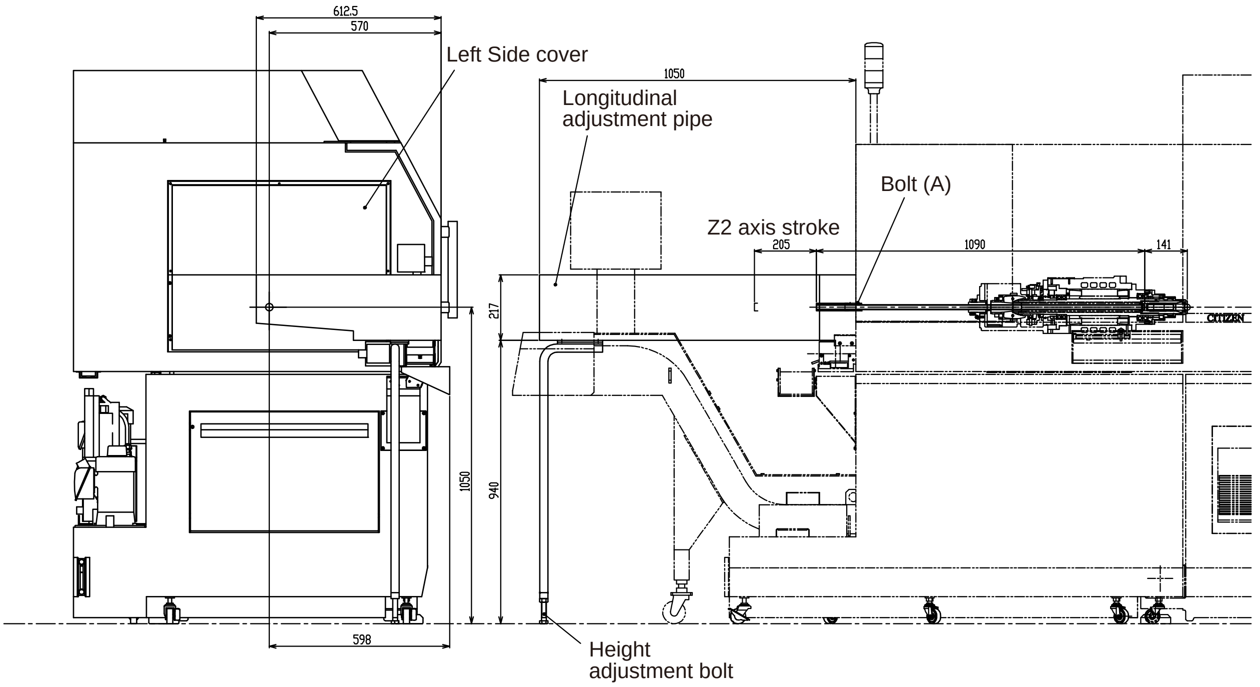
1. Remove the right side cover.

2. Mount the shelf.

If the height of the shelf does not match with the machine height, adjust the position of the shelf with the height adjustment bolt.

3. Mount the side cover for long workpiece device.

4. Keep the cover carefully that was removed in Step 1.



Long workpiece device

7.10.5 Adjusting position of longitudinal adjustment pipe

The workpiece is transferred through the workpiece stock pipe mounted in the back spindle and unloaded sequentially to the left side of the machine. You need to adjust the position of stock pipe in longitudinal direction using the adjustment pipe mounted at the exit of the stock pipe.

[Procedure]

1. Loosen the bolt (A).

   See the figure on page 7-7-53.

2. Adjust the position of longitudinal adjustment pipe so that the workpiece to be unloaded does not run out from the end of pipe.
3. Firmly tighten bolt (A), facing upward, to fix the support pipe.

> **CAUTION**
>
> Be sure to adjust the position of adjustment pipe in longitudinal direction. The workpiece to be unloaded may run out from the pipe for long workpiece machining device, and it may cause damage to the receiver shelf or machine body.

> **WARNING**
>
> Be sure to confirm that the machine has stopped before attempting to adjust the position of adjustment pipe.
> The adjustment during machine operation could result in serious personal injury.

7.11 Adjusting the Workpiece Conveyor (Option)

WARNING

Before adjusting the workpiece conveyor, make sure to turn OFF the main breaker. The work with the main breaker set in the ON position could result in serious personal injury.

7.11.1 Adjusting workpiece chute

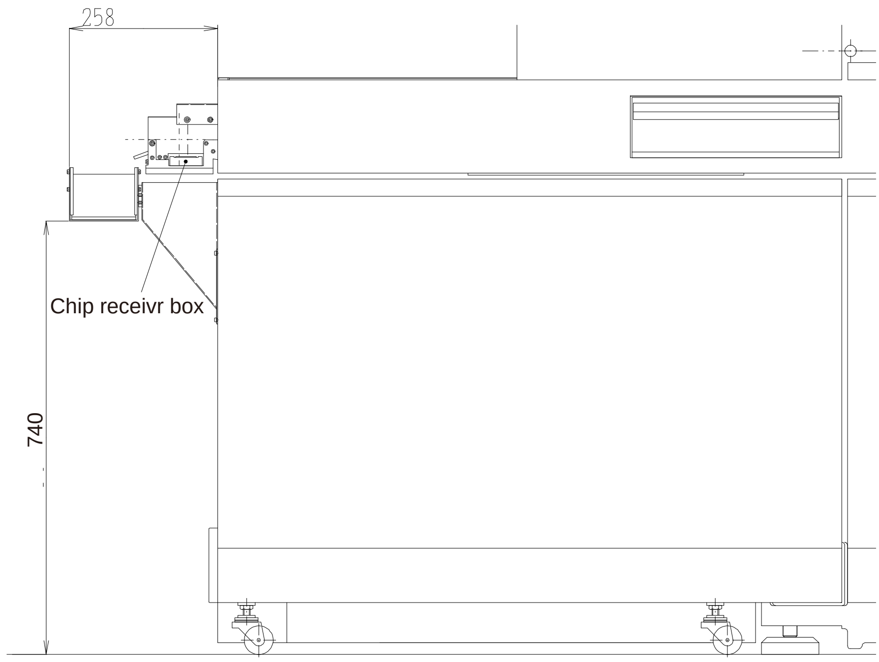
Adjust the position of the delrin plate appropriate to the workpiece, and adjust the gap between the workpiece chute and the belt.
Collecting the thin or small workpieces without adjusting the gap may reduce the collection efficiency. Therefore, it is recommended to adjust the gap every time the different size of workpiece is machined. See the figure below.

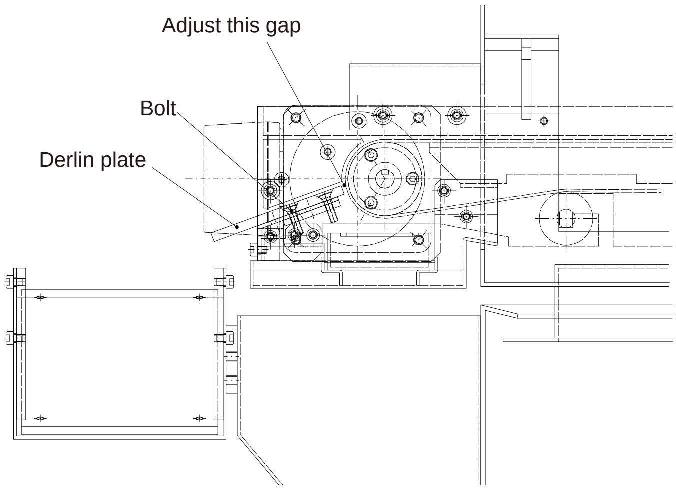
[Procedure]

1. Loosen the bolt.
2. Move the position of delrin plate to adjust gap between the chute and the belt.
3. Tighten the bolt.
4. Make sure that the workpiece chute is firmly fixed.

CAUTION

Adjust the delrin plate so that it does not contact with the belt. Otherwise, the workpiece conveyor might be damaged when running the machine.





7.11.2 Cleaning the Workpiece Conveyor (Option)

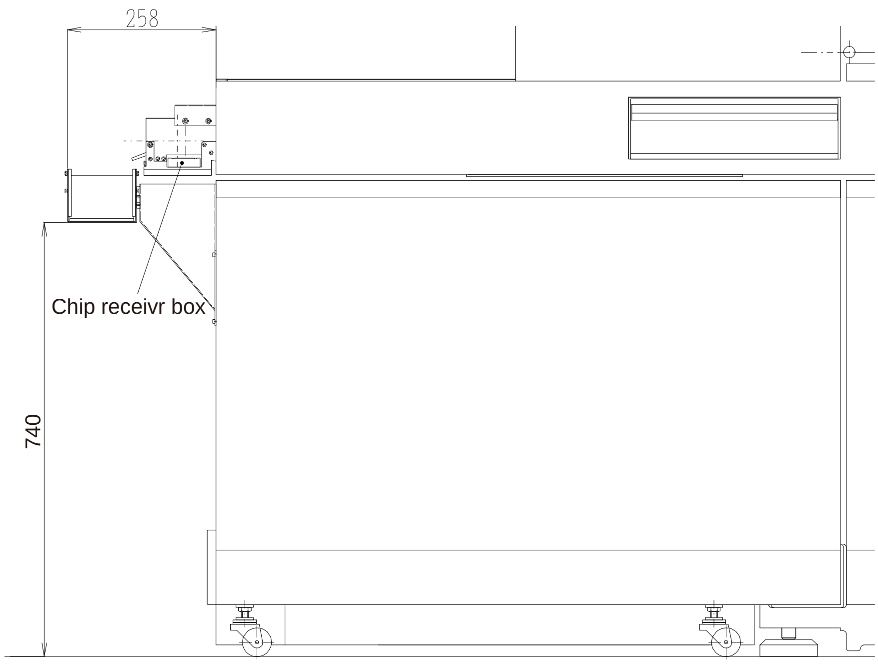
The workpiece conveyor carries chips together with workpieces, and ejects them in the chip receiver box. Remove chips accumulated in the chip receiver box.

WARNING

Before removing chips, make sure to turn OFF the main breaker. The work with the main breaker set in the ON position could result in serious personal injury.

[Procedure]

1. Remove the chip receiver box.
2. Remove chips accumulated in the chip receiver box.
3. Remount the chip receiver box.



7.12 Mounting and Removing Tools

The procedures for mounting and removing tools are given below.
See the figure below when you perform the following procedure.
If it is difficult to mount or dismount a tool from the front of the machine, you may replace a tool from the rear side of the machine.

> ⚠ **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.

7.12.1 Mounting tools

[Procedure]

1. Tool holders available are those for, 12 × 12, and 13 × 13 mm tools. For each tool holder, use the wedges supplied with the holder. Use a floor plate when 12 × 12 mm or 13 × 13 mm tools in the T01 come in close contact.
2. Insert a tool into the holder and tighten three tool fixing bolts uniformly to fix the wedge. (The tightening torque for the tool fixing bolts is about 1.5 kgm (15 Nm), which is equivalent to a force applied with a Allen wrench.
3. For the wedge in each groove without a tool, also, tighten the fixing bolts to secure the wedge.

> ⚠ **WARNING**
>
> Even in the grooves with no tool mounted on the tool holder, tighten the fixing bolts to secure the wedge. Any wedge protruding may cause an interference, resulting in severe damage to the machine.

> ⚠ **CAUTION**
>
> Be sure to tighten the tool fixing bolts securely. If the tool comes off from the holder during machine operation, it may cause a damage to the machine.

7.12.2 Removing tools

Loosen tool fixing bolts and pull the tool out of the holder. If the wedge caught in the holder can not be removed, remove a tool fixing bolt and insert in the screw hole in the center of the wedge as pointed by C in the figure above. Turn the bolt to push the wedge away from the holder.

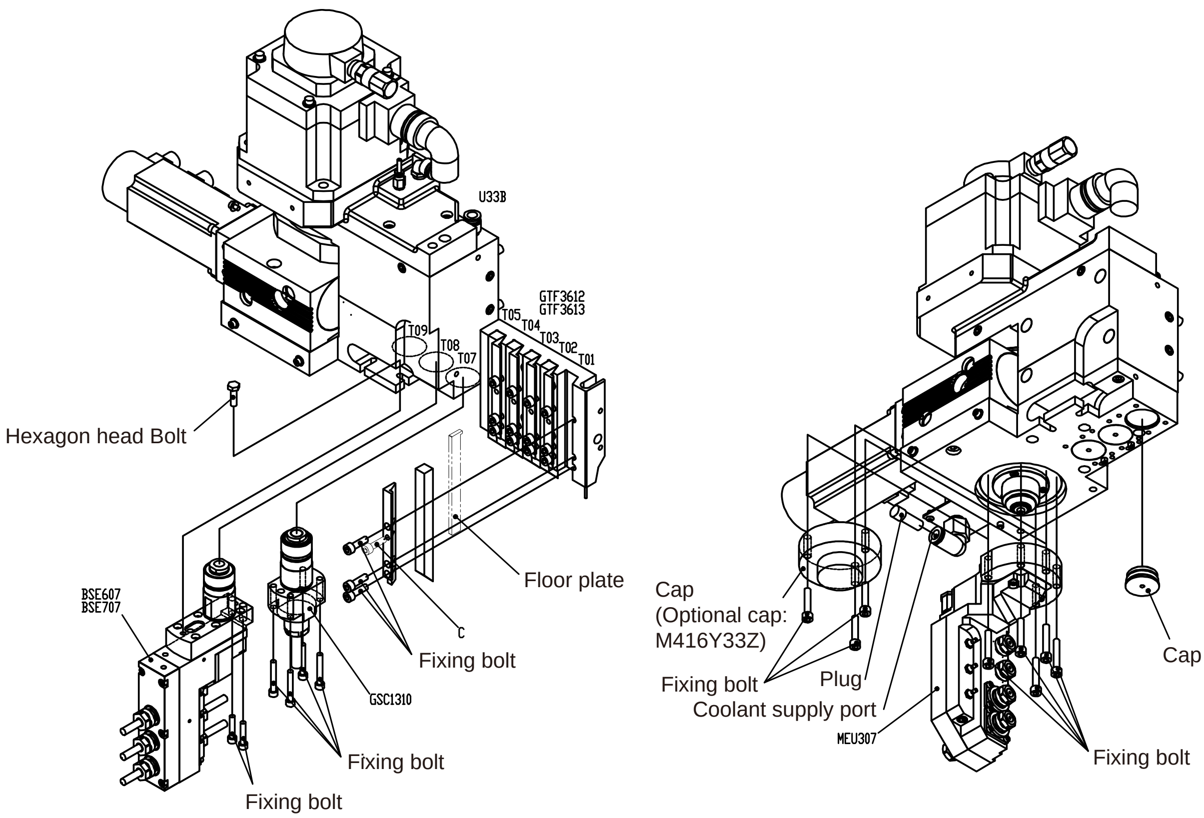
7.12.3 Installing tools on gang tool post rotary tool drive unit

[Procedure]

1. Different holders are used for different tools, refer to *<Chapter 17 Tooling>* for information on tools used on the gang tool post rotary tool drive unit.
2. Place a cap on holders that do not contain tools to cover the opening.

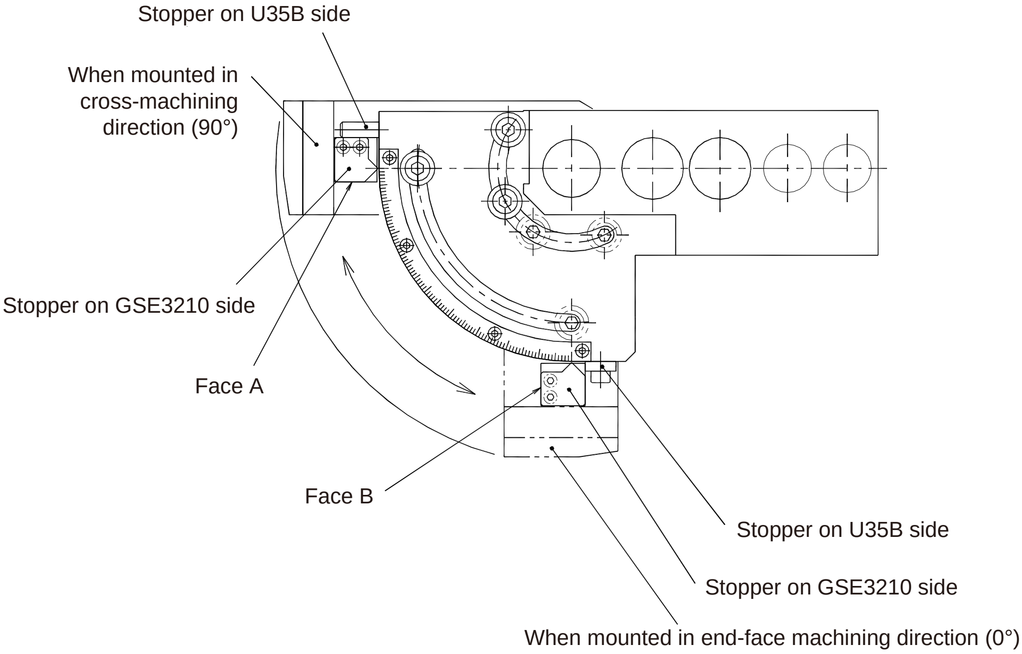
7.12.3.1 Regarding MEU307 removal

To use the machine without the MEU307, install an optional cap (M416Y33Z).
Or use the plug supplied with the U33B to cover the coolant supply port.



7.12.3.2 Mounting GSE3210

GSE3210 can be mounted steplessly either in cross-machining direction (90°) or end-face machining direction (0°) when using together with the U35B gang rotary tool drive device. When mounting GSE3210, push the stopper on GSE3210 side against the stopper on U35B side, then secure it with the fixing bolt.
To push the stopper, push the face A or face B with your finger. When mounting GSE3210 in any other angle than 0° or 90°, chuck the pin gauge with GSE3210, and check the centering by using the dial gauge.



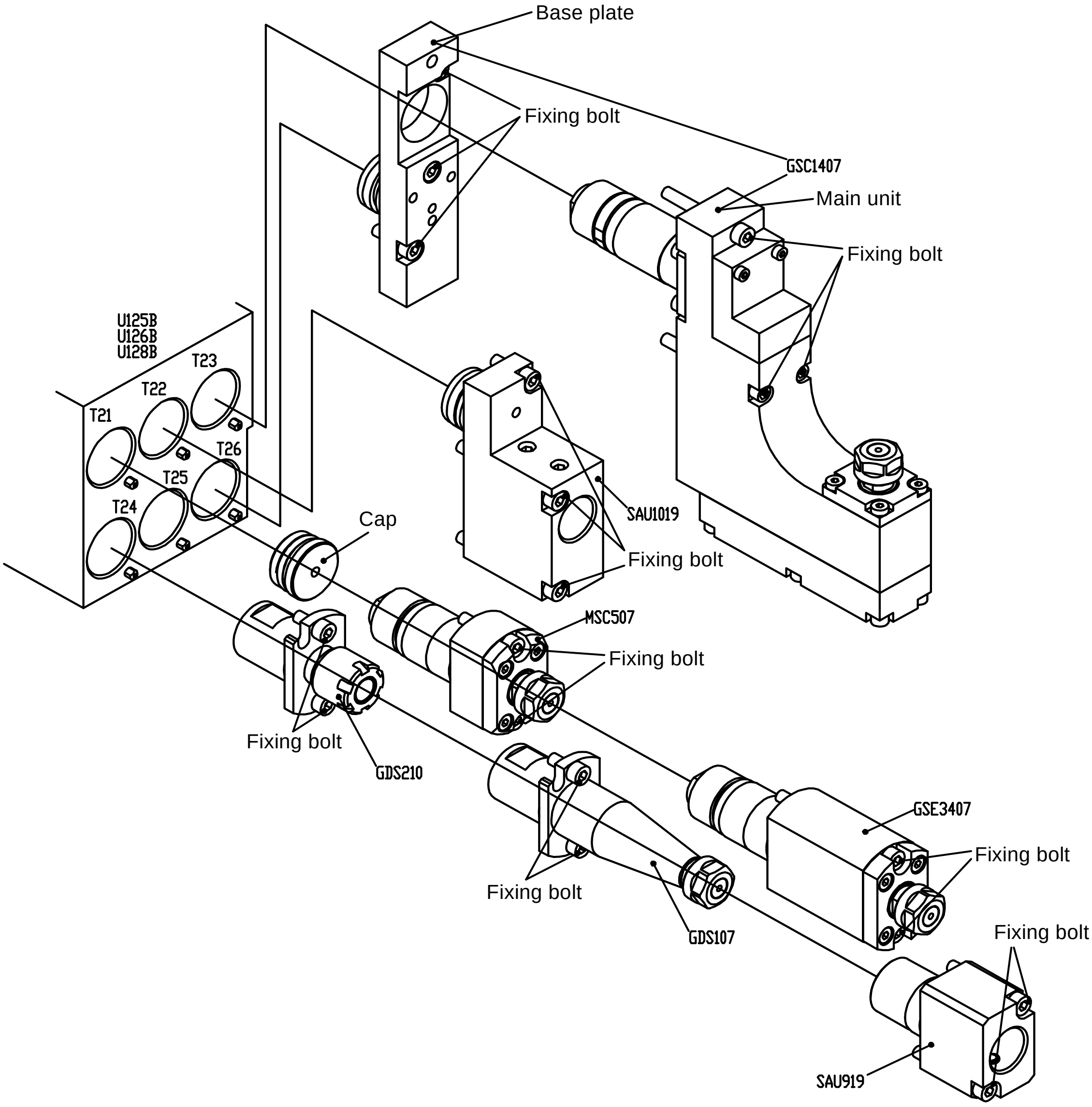
CAUTION

Do not unlock the stopper bolt. The stopper bolt is used as a guide to mount GSE3210.
Do not give any shock to the stopper when mounting or dismounting GSE3210.
Strict care must be taken when mounting or dismounting.
To push GSE3210 against the stopper, be sure to push the stopper on GSE3210 side. Never push the GSE3210 itself.

7.12.4 Installing tools on the opposite tool post

[Procedure]

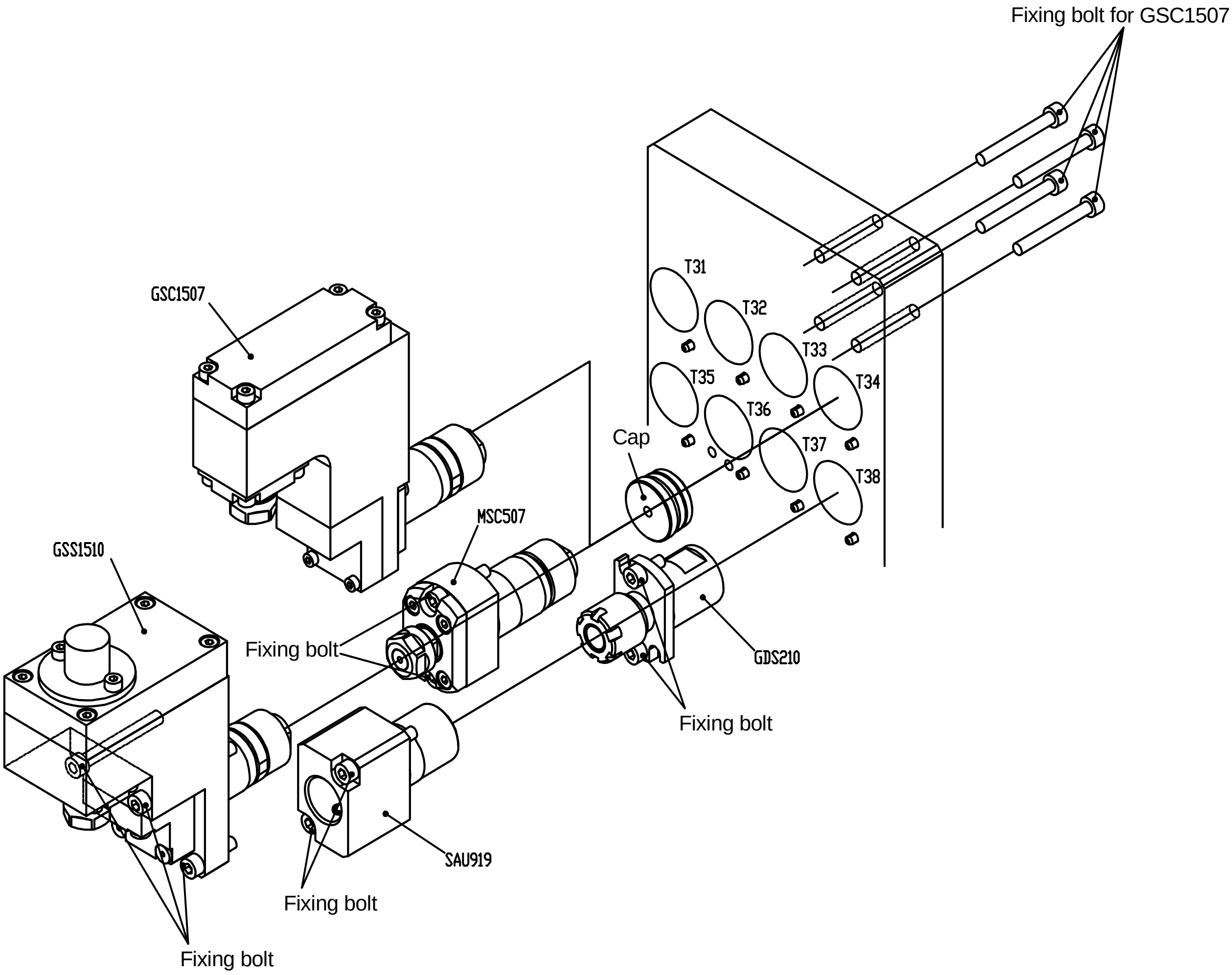
1. These are tools used on the X and XII type opposite tool post.
   The tools that can be used depend on the type of opposite tool post, refer to <*Chapter 17 Tooling*> for further information.
2. After installing the GSC1407 base plate on the opposite tool post, install the main unit on the base plate.
3. When tools T21 to T23 of the U128B will not be used, place a cap on holders that do not contain tools to cover the opening.



7.12.5 Installing tools on the back tool post

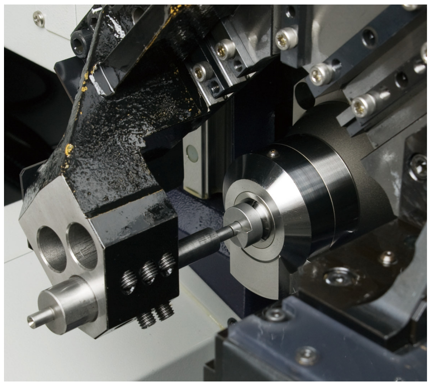
[Procedure]

1. These are tools used on the X and XII type back tool post.
   The tool positions that can be used differ with the tool, refer to *<Chapter 17 Tooling>* for further information.
2. When tools T31 to T34 will not be used, place a cap on holders that do not contain tools to cover the opening.



7.12.6 Adjustment using alignment ring (Option)

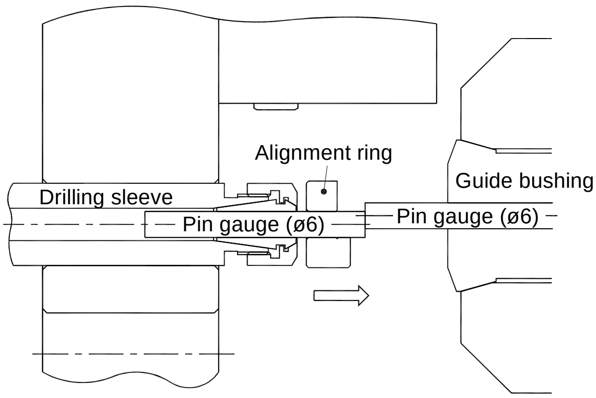
The alignment ring is used to align the position of drilling tool with the core of the workpiece. Use this tool to adjust the position of drilling sleeve holder or rotary tool with the core of guide bushing.
Call the Cincom Service Office.

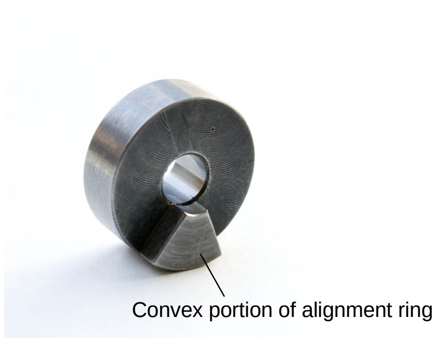


[Procedure]

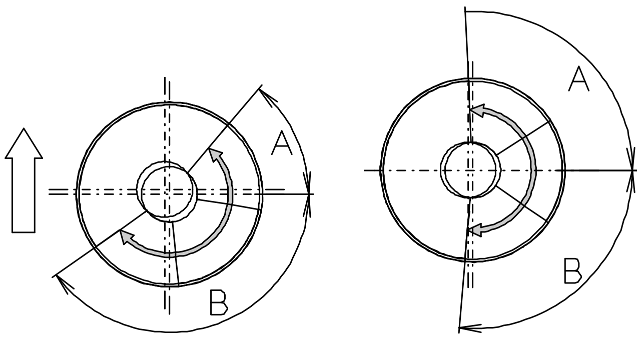
1. Set the drilling sleeve, guide bushing, pin gauge ((ø6 mm [ø0.24"]), and alignment ring (inner diameter ø6 mm [ø0.24"]), as shown in the figure below. Insert the alignment ring into the pin gauge so that its convex portion faces the guide bushing.

(The shank part of drilling tool or other material of which diameter is ø6.0 mm [ø0.24"] can be used as an alternative of pin gauge. Note, however, the backlash with the alignment ring may cause an alignment error.)



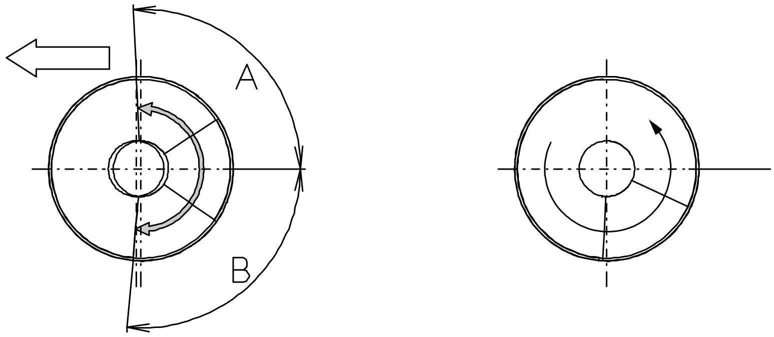


2. Perform adjustment in diameter direction (X direction).
Select the tool on which the alignment ring is mounted, select [Preparation] → [Tool Set (manual)] → [Dia.] → [Start], and perform adjustment in diameter direction (X direction).
As shown in the figure below, adjust the area A then B, so that A equals to B. When finished adjustment, press [INPUT].

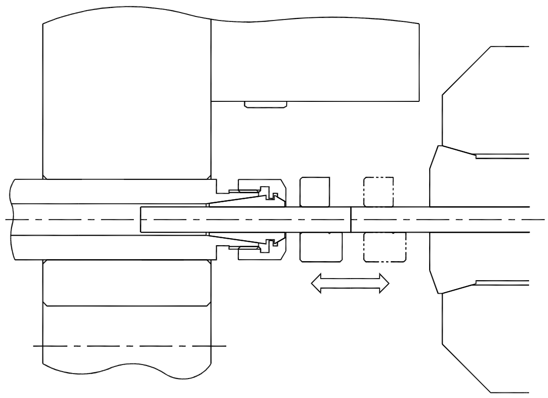


3. Perform adjustment in core direction (Y direction).
Select [Preparation] → [Tool Set (manual)] → [Core] → [Start], and perform adjustment in core direction (Y direction).
The plus (positive) direction depends on the machine model or tool layout pattern. First make sure the plus direction for core direction (Y direction).
In proportion to the movement of tool toward the direction of the ring with no convex portion, the rotating range gradually increases.

Adjust so that the ring rotates smoothly. When finished adjustment, press [INPUT].



4. Perform fine adjustment.
Perform adjustment in diameter and core directions in similar way to Steps 2 and 3. Make sure that the alignment ring moves between the two pin gauges smoothly.
Be sure to return the alignment ring to the pin gauge at drilling sleeve side completely before moving axis. Otherwise, jig and/or ring may be damaged.



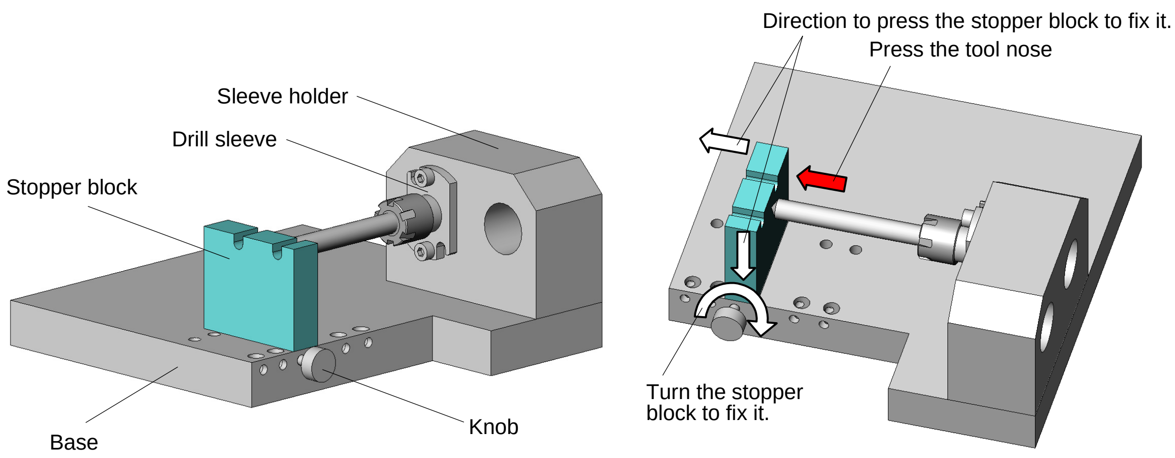
7.13 Tool presetter

Using an optional simplified tool presetter enables setting of tools for opposite tool post (U125B, U126B, U128B) and back tool post (U155B) at the outside of the machine.

[Note]

The simplified tool presetter does not assure the obligate accuracy. If the high accuracy is required in machining, adjust the offset value in the machining program.

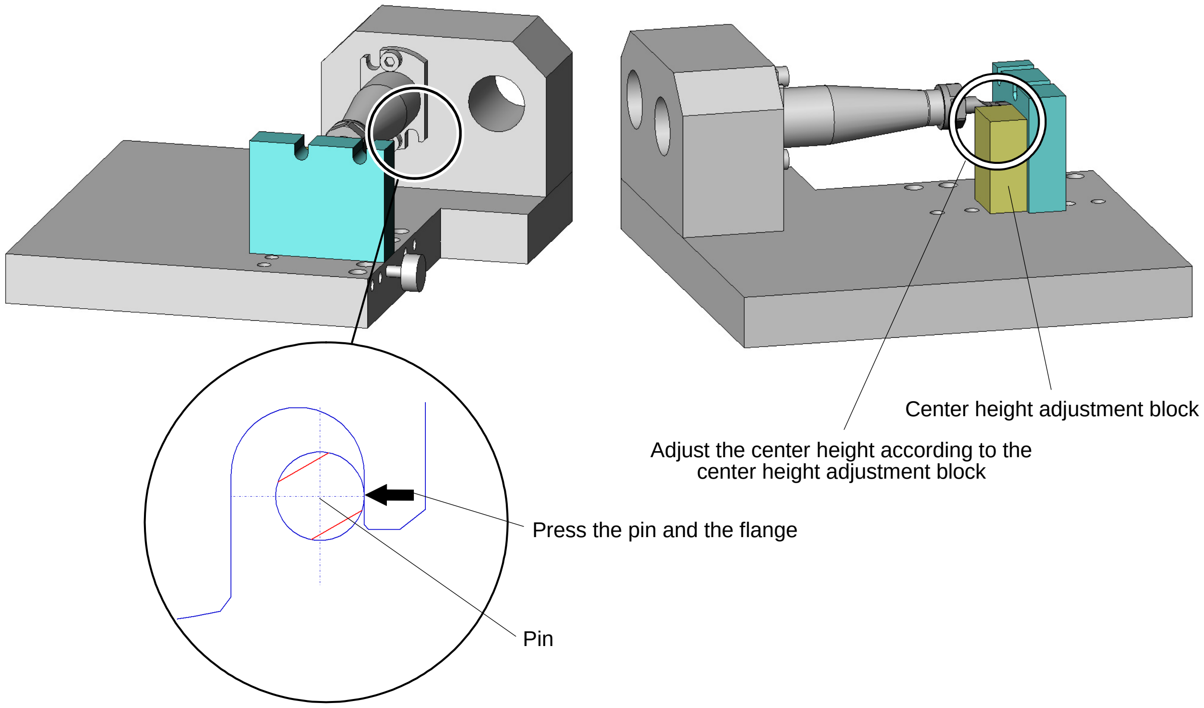
■ Tool setting for end-face drilling spindle, drill sleeve, and sleeve adapter [GSE3407, MSC507, SAU919, GDS107, GDS210]



[Procedure]

1. Fix the drill sleeve to the sleeve holder.
2. Mount the stopper block to the position Back, Front, or Deep Hole marked on the base. Tighten the knob so that the stopper block does not shake. While pushing the stopper block to the base, apply the force to the direction away from the sleeve holder. This can settle the position to press the tool nose.
3. Tighten the nut while pushing the tool nose to the stopper block. Be careful not to push the tool nose with excess force. Doing so may cause damage to the tool or stopper block.
4. Loosen the knob, move the stopper plate away from the tool nose, and then remove the stopper block from the base.

■ Setting the boring bar

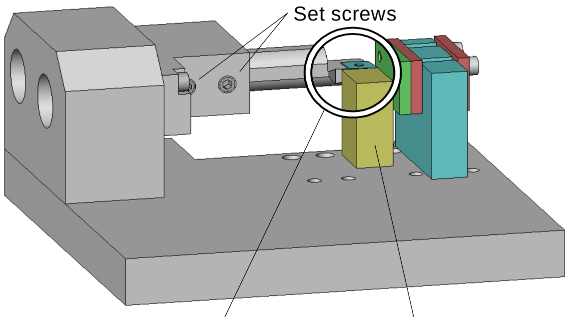


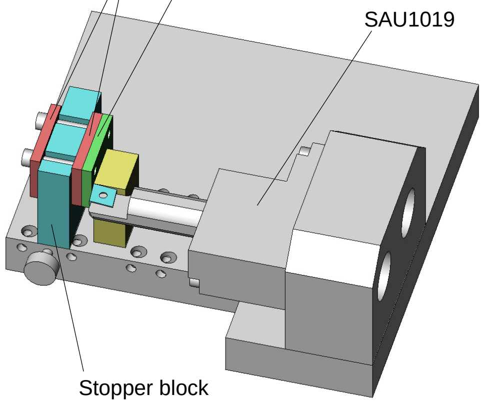
[Note]

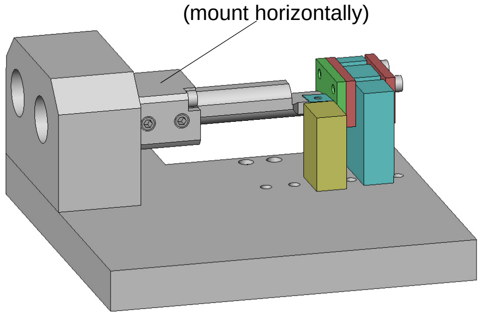
To set the boring bar, set the drill sleeve perpendicularly so that the tool nose becomes horizontal when fixing the drill sleeve in Step 1 (except for pinch turning). Set the drill sleeve while pressing the flange to the pin of the sleeve holder.

In Step 3, adjust the center height according to the center height adjustment block.

■ Tool setting for pinch turning (SAU1019, SAU919)







Pinch turning by using SAU919

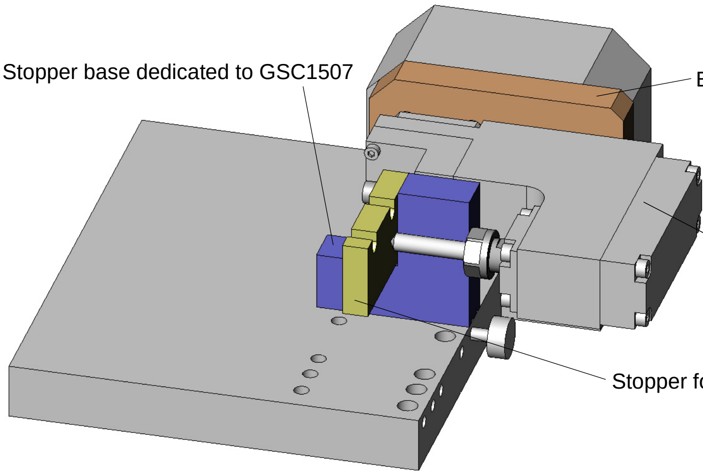
[Procedure]

1. Insert the DS holder into the sleeve adapter, and mount them to the sleeve holder.
   For pinch turning by using SAU919, set SAU919 horizontally in the sleeve holder so that the tool nose faces perpendicularly upward.
2. Mount the stopper block to the position marked as "Pinch Turn". Shift the stopper position 10 mm by using the spacer and the stopper plate. Tighten the knob so that the stopper block does not shake. While pushing the stopper block to the base, apply the force to the direction away from the sleeve holder. This can settle the position to press the tool nose.
3. Lightly press the tool nose to the stopper plate, adjust the center height according to the center height adjustment block, and then tighten the set screw. Be careful not to push the tool nose with excess force. Doing so may cause damage to the tool or stopper block.
4. Loosen the knob, move the stopper plate away from the tool nose, and then remove the stopper block from the base.

⚠ CAUTION

In pinch turning, the tool position in longitudinal direction differs from that on opposite tool post (T20's). Be sure to set the tool to the specified position.

■ Tool setting for cross spindle on back tool post [GSC1507]



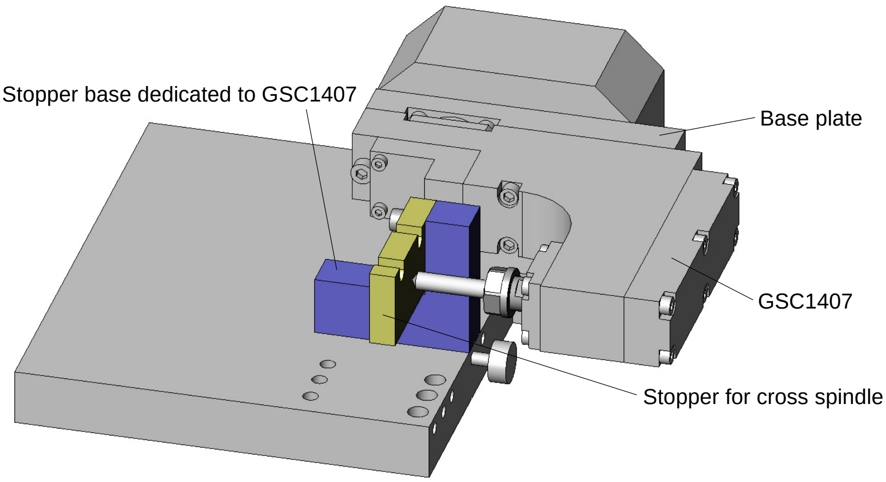
[Procedure]

1. Mount the base plate to GSC1507.
2. Set the knob to the position marked as "Back Cross", then mount GSC1507 and base plate to the sleeve holder.
3. Mount the stopper base to the position marked as "Back Cross" on the base.
   Tighten the knob so that the stopper does not shake. While pushing the stopper to the base, apply the force to the direction away from the sleeve holder. This can settle the position to press the tool nose.
4. Tighten the nut while pushing the tool nose to the stopper for cross spindle. Be careful not to push the tool nose with excess force. Doing so may cause damage to the tool or stopper.
5. Loosen the knob, move the stopper away from the tool nose, and then remove the stopper base from the base.

[Note]

The tool nose position of GSC1407 differs from that of GSC1507. Use the dedicated stopper base appropriate to the spindle.

■ Tool setting for cross spindle on opposite tool post [GSC1407]



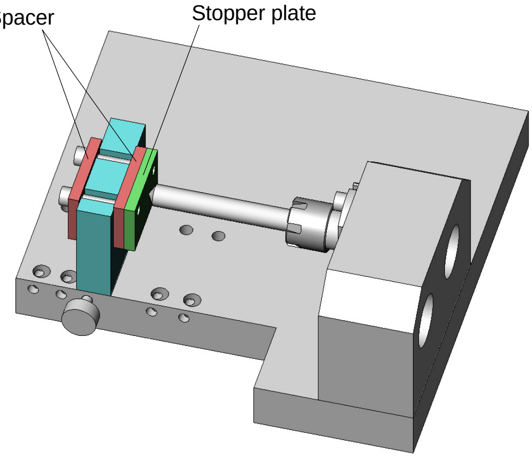
[Procedure]

1. Mount the base plate (provided with GSC1407) to the sleeve holder.
2. Set the knob to the position marked as "Front Cross", then mount GSC1407 to the base plate.
3. Tighten the knob so that the stopper does not shake.
   While pushing the stopper to the base, apply the force to the direction away from the sleeve holder. This can settle the position to press the tool nose.
4. Tighten the nut while pushing the tool nose to the stopper block for cross spindle. Be careful not to push the tool nose with excess force. Doing so may cause damage to the tool or stopper.
5. Loosen the knob, move the stopper away from the tool nose, and then remove the stopper base from the base.

[Note]

The tool nose position of GSC1407 differs from that of GSC1507. Use the dedicated stopper base appropriate to the spindle.

■ Shifting the tool nose



[Note]

To shift the tool nose from the standard position, use the spacer of 5-mm thickness and the stopper plate.

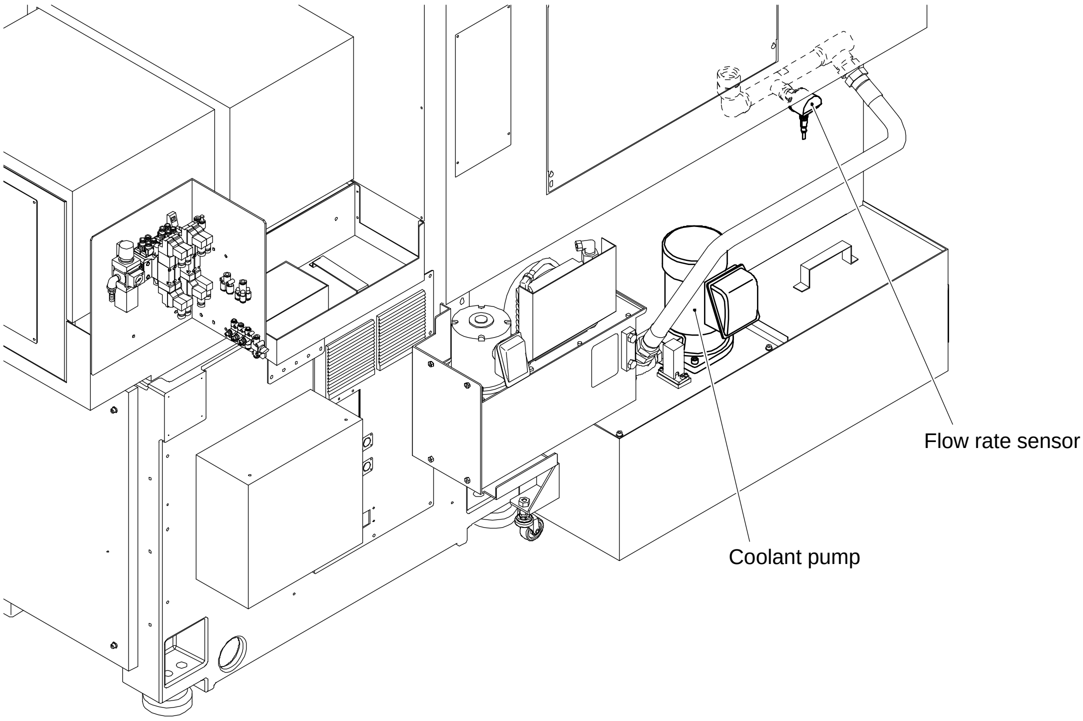
When mounting the tool with its nose shifted, pay strict attention to interference.
Be sure to press the tool nose to the stopper plate, not to the spacer.

7.14 Coolant Flow Detector (U53R)

If the coolant flowrate is dropped due to some reason, the coolant is not fed to the cutting point to likely cause fire or other disasters. This device detects the drop in coolant flowrate and automatically stops the machine operation.

[Note]

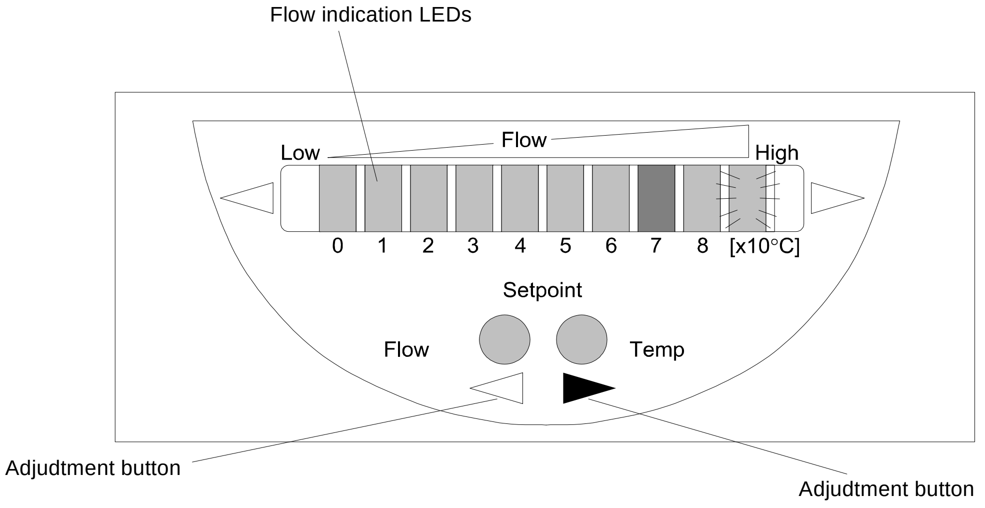
- The coolant flowrate varies depending on the temperature. To distinguish the change in flowrate under normal conditions from that in abnormal state, set the alarm flowrate (Low Rate) with the upper three coolant nozzles being closed. Define the normal flowrate as the value determined by valve full-open state.
- When the initial coolant temperature is 15°C or less, this detector does not work. Therefore, although the machine must be started early in the morning of the winter, for example, this device is turned off until the coolant temperature is elevated to at least 15°C. The operator must watch the temperature. Once the coolant temperature reaches 15°C, this detector works and maintains the active state; and even if the temperature is dropped by environment, the device keeps the function enabled.



Position of coolant flow detector

7.14.1 Names and functions of components

Coolant flow sensor



- Flow indication LEDs

  In normal operation, the LED shows the current flowrate in green LED. The Set point LED lights orange while coolant is discharged in normal rate, and red in slow rate. The LEDs are used to indicate the specified flow rate, current temperature, specified temperature, or alarm.

- Adjustment buttons

  Use these buttons to change indications and setting values.

7.14.2 Setting

■ Setting Procedure

[Procedure]

1. Set the High Flow rate.
2. Set the Low Flow rate.
3. Set the flowrate switch point.
4. Set the switch point of temperature to activate the flowrate sensor.

   : **At the shipment, the temperature to activate the flowrate sensor is set to 15°C. Accordingly, you do not need to set it again in normal operation. If you restore the default temperature setting (4°C), you need to set it to 15°C.**

■ Setting High Flow

Set the indication of maximum flowrate to the upper limit.
(All the LEDs except the switch point LED light green.)

Set High Flow in the following procedure.

[Procedure]

1. Press the Coolant button , and confirm that all nozzles discharge the coolant at maximum flowrate.
2. Hold the button for a certain time.

   LED9 goes on, then starts flashing five seconds later.
3. When LED9 starts flashing, release the button.

   The value for High Flow is set, and the system goes back to operation mode.

■ Setting Low Flow

[Note]

Be sure to set Low Flow after setting for High Flow has completed.

[Procedure]

1. Close the three nozzles at the upper part of the cutting room. Fully open the other nozzles.
2. Press the Coolant button to let the coolant flow.
3. Hold the button for a certain time.

   LED0 goes on, then starts flashing five seconds later.
4. When LED0 starts flashing, release the button.

   The value for Low Flow is set, and the system goes back to operation mode.

■ Setting Setpoint (Flowrate Monitoring)

Set the Setpoint in the following procedure:

[Procedure]

1. Press the button for a short time.

   The LED for the current switch point starts flashing.
2. Use either of the (toward left) or (toward right) button to move to the desired position. About five seconds have passed after the button is released, the position currently being indicated is set as the new switch point. Then, the machine goes back to operation mode. Set the range appropriately, using the LED7 as a guideline.

■ Setting Setpoint (Temperature Monitoring)

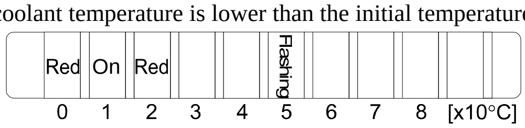
At the shipment, the temperature to activate the flowrate sensor is set to 15°C. Accordingly, you do not need to set it again in normal operation. However, if you reset the flowrate sensor, the default temperature setting (4°C) is restored. Use the following procedure to set it to 15°C.

[Procedure]

1. Press the button once to let the current temperature and temperature setting to be displayed.
2. Press the button to enter into the temperature setting mode.

   Lighting green (or orange): Indicates the current coolant temperature in unit of 10°C.
   Flashing green (or orange): Indicates the current coolant temperature in unit of 1°C.

   (When the current coolant temperature is higher than the initial temperature: Orange LED. When the current coolant temperature is lower than the initial temperature: Green LED)



Indicates the setting value in green or orange. (In the example, current setting is 15°C.)
Red LED indicates the current coolant temperature. The rightmost position shows the 10°C unit.
(In the example, current coolant temperature is in the range between 20 and 29°C.)

3. Press the (drop 1°C) or (raise 1°C) button several times until it reaches 15°C. The LED for indicating 10°C unit is turned on or off automatically.

   About five seconds have passed after the button is released, the position currently being indicated is set as the new switch point. Then, about five seconds later, the machine goes back to operation mode to monitor the flowrate.

[Note]

- If the button is not pressed for five seconds, the new setpoint is set, the temperature monitoring setpoint is displayed with new setting. After the additional five seconds later, the system goes back to flowrate monitoring mode.
- If an error is detected in setting procedure, all LEDs flash red. The system goes back to operation mode with the previous setting. When you finish setting, make sure that no LED is flashing red.

7.14.3 Operation check / Maintenance

[Procedure]

1. The machine is turned on.
   All the flow indication LEDs go on once. Afterwards, LEDs go off one by one, and the system goes to operation mode.
2. Turn the coolant discharge on.
3. Make sure the flowrate sensor shows the coolant temperature of at least 15°C.

   Press the ▶ button once to check the current coolant temperature indicated by the red LED.
   The red LED at the rightmost position indicates the coolant temperature in unit of 10°C.

   In the example on page 7-7-72, current coolant temperature is in the range between 20 and 29°C.
4. Check if the flowrate sensor works normally (by indications in operation mode).

   The green LED shows the flowrate change.

   The set point is displayed as follows.

   - Output ON: Orange LED
   - Output OFF: Red LED

[Note]

- If only LED 9 is flashing green while all other LEDs are lighting green, it indicates an excess flowrate.
- If the setpoint lights in red and LED0 lights in green, it indicates an insufficient flowrate.

■ Error indication

If the output 1 (flowrate monitoring) is short-circuited, indication of operation mode and five red LEDs (LED0 to LED4) are alternately displayed.

If the output 2 (temperature monitoring) is short-circuited, indication of operation mode and five red LEDs (LED5 to LED9) are alternately displayed.

■ Lock / Unlock

You can lock the setting value electrically to protect the settings from being modified.

Press the ◁ and ▶ buttons simultaneously for about 10 seconds lock (or unlock) the setting. (After 10 seconds, all LEDs go off for about 1 second.)

7.15 Adjustment and Handling of Medium Pressure Coolant Device

This machine comprises four systems: front machining on an opposite tool post, back machining on a back tool post, oil blower in back spindle and delivery of medium pressure coolant to the guide bushing device.

The device uptakes coolant from the coolant tank. By using the medium pressure coolant device, the coolant level may be lowered than the normal level. If you add coolant in the coolant tank, the coolant may overflow from the tank when the medium pressure coolant device is stopped. Be sure to keep the proper level of coolant when using the medium pressure coolant device.

7.15.1 Adjusting the nozzle

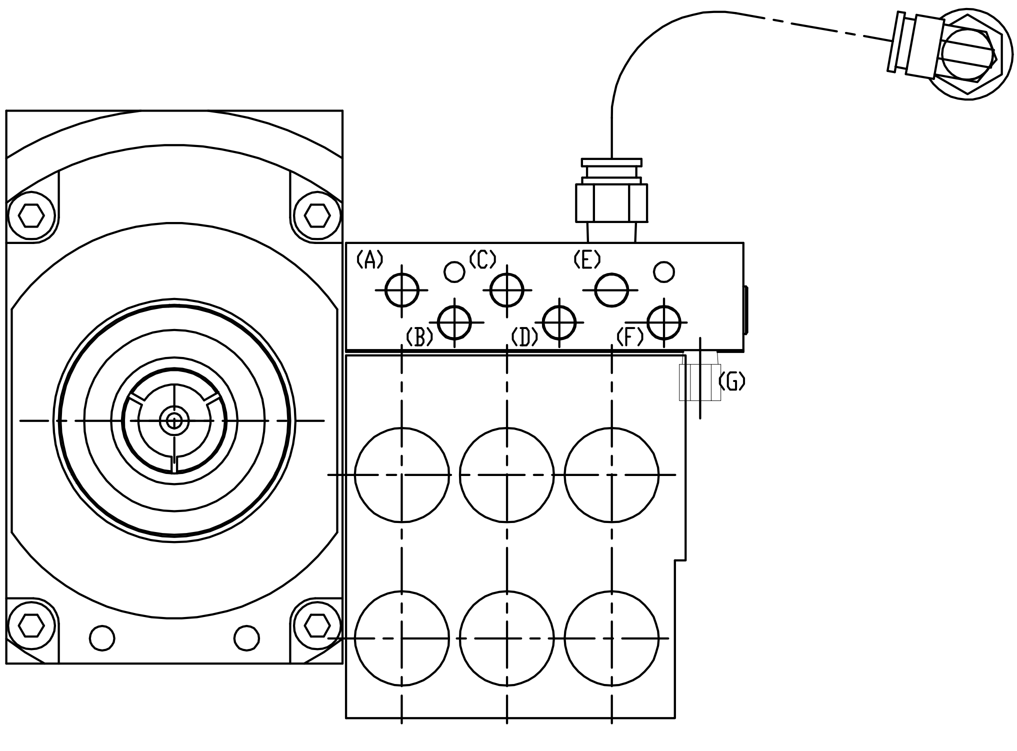
CAUTION

Be sure to confirm that the machine has stopped before attempting to adjust the position of nozzle.
The work during machine operation could result in serious personal injury.

■ Front Drill Holder

Coolant is supplied to front machining on the opposite tool post.

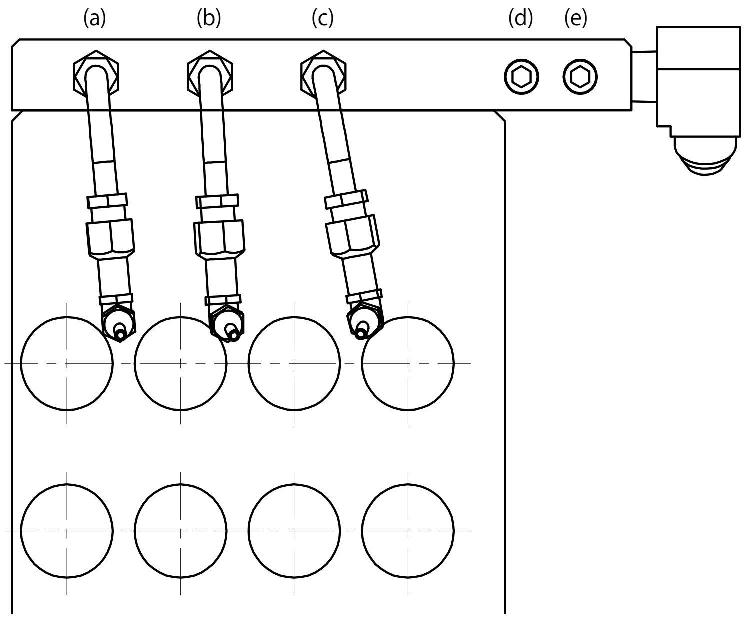
* The opposite tool post provides seven exhaust ports (A) to (G).
  Select exhaust port and joint as required by tool layout.
* Use three or fewer exhaust ports.
  Use of a large number of exhaust ports will reduce coolant exhaust pressure.
  Use the supplied plugs to cover exhaust ports that are not used.
* Fine adjust the nozzle so that coolant reaches the tip of the tool.



■ Back Drill Holder

Coolant is supplied to back machining on the back tool post.

* The back tool post provides six exhaust ports (A) to (e).
Select exhaust port and joint as required by tool layout.
* Use three or fewer exhaust ports.
Use of a large number of exhaust ports will reduce coolant exhaust pressure.
Use the supplied plugs to cover exhaust ports that are not used.
* Fine adjust the nozzle so that coolant reaches the tip of the tool.



■ Oil blower in back spindle

Coolant is discharged from the back spindle.

Use the knock-out jig supplied with the medium-coolant device. When the through-hole workpiece knock-out device is used together, use the knock-out jig provided with the through-hole workpiece knock-out device. If the blow hole is required, drill the hole additionally before using. See *<7.6.1 Mounting and replacing knock-out jig>* and *<Replacing the knock-out jig for through-hole workpieces>* of *<7.7 Rearrangement of through-hole workpiece>* for replacement of knock-out jig and dimensions of provided jigs.

When the medium pressure coolant device is mounted, air-driven back spindle blower can be used together.

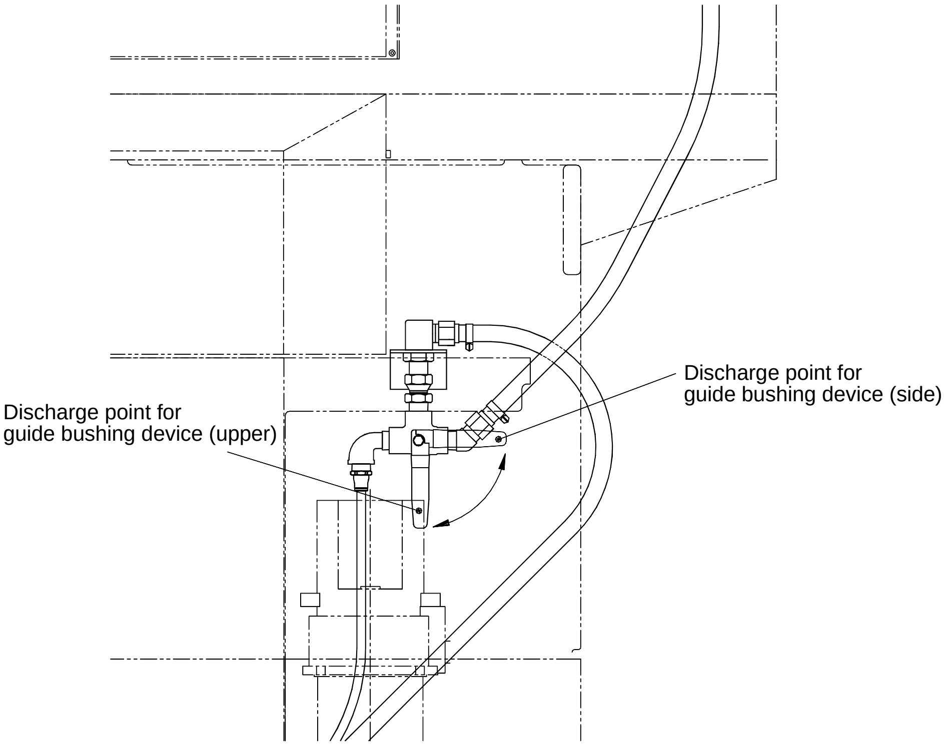
Reset the "KNOCK OUT OVERLOAD" alarm as usual according to *<7.6.3 Resetting knock-out overload alarm>*.

■ Guide bushing device

The coolant device supplies coolant to the guide bushing device (machining point).

The coolant can be supplied from the upper of the guide bushing device or from the side, by switching the valve in the machine rear side. Be sure to adjust the position of nozzle so that the coolant oil sprashes the trip of tool.

> **⚠ WARNING**
>
> Be sure to confirm that the machine has stopped before attempting to switch the valve.
> The work during machine operation could result in serious personal injury.



Guide bushing device (upper)

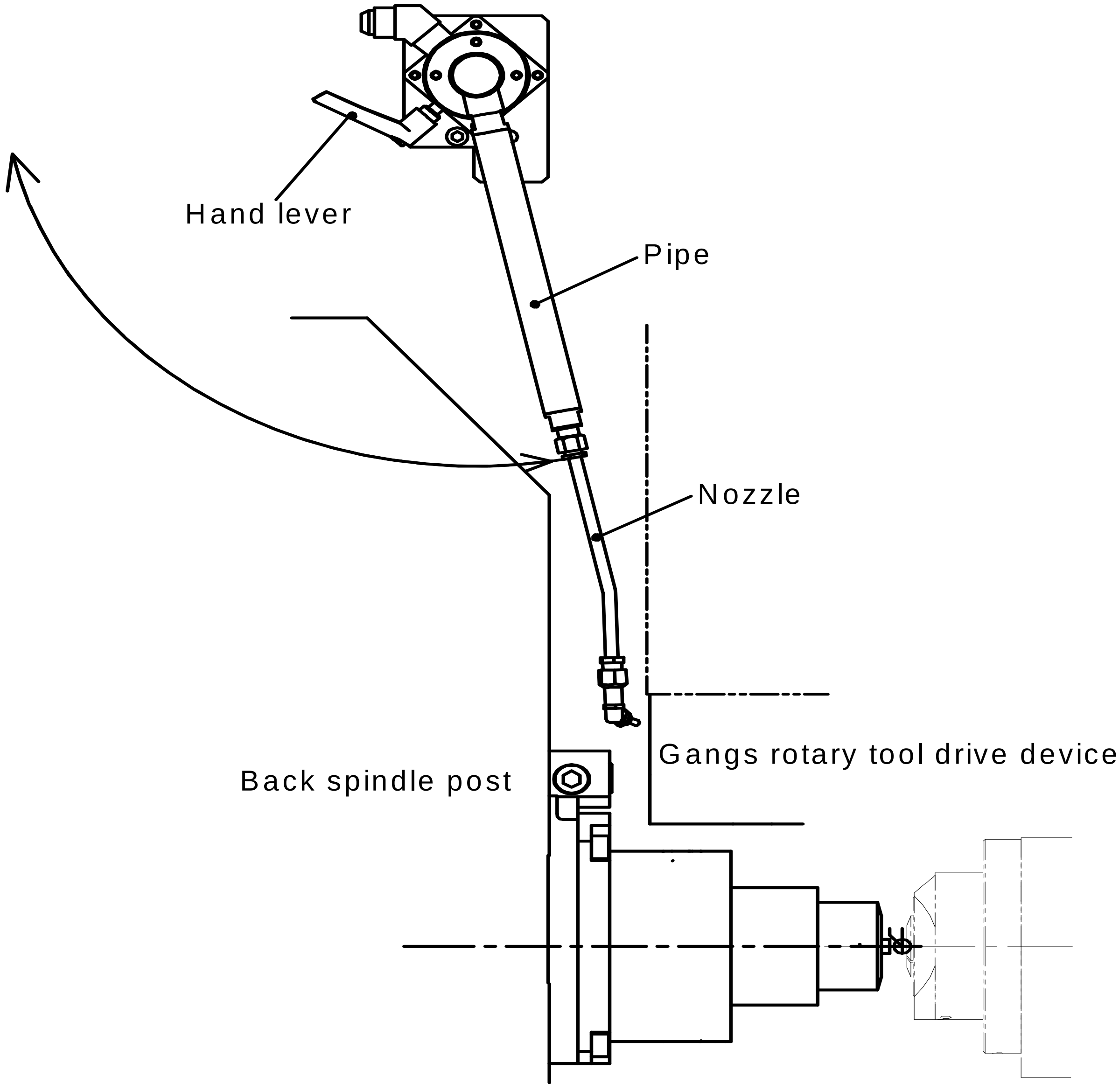
Rotate the hand lever rotates the pipe. Adjust the pipe and nozzle appropriate to the machining.

CAUTION

After adjustment of pipe, be sure to close the lever to fix the pipe. Otherwise, the pipe may fall during operation and cause damage to the machine.

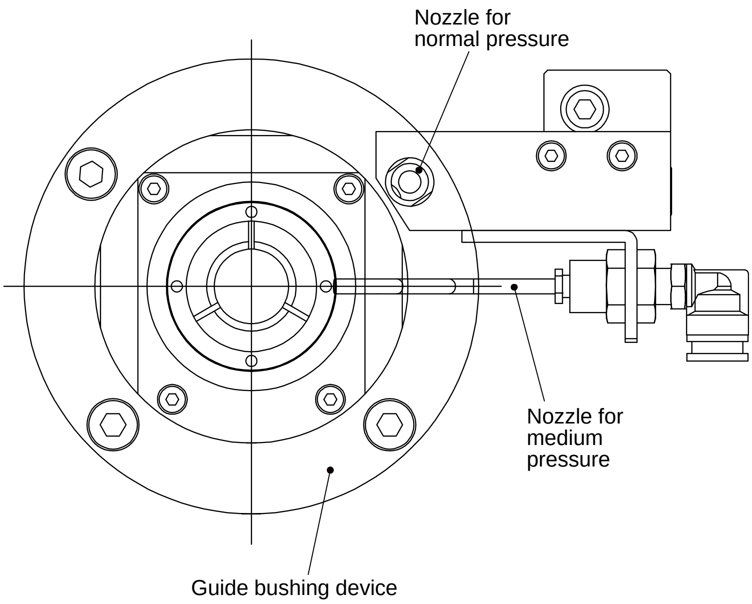
WARNING

Be sure to confirm that the machine has stopped before attempting to adjust the position of pipe.
The work during machine operation could result in serious personal injury.



Guide bushing device (side)

Adjust the position of the nozzle for supplying coolant, from the side of the guide bushing device.



7.15.2 Notes on using the separate pump unit

The medium pressure coolant device is equipped with a separate pump unit. The pump unit has many piping and wiring, e.g., suction tube from coolant tank, supply tube to the machine side, cables for connection with other devices. Be sure to arrange these piping and wiring by using the provided duct or others to secure the sufficient space where you can work safely.

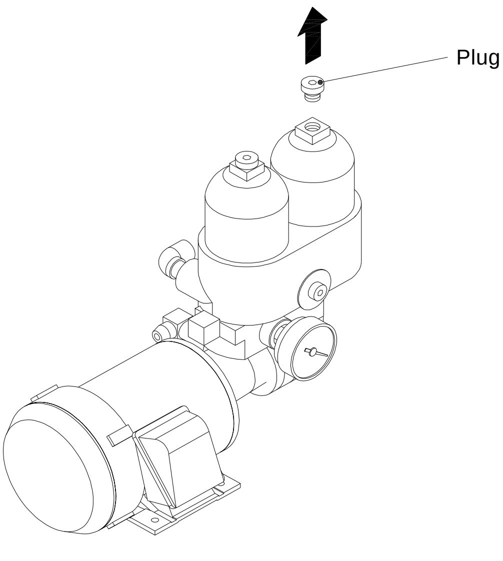
The separate pump unit has a filter to filtrate the chips and other substances contained in the coolant oil. Because the filter must be checked regularly, install the pump unit in the place where the sufficient work space for maintenance can be sucured.

WARNING

Be sure to check filter on a regular basis. Using the dirty or clogged filter may cause a serious accident such as a damage to the pump or a fire due to ignition of the pump.

Replacement of filter in trochoid pump

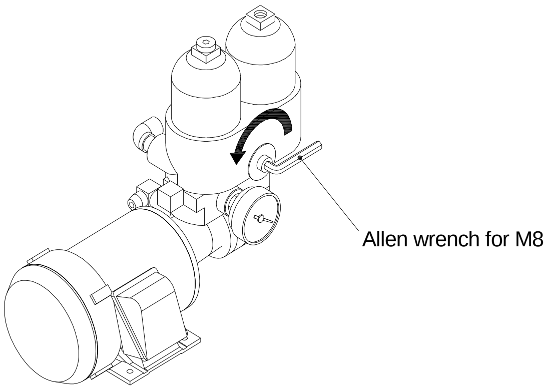
The filter in the cartridge mounted in upper portion of trochoid pump can be used repeatedly by cleaning. The time to clean the filter can be known from the indication of pressure gauge. See the Maintenance Manual, *<22.1 Periodical Check>* for details of cleaning.



[Procedure]

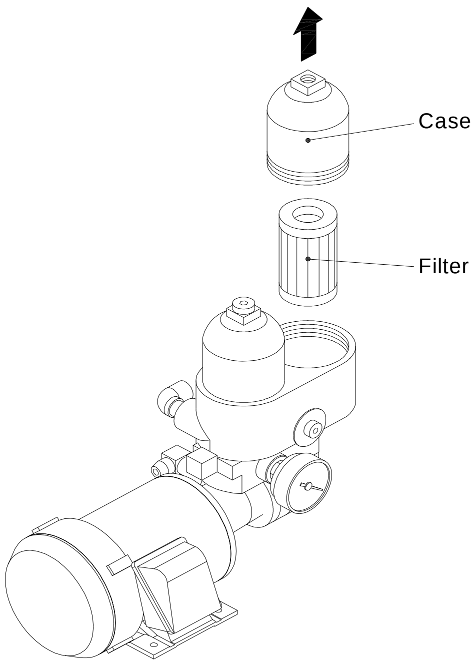
1. Stop the machine and trochoid pump.
2. Remove the plug from the top of the case.
   When this plug has removed, air is taken into the pump, and the coolant in the filter is sucked into the tank and the discharge port side. It takes about 10 seconds until the coolant in the filter is removed.

3. Switch the lever with allen wrench for M8.



4. Remove the case.

5. Take out the filter and clean it.



6. Clean the case. Remove chips and sludges adhered to the screws of the case.

7. Put the filter back to the original position.

8. Apply certain amount of grease to the screw position of the case, and mount the case.
Be careful not to damage the O-ring when mounting the case.

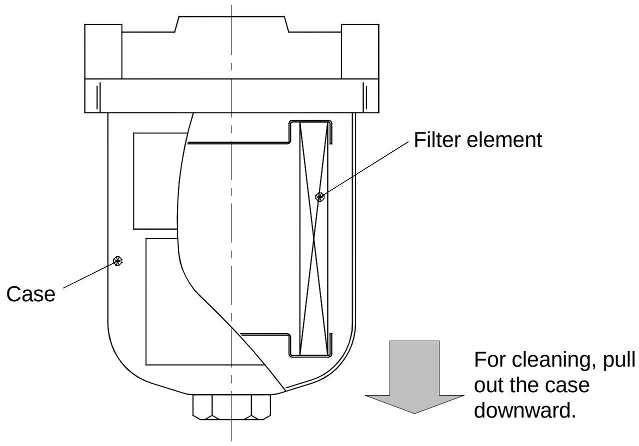
9. Mount the plug to the top of the case.

> **WARNING**
>
> If the filter has deformed or is badly clogged and hard to clean, be sure to replace the filter with new one. Using the deteriorated filter may cause a serious accident such as damage to the pump or fire due to ignition of the pump.

■ Replacing the line filter

The elements in the line filter can be used repeatedly by cleaning. The time to clean the filter can be known from the indication of pressure gauge. See <*22.1 Periodical Check*> for details of cleaning.

It is recommended to record an indication of pressure gauge when the element is clean and not clogged. For how to check the indication of pressure gauge, see <*Checking discharge pressure of trochoid pump*>.



[Procedure]

1. Stop the machine and trochoid pump.
2. Rotate the case of line filter and take it out downward.
   You will see the element.
3. Take the element out of the case, and clean it.
4. Clean the case. Remove chips and sludges adhered to the screws of the case.
5. Put the element back into the case.
6. Apply certain amount of grease to the screw position of the case, and mount the case.

> **WARNING**
>
> If the element has deformed or is badly clogged and hard to clean, be sure to replace the element with new one. Using the deteriorated element may cause a serious accident such as damage to the pump or fire due to ignition of the pump.

■ Checking discharge pressure of trochoid pump

The discharge pressure of trochoid pump can be checked by pressure gauge mounted on the line filter.

The value indicated by the pressure gauge of the line filter is different from the pressure of coolant when discarged from the nozzle tip. The pressure at the nozzle chip is reduced by the number of nozzles used, pile line resistance, and others. The pressure is also reduced due to clogged filter of trochoid pump. Use the indication of pressure gauge as a guideline.

[Procedure]

1. Stop the machine operation to prevent the valve from switching.
2. Discharge the medium pressure coolant.
3. Wait until the pointer of pressure gauge of the trochoid pump becomes steady.
4. Turn the valve.
5. You can check the pressure by viewing the pointer of the pressure gauge of the line filter.
6. After checking the pressure, turn the valve to its original position.

CAUTION

Be sure to fully close the valve for line filter pressure gauge (green) except when performing periodical check. If running the machine with the valve (green) being open, the pressure gauge may be damaged due to surge pressure.

■ Adjusting coolant nozzle length

When adjusting the length of coolant nozzle by reassembling the nozzles, follow the procedure explained below.

WARNING

When adjusting the coolant nozzle length, make sure to confirm that the machine is at a stop.

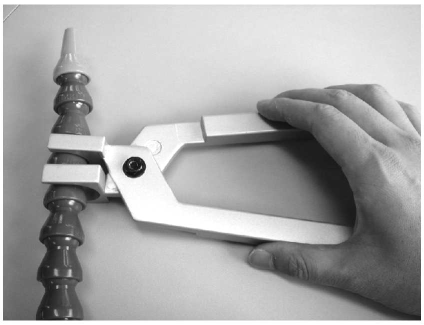
If an attempt is made to adjust the coolant nozzle length before stopping the machine, you may be caught by the machine to be injured seriously.
Use a dedicated tool when disengaging and engaging nozzles.

If you handle nozzles with bare hands, the hands may slip on the nozzles and be hit against cutting tools and/or parts in the machine to be seriously injured.

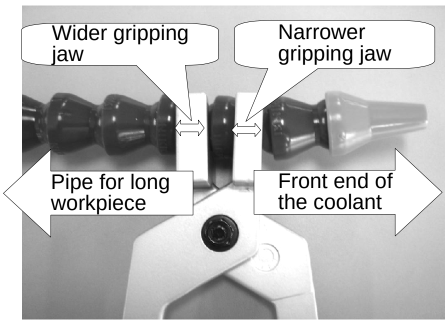
■ To remove the nozzle (using special tool)

[Procedure]

1. Insert the dedicated tool onto nozzles.

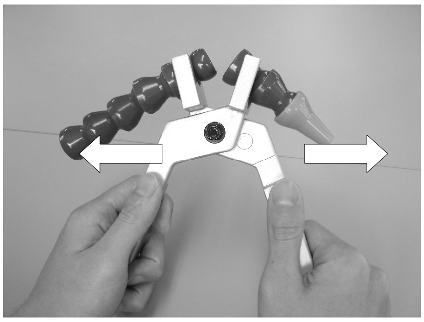


Insert the dedicated tool into nozzles in the correct direction.



2. Open the tool in the direction shown in the figure, and the nozzles are disengaged.

Be sure to operate the tool with both hands.

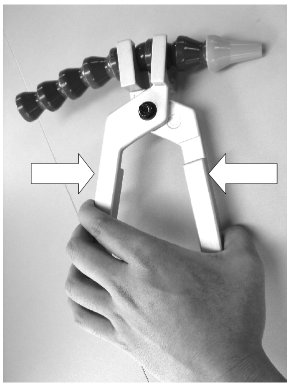


■ Engaging nozzles (using dedicated tool)

[Procedure]

1. Engage the nozzles by inserting the dedicated tool in the correct direction in the same manner as disengaging the nozzles.

   The nozzles will break if the tool is inserted in the incorrect direction.



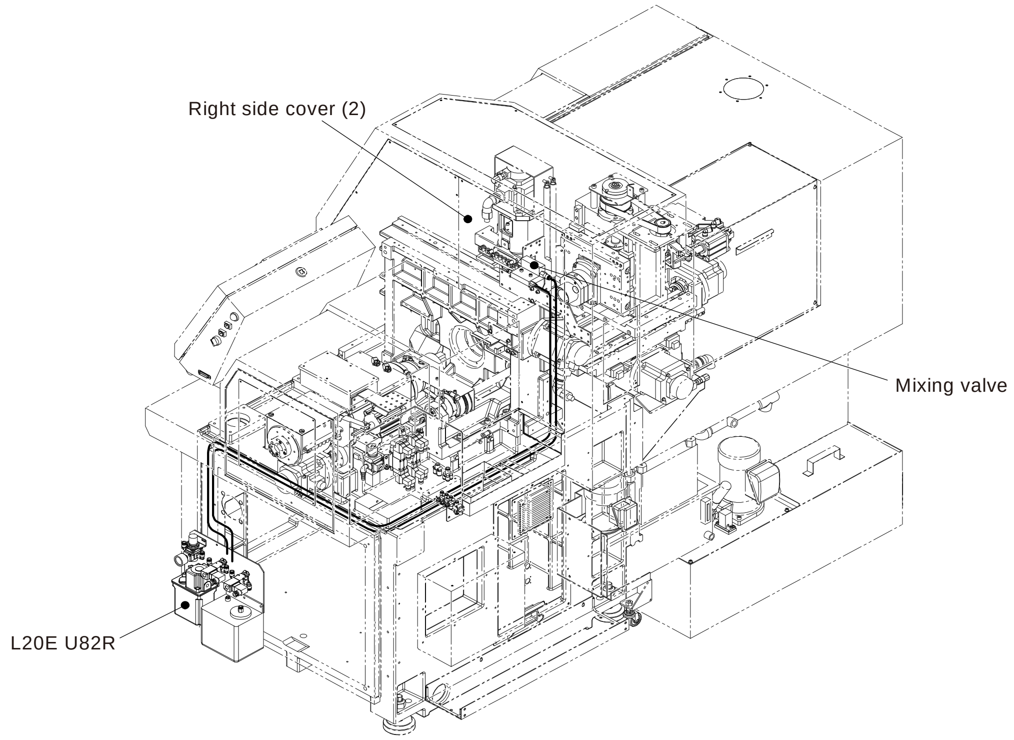
To purchase the dedicated tool, please contact our company.

Model of dedicated tool

- 26172 (Nikki Trading Crop.)

7.16 Adjustment of Oil Air Lubricant Unit (U82R)

The oil air lubricant unit blows air containing oil to the gear teeth in a rotary tool driver to keep good lubricant status.



[Note]

The unit operates in a cycle of 4-sec blow and 20-sec stop during rotary tool rotation (standard setting). If a rotary tool does not operate under automatic machine operation, the oil air blowing is not provided. However, this does not indicate the occurrence of a failure.

WARNING

Supply clean compressed air to the machine. If the compressed air contains some oil and/or foreign substances, oil air may be blown incorrectly to cause abnormal noises to occur and/or the machine life to be shorten. This may damage the machine seriously.

WARNING

After air is entered into piping in such a case as making the mixing valve empty, always exhaust the air from the piping and mixing valve. Without the air exhaust, oil air may be blown incorrectly to cause abnormal noises to occur and/or the machine life to be shorten. This may damage the machine seriously. See section *<7.16.4 Air Exhaust Procedure>* for the air exhaust procedure.

7.16.1 Names and functions of parts

The system is composed of an oil pump, a mixing valve, an electromagnetic valve for manipulating the oil pump, an electromagnetic valve for blowing oil air, and an air regulator and piping.

■ Oil pump

The oil pump neumatically feeds oil to the mixing valve with compressed air.

■ Mixing valve

The quatitative valve is intended for oil air lubricant by making oil into fine particles under transportation by air and supplying the particles to the lubricant point.

■ Air regulator

The air regulator is intended to adjust the air pressure at which oil is supplied to the lubricant point.

7.16.2 Supplying Lubricant

Check the oil quantity in the oil tank of the oil pump. If the oil quatity is close to the Low Level, supply clean lubricant oil to the High Level. The oil tank has the capacity of 1.8 litters.

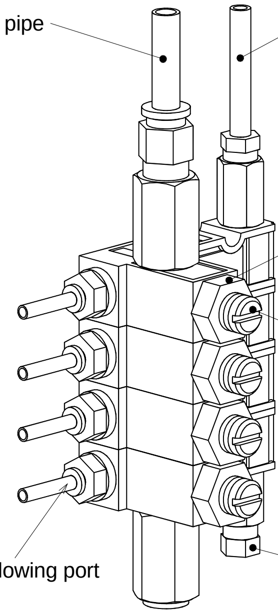
Recommended oil	:	Mobil Vactra Oil No. 2 or equivalent (viscosity: ISO VG68)
Reference	:	If a rotary tool is rotated continuously for 24 hours everyday, the full oil tank filled will become empty for about 12 days.

7.16.3 Adjusting Air Pressure

The oil air blowing pressure is normally set to 0.13 Mpa. Check the indicator of the air regulator.
Ajust the blowing pressure appropriately by turning the handle if necessary.

7.16.4 Air Exhaust Procedure

If air enters into the piping and/or mixing valve, the air must be exhaused. Remove the right side cover (2), exhaust the air from the mixing valve in the following procedure.



[Procedure]

1. Loosen the lock nuts and fasten the air valves clockwise to close them completely.
2. Exhaust the air from the main pipe. Loosen the air exhaust plug at the opposite side of the oil connection port on the mixing valve and push the manual button of the electromagnetic valve for manipulating oil pump to manipulating the pump. At first, white-turbid oil including air exudes from the air exhaust plug. The turbid gradually disappears.
3. After the air is exhausted completely, fasten the air exhaust plug. Now the air exhaust from the main pipe is completed.
4. Then exhaust the air from the mixing valve. Operate the pump until air is not exhausted any more from the blowing port on the mixing valve. When air is not found in the piping just after the blowing port, the air exhaust from the mixing valve is completed.
5. Loosen the air valves counterclockwise by 1.5 turns or more and fasten the lock nuts.
6. Check the transportation of oil by air near the piping connection at the blowing port.

(Blank Page)

Product Code	C-L220E XII
Document Code	1E1-0701
Mfg. No.	L220E/0001~
Issue Date	2014.1

8. Operating Procedures

Introduction

This manual is a summary of the Operator's Manual, provided to help you quickly search for specific operating procedures.

This manual does not include important information such as descriptions of safety precautions, operation panel controls, screen displays, preparations for machine operation, and of installation and adjustment.

Even when you have read through this manual, therefore, you must not operate the machine before reading the Operator's Manual.

Before operating the machine, be sure to read, understand, and follow the contents of the Operator's Manual, including <Chapter 2 Safety Precautions>. Periodically read safety information contained therein to make sure that the machine has been operating safely.

Citizen Machinery Miyano Co., Ltd. assumes no liability for any loss or damage, injury, or death, arising out of failure to read or follow the safety precautions and other relevant information in the Operator's Manual or out of unauthorized use or handling of the machine.

8.1 Flow of Setup

[Procedure]

1. Check the drawing for the product to be machined and determine the machining direction in consideration of the features and characteristics of the machine.
2. Determine the layout.
3. Prepare the requirements such as tools, spindle chuck, bushings, back chuck, and drill chuck.
4. Create a program.
5. Turn the machine on and load the program.
6. Input machining data.
7. Remove the tools used for the previous session of machining. They can be used as they are without being removed if they can serve for machining to be performed.

[Note]

Be sure to remove the tool if it is a back machining tool (T31 to T37).

8. Replace and adjust chucks and bushings and replace the back chuck and knock-out jig. Also replace fingers if the machine has an automatic bar loader.
9. Select the program and move to the start position.
10. Run the machine with no load a material to check the program and machine operations.
    Use On-Machine Check to make this step easier.
11. Perform tool setting.
12. Run the machine with no load a material to check operations.
13. Load a material for checking by actually cutting it.
    If back machining is required for the material without back chucking, modify the program not to chuck the workpiece at the back spindle.
14. Execute one cycle of operation to correct the diameter, longitudinal, and other required dimensions of the workpiece by modifying the program and using the compensation function.
15. When all of the front machining dimensions is corrected, reprogram the machine to execute back chucking.
16. Perform back tool setting.
17. Execute one cycle of operation to check dimensions of the workpiece and operations, then correct the dimensions by modifying the program and using the compensation function.
18. When you have finished taking the above steps, execute continuous cycles to machine several workpieces to check dimensions.
19. Set the workpiece counter, then machine them continuously.

8.2 Procedure from Turning the Power on to Automatic Operation

[Procedure]

1. Set the circuit breaker switch to the ON position.
   The Power lamp on the right side of the operation panel goes on.
2. Press the Power ON button I on the operation panel.
3. Door is locked.
4. The initial screen appears. After several ten seconds, the Preparation key PREPA. goes on and the Preparation screen and Program screen are displayed.
5. Enter the desired program number in the data setting field.
6. Press the Input key INPUT.
   The Machining Data screen appears. Check the settings, then press the Escape key ESC.
   The program number is displayed at the upper right part of the screen.
7. Use the Main Spindle Chuck key for manual operation to open the spindle chuck.
   Make sure that the Main Spindle Chuck key LED is off.
8. Press the Door open button to unlock the door, and open the door.
9. Pull out the material to the end face of the spindle.

CAUTION

If the material is left without being pull out, the material may interfere with the tool.

10. Close all doors.
11. Press the menu key [ST POS].
    The Start Position screen appears.

WARNING

Leave the safety cover closed while the lamp on the Start button is on. Opening the safety cover with the LED on may result in a serious injury or death.

12. Press the Start button .
    The lamp on the Start button blinks and the doors are locked.
13. Press the Start button .
    The lamp on the Start button lights and each axis moves to the start position.
    Upon completion of the move, the lamp on
14. Press the Door open button to unlock the door, and open the door.
15. Press the material slowly to the cut-off tool.
    (When the gap of the guide bushing is set narrowly, cut-off the end face of the material.)
16. Close all doors.
17. Press the Auto key AUTO as an Operation Mode Selection key.
    The lamp on the Auto key goes on and the Automatic Operation screen appears.
18. Press the Coolant button .
    The lamp on the Coolant button goes on and coolant is discharged.

CAUTION

The amount of discharged coolant or the coolant discharging position is inappropriate, the guide bushing or tool may be damaged.

19. Press one of the menu key [Cont.], [1 Cycle], or [1 Block], which you required.

20. Make sure that all doors are closed, and press the Start button .
The lamp on the Start button blinks and the doors are locked.

21. Press the Start button .

The lamp on the Start button goes on and the program is executed for one cycle for 1 Cycle or for one program line for 1 Block.

8.3 Procedure from Continuous Operation to Turning the Power Off

[Procedure]

1. Press the menu key [1 Cycle].

   The program is executed for only one cycle.

2. Wait until the lamp on the Start button goes off.

   The lamp will go off, indicating the end of the single cycle.

3. Press the Coolant button .

   The lamp on the coolant key goes off, indicating that coolant discharge has now been stopped.

4. Press the Power OFF button on the operation panel.

5. Set the circuit breaker switch to the OFF position.

8.4 Offset (Compensation) Data Input Procedure

[Procedure]

1. Press the Offset key .

   The last screen selected appears.
2. Press the menu key [Offset].

   The Offset screen appears.
3. If the "INC" is not displayed on the upper right corner of the screen, press the menu key [ABS/INC].

   The "INC" is reversed.
4. Use the arrow keys to move the cursor to the offset number you want to enter.
5. Use the tab keys to move the cursor to the desired input field.
6. Enter a numeric value as offset data.
7. Press the Input key .

   Check the input offset data.
   - Press the Shift key and the Line deletion key to set the offset value for one screen to 0.

8.5 MDI Operating Procedure

[Procedure]

1. Press the MDI key MDI.

   The lamp on the MDI key goes on and the MDI screen appears.
2. Use the menu key [$-Select] to select $1 (X1, Z1, Y1, C1) or $2 (X2, Z2, C2).
3. Press the menu key [1 Block] or [All Block].

   1 Block: Allows input of multiple blocks and execution per block.

   All Block: Allows input and continuous execution of multiple blocks.
4. Enter the command address and value.
5. Press the Input key INPUT.
   "Set COMP" appears at the lower right corner of the screen.

WARNING

Leave the safety cover closed while the lamp on the Start button is on. Opening the safety cover with the LED on may result in a serious injury or death.

6. Close all doors.
7. Press the Start button.
   The lamp on the Start button blinks and the doors are locked.
8. Press the Start button.

   The lamp on the Start button goes on and the MDI program is executed.
   - All the programs are deleted by pressing SHIFT CAN INPUT.

8.6 Procedure for Program Input from External Device

8.6.1 Input from Front Card Slot

[Procedure]

1. Insert the CF card into the front card slot.
2. Set the Program edit switch key to "I".
   The program is released from the protected state.
3. Press the Edit key.
   The last screen selected appears.
4. Press the menu key [List].
   The List screen appears.
5. Press the menu key [I/O].
   The last screen selected appears.
6. Press the menu key [File I/O].
   The File Access screen appears.
7. Press the menu key [Input].
8. Use the arrow keys to move the cursor to "External" folder.
   The "CF Card 1" is displayed.
9. Enter the number for "CF Card 1", then press the Input key.
   The folder is confirmed and the cursor moves to the "External" directory.
   (The screen shows the contents of the selected directory. With the new card, the contents of the directory are empty.)
10. Enter the relevant directory number.
    Example) Press the key "1".
11. Press the Input key.
    The directory is confirmed.
12. Use the arrow keys to move the cursor to "program" in "External" field. Enter the number for the target file.
13. Press the Input key.
    The program is confirmed.
14. Press the Input key.
    Now, the program input is executed.

8.6.2 Input from USB Port

[Procedure]

1. Insert the USB flash drive into the front USB port.
2. Set the Program edit switch key to "I".
   The program is released from the protected state.
3. Press the Edit key.
   The last screen selected appears.
4. Press the menu key [List].
   The List screen appears.
5. Press the menu key [I/O].
   The last screen selected appears.
6. Press the menu key [File I/O].
   The File Access screen appears.
7. Press the menu key [Input].
8. Use the arrow keys ⬆⬇ to move the cursor to "External" folder.
   The "CF Card 2" is displayed.
9. Enter the number for "CF Card 2", then press the Input key.
   The folder is confirmed and the cursor moves to the "External" directory.
   (The screen shows the contents of the selected directory. With the new card, the contents of the directory are empty.)
10. Enter the relevant directory number.
    Example) Press the key "1".
11. Press the Input key.
    The directory is confirmed.
12. Use the arrow keys ⬆⬇ to move the cursor to "Program" in "External" field.
    Enter the number for the target file.
13. Press the Input key.
    The program is confirmed.
14. Press the Input key.
    Now, the program input is executed.

8.6.3 Input Using RS232C Interface

[Procedure]

1. Connect the cable from an external device to the RS232C connector.
   The RS232C connector is provided on the right side of the operation panel.
2. Set the Program edit switch key to "|".
   The program is released from the protected state.
3. Press the Edit key.
4. Press the menu key [List].
   The List screen appears.
5. Press the menu key [I/O].
   The Input/Output screen appears.
6. Press the menu key [RS232C].
   The RS232C screen appears.
7. Press the menu key [Input].
8. Use the arrow keys to move the cursor to "Comm. Set" in "External Device" field. Enter the number for the target parameter.
9. Press the Input key.
   The communication setting is confirmed and the cursor moves to "I/O Port" in "External Device" field.
10. Enter the number for the I/O Port, then press the Input key.
    The I/O port is confirmed.
11. Press the Input key.
    Data input starts.
12. Turn the external device on and enable it for output.
    The program is input to the machine.

8.7 Procedure for Program Output to External Device

8.7.1 Output to Card Slot

[Procedure]

1. Insert the CF card into the front card slot.
2. Set the Program edit switch key to "I".
   The program is released from the protected state.
3. Press the Edit key.
4. Press the menu key [List].
   The List screen appears.
5. Press the menu key [I/O].
   The Input/Output screen appears.
6. Press the menu key [File I/O].
   The File Access screen appears.
7. Press the menu key [Output].
8. Use the arrow keys ⬆ ⬇ to move the cursor to "External" folder.
   The "CF Card 1" is displayed.
9. Enter the number for "CF Card 1", then press the Input key.
   The folder is confirmed and the cursor moves to the "External" directory.
   (The screen shows the contents of the selected directory. With the new card, the contents of the directory are empty.)
10. Enter the relevant directory number.
    Example) Press the key "1".
11. Press the Input key.
    The directory is confirmed.
12. Use the arrow keys ⬆ ⬇ to move the cursor to "NC" program and enter the O number of the relevant program.
13. Press the Input key.
    The program is confirmed.
14. Press the Input key.
    Now, the program output is executed.

8.7.2 Output to USB Port

[Procedure]

1. Insert the USB flash drive into the front USB port.
2. Set the Program edit switch key to "I".
   The program is released from the protected state.
3. Press the Edit key.
4. Press the menu key [List].
   The List screen appears.
5. Press the menu key [I/O].
   The Input/Output screen appears.
6. Press the menu key [File I/O].
   The File Access screen appears.
7. Press the menu key [Output].
8. Use the arrow keys ⬆ ⬇ to move the cursor to "External" folder.
   The "CF Card 2" is displayed.
9. Enter the number for "CF Card 2", then press the Input key.

   The folder is confirmed and the cursor moves to the "External" directory.
   (The screen shows the contents of the selected directory. With the new card, the contents of the directory are empty.)
10. Enter the relevant directory number.
    Example) Press the key "1".
11. Press the Input key.
    The directory is confirmed.
12. Use the arrow keys ⬆ ⬇ to move the cursor to "NC" program and enter the O number of the relevant program.
13. Press the Input key.
    The program is confirmed.
14. Press the Input key.
    Now, the program output is executed.

8.7.3 Output Using RS232C Interface

[Procedure]

1. Connect the cable from an external device to the RS232C connector.

2. Set the Program edit switch key to "|".
The program is released from the protected state.

3. Press the Edit key.

4. Press the menu key [List].
The List screen appears.

5. Press the menu key [I/O].
The Input/Output screen appears.

6. Press the menu key [RS232C].
The RS232C screen appears.

7. Press the menu key [Output].

8. Use the arrow keys to move the cursor to "Comm. Set" in "External Device" field. Enter the number for the target parameter.

9. Press the Input key.
The communication setting is confirmed and the cursor moves to "I/O Port" in "External Device" field.

10. Enter the number for the I/O Port, then press the Input key.
The I/O port is confirmed and the cursor moves to "NC memory" in "NC" field.

11. Enter the O number of the relevant file (program).

12. Press the Input key.
The "NC" program is confirmed.

13. Press the Input key.
Data output starts.

14. Turn the external device on and enable it for input.
The program is output from the machine.

8.8 Program Creating Procedure

[Procedure]

1. Set the Program edit switch key to "I".
The program is released from the protected state.

2. Press the Edit key.

3. Press the menu key [List].
The List screen appears.

4. Press the menu key [NEW-INP].
The Create New Program window appears.

5. Enter the number and comment for the new program.
Use the arrow keys to move to the comment field.

6. Press the Input key.

The program number edit screen appears for the new program to be created.
The program numbers currently being selected for editing is displayed at the upper right corner of the screen, and automatic operation is displayed at the upper left corner.

8.9 Program Calling Procedure

[Procedure]

1. Press the Edit key .
2. Press the menu key [List].
   The List screen appears.
3. Select the program number you want to call by using the arrow keys or enter the program number in the program number field.
4. Press the Input key .
   The called program edit screen appears.

8.10 Program Editing Procedure

[Procedure]

1. Set the Program edit switch key to "|".
   The program is released from the protected state.
2. Press the Edit key .
   The last screen selected appears.
3. Display the edit screen of the program to be edited with reference to *<8.8. Program Creating Procedure>* and *<8.9. Program Calling Procedure>*.
4. Use alphanumeric keys to enter the address and numeric values, then press the Input key .
   - When inputting or modifying the program, make sure to press the Input key.
     Otherwise, the modification is not stored.
   - Delete key : Deletes the character to the right of current cursor position.
   - Line Deletion key : Deletes the line currently containing the cursor.
   - Insert key : Toggles the input mode between "insert" and "overwrite" whenever pressed. The current input mode is displayed at the lower right corner of the screen.
   - Backspace key : Deletes the character to the left of the current cursor position.

8.11 Program Deleting Procedure

[Procedure]

1. Set the Program edit switch key O I to "I".
The program is released from the protected state.

2. Press the Edit key EDIT.
The last screen selected appears.

3. Press the menu key [List].
The List screen appears.

4. Press the menu selection key ▶ to enable the sub menu.

5. Press the menu key [Delete].
The Program Delete screen appears.

6. Enter the number of the program you want to delete.

7. Press the Input key INPUT.

8. The menu keys [Yes] and [No] are displayed. Press the menu key [Yes].
The specified program is deleted.

8.12 Machining Data Setting Procedure

Machining data is required for machining, preparation, and for tool setting. It is handled in almost the same way as machining programs; it is registered in memory. Machining data can be created or edited in the background, except the machining data currently being used for execution for machine operation.

[Procedure]

1. Set the Program edit switch key to "I". The program is released from the protected state.
2. Press the Edit key . The last screen selected appears.
3. Perform the previous procedures to *<8.8. Program Creating Procedure>* and *<8.9. Program Calling Procedure>* for the program for which you want to input machining data.
4. Press the menu selection key to enable the sub menu.
5. Press the menu key [MC-Data]. The Machining Data screen appears.
6. Use the page keys the arrow keys to move the cursor to the field for the item you want to specify.
7. Enter a numeric value.
8. Press the Input key . The cursor automatically moves to the next setting field.
9. For the holder item, enter the relevant number of the holder name you want to select.
10. Press the Input key . The cursor automatically moves to the next setting field.
11. Specify all items.

8.13 Counter Setting Procedure

[Procedure]

1. Press the Auto key AUTO as an Operation Mode Selection key. The Automatic Operation screen appears.
2. Press the menu selection key ▶ to enable the sub menu.
3. Press the menu key [Counter]. The Counter window appears.
4. Press the menu key [Cnt Set].

   The field for each item in the Counter window is enabled for set.
5. Use the arrow keys ⬆ ⬇ or the tab keys TAB ⬅ TAB ➡ to move the cursor to the desired field.
6. Enter a numeric value.
7. Press the Input key INPUT.
8. Press the menu key [Set Cmpl]. The field for each item in the Counter window is disabled for set.
9. Press the menu key [Counter]. The Counter window is closed.

8.14 Tool Setting Procedure

> **CAUTION**
>
> Described below are operation procedures when the Setting switch key is set to "O".
>
> If you are going to work inside the machine, press the Door open button to unlock the door.
>
> The door cannot be unlocked unless the following conditions are met.
>
> - All spindles have stopped.
> - All servo axes have stopped.
> - The machine is not operating.
> - The coolant is not discharged:
> - The Door open button is pressed.

8.14.1 Gang Tool Setting Procedure (T01's tool)

> **CAUTION**
>
> In the tool setting function, positioning of parts such as holders are performed based on machining data.
> Before performing tool setting, be sure to check machining data for validity. If machining data contains an error, the tools, holders, and material may interfere with one another.

[Procedure]

1. Press the Preparation key PREPA..

The Preparation screen appears.

2. Press the menu selection key ▶ to enable the sub menu.

3. Press the menu key [PRG SEL].

The Program Select screen appears.

4. Use the arrow keys ⬆ ⬇ to select the desired program number.

5. Press the Input key INPUT.

The Machining Data screen is opened automatically.

6. Check machining data.

To change any content of machine data, follow the machining data input procedure in <8.12. Machining Data Setting Procedure>.

7. Press the Escape key ESC. The Machining Data screen is closed.

8. Draw back the material beyond the end face of the guide bushing.

> **CAUTION**
>
> If the material is not drawn back sufficiently, the material and tool may interfere with each other.

9. Use the arrow keys ⬆ ⬇ to move the cursor to the number of the tool you want to set.

Use the tab keys TAB ⬅ TAB ➡ to move the cursor to the "Core" in the window.

10. Press the menu selection key ▶ to enable the main menu.

11. Close all doors.
12. Press the menu key [Man. Set]. The Manual Tool Set screen appears.
13. Press the arrow key ⬆. Two cursors are displayed and the list of the moving position is displayed.
14. Enter the number for "Core", then press the Input key INPUT.

WARNING

Do not open the safety cover or touch any tool or material while the lamp on the Start button is on. Doing either or both may result in a serious injury or death.

15. Press the Start button . The lamp on the Start button will blinks on and off, and the doors will lock.
16. Press the Start button .
The holder moves to the core adjusting position for the called tool number.
The Manual Handle screen appears.
17. Press the Door open button to unlock the door, and open the door.
18. Set the material about 10 mm ahead of the end face of the guide bushing.
19. Fit the tool shank holder securely onto the tool mounting surface, then apply the tool nose slowly to the outer circumference of the material.
20. Use the pulse handle to adjust the tool nose to fit the outer circumference of the material.
The distance adjusted by the pulse handle is displayed at the position of the core in the window.
    - If the mounted tool is right-handed, turning the pulse handle to the negative (–) side lowers the core while turning it to the positive (+) side raises the core.
21. Press the Input key INPUT. The adjusted value is input to the position of the core.
22. Press the Escape key ESC. The Manual Handle screen is closed.
23. Close all doors.
24. Press the menu key [Man. Set]. The Manual Tool Set screen appears.
25. Use the arrow keys ⬆ ⬇ to move the cursor to the "DIA" in the window.
26. Draw back the material to the end face of the guide bushing.

CAUTION

If the material is not drawn back sufficiently, the material and tool may interfere with each other.

WARNING

Do not open the safety cover or touch any tool or material while the lamp on the Start button is on. Doing either or both may result in a serious injury or death.

27. Press the Start button . The lamp on the Start button will blinks on and off, and the doors will lock.
28. Press the Start button .
The holder moves to the outer diameter set position for the called tool number.
The Manual Handle screen appears.
29. Press the Door open button to unlock the door, and open the door.
30. Set the material about 10 mm ahead of the end face of the guide bushing.
31. Fit the tool shank holder securely onto the tool mounting surface, then apply the tool nose slowly to the outer diameter of the material.
32. Secure the tool temporarily using the tool clamp screw of the holder.
33. Close all doors.
34. Press the menu key [POS PNT]. The Positioning Point screen appears.

35. Press the Start button [Start]. The lamp on the Start button will blinks on and off, and the doors will lock.

36. Press the Start button [Start]. The lamp on the Start button goes on and the tool holder moves to the positioning point.

37. Press the Door open button [Door open] to unlock the door, and open the door.

38. Tighten the tool clamp screw.

39. Draw back the material beyond the end face of the guide bushing.

40. Take steps 9. to 30. for other gang tools to set them in the same way.

[Note]

- Releasing the Start button stops axis feed in the hold state and the lamp on the Start button blinks (on for two seconds and off for one second). To restart the operation, press the Start button.
- To enable operation while the door is open, press the Door open button [Door open] to unlock the door, and set the Setting switch key to the "I" position.
  If this switch key is set to the "O" position, machine operation is prohibited while the door is open.
- The unit will only operate with the door open while the Start button [Start] is pressed.
  The axes speed at this time is limited to 2 m/min or less.
- If no operation is performed within 15 seconds after making menu selections, menu selections will automatically be canceled.

  Menu selections will also be canceled if a reset is performed during or after operations. To restart operations, be sure to again make selections with the menus.
- If the door is open:
  The machine operates only when you are holding down the Start button.
  The machine will stop when you release the key.
  The Start button lamp will blink (on for two seconds and off for one second).
  Pressing the Start button when the lamp is blinking will restart the machine.
- When performing adjustment for core (Y), diameter (X) and center (X) using the handle, selection of multiplication ratio ×100 is not possible.
- The menu key [Man. Set] is disabled if no program is selected.

8.14.2 Gang Rotary Tool Setting Procedure

If you perform this procedure for setting rotary tools, following the procedure in <8.14.1 Gang Tool Setting Procedure (T01's tool)>, you can skip steps 1. to 7. below.

[Procedure]

1. Press the Preparation key PREPA..

The Preparation screen appears.

2. Press the menu selection key to enable the sub menu.

3. Press the menu key [PRG SEL].

The Program Select screen appears.

4. Use the arrow keys to select the desired program number.

5. Press the Input key INPUT.

The Machining Data screen is opened automatically.

6. Check machining data.

To change any content of machine data, follow the machining data input procedure in <8.12. Machining Data Setting Procedure>.

7. Press the Escape key ESC.

The Machining Data screen is closed.

8. Draw back the material beyond the end face of the guide bushing.

> **CAUTION**
>
> If the material is not drawn back sufficiently, the material and tool may interfere with each other.

9. Press the menu selection key to enable the main menu.

10. Use the arrow keys to move the cursor to the number of the tool you want to set.

11. Close all doors.

12. Press the menu key [Man. Set].

The Manual Tool Set screen appears.

13. Use the arrow keys to move the cursor to the "DIA" in the window.

14. Enter the rotary tool shift amount (in the diametrical direction).

To set the amount of protrusion of the tool longer than the standard length, enter a positive (+) value. To set it shorter, enter a negative (–) value. If the normal setting is fine, you do not needed to execute this step.

- You should remember the entered shift amount because it is used in a procedure.

15. Press the Input key INPUT.

> **WARNING**
>
> Do not open the safety cover or touch any tool or material while the lamp on the Start button is on. Doing either or both may result in a serious injury or death.

16. Press the Start button.
The lamp on the Start button will blinks on and off, and the doors will lock.

17. Press the Start button.
The lamp on the Start button goes on and the holder moves to the outer diameter set position for the called tool number. .

18. Press the Door open button to unlock the door, and open the door.

19. Set the material about 20 mm ahead of the end face of the guide bushing.

20. Apply the tool nose slowly to the outer diameter of the material.

21. Tighten the cap nut for chucking the tool using the dedicated tool.

22. Close all doors.

23. Press the menu key [POS PNT].

The Positioning Point screen appears.

24. Press the Start button .
The lamp on the Start button will blinks on and off, and the doors will lock.

25. Press the Start button .
The lamp on the Start button goes on and the tool holder moves to the positioning point.

26. Press the Door open button to unlock the door, and open the door.

27. Draw back the material beyond the end face of the guide bushing.

28. Take steps 10. to 21. for other rotary tools to set them in the same way.

> **CAUTION**
>
> If the tool positioning point value as machining data is incorrect, the tool and material may interfere with each other.

[Note]

- To enable operation while the door is open, press the Door open button to unlock the door, and set the Setting switch key to the "I" position.
  If this switch key is set to the "O" position, machine operation is prohibited while the door is open.
- The unit will only operate with the door open while the Start button is pressed.
  The axes speed at this time is limited to 2 m/min or less.
- Releasing the Start button stops axis feed in the hold state and the lamp on the Start button blinks (on for two seconds and off for one second). To restart the operation, press the Start button.
- If no operation is performed within 15 seconds after making menu selections, menu selections will automatically be canceled.
  Menu selections will also be canceled if a reset is performed during or after operations. To restart operations, be sure to again make selections with the menus.
- If the door is open:
  The machine operates only when you are holding down the Start button.
  The machine will stop when you release the key.
  The Start button lamp will blink (on for two seconds and off for one second).
  Pressing the Start button when the lamp is blinking will restart the machine.

8.14.3 Front Drilling Tool Setting Procedure (T21's tool)

[Procedure]

1. Take steps 1 to 31 in <8.14.1 Gang Tool Setting Procedure (T01's tool)> in the same way to set the cut-off tool to T01.
2. Use the arrow keys ⬆⬇ to move the cursor to T01.
3. Close all doors.
4. Press the menu key [POS PNT].

   The Positioning Point screen appears.
5. Press the Start button .
   The lamp on the Start button will blinks on and off, and the doors will lock.
6. Press the Start button .
   The cut-off tool is positioned at the positioning point.
7. Set the material so that its portion to be cut off protrudes from the cut-off tool.
   Press the Door open button if you need to open the door.
8. Press the Main Spindle Chuck key .

   (Check that the spindle chuck has already been adjusted.)
   The lamp on the Main Spindle Chuck key goes on, then the material is chucked.
9. Press the menu key [Cut-Off].

   The Cut-Off screen appears.

[Note]

- To enable operation while the door is open, press the Door open button to unlock the door, and set the Setting switch key to the "I" position.
  If this switch key is set to the "O" position, machine operation is prohibited while the door is open.
- The unit will only operate with the door open while the Start button is pressed.
  The axes speed at this time is limited to 2 m/min or less.
- If no operation is performed within 15 seconds after making menu selections, menu selections will automatically be canceled.
  Menu selections will also be canceled if a reset is performed during or after operations. To restart operations, be sure to again make selections with the menus.
- If the door is open:
  The machine operates only when you are holding down the Start button.
  The machine will stop when you release the key.
  The Start button lamp will blink (on for two seconds and off for one second).
  Pressing the Start button when the lamp is blinking will restart the machine.

10. Close all doors.
11. Press the Coolant button . The lamp on the Coolant button goes on and coolant is discharged.

⚠ CAUTION

The amount of discharged coolant or the coolant discharging position is inappropriate, the guide bushing or tool may be damaged.

⚠ WARNING

Do not open the safety cover or touch any tool or material while the lamp on the Start button is on. Doing either or both may result in a serious injury or death.

12. Press the Start button . The lamp on the Start button will blinks on and off, and the doors will lock.

13. Press the Start button .
The cut-off tool moves to the positioning point and the Start button blinks on and off. Check the cut-off tool, material, coolant oil, and so on again.

14. Press the menu key [Cut-Off] and the Start button simultaneously.
The lamp on the Start button lights and the tool cuts off the material.
Wait until the lamp on the Start button goes out.

15. Press the menu key [POS PNT]. The Positioning Point screen appears.

16. Press the Start button . The lamp on the Start button will blinks on and off, and the doors will lock.

17. Press the Start button . The lamp on the Start button goes on and the tool moves to the positioning point.

18. Press the Door open button to unlock the door, and open the door.

19. Set the chuck and tool on the drill sleeve and insert it into the drilling sleeve holder.

- Set the tip of the tool near the drill sleeve.

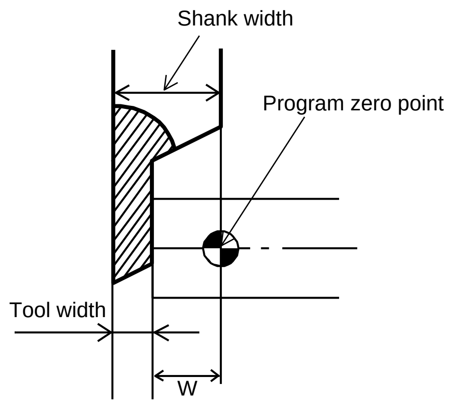
To set the right-handed cut-off tool:

Go to step 20.

To set the left-handed cut-off tool:

The end face of the material extends from the program zero point.

Calculate the value from the cut-off tool shank width and tool width, lower the spindle for the value W in the figure by pulse handle or MDI operation. Move the end face of the material to the program zero point. Then, go to step 20.



CAUTION

If the tip of the tool protrudes from the level of the end face of the back chuck cap nut, the tool and the material or guide bushing may interfere with each other.

20. Use the page keys , the tab keys TAB TAB or the arrow keys to move the cursor to the "Core" of the tool to be set.

21. Close all doors.

22. Press the menu key [Man. Set].

The Manual Tool Set screen appears.

23. Press the arrow key . Two cursors are displayed and the list of the moving position is displayed.

24. Enter the number for "Center", then press the Input key INPUT.

25. Press the Start button .
The lamp on the Start button will blinks on and off, and the doors will lock.

26. Press the Start button . The lamp on the Start button goes on. After the spindle (Z1) retracts 1 mm and the sleeve moves to the set position, the spindle (Z1) advances 1 mm.

- At this time, use the override dial to keep movement slow.

27. Press the Door open button to unlock the door, and open the door.

28. Fit the tool tip to the end face of the material, then secure it temporarily.
29. Take steps 15. to 20. for other front drilling tools to set them in the same way.
30. Tighten the clamp screw for the front drilling tool.

[Note]

- To enable operation while the door is open, press the Door open button to unlock the door, and set the Setting switch key to the "I" position.
  If this switch key is set to the "O" position, machine operation is prohibited while the door is open.
- The unit will only operate with the door open while the Start button is pressed.
  The axes speed at this time is limited to 2 m/min or less.
- Once the Start button is released, the axis motion will temporarily stop, and the lamp on the Start button will blink (on for two seconds and off for one second). To restart operations, close all doors and hold down the Start button to cancel the temporary stop of operations.
- If no operation is performed within 15 seconds after making menu selections, menu selections will automatically be canceled.

  Menu selections will also be canceled if a reset is performed during or after operations.
  To restart operations, be sure to again make selections with the menus.

8.14.4 Back Drilling Tool Setting Procedure (T31's tool)

CAUTION

In the tool setting function, positioning of parts such as holders are performed based on machining data.
Before performing tool setting, be sure to check machining data for validity.
If machining data contains an error, the tools, holders, and material may interfere with one another.

[Procedure]

1. Execute one cycle of operation to cut a workpiece dimensioned on the front side and to end up with the back chuck clamping the workpiece.
   - Before executing one-cycle operation, check that the menu key [Last PRT] on the Automatic Operation screen has not been selected. If the menu key [Last PRT] has been selected, press it again to deselect it.
2. Press the Preparation key (PREPA.). The lamp on the Preparation key goes on and the Preparation window appears.
3. Press the Door open button to unlock the door, and open the door.
4. Set the chuck and tool on the drill sleeve and insert it into the back drilling sleeve holder.

CAUTION

Make sure that the tip of the back machining tool is within 62 mm from the block end face of the back machining tool holder.
If the tool protrudes excessively, the holder, tool, and workpiece may interfere with each other.

5. Use the page keys, the tab keys (TAB ⬅ TAB ➡), and arrow keys (⬆ ⬇) to move the cursor to the number of the back tool to be set.
6. Close all doors.
7. Press the menu key [Man. Set]. The Manual Tool Set screen appears.
8. Use the tab keys (TAB ⬅ TAB ➡) to move the cursor to "Longitud" in the window.

WARNING

Do not open the safety cover or touch any tool or material while the lamp on the Start button is on. Doing either or both may result in a serious injury or death.

9. Press the Start button. The lamp on the Start button will blinks on and off, and the doors will lock.
10. Press the Start button.
    The lamp on the Start button goes on. After the back spindle moves to the center of selected tool, it advances to the set position.
    - At this time, use the Feed rate override to keep movement slow.
11. Press the Door open button to unlock the door, and open the door.
12. Fit the tool onto the end face of the workpiece then secure it temporarily.
13. Press the menu key [OPP-RET].
    The opposite tool post retract point screen appears.
14. Close all doors.
15. Press the Start button. The lamp on the Start button will blinks on and off, and the doors will lock.
16. Press the Start button.
    The lamp on the Start button goes on and the opposite tool post returns. The lamp on the Start button goes off upon completion of returning.
17. Take steps 4. to 10. for other back drilling tools to set them in the same way.

18. Press the Door open button [icon] to unlock the door, and open the door.
19. Tighten the clamp screw for the back drilling tool.

[Note]

- To enable operation while the door is open, press the Door open button [icon] to unlock the door, and set the Setting switch key [icon] to the "I" position.
  If this switch key is set to the "O" position, machine operation is prohibited while the door is open.
- The unit will only operate with the door open while the Start button [icon] is pressed.
  The axes speed at this time is limited to 2 m/min or less.
- Once the Start button is released, the axis motion will temporarily stop. And the lamp on the Start button will blink (on for two seconds and off for one second). To restart operations, press the Start button.
- If no operation is performed within 15 seconds after making menu selections, menu selections will automatically be canceled.
  Menu selections will also be canceled if a reset is performed during or after operations.
  To restart operations, be sure to again make selections with the menus.

8.14.5 Back Drilling Tool Setting Procedure (T51's tool)

> **CAUTION**
>
> In the tool setting function, positioning of parts such as holders are performed based on machining data.
> Before performing tool setting, be sure to check machining data for validity.
> If machining data contains an error, the tools, holders, and material may interfere with one another.

[Procedure]

1. Execute one cycle of operation to cut a workpiece dimensioned on the front side and to end up with the back chuck clamping the workpiece.
   - Before executing one-cycle operation, check that the menu key [Last PRT] on the Automatic Operation screen has not been selected. If the menu key [Last PRT] has been selected, press it again to deselect it.
2. Press the Preparation key [PREPA.].

   The lamp on the Preparation key goes on and the Preparation window appears.
3. Press the Door open button [ ] to unlock the door, and open the door.
4. Set the chuck and tool on the drill sleeve and insert it into the drilling sleeve holder. Set the tip of the tool near the drill sleeve.

> **CAUTION**
>
> Make sure that the front end of tools T51 to T54 of the MEU307 are within 35 mm of end face of the cap and that the front end of tools T57 to T59 of the BSE707 are within 25 mm of the end face of the cap nut.
> If either protrudes excessively, the holder, tool, and workpiece may interfere with each other.

5. Use the tab keys [TAB ←][TAB →], and arrow keys [↑][↓] to move the cursor to the number of the back tool to be set.
6. Close all doors.
7. Press the menu key [Man. Set].

   The Manual Tool Set screen appears.

> **CAUTION**
>
> When the front machining tool is set, if the T51 to T59 is called with the material protruded to the front side, the front machining tool and material may interfere with each other. Make sure to draw out the material beforehand.

8. Use the tab keys [TAB ←][TAB →] to move the cursor to "Longitud" in the window.

> **WARNING**
>
> Do not open the safety cover or touch any tool or material while the lamp on the Start button [ ] is on. Doing either or both may result in a serious injury or death.

9. Press the Start button [ ]. The lamp on the Start button will blinks on and off, and the doors will lock.

10. Press the Start button .
The lamp on the Start button goes on. After the back spindle moves to the center of selected tool, it advances to the set position.
At this time, use the override dial to keep movement slow.
11. Press the Door open button to unlock the door, and open the door.
12. Fit the tool tip to the end face of the material, then secure it temporarily.
13. Close all doors.
14. Press the menu key [OPP-RET].
The opposite tool post retract point screen appears.
15. Press the Start button . The lamp on the Start button will blinks on and off, and the doors will lock.
16. Press the Start button .
The lamp on the Start button goes on and the opposite tool post returns. The lamp on the Start button goes off upon completion of returning.
17. Take steps 4. to 10. for other back drilling tools to set them in the same way.
18. Press the Door open button to unlock the door, and open the door.
19. Tighten the clamp screw for the back drilling tool.

[Note]

- To enable operation while the door is open, press the Door open button to unlock the door, and set the Setting switch key to the "I" position.
  If this switch key is set to the "O" position, machine operation is prohibited while the door is open.
- The unit will only operate with the door open while the Start button is pressed.
  The axes speed at this time is limited to 2 m/min or less.
- Once the Start button is released, the axis motion will temporarily stop. And the lamp on the Start button will blink (on for two seconds and off for one second). To restart operations, press the Start button.
- If no operation is performed within 15 seconds after making menu selections, menu selections will automatically be canceled.
  Menu selections will also be canceled if a reset is performed during or after operations.
  To restart operations, be sure to again make selections with the menus.

8.15 Zero Point Returning Procedure

This procedure is required only when the zero point request alarm has been generated; it is not required in a series of procedures for normal operation.

> **CAUTION**
>
> Keep the safety cover closed during returning to the zero point because it involves axis movement.

[Procedure]

1. Press the Parameter key PREPA. .

   The last screen selected appears.

2. Press the menu key [ZP EXE].

   The Zero Return screen appears.

3. Check that the operation mode is "All Axes".
4. Close all doors.
5. Press the Start button .
   The lamp on the Start button will blinks on and off, and the doors will lock.
6. Press the Start button .
   The message "Executing Zero Return" is displayed for the axis currently returning to the zero point. The message "Completed Zero Return" is displayed for axes which have already returned to the zero point.

[Note]

- Zero Point Returning Procedure is not required in a daily operation. It is required only when the zero point return request is issued or when the motor is replaced.
- When All Axes Zero Return is specified, first Z1, X1, and Z2 axes return to zero point. After X1 axis has returned to zero point, Y1 axis starts to return to zero point. After Z2 axis has returned to zero point, X2 axis starts to return to zero point.

8.16 Transformation of Eco・Machine Operation Information in csv format

[Procedure]

1. Launch the Microsoft Excel to open the csv file which was output from the machine.
2. Drag with the mouse to select the data for the chart.
3. With the data being specified, press the menu bar [Insert], and select [Chart].
   The "Chart Wizard 1/4" appears.
4. Select [Column Chart] on "Standard" tab, and click [Next]. The "Chart Wizard 2/4" appears.
5. Select the "Data Range" tab, and click [Next]. The "Chart Wizard 3/4" appears.
6. Enter "Power Consumption" in the Chart Title box, "Date" in the Category (X) Axis, and "Watt" in Value (Y) Axis.
7. Select the "Titles" tab, and click [Next]. The "Chart Wizard 4/4" appears.
8. Specify the location where you want to place your chart, and click [Finish].
9. The chart is displayed.

[Note]

- In addition to the steps described above, you can create a chart based on your csv file in various way.
- "Microsoft Excel" is registered trademark or trademark of Microsoft Corporation in the United States and other countries.

8.17 Csv File Format of Eco and Machine Operation Information

[ ]: Indicates the unit of measurement.

(A) Power Consumption - Year: YYCns.csv

```
YYYY,  XXXXXXXXXX
YYYY,  XXXXXXXXXX
YYYY,  XXXXXXXXXX
  :
  :
YYYY … Year
XXXXXXXXXX … Power Consumption [Wh]
```

(B) Power Consumption - Month: MMCns.csv

```
YYYY/MM,  XXXXXXXXXX
YYYY/MM,  XXXXXXXXXX
YYYY/MM,  XXXXXXXXXX
  :
  :
YYYY/MM … Year/Month
XXXXXXXXXX … Power Consumption [Wh]
```

(C) Power Consumption - Day: DDCns.csv

```
YYYY/MM/DD,  XXXXXXXXXX
YYYY/MM/DD,  XXXXXXXXXX
YYYY/MM/DD,  XXXXXXXXXX
  :
  :
YYYY/MM/DD … Year/Month/Day
XXXXXXXXXX … Power Consumption [Wh]
```

(D) Electric Power Regeneration - Year: YYRgn.csv

```
YYYY,  XXXXXXXXXX
YYYY,  XXXXXXXXXX
YYYY,  XXXXXXXXXX
  :
  :
YYYY … Year
XXXXXXXXXX … Regenerative Energy [Wh]
```

(E) Electric Power Regeneration - Month: MMRgn.csv

```
YYYY/MM,  XXXXXXXXXX
YYYY/MM,  XXXXXXXXXX
YYYY/MM,  XXXXXXXXXX
  :
  :
YYYY/MM … Year/Month
XXXXXXXXXX … Regenerative Energy [Wh]
```

(F) Electric Power Regeneration - Day: DDRgn.csv

```
YYYY/MM/DD,  XXXXXXXXXX
YYYY/MM/DD,  XXXXXXXXXX
YYYY/MM/DD,  XXXXXXXXXX
  :
  :
YYYY/MM/DD … Year/Month/Day
XXXXXXXXXX … Regenerative Energy [Wh]
```

(G) Reduction Power - Year: YYEco.csv

```
YYYY,  XXXXXXXXXX
YYYY,  XXXXXXXXXX
YYYY,  XXXXXXXXXX
  :
  :
YYYY … Year
XXXXXXXXXX … Reduction Power [Wh]
```

(H) Reduction Power - Month: MMEco.csv

```
YYYY/MM,  XXXXXXXXXX
YYYY/MM,  XXXXXXXXXX
YYYY/MM,  XXXXXXXXXX
  :
  :
YYYY/MM … Year/Month
XXXXXXXXXX … Reduction Power [Wh]
```

(I) Reduction Power - Day: DDEco.csv

```
YYYY/MM/DD,  XXXXXXXXXX
YYYY/MM/DD,  XXXXXXXXXX
YYYY/MM/DD,  XXXXXXXXXX
  :
  :
YYYY/MM/DD … Year/Month/Day
XXXXXXXXXX … Reduction Power [Wh]
```

(J) Machine Operation - Each Item, Year: YYEchOpe.csv

```
YYYY,  aaa.a,  bbb.b,  ccc.c,  ddd.d,  eee.e
YYYY,  aaa.a,  bbb.b,  ccc.c,  ddd.d,  eee.e
YYYY,  aaa.a,  bbb.b,  ccc.c,  ddd.d,  eee.e
  :
  :
YYYY … Year
aaa.a … Automatic Operation Time [%]
bbb.b … Alarm Stop Time [%]
ccc.c … No Operation Time [%]
ddd.d … Setup Time [%]
eee.e … Power OFF Time [%]
```

[ ]: Indicates the unit of measurement.

(K) Machine Operation Information - Each Item, Month: MMEchOpe.csv

```
YYYY/MM,  aaa.a,  bbb.b,  ccc.c,  ddd.d,  eee.e
YYYY/MM,  aaa.a,  bbb.b,  ccc.c,  ddd.d,  eee.e
YYYY/MM,  aaa.a,  bbb.b,  ccc.c,  ddd.d,  eee.e
  :
  :
YYYY/MM … Year/Month
aaa.a … Automatic Operation Time [%]
bbb.b … Alarm Stop Time [%]
ccc.c … No Operation Time [%]
ddd.d … Setup Time [%]
eee.e … Power OFF Time [%]
```

(L) Machine Operation Information - Each Item, Day: DDEchOpe.csv

```
YYYY/MM/DD,  aaa.a,  bbb.b,  ccc.c,  ddd.d,  eee.e
YYYY/MM/DD,  aaa.a,  bbb.b,  ccc.c,  ddd.d,  eee.e
YYYY/MM/DD,  aaa.a,  bbb.b,  ccc.c,  ddd.d,  eee.e
  :
  :
YYYY/MM/DD … Year/Month/Day
aaa.a … Automatic Operation Time [%]
bbb.b … Alarm Stop Time [%]
ccc.c … No Operation Time [%]
ddd.d … Setup Time [%]
eee.e … Power OFF Time [%]
```

(M) Machine Operation - Time Line, Day: DDTimOpe.csv

```
YYYY/MM/DD,  HH:mm:SS,  aa
YYYY/MM/DD,  HH:mm:SS,  aa
YYYY/MM/DD,  HH:mm:SS,  aa
  :
  :
YYYY/MM/DD … Year/Month/Day
HH:mm:SS … Hour/Minute/Second
aa … Mode [–]
10 … Auto   20 … Alarm Stop
30 … No Operation
40 … Setup      50 … Power OFF
```

(N) Time rate of Cut/Non-Cut - Year: YYCut.csv

```
YYYY,  aaaaaaaa,  bbbbbbbb,  cccccccc,  dddddddd
YYYY,  aaaaaaaa,  bbbbbbbb,  cccccccc,  dddddddd
YYYY,  aaaaaaaa,  bbbbbbbb,  cccccccc,  dddddddd
  :
  :
YYYY … Year
aaaaaaaa … Total automatic opearation time [s]
bbbbbbbb … Total Cut Time: $1 [s]
cccccccc … Total Cut Time: $2 [s]
dddddddd … Total Cut Time: $3 [s]
```

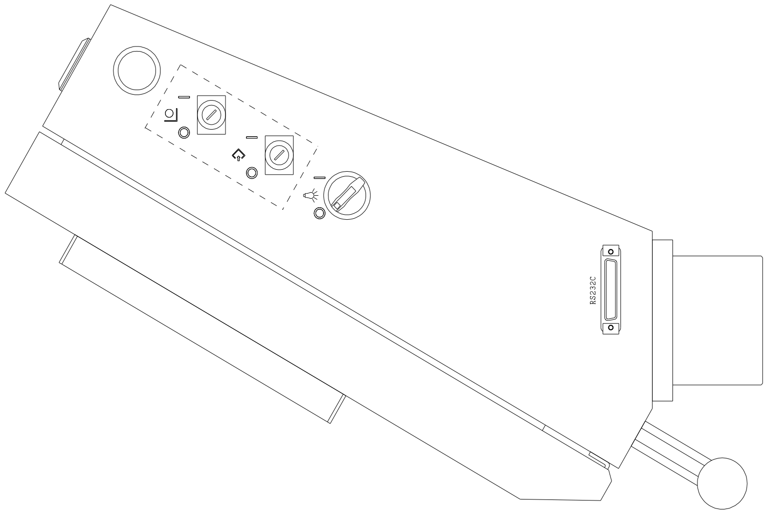
(O) Time rate of Cut/Non-Cut - Month: MMCut.csv

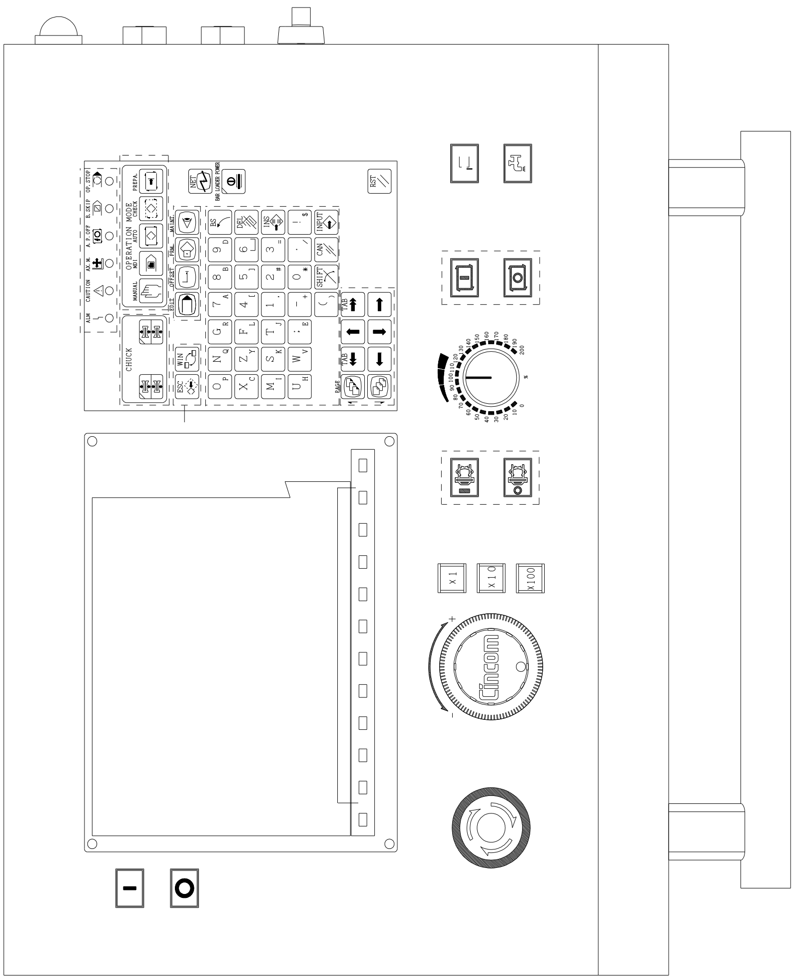
```
YYYY/MM,  aaaaaaaa,  bbbbbbbb,  cccccccc,  dddddddd
YYYY/MM,  aaaaaaaa,  bbbbbbbb,  cccccccc,  dddddddd
  :
  :
YYYY/MM … Year/Month
aaaaaaaa … Total automatic opearation time [s]
bbbbbbbb … Total Cut Time: $1 [s]
cccccccc … Total Cut Time: $2 [s]
dddddddd … Total Cut Time: $3 [s]
```

(P) Time rate of Cut/Non-Cut - Day: DDCut.csv

```
YYYY/MM/DD,  aaaaaaaa,  bbbbbbbb,  cccccccc,  dddddddd
YYYY/MM/DD,  aaaaaaaa,  bbbbbbbb,  cccccccc,  dddddddd
  :
  :
YYYY/MM/DD … Year/Month/Day
aaaaaaaa … Total automatic opearation time [s]
bbbbbbbb … Total Cut Time: $1 [s]
cccccccc … Total Cut Time: $2 [s]
dddddddd … Total Cut Time: $3 [s]
```

■ Sheet keyboard type: Symbol





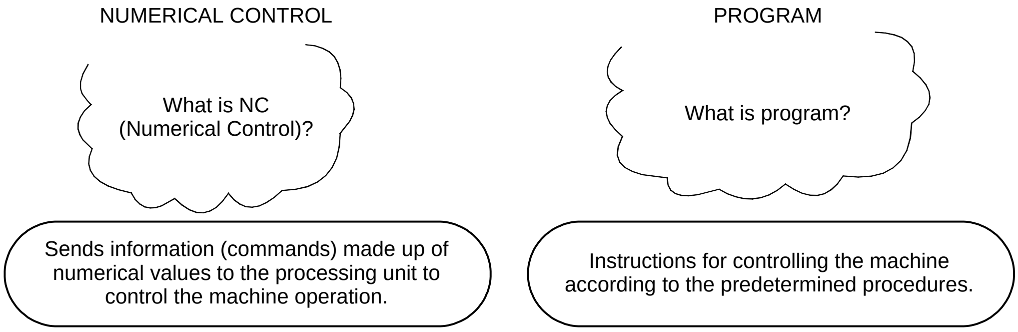
Product Code	C-L220E XII
Document Code	1E1-0800
Mfg. No.	L220E/0001~
Issue Date	2013.10

11. Fundamentals of Programming

11.1 The NC Program

The Numerical Control (NC) program is a group of commands designed to operate the machine using the NC unit.

These commands are simple alphanumeric codes designed to be readable by the NC unit as well as easy to use by the programmer.



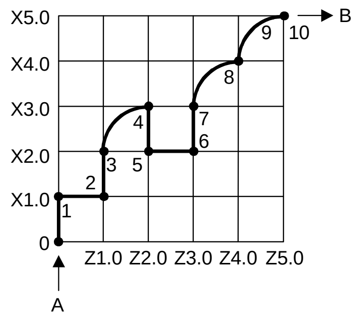
Before reading ahead for details on NC programs, try the following question as an example showing that NC programming is easy.

Q: Use the following four commands to complete the code to move from A to B in the figure.

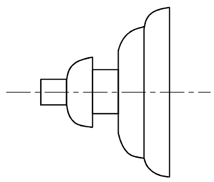
1)	G01	Move straight to ○○
2)	G02	Move circularly to ○○
3)	M03	Turn the start switch ON
4)	M02	Turn the goal switch OFF

Codes

0	**M03**			(Turn on the start switch)
1	**G01**	**X1.0**	**Z0**	(Straight move to X = 1.0, Z = 0)
2	**G01**	**X1.0**	**Z1.0**	(Straight move to X = 1.0, Z = 1.0)
3	**( )**	**X( )**	**Z1.0**	(Straight move to X = 2.0, Z = 1.0)
4	**G02**	**X( )**	**Z2.0**	(Circular move to X = 3.0, Z = 2.0)
5	**( )**	**X( )**	**Z( )**	(Straight move to X = 2.0, Z = 2.0)
6	**( )**	**X( )**	**Z( )**	(* to X = *, Z = *)
7	**( )**	**X( )**	**Z( )**	(* to X = *, Z = *)
8	**( )**	**X( )**	**Z( )**	(* to X = *, Z = *)
9	**( )**	**X( )**	**Z( )**	(* to X = *, Z = *)
10	**( )**			



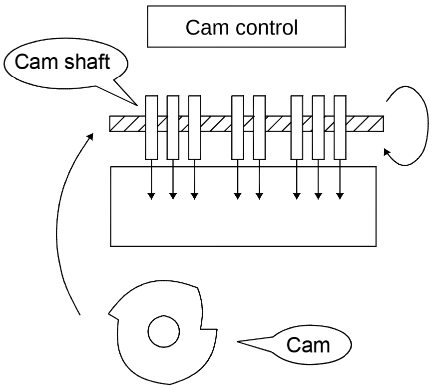
The NC program is almost the same as codes above.
(The answer to this above question is given on page 11-18.)

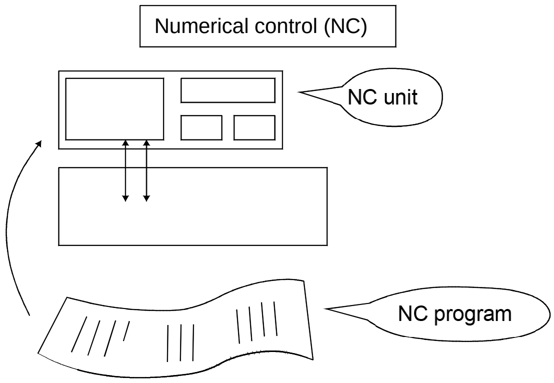


11.1.1 Difference between cam control and NC

What are the differences between cam control and NC?

Where is the brain of the machine?





How do you operate the machine?

Mechanical

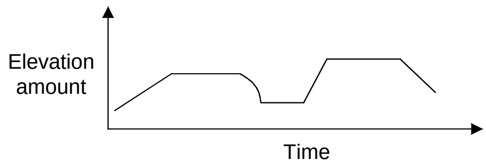
The up/down movement of the cam is transmitted directly to the mechanism via the follower to lever.

Electric

The information of the program is converted into electric signals, to operate the mechanism by driving motors and turning on/off switches.

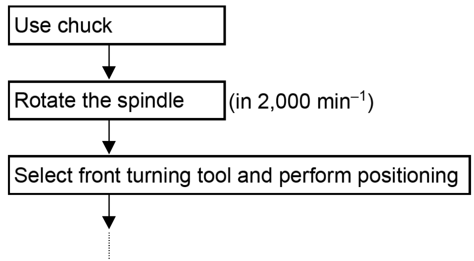
Analog

The cam is first designed. The up/down movement of the cam in relation to time is continuously given.



Digital

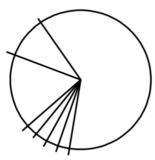
The NC program is created and the operation commands are constructed step by step according to the process.



What is a program like?

1 cycle is angular

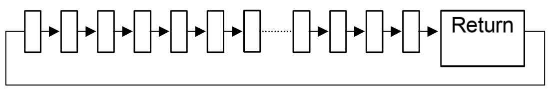
The entire process must be completed within 360°.



Limitation to complex and long-time machining.

1 cycle is linear

As many commands as required can be specified.



No limitation however long or complex the machining is.

Which is easier?

Hardware

Forming the cam, or the metal machining of the cam itself must be done first.

- Time and labor is required to set the cam.
- Correction and change is difficult.
- Storage is difficult.
- Forming the cam is costly.
- One machine can be fitted with only one cam for one workpiece.

Software

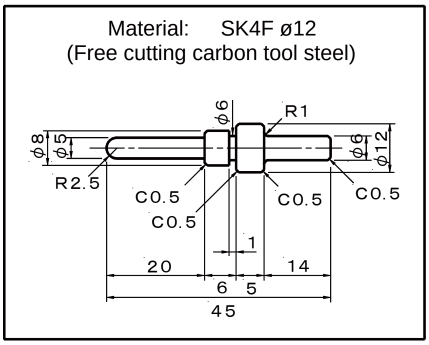
Just type the commands according to the designated manner. The program can easily be rewritten.

- Very short time is required for the preparatory stage.
- Storage does not require much space. The program can be stored as long as necessary.
- Once programming is accomplished, the program can be made easily at any workshop.
- A single machine can store many types of programs, to be selected as necessary.
- Correction and modification is very easy.

11.2 Process of Program Creation

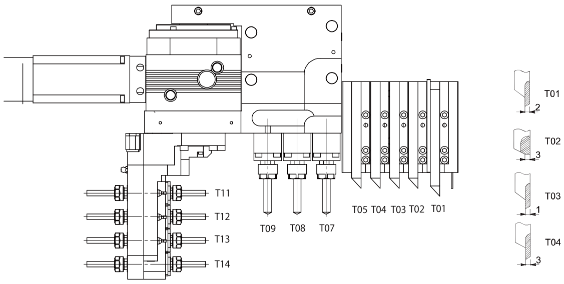
1 Workpiece Drawing

- The drawing is the basis for an NC program.



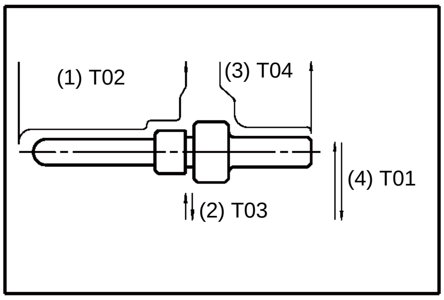
2 Creating the Tool Layout

- Select the holder to be used.
- Determine the tool layout.
- Describe the shape of the tool, the dimensions of tool width, and the tool number.



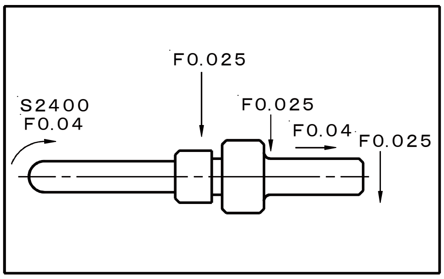
3 Creating the Machining Layout

- Specify the number of the tool used, the cutting path, and cutting procedure.



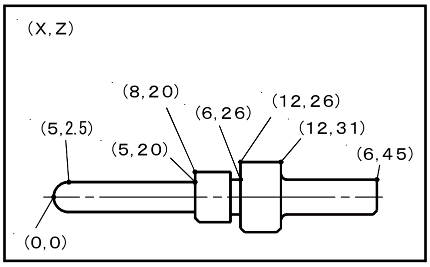
4 Setting the Spindle Speed and Feed Rate

- Select the proper cutting conditions.



5 Reading the Coordinate Values

- Read the in-feed amount (X) and longitudinal feed amount (Z) from the drawing.



6 Coding the Program

- Specify the type of the tool to be used, range of cutting, and machining conditions for cutting on paper according to the rules.

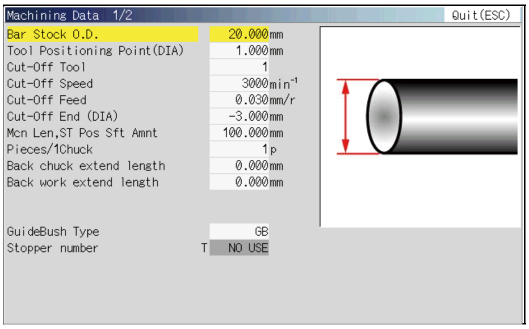
```
O0002
$1                        $2
G50 Z–0.1                 G630
(M52)                     G600
M06                       G999
G00 Z–0.6 S1=2400 M03     N999
G630                      M02
N0112 T1200               M99
Z0                        %
```

7 Entering and Saving the Program

- Enter the written program sequence on the paper into the NC device and save them.

8 Entering the Machining Data

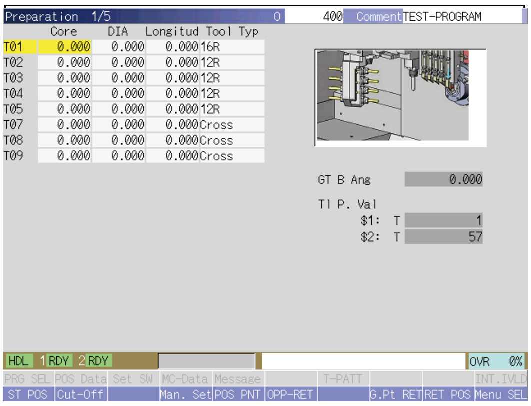
- Enter the holder name to be used, material diameter, and machining length as data into the NC in advance.



Machining data screen

9 Enter the tool setting data

- Enter prerequisite data such as the core position, diametrical direction, and longitudinal direction.



Tool set screen

10 End

- With this, the programmer's jobs are completed.

11.3 NC Program Structure

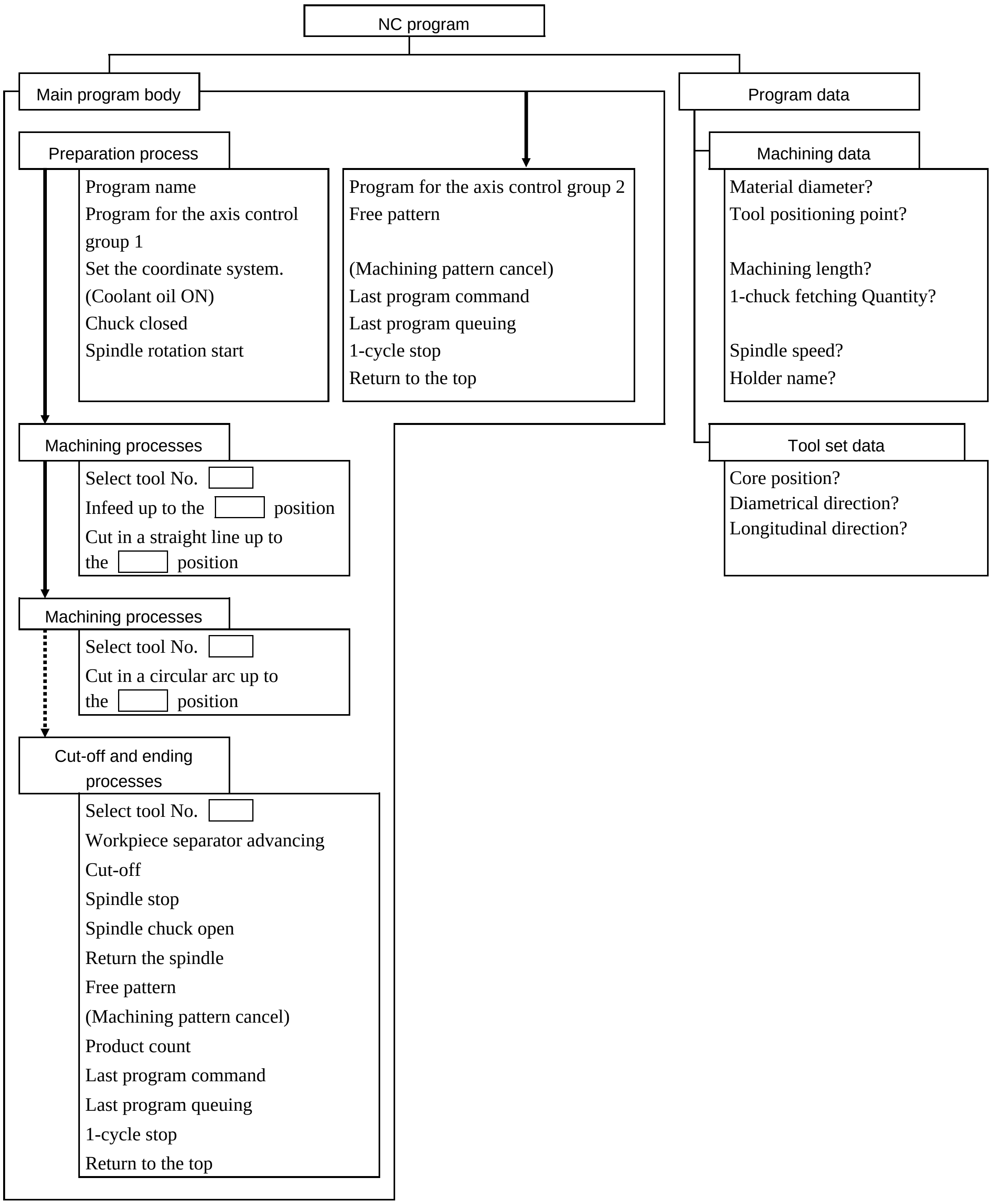
The NC program consists of a main program body and program data.

In the main program body, operation commands for a machine are arranged in the order of each machining process. This is usually called a program.

In the program data, the preconditions under which a program operates are arranged.

This program is not only required for processing on a machine, but also it serves as the communication means between the personnel.

The diagram on the next page summarizes the contents of commands used in the program body and for program data. The diagram on the page that follows lists the actually used command strings and screen displays. The machine operates under control of the program configured with the program body and program data as illustrated in the diagrams.


NC program
Main program body
Program data
Preparation process
Program name
Program for the axis control group 1
Set the coordinate system.
(Coolant oil ON)
Chuck closed
Spindle rotation start
Program for the axis control group 2
Free pattern
(Machining pattern cancel)
Last program command
Last program queuing
1-cycle stop
Return to the top
Machining data
Material diameter?
Tool positioning point?
Machining length?
1-chuck fetching Quantity?
Spindle speed?
Holder name?
Machining processes
Select tool No.
Infeed up to the position
Cut in a straight line up to the position
Tool set data
Core position?
Diametrical direction?
Longitudinal direction?
Machining processes
Select tool No.
Cut in a circular arc up to the position
Cut-off and ending processes
Select tool No.
Workpiece separator advancing
Cut-off
Spindle stop
Spindle chuck open
Return the spindle
Free pattern
(Machining pattern cancel)
Product count
Last program command
Last program queuing
1-cycle stop
Return to the top

NC program

Main program body

Preparation process

Oxxxx
$1
G50 Z0
(M52)
M06
S1=2000 M03
G630

$2
G630
G600
G999
N999
M02
M99
%

Machining processes

T
G01 X F

Machining processes

T
G02 X Z R F
F

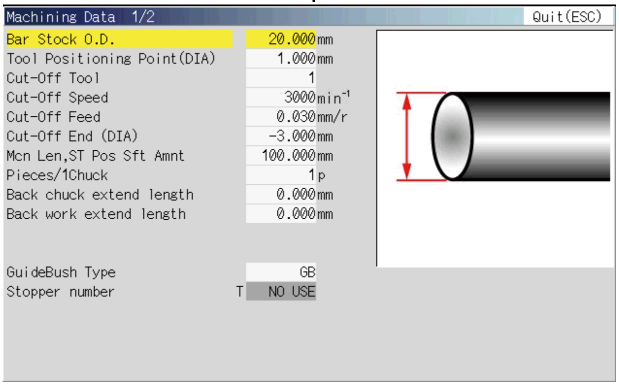
Cut-off and ending processes

T100
M320
G01 X–3.0 F
M05
M07
G00 Z0 T00
G600
M56
G999
N999
M02

M99
%

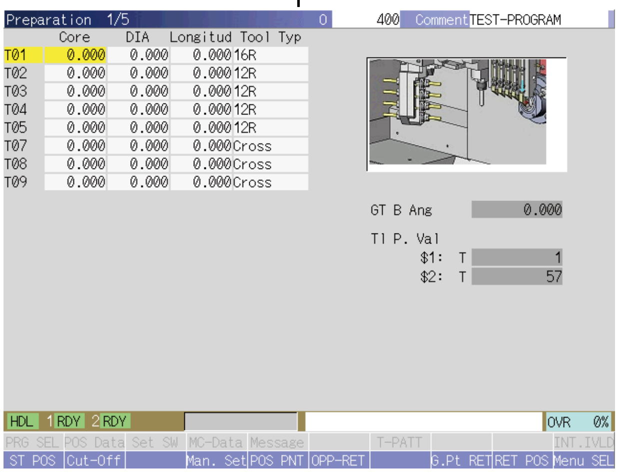
Program data

Machining data



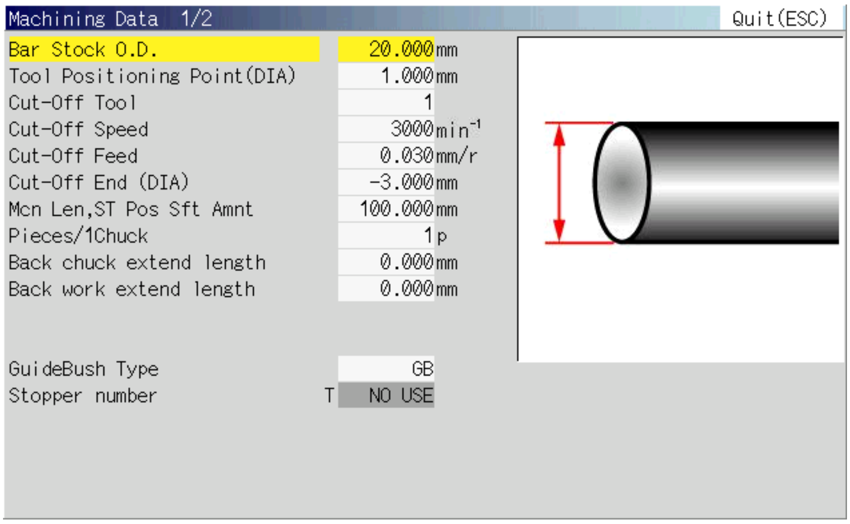
Machining data screen

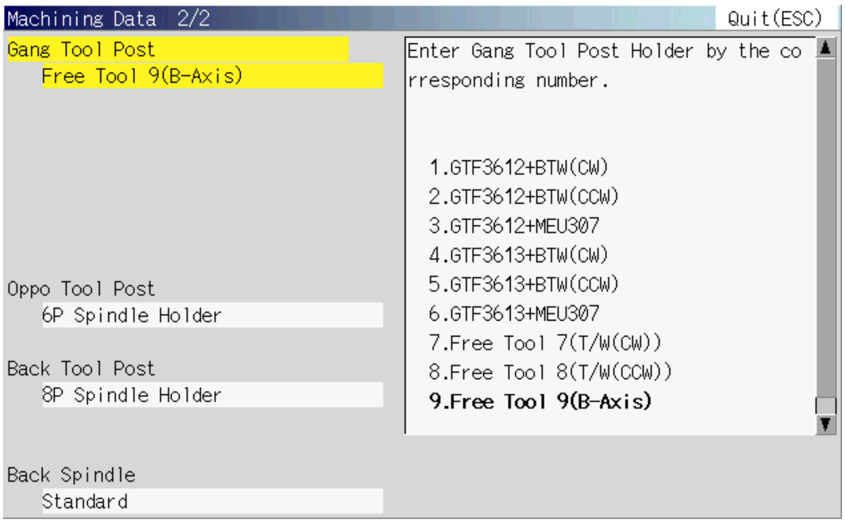
Tool set data



Tool set screen

11.3.1 Machining data (screen display)





Bar Stock O.D.

Enter the outside diameter of materials to be machined.

Tool Positioning Point (DIA) Bar Stock O.D. +

The tool positioning point is the point at which a tool is positioned when selected from among tools T01 to T14. Enter the clearance between the tool at the tool positioning point and the outer diameter of the material. That point is also the position to which the currently selected tool escapes when another tool is selected. That is, the tool positioning point is the position at which a tool is positioned when selected from among T01 to T14 and to which the tool escapes when another tool is selected.

Cut-Off Tool number

In this machine, a cut-off tool is mounted in T0100. Therefore, T1 is specified automatically.

Cut-Off Speed (min^{-1})

The tip of materials is cut at this speed specified by CUT-OFF in Preparation screen or M108 Material Exchange program.

Cut-Off Feed (mm/r)

The tip of materials is cut at this speed specified by CUT-OFF in Preparation screen or M108 Material Exchange program.

Cut-Off End (DIA)

The tip of materials is cut to the end position of this X-axis position specified by CUT-OFF in Preparation screen or M108 Material Exchange program.

This position is also used as the start position of an X-axis when the program starts.

Machining Length, Start point shift amount

The entered values are different when using the GB model and when using the GBL model + stopper enabled specifications.

＜GB machine＞

Usually, enter the Z1 axis stroke required for machining one workpiece.
(Product length + Cut-off tool width + Face-cut allowance + α (Margin))

<In GBL, when stopper specifications are available>

Enter the value <(Maximum move distance required for machining the workpiece) − (Overall length of workpiece)>.

(1)	Right-handed cut-off tool:	Enter the value (Cut-off chip width) + (Shift amount of rear turning tool). (If tool secondary machining tool is used, enter the shift amount of that tool.)
(2)	Left-handed cut-off tool:	Enter the larger value either of Cut-off tool shank width, or shift amount of secondary machining tool.

Pieces/1Chuck

Enter the number of product to be machined in the program.

Back chuck extend length

When a back long-neck chuck is used, enter the protrusion length of long-neck chuck from the back spindle cap nut. If "SUPPORT" is specified for "Back Spindle" on Machining Data screen, the "Back chuck extend length" is not displayed.

Back work extend length

When the back chuck end face or back long-neck chuck is used, enter the workpiece length extruded from the end face of back long-neck chuck.

Guide bushing type

Specify the type of guide bushing of the selected program by selecting one in the list.

Stopper number (In GBL, when stopper specifications are available)

Enter the stopper number. If the stopper is not used, enter "0"
Stopper numbers are: T01 to T05,T20.

Gang tool post

Enter the name of the gang tool holder attached to the machine.

Gang tool post for T08 through T10

Specify the holder name for cross drilling tool/end-face drilling tool mounted on T08 through T10.

Front Drill post

Enter the name of the front drilling tool holder on opposite tool post mounted on the machine.

Back Drill post

Enter the name of the back drilling tool holder on back tool post attached to the machine.

Back Spindle

Enter the type of the back spindle attached to the machine by selecting Standard, With Basket or With Support.

Wrk Length + C-off TL Width

Enter the value "work length + cut-off tool width". This item is displayed only when the synchronous automatic bar loader (CAV) is installed. Use this menu to display the rough estimate of products that can be machined with the remnant material.

[Note]

- The machining data is automatically output at the same time when the program is output. "#814=0000010000 (material outer diameter 10.0)" appears after $0 when viewing the program outside the machine.
- The value entered as the back long-neck chuck protrusion length can be reflected on programs to be created on the basis of the back chuck end face. (See *<15.15 Use of "Back Chuck Extend Length" in Machining Data>* for details.)
- Select "SUPPORT" for back spindle if the back cap nut is changed to that of the stabilizer-attached type in long workpiece machining. Selecting "SUPPORT" causes the "Back chuck extend length" to be erased from the machining data.
  When you select "SUPPORT", programs can be created on the basis of the brass part of the stabilizer-attached cap nut by automatically considering that the stabilizer-attached cap nut is longer than the normal cap nut by 10.0 mm.

11.4 Tool Layout Pattern

A more rational layout can be selected according to the number of tools to be used.

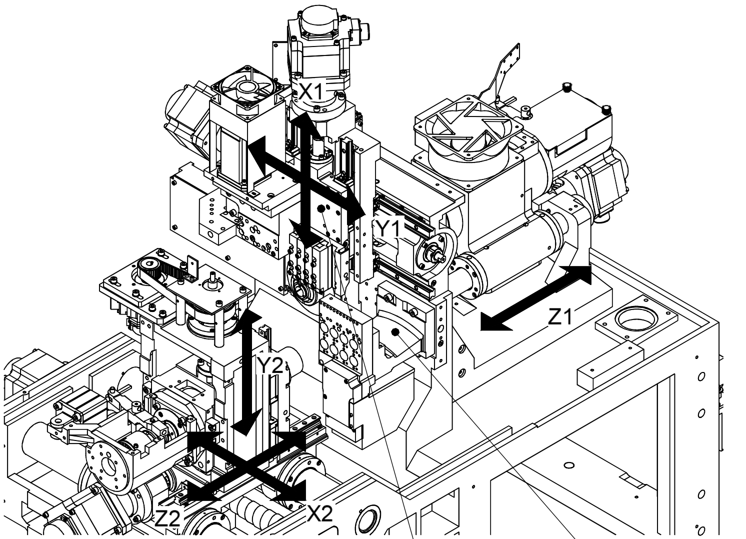
For more information, see *<Chapter 17 Tooling>*.

The type of the tool holder you have selected is displayed on the Machining Data screen. For information on holder name selection, see *<6.1.5 Entering, updating and storing the machining data>* of the Operator's Manual.

11.5 Drive Axis

11.5.1 6-axis control specification (Type XII)

- The gang tool post moves vertically. (X1 axis)
- The gang tool post moves horizontally. (Y1 axis)
- The headstock horizontally moves left or right. (Z1 axis)
- The opposite tool post and back headstock move back and forth. (X2 axis)
- The opposite tool post and back headstock moves horizontally left or right. (Z2 axis)
- The opposite tool post and back headstock moves vertically. (Y2 axis)

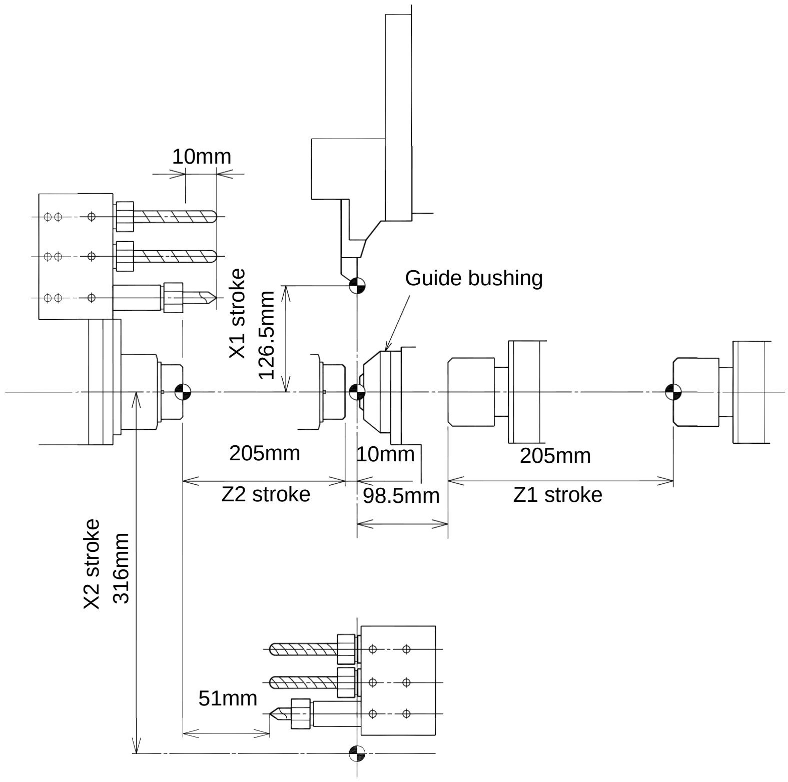


Axis \ Tool post	Name
X1	Gang tool post
Z1	Headstock
Y1	Gang tool post
B1	B axis
C1	Main spindle C axis
X2	Opposite tool post and back headstock
Z2	Opposite tool post and back headstock
Y2	Opposite tool post and back headstock
C2	Back spindle C axis
A7	Dedicated bar loader

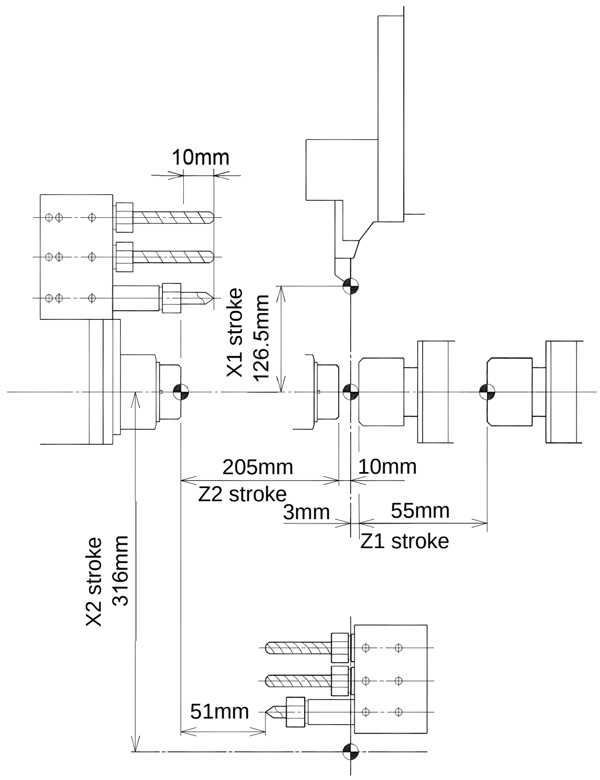
S1 = Main spindle
S2 = Back spindle
S3 = Rotary tool on gang tool post
S4 = Front rotary tool on opposite tool post (Option)
S5 = Rotary tool on back headstock

11.5.2 Stroke drawing

The fixed points shown in the figure below are set by each machining pattern. These fixed points vary according to a change in the pattern.



Stroke diagram (GB specification)



Stroke diagram (GBL specification)

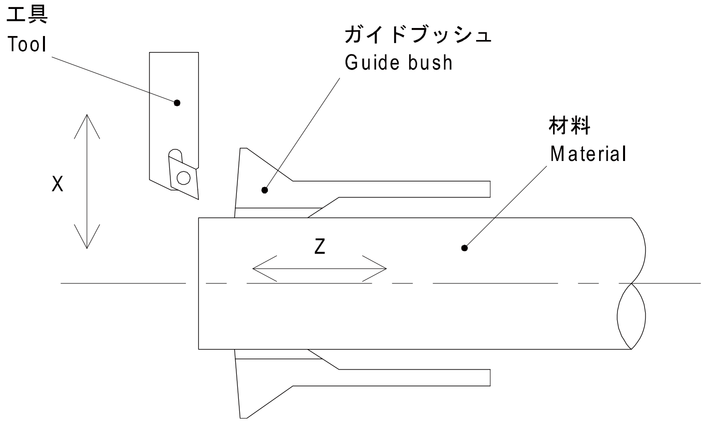
11.5.3 Multi-axis control group

Six axes move independently. These axes are functionally classified into two groups.
As a basic function, the X1, Y1, and Z1 axes are called the axis control group 1. It is symbolized in the program as "$1".
The X2, Y2, and Z2 axes are set as system 2 and appear as $2 in the program.
In general, the two groups move independently. However, the Z1 and Z2 axes can be superimposed by the simultaneous machining of outer and inner diameters or the Z1 and X2 axes can be interpolated by the $1 single machining. (See *<11.7 Machining Pattern>*.)

