Cincom A20

(2F7PL)

Programmer's Manual

CITIZEN MACHINERY CO., LTD.

(Blank page)

Remarks

- For safety operation, read thoroughly <Chapter 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 Citizen Watch Co., Ltd.
- The contents of this manual may be revised without prior notice.

  This manual applies to only the machine of the machine number shown on the back cover. Do not use manuals written for dealers and reference when operating the machine. Also, do not use this manual for other machines.
- The characteristics, functions, and operations of the machine explained in this manual do not apply to worldwide use. Some illustrations in the manual may not be identical to the actual machine.
- Citizen Watch Co., Ltd. has all copyrights regarding this manual. No part of this document may be reproduced in any form or by any means, electronic, mechanical, or photocopying, without the prior written permission of Citizen Watch Co., Ltd.
- Citizen Watch Co., Ltd. has copyrights regarding the software and programs described in this manual.
- This machine is a product controlled under the Foreign Exchange and Foreign Trade Control Law 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 us 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 Citizen beforehand.

Preface

- This Programmer's Manual is a part of the three-volume documentation prepared for the Cincom.

  It contains information necessary for programming the Cincom to perform a variety of basic machining. It explains programs and gives some useful examples.

  The remaining documentation is the Operator's Manual and the Maintenance Manual.

  The Operator's Manual is used when running the machine. It contains general information about procedures for basic daily machine operations.

  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 causes, 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.

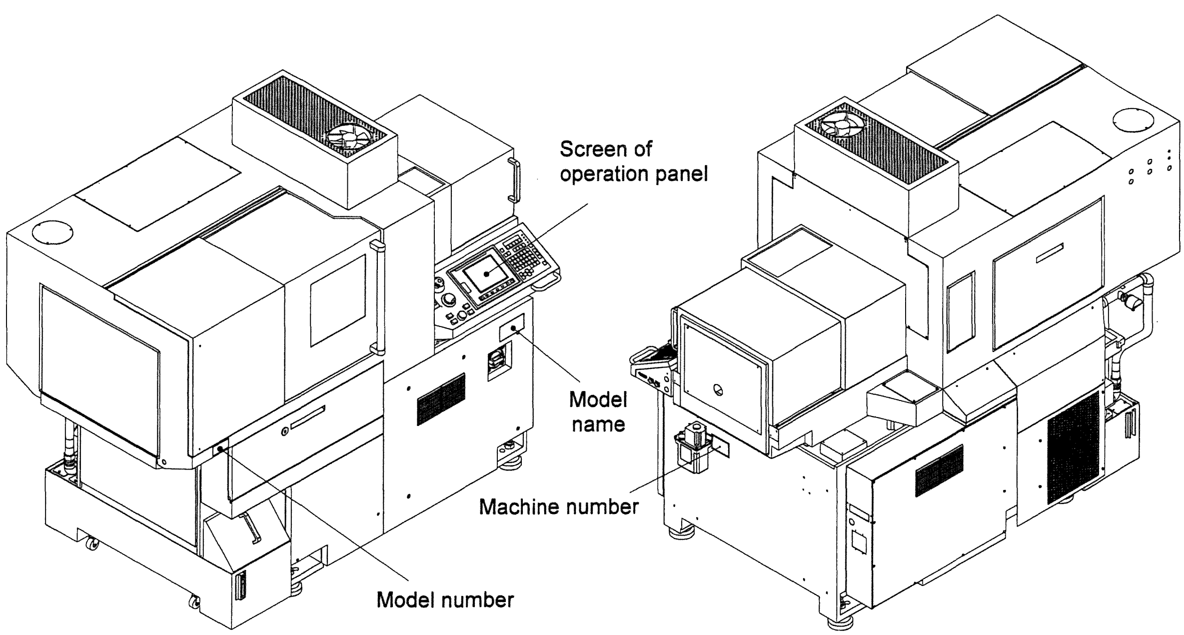
- This manual contains illustrations both for A20L and A20R. To confirm the names and locations of components, see the illustration appropriate to the machine you are using.

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	A20	Machine nameplate, operation panel, or the instruction manual
2	Model number (not indicated on some machine types)	2F7PL	Model sticker on the front panel of the machine, on the back of the instruction manual
3	Machine number	AC0001	Machine nameplate
4	NC unit	CINCOM SYSTEM FS32i	Operation panel or Machine Specification in <Section 3.3.2 NC functions> of the Operator's Manual
5	Delivery date	June, 2008	
6	Machine paint color	Silver 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
9	Software version	CNC SYSTEM G203 10A PMC (LADDER 1) L001 01 MACRO EXE 1 M001 0001 CEXE APL C001 0001 Parameter 0001 0001	On the screen of the operation panel. See <Section 5.7.7 Software Version> of the Maintenance manual

Before making inquiries about problems, please read <Chapter 5 Troubleshooting> of the Maintenance Manual and take action if necessary.



(Blank page)

Programmer's 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	–	A	2	2	0	P	L	Ⅶ						
Document Code	2	E	1	–	0	0	0	3							

1. Outline

(Blank page)

1.1 Outline of the Manual

This manual is intended to provide programmers with information that is needed to program the Cincom in order to perform a variety of basic machining.

It contains information about how to handle the machine while taking advantage of the best performance the machine can yield. Before starting the machine, read this manual and gain an adequate understanding of how to handle the machine safely.

The manual consists of the following chapters:

Chapter 1 Outline:

Explains the purpose and chapters of the manual.

Chapter 2 Safety Precautions:

Explains what is to be noted for safety during a lot of steps, including machine programming, setup adjustments, operation, and maintenance.
It also explains various safety devices provided, focusing on their use in an emergency.

Chapter 3 Fundamentals of Programming:

Gives basic knowledge needed for programming.

Chapter 4 Commands:

Explains in detail commands needed for programming.

Chapter 5 Sample Programming [Basic]

Gives simple examples of programming and adds explanations.

Chapter 6 Sample Programming [Advanced]

Gives rather complex examples of programming and adds explanations.

Chapter 7 Sample Programming [Secondary Machining]

Gives examples of secondary machining programming and adds explanation.

Chapter 8 Sample Programming [Back Machining]

Gives examples of back machining programming and adds explanation.

Chapter 9 Programming Practices:

Gives programming practices.

Chapter 10 Cutting Conditions:

Gives various cutting conditions necessary for machining.

Chapter 11 Tooling:

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

1.2 Audience

This manual is intended for programmers who want to make products using the Cincom.

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

(Blank page)

Product Code	C	–	A	2	2	0	P	L	C	L	VII			
Document Code	2	E	1	–	0	1	0	1						

2. Safety Precautions

(Blank page)

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.

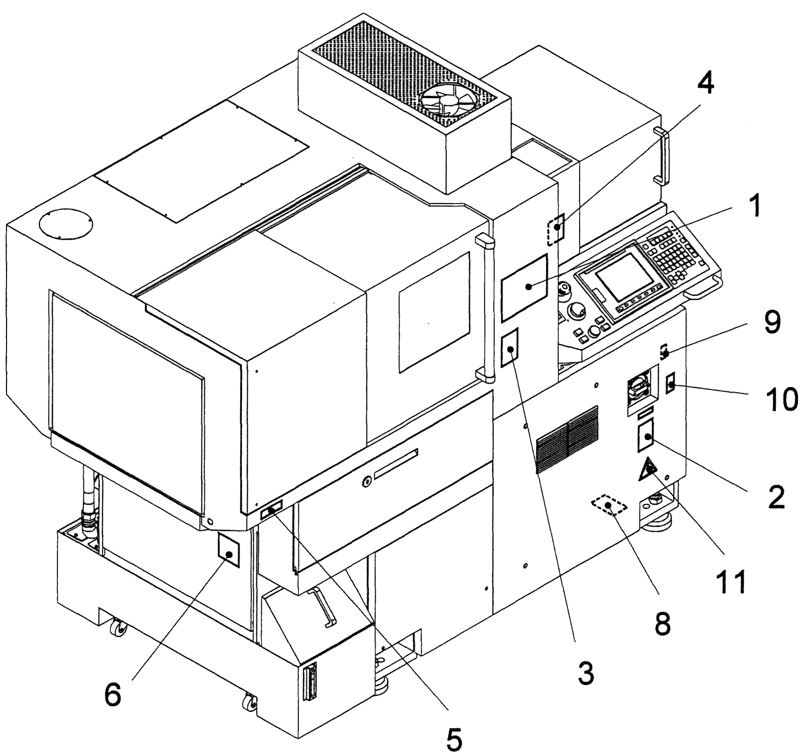
WARNING; indicates a potentially hazardous situation which, if not avoided, could result in death or serious personal injury.

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.



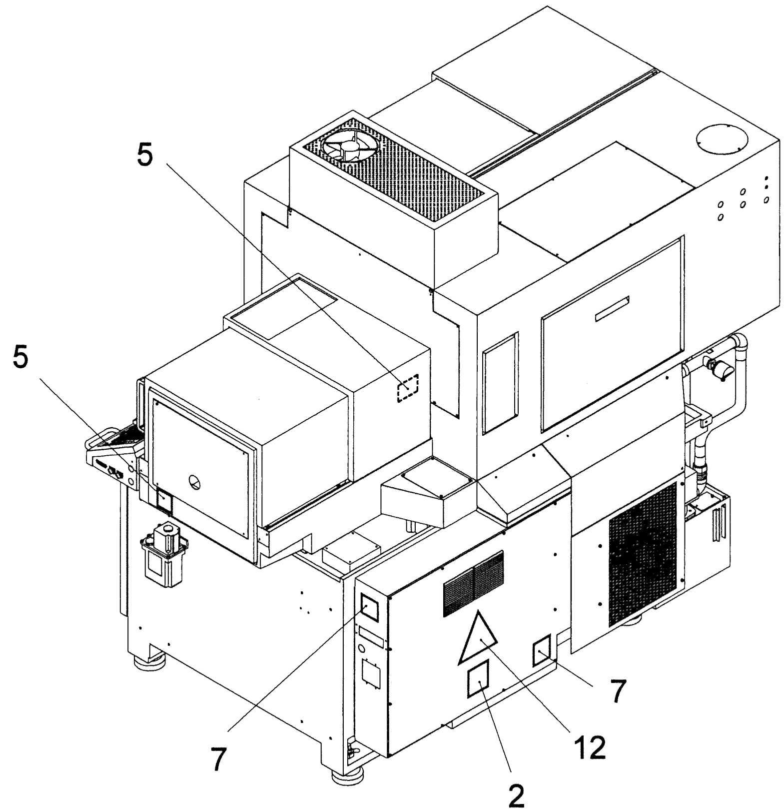


Figure 2.1-1 Locations of warning labels

1

⚠ WARNING

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

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

⚠ WARNING

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

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

EW201

2

⚠ WARNING

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

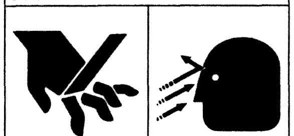
Only qualified personnel totally familiar with electrical circuits and service manual should work inside this enclosure.

Follow Lockout/Tagout.

EW01

3

⚠ WARNING



Moving part, hot chips and hot oil inside.

Keep door closed during operation.

EW02

4

⚠ WARNING

Moving parts inside.

Keep door closed during operation.

EW03

5

⚠ WARNING

Moving parts can cause serious injuries.

Keep hands and body away from moving parts.

EW04

6

⚠ WARNING

Turn off machine before removing chips inside.

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

EW05

7

⚠ WARNING

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

Turn off main breaker before connecting or disconnecting coolant pump.

EW07

8

⚠ DANGER

Hazardous voltage inside.

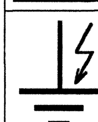
Will shock, burn, or cause death.

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

ED01

9

⚠ WARNING



GROUND

GETTING ELECTRIC SHOCK OR NOISE CAN OCCUR.

THE UNITS MUST BE GROUNDED.

(RESISTANCE VALUE MUST BE 100Ω OR LESS)

A-001

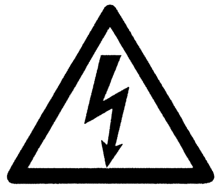
10

ATTENTION

Turn off a
lighting before
turn off a
main breaker.

EW22

11



ATTENTION high voltage!
Opening by qualified personnel only!

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 your feel dangerous while operating the machine. Note, however, that pressing these emergency stop button during machine operation might damage a tool as well as the product being processed. To reset the emergency stop state, first verify your safety, turn the locked emergency stop button clockwise to release the lock, and turn the NC power and main breaker off once and then turn them on again. Also return all the mobile sections of the machine to their return positions and then remove all the workpieces subject to machining from the machine.

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

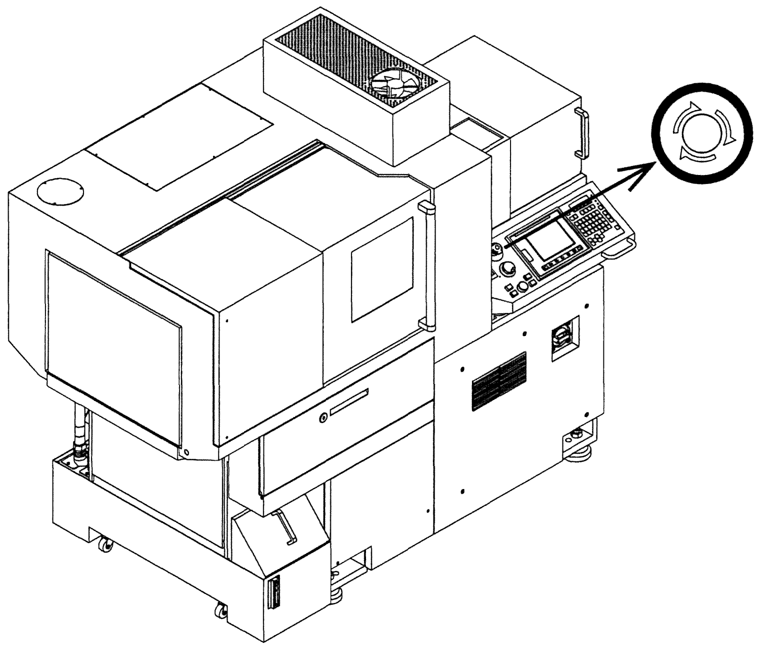


Figure 2.2-1 Location of the emergency stop button

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.

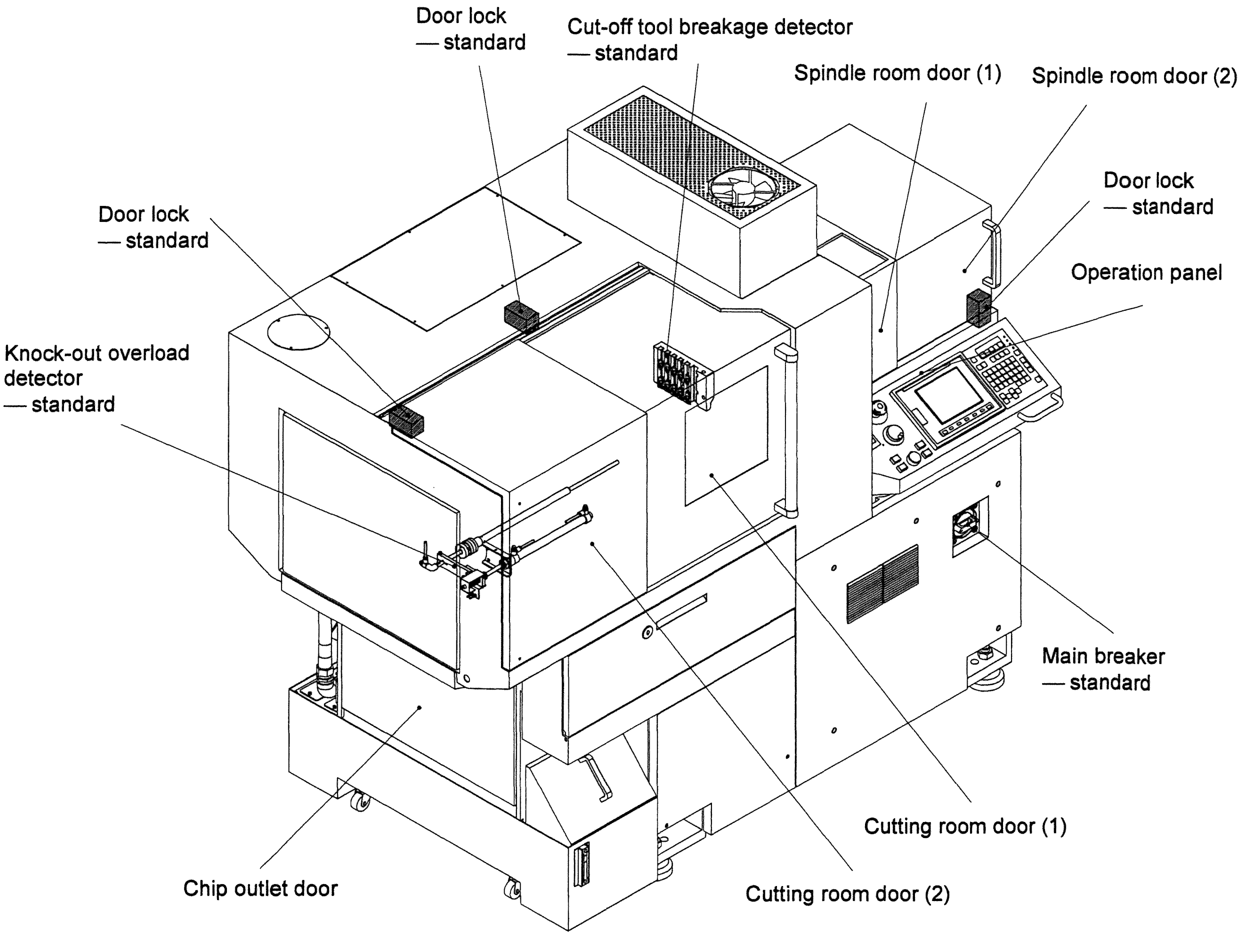


Figure 2.3-1 Locations of the safety devices (front view of the machine)

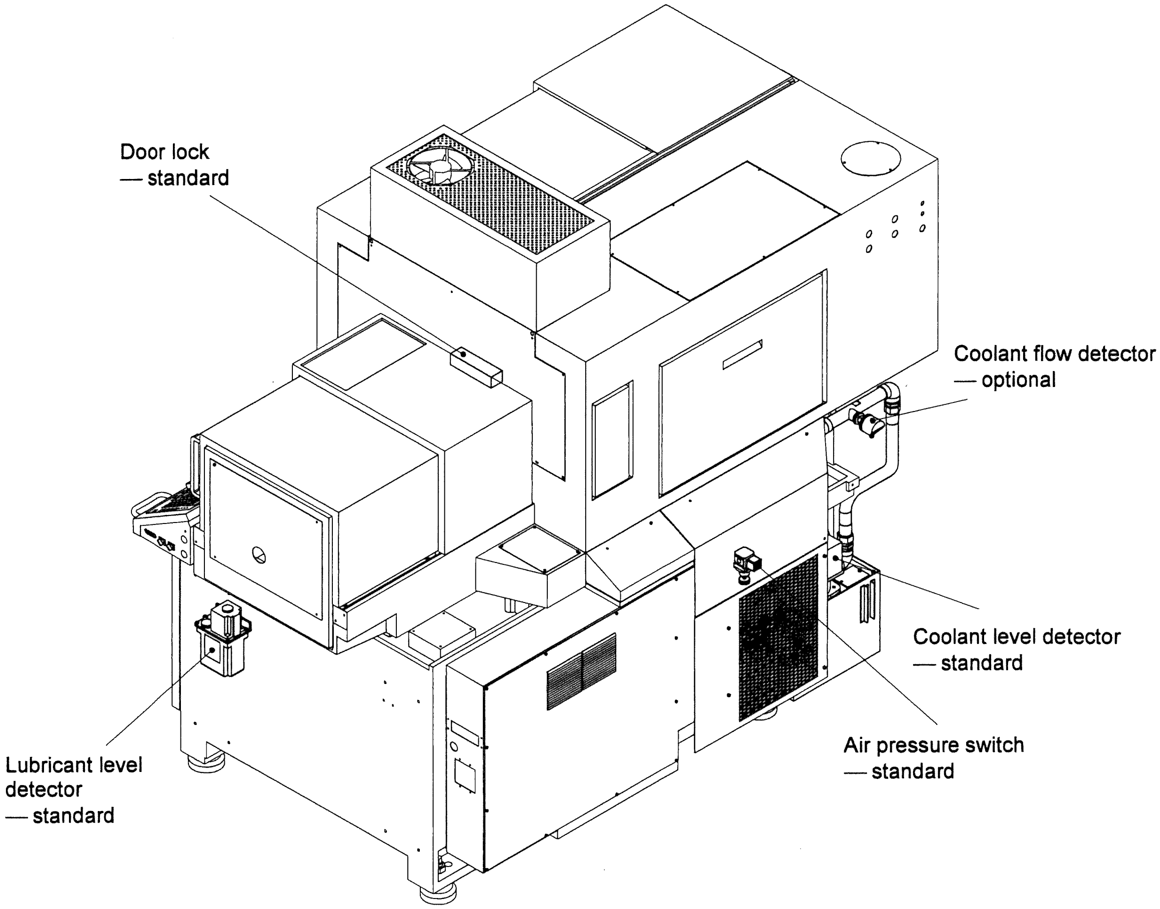


Figure 2.3-2 Locations of the safety devices (rear view of the machine)

The following are detailed descriptions of the safety devices:

Door locks — standard

Door locks prevent any person from opening the cutting side door (1), (2) and main spindle side door (1), (2) during machine operation. You will only be able to open these doors when the machine is stopped. See Figure 2.3-1.
In the Handle Feed or Preparation mode, however, operations (excluding operations regarding the main spindle) are performed at a speed of up to 2 m per minute even with such doors left open. In addition, manually opening or closing the chuck, manually rotating the spindle, and turning coolant supply on or off are performed whether the doors are open or closed.

Main breaker — standard

This device automatically shuts itself off when it detects an over current of 30A 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 "1105 TOOL BIT ALARM" 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 "1201 COOLANT OIL EMPTY" 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 "1200 LUBRICATING OIL EMPTY" is displayed and the operation of the machine is automatically stopped after completing one cycle to prevent machine damage.

Air pressure switch — standard

This switch is used to check the air pressure of main spindle and guide bushing device to prevent damage to their bearing. If an abnormal high (or low) pressure is detected, the alarm message "1132 PNEUMATIC PRESSURE ALARM" is displayed, and the machine automatically stops operation.

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 "1145 KNOCK OUT OVERLOAD" is displayed and the operation of the machine is automatically stopped to prevent a machine damage.

Coolant flow rate detector — optional

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 "1106 COOLANT OIL ALARM" 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 — optional

This function detects excessive changes in spindle speed to prevent machine damage. The alarm message "SP9002 SSPA: 02 EX DEVIATION SPEED" is displayed when it detects a change of more than 10% from the specified speed.

Note, however, that this function must be disabled when the spindle synchronization control function or the constant surface speed control function (standard function) is used and during tap and die machining.

Interference check — standard

This function checks for interference between back spindle and opposite tool post. When the function determines the possibility of interference during machine operation, the NC alarm message "INTERFERENCE: ~ " 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

For the sake of safety, the startup, stop, and operation speeds of the machine are defined as shown below.

	When a door is open: (The Door select switch key DOOR SW is set to "O".)	When all the doors are closed:
Automatic operation Program operation Program check MDI operation Preparation Cut-off process Phase adjustment	Stop and startup not permitted	Specified speed (Override enabled)
Preparation Start position Cut-off tool moves to the waiting point Tool set Return position Backward movement of gang tool Manual set Positioning point Backward movement of back spindle	Operation at a speed of up to 2 m per minute (Override enabled)	Operation at the specified speed (100%) (Override enabled)
Others Opening/closing of chuck with the SP.CHUCK key	Operation at the specified speed (100%)	Operation at the specified speed (100%)

The standard door locks is provided with the splash guard of the machine.

Note

If you open the doors when the Door select switch key DOOR SW is set to " I ", an alarm will be issued and you cannot operate the machine.

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:

- **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 key $\boxed{I}$ 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.
- **Never change the machine and control circuit.**
- **Press the emergency stop button stop all machine motion when there is an emergency situation.**
  To reset the machine after an emergency stop, make sure that the machining situation is safe, reset the pressed emergency stop button, cycle power to the machine, return all machine moving parts to their return positions, and remove all materials in progress 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 9 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 <Section 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.
- 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 front left and right 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 either front left or right doors are open during machine setup, do not select a tool, or do not operate the main and back spindles or the tool spindle drive. 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

Follow these general safety practices to prevent fires:

- Keep all open flame or sparks (cigarettes, lighters, etc.) at least one meter away from the machine.
- Always perform the necessary daily maintenance.
- Always operate the machine within its capacity and according to the procedures outlined in the Operator's Manual and in this chapter.
- 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.

Before operating the machine each day, perform these checks to help prevent fires:

- Inspect the tool for chips or cracks and make sure that the cutting capacity is adequate; the machine and tools must be suitable for the conditions and feeds of the machining program.
- Make sure that enough coolant is discharged to the machining position.
- The oil level in the coolant tank must be adequate.
- Make sure that the coolant flow path is clear.
- Make sure that the coolant hose is not twisted, and not damaged, there are no loose connections, and there is not too much chip accumulation on any curved portion of the hose.
- Inspect the chip receiver box and coolant tank for too much chip accumulation.
- Make sure all front left and right doors are securely closed.

During automatic operation, regularly perform these checks to help prevent fires:

- Inspect for chipped tools, excessive vibration and noise, and too much chip accumulation. Inspect finished workpieces for surface finish conditions that indicate tool wear or damage.
- Make sure the coolant nozzle is directed to provide optimum lubrication and chip removal.
- Check for too many jammed chips and remove as necessary. Check the coolant tank periodically for too much chip accumulation. Remove chips as necessary. Too much chip accumulation can reduce the effectiveness of the coolant pump.
- Make sure all machine covers and doors are securely closed.
- When doors are open, coolant mist or vapor may escape from the machine and catch fire outside the machine.

Prevent the following situations in which coolant supply may be insufficient, and possibly cause a fire.

- A chipped tool can induce an abnormally high friction temperature between the tool and the material.
- Chips can get caught on the machining surface and reduce the cooling effect of the coolant.
- Long chips may hit the coolant nozzle and change its position so that no coolant is applied to the machining position.
- If the coolant in the tank is at a low level or the mesh filter of the chip receiver box is blocked with chips, the coolant discharge may be insufficient.
- When the room temperature is abnormally high, the coolant may vaporize.

The following situation is especially hazardous. Stop the machine operation and shutdown the main breaker of the machine immediately, and contact the Cincom Service.

- If the machine's electrical system fails and causes a spark, it may ignite the coolant mist or vapor inside or outside around the machine.

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 switch O.

After that, press the Power on switch 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 <Section 6.5 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 <Section 6.5 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 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 <Section 9.1 Selecting the Installation Site> in the Maintenance Manual.

Product Code	C	–	A	2	2	0	P	L	C	L	VII			
Document Code	0	E	1	–	0	2	0	1						

3. Fundamentals of Programming

(Blank page)

3.1 What is NC Programming?

NUMERICAL CONTROL

What is NC ?

(Numerical control)

PROGRAM

What is program ?

Sends information (commands) made up of numerical values to the processing unit to control the machine operation.

Instructions for controlling the machine according to the predetermined procedures.

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.

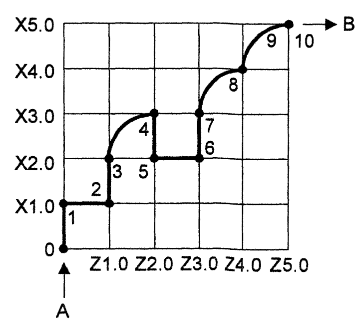
Before reading ahead for details on NC programs, try the following question as an example showing that NC programming is easy.

Q: Use the following four commands to complete the code to move from A to B in the figure.

1)	G01	Move straight to xx.
2)	G02............................	Move circularly to xx.
3)	M03	Turn on the start switch.
4)	M02	Turn off "goal-in" switch.

Code

```
 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     (Move circularly 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 (  )
```



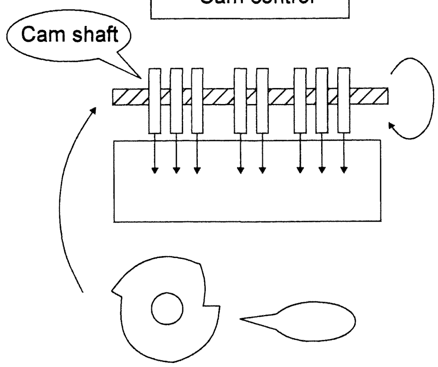
The NC program is almost the same as codes above.
(The answer to this above question is given on page 3-17.)

3.1.1 Difference between cam control and NC

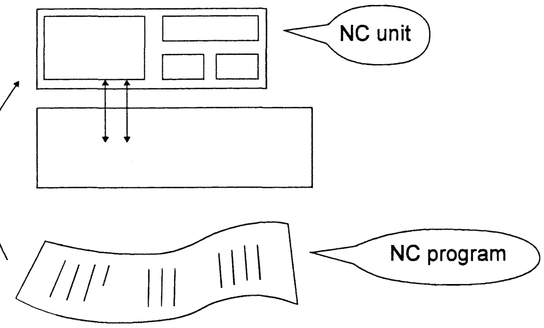
What are the differences between cam control and NC?

Where is the brain of the machine?

Cam control



Numerical control (NC)



How do you operate the machine?

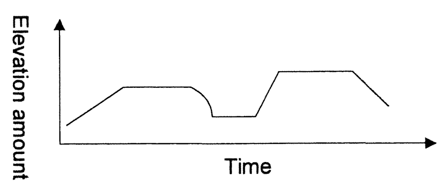
Cam control

Mechanical

The up/down movement of the cam is transmitted directly to the mechanism via the follower to lever.

Analog

The cam is first designed. The up/down movement of the cam in relation to time is continuously given.



Numerical control (NC)

Electrical

The information of the program is converted into electric signals, to operate the mechanism by driving motors and turning on/off switches.

Digital

The NC program is created and the operation commands are constructed step by step according to the process.

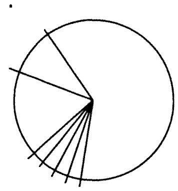
Use chuck
↓
Rotate the spindle (in 2,000 min^{-1})
↓
Select front turning tool and perform positioning
↓

What is a program like?

Cam control

1 cycle is angular

The entire process must be completed within 360°.

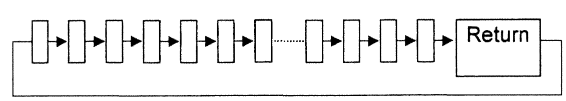


- Limitation to complex and long-time machining.

Numerical control (NC)

1 cycle is linear

As many commands as required can be specified.



- No limitation however long or complex the machining is.

Which is easier?

Cam control

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.

Numerical control (NC)

Software

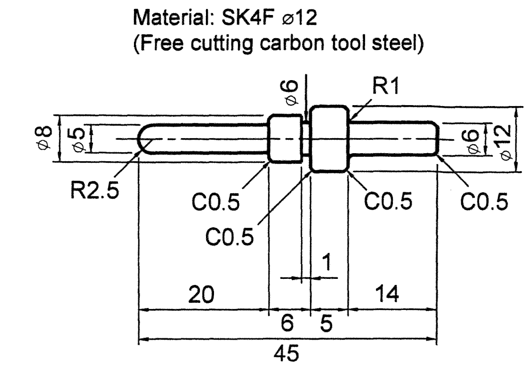
Just type the commands according to the designated manner. The program can easily be rewritten.

- Very short time is required for the preparatory stage.
- Correction and modification is very easy.
- 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.

3.2 Process of Program Creation

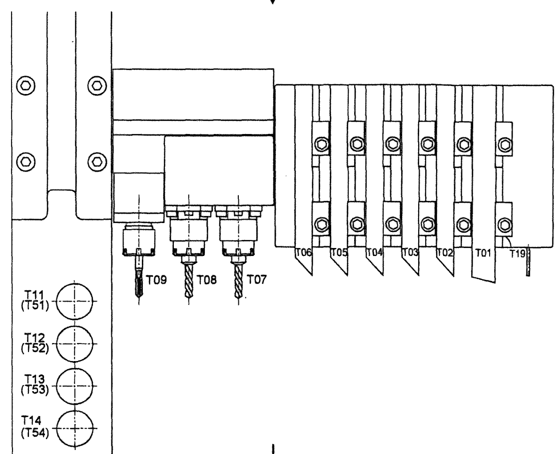
1 Drawing

- The drawing is the basis for an NC program.



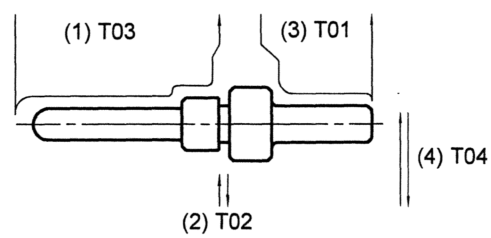
2 Creating the Tool Layout

- 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.



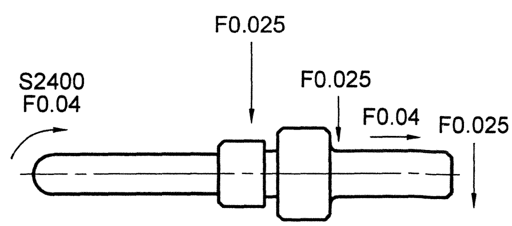
3 Creating the Machining Layout

- Specify the number of the tool used, the cutting path, and cutting procedure.



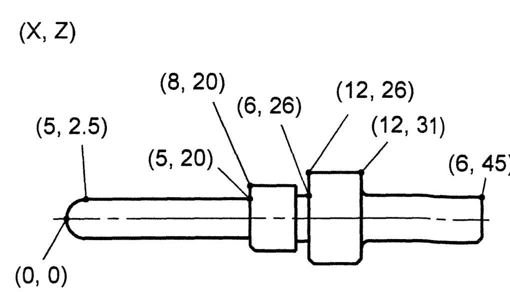
4 Setting the Spindle Speed and Feed Rate

- 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.

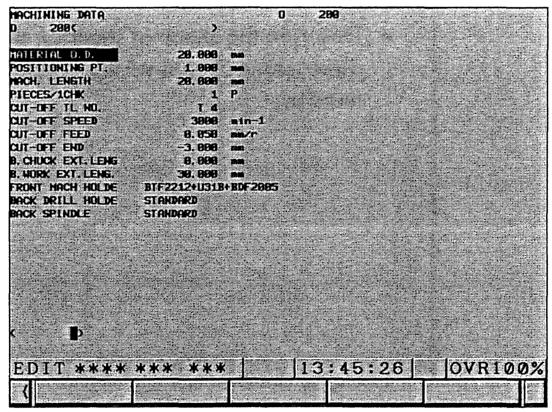
```
O0002
$1                    $2
G50 Z-0.1             G600
(M52)
M06                   G999
G00 Z-0.6             N999
S1=2400 M03           M02
G600                  %
N0103 T0300
X13.0 Z0
```

7 Entering and Saving the Program

- Enter the written program sequence on the paper into the NC unit and save them.

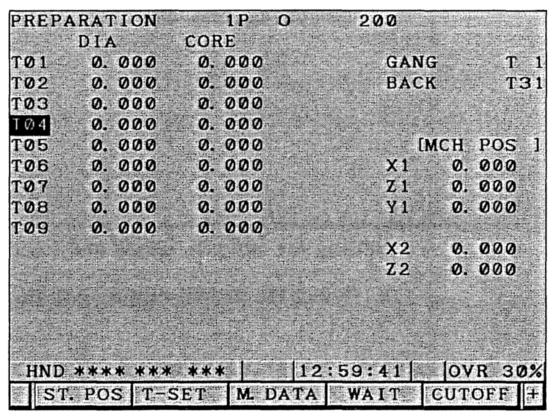
8 Entering the Machining Data

- Enter prerequisite data such as the holder name to be used, material diameter, and machining length.



9 Enter the tool setting data

- Enter prerequisite data such as the core position, diametrical direction, and longitudinal direction.



10 End

- With this, the programmer's jobs are completed.

3.3 NC Program Structure

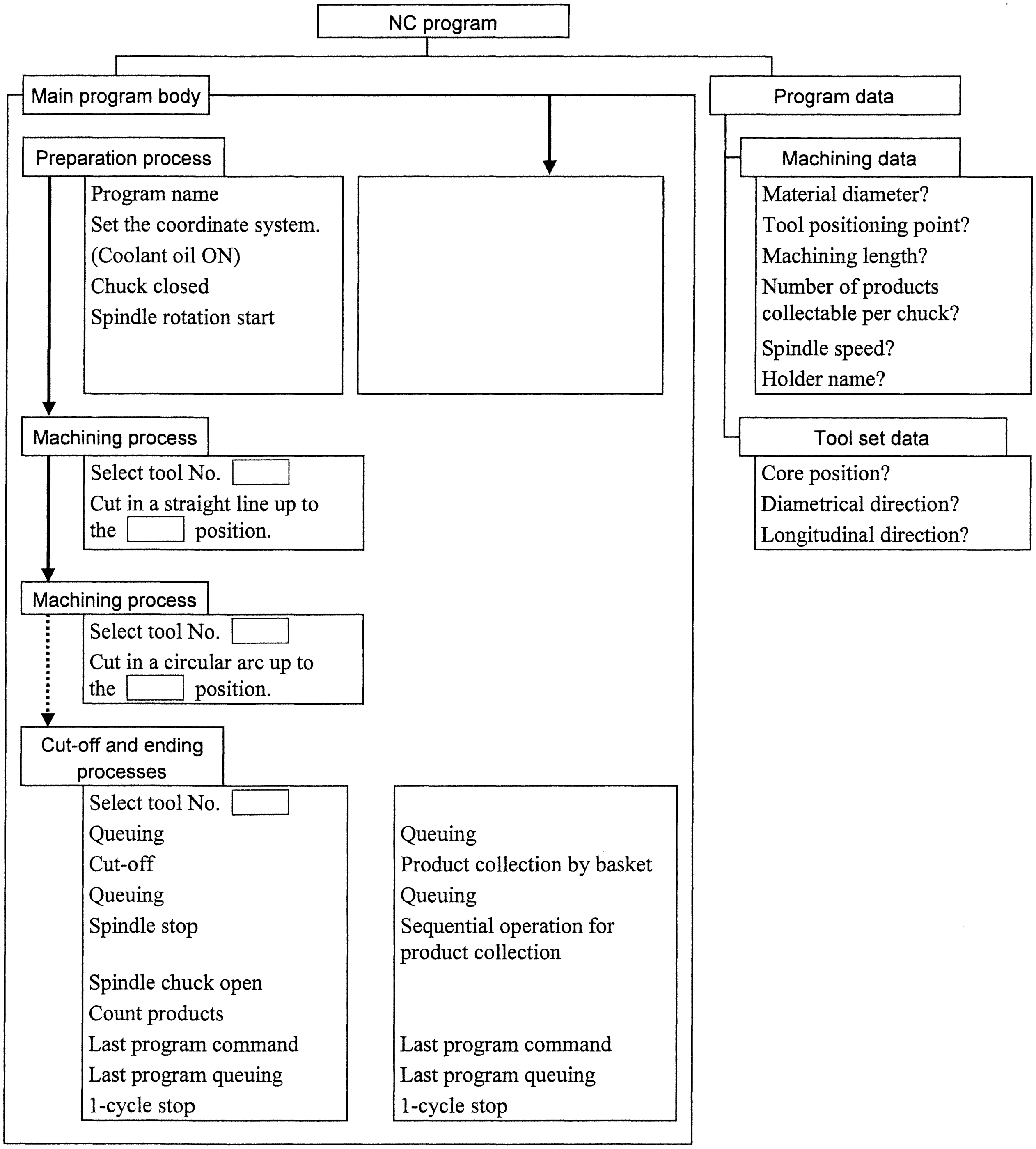
The NC program consists of a main program body and program data.

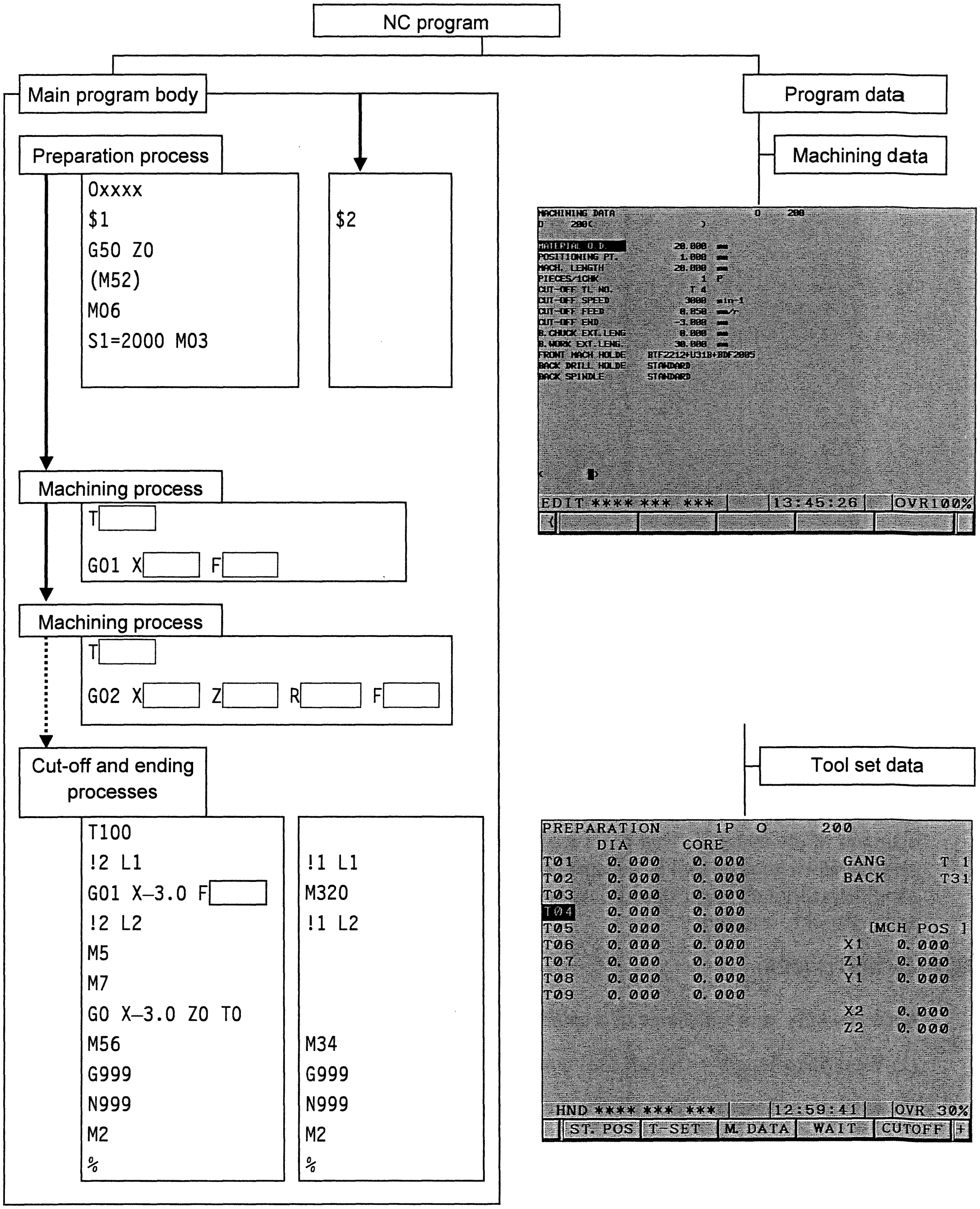
In the main program body, operation commands for a machine are arranged in the order of each machining process. This is usually called a program.

In the program data, the preconditions under which a program operates are arranged.

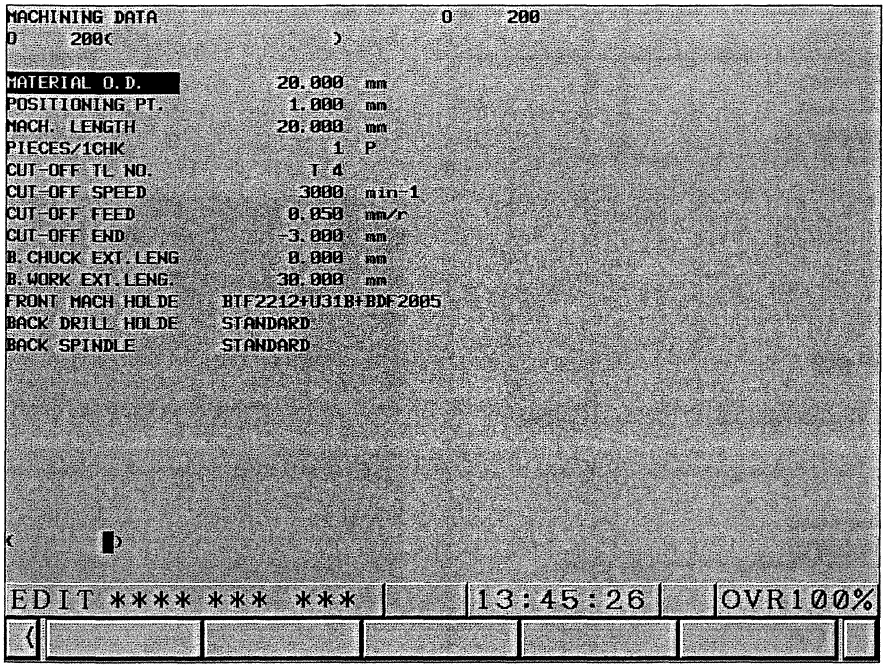
This program is not only required for processing on a machine, but also it serves as the communication means between the personnel.

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.


NC program
Main program body
Program data
Preparation process
Program name
Set the coordinate system.
(Coolant oil ON)
Chuck closed
Spindle rotation start
Machining data
Material diameter?
Tool positioning point?
Machining length?
Number of products collectable per chuck?
Spindle speed?
Holder name?
Machining process
Select tool No.
Cut in a straight line up to the position.
Tool set data
Core position?
Diametrical direction?
Longitudinal direction?
Machining process
Select tool No.
Cut in a circular arc up to the position.
Cut-off and ending processes
Select tool No.
Queuing
Cut-off
Queuing
Spindle stop
Spindle chuck open
Count products
Last program command
Last program queuing
1-cycle stop
Queuing
Product collection by basket
Queuing
Sequential operation for product collection
Last program command
Last program queuing
1-cycle stop


NC program
Main program body
Program data
Machining data
Preparation process
Oxxxx
$1
G50 Z0
(M52)
M06
S1=2000 M03
$2
MACHINING DATA
O 200
O 200( )
MATERIAL O.D. 20.000 mm
POSITIONING PT. 1.000 mm
MACH. LENGTH 20.000 mm
PIECES/1CHK 1 P
CUT-OFF TL NO. T 4
CUT-OFF SPEED 3000 min-1
CUT-OFF FEED 0.050 mm/r
CUT-OFF END -3.000 mm
B.CHUCK EXT.LENG 0.000 mm
B.WORK EXT.LENG. 30.000 mm
FRONT MACH HOLDE BTF2212+U31B+BDF2005
BACK DRILL HOLDE STANDARD
BACK SPINDLE STANDARD
EDIT **** *** *** 13:45:26 OVR100%
Machining process
T
G01 X F
Machining process
T
G02 X Z R F
Tool set data
Cut-off and ending processes
T100
!2 L1
G01 X-3.0 F
!2 L2
M5
M7
G0 X-3.0 Z0 T0
M56
G999
N999
M2
%
!1 L1
M320
!1 L2
M34
G999
N999
M2
%
PREPARATION 1P O 200
DIA CORE
T01 0.000 0.000
T02 0.000 0.000
T03 0.000 0.000
T04 0.000 0.000
T05 0.000 0.000
T06 0.000 0.000
T07 0.000 0.000
T08 0.000 0.000
T09 0.000 0.000
GANG T 1
BACK T31
[MCH POS]
X1 0.000
Z1 0.000
Y1 0.000
X2 0.000
Z2 0.000
HND **** *** *** 12:59:41 OVR 30%
ST. POS
T-SET
M. DATA
WAIT
CUTOFF

3.3.1 Machining data (screen display)



MATERIAL O.D. (#16)

Enter the outside diameter of materials to be machined.

POSITIONING PT. (#20) MATERIAL O.D.+

The tool positioning point is the point at which a tool is positioned when selected from among tools T01 to T09. 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 T09 and to which the tool escapes when another tool is selected.

MACH. LENGTH (#24)

Enter the max. move distance of a spindle required during workpiece machining.

(1) Workpiece length + Cut-off tool width or back turning tool width

(2) Workpiece length + Secondary process tool shift amount

Enter step (1) or (2) described above (as required in the program).

PIECES/1CHK (#28)

Enter the number of products to be unloaded in the program.

CUT-OFF TL NO. (#32)

In this machine, T1 is normally used as a cut-off tool.

CUT-OFF SPEED (min^{-1}) (#36)

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) (#40)

The tip of materials is cut at this feed rate specified by CUT-OFF in Preparation screen or M108 Material Exchange program.

CUT-OFF END (#44)

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.

B.CHUCK EXT.LENG (#76)

When a back long-neck chuck is used, enter the protorusion length of long-neck chuck from the back spindle cap nut.

B.WORK EXT.LENG. (#48)

Enter the protorusion length of workpiece from the end face of back spindle chuck.

FRONT MACH HOLDE (#52)

Enter the name of the gang tool holder attached to the machine.

BACK DRILL HOLDE (#60)

Enter the name of the back tool holder attached to the machine.

BACK SPINDLE (#64)

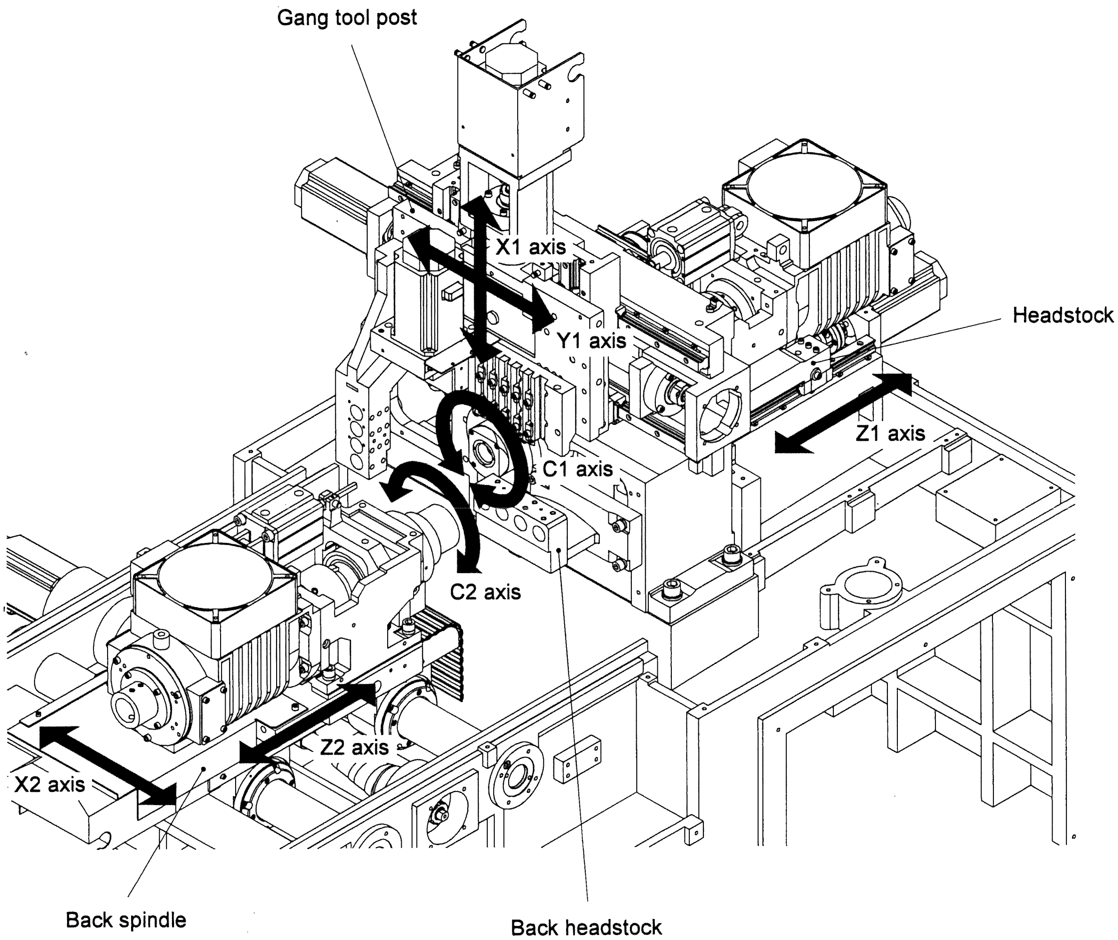
Enter the type of the back spindle attached to the machine by selecting STANDARD, BASKET or SUPPORT.

Notes

- The machining data is automatically output at the same time when the program is output. "#016=0000020000 (material outside diameter 20.0)" appears after $0 when viewing the program outside the machine.
- The value entered as the back long-neck chuck protrusion length (#76) can be reflected on programs to be created on the basis of the back chuck end face. (See <8.18 Using "B.CHUCK EXT.LENG (protrusion length of back spindle chuck)" in Machining Data> for details.)
- Select "SUPPORT" for back spindle (#64) if the back cap nut is changed to that of the stabilizer-attached type in long workpiece machining.
  Selecting "SUPPORT" causes the back long-neck chuck protrusion length (#76) 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.

3.4 Drive Axis

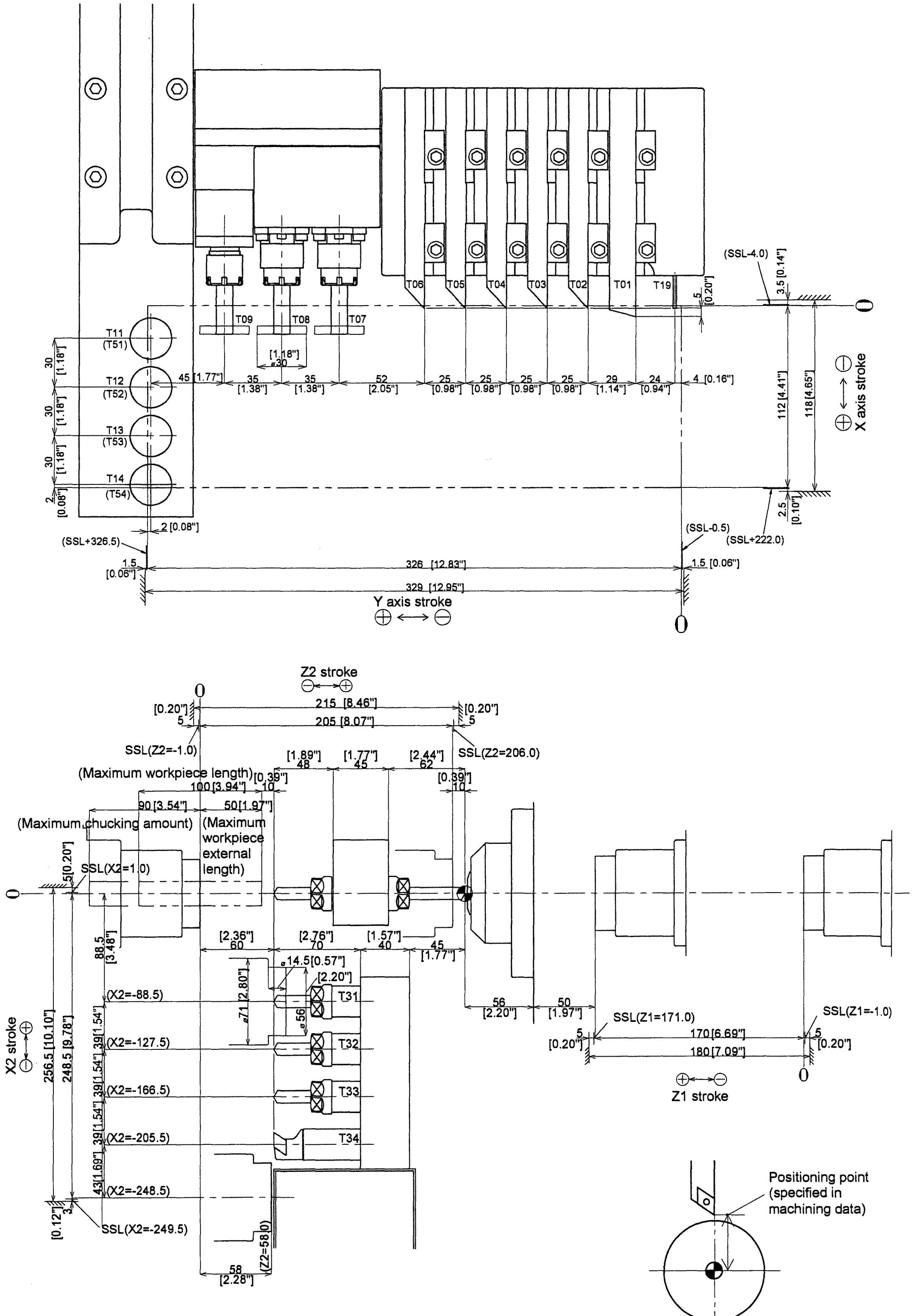
3.4.1 Drive axis



Five-axes control specification

1. Move the tool post vertically (X1 axis).
2. Move the tool post horizontally (Y1 axis).
3. Move the headstock longitudinally (Z1 axis).
4. Move the back headstock forward and backward (X2 axis).
5. Move the back headstock longitudinally (Z2 axis).
6. Index the main spindle at the specified angle (C1 axis).
7. Index the back spindle at the specified angle (C2 axis).

3.4.2 Stroke Diagram



* Back spindle stroke

When the workpiece separator is used: Advance 157 mm, retract 0 mm

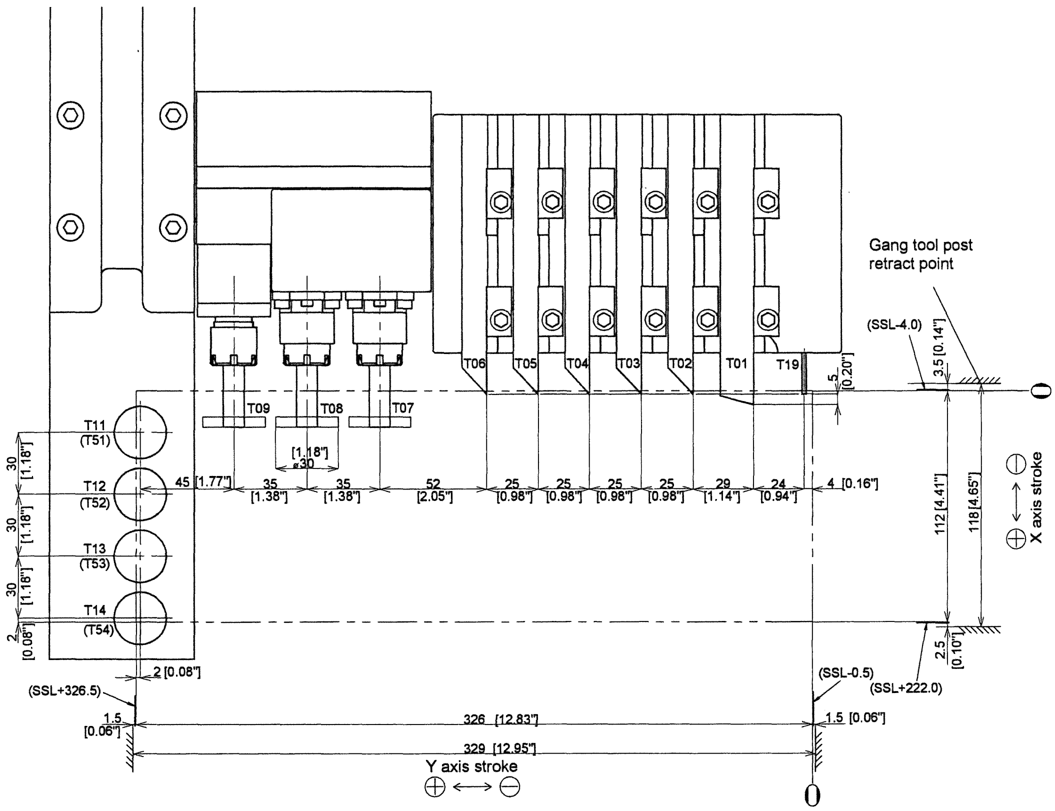
When the long workpiece device is used: Forward end position 205 mm – (tool square size) – (10 mm stabilizer), retract 0 mm

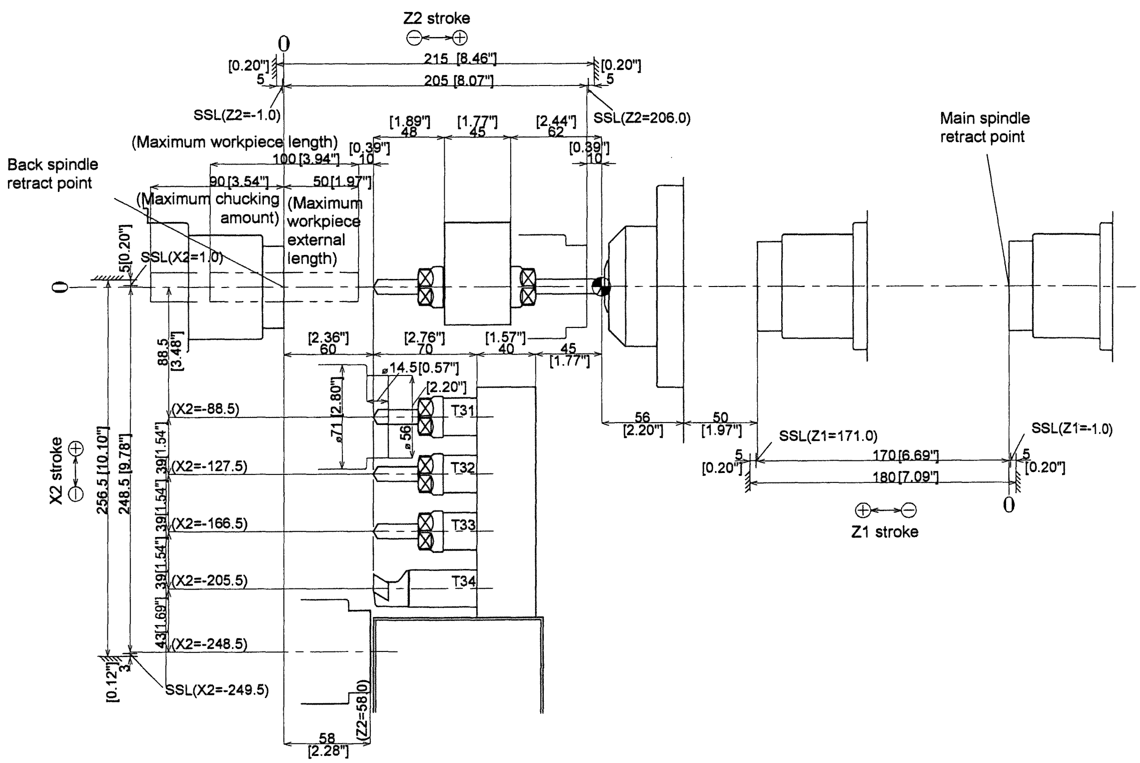
Notes

- When the workpiece separator for main spindle is used, specify "BASKET" for "BACK SPINDLE" on Machining data screen.
- When the long workpiece machining device is used, enable "LONG WORK" in Machine structure screen, and specify "SUPPORT" for "BACK SPINDLE (#64)" on Machining data screen.

Fixed points

- Gang tool post retract point (Machine coordinate value X1 0)
- Main spindle retract point (Machine coordinate value Z1 5.0)
- Back spindle retract point (Machine coordinate value Z2 0)

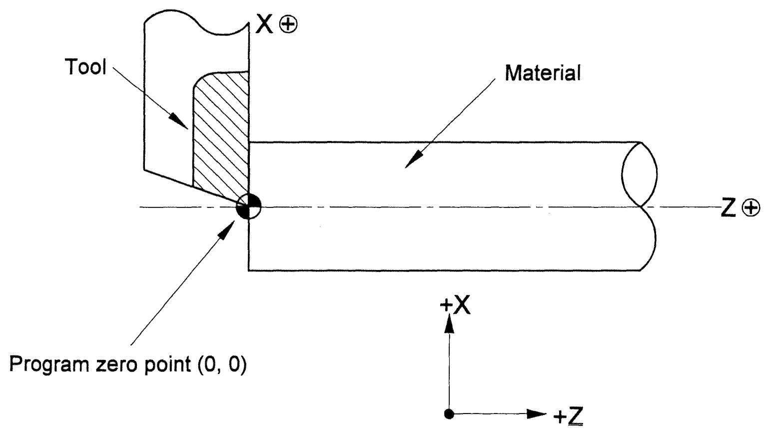




3.5 Coordinate System

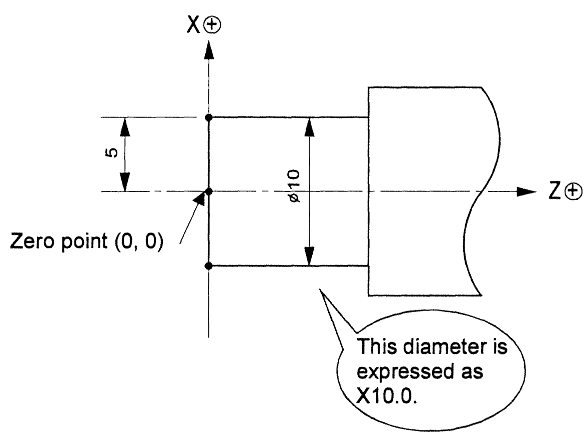
3.5.1 Coordinate axes

X axis: Infeed direction of the tool = Diameter direction
Z axis: Left/right move of spindle = Longitudinal direction

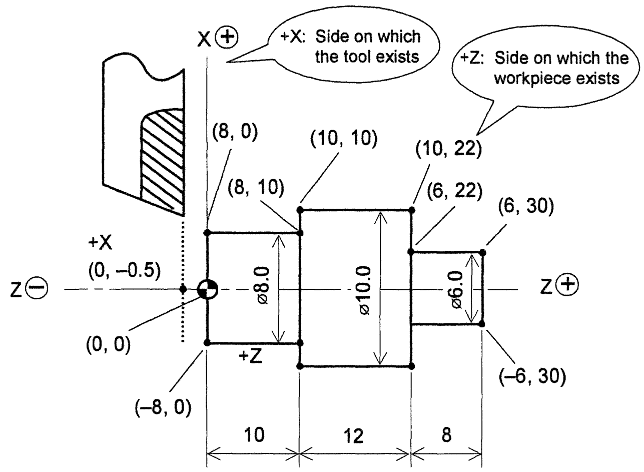


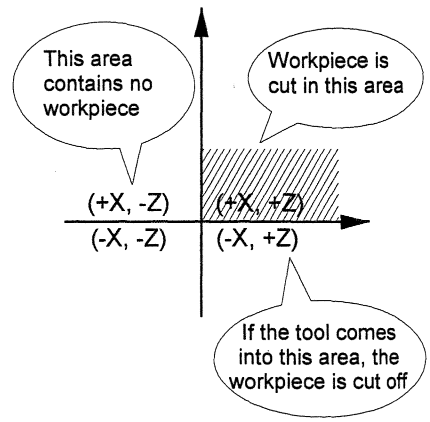
3.5.2 Diameter designation

The X coordinate value must be expressed by the diameter designation.



3.5.3 Signs and coordinate values

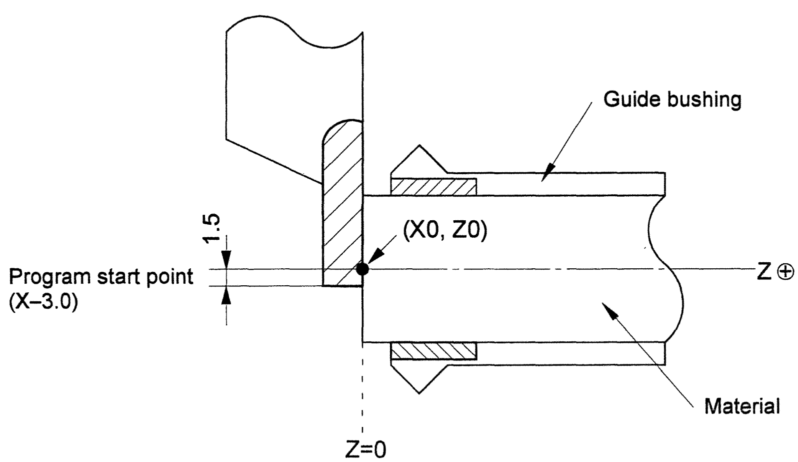




3.5.4 Coordinate system setup

The reference of the longitudinal direction can be determined.

In general, the end face of the material which is in contact with the cut-off tool is treated as the zero point (Z = 0).



To declare the above setting, use the following code:

```
G50 Z0
```

This code means "Determine the current position as Z = 0."

For the X1 axis, the reference point is the center of the material (X=0.0), accordingly, setting is not required.

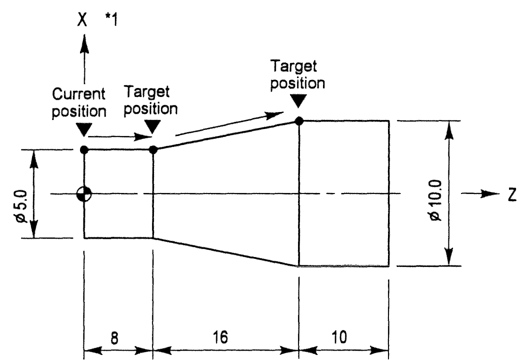
3.5.5 Absolute and incremental commands

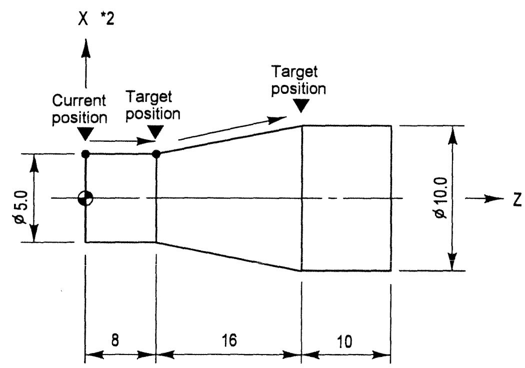
Positions may be represented by absolute coordinates which directly indicate the coordinates (X, Y, and Z) of the target position or by incremental coordinates which indicate the distances in the X, and Z directions.

Absolute command: Specifies the absolute coordinates of the target position (in the form of X□, Y□, and Z□)

Incremental command: Specifies the relative distances in the X and Z directions (in the form of U□, V□, and W□).

- X, Y and U, V values represent diameters.





It is assumed that the machine starts at Z0.

	Absolute	Incremental
	X10.0 Z0	U0 W0
	↓	↓
1	X5.0 Z8.0	U0 W8.0
	↓	↓
2	X10.0 Z24.0	U5.0 W16.0

Answers to **Q** in page 3-3.

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

3.6 Machining Patterns

To simplify a program used with a multi-axis machine, operations (machining) are grouped by purpose. An operational (machining) group is called a machining pattern. The following machining patterns are available:

- Free pattern (machining pattern cancel) (G600)

  Use the free pattern (machining pattern cancel) (G600) as the command to cancel the machining patterns of front/back parallel machining (G630), front/back simultaneous machining (G660) or pick-off/center support (G650) (in the power-on state). Use this pattern for machining with no machining pattern.

- Front/back parallel machining (G630)

  Use the front/back parallel machining (G630) as the command to simultaneously perform front outer diameter machining and front drilling on the gang tool post in axis control group 1 ($1) and back drilling on the back tool post in axis control group 2 ($2).

- Pick-off, Center-support (G650)

  This machining pattern enables the back spindle to pick off the workpiece and support the center of the workpiece. (Synchronization and superimpose functions are disabled in this machining pattern.)

- Front/back simultaneous machining (G660)

  Use this machining pattern to perform front drilling with a tool on the tool post in $1, and simultaneously perform the back drilling with a tool on the gang tool post in $2.

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

When G650 is specified, Z1 and Z2 axes are subject to superimpose control. In a program, you can specify Z1 axis as the reference axis and Z2 as the superimposed axis for synchronous operation, or move Z2 axis by superimposing on Z1 axis (reference axis). See <3.6.7 Superimpose control> for details.

Notes

- Do not execute the machining pattern commands in the MDI mode.
- Transition between all machining patterns is available.
- Superimposition can be specified only with M codes (M130/M131) independent of machining patterns. See <8.9 Z1-Z2 Axis Sumperimposition (M130, M131)> for details.

3.6.1 Machining Pattern List

Machining pattern	Command code	$1	$2
Free pattern (machining pattern cancel)	G600	• Operation X1, Y1 and Z1: No operation • Selectable tool numbers: T0100 to T0900 T1100 to T1400 T2000 * T3000 * T3100 to T3400 * T5100 to T5400 • Axes of axis control groups: X1, Y1, Z1, (C1) (Note that calling T3000 or any of T3100 to T3400 marked by * causes the axis control groups to be changed to X2 and Z2 and calling any of T5100 to T5400 causes the axis control groups to be changed to X1, Y1 and Z2.)	• Operation X2: No operation Z2: Moves to the back spindle retract point (with non-operating argument W0). • Selectable tool numbers: * T0100 to T0900 K2 T3000 T3100 to T3400 T5100 to T5400 • Axes of axis control groups: X2, Z2, (C2) (Note that calling any of T0100 to T0900 marked by * with K2 argument causes the axis control groups to be changed to X1, Y1 and Z2.) • Superimpose Cancels Z1-Z2 axis superimposition.
Front/back parallel machining	G630	• Operation X1, Y1, and Z1: No operation • Selectable tool numbers: T0100 to T0900 T1100 to T1400 • Axes of axis control groups: X1, Y1, Z1, (C1)	• Operation X2: No operation Z2: Moves to the back spindle retract position (with non-operating argument W0). • Selectable tool numbers: T3100 to T3400 • Axes of axis control groups: X2, Z2, (C2) • Superimpose Cancels Z1-Z2 axis superimposition.
Pick-off, center support	G650	• Operation X1, Y1, and Z1: No operation • Selectable tool numbers: T0100 to T0900 • Axes of axis control groups: X1, Y1, Z1, (C1)	• Operation X2: Moves to the center of the guide bushing (with non-operating argument U0) Z2: Moves to the back spindle retract position (with non-operating argument W0). • Selectable tool numbers: T3000 • Axes of axis control groups: X2, Z2, (C2) • Superimpose Superimpose Z2 on Z1
Front/back simultaneous machining	G660	• Operation X1, Y1, and Z1: No operation • Selectable tool numbers: T1100 to T1400 • Axes of axis control groups: X1, Y1, Z1, (C1)	• Operation X2 and Z2: No operation • Selectable tool numbers: T5100 to T5400 • Axes of axis control groups: X2, Z2, (C2) • Superimpose Cancels Z1-Z2 axis superimposition.

3.6.2 Machining pattern flow

Specify the machining pattern for both axis control groups $1 and $2.

Sample program

$1		$2	
G660	Front/back simultaneous machining	G660	Front/back simultaneous machining
:		:	
G630	Front/back parallel machining	G630	Front/back parallel machining
:		:	
G600	Free pattern (machining pattern cancel)	G600	Free pattern (machining pattern cancel)
:		:	
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	
%		%	

Notes

- The machining pattern is automatically entered into the queuing state.
- Be sure to cancel the coordinate system shift command and compensation command before switching the machining pattern. However, you do not have to cancel the compensation command when switching to G650.
- Describe the program for both axis control groups $1 and $2.
- Each machining pattern is a modal G code that is enabled until another machining pattern is specified.
- Each axis control group should hold the feed-per-rotation command and feed-per-minute command as modal functions.

Argument U0 and W0 for the machining pattern

U0 specifies that the X axis does not move.
W0 specifies that the Z axis does not move.
In $2, the axis that moves is the back headstock (X2 and Z2 axes).

Axis control group	$1	$2	
Argument	None	U0	W0
Mobile section	None	X2 axis	Z2 axis

3.6.3 Free pattern (machining pattern cancel) G600

The operation can be used for front or back machining done independently. In addition, the operation can cancel the machining patterns of front/back parallel machining (G630), pick-off/center support (G650) and front/back simultaneous machining (G660) in the power-on state.

Command format

$1	$2
G600	G600 W0

Axis control group

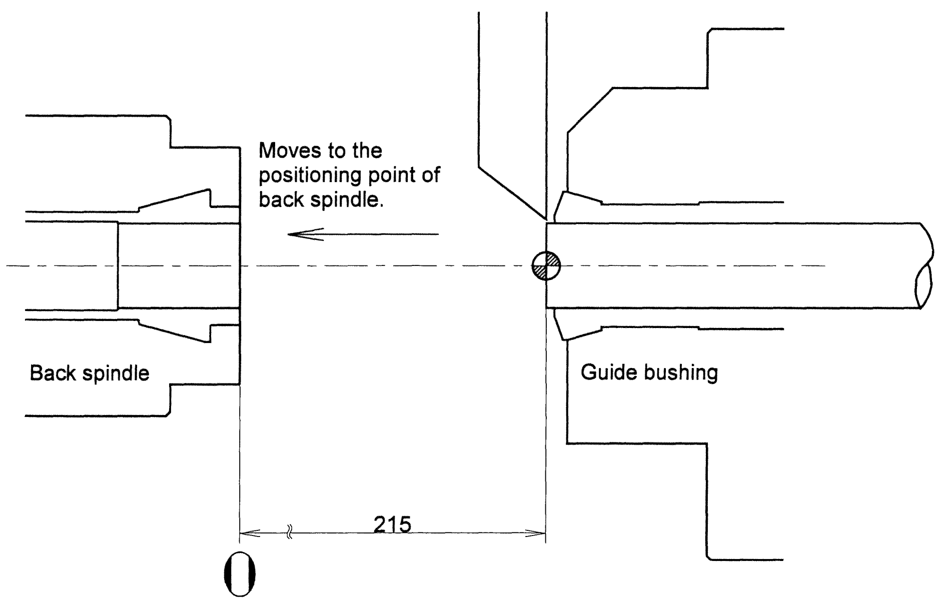
Specify this command for both axis control groups $1 and $2.

Argument

`$2 W0:` The back headstock (Z2 axis) does not move. If the argument is omitted, the back headstock moves to the positioning point.

Operation sample

The back headstock (Z2 axis) moves to the back spindle positioning point at the rapid feed rate.



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)
Superimpose	Cancel Z1-Z2 superimposition	
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 T0900 T1100 to T1400 T2000 T3000 T3100 to T3400 T5100 to T5400 T0100 to T0900 K2	T0100 to T0900 K2 T3000 T3100 to T3400 T5100 to T5400
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
	:
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	
:	
:	

Notes

- 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 T51'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.

3.6.4 Front/back parallel machining G630

Use this machining pattern to perform outer diameter machining or front drilling with the gang tool post on axis control group 1 ($1) and back drilling with the back tool post on axis control group 2 ($2).

Command format

$1	$2
`G630`	`G630 W0`

Axis control group

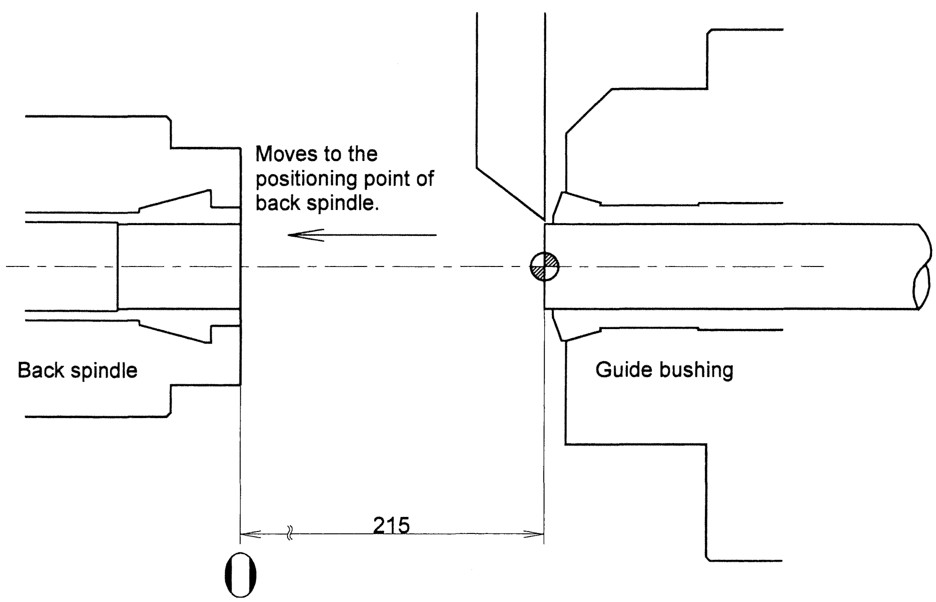
Specify this command for both axis control groups $1 and $2.

Argument

`$2 W0:` The back headstock (Z2 axis) does not move. If the argument is omitted, the back headstock moves to the positioning point.

Operation sample

The back headstock (Z2 axis) moves to the positioning point at the rapid feed rate.



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)
Superimpose	Cancel Z1-Z2 superimposition	
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 T0900 T1100 to T1400	T3100 to T3400
Others	–	–

Sample program

$1	$2
G600	G600
:	:
:	:
G630 .... Front/back parallel machining	G630 .... Front/back parallel machining
G99 M3 S1=3000	M23 S2=2500
T0300	G99
G0 X21.0 Z0	T3100
G1 X–0.5 F0.03	G0 Z–1.0
:	G1 Z2.5 F0.05
:	:
G600	G600
:	:
:	:

3.6.5 Pick-off, center support G650

This machining pattern enables the back spindle to pick off the workpiece and support the center of the workpiece.

Command format

$1	$2
G650	G650 U0 W0 Z☐

Axis control group

Specify this command for both axis control groups $1 and $2.

Arguments

$2 U0: The back headstock (X2 axis) does not move. If the argument is omitted, the X2 axis moves to the center of the main spindle.

$2 W0: The back headstock (Z2 axis) does not move. If the argument is omitted, the back headstock moves to the return point.

$2 Z : With the workpiece end face defined as zero position on front $1, the Z2 axis moves to the position specified by this Z argument.

Note

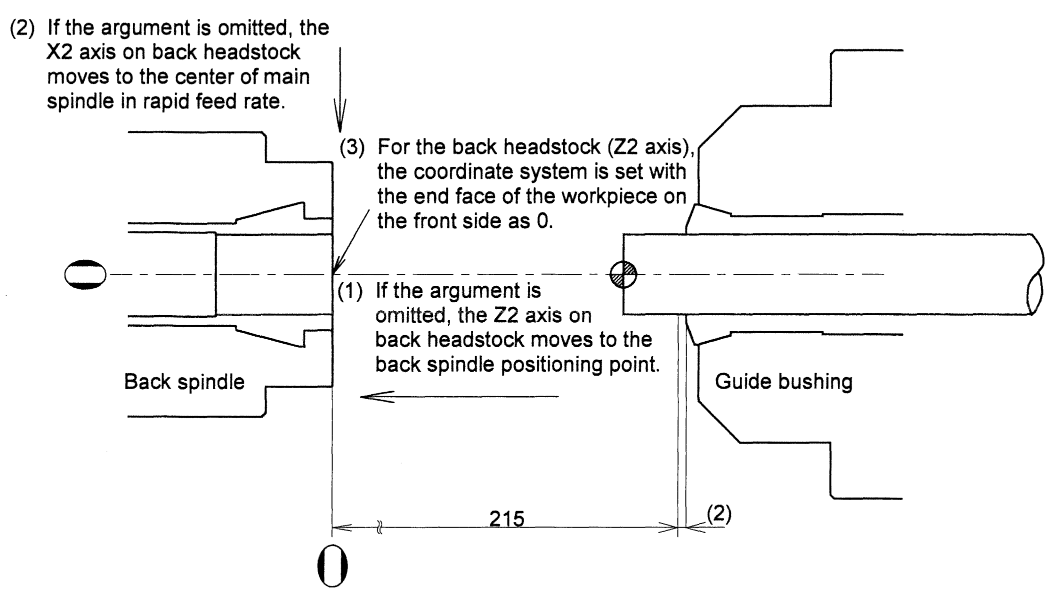
The back headstock (Z2 axis) superimposes on front headstock (Z1 axis).

Operation sample

(1) If the argument is omitted, the Z2 axis on back headstock moves to the back spindle positioning point.

(2) If the argument is omitted, the X2 axis on back headstock moves to the center of main spindle in rapid feed rate.

(3) For the back headstock (Z2 axis), the coordinate system is set with the end face of the workpiece on the front side as 0.



Macro specification

Command code	G650	
Name	The back headstock (Z2 axis) does not move.	
Axis control group	$1	$2
Axes of axis control group	X1, Z1, Y1, (C1)	X2, Z2, (C2)
Superimpose	Z2 axis superimposes on Z1 axis.	
Coordinate system		Z2: The tip of the back headstock cap nut aligns with the end face of the workpiece on the front side.
Argument		U0: The back headstock (X2 axis) does not move. W0: The back headstock (Z2 axis) does not move. Z□: The back headstock (Z2 axis) moves to the position specified by Z□.
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 T0900	T3000
Others	–	–

Sample program

$1	$2
G600	G600
:	:
:	:
G99 M03 S1=1500	:
T0100	:
G00 X17.0 Z85.0 T1	T3000
G650 .... Pick-off	G650 .... Pick-off
	G00 Z–2.0
	M24 S2=1500
	G98 G01 Z60.0 F1000
	M15
!2 L1	!1 L1
G99	
G01 X–1.0 F0.03	
G600	G600
X–3.0	:
:	:
:	

Notes

- If a numeric value is entered to machining data "back long-neck chuck protrusion length," command Z0.0 for $2 after G650 moves the Z2 axis to the position at which the front end face of a workpiece matches the back end face of the long-neck chuck.
- If "SUPPORT" is selected for machining data "back spindle," command Z0.0 for $2 after G650 moves the Z2 axis to the position at which the front end face of a workpiece matches the end face of the brass part of the cap nut for a long workpiece. This is due to the consideration that the stabilizer-attached cap nut for a long workpiece is longer than the normal cap nut by 10.0 mm.

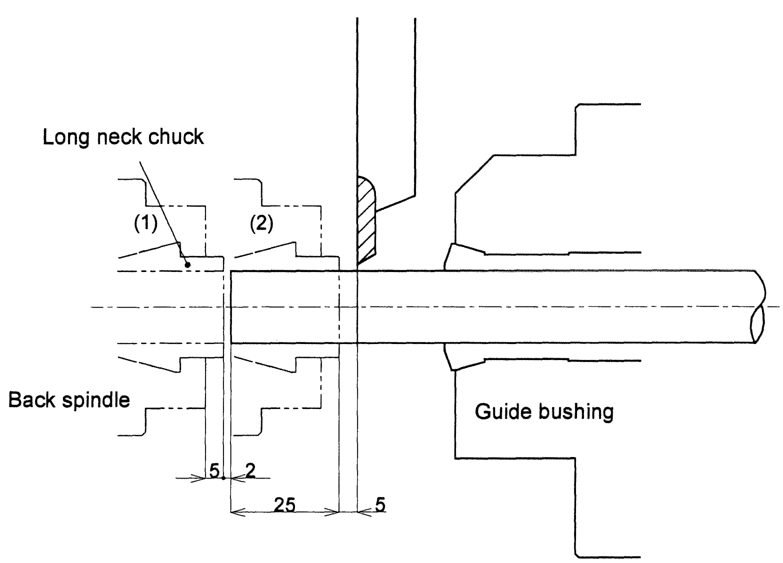
Sample program for pick-off in use of back long-neck chuck

- The long-neck chuck shall protrude from the end face of the cap nut by 5.0 mm.
- The workpiece shall be located at the position separated from the end face of the long-neck chuck by 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 long-neck chuck protrusion length" in the machining data.
  2. Enter 5.0 as "back workpiece protrusion length" in the machining data.

Sample program

$1	$2	
G600	G600	
:		
G99 M13 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	
:	:	



3.6.6 Front/back simultaneous machining G660

Use this machining pattern to simultaneously perform center hole drilling with front-back drilling holder on the gang tool post.

Command format

$1	$2
G660	G660

Axis control group

Specify this command for both axis control groups $1 and $2.

Operation sample

Command G660 does not bring any movement for each axis.

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)
Superimpose	Cancel Z1-Z2 superimposition	
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	T1100 to T1400	T5100 to T5400
Others	–	–

Sample program

$1	$2
G600	G600
:	:
:	:
G660 .... Front/back simultaneous machining	G660 .... Front/back simultaneous machining
T1100	T5100
G00 Z–0.5	G00 Z–0.5
G01 Z5.0 F0.03	G01 Z5.0 F0.03
:	:
G600	G600
:	:
:	:

Notes

- For the front/back simultaneous machining (G660) command, specify tools T11's and T51's for both axis control groups ($1 and $2). The queuing state is placed at the beginning and end of T11's and T51's. In addition, the tool specified in $1 and that in $2 must make a pair.
- The front/back simultaneous machining (G660) is simultaneous drillings at the front and back centers. Front/back superimposed drilling is disabled in which center drilling by T51's is performed together with an X or Y command by T11's (e.g. boring machining).
  However, X and Y commands are enabled on programs. Accordingly, specifying the above command by mistake may cause interference to occur. Take extra care of it.

3.6.7 Superimpose control

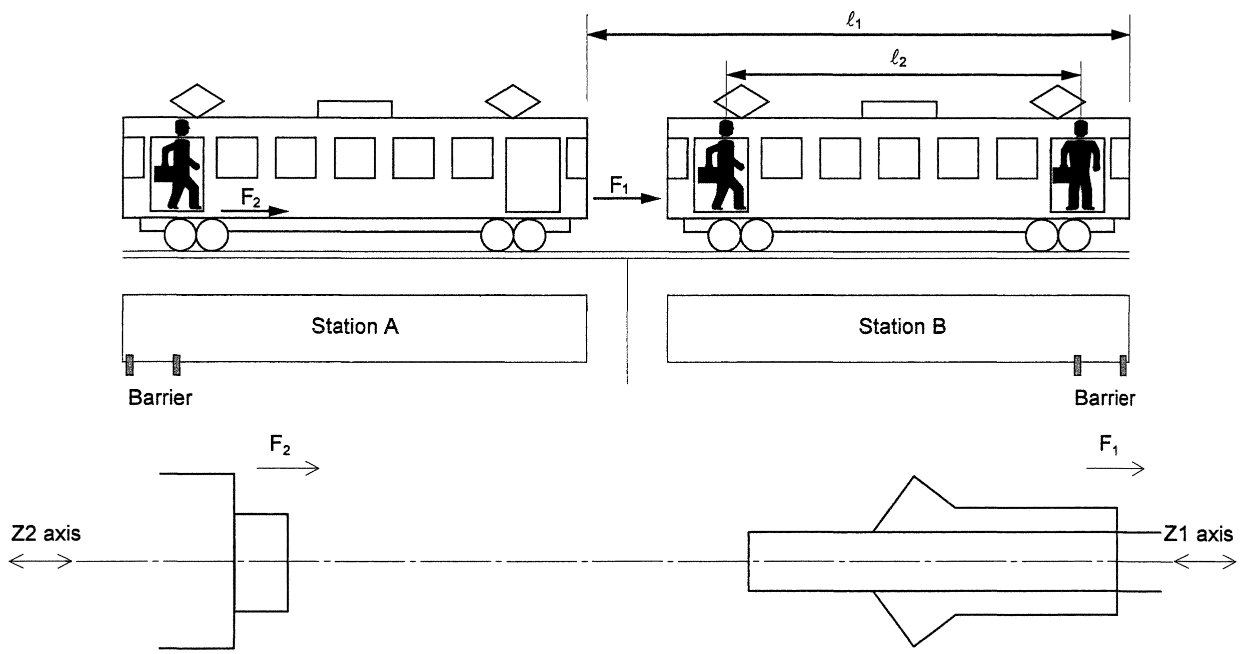
This section explains superimpose control giving an example.

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 ℓ_1) at F_1 speed, Mr. A disk his work for the distance of ℓ_2 at F_2 speed. As a result, the work $\ell_1 + \ell_2$ 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.

Product Code	C	–	A	2	2	0	P	L	VII					

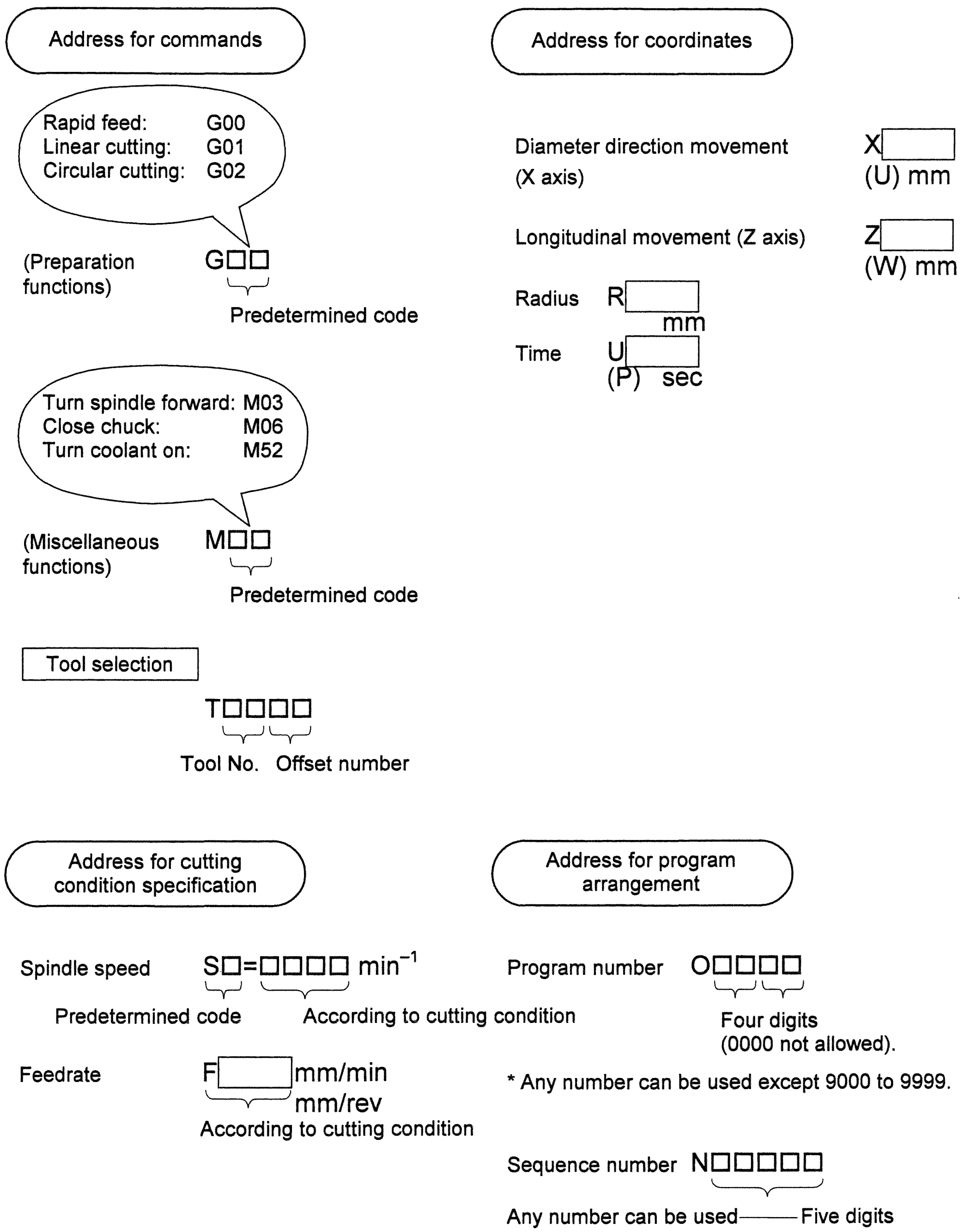
Document Code	2	E	1	–	0	3	0	1	

4. Command Codes

4.1 Addresses Used in NC ProgramsNC

NC programs is a combinations of an alphabetic character and a number.

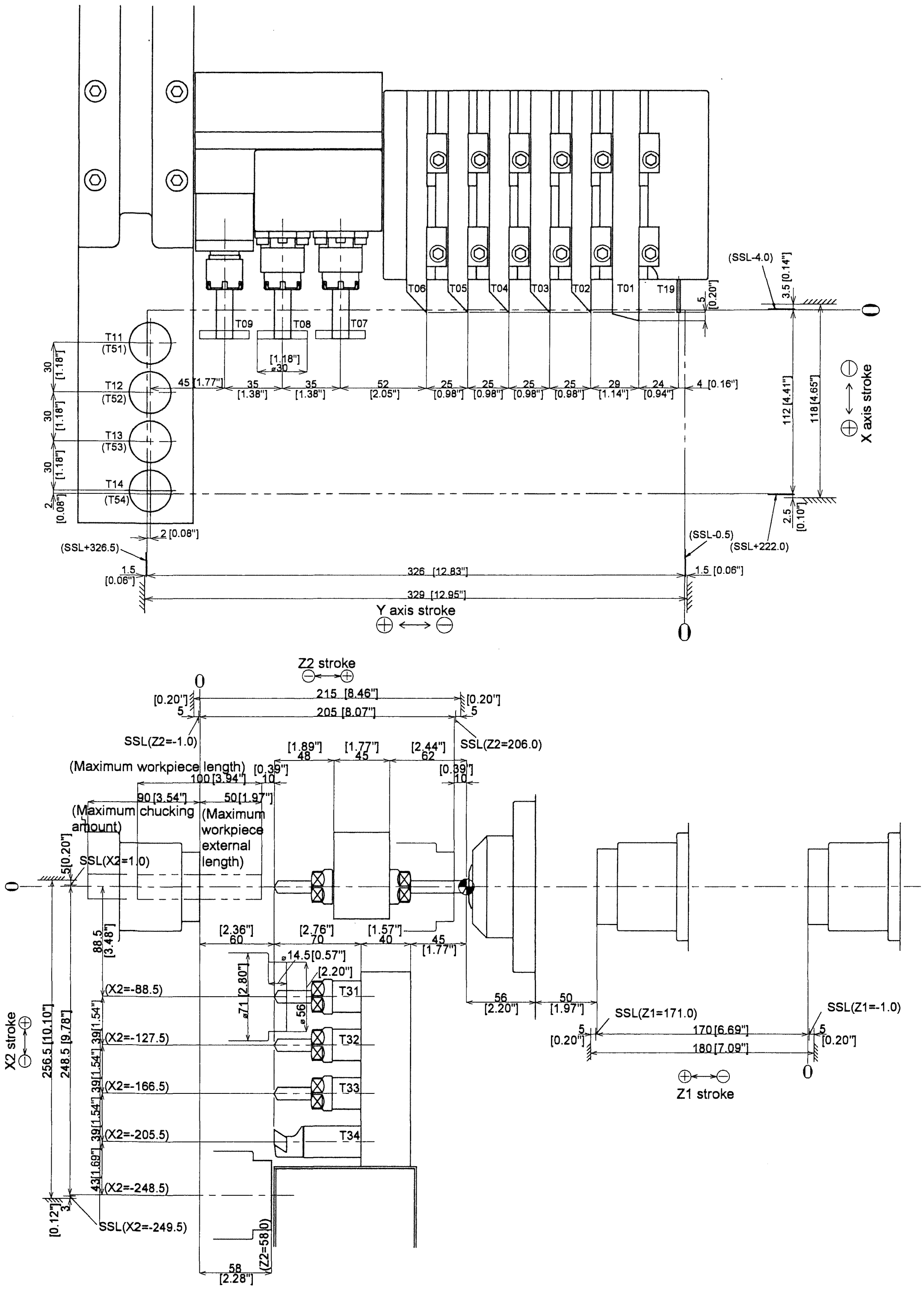
The alphabetic character is called the address.