11.6 Coordinate System

11.6.1 Coordinate axis

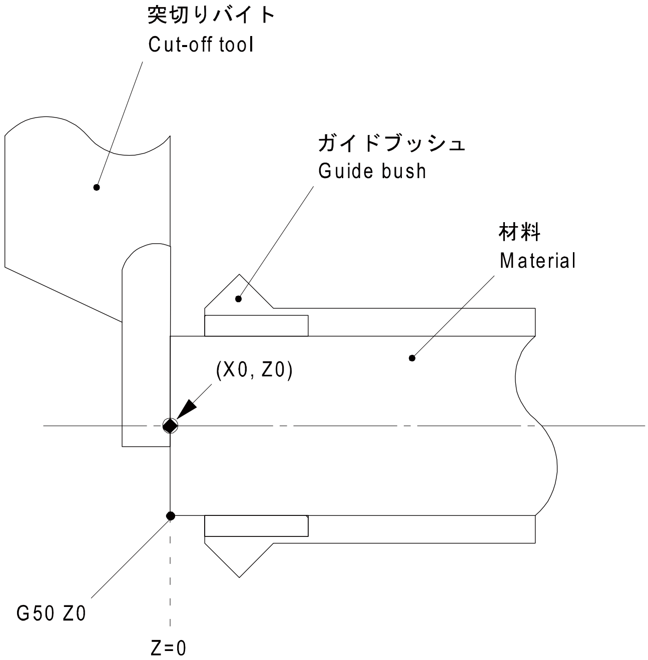
The axis along the center axis of material rotation is determined as the Z axis and the axis perpendicular to the Z axis as the X axis. In other words, the Z axis refers to the direction the material moves longitudinally and the X axis the direction in which an outer diameter turning tool moves into the material. The Z and X axes are sometimes called the longitudinal axis and diametric axis, respectively



11.6.2 Setting the coordinate system (program zero point)

For the X and Z axes defined as explained above, set the zero point of them. For the Z axis (longitudinal direction), set the position of the material end face that is in contact with the cut-off tool as the zero point (Z = 0). This setting is possible by specifying “G50 Z0” in a program. This command means “Define the current Z axis position as Z = 0.

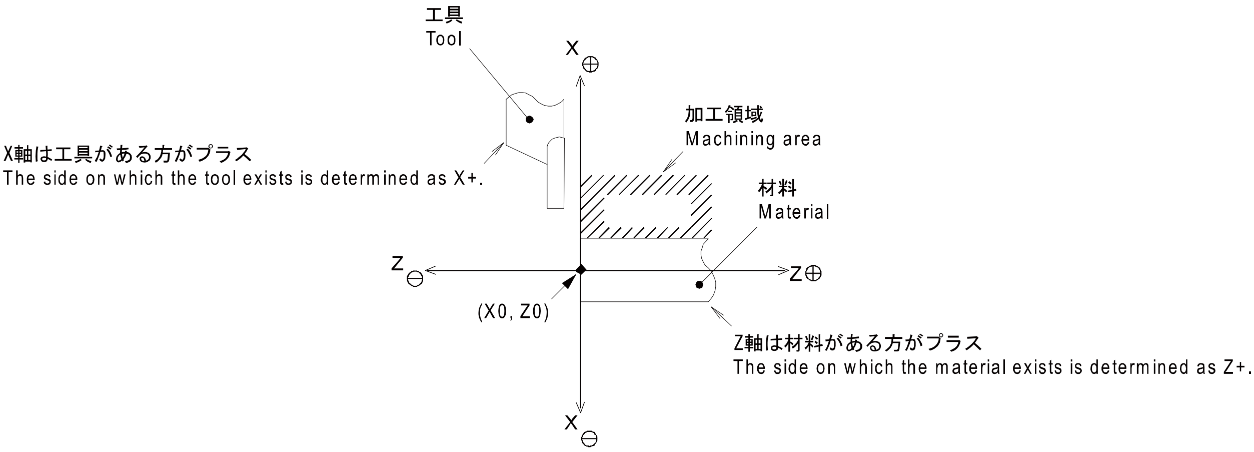
For the X axis (diametric direction), set the center of material rotation as the zero point (X = 0). For this setting, it is not necessary to specify the corresponding commands in a program because the tool setting is made so that tool nose comes to the zero point (X = 0) when a tool is mounted. Now, the center of rotation at the material end face takes the coordinate values (X0, Z0) and this point is called the program zero point



11.6.3 Signs

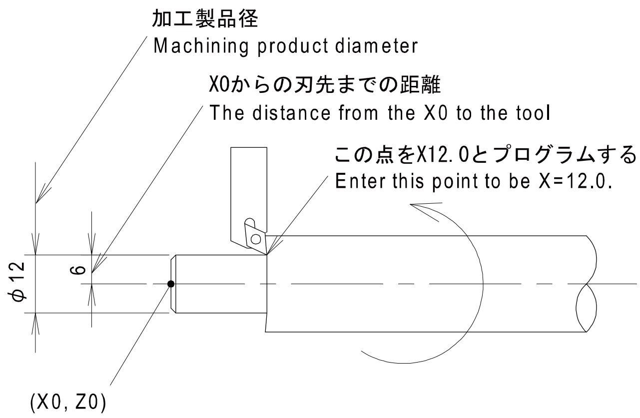
In the coordinate system, location of a point is defined by a coordinate value and a plus/minus sign. In other words, a sign indicates at which side a point in question lies in reference to the zero point; for the X axis, the side on which a tool exists is determined as X+, and for the Z axis, the side at which the material exists is determined as Z+.

Usually, machining is carried in the X+ and Z+ area.



11.6.4 Coordinate values

Along the Z axis, a coordinate value indicates the distance between a point in question and the end face of a material (Z = 0). Along the X axis, it indicates a value two times the distance between a point in question and the center of rotation of a material (X = 0). The reason for this is a material is rotated around the centerline and thus the diameter of a product to be machined is two times the distance a point lies from the center of rotation. Since a coordinate value to be specified for the X axis is two times the distance, an X axis command is called a diametric command. For a numeric value, a value in “mm” unit with a decimal point is used. Note that a decimal point must always be specified in a coordinate value.

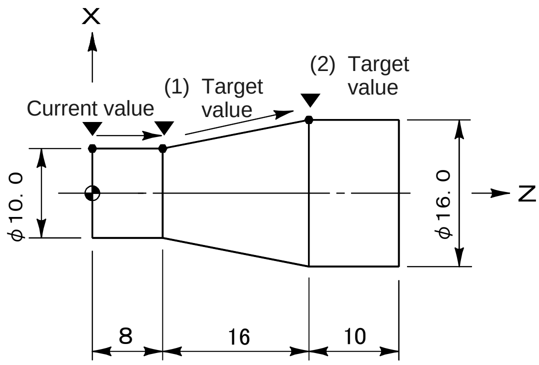


11.6.5 Absolute and incremental commands

The absolute command is used for the (X, Z) coordinate value of the target position and the incremental command is used to specify the move distances of X and Z axis directions.

Absolute command:	Specifies the (absolute) coordinate value of the target position. (X☐, Y☐, Z☐)
Incremental command:	Specifies the (relative) distance in the X, Y, and Z axis directions. (U☐, V☐, W☐)

- X and Y, and U and W are specified in diametrical values.



Regard Z0 as the start

Absolute		Incremental	
	X10.0 Z0		U0 W0
	↓		↓
(1)	X10.0 Z8.0	(1)	U0 W8.0
	↓		↓
(2)	X16.0 Z24.0	(2)	U6.0 W16.0

Answer to the question on page 11-2.

3. G01 X2.0	4. X3.0	5. G01 X2.0 Z2.0	6. G01 X2.0 Z3.0
7. G01 X3.0 Z3.0	8. G02 X4.0 Z4.0	9. G02 X5.0 Z5.0	10. M02

11.7 Machining Pattern

Machining Pattern

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.

Specify the machining pattern in both axis control groups 1 and 2.

- Free pattern (machining pattern cancel) (G600)

  This pattern cancels any of machining patterns G610 through G660 while the power is being turned on.

  In the free pattern (machining pattern cancel) mode, front/back parallel machining is possible. With the front axis control group 1, front outer diameter machining and front drilling operations are possible using the gang tool post while machining using the back tool post is possible with the back axis control group 2.

- $1 single machining (G610)

  This pattern allows the outer diameter machining by gang tool post or inner diameter machining by opposite tool post in axis control group 1.

- Inner/outer diameter simultaneous machining (Z1-Z2 axes superimpose) (G620)

  This pattern superimposes the Z2 axis on the Z1 axis, and allows outer diameter machining with gang tools in $1 and inner diameter machining with drilling tools on opposite tool post in $2, simultaneously.
  Pinch milling can be performed.

- Front/back parallel machining (G630)

  This pattern is used in front/back parallel machining. With the axis control group 1, front outer diameter machining and front drilling operations are possible using the gang tool post while machining using the back tool post is possible with the back axis control group 2.

- Pick-off, center-support (Z1-Z2 axes superimposition) (G650)

  This machining pattern enables the back spindle to rechuck the workpiece and support the center of the workpiece.

- Front/back simultaneous machining (G660)

  This pattern allows front/back simultaneous machining of the axis control groups 1 and 2 using the drilling tool holders on the gang tool post.

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 (e.g., G620, G650 machining pattern) is specified to an axis control group, its member axes that have been operating with different coordinate systems will operate synchronizing with the superimposed coordinate system.

Example: With Z1-Z2 superimposed by G650, Z1 is the reference axis and Z2 is the superimposed axis. In a program, you can specify Z2 coordinate values on the coordinate system of Z1 for synchronous or asynchronous operation. See <*11.7.9 Superimpose control*> for more information.

11.7.1 Machining pattern list

Machining pattern	Command code	$1	$2
Free pattern (machining pattern cancel)	G600	Operation X1, Z1 and Y1: No operation	Operation X2: No operation Y2: No operation Z2: Movement to return position (Argument W0 (Specifying no operation of Z2) is available)
		Selectable tool numbers: T0100 to T1400,T2000 * T5100 to T5900 (except B axis) * T0100 to T1000 K2	Selectable tool numbers: T3000 T3100 to T3800 * T5100 to T5900
		Axes of axis control groups: X1, Z1, Y1, (C1) (* Note that the axes of axis control group are switched to X1, Z2, and Y1 (C2) when T5100 to T5300 or T0100 to T1300 K2 is called.)	Axes of axis control groups: X2, Z2, (C2), Y2 (* Note that the axes of axis control group are switched to X1, Z2 and Y1 (C2)
$1 single machining	G610	Operation X1, Z1 and Y1: No operation	Operation X2: No operation Y2: No operation Z2: Movement to return position (Argument W0 (Specifying no operation of Z2) is available)
		Selectable tool numbers: T0100 to T1400 T2100 to T2600	–
		Axes of axis control groups: X1, Z1, Y1, (C1) X2, Z1, (C1), Y2	–
Inner/outer diameter simultaneous machining	G620	Operation X1, Z1 and Y1: No operation	Operation X2, Z2: No operation Y2: No operation
		Selectable tool numbers: T0100 to T0500	Selectable tool numbers: T2100 to T2300 T2100 to T2600
		Axes of axis control groups: X1, Z1, Y1, (C1)	Axes of axis control groups: X2, Z2, (C2), Y2
			Superimpose Z2 superimposed on Z1
	G621	Axes of axis control groups: X1, Z1, Y1, (C1)	Axes of axis control groups: X2, Z2, Y2
	G622	Axes of axis control groups: X1, Z1, Y1	Axes of axis control groups: X2, Z2, C1, Y2
Front/back parallel machining	G630	Operation X1, Z1 and Y1: No operation	Operation X2, Z2: No operation Y2: No operation
		Selectable tool numbers: T0100 to T1400	Selectable tool numbers: T3000 T3100 to T3800
		Axes of axis control groups: X1, Z1, Y1, (C1)	Axes of axis control groups: X2, Z2, (C2), Y2

Machining pattern	Command code	$1	$2
Pick-off, center support (Z1-Z2 superimposition)	G650	Operation X1, Z1 and Y1: No operation Selectable tool numbers: T0100 to T0500 Axes of axis control groups: X1, Z1, Y1, (C1)	Operation X2: Movement to the center of guide bushing (Argument U0 (Specifying no operation of X2) is available) Y2: Movement to the center of guide bushing (Argument V0 (Specifying no operation of X2) is available) Z2: Movement to return position (Argument W0 (Specifying no operation of Z2) is available) Selectable tool numbers: T3000 Axes of axis control groups: X2, Z2, (C2), Y2 Superimpose Z1 superimposed on Z2
Front/back simultaneous machining	G660	Operation X1, Z1 and Y1: No operation Selectable tool numbers: T0800～T1000 T1100 to T1400 Axes of axis control groups: X1, Z1, Y1, (C1)	Operation X2: No operation Y2: No operation Z2: Movement to return position (Argument W0 (Specifying no operation of Z2) is available) Selectable tool numbers: T5100 to T5900 Axes of axis control groups: X2, Z2, (C2), Y2

11.7.2 Machining pattern flow

The machining pattern should be specified for both $1 and $2.

[Sample program]

$1	$2
G610 .... $1 single machining	G610 .... $1 single machining
:	:
G660 .... Front/back simultaneous machining	G660 .... Front/back simultaneous machining
:	:
G630 .... Front/back parallel machining	G630 .... Front/back parallel machining
:	:
G650 .... Pick-off	G650 .... Pick-off
:	:
G600 .... Free pattern (machining pattern cancel)	G600 .... Free pattern (machining pattern cancel)
:	
M56	
G999	G999
N999	N999
M02	M02
M99	M99
%	%

[Note]

- Execution of machining patterns is automatically queued.
- Before selecting the desired machining pattern, be sure to cancel the coordinate system shift and offset. For selection of G650, however, it is not necessary to cancel the offset.
- Specify a machining pattern in a program for both axis control groups 1 ($1) and 2 ($2).
- Each axis control group should hold the feed-per-rotation command and feed-per-minute command as modal functions.
- A machining pattern is a modal G code that remains valid until other machining pattern is specified.

11.7.3 Free pattern (machining pattern cancel) (G600)

The free pattern can be used for front/back parallel machining. In addition, the operation can cancel the machining patterns of $1 single machining (G610), front/back parallel machining (G630), pick-off/center support (G650) and front/back simultaneous machining (G660) in the power-on state.

[Note]

$1	$2
G600	G600 W0

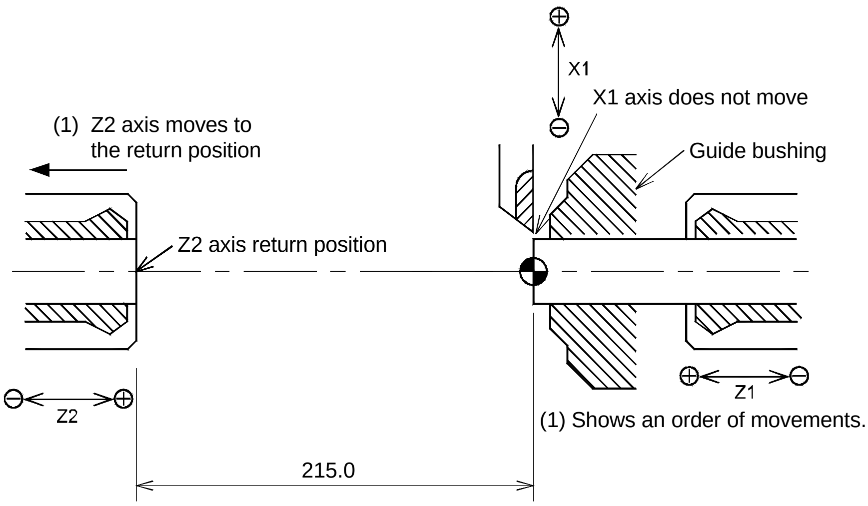
Axis control group

Specify this command for both axis control groups $1 and $2.

$2 W0: The back headstock (Z2 axis) does not move. If this argument is not specified, the back headstock (Z2 axis) moves to the return position.

Operation example

The back headstock (Z2 axis) moves to the return position at rapid feedrate.



Macro specification

Command code	G600	
Name	Free pattern (machining pattern cancel)	
Axis control group	$1	$2
Axes of axis control group	X1, Z1, Y1, (C1)	X2, Z2, (C2), Y2
Superimpose	–	–
Coordinate system	–	–
Argument	–	W0: The back headstock (Z2 axis) does not move.
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	T0100 to T1400,T2000 T5100 to T5900 T0100 to T1000 K2	T3000 T3100 to T3800 T5100 to T5900
Others	–	–

[Sample program]

$1	$2
G600 .... Free pattern (machining pattern cancel)	G600 .... Free pattern (machining pattern cancel)
	T5100
	G00 Z–0.5
:	G01 Z3.0 F0.1 T31
:	G00 Z–0.5 T00
	:
!2 L1	!1 L1
	T3100
T0300 X17.0	:
G00 X9.0 Z–0.5 T03	
G01 X12.0 Z1.0 F0.08	
Z17.8	
X15.2	
X17.0 Z18.7	
:	
:	

[Note]

- When K2 argument for T command is specified, care must be taken for interference between the back headstock, front/back drilling holder, and back tool post.
- To use a tool T50’s for back machining, be sure to use the queuing state as shown in the sample program.

  Because tool T50’s is mounted on the gang tool post, a gang tool of another tool number cannot be selected during machining with tool T50’s.

11.7.4 $1 single machining (G610)

This pattern allows the outer diameter machining by gang tool post or inner diameter machining by opposite tool post in axis control group 1.

[Note]

$1	$2
G610	G610 W0

Axis control groups

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

$2 W0: The back headstock (Z2 axis) does not move. If this argument is omitted, the Z2 axis moves to the return position.

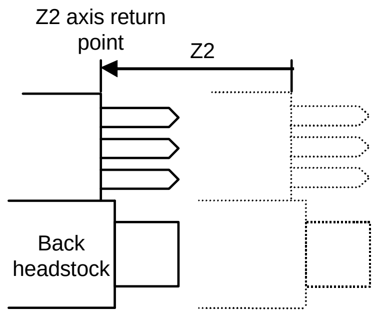
Operation example

$1: G610 does not cause any axis movement.
When a tool (T20's) is selected after the designation of G610, the gang tool post (X1 and Y1 axes) retracts, the opposite tool post (X2 axis) starts movement toward the specified tool position, and the opposite tool post (Z2 axis) moves to the forward end position. All of these operations are executed simultaneously. If the machine judges the gang tool post will interfere with the opposite tool post, it automatically controls the feed rate of the Z2 axis before the interference area and, when the gang tool post passes through the area, feeds the Z2 axis in rapid feed. If the free tool layout pattern is selected for gang tool post, the parallel machining is not performed.

CAUTION

If the machine is operated in handle mode for on-machine program check, timings of gang and opposite tool posts do not match. However, these timings will shift in a safe manner and, therefore, interference will not occur.

$2：The back headstock (Z2 axis) moves to the return position at rapid feed.



Macro specification

Command code	G610	
Name	$1 single machining	
Axis control group	$1	$2
Axes of axis control group	X1, Z1, Y1, (C1), X2, Y2	–
Superimpose	–	–
Coordinate system	–	–
Argument	–	W0: The back headstock (Z2 axis) does not move. If the argument is omitted, the Z2 axis moves to the return position.
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	–
T command	T0100 to T1400 (on gang tool post) T2100 to T2600	–
Others	–	–

[Sample program 1]

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

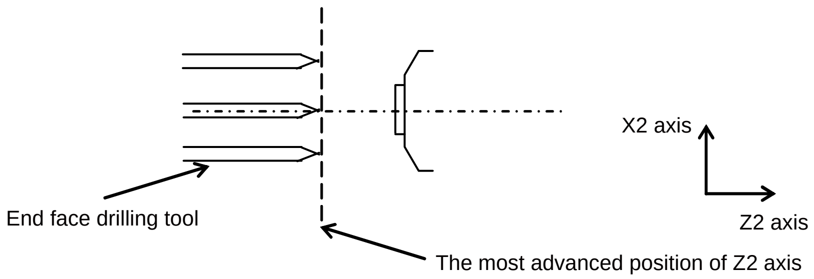
$1	$2
:	:
G610 .... $1 single machining	G610 .... $1 single machining
T2100 (X2 Z1 C1)	
:	
T2300 (X2 Z1 C1)	
:	
T0500 (X1 Z1 Y1 C1)	
:	
T2100 (X2 Z1 C1)	
:	
G630 .... Front/back parallel machining	G630 .... Front/back parallel machining
T0300 (X1 Z1 Y1 C1)	T3200 (X2 Z2 C2)
:	:

[Sample program 2]

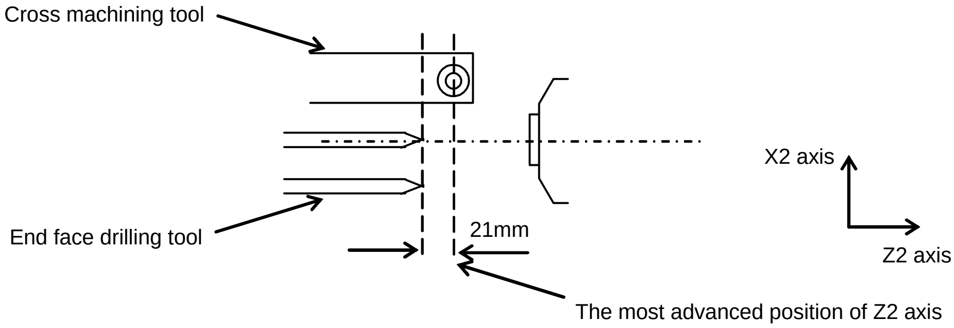
$1 single machining (G610)   Machining with the cross machining and drilling tools (L20X)

$1	$2
:	:
G610 .... $1 single machining	G610 .... $1 single machining
T2300 (X2 Z1 C1) ...... Select cross machining tool	:
:	
T2100 (X2 Z1 C1) ...... Select drilling tool	
G53 W21.0................. Z1 axis workpiece coordinate shift command	
:	
G600	G600
:	:

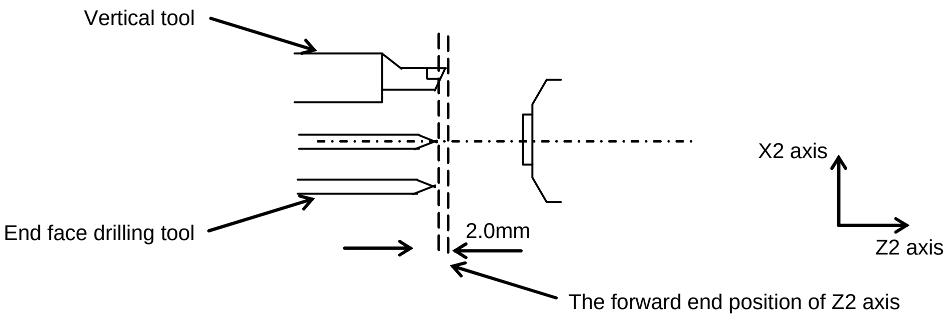
The forward end position of opposite tool post (Z2 axis) with end-face drilling tools mounted: Z2 = 205.0 mm



The forward end position of opposite tool post (Z2 axis) with cross machining tools mounted: Z2 = 184.0 mm



The forward end position of opposite tool post (Z2 axis) with vertical tool mounted:
Z2 = 203.0 mm



[Note]

- In $1, the machining by alternately using a a tool on the opposite tool post (T20's) and a tool on the gang tool post (T01's) can be performed.
- If a tool on opposite tool post (T20's) is called during $1 single machining (G610), after the X2 axis of the opposite tool post is moved, then the Z2 axis moves to the forward end position (205.0) in the Z2 axis machine coordinate). Therefore, if the workpiece is protruded from the guide bushing, interference between the workpiece and the tool may occur. After the workpiece is escaped , the tool must be called.
- The forward end position depends on the type of tools being mounted.
  Only the end-face drilling tool is mounted: 205.0 mm
  Cross machining tool is mounted: 184.0
  Vertical tool (without cross machining tool) is mounted: 203.0
- When the cross machining and drilling tools are mounted on tool post, the forward end position of Z2 axis is always 184.0 mm even if the drilling tool is called.
  In this case, shift the Z axis workpiece coordinate at the workpiece side by the difference in the longitudinal direction.

11.7.5 Inner/outer diameter simultaneous machining (G620)

With the Z2 axis superimposed on the Z1 axis, G620 simultaneously enables:

$1 to perform outer diameter machining with a gang tool, and

$2 to perform inner diameter machining with a front drilling tool on the opposite tool post

The pinch milling can be performed.

CAUTION

When you intend to use the inner/outer diameter simultaneous machining (or pinch milling machining), pay strict attention to interference. For details, see *<13.46 Simultaneous Machining>*.

[Note]

$1	$2
G620	G620 Z☐

Axis control groups

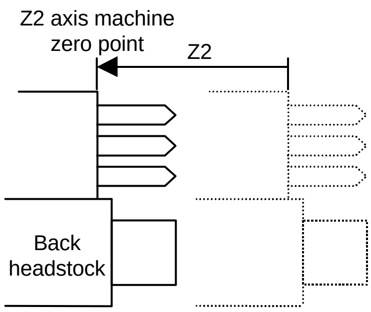
Specify this command for the axis control groups $1 and $2.

$2 Z☐	The opposite tool post (Z2 axis) moves to the specified position (in workpiece coordinate). In the Z2 axis workpiece coordinate system, the workpiece end face on the front $1 is zero (0) position. If the argument is omitted, the Z2 axis does not move.
G621	Main spindle C-axis (C1) $1 side valid
G622	Main spindle C-axis (C1) $2 side valid

Specify G621 or G622 for both axis control groups $1 and $2. If the main spindle C-axis is not mounted, these commands are ignored.

Operation sample

The axis does not move if no argument is specified.



Macro specification

Command code	G620			
Name	Inner/outer diameter simultaneous machining			
Axis control group	$1		$2	
Axes of axis control group	G620	X1, Z1, Y1, (C1)	G620	X2, Z2, (C2), Y2
	G621	X1, Z1, Y1, C1	G621	X2, Z2, Y2
	G622	X1, Z1, Y1	G622	X2, Z2, C1, Y2
Superimpose	–		Z2 superimposed on Z1	
Coordinate system	–		The coordinate system for the Z2 axis is established by defining the workpiece end face as zero position (i.e. reference).	
Argument	–		Z□: The opposite tool post Z2 axis moves to the specified position. The Z2 axis does not move if no 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	T0100 to T0500 (on gang tool post)		(on opposite tool post) T2100 to T2600	
Others	–		–	

[Sample program]

$1		$2	
G600		G600	
:		:	
G99 M3 S1=4200		:	
G620 (X1 Z1 Y1 C1)............	Inner/outer diameter simultaneous machining	G620 (X2 Z2 C2).................	Inner/outer diameter simultaneous machining
T0200		T2300	
G0 X17.0 Z–1.0		G0 Z–5.0	
G1 X4.0 F0.3		G98	
G1 Z30.0 F0.1		G1 Z4.5 F200	
		G1 Z14.5 F250	
G622 (X1 Z1 Y1)..................		G622 (X2 Z2 Y2 C1)............	
:		:	

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

[Note]

- When G620 (inner diameter/outer diameter simultaneous machining) is specified, the Z2 axis moves with superimposing on Z1 axis. If the axis movement command is specified for Z2 axis, the Z2 axis moves on the workpiece coordinate of Z1 axis.
- G621 or G622 can be specified in G620 mode only. If specified in any other mode, an alarm will occur.
- Specify G621 or G622 for both axis control groups $1 and $2
- The workpiece coordinate of Z2 axis is set as follows when G0 Z0 command is specified:

  The tool nose of cross machining tool is set to the end face of workpiece The tool nose of vertical tool is set to the end face of workpiece

11.7.6 Front/back parallel machining (G630)

Use this machining pattern to perform outer or inner diameter machining or front drilling with the gang tool post on axis control group 1 ($1) and back machining with the back tool post on axis control group 2 ($2).

[Note]

$1	$2
G630	G630

Axis control group

Specify this command for both axis control groups $1 and $2.

When a tool (T01's) is selected after the designation of G630, the opposite tool post (Z2) retracts. At the same time, the gang tool post (X1 and Y1 axes) starts movement toward the specified tool position.
These operations are executed simultaneously. If the machine judges the gang tool post will interfere with the opposite tool post during the gang tool post tool positioning, it automatically controls feed rates of X1 and Y1 axes before the interference area and, when the opposite tool post passes through the area, feeds X1 and Y1 axes in rapid feed. If the free tool layout pattern is selected for gang tool post, the parallel machining is not performed.

CAUTION

If the machine is operated in handle mode for on-machine program check, timings of gang and opposite tool posts do not match. However, these timings will shift in a safe manner and, therefore, interference will not occur.

Macro specification

Command code	G630	
Name	Front/back parallel machining	
Axis control group	$1	$2
Axes of axis control group	X1, Z1, Y1, (C1)	X2, Z2, (C2), Y2
Superimpose	–	–
Coordinate system	–	–
Argument	–	–
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	T0100 to T1400 (on gang tool post)	T3000 T3100 to T3800
Others	–	–

[Sample program]

$1	$2
G600	G600
:	:
G630 .... Front/back parallel machining	G630 .... Front/back parallel machining
T0300 (X1 Z1 Y1 C1)	T3100 (X2 Z2 C2)
G0 X21.0 Z0	G0 Z–1.0
G1 X–0.5 F0.03	G1 Z2.5 F0.05
:	:
G600	G600
:	:

[Note]

- If you want to specify the Z2 axis in $2 after the designation of G630, select the T30’s tool first. If the command is specified for the Z2 axis without selection of T30’s tool, an illegal address error will occur.
- The opposite tool post (Z2) retracts when the T01's tool is selected in $1 after the designation of G630. The opposite tool post (Z2) also retracts when the T30's tool is selected or M34 (product collection) is specified in $2.
- If the free tool layout pattern is selected for gang tool post, the opposite tool post (Z2) retracts when G630 is specified.

11.7.7 Pick-off, center-support (G650)

This machining pattern enables the back spindle to pick off the workpiece and support the center of the workpiece.

[Note]

$1	$2
G650	G650 Z☐ U0 V0 W0

Axis control group

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

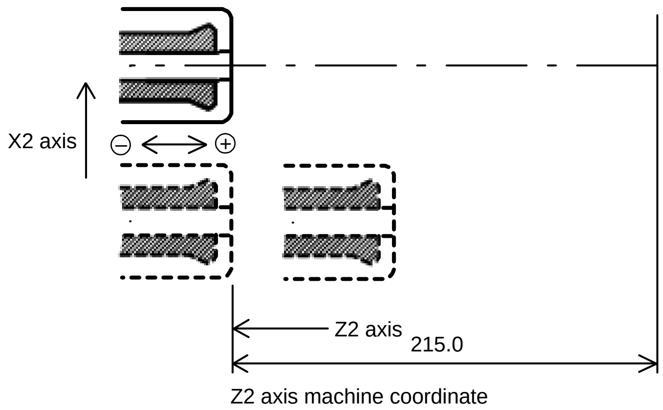
$2 Z☐ The back spindle (Z2 axis) moves to the specified position (in workpiece coordinate). In the Z2 axis workpiece coordinate system, the workpiece end face on the front $1 is zero (0) position. If this argument is not specified, the Z2 axis moves to the return position.

U0 Specify this argument not to move the back spindle (X2 axis) to the center of the guide bushing.
If this argument is not specified, the X2 axis moves to the center of the guide bushing.

V0 Specify this argument not to move the back spindle (Y2 axis) to the center of the guide bushing.
If this argument is not specified, the Y2 axis moves to the center of the guide bushing.

W0 Specify this argument not to move the back spindle (Z2 axis) to the return position.
If this argument is not specified, the Z2 axis moves to the return position. However, the back spindle (Z2 axis) does not move to the return position when the back spindle (X2 axis) is at the center of the guide bushing.

Operation example



The back spindle (Z2 axis) moves to the return position and the back spindle (X2 axis) moves to the center of the guide bushing. However, the back spindle (Z2 axis) does not move to the return position when the back spindle (X2 axis) is at the center of the guide bushing.

Set the work coordinate system for the Z2 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
Axes of axis control group	X1, Z1, Y1, C1	X2, Z2, C2, Y2
Superimpose	–	Z2 superimposed on Z1
Coordinate system	–	The coordinate system for the Z2 axis is established by defining the workpiece end face as zero position.
Argument		Z☐: The back spindle Z2 axis moves to the specified position. If this argument is not specified, the Z2 axis moves to the return position. U☐: Specify this argument not to move the back spindle (X2 axis) to the center of the guide bushing. If this argument is not specified, the X2 axis moves to the center of the guide bushing. V☐: Specify this argument not to move the back spindle (Y2 axis) to the center of the guide bushing. If this argument is not specified, the Y2 axis moves to the center of the guide bushing. W☐: Specify this argument not to move the back spindle (Z2 axis) to the return position. If this argument is not specified, the Z2 axis moves to the return position.
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	T0100 to T0500 (on gang tool post)	T3000 (Back spindle)
Others	–	–

[Sample program]

$1	$2
:	
G99 M03 S1=1500 M24 S2=1500	
T0100	
G0 X17.0 Z30.0	
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
X–3.0	
:	:

[Note]

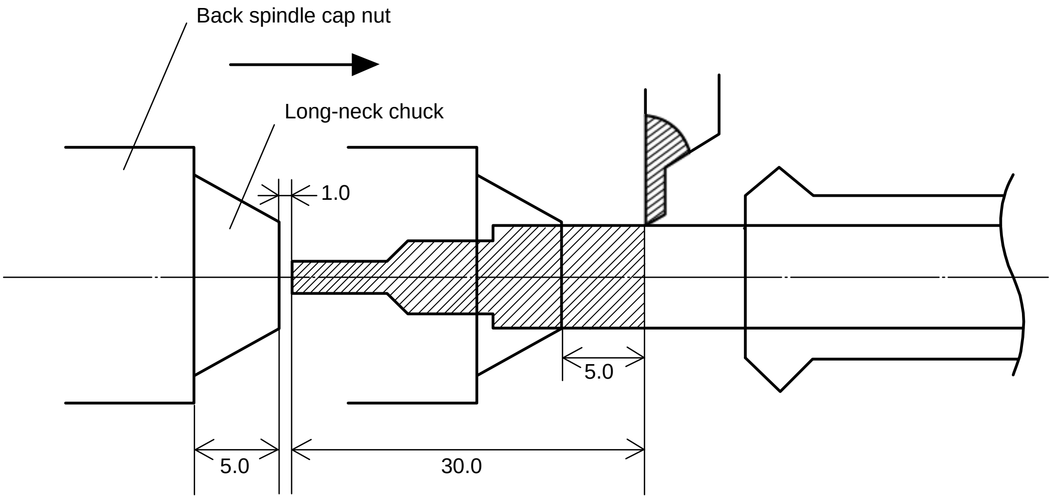
- When G650 (pick-off, center support) is specified, the Z2 axis moves with superimposing on Z1 axis. If the axis movement command is specified for Z2 axis, the Z2 axis moves on the workpiece coordinate of Z1 axis.
- Position the Z1 axis at cut-off position before specifying the G650 command.
- During pick-off operation (G650), you can move the Z2 axis to machine zero point by specifying the G231 command. *See <15.17 Pick-off Cancel (G231)>*.
- When machining the workpiece with gang rotary tool while performing center support, the gang rotary tool driving device may interfere with the coolant nozzle located at upper portion of back spindle, depending on the position of back spindle (Z2 axis).

Sample program for pick-off in use of back long-neck chuck

- The long-neck chuck shall protrude from the end face of the cap nut by 5.0 mm.
- The workpiece protrusion length from the long-neck chuck end face shall be 5.0 mm.
- The cut-off tool shall be □12 and left-handed .
- The total length of the workpiece shall be 30.0 mm.
  1. Enter 5.0 as "Back chuck extend length" in the machining data.
  2. Enter 5.0 as "Back work extend length" in the machining data.

[Sample program]

$1	$2	
G600	G600	
:		
G99 M3 S1=2000		
M24 S2=2000		
T0100		
G0 X21.0 Z42.0 T1		
G650	G650	
	M16	
	M72	
	G0 Z–1.0	1. The end face of the back chuck is positioned at the front position separated from the front end face of the workpiece by 1.0 mm.
	M77	
	G98 G1 Z25.0 F3000	2. Move the chuck to the position separated from the front end face of the workpiece by 25.0 mm (the workpiece protrusion length from the back end face of the chuck after re-chucking: 5.0 mm (total length of workpiece 30.0 – 25.0)).
	M15	
!2 L1	!1 L1	
:	:	



[Note]

- When a numeric value is specified in "Back chuck extend length" on the Machining Data screen, the Z2 axis moves to the position where the front workpiece end face and the back long-neck chuck end face are aligned by the Z0.0 command of $2 specified after G650.
- When "SUPPORT" is selected for "Back Spindle" on the Machining Data screen, the Z2 axis moves to the position where the front workpiece end face and the end face of the long workpiece cap nut brass part are aligned by the Z0.0 command of $2 specified after G650. In this control, the length of the cap nut with the support for a long workpiece which is 10.0 mm longer than the ordinary cap nut is taken into account.

11.7.8 Front/back simultaneous machining (G660)

This pattern allows front/back simultaneous machining with drilling holder on gang tool post in front $1 and back $2.

[Note]

$1	$2
G660	G660 W0 X2

Axis control group

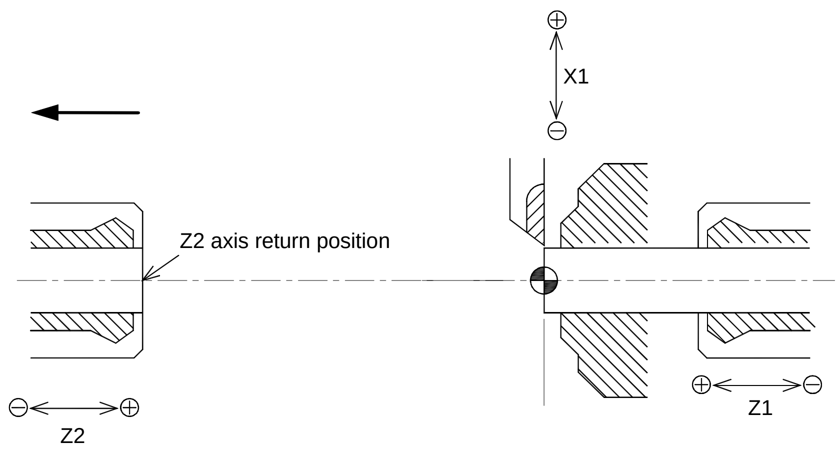
Specify this command for the axis control groups $1 and $2.

$2 W0 : The Z2 axis (back headstock) does not move. If the argument is omitted, the Z2 axis moves to the return position.

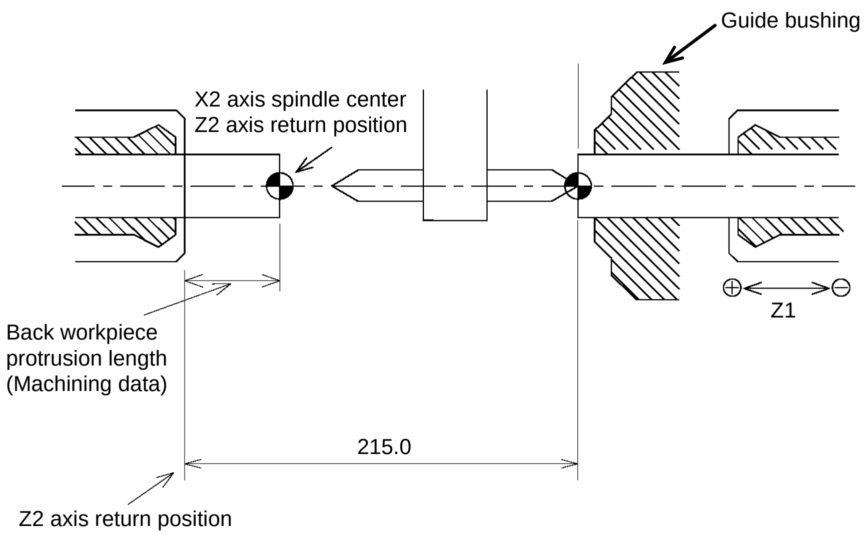
$2 X2 : The X2 axis is superimposed on the Y1 axis. Specify this argument when you intend to move the Y1 axis on front $1 in the front/back simultaneous machining. With the movement of Y1, the X2 axis moves as well.

Operation example

The back headstock (Z2 axis) moves to the return position in rapid feed rate.



After T11's or T51's tools are selected, the axis moves to the following position.



Macro specification

Command code	G660	
Name	Front/back simultaneous machining	
Axis control group	$1	$2
Axes of axis control group	X1, Z1, Y1, C1	X2, Z2, C2, Y2
Superimpose	–	The X2 axis is superimposed on the Y1 axis.
Coordinate system		Z2: The coordinate system for the Z2 axis is established by defining the position where the tool nose of the vertical end face drilling holder is aligned with the back workpiece end face as the reference position.
Argument		W0: The back spindle (Z2 axis) does not move. If the argument is omitted, the Z2 axis moves to the return position.
		X2: The X2 axis is superimposed on the Y1 axis.
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	T0800 to T1000 T1100 to T1400 (Drilling tool on gang tool post)	T5100 to T5900 (except B axis) (Drilling tool on gang tool post)
Others	–	–

[Sample program]

$1	$2
:	:
G660 .... Front/back simultaneous machining	G660........................ Front/back simultaneous machining
T1200	T5200
G00 Z–0.5 T12	G00 Z–0.5 T52
G01 Z5.0 F0.03	G01 Z5.0 F0.03
:	:

[Note]

- Specify commands to rotate the main spindle in $1, and specify commands to rotate the back spindle in $2.
- When the G660 command is executed, the queuing state is automatically placed at selection of a tool in the T01/T10's and a tool in the T50's. Select a tool in the T01/T10's and a tool in the T50's when replacing the current tool. Be sure to specify the same number as the first digit of the tool number in the T01/T10's and the tool number in the T50's.
- Front/back simultaneous machining (G660) cannot be performed on the B axis drilling spindle (MEU307).

11.7.9 Superimpose control

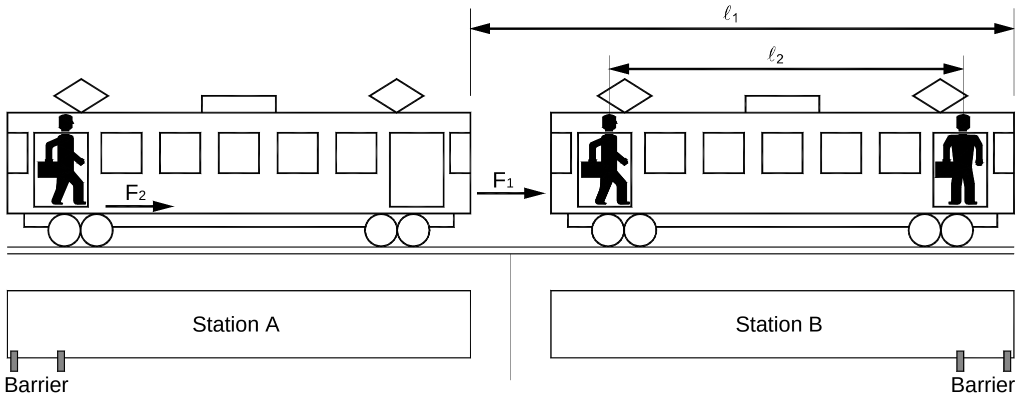
This section explains superimpose control giving an example.

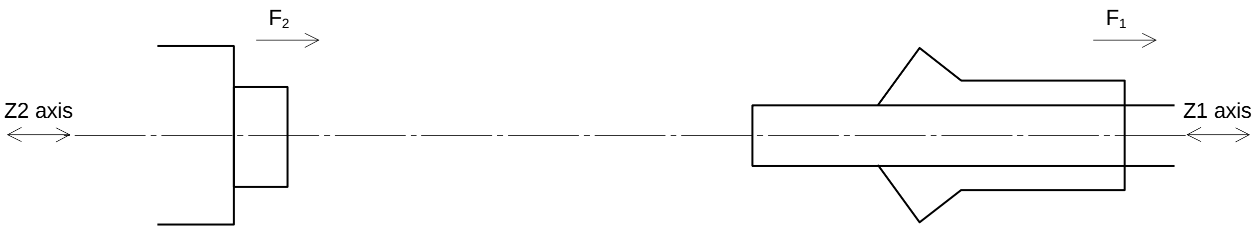
Mr. A got on the end of the train. He was getting off at the next station. The front end of the train is close to the barrier. So, he walked to the front end of the train and got off.

The following figure shows the work done by the train and Mr. A.

While the train was running the distance (work λ_1) at F_1 speed, Mr. A disk his work for the distance of λ_2 at F_2 speed. As a result, the work $\lambda_1 + \lambda_2$ was done. The train acts Z1 as the reference coordinate axis. Mr. A acts as the Z2 axis. Mr. A (Z2 axis) did his work while being on the train (Z1 axis). This is called superimpose control.

In terms of this machine, the train corresponds to the headstock (Z1 axis), and Mr. A corresponds to the back headstock tool post (Z2 axis).





Operation with superimpose control:

- If Z2 axis movement is not specified and Z1 axis movement is specified, the Z2 axis moves synchronously with the Z1 axis.
- If Z2 axis movement is specified and Z1 axis movement is not specified, only the Z2 axis moves.
- If both Z2 axis movement and Z1 axis movement are specified, the Z2 axis moves while being superimposed on the Z1 axis (being on the Z axis).

Explanation of the above figure:

When $F_1 = F_2$: The Z2 axis appears to be in stopped state.
When $F_1 < F_2$: The Z2 axis moves forward at speed of F_2-F_1.
When $F_1 > F_2$: The Z2 axis moves backward.
The queuing command is specified in the program for adjusting the timing of the Z1 and Z2 axes.

[Note]

A machining pattern command cannot be executed in MDI mode.

Product Code	C-L220E XII
Document Code	2E1-1101
Mfg. No.	L220E/0001 ~
Issue Date	2014.1

12. Command Codes

12.1 Summary of Addresses Used in NC Programs

As described in previous sections, commands used in NC programs are alphanumeric codes (alphabetic character and number) associated with some special symbols.

The alphabetic character in each command is referred to as an address.

This chapter describes the meanings and functions of addresses, $, and ! and explains how to specify their respective commands.

Command address example

Preparation function

$\mathbf{G}\underbrace{\square\square}$ Specified code

G00 Rapid feed

G01 Linear cutting

G02 Circular arc cutting

Preparation function

$\mathbf{G}\underbrace{\square\square\square}$ Specified code

G660 Front/back simultaneous machining

G630 Front/back parallel machining

Miscellaneous function

$\mathbf{M}\underbrace{\square\square}$ Specified code

M03 Spindle forward rotation

M06 Close chuck

M52 Turn on coolant supply

Tool selection command

$\mathbf{T}\underbrace{\square\square}_{\text{Tool No.}}\underbrace{\Delta\Delta}_{\text{Offset No.}}$

Coordinate value address example

Move distance of diametrical direction axis (X axis)

X☐ mm
(U)

Move distance of longitudinal direction axis (Z axis)

Z☐ mm
(W)

Radius

R☐ mm

Time

U☐ sec
(P)

Address for specifying cutting condition example

Main spindle speed

S□=☐ min^{-1}

Specified code   From cutting conditions

Feed rate

F☐ mm/min (mm/rev)

From cutting conditions

Program configuration address example

Program No.

O□□□□□□□□

Eight-digit command

This command can specify any program number within eight digits, except 8000 to 9000. Note, however, that O00000000 cannot be set.

Sequence No.

N□□□□□

Five-digit command

This command can specify any sequence number within five digits. Note, however, that N999 cannot be set.

Queuing command example

Single queuing	**!**
Queuing No. for other axis control group	**L**☐

12.2 Axis Commands (Feed Commands)

Outline

For the commands used to feed an axis, two types of command are available. They are absolute command and incremental command.

- Absolute (absolute position) command

  An absolute command specifies the axis feed target position in an absolute value referenced from the fixed position (program zero point).

- Incremental (relative position) command

  An incremental command specifies the axis feed target position as a relative position referenced from the current position.

Command Format Absolute	Command Format Incremental	Feed Axis
X□. □	U□. □	X1 or X2 axis
Z□. □	W□. □	Z1 or Z2 axis
Y□. □	V□. □	Y1 or Y2 axis
C□. □	H□. □	C1 or C2 axis

[Note]

Specify a numerical value with a decimal point.
If a numerical value is specified without a decimal point, the value is judged a micron unit value.

Example

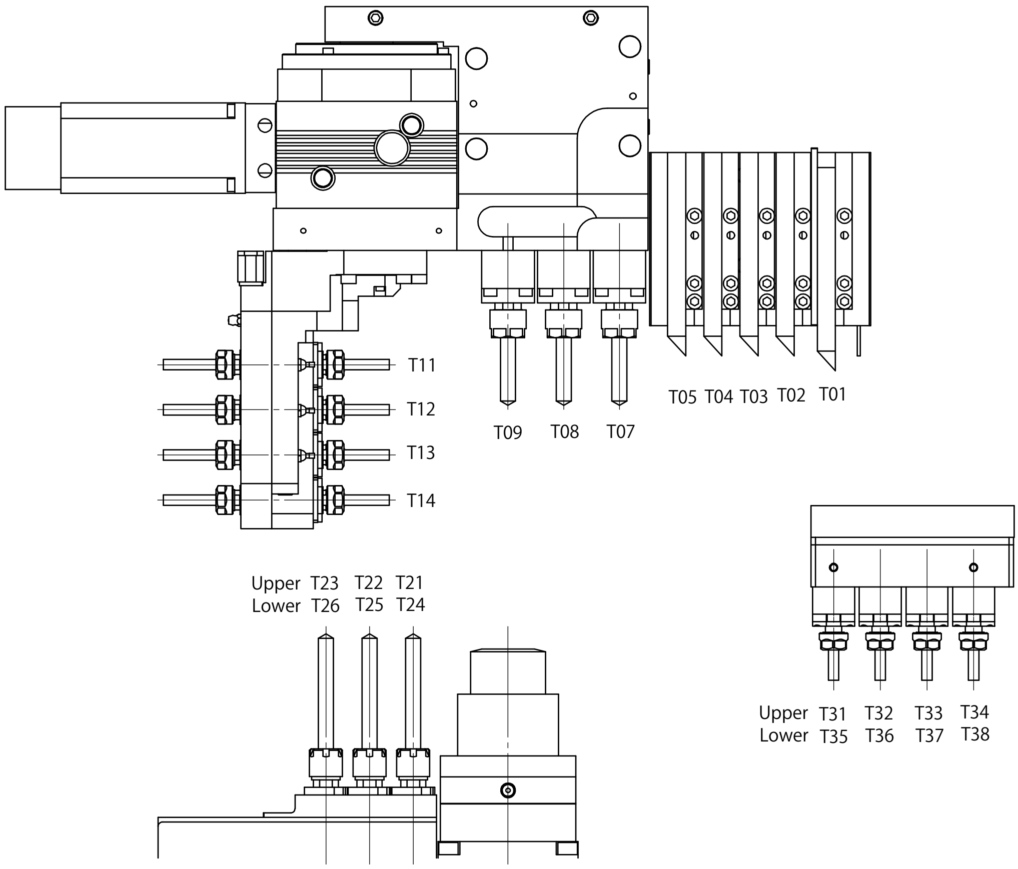
If "X10;" is specified,
The axis moves to the X0.010 position when the control specification is the micron specification.
The axis moves to the X0.0010 position when the control specification is the sub-micron specification.

12.3 T Functions (Tool Codes)

T codes can be specified to provide target tools with selections, positioning and coordinate setting easily. This enables tools to be selected and machining to be continued without specification of many commands.

- Gang tool (T01 to T05)
- Stopper (T20)
- Rotary tool on gang tool post (T07 to T14)
- Front drilling tool (T21 to T26)
- Back drilling tool (T31 to T38)
- Back spindle (T30)

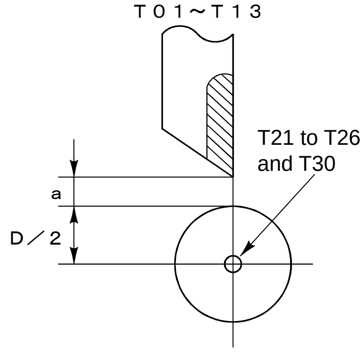
The figure below shows an example of tool layout.



Outer diameter tool (T01 to T14);
Diameter X = Position of "Material diameter D + Positioning point 2a"

Inner diameter tool (T21 to T26), Back spindle (T30);
Center position of materials

Back machining tool (T31 to T38);
Center of back spindle
(Actually, the back spindle moves.)



T-code four-digit command

[Command format]

T□□ΔΔ

□□: Tool number

ΔΔ: Tool wear offset number

A T-code command with a 4-digit number provides a tool with positioning. The first two digits correspond to the tool number. The last two digits correspond to the offset number. Specify 00 as an offset number in the offset cancel state.

T-code two-digit command

[Command format]

TΔΔ

ΔΔ: Tool wear offset number

The T-code command with a 2-digit number makes the offset valid with axis movement following the same block. Specify 00 as an offset number in the offset cancel state.

12.3.1 Tool mounting positions and machining points

Usually, use T01 as a cut-off tool.

The following list applies to GTF3613+ MEU307.

Tool post	Tool number	Outer diameter machining	Rotary tool	Inner diameter machining in front machining	Back spindle	Inner diameter machining in back machining
Gang tool post	01	T0100				
	02	T0200				
	03	T0300				
	04	T0400				
	05	T0500				
	20	T2000 (Stopper)				
	07		T0700			
	08		T0800			
	09		T0900			
	10		T1000			
	11		T1100			
	12		T1200			
	13		T1300			
	14		T1400			
	51		T5100			
	52		T5200			
	53		T5300			
	54		T5400			
Opposite tool post	21			T2100		
	22			T2200		
	23			T2300		
	24			T2400		
	25			T2500		
	26			T2600		
Back spindle	30				T3000	
Back headstock	31					T3100
	32					T3200
	33					T3300
	34					T3400
	35					T3500
	36					T3600
	37					T3700
	38					T3800

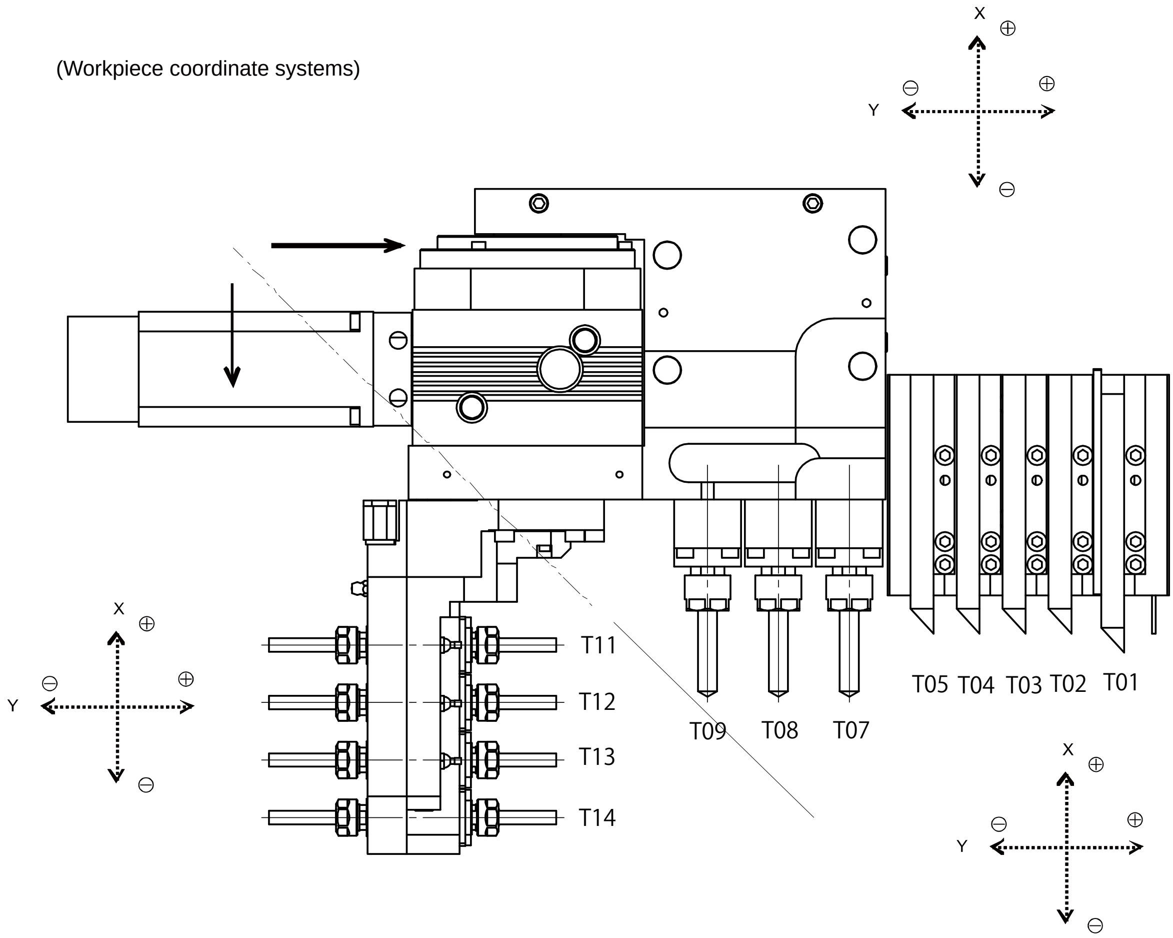
[Note]

- If the T code is specified with "No Tool Name" on Preparation screen, an alarm occurs.
- Normal tool selection is operated at the rapid feed rate. However, normal tool selections do not operate in the door-open state.
- Specify T codes in a single block.
- When arguments described below are specified for tool selection, interference between the tool and the workpiece may occur. Care for use of an argument is needed.

12.3.2 Overview of coordinate systems

If the standard tool holder (GTF3613 + MEU307) is used, work coordinate systems are different between when a tool (T0100 to T0900) on the GTF3613 is selected and when a tool (T1100 to T1000) on the MEU307 is selected.

(Workpiece coordinate systems)



Relationship between workpiece coordinate systems and operation modes

Mode	Description
Automatic operation	The system switches the workpiece coordinate system when any of T11 to T14 is selected.
Program check	The system switches the workpiece coordinate system when any of T11 to T14 is selected.
Preparation	The system switches the workpiece coordinate system when any of T11 to T14 is selected on the Tool Setting screen.
Manual operation	When the workpiece coordinate has been switched, you can feed an axis on the Handle Feed screen after any of T11 to T14 is selected.
MDI	The system switches the workpiece coordinate system when any of T11 to T14 is selected.

12.4 T Code Arguments

T code arguments allow target tools to be selected in various ways. The idling time may be omitted in tool replacement depending on uses.

CAUTION

When an argument is specified for tool selection, interference between the tool and the workpiece may occur. Care for use of an argument is needed.

12.4.1 Tools on the gang tool post (T0100 to T1400)

[Command format]

T□□ΔΔ H☐ Q1 X☐ Y☐ Z☐ K2 E☐ A☐ R☐ M1

[Argument]

H: Specify this argument to temporarily change the positioning point (relief point from material O.D. in the machining data) of the current tool. If the argument is omitted, the tool moves to the position (material diameter + tool positioning point + maximum DIA value of tool set data between current and specified tools).

Q1: Select a tool without moving the current tool to the positioning point. If the argument is omitted, select a tool after moving the current tool to the position (material diameter + tool positioning point + maximum DIA value of tool set data between current and specified tools).

X: Specify the work coordinate of X1 axis after tool selection. If the argument is omitted, the tool moves to the positioning point.

Y: Specify the workpiece coordinate of Y1 axis after tool selection. If the argument is omitted, the tool moves to the zero point of the workpiece coordinate.

Z: Specify the workpiece coordinate of Z axis after tool selection. If the argument is omitted, the Z axis does not move.
If K2 argument is specified together, the Z1 axis is used.

K2: Specify this argument for back machining by gang tool. The axes of axis control groups are set as follows after tool selection.

When T code is specified in $1:

	$1	$2
K2 argument not specified	X1, Z1, Y1 (,C1)	
K2 argument specified	X1, Z2, Y1 (,C2)	

The workpiece coordinate of the Z2 axis is set so that the tool nose directed to the back spindle becomes 0. However, the K2 argument cannot be used for T11's. If specified, an alarm occurs.

W: When the K2 argument is specified, Specify the retract point of the back headstock (Z2 axis) (incremental move distance of the back headstock (Z2 axis) from the current position). The back headstock (Z2 axis) moves to the return position (Z2 axis machine coordinate: 0.0 mm) when this argument is omitted. Specify W0 not to move the Z2 axis to the return position.

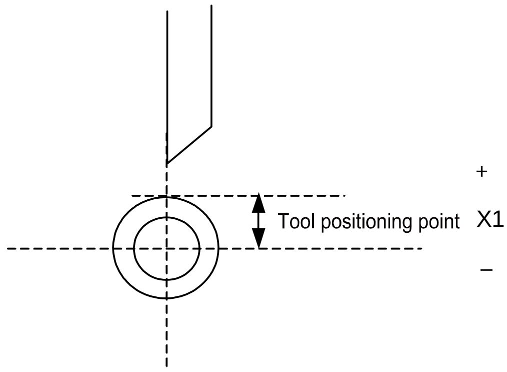
E: Specify this argument to index the spindle at the specified indexing angle during tool selection. If the argument is omitted, the spindle is not indexed. This argument is useful in the transition from turning to secondary machining process. If the K2 argument is specified together, the specification is invalid.
When the spindle C axis option is not specified: Integer between 0.0 and 359.0
When the spindle C axis option is specified: 0.000 to 359.999

R: Specify R to call a tool associated with R when GDF1207, BSE607 or BSE707 is mounted.

M1: Ignore the maximum tool set diameter alarm.

[Note]

If the tool nose of the current tool is located above the relief position (material O.D. + positioning point + maximum DIA value of tool set data between current and specified tools) of the tool post (X1 axis), the tool post (X1 axis) does not move to the tool positioning point.

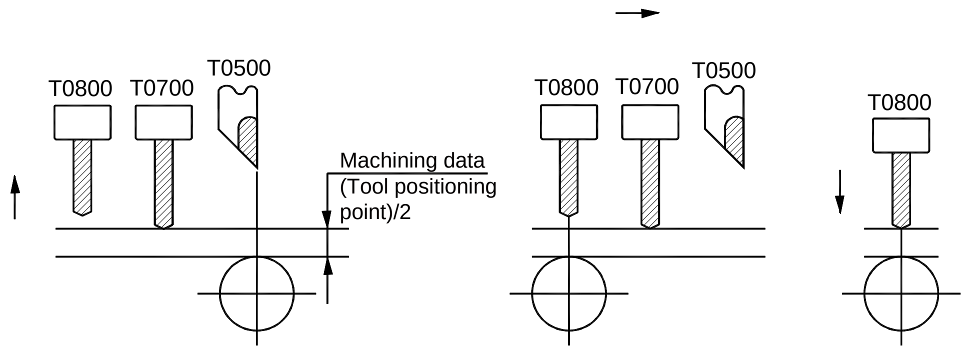


Operation example

T□□□□ Argument not specified

T0100 - T1000 to T0100 - T1000

0500→T0800



(1) When tool selection is specified, the axis automatically moves to the position determined by the following: "Tool positioning point + maximum diameter value of tool set data between currently selected tool and specified tool + maximum tool pattern value"

(2) The Y axis moves to the position of the specified tool.

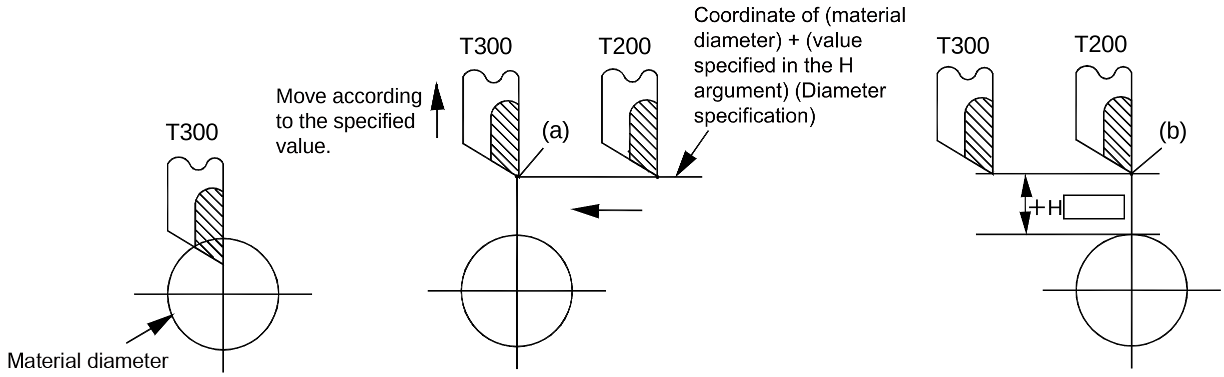
(3) The X axis of the specified tool moves to the "positioning point."

Operation example

T□□□□H□ (T code with H argument)

G00 X8.0 (When the material diameter is 20.0 mm)

T200 H12.0 Moves the current tool to the X32.0 position and switches to the next tool.



(1) Current machining tool

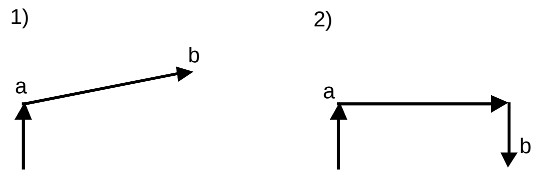
(2) After the tool moves from the position determined by the material diameter to the position (a) in the X direction specified by the H argument, the next tool moves to the position (b).

(3) The tool is selected with the positioning point (b) as specified in the H argument.

1. When the X1 axis of the currently selected tool (Point a) and the X1 axis of the specified tool (Point b) are as shown below:

If the point b is larger than point a, the machine moves as shown in (1).

If the point b is smaller than point a, the machine moves as shown in (2).

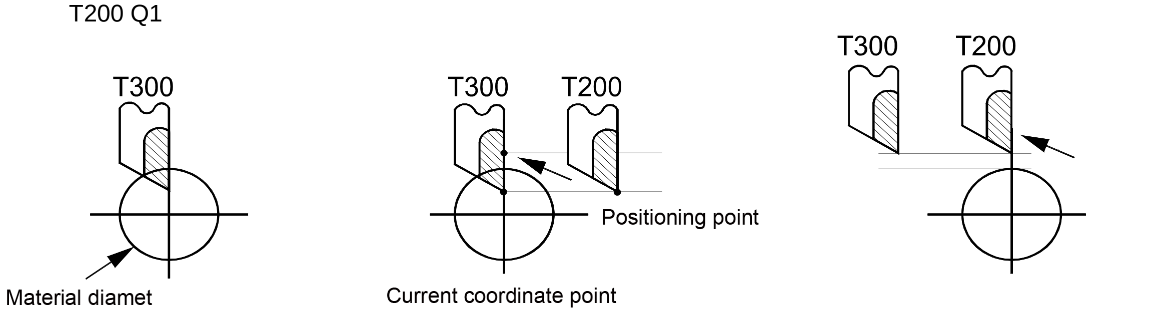


Operation example

T□□□□Q1 (T code with Q1 argument)

When the T□□□□Q1 command is issued, the tool currently being used for machining moves to the positioning point of the next tool and that next tool is selected at the positioning point in rapid feed.

T200 Q1



(1) Current machining tool

(2) The tool moves straight (the minimum distance) to the positioning point of the next tool from the current coordinate point.

(3) The new tool is selected at the positioning point.

[Note]

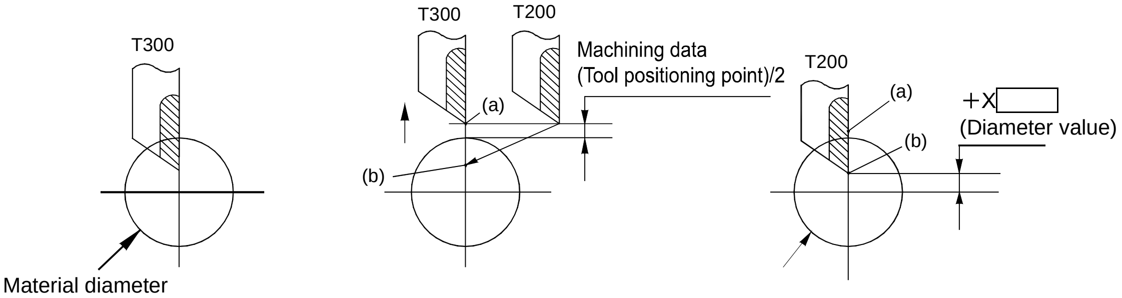
The positioning point determined at this time is "the material diameter + tool positioning point" set as machining data.

Operation example

T□□□□X☐ (T code with X argument)

The tool currently being used for machining moves to the positioning point and the next tool is positioned at the +X☐ position in rapid feed.

T□□□□ X5.0



(1) Current machining tool

(2) The tool moves automatically to the positioning point in response to the tool selection command.

(3) The next tool moves straight (the minimum distance) from point (a) to (b) by the value specified with the X argument.

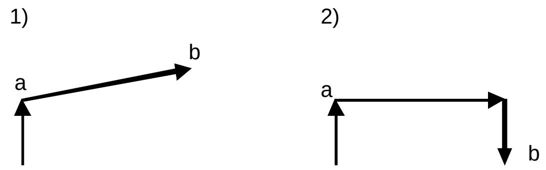
1. See the figure below for operation.

Point a: Position of currently selected tool which has been moved to the position specified by X argument

Point b: Position of the specified tool which has been moved to the position specified by X argument

If the point b is larger than point a, the machine moves as shown in (1).

If the point b is smaller than point a, the machine moves as shown in (2).



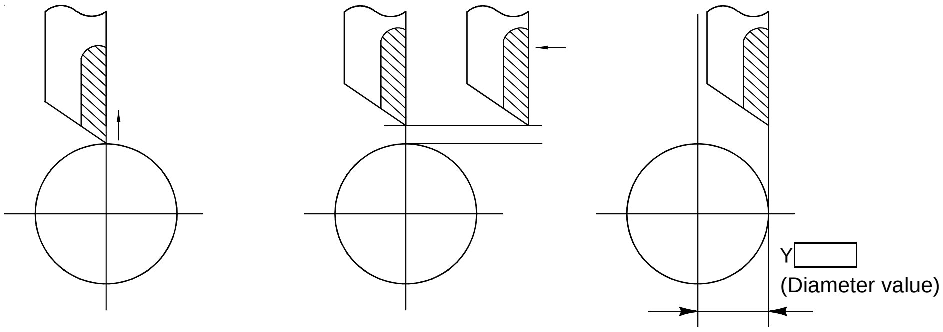
[Note]

The Z argument can be used in the same way as above.

Operation example

T□□□□Y□ (T code with Y argument)

The tool currently being used for machining moves to the positioning point and the next tool moves to the +Y□ position in rapid feed.



(1) Current machining tool

(2) The tool moves to the positioning point.

(3) The specified tool is selected at the positioning point and Y□ away from the center.

Operation example

T□□□□K2 (T code with K2 argument)

The Z2 axis (back spindle) is the target Z axis after tool selection. However, the K2 argument cannot be used for T11's. If specified, an alarm occurs.

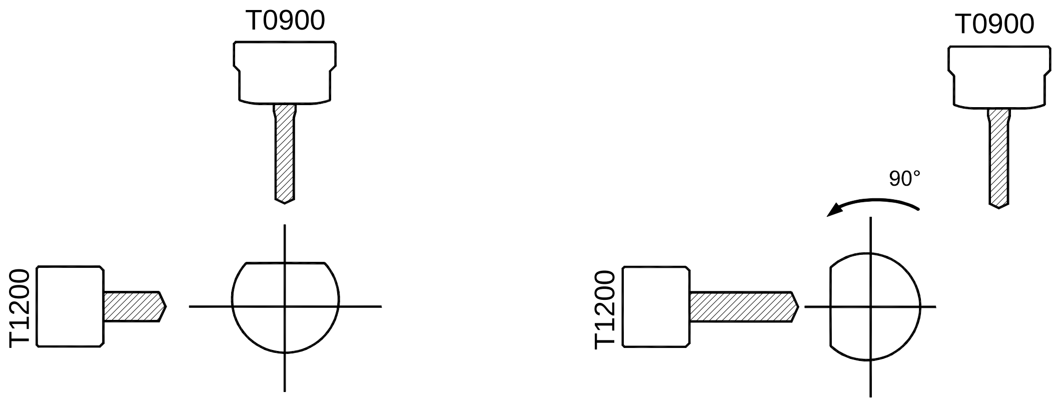
T200 K2

Operation example

T□□□□E□ (T code with E argument)

Specify this T code when you want to select a tool and index the main spindle at the same time.
For a value following E□, specify an indexing angle.

T1200 E90



(1) Current machining tool (T0900)

(2) The front spindle is indexed by 90° simultaneously when T1200 is selected.

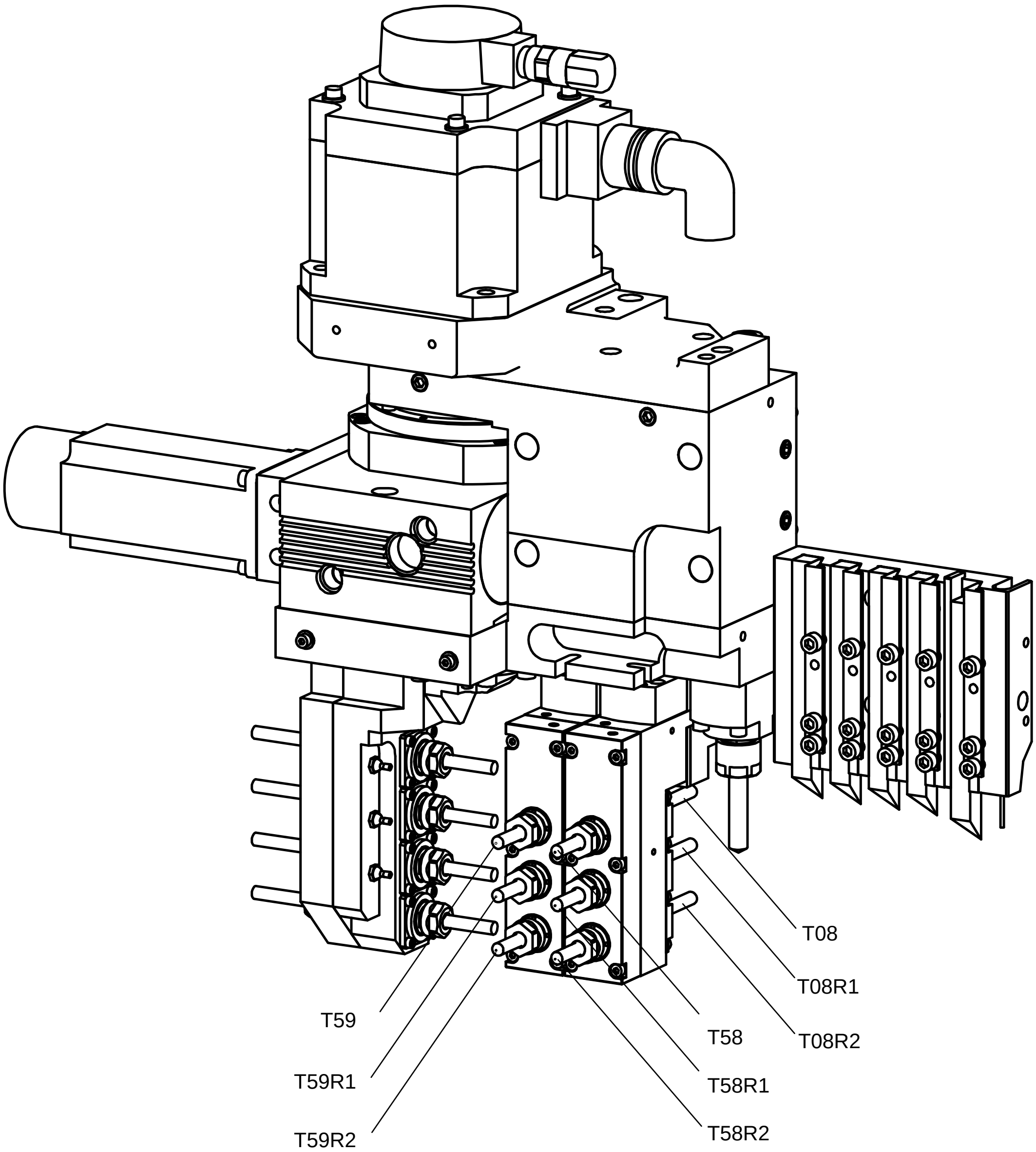
- When an end-face drilling tool is being selected or specified:

  The tip of the specified tool moves directly to the center of workpiece.

Operation example

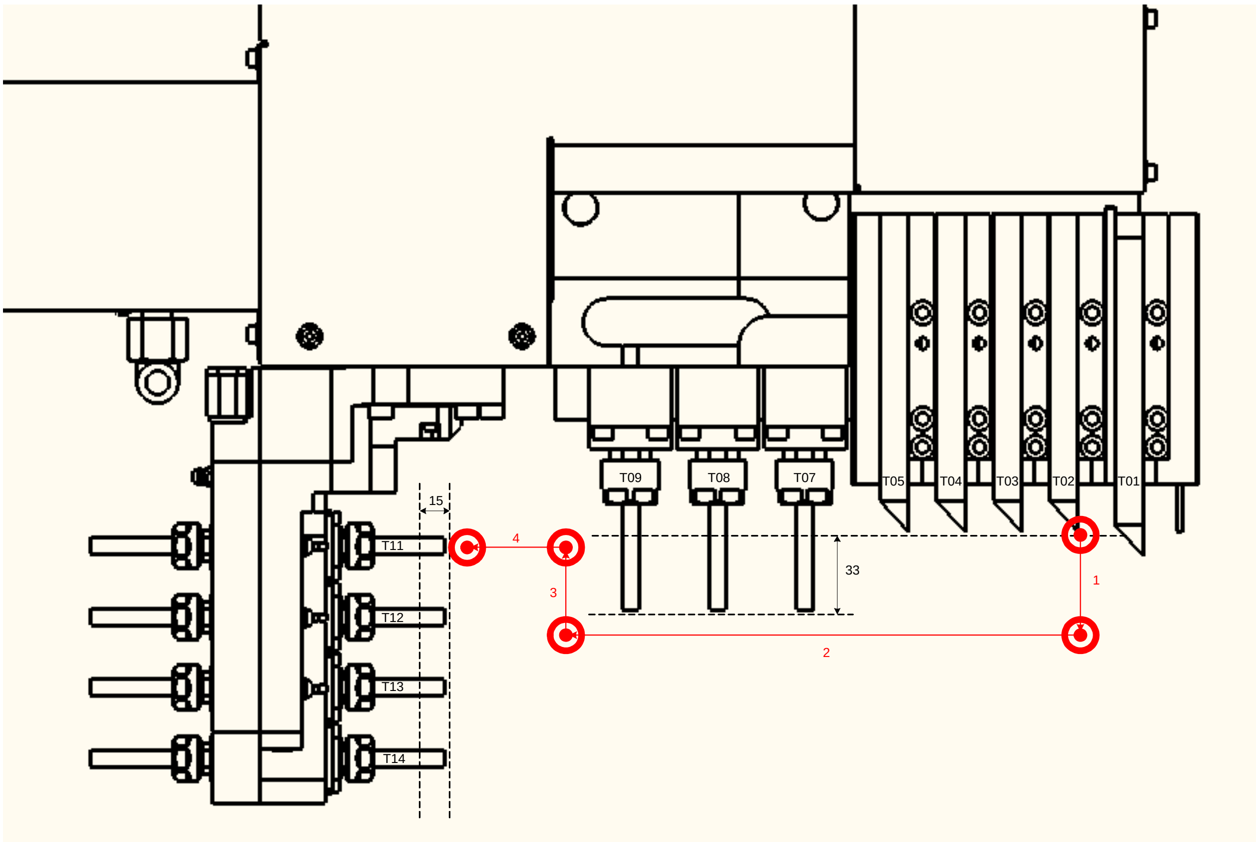
T☐☐☐☐R☐ (T code with R argument)

Specify R1 to call a tool associated with R1 when GDF1207, BSE607 or BSE707 is mounted.



3) Selection of the tool T1100 or T1200 while a tool T0100 to T1000 is selected

Example:
T0200→T1100

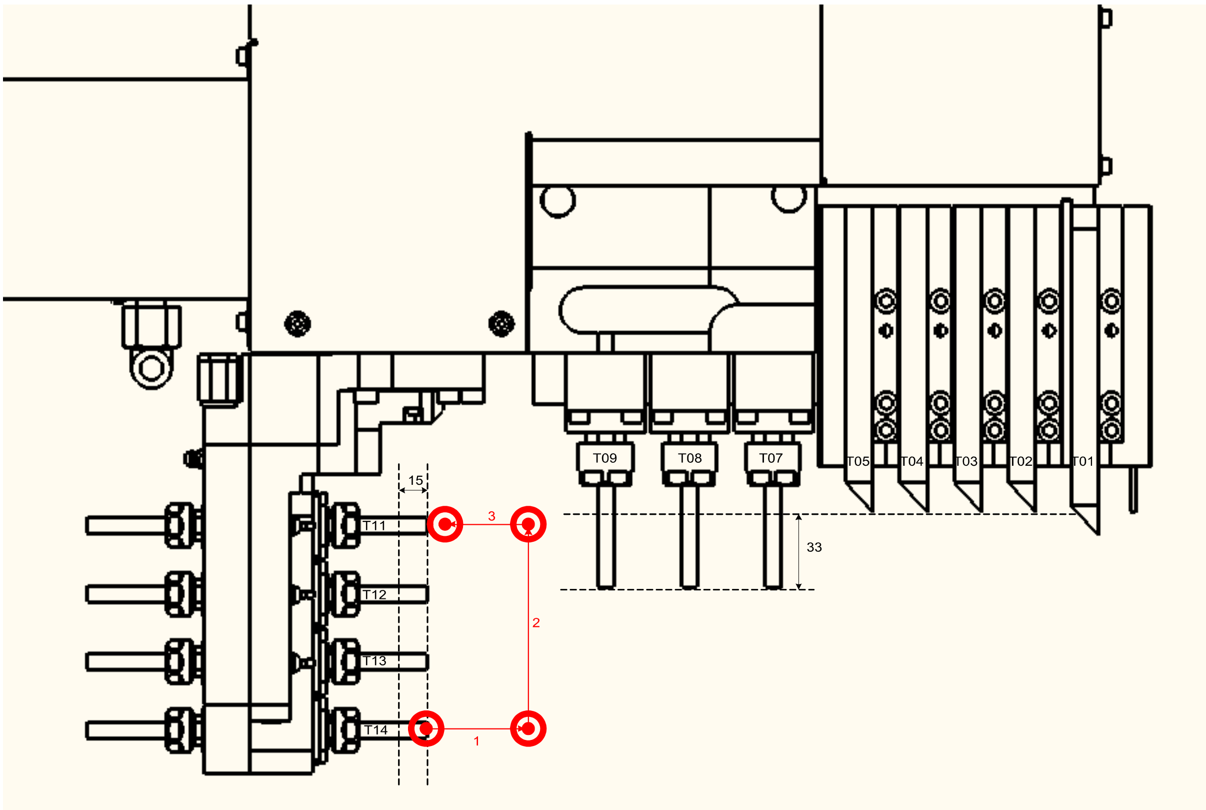


1. The X1 axis (motor X1 axis) moves to the machine coordinate 96.0 mm.
2. The Y1 axis (motor Y1 axis) moves to the tool change position (Y1 machine coordinate=530.0 mm).
3. The Y1 axis (motor X1 axis) moves to the center of command tool T11.
4. The X1 axis (motor Y1 axis) moves to the standby position of command tool T11.

4) Selection of the tool T1100 or T1200while a tool T1300 to T1400 is selected

Example:

T1400→T1100

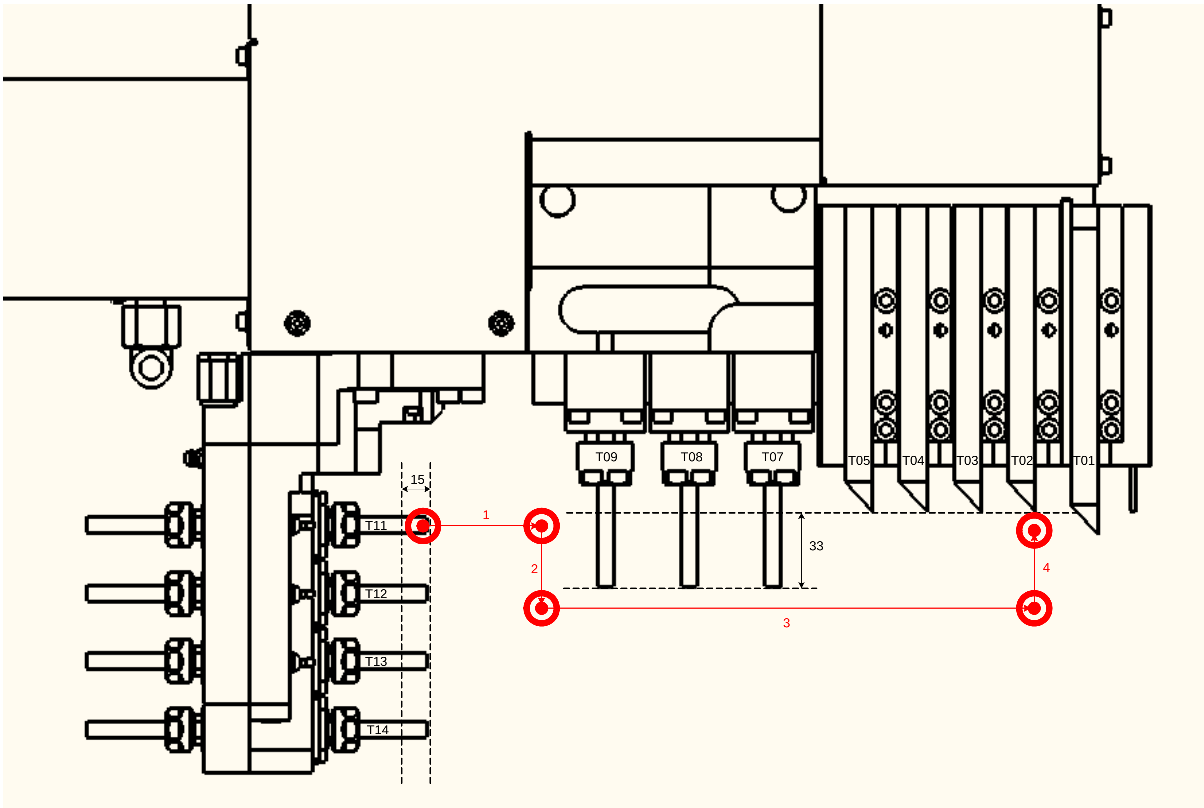


1. The X1 axis (motor Y1 axis) moves to the tool change position (Y1 machine coordinate=530.0 mm).
2. The Y1 axis (motor X1 axis) moves to the center of command tool T11.
3. The X1 axis (motor Y1 axis) moves to the standby position of command tool T11.

5) Selection of a tool T0100 to T1000 while the tool T1100 or T1200 is selected

Example:

T1100→T0200

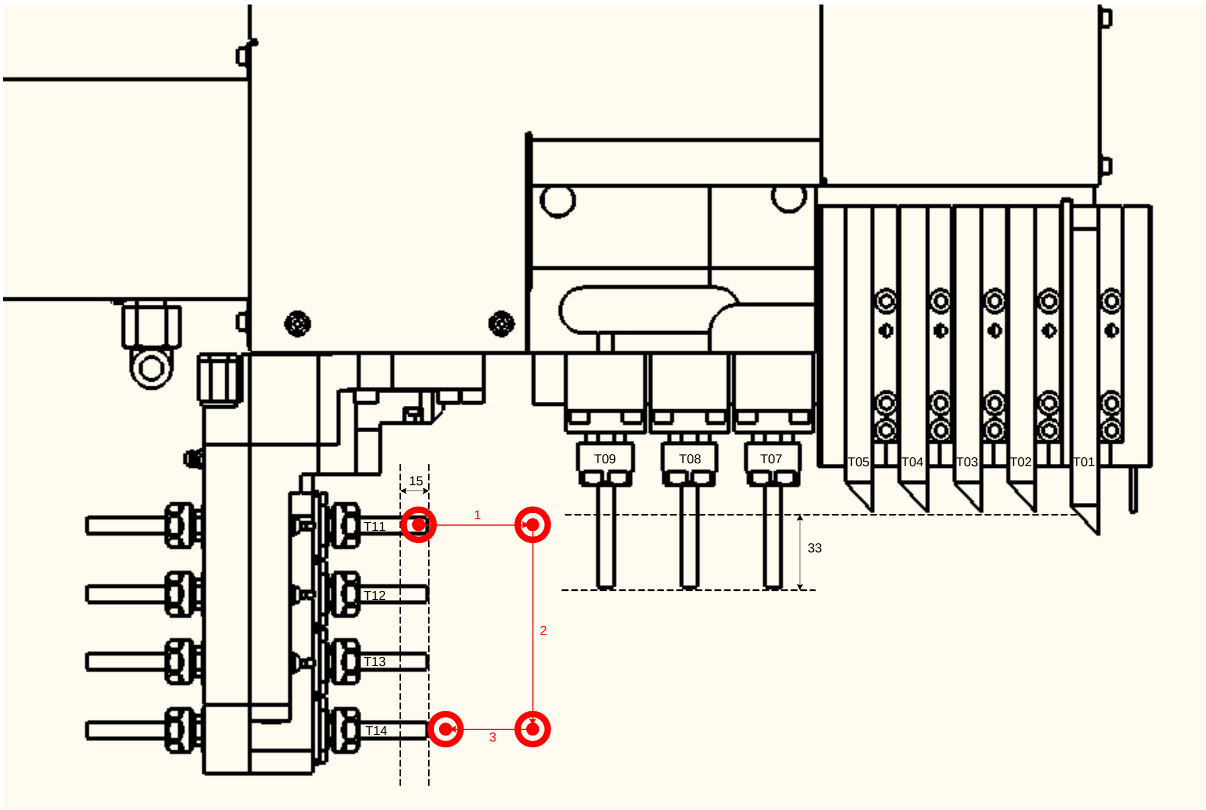


1. The Y1 axis (motor Y1 axis) moves to the tool change position (Y1 machine coordinate=530.0 mm).
2. The X1 axis (motor X1 axis) moves to the machine coordinate 96.0 mm.
3. The Y1 axis (motor Y1 axis) moves to the center of command tool T02.
4. The X1 axis (motor X1 axis) moves to the standby position of command tool T02.

6) Selection of a tool T1300 to T1400 while the tool T1100 or T1200 is selected

Example:

T1100→T1400

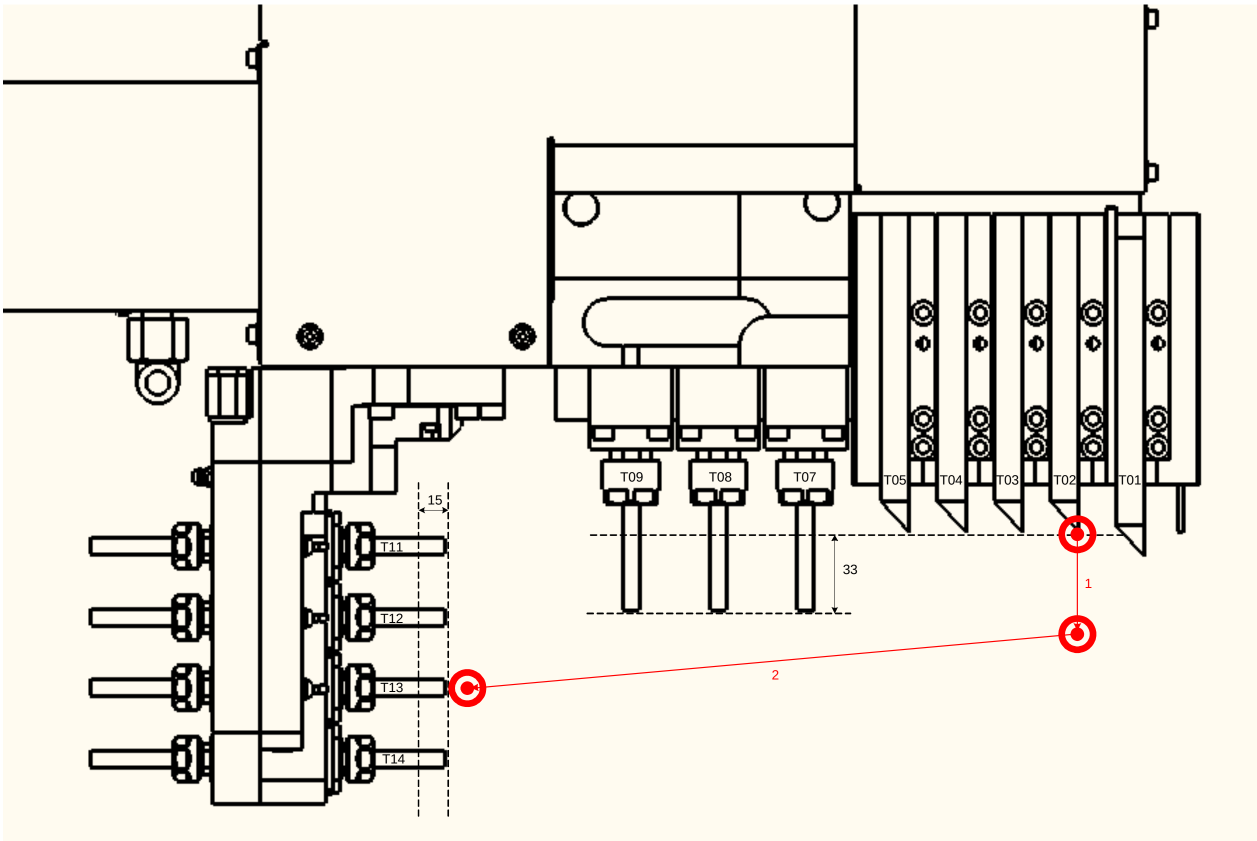


1. The X1 axis (motor Y1 axis) moves to the tool change position (Y1 machine coordinate=530.0 mm).
2. The Y1 axis (motor X1 axis) moves to the center of command tool T14.
3. The X1 axis (motor Y1 axis) moves to the standby position of command tool T14.

7) Selection of a tool T1300 or T1400 while a tool T0100 to T1000 is selected

Example:

T0200→T1300

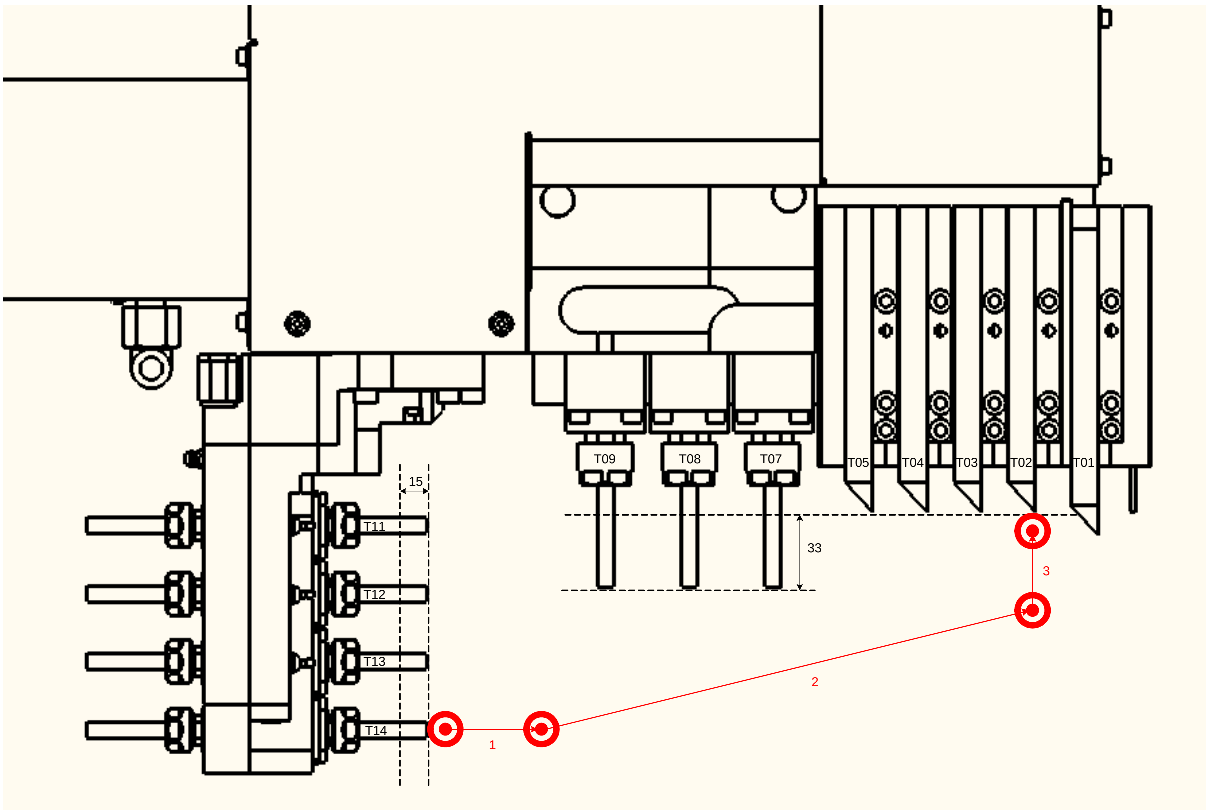


1. The X1 axis (motor X1 axis) retracts by the maximum tool setting diameter for T01 to T10.
2. The X1 axis and Y1 axis move directly to the standby position of the command tool T13.

8) Selection of a tool T0100 to T1000 while a tool T1300 or T1400 is selected

Example:

T1400→T0200

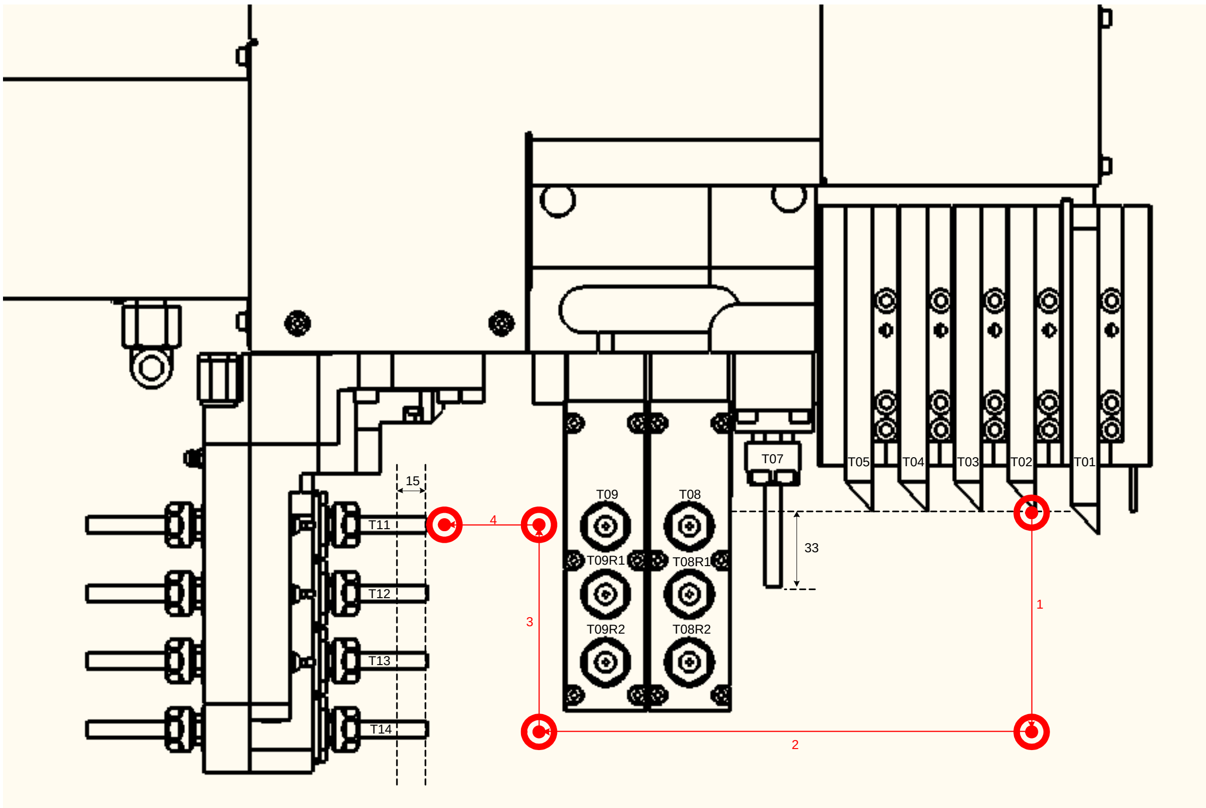


1. The X1 axis (motor Y1 axis) moves to the tool change position (Y1 machine coordinate=530.0 mm).
2. The X1 axis (motor X1 axis) moves directly to the maximum tool setting diameter position for T01 to T10, and the Y1 axis (motor Y1 axis) moves directly to the center of the command tool T02.
3. The X1 axis (motor X1 axis) move directly to the standby position of the command tool T11.

9) Selection of a tool T1100 to T1400 while a tool T0100 to T0700 or T1900 is selected (when GDF or BSE is mounted on T08 to T09)

Example:

T0100→T1100



1. When GDF or BSE is installed, if a tool is called within this operation, the X1 axis (motor X1 axis) moves to the tool change position (X1 machine coordinate = 207.0 mm).
2. The Y1 axis (motor Y1 axis) moves to the tool change position (Y1 machine coordinate=530.0 mm).
3. The Y1 axis (motor X1 axis) moves to the center of command tool T11.
4. The X1 axis (motor X1 axis) move directly to the standby position of the command tool T02.

[Note]

- If the Q1 argument is specified in the tool selection command in 3) to 9), the axis moves directly to the "positioning point" of the specified tool.
- If the value more than 66.0 mm for T0100 to T1000 or the value more than 30.0 mm for T1100 to T1300 is specified as the tool set diameter, an alarm is issued when tool T code is selected.

CAUTION

When you want to set the value more than 66.0 mm for T0100 to T1000 or the value more than 30.0 mm for T1100 to T1300 as the tool set diameter according to tool layout pattern, changing the default value of machine variable (66.0 in #26070, and 30.0 in #26071) allows the machine to operate.
However, there is a possibility of collision between the material and tool or between the tool and guide bushing.
Be sure to check operation carefully.

12.4.2 Back drilling tool on gang tool post (T5100 to T5900)

[Command format]

T□□ΔΔ X□ Y□ Z□ W□ Q1 Q3 E□ R□

[Argument]

X: Specify the workpiece coordinate of X1 axis after tool selection. Without the argument specified, the tool moves to the center of the workpiece (0.0 mm). For front/back simultaneous machining (G660), specify X argument with T11's.

Y: Specify the workpiece coordinate of Y1 axis after tool selection. Without the argument specified, the tool moves to the center of the workpiece (0.0 mm). For front/back simultaneous machining (G660), specify Y argument with T11's.

Z: Specify the position to which the back headstock (Z2 axis) is moved (absolute move distance from the machine zero point of Z2 axis). If the argument is omitted, the back headstock (Z2 axis) moves to the position separated from the maximum nose protrusion length by 5.0 mm.

W: Specify the retract point of the back headstock (Z2 axis) (incremental move distance of the back headstock (Z2 axis) from the current position). If the argument is omitted, the back headstock (Z2 axis) moves to the position separated from the maximum nose protrusion length by 5.0 mm.

Q1: Select a tool without moving the back headstock (Z2 axis). If the argument is omitted, the back headstock (Z2 axis) moves to the position separated from the maximum nose protrusion length by 5.0 mm.

Q3: Select a tool after moving the back headstock (Z2 axis) to the return position. If the argument is omitted, the back headstock (Z2 axis) moves to the position separated from the maximum nose protrusion length by 5.0 mm.

E: Specify this argument to index the back spindle at the specified indexing angle during the tool selection. If the argument is omitted, the spindle is not indexed. This argument is useful in the transition from turning to secondary machining process.

When the back spindle C axis option is not specified but the back spindle 1° indexing is specified:

Integer between 0.0 and 359.0 (with decimal point)

When the back spindle C axis option is specified:

Numeric value between 0.000 and 359.999 (with decimal point)

R: Specify R to call a tool associated with R when GDF1207, BSE607 or BSE707 is mounted.

CAUTION

If mounting the back drilling tool on gang tool post (T5100's) 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 the tool setting in the longitudinal direction. Failure to do so may cause interference.

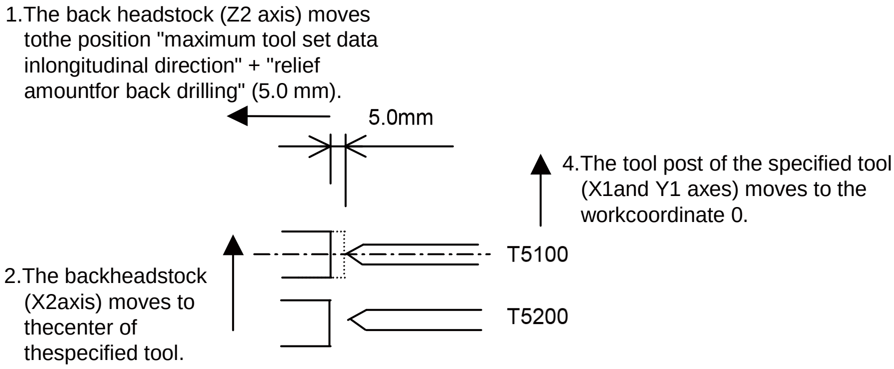
[Note]

- Arguments Z, W and Q are exclusive with each other. Accordingly, do not specify them together.
- If the large value is specified for "Longitud" in Tool Set screen, an alarm may occur at tool selection.

Operation example

T☐☐ΔΔ Argument not specified

1. The back headstock (Z2 axis) moves to the position "maximum tool set data in longitudinal direction" + "relief amount for back drilling" (5.0 mm).
2. The back headstock (X2 axis) moves to the center of the specified tool.
3. The coordinate system is set with the end face of the workpiece chucked by the back spindle as the workpiece zero position of the back headstock (Z2 axis).
4. The tool post of the specified tool (X1 and Y1 axes) moves to the workpiece coordinate 0.



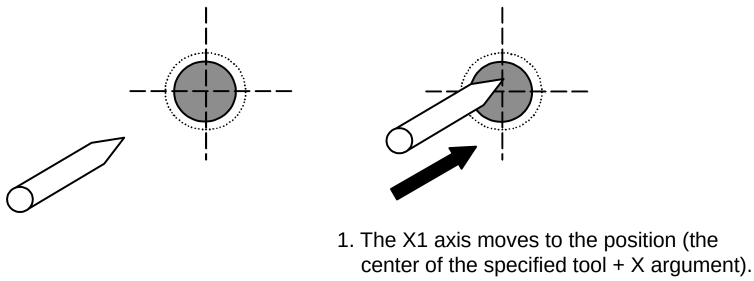
[Note]

- If the position of the back headstock (Z2 axis) has exceeded the position indicated in Step 1 (position "maximum tool set data in longitudinal direction" + "relief amount for back drilling" (5.0 mm).), back spindle (Z2 axis) does not move.
- Steps 2 and 4 are performed simultaneously.

T☐☐ΔΔ X☐ X argument

Specify the workpiece coordinate of X1 axis after tool selection. Without the argument specified, the tool moves to the center of the workpiece (0.0 mm). For front/back simultaneous machining (G660), specify X argument with T11's.

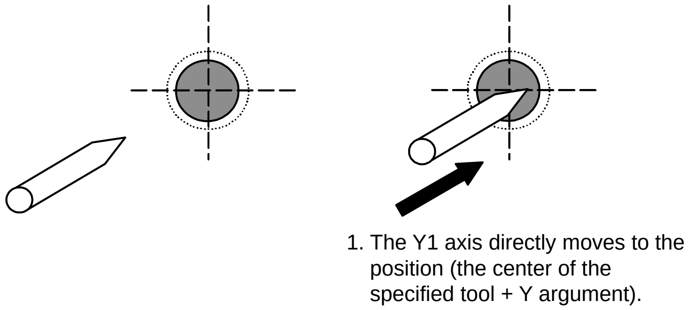
1. The X1 axis directly moves to the position (the center of the specified tool + X argument).



T□□ΔΔ Y□ Y argument

Specify the workpiece coordinate of Y1 axis after tool selection. Without the argument specified, the tool moves to the center of the workpiece (0.0 mm). For front/back simultaneous machining (G660), specify Y argument with T11's.

1. The Y1 axis directly moves to the position (the center of the specified tool + Y argument).



T□□ΔΔ Z□ Z argument

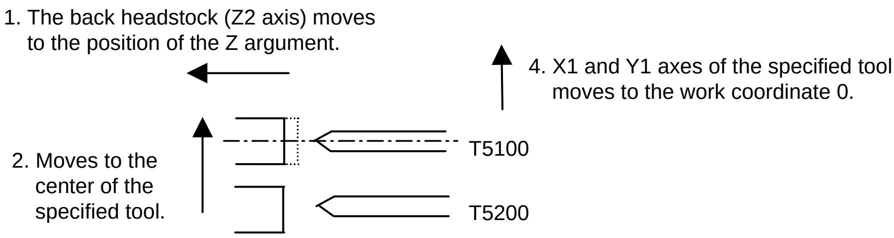
Specify the position to which the back headstock (Z2 axis) is moved (absolute move distance from the machine zero point of Z2 axis). If the argument is omitted, the back headstock (Z2 axis) moves to the position separated from the maximum nose protrusion length by 5.0 mm.

1. The back headstock (Z2 axis) moves to the position of the Z argument.

2. The coordinate system is set with the end face of the workpiece chucked by the back spindle as the workpiece zero position of the back headstock (Z2 axis).

3. The back headstock (X2 axis) moves to the center of the specified tool.

4. The tool post of the specified tool (X1 and Y1 axes) moves to the work coordinate 0.



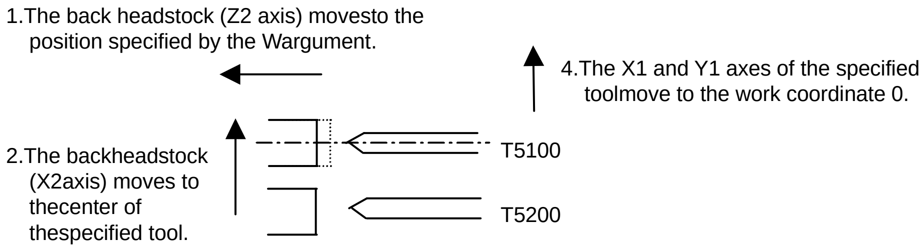
[Note]

When the Z argument is specified, note sufficiently that interference between the back headstock (Z2 axis) and the sleeve holders (X1 and Y1 axes) may not occur.

T☐☐△△ W☐ W argument

Specify the retract point of the back headstock (Z2 axis) (incremental move distance of the back headstock (Z2 axis) from the current position). If the argument is omitted, the back headstock (Z2 axis) moves to the position separated from the maximum nose protrusion length by 5.0 mm.

1. The back headstock (Z2 axis) moves to the position specified by the W argument.
2. The back headstock (X2 axis) moves to the center of the specified tool.
3. The coordinate system is set with the end face of the workpiece chucked by the back spindle as the workpiece zero position of the back headstock (Z2 axis).
4. The tool post of the specified tool (X1 and Y1 axes) moves to the work coordinate 0.



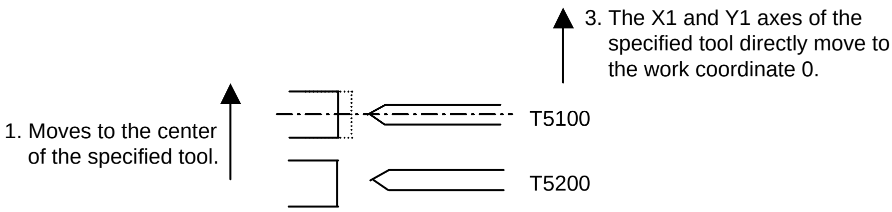
[Note]

- When the W argument is specified, note sufficiently that interference between the back headstock (Z2 axis) and the sleeve holders (X1 and Y1 axes) may not occur.
- If the W argument is specified with a positive value, an alarm occurs.

T☐☐△△ Q1 Q1 argument

Select a tool without moving the back headstock (Z2 axis). If the argument is omitted, the back headstock (Z2 axis) moves to the position separated from the maximum nose protrusion length by 5.0 mm.

1. The back headstock (X2 axis) moves to the center of the specified tool.
2. The coordinate system is set with the end face of the workpiece chucked by the back spindle as the workpiece zero position of the back headstock (Z2 axis).
3. The tool post of the specified tool (X1 and Y1 axes) directly moves to the work coordinate 0.



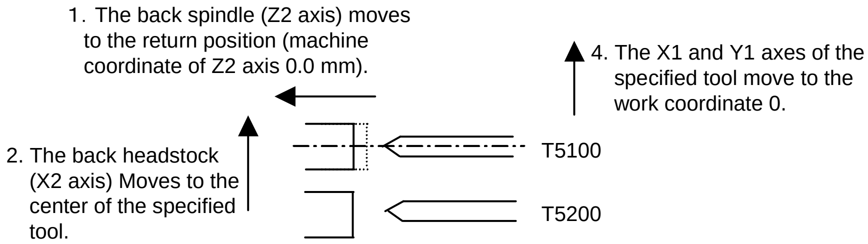
[Note]

When the Q1 argument is specified, note sufficiently that interference between the back headstock (Z2 axis) and the sleeve holders (X1 and Y1 axes) may not occur.

T□□ΔΔ Q3   Q3 argument

Select a tool after moving the back headstock (Z2 axis) to the return position. If the argument is omitted, the back headstock (Z2 axis) moves to the position separated from the maximum nose protrusion length by 5.0 mm.

1. The back spindle (Z2 axis) moves to the return position.
2. The back headstock (X2 axis) moves to the center of the specified tool.
3. The coordinate system is set with the end face of the workpiece chucked by the back spindle as the workpiece zero position of the back headstock (Z2 axis).
4. The tool post of the specified tool (X1 and Y1 axes) directly moves to the work coordinate 0.



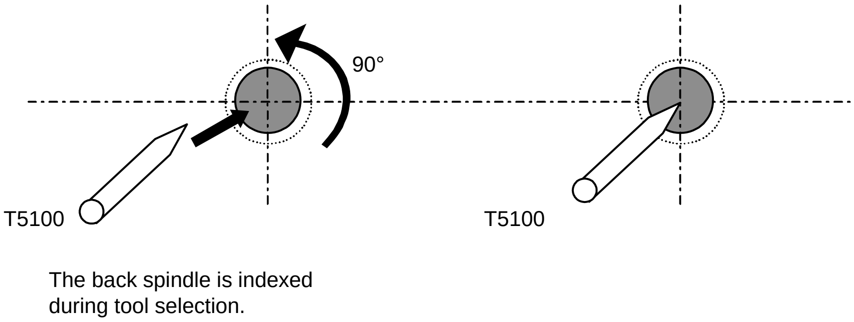
[Note]

The return position of the back headstock (Z2 axis) is Z2 axis machine coordinate 0.0 mm.

T□□ΔΔ E□   E argument

Specify this argument to index the back spindle during tool selection.
Specify the indexing angle with decimal point for E□.

T5100 E90.0



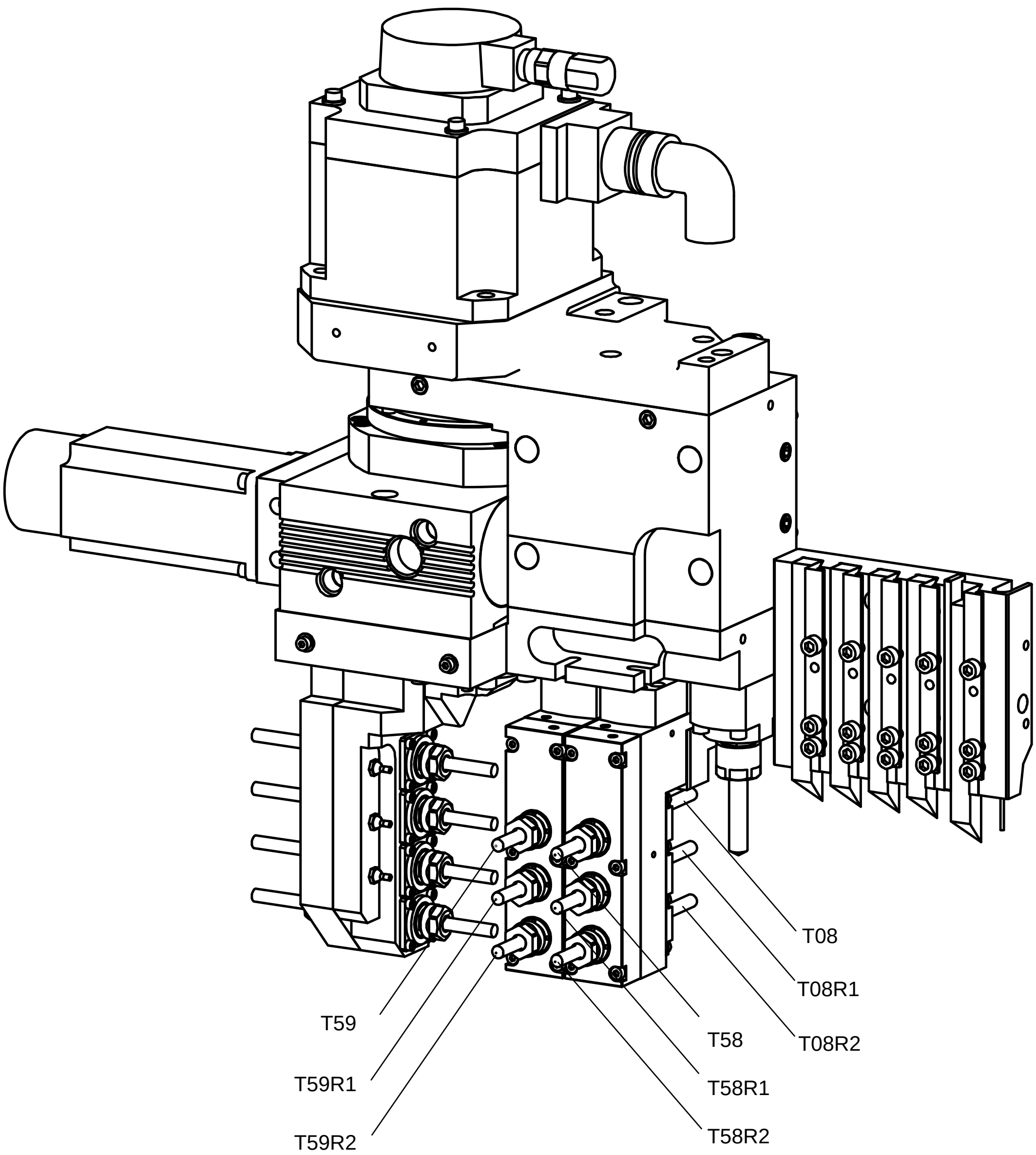
[Note]

- If the back spindle indexing option (the back spindle C axis or back spindle 1-degree indexing) is not specified, specifying the indexing angle (E argument) causes an alarm to occur.
- Specify the indexing angle (E argument) with decimal point. If omitted, an alarm occurs or an indexing at specified angle will fail.
- If the value specified for indexing angle (E argument) is out of the allowable range, an alarm occurs.

  Allowable range
  - Back spindle 1-degree indexing: Integer between 0.0 and 359.0 (with decimal point)
  - Back spindle C axis: Numeric value between 0.000 and 359.999 (with decimal point)
- If the value specified for indexing angle (E argument) is out of the valid indexing angle, an alarm occurs.
- The back spindle indexing can be performed while the back spindle is rotating.

T□□ΔΔ R1　R1 argument

Specify R to call a tool associated with R when GDF1207, BSE607 or BSE707 is mounted.



12.4.3 Front drilling tool on opposite tool post (T2100 to T2600)

[Command format]

T□□△△ Q1 W□ X□ Y□ Z□ E□ S□ M1

[Argument]

Q1: Select the tool without moving the back spindle Z2 axis of the currently used tool to the return position.

W: Specify the retract point of the back headstock (Z2 axis) (incremental move distance of the back headstock (Z2 axis) from the current position). Specifying this argument with positive value causes an alarm to occur.

If the argument is omitted:

$1: Only the opposite tool post X2 axis moves.

$2: The opposite tool post Z2 axis moves to the return position, then the X2 axis starts to move. Specify W0 not to move the Z2 axis to the return position.

X: Specify the X2 axis workpiece coordinate after tool selection. It the argument is not specified, the drilling tool moves to the center (0.0 mm) of workpiece, or the cross machining tool moves to the return position.

Y: Specify the Y2 axis workpiece coordinate after tool selection. It the argument is not specified, the tool moves to the center (0.0 mm) of workpiece.

Z: Specify the Z2 axis workpiece coordinate after tool selection. It the argument is not specified, the Z2 axis does not move.

E: Specify this argument to index the main spindle during tool selection. The specification is valid when the turning process changes to the secondary machining process. The argument is valid only for $1.

Without main spindle C-axis option:	Integer between 0.0 and 359.0
With main spindle C-axis option:	0.000 to 359.999

S: Specify the shift amount from the forward end position of the Z2 axis (Z2 axis machine coordinate 205.0) during the first selection of T20' tool after the designation of machining pattern G610. If the T20' tool is mounted in excess of the standard tool nose position, be sure to specify the shift amount with negative value. This argument is valid only for the first T20' tool after the designation of G610. If the argument is omitted, the Z2 axis moves to the forward end position.

M1: Ignores an alarm message "EX737 S2 cover may interfere with back cross-machining holder".

[Note]

- If this command is specified during $1 single machining (G610) front drilling tools (T2100 to T2600 for L20X) selection, only the X2 and Y2 axes of the opposite tool post move. So, escape the workpiece to prevent interference.
- If this T command is specified during the inner/outer diameter simultaneous machining (G620), the X2 axis starts movement after the Z2 axis moves to the return position. Since Z1 and Z2 axes are superimposed, the Z2 axis may overtravel. In this case, use the W argument.
- If this command is specified during $1 single machining (G610), after the X2 axis of the opposite tool post is moved, then the Z2 axis moves to the forward end position (Z2 axis machine coordinate 205.0). Since the Z2 axis moves to the forward end position, if the workpiece is protruded from the guide bushing , interference between the workpiece and the tool may occur. After the workpiece is escaped , the tool must be called.

  The forward end position depends on the tools being mounted.
  Only the end-face drilling tool is mounted: 205.0 mm
  Cross machining tool is mounted: 184.0
  Vertical tool (without cross machining tool) is mounted: 203.0

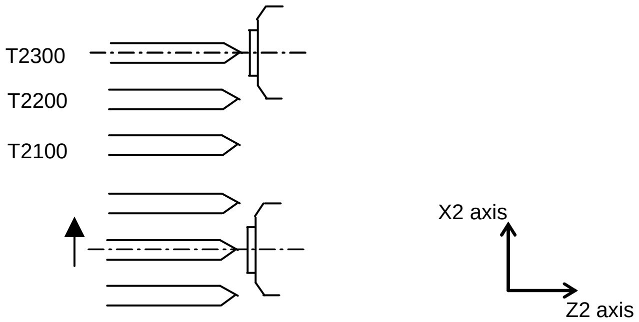
Operation example

1) Selecting drilling tool → drilling tool (turning and cross machining tools are included):

When specified in $1:

(1) In the machining program, move the Z1 axis to the position where the drilling tool will not interfere, then specify the T code command.

(2) The X2 and Y2 axes move at the same time.



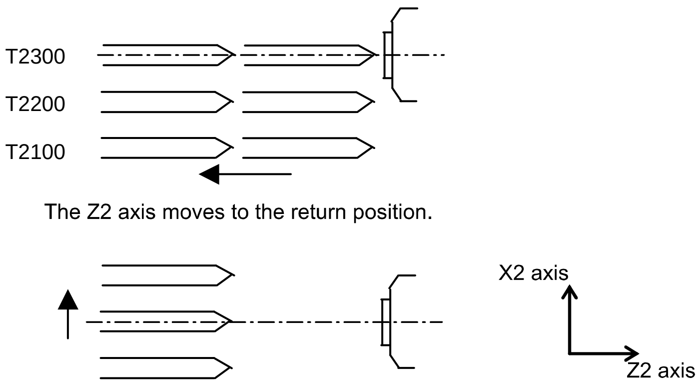
The X2 axis directly moves to the tool set position.

The X2 and Y2 axes move at the same time when the tool is selected from the lower stage to upper stage (or from the upper stage to lower stage).

When specified in $2:

(1) The Z2 axis moves to the return position.

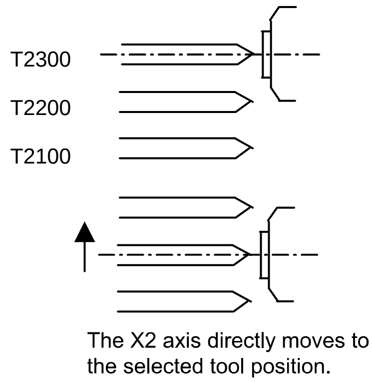
(2) The X2 and Y2 axes move at the same time.



If the vertical tool or vertical tool 1019 is selected, the axis is changed so that the tool nose faces the X axis by using the T code.

2) When specified in $2 with Q1 argument

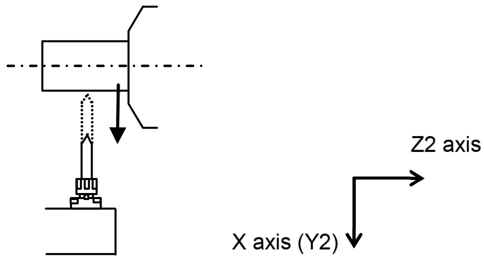
While the Z2 axis is superimposing on Z1 axis in G620 mode, the Z2 axis moves to the return position when the tool is selected. It may cause an over-travel of Z2 axis. Use the Q1 argument to avoid over-travelling.



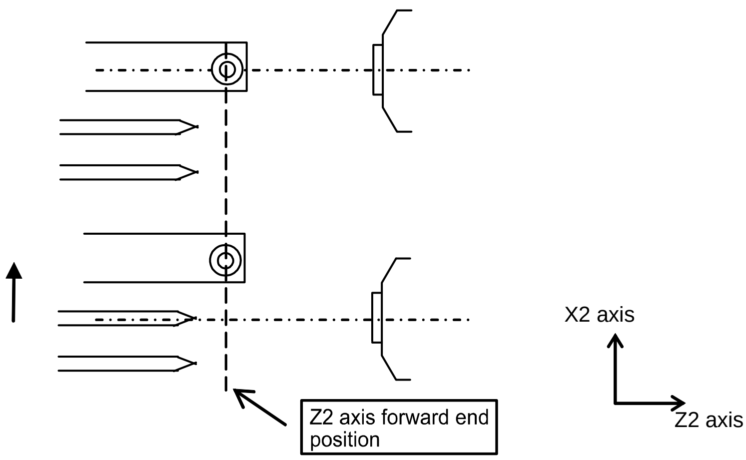
2) Selecting cross-machining tool → other tool (drilling, cross machining, turning tool)

When specified in $1:

(1) The X (or Y2) axis moves to the positioning point.



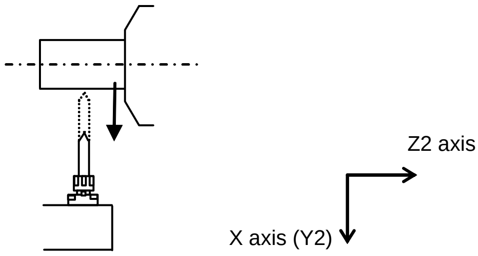
(2) If the cross-machining tool is selected, Y (X2) axis moves to the specified tool position. Then, the X (Y2) axis moves to the specified tool position. If any other tool is selected, the X and Y axes move to the specified tool position at the same time.



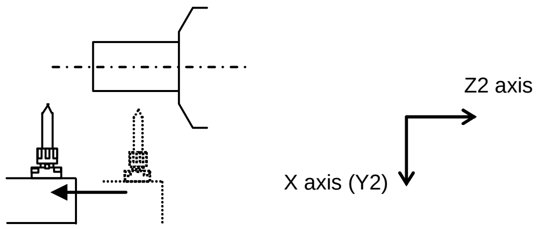
The Z2 axis moves forward only when the first selection of T20's tool in $1. If the cross machining tool is mounted, the Z2 axis moves forward to the tool nose position of the front tool post. In this case, the forward end position of Z2 axis is 184.0 mm in the machine coordinate. If no cross machining tool is mounted and the vertical tool is mounted, the forward end position is 203.0 mm. If only the end-face drilling tool or horizontal tool is mounted, the forward end position is 205.0 mm.

When specified in $2:

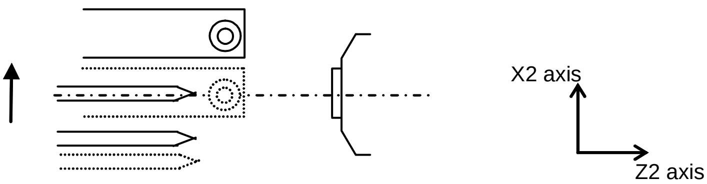
(1) The X (or Y2) axis moves to the positioning point (cross machining tool only).



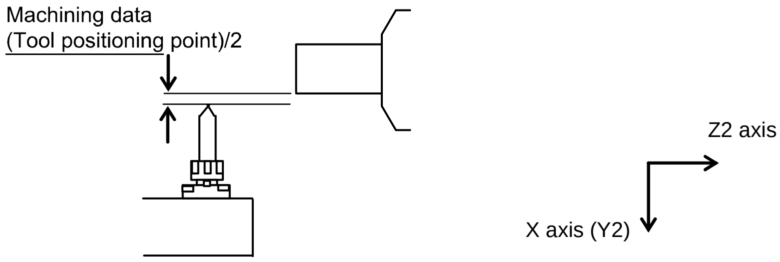
(2) The Z2 axis moves to the return position.



(3) The X2 and Y2 axes directly move to the selected tool position.



When the cross machining tool is specified, the X (or Y2) axis moves to the positioning point.



12.4.4 Back spindle (T3000)

[Command format]

T□□ΔΔ W▭ Q1 E▭

[Argument]

W: Specify the retract point of the back headstock (Z2 axis) (incremental move distance of the back headstock (Z2 axis) from the current position). The back headstock (Z2 axis) moves to the return position (Z2 axis machine coordinate: 0.0 mm) when this argument is omitted.

Q1: Select the back spindle without moving the back headstock (Z2 axis). The back headstock (Z2 axis) moves to the return position (Z2 axis machine coordinate: 0.0 mm) when this argument is omitted.

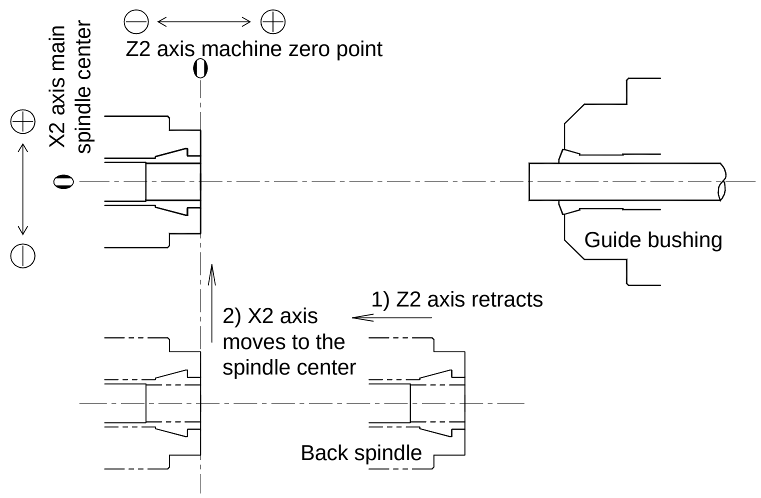
E: Specify this argument to index the back spindle at the specified indexing angle during the tool selection. If the argument is omitted, the back spindle is not indexed.

When the back spindle C axis option is not specified but the back spindle 1° indexing is specified:
Integer between 0.0 and 359.0 (with decimal point)

When the back spindle C axis option is specified:
Numeric value between 0.000 and 359.999 (with decimal point)

Operation example

1. The back headstock (Z2 axis) moves to the machine zero point (Z2 axis machine coordinate: 0.0 mm).
2. The back headstock (X2 axis) moves to the center of the main spindle.
3. The coordinate system is set with the back headstock (Z2 axis) machine coordinate.



[Note]

- Arguments Z, W and Q are exclusive with each other. Accordingly, do not specify them together.
- If the W argument is specified with a positive value, an alarm occurs.

12.4.5 Back drilling tool on back tool post (T3100 to T3800)

[Command format]

T□□ΔΔ Z□ W□ X□ Y□ Q1 Q3 E□

[Argument]

Z: Specify the retract point of the back headstock (Z2 axis) is moved (absolute move distance from the machine zero point of Z2 axis). If the argument is omitted, the back headstock (Z2 axis) moves to the position separated from the maximum nose protrusion length by 13.0 mm.

W: Specify the retract point of the back headstock (Z2 axis) (incremental move distance of the back headstock (Z2 axis) from the current position). If the argument is omitted, the back headstock (Z2 axis) moves to the position separated from the maximum nose protrusion length by 13.0 mm.

X: Specify the X2 axis workpiece position during tool selection. If the argument is omitted, the X2 axis moves to the center of the tool (X2 axis workpiece coordinate: 0) if the drilling tool is mounted. If the cross machining tool is mounted, the X2 axis moves to the positioning point.

Y: Specify the Y2 axis workpiece position during tool selection. If the argument is omitted, the Y2 axis moves to the center of the tool (X2 axis workpiece coordinate: 0).

Q1: Select a tool without moving the back headstock (Z2 axis). If the argument is omitted, the back headstock (Z2 axis) moves to the position separated from the maximum nose protrusion length by 13.0 mm.

Q3: Select a tool after moving the back headstock (Z2 axis) to the return position. If the argument is omitted, the back headstock (Z2 axis) moves to the position separated from the maximum nose protrusion length by 13.0 mm.

E: Specify this argument to index the back spindle at the specified indexing angle during the tool selection. If the argument is omitted, the spindle is not indexed. This argument is useful in the transition from turning to secondary machining process.

When the back spindle C axis option is not specified but the back spindle 1° indexing is specified:
Integer between 0.0 and 359.0 (with decimal point)

When the back spindle C axis option is specified:
Numeric value between 0.000 and 359.999 (with decimal point)

CAUTION

If mounting the back tool post back drilling tool (T3100 to T3800) 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 the tool setting in the longitudinal direction. Failure to do so may cause interference.

[Note]

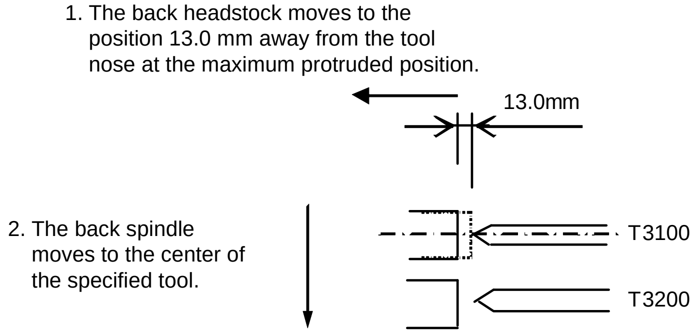
- Arguments Z, W and Q are exclusive with each other. Accordingly, do not specify them together.
- If the large value is specified for "Longitud" in Tool Set screen, an alarm may occur at tool selection.

Operation example

T☐☐△△ Argument not specified

1) When a drilling tool is selected (includes turning tool)

1. The back headstock (Z2 axis) moves to the position "maximum tool set data in longitudinal direction" + "relief amount for back drilling" (13.0 mm).
2. The back headstock (X2 axis) moves to the center of the specified tool.
The Y2 axis moves to the center of the specified tool along with the X2 axis.
3. The coordinate system is set with the end face of the workpiece chucked by the back spindle as the workpiece zero position of the back headstock (Z2 axis).

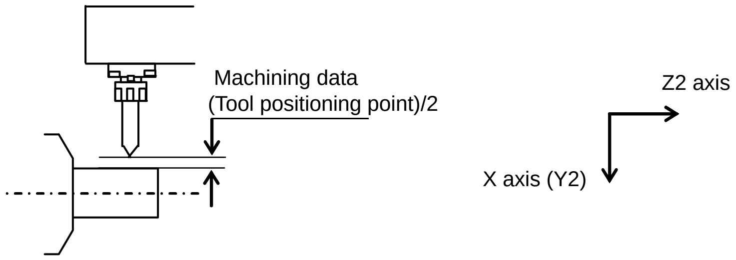


[Note]

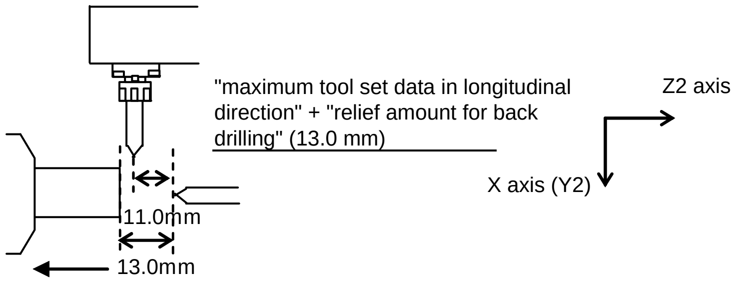
If the position of the back spindle (Z2 axis) has exceeded the position indicated in Step 1 (position "maximum tool set data in longitudinal direction" + "relief amount for back drilling" (13.0 mm).), back spindle (Z2 axis) does not move.

2) When a cross machining tool (GSC1507) is selected :

(1) The cross headstock (X axis) moves to the positioning point.



(2) The back headstock (Z2 axis) moves to the position "maximum tool set data in longitudinal direction" + "relief amount for back drilling" (13.0 mm). The "maximum tool set data in longitudinal direction" is equal to the "maximum back drilling tool length in longitudianl direction". With the cross machining tool, the tool nose is shifted 11.0 mm away from the back drilling tool.



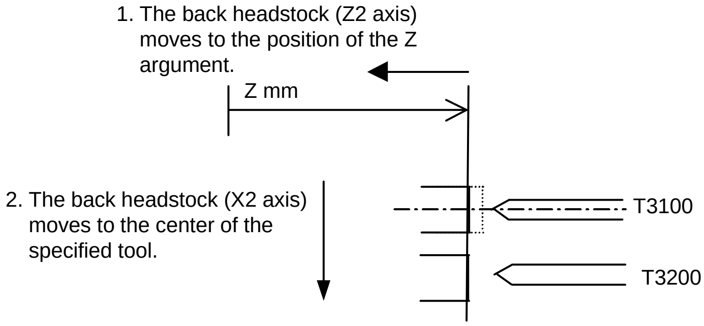
(3) The back headstock (X2, Y2 axes) move to the specified tool position at the same time.

(4) The coordinate system is set with the end face of the workpiece chucked by the back spindle as the workpiece zero position of the back headstock (Z2 axis).

T□□ΔΔ Z□ Z argument

Specify the position to which the back headstock (Z2 axis) is moved (absolute move distance from the machine zero point of Z2 axis). If the argument is omitted, the back headstock (Z2 axis) moves to the position separated from the maximum nose protrusion length by 13.0 mm.

1. The back headstock (Z2 axis) moves to the position of the Z argument.
2. The back headstock (X2 axis) moves to the center of the specified tool.
3. The coordinate system is set with the end face of the workpiece chucked by the back spindle as the workpiece zero position of the back headstock (Z2 axis).



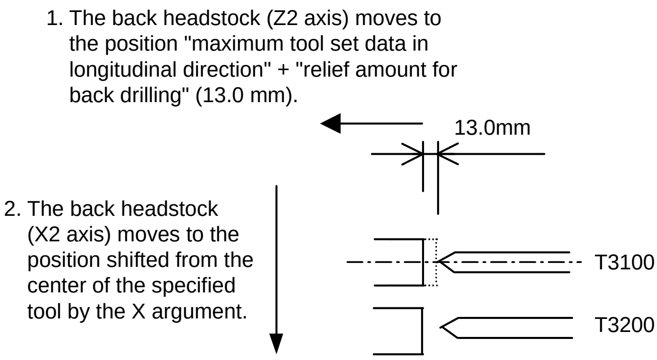
[Note]

When the Z argument is specified, note sufficiently that interference between the back headstock (Z2 and X2 axes) and the back tool post may not occur.

T□□ΔΔ X□ X argument

Specify the X2 axis workpiece position during tool selection. If the argument is omitted, the X axis moves to the center of the tool (X2 axis workpiece coordinate: 0).

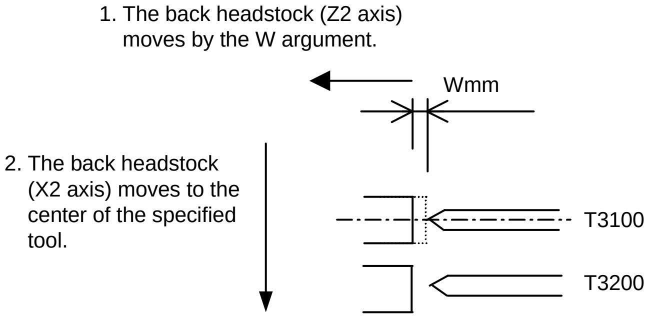
1. The back headstock (Z2 axis) moves to the position "maximum tool set data in longitudinal direction" + "relief amount for back drilling" (13.0 mm).
2. The back headstock (X2 axis) moves to the position shifted from the center of the specified tool by the X argument.
3. The coordinate system is set with the end face of the workpiece chucked by the back spindle as the workpiece zero position of the back headstock (Z2 axis).



T□□ΔΔ W[ ] W argument

Specify the retract point of the back headstock (Z2 axis) (incremental move distance of the back headstock (Z2 axis) from the current position). If the argument is omitted, the back headstock (Z2 axis) moves to the position separated from the maximum nose protrusion length by 13.0 mm.

1. The back headstock (Z2 axis) moves by the W argument.
2. The back headstock (X2 axis) moves to the center of the specified tool.
3. The coordinate system is set with the end face of the workpiece chucked by the back spindle as the workpiece zero position of the back headstock (Z2 axis).



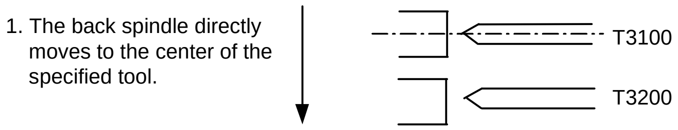
[Note]

- When the W argument is specified, note sufficiently that interference between the back headstock (Z2 and X2 axes) and the back tool post may not occur.
- If the W argument is specified with a positive value, an alarm occurs.

T□□ΔΔ Q1 Q1 argument

Select a tool without moving the back headstock (Z2 axis). If the argument is omitted, the back headstock (Z2 axis) moves to the position separated from the maximum nose protrusion length by 13.0 mm.

1. The back headstock (X2 axis) moves to the center of the specified tool.
2. The coordinate system is set with the end face of the workpiece chucked by the back spindle as the workpiece zero position of the back headstock (Z2 axis).



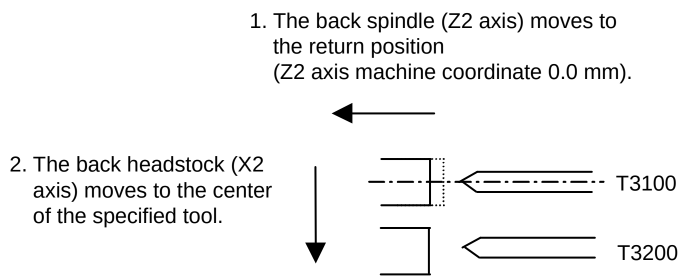
[Note]

When the Q1 argument is specified, note sufficiently that interference between the back headstock (Z2 and X2 axes) and the back tool post may not occur.

T☐☐ΔΔ Q3 Q3 argument

Select a tool after moving the back headstock (Z2 axis) to the return position. If the argument is omitted, the back headstock (Z2 axis) moves to the position separated from the maximum nose protrusion length by 13.0 mm.

1. The back spindle (Z2 axis) moves to the return position.
2. The back headstock (X2 axis) moves to the center of the specified tool.
3. The coordinate system is set with the end face of the workpiece chucked by the back spindle as the workpiece zero position of the back headstock (Z2 axis).



[Note]

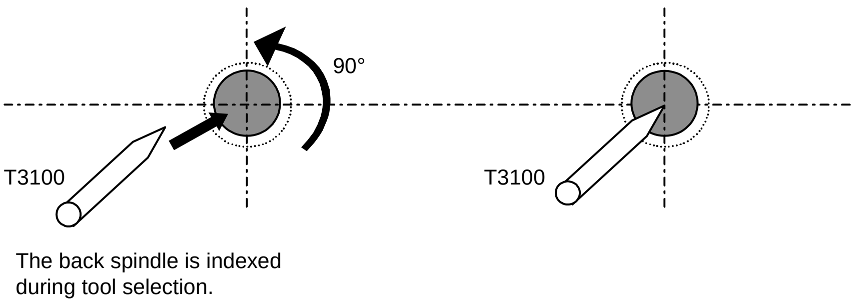
The return position of the back headstock (Z2 axis) is Z2 axis machine coordinate 0.0 mm.

T☐☐ΔΔ E☐ E argument

Specify this argument to index the back spindle during tool selection.

Specify the indexing angle with decimal point for E☐.

T3100 E90.0

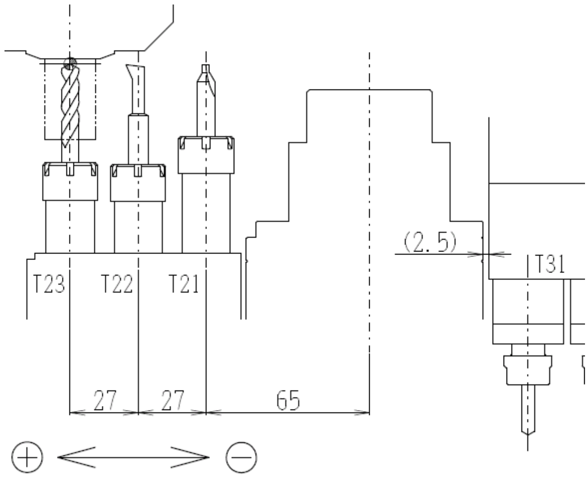


[Note]

- If the back spindle indexing option (the back spindle C axis or back spindle 1-degree indexing) is not specified, specifying the indexing angle (E argument) causes an alarm to occur.
- Specify the indexing angle (E argument) with decimal point. If omitted, an alarm occurs or an indexing at specified angle will fail.
- If the value specified for indexing angle (E argument) is out of the allowable range, an alarm occurs.

  Allowable range

  - Back spindle 1-degree indexing: Integer between 0.0 and 359.0 (with decimal point)
  - Back spindle C axis: Numeric value between 0.000 and 359.999 (with decimal point)
- If the value specified for indexing angle (E argument) is out of the valid indexing angle, an alarm occurs.
- The back spindle indexing can be performed while the back spindle is rotating.
- The signs of coordinates when using the boring tool are as shown in the figure below.



Opposite tool post viewed from the top

- Move the tool away from the end face of the material after completing the boring.
  It is dangerous to specify a tool selecting command without moving the tool away from the end face.
- The boring and inner diameter thread cutting by T23 is limited in stroke depending on the back tool post. Use T21 and T22 for boring.
- With the machine of L20X, set the boring sleeve in the opposite tool post or back tool post so that the tool nose faces horizontally (except for pinch turning).

12.4.6 T codes and arguments effective in machining patterns

The table below lists T codes and arguments effective in machining patterns. For machining patterns, see <11.7 Machining Pattern>.

Machining patterns	Axis control group	Tool type	Tool number	Arguments												
				None	X	Y	Z	W	H	Q1	Q3	K2	A	E	S	R
Free pattern (G600)	$1	Outer dia. machining tool	T0100 to T1400,T2000	○	○	○	○	○	○	○	×	○	×	○	×	×
		Front drilling tool	T0800 to T1400	○	○	○	○	×	×	×	×	×	×	○	×	○
		Back drilling tool	T5700 to T5900	○	○	○	○	○	×	○	○	×	×	○	×	○
	$2	Back spindle	T3000	○	×	×	×	○	×	○	×	×	×	×	×	×
		Back drilling tool	T3100 to T3800	○	○	○	○	○	×	○	○	×	×	○	×	×
			T5100 to T5900	○	○	○	○	○	×	○	○	×	×	○	×	○
$1 single machining (G610)	$1	Outer dia. machining tool	T0100 to T1400	○	○	○	○	×	○	○	×	○	×	○	×	×
		Front drilling tool	T2100 to T2600	○	○	○	○	○	×	○	×	×	×	×	○	×
Inner/outer diameter simultaneous machining (G620)	$1	Outer dia. machining tool	T0100 to T0500	○	○	○	○	×	○	○	×	×	×	○	×	×
	$2	Front drilling tool	T2100 to T2600	○	○	○	○	○	×	○	×	×	×	○	○	×
Front/back parallel machining (G630)	$1	Outer dia. machining tool	T0100 to T1400	○	○	○	○	○	○	○	×	○	×	○	×	×
		Front drilling tool	T0800 to T1400	○	○	○	○	×	×	×	×	×	×	○	×	○
	$2	Back spindle	T3000	○	×	×	×	○	×	○	×	×	×	×	×	×
		Back drilling tool	T3100 to T3800	○	○	○	○	○	×	○	○	×	×	○	×	×
Pick-off, center support (G650)	$1	Outer dia. machining tool	T0100 to T0500	○	○	○	○	×	○	○	×	×	×	○	×	×
	$2	Back spindle	T3000	○	×	×	×	○	×	○	×	×	×	×	×	×
Front/back simultaneous machining (G660)	$1	Front drilling tool	T0800 to T1400	○	○	○	○	×	×	×	×	×	×	○	×	○
	$2	Back drilling tool	T5700 to T5900	○	×	×	×	○	×	○	○	×	×	○	×	○

○: Specification permitted, ×: Specification not permitted

[Note]

- Any commands other than those listed in the table above cannot be specified.
- If required arguments are not specified or one or more unavailable tools or arguments are specified, an alarm occurs.

12.5 G Functions (G Codes)

The G functions, specified by 2-digit numerals after the G, are preparatory functions. They also send the axis (X1, Y1, Z1, X2, Y2, Z2) control command to the NC.

[Command format]

G□□

Example: G00 Rapid feed

G01 Cutting feed

12.5.1 G codes table

Once specified, each G code belonging to groups 01, 02, 05, 07….. in the following table remains valid until another G code in the same group is specified. That is, the G code is a modal code.

G codes table

G code	Function	Group	Remarks
G00	Rapid feed positioning	01	12.5.2
◎ G01	Linear interpolation	01	12.5.3
G02	Circular interpolation (clockwise)	01	12.5.4
G03	Circular interpolation (counterclockwise)	01	12.5.4
G04	Dwell		12.5.5
G09	Exact stop check ON		12.5.6
G12.1	Milling interpolation ON	19	14.8
G13.1	Milling interpolation OFF	19	14.8
G16	Y-Z cylindrical plane selection	02	
G17	X-Y plane selection	02	14.7.1
G18	Z-X plane selection	02	14.7.1
G19	Y-Z plane selection	02	14.7.1
G32	Thread cutting	01	13.11
G34	Variable lead thread cutting (option)	01	
G35	Circular thread cutting (clockwise)	01	13.15
G36	Circular thread cutting (counterclockwise)	01	13.15
◎ G40	Tool nose radius compensation cancel	07	13.8
G41	Tool nose radius compensation left ON	07	13.8
G42	Tool nose radius compensation right ON	07	13.8
◎ G43	Back spindle feed per revolution mode OFF	08	15.14
G44	Back spindle feed per revolution mode ON	08	15.14
G50	Coordinate system setting/Main spindle speed clamp setting		
G53	Machine coordinate system command		13.21
G65	Macro call (option)		13.38
G66	Macro modal call A (option)	14	13.38
G66.1	Macro modal call B (option)	14	13.38

G code	Function	Group	Remarks
G70	Finish machining cycle	09	
G71	Rough machining cycle in longitudinal direction	09	
G72	End face rough machining cycle	09	
G73	Closed loop machining cycle	09	
G74	End face cut-off machining cycle	09	
G75	Outer/inner diameter cut-off cycle	09	13.12
G76	Multiple repetitive threading cycle	09	
G79	Face drilling cycle	09	13.13
◎ G80	Synchronized tapping mode OFF	09	13.14
G83	Canned cycle drilling (Z axis)	09	
G84	Synchronized end face tapping mode ON	09	13.14
G85	Front/back boring cycle	09	
G88	Synchronized cross tapping mode ON	09	13.14
G90	Canned cycle longitudinal machining	09	13.17
G92	Thread cycle canned cycle	09	13.18
G94	End face machining cycle (Z axis)	09	
G96	Constant surface speed control ON	17	13.7
◎ G97	Constant surface speed control OFF	17	13.7
G98	Millimeter per minute feed (mm/min)	05	12.6
◎ G99	Millimeter per revolution feed (mm/rev)	05	12.6
G113	Spindle synchronization control cancel		15.8
G114.1	Spindle synchronization control		15.8
G114.2	Polygon machining		14.9
G114.3	Hobbing		14.10
G115	Start position queuing (Type 1)		13.29
G116	Start position queuing (Type 2)		13.30
G117	Auxiliary function output during axis feed		13.31
G140	Arbitrary axis exchange		13.32
G149	End position specification queuing		13.33
G156	Arbitrary axis superimpose		13.34
G164	Differential rotary tool function		13.16
G173	Inclined helical interpolation function		14.11.2
◎ G190	Absolute value commands for B-axis ON		
G191	Relative value commands for B-axis ON		
G231	Pick-off cancel ($1 only)		15.17
G600	Free pattern (machining pattern cancel)		11.7.3
G610	$1 single machining		11.7.4
G620	Inner/outer diameter simultaneous machining - Main spindle C-axis (C1)		11.7.5
G621	Inner/outer diameter simultaneous machining - Main spindle C-axis (C1) $1 side valid		11.7.5
G622	Inner/outer diameter simultaneous machining - Main spindle C-axis (C1) $2 side valid		11.7.5

G code	Function	Group	Remarks
G630	Front/back parallel machining		11.7.6
G650	Pick-off, center-support		11.7.7
G660	Front/back simultaneous machining		11.6.8
G813	Spindle synchronization control cancel		15.8
G814	Spindle synchronization control		15.8
G900	B-axis mode ON		
G901	B-axis mode OFF		
G910	B-axis revolution		
G950	Coordinate rotation ON		
G951	Coordinate rotation OFF		
G999	Last program		15.11

[Note]

- Multiple G codes can be specified in the same block if they belong to different groups.
- G codes cannot be specified with T☐☐☐☐in the same block.
- A three-digit G codes cannot be executed in MDI (manual data input) mode.
- Single block operation is not possible while G32, G34, G76, G79, G84, G88, G90 or G92 is valid.

  Even if single block operation is being executed, once the NC reads any one of the above G code commands, that G code will be continuously executed until canceled.
  If either one of the axis control groups ($1 or $2) operates in this condition, the other axis control group will also continue to operate in the same manner.
- G codes with ◎ mark are selected when the power is turned on.

12.5.2 Rapid feed positioning (G00)

G00 is used to move the tool rapidly to a specified X, Z coordinate and position it there.
G00 is also used to return the spindle.

Description

The G00 code is used for moving the tool or the spindle from the current position to the specified position in rapid feed.

The G00 code maintains the current state (modal) until a command (G01, G02, or G03) (see the G functions table) in the same A group and H group is specified.

The rapid feed rate has been defined in advance. This rate does not require specification in a program.

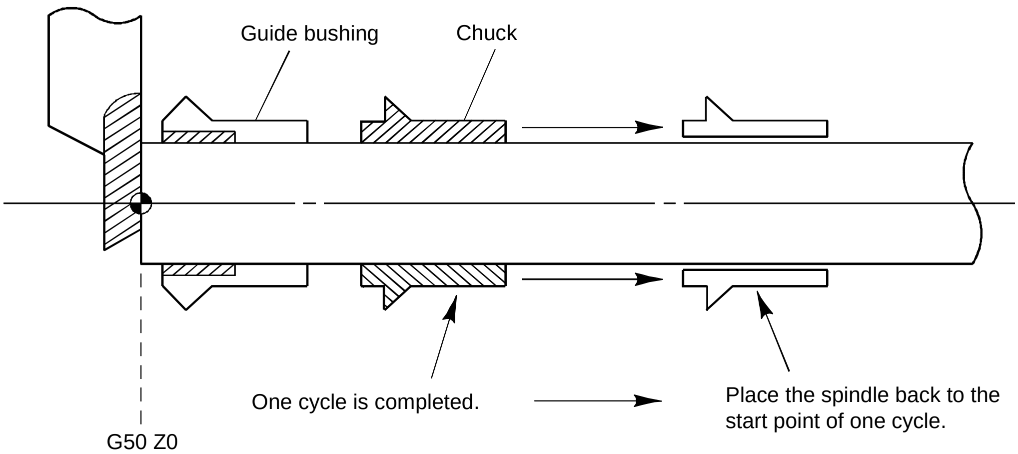
Rapid feed rate	X axis	X1 = 32 m/min, X2 = 32 m/min
	Y axis	Y1 = 32 m/min, Y2 = 8 m/min
	Z axis	Z1 = 32 m/min, Z2 = 32 m/min

Use U☐, W☐ and V☐ instead of X☐, Z☐ and Y☐ when an incremental command is specified.

[Command format]

G00 Z☐ Rapid feed in longitudinal (Z-axis) direction (X is fixed.)

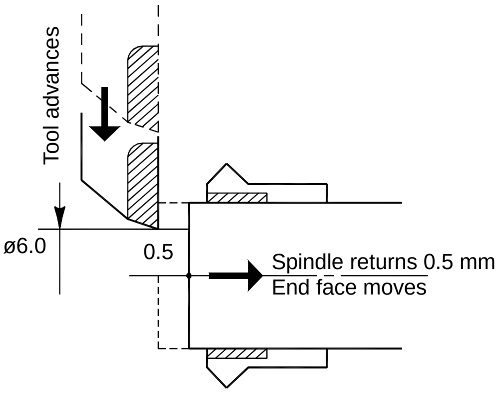
To complete one cycle and to proceed execution to the next cycle, use a G00 Z0 command to return the spindle to the start position.



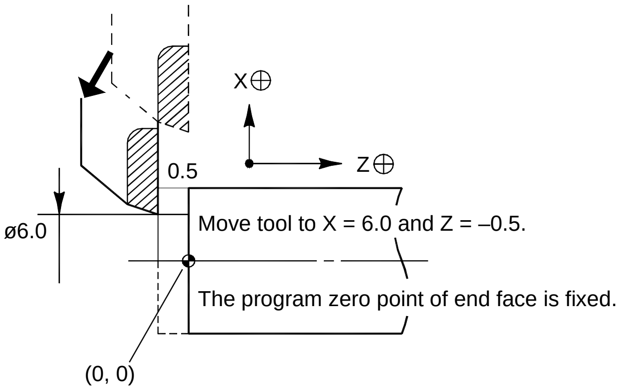
[Command format]

G00 X☐ Z☐ — Simultaneous rapid feed in diametrical (X-axis) and longitudinal (Z-axis) directions (Both X and Z axes move.)

If you want to position the tool in a diameter of 6.0 mm and set the clearance between the tool and material to 0.5 mm, specify a G00 X6.0 Z–0.5 command.



Actual movement



Conception in program

Rapid feed positioning (G00)

Normally, G00 is used for rapid feed. You can change the feed rate by adding F argument. Note, however, the F argument specified in this procedure is not held as modal functions.

[Command format]

G00 Z☐ ,F☐

12.5.3 Linear interpolation (G01)

G01 is used to perform a cutting operation while moving the tool on a line to the desired X and Z coordinates at the specified feed rate.

This function is called linear interpolation in the NC terminology.

The G01 code allows the tool to cut from the current position to the specified position in a straight line.

The G01 code maintains the current state (modal) until a command (G00, G02, or G03) (see the G functions table) in the same group is specified.

The feed rate is specified by "F□." See *<12.6 F Functions (F Codes) (Cutting Feed Rate)>*.

The unit of X□ and Z□ coordinate values is defined as shown below.

	Program command	Unit
With no decimal point.	X1	0.001mm
With a decimal point.	X1.0	1mm

* With the sub-micron specification, "X1" corresponds to 0.0001 mm.

Use U□ and W□ instead of X□ and Z□ when an incremental command is specified.

[Command format]

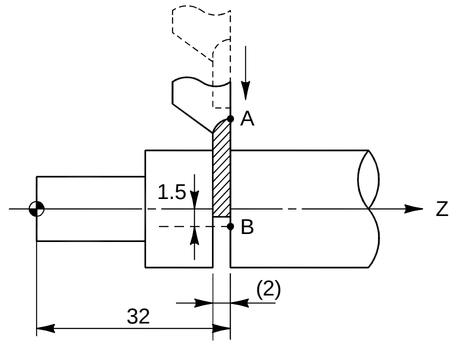
G01 X□ F□	Feed in diametrical (X axis) direction.(Z axis is fixed.)
G01 Z□ F□	Feed in longitudinal (Z axis) direction. (X axis is fixed.)
G01 X□ Z□ F□	Simultaneous feed (tapering) in diametrical (X axis) and longitudinal (Z axis) directions. (Both X and Z axes move.)

Tool feed in direction of diameter

[Command format]

G01 X☐ F☐ — Feed in diametrical (X axis) direction.(Z axis is fixed.)

Cutting from point A to point B (–3.0, 32.0)
G01 X–3.0 F☐

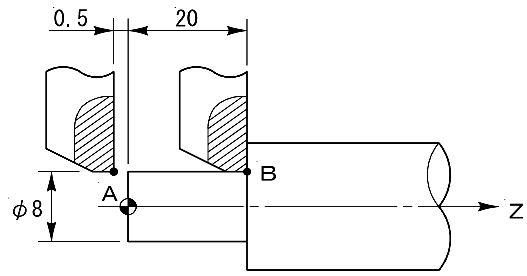


Tool feed in longitudinal direction

[Command format]

G01 X☐ F☐ — Feed in longitudinal (Z axis) direction. (X axis is fixed.)

Cutting from point A to point B (8.0, 20.0)
G01 Z20.0 F☐

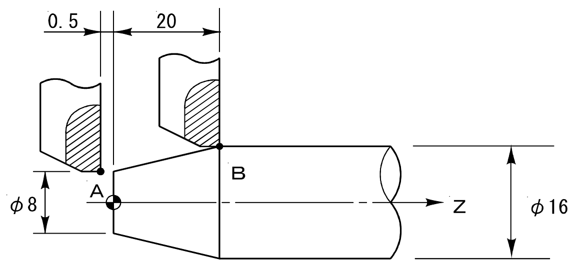


Simultaneous tool feed in diametrical and longitudinal directions (tapering)

[Command format]

G01 X☐ Z☐ F☐ — Simultaneous feed (tapering) in diametrical (X axis) and longitudinal (Z axis) directions. (Both X and Z axes move.)

Cutting from point A to point B (16.0, 20.0)
G01 X16.0 Z20.0 F☐



[Note]

Pay attention to the spindle rotation, coolant, chuck, and guide bushing states when performing the cutting feed.

12.5.4 Circular interpolation (G02, G03)

G02 and G03 are used for cutting while moving the tool on an arc with radius R from the current position to the specified position at the specified feed rate.
This function is called circular interpolation in the NC terminology.

The G02 and G03 codes enable the cutting on an arc with radius R from the current position to the specified position. In this case, it is not necessary to specify the center of the circle.

The G02 and G03 codes maintain the current state (modal) until a command (G00 or G01) (see the G code function table) in the same group is specified.

The feed rate is specified by "F□."

The unit of X□ and Z□ coordinate values is the same as for the G01 code.

[Command format]

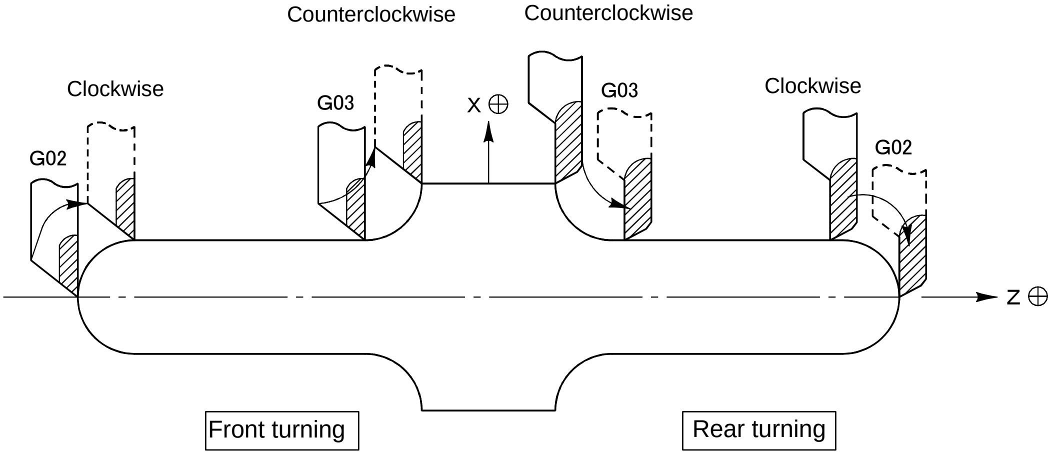
G02 X□ Z□ R□ F□	Clockwise
G03 X□ Z□ R□ F□	Counterclockwise

The G02 and G03 codes also can specify coordinates as shown below. "I" (X direction) and "K" (Z direction) indicate the distance (with signs) to the center of an arc from the start position. In this case, the radius of a circle is not required.

[Command format]

G02 X□ Z□ I□ K□ F□	Clockwise
G03 X□ Z□ I□ K□ F□	Counterclockwise

The coordinate system of G02 (clockwise) and G03 (counterclockwise) is used as shown below.



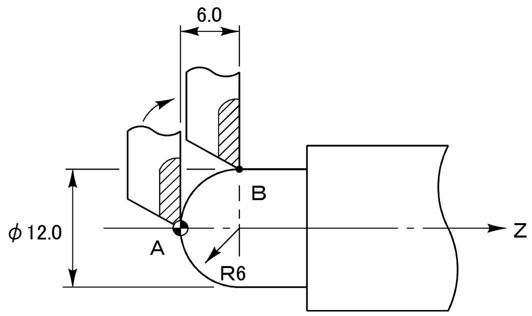
Tips: To move the tool in the + (plus) direction on the Z axis:
Specify G02 to cut the workpiece into a raised shape.
Specify G03 to cut the workpiece into a dented shape.

G02 Clockwise (CW)

Arc machining from point A to point B (12.0, 6.0)

G02 X12.0 Z6.0 R6.0 F☐

(G02 X12.0 Z6.0 K6.0 F☐)

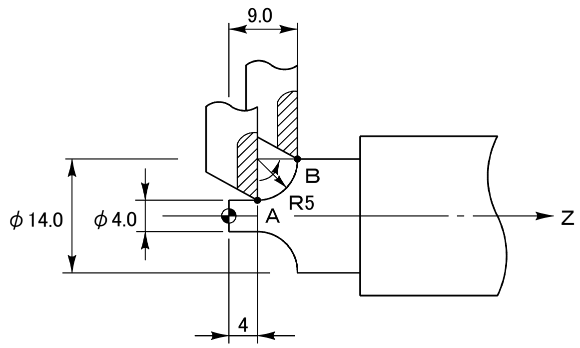


G03 Counterclockwise (CCW)

Arc machining from point A to point B (14.0, 9.0)

G03 X14.0 Z9.0 R5.0 F☐

(G03 X14.0 Z9.0 I5.0 K0 F☐)



12.5.5 Dwell (G04)

Use dwell to stop the program, holding the current operation state, for a specified period of time.