4.2 T Functions (Tool Selection)

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 T09)
- Front drilling tool (T11 to T14)
- Back drilling tool (T31 to T34, T51 to T54)
- Back spindle (T30)



T-code four-digit command

T□□ΔΔ

□□: Tool number

- Gang tool (T01 to T09)
- Front drilling tool (T11 to T14)
- Back drilling tool (T31 to T34, T51 to T54)
- Back spindle (T30)

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. For each tool number, specify an axis control method (machine coordinates, XY coordinates) with parameters.

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. If the offset is canceled, use 00.

Tool mounting positions and machining points

Tool post	Tool No.	Turning	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				
	06	T0600	(T0600)			
	07		T0700			
	08		T0800			
	09		T0900			
Gang tool post Front drilling tool	11			T1100		
	12			T1200		
	13			T1300		
	14			T1400		
Back spindle	30				T3000	
Back headstock Back drilling tool	31					T3100
	32					T3200
	33					T3300
	34					T3400
Gang tool post Back drilling tool	51					T5100
	52					T5200
	53					T5300
	54					T5400

Notes

- 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.

4.3 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.

4.3.1 T code arguments of gang tool post and rotary tools (T0100 to T0900)

Command format

T□□ΔΔ H□ Q1 X□ Y□ Z□ K2 E□

Arguments

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 workpiece 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 Z2 axis is used. In any other cases, 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	X2, Z2
K2 argument specified	X1, Z2, Y1	X2, Z1

When T code is specified in $2:

	$1	$2
K2 argument not specified	Alarm	
K2 argument specified	X2, Z1	X1, Z2, Y1

The workpiece coordinate of the Z2 axis is set so that the tool nose directed to the back spindle becomes 0.

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 valid in the transition from turning to secondary machining process.
If the K2 argument is specified together, the back spindle is indexed (the specification is limited only when the back spindle indexing option is set).

When the spindle C axis option is not specified:
Integer between 0.0 and 359.0 (with decimal point)

When the spindle C axis option is specified:
Numeric value between 0.000 and 359.999 (with decimal point)

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)

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.

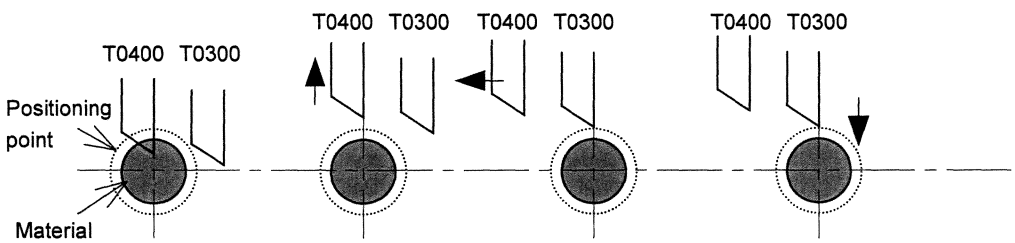
Operation sample

- T□□ΔΔ Argument not specified

Make positioning of the specified tool to the positioning point.

- T0100 to T0900 → T0100 to T0900:

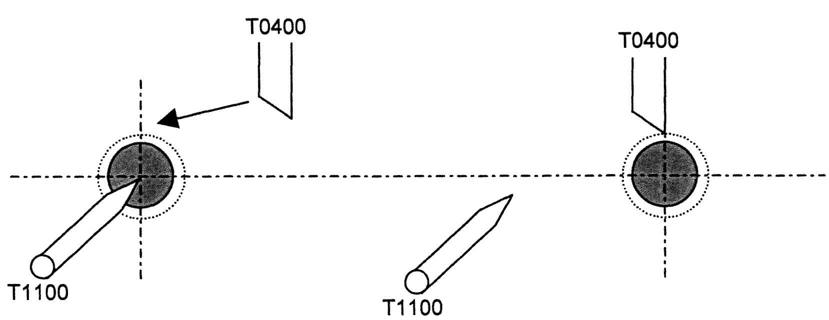
1. The X1 axis of the currently selected tool moves to the position (material O.D. + tool positioning point + maximum DIA value of tool set data between current and specified tools).
2. While keeping the X1 axis at the position in step 1, the Y1 axis moves to the workpiece coordinate 0 of the specified tool.
3. The X1 axis of the specified tool moves to the position (material O.D. + tool positioning point + DIA value of tool set data of the specified tool) (positioning point).



1. The X1 axis of the currently selected tool moves to the position (material O.D. + positioning point + maximum DIA value of tool set data between current and specified tool).
2. While keeping the X1 axis at the position in step 1, the machine moves the Y1 axis of the specified tool moves to the work coordinate 0.
3. The X1 axis of the specified tool moves to the position (material O.D. + positioning point + DIA value of tool set data of the specified tool) (positioning point).

- T1100 to T1400, T5100 to T5400 → T0100 to T0900

1. The specified tool directly moves to the position (material O.D. + tool positioning point + DIA value of tool set data of the specified tool) (positioning point).

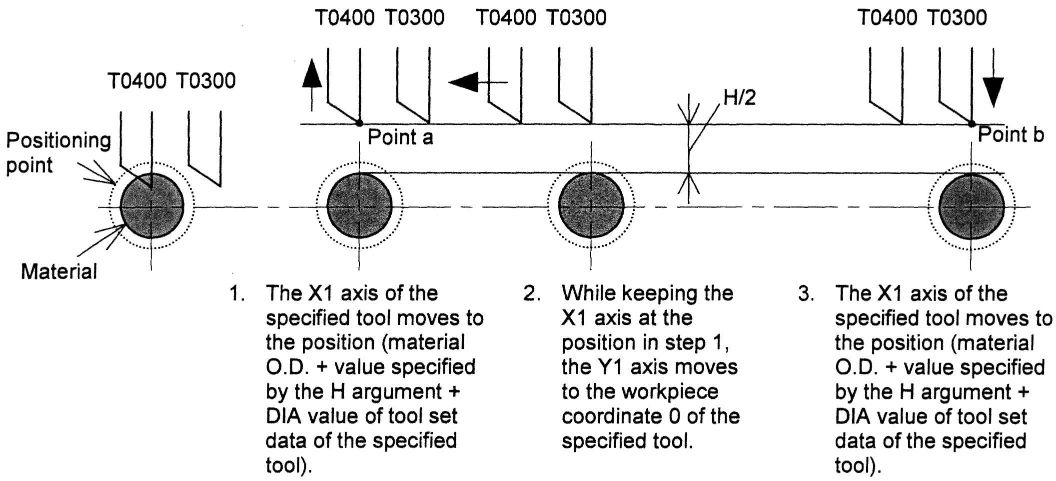


1. The selected tool moves to the position (material O.D. + positioning point + DIA value of tool set data of the specified tool) (positioning point).

- T□□ΔΔ H□ H argument

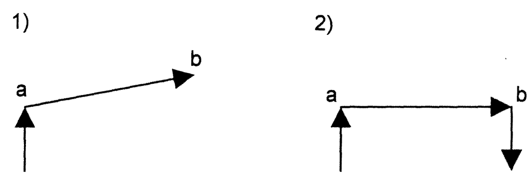
  Specify this argument to temporarily change the positioning point (relief point from material O.D. in the machining data) of the current tool.

  1. The X1 axis of the currently selected tool moves to the position (material O.D. + value specified by the H argument + DIA value of tool set data of the currently selected tool).
  2. While keeping the X1 axis at the position in step 1, the Y1 axis moves to the workpiece coordinate 0 of the specified tool.
  3. The X1 axis of the specified tool moves to the position (material O.D. + value specified by the H argument + DIA value of tool set data of the specified tool).



Notes

- 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)

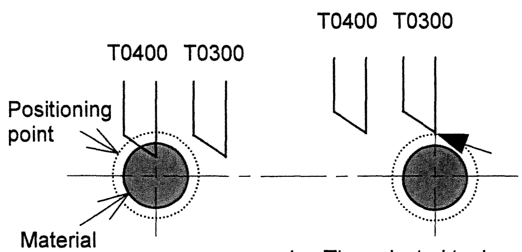


- If the H argument is specified with a negative value, an alarm occurs.
- The H argument can be specified only when the currently selected tool is a gang tool (T0100 to T0900). If the H argument is specified during selection of a tool other than gang tools, the machine operates as if the argument were not specified.
- If H and Q1 arguments are specified at the same time, the H argument is preceded.

- T□□ΔΔ Q1 Q1 argument

Select a tool without moving the current tool to the positioning point.

1. The selected tool moves to the position (material O.D. + positioning point + DIA value of tool set data of the specified tool) (positioning point).



1. The selected tool moves to the position (material O.D. + positioning point + DIA value of tool set data of the specified tool) (positioning point).

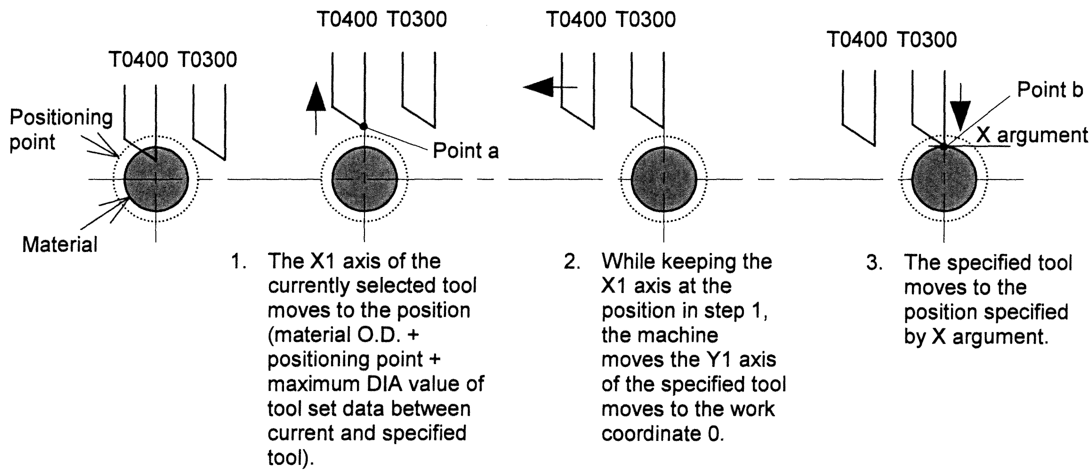
Note

If Q1 argument is specified together with one of the X, Z, and Y arguments, the axis moves from the current position of the selected tool to the position specified by X, Z, or Y argument directly.

- T□□ΔΔ X☐ X argument

Specify the position of X1 axis of the specified tool. (Work coordinate of X1 axis after tool selection)

1. The X1 axis of the currently selected tool moves to the position (material O.D. + positioning point + maximum DIA value of tool set data between current and specified tool).
2. While keeping the X1 axis at the position in step 1, the Y1 axis moves to the workpiece coordinate 0 of the specified tool.
3. The X1 axis of the specified tool moves to the position specified by X argument.



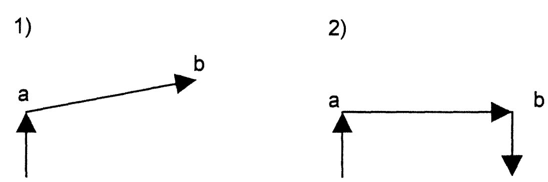
Note

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: Feed position specified by X argument of the specified tool

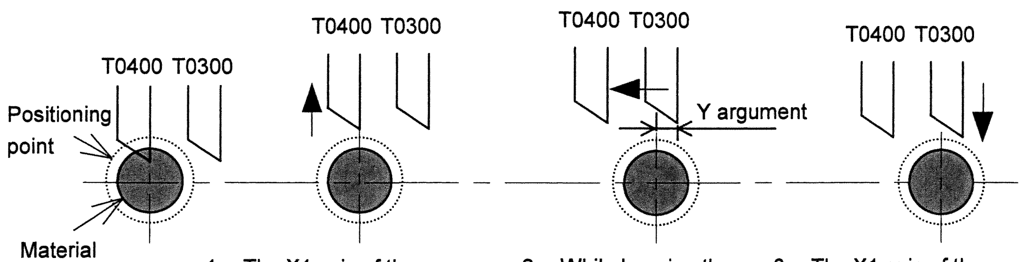
If the point b is larger than point a, the machine moves as shown in (1).
If the point b is smaller than point a, the machine moves as shown in (2)



- T□□ΔΔ Y☐ Y argument

Specify the position of Y1 axis of the specified tool. (Work coordinate of Y1 axis after tool selection)

1. The X1 axis of the currently selected tool moves to the position (material O.D. + positioning point + maximum DIA value of tool set data between current and specified tool).
2. While keeping the X1 axis at the position in step 1, the Y1 axis of the specified tool moves to the position of the Y argument.
3. The X1 axis of the specified tool moves to the position (material O.D. + tool positioning point + DIA value of tool set data of the specified tool) (positioning point).

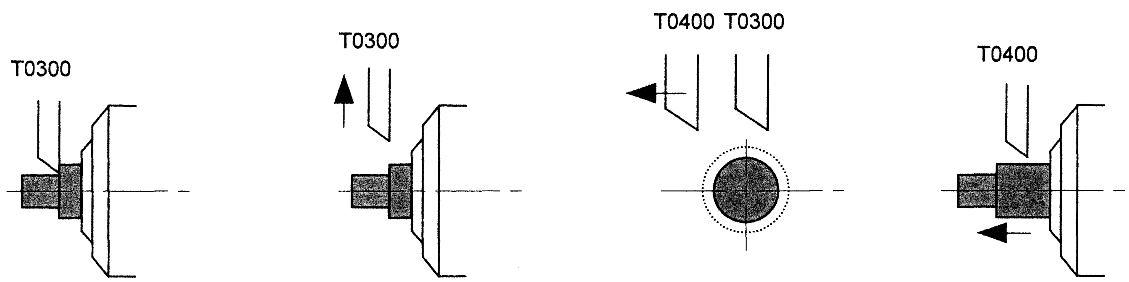


1. The X1 axis of the currently selected tool moves to the position (material O.D. + positioning point + maximum DIA value of tool set data between current and specified tool).
2. While keeping the X1 axis at the position in step 1, the machine moves the Y1 axis of the specified tool moves to the position specified by Y argument.
3. The X1 axis of the specified tool moves to the position (material O.D. + positioning point + DIA value of tool set data of the specified tool) (positioning point).

- T□□ΔΔ Z☐ Z argument

Specify the position of Z axis of the specified tool. (Work coordinate of Z1 axis after tool selection)

1. The X1 axis of the currently selected tool moves to the position (material O.D. + positioning point + maximum DIA value of tool set data between current and specified tool).
2. While keeping the X1 axis at the position in step 1, the Y1 axis moves to the position of workpiece coordinate 0 of the specified tool.
3. The X1 axis of the specified tool moves to the position (material O.D. + tool positioning point + DIA value of tool set data of the specified tool) (positioning point) and the Z axis moves to the point specified by Z argument (simultaneous operations of both X1 and Z axes).



1. The X1 axis of the currently selected tool moves to the position (material O.D. + positioning point + maximum DIA value of tool set data between current and specified tool).
2. While keeping the X1 axis at the position in step 1, the machine moves the Y1 axis of the specified tool moves to the work coordinate 0.
3. The X1 axis of the specified tool moves to the position (material O.D. + positioning point + DIA value of tool set data of the specified tool) (positioning point). The Z1 axis moves to the point specified by Z argument.

Note

If K2 argument is specified together with the specification of Z argument, the Z2 axis is intended for operation. Note the setting of the coordinate system of K2 argument. In any other cases, the Z1 axis is intended for operation.

- T□□ΔΔ K2 K2 argument

K2 argument is for back machining with a gang tool. The axes of each axis control group are defined as shown below after tool selection. With the definition, the interpolation machining with the gang tool becomes available at back spindle side.

When T code is specified in $1:

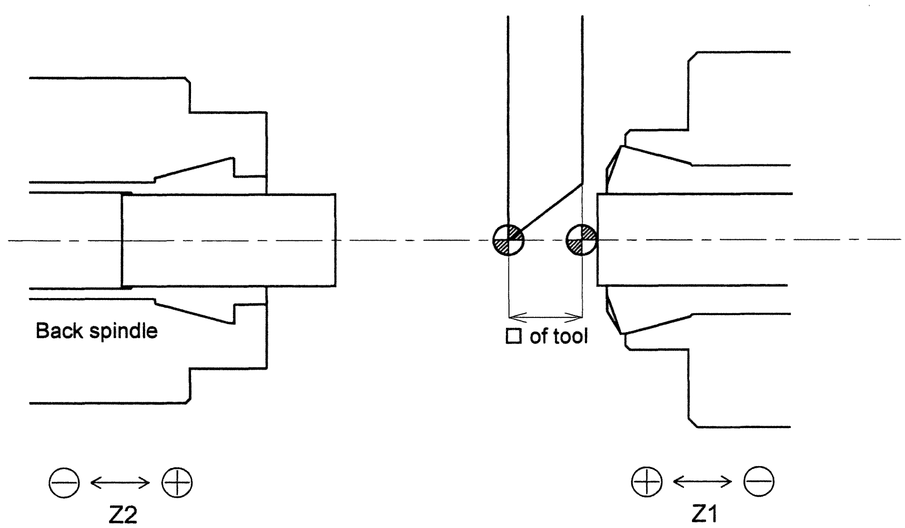
	$1	$2
K2 argument not specified	X1, Z1, Y1 (C1)	X2, Z2 (C2)
K2 argument specified	X1, Z2, Y1 (C2)	X2, Z1 (C1)

When T code is specified in $2:

	$1	$2
K2 argument not specified	Alarm	
K2 argument specified	X2, Z1 (C1)	X1, Z2, Y1 (C2)

When K2 argument is specified, set the workpiece coordinate system of the back headstock (Z2 axis) so that the tool nose of gang tool becomes zero.

Define this coordinate system setting in consideration of values □ of tools depending on the types of front machining holders. When a free tool is selected, set the coordinate system in consideration of 16 mm in the similar way as holders of □16 mm.



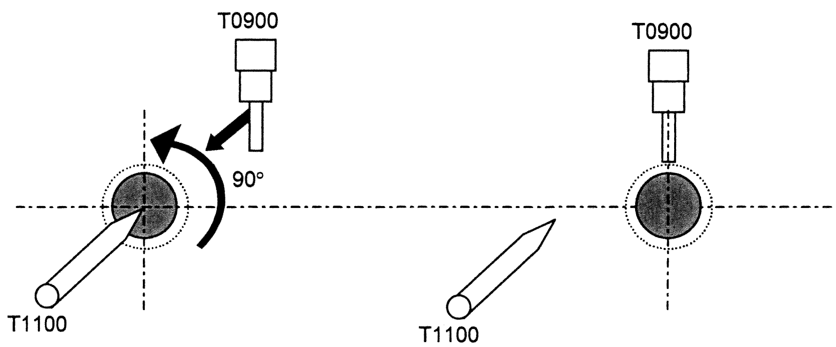
Notes

- Specifying K2 argument for any other tool than T0100 to T1000 causes an alarm to occur.
- When "BASKET" is specified for "BACK SPINDLE" in Machining data screen, an alarm occurs if K2 argument is specified.
- If K2 argument is not used under selection of tool T0100 to T1000 in axis control group 2 ($2) during execution of free pattern (G600) command, then an alarm will occur.

- T□□ΔΔ E▭ E argument

  Specify this argument to index the spindle during the tool selection.

  Specify the indexing angle with decimal point for E▭.



Index the main spindle at the specified angle during tool selection.

Notes

- Specify the indexing angle (E argument) with decimal point. If omitted, an alarm occurs or an indexing at specified angle will fail.
- When the K2 argument is specified together, the back spindle is indexed. Note, however, if the back spindle indexing option (back spindle C axis or back spindle 1-degree indexing) is not specified), an alarm occurs.
- If the value specified for indexing angle (E argument) is out of the allowable range, an alarm occurs.
- If the value specified for indexing angle (E argument) is out of the valid indexing angle, an alarm occurs.
- The main spindle (back spindle) indexing can be performed while the main spindle (back spindle) is rotating. However, for the main spindle C axis or the back spindle C axis, positioning and indexing cannot be performed concurrently because the axis returns to the zero point. (This can be performed only at the first time when the main spindle rotation mode (back spindle rotation mode) is switched to the main spindle C axis mode (back spindle C axis mode))..
- When an indexing angle (E argument) is specified, the T code cannot be run backward.
- For other notes, follow the indexing specification of each spindle.

4.3.2 T Code arguments of front drills on gang tool post (T1100 to T1400)

Command format

T□□ΔΔ X□ Y□ Z□ A□ E□ W□

Arguments

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 workpiece coordinate of Z1 axis after tool selection. Without the argument specified, the Z1 axis does not move.

A: Select either of the following for the G660 (front/back simultaneous machining) command.

- A without argument: DIA and CORE data stored in front machining tool (T11's) and back machining tool (T51's) are added to the X1 and Y1 axes.
- A1:DIA and CORE data stored in front machining tool (VII-type T11's) is added to the X1 and Y1 axes.
- A5: DIA and CORE data stored in back machining tool (VII-type T51's) is added to the X1 and Y1 axes.

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 valid in the transition from turning to secondary machining process.

When the spindle C axis option is not specified:
Integer between 0.0 and 359.0 (with decimal point)

When the spindle C axis option is specified:
Numeric value between 0.000 and 359.999 (with decimal point)

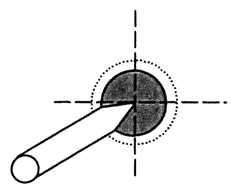
Note

Before a command among T1100 to T1400 is specified, the back headstock (Z2 axis) should be retracted to the position at which sleeve holders do not interfere.

Operation sample

- T□□ΔΔ Argument not specified

  The axis directly moves to the center of the specified tool.

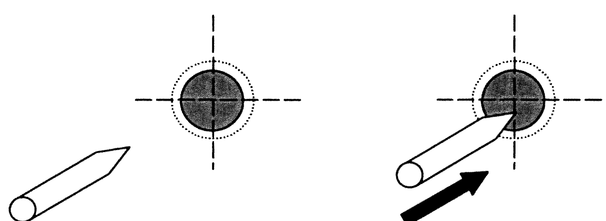


Directly moves to the center of the specified tool.

- 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.

  - The X1 axis directly moves to the position (the center of the specified tool + X argument).

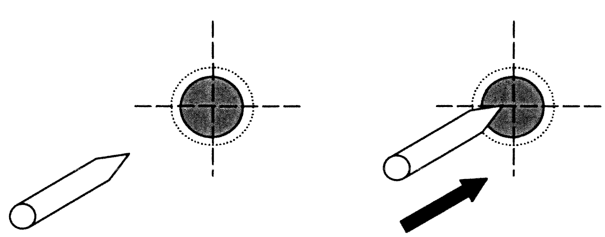


Directly moves to the position (the center of the specified tool + X argument)

- 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.

  - The Y1 axis moves to the position (the center of the specified tool + Y argument) directly.

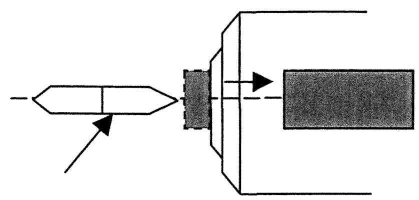


Directly moves to the position (the center of the specified tool + Y argument)

- T□□ΔΔ Z[ ] Z argument

Specify the workpiece coordinate of Z1 axis after tool selection. Without the argument specified, the Z1 axis does not move.

- The Z1 axis directly moves to the center of the specified tool. The headstock (Z1 axis) also moves to the position of Z argument.



2. The Z1 axis directly moves to the center of the specified tool. The headstock (Z1 axis) also moves to the position of Z argument.

- T□□ΔΔ A[ ] A argument

Select either of the following for the G660 (front/back simultaneous machining) command.

- A without argument: DIA and CORE data stored in front machining tool (T11's) and back machining tool (T51's) are added to the X1 and Y1 axes.
- A1: DIA and CORE data stored in front machining tool (T11's) is added to the X1 and Y1 axes.
- A5: DIA and CORE data stored in back machining tool (T51's) is added to the X1 and Y1 axes.

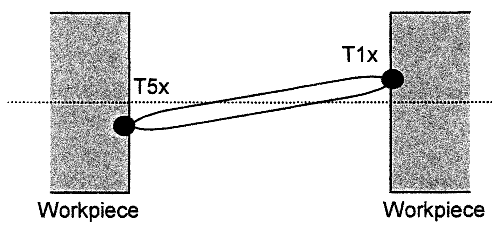


Figure 1: Middle position

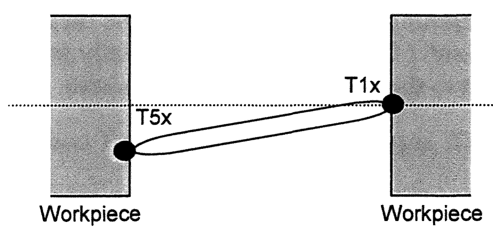


Figure 2: Front side (T11's)

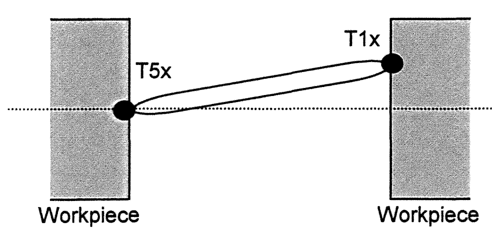


Figure 3: Rear side (T51's)

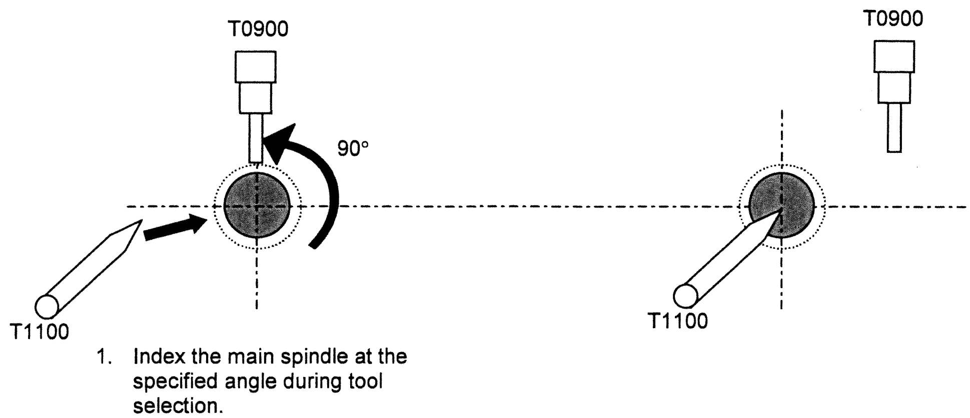
Note

Be sure to specify the A argument in axis control group 1 ($1) while executing the front/back simultaneous machining (G660) command. If not, an alarm occurs.

- T□□ΔΔ E☐ E argument

  Specify this argument to index the main spindle during the tool selection.

  Specify the indexing angle with decimal point for E☐.



Notes

- 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.
- If the value specified for indexing angle (E argument) is out of the valid indexing angle, an alarm occurs.
- The main spindle indexing can be performed while the main spindle is rotating. However, for the main spindle C axis, positioning and indexing cannot be performed concurrently because the axis returns to the zero point. (This can be performed only at the first time when the main spindle rotation mode is switched to the main spindle C axis mode).
- When an indexing angle (E argument) is specified, the T code cannot be run backward.
- For other notes, follow the indexing specification of the spindle.

4.3.3 Back drills on gang tool post (T5100 to T5400)

Command format

`T□□ΔΔ X[ ] Y[ ] Z[ ] W[ ] Q1 Q3 E[ ]`

Arguments

`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 positioning point. 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 valid 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)

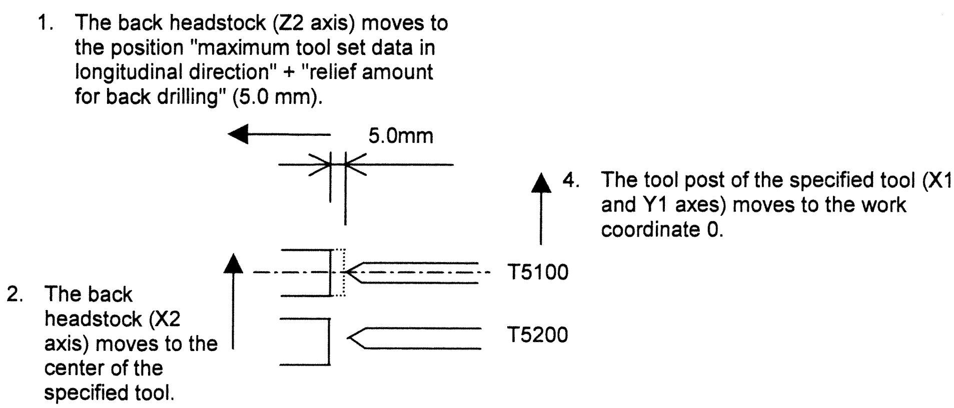
Notes

- Arguments Z, W and Q cannot be specified together. If specified, an alarm occurs.
- While the Z2 axis belongs to axis control group 1 ($1), specifying T51's command or K2 argument for T code in axis control group 2 ($2) makes the work coordinate of Z1 axis identical to the machine coordinate. Be sure to provide the specification in axis control group 1 ($1). To provide the specification in axis control group 2 ($2), first specify the free pattern (G600).

Operation sample

- 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 work coordinate 0.



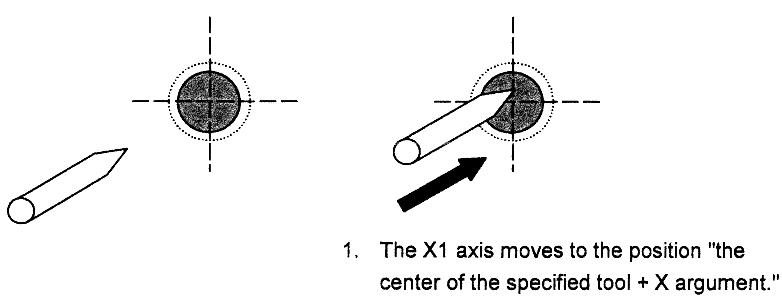
Note

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" (5.0 mm).), back spindle (Z2 axis) does not move.

- 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.

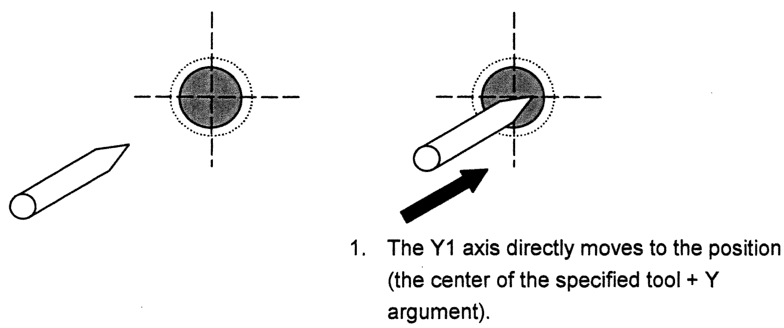
1. The X1 axis moves to the position "the center of the specified tool + X argument."



- 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.

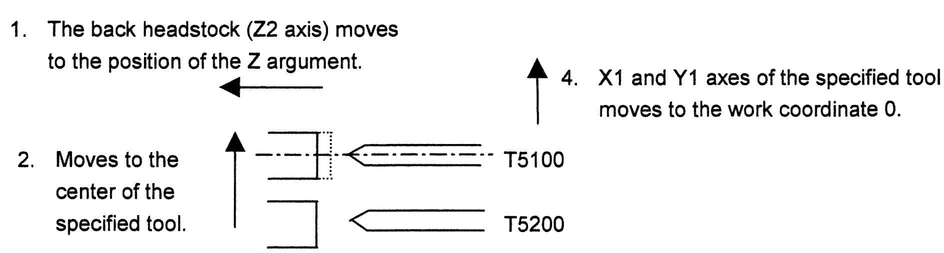
1. The Y1 axis directly moves to the position (the center of the specified tool + Y argument).



- 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.



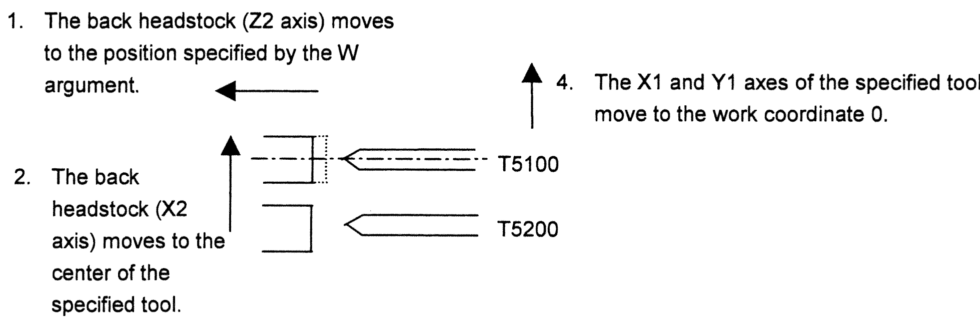
Note

When the Z argument is specified, note that interference between the back headstock (Z2 axis) and the cat hands (X1 and Y1 axes) may not occur.

- 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 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.



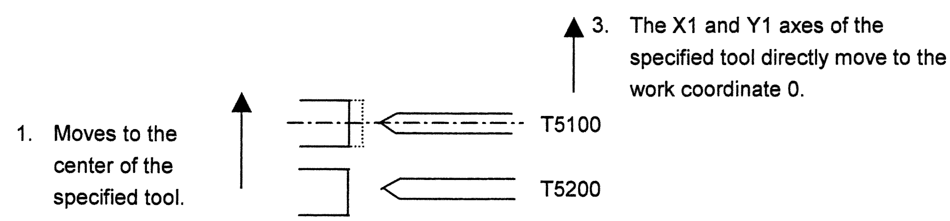
Notes

- 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□□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 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.



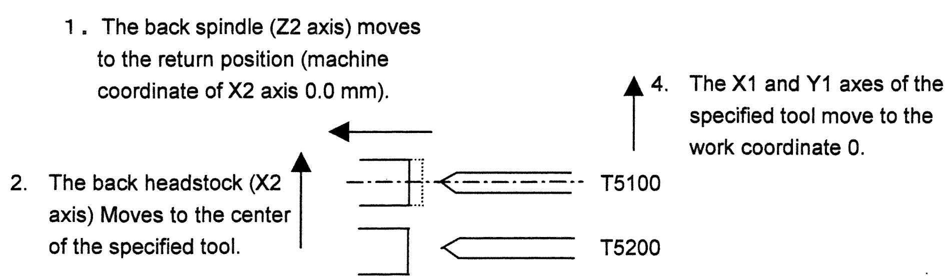
Note

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 positioning point. 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

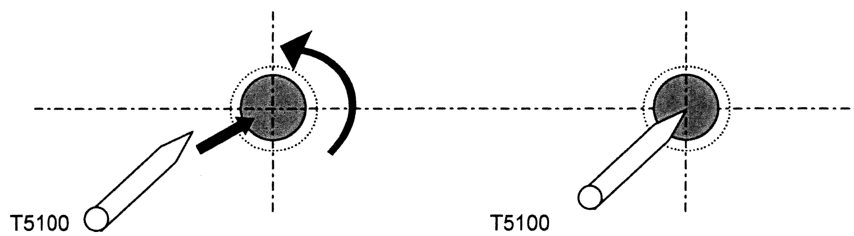
The return position of the back headstock (Z2 axis) is Z2 axis machine coordinate 0.0 mm.

- 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
  ```



The back spindle is indexed during tool selection.

Notes

- 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. However, for the back spindle C axis, positioning and indexing cannot be performed concurrently because the axis returns to the zero point. (This can be performed only at the first time when the back spindle rotation mode is switched to the back spindle C axis mode).
- When an indexing angle (E argument) is specified, the T code cannot be run backward.
- See the back spindle indexing specification for other notes.

4.3.4 Back spindle (T3000)

Command format

T□□ΔΔ Z[ ] W[ ] Q1

Arguments

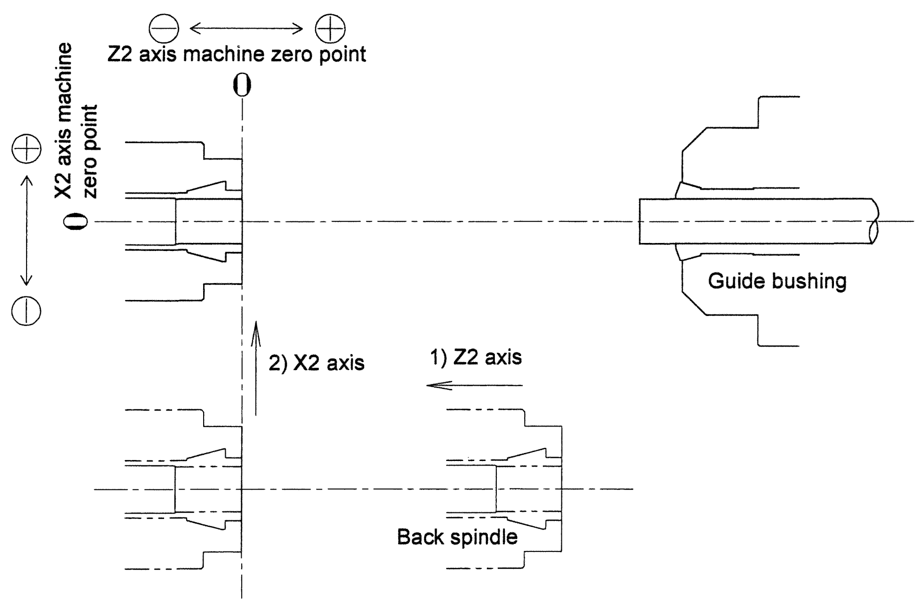
Z: Specify the position to which the back headstock (Z2 axis) is moved (absolute move distance from the machine zero point of Z2 axis). If the argument is omitted, the back headstock (Z2 axis) moves to the positioning point (Z2 axis machine coordinate: 0.0 mm).

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 positioning point (Z2 axis machine coordinate: 0.0 mm).

Q1: Select the back spindle without moving the back headstock (Z2 axis). If the argument is omitted, the back headstock (Z2 axis) moves to the positioning point (Z2 axis machine coordinate: 0.0 mm).

Operation sample

1) The back headstock (Z2 axis) is retracted to the machine zero point (Z2 axis machine coordinate: 0.0 mm).
2) The back headstock (X2 axis) moves to the machine zero point (X2 axis machine coordinate: 0.0 mm).
3) The coordinate system is set with the back headstock (Z2 axis) machine coordinate.



Notes

- Arguments Z, W and Q cannot be specified together. If specified, an alarm occurs.
- If the W argument is specified with a positive value, an alarm occurs.

4.3.5 Back drills on back tool post (T3100 to T3400)

Command format

T□□△△ Z[ ] W[ ] X[ ] Q1 Q3

Arguments

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.

X: 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).

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 positioning point. 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.

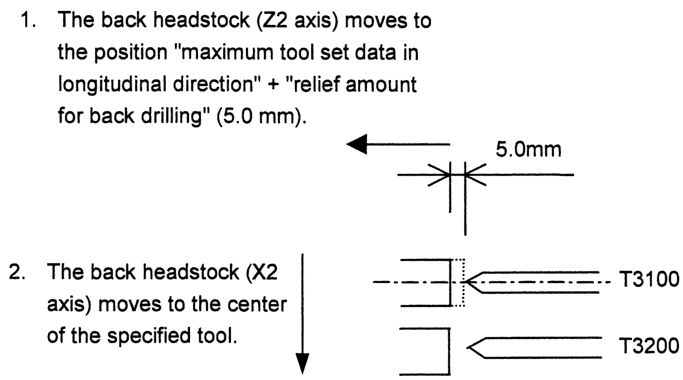
Notes

- Arguments Z, W and Q cannot be specified together. If specified, an alarm occurs.
- While Z2 axis belongs to axis control group 1 ($1), specifying T30's command or K2 argument for T code in axis control group 2 ($2) makes the work coordinate of Z1 axis identical to the machine coordinate.
  Be sure to provide the specification in axis control group 1 ($1). To provide the specification in axis control group 2 ($2), first specify the free pattern (G600).

Operation sample

- 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).



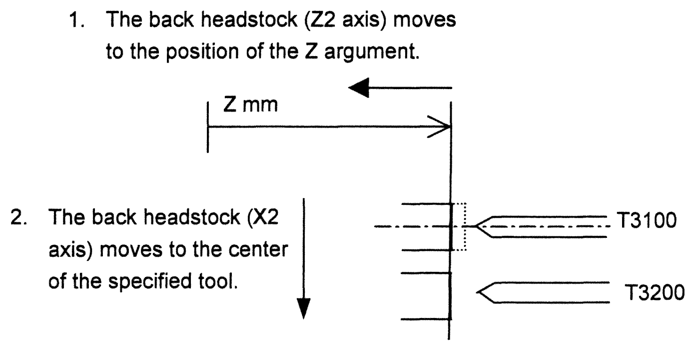
Notes

- If the position of the back spindle (Z2 axis) has exceeded the position indicated in Step 1 (the position "maximum tool set data in longitudinal direction" + "relief amount for back drilling" (5.0 mm)), back spindle (Z2 axis) does not move.
- If a circular operation causes an over travel alarm to occur on the Z2 axis, specify the Q3 argument described later.

- 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 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).



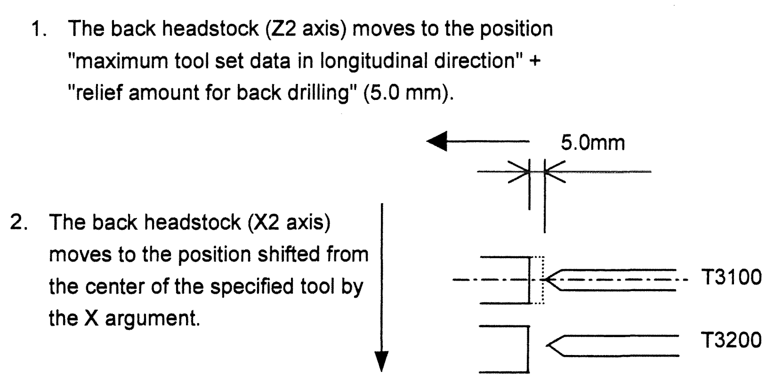
Note

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).

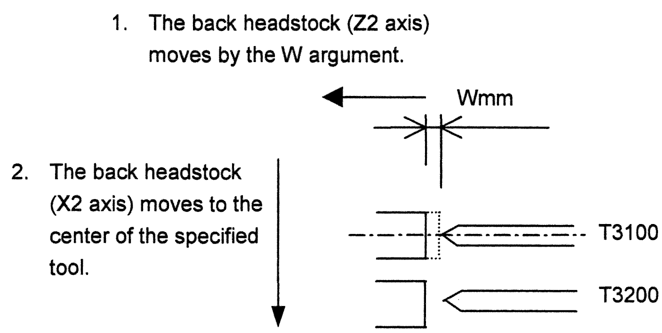
  1. The back headstock (Z2 axis) moves to the position "maximum tool set data in longitudinal direction" + "relief amount for back drilling" (5.0 mm).
  2. The back headstock (X2 axis) moves to the position shifted from the center of the specified tool by the X argument.
  3. The coordinate system is 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 5.0 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).



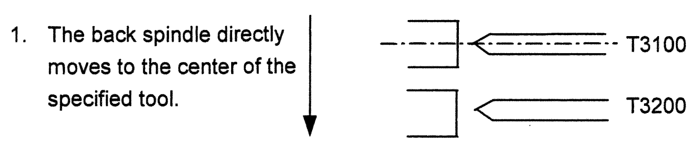
Notes

- 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 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).



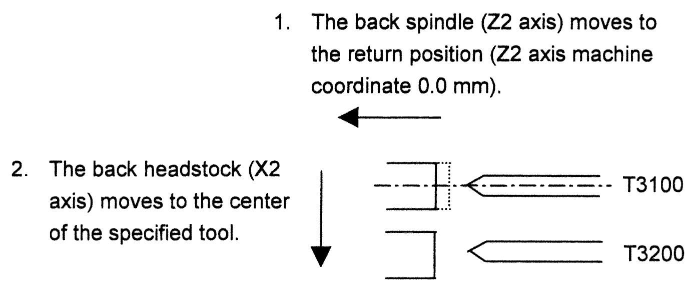
Note

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 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).



Note

The return position of the back headstock (Z2 axis) is Z2 axis machine coordinate 0.0 mm.

4.3.6 T codes and arguments effective in machining patterns

The table below lists T codes and arguments effective in machining patterns. For machining patterns, see <3.6 Machining Patterns>.

Machining patterns	Axis control group	Tool type	Tool number	Arguments											
				None	X	Y	Z	W	H	Q1	Q3	K2	A	E	M1
Free pattern (G600)	$1	Gang tool	T0100 to T1000	○	○	○	○	×	○	○	×	○	×	○	×
		Front drilling tool	T1100 to T1400	○	○	○	○	×	×	×	×	×	×	○	×
		Back spindle	T3000	○	×	×	○	○	×	○	×	×	×	×	×
		Back drilling tool	T3100 to T3400	○	○	×	○	○	×	○	○	×	×	×	×
			T5100 to T5400	○	○	○	○	○	×	○	○	×	×	○	×
	$2	Gang tool	T0100 to T1000	○	○	○	○	×	○	○	×	⊙	×	○	×
		Front drilling tool	T1100 to T1400	×	×	×	×	×	×	×	×	×	×	×	×
		Back spindle	T3000	○	×	×	○	○	×	○	×	×	×	×	×
		Back drilling tool	T3100 to T3400	○	○	×	○	○	×	○	○	×	×	×	×
			T5100 to T5400	○	○	○	○	○	×	○	○	×	×	○	×
Front/back parallel machining (G630)	$1	Gang tool	T0100 to T1000	○	○	○	○	×	○	○	×	×	×	○	×
		Front drilling tool	T1100 to T1400	○	○	○	○	×	×	×	×	×	×	○	×
		Back spindle	T3000	×	×	×	×	×	×	×	×	×	×	×	×
		Back drilling tool	T3100 to T3400	×	×	×	×	×	×	×	×	×	×	×	×
			T5100 to T5400	×	×	×	×	×	×	×	×	×	×	×	×
	$2	Gang tool	T0100 to T1000	×	×	×	×	×	×	×	×	×	×	×	×
		Front drilling tool	T1100 to T1400	×	×	×	×	×	×	×	×	×	×	×	×
		Back spindle	T3000	×	×	×	×	×	×	×	×	×	×	×	×
		Back drilling tool	T3100 to T3400	○	○	×	○	○	×	○	○	×	×	×	×
			T5100 to T5400	×	×	×	×	×	×	×	×	×	×	×	×
Front/back simultaneous machining (G660)	$1	Gang tool	T0100 to T1000	×	×	×	×	×	×	×	×	×	×	×	×
		Front drilling tool	T1100 to T1400	○	○	○	○	×	×	×	×	×	○	○	×
		Back spindle	T3000	×	×	×	×	×	×	×	×	×	×	×	×
		Back drilling tool	T3100 to T3400	×	×	×	×	×	×	×	×	×	×	×	×
			T5100 to T5400	×	×	×	×	×	×	×	×	×	×	×	×
	$2	Gang tool	T0100 to T1000	×	×	×	×	×	×	×	×	×	×	×	×
		Front drilling tool	T1100 to T1400	×	×	×	×	×	×	×	×	×	×	×	×
		Back spindle	T3000	×	×	×	×	×	×	×	×	×	×	×	×
		Back drilling tool	T3100 to T3400	×	×	×	×	×	×	×	×	×	×	×	×
			T5100 to T5400	○	○	○	○	○	×	○	○	×	×	○	×

Machining patterns	Axis control group	Tool type	Tool number	Arguments											
				None	X	Y	Z	W	H	Q1	Q3	K2	A	E	M1
Pick-off, center support (G650)	$1	Gang tool	T0100 to T1000	O	O	O	O	×	O	O	×	O	×	O	×
		Front drilling tool	T1100 to T1400	×	×	×	×	×	×	×	×	×	×	×	×
		Back spindle	T3000	×	×	×	×	×	×	×	×	×	×	×	×
		Back drilling tool	T3100 to T3400	×	×	×	×	×	×	×	×	×	×	×	×
			T5100 to T5400	×	×	×	×	×	×	×	×	×	×	×	×
	$2	Gang tool	T0100 to T1000	×	×	×	×	×	×	×	×	×	×	×	×
		Front drilling tool	T1100 to T1400	×	×	×	×	×	×	×	×	×	×	×	×
		Back spindle	T3000	O	×	×	O	O	×	O	×	×	×	×	×
		Back drilling tool	T3100 to T3400	×	×	×	×	×	×	×	×	×	×	×	×
			T5100 to T5400	×	×	×	×	×	×	×	×	×	×	×	×

⊙: Required O: Available ×: Unavailable

Notes

- If required arguments are not specified or one or more unavailable tools or arguments are specified, an alarm occurs.
- Any commands other than those listed in the table above cannot be specified.

4.4 G Functions (G codes)

Command format

G□□

G-code commands specified by a G followed by a two- or three-digit number provide preparation functions. They are given to the NC unit for controlling the X and Z axes and the spindle.

Example

G00 Rapid feed
G01 Cutting feed

4.4.1 List of G codes

Code	Description	Group	Remarks	Reference
G00	Rapid feed positioning	A		4.4.2
G01	Linear interpolation	A		4.4.3
G02	Circular interpolation (clockwise)	A		4.4.4
G03	Circular interpolation (counterclockwise)	A		4.4.4
G04	Dwell	*		4.4.5
G09	Exact stop		×	
G12.1	Milling interpolation ON (optional)		×	7.9
G13.1	Milling interpolation cancel (optional)		×	7.9
G17	XY plane			7.6
G18	ZX plane			7.6
G19	YZ plane			7.6
G25	Spindle speed fluctuation detection OFF	*		6.4
G26	Spindle speed fluctuation detection ON	*		6.4
G32	Thread cutting	A		5.7/5.8/6.13
G34	Adjustable-lead thread cutting (optional)		×	
G40	Nose R offset cancel	D		6.6/7.6.2
G41	Nose R left offset ON	D		6.6/7.6.2
G42	Nose R right offset ON	D		6.6/7.6.2
G43	Back spindle control OFF		Macro	8.16
G44	Back spindle control ON		Macro	8.16
G50	Coordinate system shift, coordinate system setting	*		5.10
G53	Machine coordinate command		Macro	6.21
G70	Multiple repetitive cycle (finish machining cycle)			6.11
G71	Multiple repetitive cycle (rough machining cycle)		Δ	6.11
G72	End-face rough turning cycle (optional)		Δ	
G73	Closed loop cutting cycle (optional)			
G74	End-face cut-off cycle (optional)			
G75	Inner/outer diameter cut-off cycle (optional)			
G76	Multiple thread cutting cycle (optional)	*		
G80	Rigid tapping mode cancel			6.8/7.7
G83	Canned drilling cycle (Z axis direction)			6.10

Code	Description	Group	Remarks	Reference
G84	Front/back rigid tapping (optional)		Macro	6.8
G85	Front/back boring cycle (optional)			
G87	Canned drilling cycle (X axis direction)			6.10
G88	Cross rigid tapping (optional)		Macro	7.7
G89	Cross boring cycle (optional)			
G90	Outer/inner diameter cutting cycle			
G92	Thread cutting canned cycle	A		5.9
G94	End-face cutting cycle			
G96	Constant surface speed control ON	B		6.5
G97	Constant surface speed control OFF	B		6.5
G98	Feed per Minute (mm/min) [inch/min]	C		4.5
G99	Feed per Rotation (mm/rev) [inch/rev]	C		4.5
G600	Free pattern (machining pattern cancel)		Macro	3.7.3
G630	Front/back parallel machining		Macro	3.7.4
G650	Pick-off / Center-support		Macro	3.7.5
G660	Front/back simultaneous machining		Macro	3.7.6
G113	Spindle synchronization cancel (optional)		Macro	8.8
G114.1	Spindle syncronization command (optional)		Macro	8.8
G813	Spindle synchronization cancel (optional)		Macro	8.8
G814	Spindle synchronization command (optional)		Macro	8.8
G899	Non-conform material phase adjustment command	*	Macro	8.12
G999	Last program		Macro	8.10

Notes

- G codes are modal except marked *.

  If the modal code is once specified, the code is remained effective until any other code to cancel the mode is specified. If the program running in automatic operation or MDI mode is stopped on the way due to reset or emergency stop, or stopped at the end of the program, the modal is reset to the default.

- The modal codes G00, G01, G02, G03, and G92 return to G01 by reset, emergency stop, or reset and rewind command.
- G codes belonging to different groups can be written in the same block.
- G codes cannot be written in a block which contains a T□□□□ code.
- G codes of G999 cannot be executed in MDI mode.
- Single block operation is disabled while G32, G76, G79, G84, G88, or G92 is enabled. If any of the G codes is called during single block operation, the operation continues until the G code is canceled.
- The code containing use macro program must be specified independently. Do not contain any other code in the same block.
- The program marked with × in Remarks column is disabled if the program pre-analysis function is enabled.
- The program marked with Δ in Remarks column is disabled on type II machine if the program pre-analysis function is enabled. See <FANUC Operator's Manual> for details.

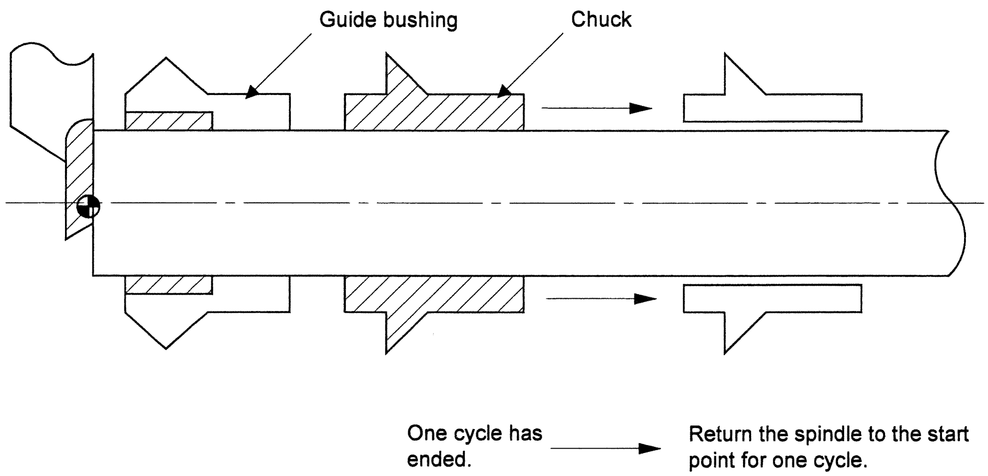
4.4.2 Positioning through rapid feed (G00)

The cutting tool is rapidly moved to the specified position (X, Z) in the coordinate system and is positioned there. This function is used when returning the spindle to the home position.

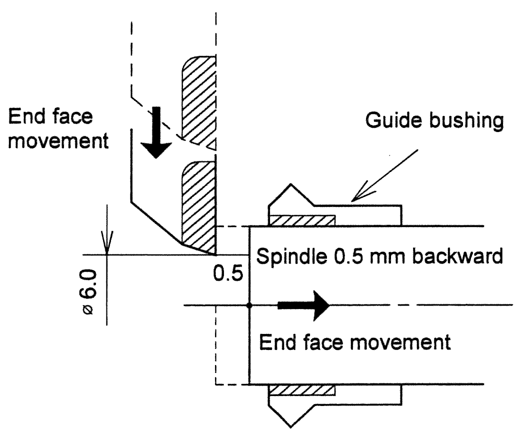
Command format

- Rapid feed in the diameter (X axis) direction `G00 X☐` (Z fixed)
- Rapid feed in the longitudinal (Z axis) direction `G00 Z☐` (X fixed)
- Rapid feed in the diameter (X axis) and longitudinal (Z axis) directions at the same time `G00 X☐ Z☐` (both X and Z moved)

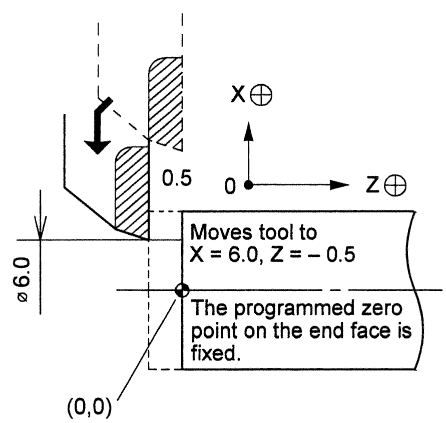
1. Specify G00 Z☐ to return the spindle to the start point when the machine has completed one cycle and is about to proceed to the next cycle.



2. Specify G00 X6.0 Z–0.5 to position the tool to ⌀6 and 0.5 mm from the material.



Actual movement



Motion assumed on the program

Note

Actually, both the tool and the spindle move as shown at left.
However, when programming, it is easier to assume that only the tool is moved while the end face of the material is fixed.

Description

- The G00 command moves the axis rapidly from the current position to the specified position.
- The G00 command is modal, meaning that it is effective until a command in the same group (G01, G02, G03, or other appropriate G code in the G code list) is commanded.
- The rapid feedrate is predetermined and need not be programmed.

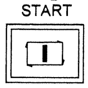
Rapid feedrate	X1 axis	18.0 m/min
	Y1 axis	32.0 m/min
	Z1 axis	32.0 m/min
	X2 axis	32.0 m/min
	Z2 axis	32.0 m/min

- For incremental inputs, specify U$\square$, W$\square$, instead of X$\square$, Z$\square$.

Note

In operation preparation mode, the rapid feedrate of all the axes is 2 m/min. when the Start key

START



is pressed with the door opened.

4.4.3 Linear interpolation (G01)

The tool is moved linearly to the specified position (X, Z) in the coordinate system at the specified feedrate while cutting the workpiece.

This function is called linear interpolation in NC terminology.

Command format

- Feed in the diameter (X axis) direction — `G01 X□ F□` (Z fixed)
- Feed in the longitudinal (Z axis) direction — `G01 Z□ F□` (X fixed)
- Taper feed (X and Z axes) — `G01 X□ Z□ F□` (both X and Z moved)

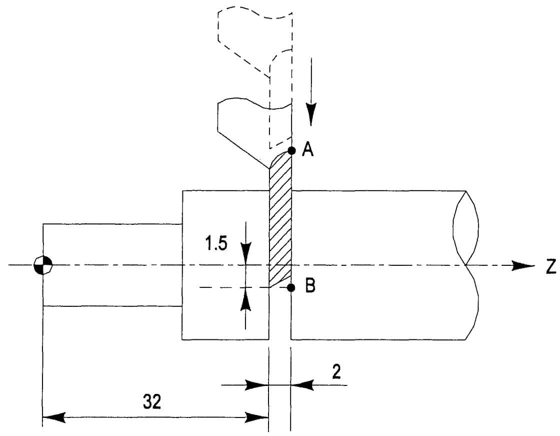
Description

- The G01 command lets the machine cut the workpiece linearly from the current position to the specified position.
- The G01 command is modal, meaning that it is effective until a command in the same group (G00, G02, G03, or other appropriate G code in the G code list) is commanded.
- The code F□ specifies the feedrate. See <Section 4.5 F Functions (F Code) (Cutting Feedrate)>.
- The units of X□ and Z□ coordinates are as follows:

	Program	Unit
Without a decimal point	X 1	0.001 mm
With a decimal point	X 1.0	1 mm

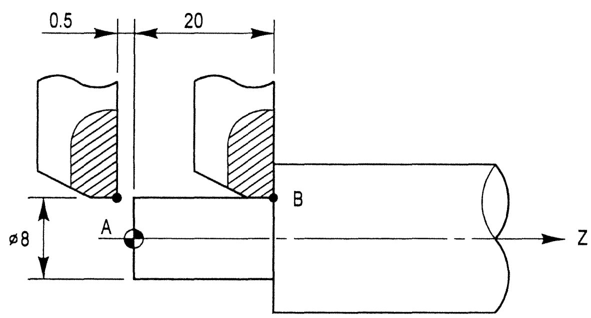
- For incremental inputs, specify U□, W□, instead of X□, Z□.

Tool feed in diameter direction



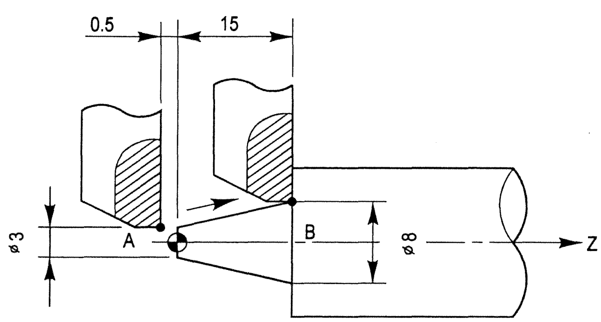
Cutting from point A to point B (–3.0, 34.0)
G01 X–3.0 F☐

Tool feed in longitudinal direction



Cutting from point A to point B (8.0, 20.0)
G01 Z20.0 F☐

Tool feed in both diameter and longitudinal directions (taper)



Cutting from point A to point B (8.0, 15.0)
G01 X8.0 Z15.0 F☐

Note

When starting a cutting feed, be careful of spindle rotation, coolant, chuck, and other conditions.

4.4.4 Circular interpolation (G02, G03)

The machine is moved in an arc of radius R to the specified position in the coordinate system at the specified feedrate while cutting the workpiece.

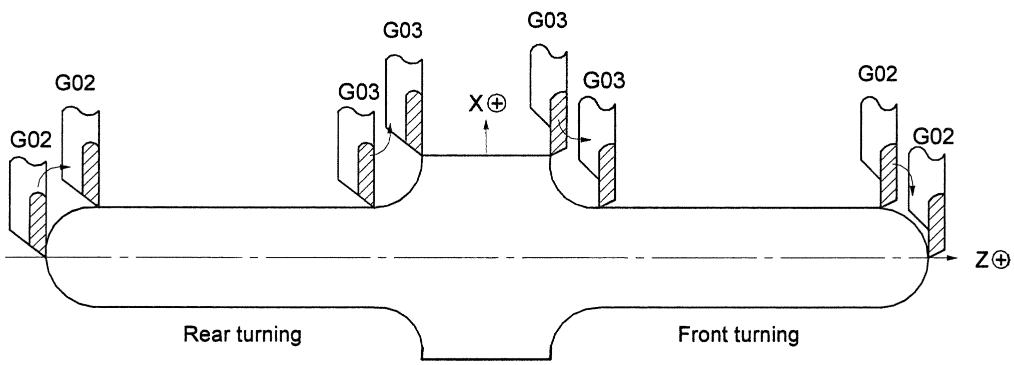
This function is called circular interpolation in NC terminology.

Command format

- G02 X☐ Z☐ R☐ F☐
- G03 X☐ Z☐ R☐ F☐

Description

- The G02 or G03 command lets the machine move in an arc with radius R from the current position to the specified position while cutting the workpiece. The center of the circle need not be specified.
- The coordinate systems for G02 and G03 are different as follows:



How to remember: To move the cutting tool in positive direction of the Z axis:
Specify G02 if the finish form is convex.
Specify G03 if the finish form is concave.

- The G02 and G03 commands are modal, meaning that they are effective until a command in the same group (G00, G01, or other appropriate G code in the G code list) is commanded.
- Specify the radius in R☐.
- The code F☐ specifies the feedrate. See <Section 4.5 F Functions (F Code) (Cutting Feedrate)>.
- The units of X☐ and Z☐ coordinates are the same as with G01.
- Coordinates for G02 and G03 can also be specified in the following way:
- The I and K codes specify the distance (signed) to the center of the arc from the starting point. If they are specified, the radius need not be specified.
- I0 and K0 can be omitted.

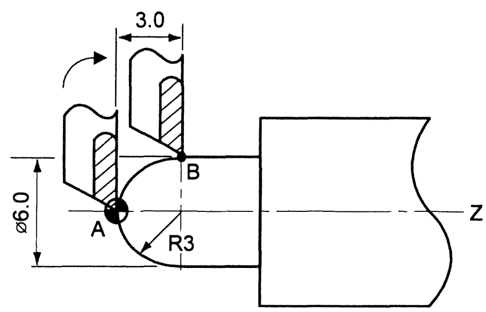
Command format

```
G02 X[    ] Z[    ] I[    ] K[    ] F[    ]
G03 X[    ] Z[    ] I[    ] K[    ] F[    ]
```

G02 (The shape after machined: convex)

Circular cutting from point A to point B (6.0, 3.0)

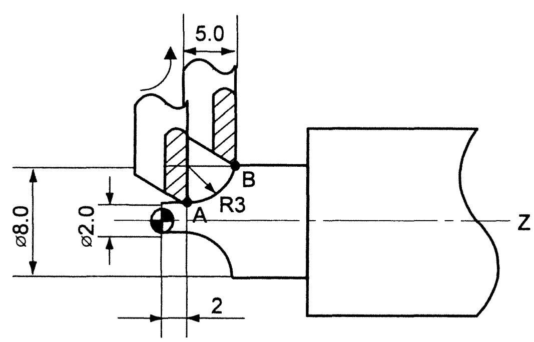
```
            G02 X6.0 Z3.0 R3.0 F[    ]
            (G02 X6.0 Z3.0 I0 K3.0 F[    ])
I0 is omitted   (G02 X6.0 Z3.0 K3.0 F[    ])
```



G03 (The shape after machined: concave)

Circular cutting from point A to point B (8.0, 5.0)

```
            G03 X8.0 Z5.0 R3.0 F[    ]
            (G03 X8.0 Z5.0 I3.0 K0 F[    ])
K0 is omitted   (G03 X8.0 Z5.0 I3.0 F[    ])
```



4.4.5 Dwell (G04)

This command halts program execution for a specified period of time, making the machine stay as is.

It is specified as follows:

Command format

`G04 U□`   □ : Time in seconds

The above command has the following effect: After executing the commands in the preceding block, the machine waits for the number of seconds specified with U□ or P□ following G04, then proceeds to the next block.

The maximum command time is 9999.999 seconds.

Sample program

To make the machine dwell for 2.5 seconds, specify `G04 U2.5` or `G04 P2500`.

Notes

- If address P is used, no decimal point can be used.
- The dwell period starts when the specified speed in the preceding block becomes 0.

Purposes of dwelling

- To stabilize groove diameter.
- To wait until the speed of rotation for threading drops to the specified value.
- For drilling (stepped) deep holes.

4.5 F Functions (F Code) (Cutting Feedrate)

This command specifies a feedrate for linear or circular cutting.

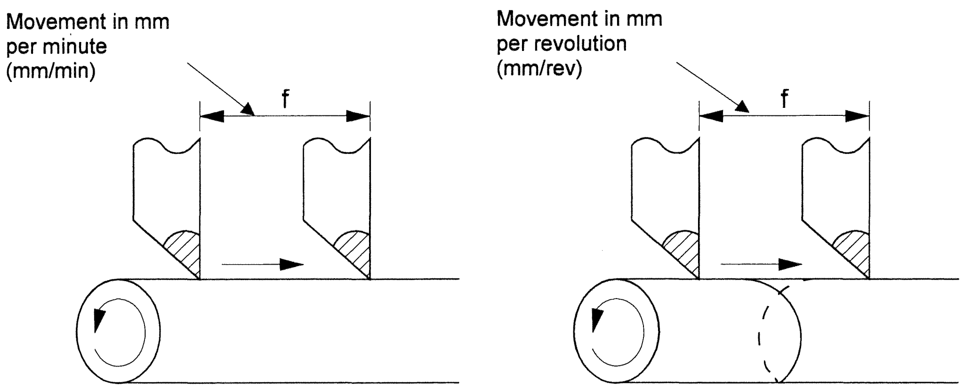
Command format

Feedrate specification F□

Description

- This command specifies a feedrate at which the machine moves along a path while cutting the workpiece as specified by G01, G02, or G03.
- One of two units can be used to specify feedrates:

  G98: mm/min (feed per minute: tool movement in millimeters in one minute)
  G99: mm/rev (feed per rotation: tool movement in millimeters in one spindle revolution)



Feed per minute   Feed per rotation

	Feed per minute	Feed per rotation
Meaning	Tool movement per minute	Tool movement per spindle revolution
Address	F	F
G code	G98	G99
Input in mm	1 mm/min to 8,000 mm/min (F1 to F8,000)	F (mm/rev) $\leq \dfrac{\text{8,000 mm/min}}{\text{spindle speed (min}^{-1}\text{)}}$
Override	Fixed to 100% for either mode of feed	

- The F□ command is modal, meaning that the specified value remains effective until another F□ value is specified.
- Determine a suitable feedrate according to the cutting condition table.

Notes

- If no F☐ command is not given, the F value used during the previous machining process is used for cutting feed.
- Immediately after the power is turned on, the condition of G99 is in effect.
- Mode switching from G99 to G98 in the same block as a G01 cutting block must be specified before the cutting command.

Sample program

```
G98  G01  X__Z__F__
```

4.6 M Functions (M codes)

4.6.1 List of M codes

M code	Function	Remarks	Reference
M00	Program Stop		
M01	Optional Stop		
M02	Cycle Stop	Macro	
M03	Main Spindle Forward Rotation		4.6.2
M04	Main Spindle Reverse Rotation		4.6.2
M05	Main Spindle Stop		4.6.2
M06	Chuck Close	Macro	4.6.5
M07	Chuck Open (Starts to open chuck when the spindle speed becomes 800 min^{-1} or slower.)	Macro	4.6.5
M08	Bar Stock Exchange ON	Macro	6.2
M09	Bar Stock Exchange OFF	Macro	6.2
M10	Knock-out Advance	Macro	8.6
M11	Knock-out Retract	Macro	8.6
M15	Back Spindle Chuck Close	Macro	4.6.6
M16	Back Spindle Chuck Open	Macro	4.6.6
M18	Main Spindle C Axis (optional)	Macro	7.2.3
M20	Main Spindle Indexing Cancel, Main Spindle C Axis Cancel	Macro	7.2.1
M23	Back Spindle Forward Rotation		4.6.3
M24	Back Spindle Reverse Rotation		4.6.3
M25	Back Spindle Stop		4.6.3
M28	Main Spindle Indexing	Macro	7.2.1
M31	Workpiece Conveyor ON	Macro	4.6.8
M33	Move Product Collection Position	Macro	8.7
M34	Sequential Operation for Product Collection	Macro	8.7
M48	Back Spindle C Axis (optional)	Macro	7.2.4
M50	Back Pick-Off Failure Detect	Macro	8.13
M51	Tool Breakage Detection	Macro	6.7
M52	Coolant ON		
M53	Coolant OFF		
M54	Bar Loader Feed Stop		6.2
M55	Bar Loader Feed Start		6.2
M56	Products Count		
M61	External M Code 1		
M62	External M Code 2		
M63	External M Code 3		
M64	External M Code 4		
M65	External M Code 5		
M72	Back Spindle Air Blower ON		8.4
M73	Back Spindle Air Blower OFF		8.4
M77	Confirm Spindle Synchronization (optional)		8.8
M78	Back Spindle Indexing (optional)	Macro	7.2.2
M79	Back Spindle Indexing Cancel, Back Spindle C Axis Cancel	Macro	7.2.2
M80	Tool Spindle Forward Rotation	Macro	4.6.4/7.2.5
M81	Tool Spindle Reverse Rotation	Macro	4.6.4/7.2.5
M82	Tool Spindle Stop	Macro	4.6.4/7.2.5

M code	Function	Remarks	Reference
M86	Cutting Block Interlock ON		
M87	Cutting Block Interlock OFF		
M88	Interference Check Disabled		4.6.12
M89	Interference Check Enabled		4.6.12
M92	Error Detect Enabled		
M93	Error Detect Disabled		
M94	Back Spindle Speed Fluctuation Detect ON	Macro	6.4
M95	Back Spindle Speed Fluctuation Detect OFF	Macro	6.4
M96	Main Spindle Speed Fluctuation Detect ON	Macro	6.4
M97	Main Spindle Speed Fluctuation Detect OFF	Macro	6.4
M98	Sub Program Call		6.3
M99	End of Sub Program Call		6.3
M108	Material Exchange	Macro	6.2
M109	General-purpose Material Change	Macro	6.2
M123	Back Spindle Torque Limit ON		
M124	Back Spindle Torque Limit OFF		
M130	Z1-Z2 Superimposition ON	Macro	8.9
M131	Z1-Z2 Superimposition OFF	Macro	8.9
M151	Gang Tool Post Retract	Macro	4.6.10
M190	C1-C2 Superimposition ON	Macro	8.10
M191	C1-C2 Superimposition OFF	Macro	8.10
M204	Thread Cutting and Chamfering ON		4.6.11
M205	Thread Cutting and Chamfering OFF		4.6.11
M241	Move Back Spindle to Return Point	Macro	8.5
M260	External M61 Relay ON		
M261	External M61 Relay OFF		
M262	External M62 Relay ON		
M263	External M62 Relay OFF		
M264	External M63 Relay ON		
M265	External M63 Relay OFF		
M266	External M64 Relay ON		
M267	External M64 Relay OFF		
M268	External M65 Relay ON		
M269	External M65 Relay OFF		
M320	Basket Advance (with basket mounted on back spindle)	Macro	4.6.7
M430	Coolant Valve 1 ON		6.19
M431	Coolant Valve 1 OFF		6.19
M432	Coolant Valve 2 ON		6.19
M433	Coolant Valve 2 OFF		6.19
M434	Coolant Valve 3 ON		6.19
M435	Coolant Valve 3 OFF		6.19
M436	Coolant Valve 4 ON		6.19
M437	Coolant Valve 4 OFF		6.19
M452	Coolant Pump ON		6.19
M453	Coolant Pump OFF		6.19
M700	Chuck Open (unconditional)	Macro	4.6.5
M780	Back spindle indexing (zero point)	Macro	4.6.7

Notes

- Execute a macro-used M code independently. Do not execute the M code in the same block as for another code.
- Up to three M codes can be specified in one block except for macro-used M code.
- The above M codes include options. For the options, see <Chapter 3 Machine Specifications> in the Operator's Manual.

4.6.2 Spindle rotation (M3, M4, M5)

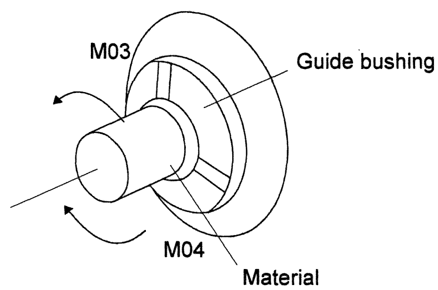
These commands rotate the spindle in the forward or reverse direction and stop it.

Command format

M03	Main spindle rotation forward
M04	Main spindle rotation reverse
M05	Main spindle rotation stop

Description

- The M03 command rotates the spindle in the forward direction; the M04 command rotates the spindle in the reverse direction; the M05 command stops the spindle.
  Counterclockwise rotation when viewed toward the spindle is regarded as forward rotation.



- If the spindle is rotating as specified by the code at the beginning of the program, its speed can be changed only by an S1=☐ command.
  The M03 and M04 codes change only the direction of rotation, leaving the speed unchanged.

- The M05 code stops spindle rotation. Pressing the All spindles stop key on the operation panel can also stop the spindle rotation.

4.6.3 Back spindle rotation (M23, M24, M25)

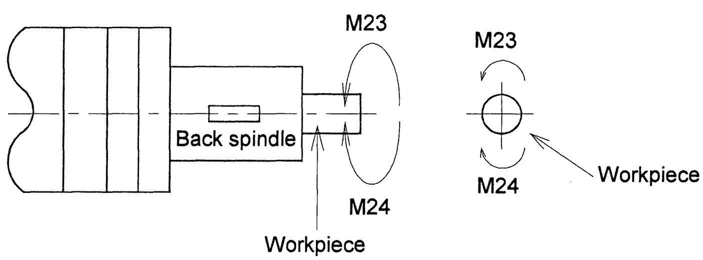
These commands rotate the back spindle in the forward or reverse direction and stop it.

Command format

M23	Back spindle rotation forward
M24	Back spindle rotation reverse
M25	Back spindle rotation stop

Description

- Counterclockwise rotation when viewed toward the back spindle is regarded as forward rotation.



- During the back spindle is rotating, its speed can be changed only by an S2=☐ command.
- Pressing the All spindles stop key on the operation panel can also stop the back spindle rotation.

4.6.4 Tool spindle rotation (M80, M81, M82)

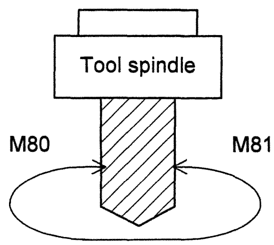
These commands rotate the tool spindle T0600 (T0700) to T0900 in the forward or reverse direction and stop it.

Counterclockwise rotation when viewed toward the spindle is regarded as forward rotation.

Pressing the All spindles stop key on the operation panel can also stop the tool spindle rotation.。

Command format

`M80 S3=□ (M80 S□)`	Tool spindle rotation forward
`M81 S3=□ (M81 S□)`	Tool spindle rotation reverse
`M82`	Tool spindle rotation stop



Notes

- To rotate the tool spindle, specify M80, M81, and speed (S□ or S3=□) in the same block. Enter the tool spindle rotation command in one block as single command.
- Issuing only M80 and M81 is not permitted. Be sure to specify speed (S3=) together with them.
- To change the tool spindle speed, specify M80, M81, and speed (S□ or S3=□) in the same block.
- If the tool spindle forward rotation, tool spindle reverse rotation, and stop commands are specified successively, incorrect operation may be performed. To issue these commands successively, provide a dwell among them.

  Example:
  ```
  M82;
  G04 U0.1;
  M80 S3=1000;
  ```

  Specify the tool spindle rotation or stop command (M80, M81, or M82) in axis control group 1 ($1). If specified in axis control group 2 ($2), an alarm occurs.
- Specify the positive value for rotation speed (S argument). If specified with negative value, an alarm occurs.
- To use the tool spindle in continuous operation, the rated speed as shown below must be specified.
  Specifying a speed exceeding the rated speed for machining may cause an overload alarm to occur.

Rotary tool drive device	Rated speed	Maximum speed
U31B (3-rotary tool model)	6,000	8,000
U32B (4-rotary tool model)	4,500	6,000

- If the speed exceeding the maximum speed is specified in S argument, an alarm occurs.
- G99 (feed per rotation) cannot make feeding. Use G98 (feed per minute) for feeding.

4.6.5 Spindle chuck open/close (M6, M7, M700)
(Must be specified in independent block.)

Command format

M06 U☐	Chuck close	
M07 U☐	Chuck open	Chuck opens when the spindle speed drops 800 min^{-1} or less.
M700 U☐	Chuck open unconditionally	Chuck opens regardless of spindle speed.

Note

With the machine equipped with the automatic bar loader, specify U argument (dwell time after chuck open) to retract the workpiece along with the main headstock (Z1 axis) after the front chuck has been opened. Use dwell time for 1.0 second as standard.

Arguments

U Dwell time after main spindle chuck has been opened/closed

Sample program

```
↓
M05 .......................... Main spindle stop
M07 .......................... Chuck open
                               Specify M07 independently in the line next to M05.
                               In the program, chuck opens when the spindle speed drops 800 min-1 or less by the
                               spindle stop command.
↓
M05
M700 ......................... Specify M700 independently in the line next to M05.
                               In the program, chuck opens regardless of spindle speed.
```

Notes

- The M700 (unconditional check open) command can be executed even while the main spindle is rotating. Sufficient care is needed to execute the M700 command (e.g., to reduce the rotation speed of main spindle).
- The M6, M7, or M700 command cannot be run backward.
  If an attempt is made to run backward M6, M7, or M700 program in on-machine program check, the machine stops before the block containing M6, M7, or M700.

4.6.6 Back spindle chuck open/close (M15, M16)
(Must be specified in independent block.)

Command format

`M15 U□`	Back chuck close
`M16 U□`	Back chuck open

Argument

U Dwell time after back spindle chuck has been opened/closed

Sample program

```
↓
M15 ..................................... Back chuck close
M23 S2=2500 ............................. Back spindle forward rotation
↓
M25 ..................................... Back spindle stop
M16 ..................................... Back chuck open
```

Note

The M15 or M16 command cannot be run backward.

If an attempt is made to run backward M15 or M16 program in on-machine program check, the machine stops before the block containing M15 or M16.

4.6.7 Move basket forward (M320, M780)

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. The M34 (sequential operation for product collection) command collects the machined workpiece in the receiver box (workpiece separator) mounted below the back spindle.

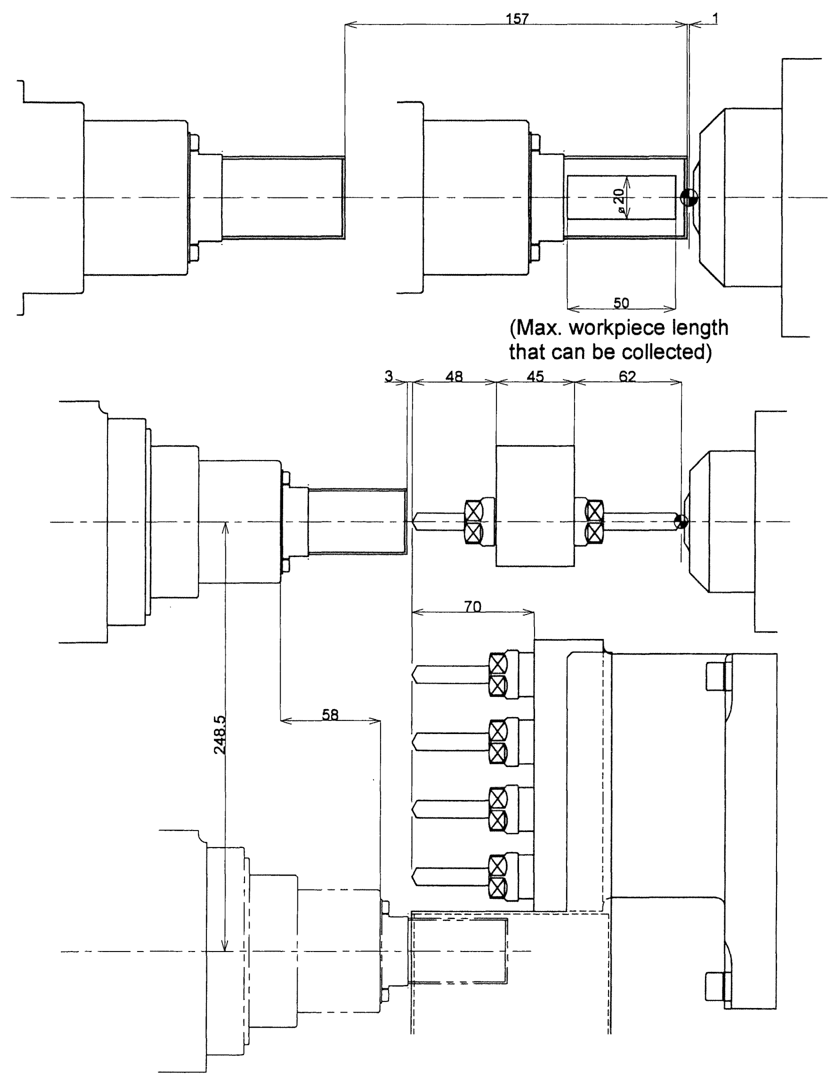
Command format

`M320 X☐ Z☐ F☐` Move basket forward

X: Specify the X1 axis position for end-position queuing. 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.

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 157.0 mm of machine coordinate.

F: Specify the feed rate (per minute) of the back headstock (Z2 axis). If F argument is omitted, the back headstock moves at the speed of 8,000 mm/min.
This argument is valid only when end-position queuing is not specified (with X argument not specified). Specify the feed rate with decimal point. If end-position queuing is specified (with X argument specified), the feed rate will be calculated automatically. Accordingly, this argument is ignored.



Sample program

- With queuing at end point

$1		$2	
G600	Free pattern (machine pattern cancel)	G600	Free pattern (machine pattern cancel)
:		:	
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 waits for Z2 axis (basket) at the point 157.0.		
G99 G1 X-3.0 F0.03	Perform cut-off machining.		
G600	Free pattern (machine pattern cancel)	G600	Free pattern (machine pattern cancel)
		M34	Sequential operation for product collection

- Without queuing at end point

$1		$2	
:		:	
T0100	Select the cut-off tool.		
!2 L1	Queuing state	!1 L1	Queuing state
G0 X20.0 Z10.0 T1	Position the cut-off tool.	M320 F8000.0	Move the basket forward (Z2=157.0)
!2 L2	Queuing state	!1 L2	Queuing state
G99 G1 X-3.0 F0.03	Perform cut-off machining.		
G600	Free pattern (machine pattern cancel)	G600	Free pattern (machine pattern cancel)
		M34 F6000.0	Sequential operation for product collection

Notes

- Before executing the M320 (move basket forward) command with end-position queuing specified (with X argument specified), be sure to select the free pattern (machining pattern cancel) (G600) command. In addition, specify the G99 (feed per rotation) and the G1 (linear interpolation) to feed the cut-off tool. Do not specify the G98 (feed per minute) and G2 or G3 (circular interpolation).
- When executing the M320 command with end-position queuing (with X argument specified), the program cannot be corrected using the on-machine program check function.
- 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.
- Before executing the M320 (move basket forward) command with end-position queuing specified (with X argument specified), specify the values so that they do not satisfy the following formula. Otherwise, an alarm PS462 occurs.
  |Value of X argument for M320| + |Value of X for cut-off process in the next block| < X offset value of cut-off tool

  * | | denotes the absolute value.
- To execute the M320 (move basket forward) command with end-position queuing not specified (with X argument not specified), specify the G98 (per minute feed) with decimal point for the feed rate (F argument) of back headstock (Z2 axis).
  Specify the G99 (per rotation feed) command for cut-off machining.
- 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.
- Specify 157.0 mm or less for Z argument. Specifying the value exceeding 157.0 causes an alarm to occur.
- If the product collection alarm is enabled on Operator's panel screen, issuing theM320 (move basket forward) command with rotary tool or drilling holder (T11 to T14) on the gang tool post being selected, causes an alarm to occur.
- Specifying end-position queuing specified (with X argument specified) while executing the Constant surface speed control ON (G96) will cause an alarm to occur.
- Reverse running of M320 (move basket forward) command is inhibited.
- When executing the program pre-analysis function with end-position queuing enabled (with X argument specified), do not set the override to 0%. Setting override to 0% will cause an alarm to occur.

Mounting and adjusting of basket on back spindle

Procedure

1. Execute the M780 (back spindle indexing, zero point) in MDI mode.
2. Press the Reset key to cancel indexing, open the door, then mount the basket in the following procedure.
3. Mount the basket so that it becomes level at the point where the back spindle is indexed. For how to mount the basket, refer to <7.9 Mounting and Adjusting Workpiece Separator [B] Workpiece receiver box collection mode (U352J)> in the Operator's Manual.
4. Specify the M780 (back spindle indexing) command in MDI mode. Check if the basket has been mounted at level.

Notes

- Select "BASKET" for "BACK SPINDLE" in machining data screen. Specifying M780 (back spindle indexing, zero point) command while "STANDARD" is selected for "BACK SPINDLE" causes an alarm to occur.
- Executing the M780 (back spindle indexing, zero point) command in any other mode than MDI causes an alarm to occur.

4.6.8 Workpiece conveyor start (M31)

This conveyor sends the products collected from the workpiece chute, to the outside of the machine.

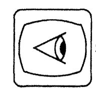
Command format

```
M31
```

Operation

The workpiece conveyor operates for 14 seconds.

Notes

- The M31 command is valid only when the workpiece conveyor switch operates intermittently.
- The M31 command is included in M34 command.
- To set the time for operating the workpiece conveyor, press the Maintenance key , press the menu keys [PARAM], [PMCMNT], and [TIMER] in the order, then change variable for T24 to 15000. The timer is set to 15 seconds.

4.6.9 Knock-out advance/retract (M10, M11)

Outline

The back spindle knock-out device advances or retracts.

Command format

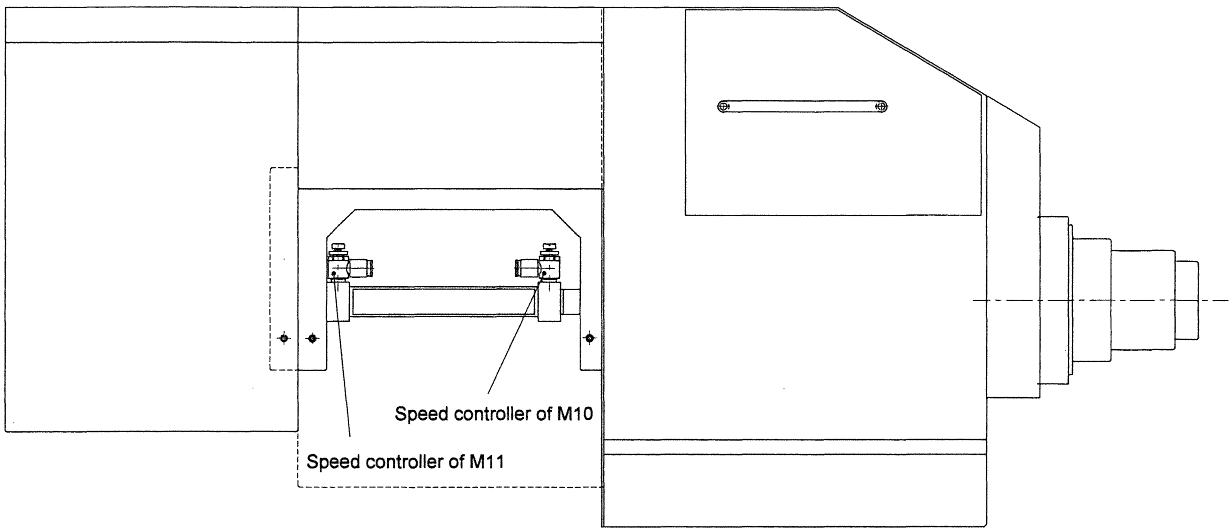
`M10`	Knock-out advance
`M11`	Knock-out retract

Command format

- Knock-out advance (M10)
  The knock-out advance (M10) command allows the knock-out to be advanced by the air cylinder.
- Knock-out retract (M11)
  The knock-out advance (M11) command allows the knock-out to be retracted by the air cylinder.

Notes

- If the back chuck is closed or the back spindle is rotating, the knock-out advance/retract (M10/M11) command causes an alarm to occur.
- Adjusting the speed controller of the air cylinder allows the speed of the knock-out advance/retract to be adjusted.



- If the knock-out pipe fails to work correctly during knock-out process due to jammed workpiece or others, the "Knock-out Overload" alarm occurs. For restting the alarm, see <5.5.11 Restoration from Other Alarms> in the Maintenance Manual.

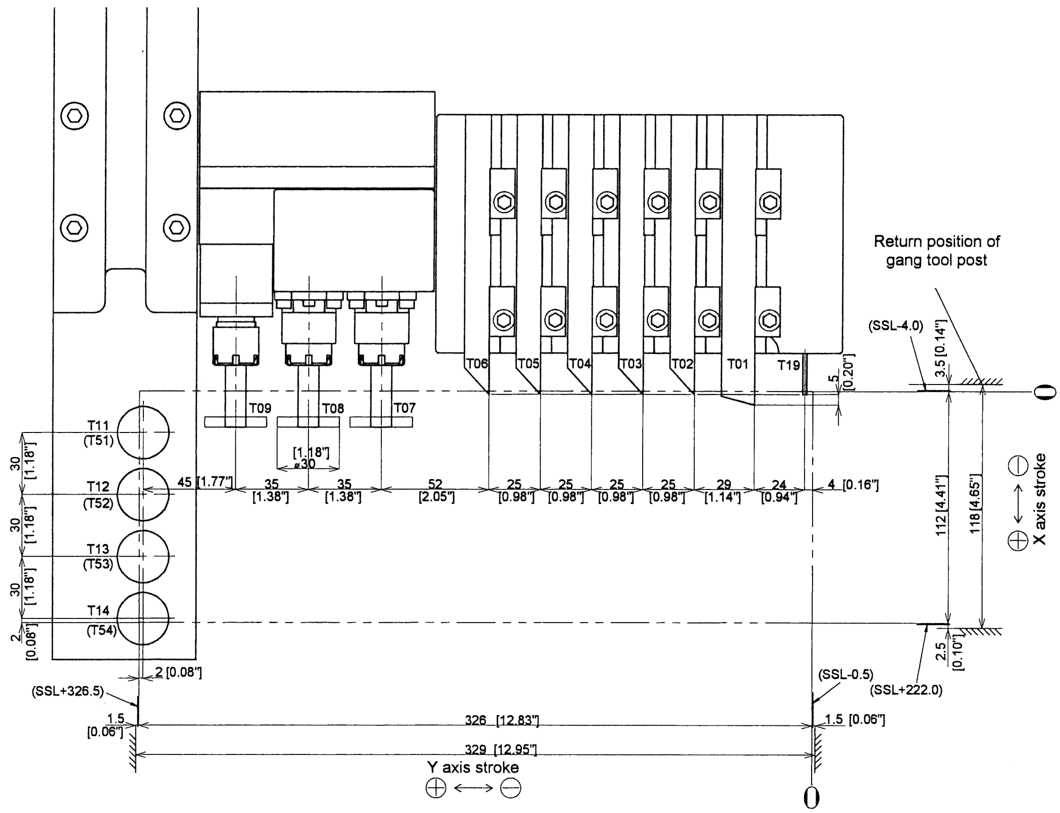
4.6.10 Gang tool post retract (M151)

Retract the gang tool post to the return point. Only the X1 axis is moved.

Command format

`M151`   Gang tool post retract

Description



Notes

- Specify M151 (Retract gang tool post) command in $1. Specifying M151 command in $2 causes an alarm to occur.
- When the X1 axis belongs to axis control group No. 2 ($2), issuing M151 (Retract gang tool post) command causes an alarm to occur.

4.6.11 Thread Cutting and Chamfering ON/OFF (M204, M205)

With conventional method, the chamfering function can be turned on or off by selecting No. 6 of Operator's panel screen. You can specify it by using the program M code.

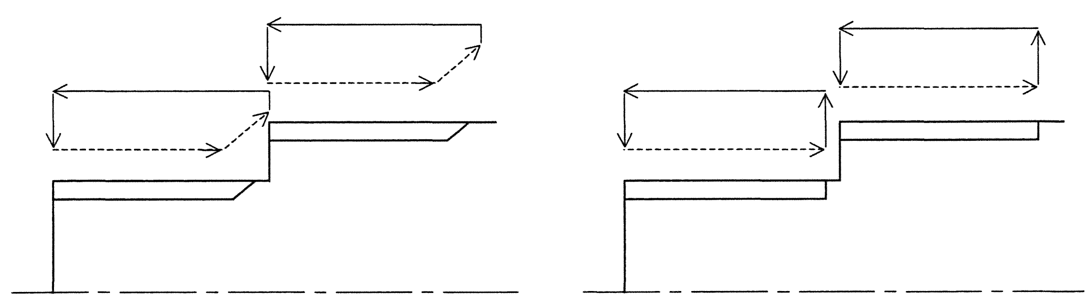
Command format

M204	Thread Cutting and Chamfering ON
M205	Thread Cutting and Chamfering OFF

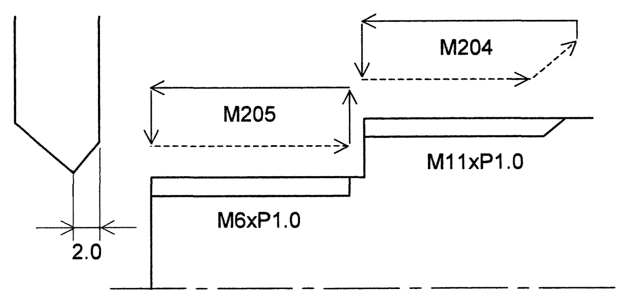
Description

Turning on by selecting No. 6 of Operator's panel screen.

Turning off by selecting No. 6 of Operator's panel screen.



Specifying M code turns on or off the chamfering function regardless of setting on No. 6 of Operator's panel screen.



Sample program

```
M3 S1=800
T300
M205 ....................................... Thread cutting and chamfering OFF
G04 U1.0
G50 W–2.0
G0 X8.0 Z–2.0 T3
G92 X5.5 Z12.0 F1.0
X5.06
:
G0 X12.0 Z–2.0
G50 W2.0
M204 ....................................... Thread cutting and chamfering ON
:
M3 S1=800
T300
G50 W–2.0
G0 X13.0 Z13.0 T3
G92 X10.56 Z20.0 F0.1
X10.26
:
G0 X13.0
```

Note

See also <5.8.3 Threading with G92>.

4.6.12 Interference Check Function (M88, M89)

The interference check function checks position data set as machining data and machine structure data to prevent machine parts 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 when the power supply is turned on.

The interference check is disabled if you either specify the "INT.CHK CAN" on Operator's panel screen or include the interference check disable command in the program. If you run the machine with this function off, therefore, check all the relevant parts in advance and use extreme care. You should enable the interference check function in all the processes to which the function is available.

Command format

M88	Disable interference check
M89	Enable interference check

Notes

- 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 a improper 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 extreme care.
- Even when the back spindle is being used in the basket collection mode, the interference check applies to the basket.

4.7 S Functions (S codes)

The S functions specified in the following formats are called speed functions. The S functions are used to specify the speeds of the main spindle and the back spindle.

Command format

S1=□□□□	Main spindle	five-digit command
S2=□□□□	Back spindle	five-digit command

The speed of the main spindle, opposite tool spindle and back spindle are calculated using the following formula. The result is rounded off to a whole number.

$$N = \frac{V}{\pi D} \times 1{,}000$$

N: Speed in rpm (min^{-1})
V: Cutting speed in m/min [inch/min]
D: Material diameter in mm [inch] or hole diameter (in the case of drilling)
π: Circular constant (approx. 3.14)

Main spindle	100~8,000min^{-1}
Back spindle	400~8,000min^{-1}

Note

Do not specify S1=□ and S2=□ in the same block.

4.8 ! (Exclamation) Function (Queuing Code)

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

Command format

!□ L□   Multi-axis control group queuing

Arguments

!: Specify the axis control group number (1 or 2) of an axis control group which enters the queuing state while the operation of another axis control group's program is in progress.

L: Queuing ID number (0 to 799, 900 to 999)

Sample program