The purpose of dwell is:

- To stabilize the groove diameter.
- To stabilize speed before starting thread cutting after the specification of a spindle speed command. See *<8.1 Thread cutting with a tap>* and *<8.4 Thread Cutting>* in [Introduction] of the Instruction Manual (Optional).
- To perform deep hole machining (stepping) using a drill.
  See *<7.5 Drilling>* in [Introduction] of the Instruction Manual (Optional).

Specifying a dwell command suspends the machine for the number of seconds specified by U□ or P□ that follows "G04" before executing the next block after executing the command in the previous block.

The dwell function starts after the axis feed speed specified in the preceding block becomes zero.

[Command format]

G04 U□ Pause dwell (indicates the unit of seconds.)

or

G04 P□ (For address P, no decimal point can be used.)

The maximum command time is 9999.999 seconds.

[Sample program]

For a dwell command of 2.5 seconds, the command format is as follows:

G04 U2.5 (2.5 sec)

or

G04 P2500 (2500 msec)

12.5.6 Exact stop check (G09)

Use this command when higher precision in edge processing is required. If the Exact stop check is enabled (ON), cutting feed proceeds to the next block after confirming the speed is completely decelerated. Thus, the desired edge precision can be obtained.

[Command format]

G09	Exact stop check ON

[Sample program]

```
           $1
 :
T0200 (Front turning)
G0 X11.0 Z-1.0 T02
X6.0
G09 G1 Z0 F0.05
G09 X7.0
G09 X8.0 Z0.5 F0.03
G09 Z10.0 F0.05
G09 X10.0
G0 X11.0 T00
 :
```

[Note]

- Exact stop check function is disabled (OFF) at power on.
- Exact stop check function is effective only when G1, G2, or G3 command is specified.
- Difference from the error detect function
  The exact stop check function is enabled only in the block where G9 is specified. Meanwhile, the error detect ON (M92) is enabled until the error detect OFF (M93) is specified.
- When the exact stop check function is enabled, the longer cycle time is required than that in disabled state because the completion of cutting feed is prolonged.

12.6 F Functions (F Codes) (Cutting Feed Rate)

12.6.1 Cutting feed rate (F)

The F code specifies the feed rate for linear or circular cutting.

[Command format]

F☐ Feed rate command

The cutting feed cannot be executed unless these F codes are specified.

The feed rate given by the F command function maintains the current state (modal) until the next command is specified.
(The cutting feed is executed by the F code used in the previous machining.)

The feed rate is specified according to the cutting shape of "G01", "G02", and "G03" codes.

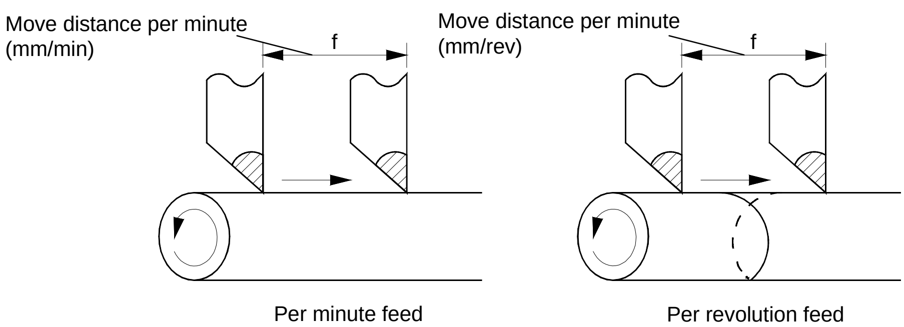
There are 2 types of the feed rate.

- G98: mm/min (per minute feed : distance of the tool in mm per minute)
- G99: mm/rev (per revolution feed : distance of the tool in mm per spindle revolution)

The G99 state is assumed when the power is turned on.
When switching the feed method from G99 to G98 in a block of G01 cutting block, the G code must come before the F code.

Example

G98 G01 X☐ Z☐ F☐



	Per minute feed	Per revolution feed
Description	The feed distance of the tool in mm per minute.	The feed distance of the tool in mm per spindle revolution.
Designation and address	F	F
Designated G code	G98	G99
Input feed rate range	1mm/min to 8000mm/min	0.001mm/rev to 999.999mm/rev
Override	Both per minute feed and per rotation feed are fixed in 100%. Feed rate changes by 0%, 1%, 3%, 5%, and in steps of 10% subsequently.	

Estimation from G98 and G99

The maximum value for F in the per rotation feed is: F (mm/rev) $\leqq \dfrac{8000\text{mm/min}}{\text{spindle speed (min}^{-1}\text{)}}$

Determine the proper feed rate by tables of the cutting conditions.

12.7 M Functions (M Codes)

12.7.1 M codes table

M code	Name	Description	See
M00	Program stop	Stops the automatic operation.	
M01	Optional stop	The automatic operation stops by pressing the switch.	
M02*	Cycle stop	Stops the automatic operation in the 1 cycle mode.	
M03	Main spindle forward rotation	Starts the main spindle forward rotation.	12.7.2
M04	Main spindle reverse rotation	Starts the main spindle reverse rotation.	12.7.2
M05	Main spindle stop	Stops the main spindle rotation.	12.7.2
M06*	Main spindle chuck close	Closes the main spindle chuck.	12.7.7
M07*	Main spindle chuck open	Opens the main spindle chuck.	12.7.7
M08	Enable bar stock exchange program	Starts the automatic material replacement program.	13.2
M09	Completed bar stock exchange program	Completed bar stock exchange program	13.2
M10*	Knock-out advance	Advances the back spindle knock-out.	15.5
M11	Knock-out retract	Returns the back spindle knock-out.	15.5
M15	Back spindle chuck close	Closes back spindle chuck.	12.7.8
M16*	Back spindle chuck open	Opens back spindle chuck.	12.7.8
M18*	Enable main spindle C axis	A series of C-axis ON operations (M18C***)	14.2.3
M20	Main spindle indexing cancel	Cancels main spindle indexing. In C-axis mode, the excitation of main spindle is not canceled.	14.2.1
M23	Back spindle forward rotation	Starts the back spindle forward rotation.	12.7.3
M24	Back spindle reverse rotation	Starts the back spindle reverse rotation.	12.7.3
M25	Back spindle stop	Stops the back spindle rotation.	12.7.3
M28*	Main spindle indexing	A series of spindle indexing operations (M28S***)	14.2.1
M31	Work conveyor ON	Workpiece conveyor start ON	12.7.9
M33*	Positioning to product separation position	Performs a series of axis feed operations. Axes move to the product separation position.	15.7
M34*	A series of operations for product separation	Performs a series of axis feed, chuck open, and knock-out operations.	15.7
M48*	Enable back spindle C axis	Turns on the back spindle C axis for a series of operations. (M48C***)	14.2.4
M50	Pick-off failure detect	After a workpiece is cut off, the main spindle rotates at low speed and the M50 command is executed.	15.12
M51*	Tool breakage detection ON	Starts cut-off tool break detection.	13.9
M52	Coolant ON	Starts the coolant supply.	
M53	Coolant OFF	Stops the coolant supply.	
M54	Turn off the bar loader torque.	Stops the feeding of the pushrod.	13.3
M55*	Start bar loader operation	Starts the bar loader material replacement.	13.3
M56	Product count	Starts the product count.	
M61 to M65	External M code	Used when optional function is added.	
M72	Back spindle air blow ON		15.4
M73	Back spindle air blow OFF		15.4
M77	Wait until spindle synchronization is completed.	Waits for completion of synchronization of the main and the back spindles.	15.9
M78*	Back spindle indexing	A series of back spindle indexing operation (M78S***)	14.2.2
M79	Back spindle indexing cancel	Cancels back spindle indexing. While in C-axis mode, does not cancel the excitation of the back spindle.	14.2.2
M80	Rotary tool on gang tool post forward rotation	Rotates the rotary tool on gang tool post in forward direction.	12.7.4

M code	Name	Description	See
M81	Rotary tool on gang tool post reverse rotation	Rotates the rotary tool on gang tool post in reverse direction.	12.7.4
M82	Stop spindle on gang tool post reverse rotation	Stops the rotation of the rotary tool on gang tool post.	12.7.4
M83	Front rotary tool on opposite tool post forward rotation	Rotates the rotary tool on opposite tool post in forward direction.	12.7.6
M84	Front rotary tool on opposite tool post reverse rotation	Rotates the rotary tool on opposite tool post in reverse direction.	12.7.6
M85	Front rotary tool on opposite tool post stop	Stops thhe rotation of the rotary tool on opposite tool post.	12.7.6
M86	Cutting start interlock enabled	Cutting is not starting until spindle speed reaches the specified value.	13.27
M87	Cutting start interlock disabled	Starts the cutting regardless of spindle speed.	13.27
M88	Interference check OFF	Disables interference check.	12.7.13
M89	Interference check ON	Enables interference check.	12.7.13
M92	Error detect ON	Turns error detection on (for higher precision in edge processing).	13.28
M93	Error detect OFF	Turns off error detect function.	13.28
M94	Back spindle speed fluctuation detection ON		13.6
M95	Back spindle speed fluctuation detection OFF		13.6
M96	Main spindle speed fluctuation detection ON	Enables abnormal rotation detection of the main spindle.	13.6
M97	Main spindle speed fluctuation detection OFF	Disables abnormal rotation detection of the main spindle.	13.6
M98	Call sub-program	Calls a subprogram.	13.5
M99	Return to main program	Be sure to insert this code at the ends of the main program and subprograms.	13.5
M108*	Sequential operation of bar material replacement	Sequential operation of bar material replacement	13.4.1
M122	Back spindle torque limit L selection (25%)		15.16
M123	Back spindle torque limit H selection (50%)		15.16
M124	Back spindle torque limit OFF (100%)		15.16
M140*	Opposite tool post advance	Advances the opposite tool post.	12.7.10
M141*	Opposite tool post retract	Retracts the opposite tool post.	12.7.10
M151*	Gang tool post retract	Moves the gang tool post X1 axis to the return position.	12.7.11
M180	Back rotary tool forward rotation	Starts the back rotary tool forward rotation.	12.7.5
M181	Back rotary tool reverse rotation	Starts the back rotary tool reverse rotation.	12.7.5
M182	Back tool spindle stop	Stops the back rotary tool rotation.	12.7.5
M190*	C1-C2 superimpose ON		15.9
M191*	C1-C2 superimpose OFF		15.9
M204	Thread cutting chamfering ON		12.7.12
M205	Thread cutting chamfering OFF		12.7.12
M211*	Mirror image ON	X (Motor Y1 axis)	14.8.7
M212*	Mirror image OFF	X (Motor Y1 axis)	14.8.7
M238*	Optional block skip		13.45
M239*	Back machining program skip		15.18
M260 to M269	External M code relay ON/OFF		
M320*	Move basket forward		15.6.1
M350*	Rapid Feed Rate Setting ON		13.26

M code	Name	Description	See
M351*	Rapid Feed Rate Setting OFF		13.26
M360*	Rapid Feed Acceleration/Deceleration Time Constant Setting ON		13.24
M361*	Rapid Feed Acceleration/Deceleration Time Constant Setting OFF		13.24
M430	Coolant Valve 1 ON		13.25
M431	Coolant Valve 1 OFF		13.25
M432	Coolant Valve 2 ON		13.25
M433	Coolant Valve 2 OFF		13.25
M434	Coolant Valve 3 ON		13.25
M435	Coolant Valve 3 OFF		13.25
M436	Coolant Valve 4 ON		13.25
M437	Coolant Valve 4 OFF		13.25
M452	Coolant Pump ON		13.25
M453	Coolant Pump OFF		13.25
M780*	Back spindle zero point indexing	(Available when a basket is used; executable only in MDI mode)	15.6.4

[Note]

- The M codes marked with an asterisk (*) used macros.
- Specify these macro M codes alone.
  Do not specify these codes along with any other code in the same block.
- If the M code belonging to the same group is specified in the same block, the later one becomes effective.
- Specifying any unused M code causes an alarm to occur.

12.7.2 Main spindle rotation and stop (M03, M04, M05)

These codes are used for rotating (forward and reverse) and stopping of the main spindle.

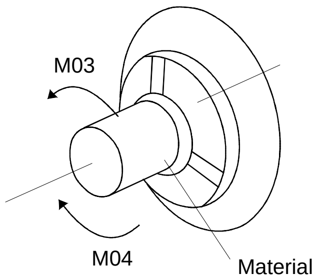
The forward rotation in this case means the counterclockwise rotation toward the spindle.

To stop the rotation of the spindle, you can also use the Spindle stop key on the operation panel.

[Command format]

M03 S1=☐	Main spindle forward rotation
M04 S1=☐	Main spindle reverse rotation
M05	Main spindle stop

The spindle speed changes when only "S1=☐" is specified during the rotation of the main spindle.



12.7.3 Back spindle rotation and stop (M23, M24, M25)

These codes are used for rotating (forward and reverse) and stopping of the back spindle.

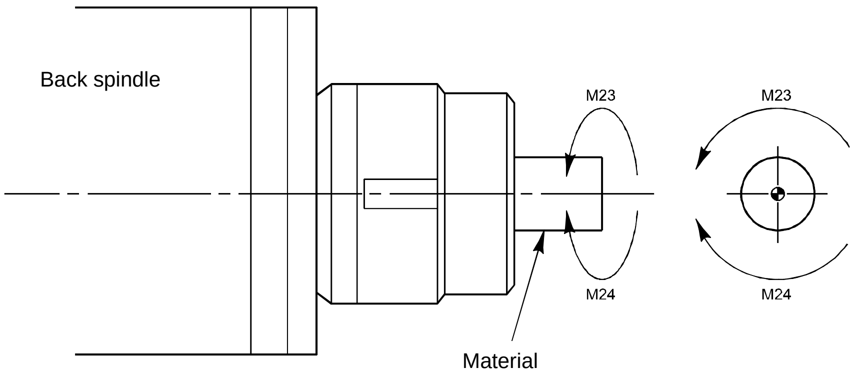
The forward rotation in this case means the counterclockwise rotation toward the back spindle.

To stop the rotation of the back spindle, you can also use the Spindle stop key on the operation panel.

[Command format]

M23 S2=$\boxed{}$	Starts the back spindle forward rotation
M24 S2=$\boxed{}$	Starts the back spindle reverse rotation
M25	Stops the back spindle rotation

The spindle speed changes when only an "S2=$\boxed{}$" is specified during the rotation of the back spindle.



[Note]

A workpiece receiver box may have been mounted when the back spindle chuck is opened. Therefore, the back spindle speed is limited to 100 min^{-1} (maximum). The back spindle rotates only at 100 min^{-1} even if M23 S2 = 2000 is specified.

If any of the following command has been specified, however, the spindle speed limit is released, regarding that the workpiece receiver box has been rarely mounted.

- Tool selecting commands T3000, T3100, T5100
- T0100 with K2 argument
- M15
- G650

12.7.4 Rotary tool on gang tool post rotation and stop (M80, M81, M82)

These codes are used for rotating (forward and reverse) and stopping of the rotary tool in a secondary process.

The forward rotation in this case means the counterclockwise rotation toward the rotary tool.

To stop the rotation of the rotary tool, you can also use the Spindle stop key on the operation panel.

[Command format]

M80 S3=☐	Rotary tool on gang tool post forward rotation
M81 S3=☐	Rotary tool on gang tool post reverse rotation
M82	Rotary tool on gang tool post stop

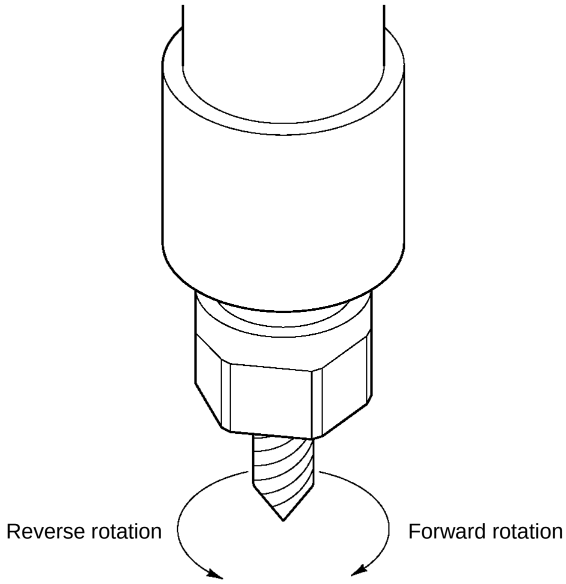
The spindle speed changes when only an "S3=☐" is specified during the rotation of the rotary tool.

Maximum speed:

B axis (T11 to T14 or T51 to T54): 8000min -1 (6000 min -1)

Other: 6000min $^{-1}$ (4500 min $^{-1}$)

The value in parenthesis indicates the rated spindle speed.



CAUTION

If the speed exceeds the rated value, the motor may be overloaded. Reduce the speed to machine the workpieces.
Machining the workpieces without reducing the speed may damage the machine components such as motor and gears.

[Note]

- A feed command must be specified in the G98 (feed per minute) mode. The G99 (feed per revolution) mode cannot be used.
- See *<14.4 Automatic Control of Rotation Direction of Rotary Tool>*. The rotation direction of rotary tool may change depending on tool layout or the selected tool number.
- For tool numbers other than B-axis tools, specifying 8000 min-1 will result in 6000 min-1 clamping.

12.7.5 Rotary tool on back tool post rotation and stop (M180, M181, M182)

This back tool post rotates the rotary tool of the back tool post, enabling eccentric drilling with a drill and end mill. The device can accept a standard drill sleeve using a sleeve adapter.

These codes are used for rotating (forward and reverse) and stopping of the rotary tool in a back machining.

The forward rotation in this case means the counterclockwise rotation toward the back rotary tool.

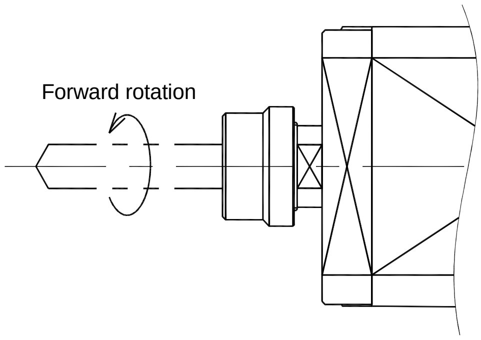
To stop the rotation of the rotary tool, you can also use the Spindle stop key on the operation panel.

[Command format]

M180 S5=☐	Back rotary tool forward rotation
M181 S5=☐	Back rotary tool reverse rotation
M182	Back rotary tool stop

The spindle speed changes when only an "S5=☐" is specified after the M180 and M181 codes.

Maximum spindle speed is 7500 min^{-1} (6000 min^{-1}). The value in parenthesis indicates the rated spindle speed.



CAUTION

The rated speed is 6000 min^{-1}. If the speed exceeds the rated value, the motor may be overloaded. Reduce the speed to machine the workpieces. Machining the workpieces without reducing the speed may damage the machine components such as motor and gears.

[Note]

- If you operate the machine in the state in which the end face drilling spindle of the back machining rotary tool is dismounted, use the sleeve adapter or the cap provided as an accessory to close the hole in order to prevent oil from going inside.
- A feed command must be specified in the G98 (feed per minute) mode. The G99 (feed per revolution) mode cannot be used.
- Refer to *<14.4 Automatic Control of Rotation Direction of Rotary Tool>*. The direction of rotation of rotary tools differ with tooling and the tool number.

12.7.6 Front rotary tool on opposite tool post rotation and stop (M83, M84, M85)

These codes are used for rotating (forward and reverse) and stopping of the rotary tool on the opposite tool post in secondary machining.

The forward rotation in this case means the counterclockwise rotation toward the rotary tool.

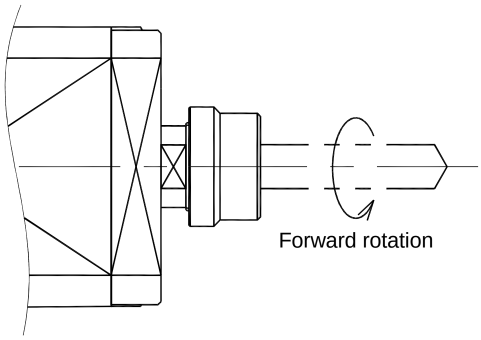
To stop the rotation of the rotary tool, you can also use the Spindle stop key on the operation panel.

[Command format]

M83 S4=☐	Front rotary tool on opposite tool post forward rotation
M84 S4=☐	Front rotary tool on opposite tool post reverse rotation
M85	Front rotary tool on opposite tool post stop

The spindle speed changes when only an "S4=☐" is specified after the M83 and M84 codes.

Maximum spindle speed is 7500 min^{-1} (6000 min^{-1}). The value in parenthesis indicates the rated spindle speed.



CAUTION

The rated speed is 6,000 min^{-1}. If the speed exceeds the rated value, the motor may be overloaded. Reduce the speed to machine the workpieces. Machining the workpieces without reducing the speed may damage the machine components such as motor and gears.

[Note]

- If you operate the machine in the state in which the end face drilling spindle of the front rotary tool is dismounted, use the sleeve adapter or the cap provided as an accessory to close the hole in order to prevent oil from going inside.
- A feed command must be specified in the G98 (feed per minute) mode. The G99 (feed per revolution) mode cannot be used.
- Refer to *<14.4 Automatic Control of Rotation Direction of Rotary Tool>*. The direction of rotation of rotary tools differ with tooling and the tool number.

12.7.7 Spindle chuck open and close (M06, M07)

These codes open and close the chuck built in the spindle.
Using the codes can allows the chuck to clamp the material to transfer the spindle rotation to the material.

[Command format]

M06	Spindle chuck close
M07	Spindle chuck open

Operations

M06	Closes the spindle chuck. Starts synchronization control between the Z1 axis and the A7 axis (bar loader).
M07	Checks the spindle status and opens the spindle chuck when the spindle speed is lowered to the specified speed or less. The specified spindle speed is set by the parameter. (Standard: 2000 min^{-1}) Stops synchronization control between the Z1 axis and the A7 axis (bar loader) and starts constant torque control.

12.7.8 Back spindle chuck open and close (M15, M16)

These codes open and close the chuck built in the back spindle.
Using the codes allows the chuck to clamp the material to transfer the back spindle rotation to the material.

[Command format]

M15	Back spindle chuck close
M16	Back spindle chuck open

Operations

M15	Closes the back spindle chuck.
M16	Opens the back spindle chuck. If M16 is specified while the back spindle is in speed acceleration/deceleration mode, the chuck starts to open when the back spindle stops or reaches the defined speed.

12.7.9 Workpiece conveyor start (M31)

This code sends the products collected from the workpiece chute on workpiece conveyor, to the outside of the machine.

[Command format]

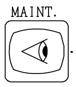
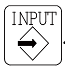
M31 Workpiece conveyor start

Operation

The workpiece conveyor operates for 14 seconds.

Setting

To change or set the workpiece conveyor operating time, follow the steps below.

1. Press the Maintenance key .
2. Press the menu key [I/F Diag].
3. Input "1001/M" and press the Input key .
4. Press the Parameter key.
5. Press the menu key [Menu SEL] repeatedly until [PLC-Data] item is displayed in the lower row of the menu display.
6. Press the menu key [PLC-Data].
   The PLC CONSTANT TIMER PARAMETERS screen is displayed.
7. Move the cursor to No. 18016 by pressing the cursor move keys.
8. Enter the workpiece conveyor operating time.
   Example: To set 15 seconds → Enter "150".
9. Press the Input key.
   The entered time is set.
10. Turn off and on the power to the operation panel.

[Note]

- The M31 command is valid only when the workpiece conveyor switch operates intermittently.
- The M31 command is included in M34 command.

12.7.10 Opposite tool post advance and return (M140, M141)

Using M codes, opposite tool post front drilling tools (T21△△△△ to T26△△) can be advanced or returned. These M codes are used to return the workpiece separator in the workpiece collection mode (with workpiece receiver box).

[Command format]

M140 W□ Z□) M1	Advances the opposite tool post.
M141 W□ Z□)	Retracts the opposite tool post.

[Argument]

M140

W□	:	Change the forward end position of the opposite tool post. Specify the difference from the forward end position of 205.0 in the Z2 axis machine coordinate. If this W argument is set to –10.0, the Z2 axis is positioned at 195.0 in the Z2 axis machine coordinate. If the argument is omitted, the Z2 axis is positioned at 205.0 in the Z2 axis machine coordinate.
Z□	:	Change the forward end position of the opposite tool post. The opposite tool post moves to the reference position of front workpiece end face (Z1 axis coordinate system). If the argument is omitted, the Z2 axis is positioned at 205.0 in the Z2 axis machine coordinate.
M1	:	Ignores an alarm message "EX737 S2 cover may interfere with back cross-machining holder".

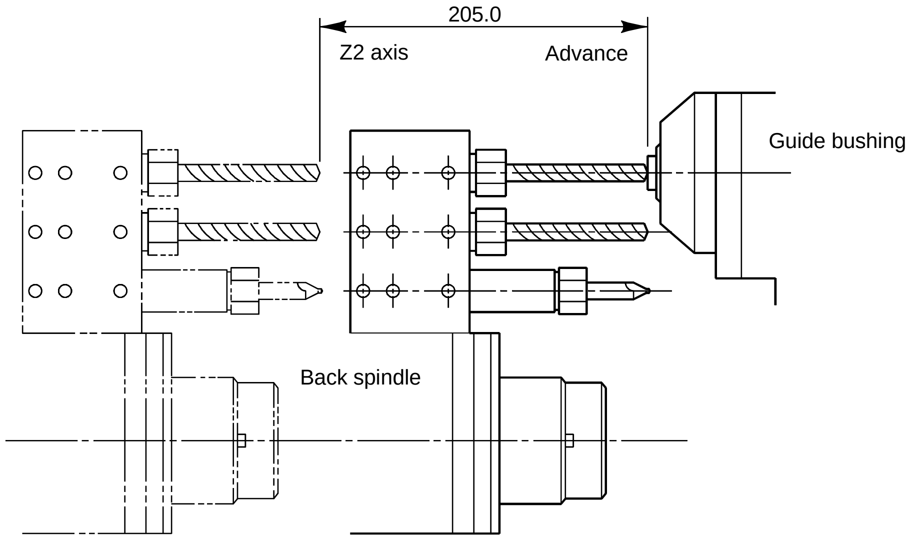
M141

W□	:	Change the return position of the opposite tool post. Specify the difference from the return position of 0 in the Z2 axis machine coordinate. If this W argument is set to 10.0, the Z2 axis is positioned at 10.0 in the Z2 axis machine coordinate. If the argument is omitted, the Z2 axis is positioned at 0 in the Z2 axis machine coordinate.
Z□	:	Change the return end position of the opposite tool post. The opposite tool post moves to the reference position of front workpiece end face (Z1 axis coordinate system). If the argument is omitted, the Z2 axis is positioned at 0.0 in the Z2 axis machine coordinate.

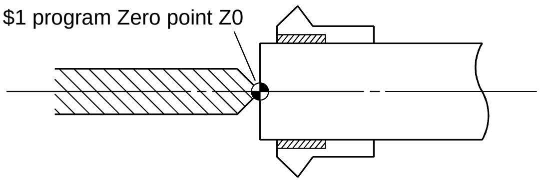
M140 and M141 commands are enabled on $1 after tools (T21△△ to T26△△) are specified.

With the machine of L20X, the forward end position depends on the type of tools being mounted.
Only the end-face drilling tool is mounted: 205.0 mm
Cross machining tool is mounted: 184.0
Vertical tool (without cross machining tool) is mounted: 203.0

If W☐ or Z☐ argument is omitted in the M140 command, the Z2 axis moves to the position of 205.0 in the Z2 axis machine coordinate.



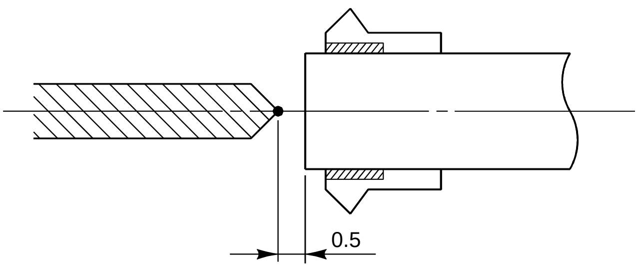
The tool set to the standard position is positioned as shown below by this code.



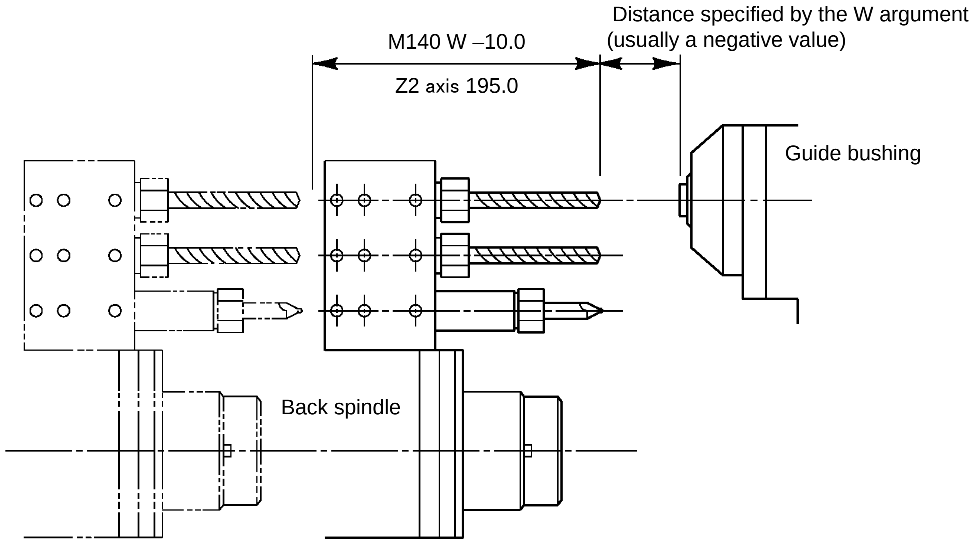
As shown in the figure above, the drill that was set without being shifted strikes against the material end face of Z = 0. Therefore, the end face is moved to Z = –0.5 so as to advance the tool as shown below.

[Sample program]

```
($1)
G0 Z-0.5
G610
T2200
M140
```



Using the M140 W☐ command, position the opposite tool post Z2 axis.



The tool returns to the machine zero point of the Z2 axis by the M141 code.

[Note]

- W☐ positioning is performed at the standard set position of the opposite tool post. Therefore, the positioning point of shifted tool must be set in axis control group 1 again by the G50 code.
- If T21△△ to T26△△ code is specified after M140 code, the tool selection is performed without retracting the opposite tool post. Pay attention to the interference with the material.
- If T31△△ to T38△△ code is specified before M140 code, the M140 code causes no operation to be performed (resulting in an alarm).
- The M141 code can also be used to return the workpiece separator advanced by the M320 command (in the workpiece receiver box collection mode).

12.7.11 Gang tool post retraction (M151)

The M151 command moves the gang tool post (X1 axis) to the return position.

[Command format]

M151 Gang tool post retraction

Axis control group

$1

[Sample program]

$1	
G600	
T0200	
:	
M151	Gang tool post retraction
:	

[Note]

- The gang tool post X1 axis cannot be specified after the M151 command is executed. To specify the X1 axis again, specify the machining pattern.
- When M151 is specified, the offset for X1 axis is canceled.

12.7.12 Thread Cutting Chamfering ON/OFF (M204, M205)

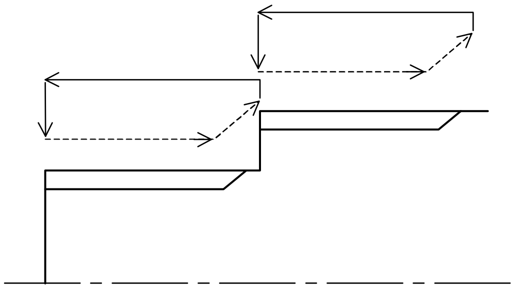
Chamfering in thread cutting operation is turned ON/OFF using the setting switch No. 17. However, M codes can also be used for this purpose.

[Command format]

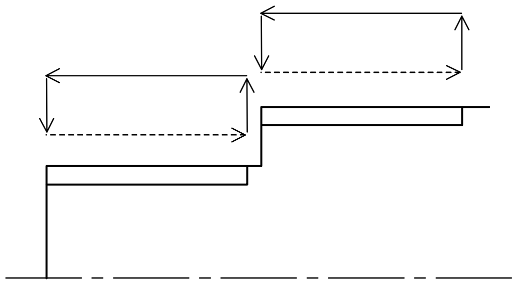
M204	Chamfering ON
M205	Chamfering OFF

Description

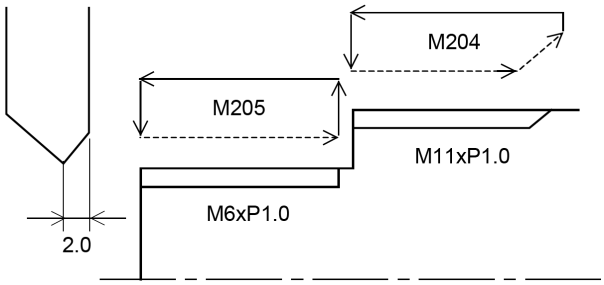
Turning thread cutting chamfering ON with the setting switch No. 17



Turning thread cutting chamfering OFF with the setting switch No. 17



By specifying the chamfering ON/OFF M code, chamfering can be turned ON and OFF disregarding the setting of the switch No. 17.



[Sample program]

```
M3 S1=[    ]
T300
M205 .......................................... Chamfering OFF
G50 W–2.0
G0 X8.0 Z–2.0 T[    ]
G92 X5.5 Z12.0 F1.0
X5.06
:
G0 X12.0 Z–2.0
G50 W2.0
M204 .......................................... Chamfering ON
:
M3 S1=[    ]
T300
G50 W–2.0
G0 X13.0 Z13.0 T[    ]
G92 X10.56 Z20.0 F0.1
X10.26
:
G0 X13.0
```

[Note]

See *<13.18 Thread Cycle Canned Cycle (G92)>*.

12.7.13 Interference check function (M88, M89)

The interference check function checks position data specified in Machining Data and Machine Structure screen to prevent machine components such as the tool post, back headstock, and product separation parts from interfering with other parts or workpieces as possible. This function raises an alarm if it detects an interference.

The interference check function is turned on usually when the power supply is turned on.

If the interference check function is disabled using the setting switch or by specifying the interference check OFF M code in the program, confirm that interference does not occur and perform the operation very carefully. Enable the interference check function as long as possible.

[Command format]

M88	Interference check disabled
M89	Interference check enabled

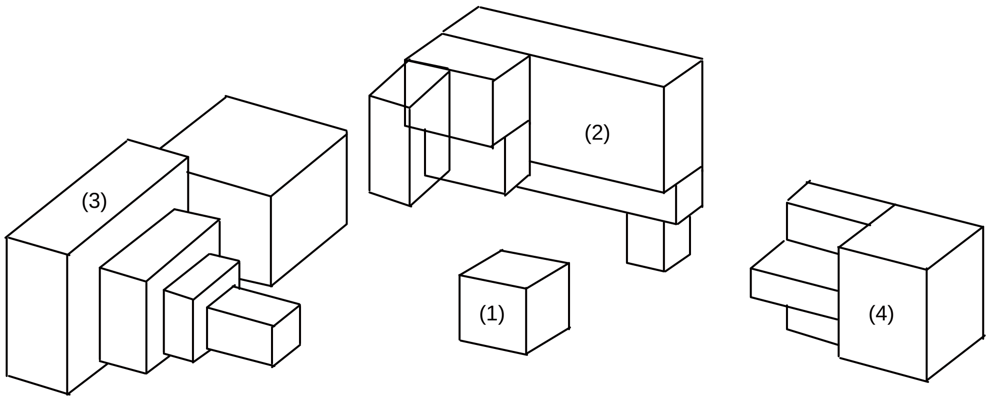
Components to be checked

(1) Guide bushing

(2) Gang tool post... The data depends on setting of "Front Mach Holder Name" in Machining Data.

(3) Back spindle including opposite tool post The data depends on setting of "Front Drill Holder Name" in Machining Data.

(4) Product collection box including back spindle .... The data depends on setting of "Back Drill Holder Name", "Back Spindle", and "Back work extend length" in Machining Data.



(Image drawing)

[Note]

- The interference check function assumes the typical settings for use under standard conditions. Note, therefore, that the function cannot check interferences when the machine is used under special conditions.
- The interference area cannot be set correctly if an incorrect holder or settings have been selected.
- If you run the machine with this function disabled by the program, check all the relevant parts in advance and use meticulous care.
- When the back spindle is being used in the workpiece receiver box collection mode, the interference check applies to the workpiece receiver box.

12.8 S Function (S Codes) (Spindle Speed Commands)

The S functions specified in the following format are also called rotation functions. These are used to specify the main spindle, back spindle, rotary tool speeds.

[Command format]

S1 = □□□□□	Main spindle	5-digit command
S2 = □□□□	Back spindle	4-digit command
S3 = □□□□	Rotary tool on gang tool post	
S4 = □□□□	Front rotary tool on opposite tool post	
S5 = □□□□	Rotary tool on back tool post	

The spindle speed is calculated from the following formula and rounded up to the nearest whole number.

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

N: Speed (min^{-1})
V: Cutting speed (m/min)
D: Material diameter (mm), or hole diameter for drilling
π: Circular constant (approx. 3.14)

Specification Ranges

Axis control group	Spindle	Speed
S1	Main spindle	200 to 10000min^{-1}
S2	Back spindle	200 to 8000min^{-1}
S3	Rotary tool on gang tool post	B axis: 200 to 8000 (6000) min^{-1} Other: 200 to 6000(4500) min^{-1}
S4	Front rotary tool on opposite tool post (L20X)	200 to 7500 (6000) min^{-1}
S5	Rotary tool on back tool post	200 to 7500 (6000) min^{-1}

CAUTION

The parenthesized figure shows rated speed. If the speed exceeds the rated value, the motor may be overloaded. Reduce the speed to machine the workpieces. Machining the workpieces without reducing the speed may damage the machine components such as motor and gears.

12.9 Queuing Function between Axis Control Groups "!"

[Command format]

!$\boxed{1}$ L$\boxed{2}$ Queuing between multi axis control group programs

$\boxed{1}$ → Enter the queuing axis control group number (1 or 2).

$\boxed{2}$ → Enter the queuing number (0 to 8999)

The queuing function enables specific parts of programs of two axis control groups ($1 and $2) to be executed simultaneously.

If the number is entered incorrectly, the program cannot advance in the queuing state.

[Sample program]

Example of multi-axis control group queuing command

O□□□□

$1	$2
:	**:**
:	**!1L1**
!2L1	**:**
:	**:**
!2L2	**!1L2**
:	**:**
:	**:**
:	**!1L3**
!2L3	**:**
:	**:**
:	**:**
:	**:**
!2L4	**!1L4**
M56	
G999	**G999**
N999	**N999**
M02	**M02**
M99	**M99**
%	**%**

(Blank Page)

Product Code	C-L220E XII
Document Code	2E1-1201
Mfg. No.	L220E/0001 ~
Issue Date	2014.1

13. Sample Programming [Advanced]

13.1 Tool Position Offset

This function compensates for the difference between actually machined dimensions and dimensions on the drawing. Coordinates on the program need not be changed. If a difference is entered in advance, dimensions are automatically corrected. Before this automatic correction can be made, an instrument for giving the difference must be provided.

[Command format]

Tool selection and offset specification at the same time:	T□□□□ (4 digits)
When only compensation value is given after tool selection:	T□□ (2 digits)

Description

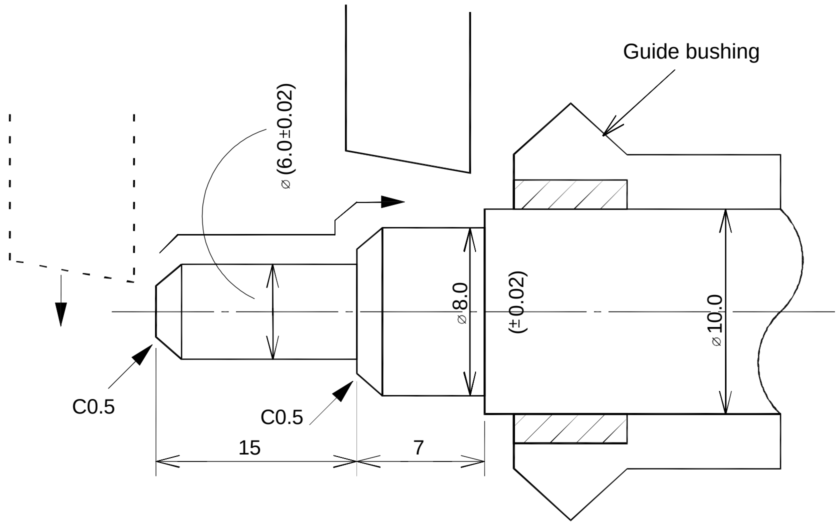
- An offset command T□ is given with a two-digit number.
  - **T□□□□** (4 digits):
    Tool selection and offset specification (only for the diameter direction) are given at the same time.
    The first two digits are a tool number and the last two digits are an offset number.
  - **T□□** (2 digits):
    When the tool has already been selected, this format is used to give or change an offset value without changing the tool. (Generally, the two-digit format is used.)
- Actual offset values are entered by the machine operator. The programmer gives offset numbers at places where coordinate correction seems necessary on the program. Generally, the following positions require an offset:
  - At the time of tool selection and positioning
  - At each step
  - At the beginning of longitudinal feed for end-face drilling or tapping
- The current offset remains in effect until the next tool selection command or another tool offset command comes.
- Any offset can be canceled by a T00. Cancellation is needed when:
  - The machine returns to the start point because cutting-off is completed.
  - The machine returns in the longitudinal direction after end-face drilling or tapping.
- Offset values are modal.
- The available offset numbers are 01 to 40. 41 through 80 are optional.
- When a **T□□□□** (4-digit) command is given, the machine moves to the position calculated with the offset value included (only in the diameter direction).

  If a **T□□** (2-digit) command is given, the offset will be in effect when the next X□ and Z□ command is given.

13.1.1 Examples of offset specification

Outer diameter machining

Outline of machining



[Sample program]

N0112 T0200

G00 X4.0 Z–0.5 $\boxed{T01}$ .... An offset command is given in a block which brings the tool close to the workpiece at the rapid feed rate.
(an offset for the dimension of a 6.0 mm diameter).

G01 X6.0 Z0.5 F0.03

Z15.0

X7.0 $\boxed{T02}$ .... If steps are involved, an offset is specified for each step.
(an offset for the dimension of an 8.0 mm diameter)

X8.0 Z15.5

Z22.0

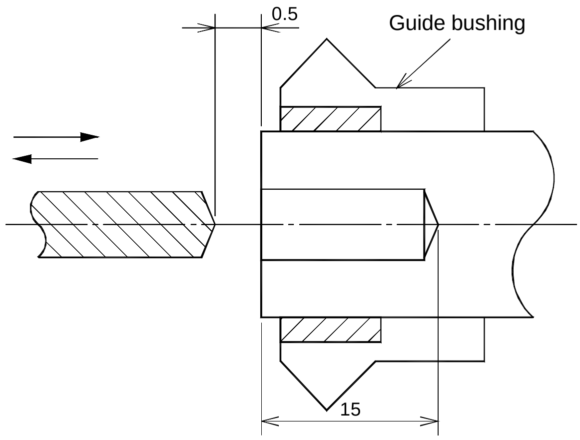
[Note]

Assumes that offset values of T01 and T02 on Z axis are the same.

If not, chamfering angle of the next block may get out of position. In some cases, it is required to enter the offset value of T02 in the preceding block, and add Z-axis offset T03 in the block containing Z22.0.

Face drilling (with a drill or tap)

Outline of machining



[Sample program]

N0323 T2300

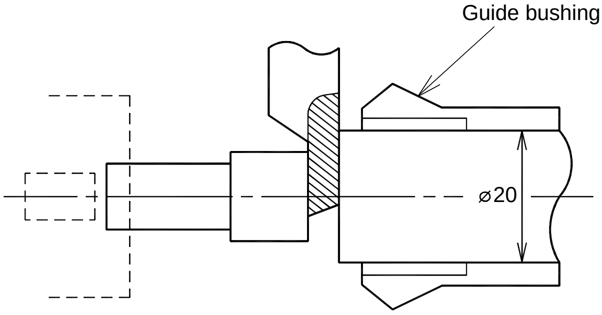
G00 Z–0.5

G01 Z15.0 F0.08 $\boxed{T13}$ .... Specifies offset value to correct the drill depth.

G00 Z–0.5 $\boxed{T00}$ .... Offset cancel (required at the time of return)

Cutting-off

Outline of machining



[Sample program]

N0401 T0100

G00 X21.0 Z22.0 $\boxed{\text{T01}}$ .... The product length is decided by the Z axis direction offset.

G650

!2 L1

G01 X–3.0 F0.02

G600

M05

M07

G00 X–3.0 Z0 $\boxed{\text{T00}}$ .... Offset cancel.
Always specify this command when returning the spindle to the start position.
If it is not specified, the position in the longitudinal direction will shift in every workpiece separation, and it finally causes a Z axis over-travel error.

M56

G999

N999

M02

%

13.2 Bar Feed Program Enable/Terminate (M08, M09)

These codes enable and terminate a bar changing program. The commands in the block (= bar changing program) enclosed between M08 and M09 are executed when the material shortage signal is received. They are otherwise skipped. (A block skip symbol "/" is used.)

The bar changing program is inserted between the cut-off and end processes.

[Command format]

Bar feed program enable/terminate

M08	Enable the bar changing program
M09	Terminate the bar changing program

[Sample program]

:	
G113	Spindle synchronization cancel
M08	Enable the bar changing program
M08	
/ ( )	Program for removing burr from the outer diameter of the material
/ ( )	
/ ( )	
/ G01 X33.0 W–25.0 F0.2	Move the cut-off tool by the material outer diameter + 1 mm upon extracting the material from the guide bushing
/ M53	Turn off coolant supply
/ M05	Stop rotating the main spindle
/ M54	Stop the machining torque of the bar loader
/ M07	Open the spindle chuck
/ M55	Issue the material replace command (to start replacing the material on the bar loader)
/ M06	Close the spindle chuck
/ G4 U2.0	Prevent bar loader torque switch time delay
/ M52	Turn on coolant supply
/ M03 S1= $\boxed{}$	Rotate the main spindle forward
/ G04 U2.0	Time for regulating the main spindle speed
/ W25.0 F0.2	Insert a material in the guide bushing
/ X–4.0 F0.02	Shortcut the tip of material
M09	Terminate the bar changing program

[Note]

- The M08 and M09 codes can be specified for both axis control groups.
- While synchronizing with the main spindle, be sure to specify G113 to cancel the spindle synchronization mode before executing bar change.

13.3 Bar Feed (M54, M55)

These codes are used to operate the bar loader.

[Command format]

Bar feed

M54	Turn off the bar loader torque
M55	Start the bar loader

The bar loader performs the following operations:

Actions of M55 (Synchronous bar loader)

Open the stabilizer.		
Move the pushrod to the material extract position.		Extract a remaining material
Turn on the material clamp.		
Move the pushrod to the material extract position (= return position).		
Turn off the material clamp.		
Turn on the material clamp.	Detect any material remaining	
Turn off the material clamp.		
Turn on the shelf material motor to move the rail backward.		Feed a shelf material.
Move the pushrod to the primary feed position.		Insert a material.
Move the pushrod to the return position.		
Turn on the shelf material motor to move the rail forward.		
Turn on the material clamp.		
Move the pushrod to the material insert position.		
Turn off the material clamp.		
Move the pushrod to the shortcut position.		Feed a material.
Close the stabilizer.		

13.4 Automatic Bar Feeder

When removing the residual material or supplying material using the automatic bar feeder unit, the following process must be inserted between the cut-off process and the ending process.

13.4.1 M108 (Material change using an argument)

Using an argument with the M code allows you to change material without writing program.

[Command format]

M108 U□ C□ D□ B□ S□ W□ F□ A□ R1 K1 M1 T□

U□	Movement of the X axis for deburring (in mm dia.). With no argument specified, this point is set at the point "Tool Positioning Point (DIA)" in the machining data. Do no omit the decimal point.
C□	Feedrate to position the deburring point (mm/min). With no argument specified, the feedrate is 150.0 mm/min. Do not omit the decimal point.
D□	Movement of Z axis for deburring (mm). With no argument specified, deburring is not performed.
B□	Movement of X axis for deburring (in mm dia.). With no argument specified, deburring is not performed.
S□	Spindle speed at withdrawing the residual material and inserting the material (min^{-1}). With no argument specified, the spindle speed is 300 min^{-1}. If the specified value exceeds 2000 min^{-1}, the spindle speed is clamped at 2000 min^{-1}.
W□	Movement of spindle at withdrawing the residual material and inserting the material (mm). With no argument specified, movement is 30.0mm.
F□	Feed rate (per minute) of the headstock (Z1 axis) when withdrawing the residual material and inserting the material. With no argument specified, feed rate is 3000.0 mm/min. Do not omit the decimal point.
A□	Dwell time after the spindle is chucked (second). With no argument specified, the dwell time is 2 seconds.
R1	Specify this argument to rotate the spindle during changing material. Omitted: The spindle stops. 1: The spindle rotates.
K1	Specify this argument to stop supplying coolant before unloading the material. When omitted, the supplying coolant is stopped.
T□	Dwell time (second) after tuning the coolant on. With no argument specified, the dwell time is 3.0 seconds. Do not omit the decimal point.
I□	This is a CAV specific argument. During inching of non-conformed materials, adjust the torque value (%) to suit the type of material and its diameter. For details, refer to the CAV instruction manual.

[Sample program 1]

All arguments necessary are input.

```
Preparation process
M09
↓
Machining Process
↓
T0100
G00 X[    ] Z[    ] T[    ]
G650
!2 L1
G01 X–3.0 F0.02
G600
M08
M08
/M108 U1.0 C150.0 D1.5 B3.0 S800 W30.0 F3000.0 A3.0 R1 K1 M1 T3.0
M09
M05
M07
G00 X–3.0 Z0 T00
M56
%
```

[Sample program 2]

Chamfering of residual material is not performed.

```
M108 S800 W30.0 F3000.0 A3.0 R1 T3.0
```

M108 Material Change Flow

1. The material shortage signal is issued from the bar feeder unit.
2. Cutting-off process is terminated at cutting-off end position in the machining data. ...Fig. (a)
3. Read the program Material Exchange (M108).
4. The spindle rotates at the "Cut-off Speed" in the machining data.
5. The tool moves to the "Bar Stock O.D." (machining data) + "U argument value" position at the feed rate specified by the C argument. ... Fig. (b)
6. The Z axis advances by the distance specified by the D argument. ... Fig. (c)
7. The residual material is chamfered according to the values specified by the D and B arguments at the "Cut-off Feed" in the machining data. ... Fig. (d)
8. If the K1 argument is omitted, Coolant OFF and Stop Medium-pressure Coolant Pump commands are issued.
9. The spindle speed is changed to the speed specified by the S argument.
10. The "Tool Positioning Point" in the machining data is shifted by the value specified by the W argument and at the same time the residual material is withdrawn from the guide bushing at the feed rate specified by the F argument. ... Fig. (e)
11. The spindle stops or keeps rotating according to the setting at the R argument.
12. Feed torque of the bar loader is turned off.
13. The main spindle chuck opens.
14. Bar stock is exchanged.

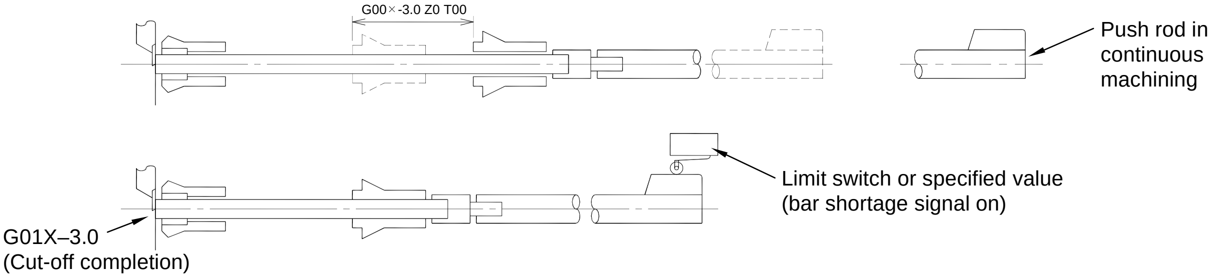
    (The residual material is brought to the retraction end of the bar loader and withdrawn by the finger chuck. The new bar stock is taken from the rack and inserted into the finger chuck. The bar loader pushes the bar stock up to the position set at the bar loader.) The bar stock exchange completed signal is returned from the bar loader to the machine.
15. The main spindle chuck closes.
16. The machine dwells for the period specified by the A argument.
17. The main spindle rotates at the speed specified by the S argument.
18. If the M1 argument is specified, phase adjustment of guide bushing is performed.
19. The bar stock is inserted into the guide bushing by the amount specified by the W argument. The bar stock is then fed forward by the front end cut length at the feed rate specified by the F argument.
20. Specify (Coolant ON) and Stop Medium-pressure Coolant Pump commands.
21. The machine dwells for the period specified by the T argument.
22. The spindle speed is changed to the speed specified at the "Cut-Off Speed" in the machining data.
23. Cutting-off is performed up to the position specified at "Cut-Off End" in the machining data at the feed rate specified at "Cut-Off Feed" in the machining data.
24. The Material Change OFF M code is read.
25. The M108 program ends.

[Note]

- When material change is specified according to Sample program 1, all steps from step 3 to step 25 are performed.
- When material change is specified according to Sample program 2, steps are performed from step 3 to step 25 omitting steps 4 to 7.
- Chamfering of the residual material is not performed unless arguments D and B are specified.
- Specify the material change operation (M108) in axis control group 1 ($1).
- In the material change operation (M108) commands, if "rotate (1)" is specified by spindle rotation selection (R argument), the spindle speed is clamped at 2000 min^{-1} when a value larger than 2000 min^{-1} is specified for the spindle speed (S argument) to be applied to residual material withdrawal and new bar stock insertion operation.

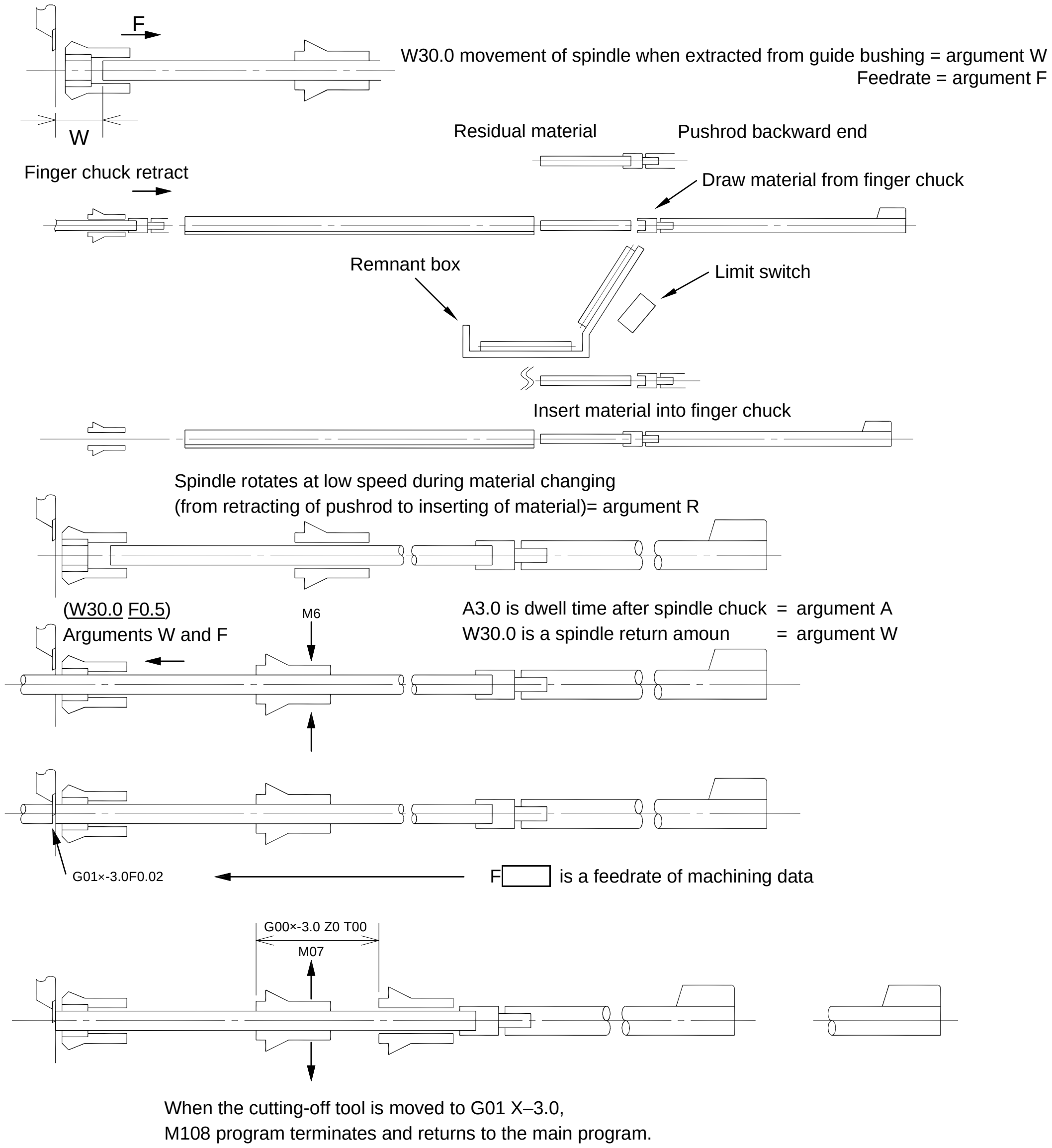
Material change flow

Material change flow when cutting-off procedure is complete, change material using an automatic bar feeder is performed in the following sequence illustrated. Chamfering of residual material can be performed as needed.



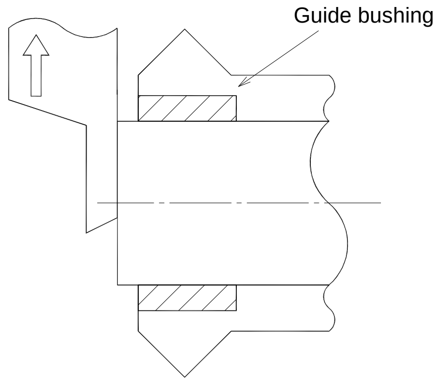
Perform chamfering of residual material (if necessary).
See (a) to (e) in the figure on the next page.

Chamfering is needed when the residual material is difficult to be withdrawn or may cause damage on the guide bushing due to burrs generated.



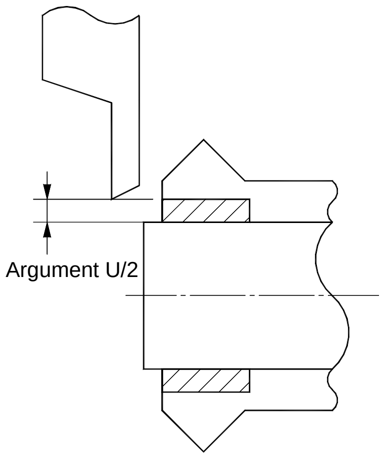
Chamfering of residual workpiece

(a)



When the steps 6 through 12 are completed.
The spindle rotates at the cutting-off speed specified in the machining data.
The tool moves upward to the position (b) at the feedrate specified by an argument C.

(b)

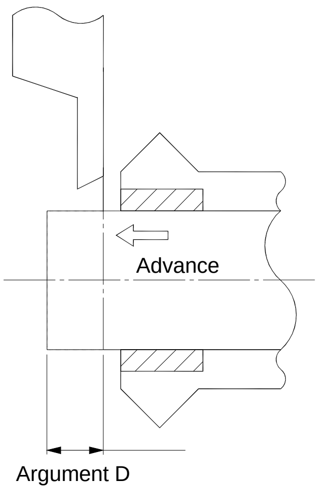


- The tool moves to the position of material outer diameter + value of argument U.

Note

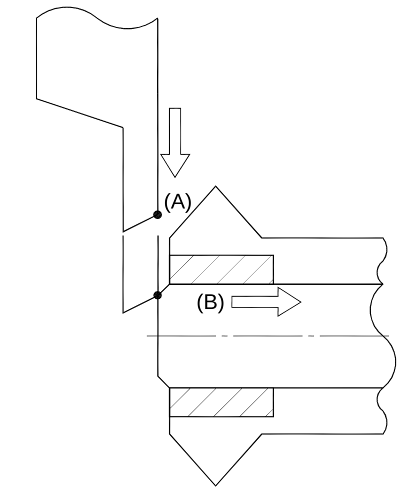
The position described above is not the tool positioning point in the machining data. If 0 is specified for argument U, the next material is fed to the point where the tool tip is positioned. So care must be taken.

(c)



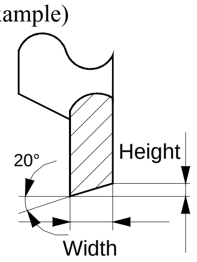
- The Z axis advances by the amount specified by the argument D at the feed rate specified by the argument C. (The material protrudes from the guide bushing.)
- Chamfering of residual material is ready.

(d)



- The tool moves downward from point (A) to (B), Z axis returns to the point specified by an argument D. Then the chamfering of residual material can be performed.
- Movement is specified by an argument B for X axis, by D for Z axis.
  (Select either axis as needed.)

(Example)

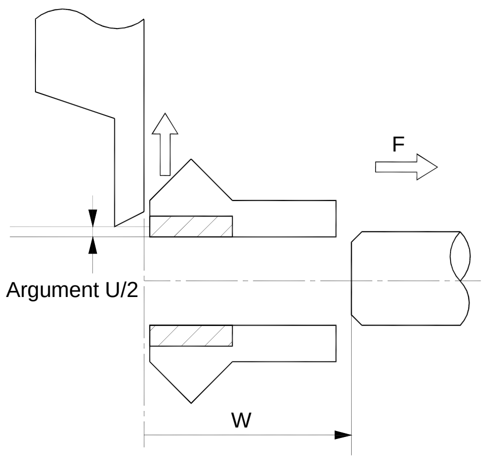


When an argument U = 1.0

Width	Height	D argument	B argument
1.0 mm	0.364 mm	1.4	2.8
1.5 mm	0.546 mm	1.55	3.1
2.0 mm	0.728 mm	1.7	3.4

In this setting chamfering of C0.5 on the residual material is performed.

(e)



- The tool moves to the position of material diameter +U argument value. At the same time, the residual material is returned as specified by an argument W at the feedrate specified by an argument F.

13.4.2 General-purpose material change program (by user program)

The material change program prepared by the user can be stored as a subprogram and called and executed by specifying M98.

[Command format]

M98 P□□□□

P□□□□ Material change program number (numbers only)

See *<13.5 Subprogram (M98)>* for details of M98.

[Sample program]

$1	$2
Preparation process	
M09	
↓	
Machining Process	
↓	
Cut-off process	
G01 X–3.0 F□	
M08	
M08	
/ M98 P□□□□	
M09	
M05	
M07	
G00 X–3.0 Z0 T00	
M56	
M02	**M02**
%	**%**

O□□□□ Material change program (prepared by user)

$1	$2	
M03 S1=800		Spindle rotates forward at 800 min^{-1}
G01 X21.0 W–30.0 F0.5		Withdraw material from the guide bushing, and at the same time, escape tool to material outer diameter
M53		Coolant OFF
M05		Spindle stop
M54		Turn off the machining torque of bar feeder
M07		Main spindle chuck open
M55		Material change command
M06		Main spindle chuck close
G04 U3.0		Dwell for 3 second
M52		Coolant ON
M03 S1=800		Spindle rotates forward at 800 min^{-1}
G01 W30.0 F0.5		Inserts the material into the guide bushing up to the short cut position.
M03 S1=2000		Spindle rotates forward at 2000 min^{-1}
G01 X–3.0 F0.02		Cut the material tip (short-cut)
M99		End of subprogram
%		

[Note]

- The material change program shown above is an example. Change values as needed.
- The program number must be 4 digits or less. Numbers 9000s cannot be used.
- The M7 (Chuck Open) command is valid when the spindle speed is 2000 min^{-1} or slower.
  Specifying the M7 command when the spindle speed exceeds 2000 min^{-1} causes an alarm to occur.

13.5 Subprogram (M98)

Once you store a coding sequence (that is going to be repeatedly used within a program) in advance as a subprogram, you can call this subprogram from the main program each time that sequence is needed in the program. And you can make the main program structure simple.

13.5.1 Subprogram call instruction

The subprogram call has the following regulations.

M98 P□ H□ L□ ,D2

P□	Program number (1 to 99999999) of the subprogram to be called (If omitted, the main program is called.)
H□	Sequence number (1 to 99999) in the subprogram to be called (If omitted, the program is executed from the beginning of the main program.)
L□	The number of times (1 to 9999) the subprogram is to be repeated. (If omitted, L1 is assumed.)
,D2	Specify this when calling a program stored in the compact flash card (CF). Specify a program number by the P□ argument. See <*13.40 Running the Program in External* Memory> for details.

13.5.2 Calling a subprogram from the main program

- The subprogram is executed only once if the number of repetition is not specified.
- The subprogram can be repeated up to 9,999 times when it is called from a main program once.
- Subprograms can be nested to eight levels.
- The subprogram call command can be specified in either of the axis control groups ($1 and $2) in the main program.
- The main program and subprogram are called for each axis control group.

(Main program)		(Subprogram)
$1	⟷	$1
$2	⟷	$2

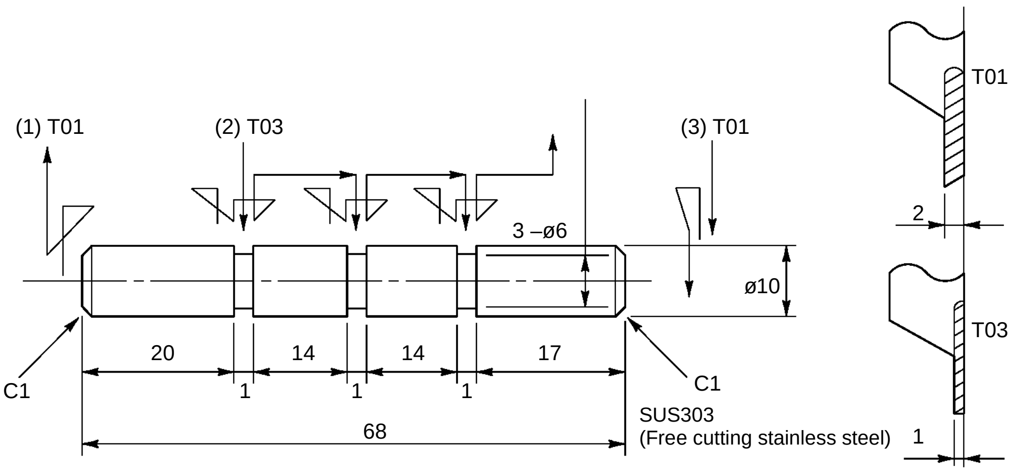
[Sample program]

Main program		Subprograms
O0001		O0002
$1	$2	$1
M98 P0002		
G00 X21.0 Z15.0		**M99**
		%
M02	**M02**	
M99	**M99**	

- The M99 command must always be placed at the end.
- The subprogram does not require M02.

13.5.3 Example of using a subprogram

Dimensions and machining layout



[Sample program]

Main program

O0001

$1	$2
G50 Z0	
M06	
G00 Z–0.5	
M03 S1=1800 G99	
G600	G600
N0101 T0100	
G00 X11.0 T01	
Z1.5	
G01 X7.0 Z–0.5 F0.015	
G00 X11.0	
N0203 T0300	
G00 X11.0 Z21.0 T03	
M98 P0002 L3	
N0301 T0100	
G00 X11.0 Z70.0 T01	
G01 X8.0 F0.015	
X11.0 F0.2	
W–1.5	
X8.0 W1.5 F0.015	
M320	
X-3.0	
G600	G600
M05	
M07	
G00 Z0 T00	
M56	
G999	G999
N999	N999
M02	M02
M99	M99
%	%

(Repeated 3 times)

Subprogram

O0002

$1	$2
G00 X11.0	
G01 X6.0 F0.01	
X11.0 F0.2	
W–0.6	
X9.8 W0.6 F0.01	
X11.0 F0.2	
W0.6	
X9.8 W–0.6 F0.01	
X11.0 F0.2	
W15.0	
M99	
%	

13.6 Spindle Speed Change Detection Function (M94, M95, M96, M97)

The spindle speed change detection function monitors the spindle speed. If the speed change exceeds a preset rate, this function automatically stops the machine. This prevents operation overload and damage to the guide bushing baking.

Both main spindle and back spindle have this spindle speed change detection function.

[Command format]

M97	Main spindle speed change detection OFF
M96	Main spindle speed change detection ON
M95	Back spindle speed change detection OFF
M94	Back spindle speed change detection ON

[Note]

- The main spindle speed change detection function is on by default when the power supply is turned on. To turn the detection function off, include the M97 or M95 command in the program.

  Normally, the program should contain M96 or M94 in the beginning to turn on the detection function for safety.
- Turn off the spindle speed change detection function to perform tapping with a dice, constant surface speed control or spindle synchronization control.

13.7 Constant Surface Speed Control (G96, G97)

While the material is cut in a diametrical direction or while cutting off the workpiece after finishing a machining process, the diameter of the material may vary depending on the portion of a workpiece, and the surface speed (the relative speed of the workpiece and tool) may also change. The control unit can detect the tool position, calculate the spindle speed, and change the spindle speed accordingly by using this command to instruct a relative speed.

[Command format]

G50 S☐ Q☐: Specifies the spindle speed limitation

The speed is controlled so that the spindle speed does not exceed the limitation during constant surface speed control.

S: Maximum spindle speed (min^{-1}) clamp value

Q: Minimum spindle speed (min^{-1}) clamp value

G96 S☐: Starts constant surface speed control

S: Constant surface speed value (m/min.)

G97 S1=☐: Ends constant surface speed control

(S2=☐)

S1=☐ min^{-1} Main spindle speed after canceling the constant surface speed control mode

S2=☐ min^{-1} Back spindle speed after canceling the constant surface speed control mode

[Note]

- If an S value is not specified when switching the mode between G96 and G97, the S value previously used is valid in the new mode.
- The spindle speed fluctuation detection cannot be turned on during constant surface speed control.
- G50 **S**☐ is effective only when the constant surface speed control command is effective by entering the G96 command.
- Specify Q, when the minimum spindle speed may change too lower.
- The tool selection command (**T**☐☐☐☐) cannot be used during constant surface speed control.
- When selecting the tool, specify the tool selection command after a cancel command (G97) is specified.

[Sample program]

$1	$2
Preparation process ↓ Front turning process ↓ Thread cutting process ↓ Other processes ↓	
(Constant surface speed control of spindle)	(Constant surface speed control of back spindle)
	M95 .... Back spindle speed fluctuation detection OFF
	M23 S2=1000
	G44 .... Back spindle feed control ON
	T3200
	G00 X16.0 Z0
	G50 S3500 .... Maximum spindle rotation clamp (3500 min^{-1})
	G96 S40 .... Constant surface speed control ON
	G01 X–1.0 F0.03
	G97 .... Constant surface speed control OFF
	M25 .... Stops the back spindle rotation.
	G43 .... Back spindle feed control OFF
	M94 .... Back spindle speed fluctuation detection ON
N501 T0100 .... Select the cut-off tool	
G00 X21.0 Z50.0 M97 .... Cut-off positioning by rapid feed Main spindle speed change detection OFF	
G50 S5000 .... Maximum clamp set command of spindle (5000 min^{-1})	
G96 S100 .... Constant surface speed control ON (Surface speed: 100 m/min)	
G01 X –3.0 F0.02 .... Cut-off	
G97 (S1=500) .... Constant surface speed control cancel (Normal rotation command of 500 min^{-1})	
M96 .... Main spindle speed change detection ON	
M05	
M07	
G00 Z□ T00	
M56	**M56**
G999	**G999**
N999	**N999**
M02	**M02**
M99	**M99**
%	**%**

* The above program does not cover product separation.

13.8 Tool Nose Radius Compensation Function

If a rounded tool-bit is used, the rounded tool nose can cause an error between the programmed form and the cutting form during taper cutting or circular cutting.

The tool nose R compensation function automatically calculates the error and compensates. The command code can fix the compensation direction.

[Command format]

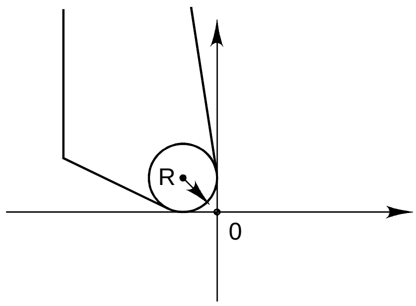
G40	Tool nose radius compensation mode Cancel
G41	Tool nose radius compensation left mode ON (Offsets the tool to the left in reference to the tool advancing direction.)
G42	Tool nose radius compensation right mode ON (Offsets the tool to the right in reference to the tool advancing direction.)

Enter tool nose R data in advance in <R> (tool nose radius value) and <P> (virtual tool nose No.) on the Tool Data screen.

Note that the meaning of <R> on the Offset screen is different from <R> on the Tool Data screen.

Virtual tool nose

- The virtual tool nose is the point of a nonexistent tip of the tool, corresponding to the zero point shown below.
- Be sure to set the tool bit in the holder as shown in the diagram.

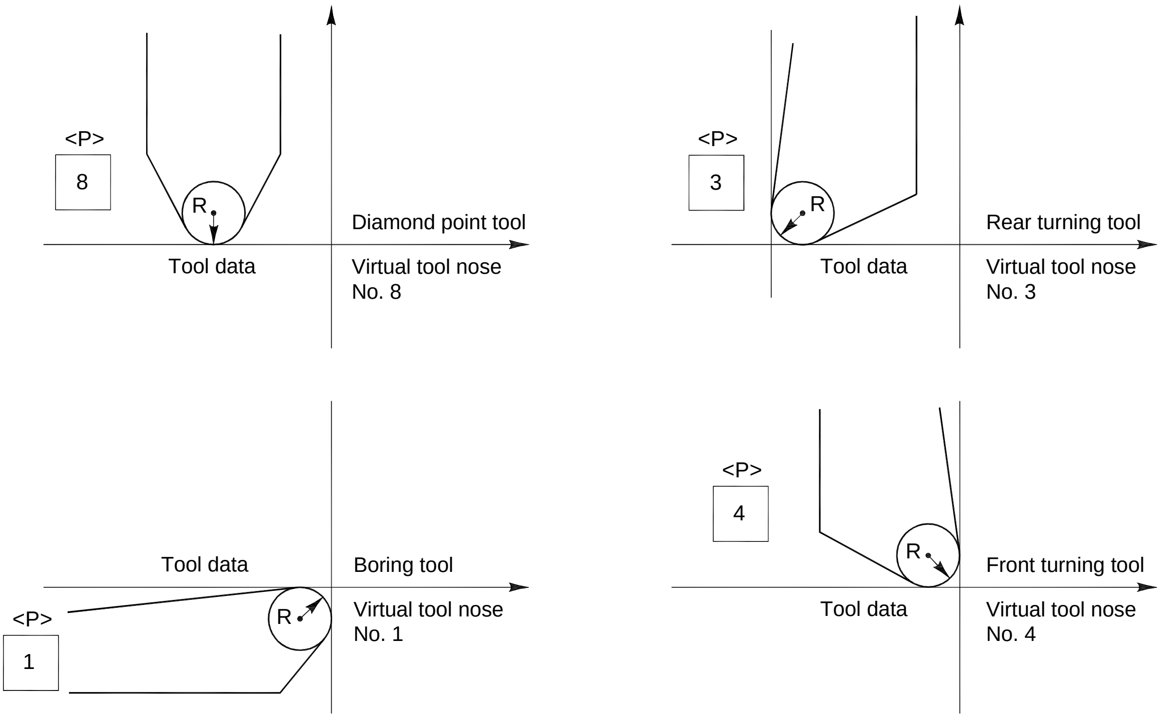


Virtual tool nose number

- The direction of the virtual tool nose viewed from the tool nose radius center is determined as the virtual tool nose number.
- Tool nose between 0 and 9 is determined according to tool noses.

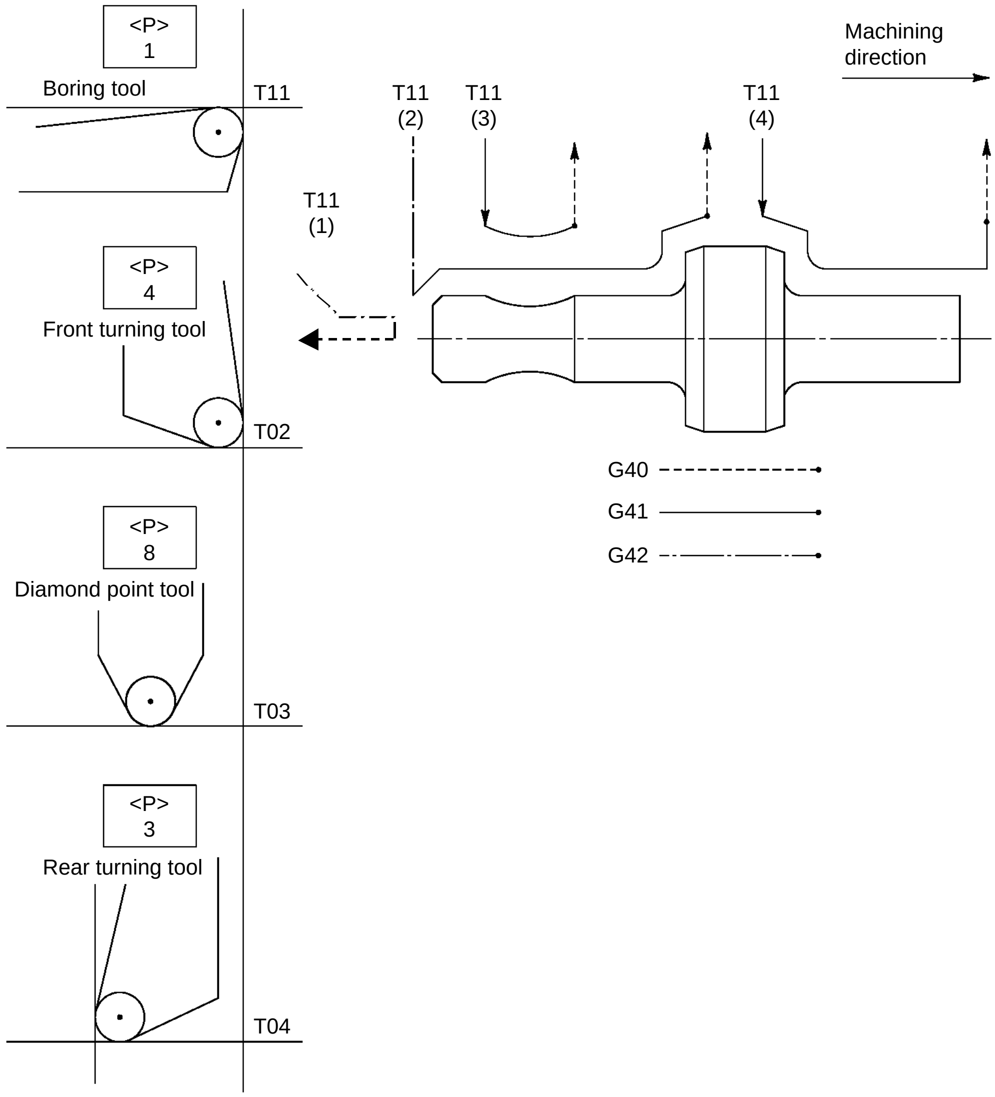
13.8.1 Tools subject to tool nose radius compensation and virtual tool nose numbers

These diagrams illustrate some commonly used tools with the associated virtual tool nose numbers.



13.8.2 Basic pattern of tool nose radius compensation G code

The conception of the tool nose radius compensation is described below.



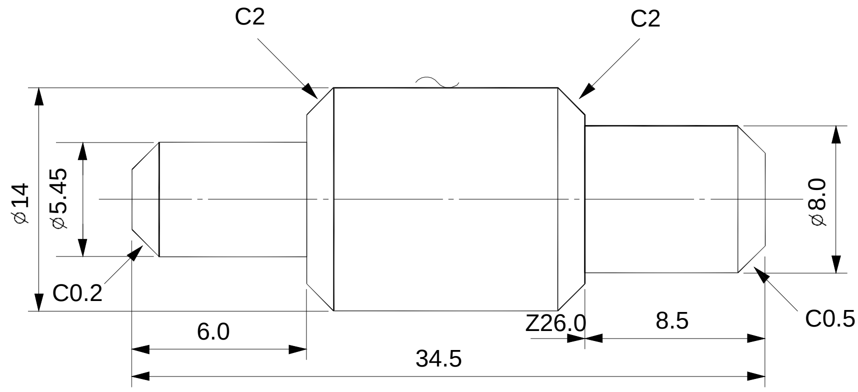
[Note]

- When positioning in rapid feed or canceling the tool nose radius compensation, pay attention to the interference with the material.

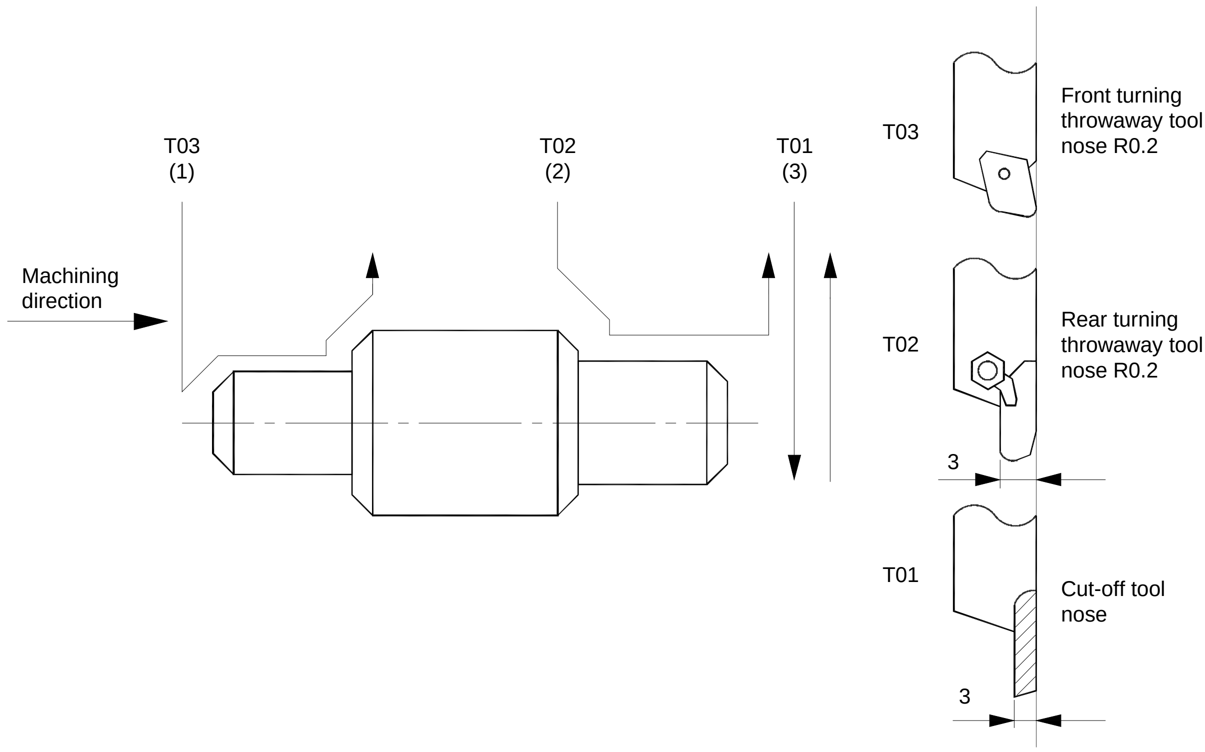
  The tool nose must be kept apart from the material by more than nose radius.
- If some improper G commands are specified during tool nose radius compensation execution, an alarm occurs.

  For the contents of the alarm, see the Instruction's manual of the NC manufacturer.
- While the tool nose radius compensation is being executed, tool exchange command (**T□□□□**) is disabled.

Machining drawing



Machining layout drawing



[Sample program]

O0300

$1	$2	
G50 Z0		
M06		
G00 X15.0 Z–0.5		
G99 M03 S1=2000		
G630	G630	.... Front/back parallel machining
N103 T0300		
G00 X15.0 Z–0.5 T03		
$\boxed{\text{G42}}$ G01 X4.05 F0.2		.... Tool nose radius compensation right mode ON
$\boxed{\text{G41}}$ X5.45 Z0.2 F0.03		.... Tool nose radius compensation left mode ON
Z6.0 F0.04		
G04 U0.3		
X10.0 F0.2		
X15.0 Z8.5 F0.03		
$\boxed{\text{G40}}$ G00 X16.0 T00		.... Tool nose radius compensation OFF
N202 T0200		
G50 W–3.0		
$\boxed{\text{G41}}$ G00 X15.0 Z23.5 T02		.... Tool nose radius compensation left mode ON
G01 X10.0 Z26.0 F0.03		
X8.0 F0.015		
Z34.0 F0.03		
X6.8 Z34.6 F0.02		
X15.0 F0.2		
$\boxed{\text{G40}}$ G00 X16.0 T00		.... Tool nose radius compensation OFF
G50 W3.0		
N301 T0100		
G50 W–3.0		
G00 X15.0 Z34.5 T01		
G01 X9.0 F0.2		
X–3.0 F0.03		
G50 W3.0		
M05		
M07		
G00 Z0 T00		
M56		
G999	G999	
N999	N999	
M02	M02	
M99	M99	
%	%	

[Note]

This program example does not contain product separation.

13.9 Cut-off Tool Breakage Detection (M51)

When the machine is equipped with a cut-off tool break detection unit, the cut-off tool breakage detection function is activated by specifying M51.

[Command format]

M51 X☐ W☐ F☐ Cut-off tool breakage detection

[Argument]

X☐ Specify the position of X1 axis to move the tip of touch sensor. If this argument is omitted, the X1 axis moves to the position –1.0 of the workpiece coordinate. In usual case, this argument is not required. Specify this argument only when the cut-off tool breakage detector cannot reach the workpiece.
Specify the X arguemnt so that the tip of the touch sensor surely pushes the material diameter.

W☐ Move distance (incremental) of workpiece on Z1 axis. If this argument is omitted, the Z1 axis does not move.

F☐ Specify the feed rate (per minute) to move the detection axis from the positioning point. If this argument is omitted, the detector moves at 2000 mm/min.

Detection method

Cut-off Tool Breakage Detection is performed as follows:

1. The tip of the touch sensor moves to the positioning point.
2. If the W argument is specified, the workpiece (Z1 axis) moves backward.
3. The touch sensor starts operating.
4. The tip of the touch sensor moves down to the position −1.0 of the workpiece coordinate. If X and F arguments are specified, the touch sensor moves down to the position specified by X argument at the feed rate specified by F argument.
   The tool post detects a cut-off tool breakage by sensing remaining workpiece. When the workpiece contacts with the touch sensor, it is determined that the cut-off tool is broken and an alarm is issued. Otherwise, the detection device proceeds to the next step.
5. The tip of the touch sensor moves to the positioning point.
6. If the workpiece (Z1 axis) has been moved by W argument in Step 2, the workpiece (Z1 axis) goes back to the position before the M51 command is issued.

CAUTION

If the workpiece is short or a large start position shift amount is set, specify the W argument to advance the spindle to the position where the sensor touches the material.
The advance amount varies according to the shift amount (shifted 4.0 mm from the zero point) shown in Fig. 2 on the next page, the start position shift amount, the workpiece length and the cut-off width. Enter an appropriate W argument value meeting the conditions.

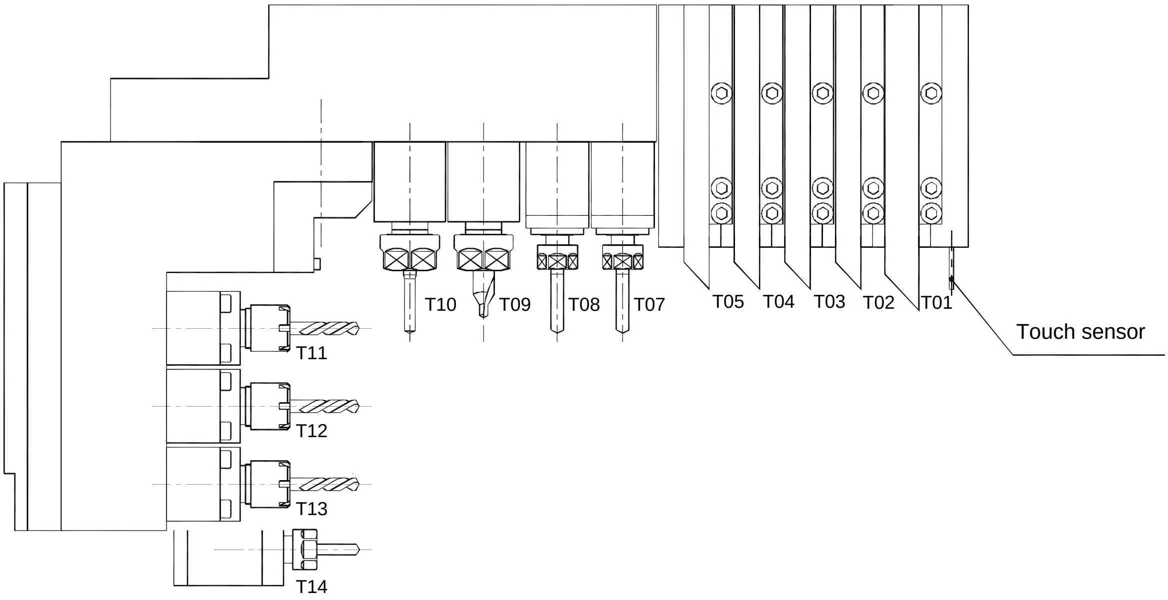


Figure 1

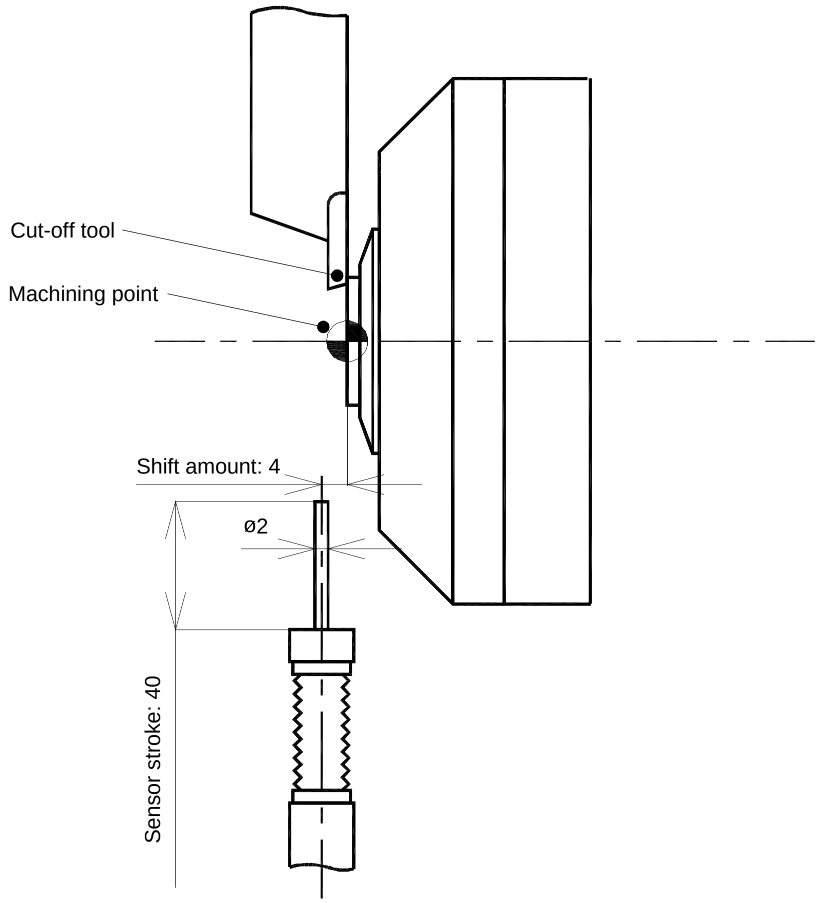


Figure 2

[Sample program]

$1	$3
G50 Z0	
G600	G600
M06	
G00 X13.0 Z–0.5	
M51 .... Cut-off tool breakage detection	
M03 S1=3000 G99	

[Note]

When returning the Z axis by specifying the W argument, the return distance must be within the range the material does not come off from the guide bushing. Burrs on the workpiece causes slippage of the chuck, and may result in machine damage.

13.10 Corner Chamfering / Corner Rounding

The corner chamfering/rounding function chamfers or rounds a corner, at the point of intersection of lines, by entering the chamfering or rounding size in the block that contains the coordinate value representing the corner.

Therefore, this function allows specification of chamfering or rounding at a corner without requiring coordinate calculation usually necessary for corner chamfering or rounding.

[Command format]

Corner chamfering:

G1 X(Z)☐ ,C☐ F☐

Corner rounding:

G1 X(Z)☐ ,R☐ F☐

X:	Specify the X coordinate of a corner.
Z:	Specify the Z coordinate of a corner.
,C:	Specify the corner chamfering size.
,R:	Specify the corner rounding size.
F:	Specified the feed rate.

[Sample program]

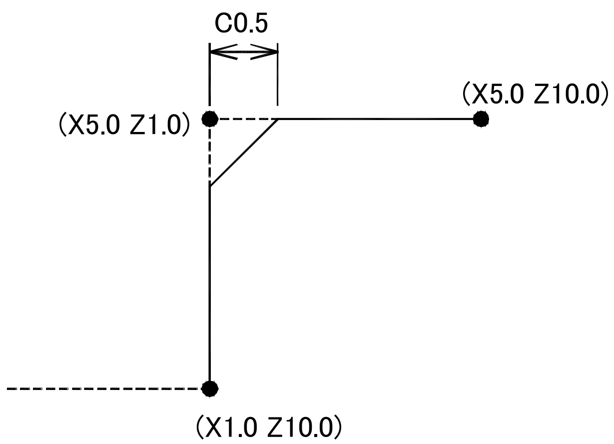
Corner chamfering:

:

G1 X1.0 Z1.0 .... Coordinate values of the point preceding the corner

X5.0 ,C0.5 .... Coordinate values representing the corner and the chamfering size

Z10.0 .... Coordinate values of the point that follows the corner to be chamfered



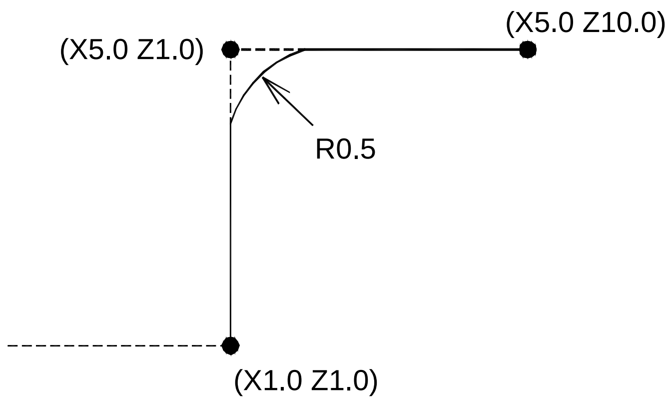
Corner rounding:

:

G1 X1.0 Z1.0 .... Coordinate values of the point preceding the corner

X5.0 ,R0.5 .... Coordinate values representing the corner and the corner rounding size

Z10.0 .... Coordinate values of the point that follows the corner to be rounded



[Note]

- This function does not compensate for tool nose R.

  To reflect the nose R to the workpiece shape by using the dimensions specified on the drawing, use the nose R compensation function.
- If both ",C" and ",R" values are specified in the same block, the command specified later is valid.
- If "," is not specified in ",C", the command is assumed as a C command.
- If "," is not specified in ",R", the command is assumed as an R command.
- Tool offset is calculated for the shape after the execution of corner chamfering or rounding.
- If the shape specified in the block that follows the corner chamfering or rounding block is not a line, program error (P382) occurs.

13.11 Thread Cutting (Equal Pitch Thread Cutting and Continuous Thread Cutting) (G32)

This function performs thread cutting by controlling the feed and the phase of main spindle rotation. Use this function to perform a straight and equal pitch screw thread cutting and tapered thread cutting. Continuous thread cutting that controls axis feed to maintain synchronization with spindle rotation with minimal errors at the shape changing point when cutting screw thread when shapes change is also possible.

This function is effective for thread cutting to the part of smaller diameter (machined in back turning process), which cannot be performed by the canned threading cycle. G32 command is used in G92 (canned threading cycle).

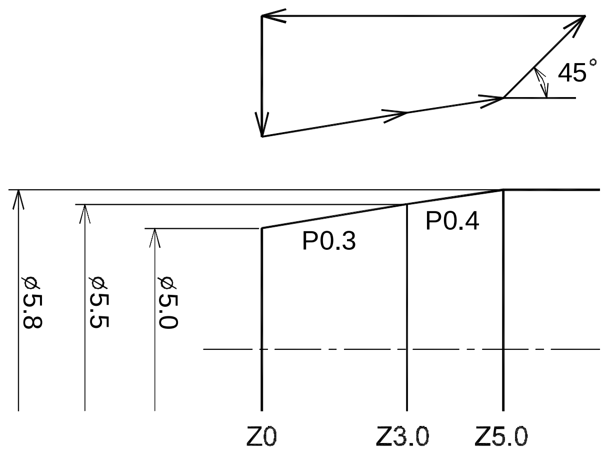
[Command format]

G32 X(U)☐ Z(W)☐ F☐ Q☐ E☐

[Argument]

X(U)☐	Specify the thread cutting infeed position. (U: Incremental command)
Z(W)☐	Specify the thread cutting end position in longitudinal direction. (W: Incremental command)
F☐	Specify the pitch (lead) in longitudinal direction.
Q☐	Enter the Q argument to specify the thread cutting start shift angle (0° to 359.999°)
E☐	Specify the number of threads per inch. The E☐ argument is used instead of the F☐ argument that specifies the longitudinal thread pitch/lead.

[Sample program]



This program is an example of machining a workpiece into a form as shown above. An outline cutting tool for front turning is used to finish-machine the top end of the workpiece into this form. Thread cutting is performed twice: the first infeed amount is 0.4 mm in the diameter, the second one is 0.2 mm in the diameter, and the final thread cutting is finished at 45°.

```
:
M3 S1=800
G99
T200
G0 X6.5 Z–1.0 T2  ............................  The tool is positioned to the thread cutting start point
X4.43
G32 X5.1 Z3.0 F0.3
X5.4 Z5.0 F0.4
X6.5 Z5.55
G0 Z–1.0
X4.23
G32 X4.9 Z3.0 F0.3
X5.2 Z5.0 F0.4
X6.5 Z5.65
G0 X6.5 Z–1.0 T0  ............................  The tool returns to the thread cutting start point
:
```

(X4.43 through X6.5 Z5.65:) Specify a thread cutting cycle in the program because the continuous threading function does not provide a canned cycle. Thread cutting is repeated twice in this program. If the thread cutting count increases, subprograms should be used.

When machining a workpiece into a generally threaded form, you can also use this function to change the amount of infeed or the angle of final thread cutting.

[Note]

- If screw thread changes in lead or geometry consecutively at very short intervals, the workpiece may not be machined correctly into the specified form.
- The program above is given as a sample, so specify machining conditions in consideration of the actual material to be machined, etc.
- G32 comamnd is canceled when G0 or G1 command is issued. To return the main spindle from the threading end position to threading start position, specify G0 or G1 at the top of every coordinate command.

13.12 Longitudinal Cut-Off Cycle (G75)

The longitudinal cut-off is a canned cycle which automatically performs grooving in outer diameter direction of workpieces by specifying the grooving end position, infeed amount, tool shift amount, and move distance of tool at fillet. This cycle is used for rough grooving on outer diameter and step machining in cutting-off operation on the fixed headstock lathes.

[Command format]

G75 R(1)
G75 X☐ Z☐ P☐ Q☐ R(2) F☐

[Argument]

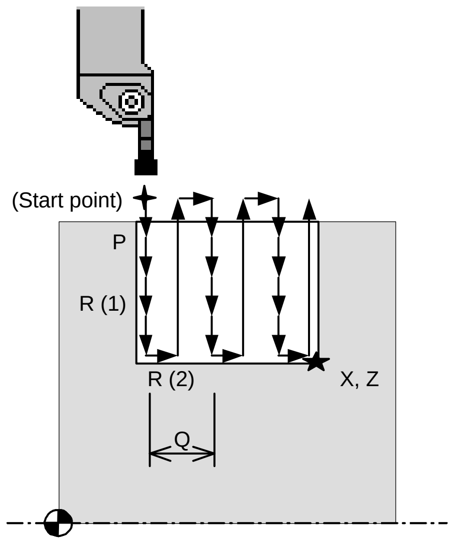
R(1)	Specify the return amount for every step.
X☐	Specify the grooving end position of X axis.
Z☐	Specify the grooving end position of Z axis.
P☐	Specify the infeed amount (steps) for every step.
Q☐	Specify the tool shift amount.
R(2)	Specify the move distance of Z axis at fillet.
F☐	Specify the cutting feed rate. (If omitted, the feed rate used in previous cycle is used.)

[Sample program]

:

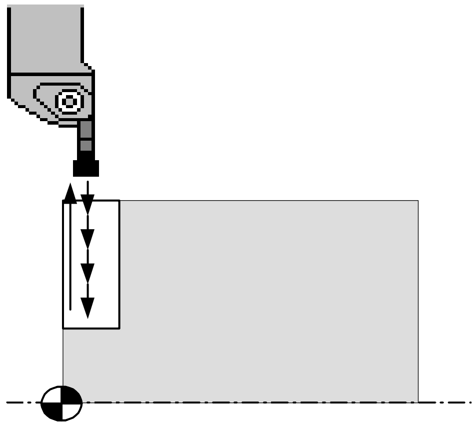
(1)	T200	Call the tool
(2)	G0 X20.0 Z7.0 T2	Position to the grooving position (start position of canned cycle)
(3)	G75 R0.2	Specify the return amount for every step
(4)	G75 X10.0 Z15.0 P2.0 Q2.5 R0 F0.1	Perform grooving to the fillet position X10.0, Z15.0: X direction: 2 mm at every step (P2.0) Z direction: Grooving 2.5 mm at every step (Q2.5) Z axis: Does not move (R0)
(5)	G0 X20.0 T0	Move the tool away (including canned cycle cancel)

:



Sample program for rough grooving to outer diameter

```
 :
T300
G0 X18.0 Z3.0 T3
G75 R0.2
G75 X5.0 P1.5 F0.08
G0 X18.0 T0
 :
```



[Note]

- Position the tool to the threading start point before specifying the cycle command.
- If Z and P arguments are omitted or 0 is specified, only the X axis moves.
- If the value of P argument is larger than the hole depth, step machining is not performed.
- The following conditios cause an alarm to occur.
  - The Z argument is specified, but the P argument is omitted or 0 is specified.
  - The value for P argument is larger than groove width.
  - The value for R(2) argument is larger than that for Q argument.
  - The value for R(1) argument is larger than that for P argument.
- The canned cycle is canceled when 01 group of G code (G0, G1, G2, or G3) is specified.

13.13 Deep Hole Drilling Cycle 2 (G79, G80)

Deep hole drilling can be specified by one line of commands when [Deep Hole Drilling Cycle 2 (G79)] is used.

The deep hole drilling cycle 2 allows specification of the first infeed depth and the second infeed depth independently. Furthermore, drilling in the end face direction and that in the cross direction are automatically distinguished by the arguments specified in the program.

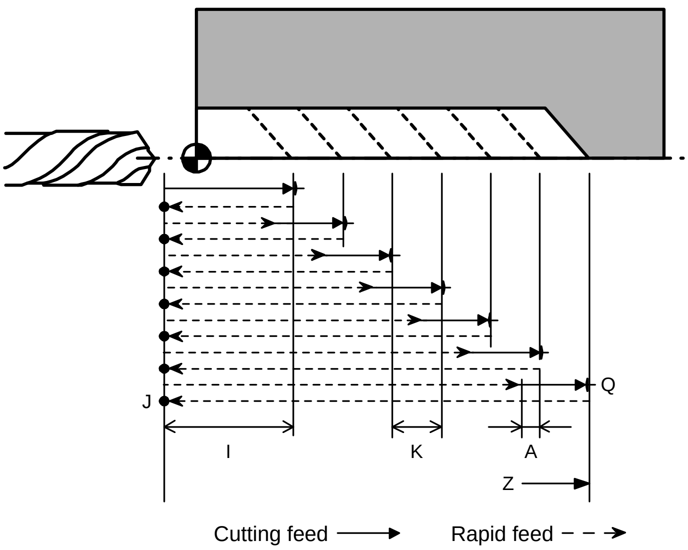
[Command format]

G79 Z(X)☐ R☐ I☐ K☐ A☐ Q☐ J☐ F☐ ,F☐

G80

[Argument]

Z☐	Specify the coordinate value of the bottom (end point) of a hole machined on the end face.	Always specify either Z or X. Specification of both Z and X is not permitted.
X☐	Specify the coordinate value of the bottom (end point) of the cross hole.	
R☐	Specify the distance from the drill positioning point to the drilling cycle start position in a radial value.	
I☐	Specify the depth of first infeed in a radial value.	If neither I nor K is specified, step machining is not performed. If only either I or K is specified, I = K is assumed.
K☐	Specify the depth of the second and successive infeed in a radial value.	
A☐	Specify the drill stop safety distance in a radial value. (See the figure below.)	
Q☐	Specify the dwell at the bottom of a hole. (In each drilling cycle, dwell is performed at the hole bottom.)	
J☐	Specify the dwell at the return position. (In each start of infeed from the return position, dwell is performed at the first positioning point.)	
F☐	Specify the cutting feed rate. If **F**☐ argument is not specified, the cycle is executed at the feed rate which is valid before the start of the drilling cycle. If G98 is specified before the specification of the cycle, specify the feed rate in a feed per minute value.	
,F☐	Specify the rapid feed rate in a feed per minute value.	



[Sample program]

```
:
:
T1200
G0 Z–1.0
G79 Z20.0 I6.0 K3.0 A0.5 J500 F0.1 T12
G80
G0 Z–1.0 T0
:
:
```

[Note]

- For the dwell command (Q and J arguments), a decimal point must not be specified. To specify 0.5 seconds, for example, input "Q (J) 500". If these arguments are omitted, dwell is not performed.
- If the A argument (drill return safety distance) is omitted, the value set at "G83 Retract" in the preparation parameter is used.
- The drilling cycle is canceled when G80, or G0, G1, G2 or G3 is specified. At the same time, argument values are cleared to zero.
- When a drilling cycle is performed in the 1-block state, operation does not stop during the execution of the cycle but it stops after the completion of the cycle.
- When the Hold key is pressed during the execution of a drilling cycle, the cycle stops immediately. The operation can be restarted from the stopped position.

13.14 Synchronized Tapping Functions (G88, G84, G80)

The synchronized tapping functions perform tapping while fully controlling the feed and rotational phase.

This feature brings about merits such as the use of an ordinary drill holder instead of a floating tap holder. The thread length is easily calculated by using an ordinary drill holder.

[Note]

The tapping speed becomes slower or faster than the value specified in the program, depending on the type of the holder to be used. When using holders with which the tapping speed changes, specify the tapping speed in consideration of deceleration or acceleration. For example, if the holder of 1/2 deceleration is used, specify the value for **F**☐ argument as a half of the standard value. Specify the rotary tool speed **S**☐ in consideration of deceleration ratio (twice as standard value in case of 1/2 deceleration).

13.14.1 Synchronized tapping for outer circumference with a rotary tool (G88 and G80)

This function performs tapping while synchronously controlling the rotary tool and the X axis (NC axis).

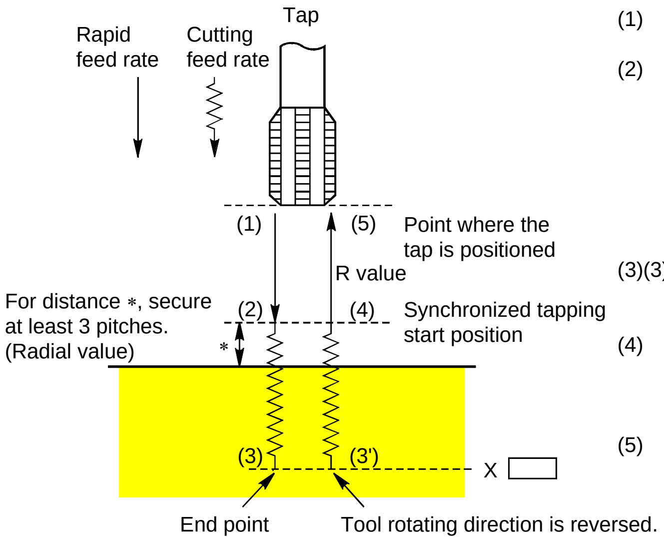
This function enables tapping for outer circumference, which makes a tapped hole in highly accurate depth.

[Command format]

G88 X☐ R☐ F☐ D±☐ S☐ Q☐, R1 or R2	Tapping cycle
G80	Tapping cycle cancel

[Argument]

X☐	Specify the tapping end position. The value must be specified for the diameter.
R☐	Specify the distance from the point where the tap is positioned to the position where synchronized tapping starts. The distance must be specified with a value for the radius.
F☐	Specify a screw pitch.
D±☐	Specify the rotary tool and the rotation in which the spindle rotates. 3: The rotary tool on gang tool post 5: The rotary tool on back spindle 4: The front rotary tool on opposite tool post. + denotes forward rotation, – denotes reverse rotation.
S☐	Specify the spindle speed.
Q☐	Specify this argument to speed up the tap when it returns. A multiple of 100 can be specified in the range 100 to 500. For example, when Q200 is specified, the tap returns twice as fast as when it heads toward the specified position. The default is Q100. In this case, the tap returns at the same speed as when it heads toward the specified position. A value "300" is recommended for the Q argument. However, the value of the Q argument depends on the conditions and the material. Determine the value in accordance with the shape of the tap. To specify the Q argument, high-speed synchronized tapping option (different from the synchronized tapping option).
,R1	Specify the synchronized tapping mode. Unless ",R1" is specified, the feed with the G98 command specified is used in the previous mode. If only "G88, R1" is specified, the mode is switched, but tapping is not performed.
,R2	Specify the synchronized tapping mode and phase adjustment. Specify this argument, for example, if you execute synchronized tapping, execute other machining process, and then execute synchronized tapping again for deburring. The starting positions of first and second synchronized tapping must be same. When using R2 argument (phase adjustment), the gear ratio of the motor and the holder must be 1:1. This argument is available only to B-axis rotary tools on the gang tool post (S3). It is not available to tools other than B-axis rotary tools on the gang tool post (S3) such as the front rotary tool on opposite tool post (S4) and the rotary tool on back tool post (S5). To use the R2 argument, the phase adjustment for synchronized tapping option (differs from the synchronized tapping) is required.



(1) Point where the tap spindle is positioned (diameter specification)

(2) Rapid feed positioning (radius specification) for the amount equivalent to the R value (synchronized tapping start position). The distance between the position (2) and the material outer diameter must be as long as 3 pitches or more.

(3)(3)' Tapping end position (Tapping end position and tool backward rotation)

(4) The tool moves by the amount equivalent to the R value and returns to the synchronized tapping start position.

(5) The tool returns to the point where the tap spindle is positioned, at the rapid feed rate.

[Sample program]

Material diameter: ø12.0

When the screw pitch is 0.7:

When using the cross machining tool on front side (L20X)

$2

```
:
M28 S0
M83 S4=0

T2300
G98 G00 X18.2 Z25.0 T13......................
G88 X0.0 R1.0 F0.7 D4 S500, R1
G80
G00 X18.2
M85
:
```

X18.2 = ø12.0 + {(1 + 2.1) × 2} = 18.2

Clearance between the workpiece and tap

Pitch × 3

Doubled because of diameter specification

[Sample program]

G84 command

When the screw pitch is 0.7:

When using the front rotary tool on opposite tool post (T20's)

$2

```
:
M28 S0
M83 S4=0
T2300
G98 G0 X0 Z-3.1 T23
G84 Z8.0 R1.0 F0.7 D4 S500, R1
Z8.0 R1.0
G80
G00 Z-3.1 T0
M85
:
```

G88 command

Material diameter: ø12.0

When the screw pitch is 0.7:

When using the front rotary tool on opposite tool post (T20's cross machining tool)

$2
:
:
M28 S0
M80 S4=0 G98
N0711 T2300
G00 X18.2 Z25.0 T23
G88 X5.0 R1.0 F0.7 D3 S500, R1
X0.0 R1.0
G80
G00 X18.2
M82
:
:

13.14.2 Synchronized tapping for the end face of a workpiece with a rotary tool (G84, G80)

While stopping the main spindle from rotating and synchronously controlling the rotary tool and the Z1 axis (or Z2 axis), this function performs tapping for the end face of the workpiece (center or eccentric). This function makes a tapped hole in highly accurate depth.

[Command format]

G84 Z□ R□ F□ D±□ S□ Q□, R1 or R2	Tapping cycle
G80	Tapping cycle cancel

[Argument]

Z□	Specify the tapping end position.
R□	Specify the distance from the point where the Z1 axis is positioned to the position where synchronized tapping starts.
F□	Specify a screw pitch.
D±□	Specify the rotary tool and the rotation in which the spindle rotates. 3: The rotary tool on gang tool post 5: The rotary tool on back spindle 4: The front rotary tool (end-face drilling tool) on opposite tool post + denotes forward rotation, – denotes reverse rotation.
S□	Specify the spindle speed.
Q□	Specify this argument to speed up the tap when it returns. A multiple of 100 can be specified in the range 100 to 500. For example, when Q200 is specified, the tap returns twice as fast as when it heads toward the specified position. The default is Q100. In this case, the tap returns at the same speed as when it heads toward the specified position. A value "300" is recommended for the Q argument. However, the value of the Q argument depends on the conditions and the material. Determine the value in accordance with the shape of the tap. To specify the Q argument, high-speed synchronized tapping option (different from the synchronized tapping option).
,R1	Specify the synchronized tapping mode.
,R2	Specify the synchronized tapping mode and phase adjustment. Specify this argument, for example, if you execute synchronized tapping, execute other machining process, and then execute synchronized tapping again for deburring. The starting positions of first and second synchronized tapping must be same. When using R2 argument (phase adjustment), the gear ratio of the motor and the holder must be 1:1. This argument is available only to B-axis rotary tools on the gang tool post (S3). It is not available to tools other than B-axis rotary tools on the gang tool post (S3) such as the front rotary tool on opposite tool post (S4) and the rotary tool on back tool post (S5). To use the R2 argument, the phase adjustment for synchronized tapping option (differs from the synchronized tapping) is required.

[Note]

The rotation direction of rotarytool may change depending on tool layout or the selected tool number. See *<14.4 Automatic Control of Rotation Direction of Tool Spindle>* for more information.

[Sample program]

When the screw pitch is 0.7:

Rotary tool on back tool post

```
$2
  :
M78 S0
G98 M180 S5=0
T3200
G0 X0 Z–3.1 T32
G84 Z8.0 R1.0 F0.7 D5 S500, R1
G80
G0 Z–3.1 T0
M182
  :
```

When the screw pitch is 0.7:

Rotary tool on gang tool post

```
$2
  :
M28 S0
M83 S4=0
T2100
G98 G00 X0 Z–3.1 T32
G84 Z8.0 R1.0 F0.7 D4 S500, R1
G80
G0 Z–3.1 T0
M85
  :
```

13.14.3 Synchronized tapping for the center of the end face of a workpiece (main or back) (G84, G80)

While rotating the spindle (main or back) and synchronously controlling the spindle and the Z axis (Z1 axis for main spindle, Z3 axis for back spindle), this function performs tapping for the center of the end face of the workpiece (main or back). This function makes a tapped hole in highly accurate depth.

[Command format]

M97(M95)	Main spindle speed change detection OFF (Back spindle speed change detection OFF)
G84 Z☐ R☐ F☐ D±☐ S☐ Q☐, R1 or R2	Tapping cycle
G80	Tapping cycle cancel
M96(M94)	Main spindle speed change detection ON (Back spindle speed change detection ON)

[Argument]

Z☐	Specify the tapping end position.
R☐	Specify the distance from the point where the Z1 axis is positioned to the position where synchronized tapping starts.
F☐	Specify a screw pitch.
D±☐	Specify the spindle and the rotation in which the spindle rotates. 1: Main spindle 2: Back spindle + denotes forward rotation, – denotes reverse rotation.
S☐	Specify the spindle speed.
Q☐	Specify this argument to speed up the tap when it returns. A multiple of 100 can be specified in the range 100 to 500. For example, when Q200 is specified, the tap returns twice as fast as when it heads toward the specified position. The default is Q100. In this case, the tap returns at the same speed as when it heads toward the specified position. A value "300" is recommended for the Q argument. However, the value of the Q argument depends on the conditions and the material. Determine the value in accordance with the shape of the tap. To specify the Q argument, high-speed synchronized tapping option (different from the synchronized tapping option).
,R1	Specify the synchronized tapping mode.
,R2	Specify the synchronized tapping mode and phase adjustment. Specify this argument, for example, if you execute synchronized tapping, execute other machining process, and then execute synchronized tapping again for deburring. The starting positions of first and second synchronized tapping must be same. When using R2 argument (phase adjustment), the gear ratio of the motor and the holder must be 1:1. To use the R2 argument, the phase adjustment for synchronized tapping option (differs from the synchronized tapping) is required.

[Note]

Be sure to execute the M97 (M95) command (spindle speed change detection OFF) before the G84 command. If the spindle speed change detection is not OFF, synchronized tapping ends up with an alarm.

[Sample program]

When the screw pitch is 0.7:

Front side

$1	
:	Clearance between the workpiece and tap
G99 M3 S1=0	Pitch × 3
M97	
T2300	
G0 X0 Z–3.1 T23	Z–3.1= –(1 + 2.1) = –3.1
G84 Z10.0 R1.0 F0.7 D1 S500, R1	
G80	
G0 Z–3.1 T0	
M5	
M96	
:	

Back side

$2
:
G44
G99 M23 S2=0
M95
T3300
G0 X0 Z–3.1 T33
G84 Z10.0 R1.0 F0.7 D2 S500, R1
G80
G0 Z–3.1 T0
M25
M94
:

13.14.4 Continuously synchronized tapping

This function performs synchronized tapping continuously. If ordinary synchronized tapping is unable to make a tapped hole in desired depth because the cutting load, this function achieves the desired depth by performing synchronized tapping continuously while changing the amount of infeed.

[Command format]

Synchronized tapping to end face of the workpiece:

G84 Z□ R□ F□ D±□ S□ Q□, R1 or R2	Tapping cycle
Z□ R□	Continuous tapping cycle

Synchronized tapping to outer circumference of the workpiece:

G88 X□ R□ F□ D±□ S□ Q□, R1 or R2	Tapping cycle
X□ R□	Continuous tapping cycle
G80	Tapping cycle cancel

[Argument]

X□	Specify the tapping end position. The value must be specified for the diameter.
R□	Specify the distance from the point where the tap is positioned to the position where synchronized tapping starts. The distance must be specified with a value for the radius.
F□	Specify a screw pitch.
D±□	Specify the rotary tool and the rotation in which the spindle rotates. 3: The rotary tool on gang tool post rotates forward. 4: The front rotary tool on opposite tool post rotates forward. 5: The rotary tool on back spindle rotates forward. + denotes forward rotation, – denotes reverse rotation.
S□	Specify the spindle speed.
Q□	Specify this argument to speed up the tap when it returns. A multiple of 100 can be specified in the range 100 to 500. For example, when Q200 is specified, the tap returns twice as fast as when it heads toward the specified position. The default is Q100. In this case, the tap returns at the same speed as when it heads toward the specified position. A value "300" is recommended for the Q argument. However, the value of the Q argument depends on the conditions and the material. Determine the value in accordance with the shape of the tap. To specify the Q argument, high-speed synchronized tapping option (different from the synchronized tapping option).
,R1	Specify the synchronized tapping mode. Unless ",R1" is specified, the feed with the G98 command specified is used in the previous mode. If only "G88, R1" is specified, the mode is switched, but tapping is not performed.
,R2	Specify the synchronized tapping mode and phase adjustment. Specify this argument, for example, if you execute synchronized tapping, execute other machining process, and then execute synchronized tapping again for deburring. The starting positions of first and second synchronized tapping must be same. When using R2 argument (phase adjustment), the gear ratio of the motor and the holder must be 1:1. This argument is available only to B-axis rotary tools on the gang tool post (S3). It is not available to tools other than B-axis rotary tools on the gang tool post (S3) such as the front rotary tool on opposite tool post (S4) and the rotary tool on back tool post (S5). To use the R2 argument, the phase adjustment for synchronized tapping option (differs from the synchronized tapping) is required.

[Sample program]

Material diameter: ø12.0

When the screw pitch is 0.7:

$1
:
:
M28 S0
S3=0 M80 G98
N0711 T1300
G00 X18.2 Z25.0 T13
G88 X5.0 R1.0 F0.7 D3 S500, R1
X0.0 R1.0
G80
G00 X18.2
M82
:
:

Material diameter: ø12.0

When the screw pitch is 0.7:

Front rotary tool on opposite tool post

$2
:
:
M28 S0
M83 S4=0
N0711 T2300
G98 G00 X18.2 Z25.0 T13
G88 X5.0 R1.0 F0.7 D4 S500, R1
X0.0 R1.0
G80
G00 X18.2
M85
:
:

13.15 Arc Threading (G35, G36)

The arc threading function performs arc threading through circular interpolation while controlling the feed of the tool synchronized with the spindle. Use of the continuous threading function enables the switching of machining - for example, from arc threading to linear threading or from arc threading to taper threading.

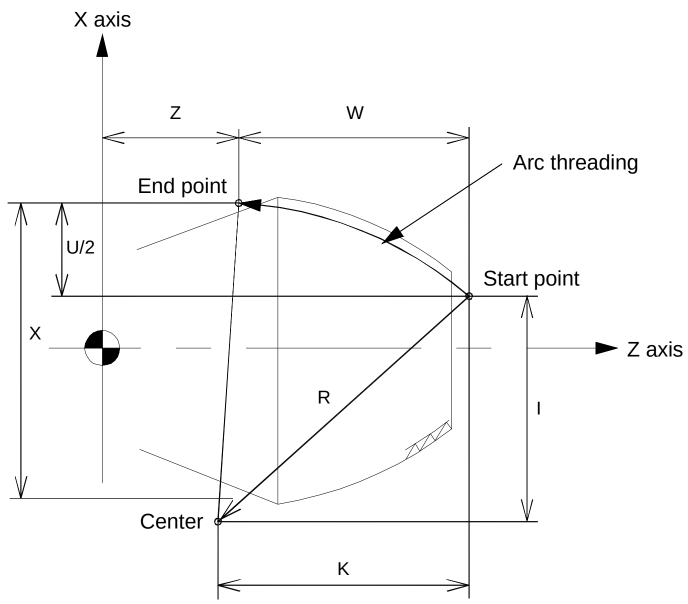
[Command format]

G35(G36) X(U)□ Z(W)□ I□ K□ (R□) F(E)□ Q□

G35	Clockwise threading
G36	Counterclockwise threading

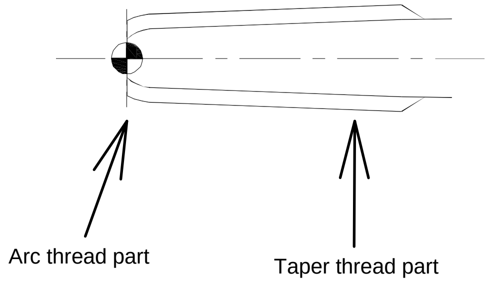
[Argument]

X(U)□	X-axis coordinate as the arc end point
Z(W)□	Z-axis coordinate as the arc end point
I□	X-axis coordinate as the center of the arc (increment to the center of the arc when viewed from the start point)
K□	Z-axis coordinate as the center of the arc (increment to the center of the arc when viewed from the start point)
R□	Radius of the arc (Specify the argument I, K, or R.)
F(E)□	Lead (F: General lead screw, E: Precision lead thread or inch screw thread)
Q□	Shift angle at which threading starts (0.001 to 360.000°)



[Sample program]

$1
T300
G50 W–10.0
G0 X11.0 Z–1.0 T3
X0.98
G35 U1.638 W4.48 R12.66 F0.9.......... Arc threading
G32 U0.682 W4.52 F0.9 .... Taper threading
G0 X5.0
Z–1.0
G50 W10.0



13.16 Differential Rotary Tool Function (G164)

This function controls spindles by superimposing the speed of a spindle on the speed of another spindle.
Use this function when you need to rotate a rotary tool by imposing it on the rotation of the main spindle. For example, the function performs tapping at the center of the workpiece with the rotary tool while the workpiece chucked by the main spindle is rotating.
The G164 command specifies the reference spindle and superimposed spindle, and places the two specified spindles in the superimposed status.
The G113 command frees the two spindles from the superimposed status in which they are rotating by the differential rotary tool command.

> **CAUTION**
>
> To specify G164 (differential speed rotary tool function), specify it after the tool has been selected. See <*14.4 Automatic Control of Rotation Direction of Rotary Tool*>.

[Command format]

G164 H□ D±□	Differential rotary tool command
G113	Differential rotary tool cancel command

[Argument]

H□	Specify the reference spindle. 1 is the main spindle, 2 is the back spindle.
D±□	Specify the rotation direction of the axis relative to superimposed and reference spindles. –3: The rotary tool on gang tool post –4: The front rotary tool (end-face drilling tool) on opposite tool post –5: The rotary tool on back spindle

[Sample program]

$1	
M3 S1=3000	
M81 S3=2000	
T1200	
G164 H1 D–3	Differential rotary tool command
M77	
G0 Z–2.0	
G84 X0.0 Z10.0 R1.0 F1.0 D3 S2000 ,R1	Synchronized tapping
G80	
G113	Differential rotary tool cancel command

$2	
M23 S2=5000	
M81 S3=1000	
T5200	
G164 H2 D–3	Differential rotary tool command
M77	
G0 Z–2.0	
G84 X0.0 Z10.0 R1.0 F1.0 D3 S1000 ,R1	Synchronized tapping
G80	
G113	Differential rotary tool cancel command

Front rotary tool on opposite tool post

$2	
M3 S1=3000	
M83 S4=2000	
T2100	
G164 H1 D–4	Differential rotary tool command
M77	
G0 Z–2.0	
G84 X0.0 Z10.0 R1.0 F1.0 D4 S2000 ,R1	Synchronized tapping
G80	
G113	Differential rotary tool cancel command

[Note]

- Be sure to specify the command M77 (spindle synchronization completion queuing) after specifying the command G164.
- The spindle rotating by the differential rotary tool function stops when the machine enters the emergency stop state. The differential rotary tool mode is canceled at the same time.
- Be careful of the maximum spindle speed clamp when specifying the differential rotary function. While the maximum spindle speed clamp command is active, the superimposed spindle is unable to maintain the speed difference specified for the reference spindle.
- Indexing for the reference spindle is not permitted in the differential rotary tool mode.
- To index the reference spindle, cancel the differential rotary tool mode.
- An alarm is issued if the spindle speed is clamped when a synchronized tapping command is executed in the differential rotary tool mode.

  ```
  M181 S5=4000
  T3100
  G4 H2 D-5
  M77
  G0 Z-2.0
  G84 X0.0 Z10.0 R1.0 F1.0 D–5 S4000 ,R1
  G80
  G113
  ```

  In the example above, the speed of the superimposed spindle ranges from 1000 min^{-1} (tapping) to 9000 min^{-1} (tap returning). Since some values exceed clamp speeds of the superimposed spindle (rotary tool on gang tool post): 5000 min^{-1}, an alarm is generated before tapping.
- In the differential rotary tool mode, specification of “,R2” (synchronized tapping phase adjustment) is not permitted.
- The rotation direction of rotary tool may change depending on tool layout or the selected tool number. See *<14.4 Automatic Control of Rotation Direction of Rotary Tool>* for more information.

13.17 Canned Cycle Longitudinal Machining (G90)

When machining a workpiece with simple shape, usually the linear interpolation (G1) function is used. However, if the workpiece has large allowance, the turning fixed canned cycle is used. The following explains the canned cycle longitudinal machining (G90).

[Command format]

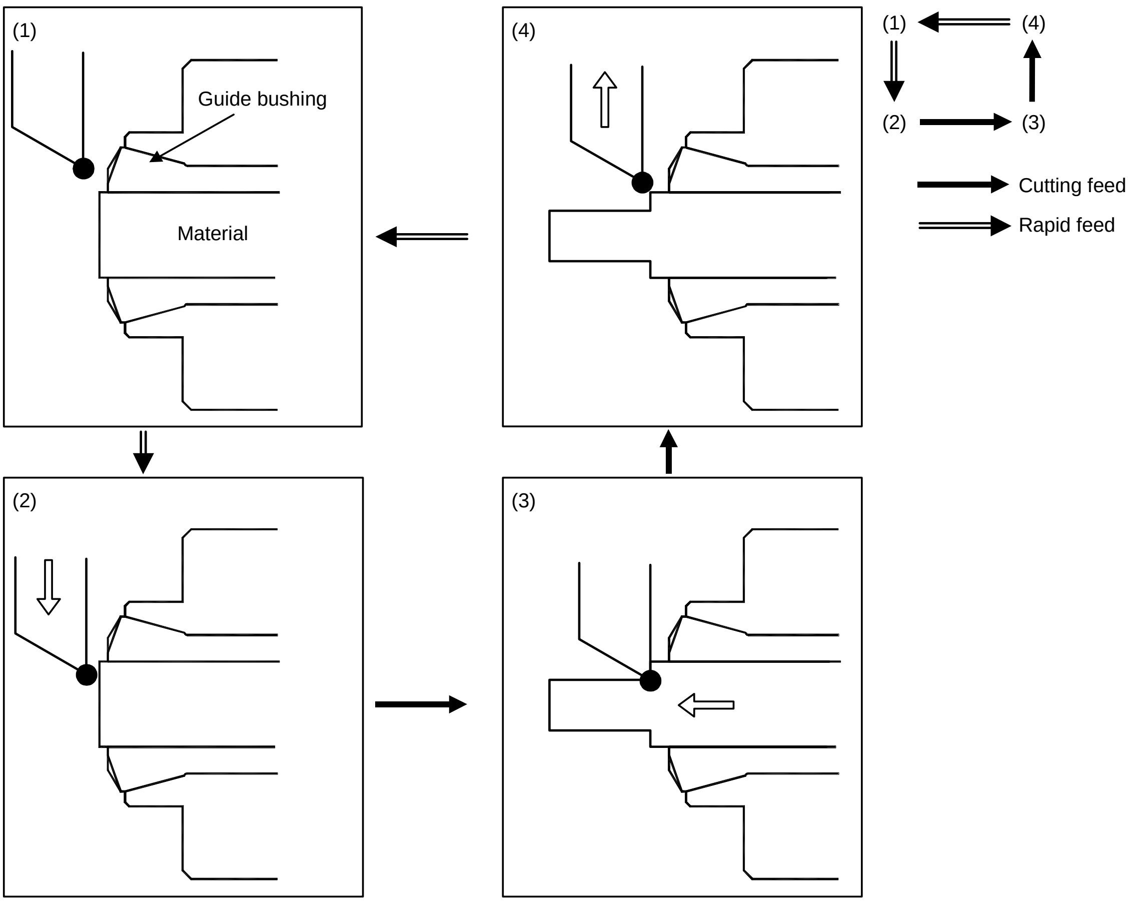
G90 X(U)☐ Z(W)☐ R☐ F☐

[Argument]

X☐	Coordinate value of the end point in the diametric direction (X axis) (Absolute command)
U☐	Coordinate value of the end point in the diametric direction (X axis) (Incremental command)
Z☐	Coordinate value of the end point in the longitudinal direction (Z axis) (Absolute command)
W☐	Coordinate value of the end point in the longitudinal direction (Z axis) (Incremental command)
R☐	Taper amount (in a radial value)
F☐	Feed rate (mm/rev)

Outline of machining

Straight and taper cutting in the longitudinal direction is performed in the cycle as shown below.



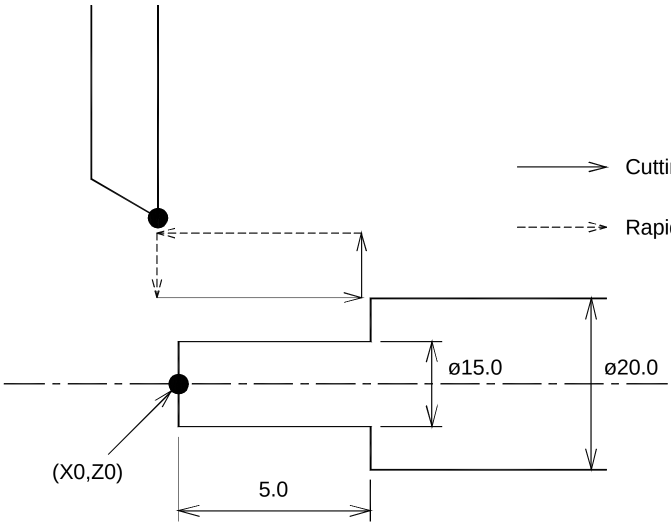
Straight cutting cycle

[Sample program]

Cutting ø20.0 mm to ø15.0 mm material × 5.0 mm long product

Infeed depth: 0.5 mm/infeed

G90 X(U)☐ Z(W)☐ F☐



G99 M3 S1=4000;	Feed per revolution mode, spindle forward rotation, 4000 min^{-1}
T500;	T05 tool selection
G0 X21.0 Z-1.0 T5;	Positioning point, offset
G90 X19.0 Z5.0 F0.1;	1st infeed
X18.0;	2nd infeed
X17.0;	3rd infeed
X16.0;	4th infeed
X15.0;	5th infeed
G0 X21.0 Z-1.0 T0;	Tool returns, offset is canceled

:

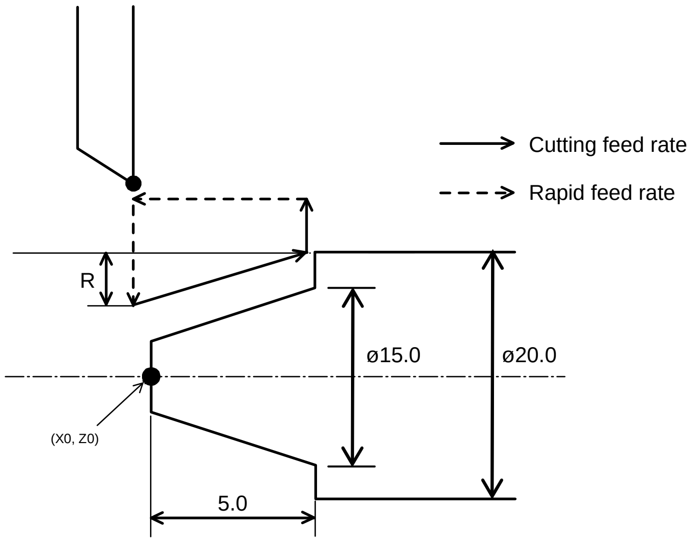
Taper cutting cycle

[Sample program]

Cutting ø20.0 mm material × 5.0 mm long product

Infeed depth: 0.5 mm/infeed

G90 X(U)☐ Z(W)☐ R(I)☐ F☐



G99 M3 S1=4000;	Feed per revolution mode, spindle forward rotation, 4000 min^{-1}
T500;	T05 tool selection
G0 X21.0 Z-1.0 T5;	Positioning point, offset
G90 X20.0 Z5.0 R-1.0 F0.1;	1st infeed
X19.0;	2nd infeed
X18.0;	3rd infeed
X17.0;	4th infeed
X16.0;	5th infeed
X15.0;	6th infeed
G0 X21.0 Z-1.0 T0;	Tool returns, offset is canceled
:	

[Note]

The canned cycle longitudinal machining is canceled by G0 and G1. After the completion of the canned cycle, always specify G0 or G1 in the coordinate value command that specifies tool return operation.

13.18 Thread Cycle Canned Cycle (G92)

This thread cutting cycle machines thread by controlling axis feed and spindle phase.

[Command format]

G92 X☐ Z☐ R☐ Q☐ F☐

[Argument]

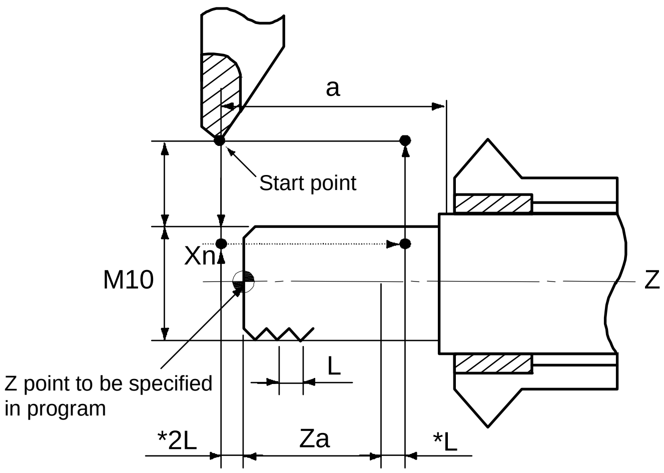
X☐	Specify the thread cutting infeed position in the X axis direction.
Z☐	Specify the thread cutting infeed position in the Z axis direction.
R☐	Specify the taper size (r) for taper thread cycle canned cycle.
Q☐	Enter the Q argument to specify thread cutting start shift angle (0.001 - 360.000 degrees)
F☐	Specify a screw pitch.

Example of machining with M10 P=1.5

Machining a non-ferrous material (right-hand thread)

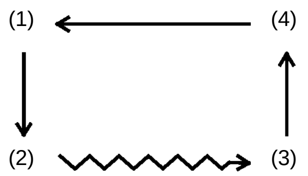
G92 X☐ Z☐ F☐

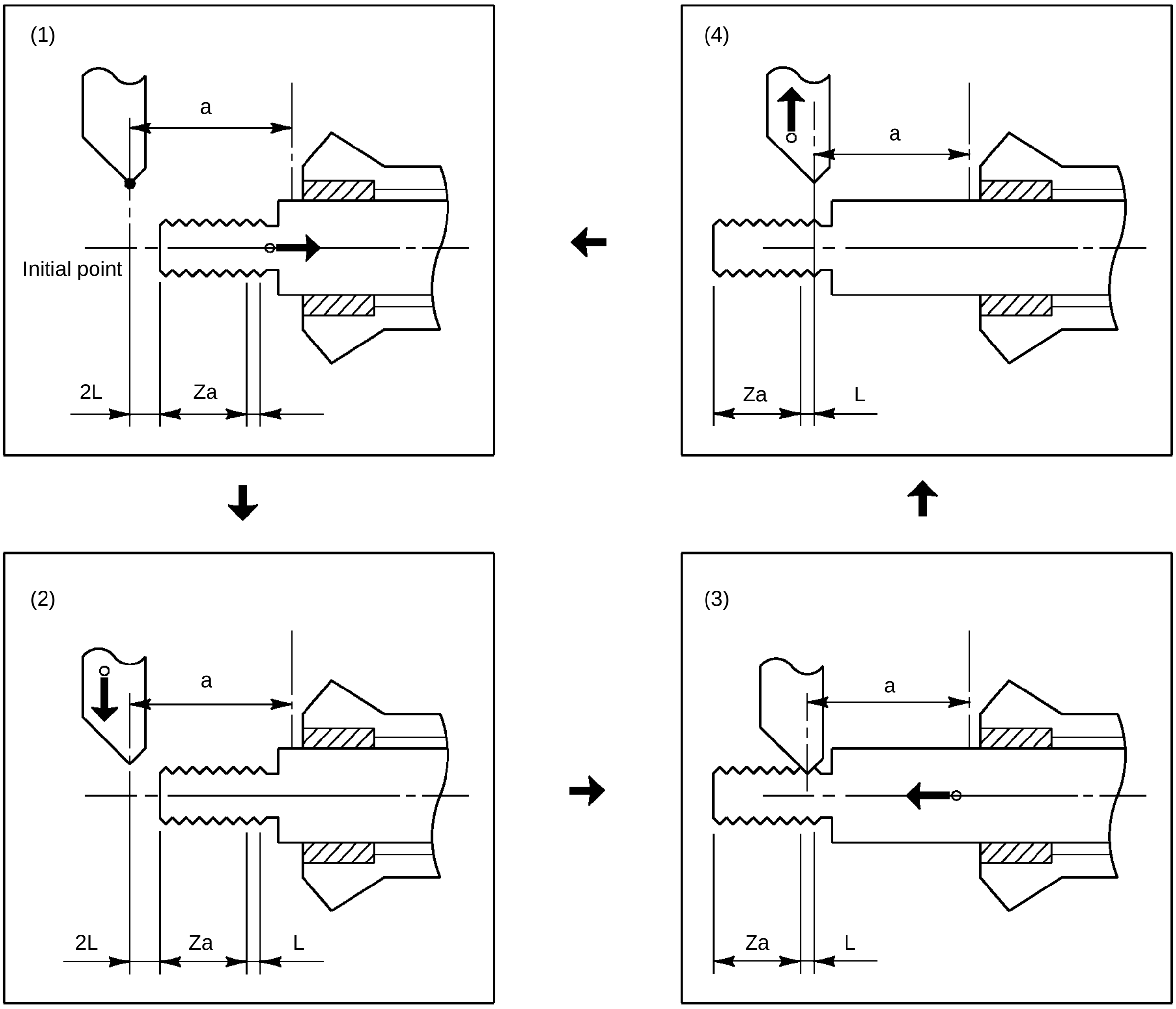
L (1.5 mm): Thread lead
Za (12 mm): Effective length
a (10.5 mm): Tool shift
Xn: In-feed
* Uneven thread ridge portion
For the details on the in-feed, see <9.3 Thread Cutting Count> in [Introduction] of the Instruction Manual (Optional).



Outline of machining

Perform threading with a tool in the following cycle:





[Sample program]

T0300	Selects the T03 tool
G50 W-10.5	
M03 S1=1500	Specifies spindle forward rotation at 1500 min^{-1}
G4 U1.0	Makes the machine dwell for 1.0 second for stable rotation.
G00 X13.0 Z-3.0 T□	Positions the tool at the initial point (Z = 2L)
G92 X9.3 Z15.0 F1.5	1st in-feed (X1 0.35) (Z = Za) (F = L)
X8.8	2nd in-feed (X2 0.25)
X8.48	3rd in-feed (X3 0.16)
X8.28	4th in-feed (X4 0.1)
X8.16	5th in-feed (X5 0.06)
X8.06	6th in-feed (X6 0.05)
X8.06	0 cut (same as final in-feed)
G00 X□ Z□	To initial point of next process
G50 W10.5	

- The spindle speed is limited as specified by the following equation:

$$N\ (\mathrm{min}^{-1}) \leqq \frac{8000\ (\mathrm{mm/min})}{L\ (\mathrm{mm})}$$

N: Main spindle speed

L: Thread lead (mm)

8000: Maximum feed rate

- The standard spindle speed for thread cutting is 500 to 2000 min^{-1}.
- Uneven thread ridges are produced at the threading start and end points due to the delay of the servo system. The length of uneven thread portions at a spindle speed of 1,500 min^{-1} is approximately two times of the thread lead length (2L) at the entry area, and the thread lead length (L) at the exit area. This uneven thread ridges, length becomes shorter as the spindle speed lowers.
- When selecting right-handed or left-handed threading, reverse the rotation of the main spindle either by changing the Z-axis start position or by using the reverse holder.
- For chamfering (round-up on the screw trailing end), 0 to 89 degrees can be set using a parameter. (See the Instruction Manual of the NC manufacturer.)

[Note]

- For the details on the in-feed, see <*9.3 Thread Cutting Count*> in [Introduction] of the Instruction Manual (Optional).
- In the G92 thread cutting programming, if EOB (;) is specified for an infeed command, the infeed command in the preceding block is used as modal command.

```
G92 X11.0 Z□ F□;
X10.0;
X9.0;
; (Thread cutting at X9.0)
; (Thread cutting at X9.0)
G0 □;
```

G92-Taper thread cutting canned cycle

G92 can be used for taper thread cutting.

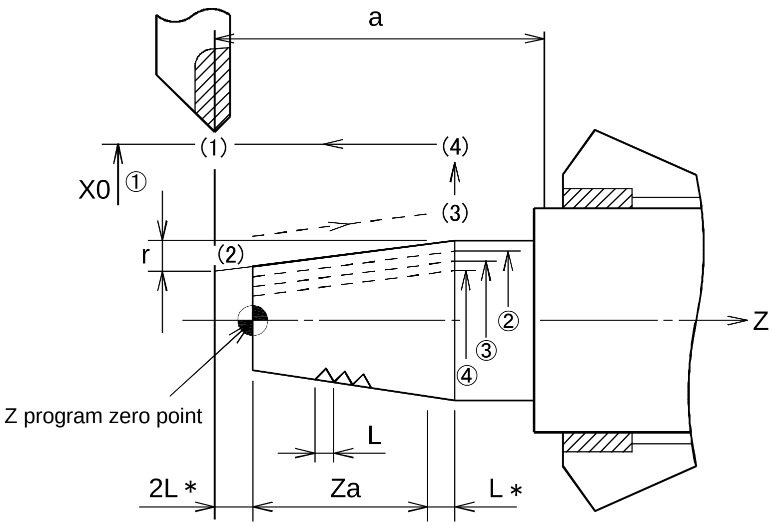
Taper thread cutting is performed in a canned cycle from (1) to (2), (3) to (4), then back to (1). The coordinate reference point is (3).

The sign (+ or –) of the slope "r" indicates the direction of the location of the start point from the end point.

Outline of machining

G92 X☐ Z☐ R $\boxed{r}$ F☐

L: Thread lead
Za: Effective length
a: Tool shift
r: Slope
* Uneven thread ridge portion



[Sample program]

T☐	Select threading tool
M☐ S1=☐	Command for spindle forward or reverse rotation at a spindle speed of ☐ min^{-1}
G4 U1.0	1.0 second dwell for stabilize rotation
G00 X $\boxed{①}$ Z☐ T☐	Positioning tool at initial point. X (thread diameter + 2L), Z (a –2L)
G92 X $\boxed{②}$ Z☐ R $\boxed{–r}$ F☐	1st in-feed
X (③)	2nd in-feed
X (④)	3rd in-feed
X (④)	Zero cut
G00 X☐ Z☐	Move to initial point of next process. (Threading cycle OFF)

13.19 Link-Thread Machining

With a machine equipped with guide bushing, if the thread length in Z direction is longer than the hold length of inner diameter of guide bushing, the workpiece becomes apart from the guide bushing when it returns to the thread positioning point. Thus, the machining is discontinued. In such a case, repeat a cycle of "outer diameter turning to thread cutting" several times to link the thread to subsequent threads. It enables thread cutting in Z direction without being broken.

This function performs thread cutting by controlling the feed and the phase of main spindle rotation. Using this function enables link-thread cutting by specifying the link position in the same Z direction.

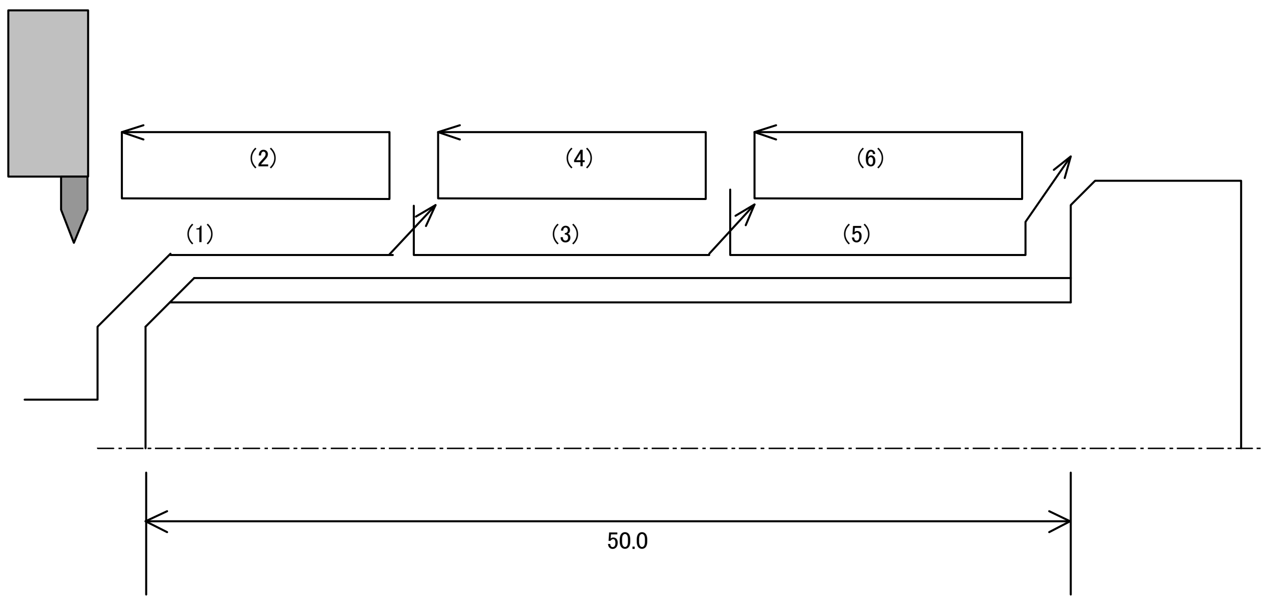
[Sample program]

Hold length of guide bushing inner diameter: 25.0 mm

Material diameter ø20.0

Thread shift amount 1.5 mm

M12 × pitch 1.5



```
T300 (Front turning (1))
G0 X21.0 Z–0.5 T3
G1 X9.0 F0.1
G1 X12.0 Z1.0 F0.05
Z19.0 F0.1
X18.0
X21.0 W1.5
   :
T400 (Thread cutting (2))
G50 W–1.5
G0 X21.0 Z–1.5 T4
M98 P0007 ................................ To subprogram
G0 X21.0 Z–1.5 T0
G50 W1.5

T300 (Front turning (3))
G0 X21.0 Z18.0 T3
G1 X14.0 F0.1
G3 X12.0 Z19.0 R1.0 F0.05
G1 Z33.0 F0.1
X18.0
X21.0 W1.5
   :
T400 (Thread cutting (4))
G50 W-1.5
G0 X21.0 Z12.0 T4 ..................
```

Z: Distance from the first threading position multiple of pitches
First threading position Z –1.5 to Z12.0
13.5 mm
13.5 / 1.5 = 9 Divided out

```
M98 P0007
G0 X21.0 Z12.0 T0
G50 W1.5

T300 (Front turning (5))
G0 X21.0 Z32.0 T3
G1 X14.0 F0.1
G3 X12.0 Z33.0 R1.0 F0.05
G1 Z47.0 F0.1
X18.0
X21.0 W1.5
   :
T400 (Thread cutting (6))
G50 W–1.5
G0 X21.0 Z25.5 T4
M98 P0007 ............................ To subprogram
G0 X21.0 Z25.5 T0
G50 W1.5
   :
   :
```

Subprogram

```
O0007
G0 X14.3
G32 X11.3 W1.5 F1.5
W15.0 (*)
U3.0 W1.5
G0 W–18.0
X13.8

G32 X10.8 W1.5 F1.5
W15.0
U3.0 W1.5
G0 W–18.0
X13.48

G32 X10.48 W1.5 F1.5
W15.0
U3.0 W1.5
G0 W–18.0
X13.28

G32 X10.28 W1.5 F1.5
W15.0
U3.0 W1.5
G0 W–18.0
X13.16

G32 X10.16 W1.5 F1.5
W15.0
U3.0 W1.5
G0 W–18.0
X13.06

G32 X10.06 W1.5 F1.5
W15.0
U3.0 W1.5
G0 W–18.0
X13.06

G32 X10.06 W1.5 F1.5
W15.0
U3.0 W1.5
G0 W–18.0

M99
```

[Note]

- The program is an example for the reference. Specify an appropriate depth of cut and the cutting condition, according to the workpiece and the tool.
- Specify the thread length so that the workpiece will not be apart from the guide bushing. Note that the length to be held by the guide bushing depends on the material diameter.
- Be sure to deburr from the workpiece diameter at the end of front turning. If the burr is remained, the workiece cannot be returned into the guide bushing. It may cause an accident and the pitch may be shifted.
- Determine the Z coordinate position to connect the second and the subsequent screws by calculating the distance (the first threading point + multiple of pitch). The length of thread cutting (shown by (*) in the sample program) must be a multiple of pitch. If these values are inaccurate, a pitch shift may occur.
- If the chuck opens during thread cutting, the shift phase of thread occurs and it causes link-thread cutting to fail. Be sure to perform link-thread cutting within one chuck.
- The program is an example when machining a thread of 1L. Take 2L or more margin for first and the final thread cuttings.
- It is recommended that the second and the subsequent infeed and the final thread cutting is finished at 45°.

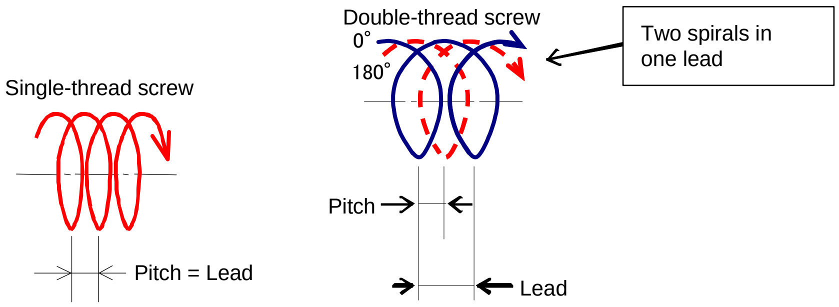
13.20 Multi-thread Screw Cutting

Multi-thread screw

Generally used screw is called as a single-thread screw, which has a spiral in one pitch, and advances only one pitch per one revolution. With the single-thread screw, the lead (distance per one rotation) is equal to the pitch. In contrary, the screw that has two or three spirals within one lead is called multi-thread screw. With the multi-thread screw, the lead is multiple of pitches.

Pitch × Number of threads = Lead

Pitch 0.75 × double-thread screw = Lead 1.5



[Sample program]

The G32 command performs thread cutting by controlling the feed and the phase of main spindle rotation.

The position of spindle rotation direction where the infeed starts with G32 command is always the same. Define this position (angle) as 0°. If the infeed of first thread starts from 0°, start the infeed of second thread from the position shifted by 180°. This can make the multi-thread screw.

* See <*13.11 Thread Cutting (Equal Pitch Thread Cutting and Continuous Thread Cutting)*> for command format.

Material diameter	ø20.0
Thread shift amount	1.5 mm

Machining of double-thread screw of M12 × Pitch 0.75 (Lead = 1.5)

Front turning process

```
:
:
:
T300 (Multi-thread cutting)
G50 W–1.5
G0 X21.0 Z–3.0 T3
X14.0
M98 P0077          To subprogram 1
G0 X14.0 Z–3.0
M98 P0777          To subprogram 2
G0 X21.0 Z–3.0 T0
G50 W1.5
:
:
:
```

Subprogram 1

```
O0077
G0 X11.76
G32 W15.0 F1.5 Q0
U5.196 W1.5 F1.5
G0 W–16.5
:
G0 X11.56
G32 W15.0 F1.5 Q0
U5.196 W1.5 F1.5
G0 W–16.5
:
G0 X11.36
G32 W15.0 F1.5 Q0
U5.196 W1.5 F1.5
G0 W–16.5
:
G0 X11.2
G32 W15.0 F1.5 Q0
U5.196 W1.5 F1.5
G0 W–16.5
:
G0 X11.1
G32 W15.0 F1.5 Q0
U5.196 W1.5 F1.5
G0 W–16.5
:
G0 X11.1
G32 W15.0 F1.5 Q0
U5.196 W1.5 F1.5
G0 W–16.5
:
M99
```

Subprogram 2

```
O0777
G0 X11.76
G32 W15.0 F1.5 Q180000
U5.196 W1.5 F1.5
G0 W–16.5
:
G0 X11.56
G32 W15.0 F1.5 Q180000
U5.196 W1.5 F1.5
G0 W–16.5
:
G0 X11.36
G32 W15.0 F1.5 Q180000
U5.196 W1.5 F1.5
G0 W–16.5
:
G0 X11.2
G32 W15.0 F1.5 Q180000
U5.196 W1.5 F1.5
G0 W–16.5
:
G0 X11.1
G32 W15.0 F1.5 Q180000
U5.196 W1.5 F1.5
G0 W–16.5
:
G0 X11.1
G32 W15.0 F1.5 Q180000
U5.196 W1.5 F1.5
G0 W–16.5
:
M99
```

By applying this program, change the infeed start angle to make triple- or quad-thread screw.

[Note]

- The program is an example for the reference. Specify an appropriate depth of cut and the cutting condition, according to the workpiece and the tool.
- Specify the thread length so that the workpiece will not be apart from the guide bushing.
- Be sure to deburr from the workpiece diameter at the end of front turning. If the burr is remained, the workiece cannot be returned into the guide bushing. It may cause an accident and the pitch may be shifted.
- If the chuck opens during thread cutting, the shift phase of thread occurs and it causes multi-thread cutting to fail. Be sure to perform multi-thread cutting within one chuck.
- Consider the lead angle to perform high-lead threading, and select an appropriate tool.
- For the high-lead threading, specify the cutting condition not to exceed the maximum feedrate.

13.21 Machine Coordinate System Command (G53)

The machine coordinate system command (G53) function specifies the positioning point in the machine coordinate system.

[Command format]

G53 X☐ Z☐ Y☐ ,F☐

X: Specify the X axis coordinate in the machine coordinate system. The axis in the specified axis control group moves to the machine coordinate specified by X.

Z: Specify the Z axis coordinate in the machine coordinate system. The axis in the specified axis control group moves to the machine coordinate specified by Z.

Y: Specify the Y axis coordinate in the machine coordinate system. The axis in the specified axis control group moves to the machine coordinate specified by Y.

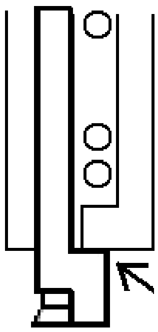
,F: Specify the feed rate. If this argument is omitted, the axis moves in rapid feed rate.

13.22 Y-axis Holder

The Y-axis holder performs cutting in Y direction. The cutting chips are dropped naturally from the cutting position, and effective for the chip removal.

[Procedure]

1. Press the Preparation key to display the Preparation screen.
2. Mount the Y-axis holder while pushing its shoulder part to the gang tool holder.



3. Move the cursor to "Tool Typ" of the tool for which the Y-axis holder is mounted and set "L Y" for the tool name.

   If the "Tool Type" was "12R", change it to "12L Y".

   *0 Usually the tool name is "R". By changing this to "L Y", the tool is positioned at the position shown in Figure A or Figure B when "DIA" or "Core" is executed.
4. Press the menu key [DIA], and press the Start key several times until the cursor reaches the position "DIA". Then, align the outer diameter of the material with the tool nose. (Figure A)
5. Move the tool away from the material, and move the material backward so that it is placed behind the end-face of guide bushing.
6. With the menu key [Core DWN] being selected, press the Start key continuously until the cursor reaches the position "Core DWN". Then, align the outer diameter of the material with the tool nose. (Figure B)

   * When adjusting the position by "Core DWN", pay attention to interference between the tool at left and the material.
7. Move the tool away from the material, and move the material backward so that it is placed behind the end-face of guide bushing.
8. Repeat Step 5. Now the setting completes.

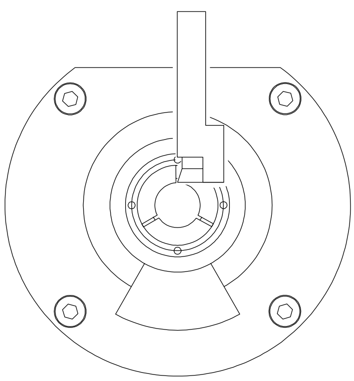


Figure A

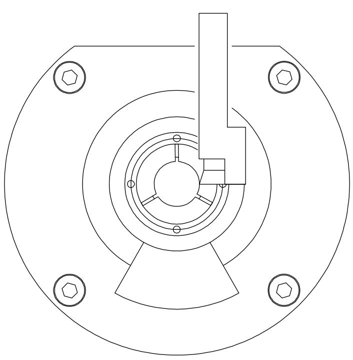


Figure B

[Sample program]   Material dia.: 20 mm

```
T500    (Front turning) ;
G0 X21.0 Z0 T5 ;
G1 X-0.5 F0.1 ;
X18.0 ;
Z15.0 ;
X19.2 ;
X20.2 W0.5 ;
G0 X21.0 W0 T0 ;
T300 (Y axis back turning)      ← Moves to the position shown in Figure A.
G50 W-3.0 ;
G0 Y21.0 Z20.0 T3 ;
X0.0 ;                          ← Moves to the position shown in Figure B.
G1 Y16.0 F0.05 ;                ← Cutting is performed in Y direction.
Z30.0 ;
Y15.4 W0.3 ;
Y21.0 ;
X21.0 ;
G0 U0 V0 W0 T0
G50 W3.0 ;
```

[Note]

- The Y-axis holder size differs according to the machine model. Select the holder meeting the machine model.
- Do not mount a Y-axis holder next to a rotary tool. If mounted, the tool may interfere with the material during machining depending on the material diameter.

13.23 Multi-piece Machining

In ordinary operation, material is re-chucked after machining of one workpiece is completed. In the multi-piece machining, however, multiple workpieces are machined continuously in single material chucking operation.

[Note]

Machining data

Material diameter	10.0 mm
Machining length	27.6 mm Total workpiece length + Cut-off tool width + Shift amount for end face turning + Margin = 25.0 + 2.0 + 0.1 + 0.5
Number of workpieces/chucking	3 pcs.
Back spindle chuck position	20.0 mm

Setting the machining data

After setting the machining length of a workpiece to "Machining Length" on the Machining Data screen, set the number of workpieces to be machined in one chucking to "Pieces/1Chuck".

Main Program

O0002

$1		$2	
G50 Z0			
M6			
G99			
!2L15		!1L15	
M98 P5 L3	Calling subprogram O0005 Number of repetitive calling times: 3	M98 P5 L3	Same as $1
!2L20		!1L20	
M8			
M8			
/M98 P7000	Material change program		
M9			
M5			
M7			
G0 X-3.0 W-$\boxed{81.3}$* T00	Move to the starting point. W value = (Total workpiece length + Cut-off tool width + Shift amount for end face turning) × (Number of products / chucking)		
G999		G999	
N999		N999	
M2		M2	
M99		M99	

Subprogram

O0005

G50 Z-0.1	Shift amount for end face turning: 0.1 mm	
M9		
M6		
G113		
G0 X11.0 Z-1.0		
:		:
:		:
[Front machining]		[Back machining]
:		:
:		:
		T3000
!2L1		!1L1
M24 S2=2500 M3 S1=2500		
G114.1 H1D-2		
T100 (CUT-OFF)		
G0 X11.0 Z27.0 T1	Z27.0 = Total workpiece length + Cut-off tool width	
G650		G650
		G0 Z-1.0
		M72
		G98 G1 Z5.0 F2000
		M77
		G4 U0.3
		M15
		M73
		G4 U0.2
!2L10		!1L10
G99 G1 X-1.0 F0.03		
G600		G600
G98 G1 X-3.0 F0.05		M25
G113		
M5		
M56		
M99		M99

[Note]

- The Z1 axis starts retracting from the forward end position by "Machining Length × Pieces/1Chuck" when the start point operation is executed.
- When 1-cycle operation completes, make sure that the start point is identical to the end point by executing the start point operation.
  If they do not match, over-travel or interference may occur.

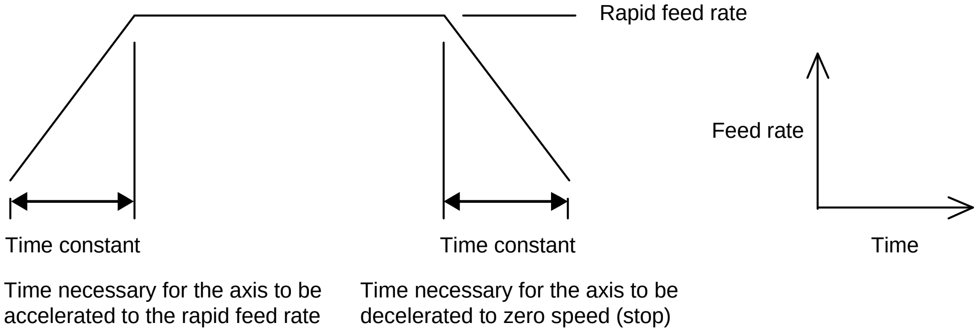
13.24 Rapid Feed Acceleration/Deceleration Time Constant Setting ON/OFF (M360/M361)

While running a program, rapid feed acceleration/deceleration time constant of individual axis (X1, Z1 and Y1 axes) can be changed. This feature is especially effective in G630 front/back parallel machining. By changing the acceleration/deceleration time constant of the gang tool post (X1 and Y1 axes) the workpiece can be finished at high accuracy in back machining.

To the contrary, rapid feed acceleration/deceleration will be slowed in tool selection for the gang tool post when the acceleration/deceleration time constant is changed, causing the cycle time to be increased.

About rapid feed acceleration/deceleration time constant

An axis for which rapid feed (G0) is specified is accelerated to the rapid feed rate at a constant slope and then decelerated at a constant slope to stop at the target point of positioning. The duration in which the axis is accelerated or decelerated is called the acceleration/deceleration time constant. Note that the acceleration time and deceleration time are the same time.



Initial Value of Rapid Feed Acceleration/Deceleration Time Constant of Each Axis

Axis Name	Acceleration/Deceleration Time Constant
X1	45 msec
Z1	100 msec
Y1	100 msec
X2	120 msec
Z2	90 msec
Y2	60 msec

[Command format]

M360 X☐ Z☐ Y☐ Rapid feed acceleration/deceleration time constant setting ON

X Specify an integral multiple of the initial value of X1 axis acceleration/deceleration time constant. Specify a value with a decimal point. Specification range: 1 to 6

Z Specify an integral multiple of the initial value of Z1 axis acceleration/deceleration time constant. Specify a value with a decimal point. Specification range: 1 to 6

Y Specify an integral multiple of the initial value of Y1 axis acceleration/deceleration time constant. Specify a value with a decimal point. Specification range: 1 to 6

M361 Rapid feed acceleration/deceleration time constant setting OFF

The acceleration/deceleration time constant of the X1, Z1 and Y1 axes is reset to the initial value.

When execution of a program is terminated halfway, it is possible to reset the acceleration/deceleration time constants to the initial values by pressing the Reset key.

[Sample program]

In the program below, M361 and M360 are specified before and after the tool selection command in $1 in the operation where back finish machining in $2 and tool selection in $1 are overlapped. Tool selection operation in $1 is performed at a restricted speed and finishing accuracy in back machining is secured.

$1		$2
:		
:		
M360 X4.0 Y4.0	Multiplies the X1 and Y1 axis acceleration/deceleration time constant by four	:
T300...	Select the tool	Back finish machining
M361...	Resets the X1 and Y1 axis acceleration/deceleration time constant to the initial values	:
:		
:		

[Note]

- Specify axis control group 1 ($1) as rapid feed acceleration/deceleration time constant setting (M360/M361) for X1, Z1 and Y1 axes and specify axis control group 2 ($2) as rapid feed acceleration/deceleration time constant setting for (M360/M361).
- Changing the acceleration/deceleration time constants may increase a cycle time.
- Determine the multiplication value for changing the acceleration/deceleration time constants according to the machining accuracy required in the back machining.
- To achieve better finishing accuracy in back machining, use M350/M351 (rapid feed rate setting ON/OFF) together with M360/M361.
  See *<13.26 Rapid Feed Rate Setting ON/OFF (M350/M351)>* for the use of M350/M351.
- M360/M361 cannot be specified in the superimposition (G620 or G650) mode.

13.25 Medium-pressure Coolant Device

Medium-pressure coolant is available by opening the valve to be used and operating the separately installed pump. This is effective to remove chips by supplying powerful coolant.

[Command format]

Trochoid pump (separately installed pump)

M452	Trochoid pump ON
M453	Trochoid pump OFF

Valves

M430	Back spindle oil blow ON
M431	Back spindle oil blow OFF
M432	Opposite tool post ON
M433	Opposite tool post OFF
M434	Back tool post ON
M435	Back tool post OFF
M436	Guide bushing or Ceiling ON
M437	Guide bushing or Ceiling OFF

[Note]

- Open any of the valves before turning on the trochoid pump.
- If the valve is closed immediately after stopping the trochoid pump, pressure remains in the coolant line. Therefore, make a program so that the valve is closed allowing a certain interval after stopping the trochoid pump.
- At the end of the program, open all valves to release internal pressure. If residual internal pressure remains, the pump motor or a valve may be damaged.
- The M codes described above show the position to where the coolant is to be discharged when U74R (medium-pressure coolant device, optional) is installed. The position may differ depending on customer's requirement.