```
O□□□□
$1                    $2
.                     .
.                     !1 L1
!2 L1                 .
.                     .
!2 L2                 !1 L2
.                     .
.                     .
.                     !1 L3
!2 L3                 .
.                     .
.                     .
.                     .
!2 L4                 !1 L4
M56
G999                  G999
N999                  N999
M02                   M02
%                     %
```

Note

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

Product Code	C	–	A	2	2	0	P	L	VII					
Document Code	2	E	1	–	0	4	0	1						

5. Sample Programming [Basic]

5.1 Program Structure

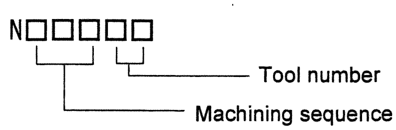
Command format

• Program number	O□□□□	
• Axis control group number	$1	$2
• Sequence number	N□□□□□	N□□□□
• Product count	M56	
• Last program execution command	G999	G999
• Sequence number	N999	N999
• One cycle stop	M02	M02
• Stop code	%	%

Description

- The program number attached at the beginning of the program helps identify the program. The available numbers are from O0001 to O8999.
- A single program for the A20 machine can contain programs for two axis control groups. The contained programs must therefore be identified by different axis control group numbers ($1 and $2).
- Adding a sequence number to the beginning of each process makes it easy to check the program. Sequence numbers are only reference numbers excluding N999. You can omit sequence numbers except N999 because they have no effect on machining.
  Each sequence number consists of the letter "N" followed by a number of up to five digits, ranging from N00001 to N99999 (except N999). Only one-digit sequence number such as "N1" can also be specified.

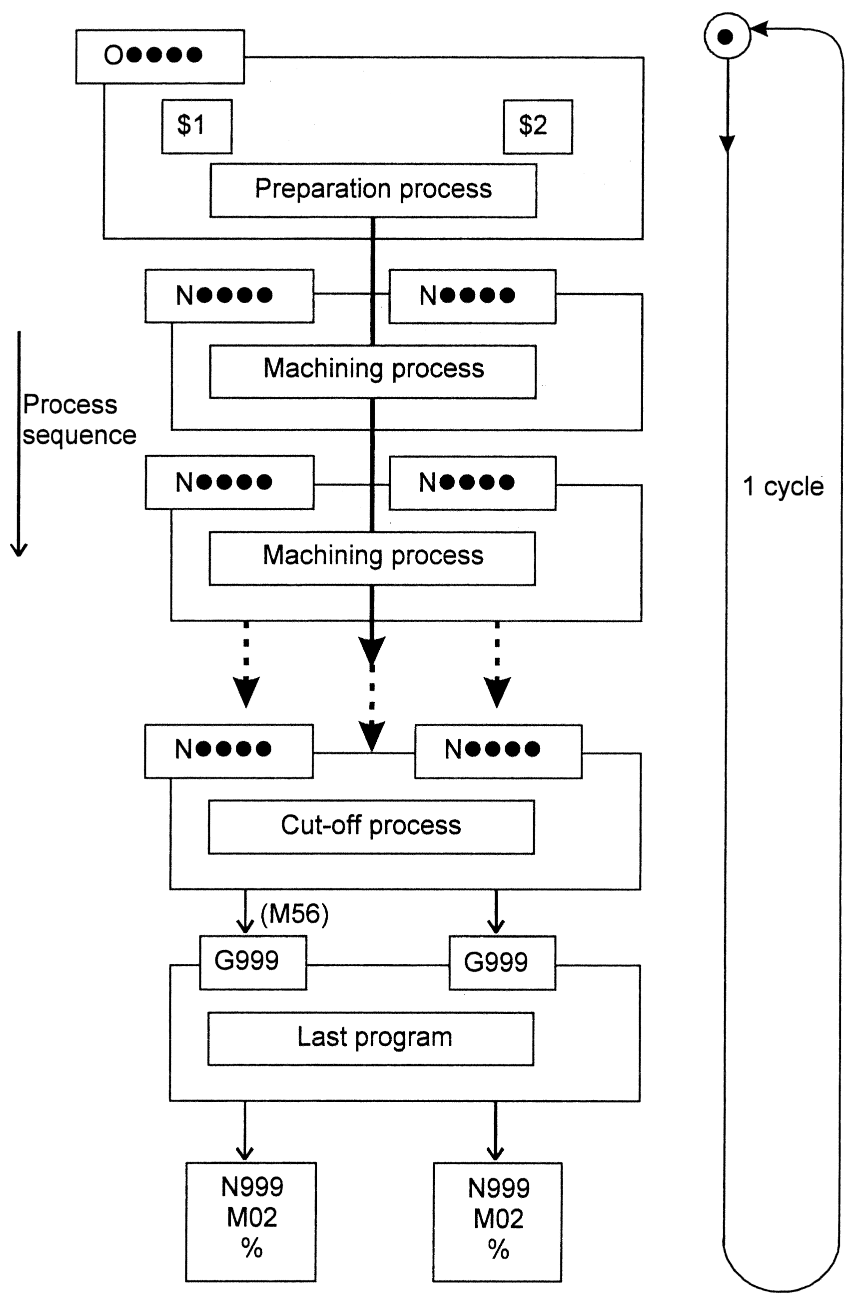
  You can assign sequence numbers in many ways. For example, using the following format for assignment makes it easy to confirm the machining sequence of each process and the tool used in the process.

  To specify the first process using tool No. 03, for example, assign a sequence number of "N00103".
- The product count command, M56, counts the number of products only during continuous automatic operation.
  Include the M56 command in either of the axis control group $1 and $2 programs.
  Note that if the M56 command is included in both of the $1 and $2 programs, two products are counted in one cycle.
- G999 must be included in each of the axis control group programs as the last program execution command. The machining pattern specified immediately after G999 is performed after queuing of each axis control group operation.
  The automatic queuing is performed by G999 specified in each axis control group program.
- N999 must be followed by the last program.
- Each axis control group program must end up with the one cycle stop command (M02) and the stop code (%).
  (Stop code is automatically entered after normal program entering.)

5.1.1 Program structure chart

- This machine allows you to program any machining process for execution after defining the axis control group ($1 or $2). Therefore, you can make your program vary depending on machining pattern.
- The cut-off process shall be programmed in a program of the axis control group 1.
- After G999 (last program command), enter the task you want to execute last in either axis control group. The task will be executed and the machining operation ended by N999 ($1 and $2).
- After N999, be sure to enter the fixed pattern.

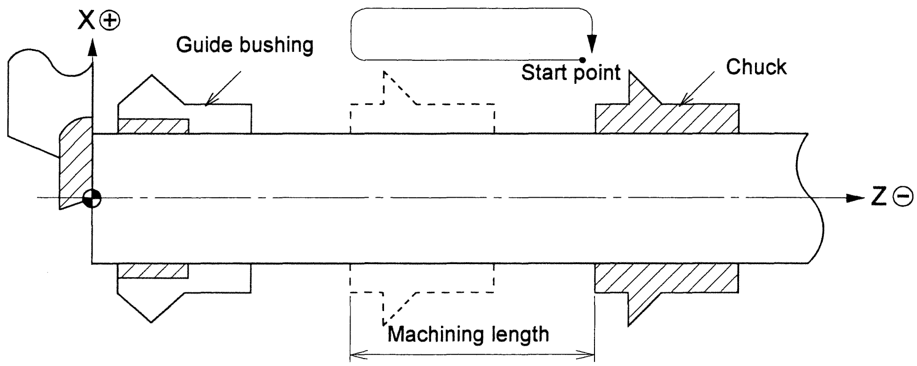


5.1.2 Guidelines for Designing a Machining Program

- Be sure to enter the center hole drilling operations such as center drilling, drilling, tapping, and boring before outer diameter cutting.
- Cutting of the outer diameter is usually done at one time. If there is groove in the outer diameter cutting portion, cut each groove individually with a grooving tool then resume the outer diameter cutting.
- A cycle is ended by cutting a workpiece with the cut-off tool.
  When your workpiece requires back machining, a cycle does not necessarily end with the cutting of the workpiece with the cut-off tool since the process varies depending on the machining pattern and designed machining sequence.
- The cut-off tool must be mounted in T01 for cut-off machining.

5.2 Preparation Processes

The preparation process sets up conditions needed for the cycle and starts the machine.



Description

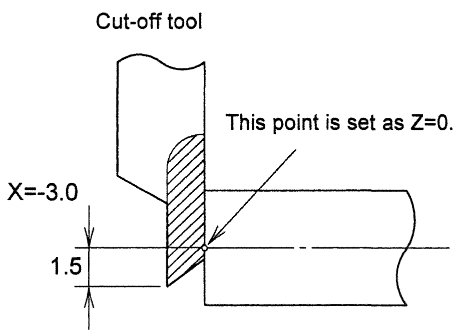
- As shown above, the program starts when the preceding cut-off process ends and the Z axis returns to its home position.
- Give a program number at the beginning of the program.
- Set up conditions.

  Whether to specify feedrates in mm/min [inch/min] (G98)
  or mm/rev [inch/rev] (G99).
  Where to set the home point of the longitudinal (Z-axis) direction (usually at Z0)

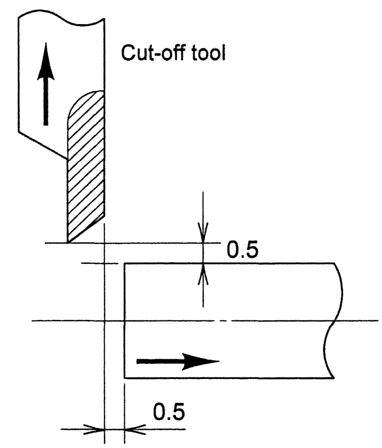
- Start the machine.
  1. Close the chuck.
  2. Move the material –0.5 mm [0.02"] in the longitudinal (Z axis) direction.
  3. Start the main spindle.
- If the back machining is required, additionally set up conditions for back spindle.

Sample program

G50 Z0



G00 X□ Z–0.5



Description

- Set the machining start position in the diametrical direction in advance so that the cut-off tool is positioned 1.5 mm down from the center at the start of the program, or at the position of X=–3.0 in diameter.
  (The X-axis value depends on the cut-off machining end position set as machining data.)
- In the same way, set the start position in the longitudinal direction so that the distance for longitudinal move required for the machining can be kept.
  (The value of Z axis varies depending on the machining length specified in the machining data.)
- After positioning the X and Z axes, bring the material in contact with the cut-off tool and start the program.
- Move the material backward and move the cut-off tool to the position of the material diameter + 1 mm. (The position changes depending on the machining data.)

Sample program

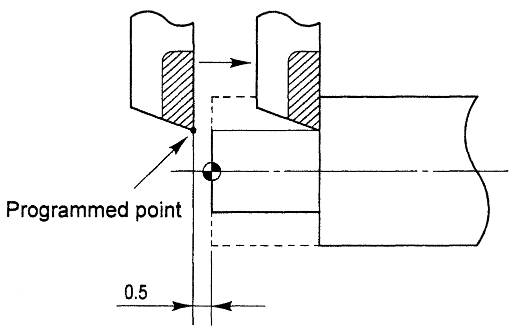
O□□□□ .... Specify the program number.

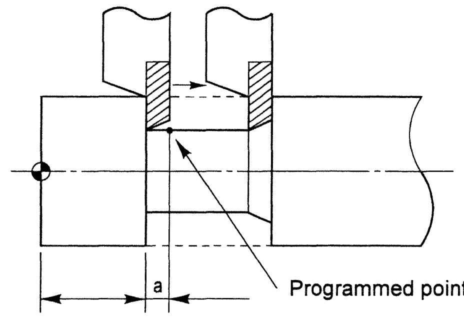
$1	$2	
		.... Specify each axis control group number.
G50 Z□		.... Set the coordinate system of the current position (machining start position) for each axis control group.
M06		.... Close the chuck.
M09		.... Material change OFF
G00 X□ Z–0.5		.... Move the cut-off tool. Generally, specify –0.5 mm in the longitudinal direction.
M3 S1=□ G99		.... Specify the main spindle (Z1-axis side) speed and forward rotation. The back spindle (Z2-axis side) command is M23 S2=□. Specify the per rotation feed. To specify the per minute feed, enter G98.

↓ Machining process

5.3 Machining Process

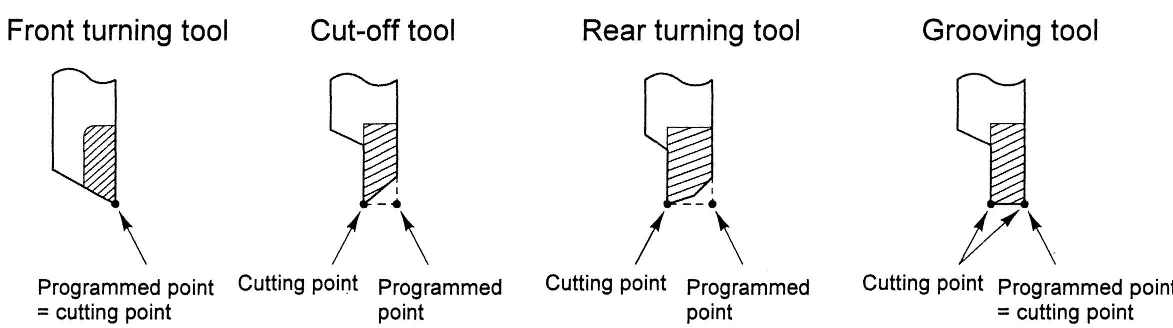
Outer diameter cutting includes front turning, rear turning, and grooving processes.





Description

- Position the front turning tool as shown above. Specify a travel through the machining length.
- As with the cut-off tool, consider the cutting tool width of the rear turning tool. Since the actual cutting point is to the left of the programmed point, the specified coordinate must be $Z=\ell$ plus cutting tool width a; otherwise, rear turning would be too short.
  Therefore, adjust the programmed point to $Z=\ell+a$ if the desired cutting point is represented by $Z=\ell$.
- Take care also of the grooving tool cutting tool width.

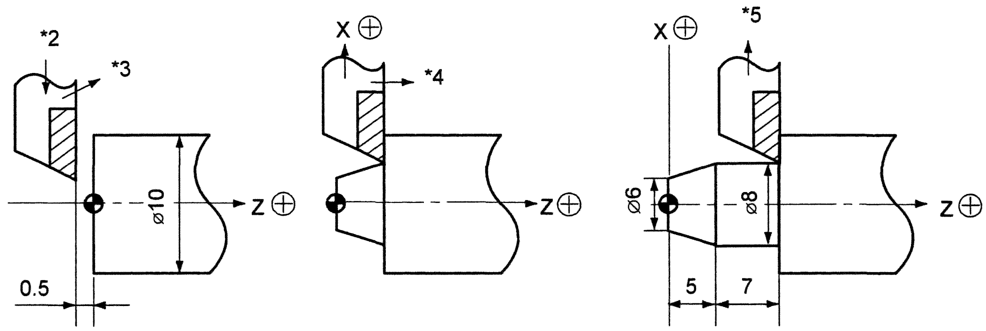


Notes

- When using rear turning and grooving tools, add the cutting tool width to the longitudinal coordinate value. It is a good idea to write down cutting tool widths in the tool layout chart.
- At the end of outer diameter cutting, be sure to move the tool to the waiting point away from the workpiece periphery.

5.3.1 Front turning process

Outline of machining

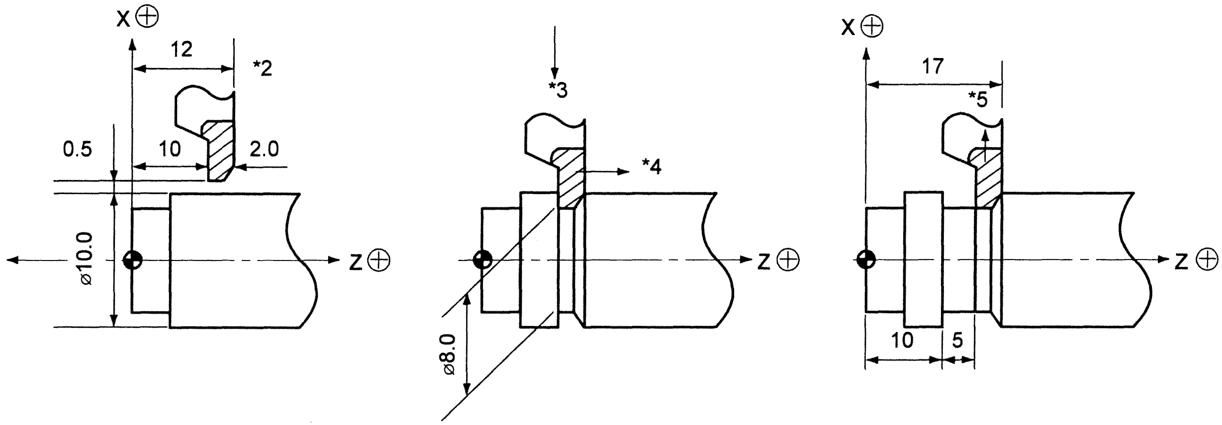


Sample program

*1	`T□□□□`	Select a front turning tool T□.
*2	`G00 X5.8 Z-0.5`	Position the tool at the specified point.
*3	`G01 X8.0 Z5.0 F□`	Cut the workpiece to a taper of 5.0 mm in the longitudinal direction.
*4	`Z12.0`	Cut the workpiece 12.0 mm in the longitudinal direction.
*5	`X11.0 F□`	Move the tool to the waiting point at the cutting feedrate.

5.3.2 Rear turning process

Outline of machining



Sample program

*1	`T□□□□`	Select a rear turning tool T[ ].
*2	`G00 X11.0 Z12.0`	Position the tool at 12 mm = 10 mm (longitudinal dimension) + 2 mm (cutting tool width).
*3	`G01 X8.0 F[    ]`	Make an in-feed to a diameter of 8 mm.
*4	`Z17.0 F[    ]`	Cut the workpiece to a longitudinal dimension of 17 mm.
*5	`X11.0 F[    ]`	Move the tool to the waiting point at the cutting feedrate.

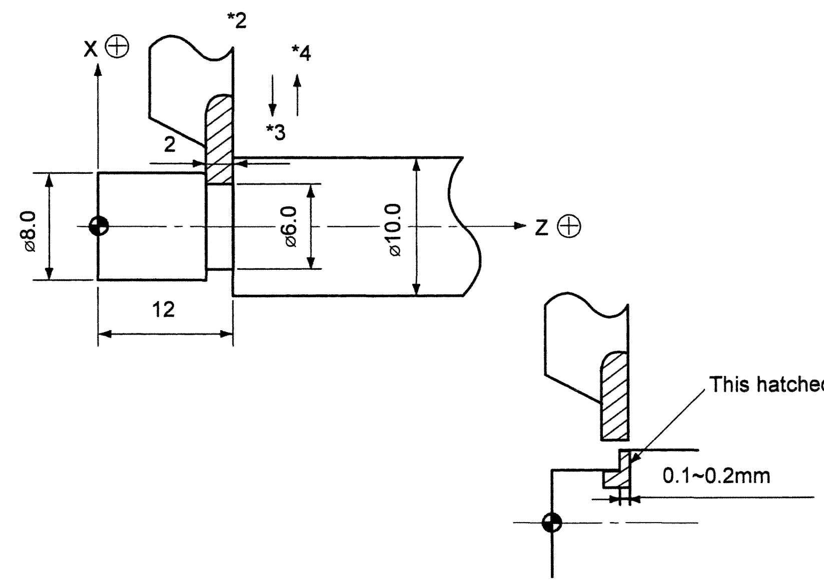
Example of coordinate system shift

*1	`T□□□□`	Select a rear turning tool T[ ].
	`G50 W–2.0`	Specify a rear turning tool coordinate system shift in the longitudinal direction. This command does not move the machine in the longitudinal direction.
*2	`G00 X11.0 Z10.0`	
*3	`G01 X8.0 F[    ]`	
*4	`Z15.0 F[    ]`	
*5	`X11.0 F[    ]`	
	`G50 W2.0`	Cancel the coordinate system shift in the longitudinal direction.

5.3.3 Grooving process

Grooving is generally performed using a grooving tool after outer diameter cutting.

Outline of machining



If grooving is to be performed after front turning, prior machining to a longitudinal dimension of no more than 0.1 to 0.2 mm should be performed at the time of front turning. In the grooving process, use the grooving tool to groove and finish the workpiece to the exact longitudinal dimension. With this method, correct longitudinal dimensions can be easily achieved.

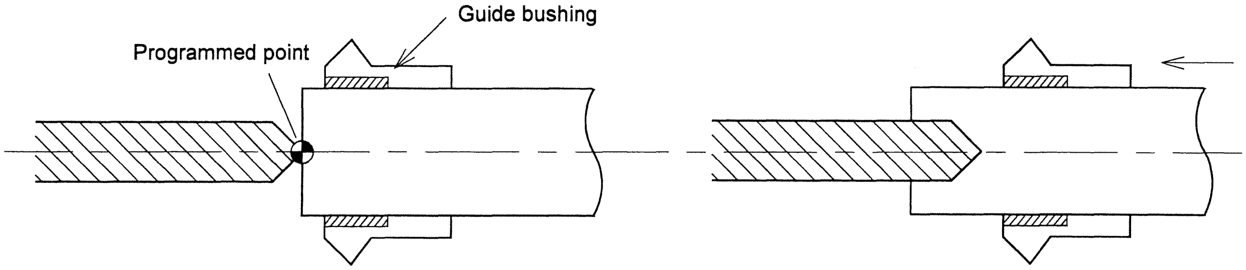
Sample program

*1	`T□□□□`	Select a grooving tool T□.
*2	`G00 X11.0 Z12.0`	Position the tool at 12 mm = 10 mm (longitudinal dimension) + 2 mm (cutting tool width).
*3	`G01 X6.0 F□`	Make an in-feed to a diameter of 6.0 mm.
*4	`X11.0 F□`	Move the tool to the waiting point at the cutting feed rate.

5.3.4 Drilling process

The center of the sleeve holder is positioned to the center of material by T□□□□ command.

Outline of machining

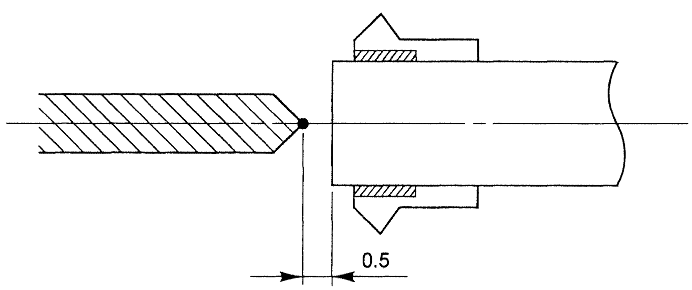


Sample program

```
T1200
G0 Z-0.5
G99 G1 Z4.0 F0.05 T12
G0 Z-1.0 T0
 :
```

As shown in the figure above, the drill contacts the material end face at the position Z=0.

Therefore, usually move the workpiece backward to Z=-0.5, then select the tool.



Note

To replace a tool after drill machining, be sure to move the drilling tool from the material end face.

5.4 Center Hole Machining

5.4.1 Spindle speed

When using a center drill or a drill, calculate the spindle speed (N) in min^{-1} using the following formula:

$$N = \frac{V}{\pi d} \times 1{,}000$$

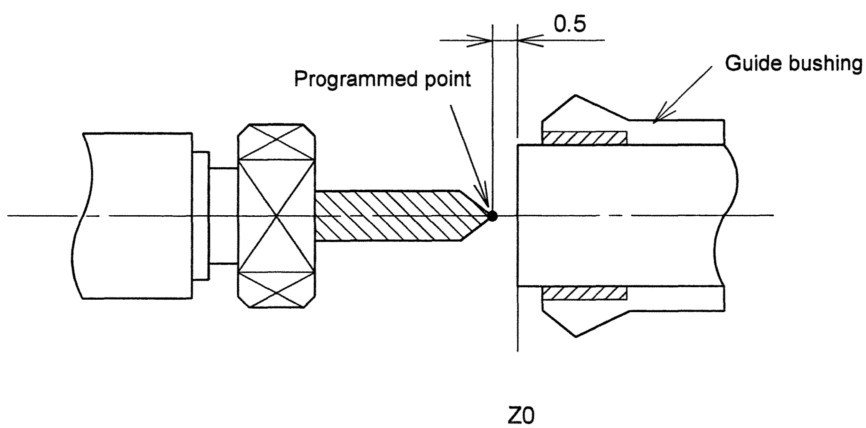
N: Spindle Speed (min^{-1})
V: Cutting speed (m/min)
d: Drill diameter (mm)
(maximum diameter of machined taper using a center drill)
π: Circular constant (approx. 3.14)

5.4.2 Programmed point and cutting point (positioning)

The programmed point of the center drill or drill is positioned in relation to the holder as follows:

For example, position the drill at the workpiece center through a tool change (selection) command as follows:

Outline of machining

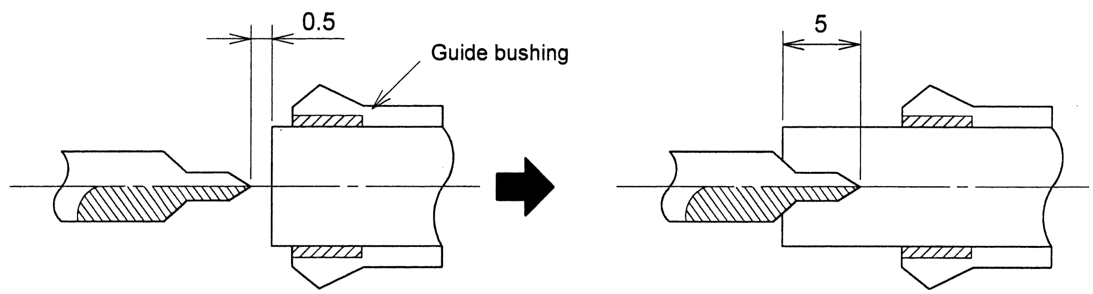


Sample program

T☐ .... Select tool T☐.
G00 Z–0.5 .... Move the end face of material by 0.5 mm.
:
: } Machining process

5.4.3 Center machining

Outline of machining



Sample program

$1		$2	
M3 S1=☐	Rotate the spindle in the forward direction.		
G600	Free pattern (machining pattern cancel)	G600	Free pattern (machining pattern cancel)
T☐	Select tool T☐.		
G00 Z–0.5	Move the end face of the material by 0.5 mm.		
G01 Z5.0 F☐ T☐	Perform the center machining of 5 mm in depth, and input the offset.		
G00 Z–0.5 T00	Return to the former position in rapid feed, and cancel the offset.		

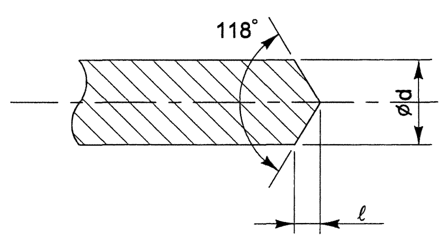
5.5 Deep Hole Drilling

Number and amounts of in-feeds

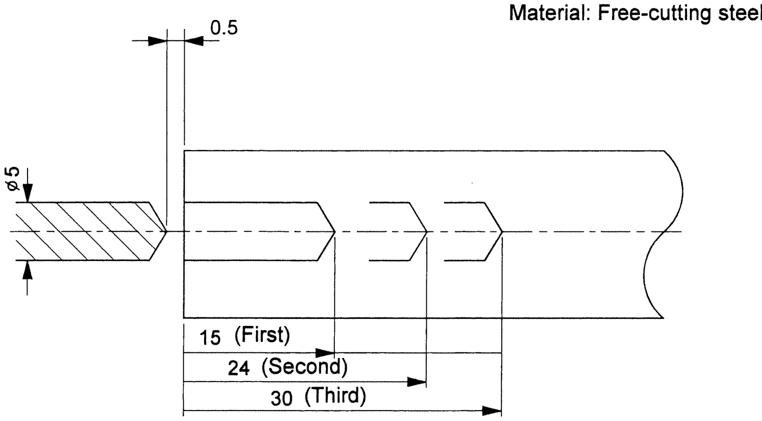
Material \ In-feed: Tool		ℓn (mm): First	
Material	Tool	First	Second and subsequent
Brass	Woodruff drill, flat drill	Whole length	
Aluminum, free-cutting steel	Twisted drill	ℓ_1=ød×3.0	$\ell_n=\ell_{n-1}$ × 0.6 (mm) ød × 1 (mm) Minimum in-feed ød × 1 (mm)
Tool steel, carbonate-steel, free-cutting stainless steel	Twisted drill	ℓ_1=ød×2.5	$\ell_n=\ell_{n-1}$× 0.5 (mm) ød × 0.9 (mm) Minimum in-feed ød × 0.9 (mm)
Stainless steel, hard-cutting material	Twisted drill	ℓ_1=ød×2.0	$\ell_n=\ell_{n-1}$ × 0.4 (mm) ød × 0.8 (mm) Minimum in-feed ød × 0.8 (mm)

Notes

- Before drilling, be sure to perform center hole machining.
  In the case of deep hole machining, as the drill goes deeper, coolant application and cuttings removal become less satisfactory.
  Scuffing on the drill as caused by the material could cause a defect on the drill.
  Therefore, use several, rather than a single, in-feeds. This is called stepping.
  See the table above for the number and amounts of in-feeds.
- The drilling depth must be the effective length plus a dimension determined considering the cutting edge length.
  Cutting edge length ℓ=0.3×ød



Outline of machining



Example of in-feed calculation

First $\ell_1 = ø5 \times 3 = 15.0$

Second $\ell_2 = \ell_{2-1} \times 0.6 = 9.0$

Third $\ell_3 = \ell_{3-1} \times 0.6 = 5.4 \approx 6.0$

$\ell_1 + \ell_2 + \ell_3 = 30.0$

Sample program

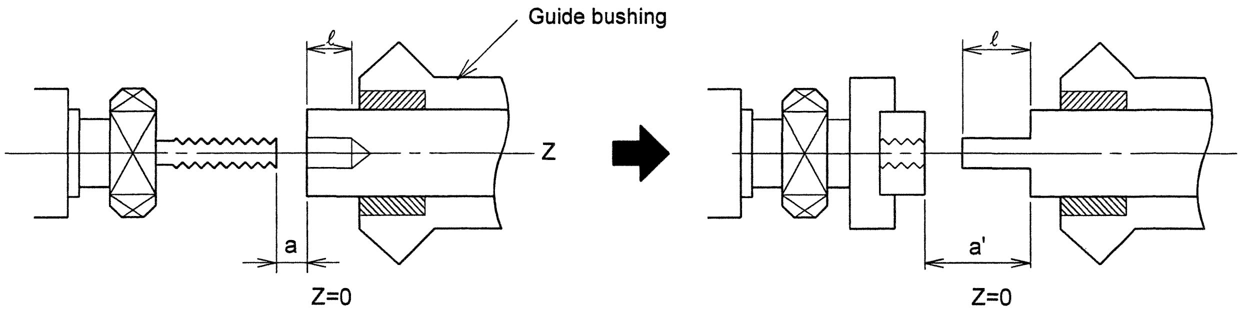
Machining process

$1		$2
M3 S1=☐	Rotate the spindle in the forward direction.	
T☐	Select tool T☐.	
G00 Z–0.5	Move the end face of the material by 0.5 mm.	
G01 Z15.0 F0.1 T☐	First in-feed and the offset input	
G00 Z–0.5	Return the drill to the former position.	
G04 U0.5	Dwell (0.5 seconds)	
Z14.5	Position the machine for the second in-feed.	
G01 Z24.0 F0.1	Second in-feed	
G00 Z–0.5	Return the drill to the former position.	
G04 U0.5	Dwell (0.5 seconds)	
Z23.5	Position the machine for the third in-feed.	
G01 Z30.0 F0.1	Third in-feed	
G00 Z–0.5 T00	Return the drill to the former position. Cancel tool offset.	

5.6 Machining with Tap and Die

Outline of machining

Position the tap and die as shown below before starting machining.



The shift amount " a' " for the die is set at "ℓ+1mm".
(ℓ: Effective length of thread)

Machining with a tap

Machining with a die

Description

- Usually, the shift amount "a" set when the tap is mounted on the holder is to be about 3 mm. If the tap may interfere with the workpiece or gang tools, set the tool with a shift amount of 10 to 30 mm.
  The shift amount must be considered in the program.
- The standard rotation speed for the tap and die machining is 300 to 1,000 min^{-1}.
  Following table is approximate criteria for spindle speeds.

Material	Nominal Diameter	Spindle Speed (min^{-1})
Non-ferrous materials	M1 to M3	600 to 1000
	M4 to M6	400 to 800
	M7 to M10	200 to 600
Ferrous materials	M1 to M3	400 to 800
	M4 to M6	200 to 600
	M7 to	to 400

- The drilling hole for tapping must be as deeper as possible. Normally, drill the hole deeper than the thread length, by the length three times the lead or by the length equal to the tap nominal diameter.
- The dwell time before the tap is extracted is calculated using the following formula:

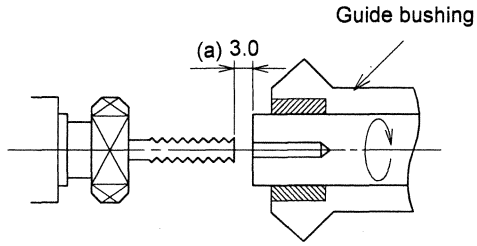
$$t \geq \frac{0.2 \times \ell}{L \times N/60} \times 1000$$

L: Lead (mm)
N: Spindle speed (min^{-1})
ℓ: Effective length of thread (mm)
t: Dwell time (s)

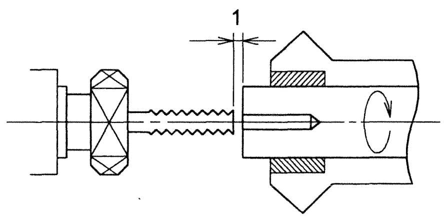
5.7 Tapping

(Machining for M6 P = 1.0)

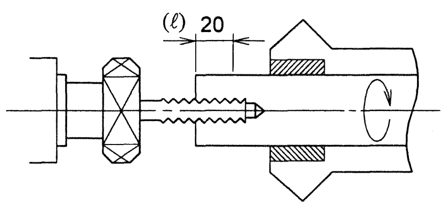
Outline of machining | **Sample program**



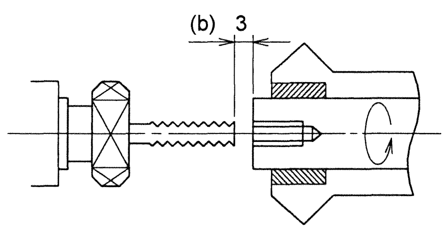
`M03 S1=300`	Specifies a spindle speed of forward rotation.
`G04 U0.5`	Makes the machine dwell for 0.5 second so that stable rotation will be reached.
`T1300`	Selects tool T13.



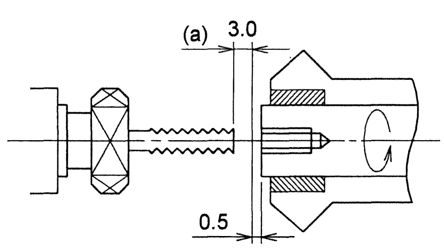
`G00 Z2.0 T□□`	Positions the tap 1 mm from the workpiece end face rapidly (shift a–1.0).



`G32 Z19.0 F0.8`	Advances the tap at the feed rate for pitch P × 0.8. The feed is 80% of the threaded length ℓ. (Z=ℓ×0.8+a, F=P×0.8)



`Z0 F1.0 M04`	Retracts the tap to the position of 3 mm (b) from the workpiece end face at the feed rate, giving a reverse rotation command and a speed at the same time. (Offset cancel) (Z=a–3.0)



`G04 U2.0`	Waits until the tap is removed. (Usually, specify U2.0.)
`G00 Z-0.5 T00`	Rapidly retracts the tap to the position shown at left to prepare for the next process.
`M03 S1=` $\boxed{\quad}$	Specifies a spindle speed of forward rotation for the next process.

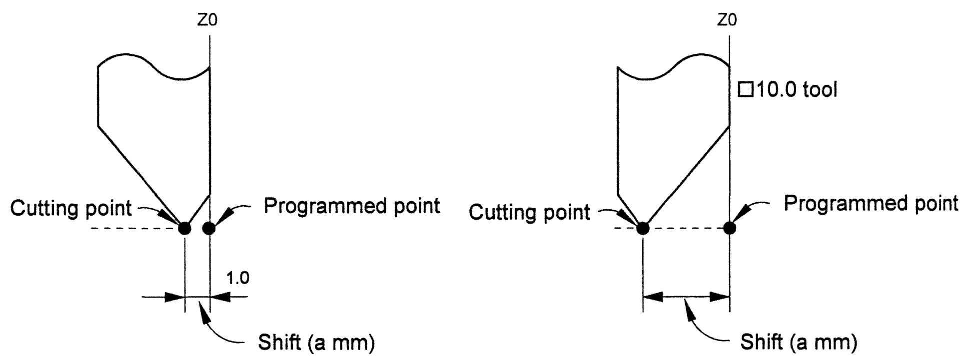
5.8 Threading with Tool

Threading is generally performed in a G92 threading canned cycle. However, special threading can also be performed by a command in a G32 block. Only threading with G92 is explained here.

5.8.1 Programmed point and cutting point

Threading tools are right-handed or left-handed. The two types should be used for different purposes. Their shifts are also different.

Generally, use a right-handed threading tool for front turning and a left-handed threading tool for rear turning.

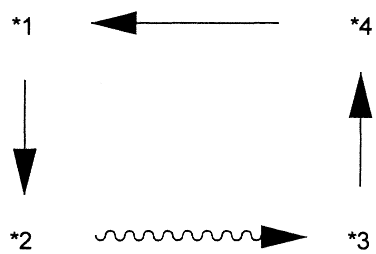


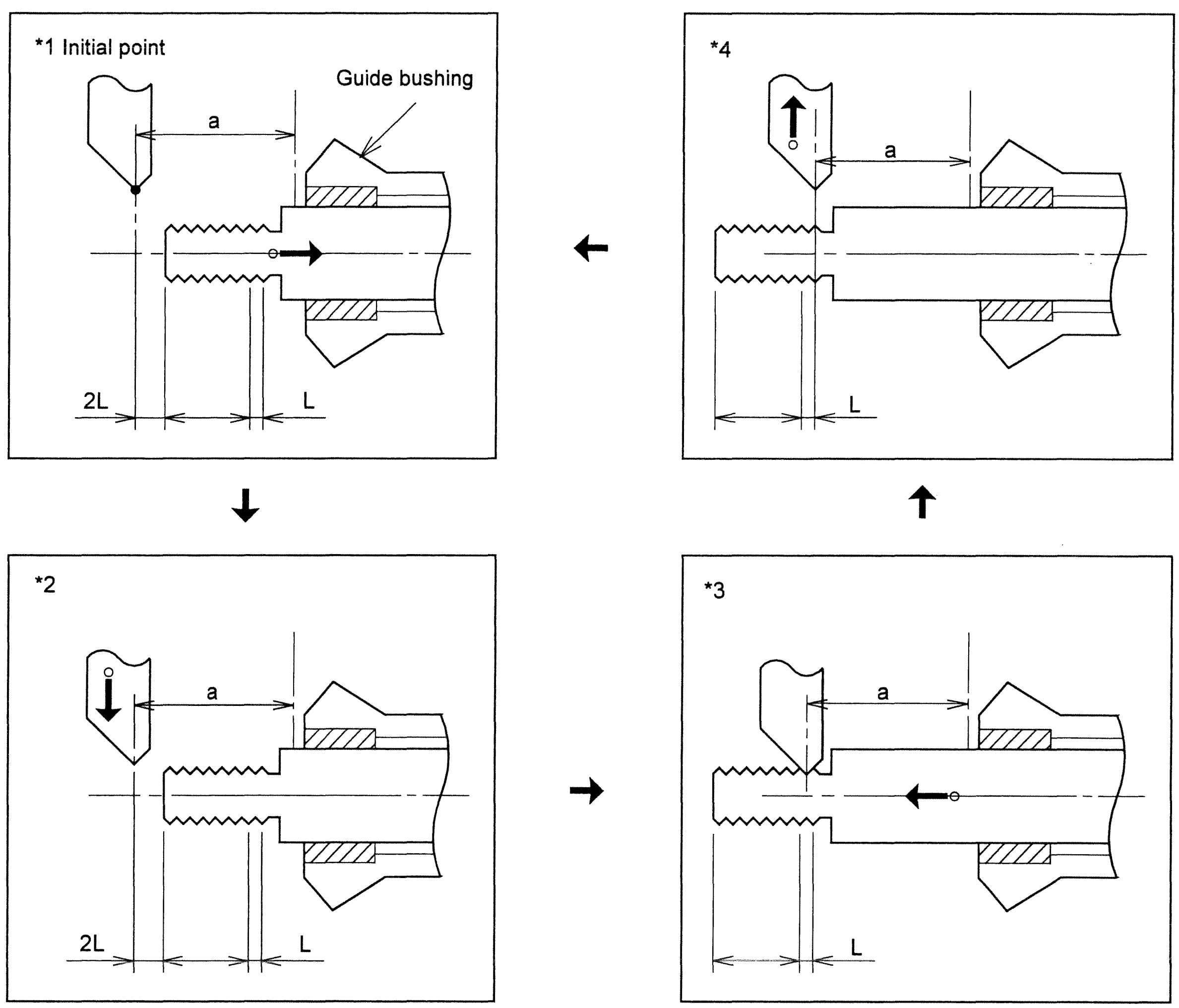
Write a program with a shift of the programmed point + a.

5.8.2 G92 threading canned cycle

Perform threading with a tool in the following cycle:

Outline of machining





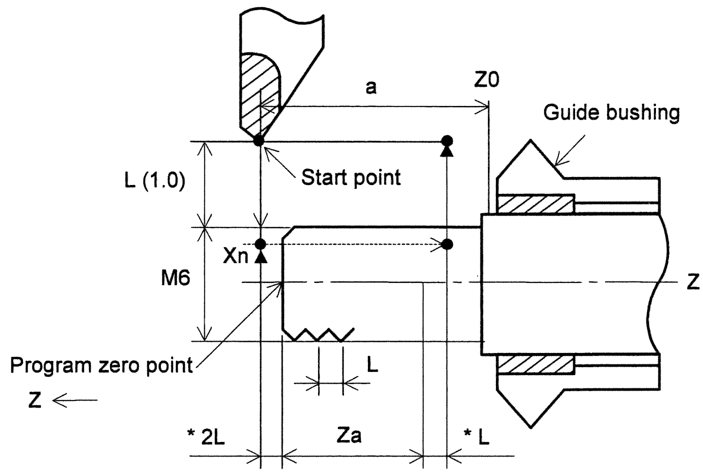
5.8.3 Threading with G92

Machining sample

M6P=1.0 Threading (right-handed) on nonferrous material

Outline of machining

G92X□ Z□ F□



Thread lead: L (1.0 mm)
Effective length: Za (4.0 mm)
Tool shift: a (7.0 mm)
In-feed: Xn
* Section with unequal crest
Values in parentheses are programming examples.

Sample program

M03 S1=1500	Specifies forward spindle rotation at 1500 min^{-1}.
T0300	Selects the T03 tool.
G04 U1.0	Makes the machine dwell for 1.0 second until stable rotation will be reached.
G00 X8.0 Z5.0 T□	Positions the tool bit at the initial point (Z=a–2L).
G92 X5.5 Z12.0 F1.0	First in-feed (X_1 0.25) (Z=a+Za+L) (L)
X5.06	Second in-feed (X_2 0.22)
X4.8	Third in-feed (X_3 0.13)
X4.7	Fourth in-feed (X_4 0.05)
X4.7	Zero cut (same as the last in-feed)
G00 X□ Z□	Moves the machine to the initial point for the next process, and cancels threading canned cycle.

Notes

- For in-feeds, see the table in <Section 10.2 Thread Cutting Count with Tool>.
- If the G92 command contains EOB code (;), the program runs with the modal of the preceding block.

```
G92 X11.0 Z□ F□;
X10.0;
X9.0;
; (Threading at X9.0)
; (Threading at X9.0)
G0 □;
```

Description

- The spindle speed is limited as follows:

$$N(\text{min}^{-1}) \leq \frac{12{,}000\ (\text{mm/min})}{L\ (\text{mm})}$$

N:	Spindle speed
L:	Thread lead
12,000:	Maximum feedrate

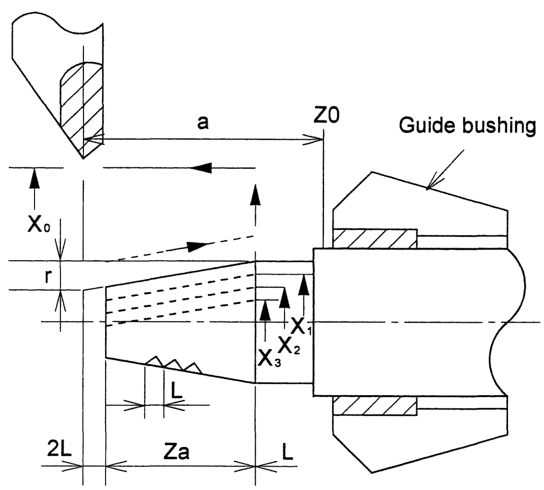
- The spindle speed for general threading is 500 to 2000 min^{-1}.
- At the beginning and end of threading, unequal crests are generated because of lags involved in the servo system. This inequality can be reduced by lowering the spindle speed on the assumption that the lead at the spindle speed of 1500 min^{-1} is about 2L at the beginning and is about L at the end.
- Select right-handed or left-handed threading by selecting the correct Z-axis start position or selecting the correct direction of spindle rotation (and using the forward or reverse rotation tool bit).
- The thread can be chamfered (to make overcuts on thread surface) in 0.1L steps in the range from 0.lL to 12.7L at the pitch of 0.1 by specifying a parameter (data No. 5130). See <the FANUC Operator's Manual>.
- Chamfering on/off can be switched at No. 6 of Operator's panel screen, or by entering M codes M204 and M205.

5.8.4 G92 taper threading canned cycle

The following taper threading can be performed:

Outline of machining

`G92 X□ Z□ R r F□`



Thread lead: L
Effective length: Za
Tool shift: a
Gradient: r
* Section with unequal crest

Taper threading is also performed in a cycle of *1→*2 to *3→*4→*1. However, the coordinate reference point at step *3 is always assumed.

The sign of the gradient r indicates the direction from the end point to the start point.

Sample program

`M□ S□`	Specifies spindle rotation in the forward or reverse direction at □ min^{-1}.
`T□`	Selects a threading tool.
`G04 U1.0`	Makes the machine dwell for 1.0 second so that stable rotation will be reached.
`G00 X X0 Z□ T□`	Positions the tool to the initial point. X (thread diameter + 2L), Z (a–2L)
`G92 X X1 Z□ R -r F`	First in-feed X (thread diameter –2 × in-feed$_1$) Z (a+Za+L), R (–gradient), F(L)
`X(X2)`	Second in-feed X (X_1–2 × in-feed$_2$)
`X(X3)`	Third in-feed X (X_2–2 × in-feed$_3$)
`X(X3)`	Zero cut with diameter X_3
`G00 X□ Z□`	Moves the tool to the initial point for the next process (threading cycle off).

Notes

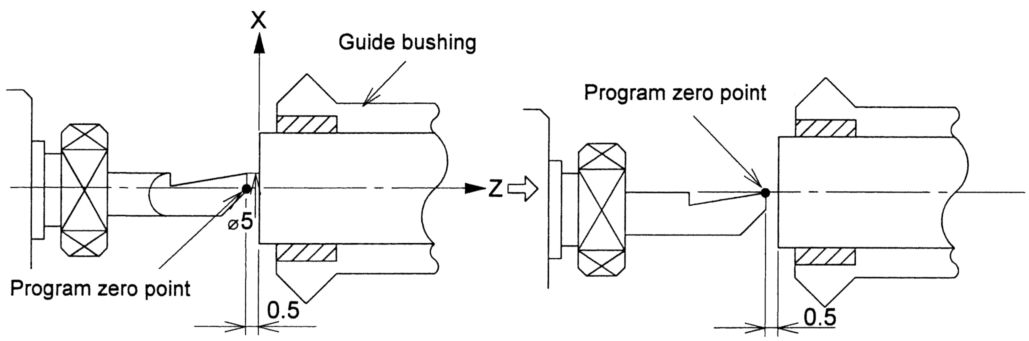
- For in-feeds, see the table in <Section 10.2 Thread Cutting Count with Tool>.
- If the G92 command contains EOB code (;), the program runs with the modal of the preceding block.