13.26 Rapid Feed Rate Setting ON/OFF (M350/M351)

The standard rapid feed rate for each axis is set to 32 m/min for the X1, Y1, Z1, X2 and Z2 axes and 8 m/min is set for the Y2 axis. Use this to change the feed rate of parts that operate at G0 (feed rate) when a canned cycle, etc. is used.

[Command format]

M350 X☐ Y☐ Z☐	Rapid feed rate setting ON
M351	Rapid feed rate setting OFF (Restores the standard rapid feed rate)

[Argument]

When specifying the function in axis control group 1 ($1)

- X☐ Specify the rapid feed rate of the X1 axis. Specification range: 0 to 32000 mm/min
- Y☐ Specify the rapid feed rate of the Y1 axis. Specification range: 0 to 32000 mm/min
- Z☐ Specify the rapid feed rate of the Z1 axis. Specification range: 0 to 32000 mm/min

When specifying the function in axis control group 1 ($2)

- X☐ Specify the rapid feed rate of the X2 axis. Specification range: 0 to 32000 mm/min
- Z☐ Specify the rapid feed rate of the Z2 axis. Specification range: 0 to 32000 mm/min

[Sample program]

$1	
:	
:	
M350 Z18000	Sets the Z1 axis rapid feed rate at 18000 mm/min (18 m/min)
T1200	
G0 Z-1.0	
G79 Z20.0 I6.0 K3.0 A0.5 J500 F0.1 T12	The Z1 axis operates at 18 m/min until M351 is specified
G80	
G0 Z-1.0 T0	
M351	Turns off the rapid feed rate setting function for the Z1 axis and restores the standard rapid feed rate
:	
:	

[Note]

- After turning on the rapid feed rate setting function by specifying M350, make sure to turn the function off by specifying M315 when the function is no more necessary. If the function is not turned off, the axis moves at the specified rapid feed rate in tool selection and the G0 mode, causing the operation to be executed slower than usual. There are cases the cycle time is considerably increased.
- Pressing the Reset key restores the standard rapid feed rate.
- M350/M351 cannot be specified in the superimposition (G620 or G650) mode.

13.27 Cutting Start Interlock Enabled/Disabled (M86, M87)

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.

[Command format]

M86 Cutting start interlock enabled
(The cutting start interlock of a specified axis control group is set back to original.)

M87 Cutting start interlock disabled (All the cutting start interlocks of a specified axis control group are disabled.)

Relationship between the machining patterns and cutting start interlocks:

Machining pattern	S1					S2				
	S1	S2	S3	S4	S5	S1	S2	S3	S4	S5
G600	√		√	√			√			√
G610	√			√						
G620	√		√			√			√	
G630	√		√				√			√
G650	Depending on the previous machining pattern									
G660	√		√				√	√		

√: Cutting start interlock enabled

[Note]

- Cutting start interlock function may not work correctly if the cutting feed is specified immediately after the spindle or rotary tool rotation command, even if this function is enabled. In this case, insert two or mroe G4 codes between the rotation command and the cutting block.

13.28 Error Detect ON/OFF (M92, M93)

Use this command when higher precision in edge processing is required. If the error detect is enabled (ON), cutting feed proceeds to the next block after confirming the speed is completely decelerated. Thus, the desired edge precision can be obtained.

[Command format]

M92	Error detect ON
M93	Error detect OFF

[Sample program]

$1
:
T0200 (Front turning)
G0 X11.0 Z-1.0 T02
X6.0
M92 (ON)
G1 Z0 F0.05
X7.0
X8.0 Z0.5 F0.03
Z10.0 F0.05
X10.0
G0 X11.0 T00
M93 (OFF)
:

[Note]

- Error detect function is disabled at power on (error detect OFF (M93)).
- Error detect function is effective only when G1, G2, or G3 command is specified.
- Difference from the exact stop check (G9) function

  The exact stop check function is enabled only in the block where G9 is specified. Meanwhile, the error detect ON (M92) is enabled until the error detect OFF (M93) is specified.
- When the error detect function is enabled, the longer cycle time is required than that in disabled state because the completion of cutting feed is prolonged.

13.29 Start Position Queuing (Type 1) (G115)

[Command format]

!L☐ G115 X☐ Z☐ C☐

Operation example

- This command specifies a start position to provide a queuing point halfway through a block.

(1) When queuing is specified in the program for an axis control group (local axis control group), the other axis control group (remote axis control group) starts moving first.

(2) The local axis control group starts moving when the remote axis control group reaches the specified start position.

(3) If the start position specified by G115 does not exist on the next-block moving path of the remote axis control group, the local axis control group starts moving when the remote axis control group reaches all axis coordinate values of the commanded position.

(4) The start position check applies only to the axis specified by G115.

(5) If the start position is not found when the remote axis control group moves to the next block, the local axis control group waits until the remote axis control group reaches the start position by moving beyond the next block.

(6) If the two axis control groups overlap by the G115 command, they remain queued.

(7) When specifying the start position, use the work coordinate values of the remote axis control group.

(8) Specifying G115 for more than three axis control groups results in the program error P33.

(9) The G115 block is not subject to single-block stop.

(10) If two or more G115 blocks are specified consecutively, the block specified last is valid.

(11) The addresses that follow G115 are the X-axis, Z-axis, and C-axis workpiece coordinates.

13.30 Start Position Queuing (Type 2) (G116)

[Command format]

!L☐ G116 X☐ Z☐ C☐

Operation example

- When queuing is specified in the program for an axis control group (local axis control group), the local axis control group starts moving first.

(1) The other axis control group (remote axis control group) starts moving when the local axis control group reaches the specified start position.

(2) If the start position specified by G116 does not exist on the next-block moving path of the local axis control group, the remote axis control group starts moving when the local axis control group reaches all axis coordinate values of the commanded position.

(3) The start position check applies only to the axis specified by G116.

(4) If the start position is not found when the local axis control group moves to the next block, the program error P33 occurs before the local axis control group starts moving.

(5) If the two axis control groups overlap by the G116 command, they remain stopped.

(6) When specifying the start position, use the work coordinate values of the remote axis control group.

(7) If G116 is specified for more than three axis control groups, multiple remote axis control groups start moving simultaneously.

(8) The G116 block is not subject to single-block stop.

(9) If two or more G116 blocks are specified consecutively, the block specified last is valid.

(10) The addresses that follow G116 are the X-axis, Z-axis, and C-axis work coordinates.

13.31 Auxiliary Function Output during Axis Feed (G117)

[Command format]

G117 X□ Z□ C□ □□□□

Auxiliary function

Operation example

This command specifies an intermediate point and the auxiliary function to be output at that point, allowing the auxiliary function to be executed during movement.

(1) This command is placed independently immediately before the moving command block in which you want the auxiliary function to be executed.

(2) This command is not subject to single-block stop.

(3) In the G117 block, auxiliary functions can be specified within the limit indicated below.
M command: 4 sets (M code used for macro: No more than 2 sets)
S command: 1 set each
T command: 1 set

(4) This command can be specified for up to two consecutive blocks.
If three blocks or more are specified consecutively, the last two blocks are valid.

(5) If the operation start position specified by G117 does not exist on the moving path, the auxiliary function is output when all axis coordinate values of the operation start position are reached. Only the specified axes are checked.

(6) At the operation start position, after checking that the previous auxiliary function has been output, the next auxiliary function is output. The PC interface can be used as usually without modification.

(7) The auxiliary function specified along with a moving command block is output before movement is started. Movement does not stop at the operation start position. Note that, at the block end position, after checking that all auxiliary functions have been output, the next block is executed.

(8) G117 should be specified in the order of operation start positions. The program error P33 occurs if the order of operation start positions is opposite to the moving sequence.
When the operation start positions match the moving sequence, auxiliary functions are output in the order in which they are specified.

(9) If the operation start position for the next block is not found, the program error P33 occurs before moving to the next block.

(10) The descriptions (8) to (9) above can be summarized as in the following table:

First block / Second block	Intermediate point found during movement	No intermediate point during movement
Intermediate point found during movement	The order in (8) is followed.	Program error occurs as in (8).
No intermediate point during movement	The order in (9) is followed for the second block.	The order in (9) is followed. Auxiliary functions are output in the order of the first block, then the second block regardless of the order of the specified points.

(11) The addresses that follow G117 are the X-axis, Z-axis, and C-axis work coordinates.

13.32 Arbitrary Axis Change (G140)

Use the G140 command to declare the axis you want to use.

Use the G140 command to declare the axes you want to superimpose on each other.

[Command format]

G140 X=X1 Z=Z1 Y=Y1	The command specifies the axes to be used in the subsequent operations as follows: X=X1, Z=Z1, Y=Y1
G140 X=X1 Z=Z2 Y=Y1	The command specifies the axes to be used in the subsequent operations as follows: X=X1, Z=Z2, Y=Y1
Gang tool post:	X1 Z1 Y1 C1 (Z1: Main headstock, C1: Main spindle C axis)
Opposite tool post:	X2 Z2 C2 (C2: Back spindle C axis) Y2

[Sample program]

$1		$2	
G600	Machining pattern cancel	G600	Machining pattern cancel
T0800			
:			
:			
!2L1		!1L1	
		G4	The G4 command is executed to prevent the read ahead of G140 while $1 is using the X1 axis and Y1 axis
		G140 X=X1 Z=Z2 Y=Y1	The arbitrary axis change (G140) command is executed after $1 finishes using the X1 axis and Y1 axis
		:	
		G140 X=_ Z=_ Y=_;	Must be returned to the previous axis.
		:	

[Note]

Before the arbitrary axis change (G140) command can be executed to declare the number of an axis being used by another axis control group, the queuing command must be executed for the axis control group to finish using the axis. At this time, the G4 command must also be executed to prevent the read ahead of G140 after the queuing command.

CAUTION

Be careful when using the G140 command (arbitrary axis change). If the G140 command is specified to declare an axis that is being used by another axis control group, the machine may encounter an interference problem.

13.33 End Position Specified Queuing (G149)

This command enables queuing between an arbitrary position in an arbitrary block in the local axis control group and an arbitrary block position in the remote axis control group.

While the standard queuing function adjusts the start timing between the queuing axis control groups, this function adjusts the end timing of the end block specified by the block ID number.

[Command format]

!L☐ G149 Q☐ X☐ Z☐ C☐ End position specified queuing

Q represents the end position block number. It is specified by BN☐.

[Sample program]

$1	$2
G98 G0 X1.1	
!2 G149 Q100 X1.0 .... Queuing block specification	!1 Q120 .... Queuing block specification
.... Queuing time	.... Queuing time
G1 X5.0 F50	
BN100 G1 X–1.0 F10	BN120 G1 Z50.0 F100

With this example, block BN120 in $2 is completed when the X axis in $1 passes a point of 1.0 in block BN100.

(1) The end position queuing position must be specified in the work coordinate system.

(2) Only the axis on which a queuing position has been specified is checked for its passing the end position.

(3) If the queuing position is specified on multiple axes, the queuing time is the time by which all the specified axes have passed the specified point.

(4) If the specified axis does not pass the point specified in the block of the specified end position block number, a program error (P33 format error) occurs when the G149 command is issued.

(5) If the end position block number of the G149-specified axis control group is not found within 10 blocks from the end position specified queuing command or if no P code has been specified, the required time is calculated assuming the moving block in which all the specified axes has passed the specified point as the end position specification block. Note, however, that a program error (P700BN No number) occurs if the moving block is not found within 10 blocks from the end position queuing command.)

(6) The program error (P700BN No number) also occurs if the end position block number of an axis control group not specified by G149 is not found within 10 blocks from the end-pint queuing command.

(7) If a macro call or subprogram call is included within the range specified for end position specified queuing, the macro call or subprogram call and M99 are counted as one block.

(8) Whichever axis control group that requires shorter run time for the range of program specified for end position specified queuing waits for processing to be started. ($2 may move first.)

(9) The end position queuing position is not aligned if the range of program specified for end position specified queuing requires run time exceeding two hours.

(10) Do not place an end position specified queuing command within 10 blocks from another end position specified queuing command. Doing so causes a program error (P700BN No number).

(11) Within the range specified for end position specified queuing, do not include any command for changing the workpiece coordinate system, shifting the local coordinate system, presetting the counter, or for milling. Otherwise, the end position specified queuing position is mis-aligned because the specified position is calculated on the workpiece coordinate system effective in the block in which end position queuing is specified.

(12) The end position may not be able to be obtained if any of the following commands is placed, in the range of program specified for end position specified queuing, for controlling the axis having the specified end position. Do not include the following commands concerning the end position specified axis.

- Arbitrary axis exchange control, Direct axis control
- Arbitrary axis superimpose control

(13) Address Q that follows G149 specifies a queued block; X1, Z1, and C specify their respective work coordinates.

(14) If an axis passes the queuing position even once in the range of the program from the block of G149 to the block immediately before the BN block, queuing is not executed correctly.

13.34 Arbitrary Axes Superimposition (G156)

Use the G156 command to declare the axes you want to superimpose on each other.

[Command format]

G156 Z2=Z1	Superimpose ON	The Z2 axis is superimposed on the Z1 axis.
G156 Z2	Superimpose OFF	The superimposition of the Z2 axis on the Z1 axis is canceled. (To cancel superimposition, specify the superimposed axis.)

Gang tool post: X1Z1Y1C1 (Z1: Main headstock, C1: Main spindle C axis)
Back spindle head stock: X2Z2C2 (C2: Back spindle C axis)

[Sample program]

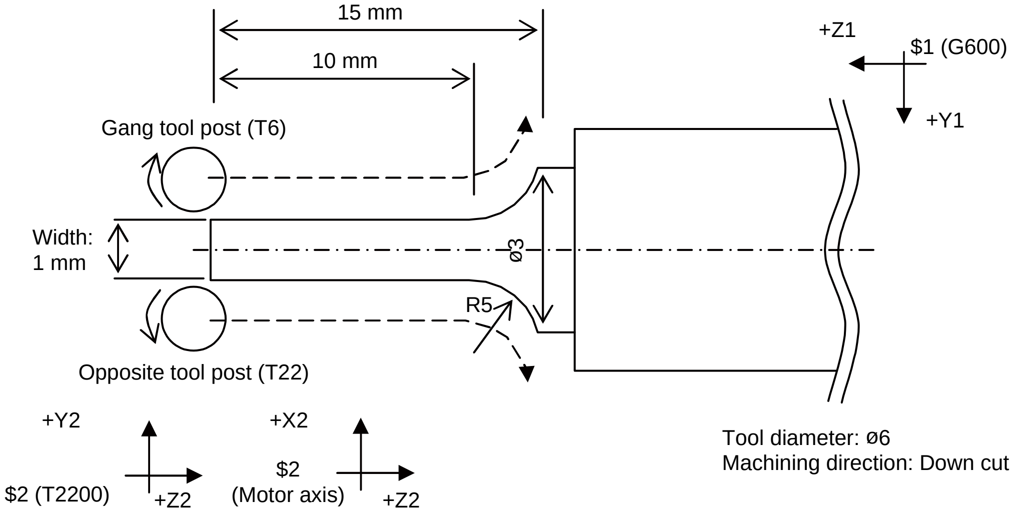
$1		$2	
G600	Machining pattern cancel	G600..............................	Machining pattern cancel
T0300		T3300	
!2L1		!1L1	
:		G156 Z2=Z1	Arbitrary axis superimpose
!2L2		!1L2	
:		G50 Z***	Coordinate system setting
:		:	
:		:	
:		:	
!2L3		!1L3	
		G156 Z2........................	Superimpose cancel
!2L4		!1L4	
:		:	

[Note]

- Before the arbitrary 1-pair axes (G156) command can be executed, the queuing command must be executed for stopping the base axis and the axis to be superimposed. The queuing command must also be executed before the superimposition cancel command.
- After executing the arbitrary 1-pair axes superimposition command (G156), set the coordinate system of the superimposed axis conforming to the coordinate system of the base axis.
- The G53 command cannot be executed for a superimposed axis during execution of the superimpose function.
- By specifying a sign preceding an axis name like "G156 Y2=－Y1", the superimposed axis can be moved in the opposite direction of the reference axis movement direction.
- To cancel superimposition, specify G156 and the superimposed axis in the program of the axis control group that contains the superimposed axis.

Pinch milling

$1		$2	
T700 (D6-ENDMILL)			
G50			
W-15.0			
G0X10.0Y-[6.0+3.0+1.0]T6.......	End mill dia. + Machining outer dia. + Clearance		
Z-[3.0+1.0]	End mill radius + Clearance		
G600	Machining Pattern Cancel	G600	Machining Pattern Cancel
M28S0		T2200 (D6-ENDMIL)	
		M88	Cannot approach unless INT. CHK is disabled.
		G0 X10.0 Y-[6.0+3.0+1.0] T22	Y coordinate is same as $1.
		G53 Z-179.0 (*1)	Position where the center of front rotary tool on gang tool post aligns with that on opposite tool post
		G1 X-5.0 F500	
!2 L10	Wait for !1 L10	!1 L10	Wait for !2 L10
		M83 S4=3000 (*2)	
G98 G19			
M80 S3=3000 (*2)			
G156 X2=Y1	Arbitrary axes superimposition X2 (motor axis) (*3) superimposes on Y axis		
(ARA)			
G1 Y-[6.0+1.1] F250	End mill dia. + Product thickness + Cutting stock		
Z10.0 F100			
G3 V-0.5 Z11.323 R2.0 F50			
G1 Y-[6.0+3.0+1.0] F1000	Retract		
G1 Z-[3.0+1.0] F5000			
(SIAGE)			
G1 Y-[6.0+1.0] F250	End mill dia. + Product thickness		
Z10.0 F100			
G3 V-0.5 Z11.323 R2.0 F50			
G1 Y-[6.0+3.0+1.0] F1000			
G1 Z-[3.0+1.0] F5000			
G0 X10.0 T0			
G50 W15.0			
!2 L112	Wait for !1 L112	!1 L122	Wait for !2 L112
M82 M85		G156 X2	Cancel superimposition
		G98 G1 X10.0 F1000	Retract (*4)
		G0 U0 V0 W0 T0	
		M1	
G610	Alternate machining pattern	G610	Alternate machining pattern Opposite tool post retracts
		M89	Enable INT. CHK



[Note]

- *1 The axis moves in raid feed rate.
  Use M350 command to change axis feed rate for safe operation as needed.

  *2 Be sure to specify the same rotational speed for both rotary tools. Otherwise, the finish surface will not be even.

  *3 If T2200 is called, the axis that moves parallel to Y1 axis (X2 of motor axis) is changed to Y2 axis.

  *4 If the machining pattern is changed after T2200 is called, the command axis is changed to motor axis.
  (In the sample program above, machining pattern is not changed.)
- Parameter for superimposition control direction of Y1 and Y2 axes needs to be changed manually.
  (Base axis parameter 1010: polar(X2)=0 must be changed to 1.)
- In tool layout, use GSE3110 in cross machining direction. If GSE3110 is used in end-face direction, it will interfere with opposite tool post. Change the tool name for T22 to "Cross" on Preparation screen.

13.35 Line Angle Command

This command is used to define the end position of a line by specifying the angle of the line and the coordinate value of either of two axes of the end position. The command automatically calculates the coordinate value of the end position of the other axis and controls axis movement accordingly.

Therefore, the line that has the desired angle can be directly specified using the angle without calculating the coordinate values of the end position.

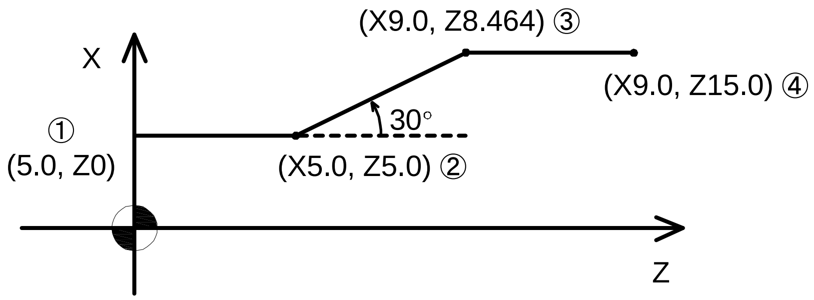
[Command format]

G1 X (Y) (Z)☐ A☐ F☐

[Argument]

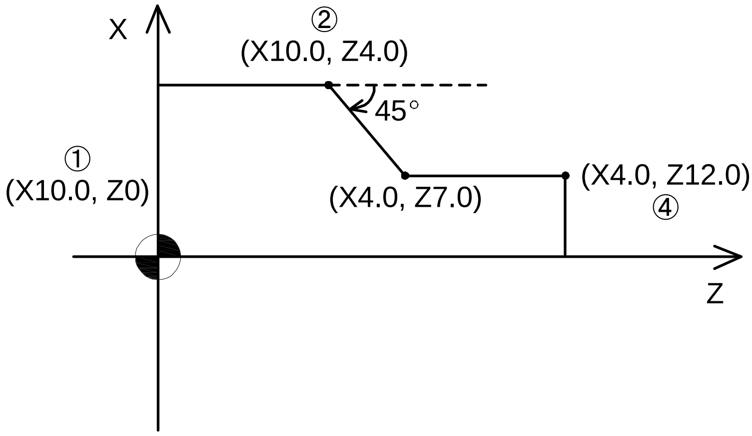
X:	Specify the X coordinate of the end position.	For the coordinate value of the end position, specify either of the axes in the selected plane.
Y:	Specify the Y coordinate of the end position.	
Z:	Specify the Z coordinate of the end position.	
A:	Specify the angle of the line.	
F:	Specify the feed rate.	

[Sample program 1]




```
  :
G1 X5.0 F0.03..........................................①
Z5.0..........................................................②
X9.0 A30.0 ...............................................③
Z15.0.........................................................④
  :
```

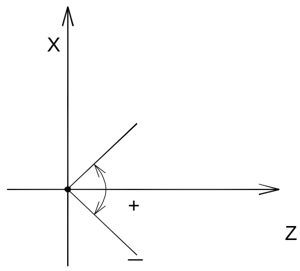
[Sample program 2]



:
G1 X10.0 F0.05..①
Z4.0..②
Z7.0 A-45.0 ....③
Z12.0..④
:

[Note]

- Specify the angle of the line by measuring it in reference to the positive direction of the horizontal axis in the selected plane (G18). The angle measured in the counterclockwise direction is expressed with the "+" (plus) sign and that in the clockwise direction with the "–" (minus) sign.



- For the end position, specify either of the axes constituting the selected plane (G18).
  If the angle and the coordinate value of both axes are specified, the angle is disregarded.
- This command is valid only in the G1 mode.
  It is not valid for other interpolation modes or positioning modes.

13.36 Geometric Command

It is possible to define the chamfering or rounding shape on the corner formed by arbitrary lines by combining the corner chamfering/rounding command, explained in <*13.10 Corner Chamfering / Corner* Rounding>, and the line angle command explained in <*13.35 Line Angle* Command>.

If the coordinate values of the point of intersection of two lines cannot be calculated easily, the geometric command function calculates the coordinate values of such a point by simply specifying the angle of the first line and the coordinate values (absolute values) of the end position and the angle of the second line. The function automatically calculates the coordinate values of the end position of the first line and controls axis movements.

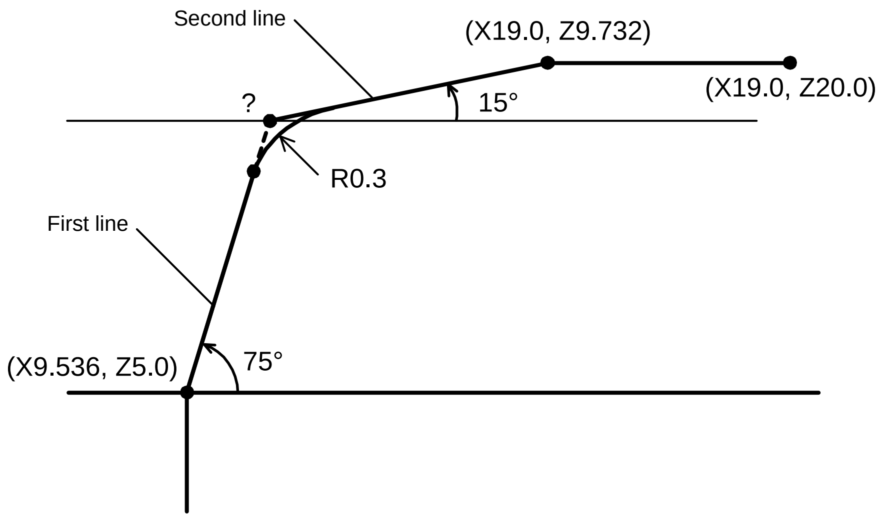
[Command format]

G1 A[①] ,R(C)[ ] F[ ];
X(Y)[ ] Z[ ] A[②];

[Argument]

A[①]	Specify the slope (angle) of the first line.
,R(C)[ ]	Specify the rounding (chamfering) amount at the corner formed between two lines. (This argument can be omitted if corner rounding (chamfering) is not necessary.)
F[ ]	Specify the feed rate.
X[ ]	Specify the X coordinate value (absolute value) of the end position of the second line.
Y[ ]	Specify the Y coordinate value (absolute value) of the end position of the second line.
Z[ ]	Specify the Z coordinate value (absolute value) of the end position of the second line.
A[②]	Specify the slope (angle) of the second line.

[Sample program]




```
  :
G1 Z5.0 F0.1;
X9.536;
G1 A75.0 ,R0.3 F0.03;
X19.0 Z9.732 A15.0;
Z20.0 F0.1
  :
```

13.37 Free Tool Layout Pattern (Holder Name = Free Tool)

This command allows the number of tools to be increased by changing the pitch between the tools on the tool post.

Usualy, 18 gang tools is available on gang tool post. Specify "Free Tool" as holder names on the Machining Data screen. However, you will need to manufacture a special holder in accordance with your requiremnet.

[Command format]

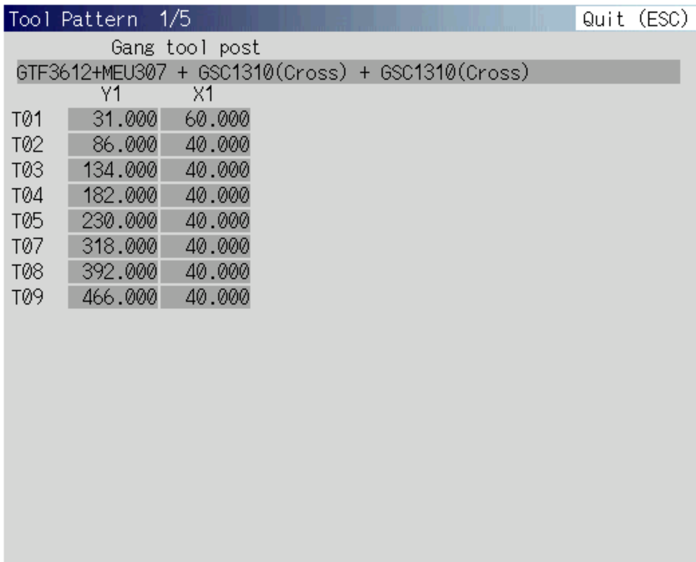
T□□□□ (Four-digit T code)

This command is much the same as an ordinary tool selection command.

Usually, T01 to T19,T21 to T29, T51 to T59, T30, and T31 to T39 can be used.

Tool Layout Pattern screen

Enter the tool position data on the Tool Pattern screen shown below.



Like the ordinary machining, an ordinary command can be issued in the X-Y coordinates after tool selection. Each program can be created as before.

[Note]

- Notice that the tool moves using the T□□□□ code.
- Confirm the contents of the following before machining when the free tool layout is used.
  1. Check whether "Free Tool" has been specified as holder name 1 to 3 on the Machining Data screen.
  2. Check the position of each tool from the guide bushing center as the reference point.
- Sufficient T codes are provided in the machine software. In actual, the tool size and other mechanical factors are limited during use.

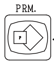
13.38 Custom Macro Program (Program Call by G code)

To call the custom macro program that has been edited and registered, you can use G code number instead of using O number. Up to 200 code numbers (between G1200 and G1399) can be registered. O8100 to O8999 are used as program numbers for this function. You may use the conventional method, calling by O number.

[Command format]

G65 P8101;	Program call by O number (conventional method)
G1201;	Program call by G code

[Procedure]

1. Press the Parameter key .
2. Press the menu key [Set Up 1].
3. Specify values according to the table below.

Preparation Parameter screen

Number	Item		Description	Available value
8071	For G1200 to G1299	Type	0:M98 P**** 1:G65 P**** 2:G66 P**** 3:G66.1 P****	0 to 3
8072		Program No.	Specify the numerical value for the hundreds place of the user program number to be called. If "3" is specified, G1200 to G1299 correspond to O8300 to O.8399. Ex.) G1256 calls O8356.	1 to 9
8073	For G1300 to G1399	Type	0:M98 P**** 1:G65 P**** 2:G66 P**** 3:G66.1 P****	0 to 3
8074		Program No.	Specify the numerical value for the hundreds place of the user program number to be called. If "5" is specified, G1300 to G1399 correspond to O8500 to O.8599. Ex.) G1356 calls O8556.	1 to 9

[Note]

- G1200 to G1399 must be specified in single block. These codes cannot be specified together with any other codes in the same block.
- A, B, C, G, M, N, O, P, S, or T cannot be used as an argument.
- The argument cannot be used when the user macro G code call type is M98.
- G, L, N, O and P cannot be used as an argument when the user macro G code call type is G65, G66 or G66.1.

13.39 DPRINT Function (POPEN, PCLOS, BPRNT, DPRNT)

Use this function to output values or characters of variables according to NC program, via RS-232C interface.

[Command format]

POPEN	Preparation for data output
PCLOS	End processing for data output
BPRNT	Outputs characters and variable values in binary format
DPRNT	Outputs characters and variable values by every digit

[Sample program]

$1
POPEN;
DPRNT [ABC #500 [53] DEF #800 [44]];
PCLOS;

Variable #500:-400.000, #800:1.2346
In [53] and [44], specify the number of digits required for integral and decimal parts.
Note: Total number of digits for integral and decimal parts must not exceed 8.
(For example, [53] represents that five digits for integral part and three digits for decimal part are specified.)

Output format to RS-232C

Communication parameter: CRLF for EOB output, 0 for feed count

DC2%CRLF

ABC-400.000DEF1.2346CRLF

%DC4

[Note]

To output data to RS-232C, be sure to specify PCLOS command to close the line. Do not terminate communication with the line being opened by POPEN command. The line is closed by Reset command, but it is not closed at 1-cycle stop. If an operation is attempted on the Input/Output screen while the line is open, machine operation is not guaranteed.

13.40 Running the Program in External Memory

Outline

This function allows the use of subprograms stored in the CF (Compact Flash) memory card. Insert the CF card into the CF card slot at the front of the operation panel.

[Procedure]

See *<6.9 Running the Program in External Memory>* in the Operator's Manual.

[Command format]

M98 P☐ H☐ L☐ ,D2

P☐	Program number (1 to 99999999) of the subprogram to be called (If omitted, the main program is called.)
H☐	Sequence number (1 to 99999) in the subprogram to be called (If omitted, the program is executed from the beginning of the main program.)
L☐	The number of times (1 to 9999) the subprogram is to be repeated. (If omitted, L1 is assumed.)
,D2	Specify this when calling a program stored in the compact flash memory card (CF).

[Sample program]

M98 P1000 L3 ,D2............................. Runs the O1000 subprogram stored in the CF card three times.

[Note]

- The programs stored in external memory can be called and executed only as subprogram. They cannot be used as a main program.
- During the operation using the subprogram stored in the CF card, do not remove the CF card from the slot.

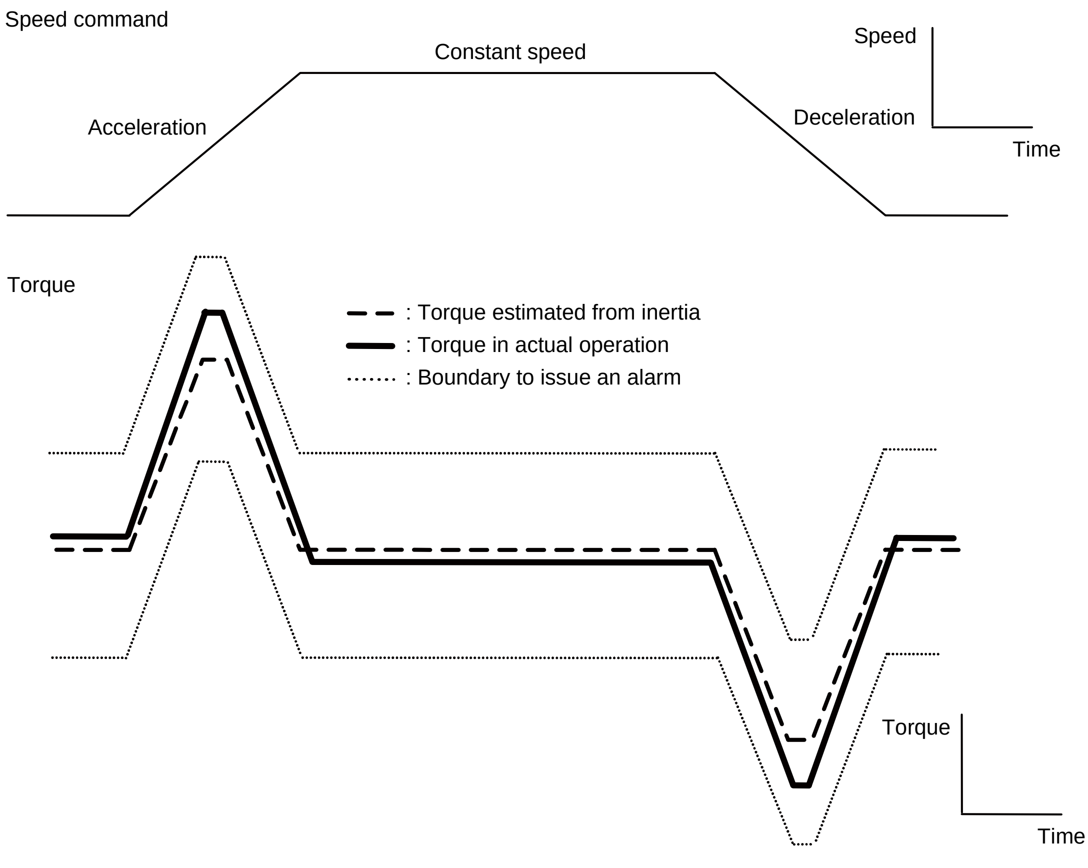
13.41 Collision Detection Function

When an interference is detected, this fucntion suppress the excessive torque of the motor to minimize the mechanical damage to the machine.

> **CAUTION**
>
> This function is aimed to minimize the machine damage due to interference. This function does not protect the machine from interference, nor guarantee the maintenance-free operation.

When the machine is running at rapid feed rate and the motor torque exceeds the boundary to detect a collision, this function reduces the speed and stops the machine, then issue an alarm. This function generates a bring back torque to pull the motor to the reverse direction by about 100 to 200 μ.

The Collision Detection Function is valid only when the machine is running at rapid feed rate (G00), not at cutting feed rate.



Torque diagram in rapid feed

The boundary for detecting collisions is set with a leeway based on the torque estimated from the inertia. If a collision alarm has occurred, one of the following problems might be causing the alarm.

- Interference has occurred among the workpiece, tools, and machine.
- There is a load on the machine’s feed mechanism.
- The clearance between the guide bushing and the workpiece is insufficient and seizure has occurred between them.

In these cases, take necessary measures such as reviewing the machining layout and programs, inspecting the machine, or adjusting the guide bushing clearance.

13.42 Optional Block Skip

13.42.1 Expansion of the optional block skip function

By specifying a number "1" to "9" following the slash code "/" at the beginning of a program block, up to nine kinds of optional block skip can be used. Specification of a multiple block skip commands in a single block is also allowed.

If only a slash code "/" is specified without a number, it is regarded as "/1".

[Procedure]

Setting valid/invalid for the optional block skip "/1" ("/") function

1. On the Automatic Operation, On-Machine Check or MDI screen, press the menu key [Skip1].
The menu item is highlighted and the optional block skip "/1" ("/") function is enabled.
To make the function invalid, press the menu key [Skip1] again. The menu item display returns to the normal display and the optional block skip "/1" ("/") function is disabled.

Setting valid/invalid for the optional block skip "/2" to "/9" function

2. Press the menu key [Set SW].
The Set SW screen is displayed.
3. On the Set SW screen, turn on the optional block skip switch for the optional block skip 2 to 9 functions to be used. This enables the selected optional block skip function.
Turn off the optional block skip switch to disable the optional block skip function.

[Command format]

/1 to /9

Specification of only a slash code "/" without a number is equivalent to "/1".

[Sample program]

When optional block skip 2 is enabled.

$2	
:	
/2 M23 S2=3000	Optional block skip 2 *1
/2 T3100	
/2 :	
/2 :	
/2 T3300	
/2 :	
/2 :	
M25	
M16	
:	

*1 Turning on of the optional block skip 2 switch on the Set SW screen skips the blocks preceded by "/2" when the program is executed.

When optional block skip 1 or 3 is enabled.

$2
:
/1 M23 S2=3000
/1 /3 T3100
/1 /3 :
/1 /3 :
/1 T3300
/1 :
/1 :
M25
M16
:

*1 (/1 /3 T3100 to /1 /3 :) *2 (/1 M23 S2=3000 to /1 :)

[*1] Turning on of the optional block skip 3 switch on the Set SW screen skips the blocks preceded by "/3" when the program is executed.

[*2] Pressing the menu key [Skip1] on the Automatic Operation screen skips the blocks preceded by "/1" or "/" when the program is executed.

[Note]

- Optional block skip 1 to 9 functions are invalid when the power supply to the NC operation panel is turned on.
- Although "/" is equivalent to "/1", use "/1" if several slash codes are to be specified in a block. Specification of several slash codes in the form of "/ /3 N20 G1 X25.0", for example, causes an alarm.
- For the slash code "/", only a number in the range from 1 to 9 is allowed. Specification of "/13 N20 G1 X25.0", for example, causes an alarm.
- For the slash code "/", a variable cannot be used instead of a number. Specification of "/#502 N20 G1 X25.0", for example, causes an alarm.
- For the slash code "/", only an integer can be used. Specification of "/1.2 N20 G1 X25.0", for example, causes an alarm.

13.42.2 Optional block skip M code (M238)

By specifying M238, it is possible to skip program blocks between M238 and a sequence number.

The optional block skip function, specified by the slash code "/", can be used together.

[Command format]

M238 A1 to 9

N9991 to N9999

[Argument]

- A number (1 to 9) that follows the A argument corresponds to the last digit of a four-digit sequence number (9991 to 9999). The combinations of an A argument and a sequence number are "A1 and N9991", "A2 and N9992", "A3 and N9993" to "A9 and N9999".
- If the A argument is omitted, it is regarded as A1.
- Sequence numbers N9991 to N9999 must not be used for other purposes. If a sequence number in this range is used incorrectly, blocks will be skipped unexpectedly.

[Sample program]

When optional block skip 2 is enabled.

$2	
:	
M238 A2	Optional block skip 2 *1 (M238 A2 to N9992)
M23 S2=3000	
T3100	
:	
:	
T3300	
:	
:	
N9992	
M25	
M16	
:	

[*1] Turning on of the optional block skip 2 switch on the Set SW screen skips blocks from M238A2 to N9992 when the program is executed.

When optional block skip 1 or 3 is enabled.

$2
:
M238 (A1)
M23 S2=3000
M238 A3
T3100
:
:
N9993
T3300
:
:
N9991
M25
M16
:

*1   *2

[*1] Turning on of the optional block skip 3 switch on the Set SW screen skips blocks from M238A3 to N9993 when the program is executed.

[*2] Pressing the menu key [Skip1] on the Automatic Operation screen skips the blocks from M238 (A1) to N9991 when the program is executed.

13.43 Thermal Displacement Correction Function

The thermal displacement correction function observes the components of machine and predicts the amount of thermal displacement of the machine. This function automatically adds the correction amount to the tool positioning coordinate.

By using this function, the amount of thermal displacement can be reduced at the cold start or during halting state.

If a tool is selected by T code, it is positioned to the point where the thermal displacement correction amount is reflected. This reflection is performed internally in the NC unit. Thus, tool wear compensation and shape compensation can be performed as usual.

Target axes

- X1 axis (T01 to T05): when 5 turning tools are used
- X2 axis (T31 to T35)

[Note]

- Correction will be performed on the target axis only.
  This function is unavailble for rotary tool on gang tool post.
- If the tool is positioned without using the T code command, the amount of thermal displacement correction will not be reflected.
- The amount of thermal displacement correction is updated at the cycle start, when M2 is specified, or when the bar stock change is executed.
  Accordingly, the amount of thermal displacement correction is not changed within a cycle operation.
- After enabling the thermal displacement correction function, be sure to perform cutting to check the size of machined workpiece.

13.43.1 Environmental requirements and restrictions on use

Required environmental conditions for thermal displacement correction function are as follows:

In an environment where the temperature is out of this range or obstruction frequently occurs, the thermal displacement correction function may not work correctly. In such a case, disable this function.

Environmental condition

- Ambient temperature at site: 10 to 30°C
- Range of temperature change in a day: Within 10°C

The default setting of thermal displacement correction function is "Enabled". However, some restrictions are imposed on using this function as described below.

* For how to enable or disable the thermal displacement correction function, see <*13.43.2 Setting thermal displacement correction function*>.

Restrictions

- The thermal displacement correction function is unavailable if water-soluble coolant is used.
  Disable the thermal displacement correction function.
- The correction results may not be constant even if the same machining program is used, due to characteristics of the thermal displacement correction function.

13.43.2 Setting thermal displacement correction function

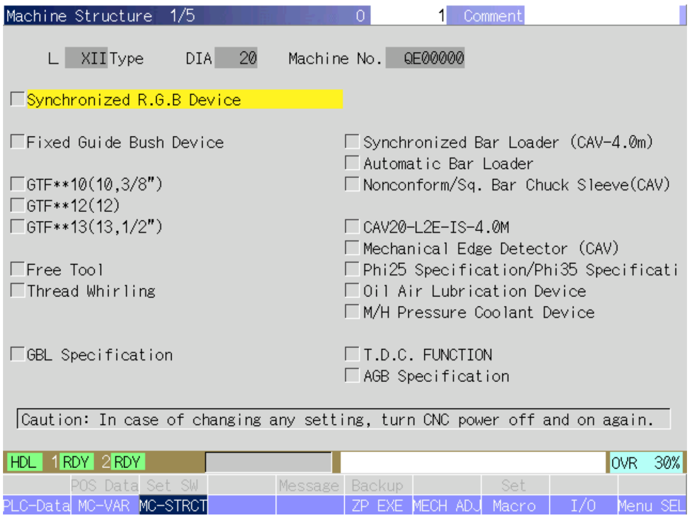
Use Machine Structure screen to enable or disable the thermal displacement correction function.

[Procedure]

1. Press the Manual select key MANUAL or Preparation key PREPA. to enter the EDIT mode.

2. Open the Machine Structure screen.
Press the Parameter key PRM. and press the Menu selection key several times until the menu key [MC-STRCT] appears. Then, press the menu key [MC-STRCT].

3. Set the thermal displacement correction function.



When the following screen appears, press the menu key [Set].
Use the Arrow keys ⬆ ⬇ to move the cursor onto "T.D.C. FUNCTION", and press the Input key INPUT to put a checkmark in checkbox.

√ : Enabled state

☐ : Shows Disabled state

4. Be sure to power on the machine to make effective the new setting.

13.44 B Code Function (MB)

The B code function is used to control external peripheral equipment via I/O.

[Command format]

MB $\boxed{}$ B code command

$\boxed{}$: Specify any integer.Specify an integer between 1 and 255.

Enter a value between 1 and 255 in the box after "MB" and the corresponding binary Y output signal will go on (1).

	Y87	Y86	Y85	Y84	Y83	Y82	Y81	Y80
MB1	0	0	0	0	0	0	0	1
MB2	0	0	0	0	0	0	1	0
MB3	0	0	0	0	0	0	1	1
MB4	0	0	0	0	0	1	0	0
⋮	⋮							
MB255	1	1	1	1	1	1	1	1

For example a "MB3" specification will result in "00000011" and the Y81 output signal (I/O:Y8.1) and the Y80 output signal (I/O:Y8.0) go on (1). And when the common X input signal X82 (I/O: X8.2) is on (1), the above Y output signal goes off (0).

[Note]

- Only 1 B code can be executed per block.
- The B code command in a multi-axis system can be executed independently for each system.

13.45 Simultaneous Machining

The Z1 and Z2 axes can be specified independently. This enables free simultaneous machining. The way to machine with superimposition and the way to machine without superimposition are available for simultaneous machining.

13.45.1 Simultaneous machining for outer and inner diameters

For the simultaneous machining, perform the outer diameter machining by T02, and the hole machining by a T21 (recommended), T22 on T03 tool. However, some restrictions for stroke exist depending on the relation between the tool shape and the diameter of front drilling tool.
Interference may occur if another combination of tools are used for simultaneous machining.

- Select tools for each axis control group ($1 (gang tool post) and $2 (opposite tool post)) by the ordinary T☐☐☐☐ command.
- After machining is complete, be sure to move the tool to the positioning point.
- The machining is performed at the feed rate (F) specified in each axis control group.

As shown in the figure below, the machining is performed by combining the vertical tools and the tools on the opposite tool post.

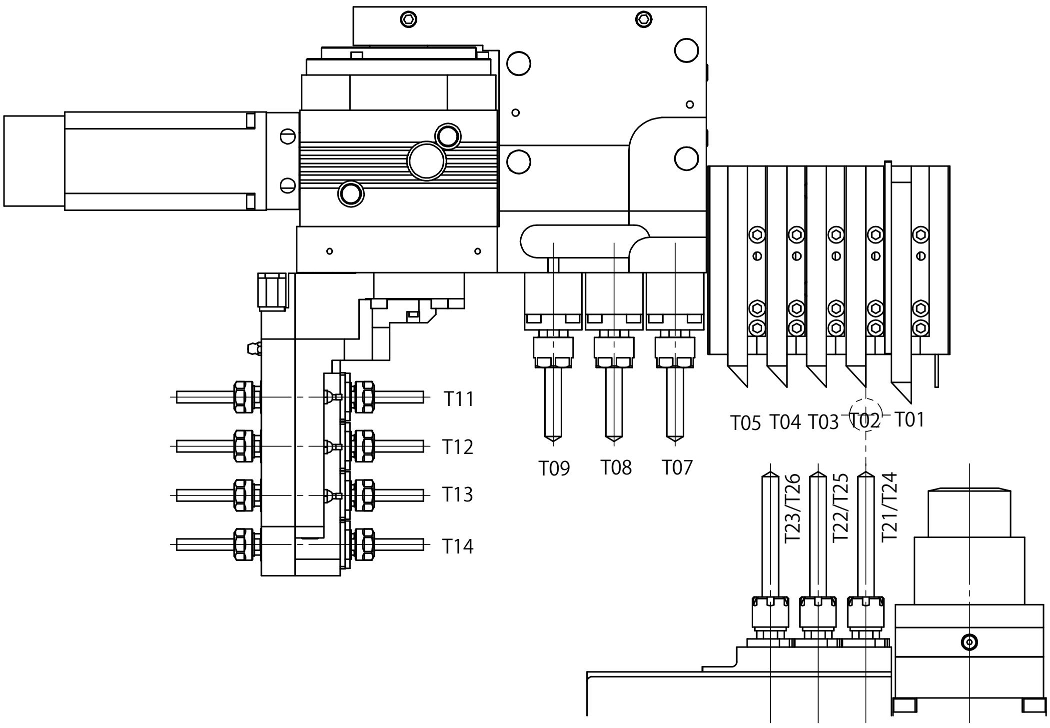


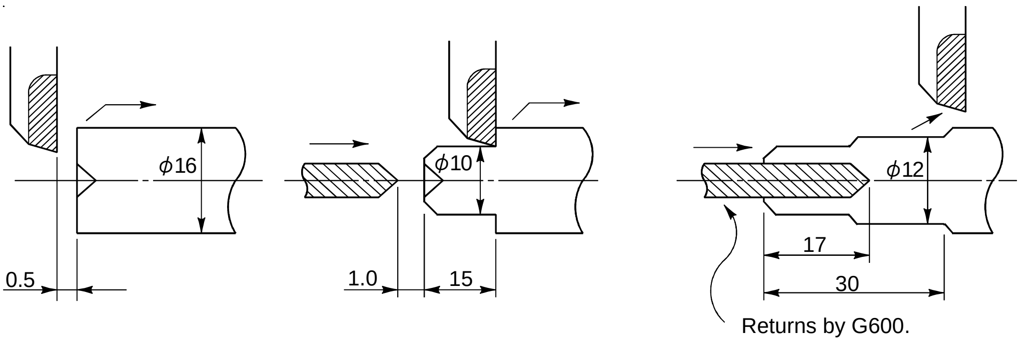
Figure 1 Simultaneous machining for outer and inner diameters

[Note]

- The opposite tool post assumes tools whose maximum drilling diameter is 10 mm. Be careful when using tools exceeding the diameter or odd-shaped tools because they have potential risk of interference.
- Be careful in simultaneous boring with the opposite tool post, when the gang tool and the tool on the opposite tool post are off-centered, causing some tools to interfere with each other.
- When GSE1407 (rotary cross-machining tool) is mounted on opposite tool post, the tool may interfere with the guide bushing. Move the Z2 axis 20 mm to the negative (–) direction to avoid an interference.

Outer diameter cutting and drilling simultaneous machining

Outline of machining



[Sample program]

Machining process

$1	$2		
T0200	T2100	....	Tool selection (in both axis control groups 1 and 2)
G00 Z–0.5		....	Rapid feed positioning (in longitudinal direction)
X8.0		....	Rapid feed positioning (in diametrical direction)
G01 X10.0 Z0.5 F☐		....	Chamfering (C0.5)
Z15.0		....	Cut to the position of 15.0 in the longitudinal direction.
G620	G620 Z–1.0	....	Z1-Z2 superimpose
X11.0	G01 Z17.0 F☐	....	
X12.0 Z15.5	G00 Z–1.0	....	
Z30.0		....	
X17.0 Z31.0		....	
G600	G600	....	Z1-Z2 superimpose command OFF (Axis control group 1) Return the opposite tool post to machine zero point. (Axis control group 2)

For axis control group 1 only (G00 Z–0.5 to G01 X10.0 Z0.5)

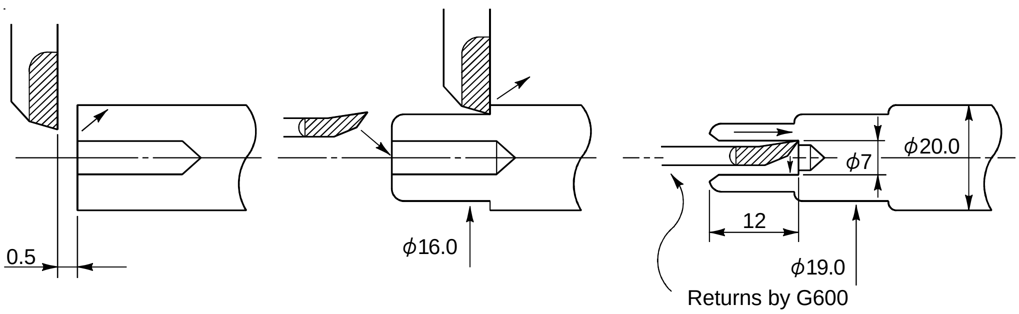
Drilling in axis control group 2 (G01 Z17.0 and G00 Z–1.0)

Outer diameter cutting in axis control group 1 (X11.0 to X17.0 Z31.0)

Machining process

Outer/Inner diameter simultaneous machining (boring)

Outline of machining



[Sample program]

Machining process

$1	$2				
T0200	**T2100**	Tool selection (in both axis control groups 1 and 2)			
G00 Z–0.5		Rapid feed positioning (in longitudinal direction)	Operates in axis control group 1 only.		
X14.0		Rapid feed positioning (in diametrical direction)			
G01 X16.0 Z0.5 F□		Chamfering (C0.5)			
Z11.0		Cut to the position of 11.0 in the longitudinal direction.			
G620	**G620 Z–0.5**	Z1-Z2 superimpose (opposite tool post positioning in longitudinal direction)			
X17.0	**G50 U□**	Cut to ø17.0		Coordinate system shift ON	
X19.0 Z12.0	**G00 X9.0**	Chamfering (C1.0)		Rapid feed positioning	
Z30.0	**G01 X7.0 Z0.5 F□**	Cut to the position of 30.0 in the longitudinal direction.	$1	Inner diameter chamfering (C0.5)	
X21.0 Z31.0	**Z12.0 F□**	Move to the outer diameter while chamfering and cutting.		Cut to the position of 12.0 in inner diameter in the longitudinal direction	$2
	X5.8 F□	Cut to ø5.8 in diameter. (Move the tool bit away.)			
	G00 Z–0.5	Return in rapid feed.			
	G50 U–□	Coordinate system shift OFF			
G600	**G600**	Z1-Z2 superimpose command OFF (Axis control group 1) Return the opposite tool post to machine zero point. (Axis control group 2)			

Machining process

13.45.2 Pinch Milling

Use the GSC1407 cross rotary tool for pinch milling using the vertical tool and the front rotary tool on the opposite post.

Mount the GSC1407 on T21 and T22 to perform pinch milling with rotary tools T07 to T10 on gang tool post.

Note, however, the axis stroke may be limited depending on the tool shape, protrusion length of front drilling tool, protrusion length of cross machining tool, and workpiece length on the back chucking device. In addition, condition to avoid interference becomes harder if any other combination of tools are used in pinch milling.

- Select tools for each axis control group ($1 (gang tool post) and $2 (opposite tool post)) by the ordinary T□□□□ command.
- After machining is complete, be sure to move the tool to the positioning point.
- The machining is performed at the feed rate (F) specified in each axis control group.

The pinch milling is performed by using the vertical tool along with the tool on opposite tool post, as shown in the figure below.

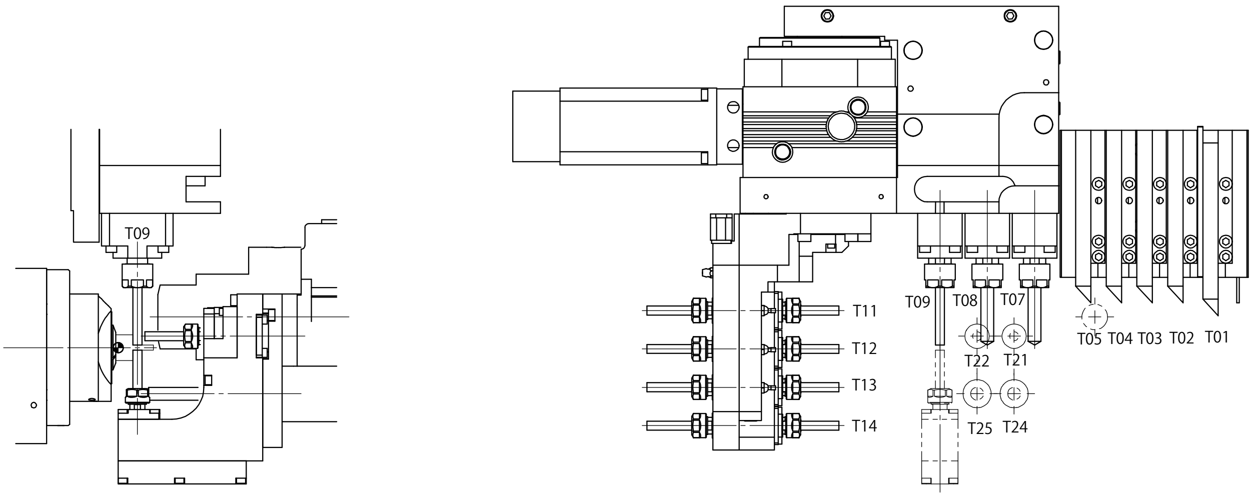


Figure Tool layout pattern for pinch milling

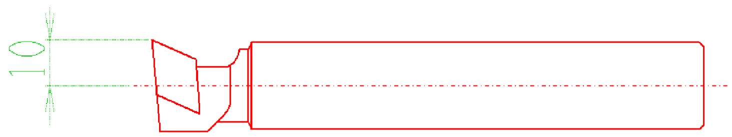
[Note]

- The tool on opposite tool post is intended to drill a hole up to ø10 mm. An interference may occur depending on tool of larger diameter or tool shape. Care must be taken.
- When GSE1407 (rotary cross-machining tool) is mounted on opposite tool post, the tool may interfere with the guide bushing. To machine the workpiece with the end-face tool on opposite tool post, move the Z2 axis 20 mm to the negative (–) direction to avoid an interference.
- The BSE607, BSE707 and GDF1207 cannot be used together with the GSE1407.
- In pinch milling, the workpiece chucked by the back spindle may interfere with the vertical tool. Pay strict attention to protrusion length of rotary cross-machining tool and outer diameter of workpiece. Collect the machined workpiece first, then perform the pinch milling.
- If the vertical tool is equipped with slitting cutter, it may interfere with the end-face machining tool on opposite tool post.
- To put the end mill through the Y axis direction, the workpice on front side may interfere with the cap nut of end-face machining tool on opposite tool post if the workpiece is protruded 45 mm or more.
- If the protrusion length of tool is shorter than 30 mm, the stroke will be limited depending on combination of vertical tool and the tool on opposite tool post, to avoid interference between tools.
- Pay strict attention to interference when machining the workpiece by moving the upper and lower milling tool in opposite direction.
- If the shift tool holder GTF3312 or GTF3313 is used, it will interfere with the workpiece chucked by the back spindle. Perform product collection before starting pinch milling.
- If U35B and GSE3210 are used together, the angle of rotary tool is limited to 45° to 90°.

13.45.3 Pinch Turning

The pinch turning with the vertical tool and the tool on opposite tool post can be performed. Mount the SAU1019 sleeve adapter on T26 and the vertical tool equipped with GTF3312 (or GTF3313) shift tool holder (15-mm shift) on T02, T03, or T04. (Mounting on T03 is recommended.)

Use the tool of 10-mm shift as shown in the figure below. Set the tool nose to the position 109 mm away from the tool mounting end face of opposite tool post.



- Select tools for each axis control group ($1 (gang tool post) and $2 (opposite tool post)) by the ordinary T□□□□ command.
- After machining is complete, be sure to move the tool to the positioning point.
- The machining is performed at the feed rate (F) specified in each axis control group.

The pinch turning is performed by using the vertical tool along with the tool on opposite tool post, as shown in the figure below.

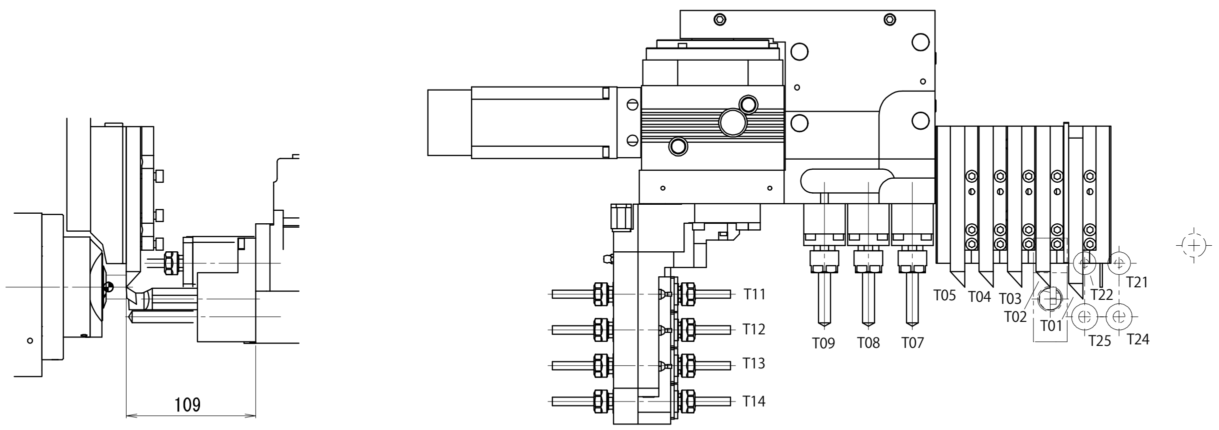


Figure Tool layout pattern for pinch turning

[Note]

- To perform the pinch turning along with the pinch milling, mount the GSC1407 rotary cross-machining tool on T21. Perform the pinch turning with the combination of T26 and T02.
  An interference will occur in any other combination of tools.
- The SAU1019 cannot be used on the machine equipped with U126B (deep-hole drilling on opposite tool post).
  Use the SAU919 to perform pinch turning.
  The stroke and the tool layout may be limited on using SAU919.
- To set the tool nose upward and perpendicularly in pinch milling, the tool nose position in longitudinal direction differs from that of T20's end-face drilling tool used generally.
  (Tool nose position in pinch milling = T20's end-face drilling tool position + 2 mm)
  Strict care must be taken when setting a tool or changing a program.

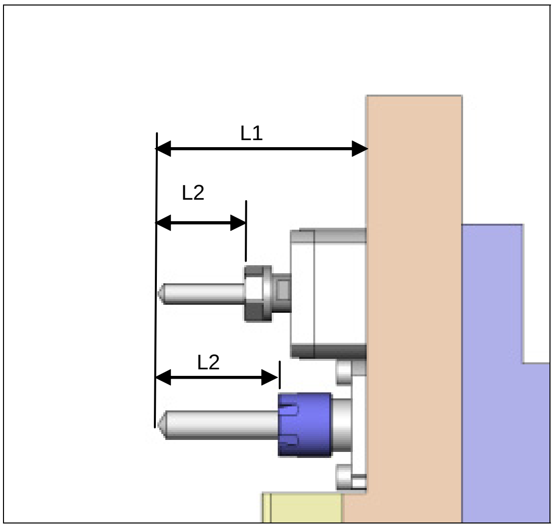
13.46 Major restrictions of tooling setup

This chapter shows the major restrictions of tooling setup.

The restriction on back tool post.

<u>The restriction of length of the tooling on back tool post.</u>

<table>
<tr><th colspan="4"></th><th>Max distance of tool nose from U155B surface
L1 [mm]</th><th>【Reference】
Max tool protrusion length
L2 [mm]</th></tr>
<tr><td rowspan="12">Back tool post
(U155B)</td><td rowspan="8">Upper position</td><td rowspan="2">T31</td><td>GDS210</td><td rowspan="4">62</td><td>30</td></tr>
<tr><td>GSE3507</td><td>20</td></tr>
<tr><td rowspan="2">T32</td><td>GDS210</td><td>30</td></tr>
<tr><td>GSE3507</td><td>20</td></tr>
<tr><td rowspan="2">T33</td><td>GDS210</td><td rowspan="4">74</td><td>44 (ø8.0、M6)</td></tr>
<tr><td>GSE3507</td><td>34 (ø5.0、M4)</td></tr>
<tr><td rowspan="2">T34</td><td>GDS210</td><td>44 (ø8.0、M6)</td></tr>
<tr><td>GSE3507</td><td>34 (ø5.0、M4)</td></tr>
<tr><td rowspan="4">Lower position</td><td>T35</td><td>GDS210</td><td rowspan="2">62</td><td rowspan="2">30</td></tr>
<tr><td>T36</td><td>GDS210</td></tr>
<tr><td>T37</td><td>GDS210</td><td rowspan="2">74</td><td rowspan="2">44 (ø8.0、M6)</td></tr>
<tr><td>T38</td><td>GDS210</td></tr>
</table>



The restriction of the toolings on opposite tool post in case cross drill spindle (GSC1507/GSS1530) is mounted on back tool post.

		Back tool post (U155B)				
		GSC1507				GSS1530
		T31	T32	T33	T34	T34
Opposite tool post (U125B,U126B,U128B)	T21 (T24)	×	×	○	○	○
	T22 (T25)	×	×	×	○	○
	T23 (T26)	×	×	×	△	×

○ : No restriction on opposite tool post.

△ : Only center drilling is possible by opposite tool post.

× : No machining is possible by opposite tool post.

Restriction of Pinch-milling

- GDF1207, BSE607, BSE707 cannot be mounted.
- Need to collect the work-piece on the back spindle before Pinch-milling if shift bite holder (GTF3312, GTF3313) is mounted.
- Position the B-axis tool (MEU307) at a 90° angle.

		Gang tool post		
		GSC1310		
		T7	T8	T9
Opposite tool post (U128B)	T21	○	○	△
	T22	○	○	△
	T23	○	○	○

○ : Need to collect the work-piece on the back spindle if its protrusion length is or is longer than 17mm.

△ : The workpiece on the back spindle and the tool on the gang tool post interfere with each other. Collect the workpiece before start of processing.

(Blankpage)

Product Code	C-L220E XII
Document Code	2E1-1301
Mfg. No.	L220E/0001 ~
Issue Date	2014.1

14. Sample Programming [Rotary tool Machining]

14.1 Rotary Tool Process Guidelines

14.1.1 Turning process and rotary tool process

In the machining operations already discussed, such as outer diameter machining, center hole drilling, and outer diameter thread cutting, the spindle is rotated. Namely, the workpiece is rotated and the tool is got in touch with it for cutting. This machining style is generally called "turning". In the rotary tool process described in this chapter, the cutting tool rotates while the material remains static.

Turning:	Machining is performed by rotating the material.
Rotary tool process:	Machining is performed by rotating the cutting tools.

14.1.2 Machining process order

Machining of a product that requires a rotary tool process is all done in a cycle in the same manner as the machining of a turning process. The rotary tool machining procedure is basically decided according to the following rule.

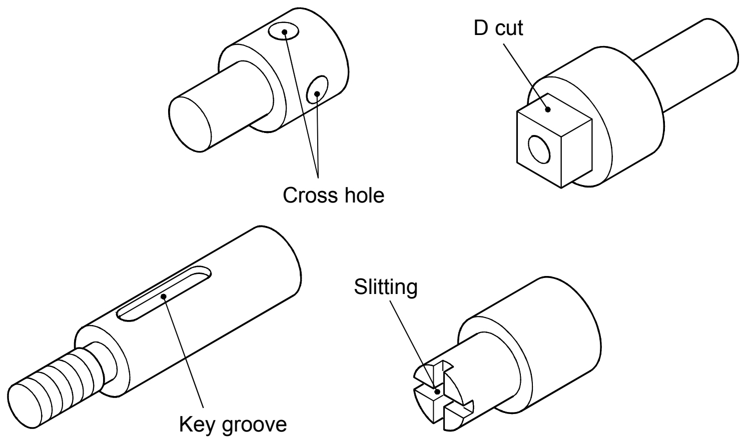
■ Basic rules for the rotary tool machining procedure

- The rotary tool process performs the machining tasks from left to right in order on the material.
- The center hole drilling and rotary tool process on the end face of a material are done before other processes.
- If turning and rotary tool process are to be performed on the same coordinate, the "turning" is done first.

The tooling layout is created based on the conception described above.

The number of tools to be mounted in a rotary tool process varies depending on the holder to be used. (For further details, see <*Chapter 17 Tooling*>).

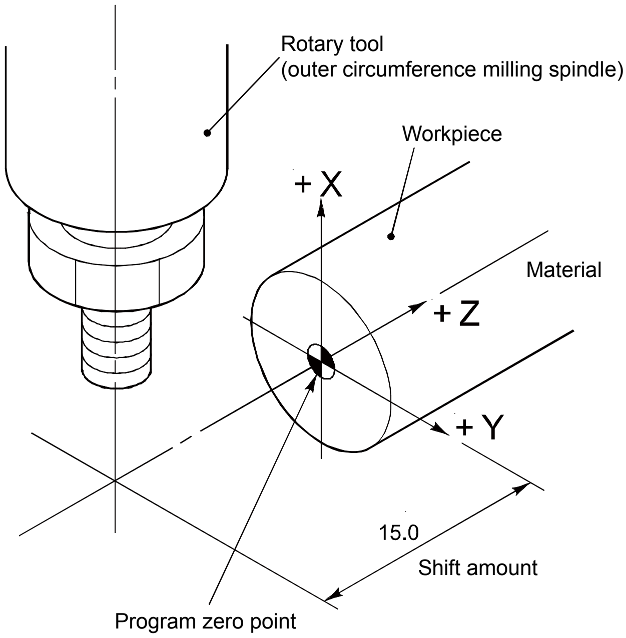
Machining example



14.1.3 The coordinate system for rotary tool process

Before creating a rotary tool process program, it is advisable to have an understanding of the coordinate system.

The diagram below shows the rotary tool in its position relative to the material.



■ Coordinate values and signs

Program coordinates for the rotary tool process are set in the same way as the coordinates for turning. The only difference is that the rotary tool process can perform machining using three axes (X, Z, and Y) since the Y axis coordinate is movable.

Values for the X and Y axis coordinate must be specified in diameter. The program zero points for the X and Z axes are the same as the ones for turning.

The Y axis zero point is set at the center of the material.

For the sign, the moving direction of the tool as shown above is defined as "+" (positive) assuming that the workpiece is fixed.

Y command can be issued even when four-digit T command is issued.

T□□□□ X[ ] Y[ ] Z[ ];

■ Tool position and coordinate system shift

The rotary tool is shifted 15 mm toward the Z axis. If a program is created without a consideration for the shift, machining will be done to a point 15 mm away from the intended position.
To avoid this displacement, shift the length between the program zero point in the Z-axis direction and rotary tool center.

This coordinate system shift must always be canceled after the rotary tool process is complete.
Write the program in $1.

[Command format]

G50 W–15.0	Coordinate system shift
G50 W15.0	Coordinate system shift cancel

14.2 Main Spindle Indexing

14.2.1 Main spindle indexing (M28, M20)

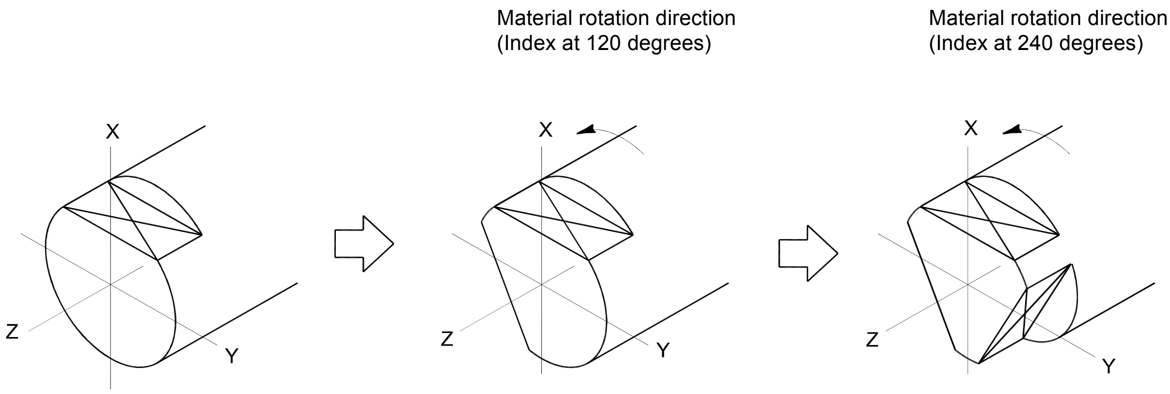
In the rotary tool process, the main spindle can be indexed in 1-degree steps.

[Command format]

M28 S□	Main spindle indexing
M20	Main spindle indexing cancel

- Use these commands in the $1 axis control group program.
- "S□" should be integer in absolute value ranging from 0 to 359.
- The spindle is indexed in the shortcut direction.
- To specify M28 during the rotation of spindle, it is not necessary to stop the spindle. Directly specify M28.
- At the first indexing after power-on, the main spindle makes two or more rotations for indexing its zero point. However, at the second or subsequent time, the indexing is performed within a single rotation.
- To return to turning during the spindle indexing, it is not necessary to use M20. Directly specify M03 S1 =□.

Machining example



Indexing at 0 degree　　Indexing at 120 degree　　Indexing at 240 degree

[Sample program]

```
    :
M28 S0 ......................................... Indexing command
G98 M80 S3=□ ....................... Rotary tool on gang tool post rotates forward, per minute feed
    :                                            Machining program
M82 ............................................ Rotary tool on gang tool post stops rotation
    :
M03 S1=□ G99 ...................... Spindle forward rotation, per rotation feed
```

14.2.2 Back spindle indexing (M78, M79)

The back spindle can be indexed in 1-degree steps.

[Command format]

M78 S☐	Back spindle indexing
M79	Back spindle indexing cancel

- Use these commands in the $2 axis control group program.
- "S☐" should be integer in absolute value ranging from 0 to 359.
- The spindle is indexed in the shortcut direction.
- To specify M78 during the rotation of back spindle, it is not necessary to stop the back spindle. Directly specify M78.
- To return to turning during the back spindle indexing, it is not necessary to use M79. Directly specify the rotation command M23 S2 =☐.

[Sample program]

```
M25................................................ Back spindle stop
G04 U0.5 ....................................... Dwell
M78 S0  ......................................... Back spindle indexing
   :                                                    Machining program
M79................................................ Back spindle indexing cancel
G04 U0.5 ....................................... Dwell
M23 S2=☐ ............................... Back spindle forward rotation
```

14.2.3 Main spindle C-axis indexing (M18)

The main spindle C axis command can be specified in the rotary tool process. (In 0.001-degree units)

[Command format]

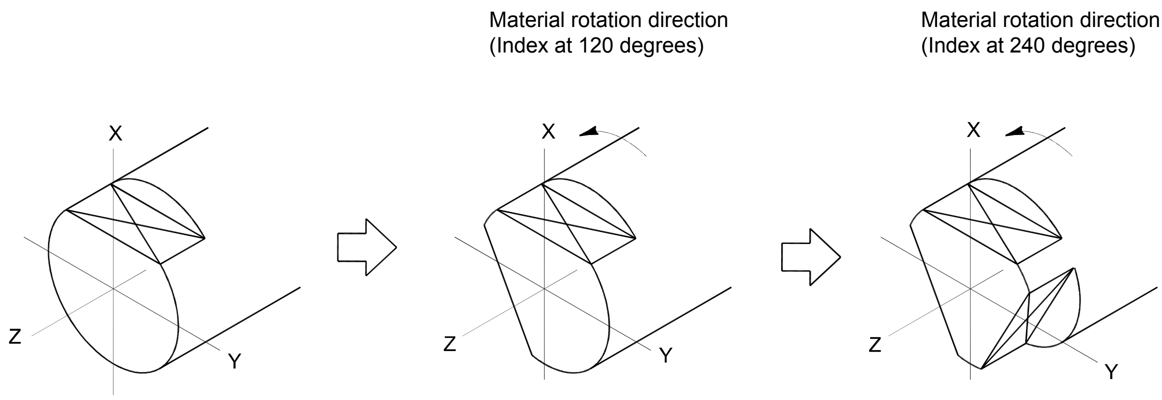
M18 C□ or G0 C□ Command of main spindle C axis

- Specify M18 in axis control group 1 ($1). Specifying M18 in $2 is allowed, however, the main spindle C axis interpolation command after M18 must be specified in $1.
- G0 C□ can directly be specified without using M18.
- M18 C□ or G0 C□ can directly be specified for the spindle C axis during the rotation of spindle.
- To return to turning during the spindle C axis control, directly specify the rotation command M03 S1=□.
- Main spindle indexing direction can be specified. Spindle indexing in the forward direction is specified by the "+" (plus) sign and that in the reverse direction is specified by the "–" (minus) sign. "Forward" and "reverse" of spindle indexing direction are defined as below by viewing the material from the end face.
  Forward: Counterclockwise
  Reverse: Clockwise

  Note that the "+" (plus) sign is omitted.
- The indexing angle for M18 C□ can be specified in the range of 0.0 to 359.999 degrees.
  The indexing angle for G50 C(H)□, G00 C(H)□, G01 C(H)□, or F□ can be specified in the range of –99999.999 to 99999.999 degrees.
- The indexing can be specified in the range of 0.001 to 99999.999 degrees.

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

Machining example



Indexing at 0 degree Indexing at 120 degree Indexing at 240 degree

[Sample program]

:

G0 C0 (M18 C0) .... Indexing command

G98 M80 S3=☐ .... Rotary tool on gang tool post rotates forward, per minute feed

: Machining program

M82 .... Rotary tool on gang tool post stops rotation

M03 S1=☐ G99 .... Spindle forward rotation, per rotation feed

Indexing can be specified in the range of 0.001 to 99999.999 degrees.

■ Indexing command

- Absolute commands

  G0 C0 (M18 C0)

  ↓

  G0 C120.001 (M18 C120.001)

  ↓

  G0 C240.0 (M18 C240.0)

- Incremental command

  G0 C0 (M18 C0)

  ↓

  G0 H120.0

  ↓

  G1 H20.0 F☐

[Note]

If the machine is not in the C axis mode and M18 C☐ or G0 C☐ is specified in any modal except for G0 modal, an alarm is generated.

14.2.4 Back spindle C axis indexing (M48)

In the rotary tool process, the back spindle C axis can be indexed. (In 0.001-degree units)

[Command format]

M48 C□ or G0 C□ Command of back spindle C axis

- Specify M18 in axis control group 2 ($2). Specifying M48 in $1 is allowed, however, the back spindle C axis interpolation command after M48 must be specified in $2.
- G0 C□ can directly be specified without using M48.
- M48 C□ or G0 C□ can directly be specified for the back spindle C axis during the rotation of back spindle.
- To return to turning during the back spindle C axis control, directly specify the rotation command M23 S2=□.
- Main spindle indexing direction can be specified. Spindle indexing in the forward direction is specified by the "+" (plus) sign and that in the reverse direction is specified by the "–" (minus) sign. "Forward" and "reverse" of spindle indexing direction are defined as below by viewing the material from the end face.
  Forward: Counterclockwise
  Reverse: Clockwise

  Note that the "+" (plus) sign is omitted.
- The indexing can be specified in the range of 0.001 to 99999.999 degrees.

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

[Note]

If the machine is not in the C axis mode and M48 C□ or G0 C□ is specified in any modal except for G0 modal, an alarm is generated.

14.2.5 C axis machining in the spindle synchronized control mode

The feature explained below is used to switch the mode to the C axis machining mode while a long workpiece is clamped at the front and back side in the spindle synchronized state. Chuck opening and closing operation can be eliminated.

[Sample program]

$1
:
M3 S1=1000 M24 S2=1000
G114.1 H1 D2 .... Spindle synchronized control ON
M77 .... Wait for the completion of spindle synchronization
:
Machining in the spindle synchronized control mode
:
G0 C90.0 .... Switching from the spindle synchronized control mode to the C axis mode (C1 - C2 synchronization) *1
:
Machining in the C axis mode
:
M3 S1=1000 .... Switching from the C axis mode to the spindle synchronized control mode
:
Machining in the spindle synchronized control mode
:
G113 .... Spindle synchronized control OFF
M5 M25
:

Specification of a C axis command in the spindle synchronized control mode is allowed. The main spindle and the back spindle decelerate in the synchronized state and the mode is switched to the C axis mode.

[Note]

- When a C axis command is specified in the spindle synchronized control mode, the C1 and C2 axes are placed in the synchronized state.
- Specification of a C2 axis command is not allowed while the C1 and C2 axes are in the synchronized state.
- If the spindle synchronized control OFF command (G113) is specified in the state the C1 and C2 axes are synchronized, both the spindle synchronized control mode and the synchronization between the C1 and C2 axes are canceled. In this case, however, the C1 axis remains in the C axis mode.

14.3 Calculating the Command Speed

14.3.1 Calculating the spindle speed value

The spindle speed value is calculated using the following formula (A).

$$N = \frac{V}{\pi D} \times 1000 \qquad (A)$$

N : Spindle speed (min^{-1})
V : Cutting speed (m/min)
D : Tool diameter (mm)
π: Circular constant (approx. 3.14)

Calculation example

Rotary tool process spindle

Use an 4-mm diameter end mill to machine a material under the following conditions;

Material: Brass
Cutting speed: 25 (m/min)
Tool diameter: ø4 (mm)

The spindle speed is obtained from formula (A) as follows:

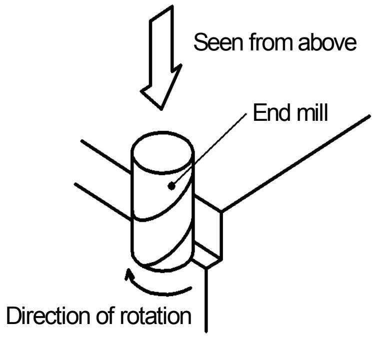
$$N = \frac{25}{3.14 \times 4} \times 1000 = 1990 \approx 2000 \ (min^{-1})$$

- The result of the calculation is rounded up to a whole number value.
- If the result of the speed calculation is more than the maximum speed, set the maximum speed.

14.4 Upcut and Downcut

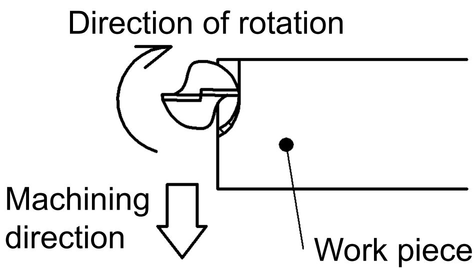
There are two machining methods for side milling using an end mill: Downcut and upcut.

The tool in a normal end mill machining turns clockwise when seen from above. In reverse milling, the tool turns in the opposite direction.



■ Upcut

When seen from above, the material appears to the left relative to machining direction of the tool. In this type of machining, cutting depth starts at 0 and gradually increases. Upcut tends to provide a better finish, but tool life tends to be shorter.



■ Downcut

When seen from above, the material appears to the right relative to machining direction of the tool. In this type of machining, cutting depth starts at maximum and gradually decreases. Downcut tends to increase tool life.

14.4.1 Calculating the tool feed rate

The tool feed rate (mm/min) is calculated using either of the following formulas based on the calculation results of formula (A). Note, however, that the formula to be used depends on the tool used.

■ Tool feed rate formula:

End mill and slitting cutter

$F = N \times fz \times Z$ (B)

- F: Feed rate
- N: Spindle speed
- fz: Feed amount per cutter
- Z: Cutters (cutter/rev)

Drill

$F = N \times fz$ (C)

- F: Feed rate
- N: Spindle speed
- fz: Feed amount per revolution

Calculation example

Rotary tool process spindle

Use an 4-mm diameter end mill to machine a material under the following conditions;

Material:	Brass
Spindle speed:	2000 (min^{-1})
Feed rate per tool:	0.065 (mm/tool)
Number of tools:	2 (teeth)

The tool feed rate is obtained from formula (B) as follows:

$F = 2000 \times 0.065 \times 2 = 260$ (mm/min)

14.5 Automatic Control of Rotation Direction of Rotary Tool

The rotation of rotary tool depends on the tool layout and the selected tool number. This function automatically controls the rotation direction of the motor so that the direction to ratate the rotary tool is made identical to that specified by G or M code. With this function, you can specify the rotation direction of rotary tool without considering the tool layout or the tool numbers.

CAUTION

The Automatic Control of Rotation Direction of Rotary Tool function is unavailable in the following cases:

– During front/back simultaneous machining (G660), when back drilling tool on gang tool post (T51's) is selected

– When a free holder is used in the machining data

14.5.1 Applicable tool layout and tool numbers

The rotation direction specified by G or M code differs from that of rotary tool in the following tool layout.

Rotary tool on gang tool post

Name		Target tool number	Type
Outer circumference milling spindle	GSE1310	T07, T08, T09	XII
When a free holder is used in the machining data	MEU307	T12, T14, T51, T53	XII
Rotary tool on gang tool post	BSE607	T08, T08R2, T09, T09R2	XII
End-face drilling spindle (triple spindles on both ends)	BSE707	T08, T08R2, T09, T09R2, T58R1, T59R1	XII

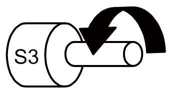
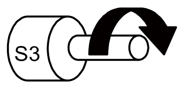
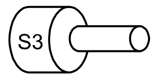
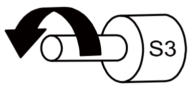
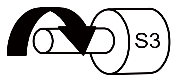
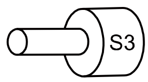
Rotary tool on back tool post

Name		Target tool number	Type
Outer circumference milling spindle	GSC1507	T31, T32, T33, T34	XII

14.5.2 Rotary tool start/stop (M80/M81/M82)

■ Rotation direction of rotary tool

The rotation direction depends on mechanical wheel block. When M80 or M81 (rotary tool rotation command) is issued, the rotation direction of motor is automatically determined by the tool number being selected. The rotation direction of tool edge is defined as follows: Counterclockwise rotation when viewed toward the spindle is regarded as forward rotation, and clockwise rotation is regarded as backward rotation.

Tool No.	M80	M81	M82
T01/T11's	Forward rotation 	Backward rotation 	Stop spindle 
T51's	Forward rotation 	Backward rotation 	Stop spindle 

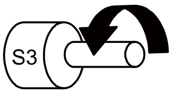
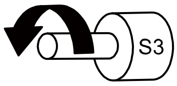
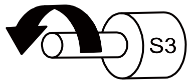
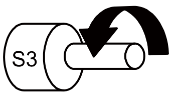
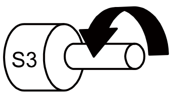
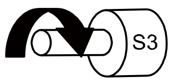
■ Rotation direction of rotary tool at tool selection

If a tool is selected (with T code) while the rotary tool is rotating, the rotation direction of rotary tool is automatically determined according to the specification described above.
For example, if the tool number T12 is selected while the tool number T11 is being selected and the rotary tool is rotating forward, the motor rotates in opposite direction. However, the tool edge keep rotating forward, you can write a program in the conventional way.

■ Rotation direction of rotary tool by machining pattern

The rotation direction of rotary tool during front/back simultaneous machining (G660) is controlled by the tool on front spindle (T01/T11's), a contrary rotation of the tool on back spindle (T51's).

[Sample program]

$1	$2
:	
G600	**G600** .... Free tool layout pattern
M80 S=☐.................. Rotary tool forward rotation	
T1200  Forward Rotation	
:	
T5200  Forward Rotation	
:	
M82	
!2L1	**!1L1**
	M80 S=☐ .... Rotary tool forward rotation
	T5200  Forward Rotation
	:
	M82
G630	**G630** .... Front/back parallel machining
M80 S=☐.................. Rotary tool forward rotation	
T1200  Forward Rotation	
:	
M82	
G660	**G660** .... Front/back simultaneous machining
M80 S=☐.................. Rotary tool forward rotation	
T1200  Forward Rotation	**T5200**  Backward Rotation
:	

[Note]

- If this function is specified in front/back simultaneous machining (G660), the rotation direction is determined based on the front machining tool (T01/T11's).
- This function can be used in synchonous tapping (G88, G84), differential rotary tool function (G164), hobbing (G114.2), and polygon machining (G114.3).
- To specify differential rotary tool function (G164), hobbing (G114.2), and polygon machining (G114.3) using this function, these machining must be performed after tool selection.
- Specify the reference direction of rotation before selecting a tool.

- If a machining program contains repetitive foward/reverse rotation at high speed, it may cause an overload alarm to occur. Rewriting the program as shown below may avoid such an alarm.

Acceleration and switching between forward and reverse rotation

M80 S3=1000	Tool spindle motor rotates forward at 1000 rpm.		M80 S3=1000	Tool spindle motor rotates forward at 1000 rpm.
T1100			T1100	
:		⇒	:	
M80 S3=5000	Accelerate to 5000 rpm from 1000 rpm.		T1200	Select a tool and tool spindle motor rotates backward.
T1200	Tool spindle motor rotates backward immediately after acceleration.		M80 S3=5000	Accelerate to 5000 rpm from 1000 rpm.
:			:	

Decelerated and switching between forward/backward rotation

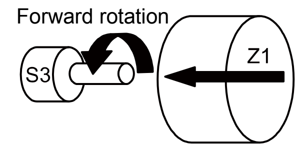
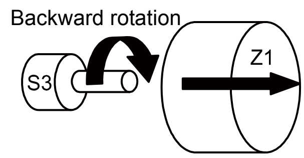
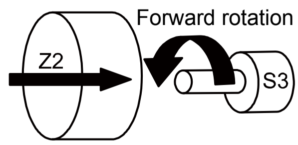
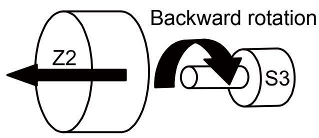
M80 S3=5000	Tool spindle motor rotates forward at 5000 rpm.		M80 S3=5000	Tool spindle motor rotates forward at 5000 rpm.
T1100			T1100	
:		⇒	:	
T1200	Select a tool and tool spindle motor rotates backward.		M80 S3=1000	Deceleration from 5000 rpm to 1000 rpm.
M80 S3=1000	Deceleration from 5000 rpm to 1000 rpm directly after switching to backward rotation.		T1200	Select a tool and tool spindle motor rotates backward.
:			:	

14.5.3 Simultaneous front/back end-face tapping (G84)

■ Axis control group

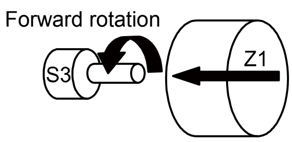
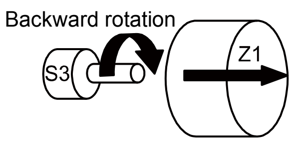
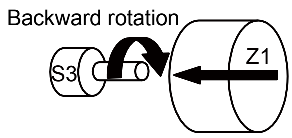
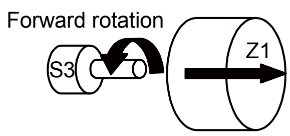
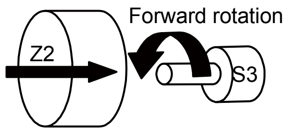
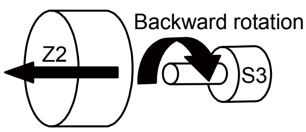
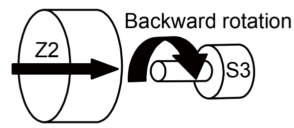
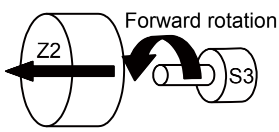
This command can be specified for the axis control groups $1 and $2.

[Sample program]

$1		$2
: T1300 (X1,Z1,Y1) G84 Z☐ F☐ S☐ D3 : G80	 	
: T5300 (X1,Z2,Y1) G84 Z☐ F☐ S☐ D3 : G80	 	

■ Rotation direction of rotary tool

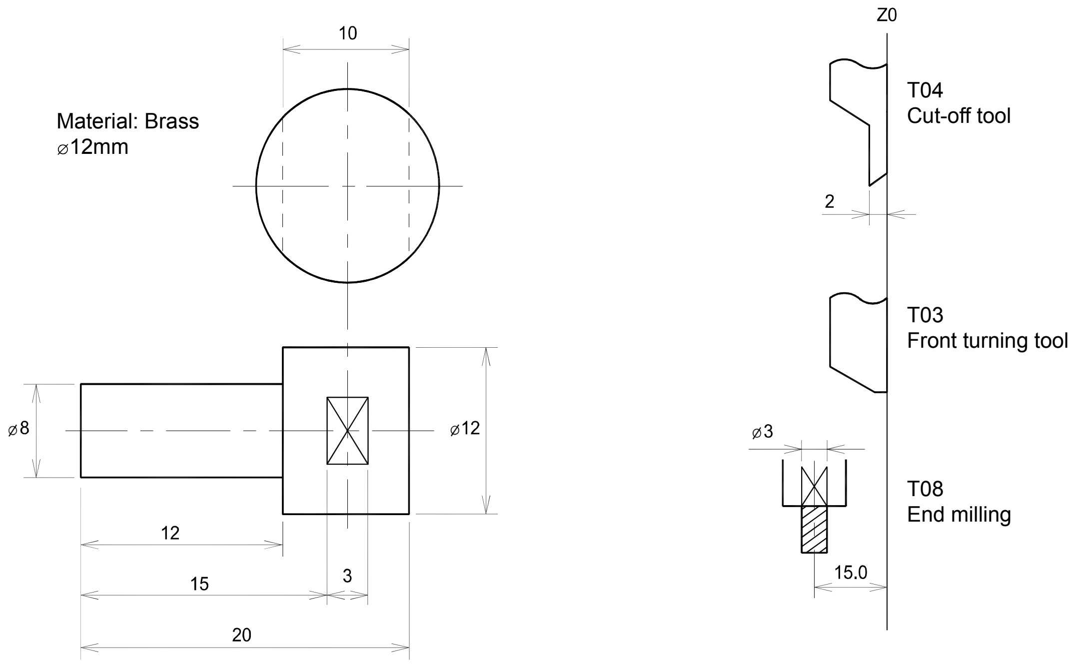
The rotary tool rotates regardless of tool number specified. If "+3" is specified for D argument, the rotary tool (S3) rotates forward at tap infeed. If "–3" is specified for D argument, the rotary tool (S3) rotates backward at tap infeed.

Tool No.	D argument "+3"	D argument "–3"
T01/T11's	 	 
T51's	 	 

14.6 Two-Surface Width Machining (Including Indexing) Program

This example is a portion of one program relating to the rotary tool process.

Machining layout



[Sample program]

T0800	Tool selection
G50 W–15.0	Longitudinal coordinate system shift
M28 S0	Main spindle indexing command
M80 S3=☐ G98	Rotary tool forward rotation command
G00 Y10.6 Z16.5 T☐	Y and Z axis rapid feed positioning
X10.0	X axis positioning
G01 Y–10.6 F☐	Two-surface width single machining
M28 S180	180-degrees indexing
Y10.6	Remaining single machining
G00 X13.0 M82	Move in the diametrical direction. (Rotary tool rotation OFF)
G50 W15.0	Longitudinal coordinate system shift cancel
M03 S1=☐ G99	

Calculation and explanation are shown in the next page.

[Calculation and explanation]

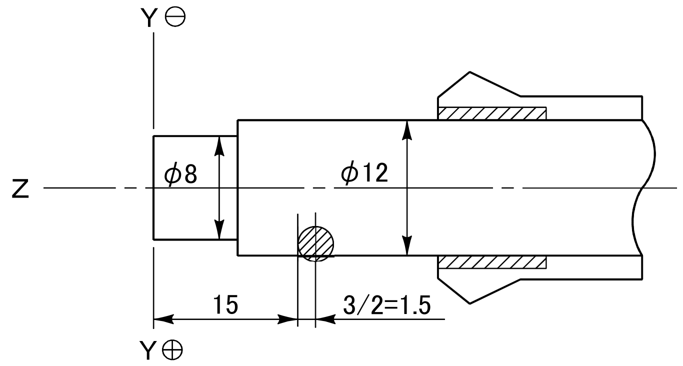
Spindle rotation "N" is given by the following expression.

$$N = \frac{25}{3.14 \times 3} \times 1000 = 2653 \approx 2700\ (\mathrm{min}^{-1})$$

Therefore, end milling feed rate "F" is given by

F = 2700 × 0.05 × 2 = 270 (mm/min)

The longitudinal direction position is set as shown in the figure below.

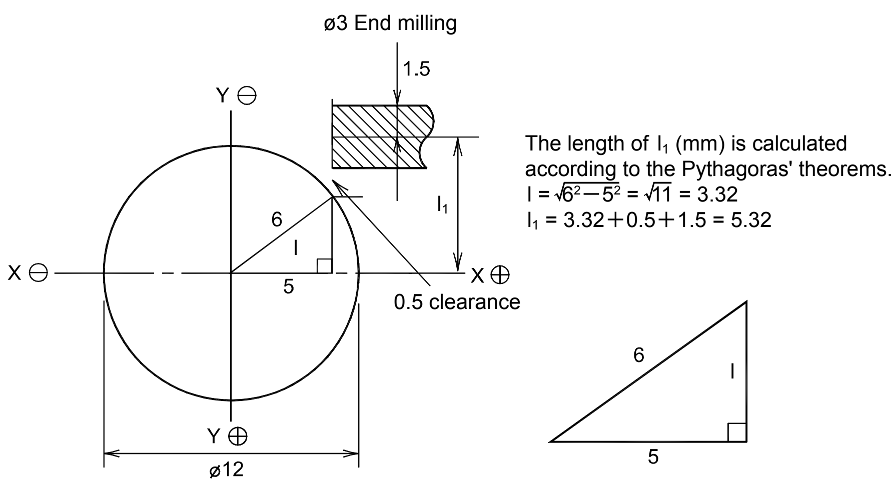


Therefore, the Z coordinates for positioning are as follows:

15 + 3 (end milling diameter) / 2 = 15 + 1.5 = 16.5

As shown below, the Y coordinates (diameter specification) are given by the following expression.

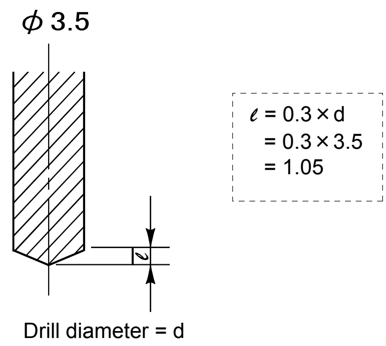
$Y = (\lambda + 0.5 + 1.5) \times 2 = (3.32 + 0.5 + 1.5) \times 2 = 10.64 \approx 10.6$



14.7 Through-Hole Machining

Explanation of this section is an example of machining which pierce the material of Ø16.0 with a cross drill of Ø3.5.

The X1 axis stroke of a gang tool is a maximum of X–5.0. The ordinary tool set cannot drill the through-hole of the material. Accordingly, the drill must be shifted so that it is elongated by the setting of the tool setting data when mounting it.



The X value of a program that considers no shift is –16.0 – (1.05 × 2) = –18.1.

If X–18.1 is specified, an X axis overrun occurs.

Since there is a stroke in X–5.0, the X value becomes 18.1 – 5.0 = 13.1.
Enter this shift amount to "DIA" in the tool setting data on the Preparation screen. This enables through-hole machining by specifying X–5.0 in the program.

In this case, the drill is shifted 15.0 mm in diameter (satisfactory if shifted 13.1 mm).

[Note]

- After entering the shift amount to "DIA" in the tool setting data on the Preparation screen, call "DIA" and then set the tool.

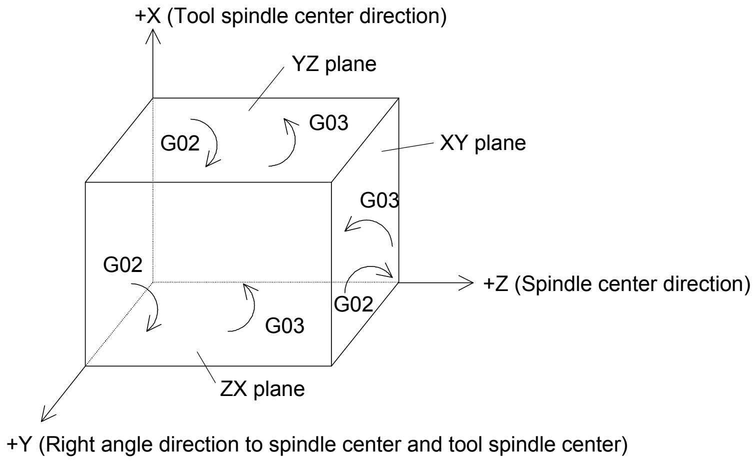
  If an incorrect value is set, the tool and the material will interfere with each other when calling a tool or executing the positioning point command.
- When selecting a tool, tool change is performed with the maximum value of "DIA" values of the gang tools set on the Preparation screen taken into account. Therefore, it is not necessary to specify unnecessarily large retraction amount or set a large value to "Tool Positioning Point (DIA)" on the Machining Data screen.

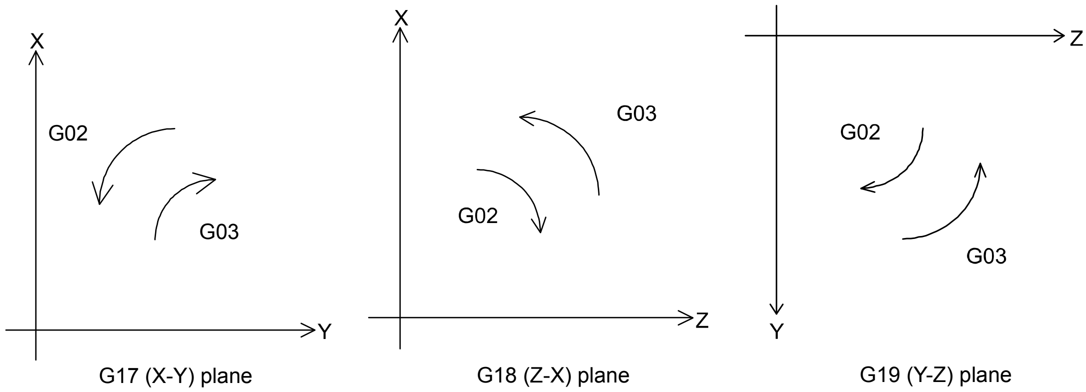
14.8 Circular Interpolation in Rotary tool Process

Circular interpolation feed for the rotary tool process is available on three planes:

[Command format]

G17	XY plane
G18	ZX plane (Plane selection mode for normal turning)
G19	YZ plane





14.8.1 Plane selection

When the machine is turned on or after resetting, the ZX plane (G18) is selected. The circular interpolations (G02 and G03) in turning are also performed on this plane.

If another plane is required for circular interpolation in the rotary tool process, enter the plane selection command G17 (XY plane) or G19 (YZ plane).

Select ZX plane (G18) again to return to turning.

14.8.2 Tool diameter compensation in rotary tool process

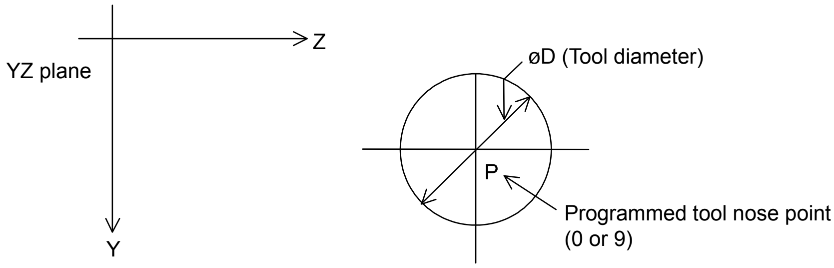
The tool diameter compensation is programmed using a tool nose R compensation function.

[Command format]

G40	Tool diameter compensation (tool nose R compensation) mode cancel
G41	Tool diameter compensation left (tool nose R compensation left) mode ON
G42	Tool diameter compensation right (tool nose R compensation right) mode ON

The relation between the tool diameter and the cutting position in a program is described below.

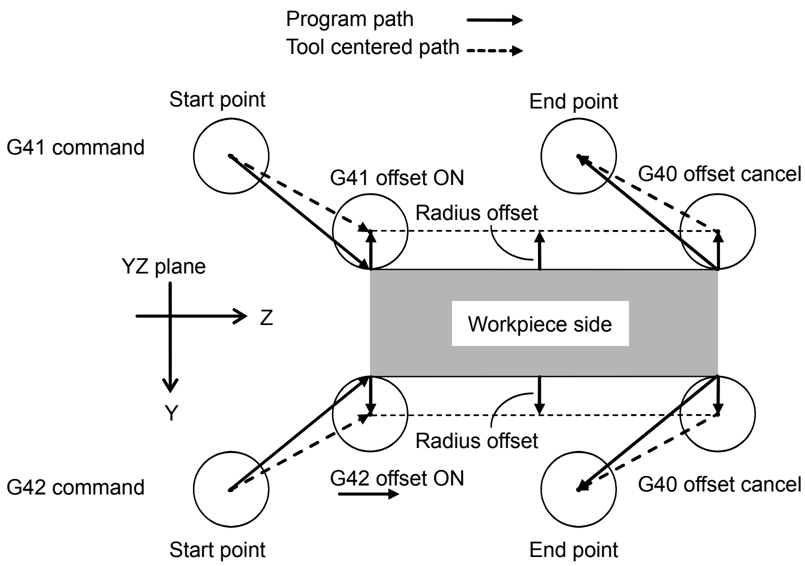
In the registration item of the tool diameter compensation, preset to "0" or "9" to the virtual tool nose number.



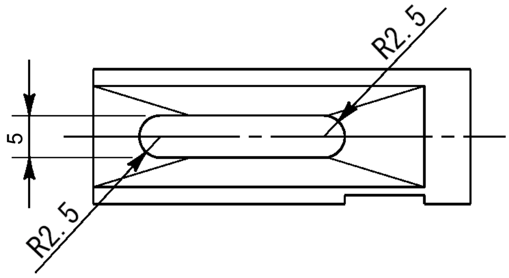
The tool cutting point and compensation are as shown below.

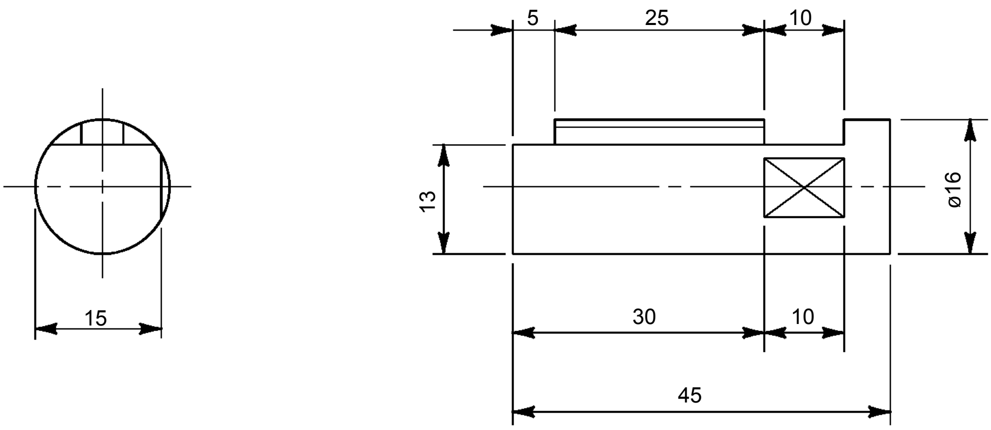
G41: Offset to the left for the tool advance direction.

G42: Offset to the right for the tool advance direction.

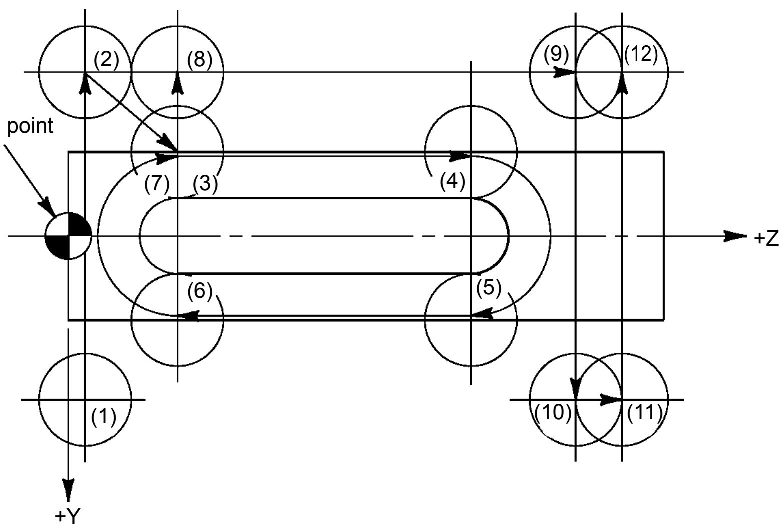


Machining layout





Tool data
Tool diameter ø6 (End mill)
Tool No. T12
Offset No. 12
Nose R radius (Tool radius value) 3.0
Virtual tool nose No. 0
Numbers (1) through (12) in the figure above indicate the programmed end mill movement.



[Sample program]

This program is described in the $1 program.

$1	
T1200	Select and compensate the tool spindle.
G50 W–15.0	Longitudinal coordinate system shift.
M28 S0	Index the spindle by 0°.
S3=$\boxed{\quad}$ M80 G98	Rotate the rotary tool in the forward direction.
G19 G00 Y22.0 Z1.0 T12	Set the plane to YZ and position the tool to the cutting start point in the YZ plane.
X10.0	Position in the cutting direction (X axis).
G01 Y–22.0 F$\boxed{\quad}$	
G41 Y–5.0 Z7.5	Compensate the tool diameter.
Z27.5	
G02 Y5.0 R2.5 F$\boxed{\quad}$	
G01 Z7.5 F$\boxed{\quad}$	
G02 Y–5.0 R2.5 F$\boxed{\quad}$	
G40 G1 Y–22.0 F$\boxed{\quad}$	Cancel the tool diameter compensation.
G00 Z34.0	
G01 Y22.0 F$\boxed{\quad}$	
G00 Z37.0	
G01 Y–22.0 F$\boxed{\quad}$	
M28 S90	Index the spindle by 90°.
G00 X14.0	
Y–18.0 Z33.0	
G01 Y18.0 F$\boxed{\quad}$	
G00 Z37.0	
G01 Y–18.0 F$\boxed{\quad}$	
G18 G00 X18.0 T00	Return the plane command to the turning mode, cancel the compensation, and return the tool to the positioning point in the diametrical direction.
M82 G99	Stop the rotary tool.
G50 W15.0	Longitudinal coordinate system shift cancel.

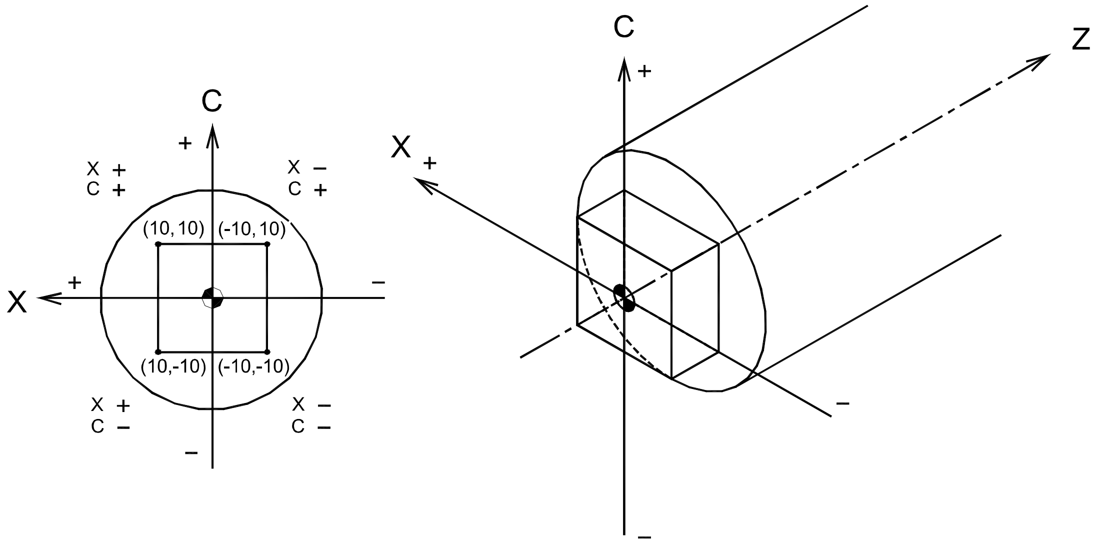
14.9 Milling Interpolation (G12.1, G13.1)

Generally, the X and Z axes are controlled as linear axes and the C axis is controlled as the rotary axis of the spindle. The milling interpolation function permits the C axis to be used as a linear axis at right angles to the X and Z axes, and enables the creation of a three-dimensional program with the three axes X, Z, and C.

14.9.1 Milling coordinates

Milling interpolation is performed between when G12.1 (milling interpolation ON) is executed and when G13.1 (milling interpolation cancel) is executed. Specify the coordinates of X (Z) and C axes with values calculated for the radius.

The plus and minus signs of X and C are as shown in the figure below.



[Command format]

G12.1 D☐ E=☐ Milling interpolation ON

G13.1 Milling interpolation cancel

[Argument]

D☐ : Selects the name of a milling virtual axis. Specify "0" to select Y axis, and specify "1" to select C axis. The default is 1(C axis).

E=☐ : Specify an axis for milling interpolation. If omitted, the axis is determined by the milling axis number specified in the basic specification parameter #1125.

For the E argument, specify "E=C" if the axis number of C axis is different between the axis control group and the tool holder.

For example, if a front drill holder is used with $1, the axis number of C axis is identical; 3 for both the axis control group and the tool holder. Thus, there is no need to specify the E argument. (Specifying the argument does not cause any problem.)

If a gang tool holder is used with $1, the axis number of C axis is different between the axis control group and the drill holder. In this case, "E=C" must be specified.

In general, you do not have to specify the D argument.

When specifying both D and E arguments, be sure to specify in the order of D and then E.

■ Relationship between the axis numbers of C axis and the axis control groups

	$1	$2
Axis number of C axis	3	2

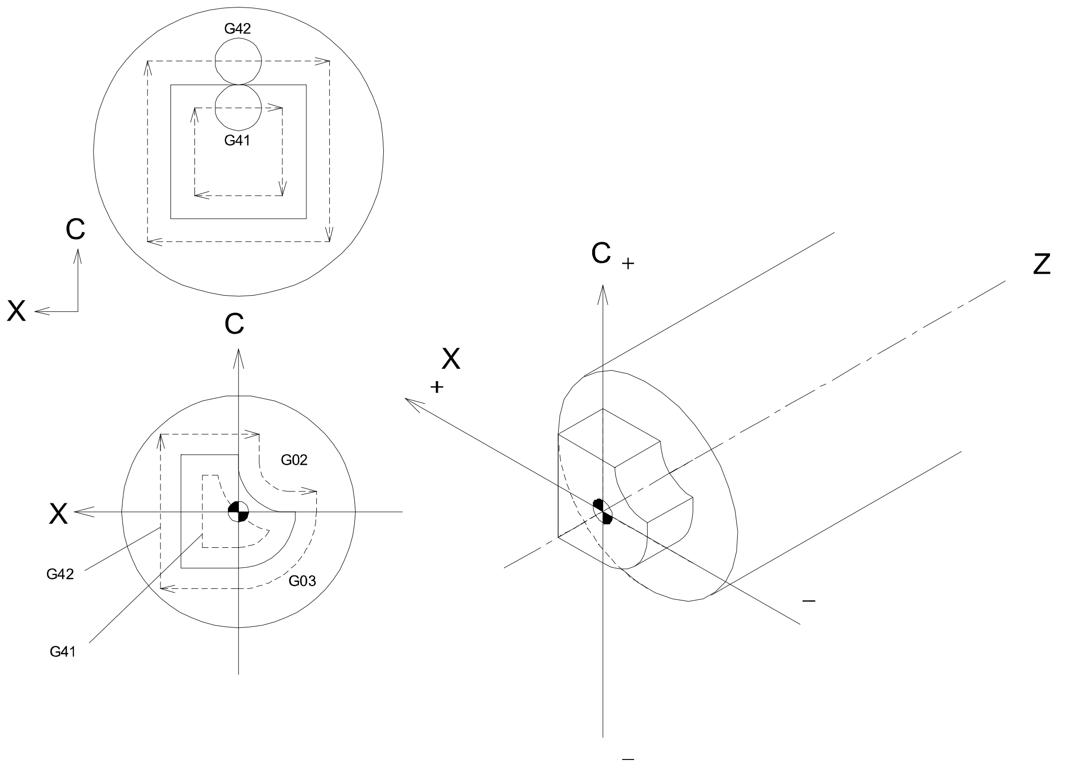
■ Relationship between the axis numbers of C axis and the tool holders

	Axis number of C axis
Gang tool holder (T0100 to T1400)	3
Front drilling holder on gang tool post (T2100 to T2300)	2
Back drilling holder on back tool post (T3100 to T3400)	2

14.9.2 Milling plane

To perform end-face machining, execute the G17 command to select the X-Y plane, and program the coordinate values viewed from the end face of the workpiece.

	At end-face machining	Compensation mode (operation)
Tool diameter compensation	G40	Compensation cancel
	G41	Right compensation
	G42	Left compensation
Circular Interpolation	G02	CCW (Counterclockwise threading)
	G03	CW (Clockwise threading)



14.9.3 Programming of a milling process

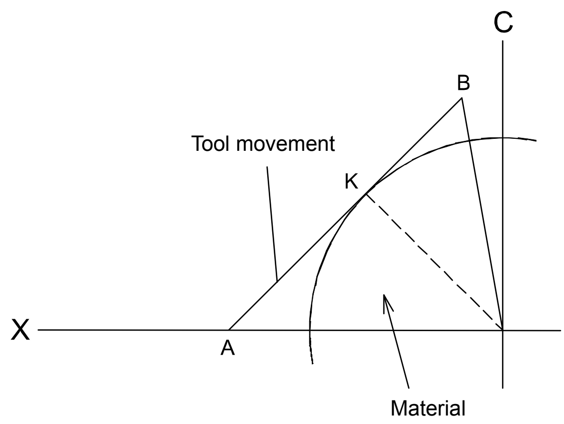
Consider the following points in programming a milling process:

When starting a milling process, move the X axis away from the workpiece to obtain an enough approach distance. The X axis makes an approach to the workpiece, and then moves to the specified position. Therefore, if the approach distance is not enough as shown in the following figure, the tool and workpiece interfere with each other.

During the positioning of the X axis from point A to point B in the following figure, the X axis makes an approach to the workpiece in the range from point A to point K, and moves away from the workpiece in the range from point K to point B.

The milling start position (point A) for avoiding interference with the workpiece must be obtained by calculation.

Point A = Ps position

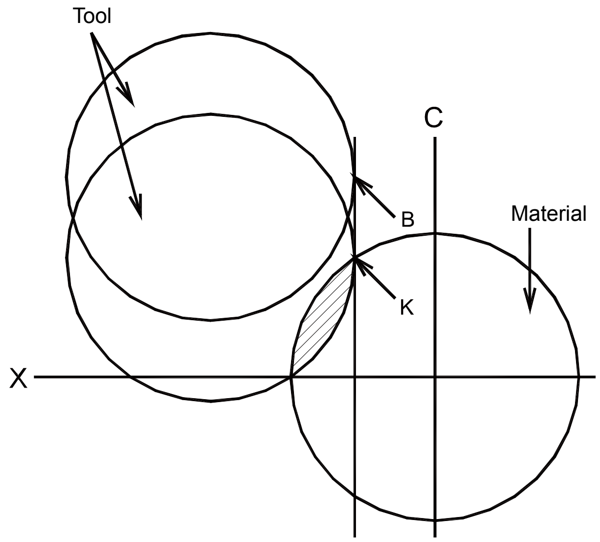


Determine the milling start position in consideration of the diameter of the rotary tool to be used and the diameter of the workpiece to be machined. If the start position is not suitable, the tool and workpiece may interfere with each other.

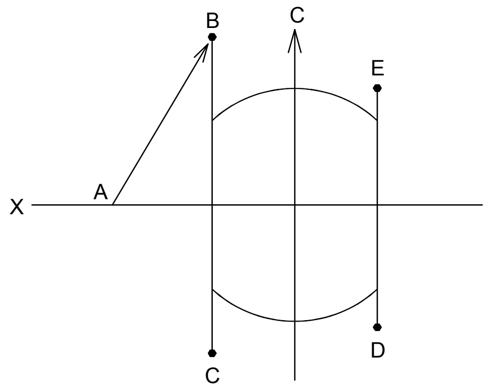
If the machining start position is point K for making the D cut as shown in the following figure, the tool and material interfere with each other in the shaded area when the X axis makes an approach to the workpiece.

To avoid the interference, obtain the machining start position (point B) by calculation.

Point B = Pa position



When performing machining both sides of a workpiece (e.g., 2-face width machining), cancel milling interpolation at completion of machining one side, rotate the C axis 180 degrees, then start machining the other side. If both sides are machined continuously without milling interpolation cancel, the tool and workpiece may interfere with each other.

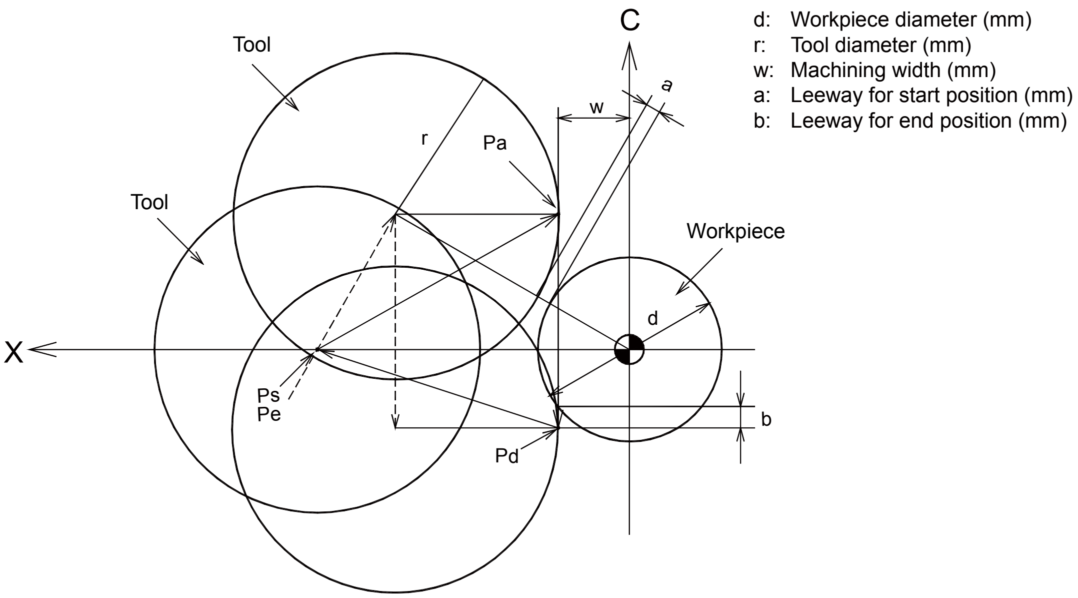


14.9.4 Calculation of milling coordinates

This section describes how to calculate the coordinates used in milling process.

To minimize the move distance, calculate the coordinates based on the leeway for start position (a) and leeway for end position (b).

This calculation for minimizing the approach distance is not always necessary. Taking a long approach distance creates less chances of interference between the tool and the workpiece.



Example: When d = 20 mm, r = 15 mm, w = 8mm, a = 2 mm, and b = 2 mm:

Ps = (63.392, 0) Pa = (8.0, 14.142)

Pd = (8.0, -8.00) Pe = (31.696, 0)

Obtain machining positions suitable for the milling process from the formulas shown in the following table:

Machining position	X axis coordinate	C axis coordinate
Milling start positions Ps = (Xs, Cs)	$Xs > \{ (d/2 + r + a)^2 / (w + r) \} \times 2$	Cs = 0
Machining start position Pa = (Xa, Ca)	Xa = w	$Ca > \sqrt{(d/2 + r + a)^2 - (w + r)^2}$
Machining end position Pd = (Xd, Cd)	Xd = w	$Cd > \sqrt{(d/2)^2 - w^2} + b$
Milling end position Pe = Ps = (Xe, Ce)	$Xe = Xs > (d/2 + r + a)^2 / (w + r)$	Ce = 0

* Xs: Two times the calculated value since it is specified in a diametric value.

[Note]

- The results of the above formulas are free from the plus and minus signs. When changing the results to coordinates, assign signs to them in consideration of the tool position.
- The coordinate represented by Ps is identical to that by Pe.
  The coordinate Xs of Ps is moved before milling interpolation ON is specified. Therefore, be sure to change the coordinate to a value calculated for the diameter when specifying it.

  The other values of Xa, Xd, and Xe are calculated for the radius. The calculated values can be used as they are.
- The description above is an example when the tool diameter compensation is enabled to move the tool from Ps to Pa. If tool diameter compensation function is not used, calculate coordinate for the center of tool, and specify it in the program.

■ Calculation procedure

Calculate the machining positions in the following procedures:

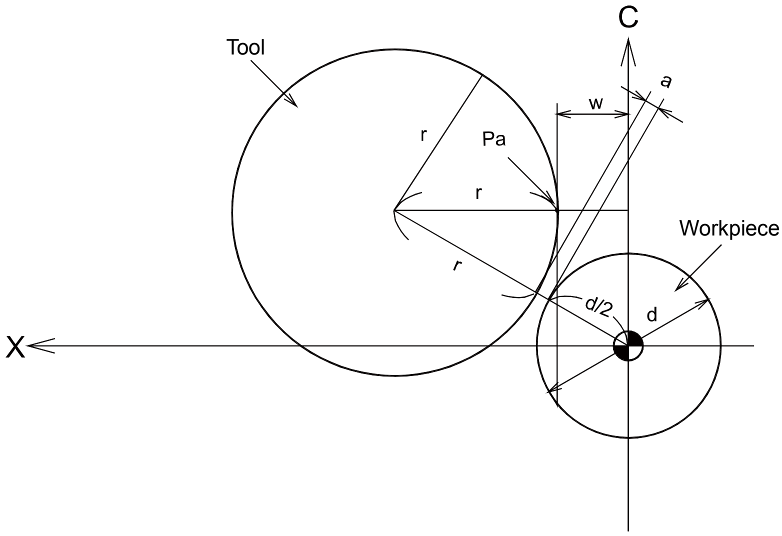
Determine the leeway for start position (a), and calculate the machining start position (Pa).

Based on the machining start position (Pa), calculate the milling start position (Ps), the first positioning point of milling.

Determine the leeway for end position (b), and calculate the machining end position (Pd).

Based on the machining end position (Pd), calculate the milling end position (Pe).

1. Determine the leeway a for the start position shown Fig. 1. Here, take value a as 2.0 mm.



Workpiece diameter d :	20.0
Tool radius r :	15.0
Machining width w :	8.0
Leeway for start position a :	2.0

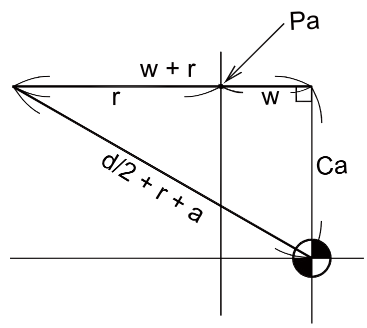


Figure 1

Triangle in Figure 1

Pa = (Xa, Ca)

Xa = w = 8.0

Ca = $\sqrt{(d/2 + r + a)^2 - (w + r)^2}$

$\approx$ 14.142 (radius)

Pa = (8.0, 14.142)

[Note]

The coodinate of Pa is used to calculate the coordinate of the cutting point based on assumption to use tool diameter compensation function (G42, G41, and G40). If tool diameter compensation function is not used, calculate the coordinate in consideration with the tool diameter.

2. The milling start position (Ps) is determined at the center of the circle which moved to position C0.0 parallel to the line between (Pa) to leeway for start position (a). When the tool moves from (Ps) to (Pa) for milling interpolation, the distance calculated here is secured.

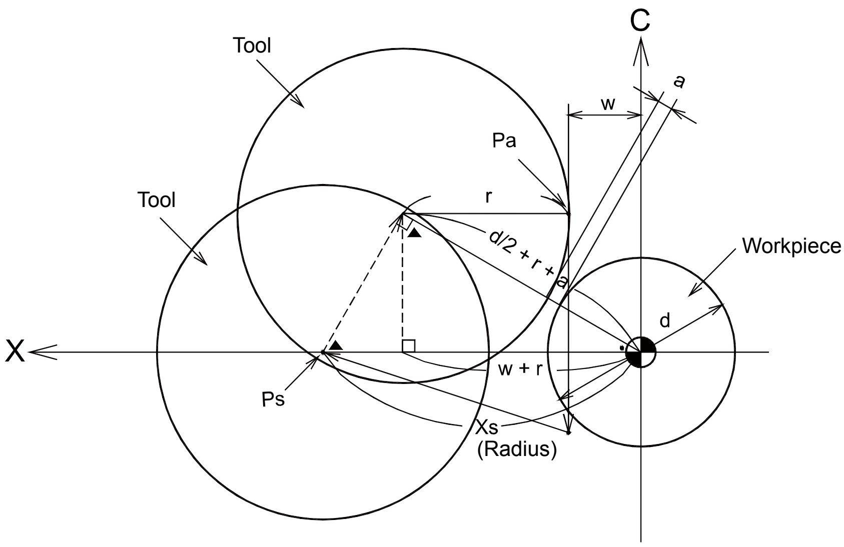
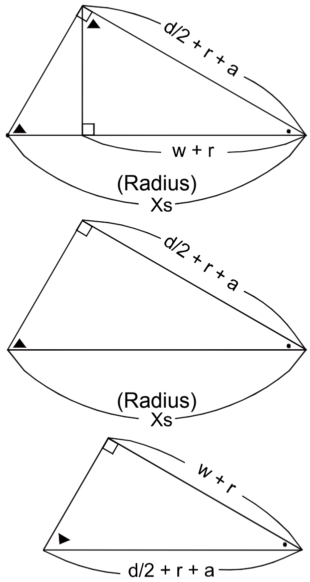


Figure 2



Triangle in Figure 2

Ps = (Xs, Cs)

d/2 + r + a : w + r = Xs : d/2 + r + a

$(w + r)Xs = (d/2 + r + a)^2$

$$Xs\ (\text{Radius}) = \frac{(d/2 + r + a)^2}{(w + r)} \approx 31.696\ (\text{Radius})$$

Diameter is: Xs = 31.696 × 2

= 63.392

Cs = 0

Ps = (63.392, 0)

[Note]

The coordinate (Ps) calculated here is the one before the tool diameter compensation is used. The coordinate of center of milling tool is calculated in diameter value.

3. Determine the leeway b for the end position shown Fig. 3. Here, take value b as 2.0 mm.

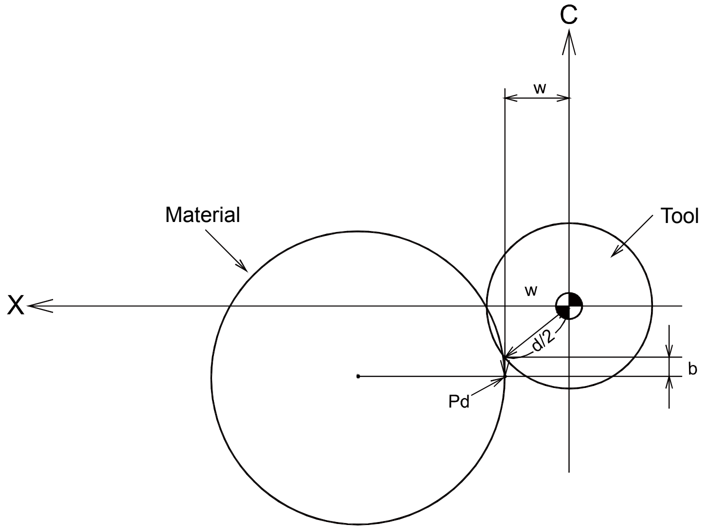
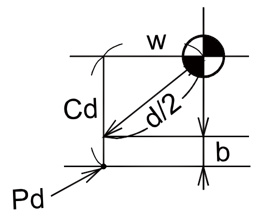


Figure 3



Triangle in Figure 3

Pd = (Xd, Cd)
Xd = w = 8.0
Cd = $\sqrt{(d/2)^2 - w^2} + b$
= 8.0 (calculated value)

* The actual tool is placed in negative area, accordingly the coordinate value becomes Cd = – 8.0.

Pd = (8.0, – 8.0)

[Note]

The coodinate of Pd is used to calculate the coordinate of the cutting point based on assumption to use tool diameter compensation function (G42, G41, and G40). If tool diameter compensation function is not used, calculate the coordinate in consideration with the tool diameter.

4. The milling end position (Pe) is equal to the milling start position (Ps).

Note, however, milling interpolation is active when the tool moves to the milling end position (Pe), the coordinates represents the radius value. The tool diameter compensation is canceled at the same time when the command to move to (Pe) is issued, thus the (Pe) is the coordinate of center of tool as well as in (Ps).

Pe = (31.696, 0)
↑
Diameter value

- Axis movement sequence

Order	Coordinates	Command	Milling interpolation	Tool diameter compensation
1	Ps = (Xs, Cs)	Diameter		
2	Pa = (Xa, Ca)	Radius	Used	Used
3	Pd = (Xd, Cd)	Radius	Used	Used
4	Pe = (Xe, Ce)	Radius	Used	

[Sample program]

The sample program below uses the values calculated from the milling coordinates.

Previous process

```
  :
M5 ....................................................  Spindle stop
G98 M80 S3=[    ] .....................................  Feed per minute. Rotate the rotary tool in forward direction
M18 C0
G50 C0 ................................................  Set C axis coordinate system
```

Milling process

```
T[    ] ...............................................  Milling tool selection
G0 X63.392 Z1.0 C0 T[    ] ............................  Go to milling start position (Ps)
G12.1 .................................................  Milling interpolation ON
G17
G41 G1 X8.0 C14.142 F[    ] ...........................  Tool diameter compensation ON, go to machining start position (Pa)
                                                         (See <14.9.2 Milling plane> and <14.8.2 Tool diameter
                                                         compensation in rotary tool process> for tool diameter
                                                         compensation.)
G1 X8.0 C–8.0 F[    ] .................................  Go to machining end position (Pd)
G40 G1 X31.696 C0 F[    ] .............................  Go to milling end position (Pe)
G13.1 .................................................  Milling interpolation OFF
M82 ...................................................  Rotary tool stop
M20 ...................................................  C axis function cancel
  :
```

14.9.5 Program format

[Sample program]

Previous process	
:	
G98 M80 S3=□	Spindle stop, feed per minute, and rotate the rotary tool in forward direction
M18 C0	
G50 C0	Set C axis coordinate system
Milling process	
N□ T□	Milling tool selection
G50 W□	Coordinate shift
G00 X□ Z□ T□	To the milling start position (Ps) (The Xs value of the milling start position Ps)
G12.1	Milling interpolation ON
G17	Plane selection
G41 G01 X□ C□ F□	Tool diameter compensation ON, and to the machining start position (Pa) (G42 may be executed instead.)
:	
:	
G01 X□ C□	Milling
:	
:	
G40 G01 X□ C□ F□	Tool diameter compensation OFF, and to the milling end position (Pe)
G13.1	Milling interpolation cancel
G50 W□	Coordinate system shift cancel
M82	Rotary tool stop
M20	C axis function cancel

[Note]

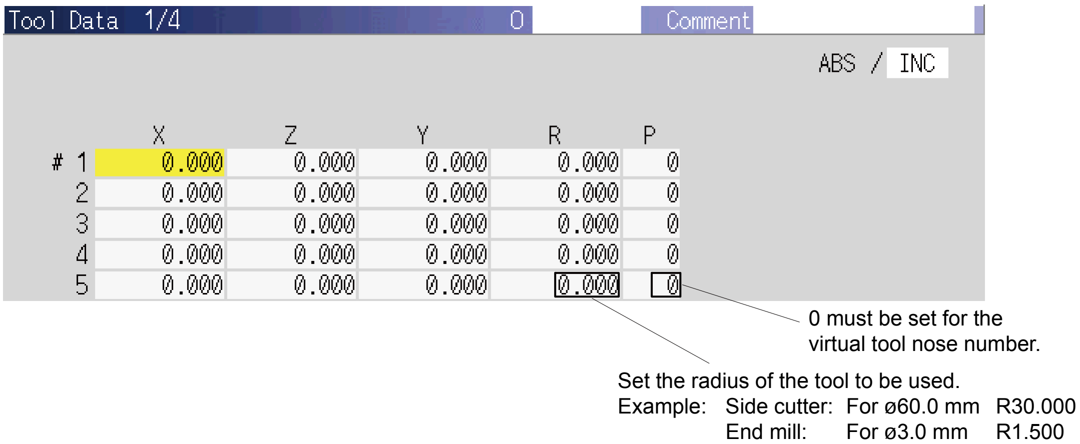
G00 command is disabled while milling interpolation is set to on (G12.1).
Use the G01 command to move an axis.

14.9.6 Supplementary explanation for tool diameter compensation

The tool nose is programmed as the center of a rotary tool for a milling process.

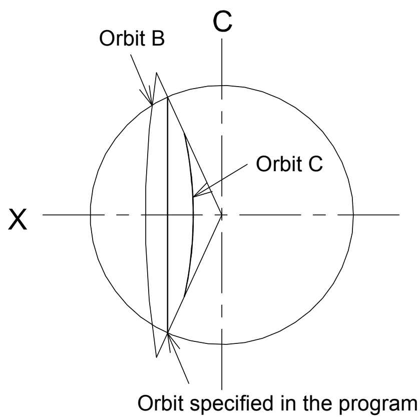
The nose R compensation commands (G40, G41, and G42) are used during the milling process. Thus, be sure to read the explanation of nose R compensation in the <Programming Guide> issued by Mitsubishi Electric Co., Ltd. The following shows the setting of nose R compensation:

■ Example: When T05 is used:



If the coordinate system shift in the diametrical direction of the tool is different from the actual shift, the workpiece is cut in an invalid shape.

For example, if linear interpolation is performed for the end face as shown in the following figure, the workpiece has the externally round shape like orbit B when the coordinate system shift is too large. Conversely, the workpiece has the internally round shape like orbit C when the coordinate system shift is too small. If linear interpolation causes a round shape, the coordinate system shift is invalid. Correct the coordinate system shift.



To compensate the dimensions of a finished workpiece, change the tool data R.

When a dimension of an actually machined workpiece is greater than the specified value, decrease the value specified in the tool data by the error.

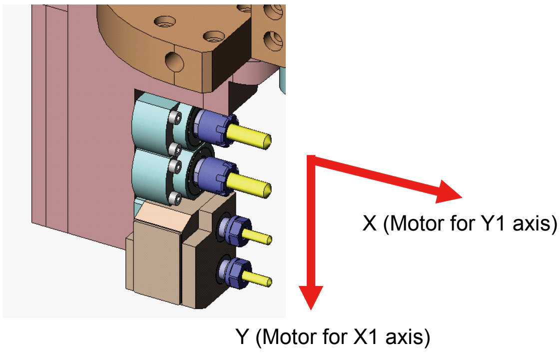
When a dimension of an actually machined workpiece is smaller than the specified value, increase the value specified in the tool data by the error.

14.9.7 Milling interpolation and mirror image

Specifying Milling Interpolation ON command (G12.1) to the motor axis that is subject to milling interpolation while the Mirror Image ON (M211) is enabled causes an alarm to occur. To avoid an alarm from occurrence, specify the Mirror Image OFF command (M212) temporarily.

The following tool layout is applicable:

- 3-tool Cross Machining/ End-face Drilling Spindle (GSE3210) (mounted in cross machining direction)



[Sample program]

$1	
:	
T1200	GSC tool selection
M212 Y1	Mirror image OFF X (motor for Y1 axis)
G50 X☐ Y☐	Coordinate system setting
G0 X☐ Y☐ T☐	Move to milling interpolation start point
G12.1	
:	Machining with milling interpolation ON
G13.1	
M211 Y1	Mirror image ON X (motor for Y1 axis)
G50 X☐ Y☐	Coordinate system setting
T1300	
:	

[Note]

- When the Mirror Image ON/OFF command (M211 or M212) is specified, the plus and the minus signs of the front horizontal axis are reversed.
- After specifying the Mirror Image ON/OFF command (M211 or M212), specify the Coordinate system setting command (G50). Outer circumference milling spindle is used for mirror image, therefore, specify G50 X [material diameter + positioning point] Y0, in normal case. For the offset, specify it when the spindle is moved to the milling start point by the G0 command.
- When machining is complete, specify M211 (mirror image ON), then proceed to the next process.

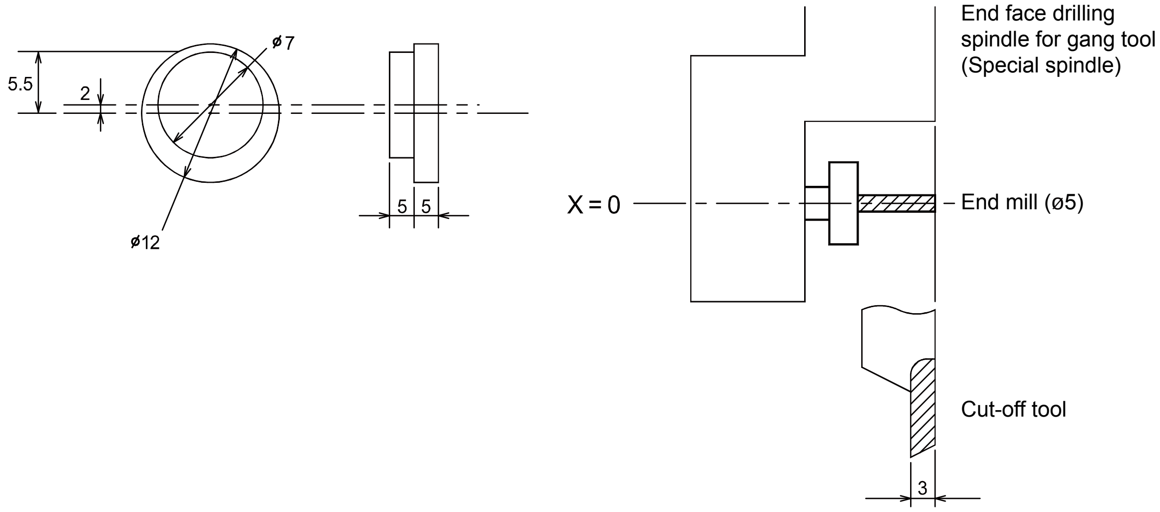
14.9.8 List of NC unit errors during milling interpolation

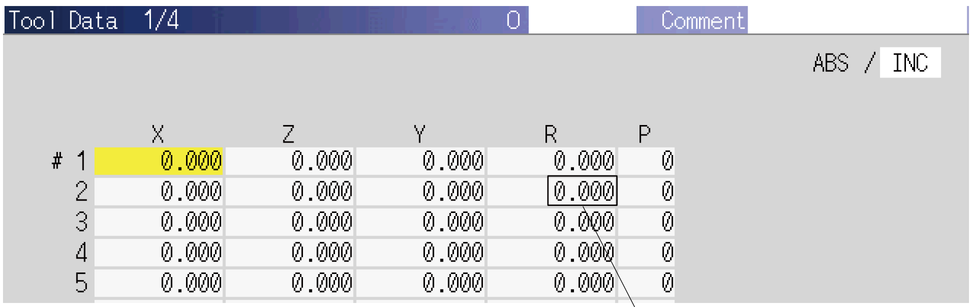
The following table lists alarms which may be issued during automatic operation. A program error is posted if a program creation error occurs or if a program is not created in accordance with the NC specifications.

Error No.	Message displayed on the screen	Description	Procedure
P480	NO. MILL SPEC	The milling command (G12.1 or G13.1) was executed when the milling function was not specified.	Check the specification.
P481	MILL ILL. G	A G code, which must not be executed during milling interpolation, was executed.	Delete the invalid G code.
P482	MILL ILL. AXIS	The command was executed for the rotary axis during milling interpolation.	Delete the command for the rotary axis.
P484	INCOMPLETE RETURN AXIS (MILL)	When the milling interpolation process starts, there is an axis which has not returned to the reference point.	Return the axis to the reference point in manual or automatic mode.
P485	INVALID MODAL (MILL)	When the milling interpolation process starts, tool diameter compensation, constant surface speed control, or the boring cycle is in progress. A T command was executed during milling interpolation.	Execute the cancel command (G40, G97, or G80) before the G12.1 command. Execute the T command before the G12.1 command.

14.9.9 Example of using the milling interpolation function 1

This section shows a program sample for performing milling interpolation with a gang tool. A special spindle is necessary in this case.





Tool Data 1/4					
	X	Z	Y	R	P
# 1	0.000	0.000	0.000	0.000	0
2	0.000	0.000	0.000	0.000	0
3	0.000	0.000	0.000	0.000	0
4	0.000	0.000	0.000	0.000	0
5	0.000	0.000	0.000	0.000	0

Enter the radius of the tool.

Example) When d=12 mm, r=2.5 mm, w=5.5 mm, a=1.0 mm, and b=1.0 mm:

Ps Xs $> (d/2 + r + a)^2 / (w + r)$

$= (12/2 + 2.5 + 1)^2 / (5.5 + 2.5) = 11.281$

11.281×2 (diameter value) $= 22.562 \approx 22.6$

Cs $= 0$ (C coordinate)

Pa Xa $= w = 5.5$

Ca $> \{(d/2 + r + a)^2 - (w + r)^2\}^{1/2}$

$= \{(12/2 + 2.5 + 1)^2 - (5.5 + 2.5)^2\}^{1/2} = 5.12 \approx 5.2$ (C coordinate)

Pd Xd $= w = 5.5$

Cd $> \{(d/2)^2 + w^2\}^{1/2} + b$

$= \{(12/2)^2 - 5.5^2\}^{1/2} + 1 = 3.39 \approx 3.4$ (C coordinate)

Pe Xe $> (w + r) + Cd^2 / (w + r)$

$= (5.5 + 2.5) + 3.4^2 / (5.5 + 2.5) = 9.44 \approx 9.5$

Ce $= 0$ (C coordinate)

[Sample program]

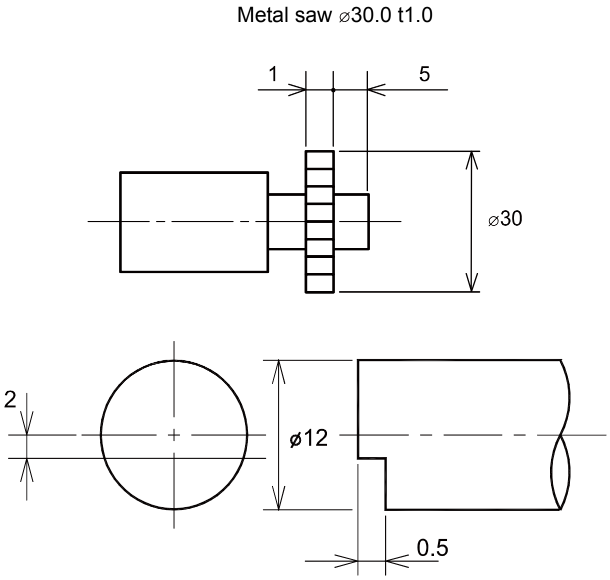
$1		$2
:		
G0 X26.0 Z–0.5		
G0 C0..	Main spindle C axis ON	
G98 M80 S3=2000		
G17..	X-Y plane selection (end face machining)	
:		
N0217 T$\boxed{\quad}$		
G0 X22.6 Z5.0 T02..................................	To the milling start position (Ps), and tool data (tool diameter compensation) call	
G12.1..	Milling interpolation ON	
G41 G0 X5.5 C5.2...................................	Right compensation, and to the machining start position (Pa)	
G1 C0 F60		
G2 X5.5 C0 R–3.5 F60	Milling interpolation machining, machining end position (Pd)	
G1 C–3.4 F300		
G40 G1 X9.5 C0.......................................	Compensation cancel, and to the milling end position (Pe)	
G13.1..	Milling interpolation OFF	
M82..	C axis release, and rotary tool stop	
M3 S1=3000 G99		
:		
G18..	X-Z plane selection (turning)	
:		

[Sample program]

Up cut

$1		$2
:		
G98 M5 M80 S3=1200		
T0700		
G0 X22.6 Z-1.0 C0 T06...........................	Milling start position (Ps)	
G12.1		
G41 G0 X5.5 C5.2..................................	Machining start position (Pa)	
G1 Z5.0 F200		
G1 C0		
G2 X5.5 C0 I–3.5 F100	Milling interpolation machining, machining end position (Pd)	
G1 C–3.4 F300		
G0 Z-1.0		
G40 G1 X9.5 C0.......................................	Milling end position (Pe)	
G13.1		
M82		
:		

14.9.10 Example of using the milling interpolation function 1 (D cut)



Ps: Xs $>((12/2 + 15 + 2)^2 / (15 + 2)) \times 2 = 62.235 \approx 63.0$

Pa: Xa = 2 (from the above figure)
Ca $> \sqrt{(12/2 + 25 + 2)^2 - (2 + 25)^2} = 18.97 \approx 19.0$

Pd: Xd = 2
Cd $> \sqrt{(12/2)^2 - 2^2} + 2 = 7.65 \approx 7.7$

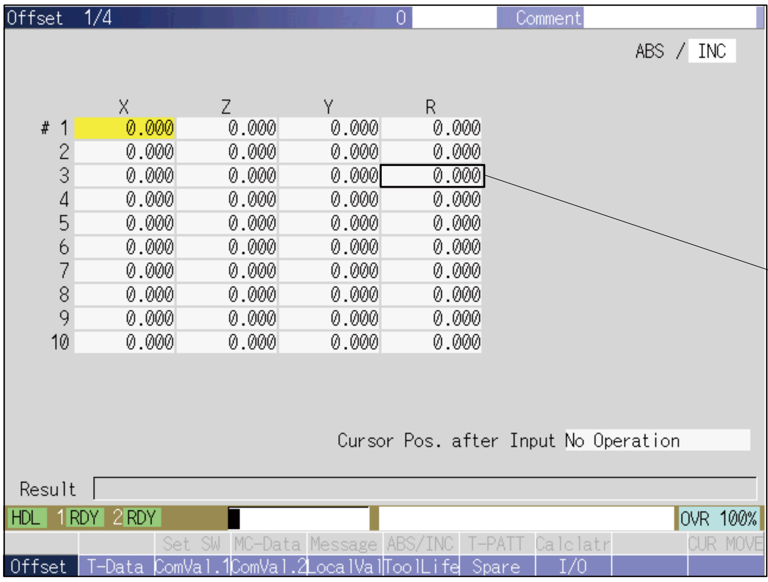
Pe: Xe $> (2 + 15) + (7.7^2 / (2 + 15)) = 20.4876$
Set about 23.0 with leeway added.

[Sample program]

$1

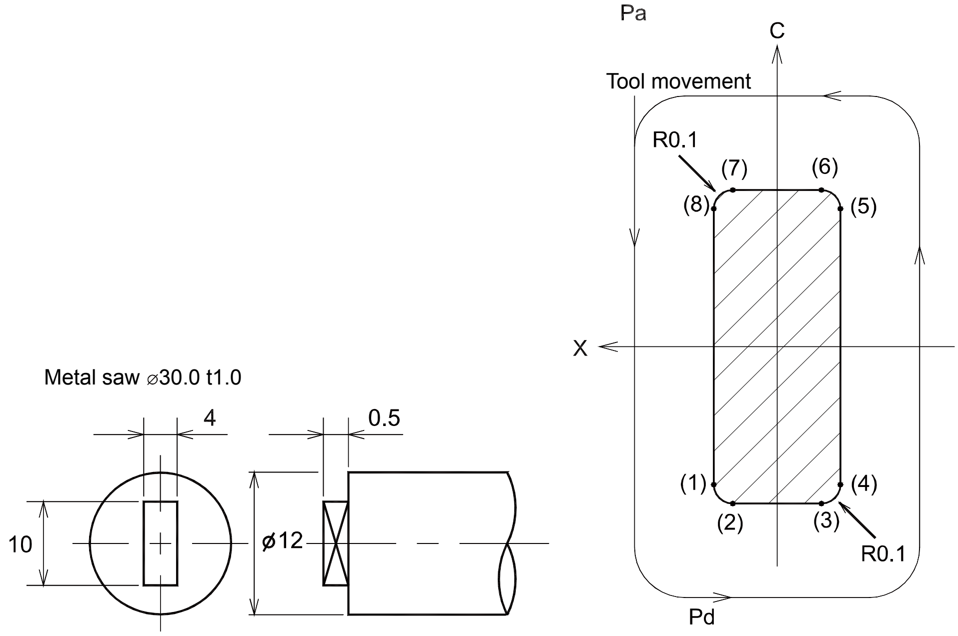
```
  :
M5
G98
M80 S3=1260
M18 C0
G50 C0

T
G50 W–5.0
G00 X63.0 Z0.5 T03 ............................. Ps milling start position
G12.1
G17
G41 G01 X2.0 C19.0 F ................. Pa machining start position
G01 C–7.7 F80 ..................................... Pd machining end position
G40 G01 X23.0 C0 F .................... Pe milling end position
G13.1
G50 W5.0
M82
M03 S1=3000 G99
  :
  :
```



Tool diameter compensation setting 15.0

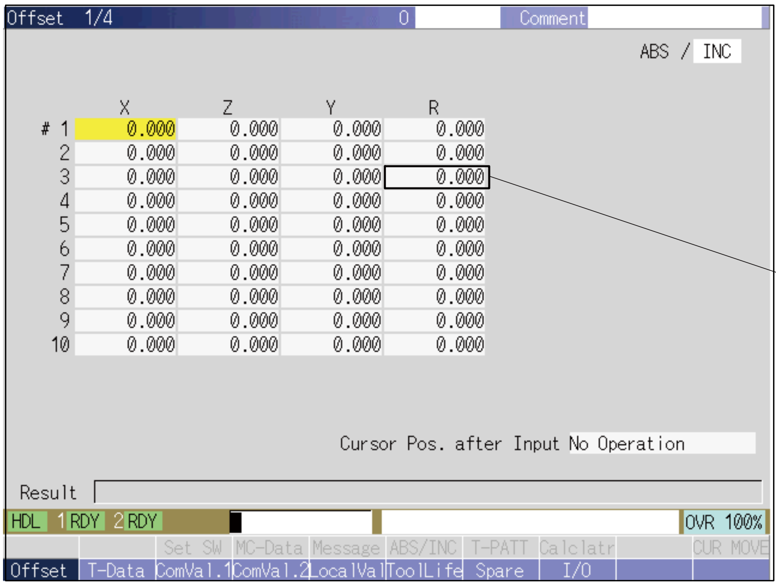
14.9.11 Example of using the milling interpolation function 2



[Sample program]

$1

```
  :
G98
M80 S3=1260
M18 C0
G50 C0
T□
G50 W–5.0
G00 X57.0 Z0.5 T03 ............................... Ps milling start position
G12.1
G17
G41 G01 X4.0 C23.0 F□ .................. Pa machining start position
G01 X2.0 C–4.9 F80 ............................... (1)
G02 X1.9 C–5.0 R0.1 F1200 .................. (2)
G01 X–1.9 F80 ....................................... (3)
G02 X–2.0 C–4.9 R0.1 F380 .................. (4)
G01 C4.9 F80 ......................................... (5)
G02 X–1.9 C5.0 R0.1 F1200 .................. (6)
G01 X1.9 F80 ......................................... (7)
G02 X2.0 C4.9 R0.1 F380 ...................... (8)
G01 X2.0 C–23.0 F500 ........................... Pd machining end position
G40 G01 X96.5 C–23.0 F□ .............. Pe milling end position
G13.1
G50 W5.0
M82
M03 S1=3000 G99
  :
```

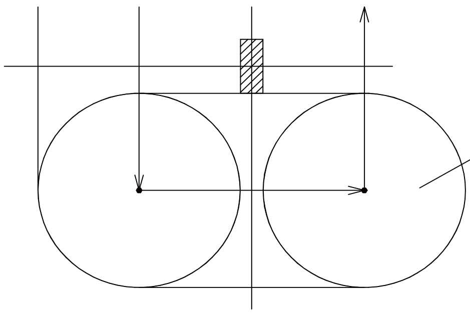


Tool diameter compensation setting
15.0

[Note]

If the R setting value of tool diameter compensation is too large, the cutter makes unnecessary movement when cutting each corner of the workpiece. As a result, machining takes more time than it should. To minimize the machining time and have corners well finished, specify such a value for R that the corners are finely rounded.

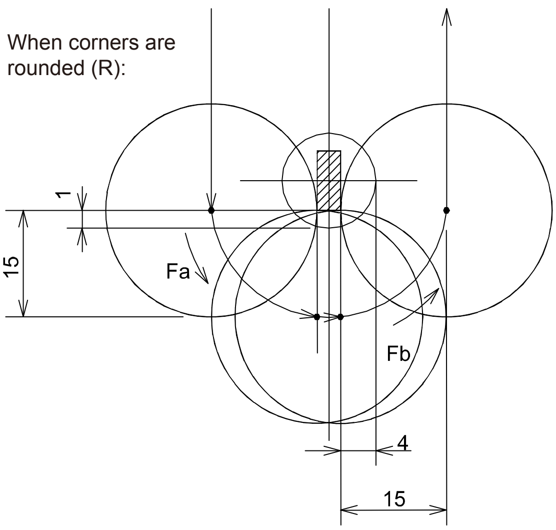
Specification for the cutter to move at right angles



Cutter

The cutter once moves away from the workpiece, and then makes an approach to the workpiece to start machining. This cutter's movement is unnecessary.

When corners are rounded (R):



Cutter's movement

To eliminate the cutter's unnecessary movement when specification is made for the cutter to move at right angles, move the cutter circularly (as if it draws arcs) as shown in the above figure.

The feed rate specified in a program indicates a speed at which the center of the cutter moves, thus differs from the actual feed rate at the machining point in the circular motion. The feed rate to be specified in the program must be calculated.

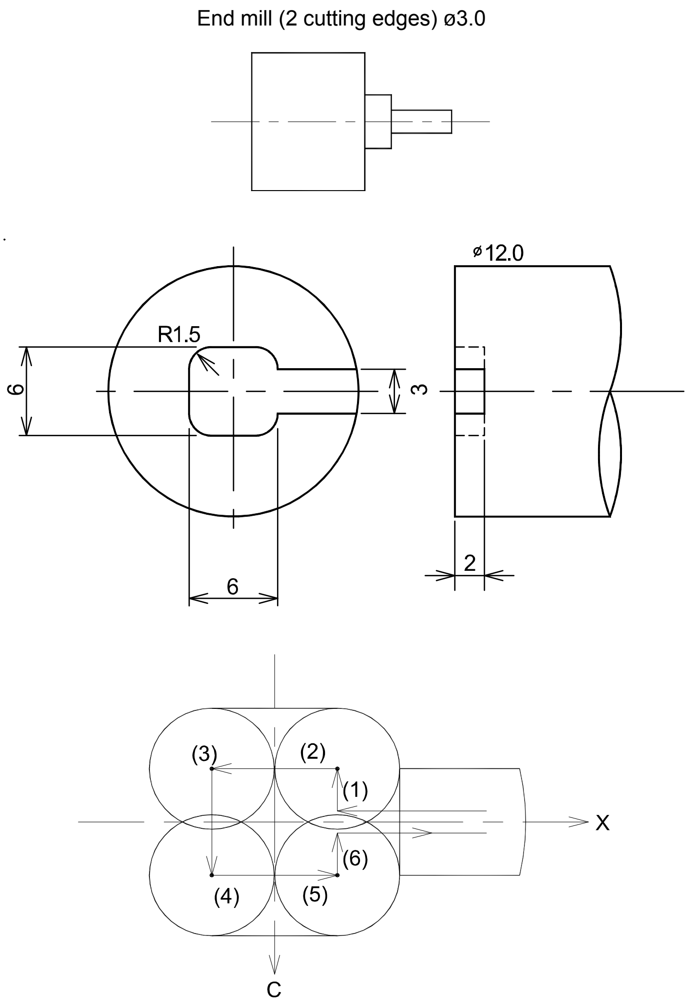
- With an arc circumscribed with the outer diameter of the cutter:
  When the machining feed rate is set as F2 = 80 mm/min, obtain the feed rates Fa and Fb to be specified in programs.

  $$Fa = 80 \times \frac{(1 + 15)}{1} = 1200$$

  $$Fb = 80 \times \frac{(4 + 15)}{4} = 380$$

- The machining feed rate (80 mm/min) is a standard value.
  If you move the cutter circularly to machine corners, the cutter cuts more part of the next machining face as the cutter diameter becomes greater. Decrease the feed rate (calculated in the above) in accordance with the allowance, then specify the decreased feed rate.

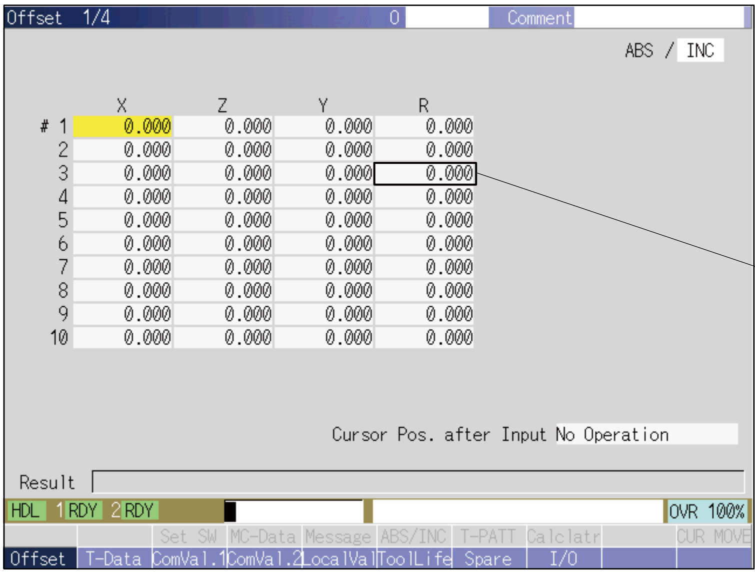
14.9.12 Example of using the milling interpolation function 3



[Sample program]

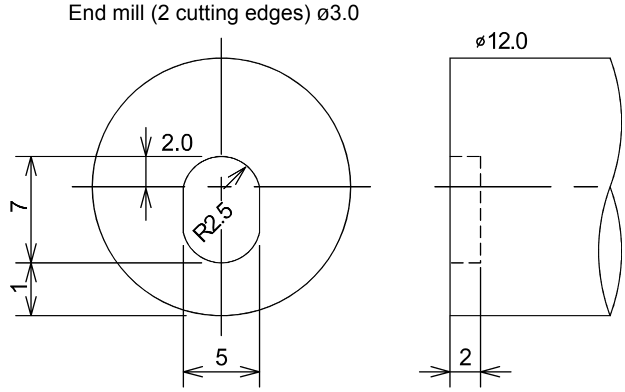
$1

```
  :
G98
M80 S3=2700
M18 C0
G50 C0
N424
T□
G00 X17.0 Z2.0 T04 ................................ Ps milling start position
G12.1
G17
G42 G01 X7.0 C–1.5 F□.................. Pa machining start position
G01 X3.0 F200 ......................................... (1)
C–3.0 ..................................................... (2)
X–3.0 ..................................................... (3)
C3.0 ....................................................... (4)
X3.0 ....................................................... (5)
C1.5 ....................................................... (6)
X7.0 F500
Z–1.0
G40 G01 X7.0 C0 F□
G13.1
G0 U0 V0 W0 T0
M82
M03 S1=3000 G99
  :
```



Tool diameter compensation setting 1.5

14.9.13 Example of using the milling interpolation function 4

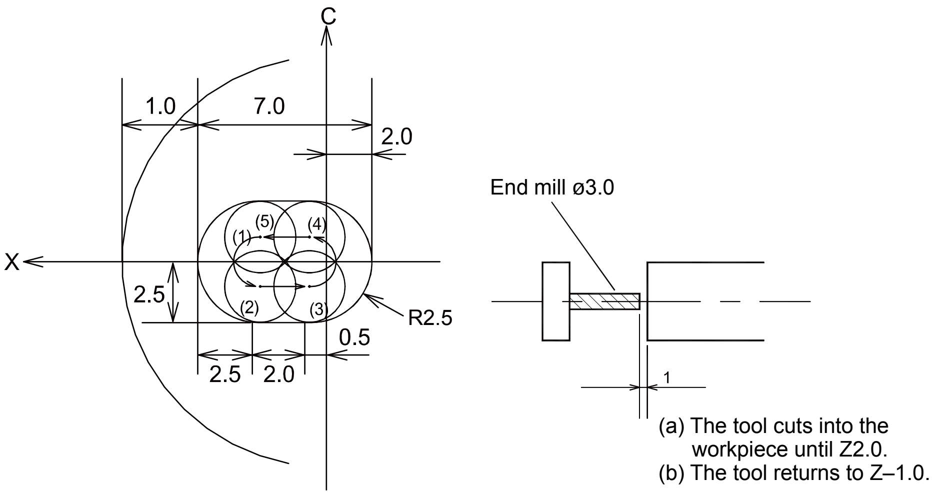


[Sample program]

$1

```
  :
G98
M80 S3=2700
M18 C0
G50 C0

T[    ]
G00 X16.0 Z–1.0 T02 ............................. Ps milling start position
G12.1
G17
G42 G01 X2.5 C2.5 F[    ]..................... (1)
G01 Z2.0 F125 ........................................ (a)
G02 C–2.5 R2.5 F160 ............................. (2)
G01 X0.5 ................................................. (3)
G02 C2.5 R2.5 ........................................ (4)
G01 X2.5 ................................................. (5)
Z–1.0 F1000 ............................................ (b)
G40 G1 U0 W0 F[    ]
G13.1
G0 U0 V0 W0 T0
M82
M03 S1=3000 G99
  :
  :
```



- With an arc inscribed with the inner diameter of the cutter:

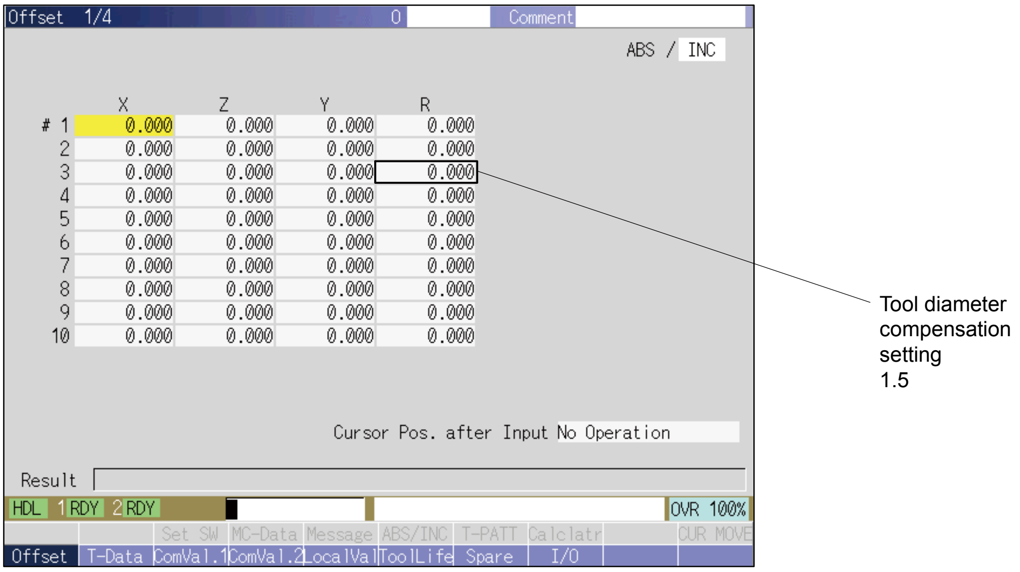
d_1	Cutter diameter	r_1	Cutter radius
d_3	Machining diameter	r_3	Machining radius

F_1 Feed rate specified in the program

F_2 Feed rate at which you want to machine workpieces

- Obtaining the feed rate for the machining diameter:

$$F_1 = F_2 \times \frac{(d_3 + d_1)}{d_3} = F_2 \times \frac{(r_3 + r_1)}{r_3}$$



	X	Z	Y	R
# 1	0.000	0.000	0.000	0.000
2	0.000	0.000	0.000	0.000
3	0.000	0.000	0.000	0.000
4	0.000	0.000	0.000	0.000
5	0.000	0.000	0.000	0.000
6	0.000	0.000	0.000	0.000
7	0.000	0.000	0.000	0.000
8	0.000	0.000	0.000	0.000
9	0.000	0.000	0.000	0.000
10	0.000	0.000	0.000	0.000

The feed rate for the machining diameter (F_2) is set as 100 mm/min.

$$F_1 = 100 \times \frac{2.5 + 1.5}{2.5} = 160$$

Note: The machining feed rate (100 mm/min) is one example.

14.10 Polygon Machining Function (G114.2, G113)

The polygon machining function enables machining for polygonal workpieces by controlling the rotary tool rotation (polygon axis) synchronously with the rotation of the spindle (workpiece axis).

This function also enables the Z axis to move under synchronization control.

CAUTION

Select a tool before specifying polygon machining (G114.2).
Before specifying polygon machining (G114.2), specify zero rotation to the main spindle and the rotary tool.

[Command format]

G114.2 H☐ D±☐ E☐ L☐ R☐	Polygon machining mode ON
M77	Wait for synchronization completion
G113	Polygon machining mode OFF

[Argument]

H☐	Specify the polygon machining axis. 3: Rotary tool on gang tool post, 5: Rotary tool on back tool post
D±☐	Specify the workpiece axis (spindle) and its rotational direction. 1: The main spindle rotates in the same direction as the rotary tool. 2: The back spindle rotates in the same direction as the rotary tool.
E☐	Specify the rotational rate (the number of teeth of the polygon machining cutter) of the polygon machining axis in the range from 1 to 999. The default is 1.
L☐	Specify the rotational rate (the number of corners of a polygon to be machined) of the spindlein the range from 1 to 999. The default is 1.
R☐	Specify the amount (angle) of the phase shift between synchronous spindles. The phase is shifted in the clockwise direction to the main spindle. If no R argument is specified, the phase adjustment is not performed. Setting range: 0 to 359.999°

[Sample program]

Workpiece axis: Main spindle (S1), rotational rate: 1

Polygon axis: Rotary tool on gang tool post (S3), number of teeth: 1, rotational rate: 6

$1	
M3 S1=0	Main spindle zero rotation in forward direction
M80 S3=0	Rotary tool on gang tool post zero rotation in forward direction
T1200...	Select rotary tool
G0 X10.0 Z–5.0	
G114.2 H3 D1 E1 L6 R0	Polygon machining mode ON
M77..	Wait for synchronization completion
S3=600	
:	
G1 Z10.0 F0.1......................................	Polygon machining
:	
G113	Polygon machining mode cancel

[Note]

- Constant surface speed control function (G96) is disabled while the polygon machining mode is active.
- The rotational rate (E and L arguments) must not exceed the maximum speed of the spindle. If the specified value exceeds the maximum, the spindle speed is clamped to the maximum speed.
- You cannot specify the polygon machining mode (G114.2) while the spindle synchronization mode (G114.1 or G814) or the hobbing mode (G114.3) is active.
- The rotation direction of rotary tool may change depending on tool layout or the selected tool number. See *<14.5 Automatic Control of Rotation Direction of Rotary Tool>* for more information.

14.11 Hobbing Function (G114.3, G113)

The hobbing function enables machining for spur gears by controlling C-axis (workpiece axis) rotation synchronously with the rotation of the rotary tool (hob cutter).
In addition, machining for helical gears can be performed by compensating the workpiece axis synchronously with the movement of Z axis.

> **CAUTION**
>
> Select a tool before specifying hobbing (G114.3).
> Before specifying hobbing (G114.3), specify zero rotation to the rotary tool.

[Command format]

G114.3 H☐ D±☐ E☐ L☐ R☐ P☐ Q☐	Hobbing mode ON
G113	Hobbing mode OFF

[Argument]

H☐	Specify the rotary tool (hobbing axis). 3: Rotary tool on gang tool post, 5: Rotary tool on back tool post
D±☐	Specify the workpiece axis (spindle) and its rotational direction. Specify 9.
E☐	Specify the rotational rate (the number of hob cutter's threads) of the hobbing axis in the range from 1 to 999. The default is 1.
L☐	Specify the rotational rate (the number of gear teeth to be machined) of the workpiece axis in the range from 1 to 999. The default is 1.
R☐	Specify the amount (angle) of the phase shift of workpiece axis. The phase is shifted in the clockwise direction to the workpiece. If no R argument is specified, the phase adjustment is not performed. Setting range: 0 to 359.999°
P☐	Specify the helix angle of a gear in the range from –89.000 to +89.000 (–89 to +89°). A spur gear is assumed unless the P argument is specified or if P0 is specified.
Q☐	Specify the gear module. In millimeter: From 100 to 25000 (0.1 to 25 mm). The minimum specification unit is 0.001 mm. In inch:From 100 to 250000 (0.1 to 25 inch-1). The minimum specification unit is 0.0001 inch-1.

[Note]

- The speed of the C axis is determined by the number of hob threads and the number of gear teeth that are specified by the G114.3 command.

  C axis speed SW (min^{-1}) = Sh $\times \frac{E}{Z}$

  Sh: Rotary tool speed (min^{-1})

  E: Number of hob cutter's threads

  Z: Number of gear teeth

- The speed of the C axis is determined by the speed of the rotary tool. Specify the speed of the rotary tool that does not exceed the speed clamp value of the C axis.

  Hobbing speed N_1 (min^{-1}) = $\frac{1000V}{\pi d}$

  C axis speed N_2 (min^{-1}) = $\frac{N_1}{Z}$ < $100min^{-1}$ (upper limit of rapid feed)

  V: Cutting speed (mm/min)

  d: Cutter diameter (φ 32 mm)

  Z: Number of teeth (number of thread ridges)

- The hobbing mode is using the C axis. Therefore, if you want to use the C axis after canceling the hobbing mode, you need to index the C axis and set the coordinate system again.
- You cannot specify the command to move the C axis while the hobbing mode is active.
- You cannot specify the hobbing mode (G114.3) while the spindle synchronization mode (G114.1 or G814) or the polygon machining mode (G114.2) is active.
- Specify P and Q arguments for machining of helical gears.
- To machine the helical gears, specify G98 (feed per minute), instead of G99 (feed per revolution).
- The rotation direction of tool spindle may change depending on tool layout or the selected tool number. See <*14.5 Automatic Control of Rotation Direction of Rotary Tool*> for more information.

14.11.1 Gear formula

Outer diameter of gear ϕD (mm) = $(Z + 2)m$

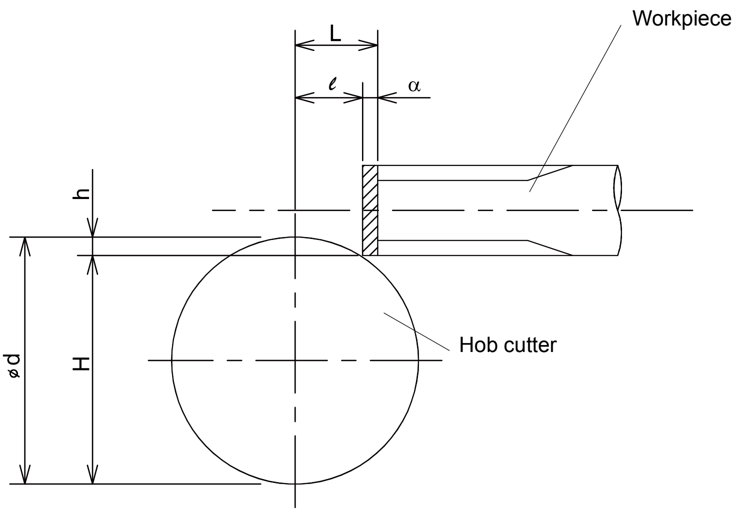
Number of teeth Z (number of thread ridges) = $\dfrac{\phi D}{m} - 2$

Module m = $\dfrac{\phi D}{(Z + 2)}$

Tooth height h = $2m + 0.2$

Inch specification: Diametral Pitch (DP) = $\dfrac{\text{Number of teeth Z}}{\text{Circle diameter of pitch (inch)}}$

When a gear is machined in actual operation, an approach distance is necessary. When calculating the dimensions of a gear, consider the approach distance.



ød : Hob cutter diameter ø32

α : Safety distance 1.5mm

$\ell = \sqrt{(d - h)\,h} = \sqrt{Hh}$

$L = \ell + \alpha$

14.11.2 Cutting condition table (reference)

Rank	Material	Module	Feed rate (mm/rev)	Cutting speed (mm/min)
1	Non-ferrous material (brass, aluminum)	0.5	0.4 to 1.0	150 to 300
		1.0	0.5 to 1.0	150 to 300
2	Free-cutting ferrous material (SUM)	0.5	0.5 to 0.75	80 to 150
		1.0	0.5 to 1.25	60 to 120
3	General ferrous material (SUS303)	0.5	0.5 to 0.75	50 to 90
		1.0	0.5 to 1.25	40 to 80
4	Heavy-cutting ferrous material (SUS304)	0.5	0.5 to 0.75	30 to 60
		1.0	0.5 to 1.25	25 to 50

14.11.3 Example of using the hobbing function

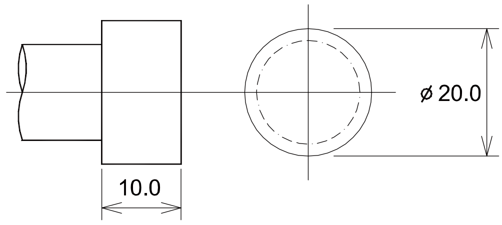
ø20

Brass

Module: m = 1.0

Number of cutting edges: 18

Spur gear



$$C(\min^{-1}) = \frac{\text{Tool spindle speed}}{\text{Number of hob teeth}}$$

$$F = \frac{\text{Tool spindle speed}}{\text{Number of hob teeth}} \times \text{millimeters per revolution feed}$$

(according to *<14.11.2 Cutting condition table (reference)>*)

$$F = \frac{1800}{18} \times 0.6 = 60$$

[Sample program]

$1	
:	
M80 S3=0 G98	Rotary tool on gang tool post zero rotation in forward direction
M18 C0	C axis zero point return
T1200	
G0 X10.0 Z–5.0	
G114.3 H3 D–9 E1 L18 R0	Hobbing mode ON
M77	Wait for synchronization completion
S3=1800	The hob cutter rotates, and the C axis rotates at the same time.
G00 X15.6 Z–9.3 T☐	
G01 Z11.0 F60	Hobbing
X22.0 F500	
G00 Z–9.3	
X15.6	
G01 Z11.0 F300	Zero cut
X22.0 F500	
M82	
G4 U0.1	
G113	Hobbing mode OFF
T☐	
:	
:	

14.12 Coordinate rotation (G950, G951)

Coordinates can be rotated to any angle relative to plane selection. Use inclined helical interpolation via the B axis or the variable tilt spindle.

[Command format]

G950 X☐ Z☐ Y☐ B☐	Coordinate rotation ON
G951	Coordinate rotation OFF

[Argument]

X☐	:	Specify the X-axis workpiece coordinate zero point after coordinate rotation.
Z☐	:	Specify the Z-axis workpiece coordinate zero point after coordinate rotation.
Y☐	:	Specify the Y-axis workpiece coordinate zero point after coordinate rotation.
B☐	:	Specify coordinate rotation angle. An angle of 0 degrees is used unless specified.

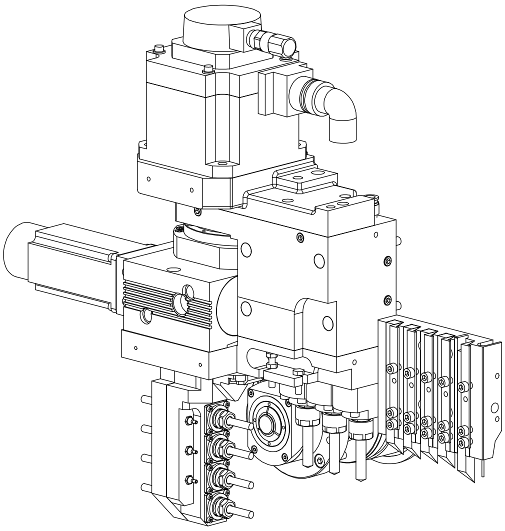
[Note]

- The X, Z and Y arguments of the coordinate rotation (G950) command can only be specified for a plane selected prior to command selection.
- After B-axis mode is turned on (G900) when B-axis tools are used, plane selection is locked in the XZ plane (G18) until the coordinate rotation (G950) command is executed. To specify helical interpolation, specify plane selection after executing the coordinate rotation (G950) command.

14.13 B-axis Function

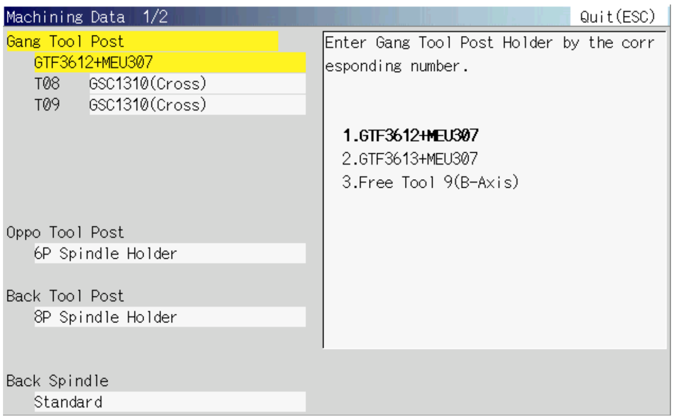
A machine equipped with a B-axis unit (type VIII) can be positioned for machining at any angle on the B-axis. Program commands make it possible to use rotary tools on a B-axis unit for cross machining, tilt machining and end-face machining at any angle position.

Contour machining with 4-axis interpolation (X, Z, C, B) is possible. Programming for contour machining requires use of a third-party CAM system.



14.13.1 Machining data setting

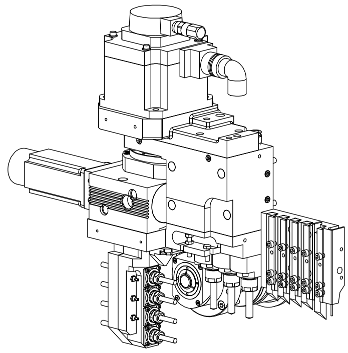
Specify the front drill holder name in the machining data for the B-axis holder of a rotary tool equipped with the B-axis function.



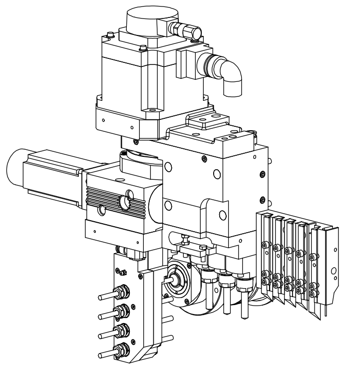
14.13.2 B-axis tool angle

The B-axis angle is 90 degrees in cross direction and 0 degrees in end face direction. The B-axis coordinate system for B-axis front machining (T11's) and back machining (T51's) is reversed.

■ B-axis front machining (T11's)

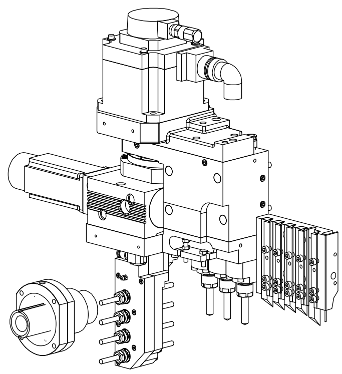


B-axis angle is at 90 degrees

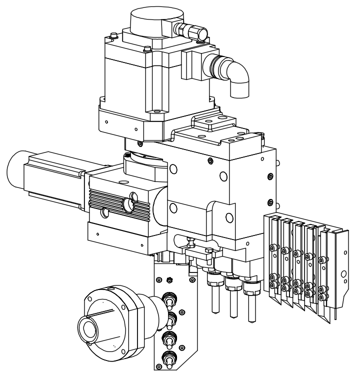


B-axis angle is at 0 degree

■ B-axis back machining (T51's)



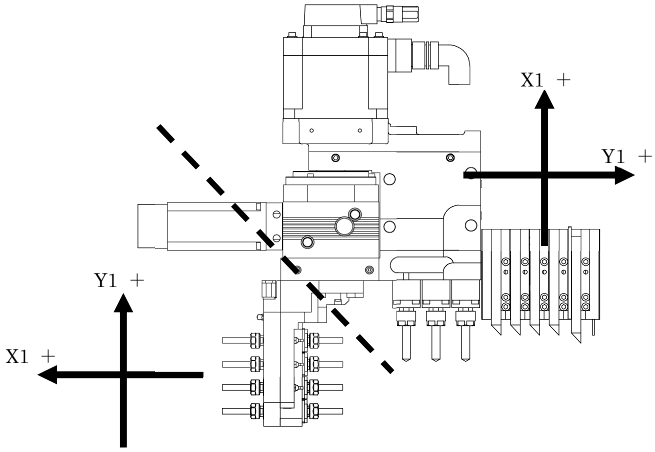
B-axis angle is at 0 degrees



B-axis angle is at 45 degree

14.13.3 B-axis tool coordinate system

In gang tool (T0100 - T1000) machining, diametrical infeed occurs along the X-axis and core infeed occurs on the Y-axis. Also on in B-axis tool (T1100 - T1400, T5100 - T5400) machining, diametrical infeed occurs along the X-axis and core infeed occurs on the Y-axis. However, when B-axis mode is on, infeed occurs along the Z-axis.



14.13.4 Commands and program examples for B-axis control

This section describes the command codes and program methods for using a B-axis tool to machine at angles along the B-axis.

■ G900/G901 B-axis mode ON/OFF

This command is used to program the tip of the B-axis tool as a supporting point. Enter the offset number (T2 digit) of the B-axis tool for machining, then run the G900 command to make the tool tip a program point. Run tool number commands when B-axis mode is in off status. Refer to <*Program examples of diagonal drilling using B-axis tools*> for information on programming.

[Command format]

T□□	Offset number command
G900 X□ Z□ Y□ B□	B-axis mode ON
G901 B□	B-axis mode OFF

[Argument]

X, Z and Y arguments :	Specify the coordinates for tip movement in the work coordinate system before turning on B-axis mode. You must specify argument X and Z. If not specified, an alarm occurs.
B arguments :	Specify B-axis rotation angle (in 0.001 degree increments). If omitted, the current B-axis angle position is used.

Machining	L220
Front machining (T11's)	–45.000(-25.000)～90.000
Back machining (T51's)	–6.000～45.000(25.000)

[Note]

- The B argument enables sub-micron (sub-inch) specifications in 0.0001 degree increments.
- Specify an offset number (T2 digit) before running a G900 command. If not specified, an alarm occurs.
- You can also enter numeric characters in the last two digits and specify an offset number in a T code 4-digit command.
- Tool selection is not available in G900 mode. You must specify B-axis mode cancellation in G901 during tool replacement.
- Using T code, machining patterns, superimpose control and spindle synchronization control commands in G900 B-axis mode will generate an alarm.
- B-axis mode is turned off at the current B-axis angle position when the B argument is omitted during G901 specification.
- $1 axes of axis control groups in B-axis mode are the five axes of X1, Z1, Y1, C1 and B1, but synchronous specification in one block is possible for four axes. It is possible to select any of the four axes that will be used, but specifying a fifth axis will generate an alarm.
- The B code command address is "B" ⇒ "MB".
- Although specifications for back machining (T51's) can theoretically be made down to the lower limit of -90.000 for the B argument range when B-axis mode is on (G900), any specification below -6.000 will generate a Y1-axis stroke end alarm.
- When an end face drilling spindle (BSE***) is installed on gang tool T09, the B-axis machine coordinates are limited to between 25.000 - 90.000 degrees.

■ G950/G951 Turning workpiece zero point specification on/off for each axis in the coordinate rotation and rotation coordinate system.

Specify this command to enable machining at any angle using B-axis tools. Use this command to incline the plane to the B-axis angle to enable operation of a machining program on an inclined plane.

[Command format]

G950 X□ Z□ Y□ B□	Specifying work coordinate zero point during or after coordinate rotation
G951	Canceling work coordinate zero point specification during or after coordinate rotation

[Argument]

X, Z and Y arguments :	The workpiece coordinate position specified by an argument becomes workpiece zero point "0" after coordinate rotation.
B arguments :	Specify coordinate rotation angle (in 0.001 degree increments). If omitted, the current B-axis angle is used as rotation angle.

Machining	L220
Front machining (T11's)	–45.000(-25.000)～90.000
Back machining (T51's)	–6.000～45.000(25.000)

[Note]

- The B argument enables sub-micron (sub-inch) specifications in 0.0001 degree increments.
- Command specified axis/offset axis for the direction of the tip of the tool becomes the Z-axis. For details, refer to *<Regarding command specified axes and offset axes of B-axis tools>*.
- When B = 0, X and Y axes specify the diameter and when B = 0, they specify the radius values for all axes. Parameter settings make it possible to change diameter specification regardless of the B argument value.

[Procedure]

Choose Parameter screen → menu key [control parameter] → No. 8162 "G176 diameter specification enable" to set "1" and the X and Y axes specify the diameter.

- Tool selection is not available during G950 coordinate rotation. You must use G951 to turn off coordinate rotation before specifying tool numbers. Use G901 to turn off B-axis mode while G900 B-axis mode is on.
- The G950/G951 coordinate rotation can be turned on/off also when the B-axis function is not available. This is useful during machining using variable tilt spindles and other fixed tilt tools.
- To specify G950 coordinate rotation in G900 B-axis mode, make the specification when the G18 (XZ) plane is selected. Making a specification in any other plane will generate an alarm. To switch to any other plane, use the G950 command before specifying. For example, to specify helical interpolation of an XY plane in G900 B-axis mode, specify G950 coordinate rotation before selecting the G17 (XY) plane.
- The X, Z and Y arguments of G950 require that the axis of the selected plane is specified first. The X and Z arguments are mandatory for a G18 (XZ) plane, the X and Y arguments are mandatory for the G17 (XY) plane and the Y and Z arguments are mandatory for the G19 (YZ) plane. If not specified, an alarm occurs. Specify the plane before running the G950 command. However, when the G900 B-axis mode is on, the G950 command can only specify the G18 (XZ) plane.
- Although specifications for back machining (T51's) can theoretically be made down to the lower limit of -90.000 for the B argument range when B-axis mode is on (G900), any specification below -6.000 will generate a Y1-axis stroke end alarm.
- When an end face drilling spindle (BSE***) is installed on gang tool T09, the B-axis machine coordinates are limited to between 25.000 - 90.000 degrees.

Coordinate system for B-axis tools after specifying G950 coordinate rotation

The G950 coordinate rotation command allows you to machine in tool nose direction using 1-axis commands without having to use 2-axis interpolation programming. At this time, regardless of the position of the B-axis angle, the Z-axis is the command specified axis for tool nose direction.

For example, if the B-axis turns at 90 degrees, the tip of the tool is pointed in cross direction and also then the Z-axis is the command specified axis for cross direction.

If G950 coordinate rotation has not been specified, the command specified axis for cross direction is the motor axis (X-axis).

If G950 coordinate rotation is specified

Set B-axis angle to 60 degrees and adjust coordinate rotation specification to B-axis angle and the X and Z-axes will take the shape shown in Figure 1.

To drill a 3.0 mm deep hole in the tapered section of a workpiece,

:

(position the tool at a 1 mm distance from the end of the tapered section of the workpiece) Specifying G98 G1 Z3.0 F200

:

will interpolate the motor axis (X, Z-axes) along with the B-axis and the tip of the tool moves in tool nose direction.

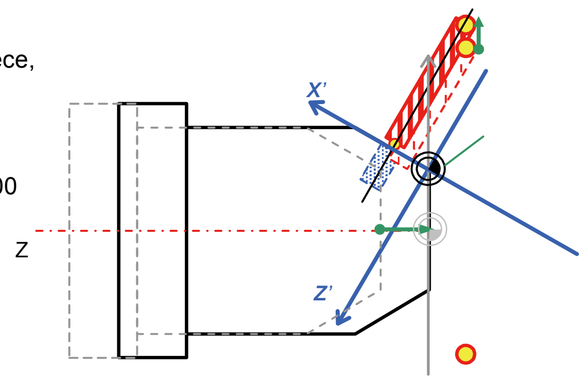


Figure 1

If G950 coordinate rotation is not specified

B-axis angle is tilted at 60 degrees, but since coordinate rotation is not performed, the X and Z-axes face the direction of the motor axis as shown in Figure 2.

To drill a 3.0 mm deep hole like above,

:

(position the tool at a 1 mm distance from the end of the tapered section of the workpiece) G98 G1 U-6.928 W2.0 F200

:

The coordinate calculation for progress at an angle of 60 degrees using the X and Z commands, the tip of the tool will move in tool nose direction.

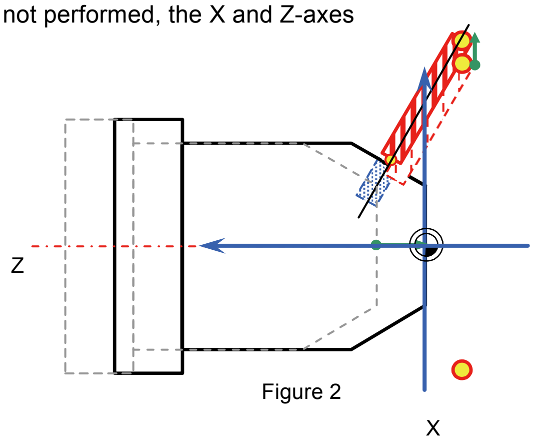


Figure 2

■ G910 independent B-axis tool rotation

This function is used to turn the angle of B-axis tools without turning on the B-axis mode.

[Command format]

G910 B□ D1

[Argument]

B arguments : Specifying B-axis rotation angle (in 0.001 degree increments).

Machining	L220
Front machining (T11's)	–45.000(-25.000)～90.000
Back machining (T51's)	–6.000～45.000(25.000)

D1 arguments : B-axis rotation occurs first. Moves in parallel according to the main system commands.

CAUTION

Since the G910 command does not rotate around the tool tip supporting point but around the center of the B-axis, care must be taken to prevent interference with the workpiece.

[Note]

- The B argument enables sub-micron (sub-inch) specifications in 0.0001 degree increments.
- B-axis rotation by G910 can only be performed using absolute value commands.
- When using the D1 argument, B-axis rotation occurs first and B-axis rotation is performed in parallel with operations in the next block. When the D1 argument will not be used, operation continues from the next block after completing rotation.
- Although specifications for back machining (T51's) can theoretically be made down to the lower limit of -90.000 for the B argument range when B-axis mode is on (G900), any specification below -6.000 will generate a Y1-axis stroke end alarm.
- When an end face drilling spindle (BSE***) is installed on gang tool T09, the B-axis machine coordinates are limited to between 25.000 - 90.000 degrees.

■ Switching G190/G191 B-axis absolute/relative value commands

Separate addresses to run absolute/relative value commands like for the X, Z, Y and C axes are not provided for the B-axis. For this reason, it is necessary to use G code G190/G191 to switch between absolute/relative value commands for the B-axis. In initial state (at power up), absolute value commands are used. G190/G191 are modal G codes and modal status can be confirmed on the Modal Information Screen.

The B-axis angle position can be confirmed in the Automatic Operation screen or the Position Data Screen.

Modal Information Screen

[Command format]

G190 Absolute value commands for B-axis

G191 Relative value commands for B-axis

Absolute/relative value command address for each axis

Each axis	Address	
	Absolute value command	Relative value command
X-axis	X	U
Z-axis	Z	W
Y-axis	Y	V
C-axis	C	H
B-axis	B (after specifying G190)	B (after specifying G191)

The absolute/relative value command address of the B-axis is "B".

[Sample program]

G900 X33.0 Z-1.0 B0 ; .... B axis mode is on

G0 B45.0 ; .... Absolute value movement. Position at a B-axis angle of 45 degrees

G191 ; .... B-axis relative value (incremental) command on

G0 B15.0 ; .... Relative value movement. Positions the tool +15.0 degrees (absolute value: 60 degrees) from the current position

.

.

.

G190 ; .... B-axis absolute value command on

G0 B20.0 ; .... Absolute value movement. Position at a B-axis angle of 20.0 degrees

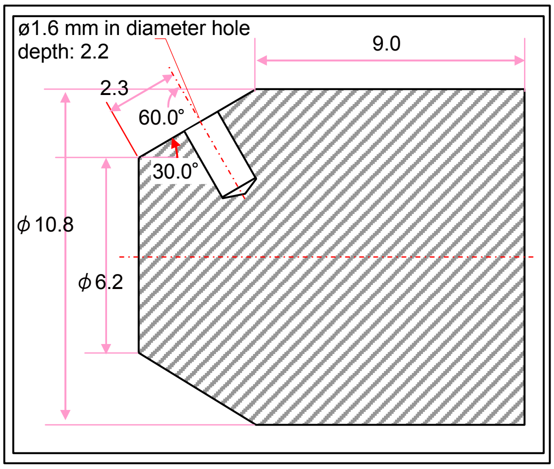
G901 B90.0 ; .... B axis mode is off

[Note]

B-axis command is enabled when the G900 B-axis mode is on. To rotate B-axis angle when B-axis mode is off, use the G910 B-axis tool independent rotation.

■ Program examples of diagonal drilling using B-axis tools

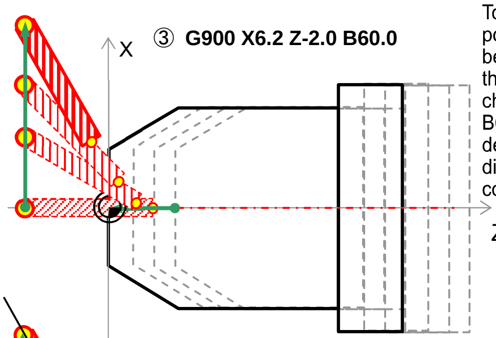
Workpiece drawing (diagonal drilling in end face direction)



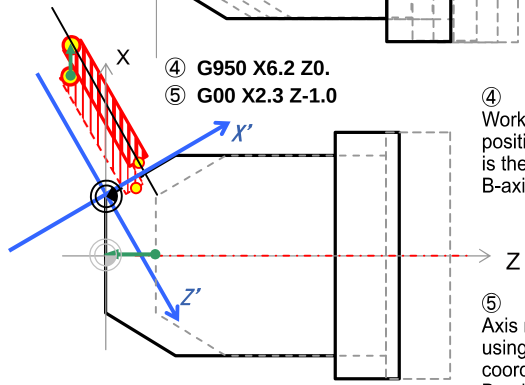

```
O100 (TEST)
G50 Z0
:
① T1100..........................................(Tool number selection)
② G00 Z-2.0 T11........... (Tool length number selection)

③ G900 X6.2 Z-2.0 B60.0 ................(B axis mode is on)
④ G950 X6.2 Z0.(B60.0)
                         (Specifying workpiece zero
                         point in the coordinate
                         rotation and rotation
                         coordinate system)
⑤ G00 X2.3 Z-1.0
                  (Movement using the radius specification
                        and B-axis coordinate system)
⑥ G83 Z2.2 P0.5 F100
⑦ G00 Z-2.0          (Movement using B-axis coordinate
          .........                                  system)
⑧ G951         (With the coordinate rotation and rotation
                 ..................................coordinate system off)
⑨ G901 B0.                            (B axis mode is off)
T1200
:
```

* ⑥, ⑦ and ⑧ are radius values for all axes after the G950 command in step ④.

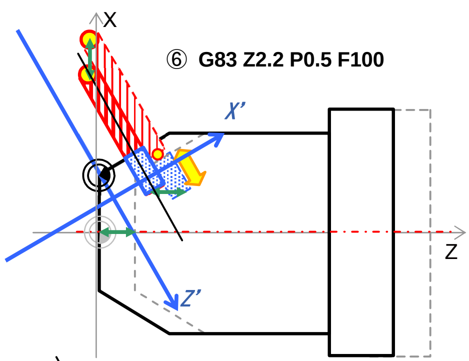


③
To make sure that the positional relationship between the tool tip and the material does not change to X6.2 Z-2.0 B60, turn the B-axis 60 degrees (with tool direction and tool length compensation on).

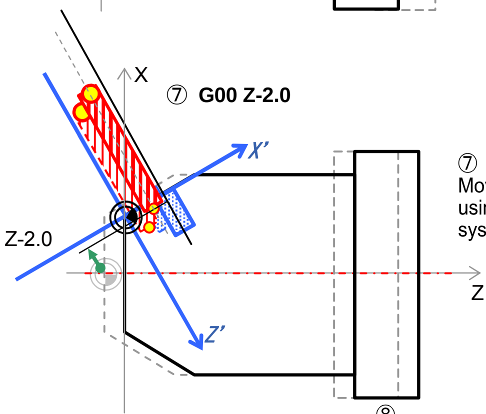


④
Workpiece zero point position (X6.2 Z0 position is the zero point) after B-axis rotation.

⑤
Axis move command using the workpiece coordinate system after B-axis rotation.

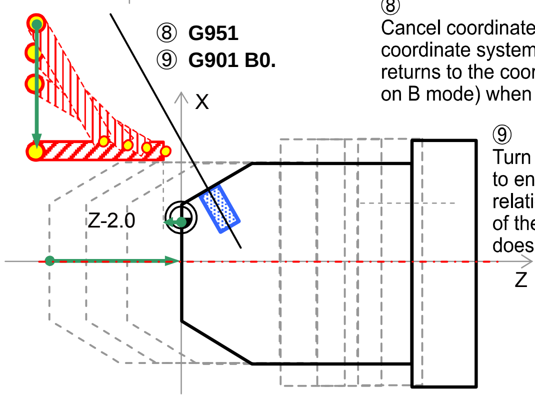


⑥
Executing a diagonal canned drilling cycle.



⑦
Movement to Z-2.0 using B-axis coordinate system.

⑧
Cancel coordinate rotation and coordinate system (the zero point returns to the coordinates prior to turning on B mode) when the B-axis mode is on.



⑨
Turn the B-axis to 0 degrees to ensure that the positional relationship between the tip of the tool and the material does not change.

14.13.5 Command specified axes and offset axes of B-axis tools

■ Regarding command specified axes and offset axes of B-axis tools

In G900 B-axis mode, the command specified axis and offset axis of infeed direction changes depending on whether or not G950 coordinate rotation is used.

List of command specified axis and offset axis of infeed direction for B-axis tools

Tool names when setting tools	G900 B-axis mode	G950 Coordinate rotation	B-axis machining angle	Command specified axis for infeed direction	Offset axis for infeed direction	Pattern
B-axis cross machining (B-axis 90 degrees)	ON	ON	–46.000～90.000°	Z-axis (B-axis angle direction)	Z-axis (B-axis angle direction)	①
		OFF	0.000°	Z-axis	Z-axis	②
			–46.000～-0.001 , 0.001～89.999°	X, Z-axis	X, Z-axis	③
			90.000°	X-axis	X-axis	④
	OFF	OFF	90.000°	X-axis	X-axis	⑤
B-axis end face drilling (B-axis 0 egree)	ON	ON	–46.000～90.000°	Z-axis (B-axis angle direction)	Z-axis (B-axis angle direction)	⑥
		OFF	0.000°	Z-axis	Z-axis	⑦
			–46.000～-0.001 , 0.001～89.999°	X, Z-axis	X, Z-axis	⑧
			90.000°	X-axis	X-axis	⑨
	OFF	OFF	0.000°	Z-axis	Z-axis	⑩

* Regarding command direction along the X, Z and Y axes, patterns ② - ④ and ⑦ - ⑨ operate using a rectangular coordinate system regardless of B-axis angle while patterns ① and ⑥ operate in B-axis angle direction.
* Tool length (when the tool name is set as "B-axis cross machining" it refers to a "diameter" value and when it is set as "B-axis end face drilling" it refers to "longitude" value and is used as tool length) when G900 B-axis mode is on is reflected to nose direction.

■ How to machine in tool set direction without turning the B-axis angle

To set tools, select cross direction (90 degree B-axis angle) or end face direction (0 degree B-axis angle) in the Preparation screen. For details, refer to the "Operator's Manual". When the workpiece that will be machined does not require B-axis rotation during tool setting, you can use B-axis tools in the traditional manner without having to use the B-axis mode or coordinate rotation commands described so far. That is operation using patterns ⑤ and ⑩ in the list above.

14.13.6 Offset for B-axis tools

■ Offset

You can also place the offset number in the last two digits of TOOΔΔ to call tools. For information on the direction for entering offsets (axis), refer to the list of *<Regarding command specified axes and offset axes of B-axis tools>*.

Offset cancel (T0 command) cannot be specified in G900 B-axis mode or when G950 coordinate rotation is on. Be sure to turn G901 B-axis mode off and G951 coordinate rotation off before canceling offset. Attempting to cancel offset when G900 B-axis mode and G950 coordinate rotation are on will generate an alarm.

■ Offset number changes during B-axis mode

To change the offset number in B-axis mode do not change when G950 coordinate rotation on. As shown in the following program example, turn off G951 coordinate rotation, change the offset number and then specify G950 again. To enable only compensation without moving the X and Y axes, use the incremental command to specify a move distance of 0.

[Sample program]

T1100
G0 **** T11 .... Specifies offset numbers
G900 X** Z** B60.0.............................. B axis mode is on
G950 X** Z** (B60.0) .... With the coordinate rotation and rotation coordinate system on
Coolant process
G951 .... With the coordinate rotation and rotation coordinate system off
G0 U0 W0 T12..................................... Changes offset number
G950 X** Z** (B60.0) .... With the coordinate rotation and rotation coordinate system on

14.13.7 How to avoid problems in B-axis mode

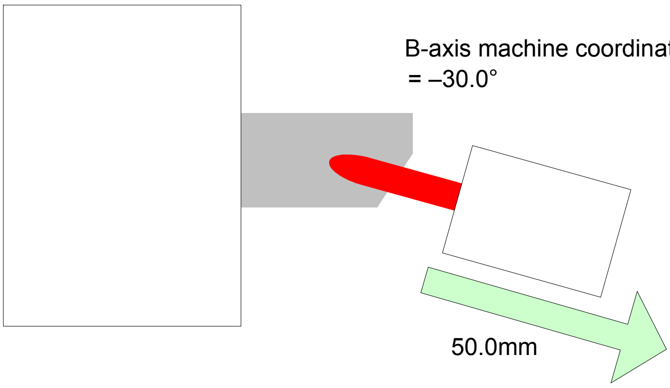
■ B-axis front machining (T11's)

If, for some reason, operation stops during front machining using a B-axis tool, and program operation is reset, confirm that the "motor axis" menu key is not selected in the Handle Feed screen. If not set, it will be possible to move the tool in B-axis angle direction using Z-axis movement.

For example, if program operation is reset while using B-axis tools to drill at a B-axis angle of 45 degrees, this method enables a reset without damaging tool tips. For details, refer to the "Operator's Manual".

■ B-axis back machining (T51's)

If, for some reason, operation stops during back machining using a B-axis tool, and program operation is reset, use MDI program operation to remove the tool from the workpiece, instead of using the Handle Feed screen in B-axis front machining (T11's) like in the example above.



[Sample program]

To move the B-axis back machining tool 50 mm away from the workpiece in the above condition (B-axis machine coordinates -30.0)

M88 .... Interference Check Off
G140 Z=Z2 Y=Y1 .... Change function
G98 G1 Ww Vv F $\boxed{}$............... Return the Z2Y1 axis
:

w = – (50.0mm × cos30°) = –43.301

v = 50.0mm × sin30° × 2 (Diametrical value) = 50.0

14.13.8 Precautions to be heeded during B-axis tool use

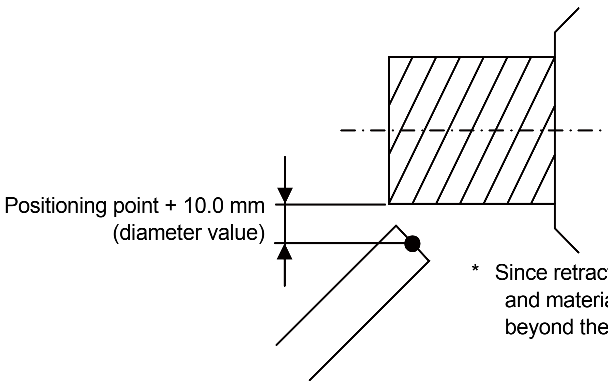
■ Precautions to be heeded during B-axis tool selection

When a tool is selected at a B-axis angle of 0 degrees, the center of the tool moves to the center of the end face of the material since the tip of the tool faces the end face direction. The material must be called at a position away from program zero point. If the material is called when it protrudes, there is a risk of interference with the tip of the tool.

When B-axis angle is between -46.000 and -0.001 degrees or 0.001 and 89.999 degrees, it is assumed that the tip of the tool is moving in cross direction. After the X-axis moves 10.0 mm beyond the positioning point of the machining data (diameter value), the tool is positioned in Y-axis direction.

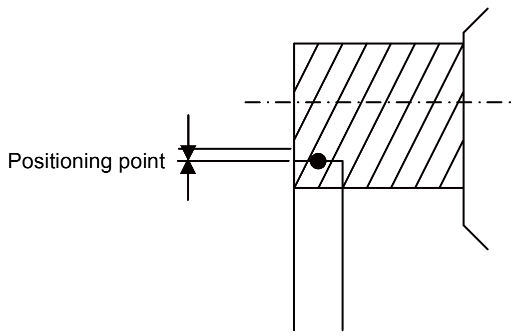
When at a 90 degree angle of the B-axis (cross direction), the tool is positioned at the positioning point.

A tool selection position at a B-axis angle of -46.000 to -0.001 degrees, or 0.001 to 89.999 degrees (the figure below shows a 45 degree angle)



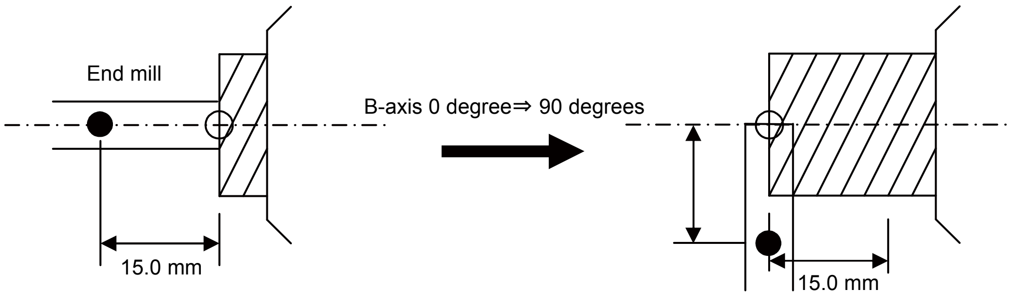
* Since retracting to the positioning point may not be enough to prevent tool and material interference, position the tool at a point that is 10.0 mm beyond the positioning point (diameter value).

A tool selection position at a B-axis angle of 90 degrees (cross direction)



■ Precautions to be heeded during B-axis tool rotation

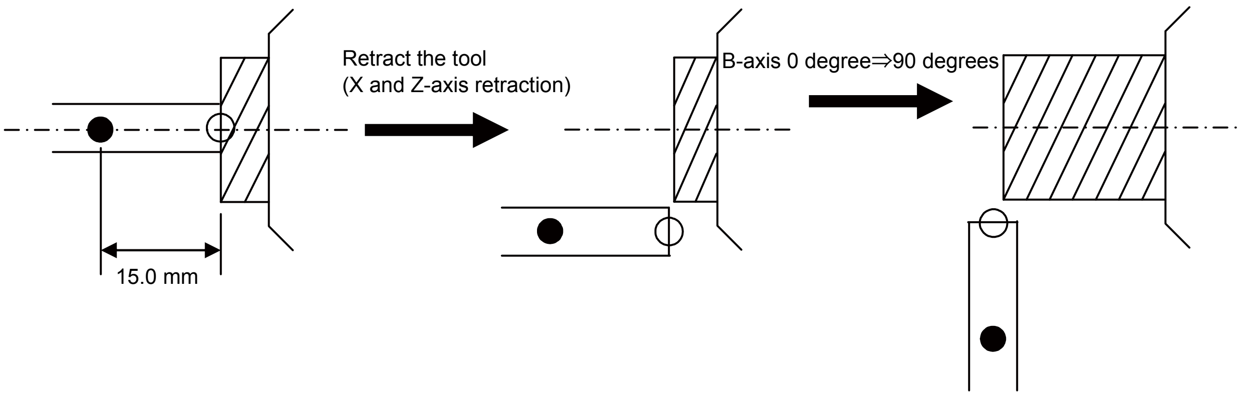
If the tool is not retracted when G900 B-axis mode is on and the tool is rotated at a B-axis angle from 0 to 90 degrees, the material and the tip of the tool will interfere as shown in the diagram below.



● : B-axis rotation center
○ : Tool tip position

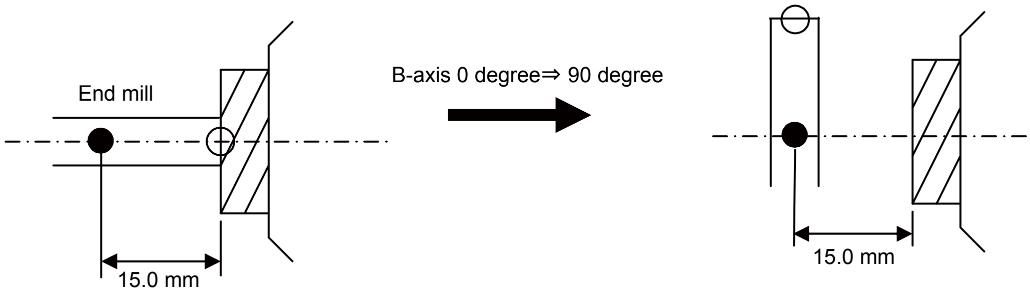
* Since the B-axis rotates around the supporting point of the tool tip in B-axis mode, it will interfere with the material.

For that reason, the tool should be retracted along the X and Z-axes before it rotates the B-axis.



* The tool rotates around the tool tip supporting point, but there is no tool and material interference
* The material (Z1) protrudes because this is a sliding headstock type machine

The diagram below shows what happens when the B-axis angle is rotated from 0 to 90 degrees (G910 B0) when B-axis mode is off.



● : B-axis rotation center
○ : Tool tip position

* Rotation around B-axis rotation center
* Tool tip and material relationship are out of contact

14.14 Helical Interpolation

The machine can provide the circular interpolation for arbitral two axes among the three axes (X1, Y1, and Z1 or Z2 axes) which are perpendicular with one another and also the linear interpolation for the remaining axis in synchronization with the circular rotation. This function enables drilling of large orific size.

14.14.1 End-face helical interpolation process

[Command format]

G2/G3 X(U)☐ Y(V)☐ Z(W)☐ I☐ J☐ K☐ P☐ F☐

14.14.1.1 Relation between circular plane and linear interpolation axis

	Circular plane	Linear interpolation axis
G17	XY plane	Z axis
G18	ZX plane	Y axis
G19	YZ plane	X axis

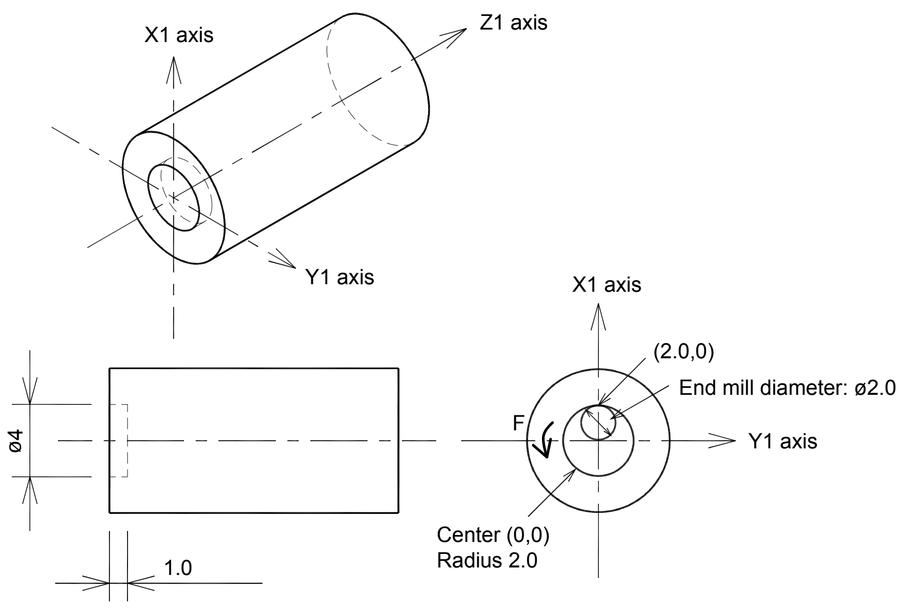
[Argument]

For circular plane G17 (X-Y plane):

X(U)☐	Specify the end-point coordinate of the circular arc based on the diameter.
Y(V)☐	Specify the end-point coordinate of the circular arc based on the diameter.
Z(W)☐	Specify the end-point coordinate of the linear axis based on the radius.
I☐	Specify the center of the circular arc with the radius command increment in the X coordinate of the center viewed from the start point.
J☐	Specify the center of the circular arc with the radius command increment in the Y coordinate of the center viewed from the start point.
P☐	Specify the number of pitches. When no P argument is specified or P0 is specified, it causes the helical interpolation from the start to end points within a single circuit.
F☐	Specify the feed rate.

[Sample program]

$1	
:	
G98 M80 S3=4000	Rotary tool on gang tool post rotates forward.
M28 S0	Spindle 0-degree indexing
T1200	
G0 Z–5.0 T12	
X0	
G17	X-Y plane selection
G42 G0 X4.0 Y0 Z-1.0	Positioning
	Nose R compensation mode ON (Specify R of tool data T12 to end mill radius 1.0.)
G2 X4.0 Y0 Z0 I-2.0 F300	Moves a tool to start point by helical interpolation.
G2 X4.0 Y0 Z1.0 I-2.0 P10 F60	Cuts in 10 pitches of helical interpolation.
G2 X4.0 Y0 I-2.0	Circulates tool by a single circle on the bottom of hole.
G40 G1 X0 Y0	Nose R compensation mode cancel.
G0 Z-1.0	
G18	Z-X plane selection
G0 X0 Z–1.0	
G50 W5.0	
M82	Rotary tool on gang tool post stops.
M3 S1=3000 G99	
:	
:	



[Note]

X(U) and Y(V) should be specified with diameter values.

14.14.2 Inclined helical interpolation process (G173)

The inclined rotation coordinate instruction allows the coordinate system to be rotated at the specified angle with the Y1 axis set to the center axis on the Z-X plane and the X1 axis set to the center axis on the Y-Z plane. The inclined rotation coordinate instruction can be combined with the helical interpolation to provide the inclined helical interpolation process.

■ Setting of inclined rotation coordinate

[Command format]

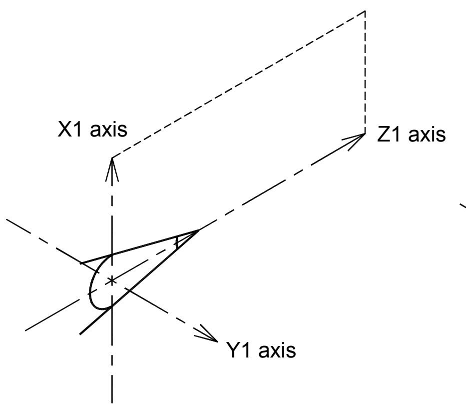
G173 X☐ Z☐ D☐	Inclined rotation coordinate command for rotating Z-X plane with Y1 axis set to the center axis
G173 Y☐ Z☐ D☐	Inclined rotation coordinate command for rotating Y-Z plane with Z1 axis set to the center axis
G173	Canel Inclined rotation coordinate command

[Argument]

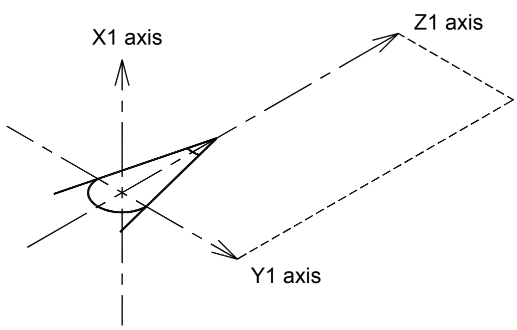
X☐	Specify the center-of-rotation coordinate on X1 axis. This argument should be specified for the rotation with Y1 axis set to the center axis.
Y☐	Specify the center-of-rotation coordinate on Y1 axis. This argument should be specified for the rotation with X1 axis set to the center axis.
Z☐	Specify the center-of-rotation coordinate on Z1 axis.
I☐	Specify the rotation angle. If no argument is specified, 0° is assumed.

Angular rotation direction

When rotating Z-X plane with Y1 axis set to the center axis



When rotating Y-Z plane with X1 axis set to the center axis



■ Helical interpolation command when inclined rotation coordinate is specified

[Command format]

G2/G3 X(U)☐ Y(V)☐ Z(W)☐ I☐ J☐ K☐ P☐ F☐

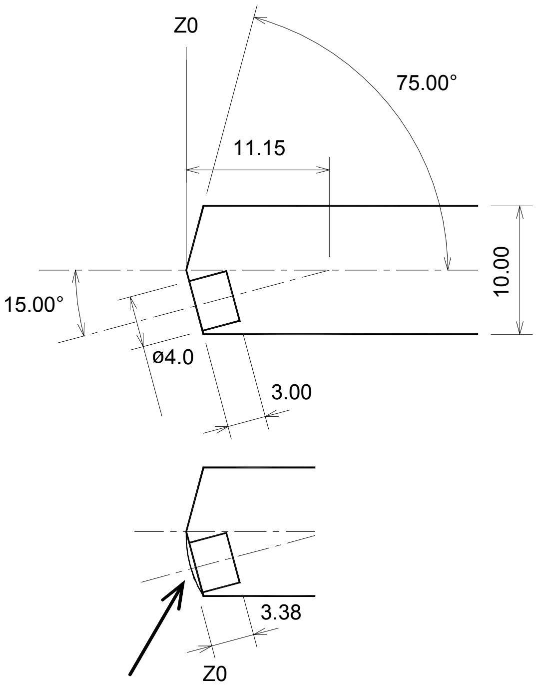
[Argument]

For circular plane G17 (X-Y plane):

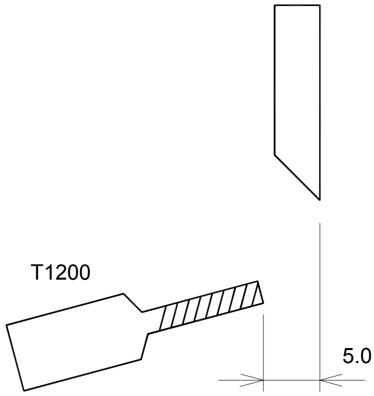
X(U)☐	Specify the end-point coordinate of the circular arc based on the radius.
Y(V)☐	Specify the end-point coordinate of the circular arc based on the radius.
Z(W)☐	Specify the end-point coordinate of the linear axis based on the radius.
I☐	Specify the center of the circular arc with the radius command increment in the X coordinate of the center viewed from the start point.
J☐	Specify the center of the circular arc with the radius command increment in the Y coordinate of the center viewed from the start point.
P☐	Specify the number of pitches in the range 0 to 99. If no P argument is specified or P0 is specified, it causes the helical interpolation from the start to end points within a single circuit.
F☐	Specify the feed rate.

[Sample program]

$1	
:	
G98 M80 S3=4000	Rotary tool on gang tool post rotates forward.
M28 S0	Spindle 0-degree indexing
T1200	
G50 W–5.0	
G0 X0 Y0 Z-3.0 T12	Position the tool at center of workpiece.
G173 Y0 Z11.15 D15.0	Coordinate system rotate command Rotate the Y-Z plane 15 degrees at the position Z11.15.
G17	
G1 X0 Y0 Z-2.0 F300	Move the tool to the center of inclined coordinate system.
G42 X2.0 Y0	Move the tool to the positioning point while compensating the tool diamter.
G2 X2.0 Y0 Z0 I-2.0 P1.0 F300	Circulates tool helically by a single circle to the position Z0 of inclined coordinate system (air cut).
G2 X2.0 Y0 Z3.38 I-2.0 P10.0 F60	Cuts in 10 pitches of helical interpolation to the position Z3.38.
G2 X2.0 Y0 I-2.0	Circulates tool by a single circle on the bottom of hole.
G40 G1 X0 Y0	Move the tool to the center of workpiece while canceling the tool diameter compensation.
Z-3.0 F1500 T0	Move the Z axis away from the tool.
G173	Cancel coordinate system rotate command.
G0 X0 Z-3.0	Position the tool at center of workpiece.
G50 W5.0	
G18	
G0 Z–1.0	
M82	
M3 S1=3000	
T200	
:	
:	



The inclined position is defined as Z0.



Swing type end-face rotary tool
(End mill diameter: ø2.0, angle: 4.8°)

[Note]

- The axis addresses in the G173 command should be specified based on radius.
- The circular plane should be changed from one to another after G173 command is issued. The initial value of the circular plane is G17 (X-Y plane). After the inclined rotation coordinate command is canceled, the circular plane is automatically returned to G18 (Z-X plane).

14.14.3 Inclined helical interpolation process (G950, G951)

Inclined helical interpolation machining can also be performed using the coordinate rotation command (G950, G951). Use a rotary tool with B-axis or a variable tilt spindle. For information on the coordinate rotation command (G950, G951), refer to *<14.12 Coordinate rotation (G950, G951)>*.

[Sample program]

Performing inclined helical interpolation using B-axis tools on a gang tool post or variable tilt spindle

$1	
:	
M5 M80 S3=4000 G98.............................	Rotary tool on gang tool post rotates forward.
T1200 E0.0	Tool selection, spindle 0-degree indexing
G0 X0 Z-2.0 T12	
G900 X0 Z-0.907 B4.8	Turn on B-axis mode, position the tool at a B-axis angle of 4.8 degrees and the tool tip at X 0 and Z -0.907
G950 X0 Z0 B4.8	Coordinate rotation ON, Specify X0Y0Z0 as the zero point
G17	X-Y plane selection
G41 G0 X1.409 Y0 Z-0.907	End mill positioning Tool compensation on End mill radius 1.0 at tool data T12 R
G3 X1.409 Y0 Z0 I-1.1 F300....................	Move to start point using helical interpolation
G3 X1.409 Y0 Z0.995 I-1.1 P25 F60........	Cut using a helical interpolation pitch of 25 mm
G3 X-0.791 Y0 W0.005 I-1.1 F100	
G3 X-0.791 Y0 I1.1 F100	Circulates tool by a single circle on the bottom of hole.
G3 X-0.791 Y0 Z-0.907 I1.1 F500............	End mill retracts
G40 G0 X-0.791 Y0 Z-0.907	Tool compensation off
G951	Coordinate rotation OFF
G901 B0...	B-axis mode off
G0 U0 V0 W0 T00....................................	Offset cancel
G18	X-Z plane selection
M82	Rotary tool on gang tool post stops rotation
M3 S1=3000 G99	
:	

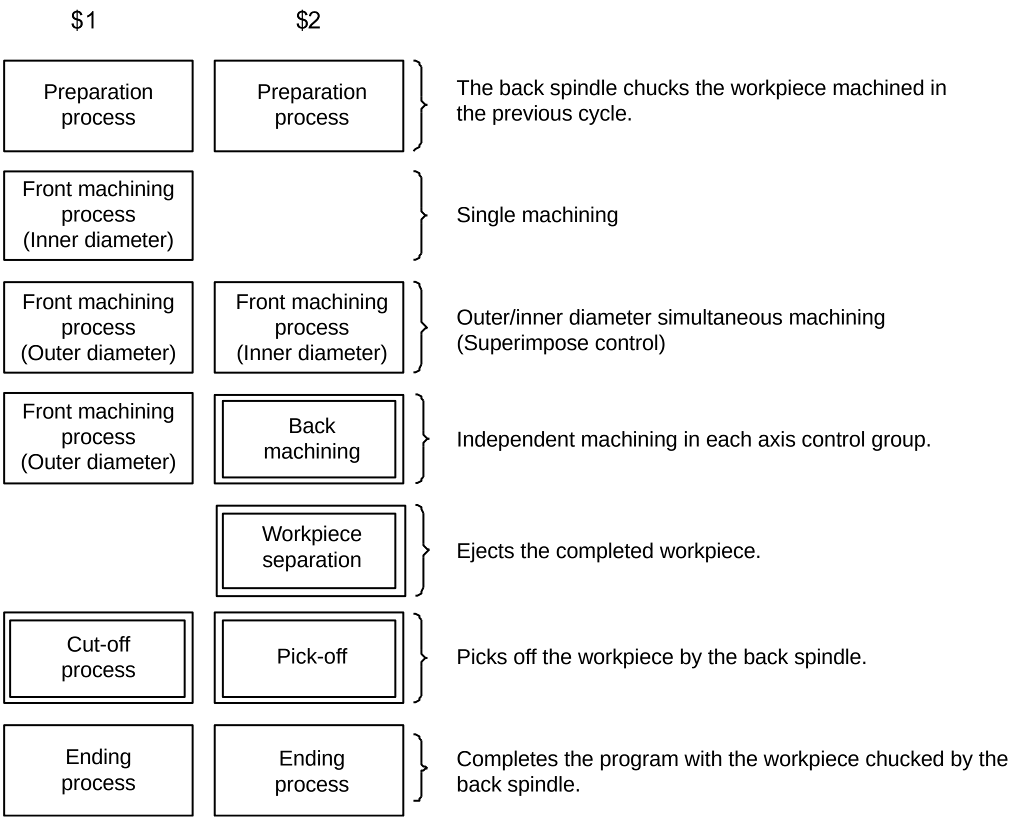
(Blank Page)

Product Code	C-L220E XII
Document Code	2E1-1401
Mfg. No.	L220E/0001~
Issue Date	2014.1

15. Sample Programming [Back Machining]

15.1 Back Machining Program

The back machining is performed by combining a front machining program ($1) and back machining program ($2).



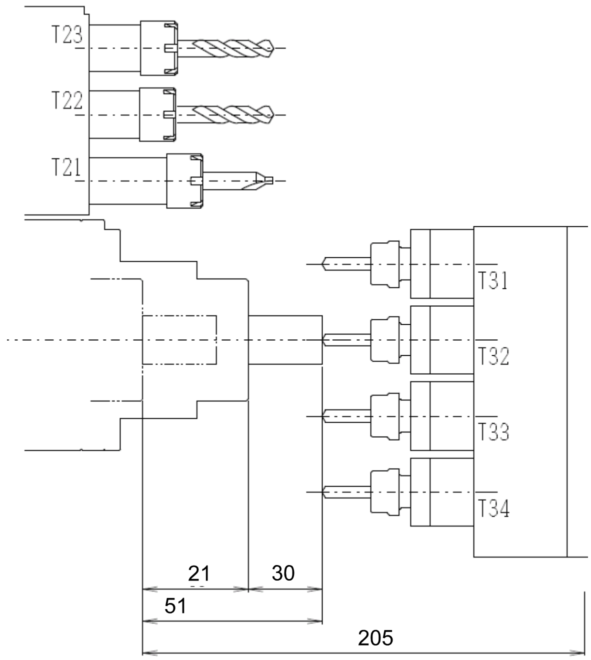
As shown above, the workpiece is finished after two cycles of program execution.

Note that inserting the back machining program at the last program execution (G999) can execute workpiece back machining after cut-off machining both in the first cycle, thereby finishing the workpiece in one cycle. The program is not however be executed unless the menu key [Last PRT] at the bottom of the screen is set to ON. The program is executed when [Last PRT] in the first cycle has been ON.

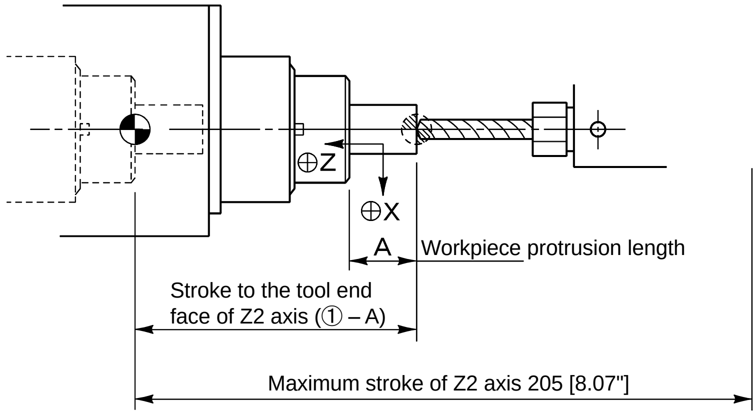
15.2 Back Machining Coordinates

The coordinates in programming must be understood according to the back spindle movement during program creation.

The positional relationship among the back spindle, the back machining tools, and the workpiece in the back spindle chuck is shown in the figure below.



The same conception as in *<11.6 Coordinate System>* can be applied as the coordinate axis. Programming is performed on the assumption that the material is fixed.



[Note]

- The length of the workpiece that protrudes from the end face of the back spindle chuck plus protrusion length of long-neck chuck is 30.0 mm (maximum).
  For more than 30.0 mm, the workpiece interferes with the workpiece separator (product chute) during workpiece separation.
- The length of the workpiece protruding from the end face of the back spindle chuck must be entered in the Back work extend length in machining data.
  (Protruding workpiece length = entire workpiece length – programmed back chucking position)
  If no value is specified, the back machining tool interferes with the workpiece protruding from the back spindle when the workpiece is chucked because the end face of the back spindle is assumed as the zero point.

15.3 Back Machining Process

- The back machining is performed by tools T31's and T51's. This machining is basically the same as for front machining.
- After the selection of a tool of any of T31's and T51's, the program zero point of the Z2 axis is taken at the end face position of the back workpiece that has the "Back work extend length" (plus "Back chuck extend length") set on the Machining Data screen.
  Enter the extend length of the workpiece, the length the workpiece end face extends from the back spindle chuck end face, to "Back work extend length" on the Machining Data screen. When a long-neck chuck is used, also enter the data to "Back chuck extend length".
- Set the back machining tool to the position within 62 mm (U153B, U155B) or 70 mm (U154B) from the end face of sleeve holder.
- If "Z–1.0" is specified after tool selection of T31's or T51's, the workpiece end face on the back spindle is positioned at the point where the clearance between this workpiece end face and the back machining tool is 1.0 mm. If the sleeve holder for back machining protrudes more than 62 mm (U153B, U155B) or 70 mm (U154B), interference may occur. Pay attention when setting the tool.
- The amount of the workpiece that protrudes from the back spindle cap nut must be within 30 mm.
- Write a program in $2.
- For "Back work extend length" of the machining data, enter the amount protruding between the end face of the back spindle chuck and the end face of workpiece.
  (Back workpiece protruding amount) = (whole workpiece length) – (programmed back chuck position)
- For "Back chuck extend length" of the machining data, enter the data when the long-neck chuck is used. The value to be entered is the protrusion length of the long-neck chuck from the end face of the back spindle cap nut. This sets the position of "Back chuck extend length" + "Back work extend length" as the program zero point.
- The GSC1507 rotary cross-machining tool can be mounted on U155B rotary tool spindle.
  Mount the GSC1507 on T34. When two or more rotary cross-machining tool are mounted, the stroke is limited on T20's tools.

[Note]

- During back machining, the stationary tools can be moved into the back spindle up to the position 30 mm from the back spindle end face, and the rotary tool for end face machining can be moved into the back spindle up to the position 20 mm from the back spindle end face. Machining with the tool moved further into the back spindle results in interference between the cap nut of the adjacent tool sleeve and the cap nut of the back spindle. When GDF1207 is mounted, the tool for back machining cannot move into the back spindle.
- Set the boring sleeve in the opposite tool post or back tool post so that the tool nose faces horizontally (except for pinch turning).
- Boring by T33 or T34 cannot be performed because tools T21, T31 and the back spindle and the workpiece separator (workpiece chute) interference each other.
  Boring up to the inside of back spindle may be limited with the stroke depending on the relation between the cap nut position of the adjacent and T21 tool sleeve and the cap nut outer diameter. ER11 and AR11 are recommended for the tools T31 to T34.
- If three or more GSC1507 rotary cross-machining tools are mounted and protrusion length of tool exceeds 25 mm, an interference with the end-face machining tool on opposite tool post will occur. In such a case, remove the tool that interferes to proceed machining.
- When GSC1407 is mounted on opposite tool post, an interference between tools may occur if the back tool post moves a great distance in the X2-axis direction.
- If GSC1407 is mounted on T21 of the opposite tool post, back machining with tools T34 or T38 causes an interference between GSC1407 and U155B.
- If slitting spindle GSS1530 is mounted on back tool post U155B, the opposite tool post T23/T26 cannot be used.

[Sample program]

(Center hole and drilling)

$2	
G600	
G99 M23 S2= □	Specify per rotation feed and forward rotation of back spindle.
(Center hole)	
T3□	Select the back machining tool.
G00 Z–1.0	Position the tool 1.0 mm away from the workpiece end face.
G01 Z5.0 F0.05 T□□	Cut to the specified depth in cutting feed.
G00 Z–1.0 T00	Cancel the compensation while returning to the position 1.0 mm away from the workpiece in rapid feed.
(Drilling)	
T3□ W–1.0	Select the back machining tool at the position –1.0 mm away from the current back spindle position.
G00 Z–1.0	Position the tool 1.0 mm away from the workpiece end face.
G01 Z15.0 F0.08 T□□	Cut to the specified depth in rapid cutting feed.
G00 Z–1.0 T00	Cancel the compensation while returning to the position 1.0 mm away from the workpiece in rapid feed.
M25	Stop the back spindle.
M34	A series of operations of product separation is performed after back machining is completed.

15.4 Back Spindle Air Blow (M72, M73)

Blowing air from the tip of the fixed knock-out pipe in the back spindle can be controlled by these commands.

[Command format]

M72	Back spindle air blow ON (Starts air blow.)
M73	Back spindle air blow OFF (Stops air blow.)

15.5 Knock-out Advance/Retract (M10, M11)

■ Outline

The back spindle knock-out device advances or retracts.

[Command format]

M10	Knock-out advance
M11	Knock-out retract

[Operation]

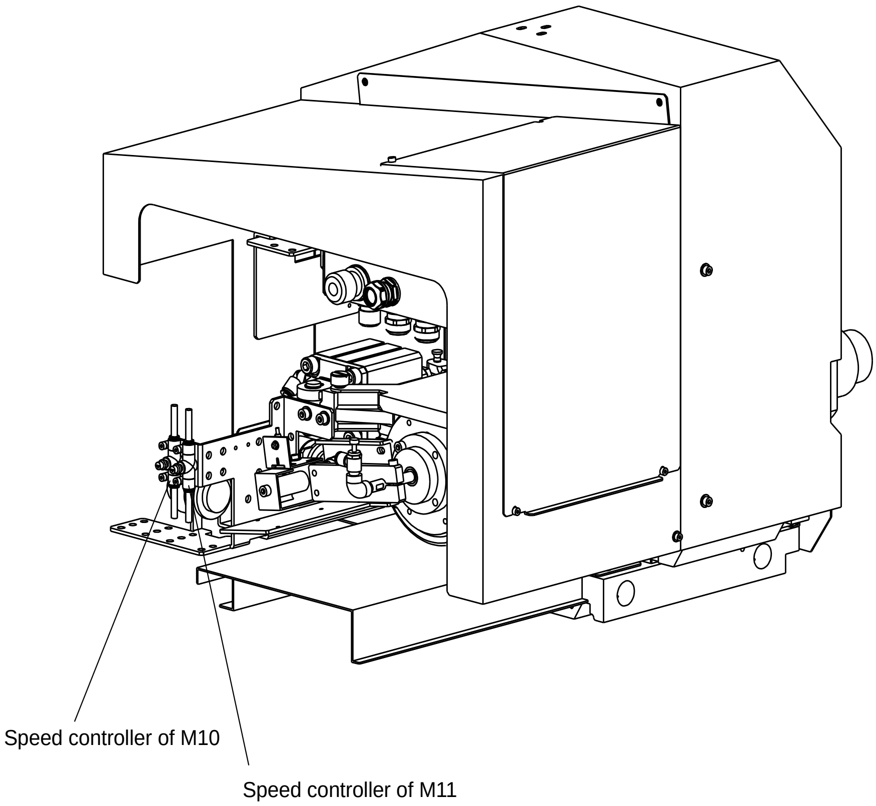
- Knock-out advance (M10)

  The knock-out advance (M10) command allows the knock-out device to be advanced by the air cylinder.
- Knock-out retract (M11)

  The knock-out advance (M11) command allows the knock-out device to be retracted by the air cylinder.

[Note]

- If the knock-out advance/retract (M10/M11) command is specified while the back spindle chuck is closed or the back spindle is operating (during rotation, while controlled in C-axis mode, during rigid tapping, during orientation, or while controlled in spindle synchronized mode), an alarm occurs.
- Adjusting the speed controller of the air cylinder allows the advance/retract speed of the knock-out device to be adjusted.



15.6 Product Collection by Basket (M320, M33, M34, M780)

15.6.1 Move basket forward (M320)

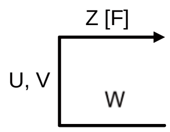
The M320 (move basket forward) command moves the back headstock (Z2 axis) to the basket advance position if "BASKET" is specified for Back spindle in machining data. If the M34 (series of operations for product separation) command is specified, the products received by the basket are collected into the product receiver box.

[Command format]

M320 X□ W□ U□ V□ Z□ R1 F□ Move basket forward

[Argument]

X□	End position queuing X position. When the X1 axis reaches the end position during cut-off process, the basket on the back headstock (Z2 axis) finishes advance movement. If X argument omitted, the end-position queuing is not performed.
W□	Specify the position to retract the Z2 axis (distance from zero point of Z2 axis). When this argument is omitted, the Z2 axis moves to 0.0 mm.
U□	Specify the position to which the X2 axis advances. When this argument is omitted, the X2 axis moves to the point 632.0 mm.
V□	Specify the position to which the Y2 axis advances. When this argument is omitted, the Y2 axis moves to the point 0.0 mm.
Z□	Specify the position to which the basket advances (absolute command). If Z argument is omitted, the back headstock (Z2 axis) moves to the point 187.0 mm of machine coordinate. When Z argument is specified, R1 argument is ignored even if specified.
R1	Specify "1" when shifted tool is used for cut-off tool. 1: The Z2 axis advances to the position 135.0 mm. Omitted: The Z2 axis advances to the position 187.0 mm.
F□	Specify the feed rate (per minute) of the back headstock (Z2 axis). If F argument is omitted, the back headstock (Z2 axis) moves in rapid feed. Specify the value with decimal point.



[Note]

When specifying the M320 (move basket forward) command with an X argument (end position queuing), the X axis must reach the specified end position during the execution of 8 blocks of commands that follow the M320 command. If the X axis does not reach the end position within this period, an alarm occurs.

[Sample program]

- With queuing at end point

$1		$2	
G600	Free pattern (machining pattern cancel)	G600	Free pattern (machining pattern cancel)
M34	A series of operations for product separation		
:		:	
T0100	Select the cut-off tool		
G0 X20.0 Z10.0 T1	Position the cut-off tool		
M320 X13.0	X1 axis (cut-off tool) at the point 13.0 mm waits for Z2 axis (basket) at the point 187.0 mm.		
G99 G1 X–3.0 F0.03	Perform cut-off machining		
M141	Return the basket		

[Note]

- Before executing the M320 (move basket forward) command with end-position queuing specified (with X argument specified), set the override to 100%. If the override setting is changed during operation, an end-position queuing will fail.
- When specifying the M320 (move basket forward) command with an X argument (end position queuing), the X axis must reach the specified end position during the execution of 8 blocks of commands that follow the M320 command. If the X axis does not reach the end position within this period, an alarm occurs.
- Select "BASKET" for "Back Spindle" in Machining Data screen. Specifying M320 (move basket forward) command while "Standard" is selected for "Back Spindle" causes an alarm to occur.

15.6.2 Basket collection mode - Move product collection position (M33)

[Command format]

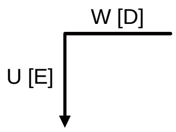
M33 Move product collection position

M33 W□ D□ U□ E□

[Argument]

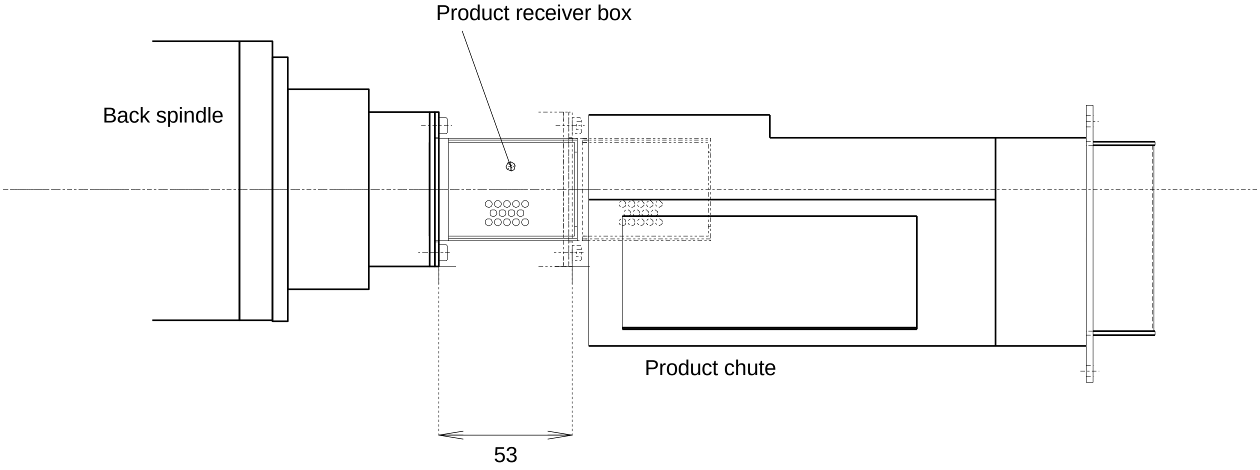
- W□ Specify the position to retract the Z2 axis (distance from zero point of Z2 axis). When this argument is omitted, the Z2 axis moves to 0.0 mm.
- D□ Specify the feed rate (per minute) to retract the Z2 axis. When this argument is omitted, the Z2 axis moves in rapid feed. Specify the value with decimal point.
- U□ Specify the position where the separator on X2 axis collect products. Specify the value with decimal point. When this argument is omitted, the X2 axis moves to 0.0 mm.
- E□ Specify the feed rate (per minute) when the separator on X2 axis moves. Specify the value with decimal point. When this argument is omitted, the X2 axis moves in rapid feed.

W□ D□ U□ E□: Can be omitted.



[Sample program]

$2	
M88	
M33	Move the product collection position. • If M33 is specified without argument specified, X2 and Z2 axes move to the zero point. • After the X2 and Z2 axes movement is completed, the coordinate system setting for the Z2 axis is executed. • Specify a program that puts the finished workpiece in the product box in the next line of the M33 code.
G00 Z□.............................	Move the back spindle to the workpiece collection position in rapid feed.
M23 S2=100	Rotate the back spindle to put the product on the workpiece separator.
G04 U1.0	Dwell
M25	Stop the rotation of back spindle.
G0 Z0	
M31	Turn on the workpiece conveyor.
T3000	Move the back spindle at the center of main spindle.
M89	



[Note]

- Specify "STANDARD" in "BACK SPINDLE" of the machining data.
- The above program example selects T3000 after product collection.
- Compared to M34 (sequential operation of product collection), M33 allows individual operations for product collection to be programmed. M33 can therefore save product collection time by efficient programming.
- Decimal point is required for argument for feed rate.

  Ex.) M33 D3000.0 E5000.0

15.6.3 Basket collection mode - sequential operation of product collection (M34)

The product received by the baskete is transferred to the production collection box.

[Command format]

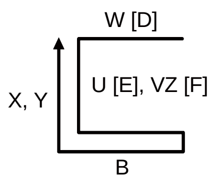
M34 W□ D□ U□ V□ E□ Z□ F□ T□ S□
B□ X□ Y□

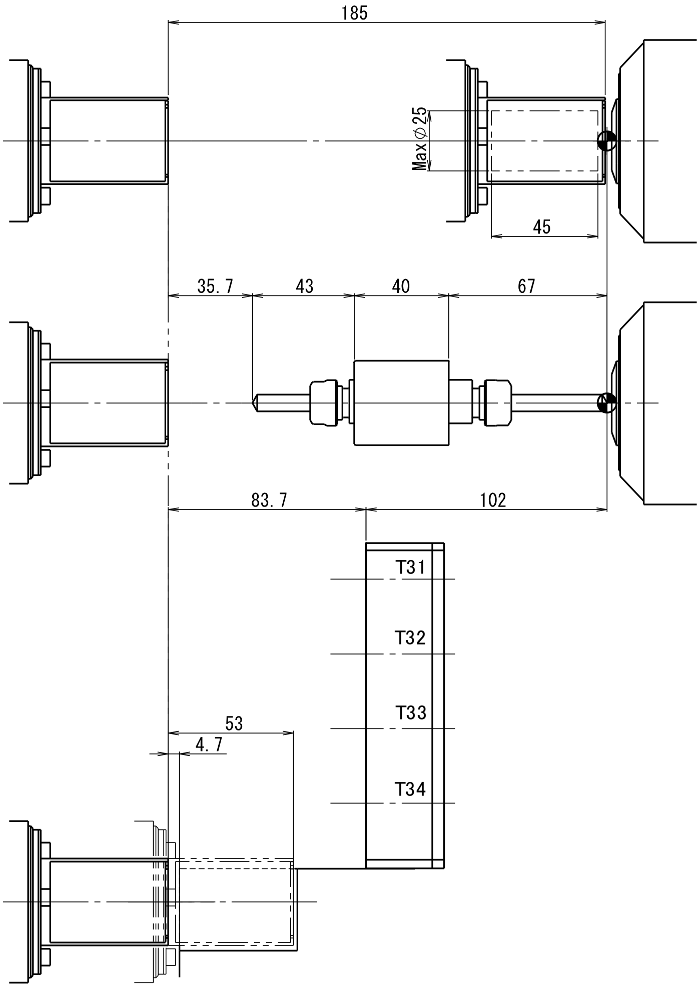
[Argument]

W□	Specify the position to retract the Z2 axis (distance from zero point of Z2 axis). When this argument is omitted, the Z2 axis moves to 0.0 mm.
D□	Specify the feed rate (per minute) to retract the Z2 axis. When this argument is omitted, the Z2 axis moves in rapid feed. Specify the value with decimal point.
U□	Specify the position where the separator on X2 axis collect products. When this argument is omitted, the X2 axis moves to 0.0 mm.
V□	Specify the position where the separator on Y2 axis collect products. When this argument is omitted, the Y2 axis moves to 0.0 mm.
E□	Specify the feed rate (per minute) when the separator on X2 axis moves. Specify the value with decimal point. When this argument is omitted, the X2 axis moves in rapid feed.
Z□	Specify the position to advance Z2 axis. When this argument is omitted, the Z2 axis moves to 53.0 mm in usual case.
F□	Specify the feed rate (per minute) when Z2 axis advances. Specify the value with decimal point. When this argument is omitted, the Z2 axis moves in rapid feed.
T□	Specify the time to rotate the back spindle at product collection. When this argument is omitted, the back spindle rotates for two seconds.
S□	Specify the back spindle speed at product collection. When this argument is omitted, the back spindle rotates at 30 min^{-1}.
B□	Specify the position to retract the Z2 axis after product collection. When this argument is omitted, the Z2 axis moves to the position 0.0 mm.
X□	Specify the position to move the X2 axis after product collection. When this argument is omitted, the X2 axis moves to the center of the spindle.
Y□	Specify the position to move the Y2 axis after product collection. When this argument is omitted, the Y2 axis moves to the center of the spindle (L20X).

W□ D□ U□ V□ E□ Z□ F□

T□ S□ B□ X□ Y□: Can be omitted.





15.6.4 Mounting and adjusting the basket on back spindle (M780)

[Procedure]

1. Specify M780 (Back spindle zero point indexing) command in MDI mode.
2. Press the Reset key.
3. The back spindle is indexed at the zero point. Mount the basket on a level with the back spindle.

[Note]

- Select "BASKET" for "Back Spindle" in Machining Data screen.
- Note that the back spindle is not excieted when mounting the basket.

15.7 Pick-off Collection of Products by Back Spindle (M33, M34)

The following explains a series of operation to collect the products, picked off by the back spindle, into the product receiver box.

15.7.1 Move product collection position (M33)

[Command format]

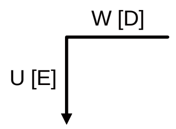
M33 Move product collection position

M33 W□ D□ U□ E□

[Argument]

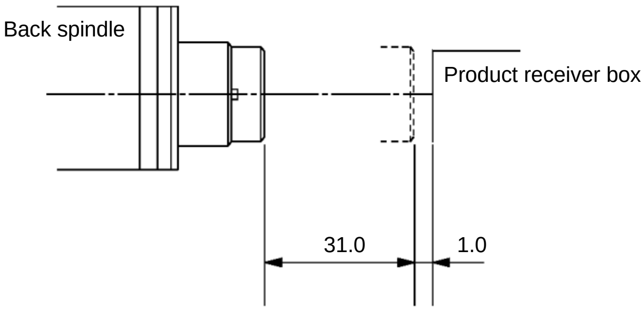
W□	Move position (distance from the zero point of the Z2 axis in mm) when the Z2 axis is retracted. Moves the axis to the return position (0.0 mm) when omitted.
D□	Feed rate (per minute) when the Z2 axis returns. Moves the axis in rapid feed when omitted. Specify the value with decimal point.
U□	Move position (mm) when the X2 axis separation position moves. Moves to the zero point (0.0 mm) when omitted.
E□	Feed rate (per minute) when the X2 axis separation position moves. Moves the axis in rapid feed when omitted. Specify the value with decimal point.

W□ D□ U□ E□: Can be omitted.



[Sample program]

$2	
M25	Stop the back spindle rotation
M33	Move the product collection position. • If M33 has no argument specified, the X2 and Z2 axes move to the zero point and terminates operation with the back spindle chucking the workpiece. • After the completion of axis movement, the machine coordinate system is set for the Z2 axis. • Specify a program that puts the finished workpiece in the product box in the next line of the M33 code.
G00 Z32.5	Move the back spindle to the workpiece separation position in rapid feed.
M16	Open the back spindle chuck.
M10	Advance the knock-out bar.
G04 U0.5	Dwell
G00 Z0	Return to the back spindle zero point.
M72	Turn the back spindle air blow on.
M11	Return the back spindle knock-out bar.
T3000	Move the back spindle at the center of main spindle.
M73	Turn off the air blow of the back spindle.



[Note]

- Specify "Standard" for "Back Spindle" in the Machining Data screen.
- The above program example selects T3000 after product separation.
- Compared to M34 for sequential operation for product collection, M33 allows individual operations for product separation to be programmed. M33 can therefore save product separation time by efficient programming.

15.7.2 A series of operations for product separation (M34)

[Command format]

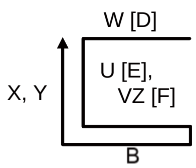
M34 X□ Y□ W□ D□ U□ V□ E□ Z□ F□ T□ B□

[Argument]

X□	Specify the X2 axis movement position after product separation. Moves the axis to the center of the spindle when omitted.
Y□	Specify the Y2 axis movement position after product separation. Moves the axis to the center of the spindle when omitted (L20X).
W□	Move position (distance from the zero point of the Z2 axis in mm) when the Z2 axis is retracted. Moves the axis to the return position (0.0 mm) when omitted.
D□	Specify the feed rate (per minute) for Z2 axis returning. Moves the axis in rapid feed when omitted. Specify the value with decimal point.
U□	Specify the X2 axis unloading movement position. Moves to the zero point (0.0 mm) when omitted.
V□	Specify the Y2 axis unloading movement position. Moves to the zero point (0.0 mm) when omitted (L20X).
E□	Specify the feed rate (per minute) for X2 axis unloading movement. Moves the axis in rapid feed when omitted. Specify the value with decimal point.
Z□	Specify the Z2 axis forward end movement position. Moves to the position of 31.0 mm when omitted.
F□	Feed rate (per minute) during Z2 axis advance. Moves the axis in rapid feed when omitted. Specify the value with decimal point.
T□	Air blow time after product separation. Blows air for one second when omitted.
B□	Specify the Z2 axis return position after product separation. Moves the axis to the return position (0.0 mm) when omitted.

W□ D□ U□ V□ E□ Z□ F□

T□ B□ X□ Y□: Can be omitted.



[Sample program]

$1	$2
G630 .... Front/back parallel machining	G630 .... Front/back parallel machining
T0200	T3100
:	:
T0300	M25 .... Stop the back spindle rotation.
:	M34 .... A series of operations for product separation

[Note]

Specify "Standard" for "Back Spindle" in the Machining Data screen.

15.7.3 Cut-off and product collection

The product is separated on the back spindle side.

Adjust the rotation of the main spindle and back spindle in the same direction before the back spindle chuck is closed.

[Sample program]

$1		$2	
M3 S1=1500		M34	Specify a series of operations for the product separation.
		M16	Open the back spindle chuck.
		M24 S2=1500	Specify 1500 for the back spindle reverse rotation speed.
!L1	Queuing command	!L1	
G114.1 H1D-2	Rotation synchronization of main spindle and back spindle		
N11 T0100	Select the cut-off tool		
G00 X26.Z $\boxed{*1}$ T01	Cut-off tool positioning		
G650	Automatic queuing.	G650	Pick-off
		M72	Turn the back spindle air blow on.
		G00 Z–1.0	Position the back spindle 1.0 mm before the workpiece by rapid feed.
		G98 G01 Z $\boxed{*2}$ F1000	Advance the back spindle to the chuck position.
		G04 U1.0	Specify a dwell pause command of one second at the chuck position.
		M77	Wait until spindle synchronization is completed.
		M15	Close the back spindle chuck.
		M73	Turn off the air blow of the back spindle.
!L2	Queuing command	!L2	
G99 G01 X–1.0 F0.03	Workpiece cut-off		
G600	Automatic queuing.	G600	Cancel the machining pattern (superimpose control).
X–3.0 F0.03			
G113	Spindle synchronization mode cancel		
M05			
M07			
G00 X–3.0			
Z $\boxed{*3}$ T00	Return to the start point.		
M56		M25	Stop the back spindle rotation.
G999	Automatic queuing.	G999	
N999	Automatic queuing.	N999	
M02		M02	
M99		M99	
%		%	

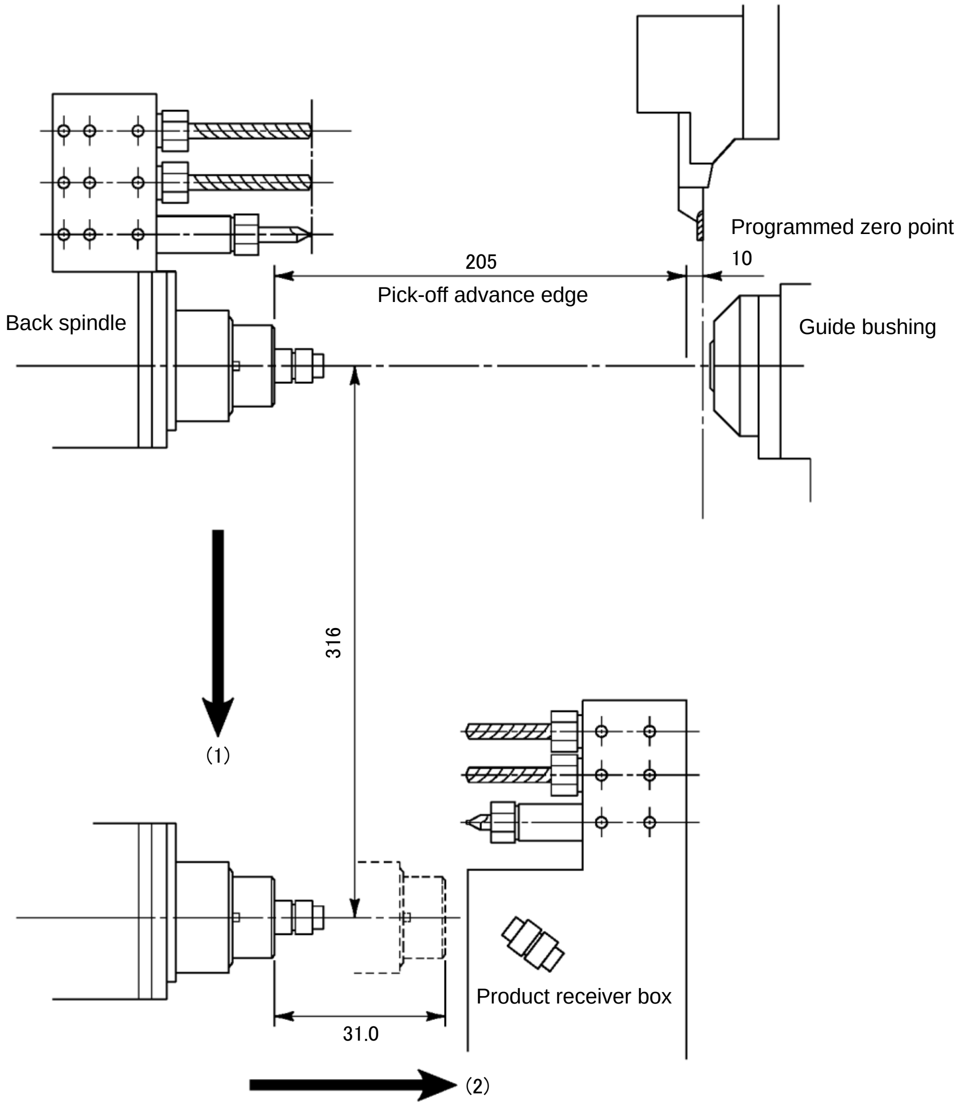
*1 : L (product length) + ℓ_3 (tool shift amount) + (tool width)

*2 : ℓ_1 (chuck amount) Move distance required until the workpiece is chucked by the back spindle chuck.

*3 : ℓ_3 (tool shift amount) Return to the program start point.

Home position of product separation, and serial operation

The product after cut-off machining or back machining is collected to a workpiece receiver box at this stroke during the operation.



Description

- The following operations are executed when product separation is specified by M34.
  1. The Z2 axis returns to the zero point.
  2. The X2 axis moves to the product collection position.
  3. The Z2 axis advances and moves to the workpiece separation position.
  4. The Z2 axis returns to the zero point after product knock-out.
  5. The X2 axis moves to the center of the main spindle.
- When an M33 command is specified, the X2 axis in Step 2 returns to the zero point to complete the operation for product collection. The subsequent collection must be programmed.
- Specify M33 in $2.

15.8 Spindle Synchronization Control (G114.1, G814, G113, G813, M77)

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 pick-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 non-conform workpiece is picked-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 places the two specified spindles in the synchronized status.

The G113 command cancels the synchronization of two spindles that was specified by a spindle synchronization control command.

[Command format]

G814 R☐	Spindle synchronization control
G813	Spindle synchronization control cancel
G114.1 H☐ D±☐ R☐	Spindle synchronization control
G113	Spindle synchronization control cancel
M77	Wait until spindle synchronization is completed

[Argument]

H☐	Specify the reference spindle. 1 is the main spindle.
D±☐	Specify the spindle to be synchronized and the direction in which the spindle rotates. When –2 is specified, the back spindle rotates backward.
R☐	Specify the amount (angle) of the phase shift between synchronous spindles. The R argument is automatically added to the phase difference calculated and stored by the non-conform material phase adjustment command in the Preparation screen. Specify a positive R argument for the phase shift in the clockwise direction (the direction of forward rotations of the main spindle). Setting range: 0 to 359.999°

[Note]

- Before specifying G114.1 or G814, be sure to specify the speed commands M3 S1=$\boxed{}$ and M24 S2=$\boxed{}$.
- If executing re-chucking, be sure to specify G114.1 or G814.
- 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 disabled in the spindle synchronization control mode. To index either spindle, cancel the spindle synchronization control mode.
- To let the back spindle to chuck 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, the value stored by executing the non-conform material phase adjustment command is used as the amount of shift.
- The spindle synchronization control is canceled by pressing the Reset key when all spindles stop (0 min^{-1}, indexing excluded).
- While synchronizing with the main spindle, be sure to specify G113 to cancel the spindle synchronization mode before executing bar change.
- Specifying the spindle synchronization command while the main and the back spindles do not rotate once causes an alarm to occur. Be sure to rotate the main and back spindles at least once before specifying the spindle synchronization command.

[Sample program]

$1		$2	
!2L1	Queuing	!1L1	Queuing
M3 S1=1000	Main spindle forward rotation	M24 S2=1000	Back spindle reverse rotation
!2L2	Queuing	!1L2	Queuing
T0100	Select the cut-off tool		
G0 X□ Z□ T01	Cut-off tool positioning		
M97	Main spindle speed fluctuation detection OFF		
M95	Back spindle speed fluctuation detection OFF		
G96 S150	Constant surface speed control command		
G50 S2500	Maximum spindle speed clamp command		
G814 R0	Spindle synchronization control (with angular shift adjustment)		
G650	Pick-off	G650	Pick-off
		G00 Z–1.0	
		M77	Wait until spindle synchronization is completed
		G98 G01 Z□ F1000	
		G4 U0.5	
		M15	Back spindle chuck close
!2L3	Queuing	!1L3	Queuing
G1 X–1.0 F0.03	Cut-off machining		
G600	Machining pattern cancel	G600	Machining pattern cancel
X–3.0 F0.05	Cut-off machining		
G97	Constant surface speed control cancel		
M96	Main spindle speed fluctuation detection ON		
M94	Back spindle speed fluctuation detection ON		
G113	Spindle synchronization control cancel		
M5 M25	The spindle stops, the back spindle stops		

Spindle synchronization control command (G814 R0)

When non-conform material phase adjustment is performed to re-chuck the workpiece in the back spindle, "G814 (G114.1 H1 D–2) R0" must always be specified. If the R argument is not specified, the workpiece is not inserted into and cannot be correctly re-chucked by the back spindle chuck since the spindles rotate synchronously without matching the angular phases.

[Note]

- The sample program does not contain product separation.
- M77 (waiting for completion of spindle synchronization) command position
- The following conditions must be satisfied for picking-off a non-conform material: the phase adjustment of both spindle must be completed, and then the back spindle is moved to the position where it chucks the workpiece. In the program sample, the M77 command is executed immediately before the back spindle moves to the position where it chucks the workpiece.
  To pick-off a round material, the M77 command can also be executed immediately before the M15 (back spindle chuck close) command.
- Specify a dwell command (G4 U0.5) after the back spindle has moved to the product chucking position.
- The operation enclosed in the broken lines is performed under constant surface speed control.

15.9 C1-C2 Axis Superimposition (M190, M191)

This C1-C2 axis superimpose control function controls the back spindle (C2 axis) by superimposing its movement over the main spindle (C1 axis) movement.

[Command format]

M190 C1-C2 axis superimpose control ON
The back spindle (C2 axis) moves along with the move command for the main spindle (C1 axis).

M191 C1-C2 axis superimpose control OFF
Cancels superimposition of the back spindle (C2 axis) from the main spindle (C1 axis).

[Sample program]

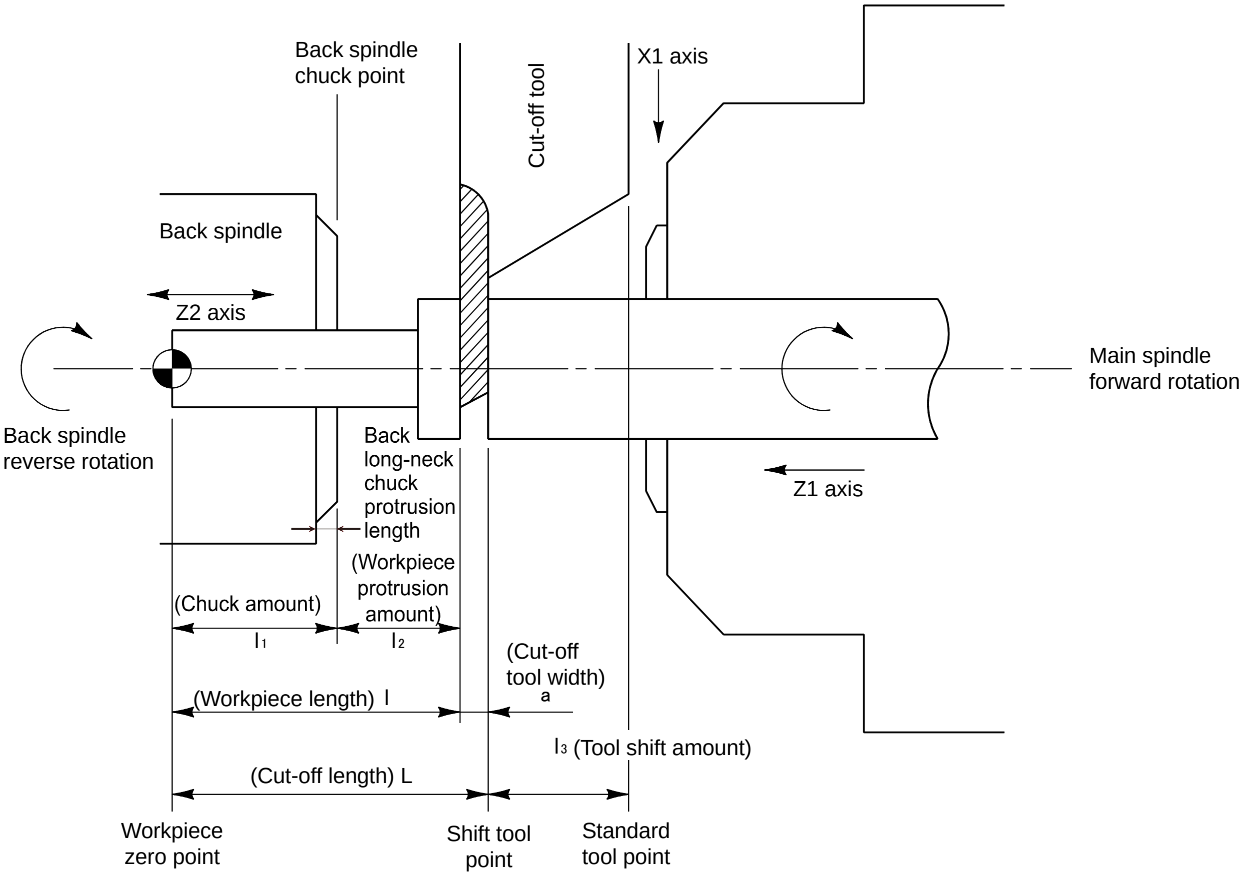
$1	$2
:	
M80 S3=4000	
G98	
T1100	
G0 X21.0 Z77.0 T11	
G650	G650
M18 C0 .... Main spindle C axis indexing	M48 C0 .... Back spindle C axis indexing
	M72
	M16
	G0 Z–1.0
	G98 G1 Z30.0 F5000
	G4 U0.3
	M15
	M73
!2L1	!1L1
M190 .... C1-C2 axis superimpose control ON	
G50 W–15.0	
G0 X21.0 Z62.0 T11	
:	
:	
G0 X21.0 T0	
G50 W15.0	
M82	
M16	
!2L2	!2L1
	M191 .... C1-C2 axis superimpose control OFF
	M20 .... Cancel main spindle C axis function
	M79 .... Cancel back spindle C axis function
:	:
:	:

[Note]

- The M190/M191 (C1-C2 axis superimpose control ON/OFF) command is enabled when the optional main spindle and back spindle C-axis functions are selected.
- Specify M18 (main spindle C axis function) in $1 and M48 (back spindle C axis function) in $2 before specifying the C1-C2 axis superimpose control command.
- C1-C2 axis superimpose control function (M190, M191) can be performed only in automatic operation mode. It cannot be performed in manual operation mode.
- Specify the M191 (C1-C2 axis superimpose control OFF) command only in the axis control group that contains the C2 axis. To cancel the C1-C2 axis superimpose control mode, execute queuing and specify the M191 command in $2. If it is specified in $1, an alarm " M01 G156 command at illegal system 1103" occurs.

15.10 Pick-Off and Ending Process

The back spindle picks off the workpiece in a cut-off process.
After that, the back spindle returns to the return position (Z2-axis machine zero point) to execute the ending process.



Description

1. Select the cut-off tool.
2. Select the back spindle.
3. Rotate the main spindle in the forward direction and the back spindle in the reverse direction at the same speed.
4. Open the back spindle chuck.
5. Move the Z1 axis to the cut-off position. The shift amount (ℓ_3) and cut-off tool width (a) of the cut-off tool must be considered.

   Z = ℓ (workpiece length) + a (cut-off tool width) + ℓ_3 (tool shift amount)
6. Move the back spindle to the back spindle chuck point. (Z2 axis)
7. Make the back spindle chuck.
8. Cut off the workpiece by the cut-off tool.
9. Return the back spindle to the return position.
10. Execute the ending process.

[Note]

- Advance the back spindle up to the point 10 mm from the guide bushing. The most advanced position of the back spindle depends on the size of the tool and on the tool holder.
  The most advanced position of the back spindle is 10 mm from the standard tool point.
  Therefore, the tool shift amount (ℓ_3) is determined in consideration of the above limitation.
- When cutting off the workpiece with chucking the back spindle, perform the reverse rotation command (M24) if the main spindle rotates in the forward direction (M03). Also use a forward rotation command (M23) if the main spindle rotates in the reverse direction (M04).

15.10.1 Pick-off process

[Sample program]

	$1		$2
			T3000
(1)	M03 S1=3000		M24 S2=3000 G98
(2)	T0100		
(3)	G00 X7.0 Z32.0 T11		
(4)	G650	(4)	G650
		(5)	G00 Z–1.0
		(6)	G01 Z20.0 F1000
		(7)	G04 U0.5
		(8)	M15
		(9)	G04 U0.5
(10)	!L1	(10)	!L1
(11)	G01 X–0.5 F0.015		
(12)	G600	(12)	G600
	X–3.0 F0.03		
	M05		
	M07		
	G0 Z0 T0		
	M56		M25
	G999		G999
	N999		N999
	M02		M02
	M99		M99
	%		%

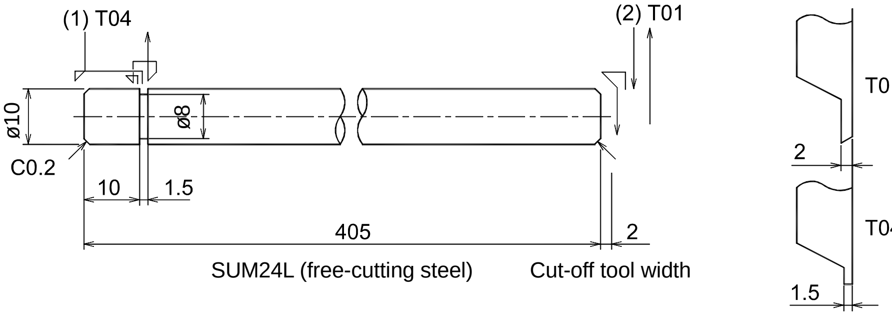
(Explanation of each process)

(1) Adjust the main spindle speed to the back spindle speed. (Examine in advance by MDI.)
(2) Select the cut-off tool.
(3) Position the cut-off tool.
(4) The Z2 axis is superimposed on the Z1 axis.
(5) The back spindle is positioned 1 mm before the end face of the workpiece in rapid feed.
(6) The back spindle advances 20 mm from the end face of the workpiece in cutting feed.
(7) Dwell for 0.5 second
(8) Back spindle chuck close
(9) Dwell for 0.5 second
(10) Queuing
(11) Cut-off machining
(12) Superimpose cancel. The Z2 axis moves to the zero point in rapid feed.

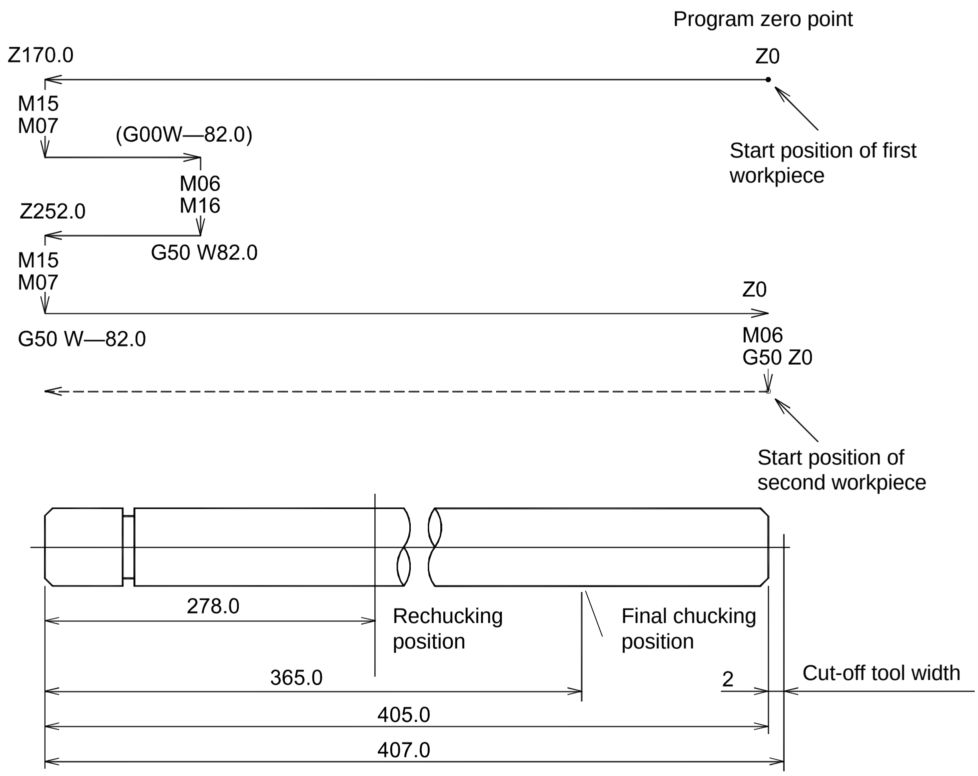
15.10.2 Re-chucking of a long workpiece

In general, the machining length of a workpiece must be within the Z1 axis stroke. If the machining length of a workpiece exceeds the Z1 axis stroke, the workpiece needs to be re-chucked by the main spindle and the back spindle. In this case, create the program including the re-chucking process as shown below.

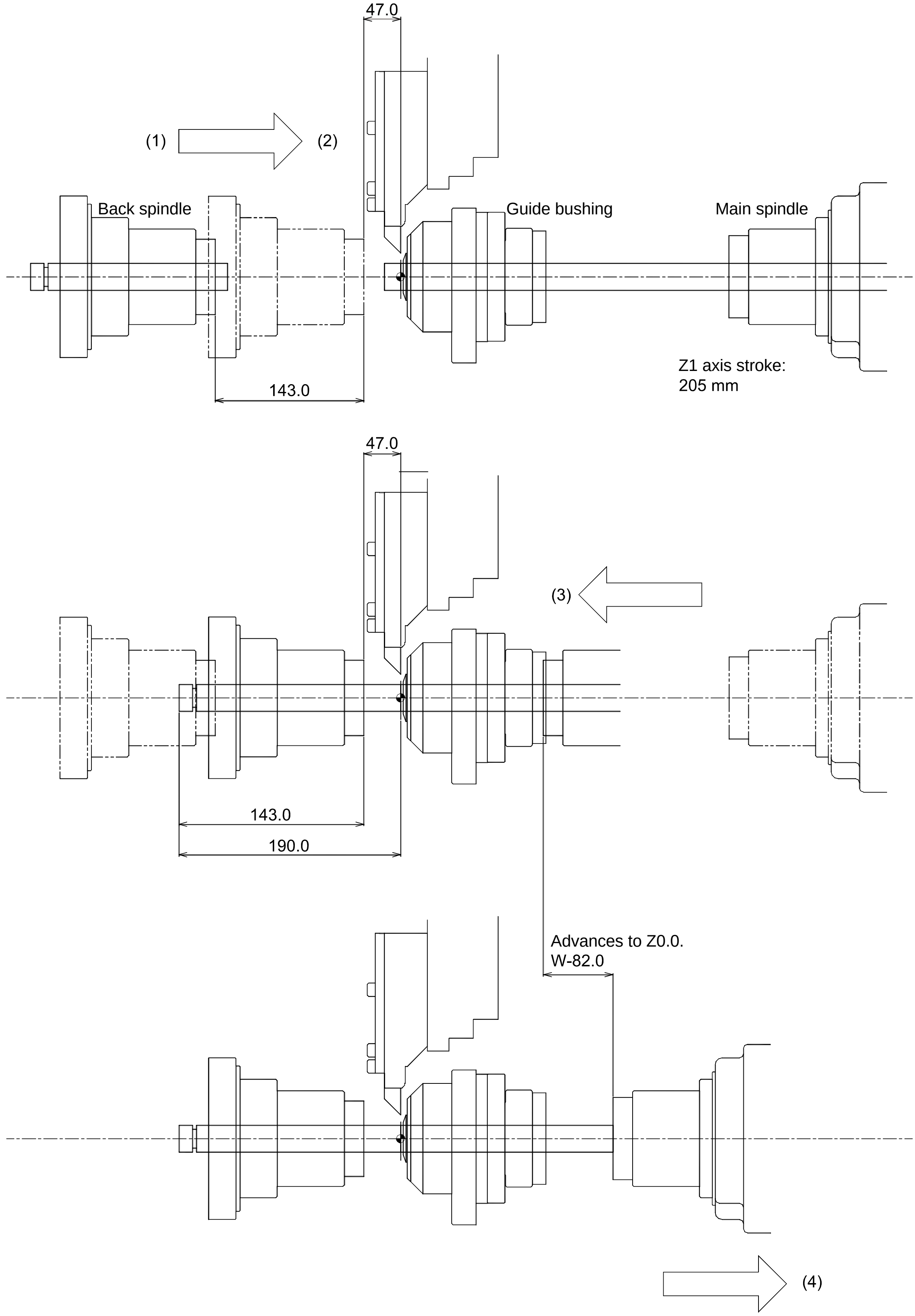
Machining layout



Operation sequence



Operation example



Requirements for program creation:

- Specify the total length of the workpiece + the cut-off tool width, as the maximum move distance. (Including end-face turning and cut-off machining which is performed twice)
- The move distance of a chuck must be within the maximum machining length not including the machining position when the workpiece is re-chucked.
- The workpiece machining completion point must be identical to the program zero point of the start point.

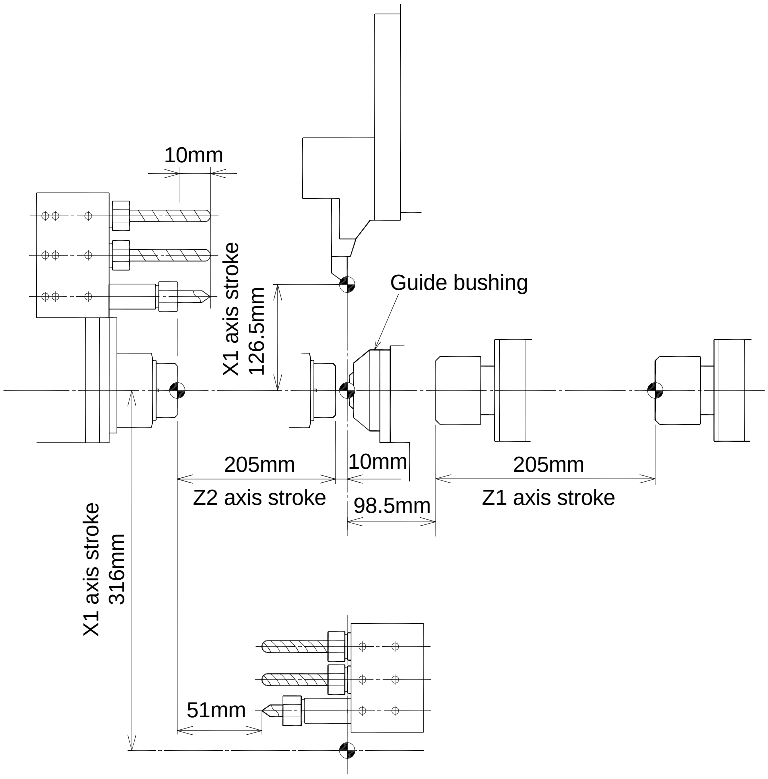
[Sample program]

O0001

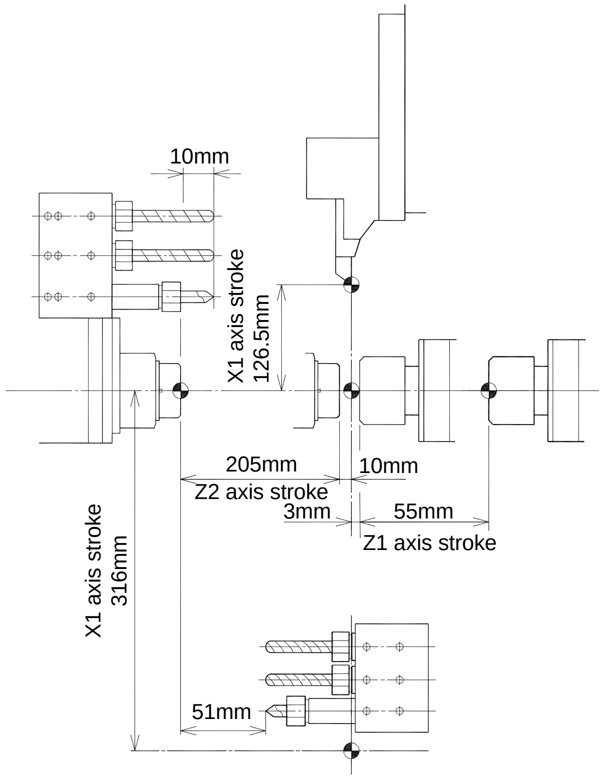
$1		$2	
G50 Z0 M06 G00 X11.0 Z–0.5 G99 M3 S1=3300 N0101 T0400 G00 X11.0 Z0.7 T04 G01 X8.6 Z–0.5 F0.03 X11.0 F0.2 Z11.5 X8.0 F0.03 X11.0 F0.2 W–0.7 X9.6 W0.7 F0.03 X11.0 F0.2 W0.7 X9.6 W–0.7 F0.03 X11.0 F0.2			
N0204 T0100	Select the cut-off tool		
M05	Spindle stop		
!2 L1	Waiting for !1L1	!1 L1	Waiting for !2L1 (1)
		T3000	
		M16	Back spindle chuck open
		G50 Z0	
		G0 Z☐	The back spindle advances ☐ mm from the Z2 axis zero point (2)
		G04 U0.5	
!2 L2	Waiting for !1L2	!1 L2	Waiting for !2L2
G00 Z–205.0 T00	Determines rechucking position, and cancels offset (3)		
G04 U0.5			
M15	Back spindle chuck close		
G04 U0.5			
M07	Main spindle chuck open		
G04 U0.5			
G00 W–82.0	The Z1 axis retracts to –82.0 (4)		
M06	Main spindle chuck close		
G04 U0.5			
M16	Back spindle chuck open		
G04 U0.5			

$1	$2
G50 W82.0 .... Z axis coordinate system setting	
S1=3300 M03 .... The main spindle rotates forward at the speed of 3300 min^{-1}	
S2=3300 M24 .... The back spindle rotates backward at the speed of 3300 min^{-1}	
G00 X11.0 Z407.0 T01	
G650	G650 W0 .... Z1-Z2 superimpose
	G98 G1 Z□
	0 F3000
	G04 U0.5
	M15
!2 L3	!1 L3
G01 X9.6 F0.03	
X11.0 F0.2	
W–0.7	
X9.6 W0.7 F0.03	
X–0.5	
G600	G600
X–3.0	
M05	
M25	
M07	
G50 W–82.0	
G0 X–3.0 Z0 T00	
M56	
G999	G999
N999	N999
M02	M2
%	%

□: Program portion for long workpieces.



Stroke diagram (GB Specification)



Stroke diagram (GBL Specification)

Back spindle stroke

When long workpiece device is used:
The forward end position 205 – (Number of tool square) – (10 mm stabilizer) retract 0 mm

[Note]

When the back spindle cap nut is replaced with the one for the long workpiece device, the stroke changes by the amount of the stabilizer (made of brass). In this case, set "SUPPORT" for "Back Spindle" on the Machine Data screen.

15.11 Executing the Last Program (G999)

This command must be specified in the last portion (end process) of each axis control group ($1, $2) 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 Last program execution

Axis control group

Specify this command for both the axis control groups $1 and $2.

[Sample program]

$1	$2
:	:
!2 L20	!1 L20
G1 X–1.0 F0.03 .... Cut-off process	
G600	G600
G1 X–3.0 F0.05	
/G813	
M5	
M25	
M56	
G999	G999
G0 X11.0 W[***–1.0] .... Move the cut-off tool and workpiece (1)	
!2 L100	!1 L100
	M23 S2=2000 .... Back machining process
	G99
	G44
	T5100
	:
	M141
	T3300
	:
	M25
	M34

$1		$2
G600		**G600**
M3 S1=3000 G99		
T100	Cut-off process (2)	
G0 X11.0 W[* +1.1]**		
G1 X–3.0 F0.03		
N999		**N999**
M5		
M7		
G0 X–3.0 Z[*] T0**	[***] represents the workpiece coordinate set at start time	
M2		**M2**
%		**%**

[Note]

- The last program is not executed unless the menu key [Last PRT] is set to ON.
- Be sure to close the front chuck before selecting a tool in the last program. If selected with the chuck opened, the material may extrude from the guide bushing.
- Be sure to terminate the last program with cut-off the workpiece. If the T51's holder (sleeve holder) is not specified in the last program and only the T31's holder is specified, you need not move the cut-off tool. Accordingly, the processes (1) and (2) can be omitted.
- Be sure to coincide with the machine coordinates of X1, Y1, Z1, and Z2 axes before calling the last program and after it was called when terminating the last program.
- Cancel the coordinate system shift command and offset command before executing the G999 command.
- The M56 (product count) command must be specified before the G999 command.

15.12 Pick-off Failure Detect (M50)

Use this command to detect a cut-off tool breakage. After a workpiece is re-chucked, the main spindle rotates at low speed (approx. 200 min^{-1}) and detects the back spindle rotation to determine if the cut-off tool is broken or not.

[Command format]

M50 Pick-off failure detect

[Sample program]

$1	$2
:	
M3 S1=2000 M24 S2=2000	
G114.1 H1 D-2	
T0100	
G0 X21.0 Z20.0 T01	
G650	G650
	G0 Z-1.0
	G98 G1 Z10.0 F3000
	M77
	M15
!2 L1	!1 L1
G1 X-3.0 F0.03	
G113	
M5 M25	
M3 S1=200	
G4 U2.0	
M50	
G600	G600
:	:

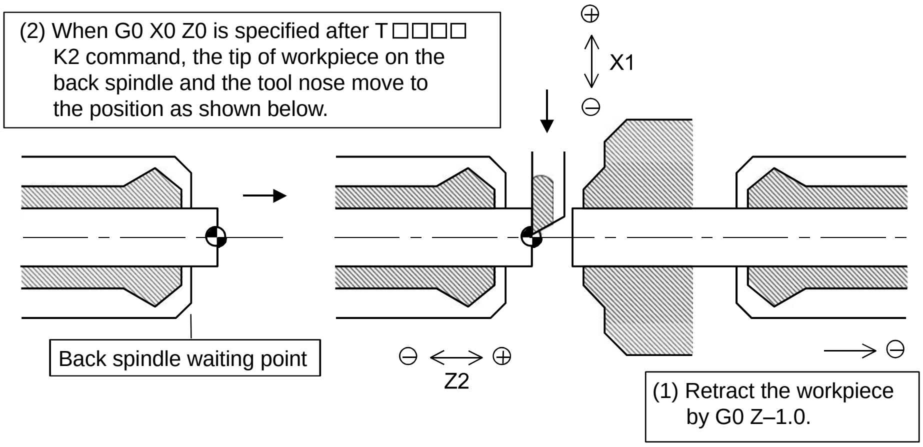
[Note]

- Specify M50 after the main spindle rotation becomes constant.
- The main spindle rotation shall be approx. 200 min^{-1}.
- M50 may fail to detect a cut-off tool breakage even when the tool has been broken if the workpiece on front spindle is apart from the workpiece chucked at back spindle due to some reasons.

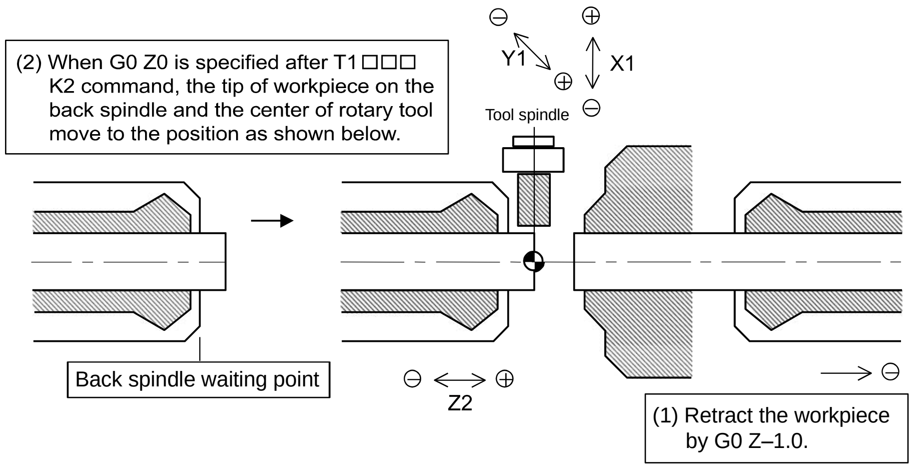
15.13 Machining with Tool or Rotary Tool in Back Machining

Operation example

Machining with tool



Machining with rotary tool



[Sample program]

```
$1
:
T300 K2 ..............................
G0 X21.0 Z0 T7
G1 X0 F0.1
:
G0 X21.0 ............................
G0 Z–10.0 T0
T□□□□ K2 .......................
:
```

[Note]

- The positioning point of Z2 axis is changed depending on the squares of tool (□12 or □13) selected in tool layout pattern. (The position of the rotary tool Z2 axis is not changed.)
  In the free tool layout, □13 is used.
- Specify the command to stop rotation of rotary tool in $1. (See *<12.7.4 Rotary tool on gang tool post rotation and stop>*.)
- To perform machining with rotary tool in $1, specify M78 and M48 after issuing T□□□□ K2 command.
  (See *<12.3 T Functions (Tool Codes)>*.)
- When machining the workpiece on back spindle by using the rotary tool, the back spindle may interfere with the gang tool post. If "interference check" function is disabled (see *<12.7.13 Interference check function (M88, M89)>*, pay strict attention to interference between the gang tool post and the back spindle when machining the workpiece.
- When machining workpiece in longitudinal direction, pay attention to interference between the workpieces on back and main spindles.
- When machining the workpiece with gang rotary tool while performing center support on the machine, the gang rotary tool driving device may interfere with the coolant nozzle located at upper portion of back spindle, depending on the position of back spindle (Z2 axis).

15.14 Back Spindle Feed Per Rotation ON/OFF (G44, G43)

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

The feed rate of feed per rotation is generally determined by the main spindle speed. When the G44 is specified for machining a workpiece at the back spindle, feed per rotation is performed at the back spindle speed in the specified axis control group.

If G600's machining patterns are used, the machining pattern command automatically controls the back spindle in the feed per rotation mode and, therefore, G43 and G44 are usually not necessary.

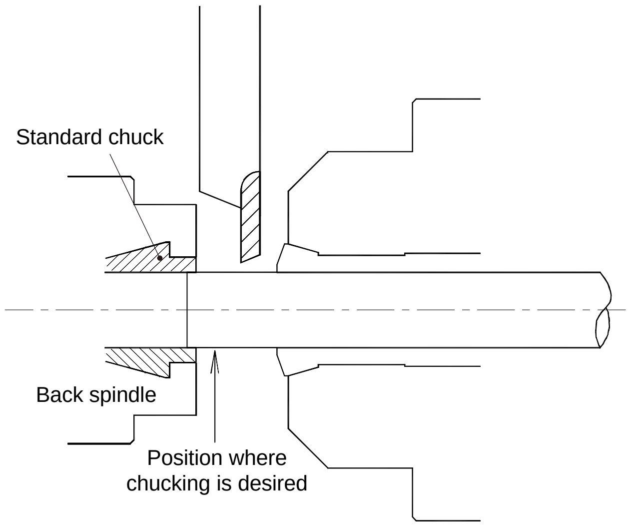
The following table lists back spindle controls for individual machining patterns.

Machining pattern	$1	$2
Free pattern (machining pattern cancel) (G600)	–	ON
$1 single machining (G610)	–	ON
Inner/outer diameter simultaneous machining (G620)	–	OFF
Front/back parallel machining (G630)	–	ON
Pick-off (G650)	–	OFF
Front/back simultaneous machining (G660)	–	ON

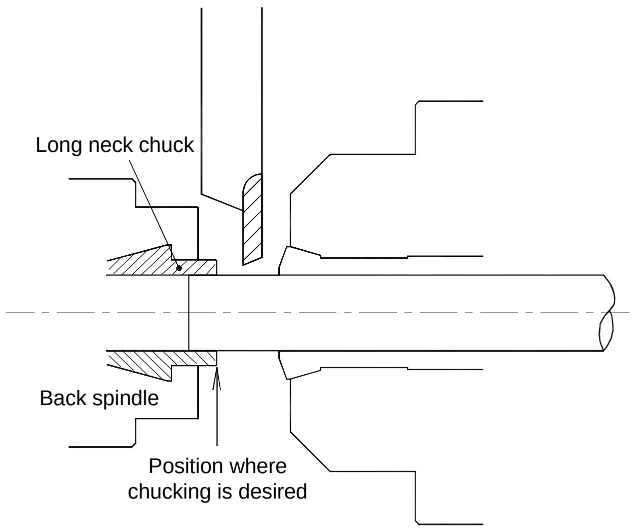
15.15 Use of "Back Chuck Extend Length" in Machining Data

A long-neck chuck may be used if right-handed cut-off tool must be used for machining and the back chucking position is close to the cut-off position.

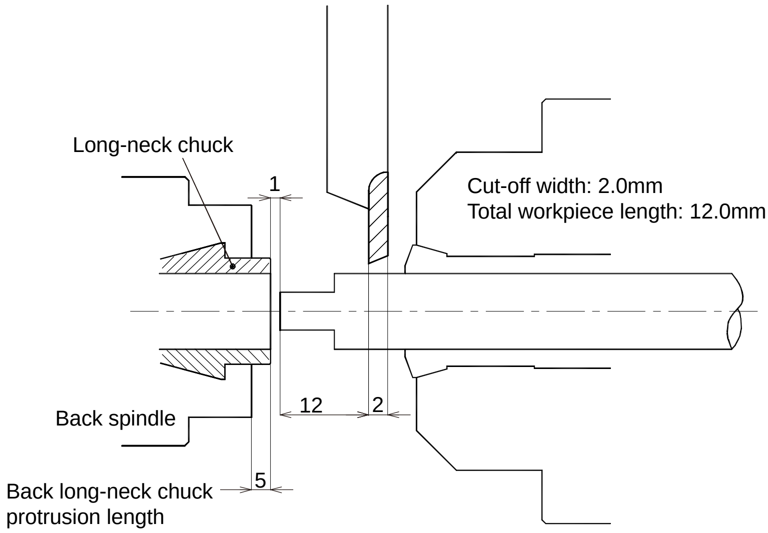
- Use of standard chuck



- Use of long-neck chuck



When a long-neck chuck is used, programming can be easier by entering the protrusion length of the long-neck chuck from the cap nut end face in machining data "Back chuck extend length."



In the figure above, the long-neck chuck protrusion length is 5.0 mm. Thus, enter "5.0" as the machining data.

Then if the machining data is added in pick-off (G650) and G0 Z–1.0 is specified in $2, the end face of the long-neck chuck front end is positioned in front of the front workpiece end face by 1.0 mm.

In addition, enter the workpiece protrusion length from the chuck end face as machining data "Back work extend length." In this example, enter "3.0".

[Sample program]

$1	$2
:	
T100	
G0 X21.0 Z14.0 T1	
G650	G650
	M72
	M16
	G0 Z–1.0 .... The back spindle moves to the position shown in the figure above.
	M77
	G98 G1 Z9.0 F3000
	G4 U0.2
	M15
!2 L1	!1 L1
G99 G1 X–1.0 F0.05	M73
:	:

[Note]

- Select the long-neck chuck and the cut-off tool properly by calculating their shapes which may not cause any interference between them to occur.
- To use a program for an existing machine as a program using long-neck chucks, change machining data "Back chuck extend length" and the pick-off part in the program.

15.16 Back Spindle Torque Limit (M122, M123, M124)

This command enables the back spindle to re-chuck workpiece that has been chucked by the main spindle. When performing re-chucking without using the Spindle synchronization control (G114.1), the back spindle overload error (S03 SV alarm NR0051 S2) occasionally occurs. The back spindle torque limit command is effective in such a case.

[Command format]

M122	Select back spindle torque limit L (25%)
M123	Select back spindle torque limit H (50%)
M124	Back spindle torque limit OFF (100%)

[Sample program]

$1	$2
:	
M3 S1=2000 M24 S2=2000	
T100 (Cut-off)	
G0 X11.0 Z32.0 T1	
G650	G650
	M72
	G0 Z-1.0
	G98 G1 Z10.0 F3000
	M123 Back spindle torque limit H
	G4 U0.5
	M15
! L10	! L10
G1 X-1.0 F0.03	
G600	G600
X-3.0 F0.05	
M124 (OFF)	
M5 M25	
M7	
:	:

[Note]

Specify the back spindle torque limit commands (M122 and M123) only while the back spindle rotates without speed changes. If the back spindle speed is changed (accelerated or decelerated) in the back spindle torque limit enabled state, back spindle rotation will become unstable.

15.17 Pick-off Cancel (G231)

This command can move the back headstock (Z2 axis) to the fixed point with the X1 axis staying in an arbitrary position while the cut-off is being executed with the pick-off command G650.

[Command format]

G231 X☐ F☐ A☐ Z☐

Axis control group

Specify this command for $1.

[Argument]

X☐	Specify the X1 axis workpiece 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 workpiece coordinate specified by the A argument during cut-off machining, the back headstock (Z2 axis) start moving.
Z☐	Specify the retract position (in machine coordinate) of the back spindle (Z2 axis). If the argument is omitted, the Z2 axis moves to the return position.

[Sample program]

$1	$2
G600	G600
:	:
G99 M03 S1=1500 M24 S2=1500	:
G114.1 H1 D–2	:
T0100	:
G00 X13.0 Z85.0 T01	
G650 .... Pick-off	G650 .... Pick-off
	G00 Z–2.0
	G98 G01 Z10.0 F1000
	G4 U0.5
	M77
	M15
!2L1	!1L1
G231 X–3.0 F0.03 A–1.0	
.... Pick-off cancel (When the X1 axis reaches the workpiece coordinate – 1.0, the back headstock moves to the backward end position.)	
:	:

[Note]

- The superimposition of the Z1 and Z2 axes is canceled when the G231 command is executed.
- The G231 does not permit arc cut-off machining.

15.18 Back Machining Program Skip Function (M239)

The function checks whether a workpiece is clamped or not in the back spindle and if it judges there is no workpiece in the back spindle, the program blocks preceded by "M239" and followed by "N19999" are skipped. This judgment is made only in the first cycle after executing the [ST POS] on the Preparation screen and not made in the second and later cycles. The function is effectively used for the machine equipped with the knock-out jig for through-hole workpieces so that the back machining is not performed mistakenly although a workpiece is not held in the back spindle to prevent the knock-out jig for through-hole workpieces from being cut erroneously.

CAUTION

Whether a workpiece is clamped in the back spindle chuck or not is checked by advancing the knock-out rod. Since the rod is pushed against the workpiece that is clamped in the chuck, the workpiece may fall, move, or is damaged if chucking depth is short or chucking force is weak. Therefore, the function cannot be used if a workpiece is not securely clamped in the back spindle chuck.

In dry run operation (warm-up operation) that does not insert the material into the guide bushing, use the optional block skip function.

[Command format]

M239 K1☐	Back machining program skip
N19999	Back machining program skip end

[Argument]

K1: Whether a workpiece exists in the back spindle chuck is checked in the cycles that follow the first cycle operation performed after the execution of the [ST POS] on the Preparation screen. When the argument is not specified, existence of a workpiece is checked only for the first cycle operation.

K2: In the first cycle after executing start position operation, the M239 command that specified this command detects whether or not a workpiece is clamped in the back headstock using knock-out. Subsequent M239 commands use the result of detection to determine whether to skip program blocks. When not specified, the M239 command in the first cycle is the only command performed and knock-out is used to determine whether a workpiece is clamped in the back headstock.

[Sample program]

$2	
M239	Back machining program skip *1
M23 S2=3000	
T3100	
:	
T3300	
:	
N19999	
M25	
M16	
:	

[*1] Existence of a workpiece, clamped in the back spindle chuck, is checked by M239. If judgment is "no workpiece", the function skips the program blocks preceded by M239 and followed by N19999.

■No K argument

$2	First cycle	On and after second cycle
M239	Knock-out check No workpiece: Skip to N19999 Workpiece present: To the next block	No knock-out check To the next block
:		
N19999		
:		
M239	Knock-out check No workpiece: Skip to N19999 Workpiece present: To the next block	No knock-out check To the next block
:		
N19999		

■K1 argument

$2	First cycle	On and after second cycle
M239 K1	Knock-out check No workpiece: Skip to N19999 Workpiece present: To the next block	No knock-out check To the next block
:		
N19999		
:		
M239 K1	Knock-out check No workpiece: Skip to N19999 Workpiece present: To the next block	No knock-out check To the next block
:		
N19999		

■K2 argument

$3

	First cycle	On and after second cycle
M239 K2	Knock-out check No workpiece: Skip to N19999 Workpiece present: To the next block	No knock-out check To the next block
:		
N19999		
:		
M239 (K2)	Knock-out check The result of the previous cycle No workpiece: Skip to N19999 Workpiece present: To the next block	No knock-out check To the next block
:		
N19999		

[Note]

- Sequence number N19999 cannot be used for other purposes.
- If the back spindle chuck is open when the back machining program skip function (M239) is executed, the back machining program is skipped unconditionally.

Product Code	C-L220E XII
Document Code	2E1-1500
Mfg. No.	L220E/0001 ~
Issue Date	2013.10

16. Program Example

16.1 Program Creation

Following programs are only examples. During actual machining, the optimum spindle speed and feed must be selected in consideration of various conditions.

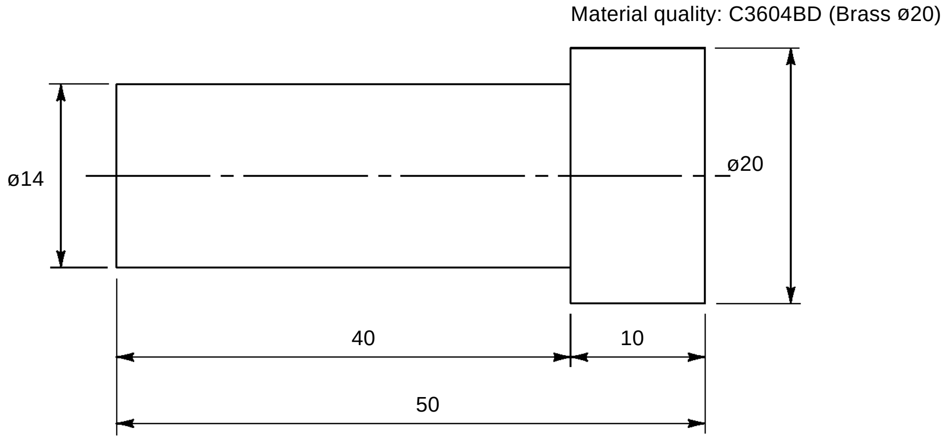
The sample programs 1 and 2 show programs created with the machining pattern and those created without the machining pattern.

- Example 1: Front machining
- Example 2: Front machining with opposite tool post
- Example 3: Back machining
- Example 4: Front/back simultaneous machining
- Example 5: Machining with rechucking long workpiece
- Example 6:
- Example 7:

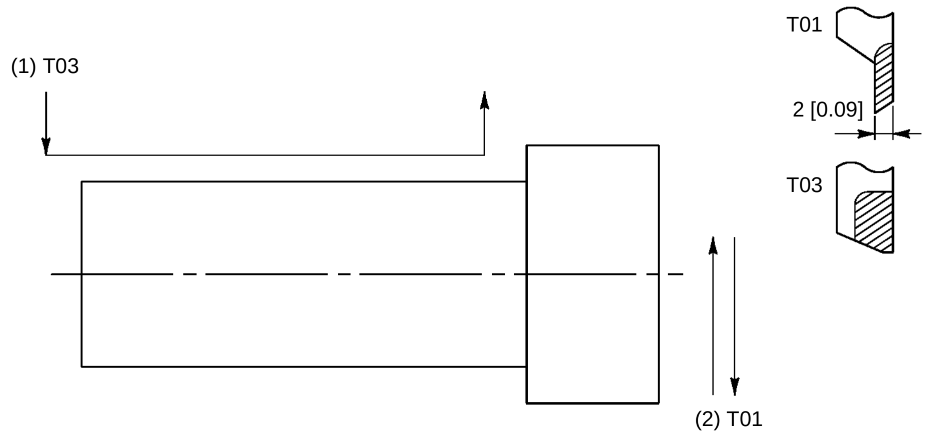
16.1.1 Example 1 (Front machining)

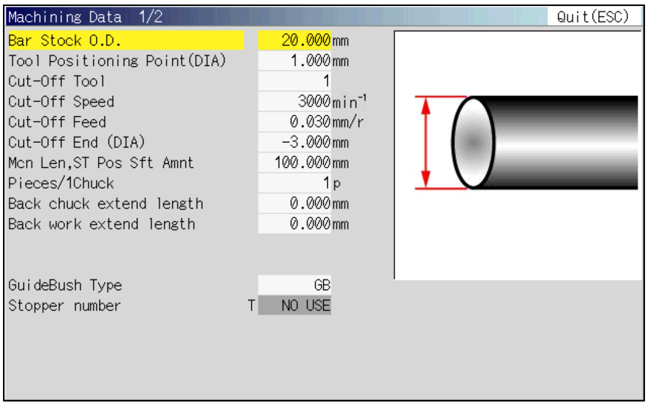
This example assumes the workpiece receiver box collection mode.

Machining drawing

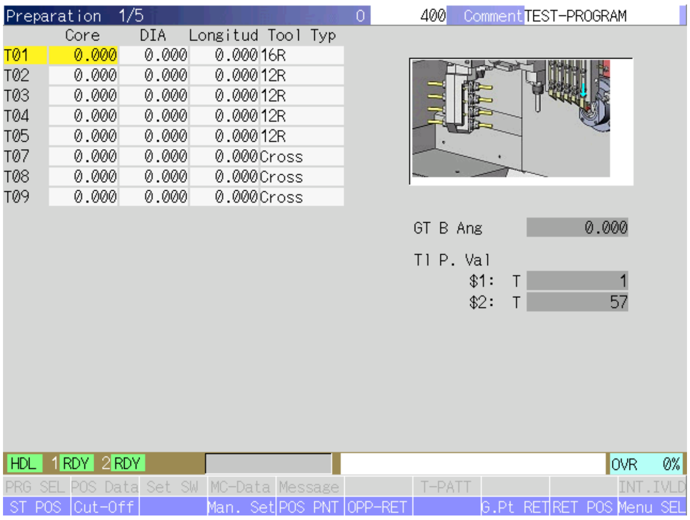


Tool layout

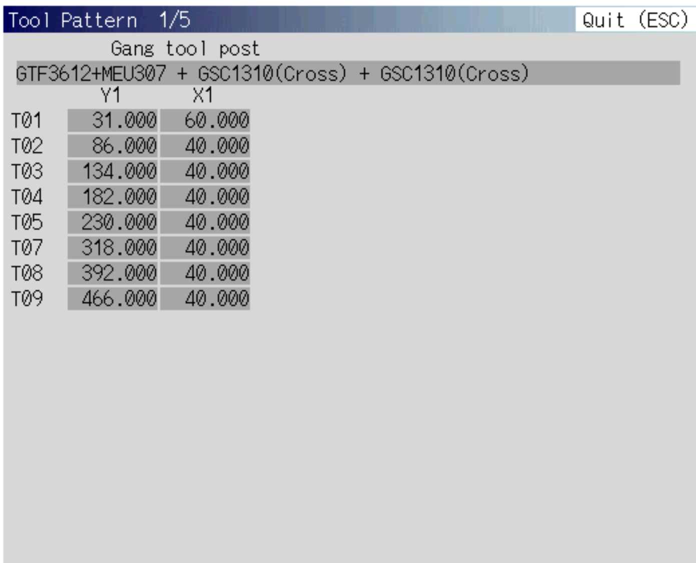




Machining data



Tool set



Tool layout pattern

[Sample program]

Machining pattern G600s included, two axis control groups used

O0001 .... Program No.

$1	$2	
		.... Axis control group
G50 Z0		.... Coordinate system setting (Figure 1)
M06		.... Chuck close
G00 X21.0 Z–1.0		.... Move the material from the tool. (Figure 2)
M03 S1=2500 G99		.... Main spindle forward rotation (2,500 min^{-1}) Per rotation feed
G610	G610	
N0103 T0300		.... Sequence number and T03 tool selection
	M34	.... A series of operations for product separation
G00 X21.0 Z–1.0 T01		.... Positioning to ø14 in rapid feed Compensation No. 1 command (Figure 3)
G01 X14.0 F0.5		
Z40.0 F0.05		.... Cut to 40 mm in longitudinal direction (Figure 4)
X21.0		.... Cut to the material outer diameter + 1.0 mm (Figure 5)
!2 L1	!1 L1	
N0201 T0100		.... Sequence number and T01 tool selection (Figure 6)
G00 X21.0 Z52.0 T02		.... Cut-off positioning compensation No. 2 command in rapid feed (Figure 7)
M320		.... Advance the back spindle to the workpiece separator position (Figure 7)
G99 G01 X–1.0 F0.03		.... Workpiece cut-off (Figure 8)
X–3.0 F0.05		
M141		.... Return the back spindle workpiece separator to Z2 axis zero point (Figure 9)
M05		.... Stop rotating the main spindle
M07		.... Chuck open
G00 X–3.0 Z0 T00		.... Return to the start point.
M56		.... Product count
G999	G999	.... Last program execution command
G610	G610	.... $1 single machining command ON
M34		.... A series of operations for product separation
G600	G600	.... Free pattern (machining pattern cancel)
N999	N999	.... Last program queuing
M02	M02	.... 1-cycle stop
M99	M99	.... Return to the top of a program.
%	%	.... Stop code

[Sample program]

No machining pattern, single axis control group used

O0001
$1
G50 Z0 M06 G00 X21.0 Z–1.0 M03 S1=2500 G99 N0103 T0300 M34 G00 X21.0 Z–1.0 T01 G01 X14.0 F0.5 Z40.0 F0.05 X21.0 N0201 T0100 G00 X21.0 Z52.0 T02 M320 G99 G01 X–1.0 F0.03 X–3.0 F0.03 M141 M05 M07 G00 X–3.0 Z0 T00 M56 G999 M34 N999 M02 M99 %

A parameter setting switch of $2 Cycle Start OFF must be set to ON before the above program can be run.

Main spindle speed

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

$$= \frac{160}{3.14 \times 20} \times 1000$$

$$\approx 2546$$

$$\approx 2500\ \text{min}^{-1}$$

V: 160 m/min

D: Ø20 mm

π: 3.14

From cutting condition table in Chapter 10

F_X = 0.03 mm/rev

F_Z = 0.05 mm/rev

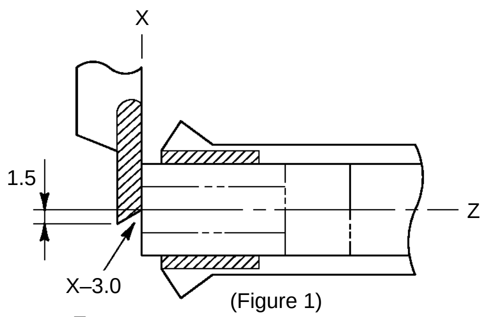
For conversion into per minute feed

F (mm/min) = Spindle speed × F (mm/rev)

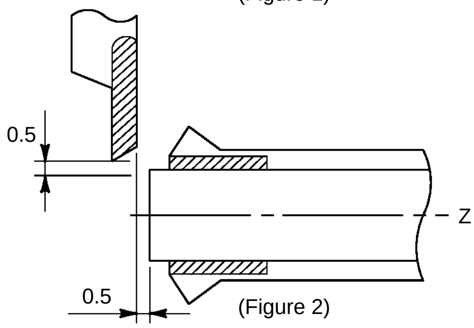
F_X = 2500 × 0.03 = F75

F_Z = 2500 × 0.05 = F125

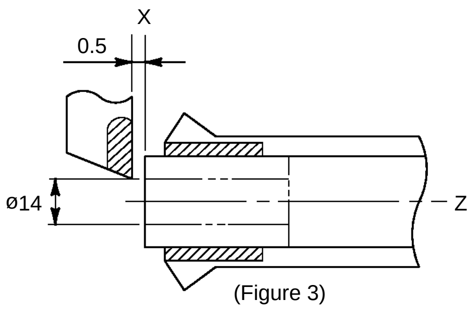
Operation diagram



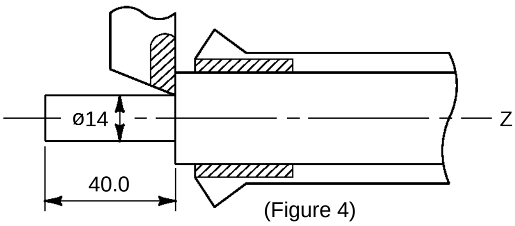
(Figure 1)



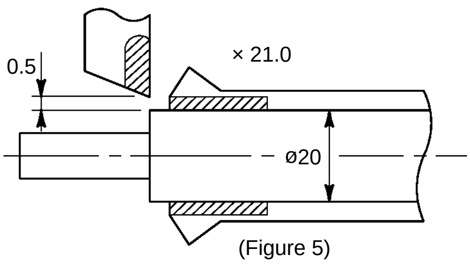
(Figure 2)



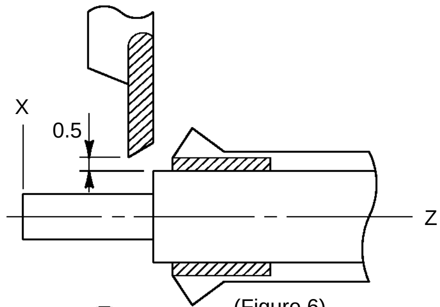
(Figure 3)



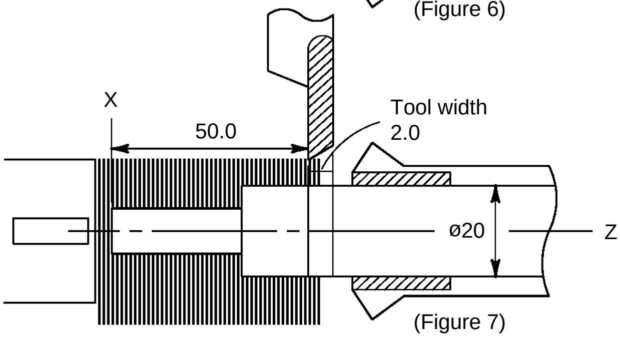
(Figure 4)



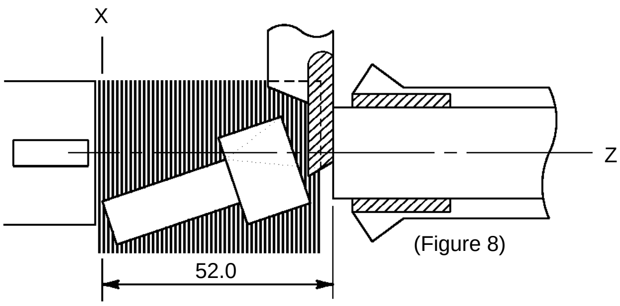
(Figure 5)



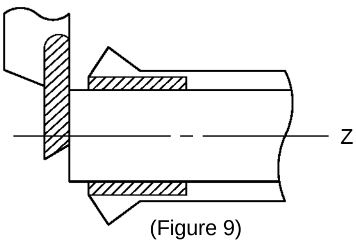
(Figure 6)



(Figure 7)



(Figure 8)

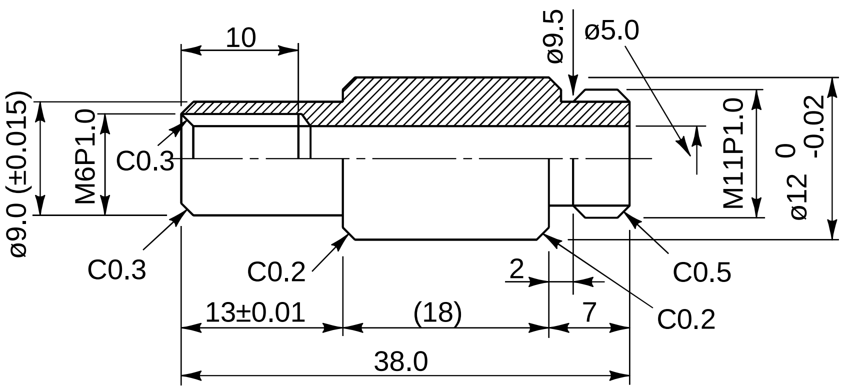


(Figure 9)

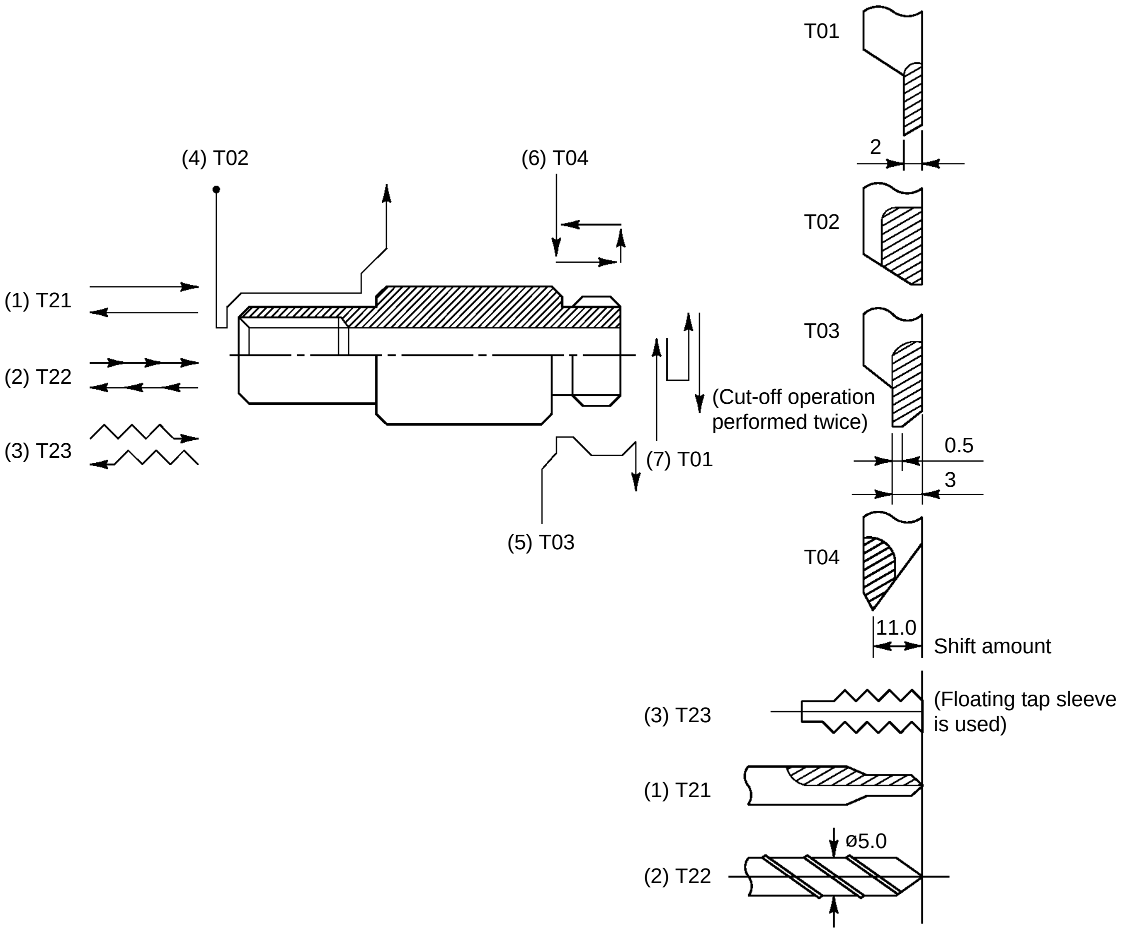
16.1.2 Example 2 (Front machining with opposite tool post)

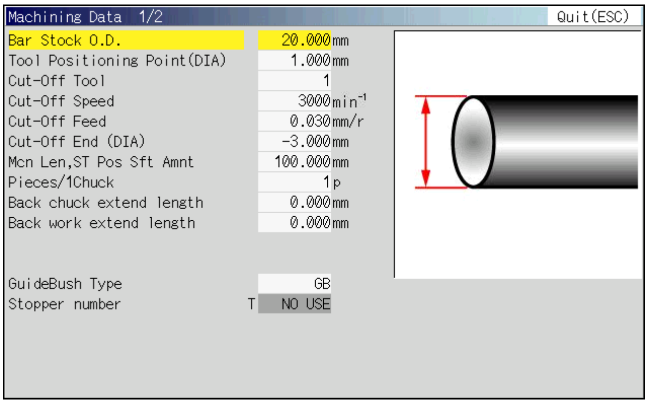
Machining drawing

Cut material: SUM24L (free cutting steel) (diameter 12 mm)

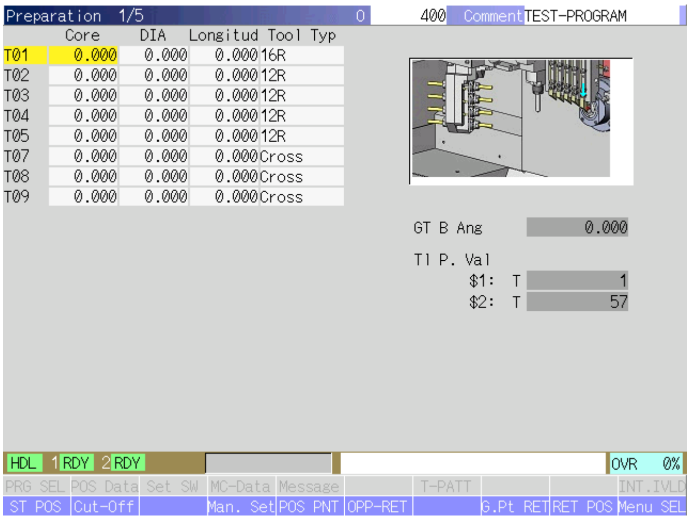


Tool layout

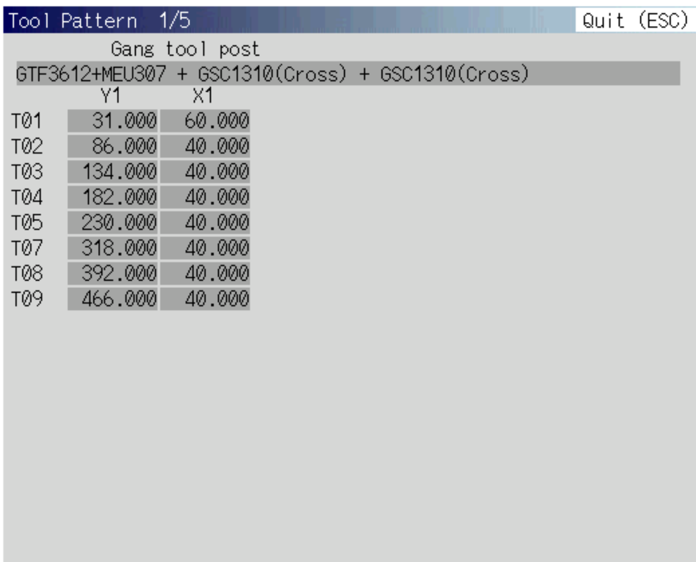




Machining data



Tool set



Tool layout pattern

[Sample program]

Machining pattern G600s included, two axis control groups used

O0002

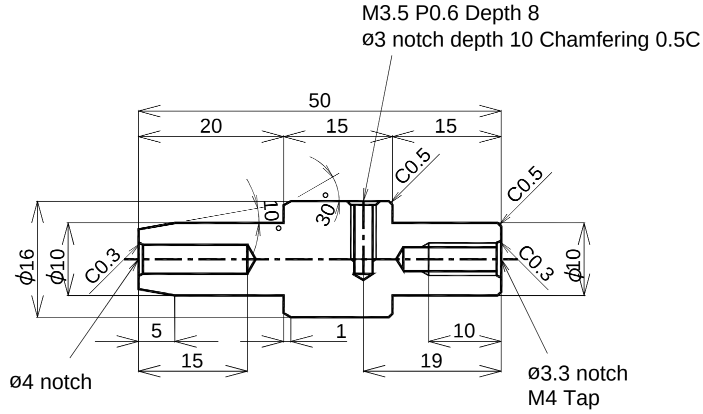
$1	$2	
G50 Z–0.1		
M06		
G113		
G00 X13.0 Z–1.0		
M51		
M03 S1=1800 G99		.... Main spindle forward rotation (2,500 min^{-1}) Per revolution feed (mm/rev)
G610	G610	.... $1 single machining command ON
N0121 T2100		.... The opposite tool post advances while the gang tool post is retracting.
G00 Z–1.0		
G01 Z6.0 F0.08 T1		
G00 Z–1.0 T00		
N0222 T2200		
G0 Z–1.0		
G01 Z16.0 F0.1 T02		.... Start drill in-feed operation
G00 Z–1.0		
G04 U0.5		
Z15.5		
G01 Z27.0 F0.09		
G00 Z–1.0		
G04 U0.5		
Z26.5		
G01 Z34.0 F0.08		
G00 Z–1.0		
G04 U0.5		
Z33.5		
G01 Z40.0 F0.07		.... Last drill in-feed operation
G00 Z–1.0 T00		

$1	$2	
N0323 T2300		
G0 Z–3.0		
M97		
S1=400		
G04 U1.0		
G32 Z12.0 F0.8 T03		.... Forward in-feed operation for M6 thread cutting
Z–3.0 F1.0 M03 T00		.... Reversing
G04 U2.0		
G00 Z–1.0		
M03 S1=2800		
M96		
G630	G630	
N402 T0200	M34	.... Gang tool post is positioned while the opposite tool post is retracting.
G00 X13.0 Z–1.0 T04	M16	
G1 X5.0 F0.5		
Z0 F0.06		
X8.4	M24 S2=2800	
X9.0 Z0.3		
Z13.0		
X11.6		
X13.0 Z13.7		
G0 U0 W0 T0		
N503 T0300		
G00 X13.0 Z33.3 T05		
G01 X11.6 Z34.0 F0.03		
X9.5		
Z35.5 F0.06		
X11.0 Z36.25		
Z40.5		
X10.0 Z41.0 F0.03		
Z42.0 F0.06		
X13.0 F0.2		
G0 U0 W0 T0		
S1=1000		
G4 U1.0		

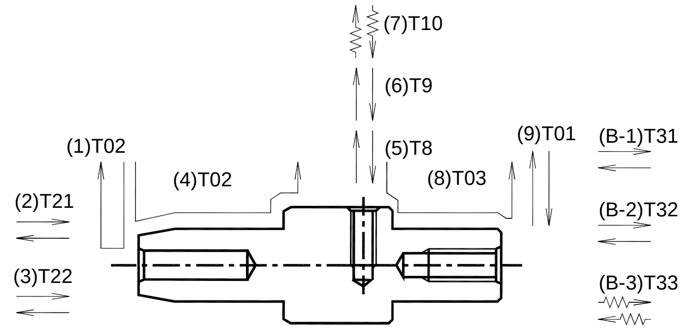
$1	$2
N604 T0400	
G00 X13.0 Z42.0 T06	
G92 X10.56 Z49.0 F1.0	
X10.26	
X10.04	
X9.86	
X9.76	
X9.7	
X9.7	
G00 X13.0 W0 T0	
S1=2800	
G114.1 H1 D–2	
N0701 T0100	
G00 X13.0 Z40.0 T07	
G650	G650
	M72
	G00 Z–1.0
	G98 G01 Z12.0 F1000
	G04 U1.0
	M77
	M15
	M73
!L1	!L1
G01 X4.0 F0.03	
G600	G600
G00 X13.0 W–0.5	
Z42.0	
G01 X–3.0 F0.03	
G113	
M05 M25	
M07	
G00 G0 X–3.0 Z–0.1 T00	
M56	
G999	G999
	M34
N999	N999
M02	M02
M99	M99
%	%

16.1.3 Example 3 (Back machining)

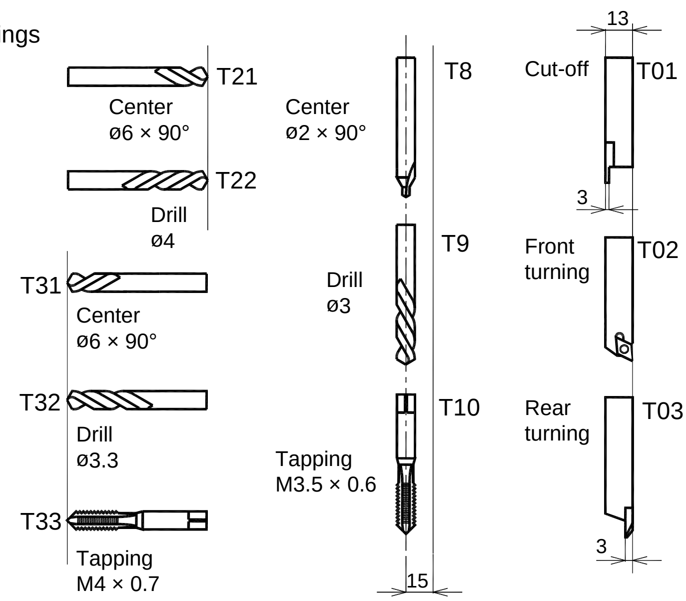
Machining drawing



Tool layout



Cutting tool drawings



Machine condition settings

Machining data settings

Material outer diameter:	16.000mm
Machining length:	70.000mm
Product count per chuck:	1
Tool positioning point:	1.0mm
Back spindle chuck position:	16.0mm

Mount the tool T9 (drill diameter: 3 mm) after specifying 5 mm in DIA field in Manual Tool Set menu.

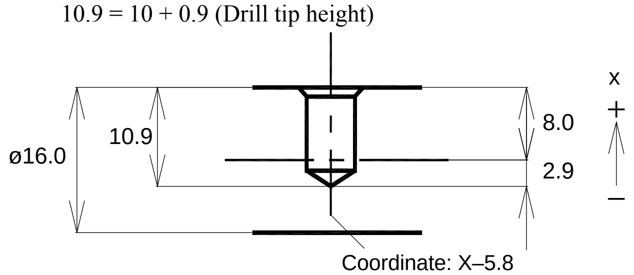
[Sample program]

Machining pattern G600s included, two axis control groups used

O0003

$1	$2	
G50 Z9.9		.... Z9.9 = 13 square – 3 mm width – machining allowance 0.1
G99		
M06		
G113		
G0 X17.0 Z–1.0 M3 S1=2400		
G610	G610	.... $1 single machining
G50 Z9.9		
N1 T0200		
G0 X17.0 Z0 T02		
G1 X–0.5 F0.1		
Z–1.0		
G0 X17.0		
U0 W0 T0		
S1=1061		
N2 T2100		.... The opposite tool post advances while the gang tool post is retracting.
G0 Z–1.0		
G1 Z2.7 F0.08 T21		
G0 Z–1.0 T0		
S1=1600		
N3 T2200		
G0 Z–1.0		
G1 Z12.0 F0.06 T22		
G0 Z–1.0		
G4 U0.5		
Z11.5		
G1 Z16.15 F0.06		
G0 Z–1.0 T0		
G630	G630	.... Front/back parallel machining
S1=2400	M98 H1	.... "H" calls the sequence number; "H1" calls N1.
	G99 M24 S2=1600	.... Gang tool post is positioned while the opposite tool post is retracting.