```
G92 X11.0 Z□ F□;
X10.0;
X9.0;
; (Treading at X9.0)
; (Treading at X9.0)
G0 □;
```

5.9 Boring (Coordinate System Shift)

The center line of the boring tool corresponds to the center line of the nominal cutting diameter. This means that the actual cutting point is deviated by the tool radius. The program must contain a scheme to eliminate this deviation.

Outline of machining



Description

Position the boring tool as shown above, then shift the tool by the cutting tool radius. Then shift the tool in diameter direction. This compensates for the deviation equal to the cutting tool radius, thus facilitates coordinate value calculations. For this purpose, use the following commands:

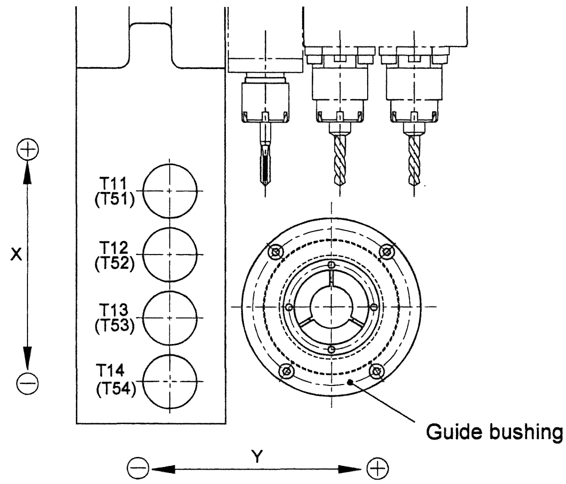
```
T□□□□
G50 U[    ]
G00 X□ Z□ T□□
```

Note

Move the tool away from the end face of the material after completing the boring.
Specifying a tool selecting command without moving the tool away from the end face causes an interference between the material and the tool. It is dangerous.

5.9.1 Boring Tool Signs

The signs of coordinates when using the boring tool are as shown in the figure below.



Sleeve holder viewed from operator side

5.9.2 Diameter direction coordinate system shift and shift cancel

Coordinate system shift `G50 U5.0` (if the cutting tool diameter is ⌀5)

For boring, insert a coordinate system shift command to facilitate coordinate value calculations. For this purpose, include a G50 coordinate system setup command in the program as shown above: The value 5.0 following the address U comes from the diameter (⌀5 mm) of the boring tool. Therefore, if a 6 mm diameter boring tool is used, the coordinate system shift command must be G50U6.0.

Coordinate system shift cancel `G50 U–5.0` (if the cutting tool diameter is ⌀5 mm)

Be sure to cancel the coordinate system shift when the boring tool have moved to the position far away from the workpiece end face as shown above:

The value –5.0 indicates the shift of 5.0 is to be canceled.

The above shift and cancel are in the diameter direction which is used for boring and other machining. In addition, there are longitudinal coordinate system shifts and cancels which are used for rear turning and cut-off tools.

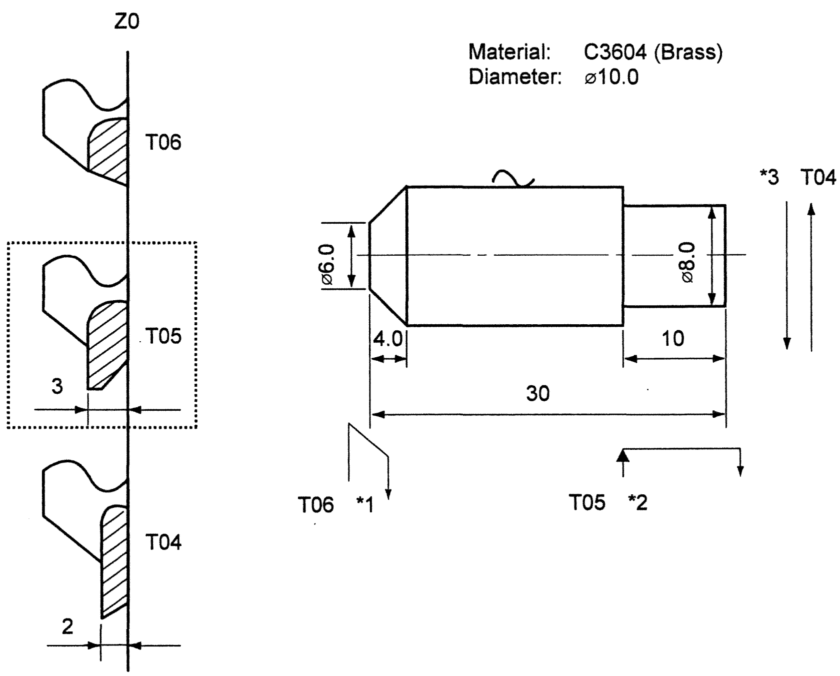
Coordinate system shifts and cancels for rear turning are explained in the next subsection.

5.9.3 Longitudinal coordinate system shift and shift cancel

Since machining with a rear turning tool bit involves a shift of the cutting point in relation to the programmed point, the addition of the cutting tool width in the longitudinal (Z axis) direction is required in the program.

The programming explained below allows the entry of dimensions on the drawing without the addition of the cutting tool width in the machining program.

Outline of machining



Coordinate system shift `G50 W-3.0` (if the cutting tool width is 3 mm)

Coordinate system shift cancel `G50 W3.0` (if the cutting tool width is 3 mm)

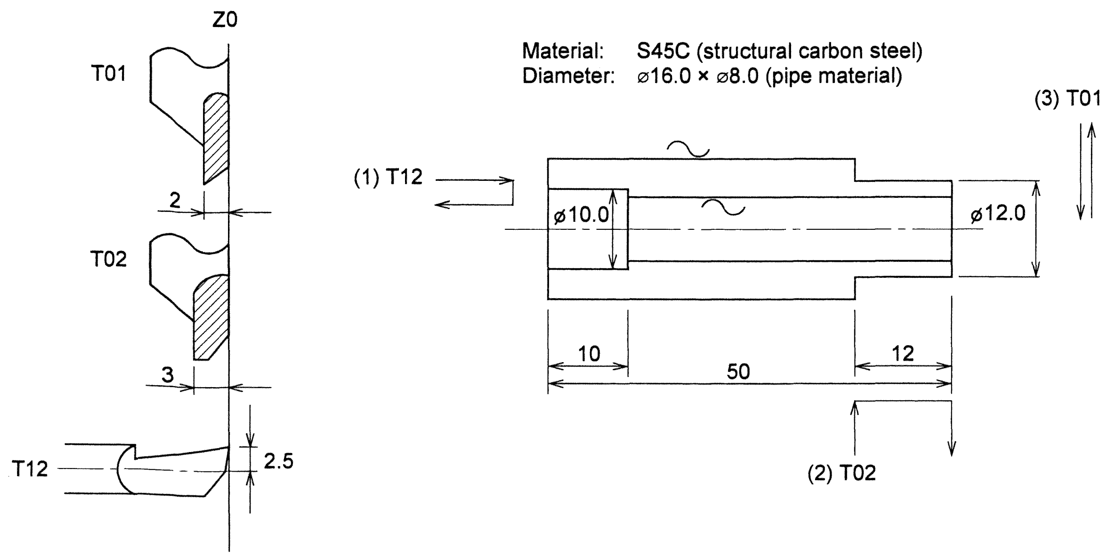
Idea: The sign is determined according to whether the programmed point is positive or negative with respect to the cutting point.

Sample program

`G50 W-3.0`	Specifies a rear turning tool coordinate system shift in the longitudinal direction.
`G00 X11.0 Z20.0`	Positions the machine rapidly (to the point calculated with the shift included).
`G01 X8.0 F☐`	Cuts the workpiece in the diameter direction.
`Z30.5 F☐`	Cuts the workpiece to 30 mm + 0.5 mm in the longitudinal direction.
`X11.0 F0.2`	Moves the tool away from the workpiece periphery.
`G50 W3.0`	Cancels the coordinate system shift. This command does not move the machine in the longitudinal direction, but will do when the next Z command is encountered.

5.9.4 Diameter/longitudinal direction coordinate system shift and shift cancel

Outline of machining



Sample program

This program example does not contain product separation.
For product separation, see the program example in the section covering program separation.

Preparation process

O[ ]	
$1	$2
G50 Z[ ]	
M06	
G00 X17.0 Z–0.5	G999
M03 S1=[ ] G99	N999
	M02
	%

Boring process

```
N0122 T1200
G50 U5.0 .................................. Boring coordinate system shift (direction of diameter)
G00 X10.0 Z–0.5 T[ ]
G01 Z10.0 F[ ]
X7.0 F[ ]
G00 Z–0.5
G50 U–5.0 ................................. Boring coordinate system shift cancel (direction of diameter)
```

Rear turning process

```
N0202 T0200
G50 W-3.0 ............................... Rear turning coordinate system shift (longitudinal direction).
G00 X17.0 Z38.0 T[    ]
G01 X12.0 F[    ]
Z50.1 F[    ]
X17.0 F0.2
G50 W3.0 ................................ Rear turning coordinate system shift cancel (longitudinal direction).
```

Cut-off process

```
N0304 T0100
G00 X17.0 Z52.0 T[    ]
G01 X13.0 F0.2
X7.0 F[    ]
X-3.0 F[    ] ......................... This can be rapid feed because pipe material cut-off operation has been
                                        completed.
M05
M07
G00 Z0 T00
M56
G999
N999
M02
%
```

Notes

- When T11's (front drilling) is called in the program, the coordinate system shift in the diameter direction is automatically cancelled.
  Note that the coordinate system shift in the longitudinal direction remains modal.
- When T31's or T51's (back drilling) is called in the program, the coordinate system shifts both in the diameter and longitudinal directions are automatically cancelled.
- Note that, if one or more of T01 to T09 (gang and rotary tools) are called in the program, the coordinate system shift in the longitudinal direction remains modal.

Product Code	C	–	A	2	2	0	P	L	VII					
Document Code	2	E	1	–	0	5	0	1						

6. Sample Programming [Advanced]

6.1 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)
Tool selection and offset specification at the same time: 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 32. 33 through 49 are optional.

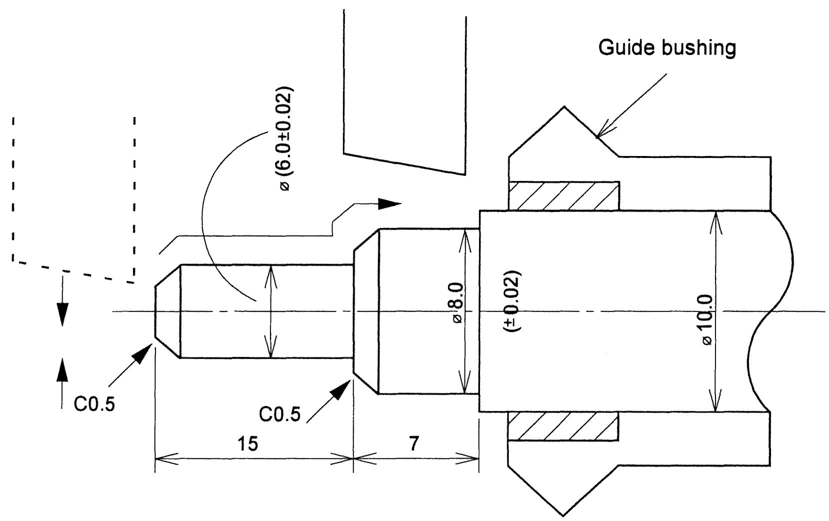
- When a T□□□□ (4-digit) command is given, the machine moves to the position calculated with the offset value included (only in the diameter direction).

  If a T□□ (2-digit) command is given, the offset will be in effect when the next X☐ and Z☐ command is given.

6.1.1 Examples of offset specification

Outer diameter machining

Outline of machining



Sample program

```
N0112 T0200
G00 X4.0 Z-0.5 T01 ...............
G01 X6.0 Z0.5 F0.03
Z15.0
X7.0 T02 ..................................
X8.0 Z15.5
Z22.0
```

- 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).
- 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)

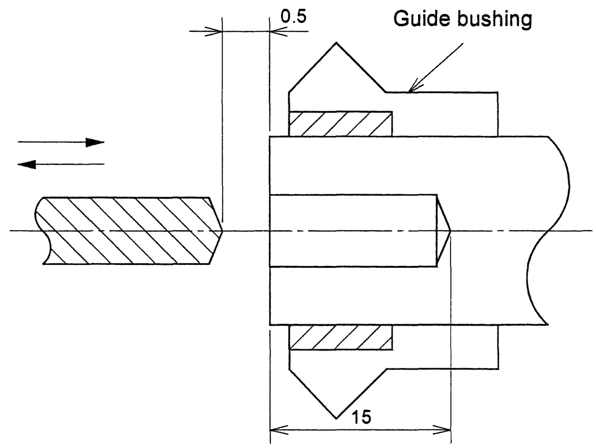
Note

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

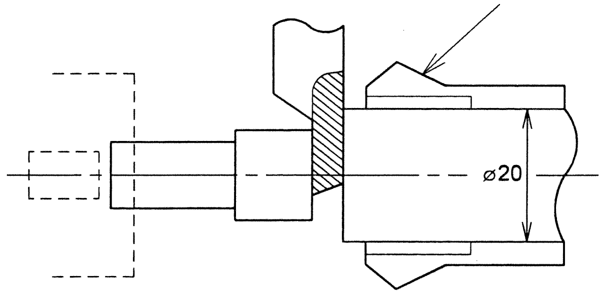


Sample program

```
N0323 T1300
G00 Z-0.5
G01 Z15.0 F0.08 [T13] ............. Specifies offset value to correct the drill depth.
G00 Z-0.5 [T00] ......................... Offset cancel (required at the time of return)
```

Cutting-off

Outline of machining



Sample program

```
N0401 T0100
G00 X21.0 Z22.0 [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 Z0 [T00] ....................... Offset cancel
                                     Always specify this command to return 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
%
```

6.2 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.

6.2.1 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☐ R☐ T☐`

`U:` Positioning point for deburring of residual material (in mm dia.). With no argument specified, the positioning point is 4.0 mm.

`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 800 min^{-1}, the spindle speed is clamped at 800 min^{-1}.

`W:` Movement of spindle at withdrawing the residual material and inserting the material (mm). With no argument specified, movement is 30 mm.

`F:` Feedrate at withdrawing the residual material and inserting the material (mm/rev). With no argument specified, the feedrate 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 3 seconds. Do not omit the decimal point.

`R:` Spindle rotation during material changing, undefined: stop, 1: rotate
Machining data: Feedrate at deburring of residual material (chamfering)
Feedrate at cutting the tip of new material (short-cut)
Rotation speed at deburring of residual material (chamfering)
Rotation speed at cutting the tip of new material (short-cut)

`T:` Dwell time after the coolant discharge is turned on (second). With no argument specified, the dwell time is 3 seconds. Do not omit the decimal point.

Sample program 1

All arguments necessary are input.

```
Preparation process
M09
↓
Machining process
↓
T0100 ........................................................ Cut-off tool
G00 X[    ] Z[    ] T[    ]
G650
!2 L1
G01X-3.0 F0.02
G600
M08
M08
/M108 U1.0 C150.0 D1.5 B3.0 S800 W30.0 F3000.0 A3.0 R1 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 tool moves to the position determined by (material outer diameter in the machining data + positioning point for deburring a bar remnant specified by U argument) at the feedrate specified by C argument (feedrate for positioning the deburring position). ...Fig. (b)
5. The spindle rotates at the specified by cut-off speed in machining data. If the B argument (move distance of X axis for deburring) or D argument (move distance of Z axis for deburring) is not specified, this step is skipped.
6. The main spindle (Z1 axis) advances the distance specified by D argument (move distance of Z axis for deburring) at the speed specified by C argument (feedrate for positioning the deburring position). If the B argument (move distance of X axis for deburring) or D argument (move distance of Z axis for deburring) is not specified, this step is skipped. ...Fig. (c)
7. The machine executes the chamfering function for the bar remnant at the Cut-Off Feed in the machining data, depending on the values of the D (move distance of Z axis for deburring) and B arguments (move distance of X axis for deburring). If the B argument (move distance of X axis for deburring) or D argument (move distance of Z axis for deburring) is not specified, this step is skipped. ...Fig. (d)
8. Specify M53 (Coolant OFF) command.
9. If the command to rotate the main spindle during exchange (without R argument) is not specified, the main spindle stops. If the command to rotate the main spindle during exchange (with R argument) is specified, the S argument (the speed at which the main spindle rotates when drawing the bar remnant and when supplying bars) is used as a spindle speed. If the S argument is omitted, the default (800 min^{-1}) is used. If the specified value for S argument exceeds 800, the speed is clamped at 800 min^{-1}.
10. The X1 axis retracts the distance specified by the B argument (move distance of X axis for deburring), and the Z1 axis retracts the distance specified by the W argument (distance by which the Z axis moves when drawing the bar remnant and when supplying bars), the bar remnant is drawn from the finger chuck at the feedrate specified by the F argument (feed rate at which the Z axis moves when drawing the bar remnant and when supplying bars). If the B argument (move distance of X axis for deburring) or D argument (move distance of Z axis for deburring) is not specified, only the Z1 axis retracts. ···Fig. (e)
11. Specify M54 (bar feeder stop) command.
12. The main spindle chuck opens (M7).
13. Specify the M55 (bar feeder start) command to exchange the bar stock.
    (Move the bar remnant to the retract edge of the bar feeder, pull out the remnant bar from the finger chuck, and insert the stock bar into the finger chuck. The bar feeder pushes the material to the position specified by the bar feeder.) The exchange completion signal is sent from the bar feeder to the machine (A220).
14. Close the main spindle chuck (M6).
15. Dwell time specified by the A argument (dwell after chuck).

16. Insert the material by the amount specified by the W argument (distance by which the Z axis moves when drawing the bar remnant and when supplying bars), and move the main spindle (Z1 axis) by the distance specified in CUT-OFF END of machining data at the feedrate specified by F argument (feed rate at which the Z axis moves when drawing the bar remnant and when supplying bars).
17. Turn the coolant ON (M52).
18. Dwells for the time specified by an argument T.
19. The spindle speed is changed according to the cutting-off speed specified in the machining data.
20. Cutting-off is performed until the cutting-off end point reaches at the feedrate specified in the machining data.
21. Load the material exchange program OFF (M9) program.
22. Terminates the Material Exchange (M108) command program.

Notes

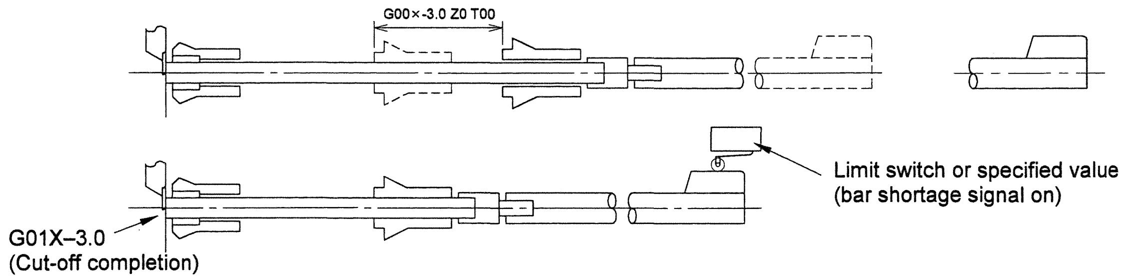
- Specify the M10 (Material Exchange) commandin $1. Specifying the M10 command in $2 causes an alarm to occur.
- If you specify M108 (Mateial Exchange) with the Sample program 1, whole steps 1 through 21 are executed.
- If you specify M108 (Mateial Exchange) with the Sample program 2, steps 5 through 7 and a part of step 10 are skipped.
- If you specify M108 (Mateial Exchange) with the Sample program 3, steps 5 through 7 and a part of step 10 are skipped. However, S, W, F, A, and R arguments are executed with the default value.
- Chamfering is not performed unless arguments D and B are not specified.
- If the specified value for S argument　If the specified value for the S argument (the speed at which the main spindle rotates when drawing the bar remnant and when supplying bars) exceeds 800^{-1}, the speed is clamped at 800 min^{-1}.

CAUTION

Be sure to use M108 or M109 code to create a material change program (see <6.2.2 General-purpose Material Change Program>). If the material change program is created by any other way, the machine may continue operation without changing material. It will be very dangerous.

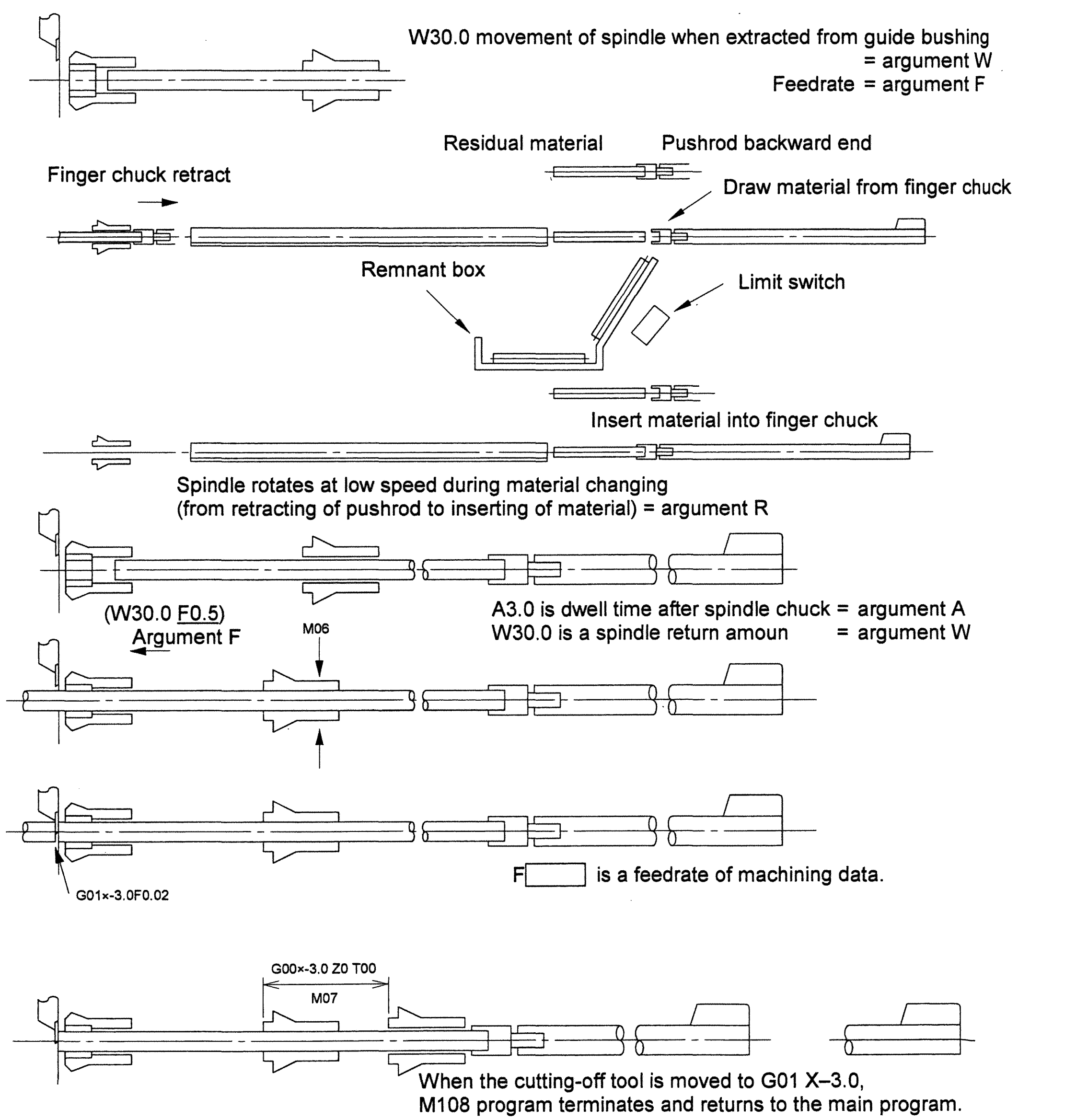
Material change flow

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.



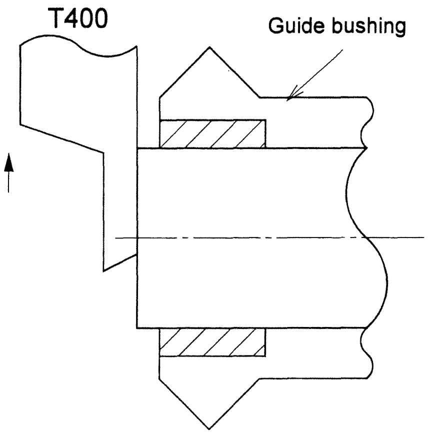
Perform chamfering of residual material (if necessary).
See (a) to (e) in the figure on the next page.

Chamfering is needed when the residual material is difficult to be withdrawn or may cause damage on the guide bushing due to burrs generated.



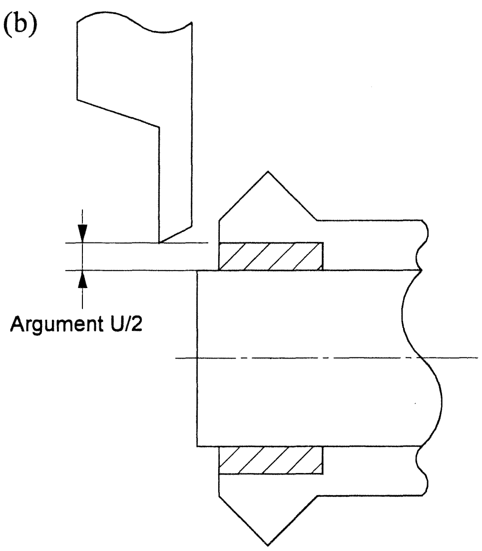
Chamfering of residual workpiece

(a)



- When the step 2. is 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)

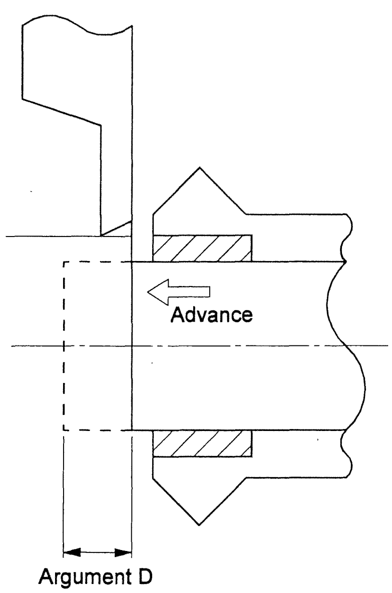


- The tool moves to the position of material outer diameter + value of argument U.

Note

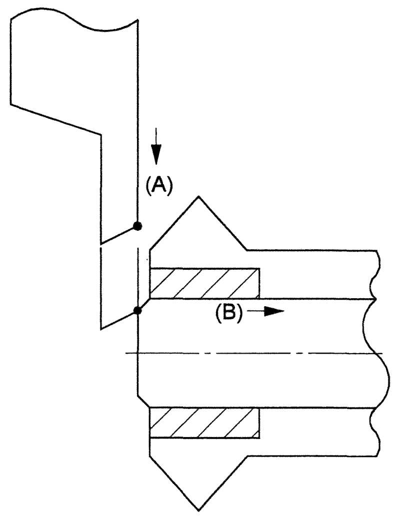
The position described above is not the tool positioning point in the machining data.
If 0 is specified for argument U, the next material is fed to the point where the tool tip is positioned. So care must be taken.

(c)



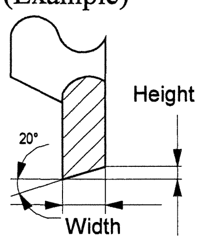
- Z axis advances for the amount specified by an argument D at the feedrate specified by an argument C. (The material comes out from the guide bushing.)
- Chamfering of residual material is ready.

(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)

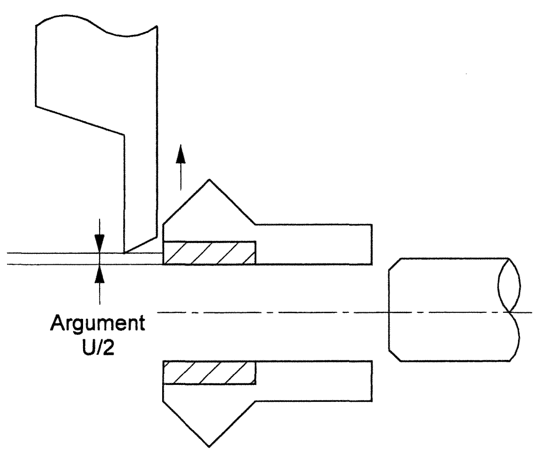


When an argument U = 1.0

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.

6.2.2 General-purpose Material Change Program (by user program)

M109 executes material change using the program, which is prepared by the user, as a subprogram. M109 can be used in material change program only.

Command format

```
M109 Q□□□□
```

Q: Material change program number (numerics only)

Sample program

$1	$2
Preparation process	
M09	
↓	
Machining process	
↓	
Cutting-off process	
G01 X-3.0 F□	
M08	
M08	
/M109 Q□□□□	
M09	
M05	
M07	
G00 X-3.0 Z0 T00	
M56	
M02	M02
%	%

O□□□□ — Material change program (prepared by user)

$1	$2	
M03 S1=500		Spindle rotates forward at 500 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		Main 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 3 seconds
M52		Coolant on
M03 S1=500		Main spindle rotates forward at 500 min^{-1}.
G01 W30.0 F0.5		Insert material into guide bushing and move to the position where the tip is cut.
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
%		

Notes

- 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 800 min^{-1} or slower. Specifying the M7 command when the spindle speed exceeds 800 min^{-1} causes an alarm to occur.
- The start and end points before and after of M109 command (material change program) must be equal in the program. If the end point does not match the start point, an alarm may occur.

⚠ CAUTION

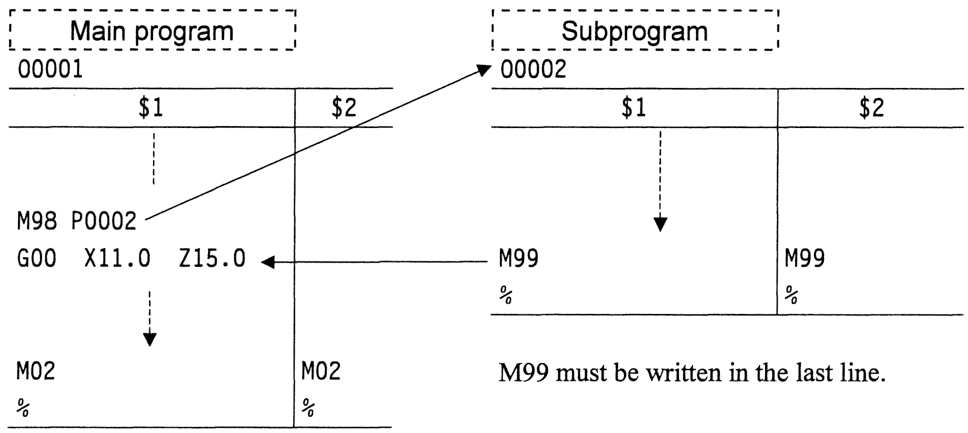
If the program pre-analysis function is enabled, do not use the subprogram (M98) to create a material change program. Use general-purpose material change program (M109) instead.
If the subprogram (M98) is used in the material change program, the program runs without changing material. Be sure to specify M109 Q□□□□ in the program.

6.3 Subprograms

Suppose that the program contains fixed sequences or repeated patterns. If such sequences and patterns are placed as subprograms in memory, the program can be written very easily.

Subprogram call from main program

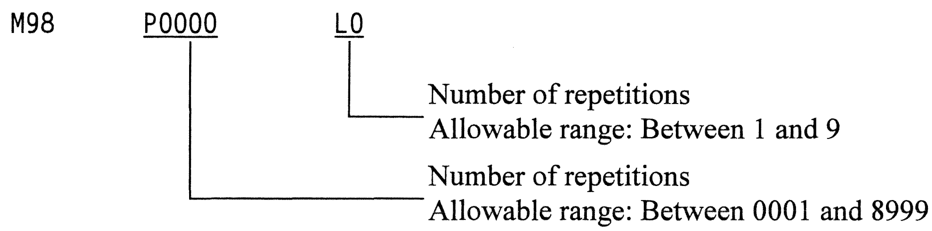
Sample program



M99 must be written in the last line.

Subprogram call pattern

The following rule must be observed when calling a subprogram:

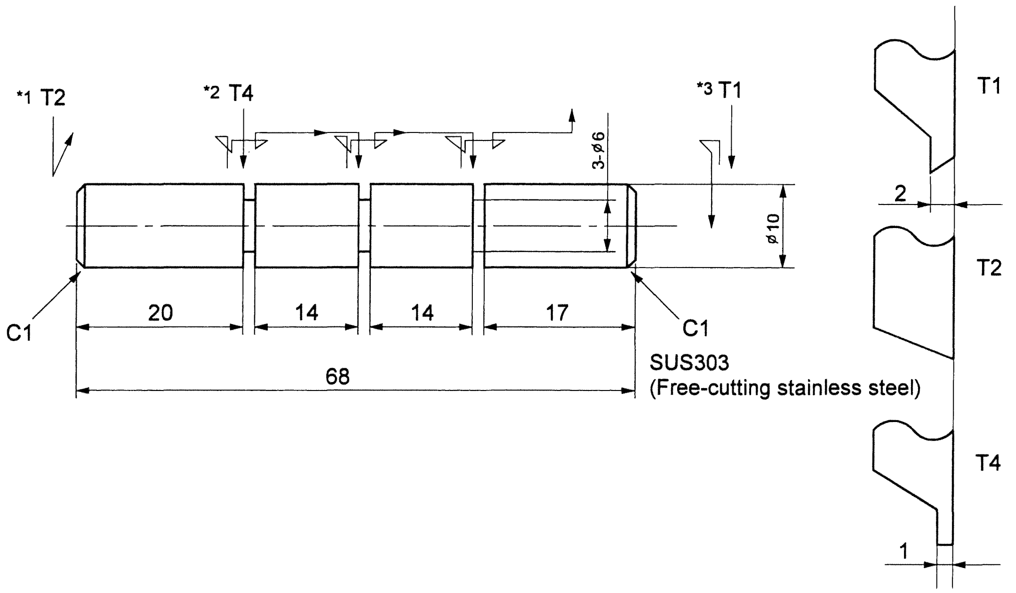


Notes

- If the number of repetitions is omitted, the subprogram is called only once.
- The subprogram can be repeated up to 9 times when it is called once.
- The maximum nesting depth from the main program is 2.
- Machining data is not supplied to subprograms.
- The execution program group which is composed of the main program and subprograms is restricted in capacity. Therefore, there may be restriction on the number of repeated subprogram runs depending on the total program size.
- If the program pre-analysis function is enabled, do not use the subprogram (M98) to create a material change program. Use general-purpose material change program (M109) instead.
- If the program pre-analysis function is enabled, do not create a program containing a subprogram (M98) in which processes change at every cycle.
- If the program pre-analysis function is enabled, too many M98 commands may cause the block count over alarm to occur. The capacity for execution program groups is limited.

Examples of using subprograms

Machining layout



Sample program

Main program

O0001

$1	$2
G50 Z0	
M06	
G99	
G00 Z-0.5	
M03 S1=1800	
G26	
N0102 T0200	
G00 X11.0 Z-0.5 T01	
X7.0	
G01 X11.0 Z1.5 F	
N0201 T0400	
G00 X11.0 Z21.0 T02	
M98 P0002 L3	
N0304 T0100	
G00 X11.0 Z70.0	
G01 X8.0 F0.015	
X11.0 F0.2	
W-1.5	
X8.0 W1.5 F0.015	
X-3.0	
G25	
M05	
M07	
G00 X-3.0 Z0 T00	
M56	
M02	M02
%	%

Execute the subprogram three times, then returns.

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	M99
%	%

6.4 Spindle Speed Fluctuation Detection Function (G25, G26, M94, M95, M96, M97)

This function issues an overheat alarm when the actual spindle speed becomes lower or higher than the specified speed, thus preventing the guide bushing from burning.

Command format

G26	Spindle speed fluctuation detection ON
G25	Spindle speed fluctuation detection OFF

or

M94	Back spindle speed fluctuation detection ON
M95	Back spindle speed fluctuation detection OFF
M95	Main spindle speed fluctuation detection ON
M96	Main spindle speed fluctuation detection OFF

Position of commands

- Place a block containing only G26 after the spindle speed command as follows:

  M03 S1=$\boxed{}$
  G26

- Place a block containing only G25 before the spindle speed stop command as follows:

  G25
  M05

Data setup

Default values for the threshold deviation and other quantities are assigned to parameters. They can be changed on programs.

[Default]					Parameter No.
p	:	3000	3 sec.	(time that elapses before checking starts)	No.4914
q	:	10	±10%	(deviation within which checking will not start)	No.4911
r	:	15	±15%	(deviation beyond which an alarm is issued)	No.4912

Notes

- Spindle speed fluctuation detection function is automatically enabled when running the program.
- Disable the spindle speed fluctuation detection function while the constant surface speed control is executed.
- Disable the spindle speed fluctuation detection function while the rigid tapping is executed.
- To enable or disable the spindle speed fluctuation detection function, enter a command in $1 for main spindle (S1) and in $2 for back spindle (S2).
- The default values can be changed on programs in the form of:
  G26 Pp Qq Rr EOB
  This value cannot be changed with M94, M95, or M96.
- The values set by the program will remain effective even when the power is turned off.
- The values changed by program will be reflected at automatic operation.
- Disable the spindle speed fluctuation detection function while the spindle rotates at 400min–1 or slower, because the spindle speed becomes unstable.
- Disable the spindle speed fluctuation detection function before changing the spindle rotation direction (forward to reverse, vice versa). Change rotatio direction, then enable the spindle speed fluctuation detection function.
- If an alarm is issued, the machine will stop at the end of current block (single block stop). A spindle overheat alarm is displayed on the CRT.
- The G25 and G26 codes cannot be written in a block which contains another G code or an X or Z coordinate value.
- Disable the spindle speed fluctuation detection function during spindle synchronization control.

6.5 Constant Surface Speed Control (Main and Back Spindle)

When the tool is fed into the workpiece in the diameter direction as in the process of cutting-off, the diameter of the workpiece changes. This also causes changes in the surface speed (workpiece speed relative to the tool). A command which specifies a surface speed can be used to read the tool position in the control unit, calculate the spindle speed, and change the spindle speed.

`G50 S☐ Q☐`	Clamp value of maximum spindle speed. This function prevents the spindle speed from increasing above this value while control for constant surface speed is in use. `S:` maximum spindle speed in min^{-1} clamped `Q:` minimum spindle speed in min^{-1} clamped
`G96 S1=☐` `S2=☐`	Starts control for constant surface speed. `S1=☐` (m/min) Surface speed of main spindle `S2=☐` (m/min) Surface speed of back spindle
`G97 S1=☐` `S2=☐`	Cancels control for constant surface speed. `S1=☐` (m/min) Main spindle speed after constant surface speed control is canceled `S2=☐` (m/min) Back spindle speed after constant surface speed control is canceled

Notes

- If a G96 code is given without an S specification, the S value previously specified in G96 will be in effect.
- If a G97 code is given without an S specification, the spindle will rotate at a speed calculated from the surface speed specified in G96.
- A G50 data is valid only when G96 (control for constant surface speed) is in effect.
- While control for constant surface speed is in effect, four-digit tool selection command (T☐☐☐☐) cannot be issued.
  If tool selection is required, issue a cancel command (G97), then issue a tool selection command.
- While a G96 code (control for constant surface speed) is in effect, a command to rotate another spindle or change the speed of another spindle cannot be issued.
  Such a command can be issued only after a G97 (canceling control for constant surface speed) command is issued.
- While control for constant surface speed is in effect, disable the spindle speed fluctuation detection function.

Sample program

```
O0001
```

Preparation process
↓
Front turning process
↓
Threading process
↓
Other process

N511 T0100	Selects a cut-off tool.
G00 X12.0 Z50.0 T□□	Positions the cut-off tool positioning point in rapid feed.
$\boxed{\text{G50 S5000}}$	Command for the maximum clamp setting of the main spindle (5000 min^{-1})
$\boxed{\text{G96 S1=100}}$	Get the main spindle controlled for constant surface speed (100 m/min.)
M320 X1.0	
G01 X-3.0 F0.02	Cutting-off
M241	
$\boxed{\text{G97}}$	Cancel the main spindle from control for constant surface speed
M05	
M07	
G00 X-3.0 Z$\boxed{}$ T00	
M56	
M02	
%	

6.6 Tool Nose R Compensation

If the cutting tool tip is rounded, this radius causes errors between programmed shapes and actually machined shapes during taper or circular cutting.

Given a radius value specified, the function for tool nose R offset calculates the error automatically and compensates for the error. The direction of offset can be specified by the following codes:

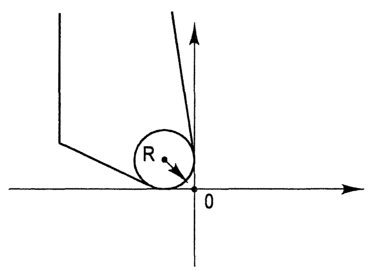
Command format

G40	Nose R offset cancel
G41	Nose R offset left mode ON
G42	Nose R offset right mode ON

Nose R data must be given in advance as part of the tool data. Actually, <R> tool nose R value and <TIP> virtual tool nose number are such data.

Virtual tool nose

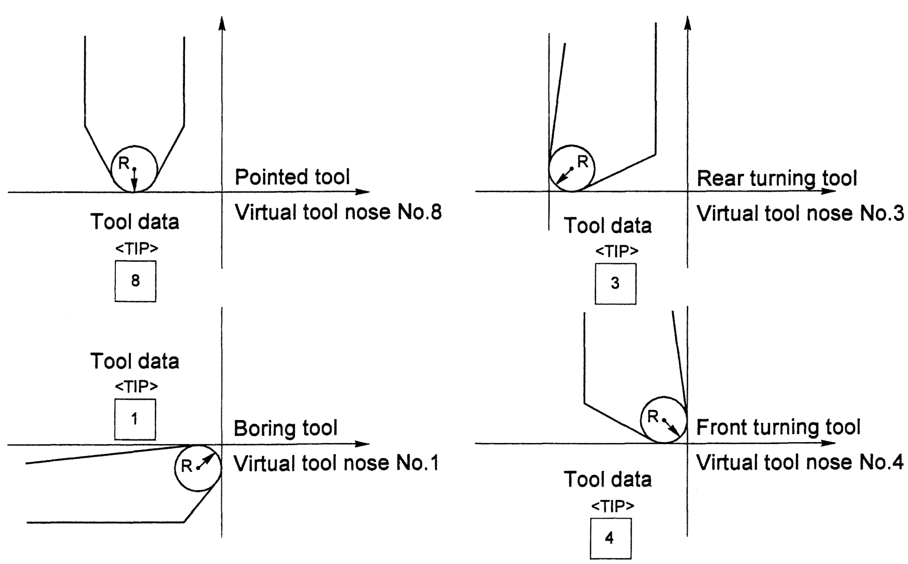
- The virtual tool nose refers to the zero point in the figure below. It does not exist actually.
- When setting the tool in the holder, make sure that it is set as shown below.



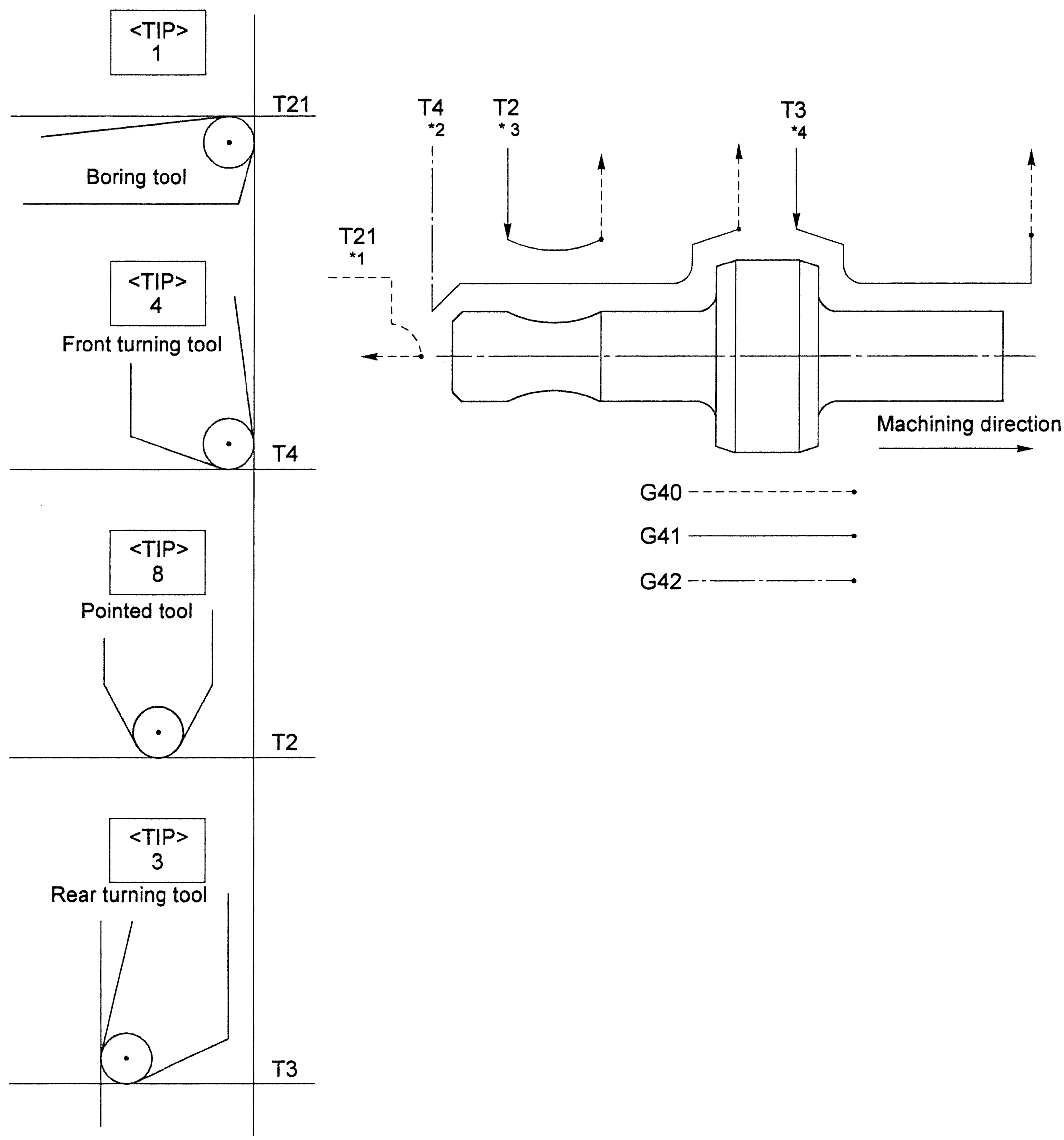
Virtual tool nose number

- The direction of the virtual tool nose when viewed from the center of the cutting tool radius is specified by the virtual tool nose number.
- The available virtual tool nose numbers are 0 to 9. Select a number according to the type of the tool as follows:

Virtual tool nose numbers given to commonly used tools requiring tool nose R compensation



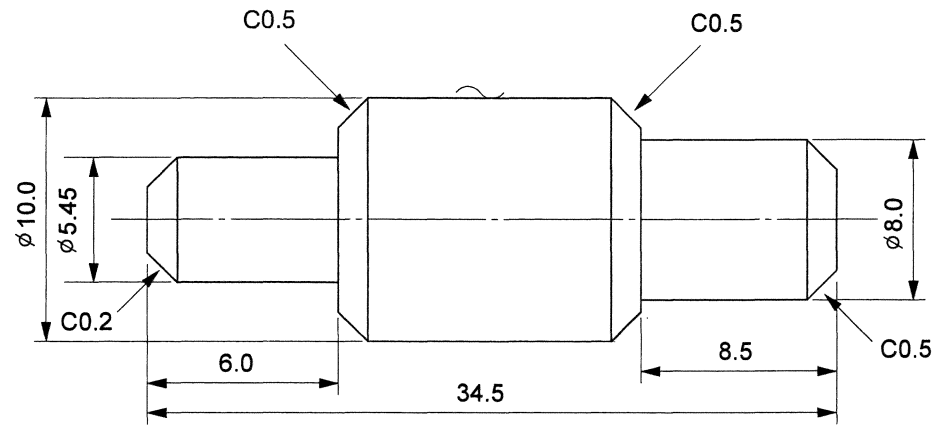
Basic pattern of G codes involving tool nose R offset



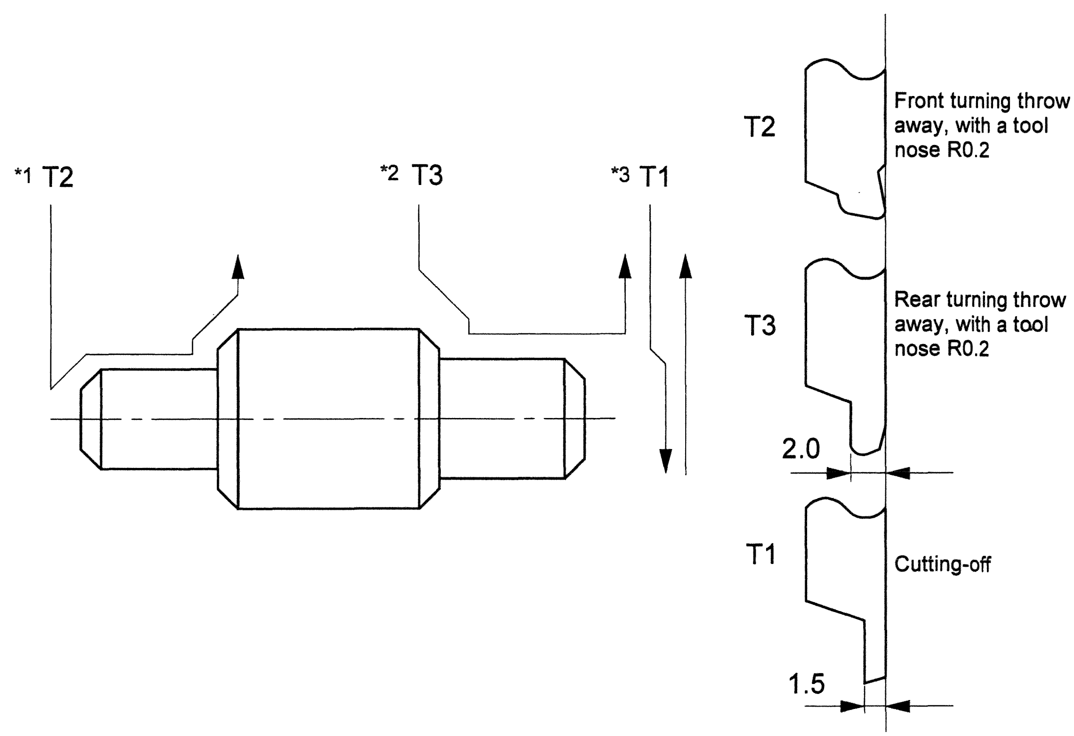
Notes

- Be careful when positioning the machine in rapid feed or canceling tool nose R offset. In general, move the tool away from the workpiece by at least the tool nose R.
- An alarm is issued when a prohibited G code is given while tool nose R offset is in progress.
- For such alarms, refer to <the instruction manual issued by FANUC, Ltd.>

Machining drawing



Machining layout



Sample program

O0300

$1		$2	
G50 Z0			
M06			
G99 G00 X15.0 Z–0.5 S1=2000 M03			
G600	Free pattern (machine pattern cancel)	G600	Free pattern (machine pattern cancel)
N0102 T0200			
G00 X15.0 Z–0.5 T02			
$\boxed{\text{G42}}$ G01 X4.05 F0.2	Nose R offset right mode ON		
$\boxed{\text{G41}}$ X5.45 Z0.2 F0.03	Nose R offset 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	Nose R offset cancel		
N0203 T0300			
G50 W–3.0			
$\boxed{\text{G41}}$ G00 X15.0 Z23.5 T03	Nose R offset 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	Nose R offset cancel		
G50 W3.0			
N0304 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	
%		%	

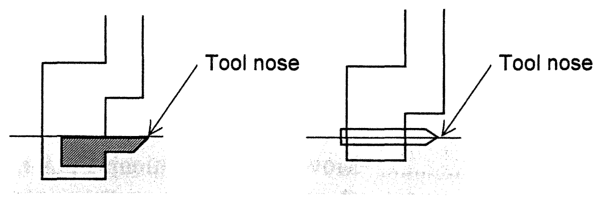
Note

This program does not contain a product separation program.