$1	$2	
N4 T0200		
G0 X17.0 Z–1.0 T02		
G1 X8.06 F0.5		
X10.0 Z5.0 F0.03		
Z20.0 F0.05		
G4 U0.2		
X14.845		
X16.0 Z21.0 F0.03		
W0.5 F0.1		
X17.0		
M5		
G98		
M80 S3=1300		
N5 T0800		
M28 S0		
G50 W–15.0		
G0 X17.0 Z31.0 T08		
G1 X7.6 F50		.... X7.6 = ø16 – 4.2 × 2
G0 X17.0 T0		
G50 W15.0		
M80 S3=1600		
N6 T0900		
G50 W-15.0		
G0 X17.0 Z31.0 T09		
G1 X-5.8 F80		.... X-5.8 = ø16 – 10.9 × 2 10.9 = 10 + 0.9 (Drill tip height)
G0 X17.0 T0		
G50 W15.0		
M80 S3=0		



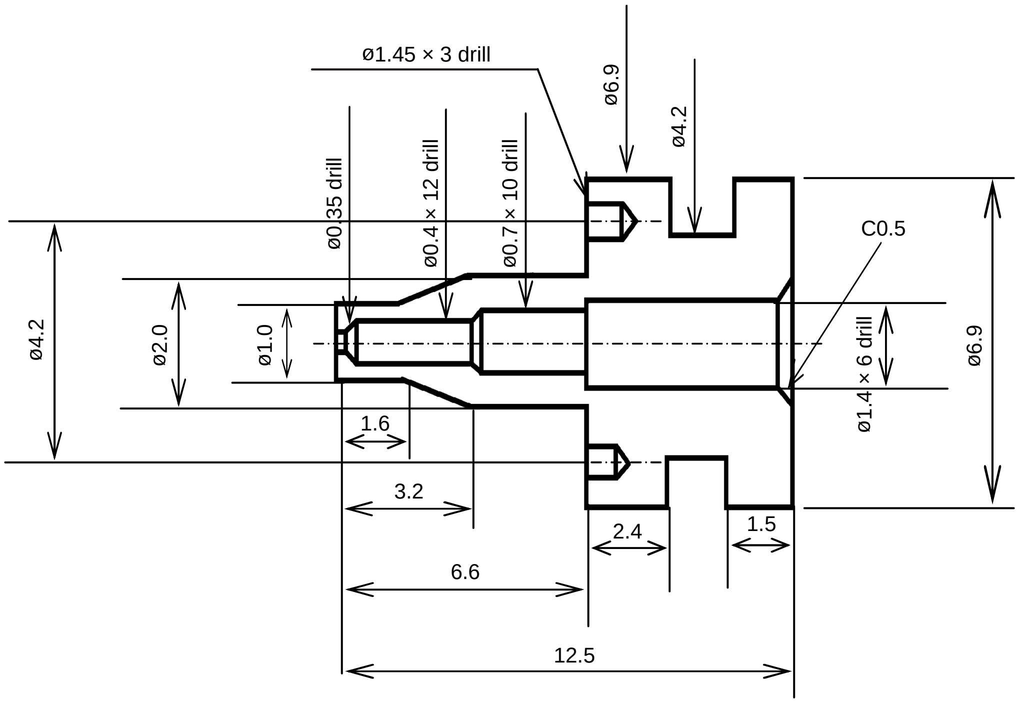
$1	$2	
N7 T1000		
G50 W-15.0		
G00 X21.6 Z31.0 T10		.... X21.6 = ø16 + {(1 + 1.8) × 2} = 21.6 1.8 = 0.6 (Pitch) × 3
G88 X0 R1.0 F0.6 D3 S500, R1		
G80		
G0 X22.0 T0		
G50 W15.0		
M82		
M20		
G99		
M3 S1=2000		
N8 T0300		
G50 W–3.0		
G0 X17.0 Z34.0 T03		
G1 X15.0 Z35.0 F0.015		
X10.0		
Z49.5 F0.03		
X8.8 Z50.1 F0.015		
X17.0 F0.1		
G50 W3.0		
S1=1600		
!2L1	!1L1	
G114.1 H1 D–2		
N9 T0100		
G0 X17.0 Z63.0 T01		
G650	G650	.... Pick-off, Center support
	M72	

$1	$2
	G0 Z–1.0
	G98 G1 Z34.0 F1000
	G4 U0.5
	M77
	M15
	M73
!2L2	!1L2
G1 X-1.0 F0.02	
G600	G600
G1 X-3.0 F0.05	
G113	
M5 M25	
M7	
G0 X-3.0 Z9.9 T0	
M56	
G999	G999
	M98 H1
N999	N999
M2	M2
M99	M99
	N1
	G99 M23 S2=1061
	T3100
	G0 Z–1.0
	G1 Z2.3 F0.05 T31
	G0 Z–1.0 T0 S2=1900

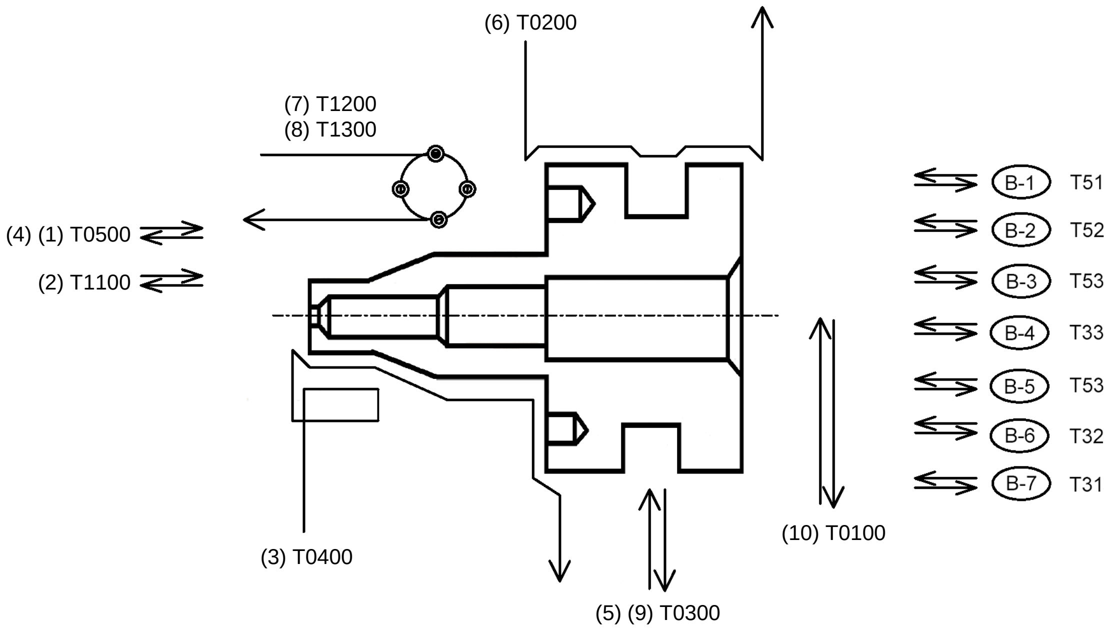
$1	$2	
	T3200	
	G0 Z–1.0	
	G1 Z6.6 F0.05 T32	
	Z–1.0 F1.0	
	G4 U0.5	
	Z6.1	
	Z8.6 F0.05	
	Z–1.0 F1.0	
	G4 U0.5	
	Z8.1	
	Z10.6 F0.05	
	Z–1.0 F1.0	
	G4 U0.5	
	Z10.1	
	Z12.6 F0.05	
	Z–1.0 F1.0	
	G4 U0.5	
	Z12.1	
	Z14.6 F0.05	
	Z–1.0 F1.0 T0	
	S2=500	
	G4 U1.0	
	T3300	
	G0 Z–1.5	
	G01 Z12.1 F0.56 T33	.... With tapping sleeve used Without back spindle chasing function (option)
	Z–1.5 F0.7 T0 M24	
	G4 U2.0	
	G0 Z–5.0	
	M25	
	M34	.... A series of operations for product separation
M99	M99	
%	%	

16.1.4 Example 4 (Front/back simultaneous machining)

Workpiece drawing



Tool layout

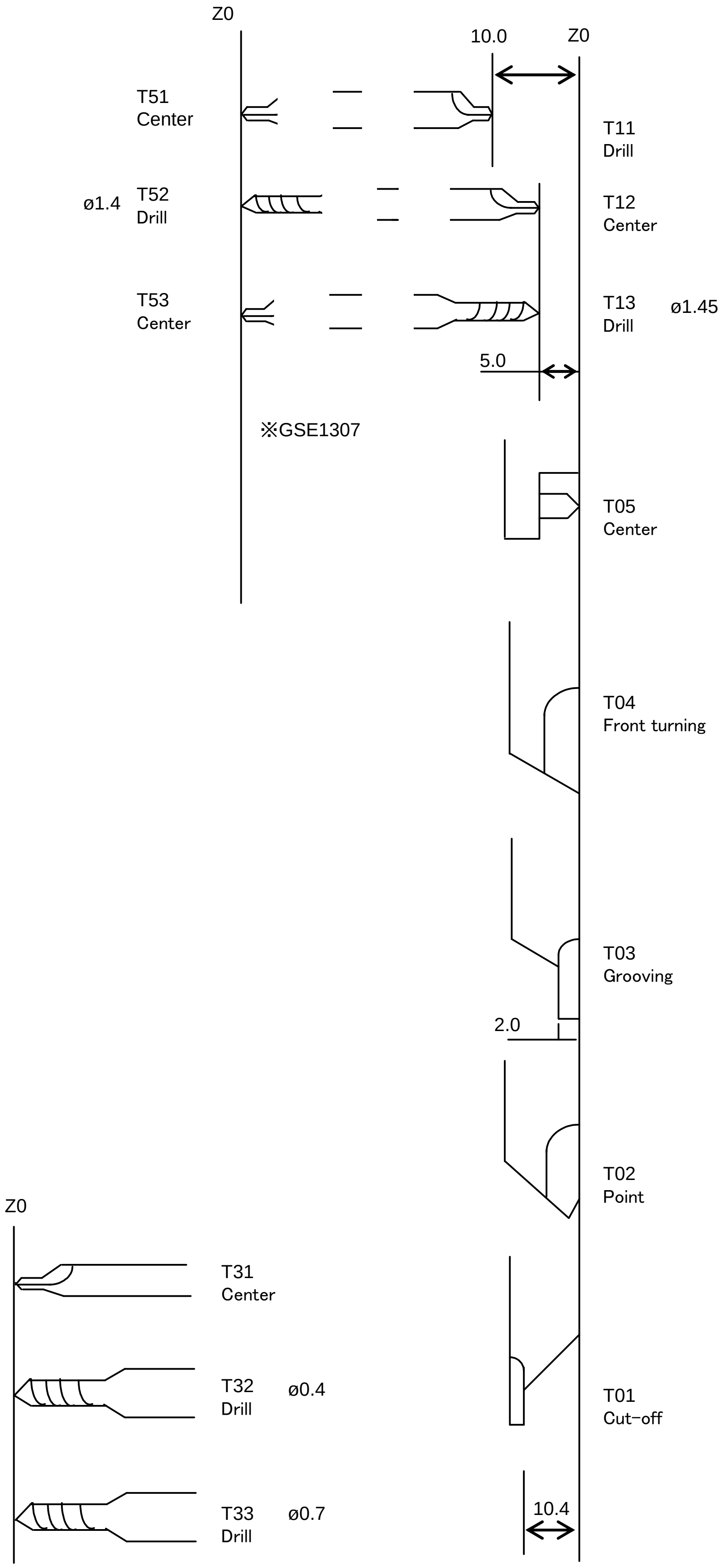


Machine condition settings

Machining data settings

Material outer diameter:	8 mm
Machining length:	20 mm
Product count per chuck:	1
Tool positioning point:	1 mm
Back spindle chuck position:	4 mm

Tool layout



Z0
10.0
Z0
T51
Center
T11
Drill
ø1.4
T52
Drill
T12
Center
T53
Center
T13
Drill
ø1.45
5.0
※GSE1307
T05
Center
T04
Front turning
T03
Grooving
2.0
T02
Point
Z0
T31
Center
T32
Drill
ø0.4
T01
Cut-off
T33
Drill
ø0.7
10.4

[Sample program]

Front/back simultaneous machining G600s, two axis control groups used

O0001

$1	$2	
G50 Z10.4		
M06		
M09		
G113		
G00 X9.0 Z–1.0		
M51 X9.0		
G98 M80 S3=4000		
G600	G600	
	/M98 H100	
	/M98 H1	
	/M25	
	/M98 H200	
G630	G630	
	/M98 H2	
G99 M03 S1=4000		
N01 T500		.... Center
G00 X0 Z–1.0		
G01 Z0.2 F0.02 T5		
Z–1.0 F0.2 T0		
M28 S0		
G98 M80 S3=4000		
N02 T1100		.... Drilling 0.35
G50 W–10.0		
Z–1.0		
G01 Z0.5 F25 T11		

$1	$2
Z–1.0 F500	
G50 W10.0	
G1 Z–1.0 F1000 T0	
M82	
M20	
G99 M03 S1=5000	
N03 T400	
G00 X9.0 Z0 T4	
G01 X–1.0 F0.02	
X3.0 F0.5	
Z3.2 F0.04	
X7.0 F0.03	
X7.9 W0.5 F0.01	
G03 U1.0 W0.5 R0.5 F0.02	
G00 X9.0	
G01 Z–1.0 F1.0	
X–1.0 F0.5	
Z0 F0.01	
X0.8 F0.03	
X1.0 Z0.1 F0.005	
Z1.6 F0.015	
X1.975 Z3.2	
Z6.6 F0.02	
X6.705 F0.03	
X6.905 W0.1 F0.02	
X7.0	
X7.9 W0.5 F0.01	
G03 U1.0 W1.0 R1.0 F0.02	
G00 X9.0	
G01 Z–1.0 F1.0 T0	
S1=4000	

$1	$2	
T500		.... Center
G00 X0 Z–1.0		
G01 Z0.2 F0.01 T5		
G04 U0.1		
G00 Z–1.0 T0		
T300		.... Grooving
G00 X9.0 Z11.0 T3		
G01 X4.2 F0.02		
G00 X9.0		
T200		.... Pointed tool
G50 W–5.5		
G00 X9.0 Z6.1 T2		
G1 X5.7 F0.5		
G01 X6.9 Z6.7 F0.01		
Z8.9 F0.02		
X6.7 Z9.0 F0.015		
Z11.0 F0.03		
X6.9 W0.1 F0.02		
Z12.4 F0.025		
X6.7 W0.1 F0.015		
X9.0 F0.5		
G50 W5.5		
G00 Z3.0 T0		
G50 W–5.5		
G00 X9.0 Z6.1 T2		
M28 S0		
G98 M80 S3=4000		
N04 T1200		.... End face center
G50 W–5.0		
G00 X4.2 Z–2.0 T12		

$1	$2	
G00 Z5.5		
G01 Z7.125 F50		
G00 Z5.5		
M28 S90		
G01 Z7.125 F50		
G00 Z5.5		
M28 S180		
G01 Z7.125 F50		
G00 Z5.5		
M28 S270		
G01 Z7.125 F50		
G01 Z–1.0 T0		
G50 W5.0		
G00 Z5.5		
G01 Z7.125 F50		
N05 T1300		.... End face drilling 1.45
G50 W–5.0		
G00 X4.2 Z–1.0 T13		
G00 Z5.5		
G01 Z8.535 F65		
G00 Z5.5		
M28 S0		
G01 Z8.535 F65		
G00 Z5.5		
M28 S90		
G01 Z8.535 F65		
G00 Z5.5		
M28 S180		
G01 Z8.535 F65		
G0 Z–1.0		
G50 W 5.0		
G00 Z–1.0 T0		
M82		
M20		
G99 M03 S1=4500		
T300		.... Grooving
G00 X9.0 Z11.0 T3		
G01 X4.2 F0.02		
G00 X9.0		

$1	$2	
M03 S1=2500 M24 S2=2500		
G114.1 H1 D–2		
N9 T100		.... Cut-off
G00 X9.0 Z24.50 T1		
G650	G650	
	M16	
	M72	
	G00 Z–1.0	
	M72	
	G98 G01 Z8.5 F2000	
	G04 U0.1	
	M77	
	M15	
	M73	
	G04 U0.1	
!L1	!L1	
G231 X–3.0 F0.03 A–1.0		
G113		
M05 M25		
M07		
M56		
G01 X–3.0 Z10.4 T0		
G999		
M06		
G04 U0.2		
G00 X9.0 Z–1.0 T0		
G98 M80 S3=4000		
G600	G600	
	M98 H100	
	M98 H1	
	M25	
	M98 H200	
G630	G630	
	M98 H2	

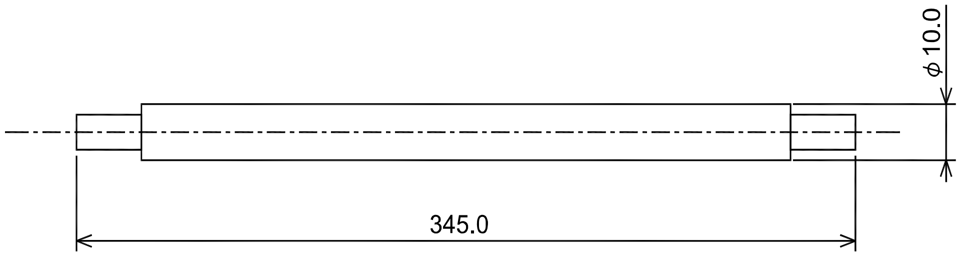
$1	$2	
T100		
G99 M03 S1=2500		
G00 X9.0 Z10.8		
G01 X–1.0 F0.03		
G01 X–3.0 F0.06		
M05		
M07		
G00 X–3.0 Z10.4 T0		
N999	N999	
M02	M02	
M99	M99	
	N100	
	G43	
	G98 M80 S3=4000	
	M78 S0	
	T5100	.... Back center
	G00 Z–1.0	
	G01 Z1.1 F100 T21	
	G04 U0.1	
	Z–1.0 F2000 T0	
	T5200	.... Back drilling 1.4
	G00 Z–1.0	
	G01 Z3.6 F70 T22	
	Z–1.0 F1000	
	G04 U0.05	
	G01 Z3.1	
	Z6.42 F70	
	Z–1.0 F1000 T0	
	T5300	.... Back bottom center
	G00 Z–1.0	
	G01 Z5.5 F100 T23	
	Z6.7 F20	
	G04 U0.05	
	Z–1.0 F1000 T0	
	M60	
	M82	
	M79	
	M99	

$1	$2	
	N1	
	G44	
	G99 M23 S2=5000	
	T3300	.... Back drilling 0.7
	G00 Z–1.0	
	G01 Z5.5 F0.1 T33	
	Z8.2 F0.006	
	G04 U0.02	
	Z–1.0 F0.1	
	G04 U0.1	
	G01 Z10.01 F0.006	
	G04 U0.02	
	G01 Z–1.0 F0.1 T0	
	M99	
	N200	.... Back center
	T5300	
	G43	
	G98 M80 S3=4000	
	M78 S0	
	G00 Z–1.0	
	G01 Z9.3 F80 T23	
	G04 U0.05	
	Z10.2 F20	
	G04 U0.05	
	Z–1.0 F1000 T0	
	M60	
	M82	
	M79	
	M99	
	N2	
	G99 M23 S2=5000	
	T3200	.... Back drilling 0.4
	G00 Z–1.0	
	G01 Z9.0 F0.2 T32	

$1	$2	
	Z10.5 F0.006	
	G04 U0.02	
	Z–1.0 F0.1	
	G04 U0.05	
	G01 Z10.0 F0.1	
	Z11.0 F0.006	
	Z–1.0 F0.1	
	G04 U0.1	
	G01 Z10.5	
	Z11.5 F0.006	
	Z–1.0 F0.1	
	G04 U0.1	
	G01 Z11.0	
	Z12.0 F0.006	
	Z–1.0 F0.1	
	G04 U0.1	
	Z12.02 F0.006	
	G04 U0.1	
	Z–1.0 F0.1 T0	
	S2=3500	
	T3100	.... End face separation
	G50 U–0.5	
	G00 X3.4 Z–0.5	
	G01 X1.3 Z1.3	
	Z –0.5 T0	
	G50 U0.5	
	M25	
	M34	
	M99	

16.1.5 Example 5 (Machining with rechucking long workpiece)

Workpiece drawing

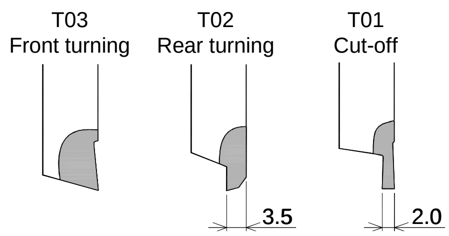


Machine condition settings

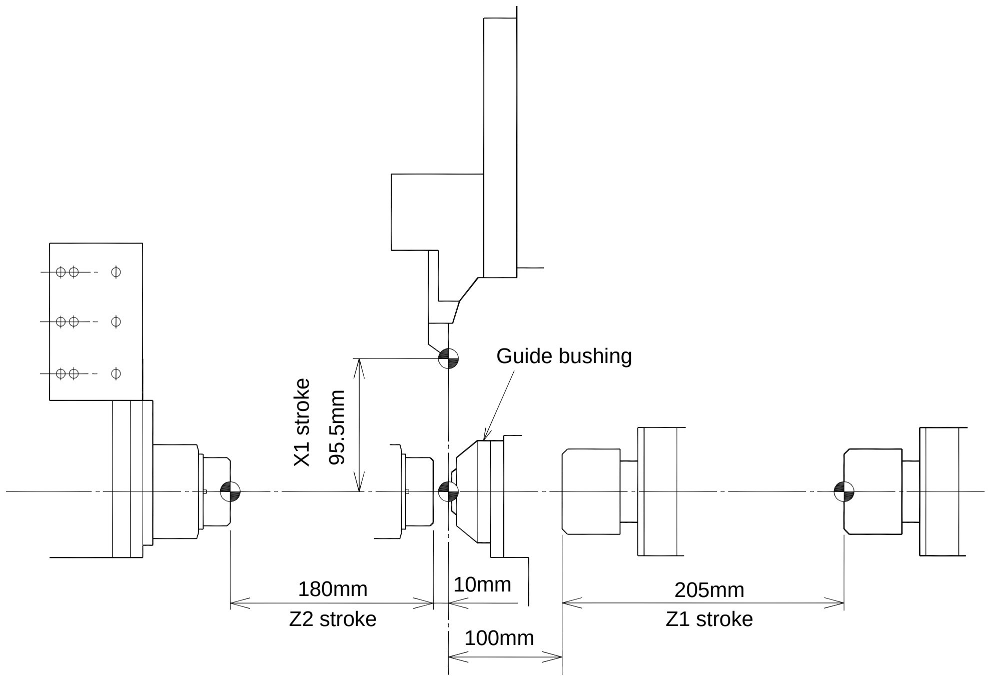
Machining data settings

Material outer diameter:	10 mm
Machining length:	200 mm
Product count per chuck:	1
Tool positioning point:	1 mm
Back spindle chuck position:	30 mm

Cutting tool



Stroke drawing

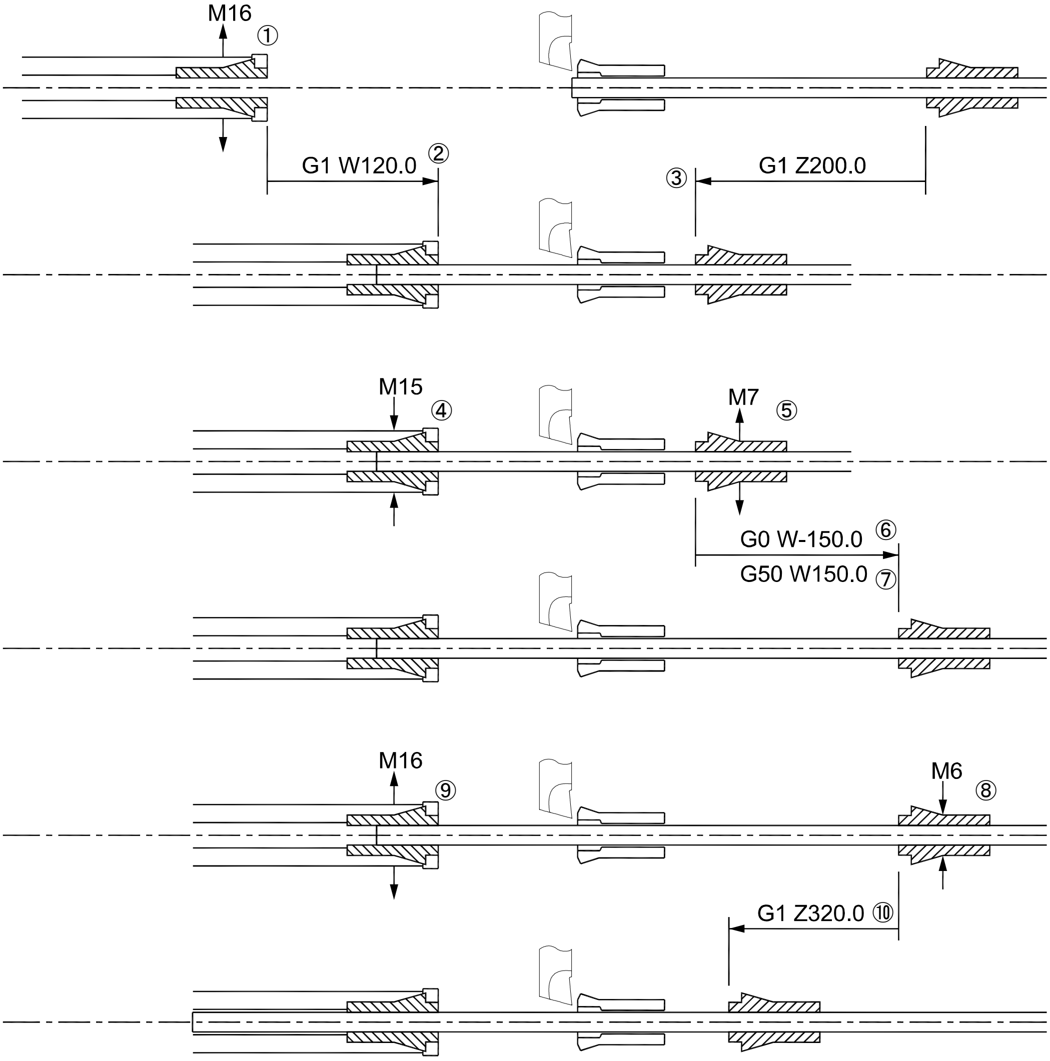


[Sample program]

$1		$2	
(Preparation process)			
G630		G630	
T300	Front turning	M98 H1	Back machining
G0 X11.0 Z-1.0 T3		T3000	
:		M16	(1) Back spindle chuck open
(Machining)		M25	
:			
M5			
:			
!L11		!L11	
G98 G1 Z200.0 F2000 T0	(3)	G98 G1 W120.0 F2000	(2) Specify the move distance according to the position chucked by back spindle chuck
!L12		!L12	
G4 U0.2			
M15	(4) Back spindle chuck close		
G4 U0.5			
M7	(5) Main spindle chuck open		
G4 U0.2			
G0 W-150.0	(6) Spindle move		
G50 W150.0	(7) Coordinate system shift for move distance		
G4 U0.2			
M6	(8) Main spindle chuck close		
G4 U0.5			
M16	(9) Back spindle chuck open		
G4 U0.2			
G98 G1 Z320.0 F2000 T0	(10)		
M3 S1=1000			
M24 S2=1000			
G114.1 H1 D–2			
G650		G650 W0	
M122	Back spindle torque limit L selection (as needed)	G98 G1 Z275.0 F1000	
		M77	
		M15	Back spindle chuck close
		G4 U0.2	

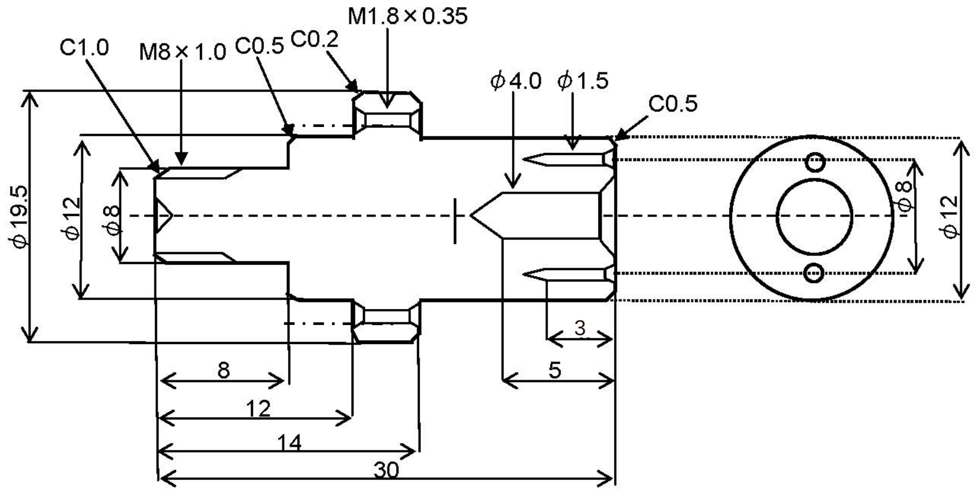
$1	$2
!L13	!L13
G99 M3 S1=2500 .... Spindle rotation for machining	
T200 .... Rear turning	
G50 W-3.5	
G0 X11.0 Z320.0 T2	
:	
(Machining)	
:	
G50 W3.5	
G99 M3 S1=2000 .... Spindle rotation for cut-off	
T200 .... Cut-off, width 2 mm	
G0 X11.0 Z347.0 T1	
!L14	!L14
	M16 .... Back spindle chuck open
	G98 G1 Z315.0 F1000
	G4 U0.2
	M15 .... Back spindle chuck close
	G4 U0.2
!L15	!L15
G99 G1 X–1.0 F0.03	
G600	G600
G113	M25
G1 X–3.0 F0.03	M124 .... Back spindle torque limit OFF
M5	
M7	
M56	
G50 W–150.0 .... Shift cancel	
G0 X–3.0 Z–0.1 T0	
M2	
M99	
%	

Operation diagram

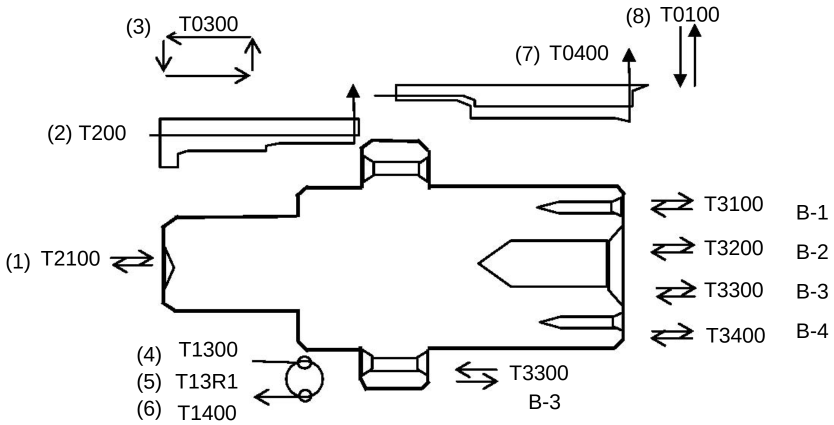


16.1.6 Example 6

Machining drawing

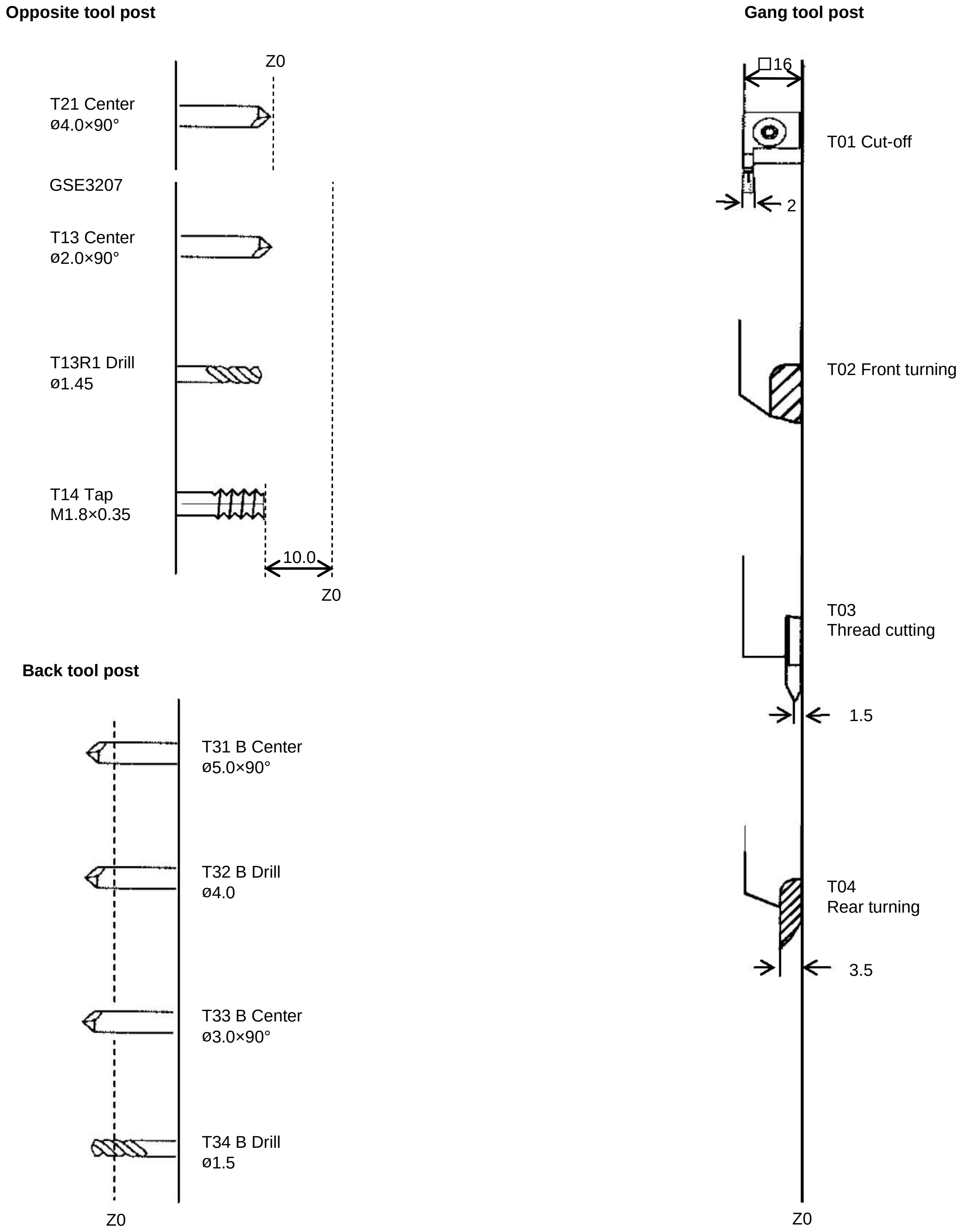


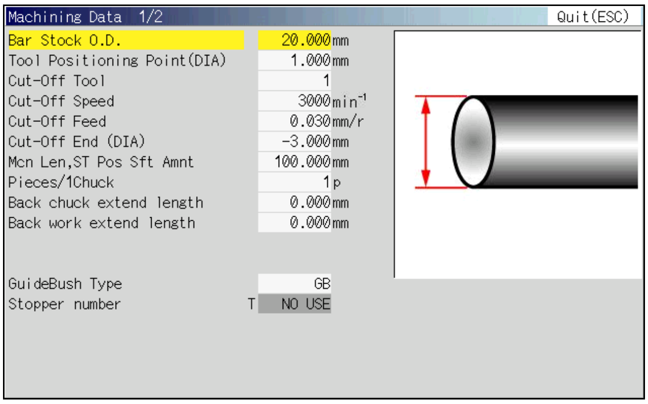
Machining layout



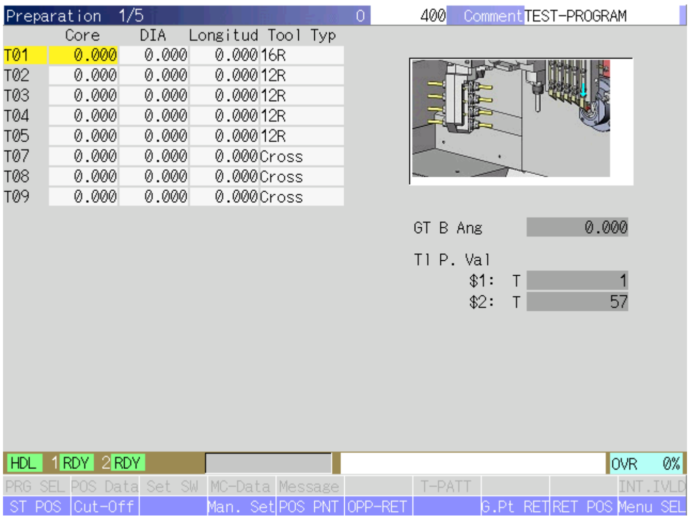
Workpiece diameter: ø20.0

Tool layout

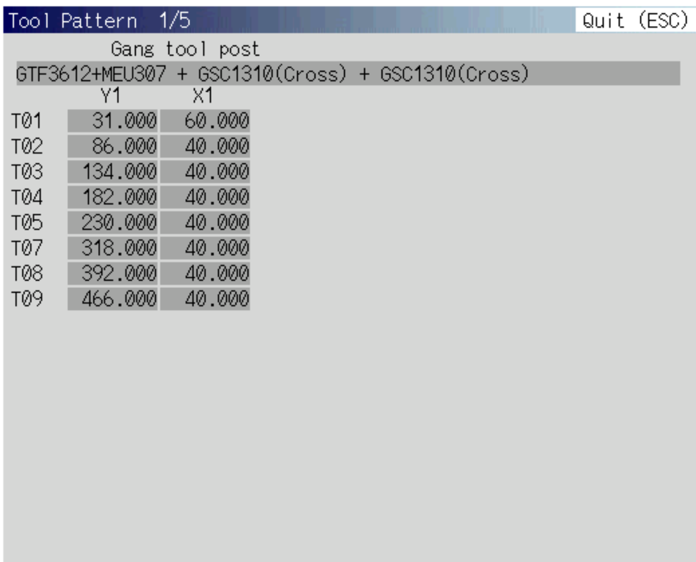

Opposite tool post
Z0
T21 Center
ø4.0×90°
GSE3207
T13 Center
ø2.0×90°
T13R1 Drill
ø1.45
T14 Tap
M1.8×0.35
10.0
Z0
Back tool post
T31 B Center
ø5.0×90°
T32 B Drill
ø4.0
T33 B Center
ø3.0×90°
T34 B Drill
ø1.5
Z0
Gang tool post
□16
T01 Cut-off
2
T02 Front turning
T03
Thread cutting
1.5
T04
Rear turning
3.5
Z0



Machining data



Tool set



Tool layout pattern

[Sample program]

O412

$1	$2
G50 Z13.9	
M9	
M6	
G113	
G0 X21.0 Z-1.0	
M51	
G99 M3 S1=1500	
G610	G610
T2100(CENTER-4.0)	
G0 X0.0 Z-1.0 T12	
G1 Z2.0 F0.05	
G0 Z-1.0T0	
G630	G630
G99 M3 S1=3000	/M98 H111
T0200(FRONT-T)	M16
G0 X21.0 Z-1.0 T2	M24 S2=1500
G1 X14.01 F1.0	
Z12.0 F0.01	
X19.0 F0.01	
X21.0 W1.0 F0.01	
G0 Z-1.0	
G1 X-0.5 F1.0	
G1 Z0 F0.01	
X6.0 F0.01	
X8.0 W1.0 F0.01	
Z8.0 F0.01	
X11.0 F0.01	
X12.0 W0.5 F0.01	
Z12.0 F0.01	
X21.0 F0.01	
G0 Z-1.0 T0	

$1	$2
M3 S1=1500 T300(THREAD-M8-1.0) G50 W-1.5 G0 X9.0 Z-2.0 T3 G92 X7.56 Z5.0 F1.0 X7.26 X7.04 X6.86 X6.76 X6.7 G0 X21.0 Z-1.0 T0 G50 W1.5 M5 G98 M80 S3=4800 M18 C0 T1300(CENTER-2.0) G50 W-10.0 G0 X15.5 Z-1.0T13 G0 Z11.0 G1 Z13.0 F150 G0 Z11.0 M18 C180.0 G4 U0.5 G1 Z13.0 F150 G0 Z-1.0 T0 G50 W10.0 G0 Z-1.0	

$1	$2
T1300 R1(DRILL-1.45) G50 W-10.0 G0 X15.5 Z-1.0 T23 G0 Z11.0 G1 Z13.5 F220 G0 Z11.0 G4 U0.3 G0 Z13.0 G1 Z15.0 F220 G0 Z11.0 G4 U0.3 G0 Z14.5 G1 Z16.5 F220 G0 Z11.0 G4 U0.3 G0 Z16.0 G1 Z18.0 F220 G0 Z11.0 M18 C0 G4 U0.5 G0 Z11.0 G1 Z13.5 F220 G0 Z11.0 G4 U0.3 G0 Z13.0 G1 Z15.0 F220 G0 Z11.0 G4 U0.3 G0 Z14.5 G1 Z16.5 F220 G0 Z11.0 G4 U0.3 G0 Z16.0 G1 Z18.0 F220 G0 Z-1.0 G50 W10.0 G0 Z-1.0	

$1	$2
G98 M80 S3=0	
T1400(TAP-M1.8-0.35)	
G50 W-10.0	
G0 X15.5 Z-1.0 T14	
G0 Z11.0	
G84 Z16.0 F0.35 D3 S500, R1	
G80	
G0 Z11.0	
M18 C180.0	
G4 U0.5	
G84 Z16.0 F0.35 D3 S500, R1	
G80	
G0 Z-1.0 T0	
G50 W10.0	
G0 Z-1.0	
M82	
M20	
G99 M3 S1=2500	
T0400(REAR-T)	
G50 W-3.5	
G0 X21.0 Z11.0 T4	
G0 X19.5	
G1 Z13.75 F0.03	
X18.8 Z19.0 W0.35 F0.03	
X16.0 F0.03	
Z31.0 F0.04	
X19.0 F0.05	
X21.0 W1.0 F0.03	
G0 Z14.0	
G1 X12.0 F0.03	
Z29.5 F0.04	
X10.0 W1.0 F0.03	
G0 X21.0 T0	
G50 W3.5	
M24 S2=1500	
M3 S1=1500	
G114.1 H1 D-2 R0	

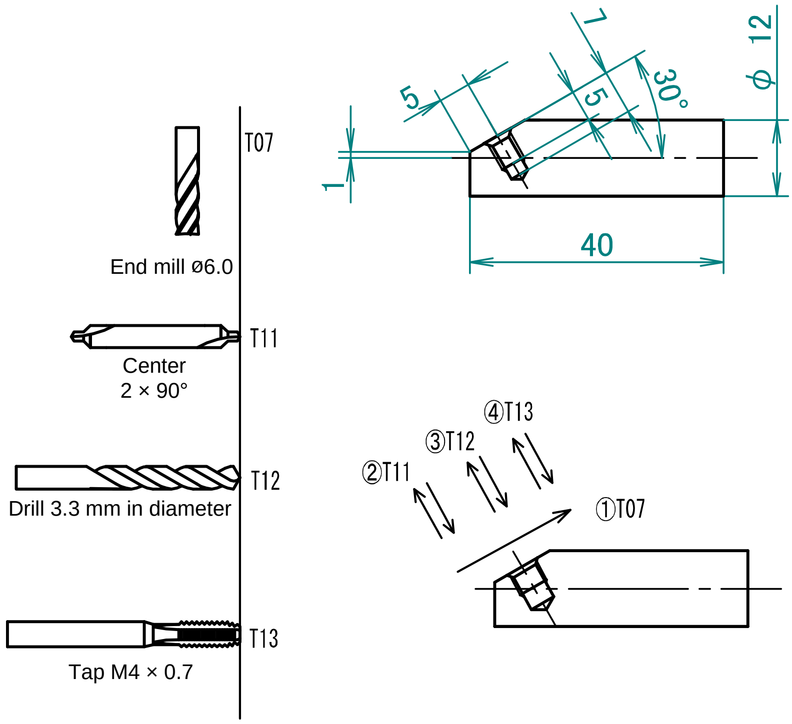
$1	$2
N9 T100(CUT-OFF)	
G0 X21.0 Z46.0 T1	
M1	
G650	G650
	G0 Z-1.0 M72
	G98 G1Z11.0 F2000
	M77
	G4 U0.3
	M15
	M73
	G4 U0.2
!L1	!L1
G1 X-1.0 F0.03	
G600	G600
G1 X-3.0 F0.05	M25
G113	
M5	
M7	
M56	
G0 X-3.0 Z13.9 T00	
G999	G999
G630	G630
	M98 H111
N999	N999
M2	M2
M99	M99
	N111
	M25
	G98 M180 S5=3800
	M48 C0
	T3100(B.CENTER-5.0)
	G0 X0.0 Z-1.0 T31
	G1 Z2.7 F150
	G4 U0.1
	G0 Z-1.0 T0

$1	$2
	G98 M180 S5=2700
	T3200(B.DRILL-4.0)
	G0 X0.0 Z-1.0 T32
	G1 Z5.2 F200
	G0 Z-1.0 T0
	G98 M180 S5=3800
	T3300(B.CENTER-3.0)
	G0 X-15.5 Z-1.0 T33
	G0 Z15.0
	G1 Z16.2 F150
	G4 U0.1
	G0 Z15.0
	M48 C180.0
	G4 U0.5
	G0 Z15.0
	G1 Z16.2 F150
	G4 U0.1
	G0 Z-1.0
	G0 X8.0 Z-1.0
	G1 Z1.0 F150
	G4 U0.1
	G0 Z-1.0
	M48 C0
	G4 U0.5
	G1 Z1.0 F150
	G4 U0.1
	G0 Z-1.0 T0

$1	$2
	T3400(B.DRILL-1.5)
	G0 X8.0 Z-1.0 T34
	G1 Z1.8 F250
	G0 Z-1.0
	G0 Z1.3
	G1 Z3.6 F250
	G0 Z-1.0
	M48 C180.0
	G4 U0.5
	G1 Z1.8 F250
	G0 Z-1.0
	G0 Z1.3
	G1 Z3.6 F250
	G0 Z-1.0 T0
	M182
	M79
	M34
	M99

16.1.7 Example 7

Machining layout



[Sample program]

O1227

$1	$2
G50 Z0	
G113	
M9	
M6	
G0 X29.0 Z-1.0	
G635	G635
M18 C0	G637
G98 M80 S3=1000	M25
	T2000
T0700 (ENDMILL)	M98 H11 (UKEKAGO)
G50 W-10.0	
G0 X13.0 Y4.88 Z-3.5 T7	
G1 X10.0 F1000	
G1 Y9.5 Z8.22 F100	
G1 X8.0 F1000	
G1 Y4.88 Z-3.5 F100	
G1 X4.0 F1000	
G1 Y9.5 Z8.22 F100	
G1 X0 F1000	
G1 Y4.88 Z-3.5 F100	
G1 X-4.0 F1000	
G1 Y9.5 Z8.22 F100	
G1 X-8.0 F1000	
G1 Y4.88 Z-3.5 F100	
G1 X-10.0 F1000	
G1 Y9.5 Z8.22 F100	
G1 X-12.0 F1000	
G1 Y4.88 Z-3.5 F100	
G1 X13.0 F1000	
G50 W10.0	
M80 S3=1000	
G680	G680
T1100 (CENTER)	
G0 X13.0 Y0 Z-2.0 T11	
G900 X2.0 Z-2.0 B60.0	
G950 X2.0 Z0	
G0 X5.0 Z-1.0	
G1 Z3.5 F50	
G4 U0.05	
G1 Z-3.0 F1000	
G951	
G901 X13.0 Z-3.0 B60.0	

$1	$2
T1200(DRILL)	
G0 X13.0 Y0 Z-2.0 T12	
G900 X2.0 Z-2.0 B60.0	
G950 X18.0 Z0	
G0 X5.0 Z-1.0	
G1 Z8.0 F50	
G1 Z-3.0 F1000	
G951	
G901 X13.0 Z-3.0 B60.0	
M80 S3=1000	
T1300(TAP)	
G0 X13.0 Y0 Z-2.0 T13	
G900 X2.0 Z-2.0 B60.0	
G950 X2.0 Z0	
G0 X5.0 Z-2.0	
G84 Z6.4 F0.7 D3 S800,R1	
G80	
G1 Z-3.0 F1000	
G951	
G901 X13.0 Z-3.0 B90.0	
M82	
!2!3L2	!1!3L2
G99 M3S1=1000 M24S2=1000	
G114.1 H1 D-2	
T0100	
G0 X13.0 Z42.2 T1	
G650	G650
!2!3L4	!1!3L4
G99 G1 X-1.0 F0.05	
G600	G600
G1 X-3.0 F0.05	
G113	
M8	
M8	
/M155	
M9	
M5	
M25	

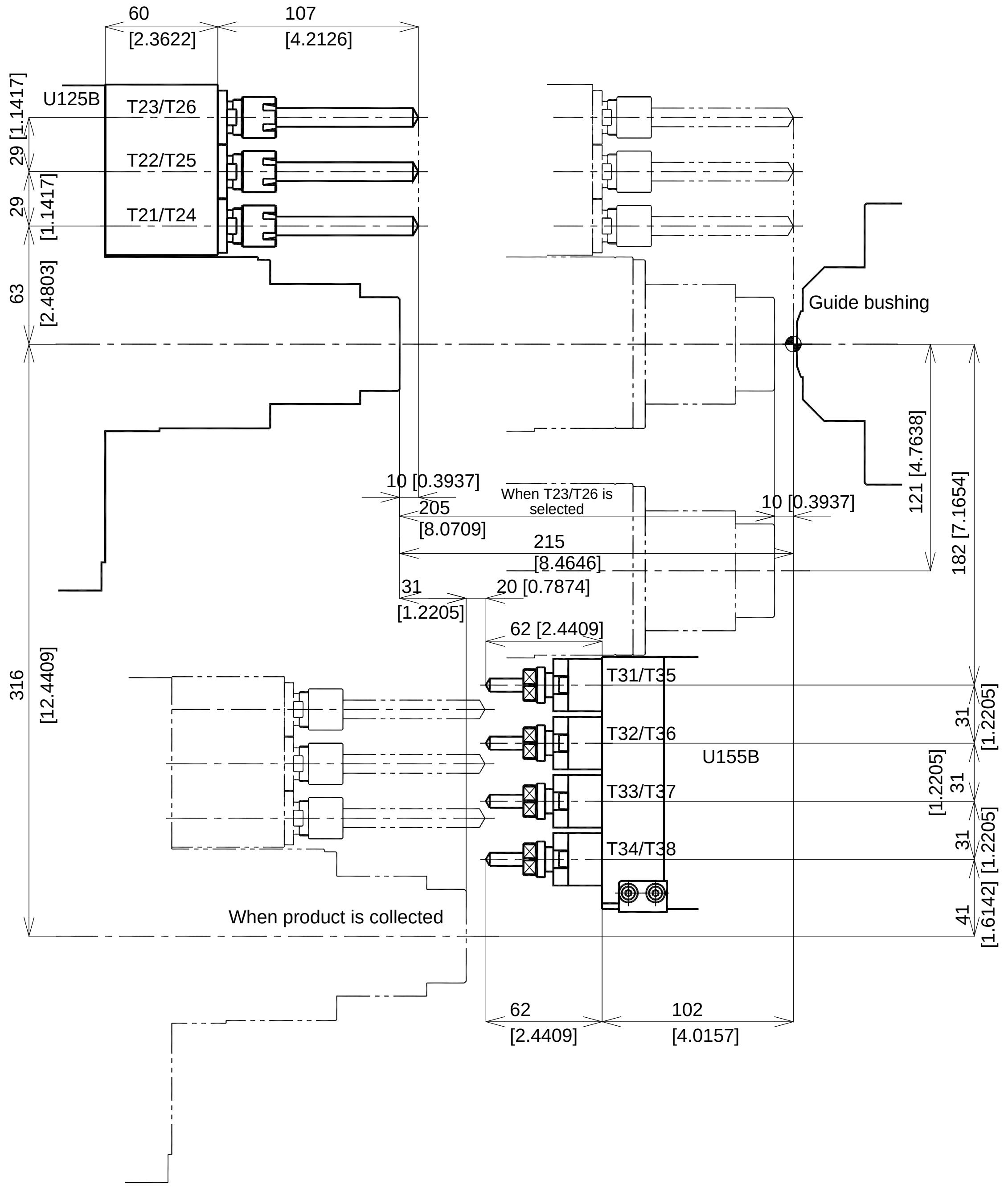
$1	$2
M82	
M7	
G4 U0.3	
M56	
G0 X-3.0 Z0 T0	
G999	G999
G635	G635
	G637
	M25
	T2000
	M98 H11 (UKEKAGO)
G600	G600
N999	N999
M2	M2
M99	M99
	N11
	M32 Z20.0
	M16
	G4 U1.0
	M10
	M72
	G4 U0.5
	M11
	M33 T2500 H6000
	M89
	M31
	M73
	T2500
	M99
%	%

Product Code	C-L220E XII
Document Code	2E1-1601
Mfg. No.	L220E/0001 ~
Issue Date	2014.1

17. Tooling

17.1 Structure of Tooling System

L20 Standard tool layout (with U125B, U155B)



The standard tool layout and stroke of opposite tool post device of L20 is shown above. It is not always necessary to follow the above layout. Machining shall be done by optimum layout for the workpiece.

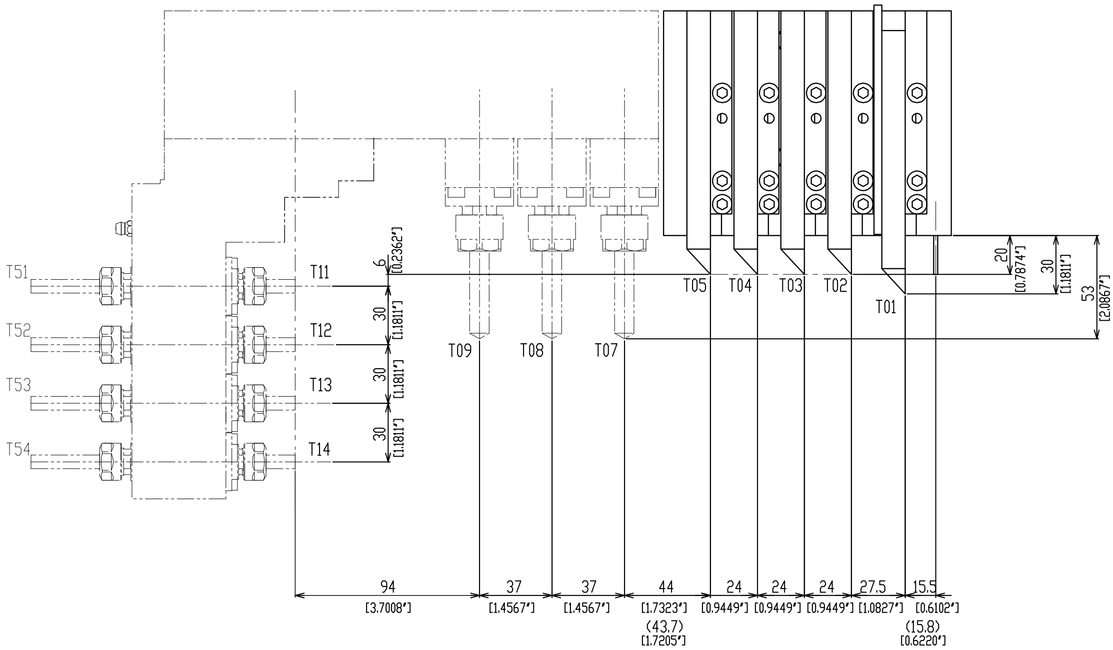
17.2 Details of Tooling System

Tool Holder

This tool holder is provided for outer diameter cutting.

GTF3612 5-Tool Vertical Holder (5-tool, □12)

GTF3613 5-Tool Vertical Holder (5-tool, □13)



* ( ) indicates GTF3613 when □1/2" tools are used
* This figure shows type XII.

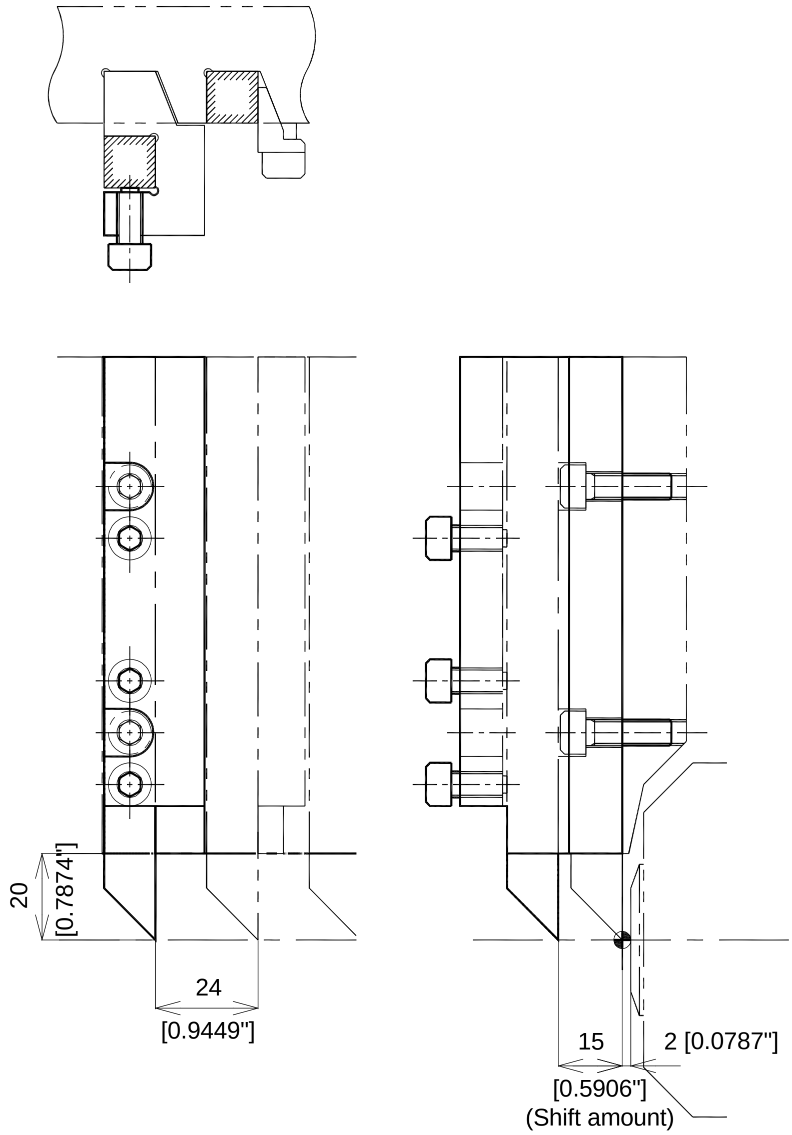
[Note]

- Cut-off tool shall be mounted on the right-most position (T01).
- Use the provided spacer to mount a tool of □12 (or □13) at the position T01.

Specification	Description	
Tool holder name	GTF3612	GTF3613
Machine type	VIII, IX, X, XII	VIII, IX, X, XII
Usage	All types of tools	All types of tools
Tool size	12×12×120 mm (T01 to T05), 16×16×120 mm (T01)	13×13×120 mm (1/2"×1/2"×4"-3/4") (T01 to T05), 16×16×120 mm (T01)

This tool holder is provided for the turning tool. The tool can be mounted with 15 mm [0.59"] longitudinal shift from standard fitting point. This tool holder can be mounted at the turning tool station positions except T01 with the vertical holder-GTF3612.

GTF3312 Shift Tool Holder (Adapter type)



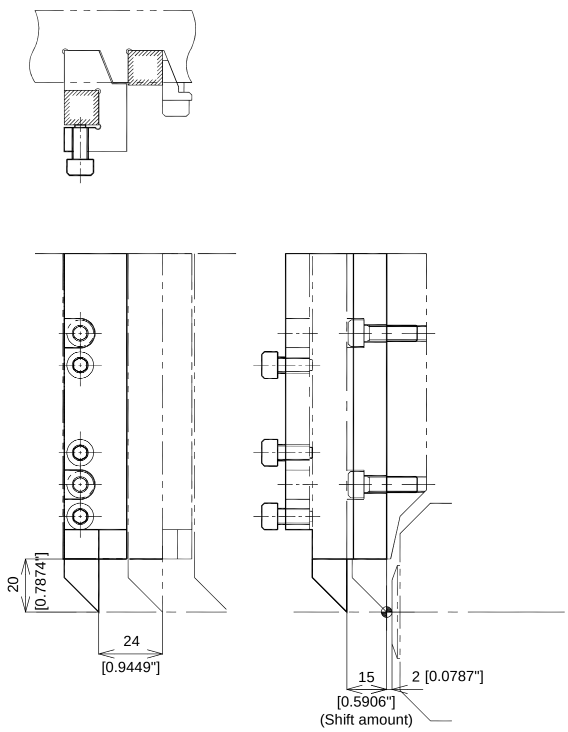
[Note]

If this holder is mounted, be careful for the holder not to interfere with the back spindle or the sleeve on the opposite tool post.

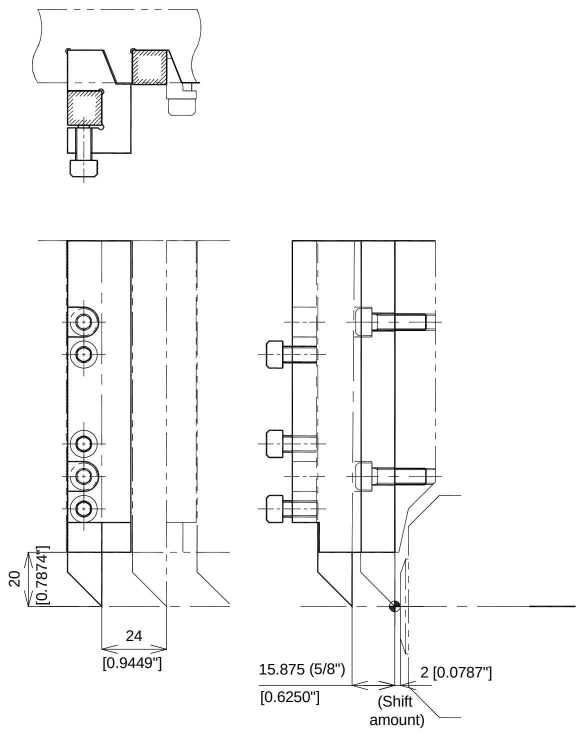
Specification	Description
Tool holder name	GTF3312
Machine type	VIII, IX, X, XII
Usage	Threading and others
Tool size	12×12×120 mm (Shifted by 15 mm [0.59"])
Provided set screws	Two M6×20 screws

This tool holder is provided for the turning tool. The tool can be mounted with 15 mm [0.59"] (or 15.875 mm [5/8"]) longitudinal shift from standard fitting point. This tool holder can be mounted at the turning tool station positions except T01 with the vertical holder-GTF3613.

GTF3313　Shift Tool Holder (Adapter type)



GTF3313L　Shift Tool Holder (Adapter type)

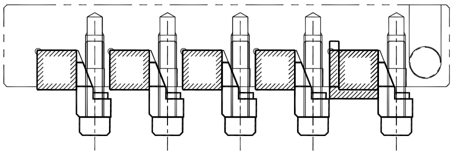


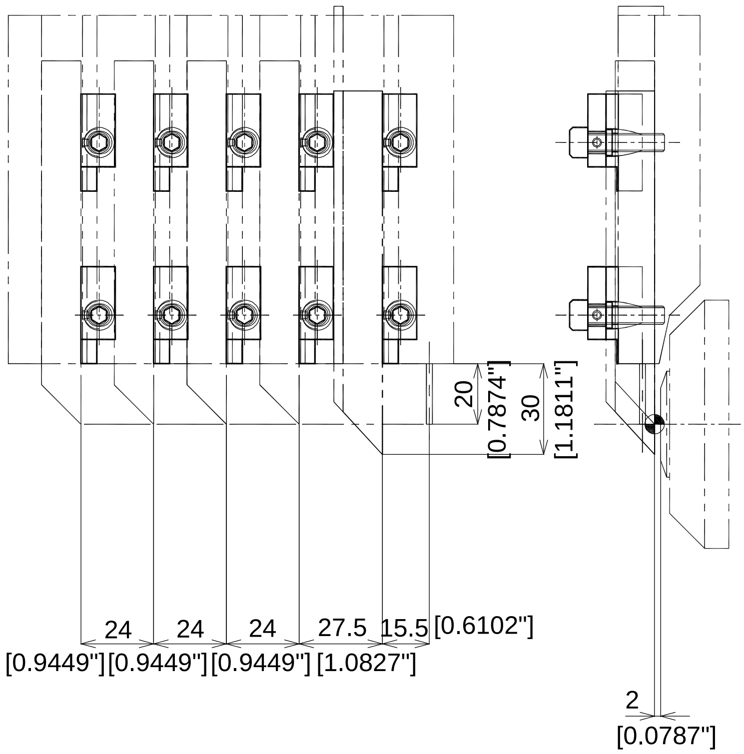
[Note]

If this holder is mounted, be careful for the holder not to interfere with the back spindle or the sleeve on the opposite tool post.

Specification	Description	
Tool holder name	GTF3313	GTF3313L
Machine type	VIII, IX, X, XII	VIII, IX, X, XII
Usage	Threading and others	Threading and others
Tool size	13×13×120 mm (Shifted by 15 mm [0.59"])	1/2"×1/2"×4"-3/4" (Shifted by 5/8")
Provided set screws	Two M6×20 screws	Two M6×20 screws

TJU412　CS Quick Wedge





[Note]

This quick wedge is used for holding the tool.

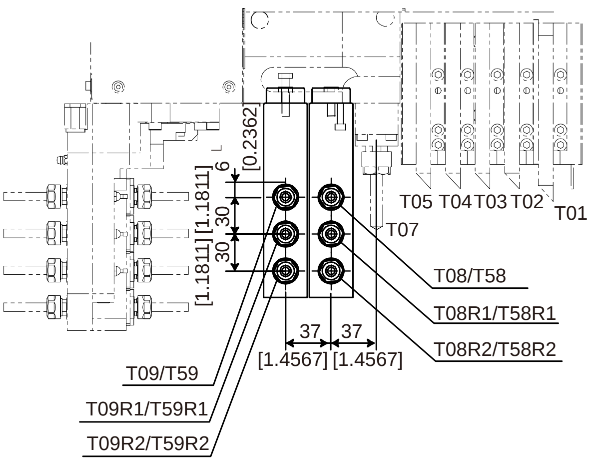
This can be used by replacing the wedge.

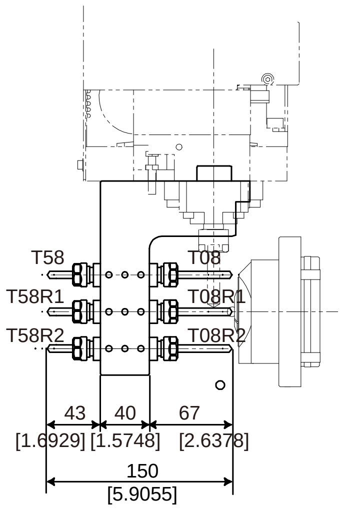
Specification	Description
Tool holder name	TJU412
Machine type	VIII, IX, X, XII
Tool holder	GTF3612, GTF3613
Tool size	12×12×120 mm (T01 to T05), 13×13×120 mm (T01 to T05), 16×16×120 mm (T01)

Sleeve

This sleeve holder is to be mounted on rotary tool driving device (U33B), and is used for end-face drilling. Up to three arbitrary sleeves can be mounted.

GDF1207　3-tool Vertical Sleeve Holder





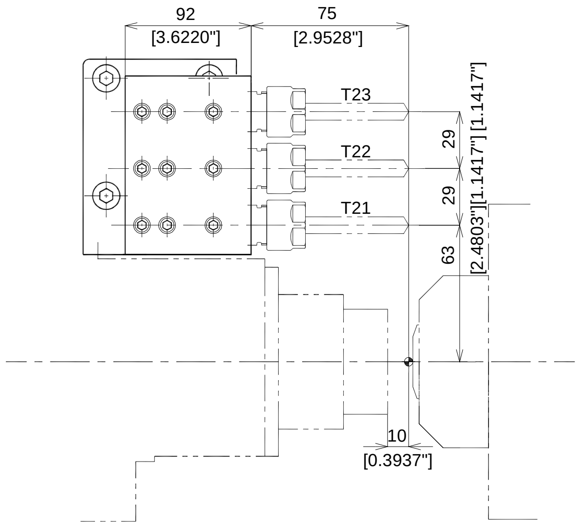
* This figure shows type XII.

[Note]

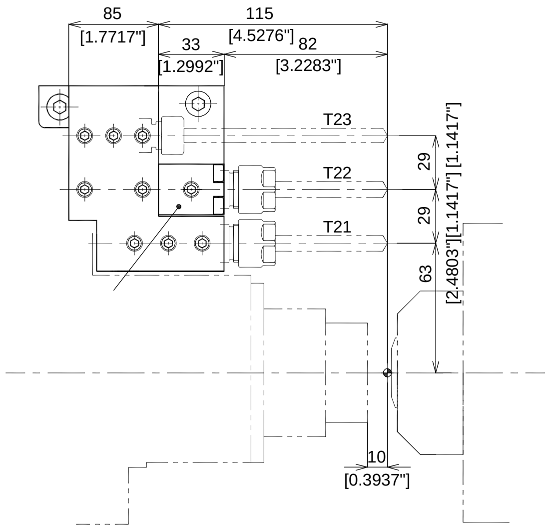
- This holder can be mounted only on T08~T10.
- When performing milling by the rotary tool mounted next to this holder, pay attention to the interference between the workpiece and this holder.
- When this holder is mounted, the back spindle and the opposite sleeves may interfere with this holder. Care must be taken.
- If GSE3210 is mounted in end-face direction, the back machining is unavailable.
- If GDF1207 is mounted on type X, XII the pinch milling and pinch turning cannot be performed.

Specification	Description
Sleeve name	GDF1207
Machine type	VIII, IX, X, XII
Available sleeve	GDS110, BDS507, BDS607, VDS506, HDS5406, HDS5506, HDS5706, HDS5806, BNS407, BNS507, VNS406, VNS506
Sleeve mounting hole diameter	Ø19.05 mm (Ø3/4")
Provided set screws	Hexagon socket head bolts two M5×25 screws, Hexagon head bolt one M6×20

U123B Front 3-tool Tool Holder



U124B Front 3-tool Tool Holder (For Deep Hole Machining)

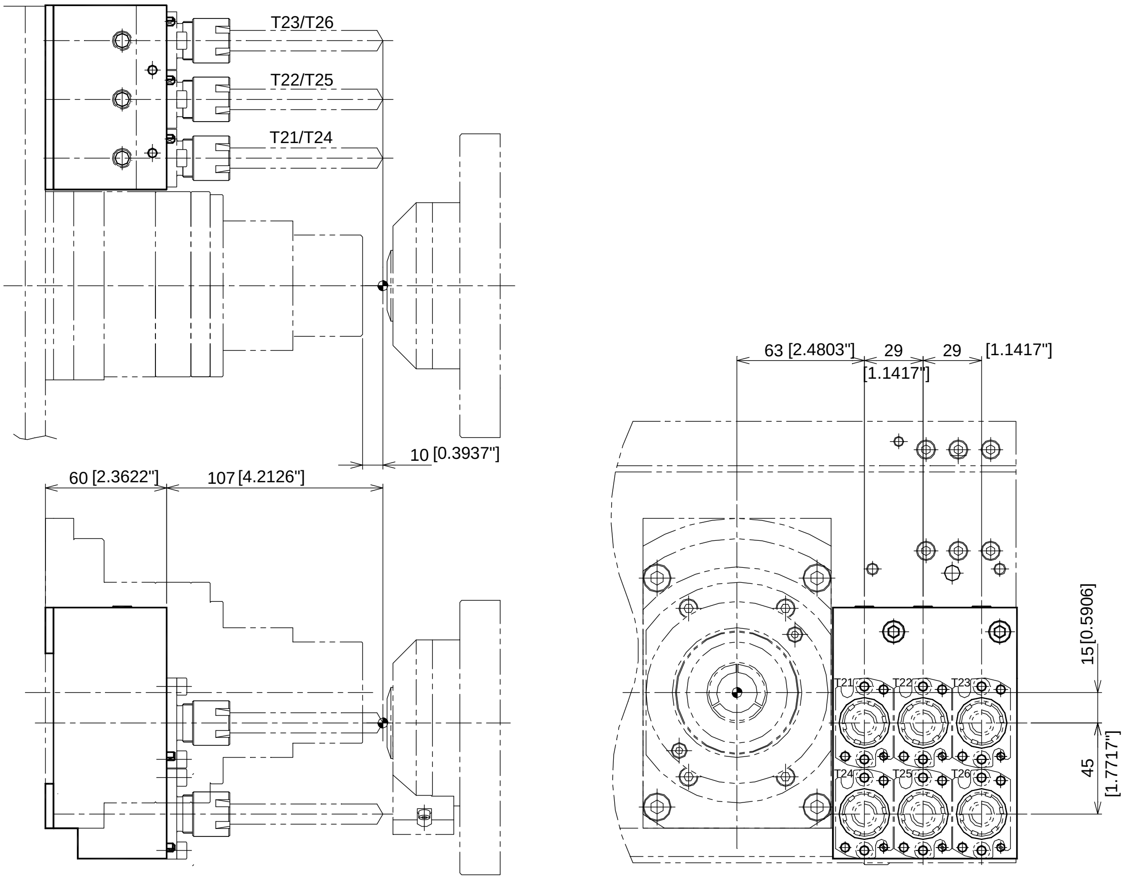


[Note]

Mount an adapter when using T22 or T23 with shorter sleeve length.
Temporarily tighten the adapter after inserting the sleeve, then fix the adapter so that it is seated along with the sleeve core. Fixing only the adapter to the 3-spindle holder may cause deviation of the sleeve core (U124B only).

Specification	Description	
Tool holder name	U123B	U124B
Machine type	VIII, IX	VIII, IX
Available sleeve	GDS110, BDS507, BDS607, VDS506, LDS107, BDS610, VDS110, LDS110, VNS406, BNS407	GDS110, BDS507, BDS607, VDS506, LDS107, BDS610, VDS110, LDS110, VNS406, BNS407
Sleeve mounting hole diameter	ø19.05 mm (ø3/4")	ø19.05 mm (ø3/4")
Provided set screws	–	Four M5×40 screws

U125B Front 6-tool Tool Holder

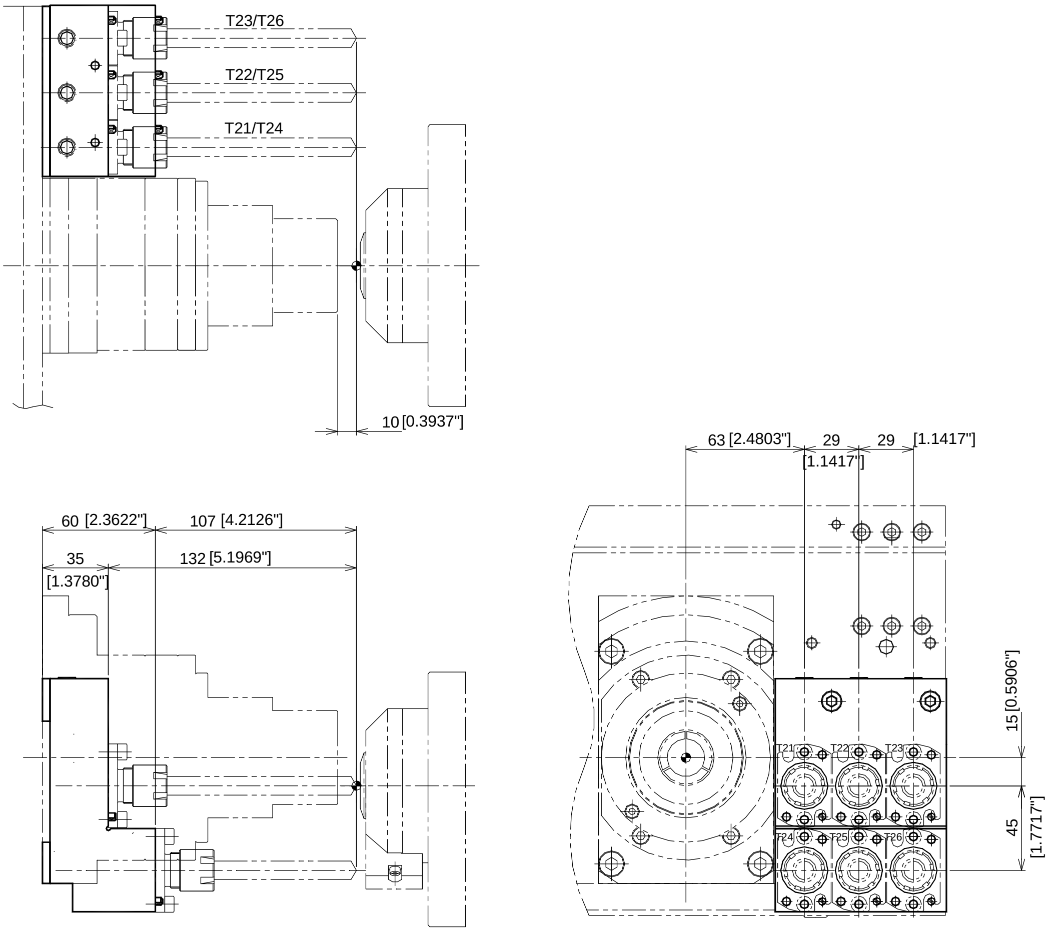


[Note]

Caps may not be placed in the same row depending on the cap diameter of the sleeve. To place the caps in the same row, use GDS110.

Specification	Description
Tool holder name	U125B
Machine type	X, XII
Available sleeve	GDS107, GDS210, SAU919, SAU1019, GDS110
Sleeve mounting hole diameter	ø25.0 mm [ø0.9843]
Provided set screws	–

U126B Front 6-tool Tool Holder (For Deep Hole Machining)



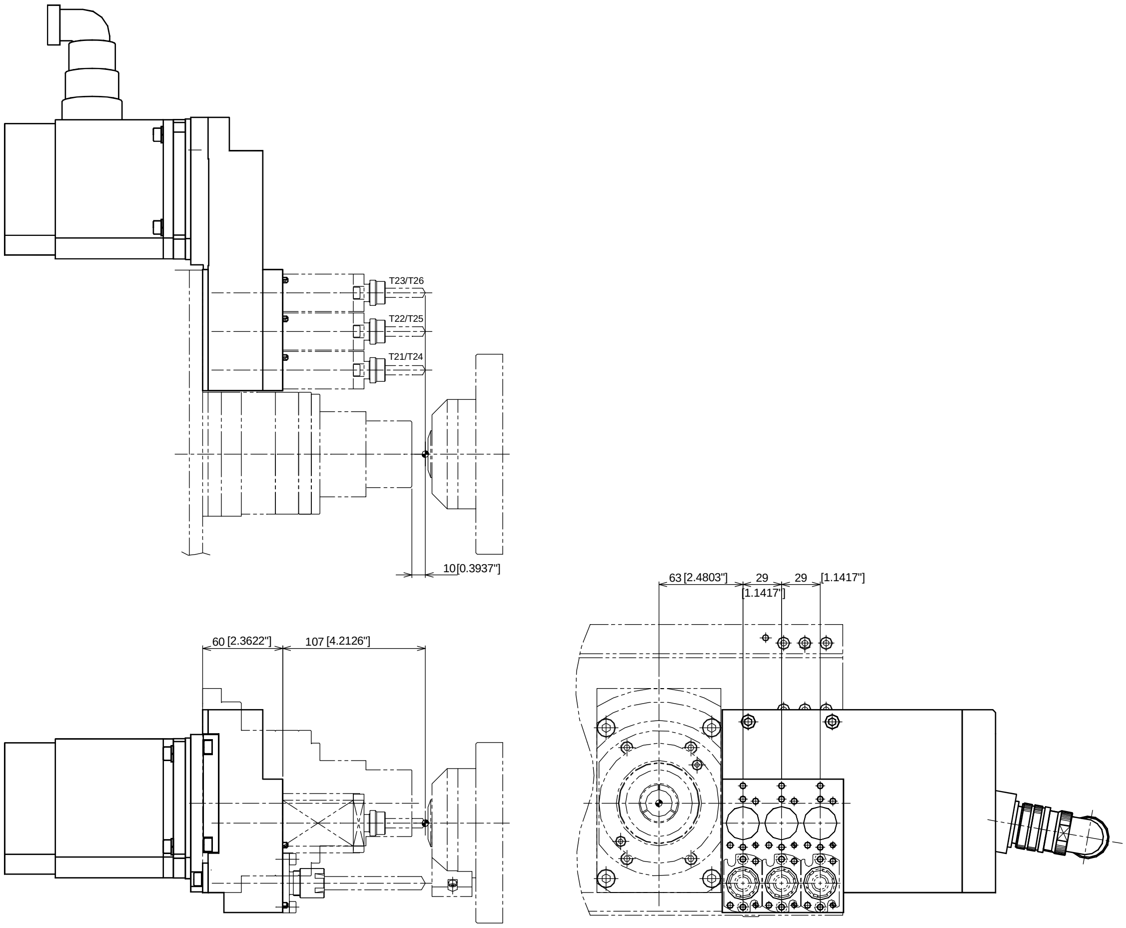
[Note]

Mount an adapter SAU919 when using T21 to T23 with shorter sleeve length.

Specification	Description
Tool holder name	U126B
Machine type	X, XII
Available sleeve	GDS107, GDS210, SAU919
Sleeve mounting hole diameter	Ø25.0 mm [Ø0.9843]
Provided set screws	–

This device performs drilling or end milling to the end face of workpiece at front spindle side with up to three GSE3407's or MSC507's mounted on upper stage. Three fixed tools can be mounted on lower stage.

U128B Front Rotary Tool Drive Device



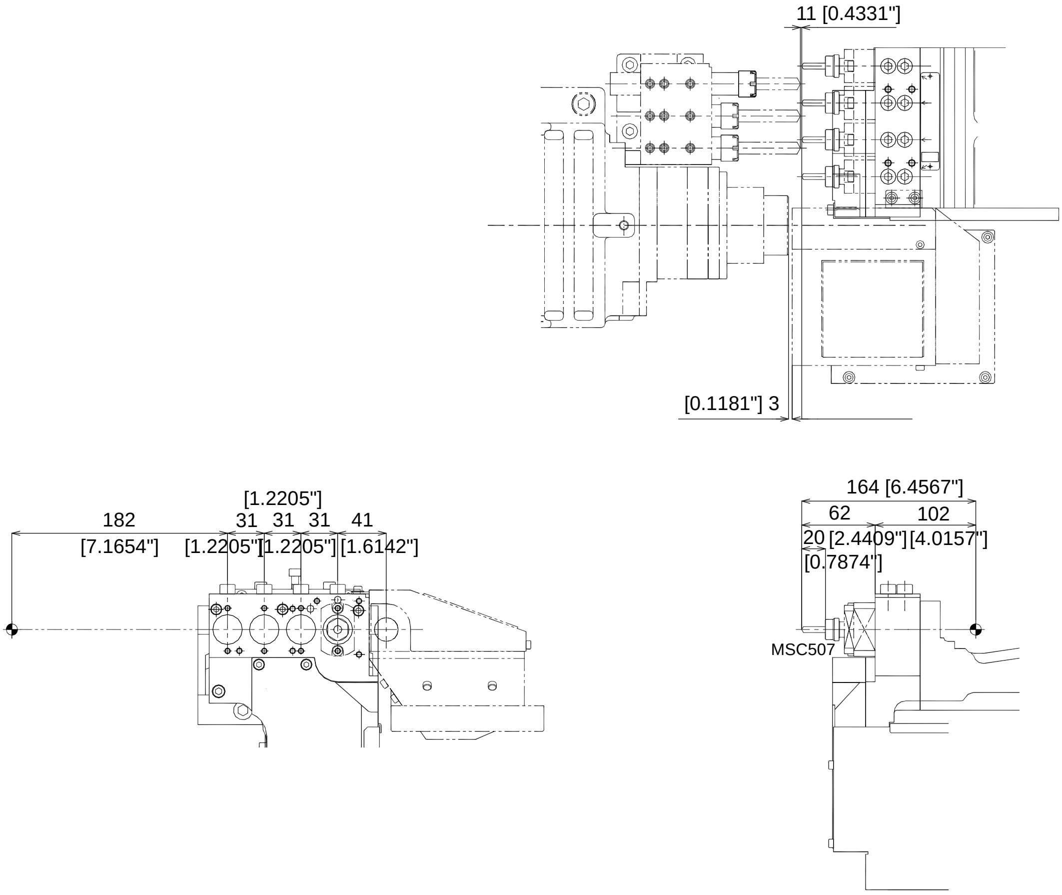
[Note]

When GSC1407 is mounted, see "GSC1407" for detailed information.

Specification	Description
Tool holder name	L220E U128B
Machine type	X, XII
End face milling spindle	GSE3407, MSC507
Cross milling spindle	GSC1407
Sleeve holder (for stationary tool)	SAU919, SAU1019, GDS107, GDS210
Maximum speed / Rated speed	7,500min^{-1}/6,000min^{-1}

This device performs drilling or end milling to the end face of workpiece at back spindle side with up to four MSC507's.

U153B　Back Rotary Tool Drive Device



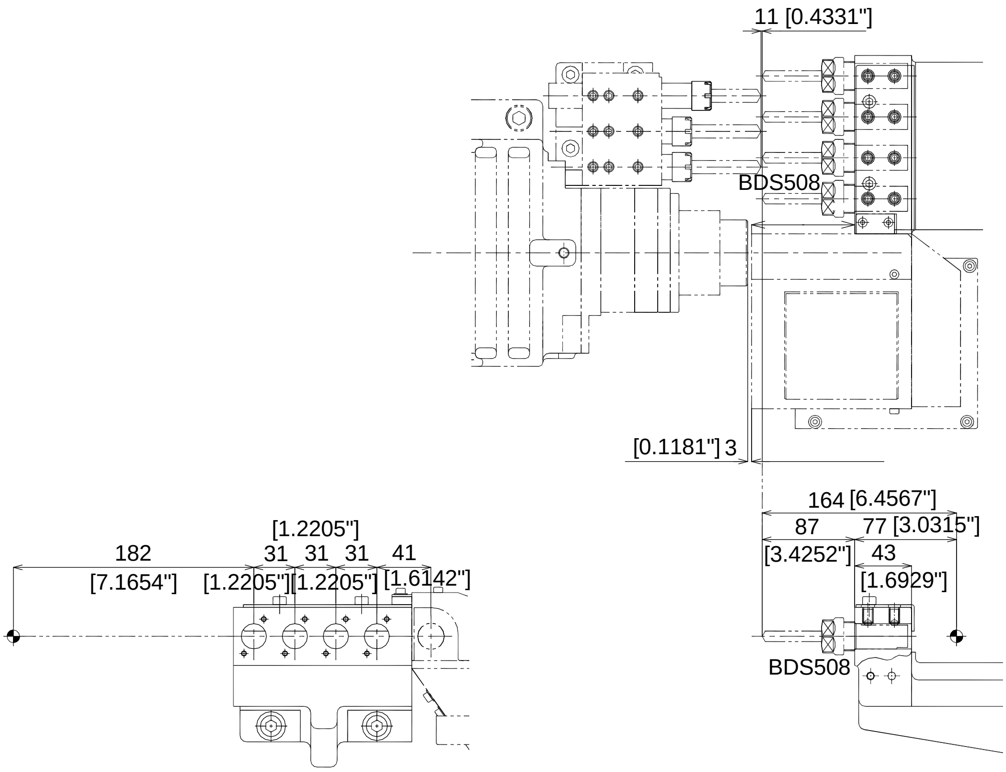
[Note]

When GSS1530 is mounted, see "GSS1530" for detailed information.

Specification	Description
Tool holder name	U153B
Machine type	VIII, IX
End face milling spindle	MSC507
Slitting cutter	GSS1530
Sleeve holder (for stationary tool)	SAU819
Maximum speed / Rated speed	7,500min^{-1}/6,000min^{-1}

Up to four sleeves can be mounted. This holder is used for back drilling.

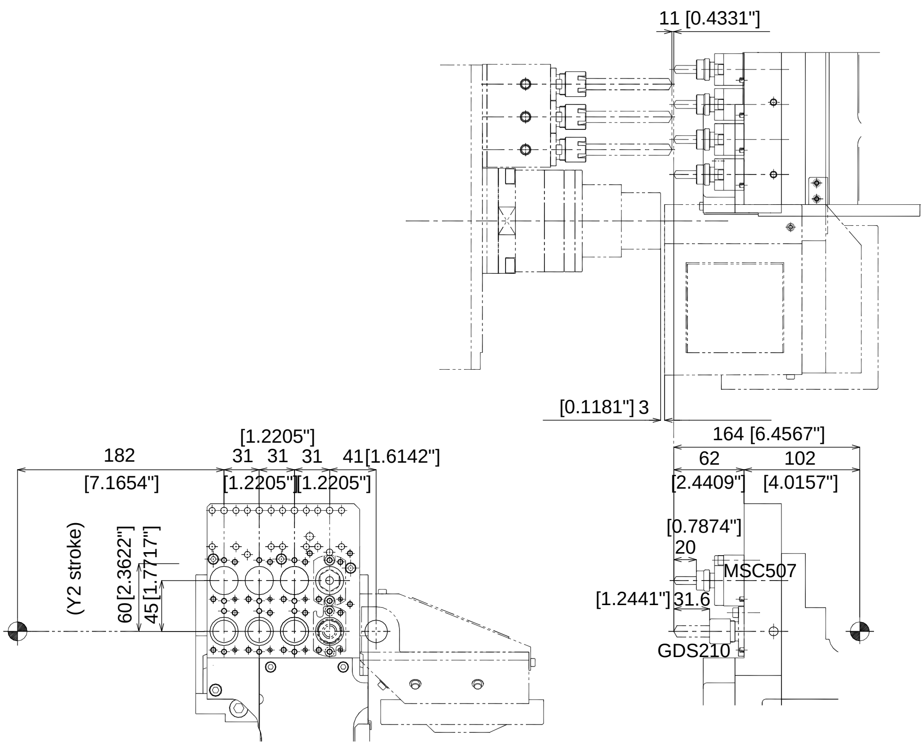
U154B Back 4-Spindle Holder (ø19.05)



Specification	Description
Tool holder name	U154B
Machine type	VIII, IX
Available sleeve	BDS508
Sleeve mounting hole diameter	ø19.05 mm (ø3/4")

This device performs drilling or end milling to the end face of workpiece at back spindle side with up to four MSC507's mounted on upper stage. Four fixed tools can be mounted on lower stage.

U155B Back Rotary Tool Drive Device



[Note]

- When GSS1530 is mounted, see "GSS1530" for detailed information.
- When GSC1507 is mounted, see "GSC1507" for detailed information.

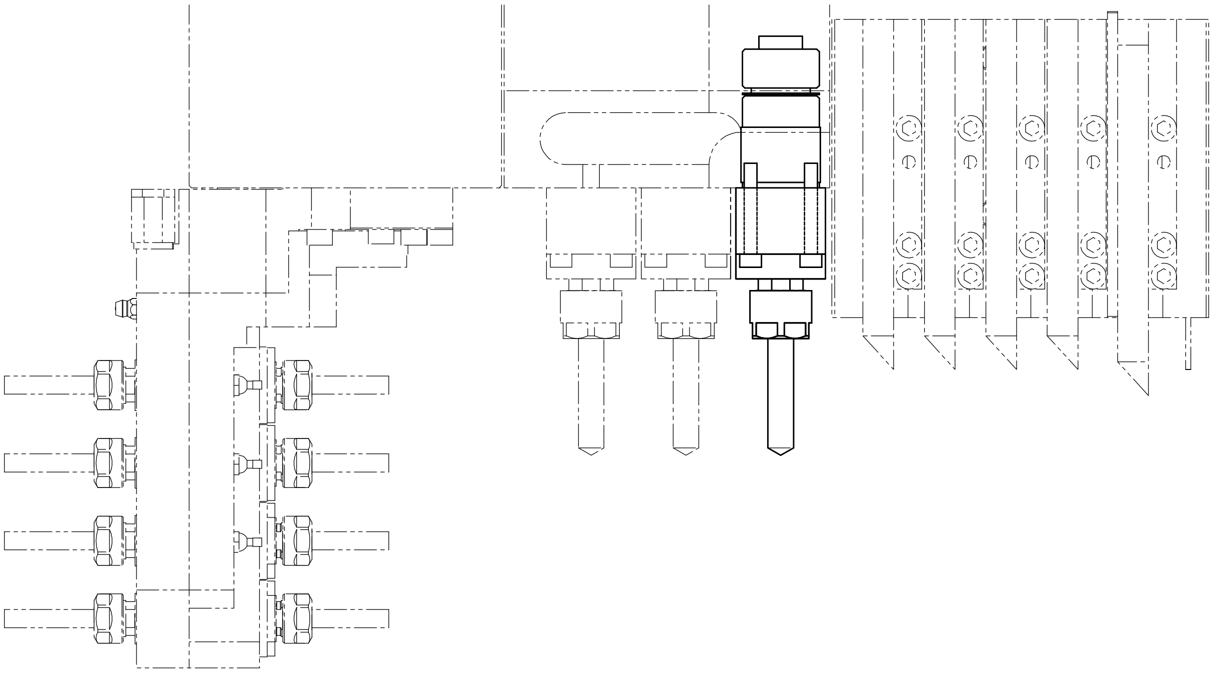
Specification	Description
Tool holder name	U155B
Machine type	X, XII
End face milling spindle	MSC507
Cross milling spindle	GSC1507
Slitting cutter	GSS1530
Sleeve holder (for stationary tool)	SAU919, GDS210
Maximum speed / Rated speed	7,500min^{-1}/6,000min^{-1}

Rotary Tools

Outer Circumference Milling Spindle

This spindle is used to perform drilling on the outer diameter of the workpiece, or to perform key grooving by the end mill, after stopping the spindle.

GSC1310　Outer Circumference Milling Spindle (Rego Type Chuck)

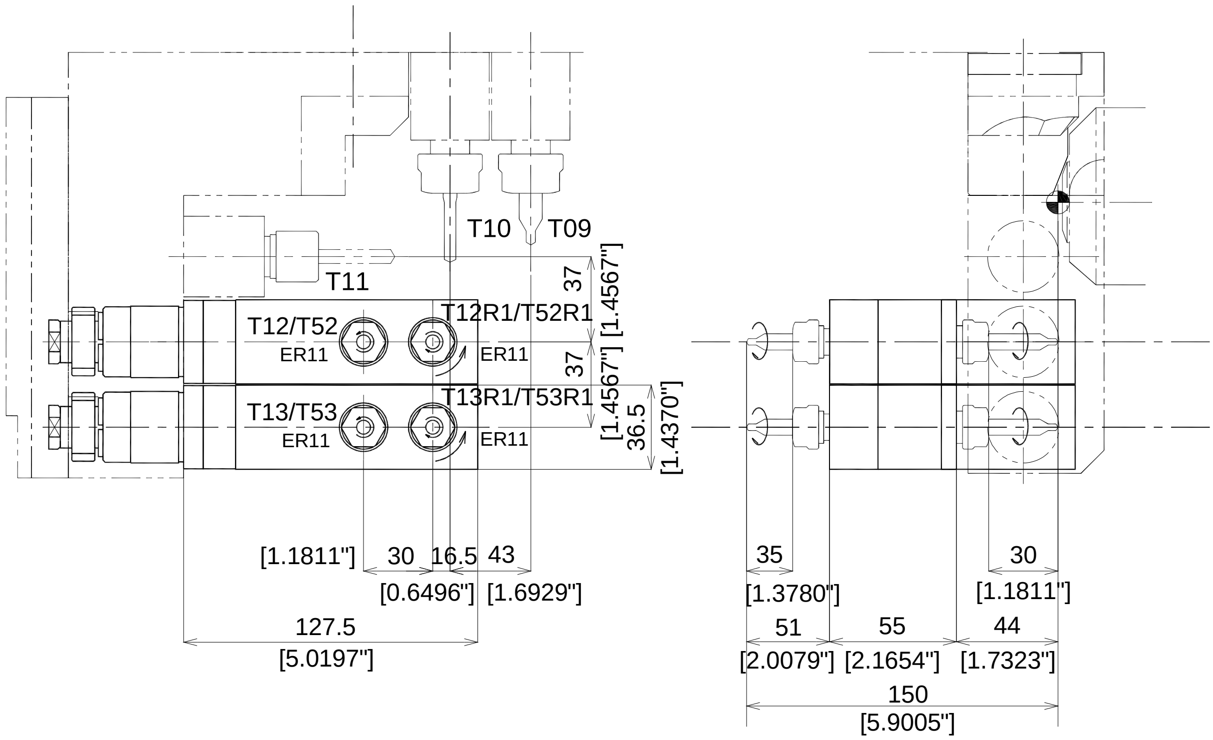


Specification	Description
Tool holder name	GSC1310
Machine type	VIII, IX, X, XII
Max. chuck dia.	ø10 mm [ø0.3937"]
Spindle speed	200 to 6,000min^{-1}
Chuck type	ER16, AR16
Provided set screws	Four M5×35 screws
Provided tool	–

End-face Drilling Spindle

GSE3707 End-face Drilling Spindle (Double spindles for both ends)

This spindle is provided for drilling or milling on the end-face while the main spindle is stopped.
Fix the mounting plate to GSE3210, and then fix GSE3207 to the mounting plate.



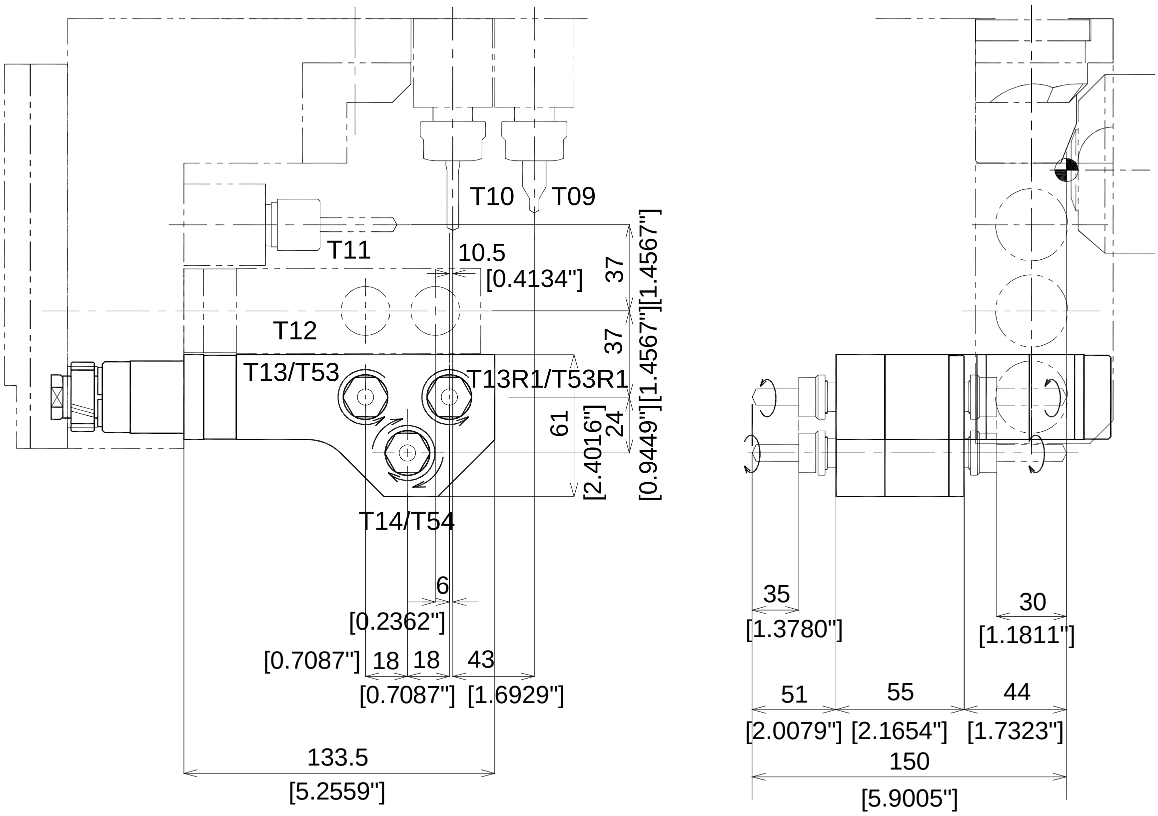
[Note]

- Up to three spindles can be mounted on GSE3210 (T11 to T13). Note that T10 becomes unavailable if this spindle is mounted on T11. When this spindle is mounted on T12 or T13, care must be taken for setting T10 in diameter direction.
- This spindle becomes unavailable if GSE3210 is mounted in the end-face direction.
- When this spindle is mounted, the back spindle and the opposite sleeves may interfere with this spindle. Care must be taken.
- When mounted on type X, the axis stroke is restricted in pinch milling.

Specification	Description
Tool holder name	GSE3707
Machine type	VIII, X
Max. chuck dia.	ø7 mm [ø0.2756"]
Spindle speed	200 to 6,000min^{-1}
Chuck type	ER11, AR11
Spindle rotation direction	Direction shown by the arrow in the figure above for a forward rotation command to T07
Provided set screws	Four M5×10 screws, and four M5×16 screws

GSE3807 End-face Drilling Spindle (Triple spindles for both ends)

This spindle is provided for drilling or milling on the end-face while the main spindle is stopped.
Fix the mounting plate to GSE3210, and then fix GSE3807 to the mounting plate.



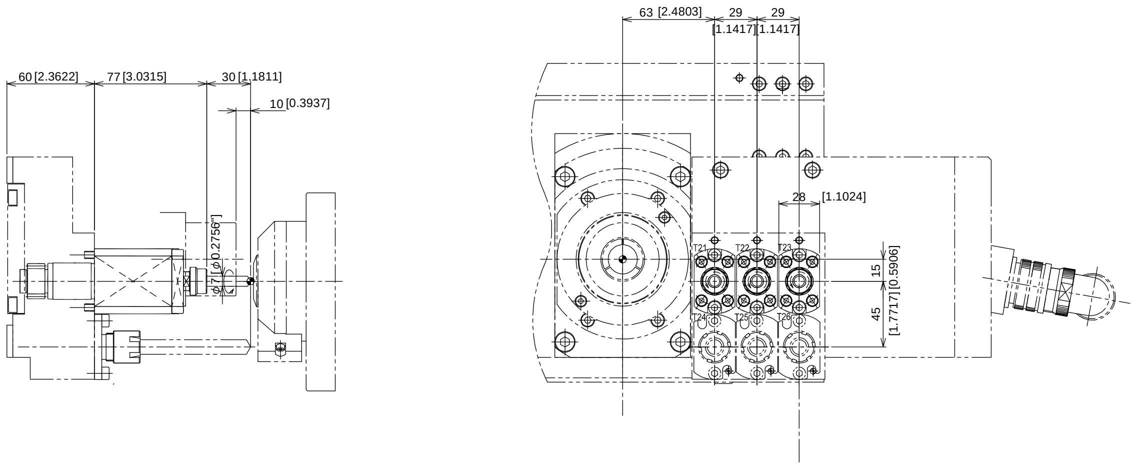
[Note]

- This spindle can be mounted on GSE3210 (T13). When this spindle is mounted on T13, care must be taken for setting T10 in diameter direction.
- This spindle becomes unavailable if GSE3210 is mounted in end-face direction.
- When this spindle is mounted, the back spindle and the opposite sleeves may interfere with this spindle. Care must be taken.
- When mounted on type X, the axis stroke is restricted in pinch milling.

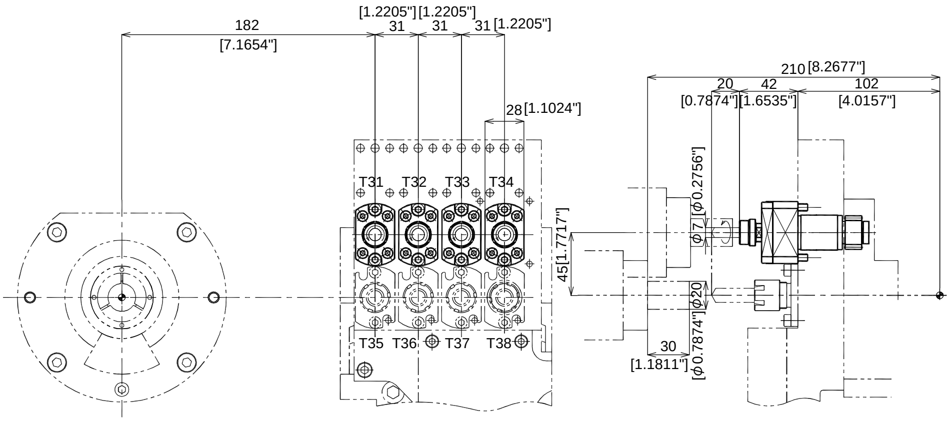
Specification	Description
Tool holder name	GSE3807
Machine type	VIII, X
Max. chuck dia.	ø7 mm [ø0.2756"]
Spindle speed	200 to 6,000min^{-1}
Chuck type	ER11, AR11
Spindle rotation direction	Direction shown by the arrow in the figure above for a forward rotation command to T07
Provided set screws	Four M5×10 screws and four M5×16-4 screws

This spindle is to be mounted on end-face and is used for drilling or milling on the end-face.

GSE3407　End-face Drilling Spindle (mounted on U128B)



MSC507　End-face Drilling Spindle (mounted on U155B)



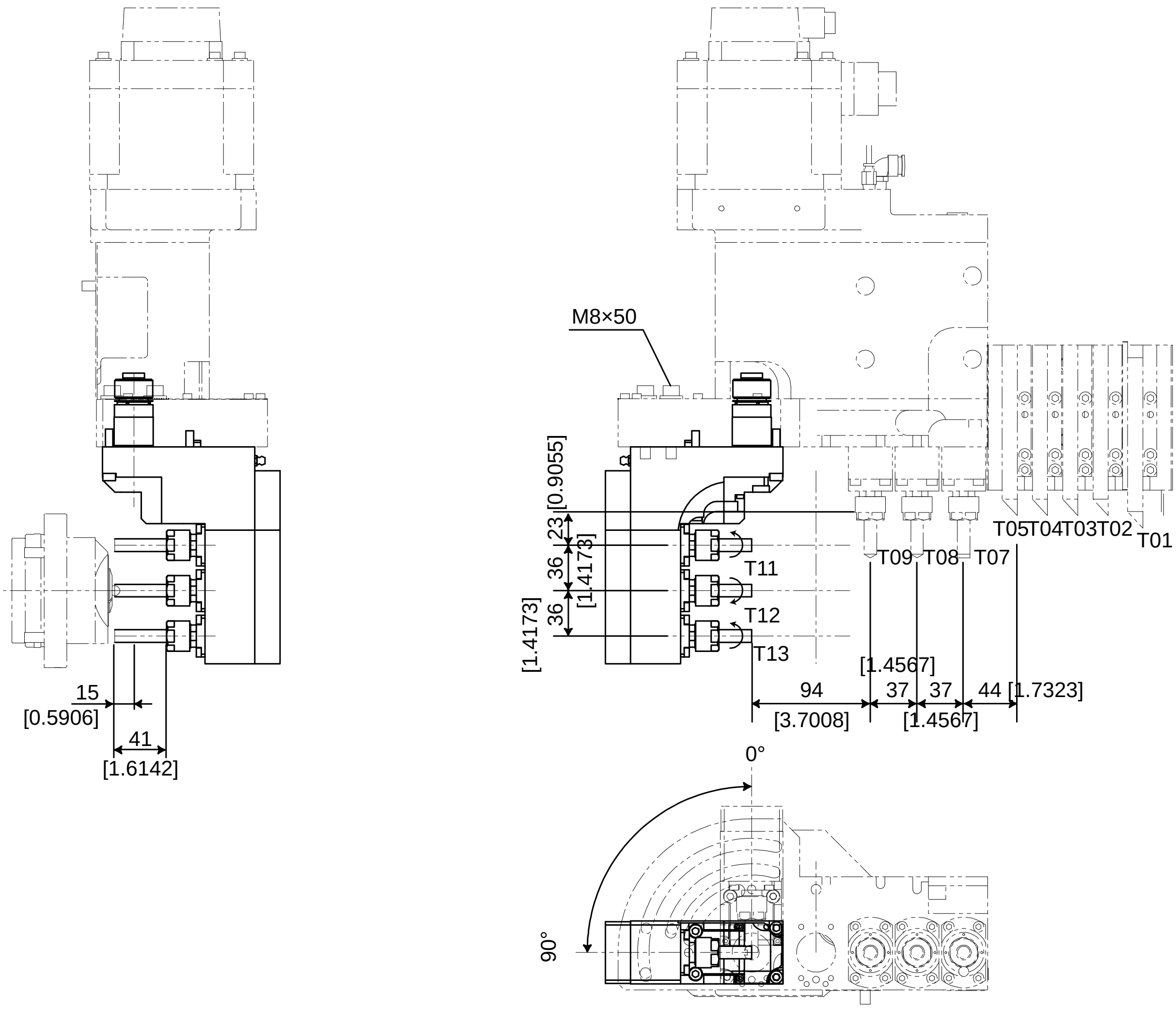
[Note]

- GSE3407 is tooling for the U128B.
- MSC507 is for U128B, U153B or U155B back rotary tool driving device.
- To use this spindle on U153B, block the M8 holes (2 places) on the upper part of U153B with hexagon socket head bolts M8×8.

Specification	Description	
Tool holder name	GSE3407	MSC507
Machine type	X, XII	VIII, IX, X, XII
Max. chuck dia.	Ø7 mm [ø0.2756"]	Ø7 mm [ø0.2756"]
Spindle speed	300 to 7,500min^{-1} (U128B)	200 to 7,500min^{-1} (U128B, U155B)
Chuck type	ER11, AR11	ER11, AR11
Provided set screws	–	Two M5×25 screws

GSE1510 3-Tool Drilling Spindle

This spindle is used to perform drilling on the end-face or outer circumference of the workpiece, or to perform machining by the end mill, after stopping the spindle.



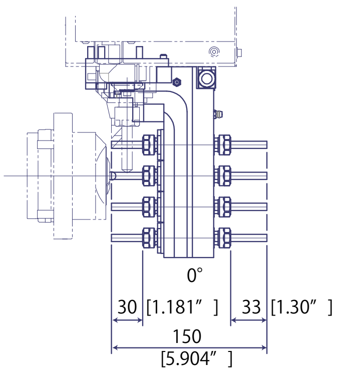
[Note]

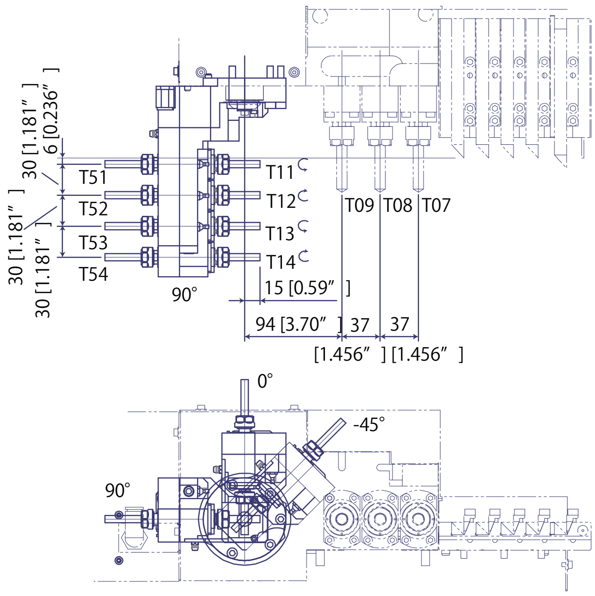
- This holder can be mounted only on T10.
- It can be adjusted to 0, to 90 degrees.
- When an angle between 12 and 66 degrees is used on the T10, interference with the T09 tooling is possible. If this happens, remove the T09 tooling.

Specification	Description
Tool holder name	GSE1510
Machine type	VIII, X
Max. chuck diameter	ø 10 mm [ø0.3927"]
Spindle speed	Max. 6,000min^{-1}
Chuck type	ER16, AR16
Spindle rotation direction	Direction shown by the arrow in the figure above for a forward rotation command to T11
Provided set screws	Three M8×50 screws (With T11 mounted)

MEU307　Drilling Spindle (Both Sides 4-Tool)

This spindle is used to perform drilling on the end-face or outer circumference of the workpiece, or to perform machining by the end mill, after stopping the spindle. The spindle can be mounted at an arbitrary angle between –45 to 90 degrees against the workpiece.





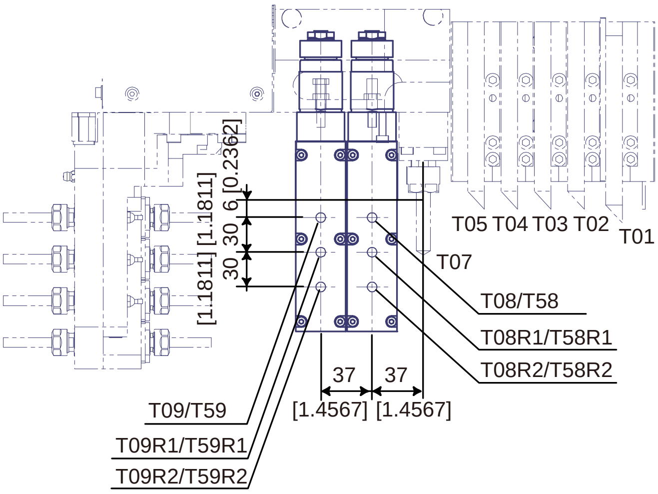
[Note]

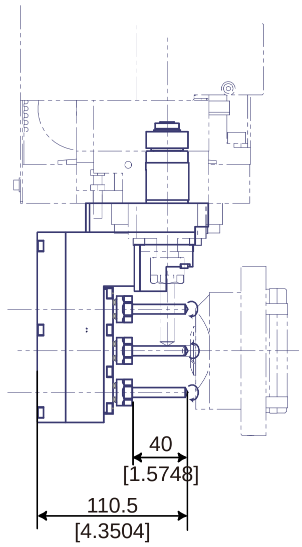
- When used at an angle other than 90°, make sure that it does not interfere with the back headstock and the opposite sleeves.
- Mounting the BSE607 and BSE707 on the T09 will restrict angle of movement.

Specification	Description
Tool holder name	MEU307
Machine type	IX, XII
Max. chuck diameter	ø7 mm [ø0.2756"]
Spindle speed	Max. 8,000 min^{-1}
Chuck type	ER11, AR11
Spindle rotation direction	Direction shown by the arrow in the figure above for a forward rotation command to T07
Provided set screws	Five M6×35 screws

BSE607 End-face Drilling Spindle (3-Tool)

This spindle is mounted on rotary tool drive device, and used for end-face drilling.





* This figure shows type XII.

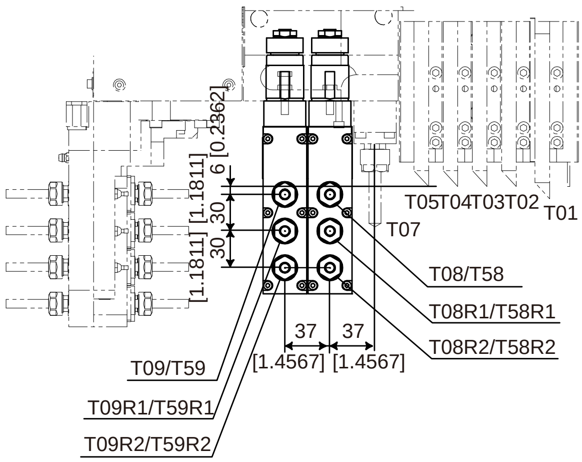
[Note]

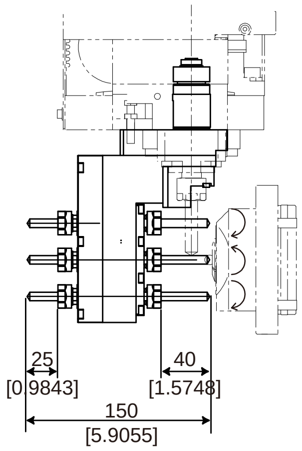
- Make sure that it does not interfere with the back headstock and the opposite sleeves.
- Take care as the amount of workpiece protrusion may result in interference with the tooling.

Specification	Description
Tool holder name	BSE607
Max. chuck dia.	ø7 mm [ø0.2756"]
Spindle speed	Max. 6,000 min^{-1}
Chuck type	ER11, AR11
Spindle rotation direction	Direction shown by the arrow in the figure above for a forward rotation command to T07
Provided set screws	Hexagon socket head bolts two M5×25, Hexagon head bolt one M6×20

BSE707　Both end-face Drilling Spindle (3-Tool)

This spindle is mounted on rotary tool drive device, and used for end-face drilling.





* This figure shows type XII.

[Note]

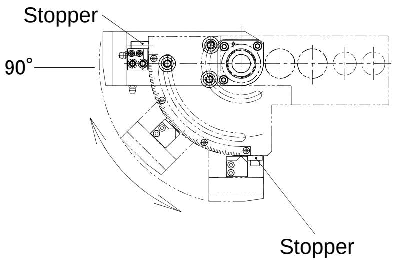
- Make sure that it does not interfere with the back headstock and the opposite sleeves.
- Take care as the amount of workpiece protrusion may result in interference with the tooling.

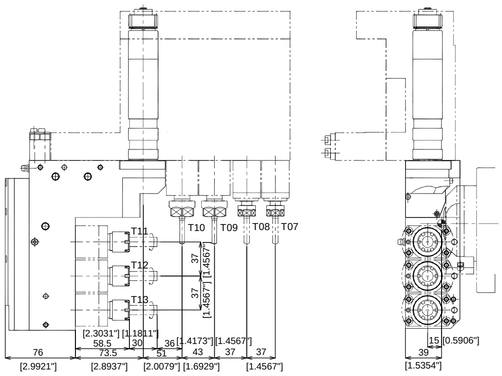
Specification	Description
Tool holder name	BSE707
Max. chuck dia.	ø7 mm [ø0.2756"]
Spindle speed	Max. 6,000 min^{-1}
Chuck type	ER11, AR11
Rotary tool drive device	U33B, U35B
Spindle rotation direction	Direction shown by the arrow in the figure above for a forward rotation command to T07
Provided set screws	Heaxagon socket head bolts two M5×25, Hexagon head bolt one M6×20

Outer Circumference Milling Spindle, End-face Drilling Spindle

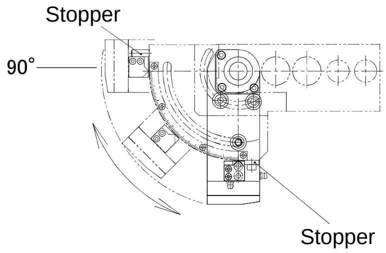
This spindle is to be mounted on U35B rotary tool in the end-face or cross-machining direction, and is used for outer circumference and end-face machining including drilling, tapping, or milling with rotary tools.

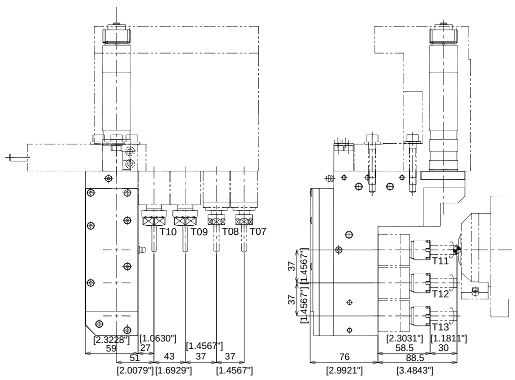
GSE3210 3-tool Cross Machining/ End-face Drilling Spindle (mounted in cross machining direction)





GSE3210 3-tool Cross Machining/ End-face Drilling Spindle (mounted in end-face machining direction)





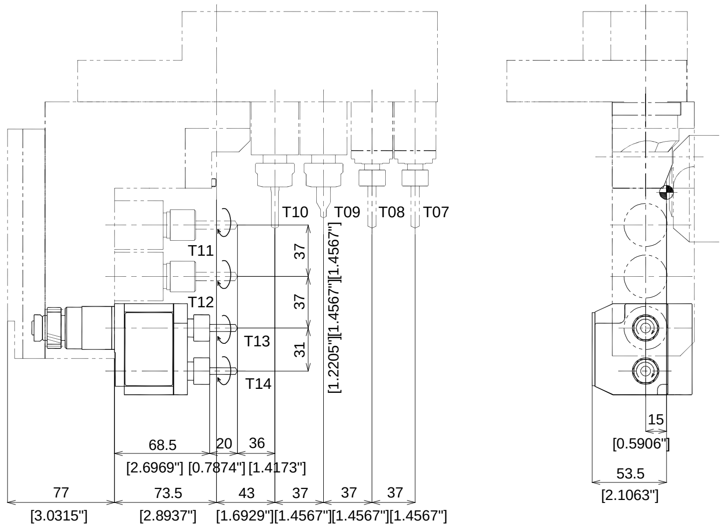
[Note]

- When performing milling by the rotary tool mounted next to this holder, pay attention to the interference between the workpiece and this holder.
- When this spindle is mounted, the back spindle and the opposite sleeves may interfere with this spindle. Care must be taken.
- If this holder is used for end-face direction, GSE3707 and GSE3807 cannot be used.

Specification	Description	
Tool holder name	GSE3210 (when mounted in cross machining direction)	GSE3210 (when mounted in end-face machining direction)
Machine type	VIII, X	VIII, X
Outer dia. of sleeve	Ø31 mm [ø1.2205"]	Ø31 mm [ø1.2205"]
Spindle speed	200 to 6,000min^{-1}	200 to 6,000min^{-1}
Rotary tool	GSC1310 (T11 to T13), GSC1607 (T13), GSE3707 (T12, T13), GSE3807 (T13)	GSC1310 (T11 to T13), GSC1607 (T13)
Provided set screws	Three M8×40 screws	–

This spindle is provided for drilling on the outer circumference or milling a key groove while the main spindle is stopped.

GSC1607　Outer Circumference Milling Spindle



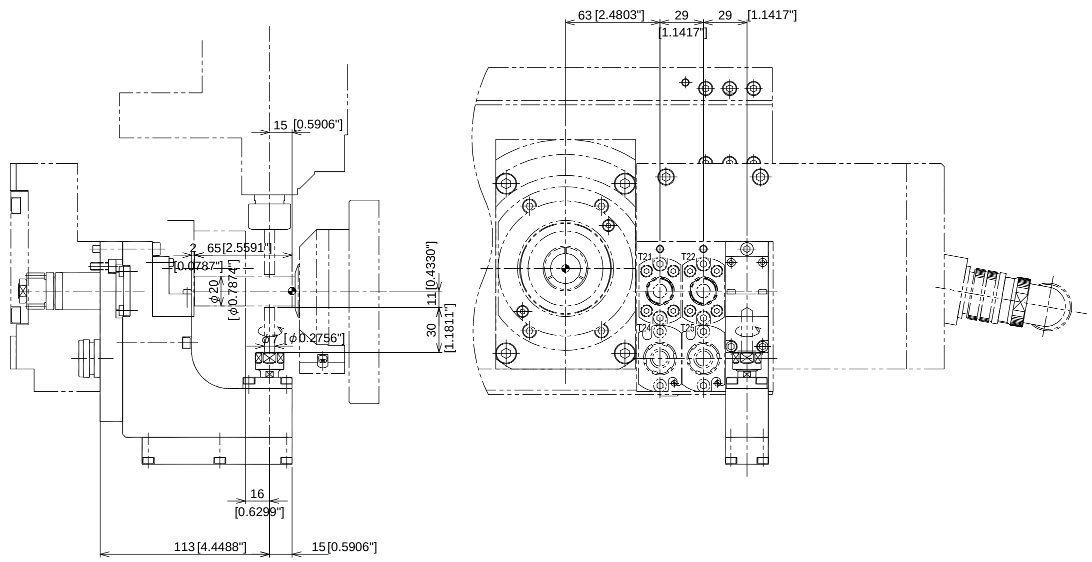
[Note]

The spindle can be mounted only on T13 of GSE3210.

Specification	Description
Tool holder name	GSC1607
Machine type	VIII, X
Outer dia. of sleeve	Ø7 mm [ø0.2756"]
Spindle speed	200 to 6,000min^{-1}
Rotary tool	ER11, AR11
Spindle rotation direction	Direction shown by the arrow in the figure above for a forward rotation command to T07
Provided set screws	Two M5×25 screws, one M5×35 screw

This spindle is to be mounted on front rotary tool driving device (U128B), and is used for pinch milling. Fix the base plate to U128B, and then fix GSC1407 to the base plate.

GSC1407　Outer Circumference Milling Spindle



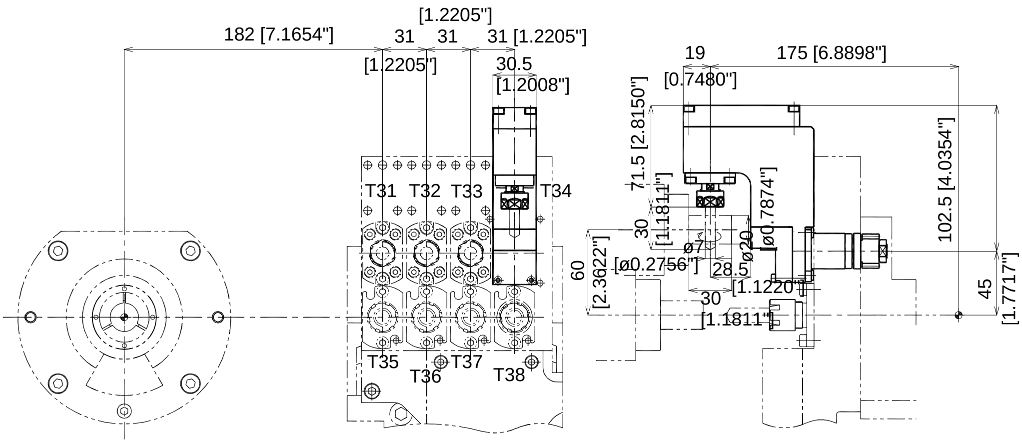
[Note]

- This spindle can be mounted on T21, T22, or T23. At this time, the tool post on lower stage becomes unavailable. (When mounted on T23, T26 becomes unavailable.)
- Pay attention to protrusion length to avoid an inteference with the front gang tool.
- With this spindle is mounted on U128B, to machine the workpiece with the other end-face tool, move the Z2 axis 20 mm to the negative (–) direction to avoid an interference with the guide bushing.
- When using an optional simplified tool presetter for setting tools, a dedicated jig (U401X) is required.

Specification	Description
Tool holder name	GSC1407
Machine type	X, XII
Max. chuck dia.	ø7 mm [ø0.2756"]
Spindle speed	200 to 7,500min^{-1}
Chuck type	ER11, AR11
Spindle rotation direction	Direction shown by the arrow in the figure above for a forward rotation command
Provided set screws	Two M5×16 screws, one M5×12 screw, two M5×50 screws, and one M5×40 screw

This spindle is to be mounted on back rotary tool driving device (U155B), and is used for back cross machining.

GSC1507 Outer Circumference Milling Spindle



[Note]

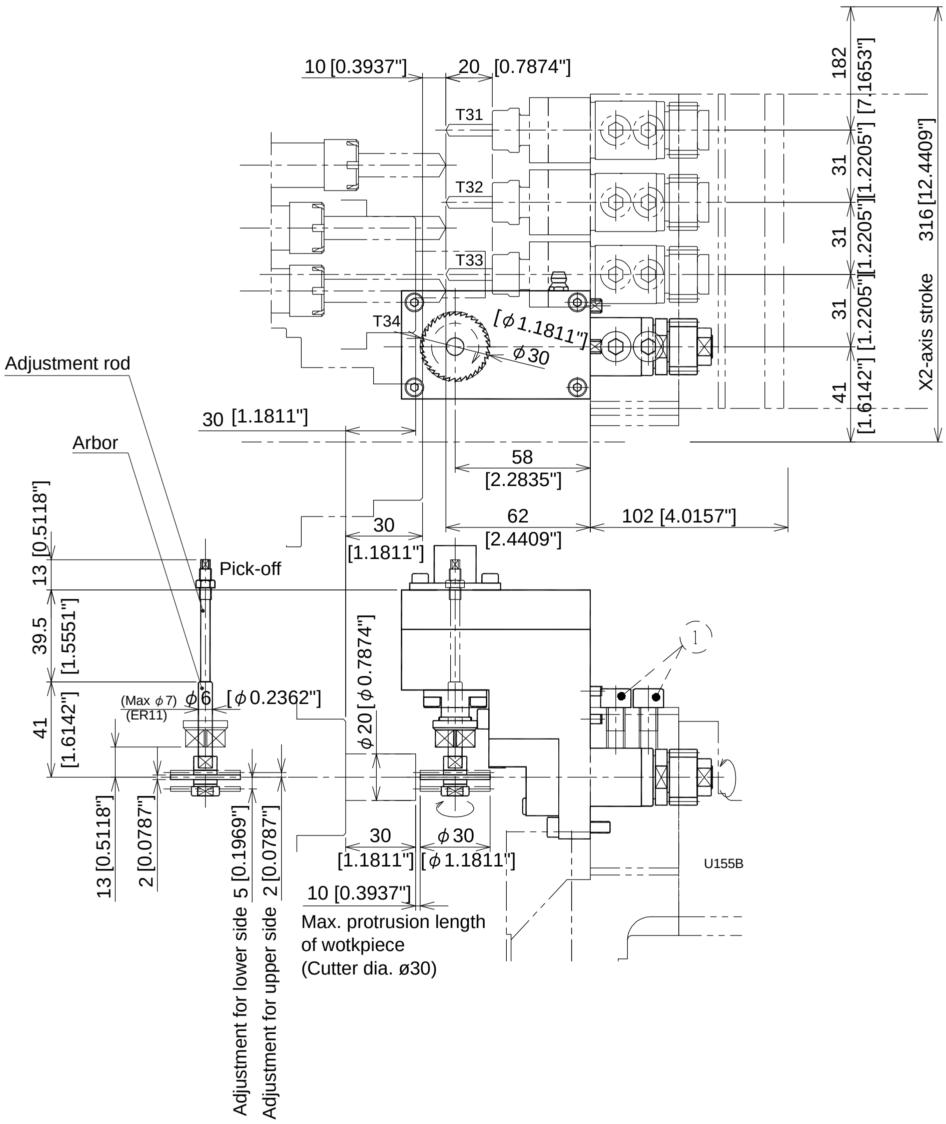
- Pay attention to protrusion length to avoid an inteference with the workpiece on back spindle.
- For details, see *<13.47 Major restrictions of tooling setup (L20X) >* in Programmer's Manual.
- When using an optional simplified tool presetter for setting tools, a dedicated jig (U402X) is required.

Specification	Description
Tool holder name	GSC1507
Machine type	X, XII
Max. chuck dia.	ø7 mm [ø0.2756"]
Spindle speed	200 to 7,500min^{-1}
Chuck type	ER11, AR11
Spindle rotation direction	Direction shown by the arrow in the figure above for a forward rotation command
Provided set screws	Four M5×40 screws

Slitting Spindle

This spindle is to be mounted on U153B or U155B back rotary tool driving device, and is used for screw slitting on back side.

GSS1530 Slitting Spindle (with slitting cutter)



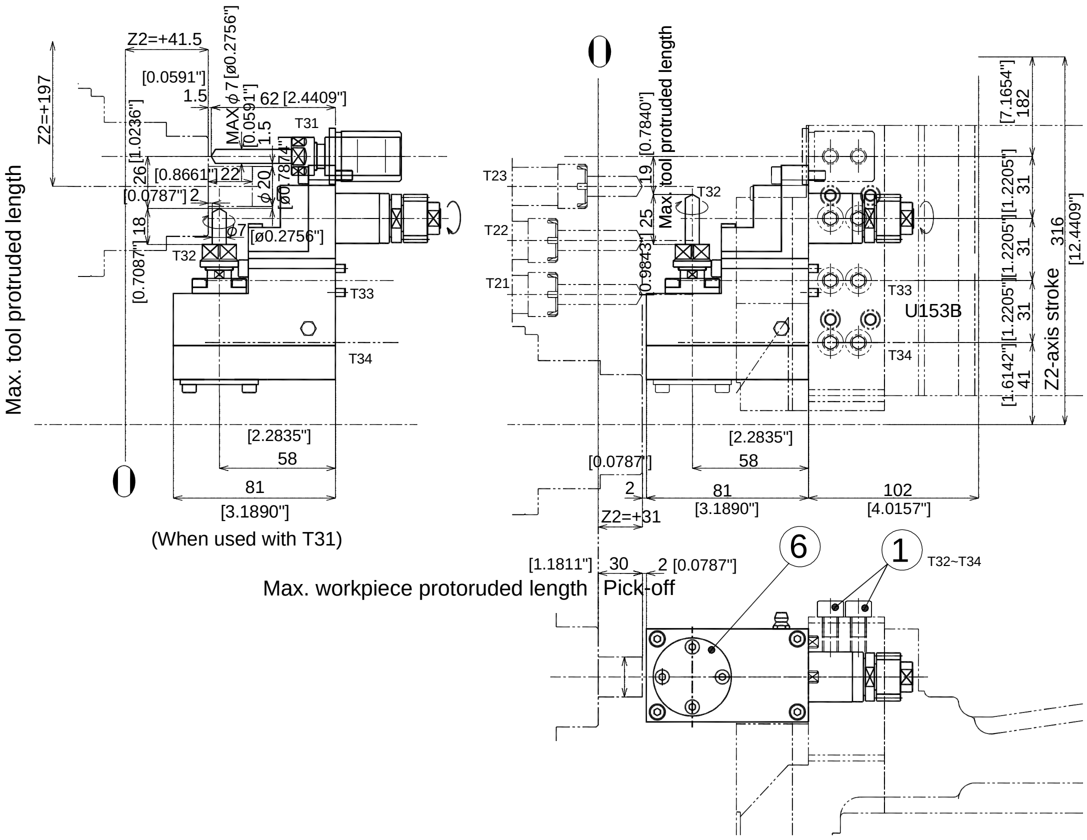
[Note]

- This spindle can be mounted on T34 only.
- To use this spindle on U153B, block the M8 holes (8 places) on the upper part of U153B with hexagon socket head bolts M8×8 (shown by ① in the figure shown above).
- If the cutter of its diameter exceeds 30 mm, the workpiece on the back spindle may interfere with the slitting cutter. Pay attention to the protruded length of workpiece.
- If the cutter is used at the center of the workpiece on the back spindle, the cutter thickness must be less than 2 mm. Otherwise, the cutter will interfere with the machine.
- The height can be defined by hitting the arbor against the adjustment rod.
- The distance between the end-face of the stopper and center of the workpiece is set to 41 mm when the adjustment rod height is 13 mm above the top face of the machine.
- The adjustment rod moves 0.5 mm per a rotation, that allows fine adjustment.

Specification	Description
Tool holder name	GSS1530
Machine type	VIII, IX, X, XII
Max. chuck dia.	ø7 mm [ø0.2756"]
Spindle speed	120 to 4,500min^{-1}
Chuck type	ER11, AR11
Spindle rotation direction	Direction shown by the arrow in the figure above for a forward rotation command
Max. cutter diameter	ø30 mm [ø1.1811"]
Max. cutter thickness	2 mm [ø0.0787"]
Provided set screws	One M5×20 screw and two M5×50 screws

This spindle is mounted on back rotary tool driving device (U153B) and used for back cross machining.

GSS1530 Slitting Spindle (with cross machining tool)



[Note]

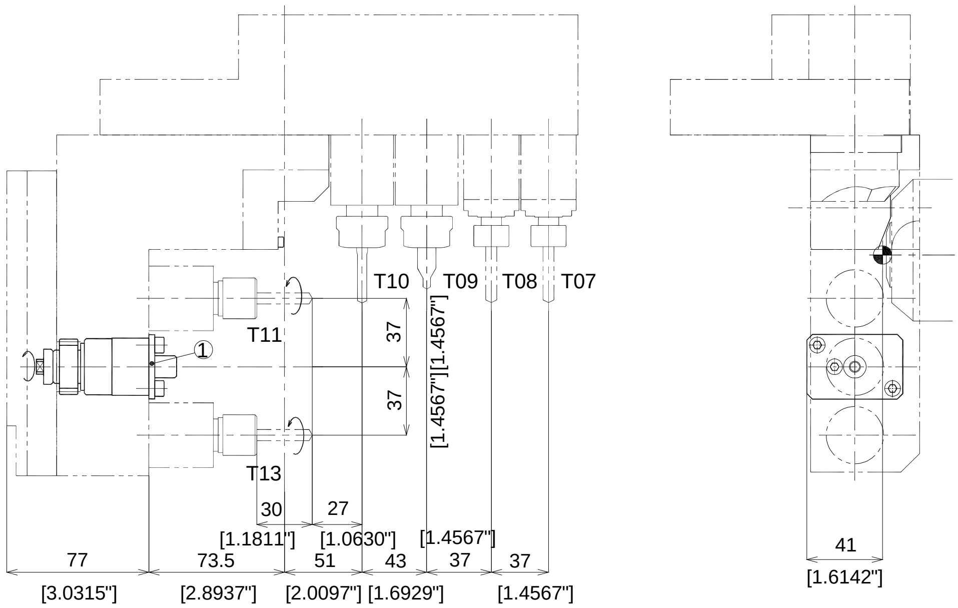
- This spindle can be mounted on T32 only. Cannot be mounted on T33 and T34. Attach a cap provided with U153B. MSC507 cannot be used.
- To use this spindle, block the M8 holes (8 places) on the upper part of U153B with hexagon socket head bolts M8×8 (shown by ① in the figure shown above).
- To use this spindle in cross machining direction, remove the adjusting rod and mount ⑥. (Use the four M4×8 screws provided.).
- The spindle may interfere with the workpiece on back spindle. Pay attention to workipiece protrusion.

Specification	Description
Tool holder name	GSS1530
Machine type	VIII, IX
Max. chuck dia.	ø7 mm [ø0.2756"]
Spindle speed	120 to 4,500min^{-1}
Chuck type	ER11, AR11
Spindle rotation direction	Direction shown by the arrow in the figure above for a forward rotation command
Provided set screws	–

Idle Gear Unit

This unit can be mounted on GSE3210. Mount this unit on T11 or T12 if the rotary tool unit is mounted on T12 or T13.

GSD107　Idle Gear Unit



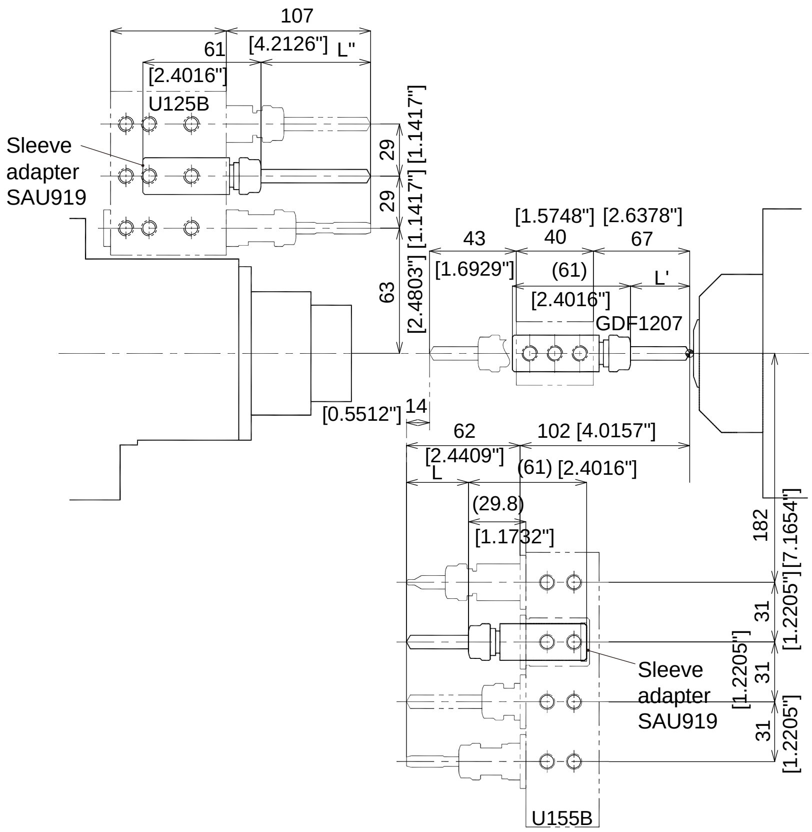
Specification	Description
Tool holder name	GSD107
Machine type	VIII, X
Provided set screws	Two M5×8 screws

Sleeve

Boring and Drill Sleeve

BDS507 Boring and Drill Sleeve (Rego Type Chuck, ~ø7)

This sleeve chucks the drill and other tools with the straight shank used for the front and back machining by the rego type chuck. L and L' or L" dimension is adjustable by putting sleeve in and out.



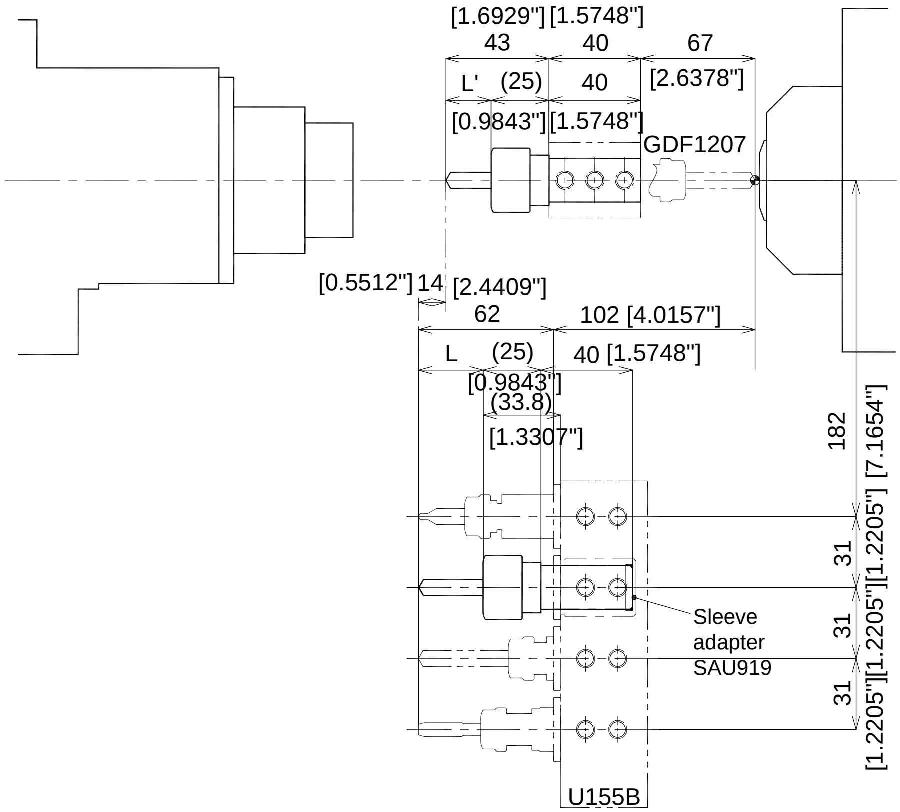
[Note]

- To use this sleeve on U153B machine, SAU819 is required.
- To use this sleeve on U125B, U126B, U128B and U153B machine, SAU919 is required.

Specification	Description
Tool holder name	BDS507
Machine type	VIII, IX, X, XII
Max. chuck dia.	ø7 mm [ø0.2756"]
Chuck type	ER11, AR11
Outer dia. of sleeve	ø19.05 mm [ø3/4"]

BDS508 Boring and Drill Sleeve (Rego Type Chuck, ~ø10)

This sleeve chucks the drill and other tools with the straight shank used for the back machining by the rego type chuck. L or L' dimension is adjustable by putting sleeve in and out.



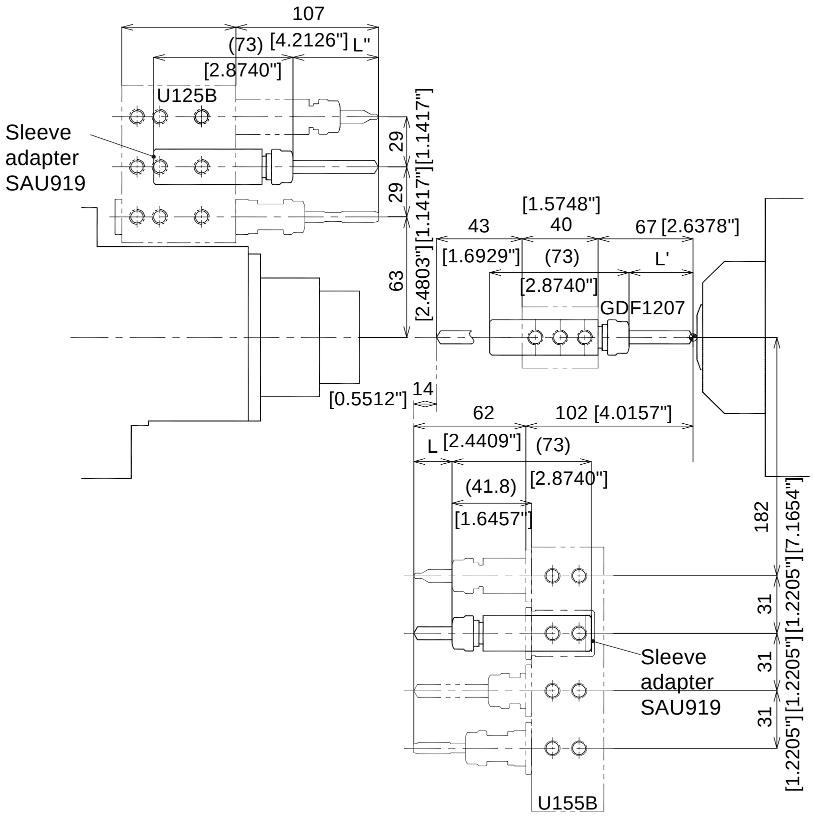
[Note]

- Depending on diameter of sleeve cap, sleeves may not be aligned in same stage.
- SAU819 is required on U153B machine, and SAU 919 is required on U125B, U126B, U128B and U155B machine.

Specification	Description
Tool holder name	BDS508
Machine type	VIII, IX, X, XII
Max. chuck dia.	ø10 mm [ø0.3937"]
Chuck type	ER16, AR16
Outer dia. of sleeve	ø19.05 mm [ø3/4"]

BDS607 Boring and Drill Sleeve (Rego Type Chuck, ~ø7)

This sleeve chucks the drill and other tools with the straight shank used for the front and back machining by the rego type chuck. L and L' or L" dimension is adjustable by putting sleeve in and out.



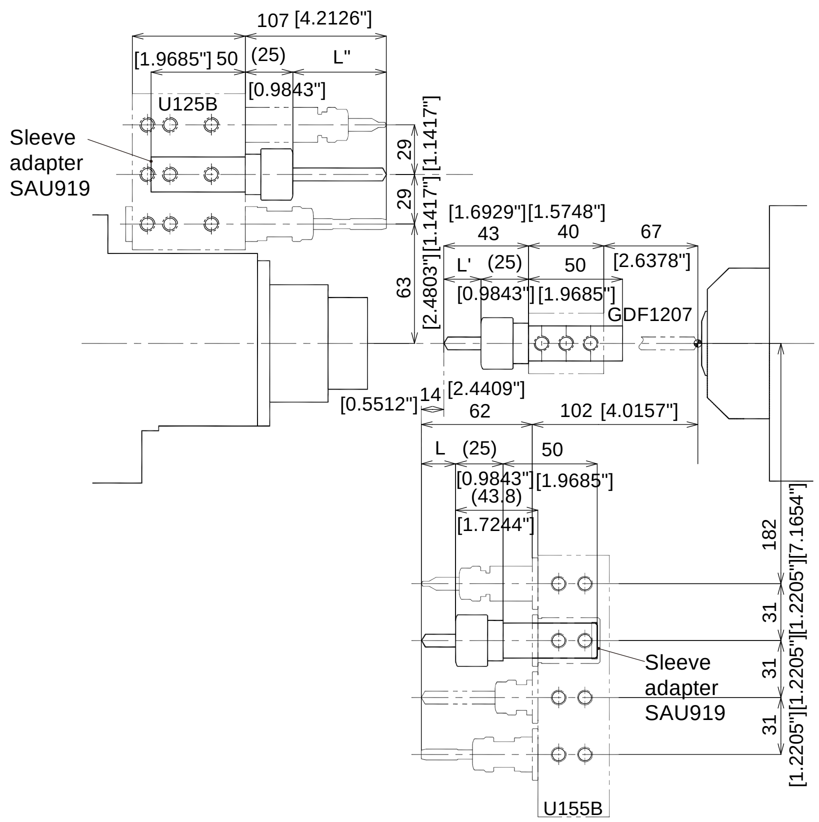
[Note]

- To use this sleeve on U153B machine, SAU819 is required.
- To use this sleeve on U125B, U126B, U128B and U155B machine, SAU919 is required.

Specification	Description
Tool holder name	BDS607
Machine type	VIII, IX, X, XII
Max. chuck dia.	ø7 mm [ø0.2756"]
Chuck type	ER11, AR11
Outer dia. of sleeve	ø19.05 mm [ø3/4"]

BDS610 Boring and Drill Sleeve (Rego Type Chuck, ~ø10)

This sleeve chucks the drill and other tools with the straight shank used for the front machining by the rego type chuck. L and L' or L" dimension is adjustable by putting sleeve in and out.



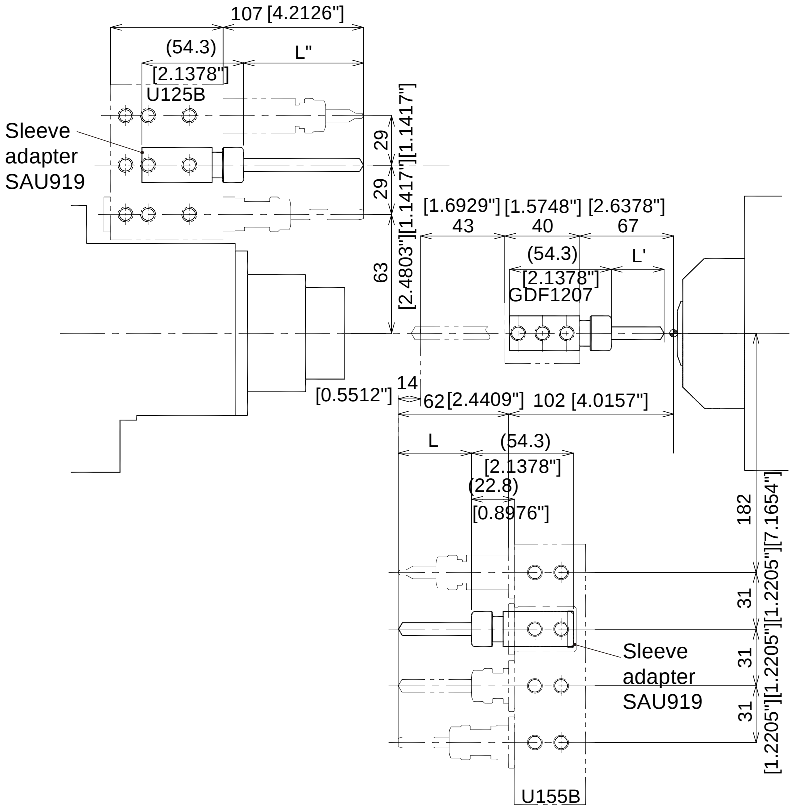
[Note]

- Depending on diameter of sleeve cap, sleeves may not be aligned in same stage.
- To use this sleeve on U153B machine, SAU819 is required.
- To use this sleeve on U125B, U126B, 128B and U155B SAU919 is required.

Specification	Description
Tool holder name	BDS610
Machine type	VIII, IX, X, XII
Max. chuck dia.	ø10 mm [ø0.3937"]
Chuck type	ER16, AR16
Outer dia. of sleeve	ø19.05 mm [ø3/4"]

BDS707 Boring and Drill Sleeve (Rego Type Chuck, ~ø7)

This sleeve chucks the drill or other tools with the straight shank used for the front and back machining by the rego type chuck. L and L' or L" dimension is adjustable by putting sleeve in and out.



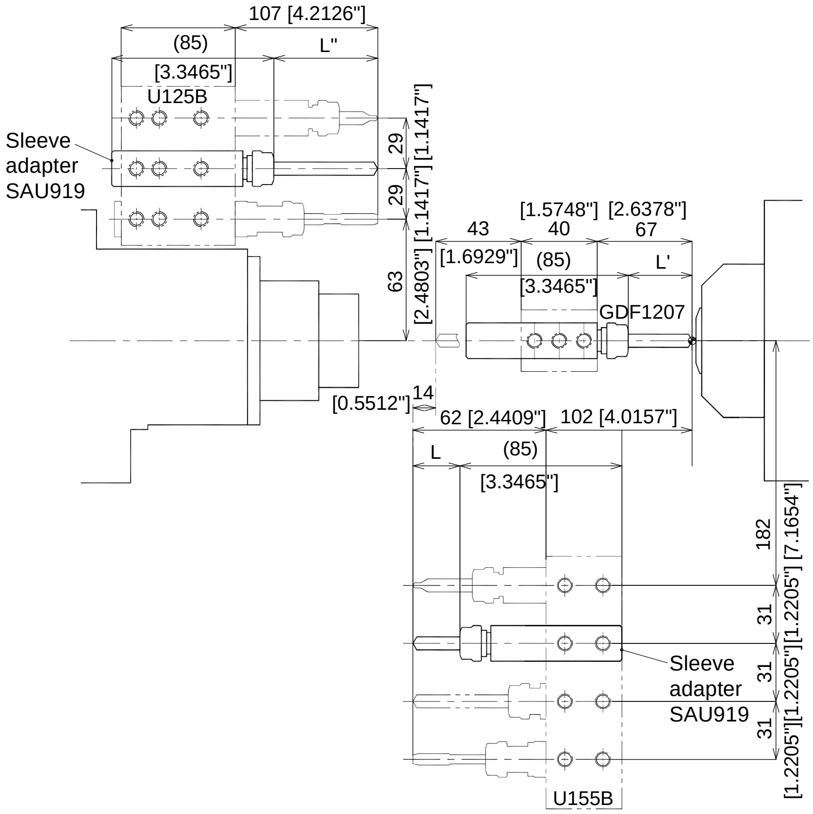
[Note]

- To use this sleeve on U153B machine, SAU819 is required.
- To use this sleeve on U125B, U126B, U128B and U155B machine, SAU919 is required.

Specification	Description
Tool holder name	BDS707
Machine type	VIII, IX, X, XII
Max. chuck dia.	ø7 mm [ø0.2756"]
Chuck type	ER11, AR11
Outer dia. of sleeve	ø19.05 mm [ø3/4"]

VDS506 Boring and Drill Sleeve (Rego Type Chuck, ~ø7)

This sleeve chucks the drill or other tools with the straight shank used for the front and back machining by the rego type chuck. L and L' or L" dimension is adjustable by putting sleeve in and out.



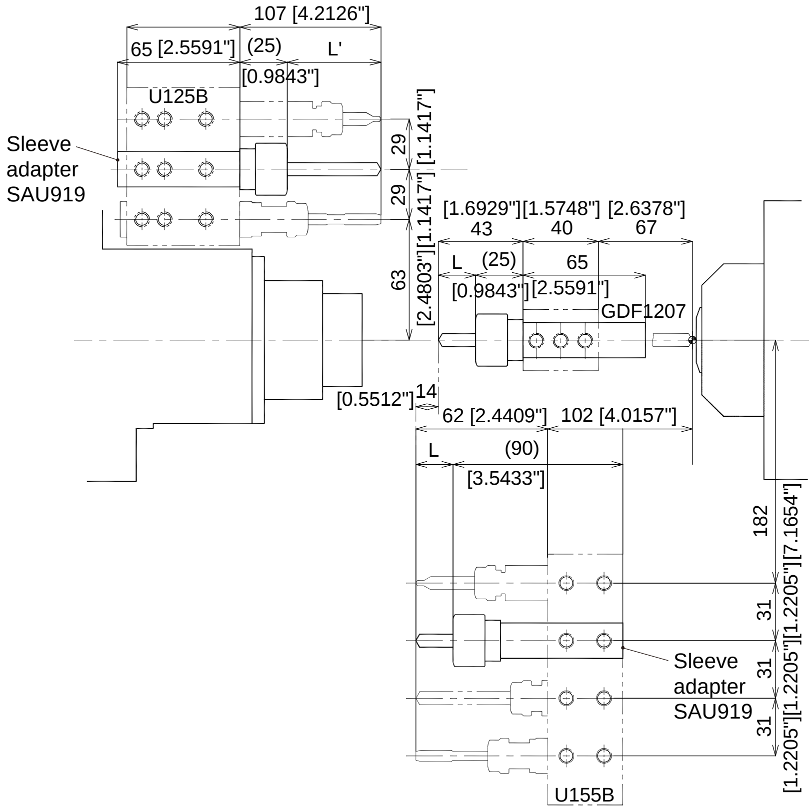
[Note]

- To use this sleeve on U153B machine, SAU819 is required.
- To use this sleeve U125B, U126B, U128B and U155B machine, SAU919 is required.
- VDS506 may not be mounted depending on extended tool length L, when VDS506 is used together with SAU819 or SAU919.

Specification	Description
Tool holder name	VDS506
Machine type	VIII, IX, X, XII
Max. chuck dia.	ø7 mm [ø0.2756"]
Chuck type	ER11, AR11
Outer dia. of sleeve	ø19.05 mm [ø3/4"]

VDS110 Boring and Drill Sleeve (Rego Type Chuck, ~ø10)

This sleeve chucks the drill or other tools with the straight shank used for the front and back machining by the rego type chuck. L or L' dimension is adjustable by putting sleeve in and out.



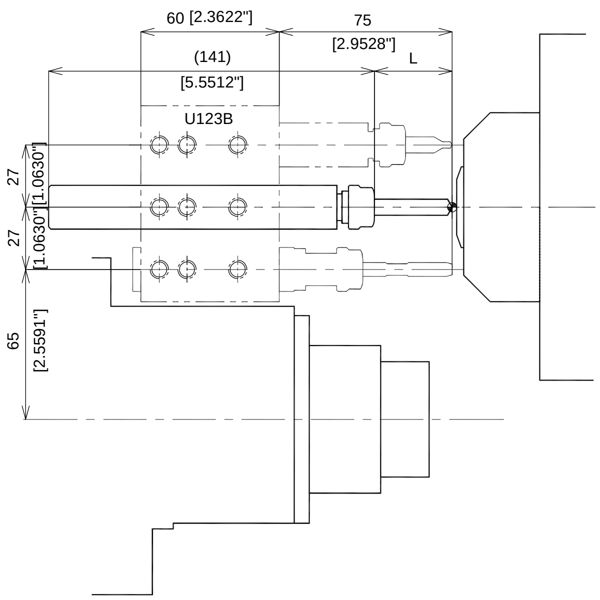
[Note]

- Depending on diameter of sleeve cap, sleeves may not be aligned in same stage.
- To use this sleeve on U153B machine, SAU819 is required.
- To use this sleeve U125B, U126B, U128B and U155B machine, SAU919 is required.
- This sleeve may not be mounted depending on extended tool length L, when this sleeve is used together with SAU819 or SAU919.

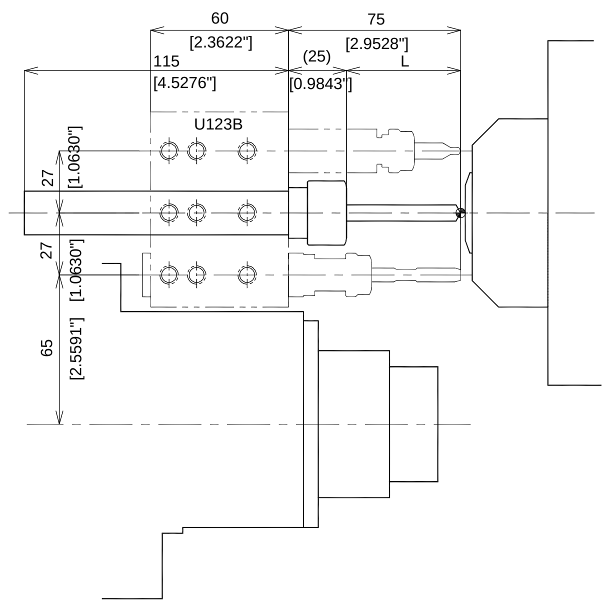
Specification	Description
Tool holder name	VDS110
Machine type	VIII, IX, X, XII
Max. chuck dia.	ø10 mm [ø0.3937"]
Chuck type	ER16, AR16
Outer dia. of sleeve	ø19.05 mm [ø3/4"]

This sleeve chucks the drill or other tools with the straight shank used for the front machining by the rego type chuck. L dimension is adjustable by putting sleeve in and out.

LDS107 Boring and Drill Sleeve (Rego Type Chuck, ~ø7)



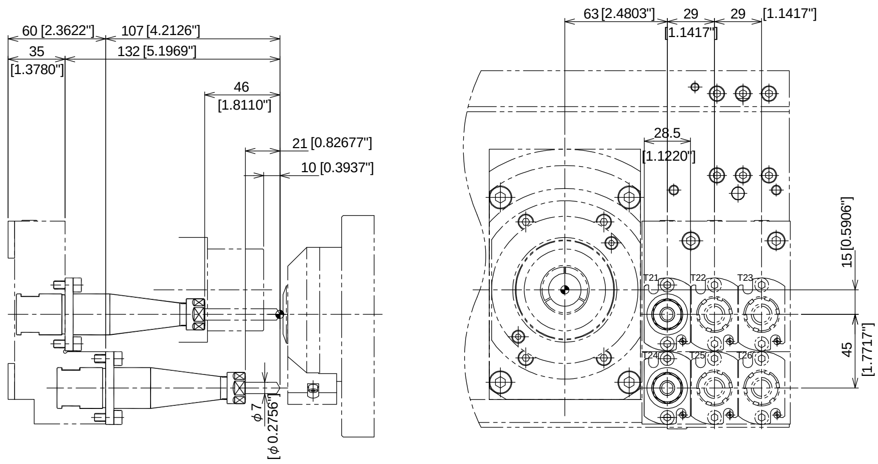
LDS110 Boring and Drill Sleeve (Rego Type Chuck, ~ø10)



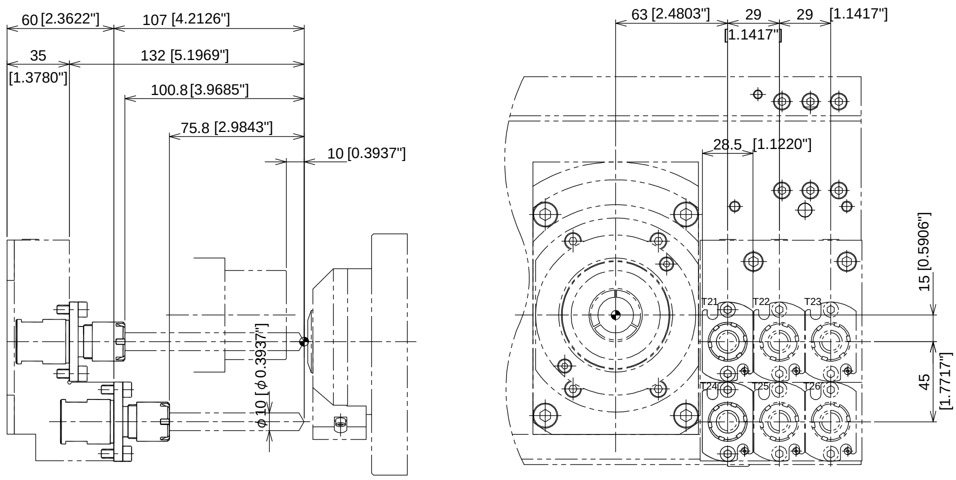
Specification	Description	
Tool holder name	LDS107	LDS110
Machine type	VIII, IX	VIII, IX
Max. chuck dia.	ø7 mm [ø0.2756"]	ø10 mm [ø0.3937"]
Chuck type	ER11, AR11	ER16, AR16
Outer dia. of sleeve	ø19.05 mm [ø3/4"]	ø19.05 mm [ø3/4"]

This sleeve chucks the drill and other tools with the straight shank used for the front and back machining by the rego type chuck. The end face of the sleeve is fixed, therefore, the longitudinal dimension can be adjusted by putting the tool in and out.

GDS107 Boring and Drill Sleeve (Rego Type Chuck, ~ø7)



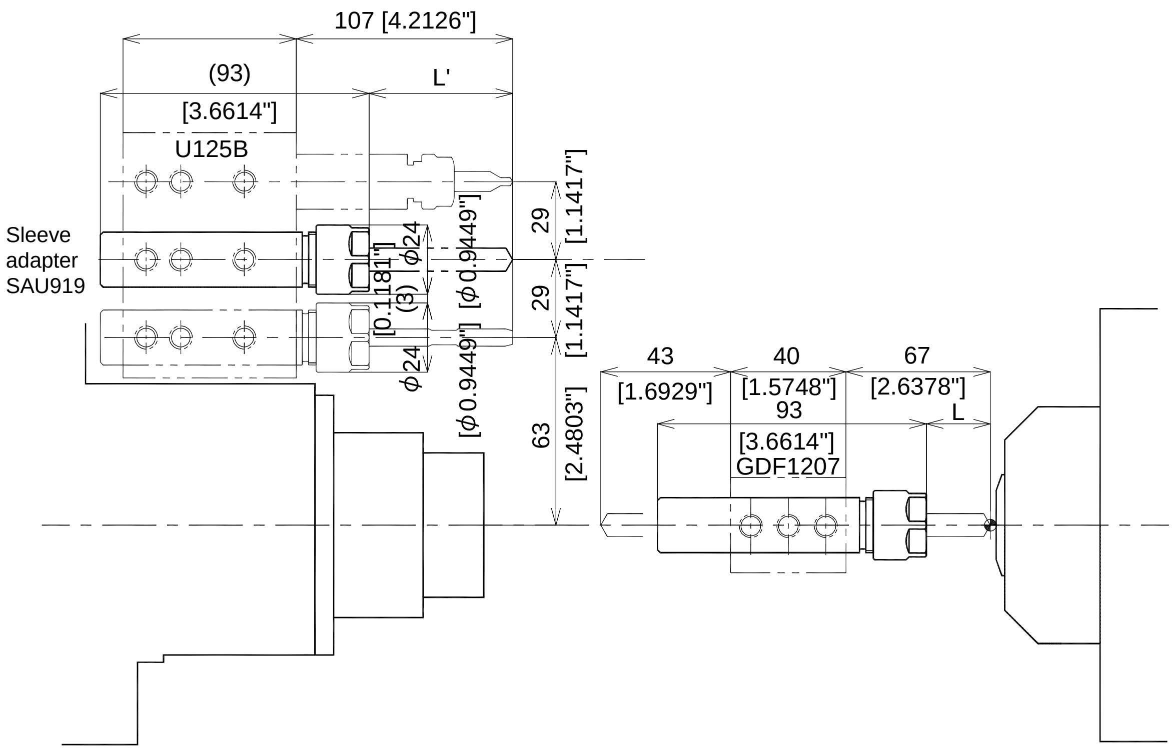
GDS210 Boring and Drill Sleeve (Rego Type Chuck, ~ø10)



Specification	Description	
Tool holder name	GDS107	GDS210
Machine type	X, XII	X, XII
Max. chuck dia.	ø7 mm [ø0.2756"]	ø10 mm [ø0.3937"]
Chuck type	ER11, AR11	ER16, AR16
Outer dia. of sleeve	ø25.0 mm [ø0.9843"]	ø25.0 mm [ø0.9843"]
Provided set screws	Two M5×12 screws	Two M5×12 screws

GDS110 Boring and Drill Sleeve (Rego Type Chuck, ~ø10)

This sleeve chucks the drill and other tools with the straight shank used for the front and back machining by the rego type chuck. L or L' dimension is adjustable by putting sleeve in and out.



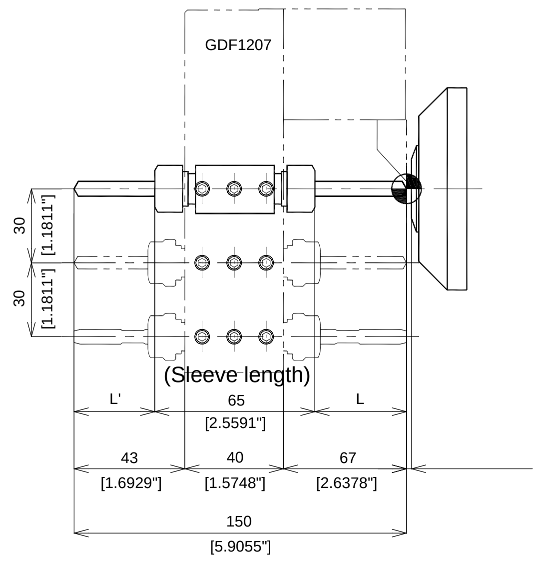
[Note]

- The diameter of sleeve cap is 24 mm.
- This sleeve cannot be used on back tool post.
- To use this sleeve on U128B, SAU919 is required.

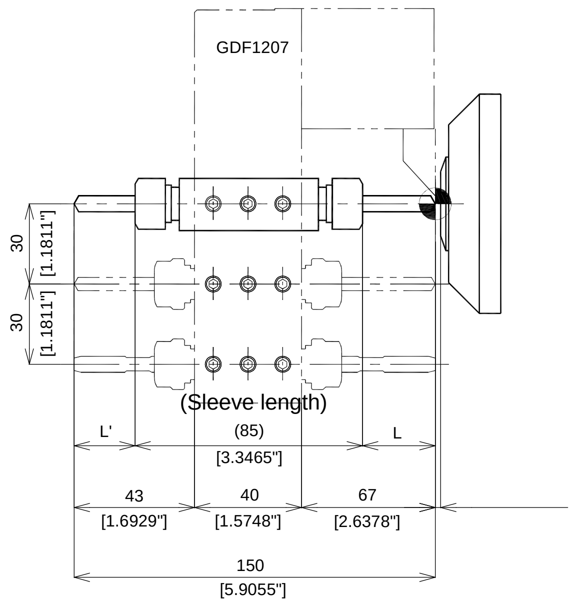
Specification	Description
Tool holder name	GDS110
Machine type	VIII, IX, X, XII
Max. chuck dia.	ø10 mm [ø0.3937"]
Chuck type	ER16, AR16
Outer dia. of sleeve	ø19.05 mm [ø3/4"]

This sleeve chucks the drill or other tools with the straight shank used for the front and back machining by the rego type chuck. L or L' dimension is adjustable by putting sleeve in and out.

HDS5406 Boring and Drill Sleeve
(Both ends, Rego Type Chuck, ~ø7, Short type)



HDS5506 Boring and Drill Sleeve
(Both ends, Rego Type Chuck, ~ø7)



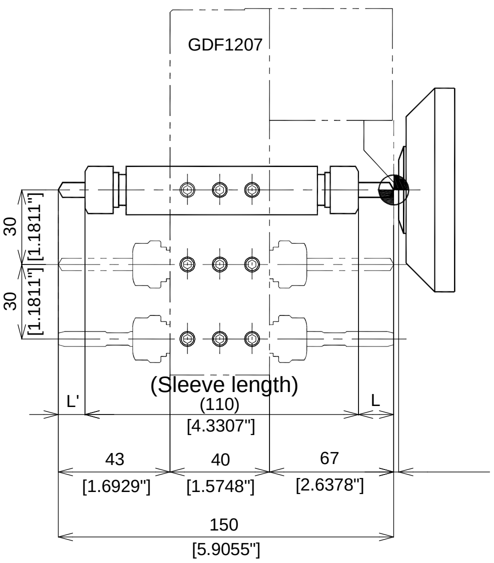
[Note]

Pay attention to the interference between the back spindle including the opposite sleeves, when mounting this sleeve.

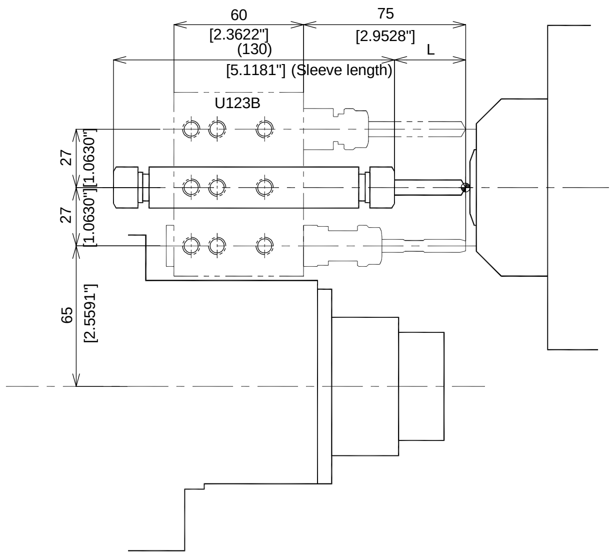
Specification	Description	
Tool holder name	HDS5406	HDS5506
Machine type	VIII, IX, X, XII	VIII, IX, X, XII
Max. chuck dia.	ø7 mm [ø0.2756"]	ø7 mm [ø0.2756"]
Chuck type	ER11, AR11	ER11, AR11
Outer dia. of sleeve	ø19.05 mm [ø3/4"]	ø19.05 mm [ø3/4"]
Sleeve holder	GDF1207	GDF1207
Provided set screws	–	–

This sleeve chucks the drill or other tools with the straight shank used for the front and back machining by the rego type chuck. L or L' dimension is adjustable by putting sleeve in and out.

HDS5706 Boring and Drill Sleeve
(Both ends, Rego Type Chuck, ~ø7
Semi-long type)



HDS5806 Boring and Drill Sleeve
(Both ends, Rego Type Chuck, ~ø7
Short type)



[Note]

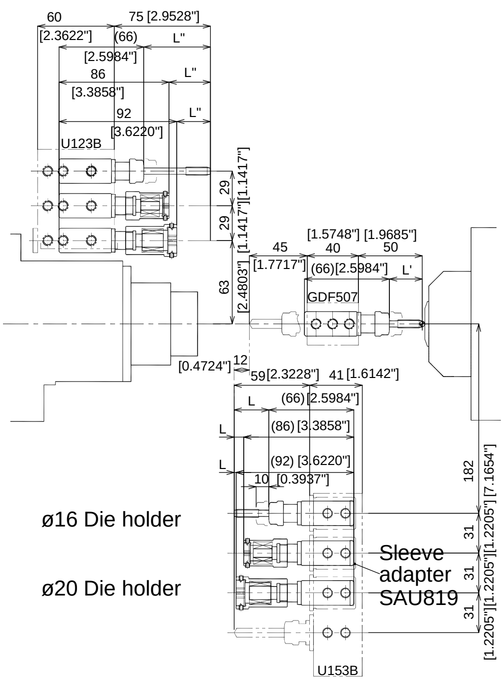
Pay attention to the interference between the back spindle including the opposite sleeves, when mounting this sleeve.

Specification	Description	
Tool holder name	HDS5706	HDS5806
Machine type	VIII, IX, X, XII	VII, IX
Max. chuck dia.	ø7 mm [ø0.2756"]	ø7 mm [ø0.2756"]
Chuck type	ER11, AR11	ER11, AR11
Outer dia. of sleeve	ø19.05 mm [ø3/4"]	ø19.05 mm [ø3/4"]
Sleeve holder	GDF1207	U123B
Provided set screws	–	–

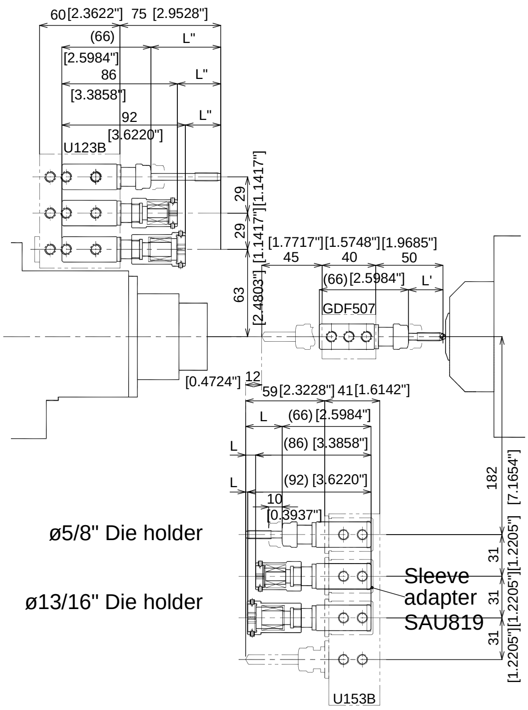
Tapping and Die Sleeve

This sleeve is used to hold the tap and die. Tap can be hold by rego type chuck. The die is hold by die holder mounted in place of cap nut of rego type chuck. L and L' or L" dimension is adjustable by putting sleeve in and out.

BNS407 Tapping and Die Sleeve (Rego Type Chuck, ~ø7) (Die ø16, ø20)



BNS507 Tapping and Die Sleeve (Rego Type Chuck, ~ø7) (Die ø5/8", ø13/16")



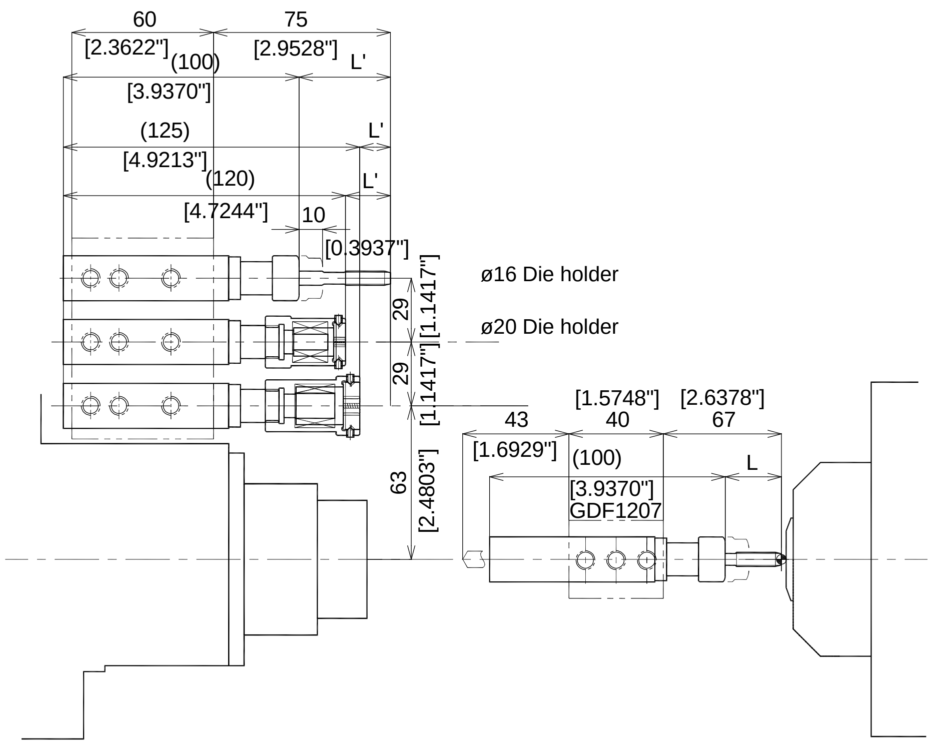
[Note]

- In case of installing the sleeve to the sleeve holder GDF1207, the die holder is not available for use.
- To use this sleeve on U153B machine, SAU819 is required.
- To use this sleeve on U125B, U126B, U128B and U155B machine on U153B, SAU919 is required. Note that, in back machining, this sleeve can be used with T34 or T38 only. It cannot be used with any other tools.

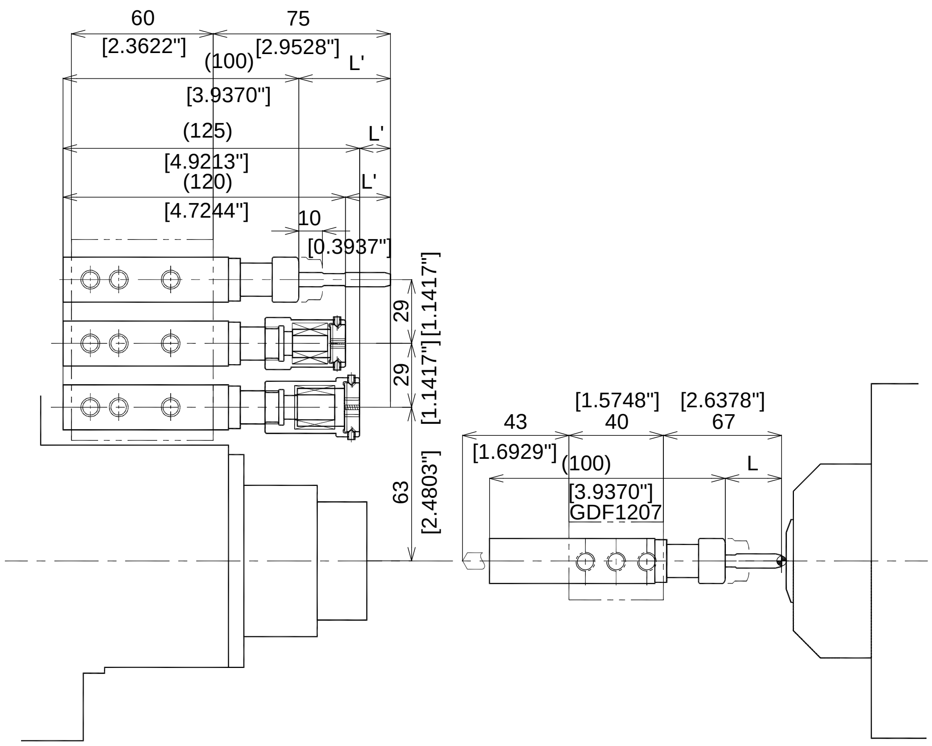
Specification	Description	
Tool holder name	BNS407	BNS507
Machine type	VIII, IX, X, XII	VIII, IX, X, XII
Max. chuck dia.	ø7 mm [ø0.2756"]	ø7 mm [ø0.2756"]
Chuck type	ER11, AR11	ER11, AR11
Die size	ø16×5 mm, ø20×7 mm	ø5/8"×1/4", ø13/16"×1/4"
Outer dia. of sleeve	ø19.05 mm [ø3/4"]	ø19.05 mm [ø3/4"]
Provided set screws	–	–

This sleeve is used to hold the tap and die. Tap can be hold by rego type chuck. The die is hold by die holder mounted in place of cap nut of rego type chuck. L or L' dimension is adjustable by putting sleeve in and out.

VNS406 Tapping and Die Sleeve (Rego Type Chuck, ~ø7) (Die ø16, ø20)



VNS506 Tapping and Die Sleeve (Rego Type Chuck, ~ø7) (Die ø5/8, ø13/16")



[Note]

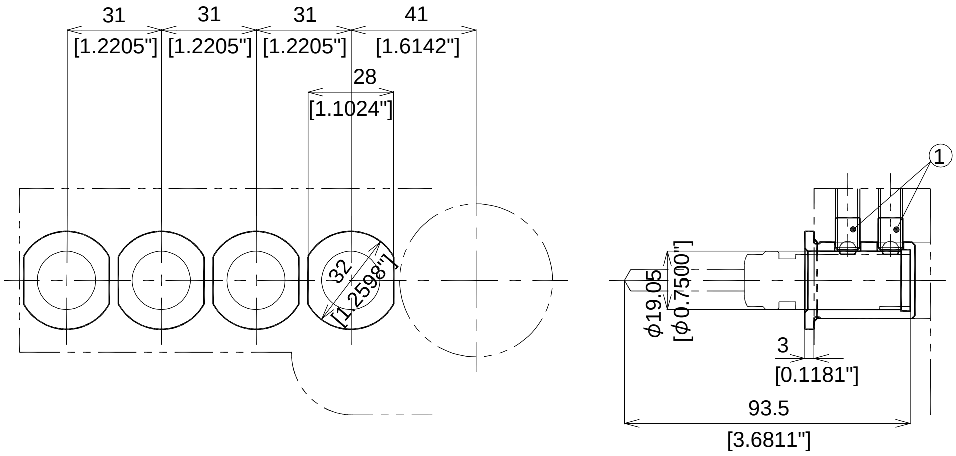
- In case of installing the sleeve to the sleeve holder GD1207, the die holder is not available for use.
- To use this sleeve on U125B, U126B, U128B machine, SAU919 is required. Note that this sleeve may not be mounted depending on extended tool length L.

Specification	Description	
Tool holder name	VNS406	VNS506
Machine type	VIII, IX, X, XII	VIII, IX, X, XII
Max. chuck dia.	ø7 mm [ø0.2756"]	ø7 mm [ø0.2756"]
Chuck type	ER11, AR11	ER11, AR11
Die size	ø16×5 mm, ø20×7 mm	ø5/8"×1/4", ø13/16"×1/4"
Outer dia. of sleeve	ø19.05 mm [ø3/4"]	ø19.05 mm [ø3/4"]
Provided set screws	–	–

Sleeve Adapter

SAU819 Sleeve Adapter (ø19.05 mm)

This adapter is provided for using the shank dia ø19.05 mm sleeve with the back rotary tool driving device U153B.

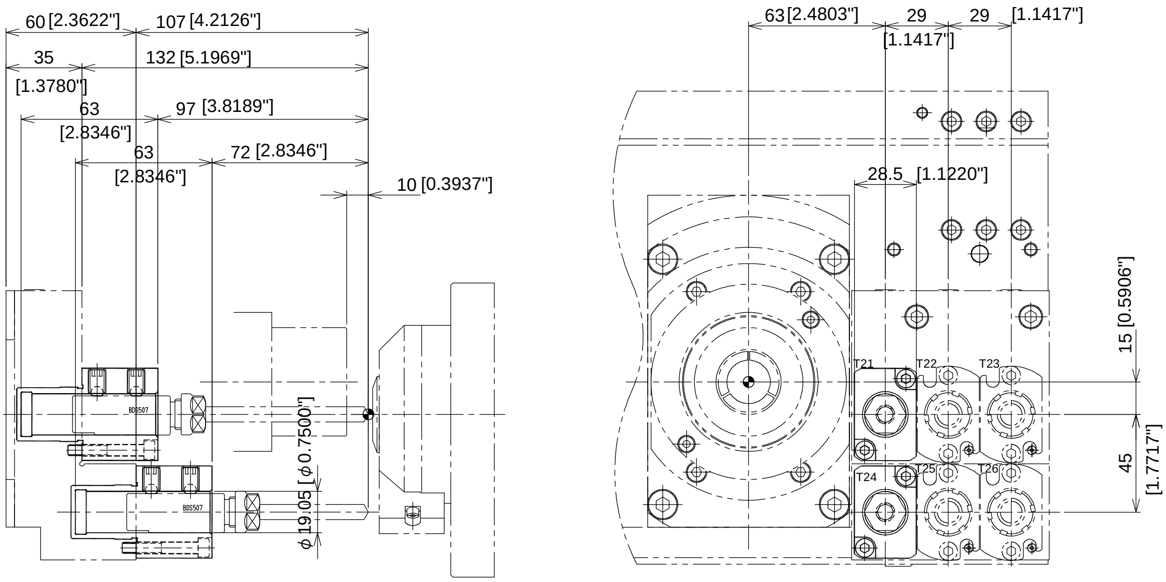


[Note]

To use this adapter, remove the hexagon socket head bolts M8×8 from the upper part of stations of U153B, and fix the sleeve with the provided stopper (A-STOP, ASF812), as shown by ① in the figure above.

SAU919 Sleeve Adapter (ø19.05 mm)

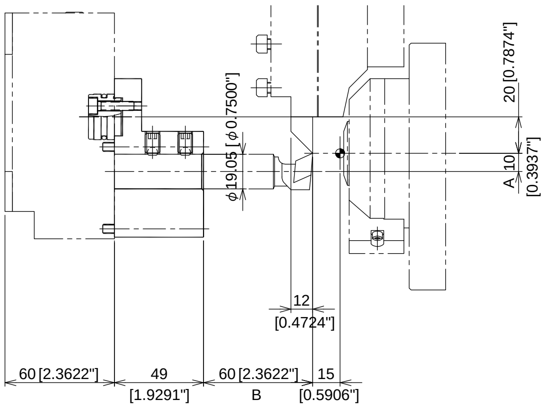
SAU919 is provided for using the sleeve having shank dia ø19.05 mm with U125B, U126B, U128B, or U155B.

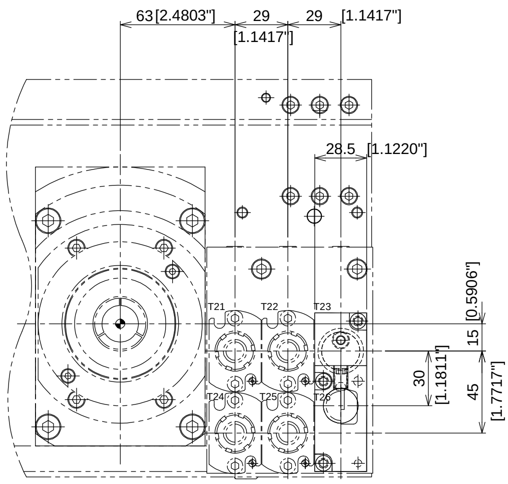


Specification	Description	
Adapter name	SAU819	SAU919
Machine type	VIII, IX	X, XII
Inner dia. of adapter	ø19.05 mm [ø0.7500"]	ø19.05 mm [ø0.7500"]
Outer dia. of adapter	ø25.0 mm [ø0.9843"]	ø25.0 mm [ø0.9843"]
Provided set screws	–	Two M5×35 screws

SAU1019 Sleeve Adapter (ø19.05 mm)

This adapter is provided for pinch turning with the boring sleeve having shank dia Ø19.05 mm.





[Note]

- Use the boring sleeve of which tool nose shift amount is 10 mm (dimension A in the figure) for this adapter. Set the protrusion length of tool at the position 60 mm away from the end face of adapter (dimension B).
- Can be used for U125B and U128B. Cannot be used for U126B.
- In pich turning, use GTF3312 or GTF3313 together with this adpater.
- To set the tool nose upward and perpendicularly in pinch milling, the tool nose position in longitudinal direction differs from that of T20's end-face drilling tool used generally.
  (Tool nose position in pinch milling = T20's end-face drilling tool position + 2 mm)
  Strict care must be taken when setting a tool or changing a program.

Specification	Description
Adapter name	SAU1019
Machine type	X, XII
Inner dia. of adapter	Ø19.05 mm [ø0.7500"]
Outer dia. of adapter	Ø25.0 mm [Ø0.9843"]
Provided set screws	One M5×16 screw and two M5×50 screws

Notes on using the end face drilling sleeve holder to be mounted on the vertical holder

The dimension of tools is uneven. If the core of tool is not aligned with that of workpiece, perform adjustment using an alignment ring. (See the Operator's Manual *<7.8.5 Adjustment using alignment ring>* .)

(Blank page)

Product Code	C-L220E XII
Document Code	2E1-1701
Mfg. No.	L220E/0001~
Issue Date	2014.1

21. Machine Structure

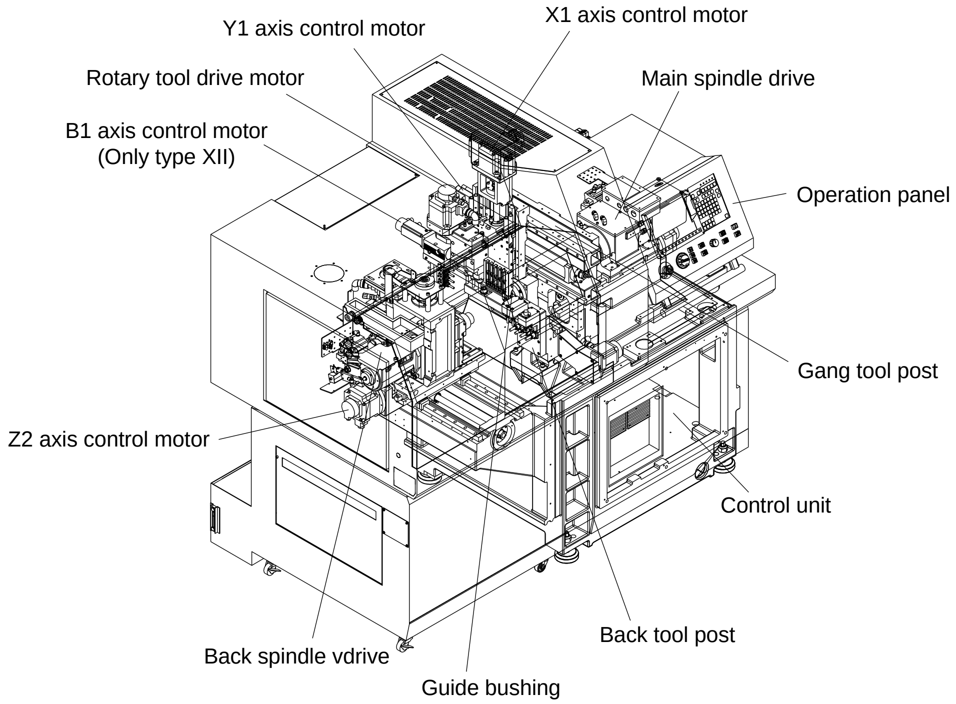
21.1 Machine Overview

The user's cooperation is indispensable for maintaining the machine in good onditions and for recovering the machine immediately if trouble occurs.

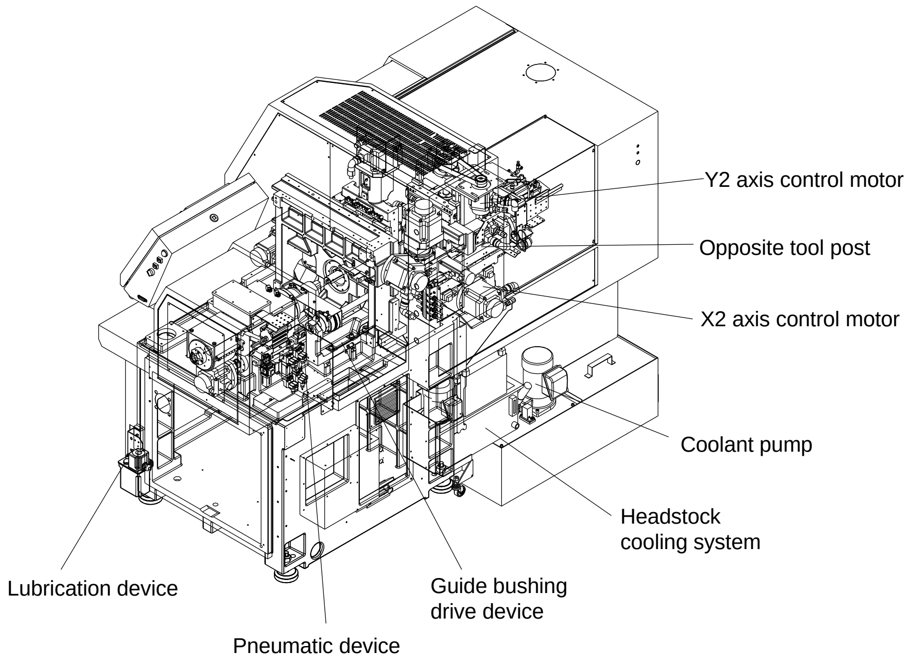
To provide an mechanical overview, this chapter illustrates the major mechanisms and components of the machine. See < *Wiring Diagrams*> for electric devices.

21.2 Main Components of the Machine

Shown below are the front and rear views of the machine with the names of its main components.



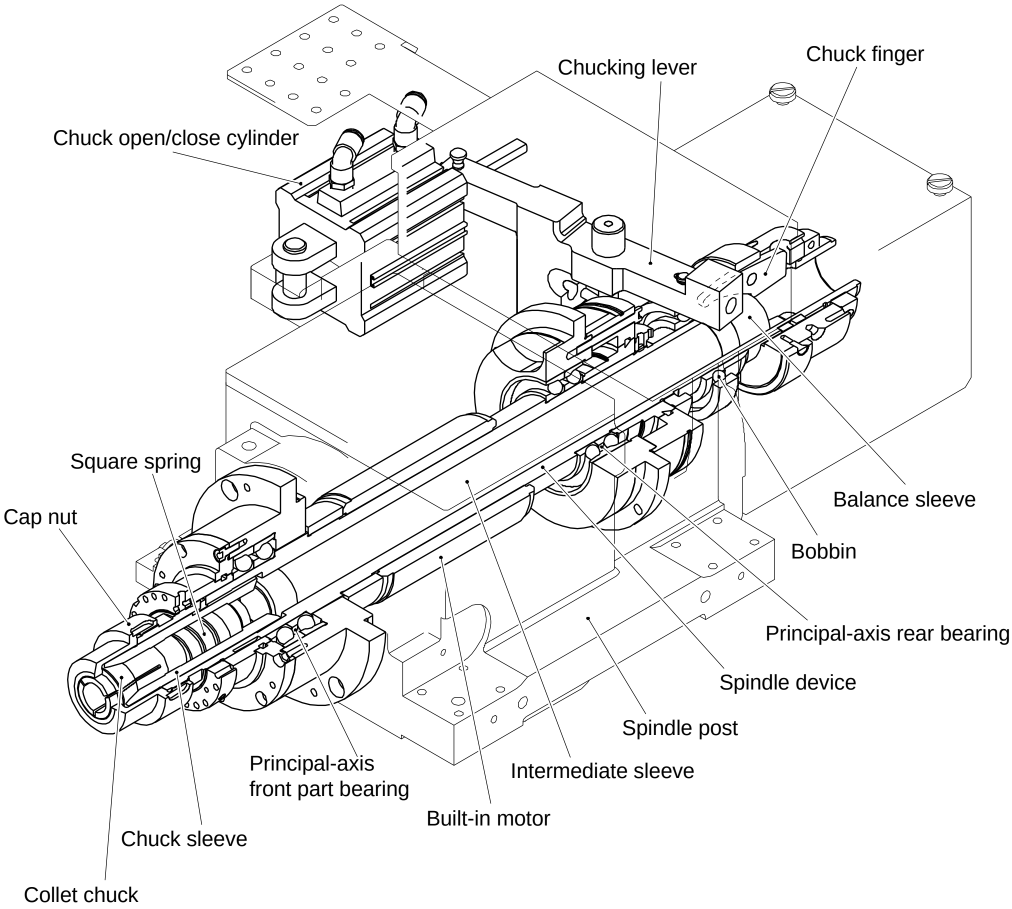
Front view of machine



Rear view of machine

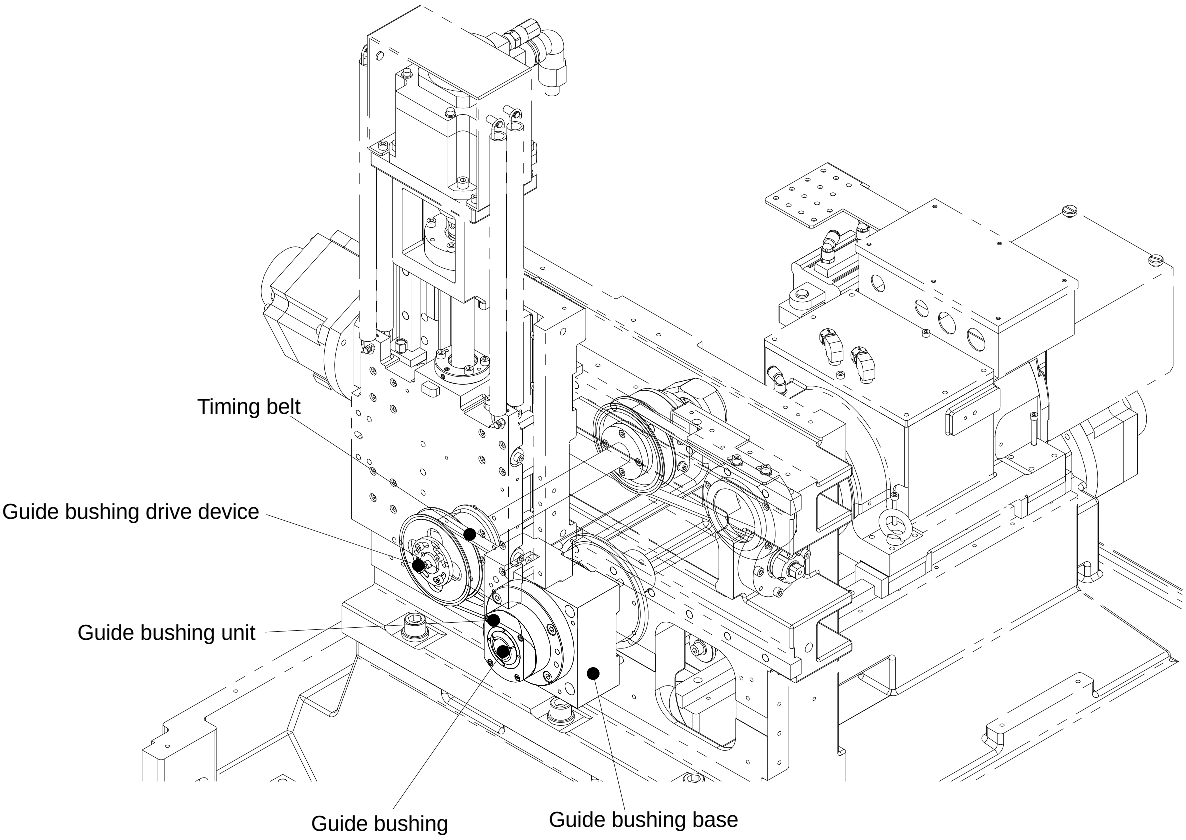
21.3 Spindle Device

Shown below are the names of the components of the spindle device.



21.4 Guide Bushing and Guide Bushing Drive Device

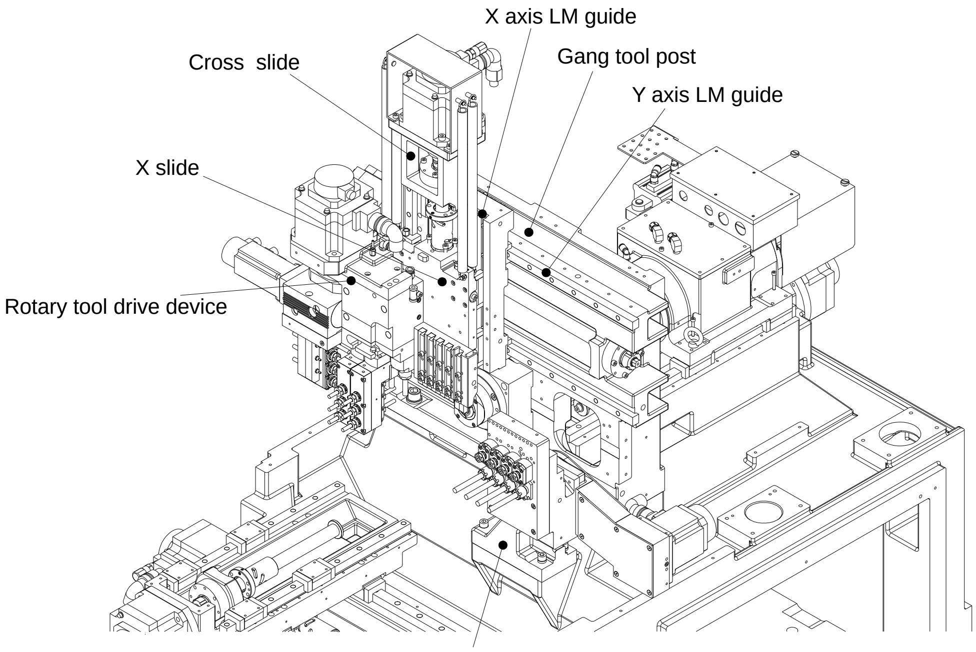
Shown below are the names of the components of the guide bushing and the guide bushing drive device.



Rotary guide bushing

21.5 Tool Post

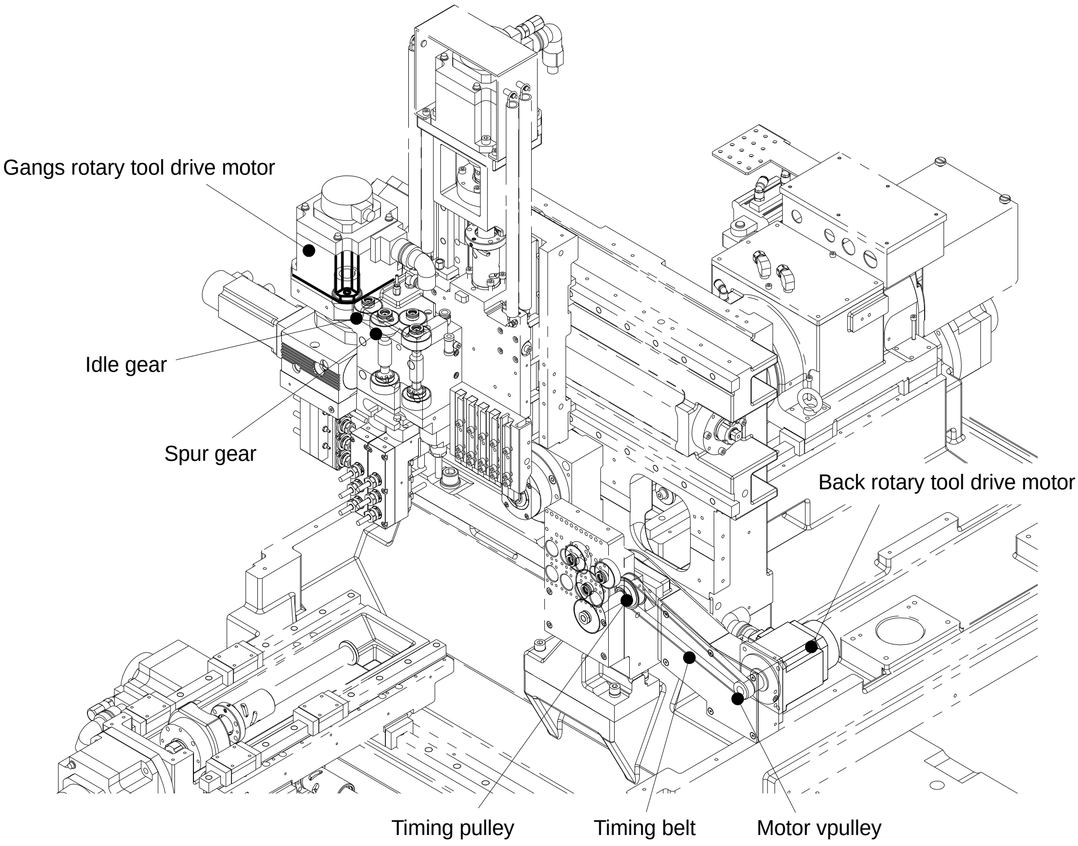
Shown below is the external view of the tool post with the names of its components.



Tool post (L20 Type XII)

21.6 Rotary Tool Drive Device

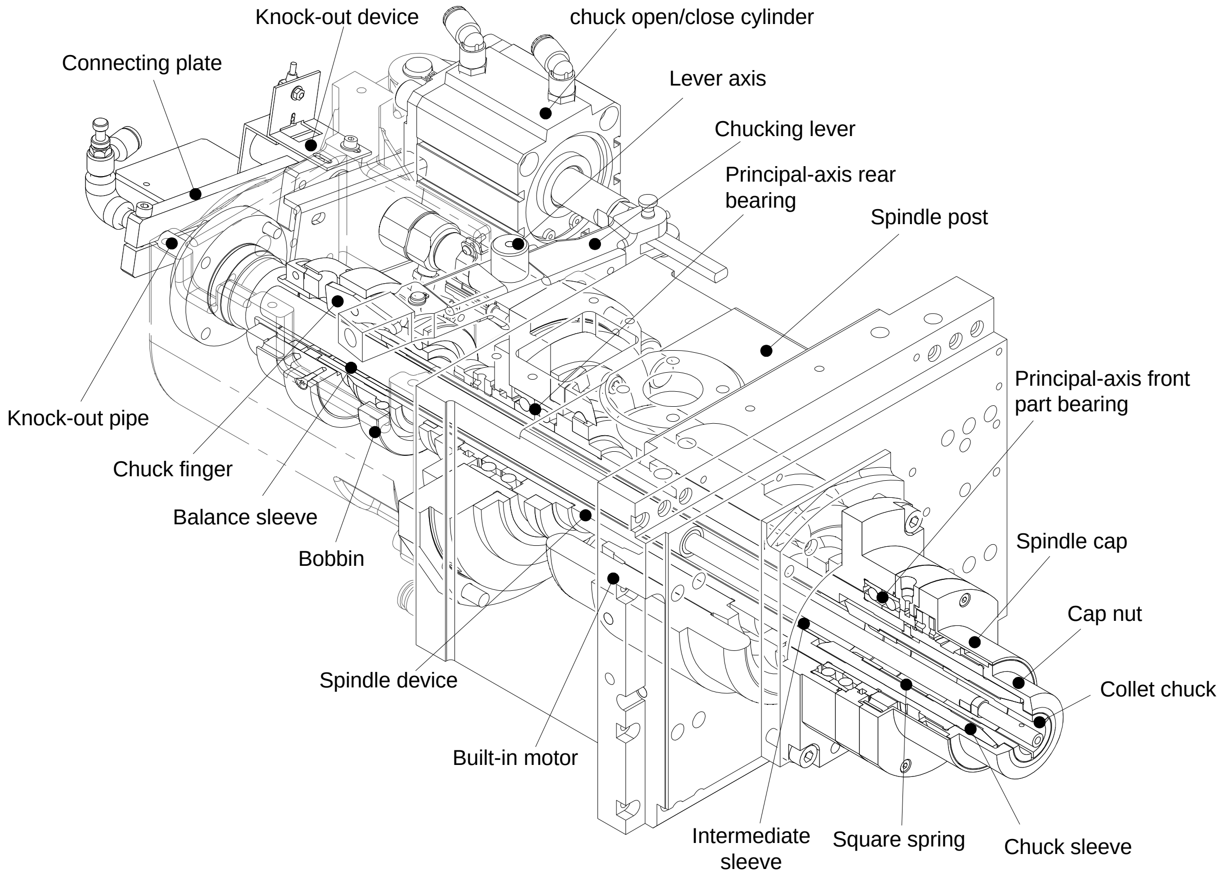
Shown below is the external view of the rotary tool drive device with the names of its components.



Rotary tool drive device (Type XII)

21.7 Back Spindle Device

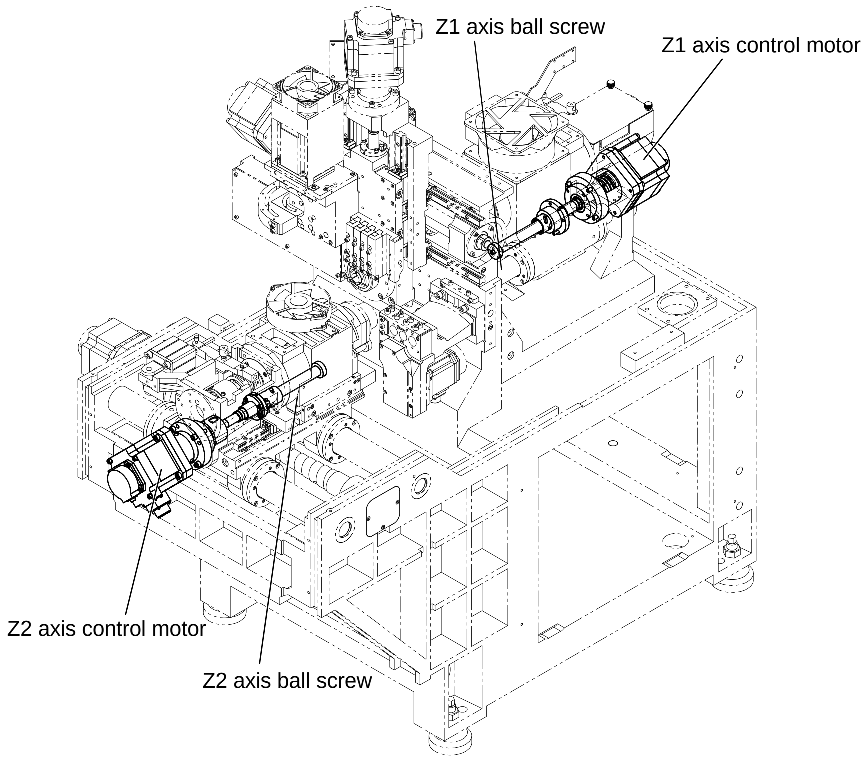
Shown below are the names of the components of the back spindle device.



Back Spindle Device (L20 Type XII)

21.8 Z-Axis Feed Mechanism

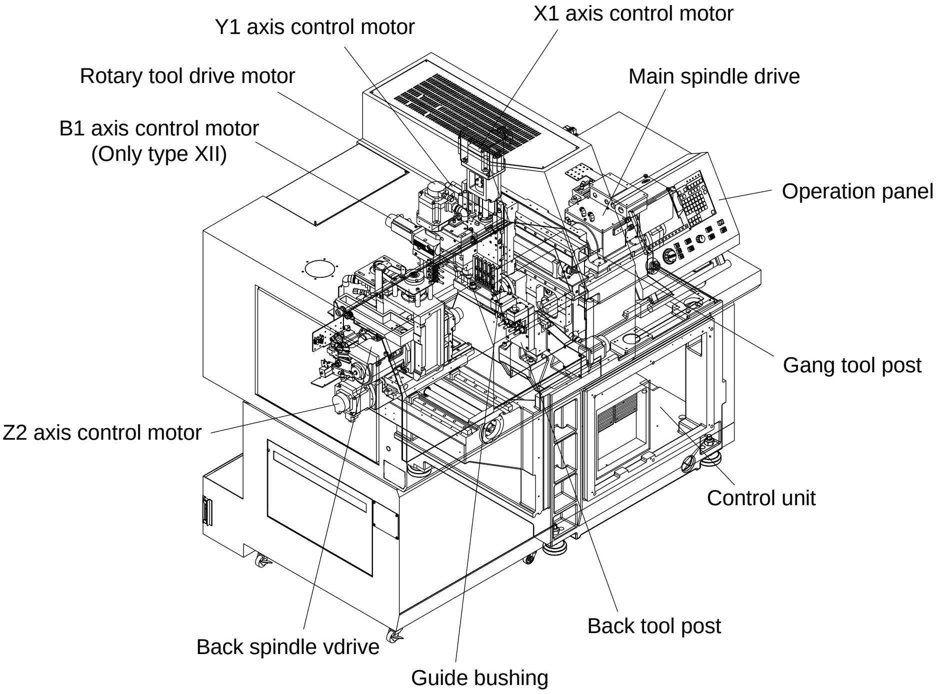
Shown below is the perspective view of the Z-axis feed mechanism with the names of its components.



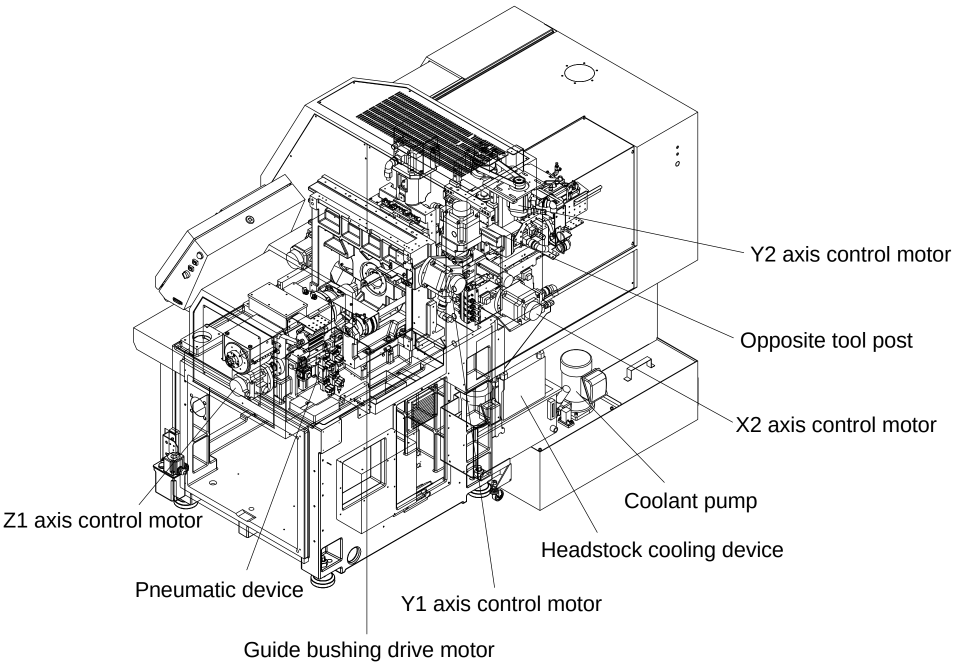
Z-axis feed mechanism (L20 Type XII)

21.9 X, Y-Axis Feed Mechanism

Shown below is the perspective view of the X,Y-axis feed mechanism with the names of its components.



X, Y-axis feed mechanism front view of machine (L20 type XII)

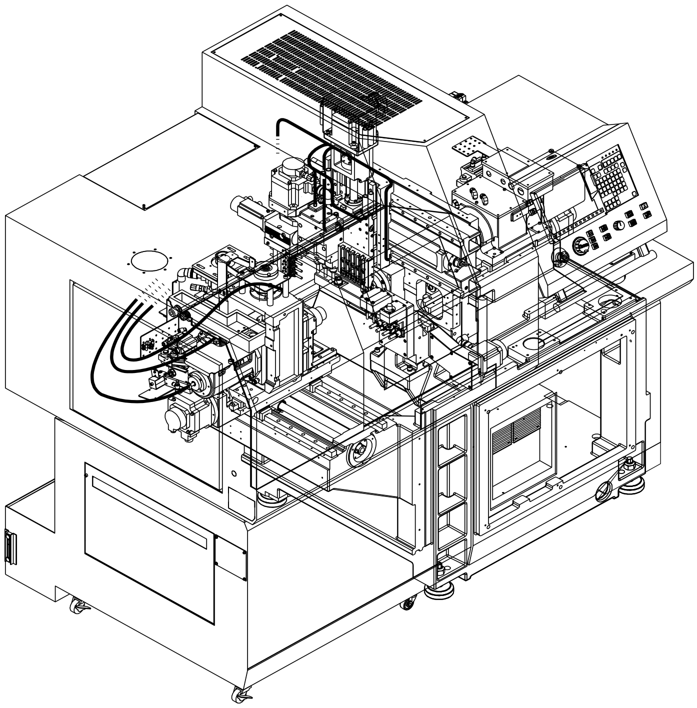


X, Y-axis feed mechanism rear view of machine (L20 type XII)

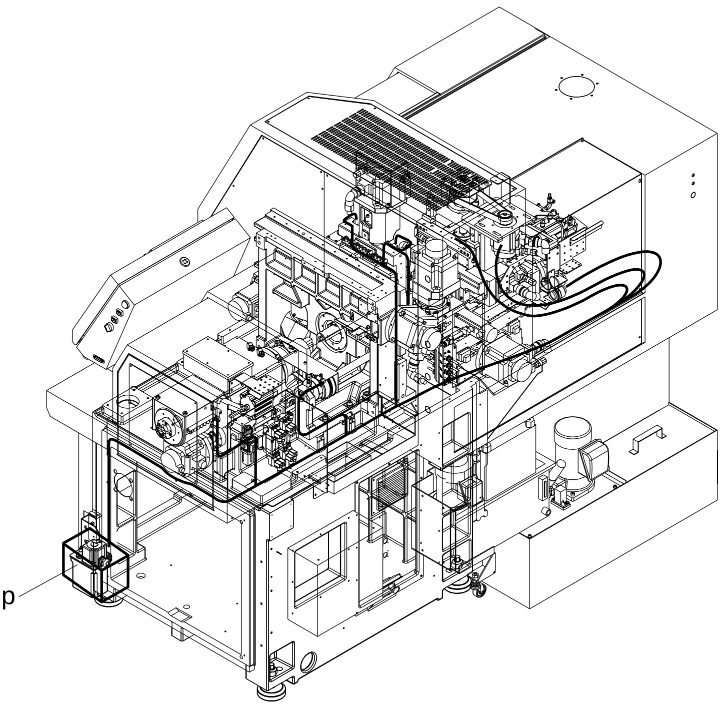
21.10 Lubricating Oil Device

Shown below are the front and rear views of the Lubricating oil device, and the Lubricating oil piping diagram.

■ Lubricating oil piping diagram

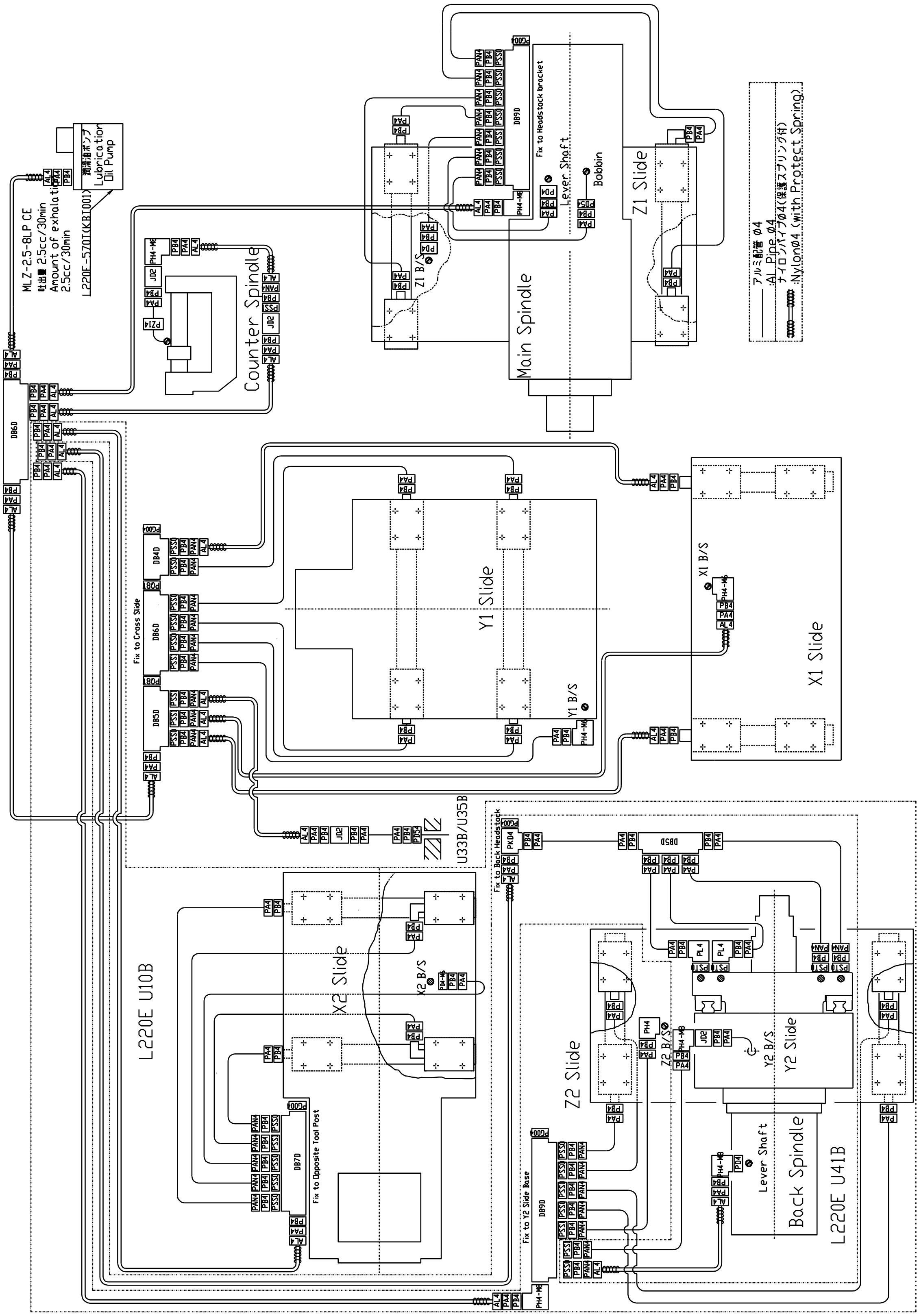


Front view of machine



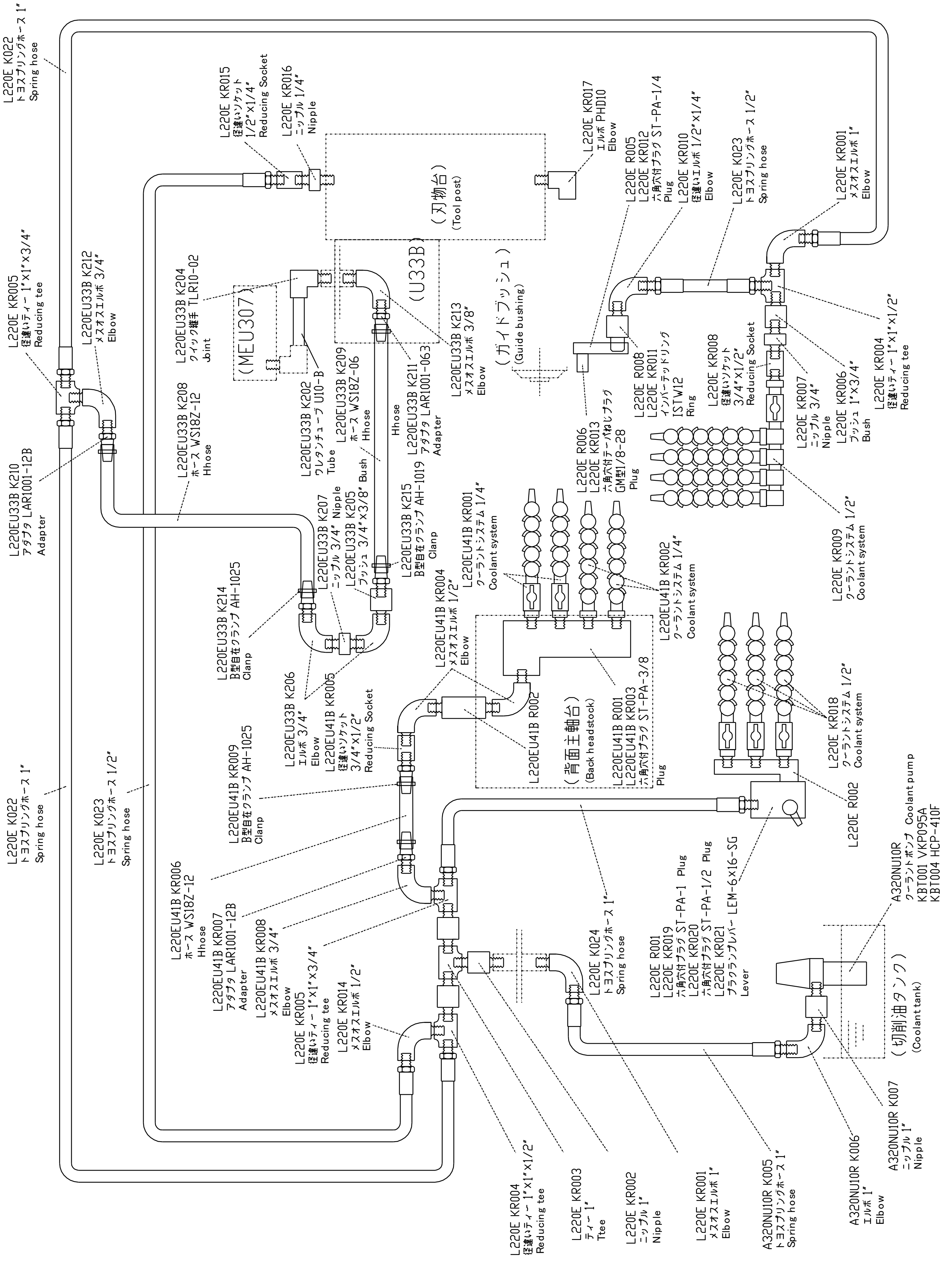
Rear view of machine

■ Lubricating oil circuit diagram



21.11Coolant Device

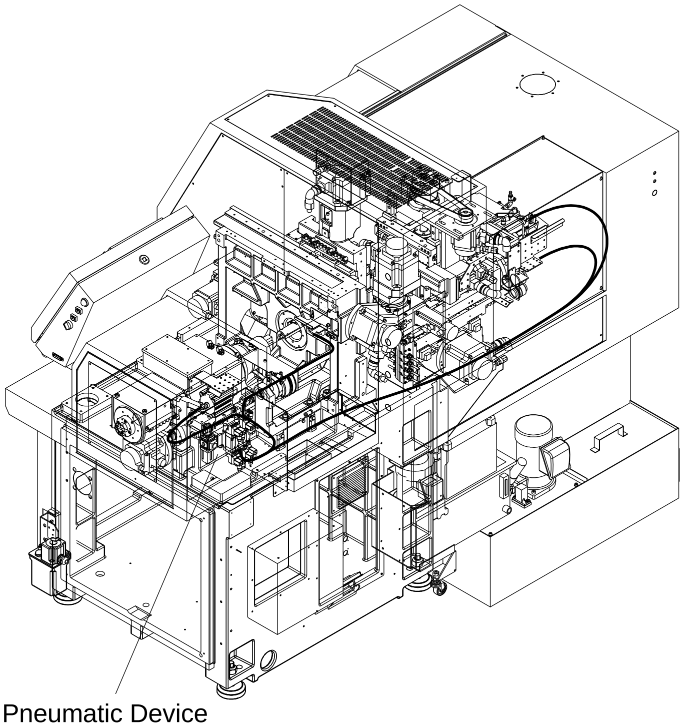
Shown below are coolant piping diagrams.



21.12Pneumatic Device

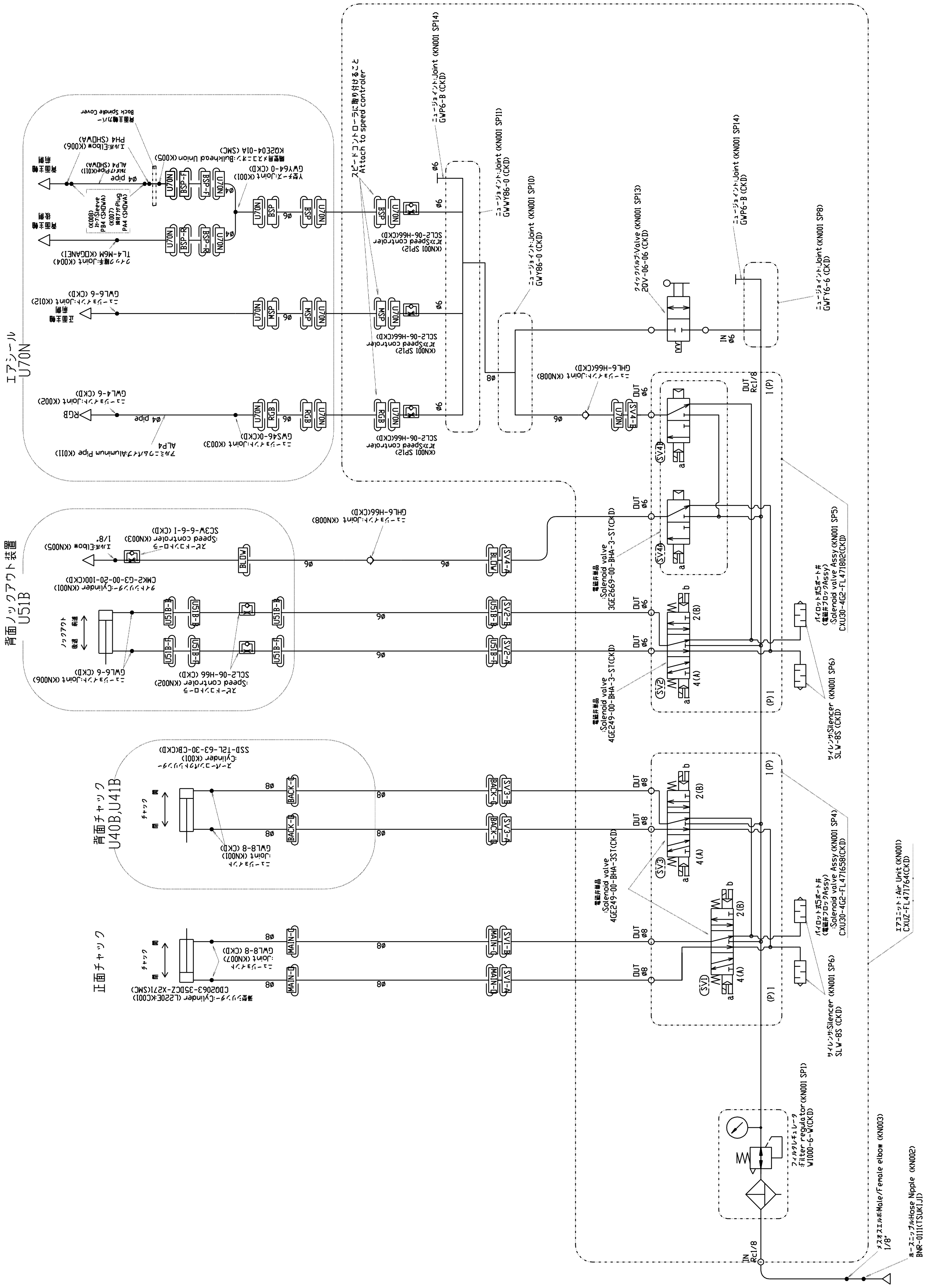
Shown below are the pneumatic device piping diagram and pneumatic piping diagram.

■ Pneumatic piping diagram



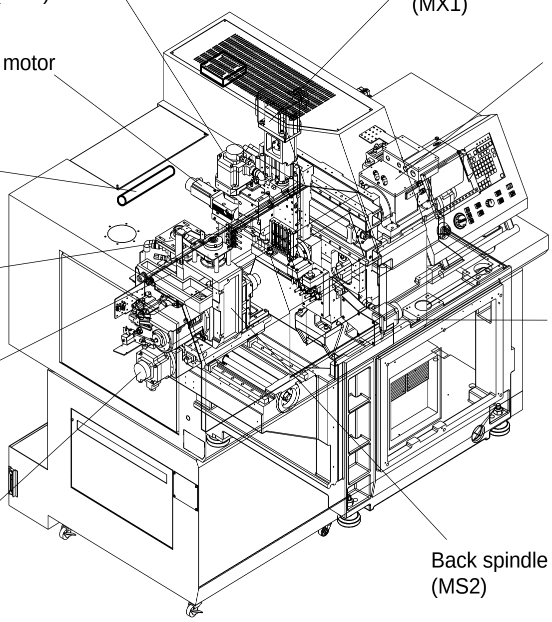
Rear view of machine

■ Pneumatic circuit diagram



21.13 Electric Device

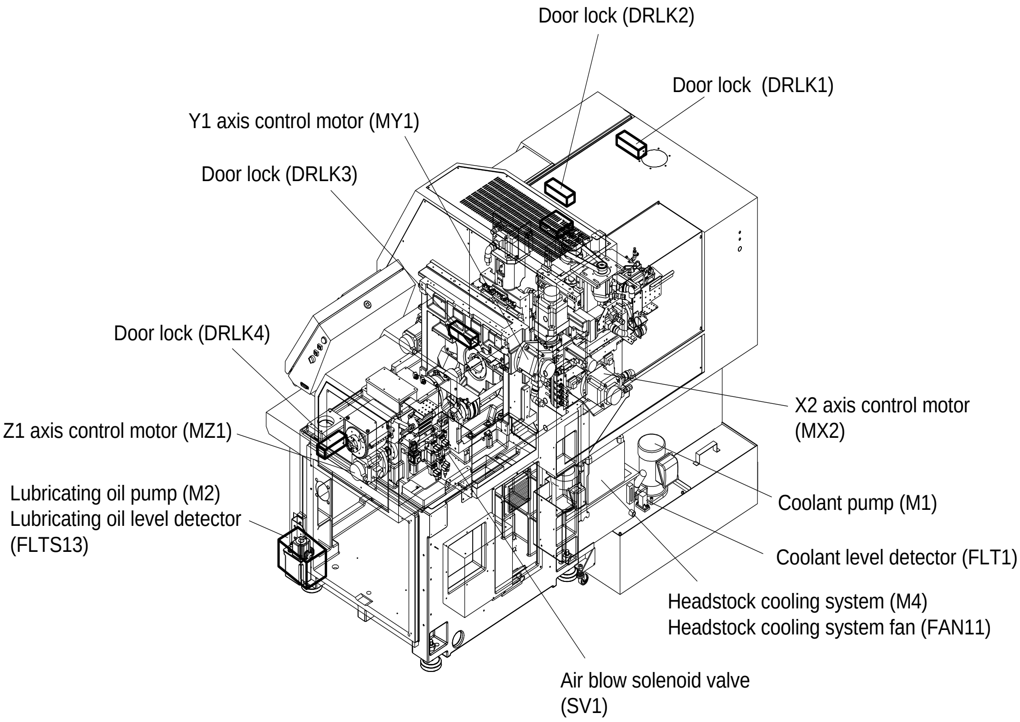
Shown below are the names and locations of electric components (motors, switches, and sensors) of the L20E and L20X.



Electric Device (front view) (L20 Type XII)

Parameter axis name list

Item	Parameter axis name
Main spindle drive motor (MS1)	S1
Back spindle drive motor (MS2)	S2
Rotary tool drive motor (MS3)	S3
Front rotary tool drive motor (MS4)	S4 (option)
Back rotary tool drive motor (MS5)	S5 (Type X and XII: standard)
X1 axis control motor (MX1)	X1
Z1 axis control motor (MZ1)	Z1
Y1 axis control motor (MY1)	Y1
X2 axis control motor (MX2)	X2
Z2 axis control motor (MZ2)	Z2
Y2 axis control motor (MY2)	Y2 (Only type X and XII)
Gang tool B axis motor (MA6)	A6 (Only type XII)



Electric Device (rear view) (L20 Type XII)

Product Code	C-L220E XII
Document Code	3E1-2100
Mfg. No.	L220E/0001 ~
Issue Date	2013.10

22. Periodical Check

22.1 Periodical Check

The machine must be checked at regular intervals regardless of presence or absence of a machine fault. Periodical checks enable you to predict a machine fault or detect it at an early stage. Consequently, you can minimize the loss of production.

This section explains periodical checks (daily, monthly, and semi-annually).

CAUTION

Be sure to make periodical checks. Otherwise, the machine may seriously be damaged.

WARNING

Be sure to conduct any work in emergency stop state other than operation check. Working with the machine during operation causes an accident which could result in death or serious personal injury.

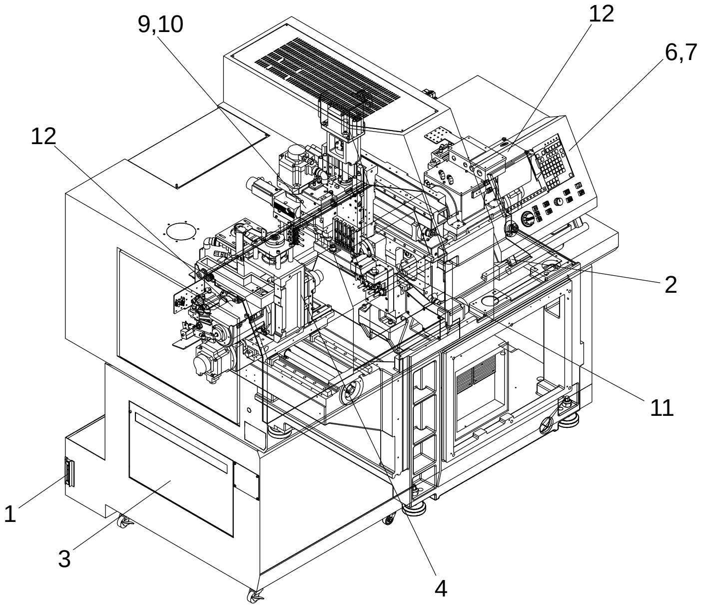
22.1.1 Daily Check

Daily check is required every day before the machine is operated or while the machine is operating.

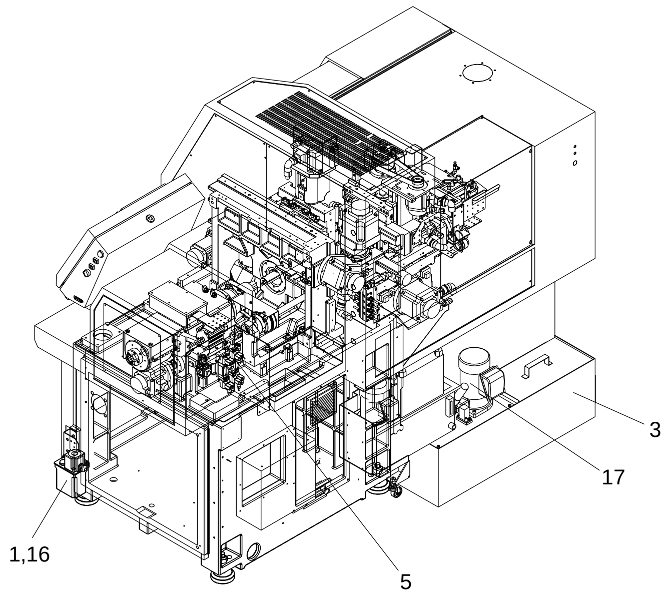
Check the parts indicated by numbers in figures below. The numbers correspond to those in the daily check item tables below. In addition to the check items, you can also check other items (e.g., sound, temperature, scratch, and odor) using your five senses during operation. You must know how the machine operates normally to predict a machine fault using your five senses. Prediction of a machine fault in this way is considered as the most effective method of daily check.

Your attitude to the machine as described above is very important when you maintain the machine.

■ Daily check points



Front view of machine



Rear view of machine

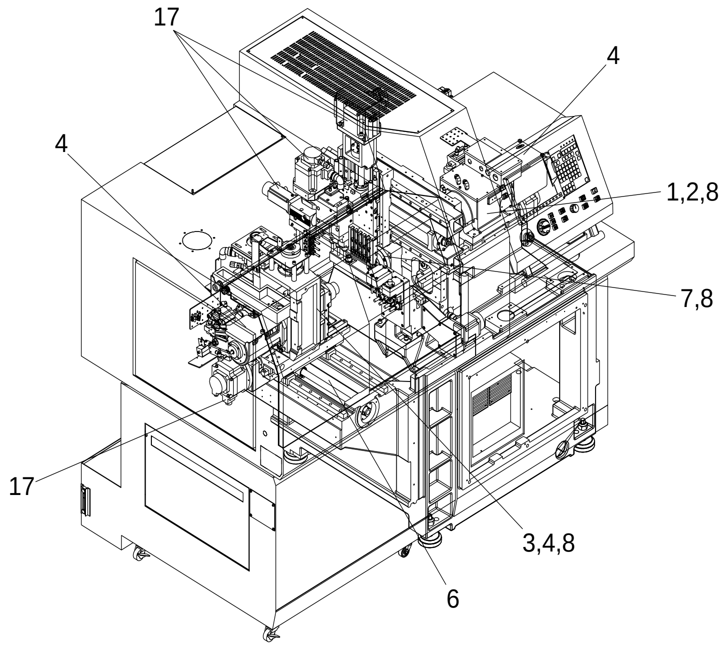
■ Daily check items

Type	No.	Check item	Check method or point
Before power-on	1	Coolant level and lubricant level.	Check the coolant level gage. The Lubricant level must be at least as a quarter of the transparent tank.
	2	Check if the tools and measuring tools are back in place.	Confirm that no tool or measuring equipment is left on a moving section.
	3	Check if chips are removed.	Remove chips from the coolant tank as soon as possible. Take out the chip receiver box and remove accumulated chips. Be sure to remove chips according to <*22.1.4 Removing Chips*>.
	4	Check if the cutting room door opens and closes normally.	Open and close the cutting room door to check if the door closes firmly.
	5	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 power-on	6	Check if an alarm lamp on the operation panel is on.	Confirm that the LCD screen shows no alarm and that no alarm lamp on the machine operation panel is on.
	7	Check if lamps on the operation panel are in normal state.	Check if the push-button switches and lamps are normal.
After setup	8	Check if the tool is clamped correctly.	Confirm that the screws securing the tool to the holder and the holder to the tool post are firmly tightened.
	9	Confirm that the tool is not damaged.	Check the tool edge.
	10	Check if the guide bushing and the chuck are adjusted correctly.	Check the clearance between the material and the guide bushing. Confirm that the collet chuck holds the material smoothly.
	11	Check if the sliding part of the bobbin is lubricated correctly.	Apply lubricant manually if the surface is dry.
During operation	12	Confirm that there is no abnormal noise.	Check if the rotary parts and slides sound normal.
	13	Confirm that the machine is in normal state.	Check the machine using your five senses (e.g., noise, temperature, scratch, and odor).
	14	Check if the lubrication pump is working.	Normal if the motor shaft in the transparent tank takes 60 min to make a full rotation
	15	Check if the coolant pump is working.	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.

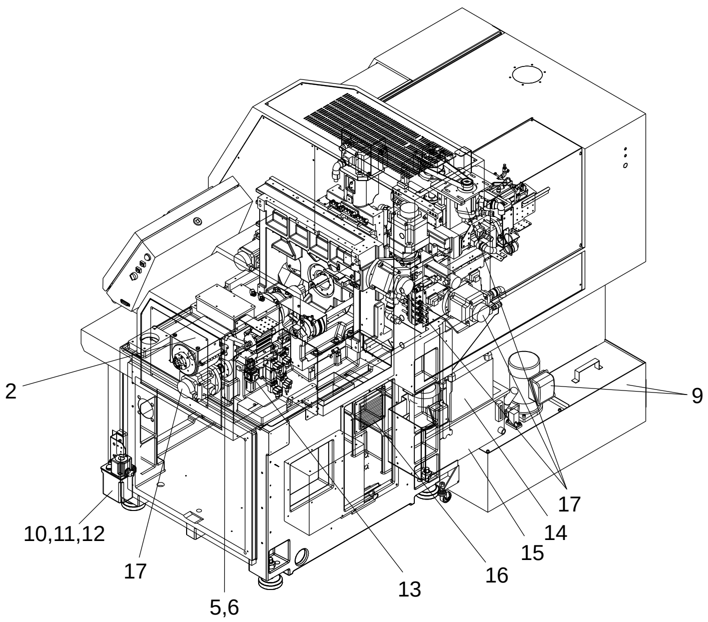
22.1.2 Monthly Check

Monthly check is required for predicting a machine fault (or detecting it at an early stage) which may be missed through daily check. Must be carried out.

■ Monthly check points

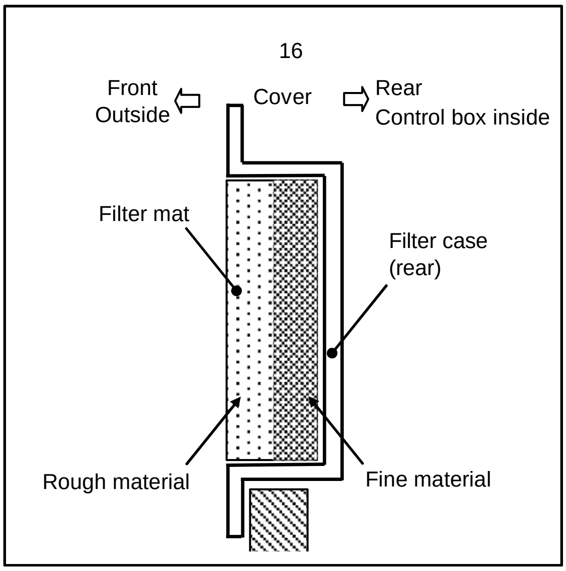


Front view of machine

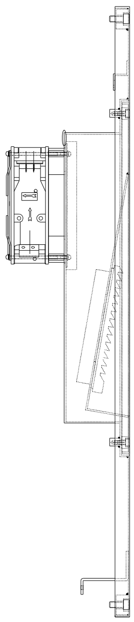


Rear view of machine

Replacement of filter mat



Filter mat assy



Detailed drawing for monthly check item No. 16

■ Monthly check items

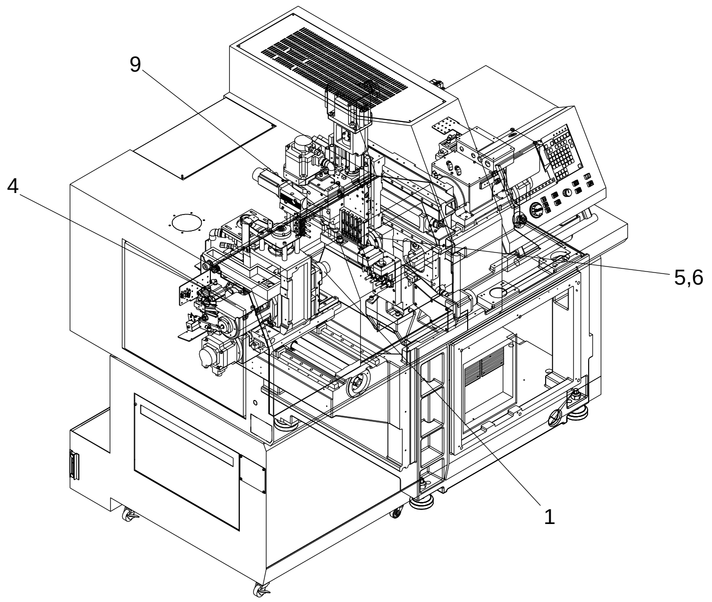
Type	No.	Check item	Check method	Criteria and action
Spindle	1	Heating of the bearing	Rotate the spindle at 3,000 min^{-1} for 30 min.	Touch the housing by hands. Normal if the housing is only warm.
	2	Wear of the chuck finger	Visually check the chuck fingers.	If the chuck finger is remarkably worn out, replace the chuck finger.
Back spindle	3	Heating of the bearing	Rotate the spindle at 3,000 min^{-1} for 30 min.	Touch the housing by hands. Normal if the housing is only warm.
	4	Wear of the chuck finger	Visually check the chuck fingers.	If the chuck finger is remarkably worn out, replace the chuck finger.
Headstock and tool post feed structure	5	Lubrication of the ball screw	Visual check	Clean the screw shaft if it is stained.
	6	Condition of slide cover of feed axis	Visually check the slide cover.	Replace them if they are damaged.
Guide bushing device	7	Inner hole of the guide bush sleeve	Visual check and finger touch	No scratch or burr
	8	Chuck and guide bushing statues	Dismount the collect chuck, remove the springs, and then dismount the chuck sleeve, intermediate sleeve, and balance sleeve.	Clean the inside of the spindle and the removed parts. Similarly, clean the guide bushing and the removed parts.
Coolant device	9	Check for chip accumulation in the areas around the coolant pump and in the coolant tank.	Remove the partition from the side on which the coolant pump is mounted. See <*22.1.4 Removing Chips*>. Draw the coolant tank and check for chip accumulation in the tank.	Remove chip accumulation from the coolant tank and the areas around and under the coolant pump.
Lubrication device	10	Discharge amount of oil from the pump unit	Check how fast the manual handle drops.	2.5 cc in 60 min Check the piping if the manual handle drops too fast.
	11	Parts for lubrication piping and oil leakage	Visual check of the piping and tubes	Retighten the loosened parts if it is oily.
	12	Check how oil is discharged.	Determine if oil replenishment is necessary.	Check the motor shaft revolution.
Filter of pneumatic device	13	Confirm that the filter is not stained or clogged.	Visual check and finger touch	Replace the filter with neutral detergent if it is stained or clogged.
Cooling device	14	Check if each cooling fan is working normally	A cooling fan is provided for the spindle cooling device and each amplifier in the electric box. Place your palm near the air outlet of each fan unit to see if the fan is operating normally.	If a fan is not working normally, replace it.
	15	Level of cooling oil for headstock	Check the oil level gauge on headstock cooling system	If the oil is insufficient, pour clean oil into the tank until the tank is at full capacity.

Type	No.	Check item	Check method	Criteria and action
Control unit filter	16	• Grimed and clogged state of the filter • Grimed and clogged state of the oil pan	• Check visually and also by touching • Check the oil pan visually.	• If the filter is grimed and clogged, wash the filter with neutral detergent. Note) When washing the filter use only neutral detergent. If other cleaning agent is used, the filter may be damaged. • If the oil pan is filled with the waste oil, clean it. Note) If the oil pan is grimed, the filter may be clogged. Check the filter.
Checking the motor wiring	17	Connector status of each motor	Check if the connectors of each motor are firmly tightened.	Retighten loose connectors if any.
B1 axis revolving assembly (2M9 and 2M12 model)	18	Lubricant level	Visually check the oil gauge of the B1 axis revolving assembly.	If there is not enough oil, add oil.

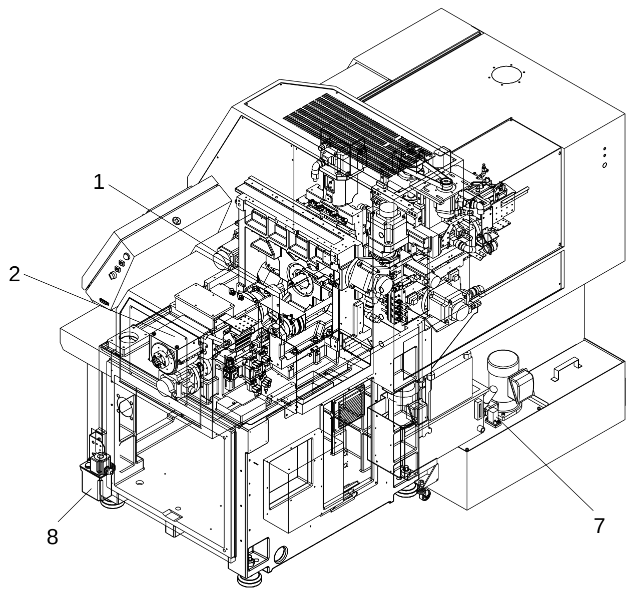
22.1.3 Semi-Annual Check

Semi-annual check is carried out for parts which seldom fail or for determining when to replace consumable parts. Do not neglect the check items. Otherwise, the machine may seriously be damaged.

■ Semi-annual check parts



Front view of machine



Rear view of machine

■ Semi-annual check items

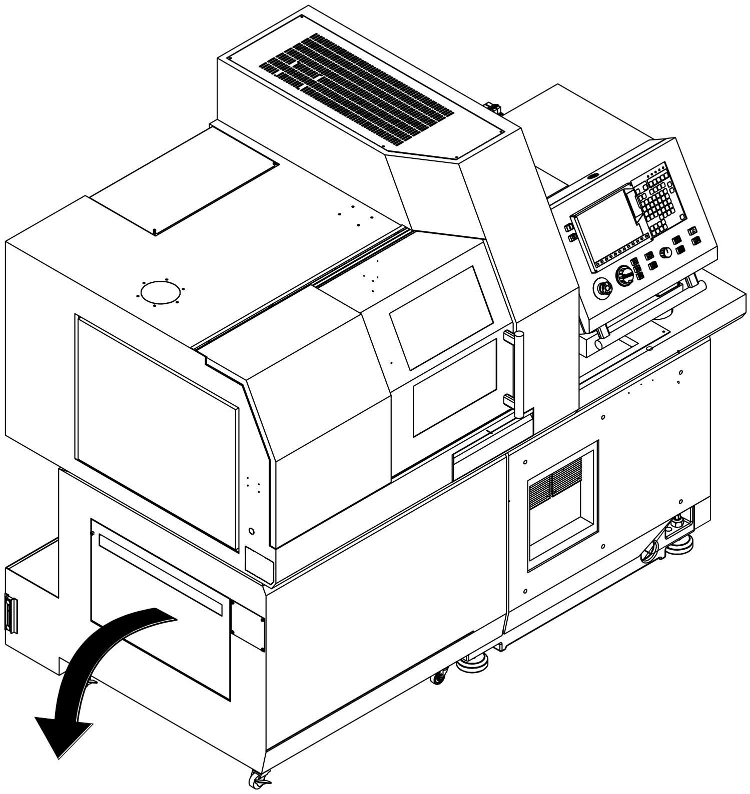
Type	No.	Check item	Check method	Criteria and action
Spindle	1	Deflection of the spindle straight hole	Attach the dial indicator to the hole into which the spindle chuck sleeve is inserted, and give a full rotation to the spindle softly.	Normal if the indicator shows a value which is within the tolerance of the inspection table provided with the machine (TIR0.005).
	2	Wear of the chuck bobbin	Visual check of the outer circumference	The chuck bobbin can be used if not severely worn out.
Back spindle	3	Deflection of the back spindle straight hole	Attach the dial indicator to the hole into which the back spindle chuck sleeve is inserted, and give a full rotation to the spindle softly.	Normal if the indicator shows a value which is within the tolerance of the inspection table provided with the machine (TIR0.005).
	4	Wear of the chuck bobbin	Visual check of the outer circumference	The chuck bobbin can be used if not severely worn out.
Guide bushing device	5	Heating of the bearing, abnormal noise, and thrust rattling	• Rotate the spindle at 3,000 min^{-1} for 30 min. • Push and pull the sleeve to the thrust direction by hand.	• Touch the housing. Take appropriate action if it is too hot or you hear abnormal noise during rotation. • Check if rattling condition is observed.
	6	Tension on and wear of 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.
Coolant device	7	Float switch in the tank	Remove oil from the tank.	Normal if the alarm lamp on the operation panel goes on after one cycle
Lubrication device	8	Float switch of the pump unit	Remove oil from the pump unit tank.	Normal if the alarm lamp on the operation panel goes on after one cycle
Rotary tool spindle	9	Abnormal noise and thrust rattling	• Rotate the tool spindle at 2000 min^{-1} for 30 min. • Push and pull the spindle to the thrust direction by hand.	• Check if the rotary spindle generates abnormal noise during the rotation. • Check if rattling condition is observed.

22.1.4 Removing Chips

If you are negligent to remove accumulated chips from coolant tank, it causes an unexpected trouble such as coolant overflow. This section explains how to remove chips from chip chute and to clean the coolant tank.

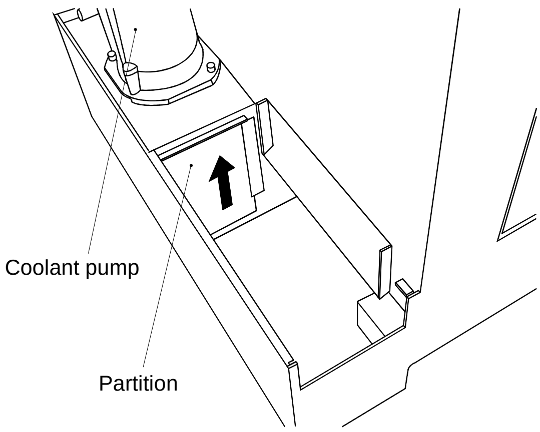
Provide an area large enough to pull out the chip chute from the machine.

1. Remove larger chips. Open the chip outlet door provided on the lower part of the left side of the machine, rake 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. When the machine is used for a long time period, very fine chips may accumulate at the bottom of the coolant tank.
Draw the coolant tank, remove the chip-accumulated floor plate, and clean it as required.
Be sure to disconnect the wiring connectors and the piping joints before separating the coolant tank from the machine.

WARNING

Never discharge coolant for machining with fine chips accumulated at the bottom of the coolant tank.
Chips in coolant may cause a fire or damage to the coolant pump.

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.

CAUTION

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 may result in death or serious personal injury due to an electric shock.

22.2 Oil and Grease Supply

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 A20 machine.

22.2.1 Lubrication list

For details of lubrications, see *<22.2.2 Precautions on Lubrication and Replacement of Lubricant>*.

Lubricating Position	Frequency	Volume	Lubrication type and name	Status of oils and greases
Slide lubricating oil pump	a	0.8 liter	Mobil Vactra Oil No.2 or its equivalent (Viscosity: ISO VG68)	Liquid
Coolant tank	a	Depends on tank capacity	See *<22.2.3 Coolant>*	Liquid
Tooling spur gear	b	Adequate	Mobilith SHC220 or its equivalent (Grease of wide application temperature ranges for rolling bearing) (DIN KPHC2N-40)	semi-solid (grease)
Bobbin sliding portion	a	Adequate	Mobilith SHC220 or its equivalent (Grease of wide application temperature ranges for rolling bearing)	semi-solid (grease)
Oil air lubricating device	a	1.8 liter	Mobil Vactra Oil No.2 or its equivalent (Viscosity: ISO VG68)	Liquid
Spindle cooling device	b	15 liter (Tank capacity 8 liter)	Mobil Velocite Oil No.3 or its equivalent (Viscosity: ISO VG02)	Liquid
B1 axis revolving assembly	b	0.2 liter	Mobilith SHC220 or its equivalent (SHC629)	Liquid

- Be sure to lubricate the parts 'a' as required during the daily check and the parts 'b' as required during the monthly check, respectively.
- Contact the manufacturer for equivalent to recommended lubricant.

[Note]

- 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 2.5 cc/60min.
- The spindle bearing is 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.
- Dispose of the waste oil according to all national laws and regulations.
- Applying grease to the tool spindle gear and tools excessively causes heat generation.
- When replacing the quill-drive tool, apply a proper amount of grease to the teeth portion of gear before inserting the tool.

22.2.2 Precautions on Lubrication and Replacement of Lubricant

■ Lubricant for slides

- Fill the lubricant tank with new oil when the lubricant level for the slides is nearing the lower limit.
  There is no need to replace all the oil if you supply the oil correctly.

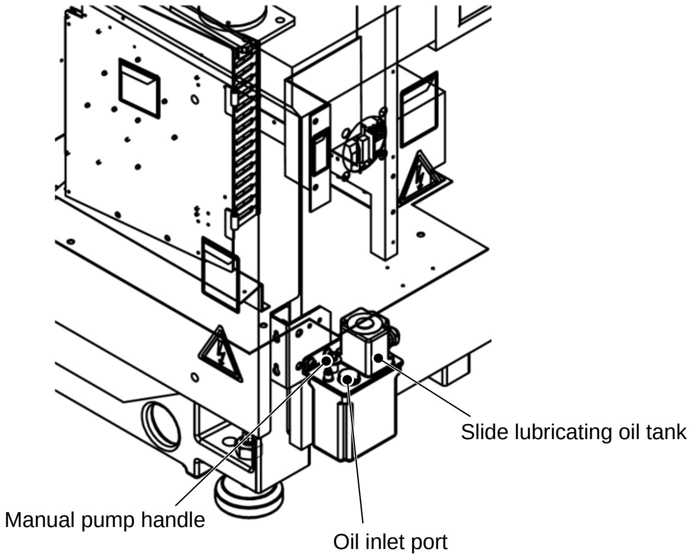
  If the machine has not been used for a long time, conduct the following sequence before starting the machine:
- Pull up the pump manual handle and release it. Repeat this operation until the lubricant is well supplied to each part.

[Note]

Ask the manufacturer for lubricant for slide equivalent to Exxon Mobil Vactra Oil No. 2.

CAUTION

Never push down the pump manual handle.
Otherwise, the lubrication pump may be damaged.



Lubrication pump unit

■ Coolant

Coolant is frequently disposed with chips. Therefore, the coolant level lowers rather fast. Pay attention to the coolant level gage at work in order to supply the coolant whenever necessary.

WARNING

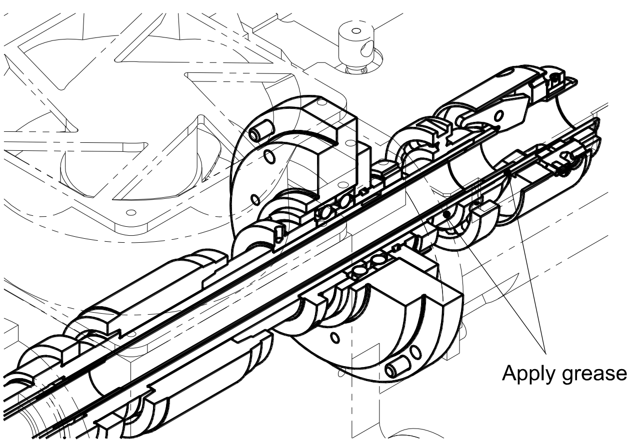
Monitor the level and condition of the coolant in the machine every day. Low or ineffective coolant can result in damage to the tool and a possible fire.

■ Tool post spindle spur gear lubricating

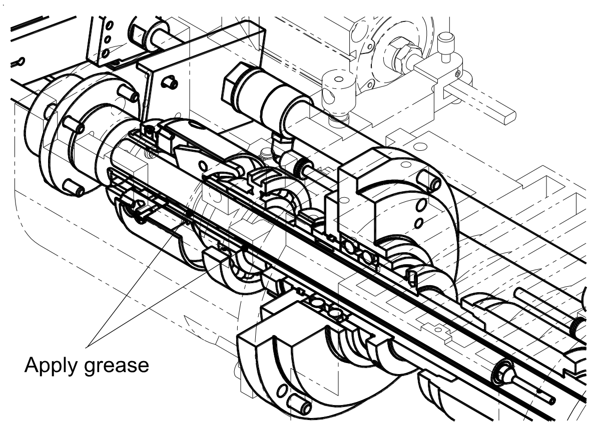
If the tool spindle is equipped with the grease nipple, use the grease gun to apply grease to the teeth of the spur gear located inside the tool spindle.

■ Bobbin sliding portion

Apply grease on the sliding portion of the spindle bobbin.
Apply grease on each location with the chuck both opened as well as closed.



Spindle Bobbin Sliding Portion

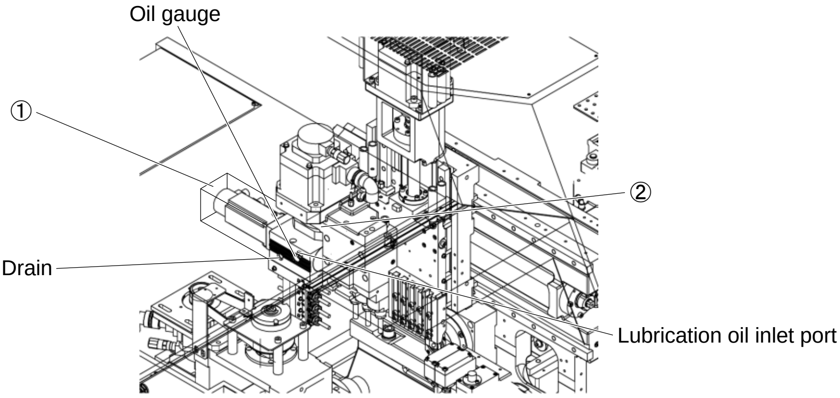


Back Spindle Bobbin Sliding Portion

> **CAUTION**
>
> Avoid "empty chucking" (chucking with no bar material).
> Doing so may cause the machine to malfunction or damage the collet chuck.

> **WARNING**
>
> Be sure to make the machine in emergency stop status before applying grease to the bobbin or spindle.
> Otherwise, your hand may be caught in the machine, resulting in a severe injury.

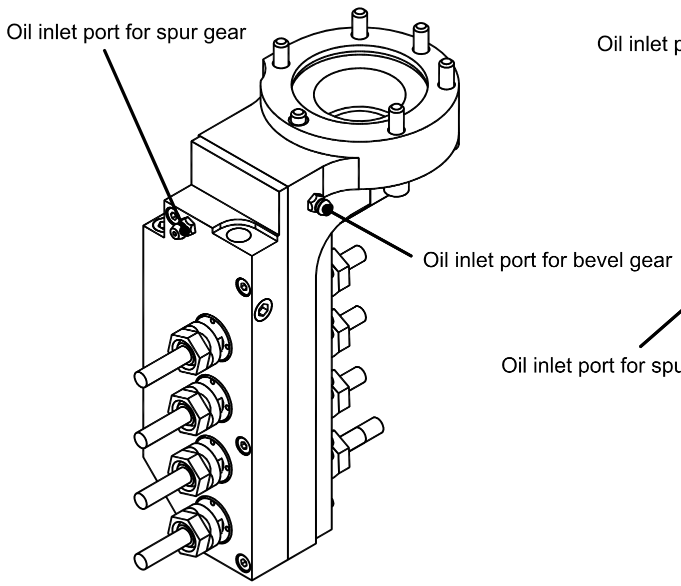
■ B1 axis revolving assembly (XII model)



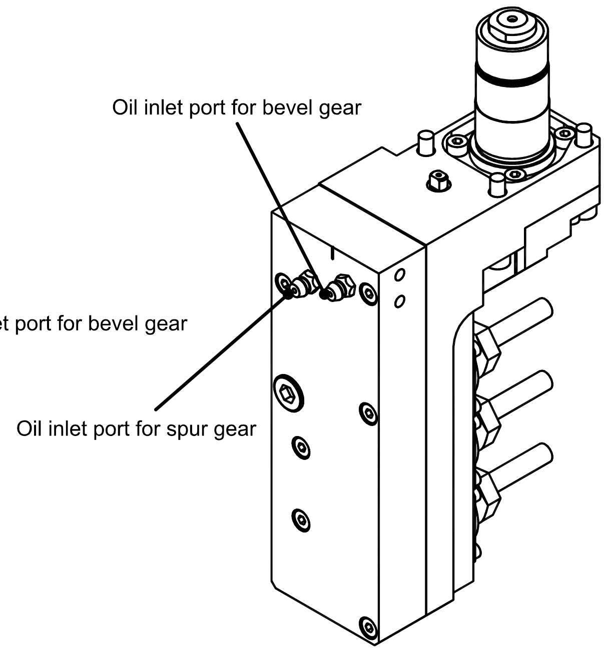
1. Set the B1 axis to B1 = 90 degrees.
2. Move the X1 axis downwards.
3. Remove covers 1 and 2.
4. Remove the plug and the old lubricating oil.
5. Remove the drain side plug.
6. Pour lubricating oil into the oil inlet port until it reaches the center of the oil level gauge.
7. Install the covers.

Lubricating a tooling gear equipped with a grease nipple

If the tooling is equipped with a grease nipple, use a grease gun to apply grease to the teeth of the gears located inside the tooling via the grease nipple.



MEU307



BSE607,707

22.2.3 Coolant

The applicable machine can use the following coolants.

- Water insoluble coolant (oilness) : Usable with the machine of standard specification
- Water soluble coolant : Usable with the machine of special specification (e.g., air sealing)

■ Water insoluble coolant (oilness)

Selecting a coolant

- Copper corrosion (100°C, 1h) 1 : Usable with no problem.
- Copper corrosion (100°C, 1h) 2 : Basically usable, however, some products have high activity. Contact the manufacturer of coolant before selecting a coolant.
- Copper corrosion (100°C, 1h) 3, 4 : Do not use. The machine (including electric and electronics component, resin, and sealant) will remarkably be damaged.

Precautions on the use of coolants

The chemical admixture contained in coolant may be deteriorated by evaporation due to heat generated during cutting or others. Accordingly, the initial performance may not be expected. Provide an appropriate maintenance such as replacement interval according to the instruction of manufacturer.

■ Water soluble coolants

Selecting a coolant

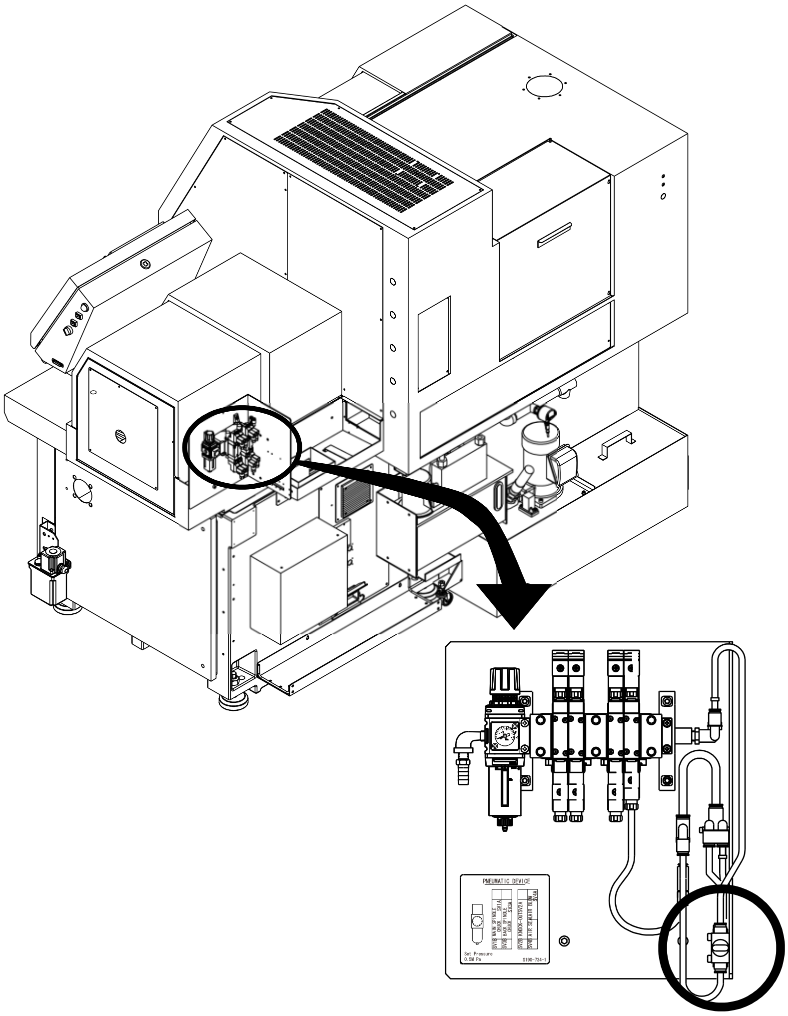
- Emulsion (milky white) type : Usable with no problem
- Soluble (translucent and transparency) type : Basically usable, however, some products have high permeability. Contact the manufacturer of coolant before selecting a coolant.
- Solution (transparency) type : Do not use. The machine (including painted portion, rotating section, resin, and sealant) will remarkably be damaged. Some products are called as a synthetic type.

Notes on using water-soluble coolant

- To prevent the adherence of rotary section, always open the manual cock of the air seal device when using a water soluble coolant.
- The dilution methods and ratio of coolants vary depending on the type of coolant. Follow the instructions provided by the manufacturer of each coolant. Be sure to check the coolant diluted state every day to keep the appropriate strength of the coolant solution. A rust may be generated with low pH (8.0 or less).
- Water-soluble coolants offer protection from rust while the machine surface is wet. Once the machine surface dries, however, it could rust. Apply the rustproof oil while the machine surface is still wet, to prevent rust.
- If the surface of the water-soluble coolant appears to be unclean (abnormal), replace the coolant immediately.
- Before starting and after ending operations, remove chips and lightly apply lubricating oil on the sliding parts of the machine.

CAUTION

Water-soluble coolants have high degreasing strength due to their alkaline content and this may cause skin inflammation. Be sure to wash your hands with neutral detergent after ending operations. Apply a protective cream especially if you have sensitive skin.



22.3 Consumable Parts

Some parts reach their life faster than the others. It is unavoidable to use short-life parts in terms of the machine structure. These wearing parts must be checked regularly and replaced if defect is found. This section describes consumable parts.

Consumable parts

For the information on how to replace the parts listed below, refer to <*Chapter 7 Mounting and Adjustment* > of the Operator's Manual. For the replacement of filter mat, see <*22.1.2 Monthly Check*>.

Name	Type	Manufacturer	Check interval	Criteria for replacement
Main spindle collet chuck	FC034-M TF25 40.005 76-24	Citizen S&M NEUCOMM SCHAUBLIN	Every 3 months in continuous use	• Fatigue • Damage in the chuck grip • Abnormal wear
Main spindle chuck finger	L220E: C110	Citizen	Monthly	Abnormal wear
Guide bushing drive timing belt	655-U5GY-28 U3X9	Unitta	Every 6 months	• Damage • Abnormal wear
Guide bushing	WFG206-M, DFG206-M, 0201, TD25NS 61.002, 22.001 B238	Citizen S&M NEUCOMM DUNNER	Every 6 months	• Damage • Abnormal wear
Back spindle collet chuck	FC034-M-K TF25 40.005 76-24	Citizen S&M NEUCOMM SCHAUBLIN	Every 3 months in continuous use	• Fatigue • Damage in the chuck grip • Abnormal wear
Back spindle chuck finger	L220EU41B: 116	Citizen	Monthly	Abnormal wear
Y2 axis drive timing belt	645-5GT-25	Unitta	Every 6 months	• Damage • Abnormal wear
Element for pneumatic device (W1000-6-W)	W1000-ELEMENT-ASSY	CKD	Monthly	• Stained and clogged • Discolored
Back rotary tool drive timing belt	615-3GT-12	Unitta	Every 6 months	• Damage • Abnormal wear
Filter mat	SK3170100	RITTAL	Monthly	• Fatigue • Stainer

(Blank Page)

Product Code	C-L220E XII
Document Code	3E1-2200
Mfg. No.	L220E/0001~
Issue Date	2013.10

23. Troubleshooting

23.1 Failure Detection

The machine may cause a failure even though proper maintenance, including specified periodic checks, has been performed sufficiently. The machine must be recovered from the failure even in that case. This chapter describes the procedure for identifying the causes of relatively simple machine failures and the action required for recovering the machine from them.

23.1.1 Introduction

Once the machine causes a failure, you must take immediate action to prevent the failure from resulting in a serious accident. For this purpose, you must first use your senses to check for abnormal appearance of the machine such as damages to its components, abnormal heat generation, parching smells, abnormal noise.

If a trouble is serious and dangerous, immediately turn off the main circuit breaker.

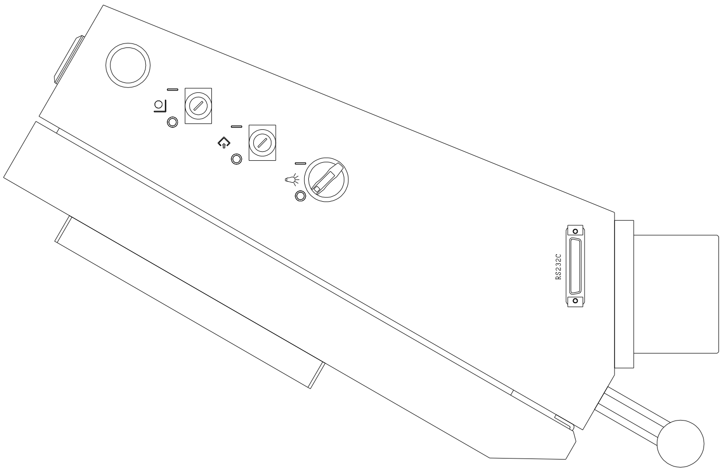
23.1.2 Failure detecting procedure

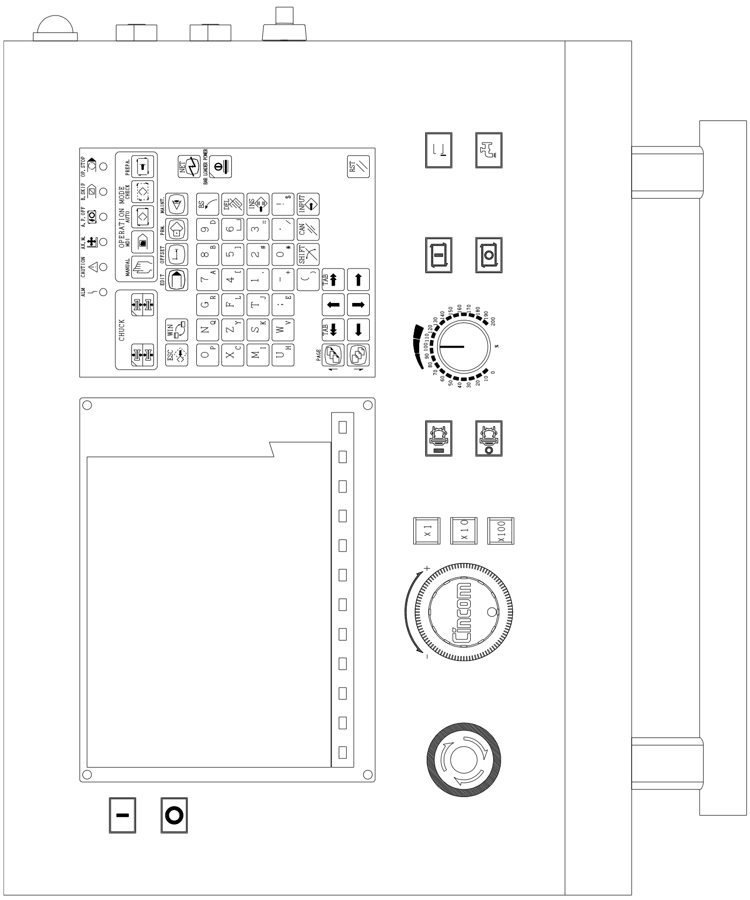
Detection with alarms
If the machine detects an error or failure, the alarm indicator on the operation panel goes on or blinks. In this case, see *<23.3 Alarms>*.

Detection by interface diagnosis
If the machine causes an unexpected or unknown fault or failure, it may adversely affect machine operation without generating an alarm. If this happens, signals of interface (I/F) diagnosis find the failure with checking input/output. See *<23.5 Mechanical Errors>*. I/F diagnosis is also available even when a failure has been detected with an alarm. For using I/F diagnosis, see *<23.2 Interface Diagnosis>*.

Detection by checking the mechanism
If a failure occurs while input/output signals are normal, the possible cause is assumed to be a mechanical part. Check the mechanical parts to see if the failure is a mechanical transmission fault caused by a loose or damaged belt or key or a mechanical movement fault caused by a galling slide or damaged bearing.

■ Sheet keyboard type: Symbol





23.2 Interface Diagnosis

The machine is controlled by a programmable logic controller (PLC) built into the NC device. The PLC receives sensor signals (as input signals) and drives parts such relays (using output signals) Checking the states of input and output signals to and from the PLC is called interface (I/F) diagnosis and is performed on the I/F DIAGNOSIS screen. Although I/F diagnosis is very effective to detect machine trouble, the user requires some technical knowledge. This section provides general information for effective use of I/F diagnosis.

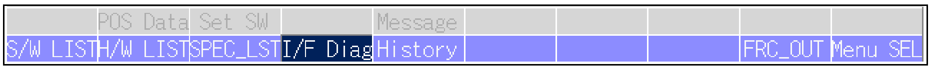
23.2.1 Operating the I/F DIAGNOSIS screen

Display the I/F DIAGNOSIS screen by following procedure.

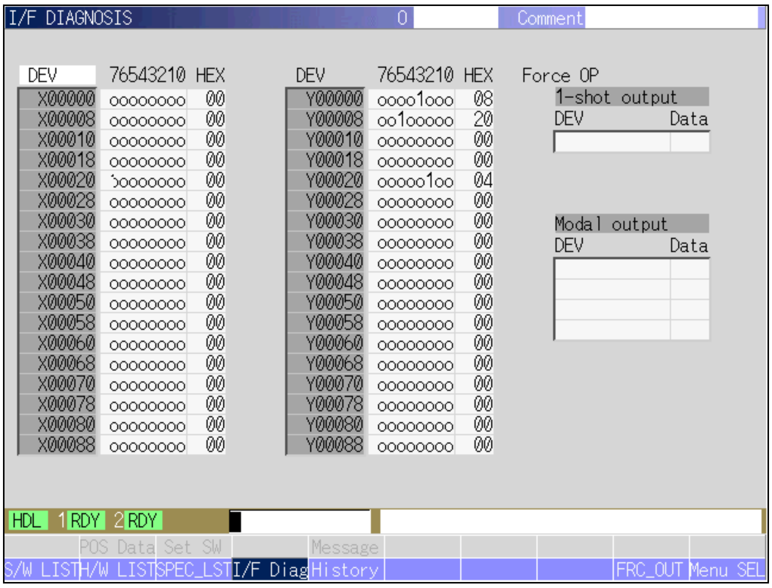
[Procedure]

1. Press the Diagnosis key MAINT..

The Diagnosis menu appears.



2. Press the menu key [I/F Diag]. (Skip this step when the I/F DIAGNOSIS screen has already been displayed.) The I/F DIAGNOSIS screen will appear.



3. Enter device numbers (X○○ for input and Y○○ for output) in the Device input fields on screen using <alphanumeric keys>.

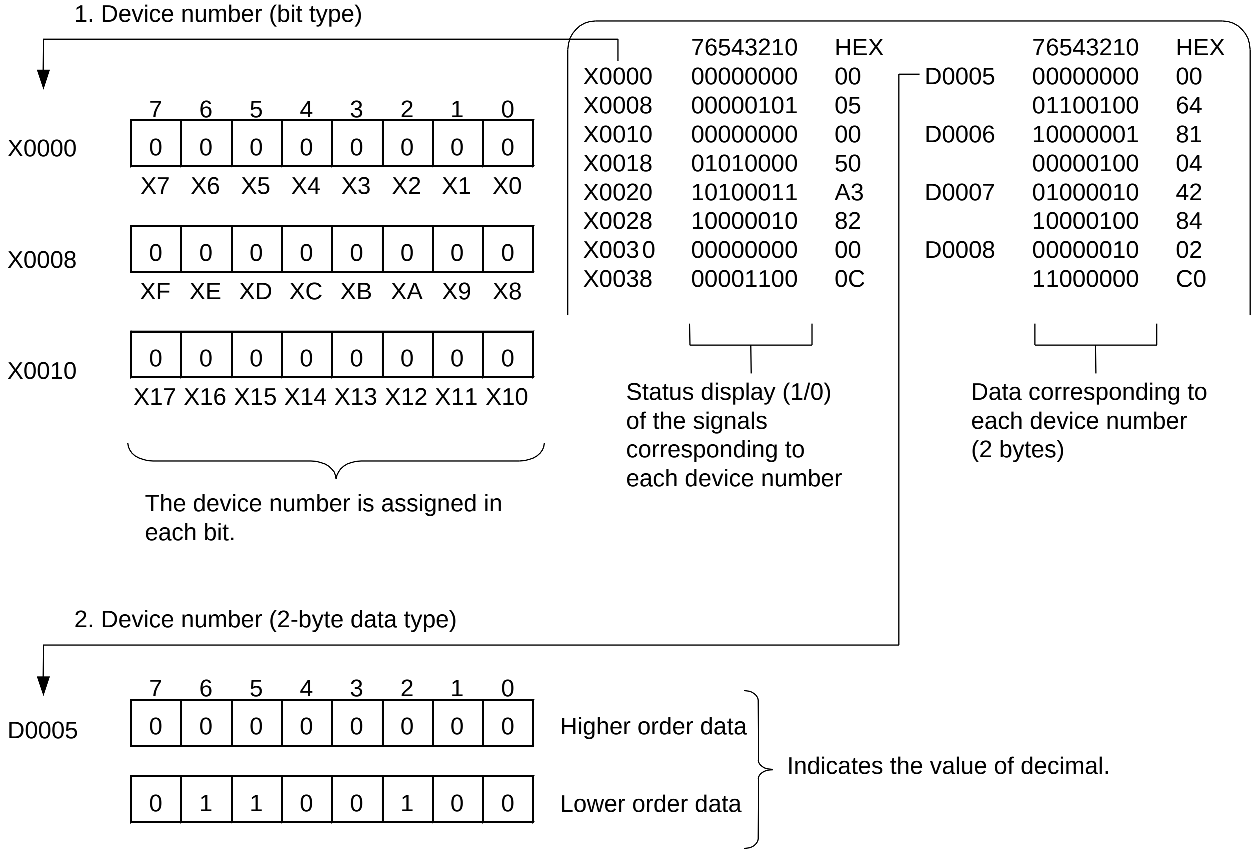
4. Press the input key INPUT.

The entered device numbers are displayed.

23.2.2 Device numbers and display data

The device number is made up of a letter from the alphabet and a number or numbers following that letter. Each device number has a specific meaning in the PLC.
There are two kinds of device numbers: bit type device numbers such as X and Y, and data type device numbers such as D and R. Following is an explanation of the device numbers.

1. Device number (bit type)



A signal changes according to the machine state and thus you can diagnose the machine state based on the signal.

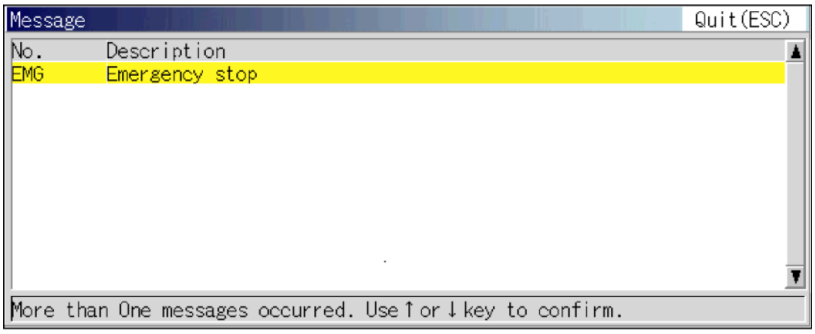
23.3 Alarms

23.3.1 Error and alarm messages

If an error or alarm occurs, the warning or alarm indicator on the operation panel goes on or blinks and the Message screen appears automatically. Check the message on the screen. The Message screen may not appear automatically depending on the content of the error or alarm. In that case, press the menu key [Message] to display the message.

[Procedure]

1. Press the Menu Up/Down selection key ▶ to enable the submenu (upper row) (Skip this step when the submenu has already been selected.)
2. Press the menu key [Message].
   The Message window will then appear.



23.3.2 Alarm lists

23.3.2.1 List of messages when the error lamp blinks

Alarm No.	Message	Description	Action
M01	OPERATION ERROR Error No. nnnn	Alarms during the NC run caused by operator errors or machine failures are displayed. Use the error number to check details.	See the Instruction manual of the NC manufacturer.
M02	NEED R.P. RTN Error No. nnnn	Absolute position detector malfunctions, such as deleting the absolute position data or the detector failure, occurred. Use the error number to check details.	See the Instruction manual of the NC manufacturer.
M03	COLLISION ALARM	Specified command causes an interference between two parts.	See *<23.4.1 Recovery from interference check alarm>*.
M04	AREA ALARM	Two parts are in the area where an interference may occur.	See *<23.4.1 Recovery from interference check alarm>*.

Alarm No.	Message and symptom	Action
OP001	Work counter is full. Auto operation stops end of cycle.	Press the Reset key after completion of the current cycle.
OP002	Lubrication oil is empty.Auto operation stops end of cycle.	Add lubricating oil.
OP003	Chip conveyor is overloaded.	See *<23.4.11 Recovery from external device alarms>*.
OP004	Coolant oil is empty. Auto operation stops end of cycle.	Add coolant.
OP005	External error. Auto operation stops end of cycle.	The additional external device is malfunctioning. Check the device.
OP006	Bar stock is empty. Auto operation stops end of cycle.	Add materials to the bar loader shelf.
OP008	Zero return mode. Other mode is disable	Press a key such as Manual Operation, MDI, or Auto.
OP009	Mechanical adjustment mode. Other mode is disable.	Press a key such as Manual Operation, MDI, or Auto.
OP011	Main and back spindle synchronous mode.	Spindle synchronization is cancelled by specifying G113 in the program or by pressing the Reset key when the back spindle stops.
OP012	Bar loader is disconnected. Setting switch No.7 is ON.	Set the setting SW 7 to OFF to enable completion of preparation of the bar loader.
OP013	Override switch is 0%. Then machine can not be moved.	Set the override switch appropriately.
OP014	Tool life is reached. Auto operation stops end of cycle.	Press the Reset key to release the machine from the error. Replace the worn tool.
OP015	Interference check is disable. Setting switch No.11 is ON.	Move the axis outside the interference area. Then, press the menu key [INT.IVLD] or remove the check mark from the setting switch "11 Interfere Check Off", to enable the interference check.
OP040	T.D.C. sensor alarm. Stops end of cycle. (**ch)	See <*23.4.14 Recovery from thermal displacement correction device* alarm>.
OP041	Door can not be locked. Please shut the door.	Tightly close the door so that it can be locked.
OP045	Please lock a door by a start key.	To re-start the main spindle, the door must be lockedPress the Start key to lock the door.
OP046	Spindle not restarted. Press SPINDLE START button.	The spindle is standstill. Restart the spindle with the Spindle start key and run the program with the Start key.
OP050	Pause during axis movement	Press the Start key.
OP056	Loader not ready. Machine stops after this cycle.	The bar loader is not ready. After the machine stops, check the bar loader for condition.
OP057	Dry run valid. Setting switch No. 4 is ON.	The dry run is enabled. Check the switch setting.
OP059	MST lock ON. Setting switch No. 14 is ON.	The auxiliary function is locked. Check the switch setting.
OP063	Bar loader door is open. Close the door.	Close the bar loader door. (For dedicated bar loader)
OP076	Device battery low level	See <*23.4.12 Troubleshooting after OP076 Device battery low level* Alarm>.

Alarm No.	Message and symptom	Action
OP077	Top cover fan alarm. The fan or the sensor is not working normally.	Check the fan or the sensor.
OP079	Cycle Start Off on the Set SW screen is effective.	Disable the Cycle Start Off function.
OP080	Single Block Off on the Set SW screen is effective.	Disable the Single Block Off function.
OP083	B axis is being set up. The handle cannot be operated.	Turn the power off and then on again if the warning does not go off.
OP087	Remote Access is permitted on the Set SW screen.	Remote Access is enabled. Check the switch setting.
OP088	It fails in the update of Eco/ope. info. file.	See *<23.5.6 If "It fails in the update of OP088 Eco/ope. info. file." alarm occurred>*
OP089	Execute "ST POS" on the PREPARATION mode.	Perform start position operation for prepraration before starting operation after idling is stopped.
OP093	Execute "ST_POS" on the PREP_mode after power on.	After power on, execute Start Position operation on Preparation screen before running the machine.
OP094	Close the door.	Close the door before starting operation.
OP095	Option effective time is less than.	Confirm the enable/disable status and remaining effective period of option.
OP097	Call local service center.Machine will stop in xx hours.	Call the Cincom Service Office.
OP098	Call local service center.Machine will stop in xx hours.	Call the Cincom Service Office.
OP100	Thermal displacement correction function is not working normally. The machine will stop after this cycle.	See *<23.4.14 Recovery from thermal displacement correction device* alarm>
OP101	The overheat in the control panel. It is high temperature.	The temperature in the control unit is abnormally high. Check the cause of abnormal temperature rise.
OP102	There is not enough oil air lubrication. The machine will stop after this cycle.	When the cycle stops, replenish lubrication oil when it is safe to do so.
OP200	Illegal interference check data. Reselect tool holder.	The interference check data is illegal. Reset "Bar Stock O.D.", "Back Spindle Chuck POS", "Front Mach Holder Name", "Front Drill Holder Name", and "Back Spindle" in the machining data.
OP201	Guide bush and work on back chuck are interfered.	Continuing the operation may cause mechanical interference. Review the program or issue the M code for disabling the interference check (M89).
OP202	Guide bush and work basket on back spindle are interfered.	Same as above.
OP203	The guide bushing device and the workpiece separator interfere with each other.	Same as above.
OP204	The guide bushing device and the long workpiece device interfere with each other.	Same as above.
OP205	Back drill unit and work on back chuck are interfered.	Same as above.
OP206	Back drill unit and back sp. work basket are interfered.	Same as above.
OP207	Back drill unit and back spindle cap are interfered.	Same as above.
OP208	Back drill unit and back spindle cap cover are interfered.	Same as above.
OP209	Back drill unit and back spindle are interfered.	Same as above.
OP212	Back drill sleeve and work on back chuck are interfered.	Same as above.
OP213	Back drill sleeve and back sp. work basket are interfered.	Same as above.
OP214	Back drill sleeve and back spindle cap are interfered.	Same as above.
OP215	Back drill sleeve and back sp. cap cover are interfered.	Same as above.
OP216	Back drill sleeve and back spindle are interfered.	Same as above.
OP218	Back drill sleeve and front drill sleeve are interfered.	Same as above.
OP234	Front drill holder and back sp. work basket are interfered.	Same as above.
OP235	Front drill holder and work on back chuck are interfered.	Same as above.
OP236	Front drill holder and back spindle cap are interfered.	Same as above.
OP237	Front drill holder and back Sp. cap cover are interfered.	Same as above.
OP239	The front drill holder and the workpiece separator interfere with each other.	Same as above.
OP240	The front drill holder and the long workpiece device interfere with each other.	Same as above.
OP249	The workpiece separator and the long workpiece device interfere with each other.	Same as above.
OP266	The work chute and the opposite tool post interfere with each other.	Same as above.
OP267	Work chute and work on back spindle chuck are interfered.	Same as above.

Alarm No.	Message and symptom	Action
OP268	Work chute and back spindle work basket are interfered.	Continuing the operation may cause mechanical interference. Review the program or issue the M code for disabling the interference check (M89).
OP269	Work chute and back spindle cap are interfered.	Same as above.
OP270	The work chute and the front drill sleeve interfere with each other.	Same as above.
OP271	The work chute and the long workpiece device interfere with each other.	Same as above.
OP272	The work chute and the workpiece separator interfere with each other.	Same as above.
OP273	Front drill holder and front drill sleeve are interfered.	Same as above.
OP275	Front holder and B.SP. slide cover are interfered.	Same as above.
OP284	The back machining tool and the workpiece chucked by the back spindle interfere with each other.	Same as above.
OP285	The back machining tool and the workpiece receiver box at the front end of the back spindle interfere with each other.	Same as above.
OP286	The back machining tool and the back spindle cap nut interfere with each other.	Same as above.
OP287	The back machining tool and the back spindle cap nut cover interfere with each other.	Same as above.
OP288	The back machining tool and the back spindle interfere with each other.	Same as above.
OP289	The back machining holder and the front-facing tool interfere with each other.	Same as above.
OP290	The front drill holder and the front-facing tool interfere with each other.	Same as above.
OP291	The front-facing tool and the guide bushing interfere with each other.	Same as above.
OP300	Press the enable switch.	Press the Enable switch and perform operation.
OP301	T.D.C. data alarm. Stops end of cycle.	See <*23.4.14 Recovery from thermal displacement correction device* alarm>.
OP302	Z1 axis stroke canceled. Setting switch No. 47 is ON.	Z1 axis stroke is canceled. Confirm it.

23.3.2.2 List of messages displayed with alarm indicator on

Alarm No.	Message and symptom	Action
EX001	200VAC over current alarm. Circuit protector is activated.	See <*23.4.7 Recovery from overcurrent alarm*>.
EX003	Main spindle motor alarm. The drive unit is in alarm status.	See <*23.4.9 Troubleshooting after main spindle, back spindle, and rotary tool alarms*>.
EX004	Back spindle motor alarm. The drive unit is in alarm status.	See <*23.4.9 Troubleshooting after main spindle, back spindle, and rotary tool alarms*>.
EX005	Gang Rotary tool Motor Alarm. The drive unit is in alarm status.	See <*23.4.9 Troubleshooting after main spindle, back spindle, and rotary tool alarms*>.
EX008	Facing rotary tool motor alarm. The drive unit is in alarm status.	See <*23.4.9 Troubleshooting after main spindle, back spindle, and rotary tool alarms*>.
EX009	Back Rotary tool Motor Alarm. The drive unit is in alarm status.	See <*23.4.9 Troubleshooting after main spindle, back spindle, and rotary tool alarms*>.
EX052	Emergency button alarm. Emergency button is pressed.	Turn the Emergency Stop button clockwise to cancel emergency stop.
EX055	Coolant or medium-press. pump overload. Thermal activated.	Remove the major cause(s) and reset the thermal relay.
EX101	Bar loader alarm. Bar loader device is in alarm status.	See <*23.4.11 Recovery from external device alarms*>.
EX102	External alarm 1. Option device is in alarm status.	Check the external device.
EX103	External alarm 3. Option device is in alarm status.	Check the external device.
EX104	External alarm 4. Option device is in alarm status.	Check the external device.
EX105	Coolant discharge alarm. Discharge amount is decreased.	See <*23.4.13.2 Troubleshooting after "EX105 Coolant Discharge Alarm*">.
EX106	Knock-Out Overload. Knockout error sensor ON by overload.	See <*23.4.13 Recovery from other* alarms>.
EX107	Tool bit breakage alarm. Cut off tool is broken.	See <*23.4.13 Recovery from other* alarms>.
EX108	Cycle time alarm. Machine stops long time.	See <*23.4.13 Recovery from other* alarms>.
EX109	Main spindle speed fluctuation alarm.	See <*23.4.10 Troubleshooting after spindle related* alarms>.
EX110	Back spindle speed fluctuation alarm.	See <*23.4.10 Troubleshooting after spindle related* alarms>.
EX113	Knock-out is not in a retract position.	Retract the knock-out. If the operation is not recovered, check the knock-out retraction end sensor.
EX114	It cannot be ordered "M10" or "M11".	M10 or M11 cannot be specified if the machine is equipped with the back bascket. Check the program.
EX115	BackSpindle rpm must be 100 or less due to back basket.	If the machine is equipped with the back basket, the back spindle speed must not be faster than 100 min^{-1}. Check the program.
EX118	Z1 is not in correct position.	Move the Z1 axis manually to the area where the machine can start.
EX122	T.D.C. device alarm	See <*23.4.14 Recovery from thermal displacement correction device* alarm>.
EX131	T.D.C. data alarm	See <*23.4.14 Recovery from thermal displacement correction device* alarm>.
EX132	T.D.C. sensor alarm	See <*23.4.14 Recovery from thermal displacement correction device* alarm>.
EX133	Bar loader is not ready. Bar loader power is not turned on.	See <*23.4.11 Recovery from external device alarms*>.
EX136	The bar loader is not ready. Close channel/stabilizer.	Close stabilization in the bar loader operation screen.
EX137	Open the clamp.	Use the bar loader operation screen to move the material clamp to the open position.
EX138	Move the rail to the back.	Use the bar loader operation screen to move the push arrow to the lowered position.
EX139	Raise the bar rest.	Use the bar rest operation screen to move the bar rest to the raised position.

Alarm No.	Message and symptom	Action
EX140	Back spindle error.	The back spindle motor fails. Check if it works normally.
EX144	Knock-out sensor is abnormal.	See *<23.4.2.2 Resetting "EX144 Knock-out Sensor Illegal"* Alarm>
EX146	Main Spindle chuck unjust operation.	See *<23.4.3 Recovery from spindle chuck illegal state>*.
EX147	Back Spindle chuck unjust operation.	See *<23.4.4 Recovery from back spindle chuck illegal state>*.
EX148	Please retreat knock-out.	Retract the knock-out device on Mechanical Adjustment screen, then start operation.
EX149	Please advance knock-out.	Advance the knock-out device on Mechanical Adjustment screen, then start operation.
EX156	Middle pressure coolant valve is not open.	Open valve before turning on the pump.
EX157	Middle pressure coolant pump is turning on.	Close valve before turning off the pump.
EX158	The Main Spindle overheated.	If you feel odor or abnormal overheat from the main spindle, turn the main power off and contact the Cincom Service immediately.
EX159	The Back Spindle overheated.	If you feel odor or abnormal overheat from the back spindle, turn the main power off and contact the Cincom Service immediately.
EX165	Phase adj. disabled. Turn main/back Sp. more than 1 rev.	Run both spindle and back spindle by at least one rotation, and specify G899 again.
EX200	Work counter full. Counter is reached required quantity.	Press the Reset key.
EX201	Bar stock empty. No bar stock.	See *<23.4.11 Recovery from external device alarms>*.
EX202	Lubrication oil empty alarm. Supply the oil.	See *<23.4.13 Recovery from other* alarms>.
EX203	Coolant oil alarm. Supply the oil.	See *<23.4.13 Recovery from other* alarms>.
EX204	Oil air lubrication alarm. There is not enough lubrication oil. Add lubrication oil.	See *<23.4.13 Recovery from other* alarms>.
EX205	External alarm 2. Option device is in alarm status.	Check the optional device.
EX208	Chip conveyor overload. Chip is jammed in the conveyor.	See *<23.4.11 Recovery from external device alarms>*.
EX210	The bar loader failed to operate normally.	The bar loader failed to operate normally. Check for mechanical interference.
EX211	Tool life alarm. Specified tool's tool life is over.	Press the Reset key and replace the worn tool.
EX212	Door open alarm of the bar loader. Close the door.	Close the bar loader door. (special bar loader mechanism)
EX215	Option effective time is over.	Confirm the enable/disable status and remaining effective period of option.
EX225	Automatic power-OFF failure.	Automatic power-off function is disabled due to failure of circuit. Check if circuit breaker (NFB1) and relay (Ry201) work normally, or check if cables are properly connected.
EX301	Abnormal feedrate. Axis feedrate exceeded limit.	The axis feed rate exceeded the safe level with the door opened. Turn the machine main circuit breaker off and remove the cause(s). (For products shipped to EC)
EX304	Illegal door lock signal. Door lock is faulty.	Even when the door was locked, it was unlocked during the operation. After the door was locked once, the door lock signal goes off.Turn the machine main circuit breaker off and remove the cause(s).
EX305	Safety feedrate exceeded. Exceeded in PH operation.	During handle feed operation, safety speed is exceeded. Release the alarm by pressing the Reset key. (For products shipped to EC)
EX311	Contactor & Relay breakdown.	Fused contact was detected on contactor (MC1, MC2, MC3A, MC3B, MCCL) or relay (RY10, RY12). Replace the contactor or relay with new one.

Alarm No.	Message and symptom	Action
EX312	Door lock relay breakdown.	Fused contact was detected on relay RY15 or on-board RY304. Replace the relay with new one.
EX313	Feedrate is over safety speed.	Turn off the NC power or main breaker.
EX314	Sp.Rotation is over safety speed.	Turn off the NC power or main breaker.
EX401	Detection of the machine moving.	Contact your sales representative.
EX402	Device function alarm	Contact your sales representative.
EX403	Internal data alarm.	Call the Cincom Service Office.
EX404	M-Code alarm.	The specified M code is illegal.
EX407	Option is not set.	Check the program.
EX410	Last part function not executed. Check workpiece.	Last program has not been ejected. Check the workpiece. Also check the program.
EX501	Spindle chuck closed. Open spindle chuck.	Open the spindle chuck and perform start-position operation.
EX502	Cut-off tool number on the Machining Data is not set.	Specify the "Cut-Off Tool" in the machining data.
EX520	Cancel tool nose R compensation. (G40)	Specify G40 before T code to cancel tool nose radius compensation.
EX521	Cancel constant surface speed control function. (G97)	Specify G97 to cancel constant surface speed control before specifying the T code.
EX522	You cannot specify this T code.	This T code does not exist. Review the program.
EX524	You cannot specify a value less than "0" for "H argument".	When specifying the "H" argument at the selection of a tool on gang tool post, specify a value larger than "0".
EX525	A value exceeding "0" cannot be specified for "argument W".	For the "W" argument, specify a negative value.
EX526	Illegal argument is specified.	An argument that is not permitted for a T code is specified. Check the program.
EX531	Use same number for X in T1X00 and T5X00.	In the front/back simultaneous machining, specify the same "X" number for T1X00 and T5X00. Review the program.
EX535	Argument K2 cannot be specified.	The tool number is illegal. Review the program.
EX537	K2 & E argument cannot be specified simultaneously	Specify either of K2 or E argument. Review the program.
EX539	R1 argument cannot be Specified.	The R1 argument is unavailable for the specified T code.Review the program.
EX551	Argument A in G231 block is illegal.	The A argument is not specified in G231.
EX555	Designation of the machining pattern is not allowed.	Cancel the current machining pattern and specify another pattern.
EX556	Argument X in G231 block is illegal.	The "X" argument value is illegal. Check the program.
EX559	Argument exceeds the upper bound value.	The specified value is larger than the upper limit of this argument. Review the program.
EX562	Back spindle setting in machining data is illegal.	On the Machining Data screen, "Basket" is not set for "Back Spindle".
EX568	More than two M211 or M212 arguments are specified.	Review the program.
EX569	The M211 or M212 arguments are invalid.	Review the program.
EX574	You cannot specify M32 or M33.	You can only specify M32 and M33 in G611 mode $1 or in G632 and G637 mode $2.
EX580	C axis option for the main spindle is not installed.	To specify the C axis indexing command for main spindle (M18) purchase the C axis option.
EX581	C axis option for the back spindle is not installed.	To specify the C axis indexing command for back spindle (M48) purchase the C axis option.
EX582	Indexing option for the back spindle is not installed.	To specify the C axis indexing command (M78) purchase the back spindle indexing option.
EX583	Indexing angle value must be within 0 to 360.	For M28, M78, M18, and M48, specify indexing angels equal to and greater than "0" and less than "360".
EX584	Illegal indexing command specified.	Indexing unit angle: Specify a multiple of (M28: 1°, M78: 1°) as the spindle indexing angle.

Alarm No.	Message and symptom	Action
EX590	Back spindle chuck closed.	Open the back chuck before specifying advancing the knock-out bar.
EX591	Back spindle is working.	Stop the operation of the back spindle before specifying advancing the knock-out bar.
EX592	Knock-out can not be advanced to correct position.	See <*23.4.10 Troubleshooting after spindle related* alarms>.
EX598	Argument Z cannot be specified.	Z argument cannot be specified for M10. Review the program.
EX602	There is an error in R1 argument command.	There is an error in T code with R1 argument specified in G660 mode.Review the program.
EX603	Specify the A or B argument in G660 mode.	Review the program.
EX604	This M code cannot be used by the superimposed.	Cancel the superimposition mode, then specify this M code. Review the program.
EX605	Specify the same value simultaneously for X and Y arguments.	Review the program.
EX606	You cannot specify argument X.	Review the program.
EX607	You cannot specify argument Z.	Review the program.
EX609	You cannot make specifications for this system ($).	Review the program.
EX620	Remnant bar did not retract.	Return the remaining material.
EX621	Remnant bar was not removed.	Pull out the remaining material.
EX622	Bar stock empty.	Supply the bar.
EX623	Material is not in correct position.	Check the material position and set the material properly.
EX624	Abnormal bar feeder sensor. Clean the sensor.	Clean the sensor element.
EX625	Abnormal bar feeder sensor. Check the bar feeder sensor.	Check the sensor for staining.
EX701	Tool has not been selected. Select tool.	Select a tool with the cursor on the Preparation screen before executing the preparation for operation.
EX702	Core cannot be selected for $2.	
EX703	DIA cannot be selected for $2.	
EX704	Cut-off cannot be selected for $2.	Select the T code correctly on the Preparation screen to perform "Cut-off" operation.
EX706	"You cannot specify "T3000".	A T code of "T3000" can be specified only in the program of $2. Review the program.
EX710	Opposite tool post could cause interference.	Specify command M140 after T20's.
EX712	You cannot specify this T code (~"T2000").	This T code does not exist. Review the program.
EX714	You cannot specify a T code less than "T2100" for $2.	A T code that is less than "T2100" can be specified only in the program of $1. Review the program.
EX715	You cannot specify this T code (~"T2900").	This T code does not exist. Review the program.
EX724	The tool nose is over the maximum length.	Set the tool again after confirming the tool protruding length, or use the "Q1" argument when specifying a T code.
EX725	Z1-Z2 axes are superimposed. Cancel superimpose function.	Specify G810 to cancel the Z1-Z2 superimposition, and then specify T20's command.
EX729	Front mach holder on main chuck work may cause interference.	Move the Y1 axis to a safety position, and then operate the X1 axis.
EX734	You cannot specify argument E for $2.	Specify without using argument E.
EX735	You cannot specify argument X.	Argument X has not been specified for G900. Review the program.
EX736	Back Toolset DIA is required.	Specify the "Back Toolset DIA" in the Preparation screen.
EX737	S2 Cover & T3x Cross Holder may cause interference.	Continuing the operation may cause mechanical interference. Specify the M1 argument. Review the program.
EX738	You cannot specify this code.	Review the program.
EX750	B axis device has not been mounted.	Specify a B axis option.
EX751	You cannot make specifications in B axis mode (G900).	Review the program.

Alarm No.	Message and symptom	Action
EX752	You cannot make specifications in modes other than B axis mode (G900).	Review the program.
EX753	Specify using machining pattern G600, G611 or G630.	Review the program.
EX754	Specify when selecting a B axis tool.	Review the program.
EX755	Invalid tool name used in preparations.	Change the tool name in the Man. Set in preparations.
EX757	Specify argument B.	Review the program.
EX758	The plane selection is invalid.	Review the program.
EX759	Make specification for plane G18.	Review the program.
EX764	B axis angle is incorrect.	Review the program.
EX765	You cannot specify argument Y.	Review the program.

23.3.2.3 NC alarm messages

Alarm No.	Message	Symptom	Action
S01	SERVO ALARM:PR Error No. nn	Servo system error. Use the error number to check for details.	See the Instruction manual of the NC manufacturer.
S02	INT PARAM ERR Error No. nn	Error in parameters sent to the servo amplifier from the NC.	See the Instruction manual of the NC manufacturer.
S03	SERVO ALARM: NR. Error No. nn	Servo system error. Use the error number to check for details.	See the Instruction manual of the NC manufacturer.
S04	SERVO ALARM: AR Servo system error.	Servo system error. Use the error number to check for details.	See the Instruction manual of the NC manufacturer.
S51	PARAMETER ALARM Warning No. nn	Illegal value entered for the servo parameters.	See the Instruction manual of the NC manufacturer.
S52	SERVO WARNING Error No. nnnn	Servo system error. Use the error number to check for details.	See the Instruction manual of the NC manufacturer.
	Battery voltage drop 009F	The battery voltage, which is supplied to the absolute position detector of servo motor, has dropped.	
Y02	SYSTEM ALARM Error No. nnnn	Error in data transmission between the NC and the servo amplifier. Use the error number to check for details.	See the Instruction manual of the NC manufacturer.
Y03	AMP. UNEQUIPPED	Servo amplifier not equipped properly.	See the Instruction manual of the NC manufacturer.
Y05	INT PARAM ERR Error No. nnnn	Error in parameters used to turn the NC power on. Use the error number to check for details.	See the Instruction manual of the NC manufacturer.
Y06	mcp_no ERROR Error No. nnnn	When the NC power is turned on, there are discrepancies between the NC servo interface section (MCP) and the axis parameters. Use the error number to check for details.	See the Instruction manual of the NC manufacturer.
Y07	AMPLIFIER POWER OFF Servo system error.	Servo system error. Use the error number to check for details.	See the Instruction manual of the NC manufacturer.
Y51	PARAMETER ERROR Error No. nnnn	Parameter error occurred while some control axes were moving. Use the error number to check for details.	See the Instruction manual of the NC manufacturer.
Znn	xxxxx	NC system alarm. Use the error number to check for details.	See the Instruction manual of the NC manufacturer.
Z52	Battery fault xxxxx	The voltage of the battery which is mounted on the NC control unit has dropped.	See the Instruction manual of the NC manufacturer.

23.3.3 Stop codes

Stop codes represent the states in which the NC unit has been stopped in a certain condition.

Alarm No.	Message	Symptom	Action
T01	CAN'T CYCLE ST Error No. nnnn	NC can not start automatic operation in the stop status. Use the error number to find more information.	See the Instruction manual of the NC manufacturer.
T02	FEED HOLD Error No. nnnn	A certain condition caused the automatically operating NC unit to halt automatic operation. Use the error number to find more information.	See the Instruction manual of the NC manufacturer.
T03	BLOCK STOP Error No. nnnn	During automatic operation, the NC stopped program execution at the end of 1 block. Use the error number to find more information.	See the Instruction manual of the NC manufacturer.

23.3.4 File system error lists

The following explains details of file system errors that occur during screen operation.

If a file system error occurs, the error message is displayed in the format of "File system error ○ Δ (◇)".

○: Indicates the operation.

(If the corresponding code is not displayed, the error code indicates that the error occurred during the operation of a file other than the files below.)

Display	Contents of operation
1	Directory operation
2	Machining program operation
3	Tool name file operation
4	Right-hand/left-hand information file operation
5	Tool pattern file operation
6	Interference check file operation
8	Tool set 2 file operation

Δ: Indicates the processing executed at the occurrence of the error.

Display	Contents of processing
1	Failed to open the file or directory.
2	Failed to read the file or directory.
3	Failed to write the data to the file.
4	Failed to close the file or directory.
5	Failed to delete the file.
6	Failed to copy the file.
7	Failed to get the size information of the file.
8	File size is larger than the available area size
9	Illegal holder number, etc.

◇: Indicates the detail cause of the error, or the name of the file that caused the error.

Display	Details of cause
MDI	Error occurred in MDI program operation.
Program number	Error occurred in machining program operation. (Displays the program number.)
File name	Name of the file that caused the error.
Error code	Code of the error occurred as the result of operation.

Display examples of file system error

Example 1) File system error 26 (1)
Failed in copying the machining program 1.prg file.

Example 2) File system error 1 (MDI)
Failed in opening the MDI program.

Corrective Action

1. Error caused by machining program file operation

If the error occurs repeatedly when the same operation is attempted, the probable cause is "the file size is too large" or "the file is illegal".

If the error occurred with the externally input program, check the program at the input source.

If the error occurred with the program created with PC, etc., the program must be created again.

If the problem cannot be cleared even after the corrective action explained above has been taken, it is necessary to recover the SRAM. Before recovering the SRAM, back up the machining programs stored in the machine to the external memory device (memory card, etc.) and record the offset data and tool setting data.

2. Error caused by operation of a file other than the machining program file

The probable cause is "necessary file does not exist" or "SRAM area is destroyed". Please contact the Cincom Service Office.

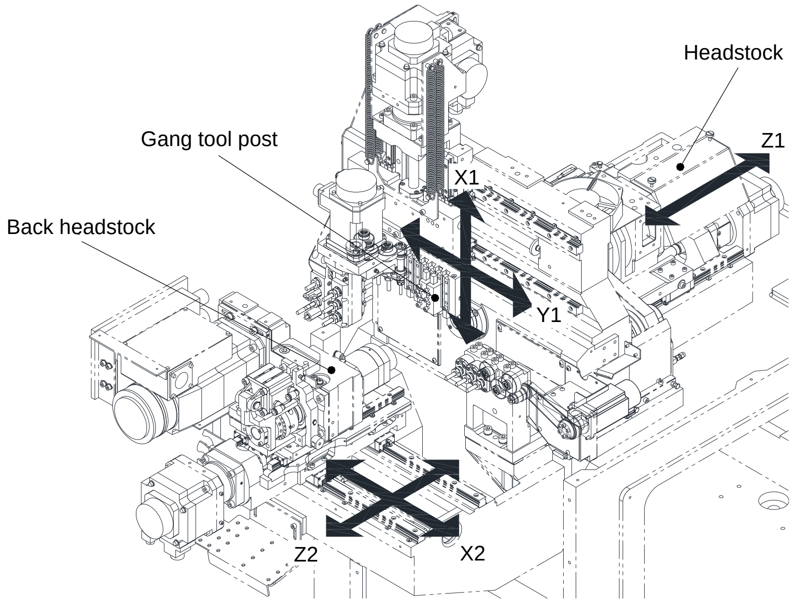
23.4 Post-Alarm Actions

This section describes the proper actions required to identify and correct the machine problem. The solutions provided are those to some of the most common problems. Call the Cincom Service Office if the problem cannot be located or is beyond the scope of this manual.

23.4.1 Recovery from interference check alarm

23.4.1.1 Interference check

This machine checks for interference among the components.



The interference check function recognizes rough shape of the machine according to the machining data and checks for interference among components by the data.

[Note]

Be sure to set the machining data correctly.

CAUTION

The NC unit does not have exact dimensional or geometrical data on the components of the machine when checking for interference between them.
Accordingly, components may interfere with each other while the interference check alarm has not been generated, or the interference check alarm may be generated while they seem to have no chance of interference.
You should therefore perform visual check for interference in combination with the interference check function of the machine.

23.4.1.2 Troubleshooting after "M03 COLLISION ALARM"

[Procedure]

1. Press the Reset key RST.
The error indication disappears.

23.4.1.3 Troubleshooting after "M04 AREA ALARM"

[Procedure]

1. Press the Manual key MANUAL.
The Manual key lamp goes on and the Handle Feed screen appears.

2. Press the Menu Up/Down selection key ► to enable the submenu (upper row) (Skip this step when the submenu has already been selected.)

3. Press the menu key [INT.IVLD].
The interference check is disabled.

4. Use the arrow keys ⬆ ⬇ to select the axis to be free from interference, then turn the handle to escape the interfering object.
The alarm indicator goes off.

5. Press the menu key [INT.IVLD].
The interference check is enabled.

[Note]

- The interference check function can be disabled for only specific components by specifying M88 (interference check disable command) and M89 (interference check enable command) in the program.
- The interference check alarm is generated by an error either in the program or in operation. Try to create and use the program without generating the interference check alarm.

23.4.2 Resetting Knock-Out Alarm

23.4.2.1 Troubleshooting after "EX106 Knock-Out Overload" (When the machine is equipped with the air-driven back knock-out device <standard>)

[Procedure]

1. Check if the workpiece is remained in back spindle chuck.

When remained:

- Remove the workpiece.
  - ➔ Take procedures described in <*7.4.3 Resetting knock-out overload*> of Operator's Manual.

When not remained:

- ➔ Go to Step 4.

2. Check if chips are remained in back spindle chuck or inside the machine.

When remained:

- Remove the remained chips.
  - ➔ Take procedures described in <*7.4.3 Resetting knock-out overload*> of Operator's Manual.

When not remained:

- ➔ Go to Step 4.

3. Check if the knock-out rod is bent.

When bent

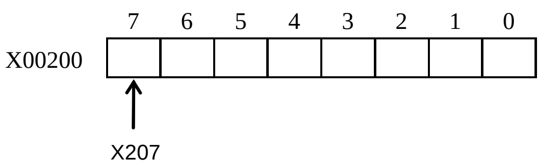
- Remove the cause that have made the rod bent. Then, replace the knock-out rod.
  - ➔ Take procedures described in <*7.4.3 Resetting knock-out overload*> of Operator's Manual.

When not bent:

- ➔ Go to Step 4.

4. Make sure that the knock-out rod is set in specified position, then make sure of X207 on I/F Diagnosis screen.

Confirmation of I/F Diagnosisscreen X00207



When X207 is zero:

- Sensor failure
- Sensor cable breakage
- Incorrect sensor mounting position

The above are probable causes of the alarm.

23.4.2.2 Resetting "EX144 Knock-out Sensor Illegal" Alarm

[Procedure]

1. Confirm that no workpiece is left in the back spindle chuck.
   - Remove the workpiece, if any.
2. Check if the knock-out device works normally.

   When the knock-out device works normally:

   ➔ Go to Step 3.

   When the knock-out device does not work normally:

   ➔ Go to Step 4.
3. Check if knock-out operation completes within the specified time (standard time: 7.5 seconds).

   Longer than the specified time:
   - If an alarm occurs while the knock-out device is moving forward, increase the value of #18064.
     [Setting time]
     PLC constant #18064: 75
   - If an alarm occurs while the knock-out device is moving backward, increase the value of #18065.
     [Setting time]
     PLC constant #18065: 75

   Not longer than the specified time:
   - Sensor failure
   - Imperfect sensor adjustment
   - Sensor cable breakage

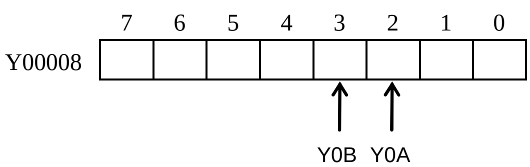
   The above are probable causes of the alarm.
4. Check the Status Screen 4 while running the following program in MDI mode.

```
G4 U
M10
G4 U
M11
M2
```

Make sure of Y0A on the Status screen while executing M10 (knock-out advance) command.

Make sure of Y0B on the Status screen while executing M11 (knock-out retract) command.

Confirmation of Statusscreen Y0A and Y0B



When Y0A and Y0B remains as 0:
- The NC unit may be faulty.

When Y0A and Y0B switches between 1 and 0:
- Pneumatic device error
- Solenoid valve trouble
- Breakage of solenoid valve cable

The above are probable causes of the alarm.

23.4.3 Recovery from spindle chuck illegal state

[Alarm message]

EX146 Main Spindle chuck unjust operation

[Procedure]

1. Check if the spindle chuck operates.

If so:

- The spindle chuck open/close sensor is not adjusted correctly.
- The spindle chuck open/close sensor is faulty.
- The sensor cable is disconnected.

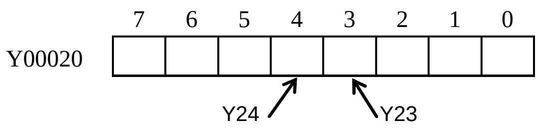
The possible cause is any of the above conditions.

If not:

➔ Go to Step 2.

2. Check Y23 and Y24 on the I/F DIAGNOSIS screen while manually operating the spindle chuck.

Checking Y0023 and Y0024 on the I/F DIAGNOSIS screen



If Y23 and Y24 remain at "0":

- The NC input unit will be faulty.

If Y23 and Y24 change between "1" and "0":

- The pneumatic device is faulty.
- The solenoid valve is faulty.

The possible cause is any of the above conditions.

23.4.4 Recovery from back spindle chuck illegal state

[Alarm message]

EX147 Back Spindle chuck unjust operation

[Procedure]

1. Check if the back spindle chuck operates.

If so:

- The back spindle chuck open/close sensor is not adjusted correctly
- The spindle chuck open/close sensor is faulty
- The sensor cable is disconnected

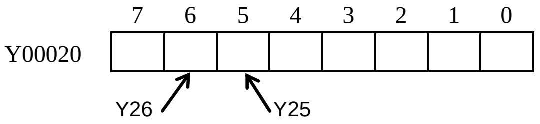
The possible cause is any of the above conditions.

If not:

➔ Go to Step 2.

2. Check Y25 and Y26 on the I/F DIAGNOSIS screen while manually operating the back spindle chuck.

Checking Y0025 and Y0026 on the I/F DIAGNOSIS screen



If Y25 and Y26 remain at "0":

The NC input unit will be faulty.

If Y25 and Y26 change between "1" and "0":

- The pneumatic device is faulty.
- The solenoid valve is faulty.

The possible cause is any of the above conditions.

23.4.5 Recovery from emergency stop button alarm

23.4.5.1 Resetting "EX052 Emergency stop button alarm."

[Procedure]

1. Check if the emergency stop button is pressed.

   The emergency stop button is pressed:
   - After confirming the safety, reset the emergency stop button by turning it clockwise.
     Turn off the NC power and turn it on again.

   The emergency stop button is not pressed:
   - Go to Step 2.
2. Check the LEDs of the safety relay (RY10) (machine of EC specification).

   Only one LED is lit:
   - Any of the circuit protectors (CP1, CP2, CP3, CP4) or the safety relay unit (RY10) is OFF or faulty.
   - The thermal relay (OLS1, OLS4 or OLSCL) is tripped or faulty.
   - Any of the following magnetic switches is melted or faulty:
     MC1, MC2, MC3A, MC3B, MC4, MCCL, RY12, RY302, RY303, RY305
   - The safety relay RY10 is melted or faulty.

   The possible cause is any of the above conditions.

23.4.5.2 Turning on/off the power

To turn on the power

[Procedure]

1. Turn on the main circuit breaker.
2. Press the Power ON button $\boxed{\text{I}}$.
   (Power is supplied to the NC)

To turn off the power

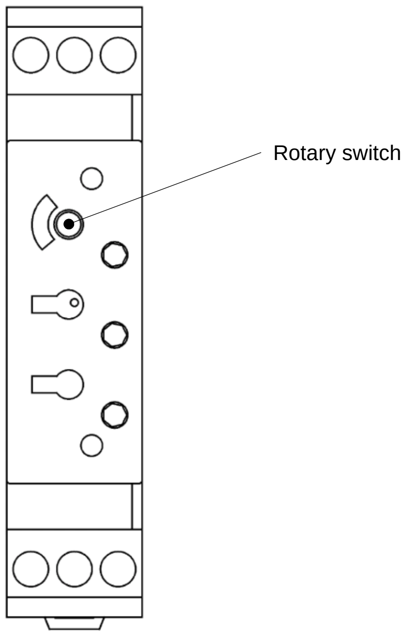
[Procedure]

1. Press the Power OFF button $\boxed{\text{O}}$.
   The screen is blank.
2. Turn off the machine light.
3. Turn off the machine circuit breaker.

> **CAUTION**
>
> Do not touch the keys on the operation panel when tuning on/off the power.
> Failure to observe this instruction can cause machine failure.

23.4.5.3 Setting value of the safety timer (Standard machine)

Power supply to the servo axes and the spindles is shut off when the Emergency Stop button is pressed. The safety timer is provided to set the delay time to shut off the power supply after the servo axes and the spindles are stopped completely.



TMR1

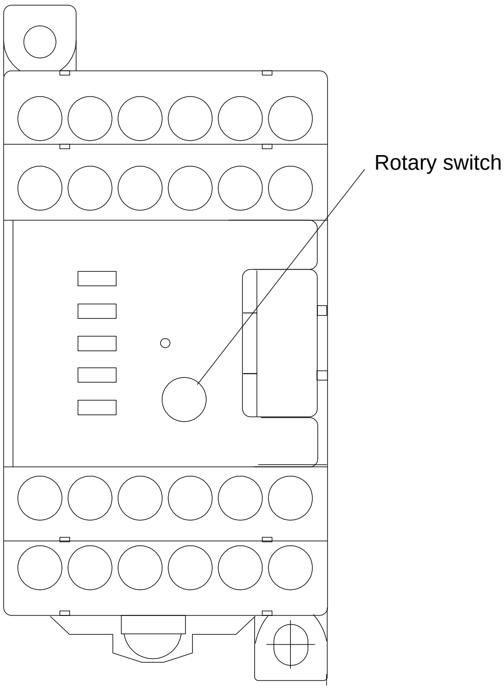
Symbol	Setting value	Use
TMR1	Time setting: 5S Time range switching: x1 Operation mode switching: D	Spindles and NC emergency stop

[Note]

Set the value indicated above to secure the time so that the servo axes and the spindles can stop completely before the power supply is shut off.

23.4.5.4 Setting value of the safety relay (EC machine)

Power supply to the servo axes and the spindles is shut off when the emergency stop button is pressed.
The safety relay has the timer that sets the delay time to shut off the power supply after the servo axes and the spindles are stopped completely.



RY10

Symbol	Setting value	Use
RY10	4.5	Spindles and NC emergency stop

[Note]

Set the value indicated above to secure the time so that the servo axes and the spindles can stop completely before the power supply is shut off.

23.4.6 Recovery from pump overload alarm

23.4.6.1 Troubleshooting after "EX055 Coolant or medium-press. pump overload"

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.

[Procedure]

1. Turn off the NC and disengage the main breaker.
2. Check the thermal relay OLS1, OLS4 and OLSCL (option).

   When it has tripped:

   - The coolant pump motor, spindle cooling device motor or intermediate pressure coolant pump (option) is overloaded. Check the amount and condition of the coolant. Check if the pump suction is injured due to cutting chips blocking.
   - The thermal relay setting is inappropriate.
     The above are probable causes of the alarm. Remove the cause and go to Step 3.

   When it has not tripped:

   ➔ Go to Step 4.
3. Disengage the main breaker, reset the thermal relay, and engage the main breaker.

   The alarm lamp goes off.
4. Check the following failures.
   - Signal cable breakage
   - Imperfect connector plugging-in
   - NC unit failure
   - Circuit protector failure

[Note]

Whether the thermal relay has tripped or not can be judged from the indication for trip status (green). When the thermal relay trips, the indication disappears. If the thermal relay has tripped, pressing the reset bar restores the machine from the trip condition.
To set the current capacity, use the adjustment volume.

Symbol	Rating	Application
OLS1	2.5A	Coolant pump
OLS4	0.56A	Spindle cooling device
OLSCL	3.6A	Intermediate pressure coolant pump

23.4.7 Recovery from overcurrent alarm

23.4.7.1 Troubleshooting after "EX001 200VAC over current alarm. Circuit protector is activated."

DANGER

Be sure to turn off the main breaker of the machine before starting the work.
Failure to do so will result in electric shock or serious personal injury.

[Procedure]

1. Turn off the NC and disengage the main breaker.
2. When circuit protectors (CP1, CP2, CP3, CP4, and CP5) detect overcurrent, they are disengaged. Open the doors of the control console and check the state of these protectors.

   When CP5 detects an over current, the NC power supply does not go on and no alarm is output. Check whether there is a short-circuit in the 200 V circuit.

   The table below shows symbols, ratings, protection voltages, line numbers, and controlled devices for respective circuit protectors.

Symbol	Capacity	Protection voltage	Line number	Controlled devices
CP1	7A	200 VAC, 3-phase	R4, S4, T4	Coolant pump, intermediate pressure coolant pump (optional)
CP2	10A	200 VAC, 3-phase	B6, S6, T6	Loader, spindle cooling device and chip conveyor (option)
CP3	2A	200 VAC, single-phase	R7, S7	Lubricating oil pump, fan motors, workpiece conveyor (optional)
CP4	2A	24 VDC	+24V1	Work light
CP5	15A	200 VAC, single-phase	R10, S10	Stabilized power supply (AVR1, AVR2), servo amplifier module control power supply

   When either CP1, CP2, CP3, or CP4 is disengaged:

   ➔ Go to Step 3.

   When either CP1, CP2, CP3, and CP4 is engaged:

   ➔ Go to Step 9.

3. The main cause must be removed.

   It depends on the circuit protector type.

   ➔ Remove the cause and go to Step 8.

   When the circuit protector (CP1) is disengaged:

   ➔ Remove the cause and go to Step 4.

   When the circuit protector (CP2) is disengaged:

   ➔ Remove the cause and go to Step 5.

   When the circuit protector (CP3) is disengaged:

   ➔ Remove the cause and go to Step 6.

   When the circuit protector (CP4) is disengaged:

   ➔ Remove the cause and go to Step 7.

4. Check whether a failure has occurred in the coolant pump.
   - Coolant pump motor or intermediate pressure coolant pump is faulty or short-circuited.
   - The chip is accumulated around the suction opening of the coolant pump or intermediate pressure coolant pump.
   - Circuit protector failure

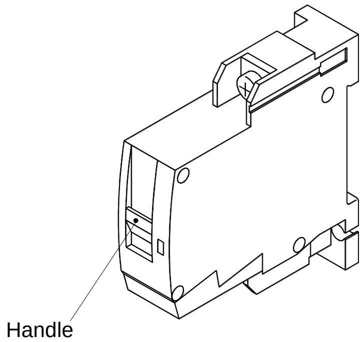
   The above are probable causes of the alarm. Remove the cause and go to Step 8.

5. A failure was detected in the loader power supply circuit.
   - The power circuit of bar loader is short-circuited.
   - There is a failure or short-circuit in the spindle cooling device motor
   - The power circuit of chip conveyor is short-circuited.
6. Check the lubrication oil pump and the fan motor for malfunctions.
   - Lubricating oil pump motor is faulty or short-circuited.
   - Workpiece conveyor is faulty or short-circuited.
   - Control unit outdoor air intake fan motor (FAN 3, 4) are not turning (are locked)
   - Amplifier cooling fan motor (FAN 2) is not turning (is locked)
   - Spindle cooling device fan motor (FAN 11) is not turning (is locked)
   - Circuit protector failure

   The above are probable causes of the alarm. Remove the cause and go to Step 8.
7. An error is detected on the internal illumination lamp.
   - 24 VDC circuit of internal illumination lamp is short-circuited.
   - The internal illumination lamp is faulty or short-circuited.
   - Circuit protector failure

   The above are probable causes of the alarm. Remove the cause and go to Step 8.
8. Disengage the circuit protectors once and engage them again, and engage the main breaker.

   The alarm lamp goes off.



9. Check the following failures.
   - Signal cable breakage
   - Imperfect connector plugging-in
   - NC unit failure
   - Circuit protector failure

23.4.8 Troubleshooting when 24 VDC power supply fails

[Procedure]

1. Turn off the main breaker of the machine according to the procedure in *<5.1 Turning On/Off the power>* in the Operator's Manual, then open the door of the front electric device box to check the fuse.

When the fuse has 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.

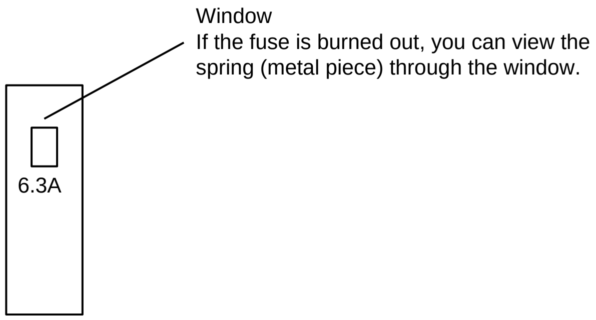
The possible cause is either of the above two conditions. Remove the cause and go to step 2.

When the fuse has 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 go to step 3.

3. Close the door of the front electric device box, then turn on the machine according to the procedure in *<5.1 Turning On/Off the power>* in the Operator's Manual.



[Note]

If the same status recurs after operation is restarted, remove the main cause of the alarm.

23.4.9 Troubleshooting after main spindle, back spindle, and rotary tool alarms

EX003	Main spindle motor alarm
EX004	Back spindle motor alarm
EX005	Rotary tool on gang tool post motor alarm
EX008	Front rotary tool on opposite tool post motor alarm
EX009	Back rotary tool motor alarm

When one of the above PC alarms occurs, the machine stops and an NC alarm also occurs. For details and causes of alarms, see the NC Alarm Table of the Instruction manual of the NC manufacturer.

23.4.10 Troubleshooting after spindle related alarms

23.4.10.1 Troubleshooting after "EX109 Main spindle speed fluctuation alarm."

[Procedure]

1. Manually rotate the main spindle with the main circuit breaker off.

If the main spindle won't turn or feels extremely heavy:

- The main spindle may have a mechanical failure.

When the main spindle rotates normally:

- Go to step 2.

2. Check if the main spindle is heavily overloaded with the tool (drill) which has been chipped, broken, or worn down.

If the tool (drill) has been chipped, broken, or worn down:

- Replace or re-polish the tool.

Normal:

- Go to step 3.

3. Turn on the power supply of the machine and manually turn the main spindle to check if the spindle speed S1 displayed on the screen is 0.

If S1 is 0:

- The encoder cable has been disconnected or the connector is loose.
- The encoder is faulty.
- The NC unit is faulty.

The possible cause is any of the above conditions.

Visually check the encoder cable for discontinuity and check for looseness of its connector.
Call the Cincom Service Office if you find no problem with the encoder cable.

If the S1 value varies:

- Go to step 4.

4. Remove the material first. Execute the spindle speed change detection OFF command (M97) in MDI mode, then execute M3 S1=1000 to turn the spindle at 1000 min^{-1} to check whether the spindle speed S1 on the screen falls within the range of 1000 $\pm$100 min^{-1}.

If the S1 value is within 1000 $\pm$100 min^{-1}:

- The spindle speed change detection level has not been set to the standard value. For changing the setting, refer to see *<13.6 Spindle Speed Change Detection Function>* in the Programmer's Manual.

If the S1 value is outside 1000 $\pm$100 min^{-1} or the main spindle motor alarm is raised:

- The encoder is faulty.
- The spindle speed has not been adjusted appropriately.
- The spindle motor is faulty.
- The encoder cable has been disconnected or the connector is loose.
- The spindle servo amplifier is faulty.
- The NC unit is faulty.

The possible cause is any of the above conditions.

Visually check the encoder cable for discontinuity and check for looseness of its connector.
Call the Cincom Service Office if you find no problem with the encoder cable or if the procedure for recovery from the main spindle motor alarm fails to cancel the alarm.

23.4.10.2 Troubleshooting after "EX110 Back spindle speed fluctuation alarm."

[Procedure]

1. Manually turn the back spindle with the main circuit breaker off.

   If the back spindle won't turn or feels extremely heavy:

   - The back spindle may have a mechanical failure.

   When the back spindle turns normally:

   - Go to step 2.

2. Check if the back spindle is heavily overloaded with the tool (drill) which has been chipped, broken, or worn down.

   If the tool (drill) has been chipped, broken, or worn down:

   - Replace or re-polish the tool.

   Normal:

   - Go to step 3.

3. Turn on the power supply of the machine and manually turn the back spindle to check if the spindle speed S2 displayed on the screen is 0.

   If S2 is 0:

   - The encoder cable has been disconnected or the connector is loose.
   - The encoder is faulty.
   - The NC unit is faulty.

   The possible cause is any of the above conditions.

   Visually check the encoder cable for discontinuity and check for looseness of its connector.
   Call the Cincom Service Office if you find no problem with the encoder cable.