6.7 Cut-off Tool Breakage Detection (M51)

After the cut-off process, the touch sensor mounted on the tool post detects a cut-off tool breakage by sensing remaining workpiece.

> ⚠ **CAUTION**
>
> **In the block before the block containing M51, specify the X value so that the tool does not contact with the material.**
> **If any object exists in the portion under the tool nose of boring or center drilling tool (shaded portion in the figure) between cut-off tool and the breakage sensor, the workpiece may interfere with the tool when detecting the cut-off tool breakage.**
>
> **In addition, the workpiece may interfere with the tool mounted to the sleeve holder when detecting the cut-off tool breakage.**



Command format

`M51 X[ ] W[ ] F[ ]` Cut-off tool breakage detection

X: Position of X1 axis to move the tip of touch sensor. With this argument not specified, the X1 axis moves to the position –1.0.
Specify X argument 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: Feed rate (per minute, with decimal point) at touch sensor.
If this argument is omitted, the cut-off tool breakage detection device moves at the speed of 2000.0 mm/min. Be sure to specify the feed rate with decimal point.

Detection method

Cut-off Tool Breakage Detection (M51) is performed as follows:

1. The tip of the touch sensor moves to the positioning point.

   1-1. Call the touch sensor (the X1 and Y1 axes move), and position the device to the position "outer diameter + tool positioning point" of the machining data (the X1 axis moves).

   1-2. If the move distance (W argument) is specified for the workpiece (Z1 axis), the workpiece (Z1 axis) moves at the same time when the X1 axis is positioned.

2. The touch sensor starts operating.

   2-1. Detection by the touch sensor is enabled. The tip of the touch sensor moves down to the detection point specified by X argument at the feed rate specified by F argument.

   2-2. 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 generated.

3. The tip of the touch sensor moves to the positioning point.

    3-1. The tip of the touch sensor moves to the position "outer diameter + tool positioning point" of the machining data.

    3-2. If the workpiece (Z1 axis) has been moved by W argument in Step 1, the workpiece goes back to the position before the M51 command is issued.

Note

The machine moves to the sensing position automatically as commanded by M51.

See Figure 1 for the sensing position and Figure 2 for the position when viewed in the longitudinal direction.

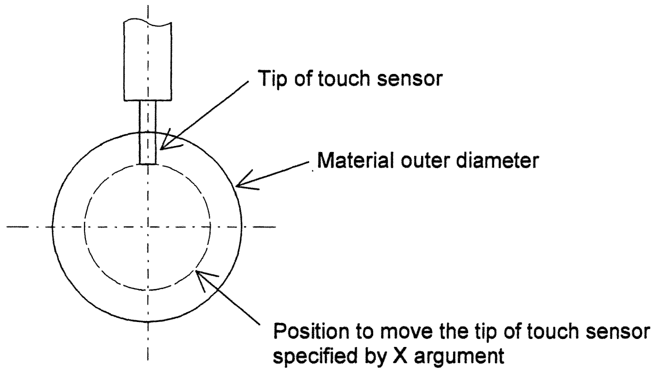
Sample program 1 (with no argument specified)

`O0007`	
`G50 Z0`	
`M06`	
`G99`	
`G00 X13.0 Z-0.5`	Moves tool to positioning point. (material diameter: ⌀12)
`M51`	Command (sequence of operations) to detect a cut-off tool breakage.
`M03 S1=2000`	
`T1100`	

Sample program 2 (with arguments specified)

`O0007`	
`G50 Z0`	
`M06`	
`G99`	
`G00 X13.0 Z-0.5`	Moves tool to positioning point. (material diameter: ⌀12)
`M51 X[ ] W[ ] F2000.0`	Command (sequence of operations) to detect a cut-off tool breakage. Specify X argument when changing the position to move the tip of touch sensor. Specify W argument if the workpiece length is short. F2000.0 represents the feed rate.
`M03 S1=2000`	
`T1100`	

⚠ CAUTION



- **Specify X argument so that the tip of the touch sensor surely pushes the material diameter. In addition, it is recommended to move the touch sensor to the position X=–1.0 to detect the remaining workpieces of various shape.**
- **If the workpiece is rather short, use M51 with W argument to advance the spindle to the position at which the spindle hits on the sensor.**
  The advance amount varies depending on the touch sensor shift amount (4.0 mm from the zero point) shown in Figure 2 in the next page, the workpiece length and the cut-off width. Enter the value of the W argument according to the conditions.

  * **The W argument should be specified if the workpiece length is equal to or less than 3.0 mm. However, the value is affected by the cut-off tool width.**

Note

Specify M51 (cut-off tool breakage detection) in axis control group 1 ($1). If specified in $2, an alarm occurs.

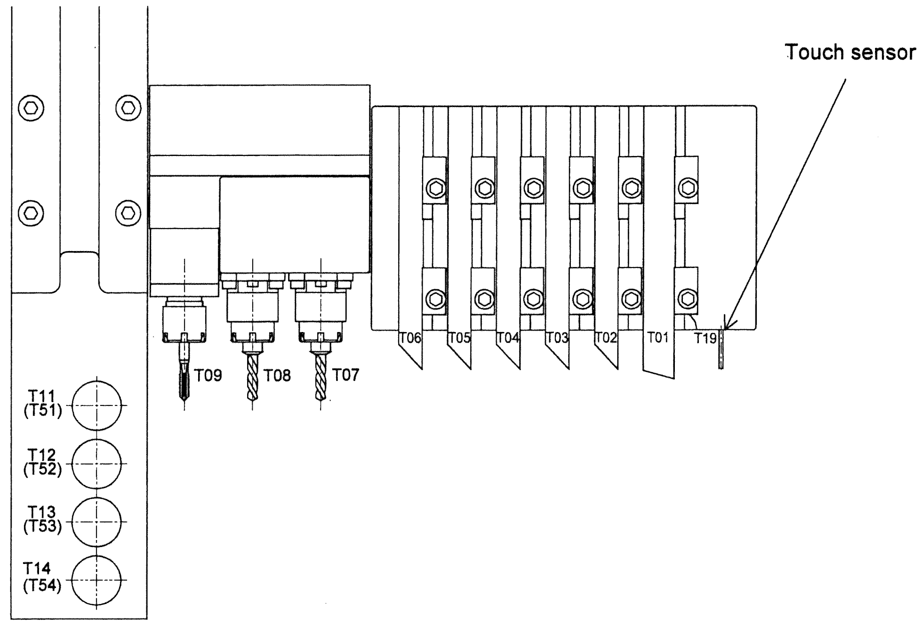


Figure 1

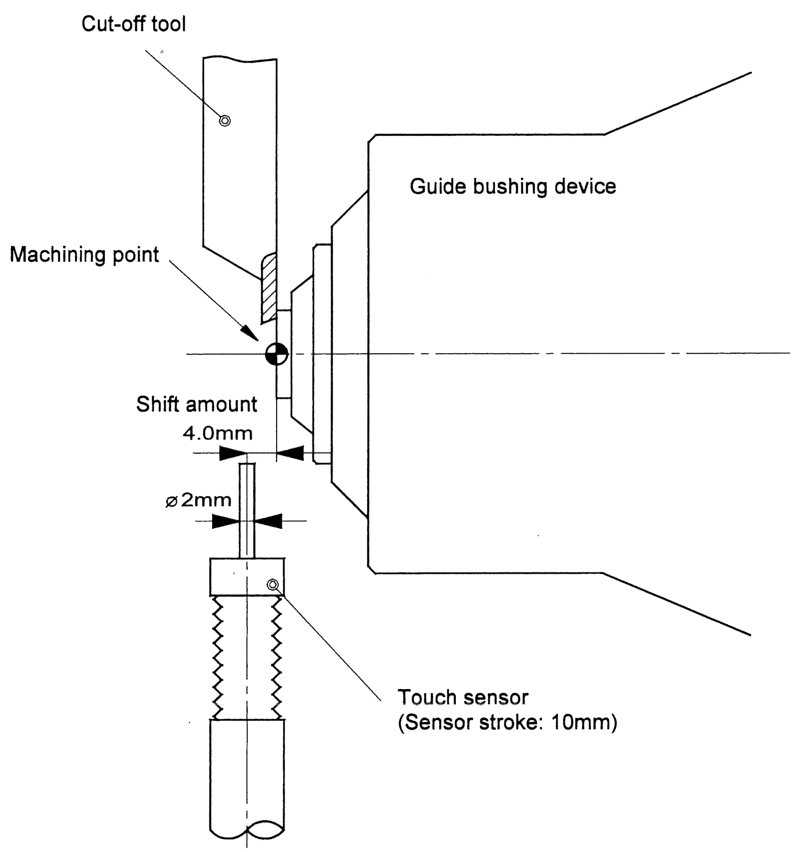


Figure 2

6.8 Front/Back End-face Rigid Tapping (G84, G80)

This command controls the spindle motor in such a way to control a servo motor to compensate the tapping spindle (Z1 axis or Z2 axis) and the spindle with each other for tapping.

This command also performs a phase adjustment (for main and back spindle only). Put a checkmark on SYNC. TAPPING PHASING FUNCTION in Machine Structure screen. To enable this function, turn off and then on the power of the operator panel.

This function is useful when you want to perform rigid tapping for deburring after performing rigid tapping and other machining process.

> ⚠ **CAUTION**
>
> **Phase adjustment may reduce an accuracy a little.**

Command format

G84 Z(W)☐ R☐ F☐ D±☐ S☐ Q☐ T☐ H1:
Front/back end-face rigid tapping

G80: Rigid tapping mode cancel

Z(W): Specify the end point of tapping.
(Incremental command used when W is specified)

R: Specify the distance from the Z-axis positioning point to the start point of synchronous tapping.

F: Specify the thread pitch.

D±: Specify the direction in which the spindle is to rotate. Specify 1 to rotate the main spindle forward. Specify 2 to rotate the back spindle forward. Specify 3 to rotate the tool spindle forward.

S: Specify the speed.

Q: Tap pull-out override specification (%). Specify this argument if you want to make the tap pull-out rate faster. The available values range from 100 to 200 in 10%. The default value is 100.
To specify the Q argument, an option other than the rigid tap function (high-speed rigid tap function) is required.

T: Specify the dwell time at the bottom of a hole in unit of 1/1000 sec. Do not omit the decimal point.
If the argument is omitted, the program does not stop (no dwell).

H1: Argument for step machining. Does not specify G80 (rigid tapping mode cancel) command at the end of G84 (front/back end-face rigid tapping) in macro program. If the argument is omitted, specify this command before exiting from the rigid tapping mode.

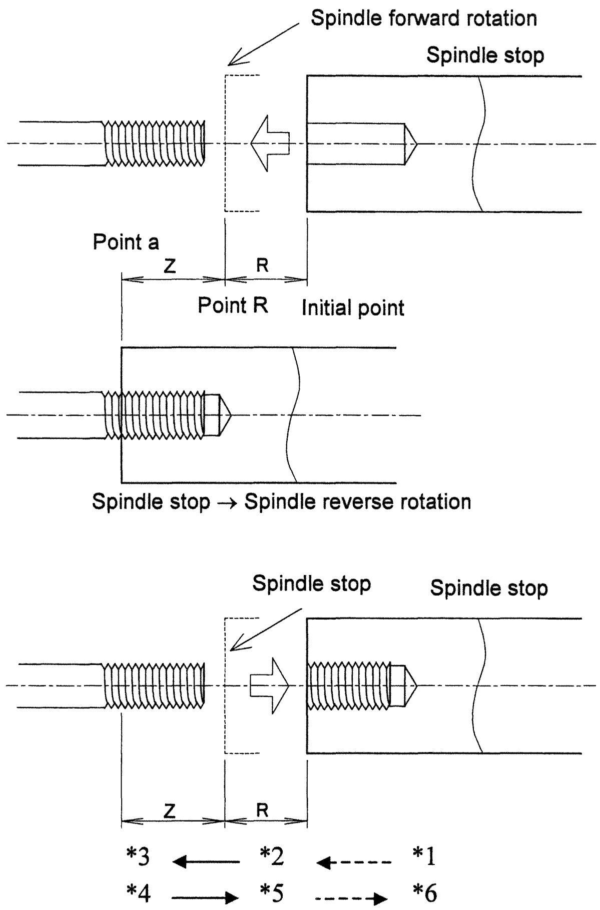
Per-minute feed: Thread lead = Feed rate ÷ Spindle speed

Per-rotation feed: Thread lead = Feed rate

Notes

- If no option for synchronous tapping function is specified, specifying G84 command (front/back end-face rigid tapping) will cause an alarm to occur.
- When the rotation direction of spindle (D argument) is ±3, specifying G84 command (front/back end-face rigid tapping) in axis control group 2 ($2) will cause an alarm to occur.
- When the rotation direction of spindle (D argument) is ±1, specifying G84 command (front/back end-face rigid tapping) in axis control group 2 ($2) will cause an alarm to occur.
- If the end point of tapping (argument Z or W) is not specified, an alarm will occur.
- If any other value than ±1, ±2, or ±3 is specified for the rotation direction of spindle (D argument), an alarm will occur.
- If the thread pitch (F argument) or speed (S argument) is not specified, an alarm will occur.
- Disable the spindle speed fluctuation detection function while the main spindle (S1) or back spindle (S2) is performing rigid tapping.
- Be sure to stop the tool spindle before specifying the G84 command (front rigid tapping by tool spindle).
- Phase adjustment is unavailable in front rigid tapping by tool spindle (G84).
- If the options for the high-speed rigid tap function are not set, the tap pull-out override (Q argument) causes an alarm to occur at the front/back end face rigid tap (G84) command.
- The tap pull-out override (Q argument) is rounded in 10%. If a value out of the available range is specified, the Q argument is clumped to 100 or 200.
- When the program pre-analysis function is enabled, issuing G84 (back end-face rigid tapping) command from axis control group 1 ($1) causes an alarm to occur.
- Reverse running of M84 (rigid tap) command is inhibited.

Outline of machining



*1 Spindle stop state

*2 Rapid feed from the initial point to the point R.

*3 The spindle rotates in the forward direction at the speed of S☐ and tapping is performed to the point a, then the spindle stops.

*4 Dwell (only if specified)

*5 The spindle rotates in the reverse direction at the speed of S☐ and the Z axis moves to the point R, then the spindle stops.

*6 Rapid feed to the initial point.

←--- Rapid feed
←—— Cutting feed

Sample program 1

```
↓
M5

G0 Z-2.0 T00

N223 T1300 ................................ Selects a tapping tool.
G84 Z5.0 R1.0 F0.5 D1 S500 ................ Rigid tapping command
G80 ....................................... Rigid tapping cancel
M3 S1=☐
↓
```

Sample program 2

```
:
M5
G0 Z-2.0 T00
N223 T1300 ................................ Selects a tapping tool.
G84 Z5.0 R1.0 F0.5 D1 S500 H1 ............. Rigid tapping command   first step
G84 Z6.0 R1.0 F0.5 D1 S500 H1 ............. Rigid tapping command   second step
:
G84 Z9.0 R1.0 F0.5 D1 S500 H1 ............. Rigid tapping command   (n-1)-th time
G84 Z10.0 R1.0 F0.5 D1 S500 ............... Rigid tapping command   n-th time
                                                                   (with H argument not specified)
G80 ....................................... Rigid tapping cancel
:
```

6.9 Corner Chamfering/Rounding

This function enables you to program using the values for corner chamfering and rounding that are written on the machining drawing.

Command format

With corner chamfering:

```
G1 X□ K□
G1 Z□ I□
```

With corner rounding:

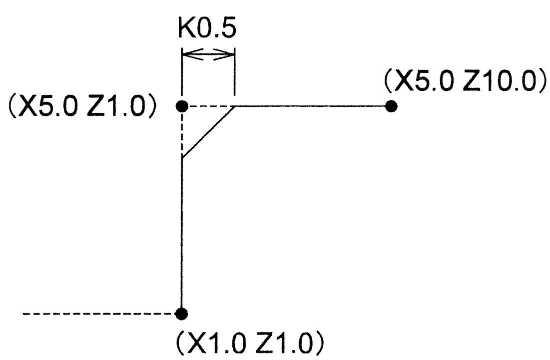
```
G1 X□ R□
```

X:	Coordinates of the corner
K:	Value for corner chamfering (for X axis)
I:	Value for corner chamfering (for Z axis)
R:	Value for corner rounding

Sample program

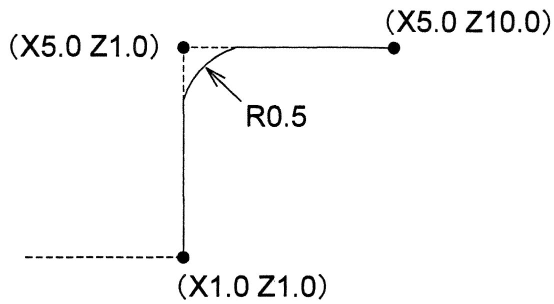
With corner chamfering:

```
:
G1 X1.0 Z1.0 ...................... Coordinates of the previous corner
(G1) X5.0 (Z1.0) K(I)0.5 .......... Coordinates of the new corner, and value for chamfering
(G1) (X5.0) Z10.0 ................. Coordinates of the next corner
```



With corner rounding:

```
:
G1 X1.0 Z1.0 .................................. Coordinates of the previous corner
(G1) X5.0 (Z1.0) R0.5 ................ Coordinates of the new corner, and value for rounding
(G1) (X5.0) Z10.0 ........................ Coordinates of the next corner
                                           (The value in parentheses can be omitted.)
```



Note

If both corner chamfering/rounding function and direct input of on-drawing dimensions function are specified, those two functions cannot be performed together in the same program. Select either of functions using parameters shown in the table below.

No.3453	0	Corner chamfering/rounding
	1	Input of on-drawing dimensions

If the program contains either one only, or does not contain both functions, the parameter No. 3453 is invalid.

6.10 Canned Drilling Cycle

This function simplifies the creation of machining programs. For example, when you want to repeat deep-hole machining, you can perform the operation by executing only one block of the program.

Command format

With the drilling tool spindle in the Z-axis direction:

G83 Z(W) ☐ Q☐ P☐ F☐

With the drilling tool spindle in the X-axis direction:

G87 X(U)☐ Q☐ P☐ F☐

Canned drilling cycle cancel

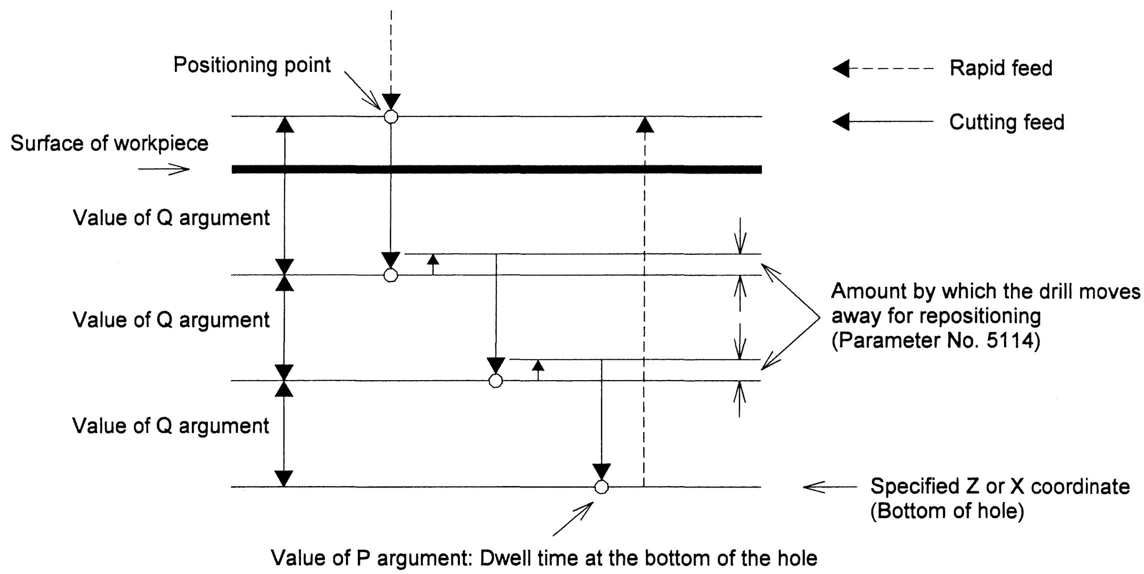
G80

Z(W):	Z coordinate of the bottom of the hole
X(U):	X coordinate of the bottom of the hole
Q:	Infeed amount at a time (Z axis: 1 mm = 1000, X axis: Diameter 1 mm = 500) *
P:	Dwell time at the bottom of the hole (1 second = 1000) *
F:	Cutting feed rate

* For operation in units of sub-microns, multiply the value by 10.

■ Parameter No. 5114:

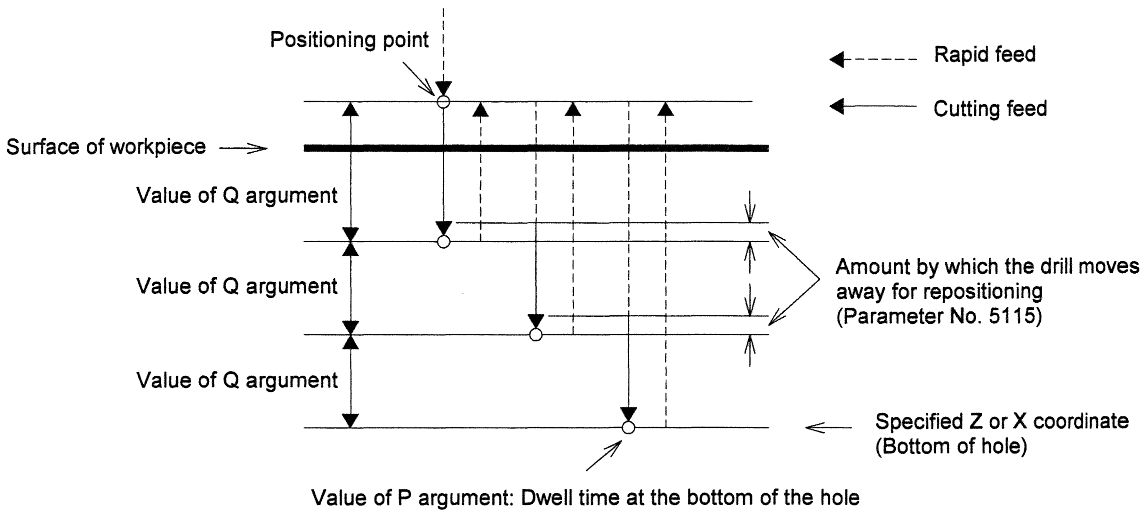
The following shows the operation when parameter No. 5101 bit 2 is 0. The initial value is 1.



Amount by which the drill moves away for repositioning. The initial value is 0.
(Z axis: 1 mm = 1.0, X axis: Diameter 1 mm = 0.5)

■ Deep hole drilling cycle

The following shows the operation when parameter No. 5101 bit 2 is 1. The initial value is 1.



Parameter No. 5115: Amount by which the drill moves away for repositioning. The default value is 0.5.
(Z axis: 1 mm = 1.0, X axis: Diameter 1 mm = 0.5)

Sample program

With the drilling tool spindle in the Z-axis direction (G83)

When the depth of the hole is 20 mm and the infeed amount at a time is 2 mm, specify 0.5 second as the dwell time at the bottom of the hole, and F0.04 as the cutting feed rate.

```
:
M3 S1=1000
G99 ............................................ Millimeters per revolution feed
                                                 (Millimeters per minute feed can be specified instead.)
G0 Z-1.0 ....................................... Move the workpiece to a safety position.
T2100 .......................................... Select a drilling tool.
G0 Z-1.0 ....................................... Position the drill in Z-axis direction.
G83 Z20.0 Q2000 P500 F0.04 ..................... Canned drilling cycle
G80 ............................................ Canned drilling cycle cancel
G0 Z-1.0
:
```

Note) For the amount by which the drill moves away for repositioning, specify the value in parameter No. 5114 or No. 5115 according to the data of bit 2 of parameter No. 5101. For operation in units of sub-microns, multiply the values of Q and P arguments by 10.

With the drilling tool spindle in the X-axis direction (G87)

When the depth of the hole is 3 mm (material diameter 12.0 mm) and the infeed amount at a time is 1 mm, specify 0.5 second as the dwell time at the bottom of the hole, and F30 as the cutting feed rate.

```
:
M80 S3=1000 .................................... Tool spindle speed
G98 ............................................ Millimeters per minute feed
G0 Z-1.0 ....................................... Move the workpiece to a safety position.
                                                 (This operation is not necessary when calling the tool spindle does not
                                                 cause any interference.)
T1100 .......................................... Select a tool spindle.
G0 X13.0 ....................................... Positioning at cutting feed rate
G87 X6.0 Q500 P500 F30 ......................... Canned drilling cycle
G80 ............................................ Canned drilling cycle cancel
G0 X13.0 ....................................... Move the tool away to a point of the material outer diameter + 1.0 mm.
:
```

Note) For the amount by which the drill moves away for repositioning, specify the value in parameter No. 5114 or No. 5115 according to the data of bit 2 of parameter No. 5101. For operation in units of sub-microns, multiply the values of Q and P arguments by 10.

Notes

- A dwell time cannot be specified for the positioning point.
- The program above is given as a sample, so specify machining conditions in consideration of the actual material to be machined, etc.
- Select the axis control group by pressing the Menu selection keys ◄ and ► at the bottom of LCD screen simultaneously, and enter the necessary parameter.

6.11 Multiple Repetitive Cycle

This function simplifies the creation of machining programs. For example, the tool path for rough machining in a middle process can be determined automatically by giving the information of only the final workpiece form.

Command format

Rough outline machining cycle

```
G71 U[    ] R[    ]
G71 P[ ns ] Q[ nf ] U[    ] W[    ] F[    ] S1(2)=[    ] T[    ]
N[ ns ]
 :
(Workpiece form)
 :
N[ nf ]
```

(Workpiece form): The final form is created as the tool moves A → A' → B in the figure below. The commands to move the tool can be specified in the blocks of the sequence numbers from ns to nf.

U : Infeed amount
Specify an unsigned value. The infeed direction is determined by the direction from A to A'. (Radius specification) This specification is modal and valid until the next value is specified. The infeed amount can also be specified in parameter No. 5132. The parameter value varies depending on program specification.

R : Distance by which the tool moves away
This specification is modal and valid until the next value is specified. The distance can also be specified in parameter No. 5133. The parameter value varies depending on program specification.

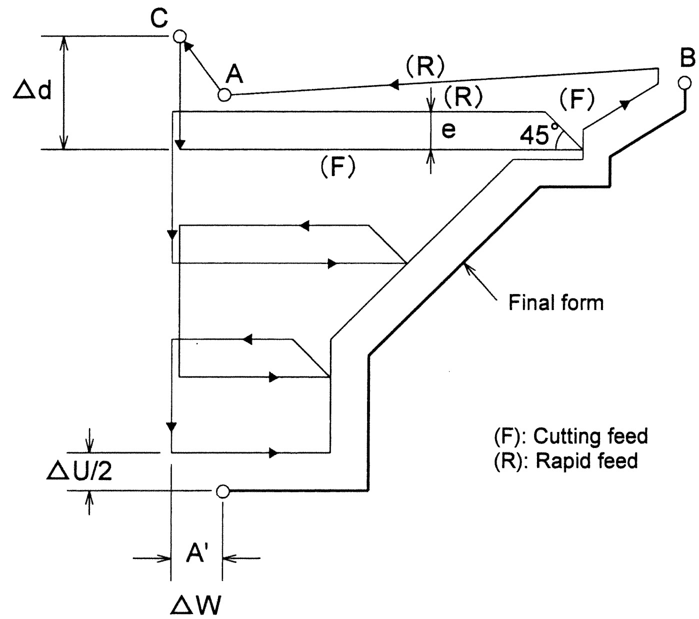
P[ns] : Sequence number of the first one of the final form blocks

Q[nf] : Sequence number of the last one of the final form blocks

U : Tool's moving distance and direction for the finishing allowance in the X-axis direction (diameter/radius specification)

W : Tool's moving distance and direction for the finishing allowance in the Z-axis direction

F S1(2)=[   ] T :
The F, S, and T functions specified in the blocks ns to nf are ignored during the rough outline machining cycle. However, the F, S, and T functions specified in the block including the G71 command are enabled.



Finish machining cycle

When the G71, G72, and G73 commands are used for rough outline machining, they can also be used for finish machining.

Command format

G70 P☐ Q☐

P: Sequence number of the first one of the final form blocks

Q: Sequence number of the last one of the final form blocks

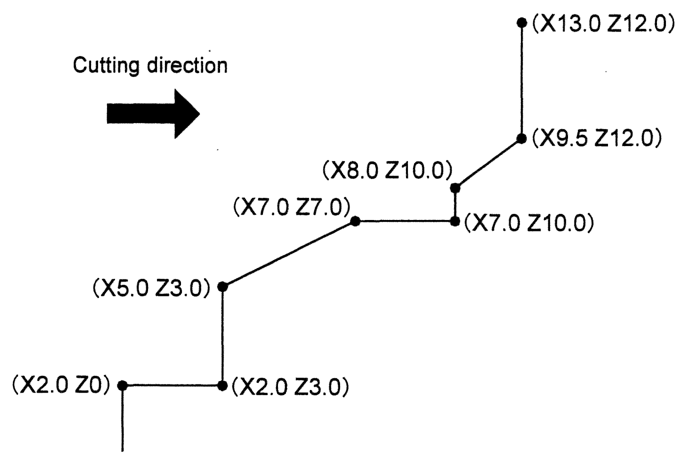
Notes

- When the G70 command is executed, the F, S, and T functions that are specified in the blocks including the G71, G72, or G73 command are ignored. However, the F, S, and T functions specified in the blocks of the sequence numbers from P☐ to Q☐ are enabled.
- When the finish machining cycle (G70) ends, the tool returns to the start point at the rapid feed rate, and the specified CNC data is loaded into the next block of the finish machining cycle (G70).
- Subprograms cannot be called from the blocks of the sequence numbers from P☐ to Q☐ that are used with the commands G70 to G73.
- When the program pre-analysis function is enabled, Type II of G71 and G72 cannot be used. For more information, refer to <the instruction manual issued by FANUC, Ltd.>

Sample program

Rough outline machining cycle (G71) and finish machining cycle (G70)

For example, use these commands to perform rough machining for part of a workpiece as shown below and finish machining. (Material diameter 12.0)



Examples of machining conditions

Δd (infeed amount)	: 1.0 (mm) (2 mm for the diameter)
e (Distance by which the tool moves away)	: 0.5 (mm) (1 mm for the diameter)
Δu (finishing allowance in the X-axis direction)	: 0.1 (mm) (Finishing allowance of 0.1 mm for the diameter is left.)
Δw (finishing allowance in the Z-axis direction)	: –0.05 (The sign is changed depending on the cutting direction. A negative value is specified for the cutting direction in this example.)

ns (sequence number of the first block of the finish machining program) : 1000

nf (sequence number of the last block of the finish machining program) : 2000

f (feed rate for rough machining) : 0.1

s (spindle speed for rough machining) : 2000

t (offset number for rough machining) : 13

```
:
M3 S1=1000
G99
G0 X13.0 Z-1.0
G71 U1.0 R0.5                                  }
G71 P1000 Q2000 U0.1 W-0.05 F0.1 S1=2000 T13   }................ Rough outline machining cycle
N1000 G0 X2.0 T12
G1 Z3.0 F0.02
X5.0
X7.0 Z7.0 ...................................................... Finish machining program
Z10.0 F0.05
X8.0
X9.5 Z12.0 F0.03
N2000 Z13.0 F0.02
G70 P1000 Q2000 ................................................ Finish machining cycle
G0 X13.0 T0
:
```

Notes

- The program above is given as a sample, so specify machining conditions in consideration of the actual material to be machined, etc.
- For further explanation such as notes, see <the instruction manual issued by FANUC, Ltd.>

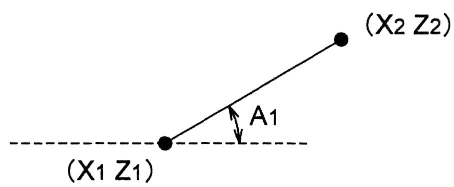
6.12 Direct Input of On-drawing Dimensions

This function simplifies the creation of machining programs. For example, you can use values such as angles of lines, chamfering, and corner rounding that are written on a machining drawing when programming. Some information of the function is given in <Section 6.9 Corner Chamfering/Rounding>. The following explains other functions:

Command format

Input of an angle

(Example) Calculate the next coordinate from a given X coordinate and angle.



Suppose that the tool is now at the coordinates (X_1, Z_1).

G1 X $\boxed{X_2}$, A $\boxed{A_1}$

When the value of Z_2 is not entered and only the angle A_1 is entered, the value of Z_2 is calculated for the tool to move.

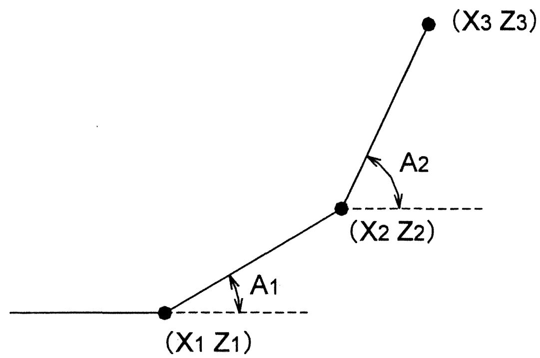
When the value of Z_2 is known but X_2 is unknown, the following is determined:

G1 Z $\boxed{Z_2}$, A $\boxed{A_1}$

A : Angle

When the tool path consists of consecutive slanting lines:

(Example)



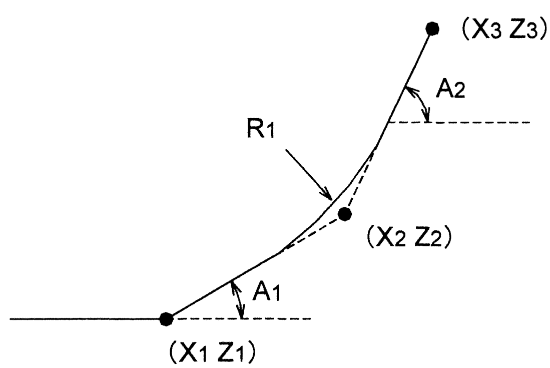
Suppose that the tool is now at the coordinates (X_1, Z_1).

G1 X$\boxed{X_2}$, A$\boxed{A_1}$ or G1 Z$\boxed{Z_2}$, A$\boxed{A_1}$

(G1) X$\boxed{X_3}$, A$\boxed{A_2}$ or (G1) Z$\boxed{Z_3}$, A$\boxed{A_2}$ (The value in parentheses can be omitted.)

When R connects the different lines and different lines of an angle

(Example)

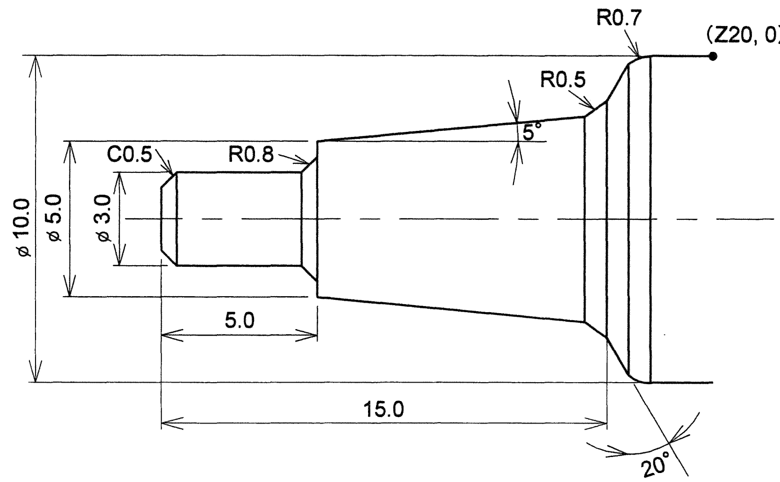


Suppose that the tool is now at the coordinates (X_1, Z_1).

G1 X$\boxed{X_2}$, A$\boxed{A_1}$, R$\boxed{R_1}$ or G1 Z$\boxed{Z_2}$, A$\boxed{A_1}$, R$\boxed{R_1}$

(G1) X$\boxed{X_3}$, A$\boxed{A_2}$ or (G1) Z$\boxed{Z_3}$, A$\boxed{A_2}$ (The value in parentheses can be omitted.)

Sample program




```
:
G1 X0 Z0 F0.02
X3.0, C0.5
Z5.0, R0.8
X5.0
Z15.0, A5.0, R0.5
X10.0, A70.0, R0.7
Z20.0
:
```

Notes

- The program above is given as a sample, so specify machining conditions in consideration of the actual material to be machined, etc.
- For further explanation such as notes, see <the instruction manual issued by FANUC, Ltd.>
- If both corner chamfering/rounding function and direct input of on-drawing dimensions function are specified, those two functions cannot be performed together in the same program. Select either of functions using parameters shown in the table below.

No.3453	0	Corner chamfering/rounding
	1	Input of on-drawing dimensions

  If the program contains either one only, or does not contain both functions, the parameter No. 3453 is invalid.

- The input of on-drawing dimensions is disabled while the program pre-analysis function is enabled.

6.13 Continuous Threading

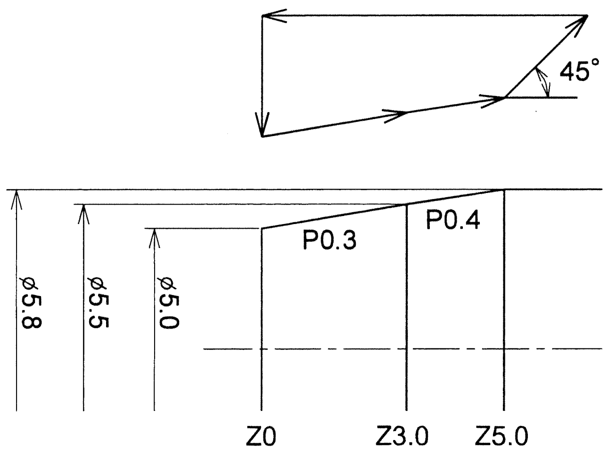
Use this function to cut special screw thread which changes in lead or geometry while thread cutting is in progress. Thread cutting is performed while synchronization with the spindle is controlled by means of connection of the changes. Unlike the G92 command, the G32 command does not provide a canned cycle, so it does not return the tool to thread cutting start point.

Command format

```
G32 X(U) Z(W)[    ] F[    ]
X(U) Z(W)[    ] F[    ]
:
```

(Consecutive coordinates (X, Z) and pitches can be specified.)

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°.

```
O0001
:
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.
:
```

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.

Notes

- 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.

6.14 Threading (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 curring and tapered thread cutting.

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☐
```

Arguments

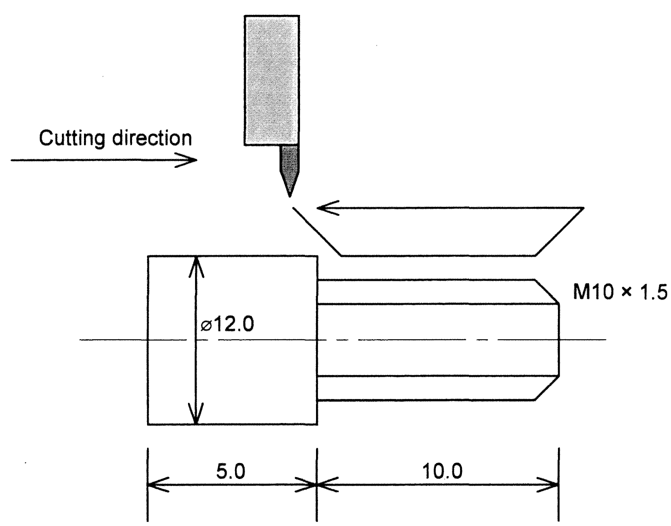
- X☐ (U☐) : Specify the infeed position.
  (U: incremental command)
- Z☐ (W☐) : Specify the threading end position in longitudinal direction.
  (W: incremental command)
- F☐: Specify the pitch (lead) in longitudinal direction.
- Q☐: Specify the shift angle at the start of threading (between 0.001 and 360.000°), if needed.

  Q is used for multi-lead threading. Decimal point is not allowed, for example, specify 90000 for 90.0°. See <6.17 Multi-thread Cutting> for details.

Sample program

```
T300
G0 X20.0 Z5.5 T3
G1 X14.696 F0.1
G32 X9.5 W1.5 F1.5
Z15.0
U5.196 W1.5
G0 Z5.5

X14.336
G32 X9.14 W1.5 F1.5
Z15.0
U5.196 W1.5
G0 Z5.5
  :
  :
```



Repeat infeed cutting to complete threading.

Notes

- 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.
- G32 is not a canned cycle, therefore, perform thread cutting by repeating the cycle (infeed → thread cutting → outfeed → return to threading start position) to finish threading. Use of the subprogram is recommended to simplify the program description.

6.15 Longitudinal Cut-Off Cycle (G75) — Option

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.

Command format

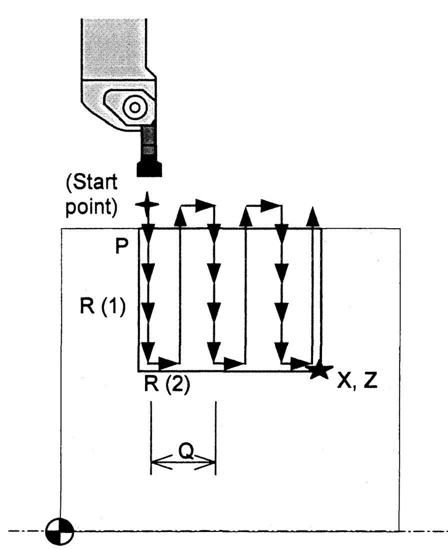
```
G75 R(1)
G75 X[    ] Z[    ] P[    ] Q[    ] R(2) F[    ]
```

Arguments

- 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.)

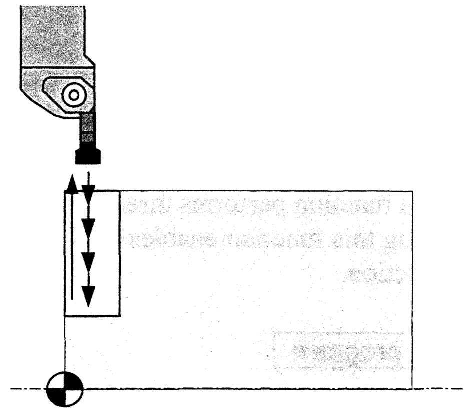
Sample program

	:	
(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 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
   :
```



Notes

- 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 operation is not performed.
- The followings will 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.

6.16 Link-Thread Machining

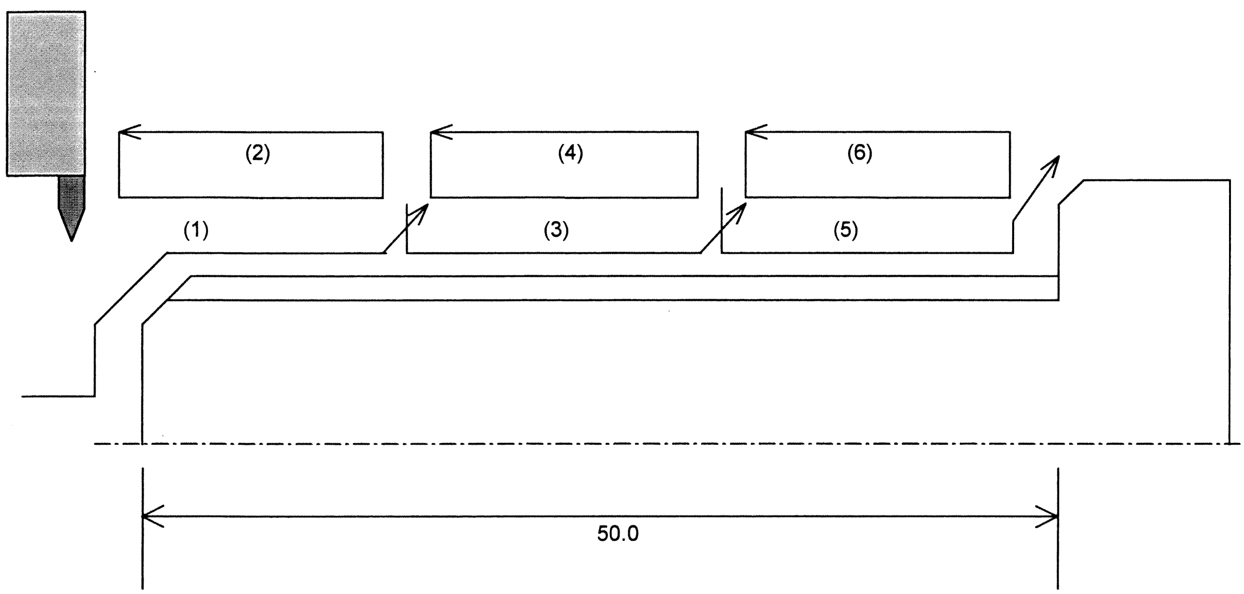
With a machine equipped with guide bushing, if the thread length in Z direction is longer than the hold length of inner diameter of guide bushing, the workpiece becomes apart from the guide bushing when it returns to the thread positioning point. Thus, the machining is discontinued. In such a case, repeat a cycle of "outer diameter turning to thread cutting" several times to link the thread to subsequent threads. It enables thread cutting in Y 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 lenth of guide bushing inner diameter: 25.0 mm

Material diameter: ⌀20.0 mm
Shift amount for thread cutting: 1.5 mm
M12 × pitch 1.5



```
T300 (Front turning (1))
G0 X21.0 Z-0.5 T3
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
X14.0 F0.1
G3 X12.0 Z19.0 R1.0 F0.05
Z33.0 F0.1
X18.0
X21.0 W1.5
  :
T400 (Thread cutting (4))
G50 W-1.5
G0 X21.0 Z12.0 T4 .............
M98 P0007
G0 X21.0 Z12.0 T0
G50 W1.5
```

Z: Distance from the first threading position multiple of pitches
First threading position Z–1.5 to Z12.013.5mm
15.0 ÷ 1.5=9 Divided out

```
T300 (Front turning (5))
G0 X21.0 Z32.0 T3
X14.0 F0.1
G3 X12.0 Z33.0 R1.0 F0.05
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 X16.496
G32 X11.3 W1.5 F1.5
W15.0 (*)
U5.196 W1.5
G0 W-18.0
X15.996

G32 X10.8 W1.5 F1.5
W15.0
U5.196 W1.5
G0 W-18.0
X15.676

G32 X10.48 W1.5 F1.5
W15.0
U5.196 W1.5
G0 W-18.0
X15.476

G32 X10.28 W1.5 F1.5
W15.0
U5.196 W1.5
G0 W-18.0
X15.356

G32 X10.16 W1.5 F1.5
W15.0
U5.196 W1.5
G0 W-18.0
X15.256

G32 X10.06 W1.5 F1.5
W15.0
U5.196 W1.5
G0 W-18.0
X15.256

G32 X10.06 W1.5 F1.5
W15.0
U5.196 W1.5
G0 W-18.0

M99
```

Notes

- 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.