   If the S2 value varies:

   - Go to step 4.

4. Remove the workpiece first. Execute the spindle speed change detection OFF command (M87) in MDI mode, then execute M23 S2=1000 to turn the spindle at 1000 min–1 to check whether the spindle speed S2 on the screen falls within the range of 1000 ±100 min^{-1}.

   If the S2 value is within 1000 ±100 min^{-1}:

   - The spindle speed change detection level has not been set to the standard value. For changing the setting, refer to see *<13.6 Spindle Speed Change Detection Function>* in the Programmer's Manual.

   If the S2 value is outside 1000 ±100 min–1 or the back spindle motor alarm is raised:

   The encoder is faulty.

   - The spindle speed has not been adjusted appropriately.
   - The back spindle motor is faulty.
   - The encoder cable has been disconnected or the connector is loose.
   - The back spindle servo amplifier is faulty.
   - The NC unit is faulty.

   The possible cause is any of the above conditions.

   Visually check the encoder cable for discontinuity and check for looseness of its connector.
   Call the Cincom Service Office if you find no problem with the encoder cable or if the procedure for recovery from the main spindle motor alarm fails to cancel the alarm.

23.4.10.3 Procedure if the message "EX592 Knock-out can not be advanced to correct position." appears

[Procedure]

1. Check if the workpiece is caught with the back spindle.

When caught:

- Remove the workpiece.

When not caught:

- Diameter of knock-out jig is larger than the chuck size.
- The knock-out rod is bent.

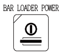
The possible cause is either of the above two conditions. Remove the cause. If you find no problem, call the Cincom Service Office.

23.4.11 Recovery from external device alarms

23.4.11.1 Troubleshooting after "EX133 Bar loader is not ready."

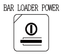
[Procedure]

For a dedicated bar loader:

1. Check the indicator on the Power key 

of the bar loader.

If the indicator is off:

- Press the Power key 

of the bar loader.

If the indicator is on:

- The NC unit may be faulty. Call the Cincom Service Office.

For another bar loader:

2. Check the power supply of the bar loader.

When the power is off:

- Turn the power on.
- Turn the bar loader torque ON (cycle start).

If the power is on:

- Turn the bar loader torque ON (cycle start).
- The cable has been disconnected or erroneously wired.
- The NC unit is faulty.
- The bar loader is faulty.

The possible cause is any of the above conditions. Follow the bar loader maintenance procedure to check the bar loader and cable. Call the Cincom Service Office if neither of them has any problem.

23.4.11.2 Troubleshooting after "EX101 Bar loader alarm"

[Procedure]

1. Check the conditions of the bar loader.

If an alarm has been raised:

- Reset the alarm.

If no alarm has been raised:

- The bar loader has been wired erroneously or the cable has been disconnected.
- The NC unit is faulty.

The possible cause is any of the above conditions. Check the cable. Call the Cincom Service Office if the cable has no problem.

[Note]

If the machine is equipped with an automatic bar loader (excluding a dedicated bar loader) contact the bar loader manufacturer.

23.4.11.3 Troubleshooting after "EX201 Bar stock empty"

[Procedure]

1. Check whether there are materials on the bar loader.

When the bar loader has no material:

- Replenish the bar loader with materials.

If the bar loader has materials:

- The stock bar detection sensor is faulty.
- The bar loader has been wired erroneously or the cable has been disconnected.

The possible cause is any of the above conditions. Check the cable and sensor. Call the Cincom Service Office if neither of them has any problem.

[Note]

If the machine is equipped with an automatic bar loader (excluding a dedicated bar loader) contact the bar loader manufacturer.

23.4.11.4 Troubleshooting after "EX208 Chip conveyor overload."

[Procedure]

1. Check if the chip conveyor has been overloaded with clogging chips or other foreign matters.

If the chip conveyor has been overloaded:

- Remove the cause.

Normal:

- Refer to the Operator's Manual for the chip conveyor.

[Note]

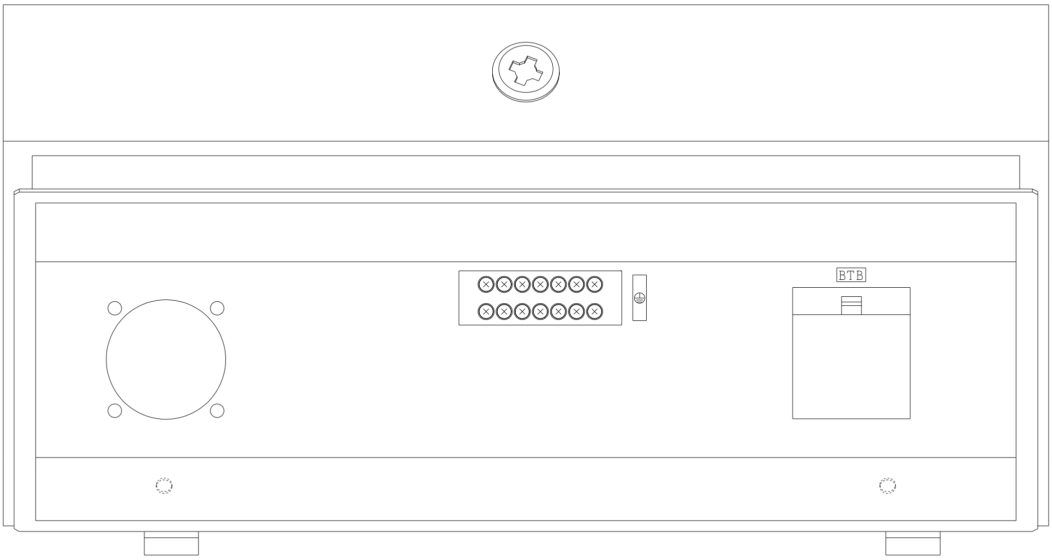
The chip conveyor requires cleaning at regular intervals. To clean the conveyor, detach it from the machine body to remove fine chips.

23.4.12 Troubleshooting after OP076 Device battery low level Alarm

> **⚠ CAUTION**
>
> Before replacing the battery, make sure that the NC power is on.
> If battery is replaced while the NC power is off, "EX401 Detection of the machine moving" alarm will occur.

[Procedure]

1. Prepare four pieces of new size AA alkaline battery.
2. Turn on the NC power supply.
3. Insert the supplied lock handle key in the lock handle (1 location) on the operation panel and turn it 90 degrees clockwise to open the operation panel.



Rear View of the Operation Panel

4. Remove the cover from the battery holder, then replace the four AA batteries in the battery holder with new ones. When putting a battery in the battery holder, make sure to put one in the correct orientation.
5. Press the reset key 

to cancel the alarm. After that ensure that the alarm does not occur again in 5 seconds or more.

When the alarm does not occur again in 5 seconds:

➔ Go to step 7.

When the alarm is not reset or the alarm occurs again:

- The battery is not set in the correct orientation (+ / – terminals).
- The set number of batteries is less than four.
- The battery terminals are not correctly engaged with the mating terminals.
- Set batteries are not new.

The possible cause is one of the above four conditions. Check the battery holder and the set batteries. Call the Cincom Service Office if neither of them has any problem.

6. Close the operation panel and insert the supplied lock handle key in the lock handle (1 location) on the operation panel and turn it 90 degrees counterclockwise.

[Note]

Although the machine can be operated continuously even after the occurrence of the alarm, alarm "EX401 Detection of the machine moving" may be detected if the machine is used for more than two weeks after the occurrence of the alarm.
(Alarm EX401 occurs when the battery voltage becomes lower than the specified limit value.)

23.4.13 Recovery from other alarms

23.4.13.1 Troubleshooting after "EX108 Cycle time alarm"

[Procedure]

1. Check if the program cycle time is exceeding the set time (the standard setting is 30 minutes).

If the cycle time exceeds the set time:

- Extend the set time.
  To measure the actual cycle time, run the program for one cycle with the cycle time check disabled (by setting the "Bit 1" of parameter #1 on the BIT SELECTION PARAMETERS screen to "1" according to see *<23.6.2 PLC bit selection parameter setting>*.

CAUTION

Disabling the cycle time check prevents the machine from raising an alarm even if the machine causes an unexpected failure. Be sure to enable the cycle time check except when measuring the cycle time for determining the setting.

When the cycle time does not exceed the set time:

➔ Go to Step 2.

2. Run the program for one block, then check the NC operation status display when the machine makes no response with the Start key lamp remains on, to pressing the Start key.

Operation Status

Symbol	Description
EMG	Emergency stop
RST	Resetting the NC
RDY	Ready for operation
	During automatic operation
SYN	Being queued for synchronization
CRS	Waiting until axes cross
STP	Being stopped
HLD	Being hold

The status display is SYN for both axis control groups 1 and 2:

- Axis control groups 1 and 2 are being stopped by different queue commands.
  Modify the program so that the queue commands for axis control groups 1 and 2 are of the same command sequence.

When one of axis control groups 1 and 2 has no status display (automatic operation) and the other has a status display of SYN, or when both of them have no status display:

- The block being executed for the axis control group with no status display contains a command for which no completion signal has been issued.
  If the same block contains multiple commands which can be specified alone, specify them in different blocks and execute them one by one to identify the command that stops operation.

If the machine stops with a cutting block such as G1, G2, or G3:

A per rotation feed command (G99) may be suspected while neither the spindle nor the back spindle is rotating.

If the cutting block contains no rotation command, insert one before that block. If the cutting block contains one, see *<23.5.2 Main spindle does not rotate during automatic operation>* to remove the cause of stopping operation, then run the program again.

If the machine has stopped with an M code:

The M code may be illegal or the condition for completing the M code may not have been satisfied. After making sure that the specified M code is not illegal, check whether the command-specified operation has been performed or not (for example, whether the chuck has been closed completely, stopped prematurely, or left open with M6 specified) Then, take action appropriately depending on the result of the check.

If the machine has stopped with any other command:

Call the Cincom Service Office.

When the program is run to the end:

- The program may not contain M02.
  Check the program and insert M02 if it is not included.

If the program sometimes results in an alarm:

The program stops running at such a G or M code as described above.

- The "door open" signal is issued because of imperfect contact of the door switch while the door has been closed.

[Note]

- Select a PLC constant of 18003 (in 0.1 seconds) to set the cycle time.
- The cycle time can be set up to 32400 (54 minutes).
- See *<23.6.1 PLC constant setting>* for how to read the PCL-Data screen.
- The M codes for resetting the cycle time count are M1, M56, M57, and M2.
- The cycle time is checked only in automatic operation mode.

23.4.13.2 Troubleshooting after "EX105 Coolant Discharge Alarm"

[Procedure]

1. Press the Reset key 

to cancel the alarm and adjust the coolant flow rate according to the adjustment instructions for the coolant flow rate detector (U54R), which are provided after this procedure.

If the alarm is raised as soon as you press the coolant switch to adjust the flow rate (Process 3):

- The coolant tank may be short of coolant. Check the oil level and replenish the tank if coolant is short.
- The filter in the coolant tank may have been clogged, preventing coolant from reaching the suction opening of the pump. Clean inside the tank.
- The flow rate sensor may be defective. Fully turn the "set value adjusting potentiometer" counterclockwise. (Process 7)

If the "alarm output indicator LED" does not come on:

- Replace the flow rate sensor because it seems to be defective.

If the "alarm output indicator LED" comes on:

- The flow rate seems to have not been adjusted correctly. Adjust the flow rate over again.

When the flow rate can be adjusted as described in the procedure:

- The filter in the coolant tank may have been clogged, taking time for coolant to return to the suction opening of the pump. Clean inside the tank.
- The flow regulating valve may have been too tightened or the "set value adjusting potentiometer" may have not been adjusted correctly. Leave the coolant supply system as it is for further observation. If the alarm persists, call the Cincom Service Office.

[Note]

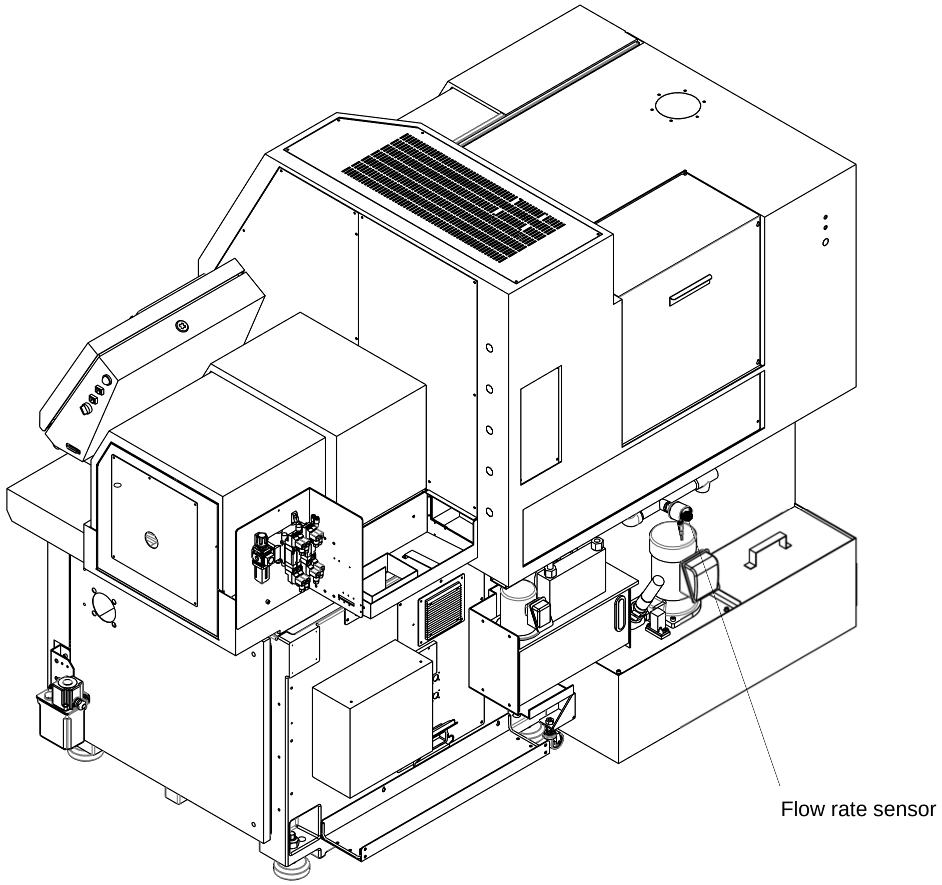
The coolant flow rate detector is an option.

Adjusting the coolant flow detector (U53R)

If the coolant flowrate decreases for some reason, coolant may not expand over the cutting point, resulting in fire or other accidents. This unit detects such decrease of flowrate to stop the machine operation automatically.

[Note]

- The coolant flowrate varies depending on the temperature. To distinguish the change in flowrate under normal conditions from that in abnormal state, set the alarm flowrate to not more than 2/3 of normal value. Define the normal flowrate as the value determined by valve full-open state.
- When the initial coolant temperature is 15°C or less, this detector does not work. Therefore, although the machine must be started early in the morning of the winter, for example, this device is turned off until the coolant temperature is elevated to at least 15°C. The operator must watch the temperature. Once the coolant temperature reaches 15°C, this detector works and maintains the active state and even if the temperature is dropped by environment, the device keeps the function enabled.

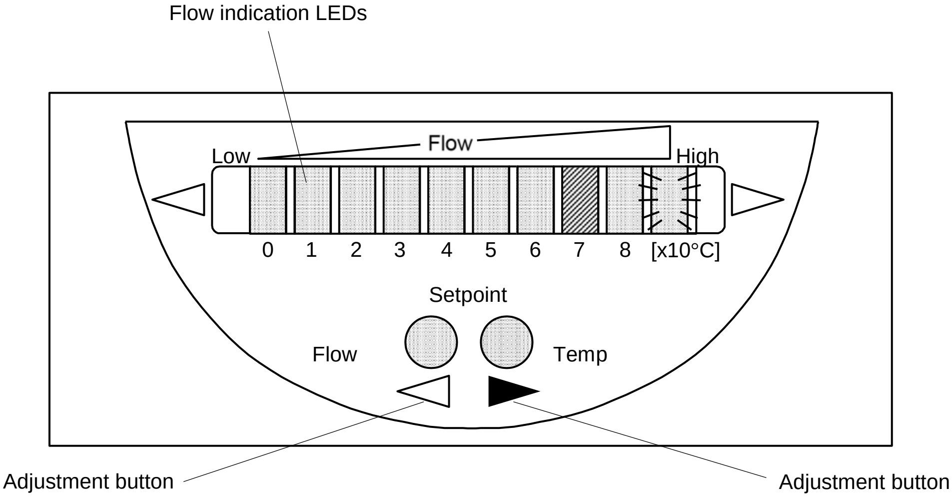


Position coolant flow detector

Names and functions of components

The coolant flow rate detector consists of the flow rate sensor and the coolant temperature setting unit.

Coolant flow sensor



- Flow indication LEDs

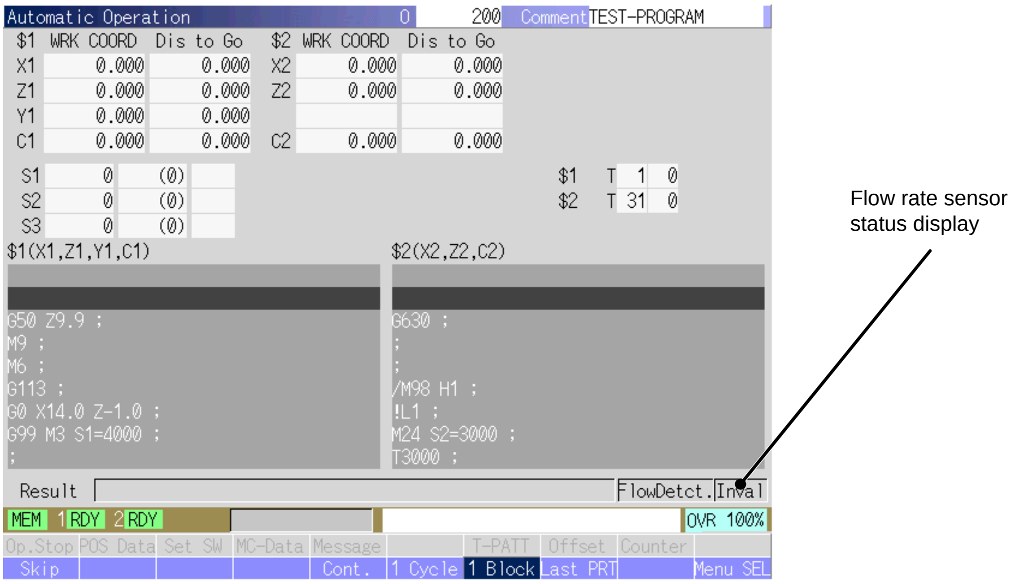
  In normal operation, the LED shows the current flowrate in green LED.
  The Set point LED lights orange while coolant is discharged in normal rate, and red in slow rate.
  The LEDs are used to indicate the specified flow rate, current temperature, specified temperature, or alarm.

- Adjustment buttons ◁ ▶

  Use these buttons to change indications and setting values.

- Active flow rate sensor status display (Lower right corner of the screen)

  Indicates the flow rate sensor function status. The flow rate is monitored while the Display is enabled, but not when the Display is disabled. (Standard temperature of this machine: 15°C or more.)
  If Display is disabled, pay strict attention to run the machine. Do not leave the machine while running.



Setting method

Setting method

1. Set the High Flow rate.
2. Set the Low Flow rate.
3. Set the flowrate switch point.
4. Set the switch point of temperature to activate the flowrate sensor.

* At the shipment, the temperature to activate the flowrate sensor is set to 15°C. Accordingly, you do not need to set it again in normal operation. If you restore the default temperature setting (4°C), you need to set it to 15°C.

[Procedure]

Setting High Flow

Set the indication of maximum flowrate to the upper limit.
(All the LEDs except the switch point LED light green.)

Set High Flow in the following procedure.

1. Press the Coolant button, and confirm that all nozzles discharge the coolant at maximum flowrate.
2. Hold the button for a certain time.
   LED9 goes on, then starts flashing five seconds later.
3. When LED9 starts flashing, release the button.
   The value for High Flow is set, and the system goes back to operation mode.

Setting Low Flow

[Note]

Be sure to set Low Flow after setting for High Flow has completed.

1. Close the three nozzles at the upper part of the cutting room. Fully open the other nozzles.
   Press the Coolant button to let the coolant flow.
2. Hold the button for a certain time. LED0 goes on, then starts flashing five seconds later.
3. When LED0 starts flashing, release the button.
   The value for Low Flow is set, and the system goes back to operation mode.

Setting Setpoint (SP1: Flowrate Monitoring)

Set the Setpoint in the following procedure:

1. Press the ◁ button for a short time.

The LED for the current switch point starts flashing.

2. Use either of the ◁ (toward left) or ▶ (toward right) button to move to the desired position. About five seconds have passed after the button is released, the position currently being indicated is set as the new switch point. Then, the machine goes back to operation mode.
(Set the range appropriately, using the LED7 as a guideline.)

Setting Setpoint (SP2: Temperature Monitoring)

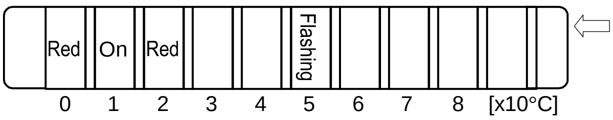
At the shipment, the temperature to activate the flowrate sensor is set to 15°C. Accordingly, you do not need to set it again in normal operation.
However, if you reset the flowrate sensor, the default temperature setting (4°C) is restored. Use the following procedure to set it to 15°C.

1. Press the ▶ button once to let the current temperature and temperature setting to be displayed.
Press the ▶ button to enter into the temperature setting mode.

Lighting green (or orange): Indicates the current coolant temperature in unit of 10°C.

Flashing green (or orange): ndicates the current coolant temperature in unit of 1°C.

(When the current coolant temperature is higher than the initial temperature: Orange LED.
When the current coolant temperature is lower than the initial temperature: Green LED)



Indicates the setting value in green or orange. (In the example, current setting is 15°C.)
Red LED indicates the current coolant temperature. The rightmost position shows the 10°C unit.
(In the example, current coolant temperature is in the range between 20 and 29°C.)

2. Press the ◁ (drop 1°C) or ▶ (raise 1°C) button several times until it reaches 15°C. The LED for indicating 10°C unit is turned on or off automatically.
About five seconds have passed after the button is released, the position currently being indicated is set as the new switch point. Then, about five seconds later, the machine goes back to operation mode to monitor the flowrate.

[Note]

- If the button is not pressed for five seconds, the new setpoint is set, the temperature monitoring setpoint is displayed with new setting. After the additional five seconds later, the system goes back to flowrate monitoring mode.
- If an error is detected in setting procedure, all LEDs flash red. The system goes back to operation mode with the previous setting. When you finish setting, make sure that no LED is flashing red.

Operation check / Maintenance

1. After the machine is turned on, all the flow indication LEDs go on once. Afterwards, LEDs go off one by one, and the system goes to operation mode.
2. Turn the coolant discharge on.
3. Make sure the flowrate sensor shows the coolant temperature of at least 15°C.

   (Press the ▶ button once to check the current coolant temperature indicated by the red LED.)
   The red LED at the rightmost position indicates the coolant temperature in unit of 10°C.

   In the example on page 23-47, current coolant temperature is in the range between 20 and 29°C.
4. Check if the flowrate sensor works normally (by indications in operation mode).

   The green LED shows the flowrate change.

Setpoint indication:	Output ON:	Orange LED
	Output OFF:	Red LED

[Note]

- If only LED 9 is flashing green while all other LEDs are lighting green, it indicates an excess flowrate.
- If the setpoint lights in red and LED0 lights in green, it indicates an insufficient flowrate.

Error indication

If the output 1 (flowrate monitoring) is short-circuited, indication of operation mode and five red LEDs (LED0 to LED4) are alternately displayed.

If the output 2 (temperature monitoring) is short-circuited, indication of operation mode and five red LEDs (LED5 to LED9) are alternately displayed.

Lock / Unlock

You can lock the setting value electrically to protect the settings from being modified.

Press the ◁ and ▶ buttons simultaneously for about 10 seconds lock (or unlock) the setting. (After 10 seconds, all LEDs go off for about 1 second.)

23.4.13.3 Troubleshooting after "EX107 Tool bit breakage alarm. Cut off tool is broken."

[Procedure]

1. Make sure there is no material left without being cut off.

   If there is material not cut off:
   - Replace the cut-off tool.

   If there is no material:

   ➔ Go to step 2.

2. Check if chips have been jammed around the guide bushing.

   If chips have been jammed:
   - Remove chips.

   When chips have not been jammed:
   - The sensor is poorly adjusted.
   - The sensor is failed.
   - The sensor cable is disconnected.

   The possible cause is any of the above conditions. Check the cable and sensor.
   Call the Cincom Service Office if neither of them has any problem.

[Note]

The cut-off tool breakage detection device is an option.

23.4.13.4 Troubleshooting after "EX202 Lubrication oil empty alarm or EX204 Oil-air lub alarm"

[Procedure]

1. Check the lubricating oil level.

   If the lubricating oil level is too low:
   - Add lubricating oil.

   When the lubricating oil level is acceptable:
   - The sensor is failed.
   - The cable has been disconnected.

   The possible cause is any of the above conditions. Check the cable and sensor. Call the Cincom Service Office if neither of them has any problem.

23.4.13.5 Troubleshooting after "EX203 Coolant oil alarm"

[Procedure]

1. Check coolant level.

If the coolant level is too low:

- Add coolant.

When the coolant level is acceptable:

➔ Go to step 2.

2. Check chips.

If clogging chips prevent the tank from collecting oil:

- Remove chips.

When chips are not clogging:

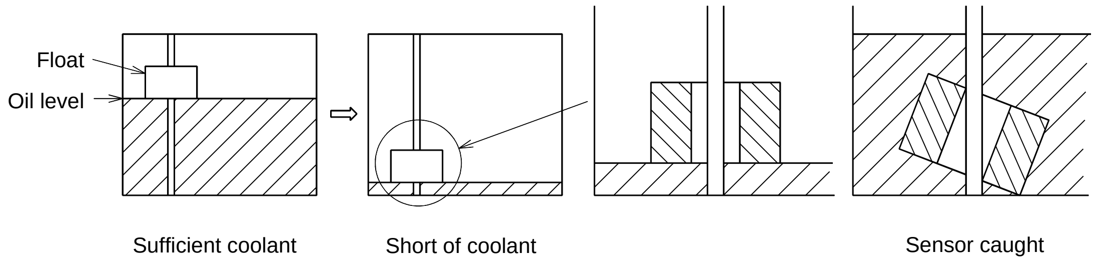
- The sensor is failed.
- The cable has been disconnected.

The possible cause is any of the above conditions. Check the cable and sensor. Call the Cincom Service Office if neither of them has any problem.

The sensor has a float which goes down as the oil level is lowered, raising an alarm just before the float reaches the bottom of the tank.

Note, however, that the float reaching the bottom of the tank with no coolant left may be slanted and not go up next time the tank is charged with coolant.

Since the sensor raises an alarm even with coolant in the tank, in this case, remove the sensor from the tank and set it back again with the float set horizontally.



23.4.13.6 Troubleshooting after "Z52 Battery fault xxxxx"

It is required to replace the battery which is used for backup of the NC parameters stored in NC control unit and the machining program.

DANGER

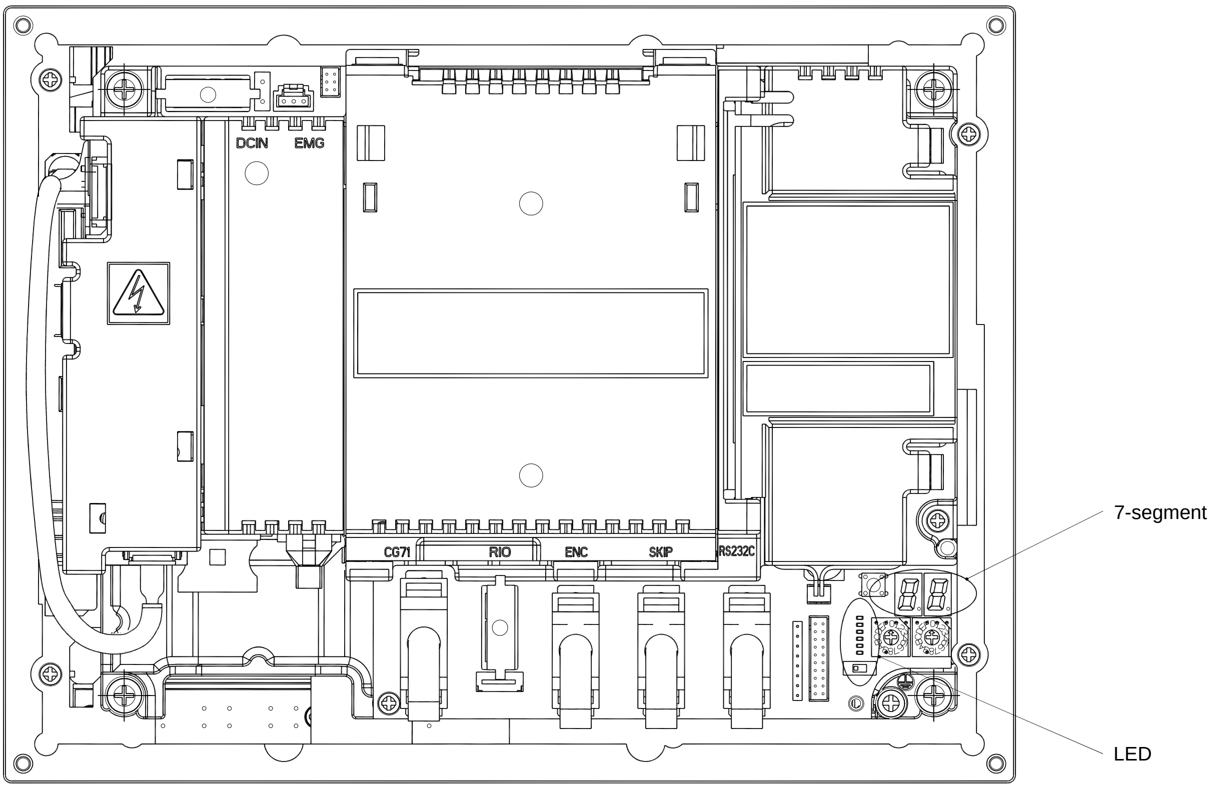
Be sure to turn off the main breaker of the machine before starting the work.
Failure to do so will result in death or serious personal injury from electric shock.

[Note]

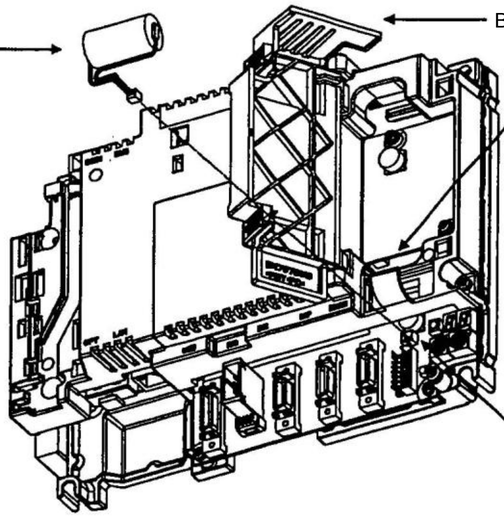
Be sure to complete replacement work within 30 minutes after powering off the NC unit.
If the new battery is not connected within 30 minutes, the backup data will be lost.

[Procedure]

1. Turn off the main breaker of the machine according to the procedure explained in *<5.1 Turning On/Off the power>* in the Operator's Manual. After that open the operation panel and confirm the location of the NC control unit.
2. Make sure that LEDs and 7-segment indicators on NC control unit are unlit.



3. Pull the right side of the front cover of the NC control unit toward you, and open the front cover.



4. Remove the connector coming from the battery out from the BAT connector on the NC control unit. Then, take the battery out of the battery holder.

5. Put the new battery into the battery holder. Then connect the connector coming from the battery to the BAT connector on the NC control unit.

6. Close the front cover of the NC control unit. Confirm that the cover is securely caught to the claw and click sound is heard.

7. Close the operation panel door.

Continous battery backup time	:45,000 hours (at ordinary temperature. Shortened by the temperature deviation)
Useful life of battery	:Approx. 5 years from manufactured date
Battery for replacement	:Q6BAT BK0-C10811H03 (made by Mitsubishi Electric)

23.4.13.7 Troubleshooting after "S52 Battery voltage drop 009F"

It is required to replace the battery which is used for backup of the position data stored in absolute position detector of AC servo motor.

[Note]

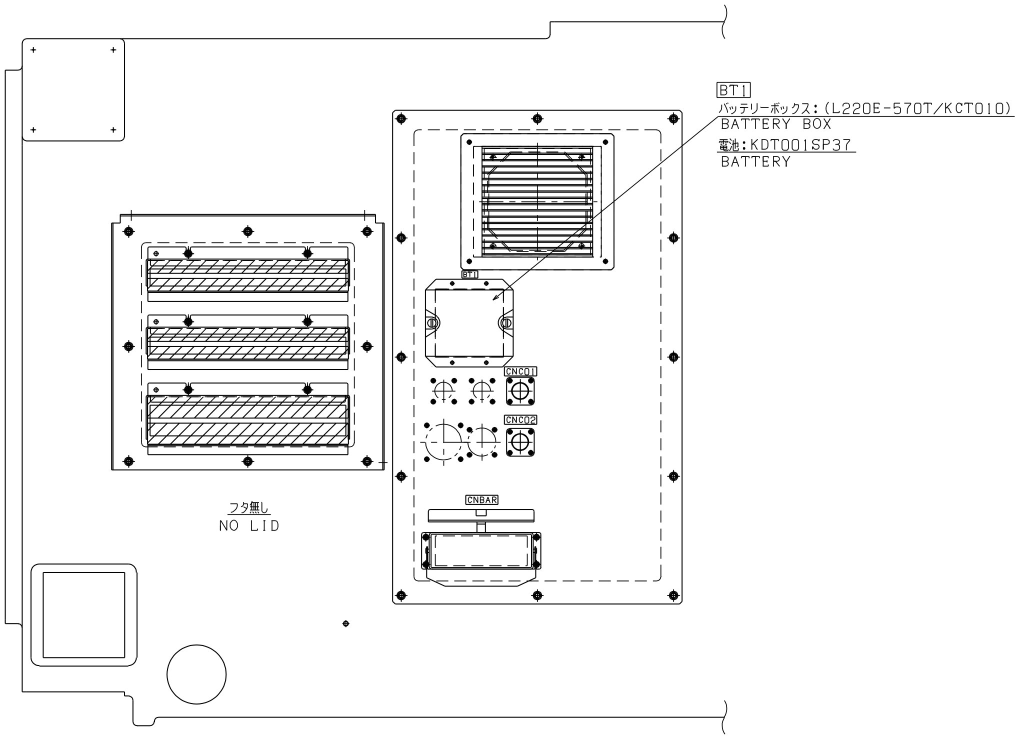
Conduct this work with the NC power being on. Otherwise, the stored absolute position data will be lost.

> **DANGER**
>
> Replacement of the battery for backup of servo motor position data shall be conducted with the powers of machine and the NC unit being on. Do not perform any other operation than the procedures described below.
> Otherwise, you may be electrically shocked, resulting in death or serious personal injury.

[Procedure]

1. Turn on the main breaker of the machine according to the procedure in *<5.1 Turning On/Off the power>* in the Operator's Manual.

2. Loosen the screw securing the battery box, and remove the cover.



3. Take the old battery out of the battery box, and put the new battery into the box. In this time, pay attention to orientation of the battery.

4. Close the cover of the battery box and tighten the screw to secure the battery box.

Continous battery backup time	:	10,000 hours (at ordinary temperature. Shortened by the temperature deviation) Replace once a year the battery.
Useful life of battery	:	Approx. 1 years from manufactured date
Battery for replacement	:	LR20 (size D alkaline dry cell) × 4

23.4.14 Recovery from thermal displacement correction device alarm

23.4.14.1 Troubleshooting after "EX122 T.D.C. device alarm", "EX131 T.D.C. data alarm"

[Procedure]

1. Press the reset key RST to cancel the alarm.
2. Turn off the NC power supply, then turn off the main breaker of the machine.
3. Turn on the main breaker, and then turn on the NC power supply.
4. Restart operation and check if any alarm occurs between the first and third cycle.

Alarm occurs:

- The device may be faulty. Call the Cincom Service Office.

23.4.14.2 Troubleshooting after "EX132 T.D.C. sensor alarm (○○ch）"

[Procedure]

1. Press the reset key RST to cancel the alarm.
2. Turn off the NC power supply, then turn off the main breaker of the machine.
3. Turn on the main breaker, and then turn on the NC power supply.
4. Restart operation and check if any alarm occurs between the first and third cycle.

Alarm occurs:

- The device may be faulty. Call the Cincom Service Office.

23.5 Mechanical Errors

This section describes the troubleshooting procedures for mechanical errors or failures which do not result in alarms. If the cause of a problem cannot be identified or the problem cannot be solved, call the Cincom Service Office.

23.5.1 Machine fails to start

[Procedure]

1. Check whether the Start button is on.

If the indicator is on:

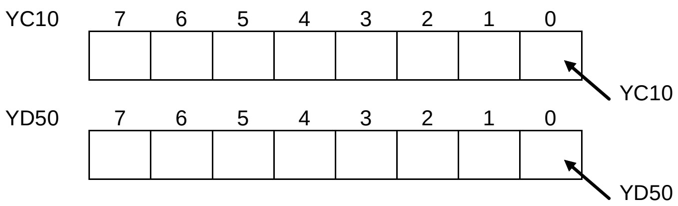
- The override may be 0%.

If the indicator is off:

➔ Go to step 2.

2. Check YC10 ($1) and YD50 ($2) on the I/F DIAGNOSIS screen with the Start button depressed.

Checking YC10 and YD50 on the I/F DIAGNOSIS screen



YC10=1, YD50=1:

- The NC unit may be faulty.

YC10=0, YD50=0:

➔ Go to step 3.

3. Check whether MDI or AUTO mode has been selected.

If none of those mode has not been selected:

- Select the mode in which the machine can be started automatically.

If MDI or AUTO mode has been selected:

➔ Go to step 4.

4. Check whether setting switches 18 ($1 cycle start) and 19 ($2 cycle start) have been checked.

If the switches have not been checked:

- Check the switches.

If the switches have been checked:

➔ Go to step 5.

5. Check whether an alarm has occurred.

If an alarm has occurred:

- Reset the alarm.

If no alarm has occurred:

- The Start button is faulty or the cable has been disconnected.

[Note]

- See <*23.2 Interface Diagnosis*> for how to read the I/F DIAGNOSIS screen.
- See <*6.8.6 Setting Switches*> in Operator's Manual for how to read the Set SW screen.

23.5.2 Main spindle does not rotate during automatic operation

[Procedure]

1. Enter the main spindle rotation command in MDI mode. (M03 S1=1000).

If the spindle rotates:

- The program may contain a command the specified main spindle rotating command may not be in the valid format (for example, with S1=○○○○ omitted).

If the spindle does not rotate:

- The NC unit is faulty.
- The motor is faulty.
- The drive unit is faulty.
- The cable has been disconnected.

The possible cause is any of the above conditions. Check the cable and sensor.
Call the Cincom Service Office if neither of them has any problem.

23.5.3 Back spindle does not rotate during automatic operation

[Procedure]

1. Enter the back spindle rotation command in MDI mode. (M23 S2=1000).

If the spindle rotates:

- The program may contain a command the specified back spindle rotating command may not be in the valid format (for example, with S2=○○○○ omitted).

If the spindle does not rotate:

- The NC unit is faulty.
- The motor is faulty.
- The drive unit is faulty.
- The cable has been disconnected.

The possible cause is any of the above conditions. Check the cable and sensor.
Call the Cincom Service Office if neither of them has any problem.

23.5.4 Rotary tool does not rotate during automatic operation

[Procedure]

1. Enter the rotary tool rotation command in MDI mode. (M80 S3=1000).

If the rotary tool rotates:

- The rotary tool rotating command specified in the program may not be in the valid format (for example, with S3=○○○○ omitted).

If the rotary tool does not rotate:

- The NC unit is faulty.
- The motor is faulty.
- The drive unit is faulty.
- The cable has been disconnected.

The possible cause is any of the above conditions. Check the cable and sensor. Call the Cincom Service Office if neither of them has any problem.

23.5.5 Guide bushing alarms

23.5.5.1 Fretting problem

[Procedure]

1. Check the cutting conditions.

1-1. Check whether the spindle speed, feed rate, coolant, and the material and grinding state of the tool are appropriate for the workpiece to be cut.

1-2. Check whether the tools and tool bits are appropriate with proper rigidity without being abnormal, such as being worn down, chipped, or damaged.

If the cutting conditions are inappropriate:

- Change cutting conditions.

When the cutting conditions are appropriate:

➔ Go to step 2.

2. Check whether the clearance and contact between the guide bushing and the material are appropriate.

If the clearance and contact states are inappropriate:

Remove and clean the guide bushing and drawbar. Mount them back and perform set clearance adjustment.

Lap the guide bushing and material to fit them together.

- Replace the guide bushing.

When the clearance and contact states are appropriate:

➔ Go to step 3.

3. Check whether the machine has no abnormal vibration with no sliding part rattling and with the rotary guide bushing unit maintained normally.

If the cutting conditions are inappropriate:

If the machine has a problem:
Problem with the rotary guide bushing
Check if:
– the driving timing belt has been worn.
– the support bearing is abnormal.

- Contact Cincom Service Office if the machine has abnormal vibration, a sliding part is rattling, or if the rotary guide bushing unit is abnormal.

23.5.5.2 Too large machining diameter fluctuations

[Procedure]

For normal level of dispersion in the diametral or longitudinal direction

1. Check the cutting conditions as in steps 1 to 3 in see <*23.5.5.1 Fretting* problem>.

[Procedure]

Dispersion found with the lapse of time

1. Large fluctuation in the initial state.
   - Check the initial wear of tools.
   - Warm up the machine before re-machining.
2. Long-time dispersion which seems to be thermal variation.
   - Check whether the room temperature is kept constant.
   - Check whether turning on and off of air conditioning has any effect on the machine.
   - Check whether the changes in temperature in the morning and evening have any effect on the machine.

[Procedure]

1. Check the following items as factors other than the above.
   - Check if ball screws in the mechanical system are abnormal in clearance and bearing.
   - Check if the spindle is abnormal such as having looseness, runout, or end face runout in the thrust and radial directions.
   - Check if the mechanical or driving system is abnormal in any other point.

   If the cutting conditions are inappropriate:

   - Call the Cincom Service Office.

23.5.5.3 Poor machining roundness

[Procedure]

1. Check the roundness of the material.
   - If the material is poor in roundness, the machined workpiece is also poor in roundness.
   - Machining a profile or polygonal material results in a workpiece poor in roundness.

   When the cutting conditions are appropriate:

   ➔ Go to step 2.

2. Check the cutting conditions as in steps 1 and 2 in *<23.5.5.1 Fretting* problem>.

   When the cutting conditions are appropriate:

   ➔ Go to step 3.

3. For the synchronous rotary guide bushing device, remove the guide bushing and check the runout and looseness of the tapered part of the opening of the sleeve.
   - Much runout degrade the roundness of the workpiece machined.

   If the cutting conditions are inappropriate:

   - Call the Cincom Service Office.

[Note]

Adjusting the clearance between the guide bushing and material changes the roundness of the material.

23.5.5.4 Abnormal noise during rotation

[Procedure]

1. Noise from the rotary guide bushing
   Check if:
   - the driving timing belt has been worn.
   - the support bearing is abnormal.

23.5.6 If "It fails in the update of OP088 Eco/ope. info. file." alarm occurred

If this message is displayed, it indicates that the file cannot be updated correctly and it is necessary to initialize the data. After the initialization, all of Eco and machine operation information related data are initialized.

[Procedure]

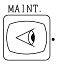
1. Check whether the backup file is available or not.

Backup file is available:

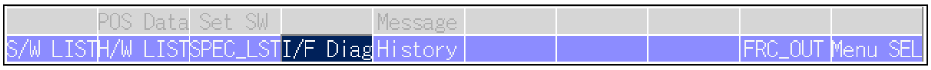
➔ Go to Step 2.

Backup file is not available:

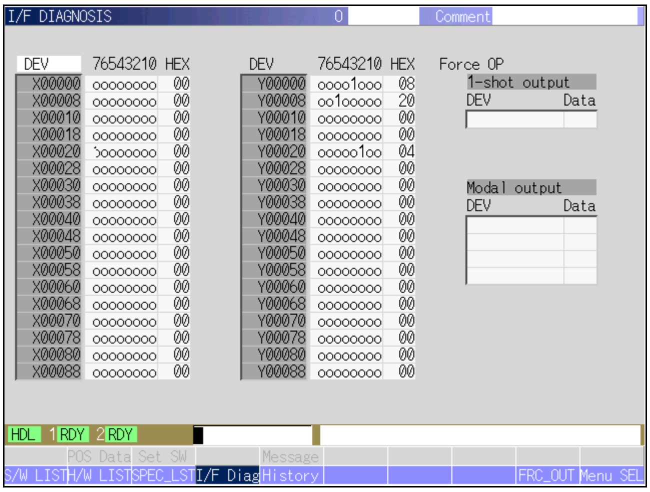
Make a backup file by referring to *<6.21.1 Output of Eco/Machine Operation Information>* in the Operator's Manual.

2. Press the Diagnosis key .

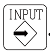
The Diagnosis menu appears.



3. Press the menu key [I/F Diag]. (Skip this step when the I/F DIAGNOSIS screen has already been displayed.) The I/F DIAGNOSIS screen will appear.

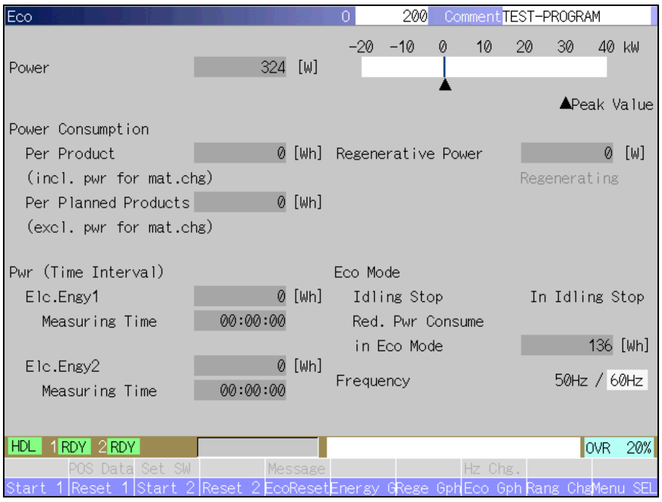


4. Use <alphanumeric> keys to enter "1001//M" in the input fields.

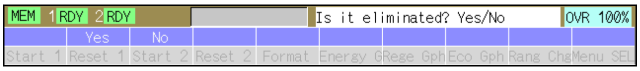
5. Press the Input key 

.

6. Press the menu key [Eco]. (Skip this step when the Eco screen has already been displayed. If the menu key [Eco] is not on the screen, press the menu key [Menu SEL] to display the menu key [Eco].)
The Eco screen will appear.



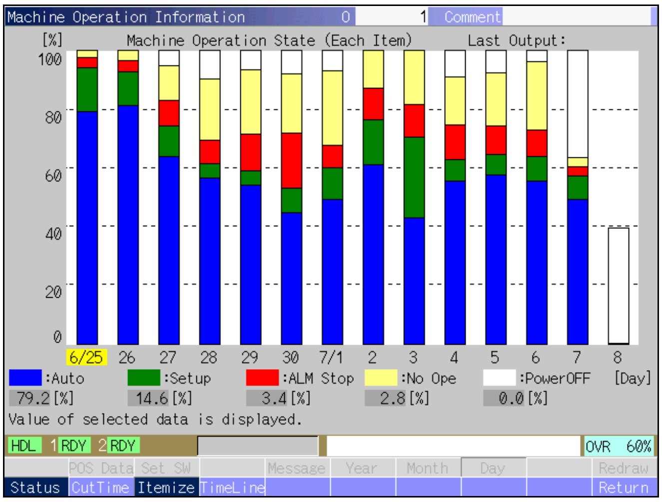
7. Press the menu key [Format]. (If the menu key [Format] is not on the screen, press the menu key [Menu SEL] to display the menu key [Format].) A message is displayed and menu keys [Yes] and [No] are displayed in the upper row menu.



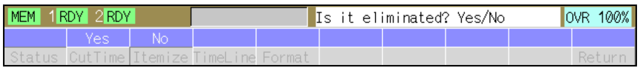
8. Press the menu key [Yes]. All of Eco related data are initialized.

9. Press the menu key [Operation Infomation]. (If the menu key [Operation Infomation] is not on the screen. press the menu key [Menu SEL] to display the menu key [Operation Infomation].)

The Machine Operation Infomation screen will appear.



10. Press the menu key [Format]. (If the menu key [Format] is not on the screen, press the menu key [Menu SEL] to display the menu key [Format].)
A message is displayed and menu keys [Yes] and [No] are displayed in the upper row menu.



11. Press the menu key [Yes]. All of machine operation information related data are initialized.

12. Turn the power supply on and off.

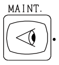
[Note]

When the data are initialized, the date for graph display is also initialized. The graph is displayed from the date when the power is turned on next.

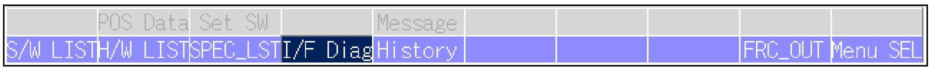
23.6 Appendix

23.6.1 PLC constant setting

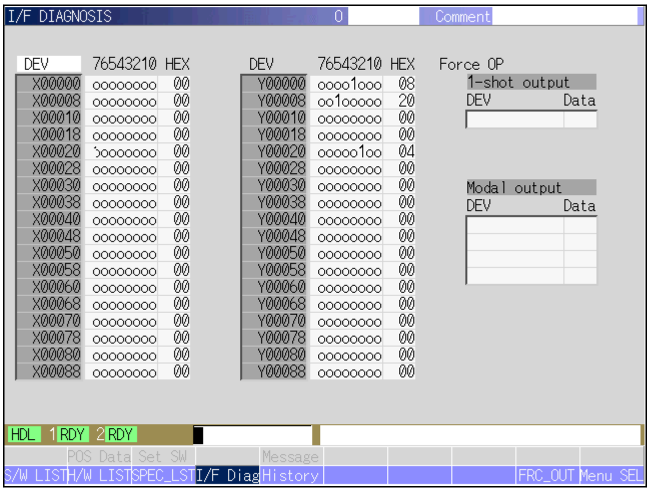
[Procedure]

1. Press the Diagnosis key .

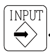
The Diagnosis menu appears.

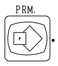


2. Press the menu key [I/F Diag]. (Skip this step when the I/F DIAGNOSIS screen has already been displayed.)
The I/F DIAGNOSIS screen will appear.



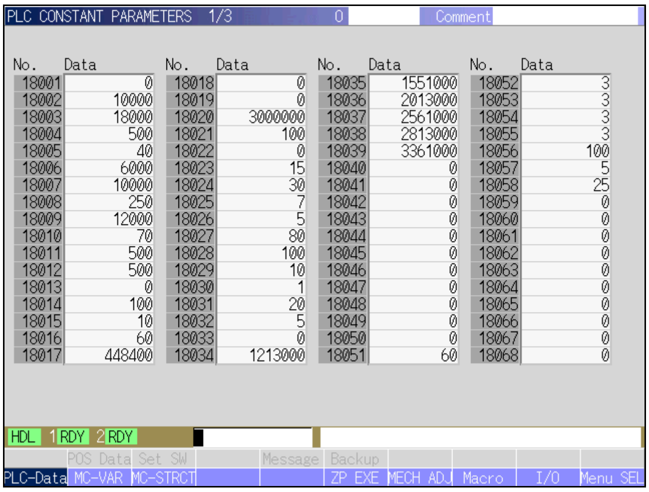
3. Use <alphanumeric> keys to enter "1001//M" in the input fields.

4. Press the Input key .

5. Press the Parameter key .

The Parameter screen appears.

6. Press the menu key [PLC-Data]. (Skip this step when the PLC CONSTANT PARAMETERS screen has already been displayed. If the menu key [PLC-Data] is not on the screen, press the menu key [Menu SEL] to display the menu key [PLC-Data].)
PLC CONSTANT PARAMETERS screen appears.



7. Move the cursor to the item (switch) to be set using the arrow keys [↑] [↓] or the tab keys [TAB ←] [TAB →].

8. Use <numeric> keys to enter the desired value.

9. Press the Input key [INPUT].

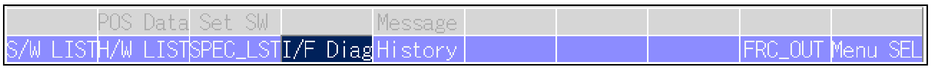
10. Repeat steps 7. to 9. until all data is set.

23.6.2 PLC bit selection parameter setting

[Procedure]

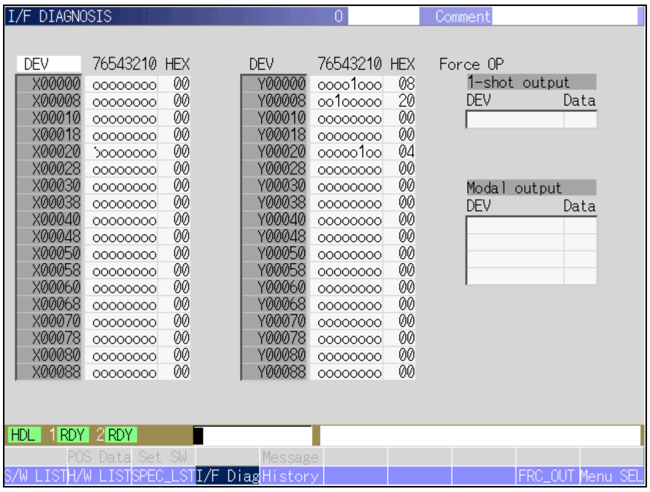
1. Press the Diagnosis key .

The Diagnosis menu appears.



2. Press the menu key [I/F Diag]. (Skip this step when the I/F DIAGNOSIS screen has already been displayed.)

The I/F DIAGNOSIS screen will appear.



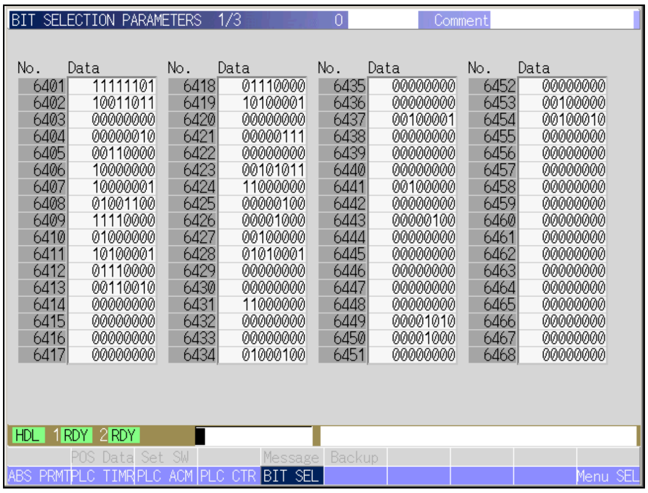
3. Use <alphanumeric> keys to enter "1001//M" in the input fields.

4. Press the Input key .

5. Press the Parameter key .

The Parameter screen appears.

6. Press the menu key [BIT SEL]. (Skip this step when the BIT SELECTION PARAMETERS screen has already been displayed. If the menu key [BIT SEL] is not on the screen, press the menu key [Menu SEL] to display the menu key [BIT SEL].)
The BIT SELECTION PARAMETERS screen appears.



No.	Data	No.	Data	No.	Data	No.	Data
6401	11111101	6418	01110000	6435	00000000	6452	00000000
6402	10011011	6419	10100001	6436	00000000	6453	00100000
6403	00000000	6420	00000000	6437	00100001	6454	00100010
6404	00000010	6421	00000111	6438	00000000	6455	00000000
6405	00110000	6422	00000000	6439	00000000	6456	00000000
6406	10000000	6423	00101011	6440	00000000	6457	00000000
6407	10000001	6424	11000000	6441	00100000	6458	00000000
6408	01001100	6425	00000100	6442	00000000	6459	00000000
6409	11110000	6426	00001000	6443	00000100	6460	00000000
6410	01000000	6427	00100000	6444	00000000	6461	00000000
6411	10100001	6428	01010001	6445	00000000	6462	00000000
6412	01110000	6429	00000000	6446	00000000	6463	00000000
6413	00110010	6430	00000000	6447	00000000	6464	00000000
6414	00000000	6431	11000000	6448	00000000	6465	00000000
6415	00000000	6432	00000000	6449	00001010	6466	00000000
6416	00000000	6433	00000000	6450	00001000	6467	00000000
6417	00000000	6434	01000100	6451	00000000	6468	00000000

7. Use the arrow keys ⬆ ⬇, tab keys TAB ⬅ TAB ➡, or Page keys to position the cursor on the item you want to set.

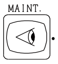
8. Use <numeric> keys to enter eight bits of data. Or, use the arrow key ⬆ or ⬇ to position the cursor on the bit you want to set, use the Delete key DEL to delete the existing value, then enter a new value (0 or 1).

9. Press the Input key INPUT.

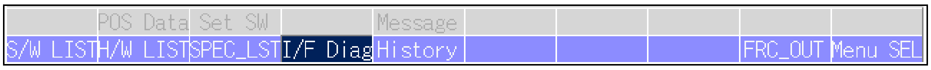
10. Repeat steps 7. to 9. until all data is set.

23.6.3 Checking the version of software

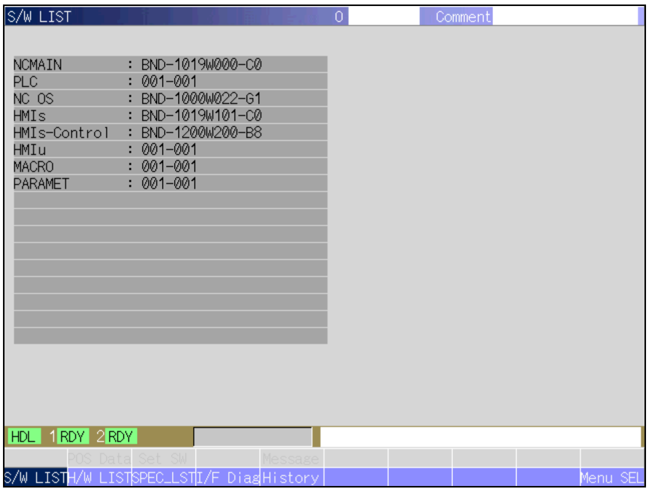
[Procedure]

1. Press the Diagnosis key .

The Diagnosis menu appears.

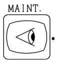


2. Press the menu key [S/W LIST]. (Skip this step when the S/W LIST screen has already been displayed.)
The S/W LIST screen appears, displaying the version of each software program.

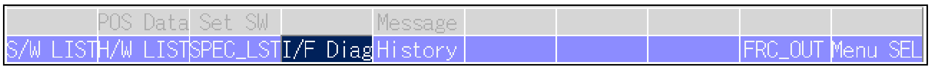


23.6.4 Checking the alarm history

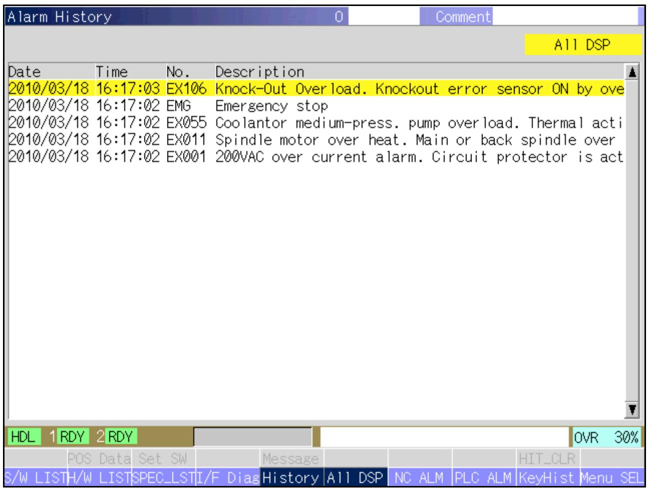
[Procedure]

1. Press the Diagnosis key .

The Diagnosis menu appears.



2. Press the menu key [History]. (Skip this step when the Alarm History screen has already been displayed.)
The Alarm History screen appears, listing the last 100 alarms generated.



[Note]

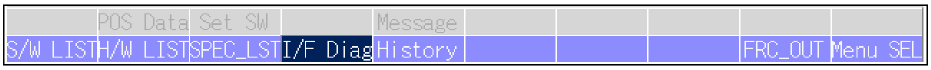
- The Alarm History screen displays the menu keys [All DSP], [NC ALM], and [PLC ALM].
- Pressing the menu key [NC ALM] displays only the alarms generated from the NC unit.
- Pressing the menu key [PLC ALM] displays only the alarms generated from the PLC (ladder diagram and macros).
- Pressing the menu key [All DSP] displays both of the NC and PLC alarms.

23.6.5 Checking the key-in history

[Procedure]

1. Press the Diagnosis key MAINT.

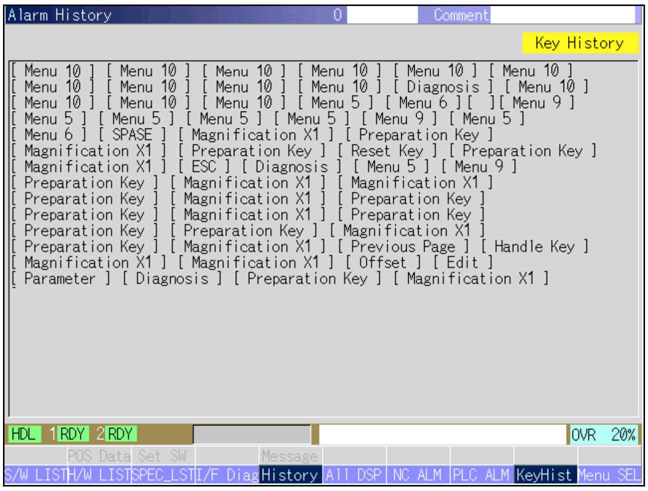
The Diagnosis menu appears.



2. Press the menu key [History]. (Skip this step when the Alarm History screen has already been displayed.)

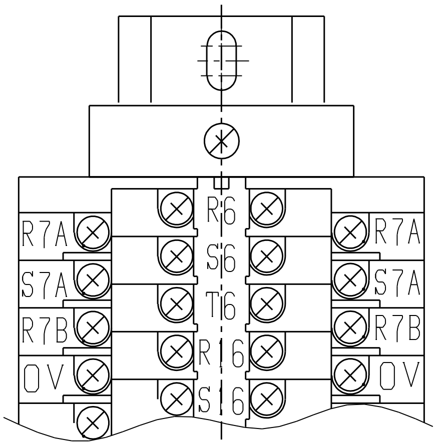
3. Press the Page key .

The Key History screen appears, listing the last 120 keys you have pressed.

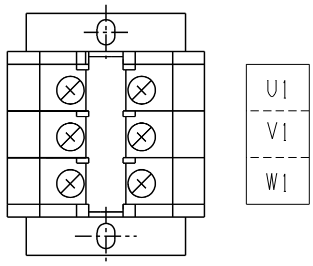


23.6.6 Terminal layout

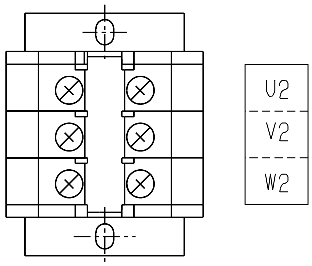
TB1

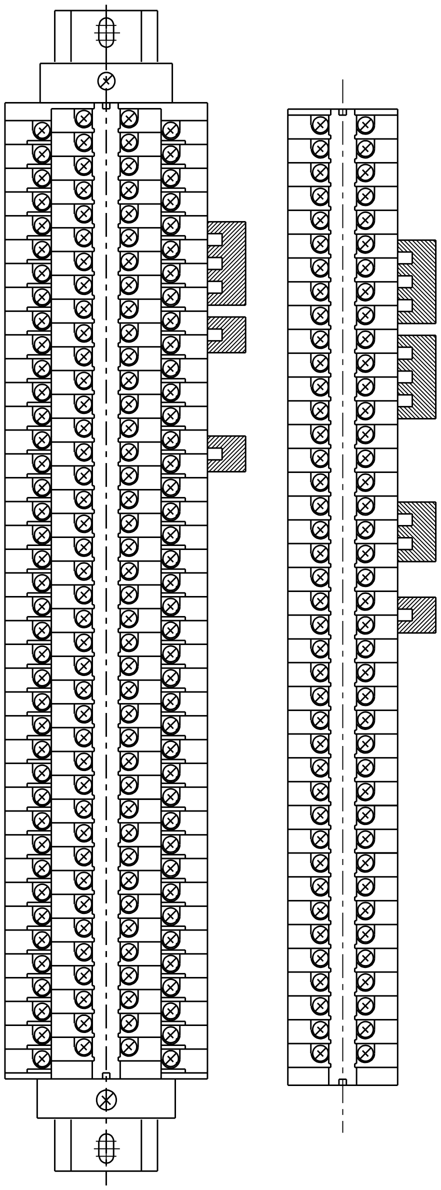


TB6



TB7

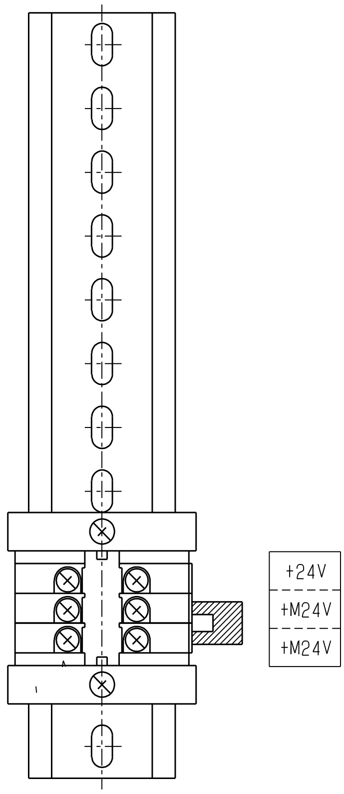


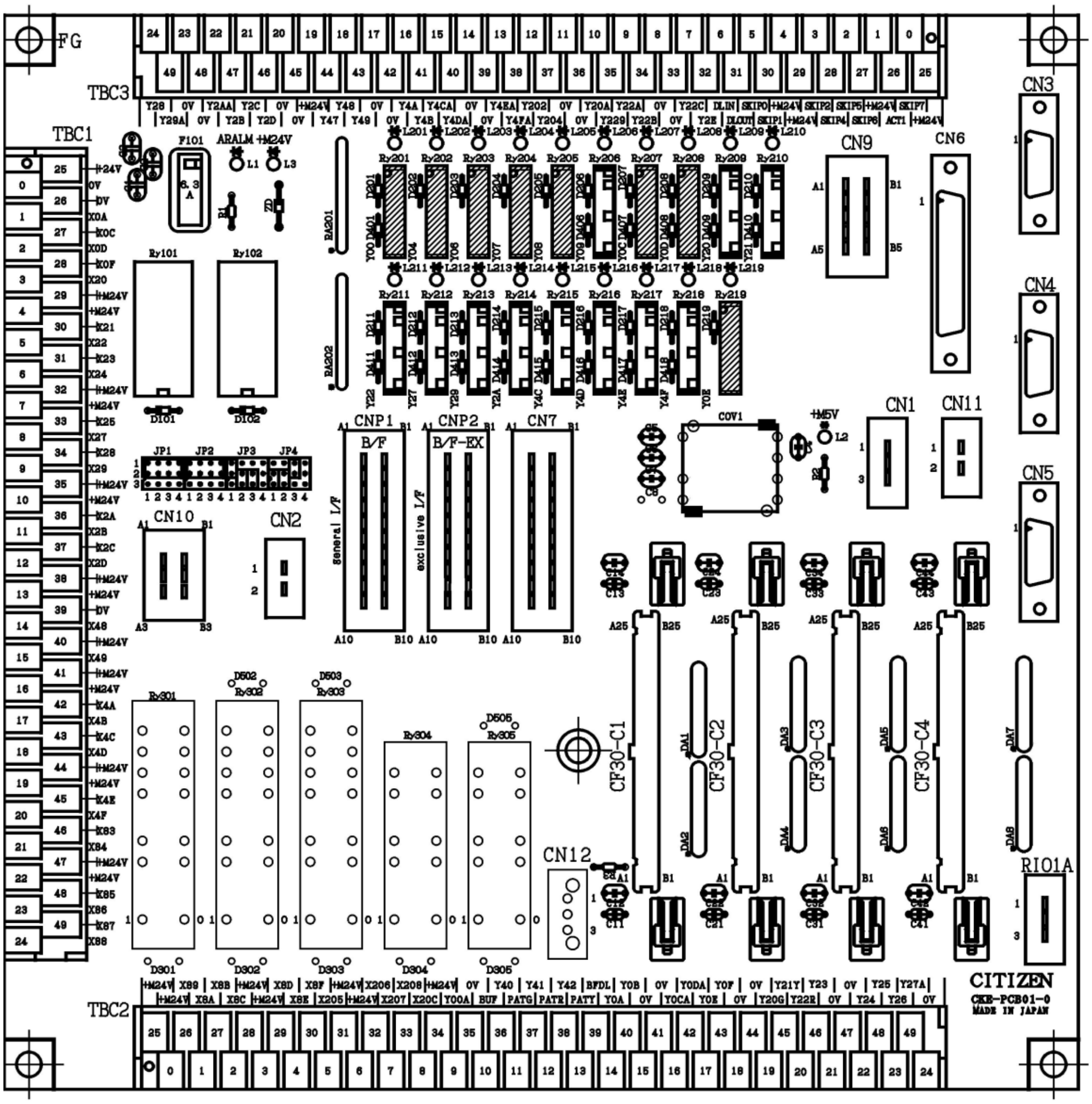


R7A	R6
S7A	S6
R7B	T6
R7D	R16
0V	S16
0V	R7
0V	R7
0V	R7
0VB	R7
0VB	S7
0VB	S7
BR1	S7
BR2	S7
24D1	FLON
24D1	+24V
PSEMG	+24V1
DRSW2	+M24V
DRSWC	+M24V
EMGB10	+M24V
EMGB11	+24VD
EMGB12	+24VB
EMGB20	+24VB
DS	ACT2
DS1	X02
DS2	X03
DS3	X04
DSA	X05
DS1A	X0B
DS2A	Y01
DS3A	SF1A
DS4A	EMGB1A
DS4	EMGB1B
DL1	EMGB1C
DL2	EMGB1D
DL3	EMGB2A
DL4	EMGB2B
DL1A	EMGB2C
DL2A	EMGB2D
DL3A	EMGB11A
DL4A	EMGB12A

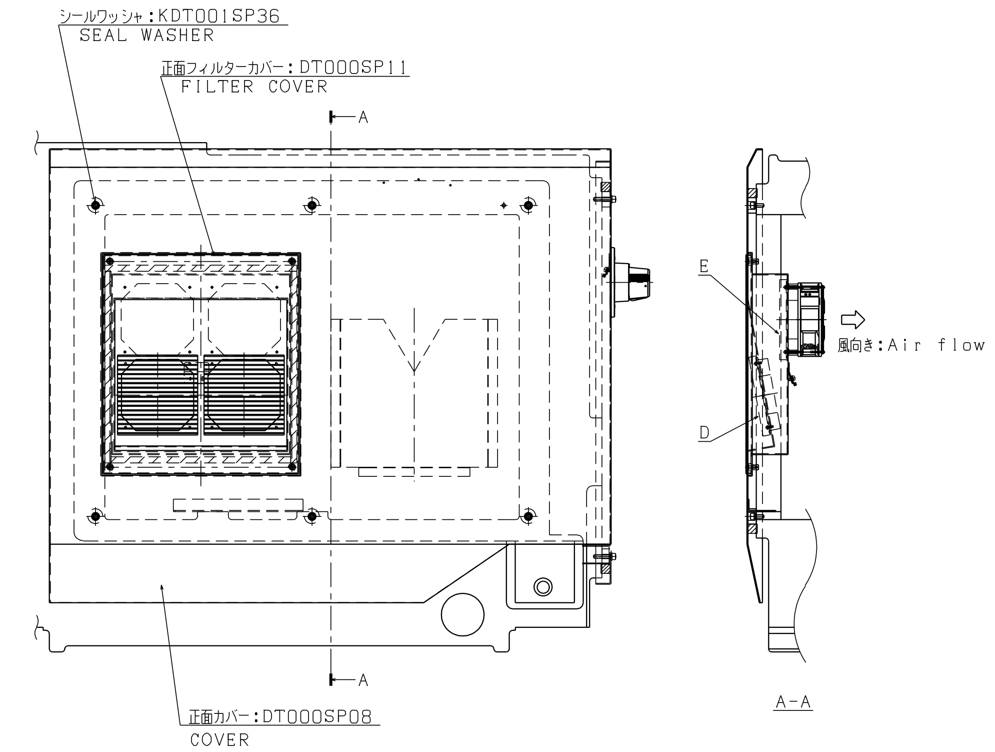
下段 Lower row 上段 Upper row

TB8

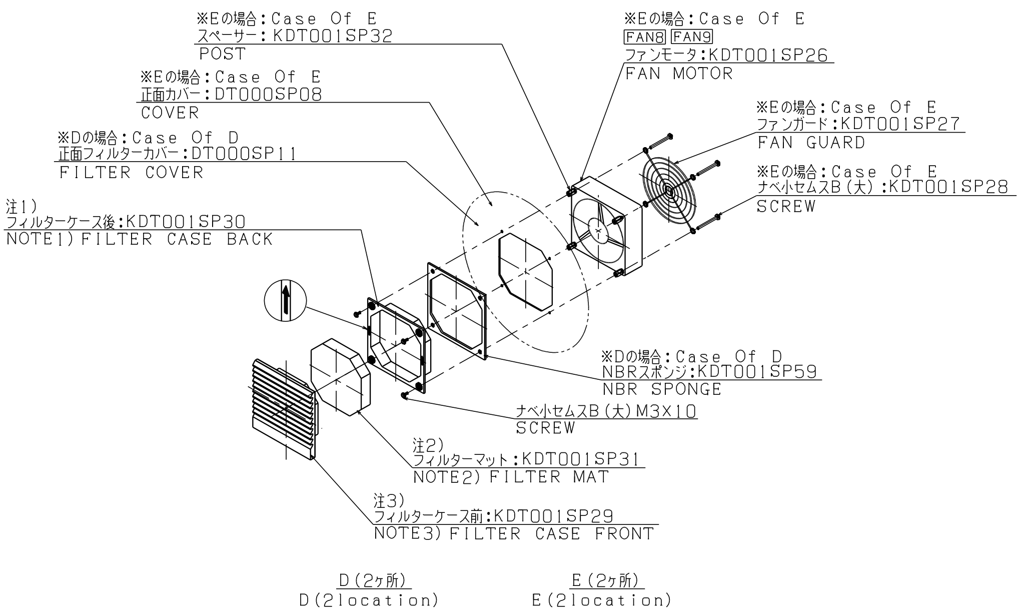




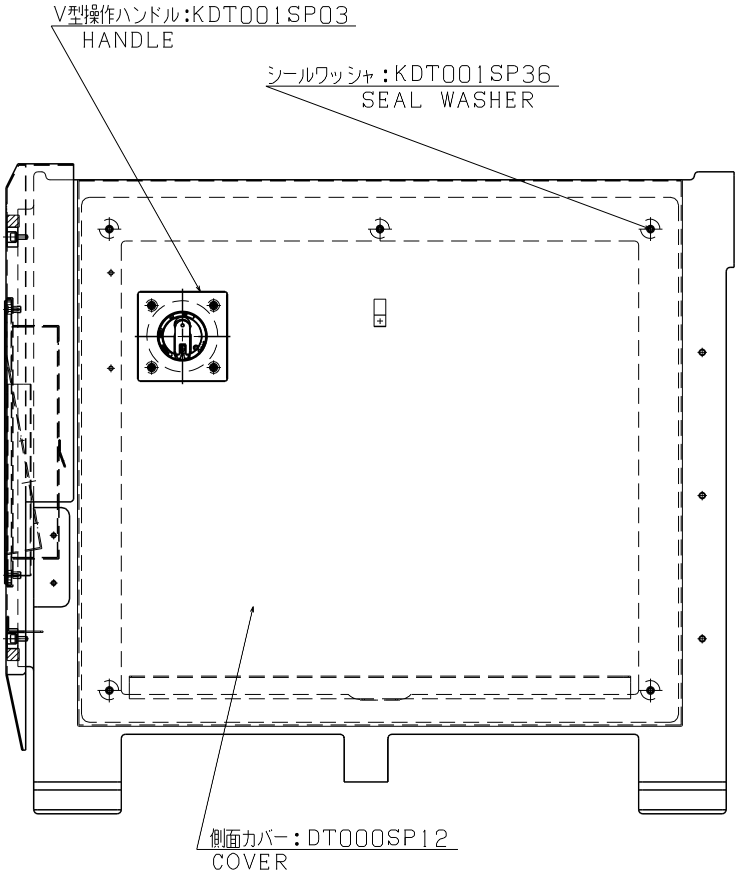
23.6.7 Control box component layout



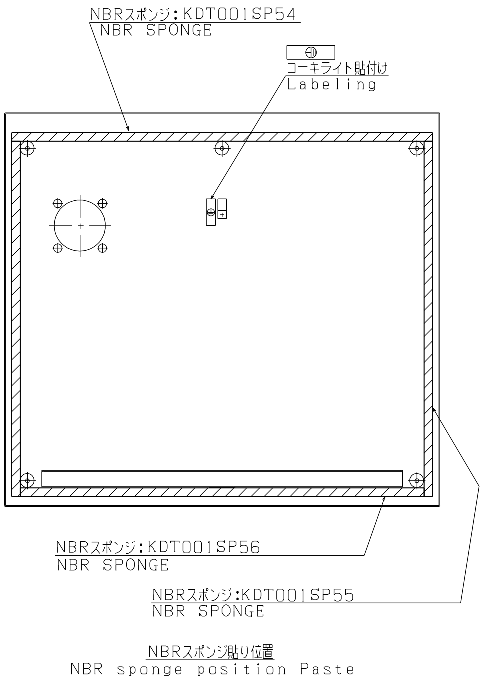
Control panel front view (with cover)



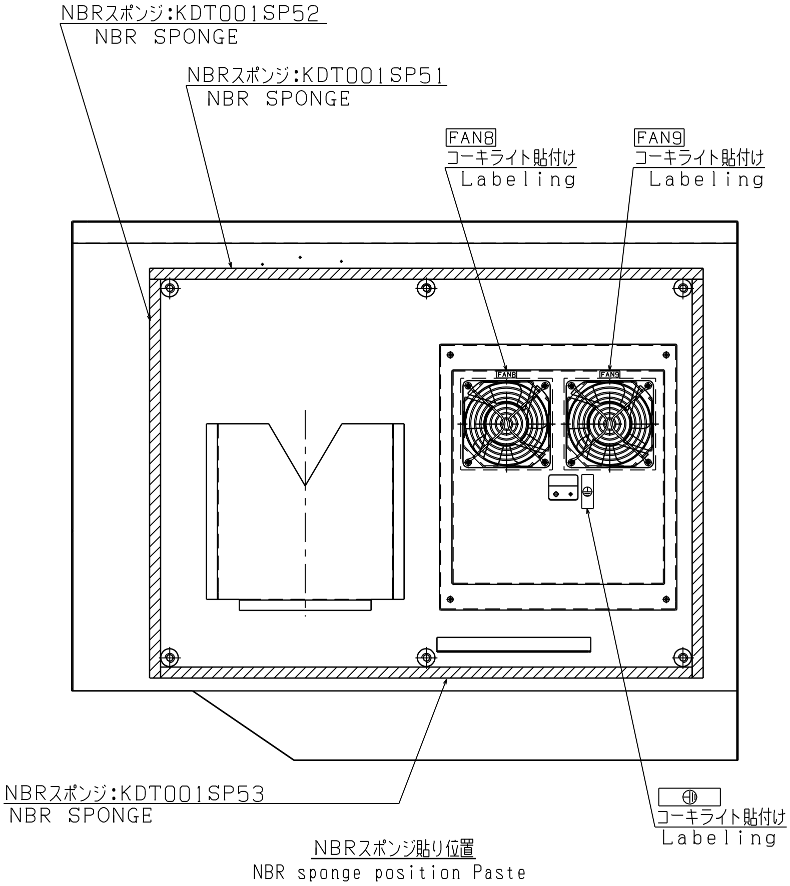
Control panel rear filter detailed view



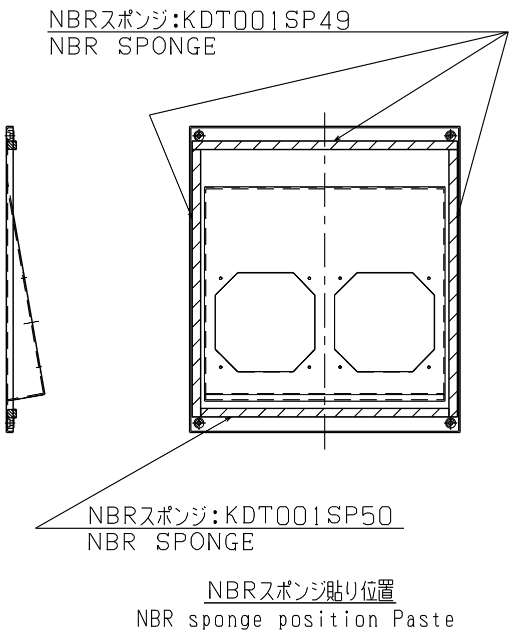
Control panel side view



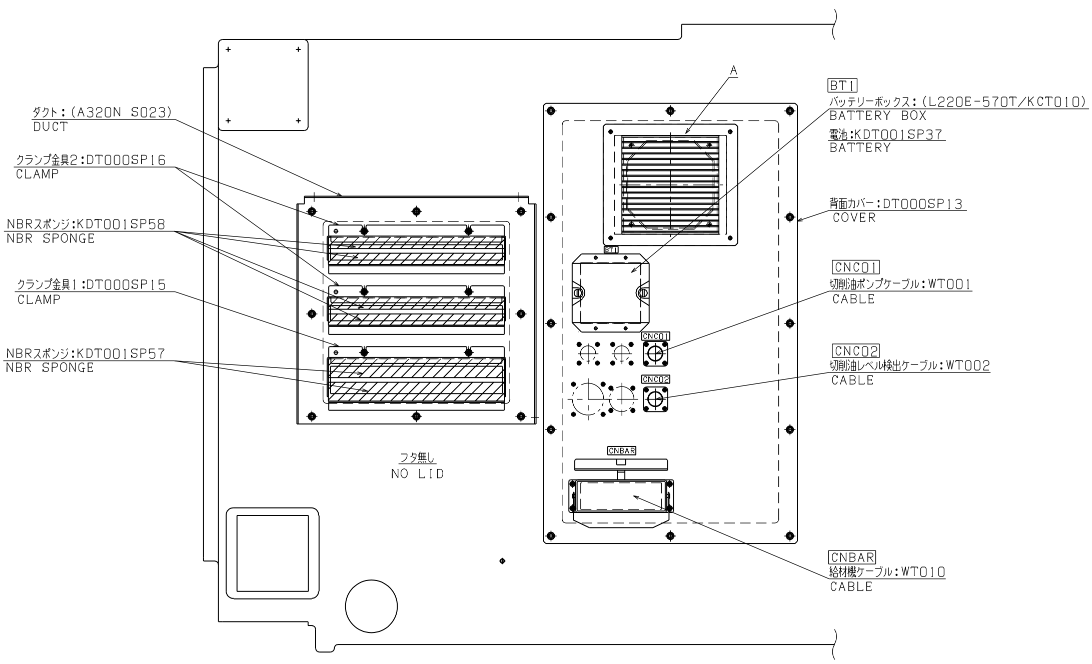
Control panel side cover detailed view



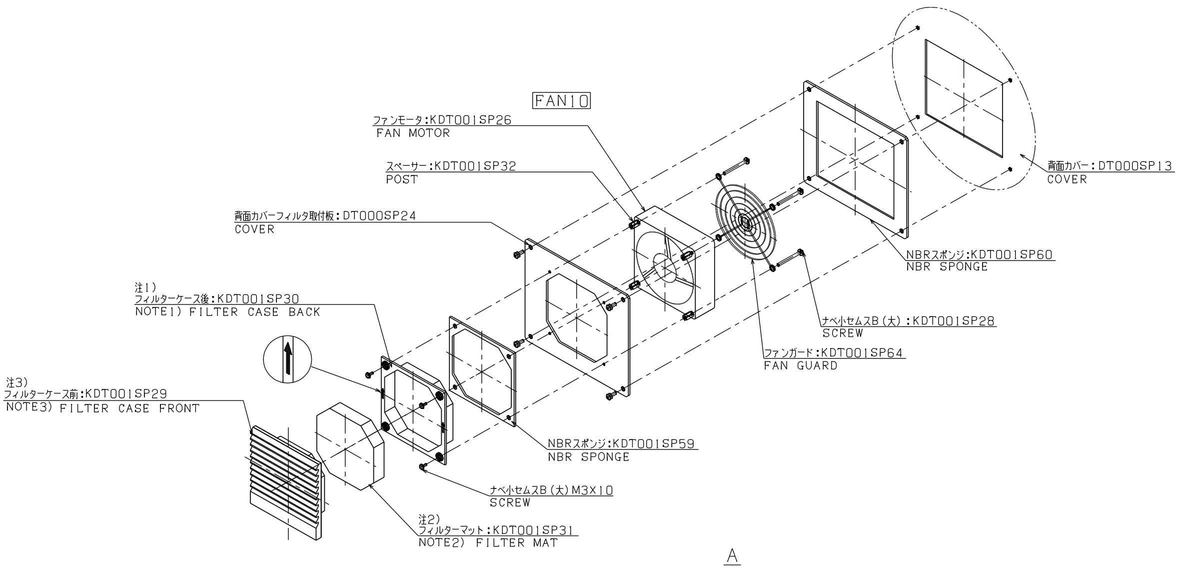
Control panel front cover view



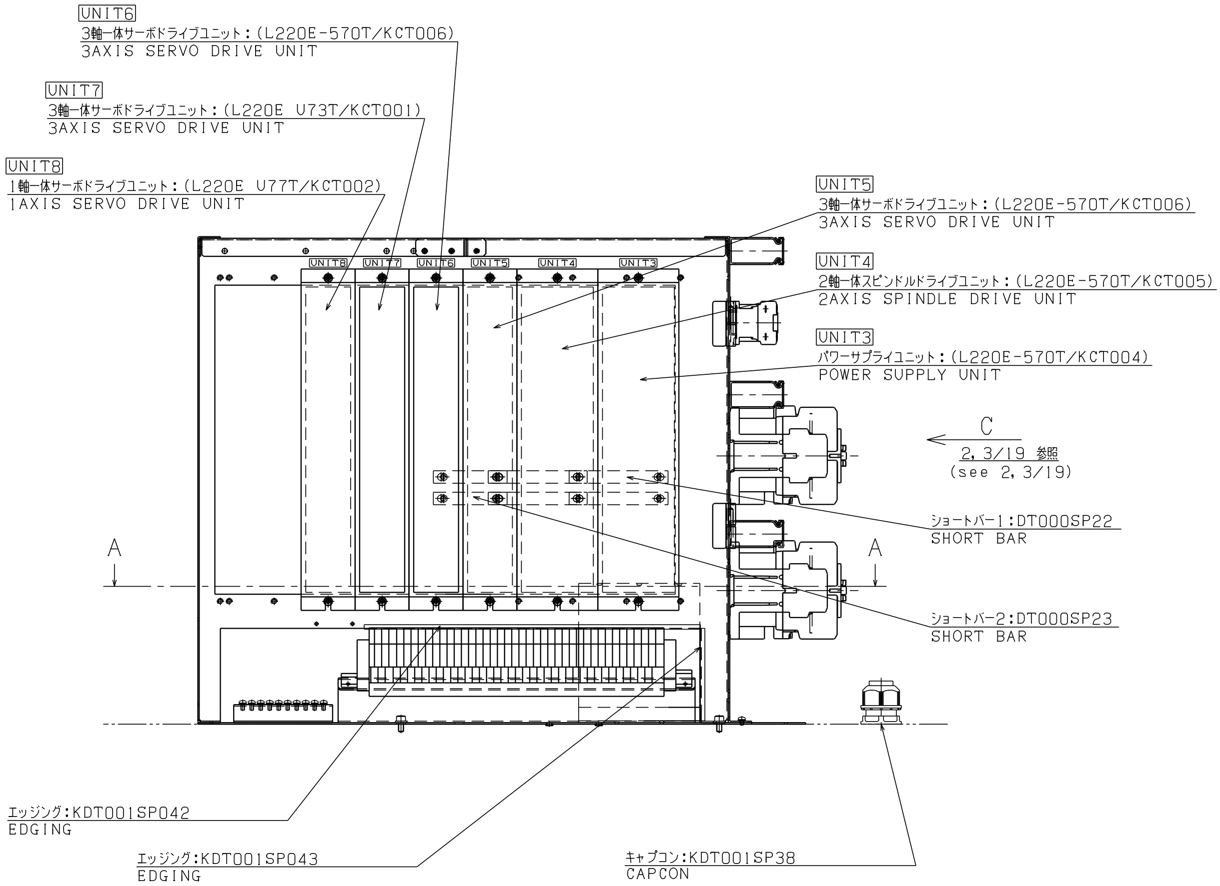
Control panel front cover detailed view



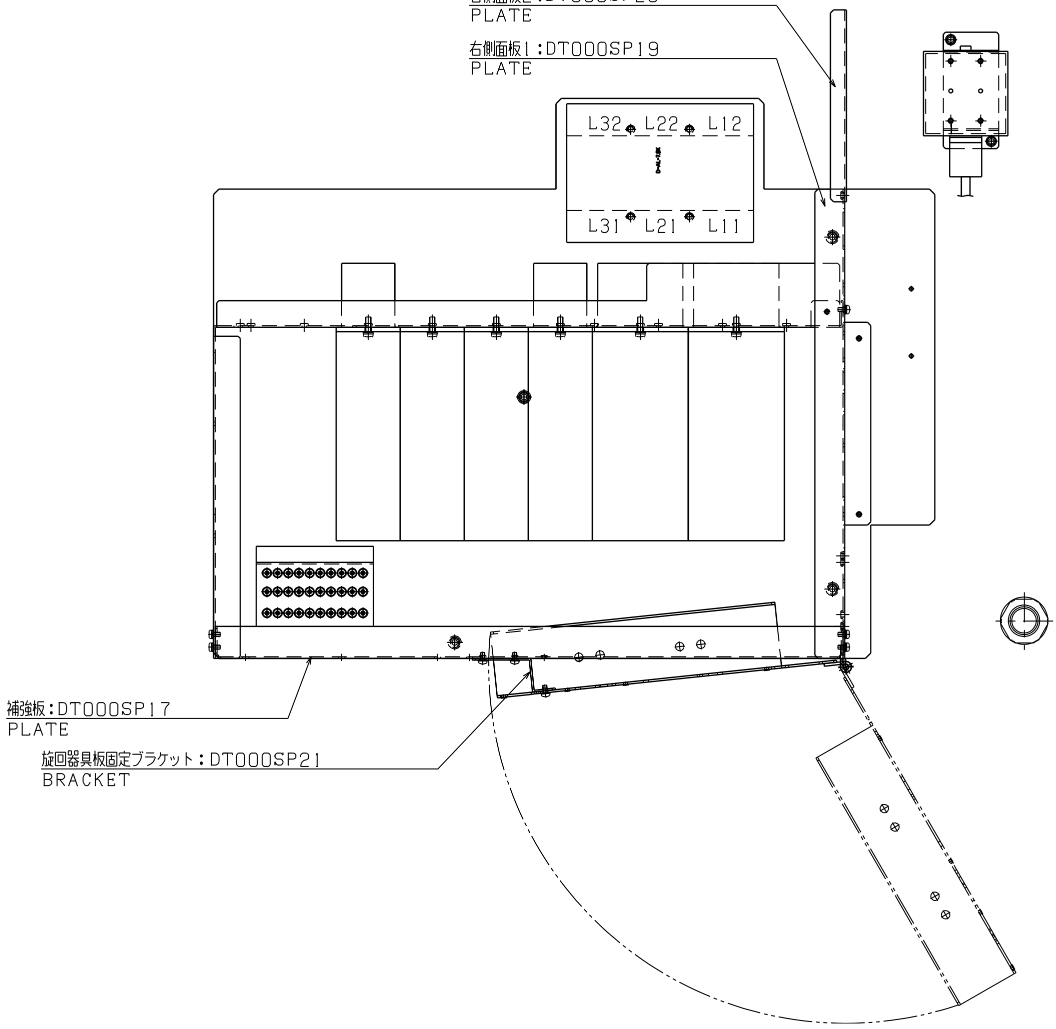
Control panel rear detailed view



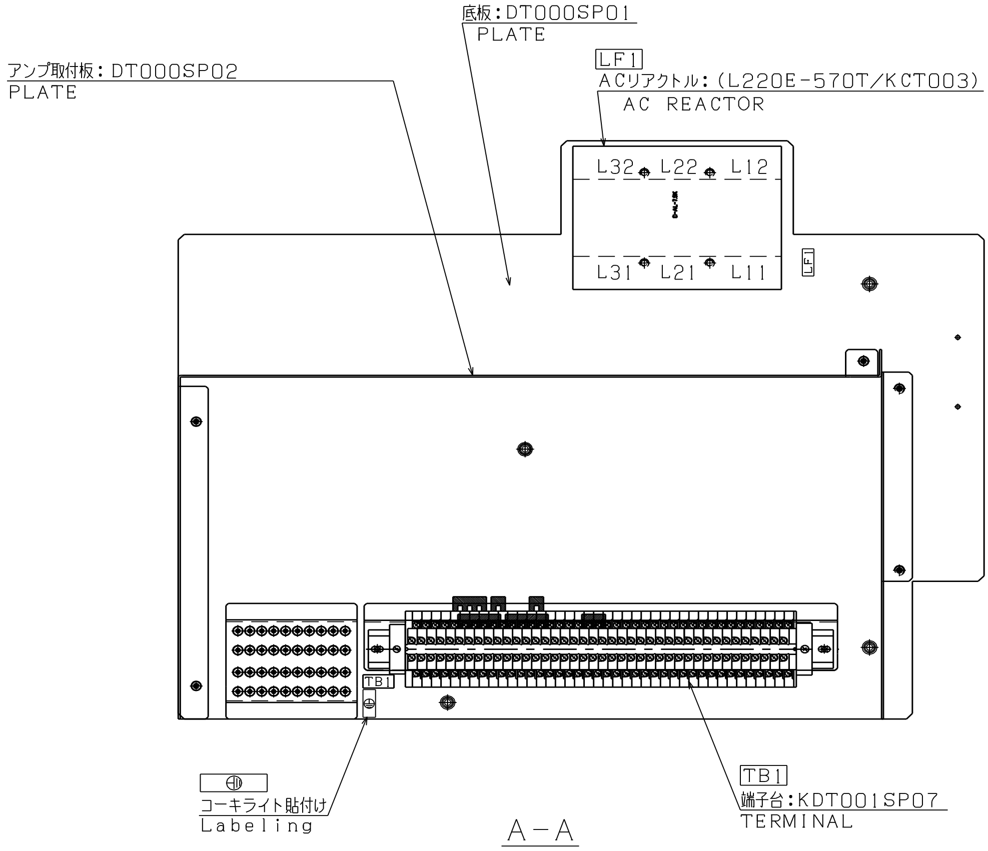
Control panel rear filter detailed view



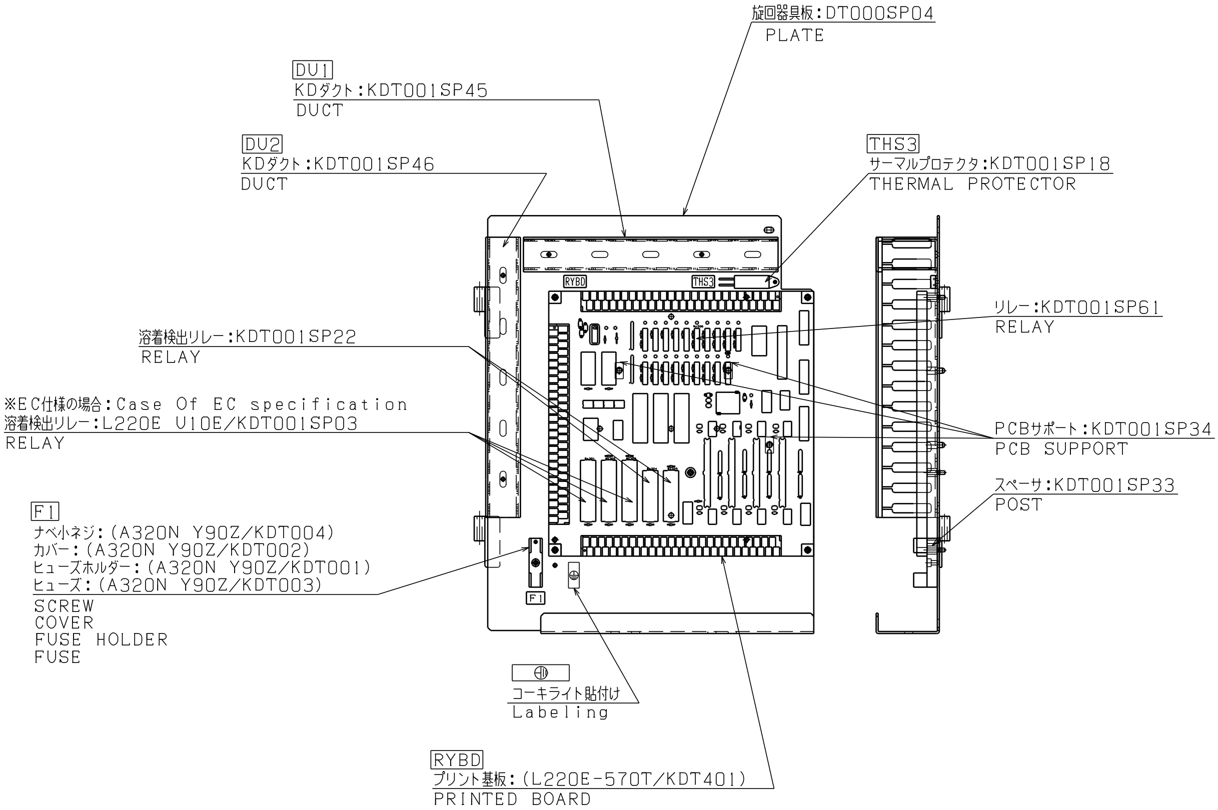
Control panel front view (without cover)



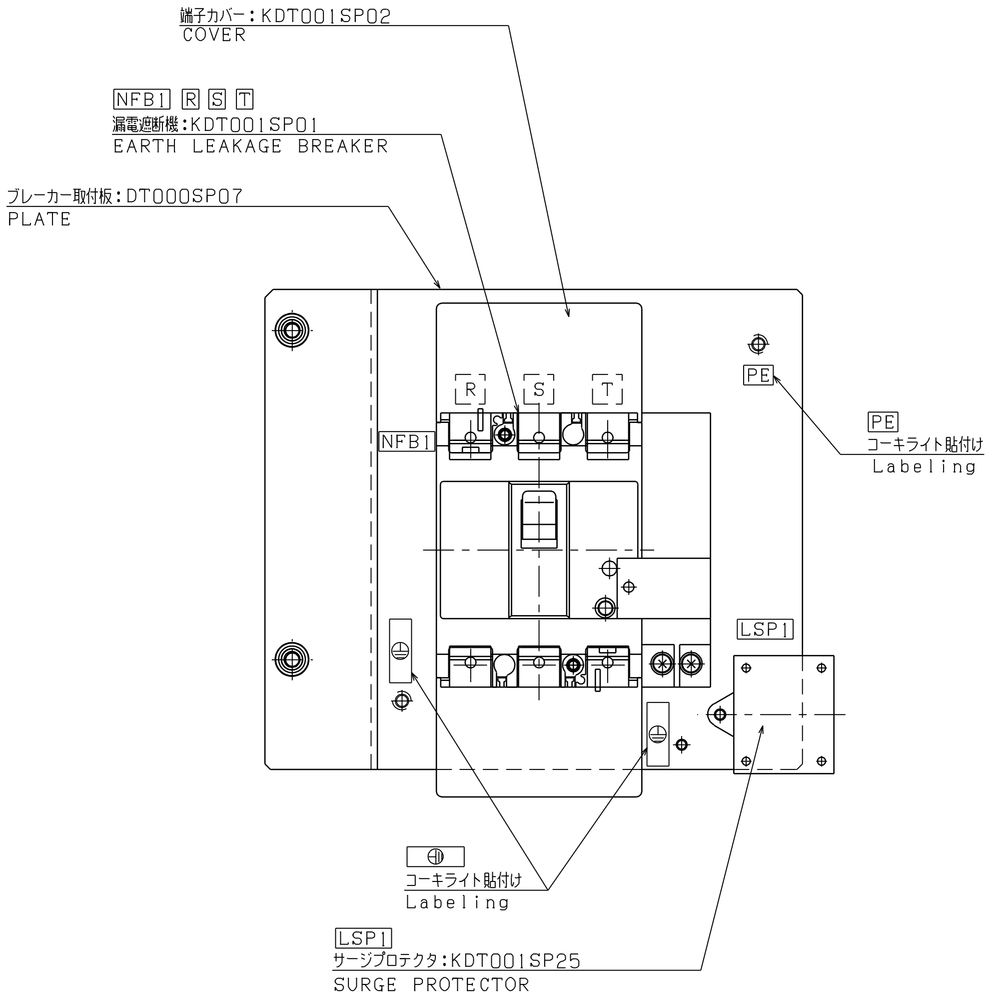
Control panel top view



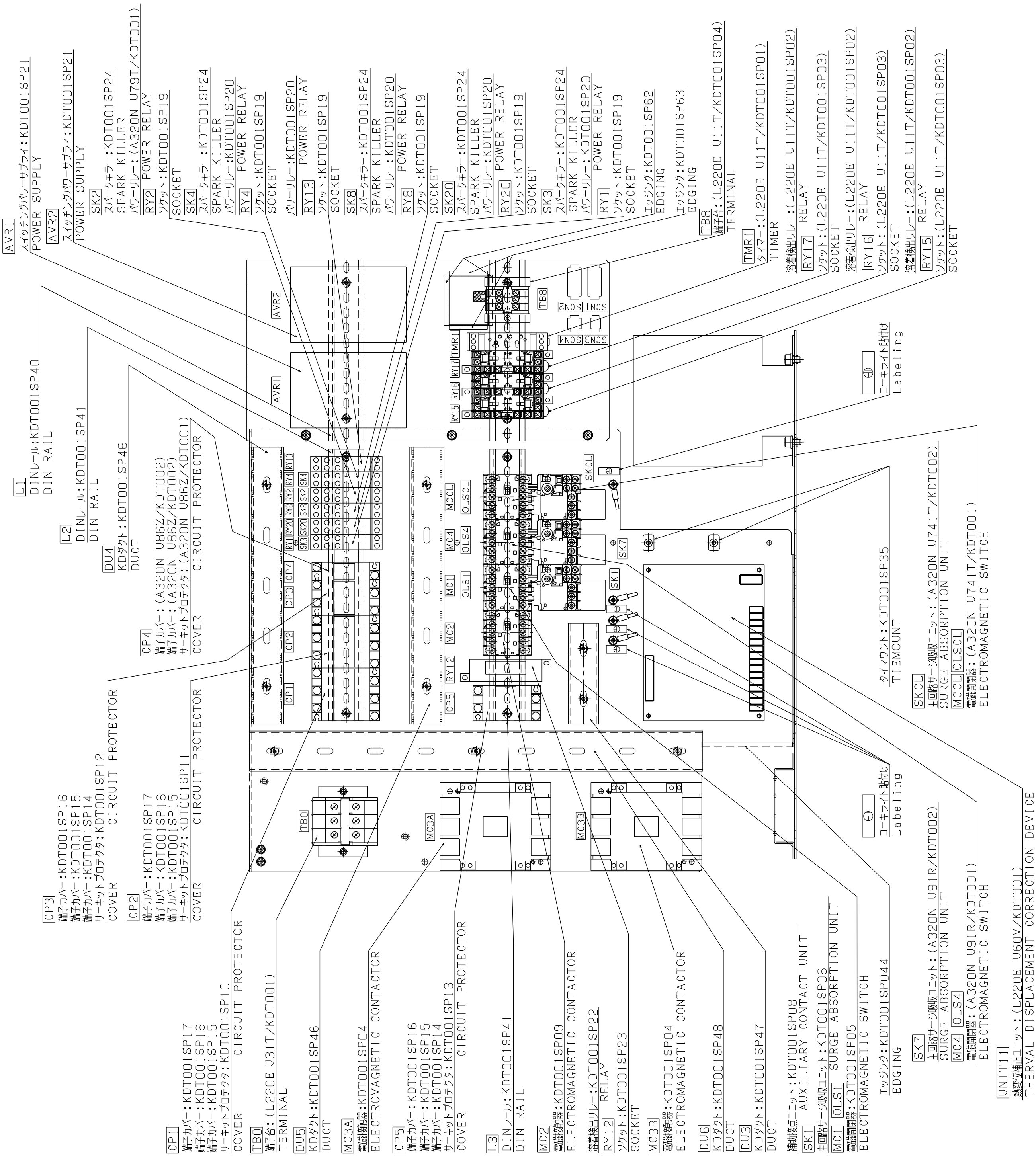
Cross-section along A-A



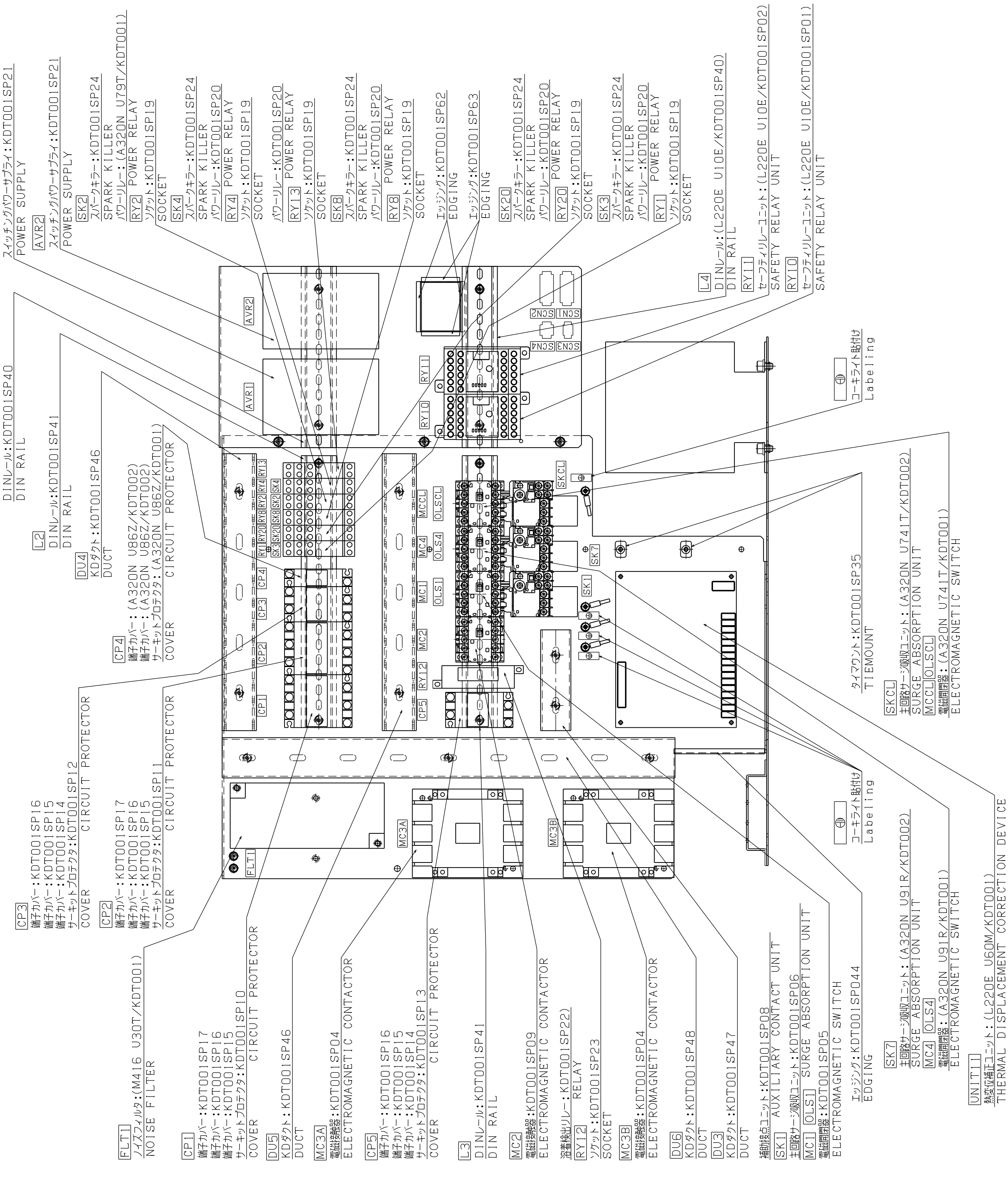
Swivel equipment panel detailed view



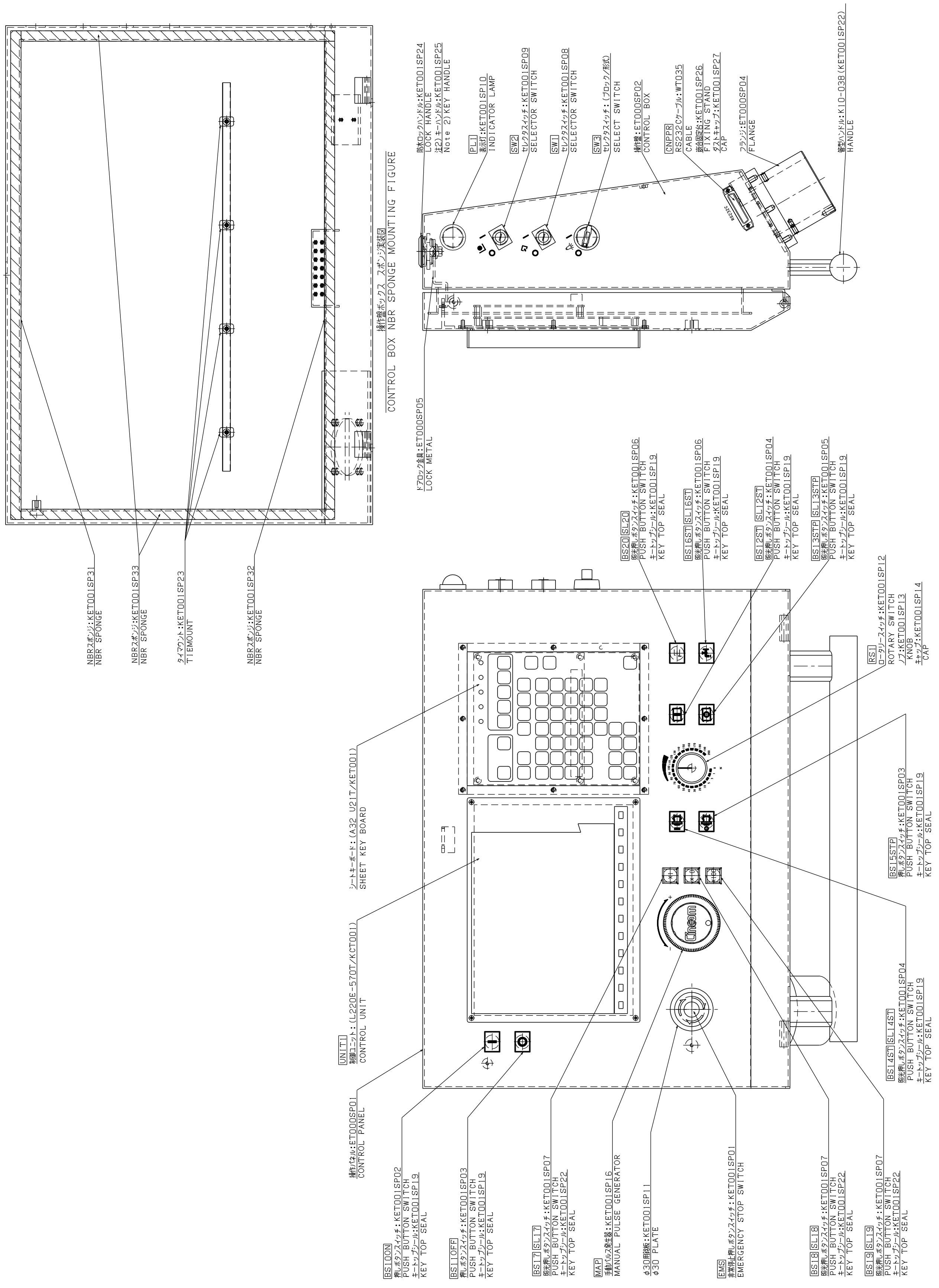
Circuit breaker detailed view



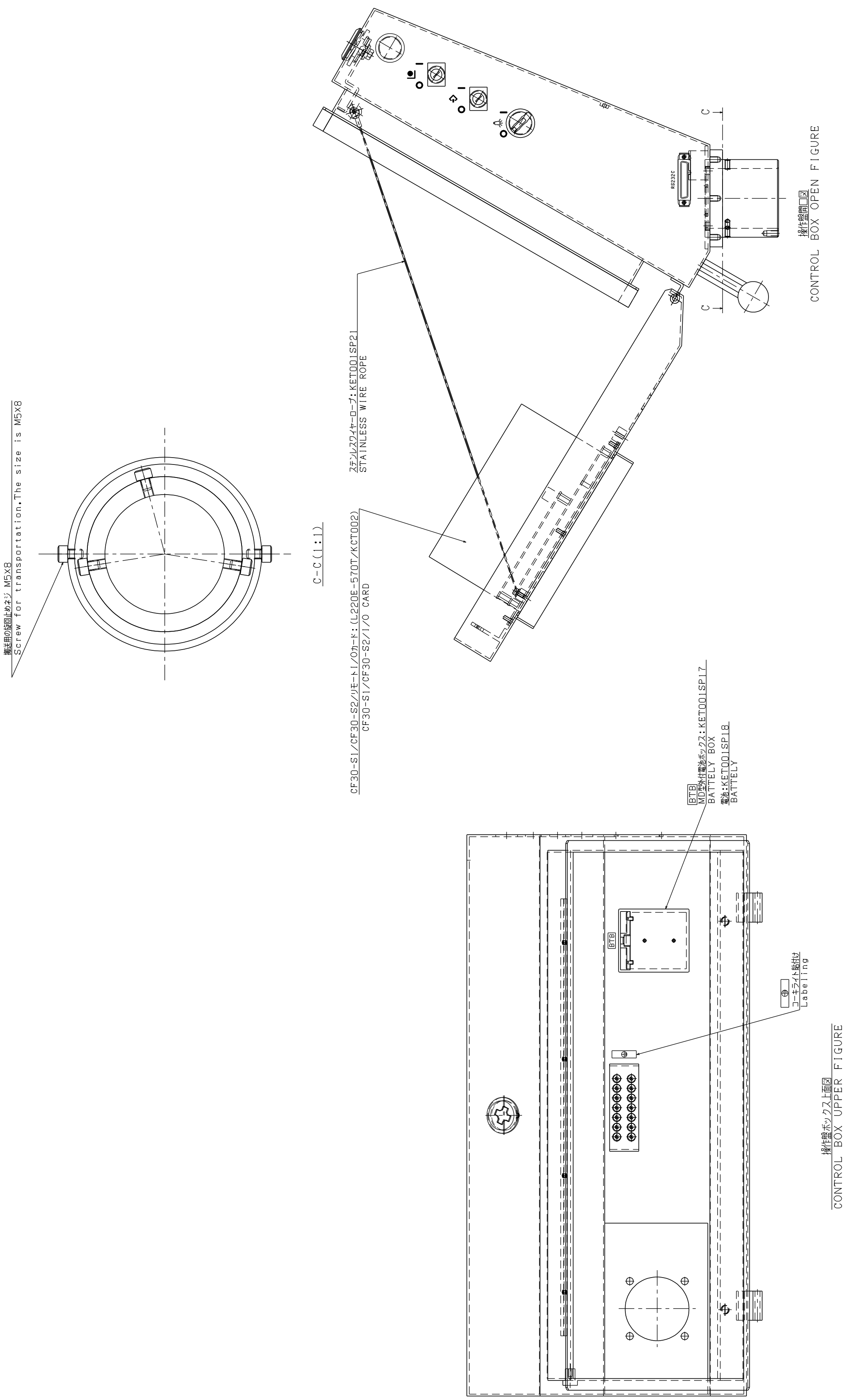
Equipment panel detailed view (standard model)



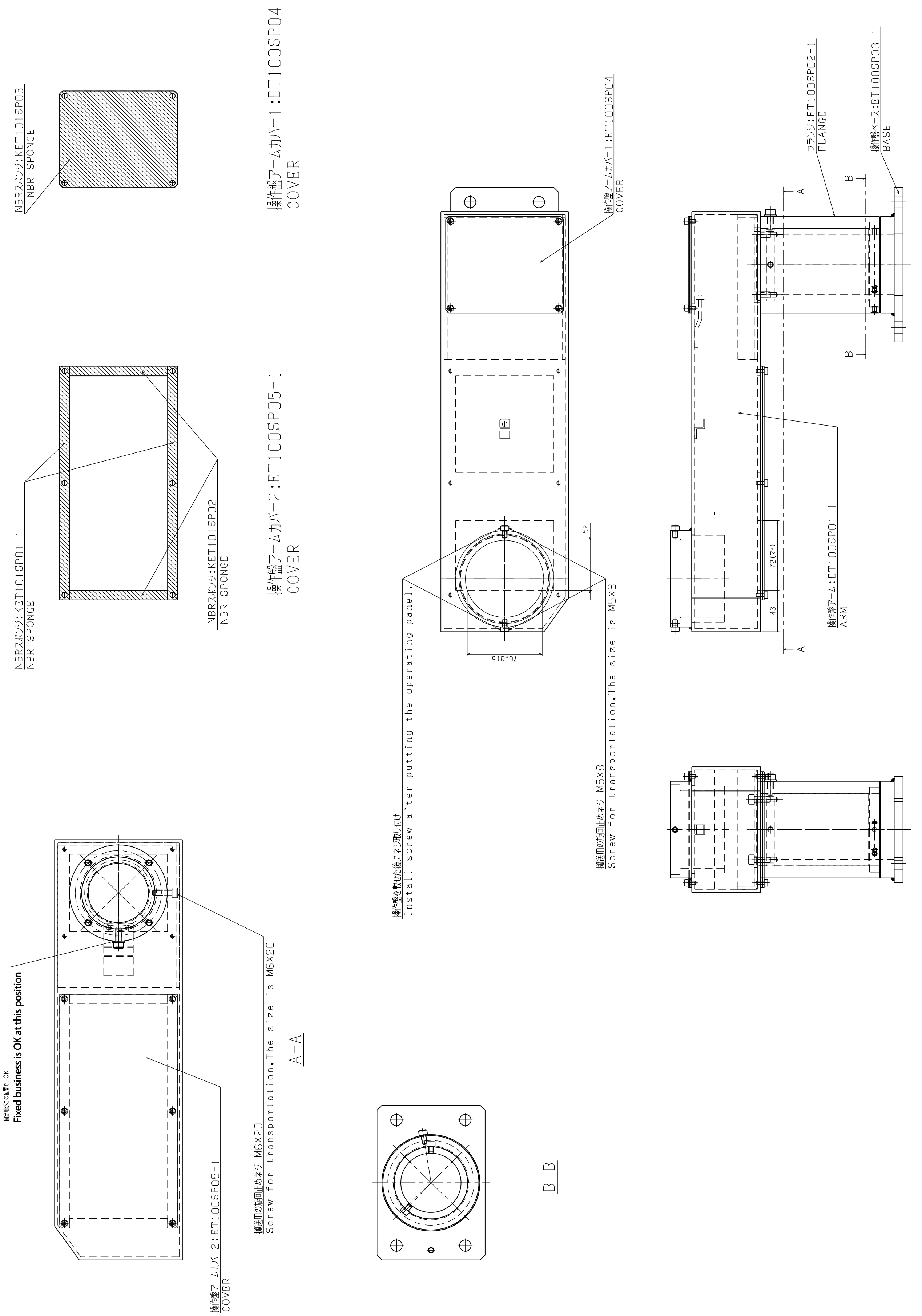
Equipment panel detailed view (EC model)



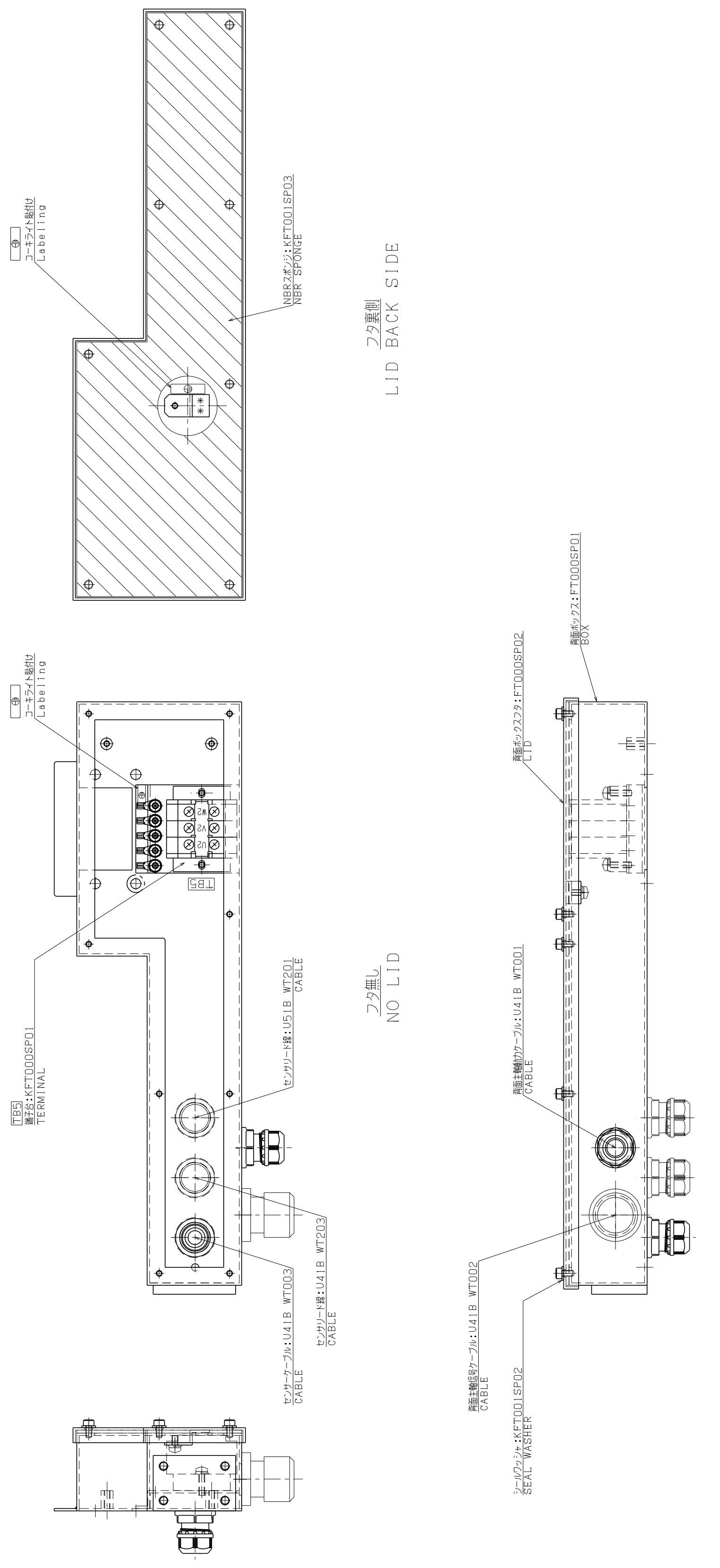
Operation panel detailed view (1)



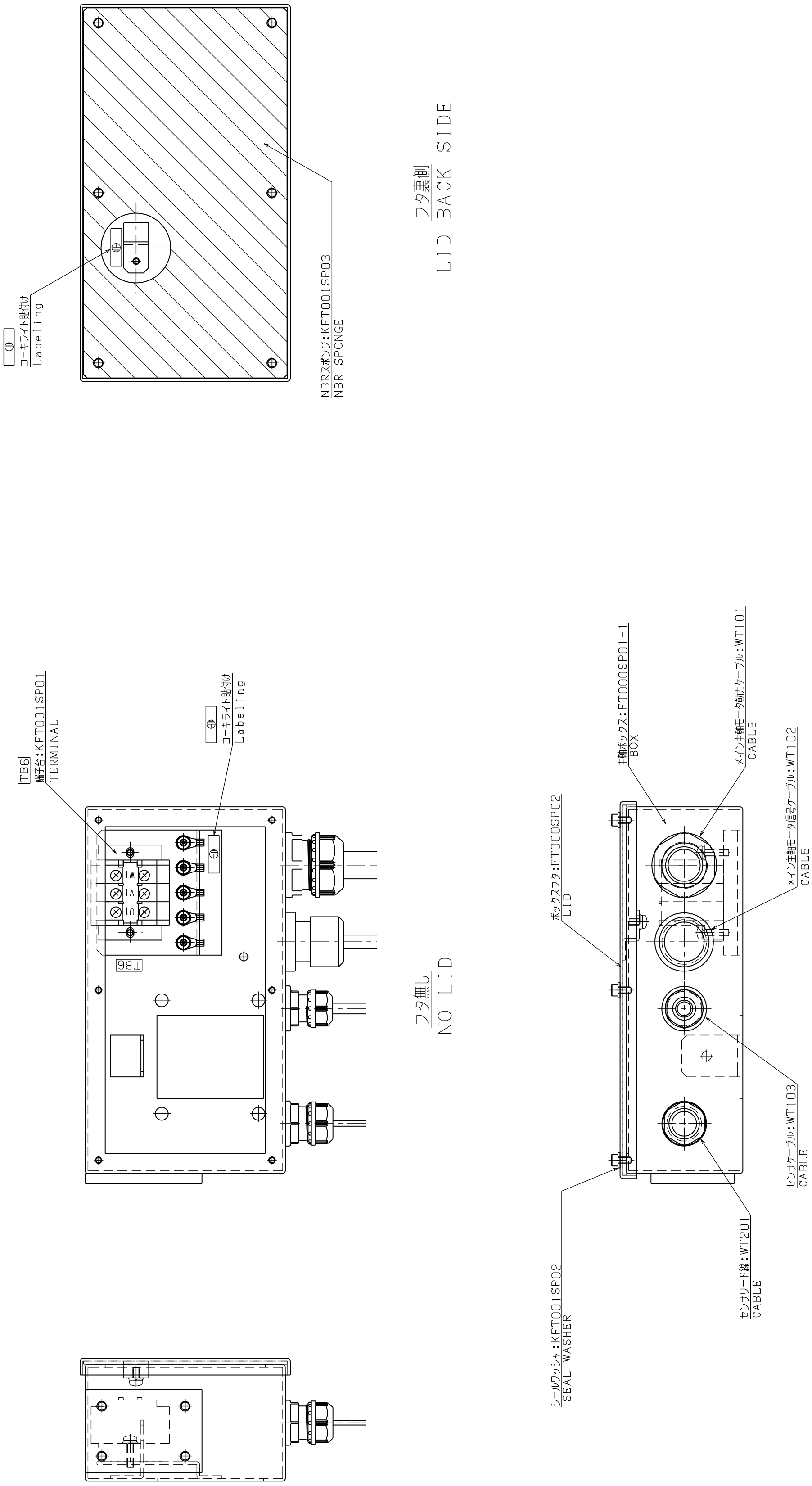
Operation panel detailed view (2)



Operation panel arm detailed view



Relay box detailed view (1)



Relay box detailed view (2)

Product Code	C-L220E XII
Document Code	3E1-2300
Mfg. No.	L220/0001 ~
Issue Date	2013.10

24. Relocating the NC Machine

24.1 Preface

It is necessary to relocate the NC machine installed by the manufacturer. This chapter explains how to safely prepare the NC machine for relocation, transport the machine, select a suitable new site, and install the machine at the new site.

Before the work, you must carefully read this chapter.

> **CAUTION**
>
> If you do not follow the instructions herein, you will damage the machine.

24.2 Selecting the Installation Site

To ensure the highly precise cutting performance for on the NC machine, take into consideration the location of power outlets, foundation strength, ambient temperature and humidity, traffic, ventilation, sunlight, nearby electrical machines and equipment generating high-frequency noise, and all other considerations that may affect the operation of your NC machine.

The site you select should meet the following requirements.

■ Site foundation:

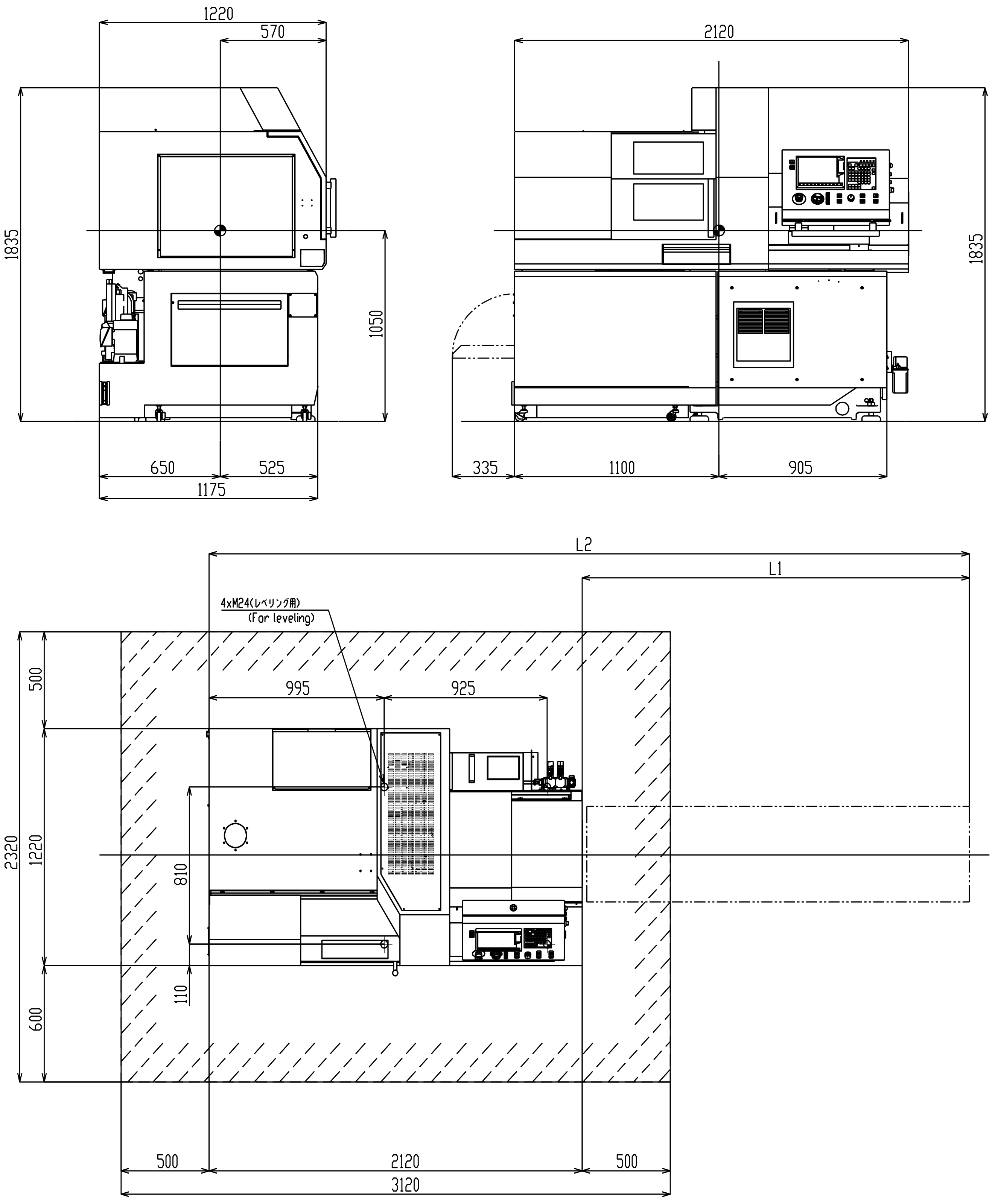
- The weight bearing capacity of the soil must be 1 ton/m^2, and the foundation thickness must be 100 mm or more.
- The foundation must extend 300 mm or more from the periphery of the NC machine.
- The NC machine must be installed on a sturdy, level surface that is not affected by vibrations of other machines.
- If you are going to prepare a trench for isolating vibrations, dig it along the edge of the machine's foundation area.
- Concrete blocks cannot be used as foundation. Never place the NC machine on concrete blocks.
- Select an installation site that provides enough clearance around the NC machine to allow movement of a cart for removing cutting chips and so that operators can perform maintenance tasks such as removing the panels without bumping into other machines.
- (External dimensions of the NC machine: depth 1,220 × width 2,120 × height 1,835 mm)

■ Site environment:

- The NC machine must not be installed in a place exposed to sharp temperature changes, for example, near air conditioners.
- The NC machine must not be exposed to dust and direct sunlight or must not be placed near a ventilation opening or heat source or in a place of high humidity.

  If it is unavoidable to install the NC machine in such the location, a heavy curtain or a panel or screen can protect the NC machine from direct sunlight or ventilation air or heat.
- The work site should be well ventilated to prevent heat build-up.
- To prevent external dust from getting into the machine, air purge (internal pressure) is applied to the air circuit.

  The air circuits for the mechanisms are equipped with mist separators and dryers to clean air. However, if air is badly polluted at the factory, the air cleaning capacity is not enough. In this case, polluted air is supplied causing a machine failure.
  - Simple method for checking the cleanness of air at factory
    Blow the air to be used against a piece of white paper for about 30 seconds using an air gun. If the paper is as white as before the test, the air can be supplied directly to the machine. If stains are detected on the paper, do not supply the air directly to the machine.
  - Procedure if air pollution is detected
    The factory facilities need to be reviewed. To increase the air cleaning capacity, install additional air dryers, mist separators, and filters into the factory piping system.



Outside dimensions of the machine

■ NC machine installation standards:

WARNING

If you do not follow the precautions herein, high-frequency noise may cause malfunction of the machine, resulting in severe injuries or deaths.

- Power line

  Use a dedicated power line (200 VAC ± 10%) of the NC machine separately from the power lines of other machines and equipment that generate high-frequency noise. (See examples below.)

- Installation site

  Install the NC machine at least 20 m away from other machines and equipment that generate high-frequency noise. (See examples below.)

- Grounding of the NC machine

  Use 5.5 mm^2 or more thick grounding wire and ground the NC machine separately from the grounds of other machines and equipment. The ground resistance of the NC machine should be 100 ohms or less (class 3 grounding work).

  If the NC machine is installed near machines and equipment that generate high-frequency noise, provide a separate ground terminal within 5 m of the NC machine. The ground resistance of the NC machine should be 100 ohms or less.

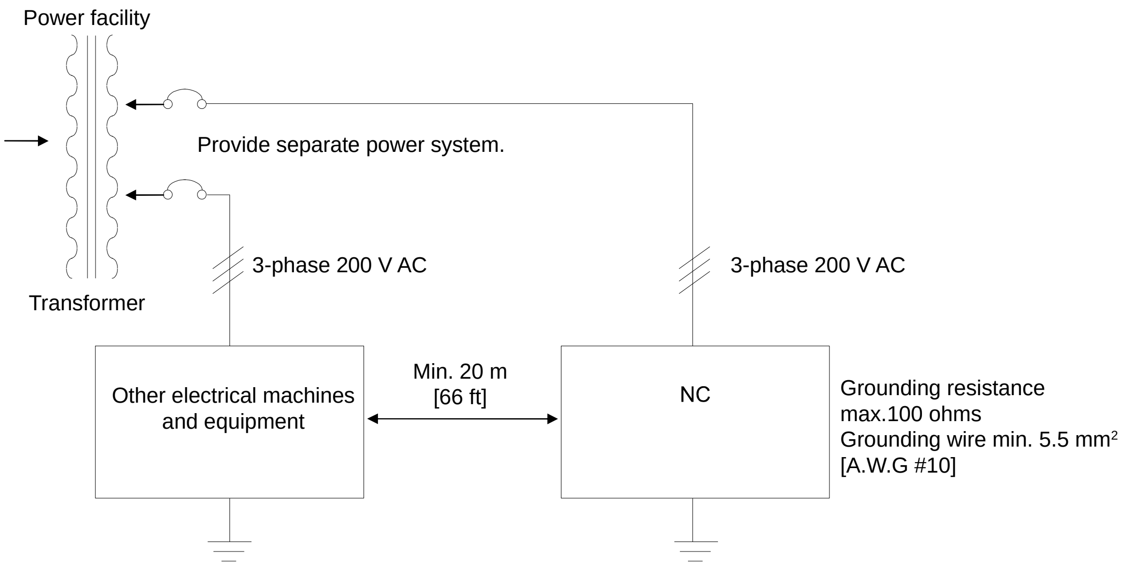
  Examples of equipment that generate high-frequency noise:

  - Arc welding machine
  - Resistance welding
  - High frequency dryer
  - Electric spark machine
  - Others

- If a circuit breaker with an electric leakage detection function is used as the power circuit breaker on the factory side, the sensitivity current must be 30 mA. If the sensitivity current is less than 30 mA, the circuit breaker may be disengaged abnormally.

■ Example of NC machine installation:

The illustration below shows an example on how to ground the NC machine installed near in the proximity of other machines or equipment.



WARNING

Operations needed at relocation should always be performed by the personnel who is authorized by the public agency, for example, in hoist slinging or folk-lift truck operation. Operations by unauthorized personnel may cause a serious accident.

24.3 Preparation for Transporting

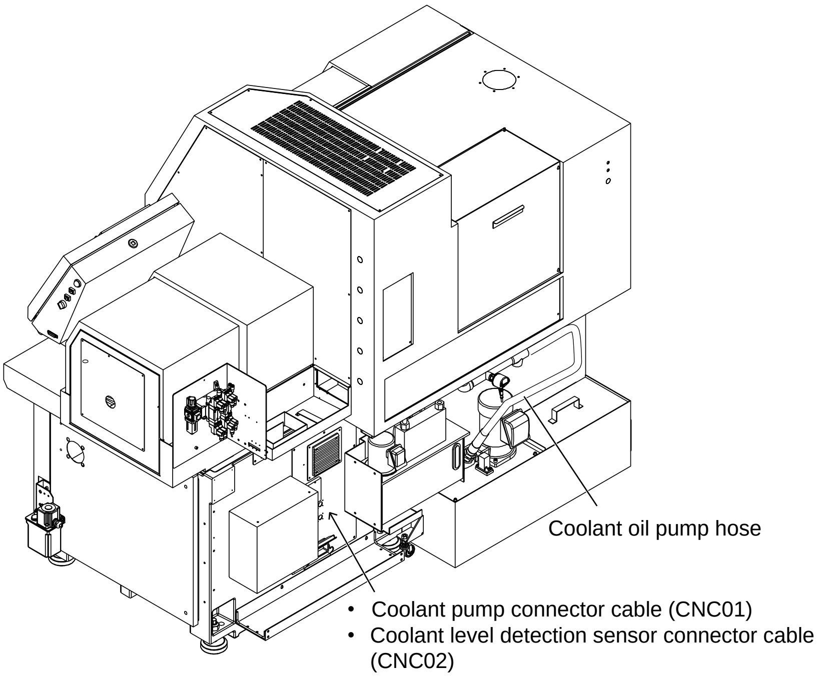
Use the following steps to prepare the machine for transportation.

[Procedure]

1. Save data stored in device memory on an external storage device as a backup copy.
   - **The NC machine will retain the data even if you disconnect it from the power outlet. However, to assure the safety of data, it is best to make backup copies before transporting the machine.**
2. Remove the external storage device.
3. Remove tools from the machine.
4. Disengage the main circuit breaker of the machine and the breaker of the factory power outlet.

> **DANGER**
>
> Make sure that the breaker of the factory power outlet and main circuit breaker of the NC machine are disengaged.
> Failure to do so may expose you to hazardous voltage that can cause severe injury or death.

5. Disconnect the coolant pump hose.
6. Disconnect the coolant pump connector cable.
7. Disconnect the coolant level detection sensor cable.



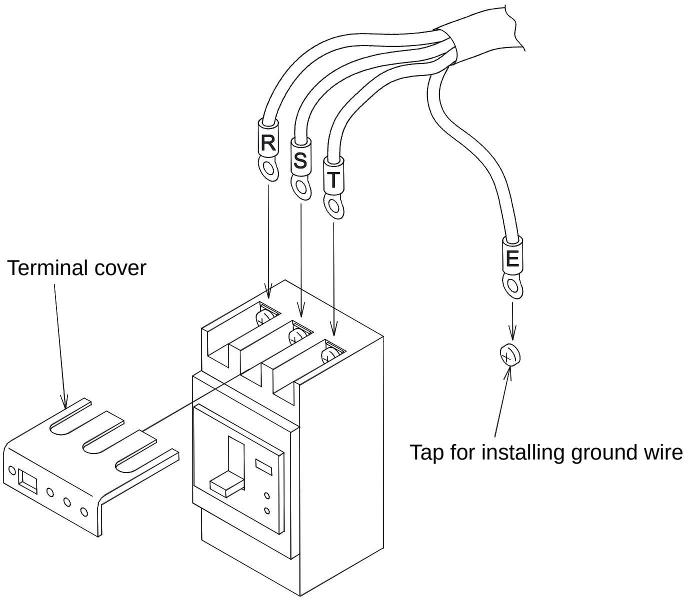
8. Pull out the coolant tank and remove the cutting chips and coolant from it.

   Note: Dispose of the waste oil according to all national laws and regulations.
9. Disconnect the power cable and grounding wire.

> **DANGER**
>
> Make sure that the breaker of the factory power outlet and main circuit breaker of the NC machine are disengaged.
> Failure to do so may expose you to hazardous voltage that can cause severe injury or death.

[Procedure]

1. Remove the cover on the right side of the control unit.
2. Remove the terminal cover.
3. Disconnect the power supply cable.
4. Disconnect the ground wire.
5. Replace the terminal cover.
6. Install the cover on the right side of the control unit.



7. If the bar feeder is mounted, disconnect its connector cable.
8. Remove cutting chips and clean the machine.
9. Secure the moving parts of the machine to prevent movement and damage during transportation.

Secure the spindles

Secure the spindles using the transfer jig provided with the machine (for Asia, USA, and Europe).
See the label "Securing the transfer jigs" attached to the machine for locations to secure.

Secure the Covers

Affix paper tape to prevent covers (such as sliding cover) and doors from moving during transportation.

CAUTION

Do not use tapes with strong adhesives. Such tapes may peel off the paint on the machine when you pull them off at the delivery site. Also, do not leave tape stuck on the machine for a long period. Otherwise, the paint on the machine may peel off when you try to remove the tape

Protect the operation panel

- Cover the LCD display with a protective material.
- Mount the transfer jig to the side of the operation panel and secure the panel to the machine.

Remove the patrol light and 3 step signal tower (optional)

- Loosen the holding nut at the base of the patrol light and 3 step signal tower.
- Unplug the connector to remove the patrol light and 3 step signal tower.

10. Loosen the four level adjustment bolts, and remove the machine from the bed while assuring safety

24.4 Transportation

Use a hoist sling or forklift truck to remove the machine from the current site and transport it to a new site or delivery vehicle.

24.4.1 Hoist sling

CAUTION

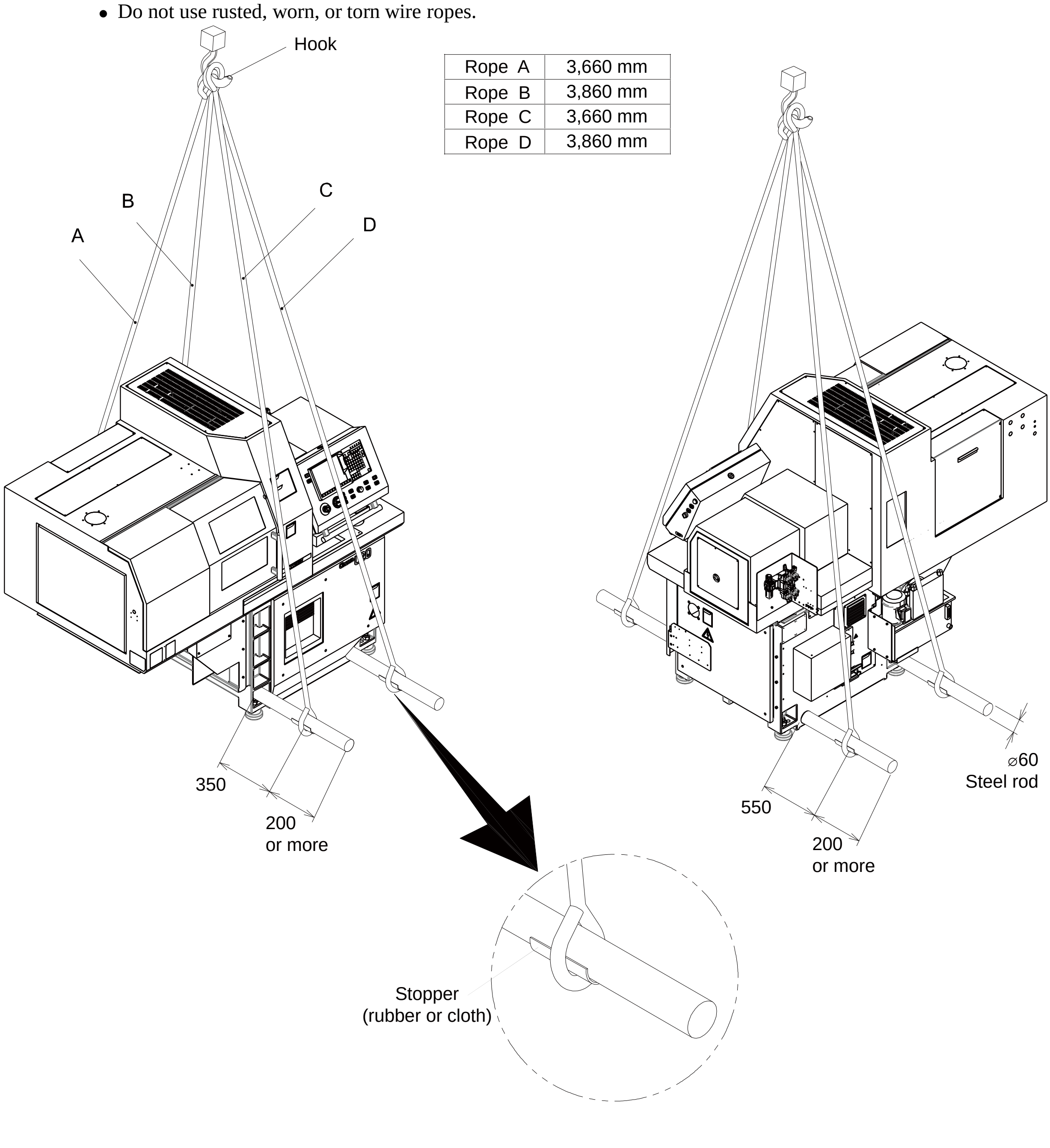
If you do not follow the precautions and instructions in this section, the machine could fall or overturn and cause severe injury or death.

If the large unit such as chip conveyor is equipped with the machine, remove it before transportation. The center of gravity will change with the large unit installed, and it may cause fall or overturn of the machine resulting in severe personal injury or death.

- The mass of the machine depends on the options mounted. The maximum weight is 2,400g.

  Nylon or wire ropes should be used for slinging the machine. They must be a minimum of 10 mm [0.39"] in diameter.
- Use a hook as shown below.
- Do not use rusted, worn, or torn wire ropes.

Rope A	3,660 mm
Rope B	3,860 mm
Rope C	3,660 mm
Rope D	3,860 mm



[Procedure]

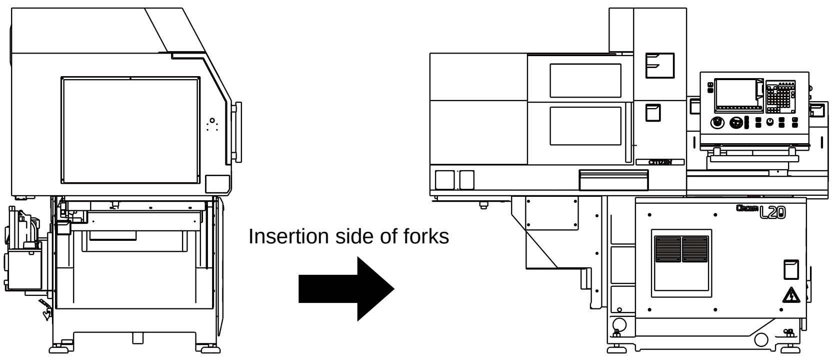
1. Insert a 60 mm [2.36"] diameter steel rod across the bolt holes on the lower bed.
2. Hook the slinging ropes on the steel rod as shown in the figure.
3. Hoist the rope gradually and stop hoisting when the wire rope becomes taught.
4. Check the sling to be sure it is safe and secure.
5. Hoist the rope until the machine is lifted of the floor.
6. Stop hoisting and check the sling again to be sure it is safe and secure.
7. If the sling is safe and secure, hoist the rope to the required height.
8. Transport the machine to a delivery vehicle or new site.
9. To place the machine on a vehicle or the ground, lower the machine gradually and stop just before the machine touches the surface.
10. Check the position of contact and adjust the position if necessary.
11. If the position is correct, lower the machine to the surface.

24.4.2 Forklift truck

WARNING

If you do not follow the precautions and instructions in this section, the machine could fall or overturn and cause severe injury or death.

- The mass of the machine depends on the options mounted. The maximum weight is 2,400g.
  Be sure to use a forklift truck that can handle this load.
- Be sure to have a supervisor attend the transport work together with a driver of the forklift truck so that projecting portions of the machine are not damaged.
- Be sure to perform a trial lifting from the front, and rear, or side before the transportation. Consider the position where the center of gravity is stable and good for lifting.



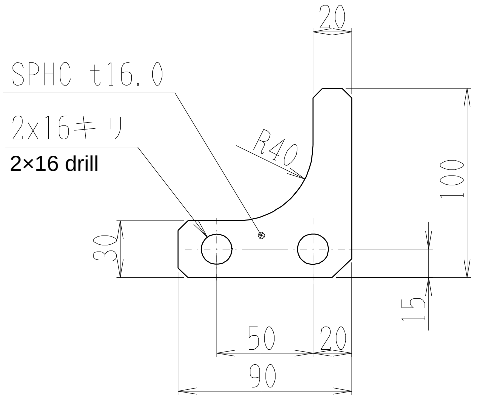
[Procedure]

1. Insert the fork into the cutout bottom position between the legs on the right side of the machine.
2. Identify the most stable position and lift the machine.
3. Transport the machine to the new site or delivery vehicle.
4. To place the machine on a vehicle or the ground, lower the machine gradually and stop just before the machine touches the surface.
5. Check the position of contact and adjust the position if necessary.
6. If the position is correct, lower the machine to the surface

24.5 Secure the machine using anchor bolts

To secure the machine using anchor bolts, refer to the following procedure. Machine configuration, site environment and installation of options may make it impossible to use the following procedure for securing a machine. Consider use of brackets specially designed for the purpose.
For details regarding machine dimensions and leveling bolt location dimensions, refer to *<3.2 Outside Dimensions and Layout of Machine>*.

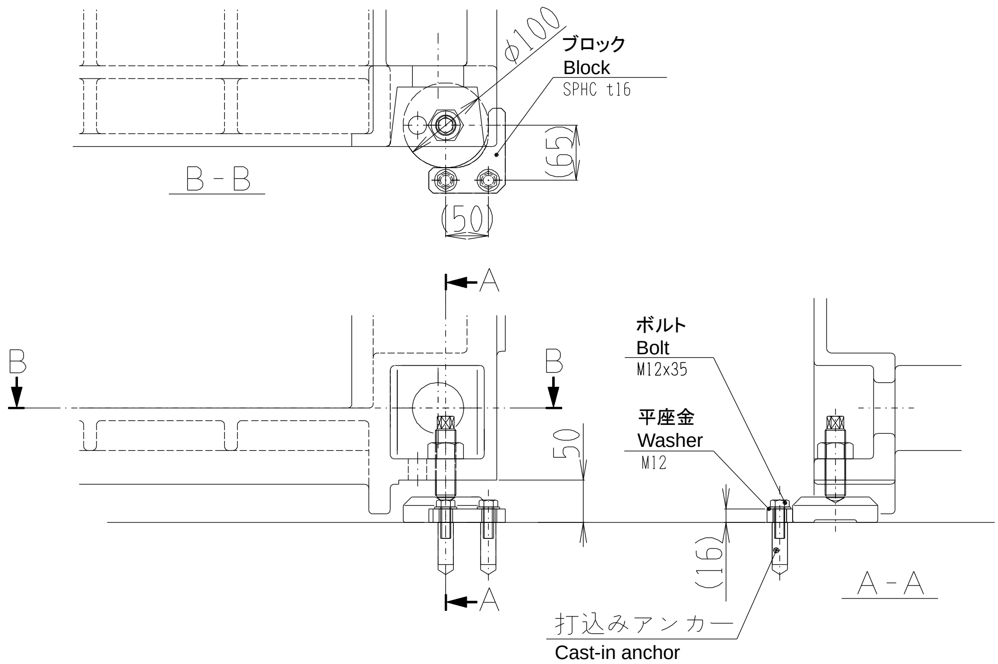
[A] Using anchor blocks
Use anchor blocks as shown below to restrain and secure the mounting seats.



形状及び寸法は参考です。
設置状況、製作上の都合に応じて
最適な寸法を決定してください。

Shape and dimension of block is for reference.
Determine the dimension appropriate to installation status and manufacturing circumstances.

Figure showing anchor block dimensions



Schematic diagram showing anchor block installation

[B] Using brackets to secure machine
Install the brackets shown below near the leveling bolts of the machine and secure the machine to the floor.
Tap holes for installing the brackets will have to be drilled.

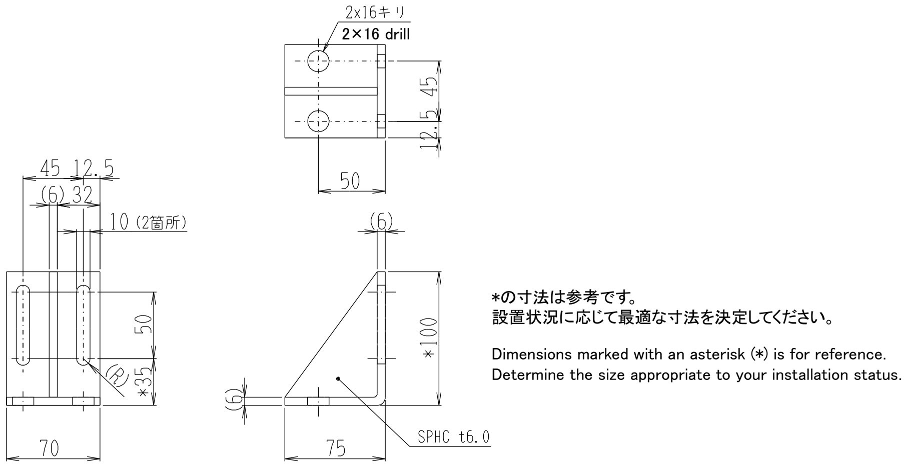
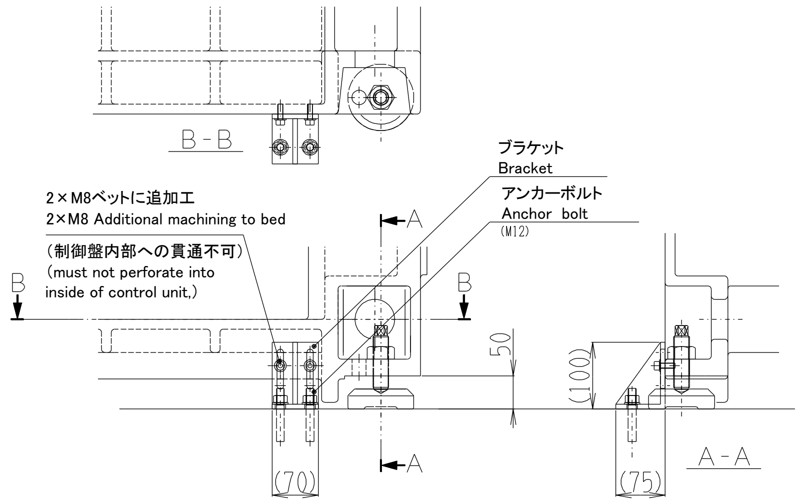


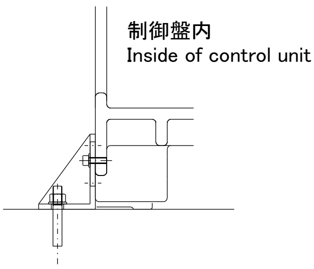
Figure showing anchor bracket dimensions

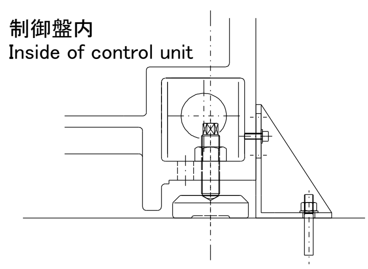


Schematic diagram showing anchor bracket installation

[Note]

Additional machining to the bed must be performed in locations where there is no risk of penetration into the bed. The control unit is inside the bed and there is a risk that additional machining could damage the inside and the resulting contamination from chips and oil could lead to machine malfunction.



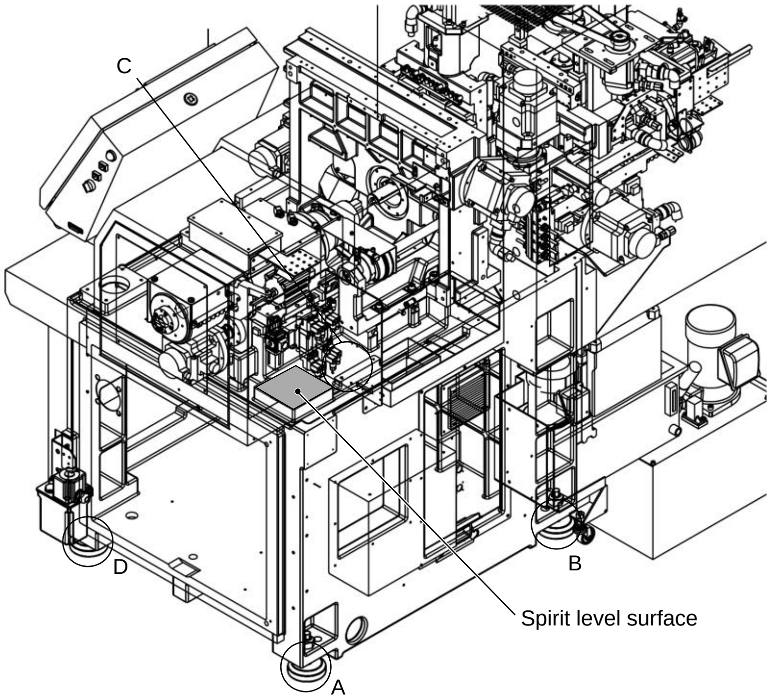


Regardless of the methods for securing the machine described above, we also design and manufacture anchor brackets for securing machines.

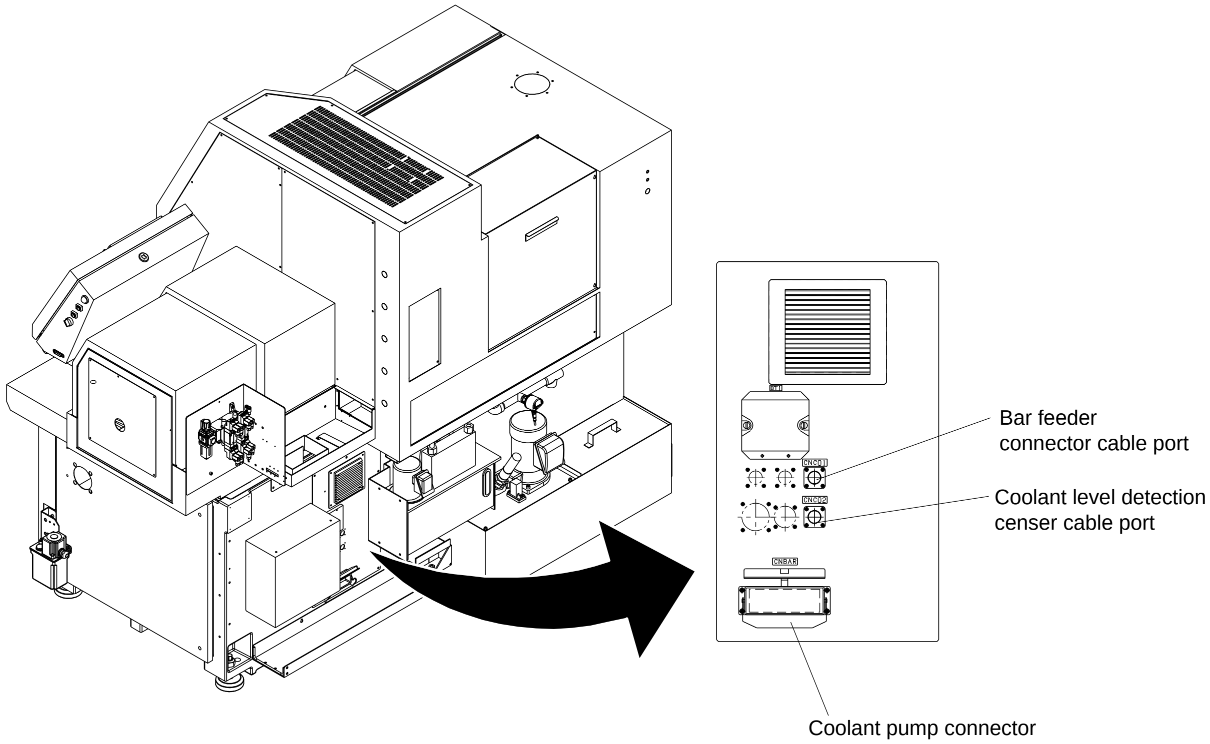
24.6 Installing the NC Machine

[Procedure]

1. Transport the machine to the installation site by hoist sling or forklift.
2. Gradually lower the machine and stop just before it reaches the floor.
3. Place the mounting seats under the machine and use leveling adjustment bolts and hexagonal bolts to secure it in four locations. Adjust the leveling adjustment bolts so that the machine is approximately level.
   - **As a guideline, place the coolant tank into position. If the coolant tank can be placed properly, the machine is approximately level.**
4. After ensuring that the machine is approximately level, remove the wire ropes and steel rods from the machine or withdraw the forklift.
5. Place a spirit level on the machining surface.
   - **Make sure that the machining surface is clean and that there is no foreign material between the machine and the level during measurement.**
   - **The spirit level sensitivity should be approximately 0.02 mm [0.0008"] per meter for each scale setting.**



6. Loosen the D leveling bolt and use the four leveling bolts and a spirit level to adjust the A, B and C leveling bolts so that the machine is level to within 0.04 mm [0.0016"] in both longitudinal and transverse direction. Finally, tighten the D bolt so that the mounting seats are firmly secured.
7. Check the level to be sure it is correct.
8. Remove the gummed paper tape and gummed tape from the sliding doors and covers.
9. Remove protective material from the liquid crystal display (LCD).
10. Reinstall the patrol light and 3 step signal tower.
11. Remove the transfer jig.
12. To mount the bar feeder, connect the connector cable.



13. Connect the coolant pump hose, coolant pump connector cable, and coolant level detection sensor cable.

14. Clean the machine by completing the following steps.

- **The machine may get stained during transportation by foreign materials such as dust, earth, and sand. Before starting, be sure to check the machine condition so that it will not be started in stained state.**

> **CAUTION**
>
> Do not use compressed air to remove dust.
> Doing so may cause a damage to the fittings due to foreign materials entered into the gaps between fittings.

Clean the machine with a dry cloth or a cloth soaked in high quality petroleum (neutral) to remove dust and foreign materials.

After cleaning, lightly apply a lubricating oil equivalent to Exxon Mobil Vactor Oil No. 2 to the finish surface.

15. Connect the power cables and grounding wire.

> **DANGER**
>
> Make sure that the breaker of the factory power outlet and main circuit breaker of the NC machine are disengaged.
> Failure to do so may expose you to hazardous voltage that can cause severe injury or death.

a. Ensuring the power breaker.

Ensure a dedicated power breaker (isolated from other machines) near the NC machine so that a machine operator can turn it on/off easily. The breaker capacity should be 40A.

b. Arranging the power cable and grounding wire.

1. Connect a grounding wire to the grounding terminal “PE” of the machine.
2. Connect the power supply cable to the breaker.

Electric wire thickness

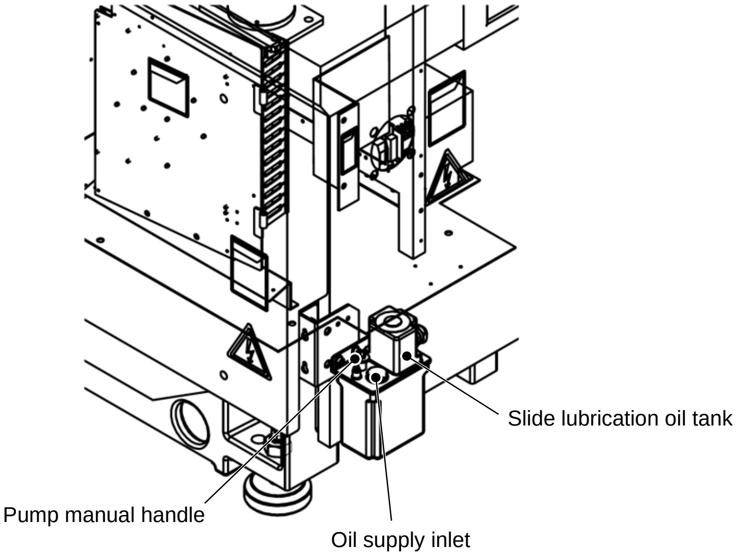
The power cable and grounding wire should conform to the following specifications.

Overall capacity	R.S.T	Grounding
8 KVA	5.5 mm^2	5.5 mm^2

- **Be sure to use the O-ring type ground jack. Do not use the Y-blade type jack, it is easily disconnected.**

16. Fill the tank of the lubrication device.

- **The tank can hold 0.8 liters. See the figure below.**



17. Fill the coolant tank on the left side of the machine with coolant.

- **The tank can hold approximately 160 liters.**

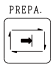
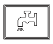
  Oil-based coolant is recommended to prevent frictional wear of material and guide bushing sliding mechanism.

24.7 Pre-operation Check

Use the following procedure to ensure that the NC machine functions properly before starting operations at the new work site.

[Procedure]

1. Engage the breaker of the factory power outlet.
2. Check the power supply voltage. The power supply rating must be 3-phase, 200 VAC.

   Measure the voltage of each 3-phase power cable and make sure the fluctuation is within 200 V ± 10%.
3. Engage the main circuit breaker of the NC machine.
4. Reset the Emergency Stop switch on the operation panel if it has been pressed.
5. Press the power ON switch on the operation panel to turn on the NC machine.
6. Select Preparation  on the operation panel and make sure the Preparation screen appears.
7. Press the Coolant button  on the operation panel to check the coolant motor is certainly turned on and off.

   Make sure the coolant oil pump rotates in the direction of the arrow indicated on the pump. If the connection is incorrect, the pump runs in reverse direction.
8. Perform the regular test operation procedures and checkups.

[Note]

The following alarm may occur after the machine has been transferred:

"EX401 DETECTION OF THE MACHINE MOVING"

"OP097 Machine relocation detected. Contact service. 00 hours until machine shutdown. "

This alarm occurs when the machine transfer detection function mounted on the machine detects the machine moving. When this alarm message appears, contact your nearest service center.

24.8 Disposal

Note the following when disposing of the machine.

- Be sure to remove all lubricant and coolant.
- Dispose of the machine and waste oil according to all national laws and regulations.

DANGER

A falling object may cause an accident resulting in death or serious personal injury

If you need to disassemble the machine, be sure to remove the headstock and the tool post or other component that might fall by gravity, then unlatch the brake.

(Blank Page)

Product Code	C-L220E XII
Document Code	3E1-2400
Mfg. No.	L220E/0001~
Issue Date	2013.10