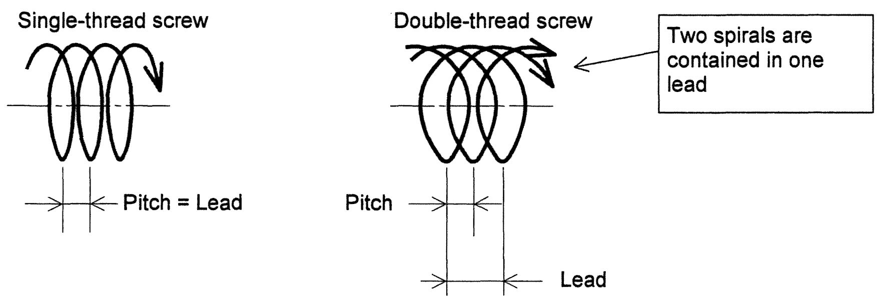
6.17 Multi-thread Cutting

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. The lead of multi-thread screw becomes multiple of (number of threads × pitch).

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 <6.14 Threading (G32)> for command format.

Material outer diameter: ⌀20.0 mm
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

```
00077
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

```
00077
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.

Notes

- 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.
- While the program pre-analysis function is enabled, multi-thread cutting is disabled.

6.18 Custom Macro

This function enables the use of variables, operation instructions, and conditional branch instructions with which you can create highly versatile programs.

Description

Variables

A variable is something that can contain numeric values. For example, when you want to assign the value 12.0 to #500 which is a variable, specify the following in the program:

#500=12.0

This assignment changes #500 to 12.0. When you specify the following after the previous specification, the tool moves to X12.0 at the rapid feed rate:

G0 X#500

Similarly, when you specify the following, the tool moves to X10.0 at the rapid feed rate:

#500=10.0
G0 X#500

Types of variables

Variables are classified into four types by variable number.

Variable number	Type of variable	Function
#0	Always null	This variable is always null, and a value cannot be assigned to it.
#1 to #33	Local variable	These variables can be used individually, and operation results are to be loaded into them. When the power is turned off, the value assigned to each local variable is nullified. When the macro is called, the argument is assigned.
#100 to #149 #500 to #531	Common variable	These variables can be commonly used for different programs. When the power is turned off, the values assigned to the variables #100 to #149 are nullified. However, the values assigned to the variables #500 to #531 are retained even when the power is turned off.
#1000 to	System variable	These variables are used for reading/writing various CNC data such as the current position and tool offset.

Variable range:

The values of local variables and common variables can be used in the range below.

-10^{47} to -10^{-29}
0
$+10^{-29}$ to $+10^{47}$

P/S alarm (No. 111) is issued when operation results are outside the range above.

Omission of a decimal point:

A decimal point can be omitted when the value to be assign to a variable is defined in a program.

Example: When #1=123; is defined, #1 is 123.000.

Formula:

Operations can be performed by using variables.

[Example 1]

```
#500=10.0
#501=20.0
#503=#500+#501
```

The formula above explains that the value 30.0 is assigned to #503.

[Example 2]

```
#500=30.0
#501=sin[#500]
```

The formula above explains that the value 0.5 is assigned to #501.

Use units of degrees to specify angles in trigonometric functions.

The following shows some operation instructions:

Sum #i = #j + #k

Difference #i = #j - #k

Product #i = #j * #k (Use an asterisk * as a multiplication sign.)

Quotient #i = #j / #k (Use a slash / as a division sign.)

Sine #i = sin[#j]

Cosine #i = cos[#j]

Tangent #i = tan[#j]

Square root of #i = SQRT[#j] (The value of $\sqrt{\#j}$ is assigned to #i.)
(i, j, and k: Given integers)

Branching and repetition

You can change a program flow by specifying GO TO statements or IF statements in the program. They are typical statements for changing a program flow.

- Unconditional branching

  The program branches to sequence number n. The sequence number ranges from 1 to 99999.

  GO TO n (n: sequence number)
  n can be a variable.

```
[Example]
N10 <------+
:          |   Returns unconditionally.
GOTO 10 ---+
```

Notes

Be careful that control does not enter the infinite loop because it unconditionally jumps.

- Conditional branch

  When the conditional expression is satisfied, the program branches to the sequence number n. When it is not satisfied, the next block is executed.

  [Example]
  When #500 is greater than 10, the program branches to the sequence number 2.

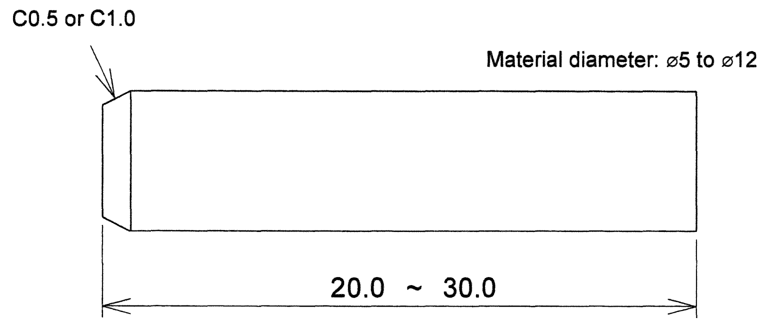
```
 :
IF [#500 GT 10] GOTO 2
G0 X12.0        When the condition is not satisfied
 :              When the condition is satisfied:
N2 G0 Z10.0
G1 X5.0
 :
```

Operators

An operator consists of two alphabetic characters that determine the condition of a conditional expression. (GT is an operator in the example above.)

Operators	Meaning	
EQ	Equal	(=)
NE	Not equal	(≠)
GT	Greater than	(>)
GE	Equal to or greater than	(≥)
LT	Smaller than	(<)
LE	Equal to or smaller than	(≤)

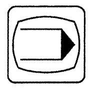
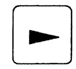
Sample program



Create a versatile program to machine workpieces in different diameters or lengths as shown above. (See the note in the next page for how to enter brackets [ ].)

O0001	Program number	
	$1	$2
#500 = 6.0	Specify the material diameter.	
#501 = 25.0	Specify the workpiece length.	
#502 = 0.5	Specify the value for chamfering.	
#503 = 1.5	Specify the width of a cut-off tool.	
G50 Z-0.1		
M6		
G99		
G0 X[#500 + 1.0] Z-0.6	Move the tool away to a point the material outer diameter + 1.0. (Since 6.0 is assigned to #500, the tool moves to X7.0 in this case.)	
M3 S1=1500		
T0300	Select a tool T03.	
M34		
G0 Z0 T3		
G1 X-0.5 F0.03		
X[#500 - #502*2-1.0] Z-0.5 F0.2 X[#500 + 1.0] Z[#502 + 0.5] F0.03	The tool bit moves to the point that is determined by the specified material outer diameter and the specified value for chamfering, and starts chamfering in the specified size.	
G0 X[#500 + 1.0] T0	Move the tool away to a point of the material outer diameter + 1.0.	
T0400	Select a tool T04.	
G0 X[#500 + 1.0] Z[#501 + #503] T11	Position the X axis to a point of the material outer diameter +1.0, and the Z axis to a point of the workpiece length + the width of cut-off tool.	
M320		
G1 X-0.5 F0.05		
M241		
G99 G1 X-1.5 F0.06		
M05		
M07		
G0 Z-1.5 Z-0.1 T0		
M56		
M02		M02
%		%

Notes

- The program above is given as a sample, so specify machining conditions in consideration of the actual material to be machined, etc. Before running a new program to operate the machine, make a thorough check on the program to prevent interference.
- Be sure to use custom macros according to <the instruction manual issued by FANUC, Ltd.>
- Change machining data in accordance with the material diameter and workpiece length.
- To enter brackets [ ], press the Edit key EDIT , and press the Menu selection key  several times until the menu key [[ ]SWT] appears. Press the menu key [[ ]SWT] to select it. If it is not selected, a right or left parenthesis is entered instead of a bracket even when you press the Bracket key while holding down Shift key.
- When the program pre-analysis function is enabled, you cannot use a custom macro which contains the process extends over several cycles (e.g., a command is executed once in several cycles).

6.19 Medium Pressure Coolant Discharge Device

This device discharges medium pressure coolant by opening the desired valve to rotate the separate pump unit. The dedicated trochoid pump of the device can discharge coolant with force, enabling chip removal effectively.

Command format

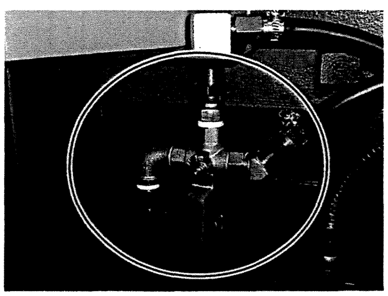
Trochoid pump (separate pump unit)

M452	Trochoid pump ON
M453	Trochoid pump OFF

M430	Valve for oil blower in back spindle ON
M431	Valve for oil blower in back spindle OFF
M432	Valve for drilling holder on front spindle side ON
M433	Valve for drilling holder on front spindle side OFF
M434	Valve for drilling holder on back spindle side ON
M435	Valve for drilling holder on back spindle side OFF
* M436	Valve for the side of guide bushing or ceiling ON
* M437	Valve for the side of guide bushing or ceiling OFF

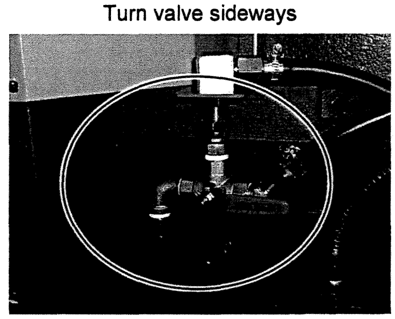
* By switching a valve on the coolant pump, the coolant is discharged from the side of guide bushing or from the pipe at ceiling.

Turn valve downward



Discharges coolant from the pipe at ceiling

Turn valve sideways



Discharges coolant from the pipe at the side of guide bushing

Notes

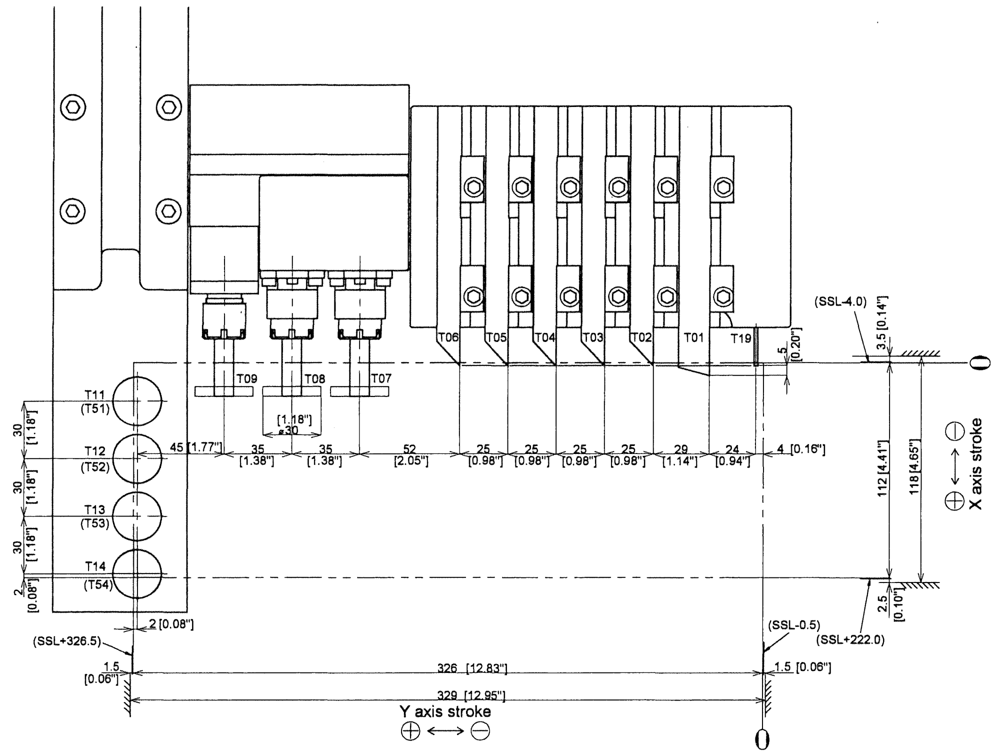
- Before the trochoidal pump can be turned on, any one of the valves should be opened.
- If the valves are closed just after trochoidal pump OFF, internal pressure remains in the machine. Therefore, create the program so that the valves are closed after a certain period has passed from the trochoidal pump OFF.
- Open all the valves to remove the internal pressure at the end of the program. If the internal pressure remains continuously, the pump motor and the valves may be defected.

6.20 Setting the Free Tool Layout Pattern

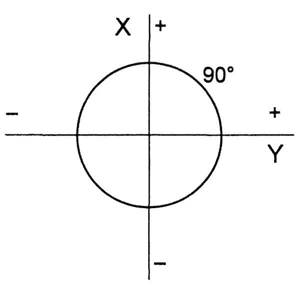
Setting free tools allows tool pattern data to be edited to increase the degree of freedom of tooling and machining.

6.20.1 Angle Setting

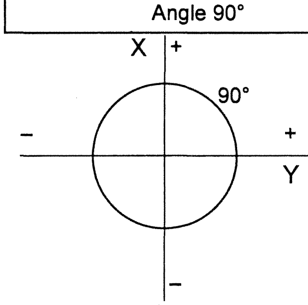
Setting tool angles allows the workpiece coordinates of tools installed on the gang tool post (X1 and Y1 axes) to be changed as shown in the figure below. The angles can range from 0° to 359°.



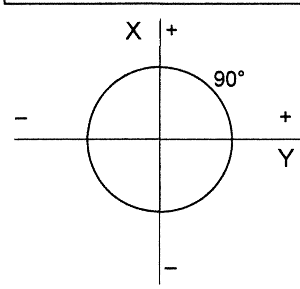
T51 to T54 (Back drilling tool)
Default setting Tool No.4 (end-face), Angle 90°

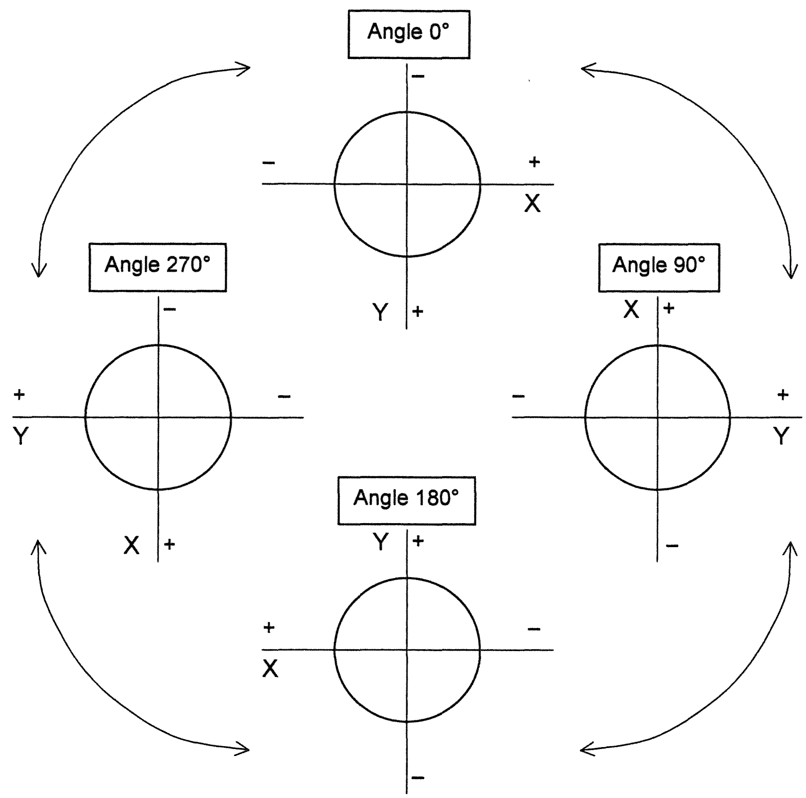


T11 to T14 (Front drilling tool)
Default setting Tool No.4 (end-face), Angle 90°



T01 to T09 (Rotary tool)
Default setting Tool No.1 (Right-handed)
Tool No.3 (Cross)
Angle 90°

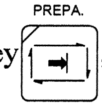
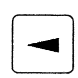
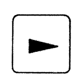
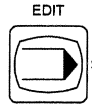
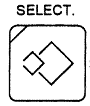


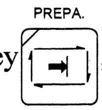


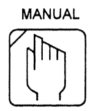
Notes

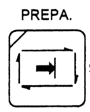
- Setting any of the tool angles to other than 90° or 270° causes the rapid feed rate on the X1 and Y1 axes to be limited to 18 m/min.
- If any of the tool angles is set to other than those in <6.20.3 Standard tool pattern data>, the selection operation of the tool is different from the normal tool selection. Use arguments appropriately to avoid interference with the material.

Procedure

1. Press the Preparation key PREPA. , press the menu selection key   to open [T-PATT], and take note of the current setting.
   (See also <6.20.3 Standard tool pattern data>.)
2. Press the Edit key EDIT , and select a program number you want to modify.
3. Press the menu key [M. DATA], and specify "Free Tool" for the front machining tool holder.
4. Press the Program select key SELECT. , and select a program number.

5. Press the Preparation key 

, and take note of the current setting. Then, specify the new angles.
6. Set tool set data for tool breakage detection device (T19).
7. Press the Manual operation key 

, put a cursor onto the tool that has been changed to confirm the settings for X and Y axes.
8. Press the Preparation key 

, and confirm that the settings for all the tools are correct.

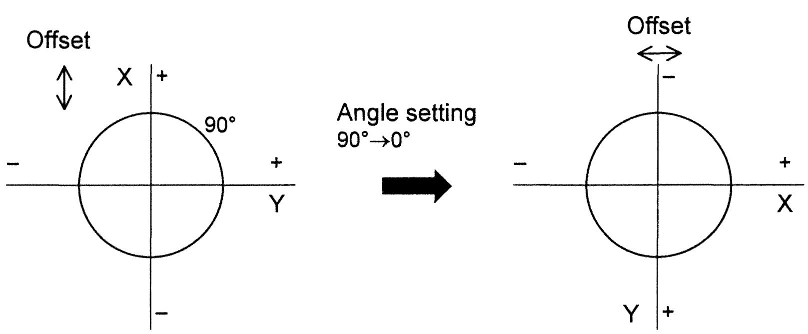
Note

In the free tool layout pattern, the reference position (zero point) for the tool holder is shifted by 16 mm if the K2 argument is specified.

CAUTION

Specifying an incorrect value or calling an undefined tool may cause an interference.

For example, when some compensation in the Y axis direction is required in the figure below, interchange the X and Y axes in the free tool angle setting. This enables operations in the horizontal and vertical directions to be performed by X and Y commands, respectively. Accordingly, an offset input in the X direction is made valid in the horizontal direction.



However, if option "Y axis offset" is not set, the offset in the Y axis direction, or the vertical direction, cannot be used.

6.20.2 Tool setting

Tools can be set as shown in the tables below. Type display numbers of mounted tools.

Number	Tool name	Description
0	No tool	No setting of tool type
1	Right	Right-handed tool
2	Left	Left-handed tool
3	Cross hole	Cross rotary tool
4	End face hole	End face drilling or end face hole rotary tool

6.20.3 Standard tool pattern data

To set a free tool, see the standard tool pattern data described below and set proper values if necessary.

Tool holder

- BTF2212, BTF2213

	X-shift value	Y-shift value	Angle	Distance	Tool type
T01	221.000	–56.000	90	0.000	1
T02	221.000	–114.000	90	0.000	1
T03	221.000	–164.000	90	0.000	1
T04	221.000	–214.000	90	0.000	1
T05	221.000	–264.000	90	0.000	1
T06	221.000	–314.000	90	0.000	1

- BTF2412, BTF2413

	X-shift value	Y-shift value	Angle	Distance	Tool type
T01	221.000	–50.000	90	0.000	1
T02	221.000	–108.000	90	0.000	1
T03	221.000	–158.000	90	0.000	1
T04	221.000	–208.000	90	0.000	1
T05	221.000	–258.000	90	0.000	1

- BTF2216

	X-shift value	Y-shift value	Angle	Distance	Tool type
T01	221.000	–64.000	90	0.000	1
T02	221.000	–124.000	90	0.000	1
T03	221.000	–184.000	90	0.000	1
T04	221.000	–244.000	90	0.000	1
T05	221.000	–304.000	90	0.000	1

Rotary tool

- U31B

(BTF2212 BTF2313)	(BTF2216)	X-shift value	Y-shift value	Angle	Distance	Tool type
T07	T06	221.000	–418.000	90	0.000	3
T08	T07	221.000	–488.000	90	0.000	3
T09	T08	221.000	–558.000	90	0.000	3
		205.000	–558.000	90	0.000	4

- U32B

(BTF2412 BTF2413)	–	X-shift value	Y-shift value	Angle	Distance	Tool type
T06	–	221.000	–348.000	90	0.000	3
T07	–	221.000	–418.000	90	0.000	3
T08	–	221.000	–488.000	90	0.000	3
T09	–	221.000	–558.000	90	0.000	3
		205.000	–558.000	90	0.000	4

Sleeve holder

- BDF2005

	X-shift value	Y-shift value	Angle	Distance	Tool type
T11	184.000	–648.000	90	0.000	4
T12	124.000	–648.000	90	0.000	4
T13	64.000	–648.000	90	0.000	4
T14	4.000	–648.000	90	0.000	4
T51	184.000	–648.000	90	0.000	4
T52	124.000	–648.000	90	0.000	4
T53	64.000	–648.000	90	0.000	4
T54	4.000	–648.000	90	0.000	4

Back sleeve holder

	X-shift value	Y-shift value	Tool type
T31	–177.000	0.000	4
T32	–255.000	0.000	4
T33	–333.000	0.000	4
T34	–411.000	0.000	4

Tool breakage detection device

- BTF2212 + U31B, BTF2213 + U31B

	X-shift value	Y-shift value	Angle	Distance	Tool type
T19	221.000	–8.000	90	0.000	1

- BTF2216 + U31B

	X-shift value	Y-shift value	Angle	Distance	Tool type
T19	221.000	–16.000	90	0.000	1

- BTF2412 + U32B, BTF2413 + U32B

	X-shift value	Y-shift value	Angle	Distance	Tool type
T19	221.000	–2.000	90	0.000	1

6.21 Machine Coordinate Value Specification (G53)

The machine coordinate value specification (G53) moves the material to the specified position on the material coordinate system.

Command format

```
G53 X[    ] Z[    ]
```

X: Specify the X axis machine coordinate. The specified axis of axis control group moves to the machine coordinate specified by X.

Z: Specify the Z axis machine coordinate. The specified axis of axis control group moves to the machine coordinate specified by Z.

Note

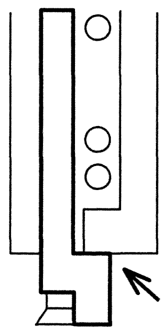
The machine coordinate value specification (G53) to the gang tool post (X1 and Y1 axes) causes an alarm to occur.

6.22 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. Press the menu key [T-PATT], and specify "2. LEFT" in [TOOL] for the tool to which the Y-axis holder is to be mounted. (The initial setting is "1. RIGHT".)

   * Initial setting for the tool is "Right". To change it to "LEFT", the axis move direction in "DIA" and "CORE" process is automatically changed as shown in Fig. A and Fig. B.

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. (Fig. 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 D] being selected, press the Start key continuously until the cursor reaches the position [CORE D]. Then, align the outer diameter of the material with the tool nose. (Fig. B)

   * When adjusting the position by [CORE D], 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.

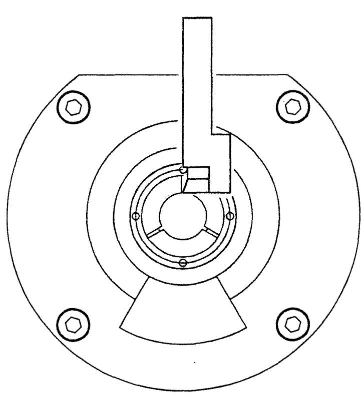


Fig. A

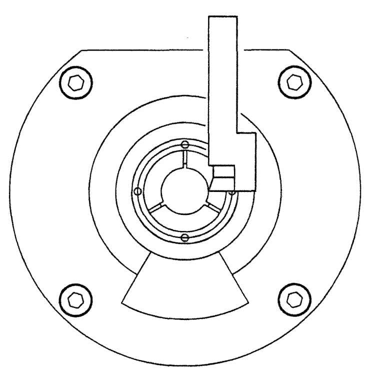


Fig. B

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 Fig. A.
G50 W-3.0
G0 X21.0 Y21.0 Z20.0 T3
X0.0 ...................................................... Moves to the position shown in Fig. 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
```

Notes

- To create a program by which the cutting is performed in Y direction as shown in the sample program, an optional "Y-axis offset" option is required to use compensation (offset) of Y direction.
- By exchanging the X axis with Y axis using the free tool layout, you can create a program for Y axis, however, the idle time for calling gang tool is greatly affected because the rapid feedrate for X1 and Y1 axes is changed to 18 m/min.

6.23 Program Pre-analysis Function

This function is intended to improve the running efficiency by analysing the machining program on NC unit prior to actual running. Using this function can reduce the idle time during running of the machining program.

6.23.1 Restrictions

The program pre-analysis function is set to [ENABLE] by the default. However, some restrictions are imposed in creating a program. If conditions described below is not satisfied, the program pre-analysis function does not work. To run such a program, you need to disable the program pre-analysis function. In addition, this function may be disabled on your machine depending on the tooling performed at our factory.

* See <6.23.3 Enabling or disabling program pre-analysis function> for enabling or disable the function.

Restrictions

- The following functions are disabled while the program pre-analysis function is enabled:

  – Input of on-drawing dimensions
  – Exact stop check (G9)
  – Milling interpolation
  – Multi-thread cutting
  – Variable-lead cutting
  – Integrated canned cycle II
  – Tool life management II

- The program pre-analysis function cannot be used in a program that contains custom macro or subprogram in which the tool path changes at every cycle.

  If the program pre-analysis function is executed, the machine repeatedly runs the program which was analyzed and stored immediately before actual running. Accordingly, if the program is to be changed at the later cycle, the machie runs the program in the first cycle repeatedly.

  For example, if you create a conditional macro program to change a tool number in the fourth cycle, the tool number will not be changed and the operatio is continued while the program pre-analysis function is enabled.

- Restriction on number of blocks

  If the program created for each axis control group contains more than 1,000 blocks, an alarm "BLOCK COUNT OVER" occurs when executing the program pre-analysis. (The number of allowable blocks depends on the program contents, 1,000 blocks is standard.)

  Consider the followings about the number of blocks.

  - Count a command in single line as one block.
  - Count the number of blocks in subprogram, if M98 (subprogram) is used.
  - Count the repetition by the custom macro.
  - The A220-specific macro (T code, M code, and G code) contains internal commands for several blocks. If these codes are used, "BLOCK COUNT OVER" alarm may occur even if the number of blocks in the program does not exceed 1,000 blocks.

- Use M109 Q□□□□ (□□□□: Program number) for general-purpose material change program.

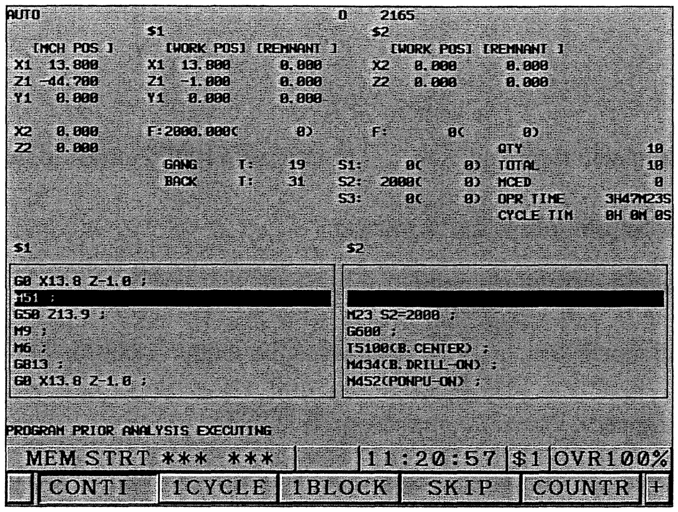
  While the program pre-analysis function is enabled, subprogram M98 cannot be used in material change program. Be sure to use M109 Q□□□□. For details, see <6.2.2 General-purpose Material Change Program (by user program)>.

M8	Material change ON
M8	Material change ON
/M109 Q□□□□	General-purpose material exchange
M9	Material change OFF

- When executing the Last Program command (G999) or Material Exchange command (M108, M109), the machine coordinate of start point must be identical to that of end point. If not, an alarm occurs in program pre-analysis process. In such a case, disable the program pre-analysis function.

6.23.2 Starting program pre-analysis function

The program pre-analysis function is launched in the following conditions:



- On-machine program check mode

  Starts immediately before the cycle operation. The function checks and analyzes overall program at every cycle to detect conflict between start position and end position or over-travel of each axis. The function analyzes the program in on-machine program check process, however, the program pre-analysis function does not work during actual program running.

- Automatic operation mode

  Starts immediately before the continuous operation. The function checks, analyzes and stores the program. Since the overall program is analyzed, the idle time may be reduced in comparison with the normal analyze that performs analysis process every one block.

- Other conditions

  – When optional block skip function is switched from enabled to disabled or vice versa.

  – When the machining program or machining data is changed

  – When menu key function is executed in Preparation mode

  – When the tool set data in Preparation mode is changed

  – When a program is selected

  – When parameter, offset, and custom macro variable data is changed

Note

If the offset data or custom macro variable data is changed during continuous operation, the program pre-analysis function restarts at the beginning of the next cycle. At that time, the automatic operation halts until the analysis completes. Upon completion of analysis, the continous operation resumes automatically.

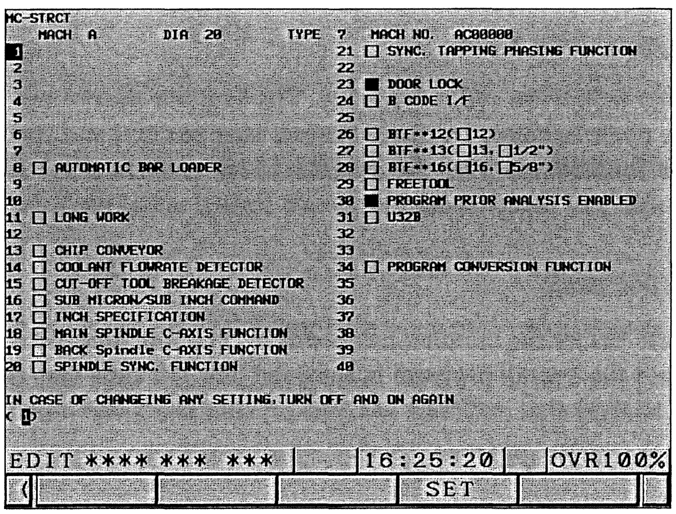
6.23.3 Enabling or disabling program pre-analysis function

Use the Machine structure menu in Maintenance screen to enable or disable the program pre-analysis function.

Procedure

1. Press the Program selection key to enter the EDIT mode. (The current mode is indicated at the bottom left of the screen.)
2. Display the Machine structure menu.

   Press the Maintenance key, then press the menu selection key until the menu key [STRCT] appears. Press the menu key [STRCT].
3. Specify parameters for the program pre-analysis function appropriately.



   When the screen shown above appears, press the menu key [SET].

   Use the cursor move key to move the cursor onto "30 PROGRAM PRIOR ANALYSIS ENABLED", then press the key to reverse the checkbox.

   ■ : Enabled, □: Disabled
4. Be sure to cycle the power upon completion of setting. In addition, be sure to select a program again.

(Blank page)

Product Code	C	–	A	2	2	0	P	L	VII					
Document Code	2	E	1	–	0	6	0	1						

7. Sample Programming [Secondary Machining]

7.1 Secondary Machining

7.1.1 Difference between Turning and Secondary Machining

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 secondary machining, contrary to this turning, the tool is rotated and the workpiece is fixed.

Turning: The workpiece is rotated and cut.

Secondary machining: The workpiece is fixed and the tool is rotated for machining.

7.1.2 Machining Process Order

In the machining process of products, all the machining steps including secondary machining are carried out in one cycle, like single turning. The machining order is fundamentally determined by the following principles.

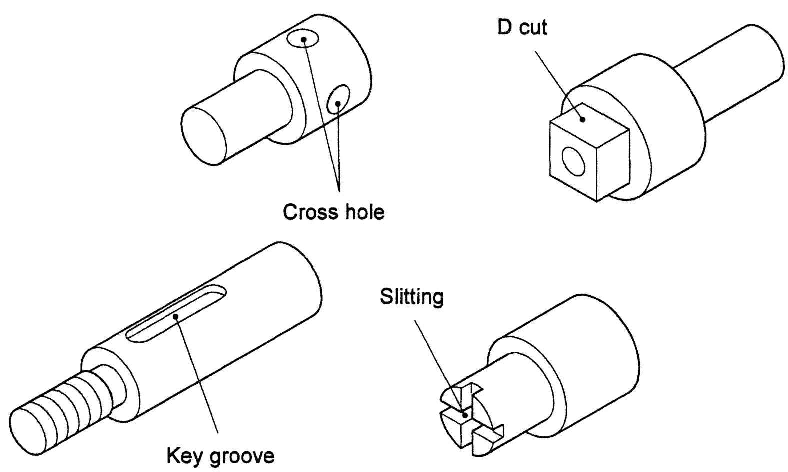
Major principles for machining order

- The workpiece is machined from left to right.
- The center hole drilling and end face secondary machining are carried out first.
- When both turning and secondary machining are needed in the same place, the secondary machining follows turning.

The tooling layout should be prepared based on the above principles.

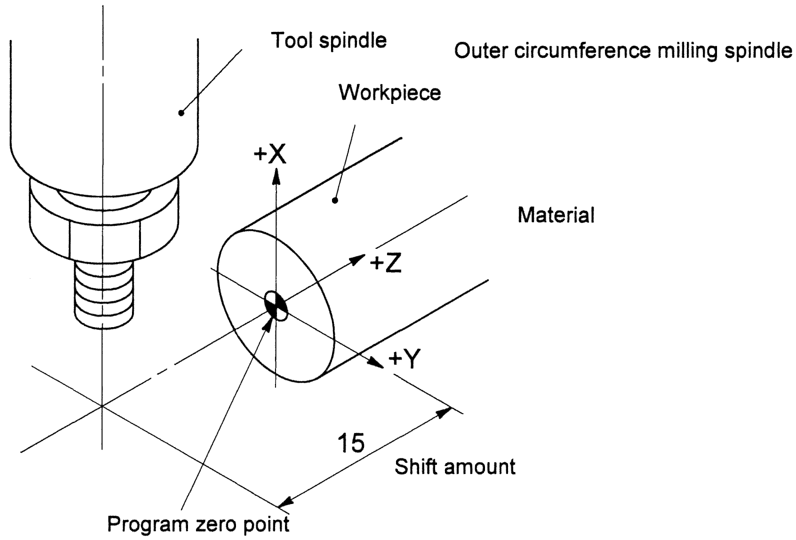
The number of tools used in the secondary machining is determined by the holder. For details, see <Chapter 11 Tooling>.

Machining example



7.1.3 Coordinates for Secondary Machining

To create a secondary machining program, it is necessary to understand the coordinates. The positional relationship between workpiece and tool spindle is as illustrated below.



Coordinates and signs

The coordinates for the secondary machining program are determined based on the same philosophy as turning. Since the Y axis can also be moved in the secondary machining, the machining process with three axes, X, Y, and Z, is possible.

To program the X and Y coordinates, be sure to use the diameter. For X and Z axes, the program zero point is usually the same as turning.

For the Y axis, the workpiece center is defined as zero point.

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 shift

The tool spindle is shifted by 15 mm on the Z axis in the above illustration. Therefore, the coordinates are programmed as they are, the workpiece is machined at the position shifted by 15 mm.

The coordinate shift function is used for the distance between the program zero point of the Z axis and the tool spindle center.

After the secondary machining, be sure to cancel the coordinate shift.

Write a program in $1.

Command format

`G50 W-15.0`	Coordinate shift
`G50 W15.0`	Coordinate shift cancel

7.2 Main Spindle Indexing

7.2.1 Main Spindle Indexing (M28, M20)

In the secondary machining, the spindle can be indexed every 15°.
(1° indexing is optional.)

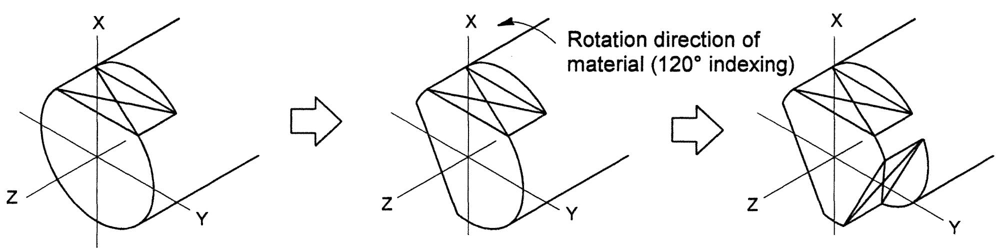
Command format

M28 S☐	Main spindle indexing
M20	Main spindle indexing cancel

Argument

S: Indexing angle (Available range: from 0 to 359)

Machining example



Sample program

M05	Main spindle stop
G04 U0.5	Dwell
M28 S0	Main spindle indexing
G98	
M80 S☐	Per minute feed and forward rotation tool spindle
⋮	Machining program
M82	Tool spindle stop
M20	Main spindle indexing cancel
G4 U0.5	Dwell
M03 S1=☐ G99	Conducts main spindle forward rotation and per minute rotation

Shift of orientation zero point

The orientation zero point can be shifted by changing the value of parameter No. 4077 ($1 side). Set the value between 0 to 360 degrees proportional to 0 to 4096.

Notes

- "S$\boxed{}$" should be integer in absolute value ranging from 0 to 359.
- During indexing, the rotating direction of the spindle is indexed by the rotating direction of the preceding spindle rotation command. If another indexing command will follow the current indexing command, the indexing is made via a short route.
- Even while the spindle is rotating, the indexing command can be issued directly without stopping the spindle.
- For the first indexing after power-on, the spindle is operated by more than one rotation to find the zero point. For the second and following indexing steps, the zero point indexing is made within one rotation.
- To return to turning, issue the main spindle rotation command (M03, M04). The indexing is canceled and the spindle rotates.
- Reverse running is inhibited for M28 (Enables main spindle indexing) and M20 (Cancels main spindle indexing).

7.2.2 Back Spindle Indexing (M78, M79)

The back spindle can be indexed at 1° intervals.

Command format

`M78 S□`	Back spindle indexing
`M79`	Back spindle indexing cancel

Argument

S: Indexing angle (Available range: from 9 to 359)

Sample program

```
M25 ...................................... Back spindle stop
G04 U0.5 ................................. Dwell
M78 S0 ................................... Back spindle indexing
 ⋮                                         Machining program
M79 ...................................... Back spindle indexing cancel
G4 U0.5 .................................. Dwell
M23 S2=□ ................................. Back spindle forward rotation
```

Shift of orientation zero point

The orientation zero point can be shifted by changing the value of parameter No. 4077 ($2 side). Set the value between 0 to 360 degrees proportional to 0 to 4096.

Notes

- Specify the absolute integer between 0 and 359 for S□.
- During indexing, the rotating direction of the back spindle is indexed by the rotating direction of the preceding back spindle rotation command. If another indexing command will follow the current indexing command, the indexing is made via a short route.
- Even while the back spindle is rotating, the indexing command can be issued directly without stopping the back spindle.
- For the first indexing after power-on, the spindle is operated by more than one rotation to find the zero point. For the second and following indexing steps, the zero point indexing is made within one rotation.
- To return to turning, issue the back spindle rotation command (M23, M24). The indexing is canceled and the back spindle rotates.
- Reverse running is inhibited for M78 (Enables back spindle indexing) and M79 (Cancels back spindle indexing).

7.2.3 Main Spindle C Axis (M18, M20)

In the secondary machining process, the main spindle C axis can be specified (in 0.001-degree units).

The command turns the spindle C axis on , indexes zero point C0, set that position as C0 in the coordinate system.

Command format

`M18 C□`	Main spindle C axis command
`M20`	Main spindle C axis command cancel

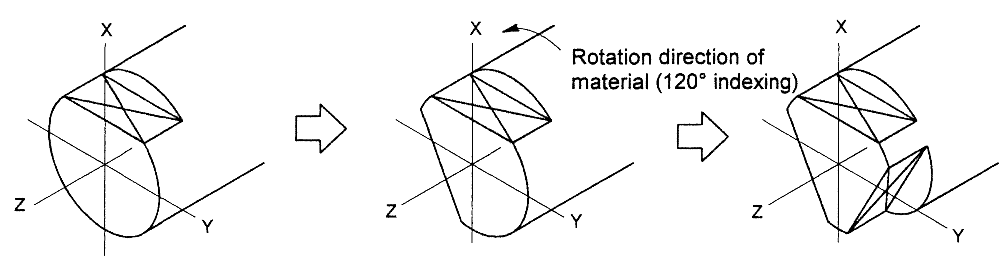
- This command rotates the material from zero point C0 to □ degrees.
- A fixed direction can be specified for the indexing.
- The spindle forward rotation is prescribed as "forward rotation" by an indexing + (positive) sign, and the spindle reverse rotation as "reverse rotation" by a – (negative) sign. Moreover, when it is viewed from the end face of the material, counterclockwise rotation is prescribed as "forward rotation", and the clockwise rotation as "reverse rotation."
  In this case, the + (positive) sign is omitted.
- After zero point "C0" is indexed, any indexing command can be specified.
  The indexing can be specified in the range of –99999.999 to 99999.999 degrees.
- After the M18 C□ command is specified, the usual C axis command will become effective.

Example: `G50 C□ (H□)` H is an incremental command.

`G00 C□ (H□)`

`G01 C□ (H□) F□`

Machining example



0° indexing 120° indexing 240° indexing

Sample program

```
M05 ..................................... Main spindle stop
M18 C0 ................................. Main spindle C axis
G98 ..................................... Feed per minute
M80 S3=[    ] ................... Tool spindle forward rotation
 ⋮                                          Machining program
M82 ..................................... Tool spindle stop
M20 ..................................... Main spindle C axis cancel
M03 S1=[    ] G99 ........... Main spindle forward rotation and per minute rotation
```

Commands for indexing

- Absolute commands

```
M18 C0
↓
M18 C120.001
↓
M18 C240.0
```

- Incremental command

```
M18 C0
↓
G0 H120.0
↓
G1 H20.0 F[    ]
```

Shift of C axis zero point

The C axis zero point can be shifted by changing the value of parameter No. 4135 ($1 side). Set the value between 0 to 360 degrees proportional to 0 to 360000.

Notes

- For the machine coordinate and workpiece coordinate for C axis, the screen can display the value ranging from –999.999 to 9999.999. If the specified value is out of range, the indication disappears, but the operation is conducted.
- Even while the main spindle is rotating, the indexing command can be issued directly without stopping the main spindle.
- Reverse running is inhibited for M18 (Main spindle C axis command) and M20 (Main spindle C axis command cancel).
- Main spindle C axis function is canceled when M5 (Main spindle stop) command is executed.

7.2.4 Back Spindle C Axis (M48, M79)

In the secondary machining process, the back spindle C axis can be specified (in 0.001-degree units).

The command turns the back spindle C axis on, indexes zero point C0, set that position as C0 in the coordinate system.

Command format

`M48 C☐`	Back spindle C axis command
`M79`	Back spindle C axis command cancel

- This command rotates the material ☐ degrees from zero point C0.
- A fixed direction can be specified for indexing.
- The back spindle forward rotation is prescribed as "forward rotation" by an indexing + (positive) sign, and the spindle reverse rotation as "reverse rotation" by a – (negative) sign. Moreover, when it is viewed from the end face of the material, counterclockwise rotation is prescribed as "forward rotation", and the clockwise rotation as "reverse rotation."
  In this case, the + (positive) sign is omitted.
- After zero point "C0" is indexed, any indexing command can be specified.
  The indexing can be specified in the range of –99999.999 to 99999.999 degrees.
- After the M48 C☐ command is specified, the usual C axis command will become effective.

Example: `G50 C☐ (H☐)` H is an incremental command.

`G00 C☐ (H☐)`

`G01 C☐ (H☐) F☐`

Shift of C axis zero point

The C axis zero point can be shifted by changing the value of parameter No. 4135 ($2 side). Set the value between 0 to 360 degrees proportional to 0 to 360000.

Notes

- For the machine coordinate and workpiece coordinate for C axis, the screen can display the value ranging from –999.999 to 9999.999. If the specified value is out of range, the indication disappears, but the operation is conducted.
- Even while the back spindle is rotating, the indexing command can be issued directly without stopping the back spindle.
- Reverse running is inhibited for M48 (Back spindle C axis command) and M79 (Back spindle C axis command cancel).
- Back spindle C axis function is canceled when M25 (Back spindle stop) command is executed.

7.2.5 Tool spindle rotation

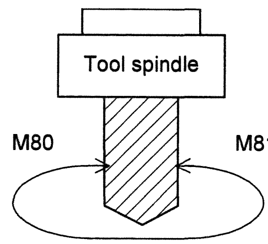
These commands rotate the tool spindle T0700 (T0600) to T0900 in the forward or reverse direction and stop it.

Counterclockwise rotation when viewed toward the spindle is regarded as forward rotation.

Pressing the All spindles stop key on the operation panel can also stop the tool spindle rotation.

Command format

`M80 S3=☐` or `M80 S☐`	Tool spindle rotation forward
`M81 S3=☐` or `M81 S☐`	Tool spindle rotation reverse
`M82`	Tool spindle rotation stop



Notes

- To rotate the tool spindle, specify M80, M81, and speed (S☐ or S3=☐) in the same block. Enter the tool spindle rotation command in one block as single command.
- Issuing only M80 and M81 is not permitted. Be sure to specify speed (S3=☐) together with them.
- To change the tool spindle speed, specify M80, M81, and speed (S☐ or S3=☐) in the same block.
- If the tool spindle forward rotation, tool spindle reverse rotation, and stop commands are specified successively, incorrect operation may be performed. To issue these commands successively, provide a dwell among them.

  Example:
  ```
  M82;
  G04 U0.1;
  M80 S3=1000;
  ```

  Specify the tool spindle rotation or stop command (M80, M81, or M82) in axis control group 1 ($1). If specified in axis control group 2 ($2), an alarm occurs.
- Specify the positive value for rotation speed (S argument). If specified with negative value, an alarm occurs.
- To use the tool spindle in continuous operation, the rated speed as shown below must be specified.
  Specifying a speed exceeding the rated speed for machining may cause an overload alarm to occur.

Rotary tool driving device	Rated speed	Maximum speed
U31B (3-rotary tool specification)	6,000	8,000
U32B (4-rotary tool specification)	4,500	6,000

- If the speed exceeding the maximum speed is specified in S argument, an alarm occurs.
- G99 (feed per rotation) cannot make feeding. Use G98 (feed per minute) for feeding.

7.3 Calculating the Command Speed

7.3.1 Calculating the Spindle Command Speed

The spindle speed can be calculated from the following formula (A).

$$N = \frac{V}{\pi D} \times 1{,}000 \quad \text{......................} \quad (A)$$

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

Calculation example

Secondary machining spindle

Assuming that the workpiece is machined with a ⌀4 end mill.

Material: Brass
Cutting speed: 25 (m/min)
Tool diameter: ⌀4 (mm)

Using formula (A), the speed is calculated as follows.

$$N = \frac{25}{3.14 \times 4} \times 1000 = 1990 \approx 2000 \ (min^{-1})$$

- The 100's order must be rounded off to get the speed as shown above.
- If the calculated speed exceeds the maximum, unconditionally use the maximum speed (6000 min^{-1}).

7.3.2 Calculating the Tool Feedrate

The tool feedrate (mm/min) can be obtained from the equations shown below based on the calculation result from formula (A). Note that the applicable equation is determined by the tool to be used.

Tool feedrate

End mill and slitting cutter

$F = N \times fz \times Z$ .... (B)

F: Feedrate
N: Spindle speed
fz: Feedrate per blade
Z: Number of blades (blade/rev)

Drill

$F = N \times fr$ .... (C)

F: Feedrate
N: Spindle speed
fr: Feedrate per rotation

Calculation example

Secondary machining spindle

Assuming that the workpiece is machined with a ⌀4 end mill.

Material:	Brass
Speed:	2000 (min^{-1})
Feedrate per blade:	0.065 (mm/blade)
Number of blades:	2 (blades)

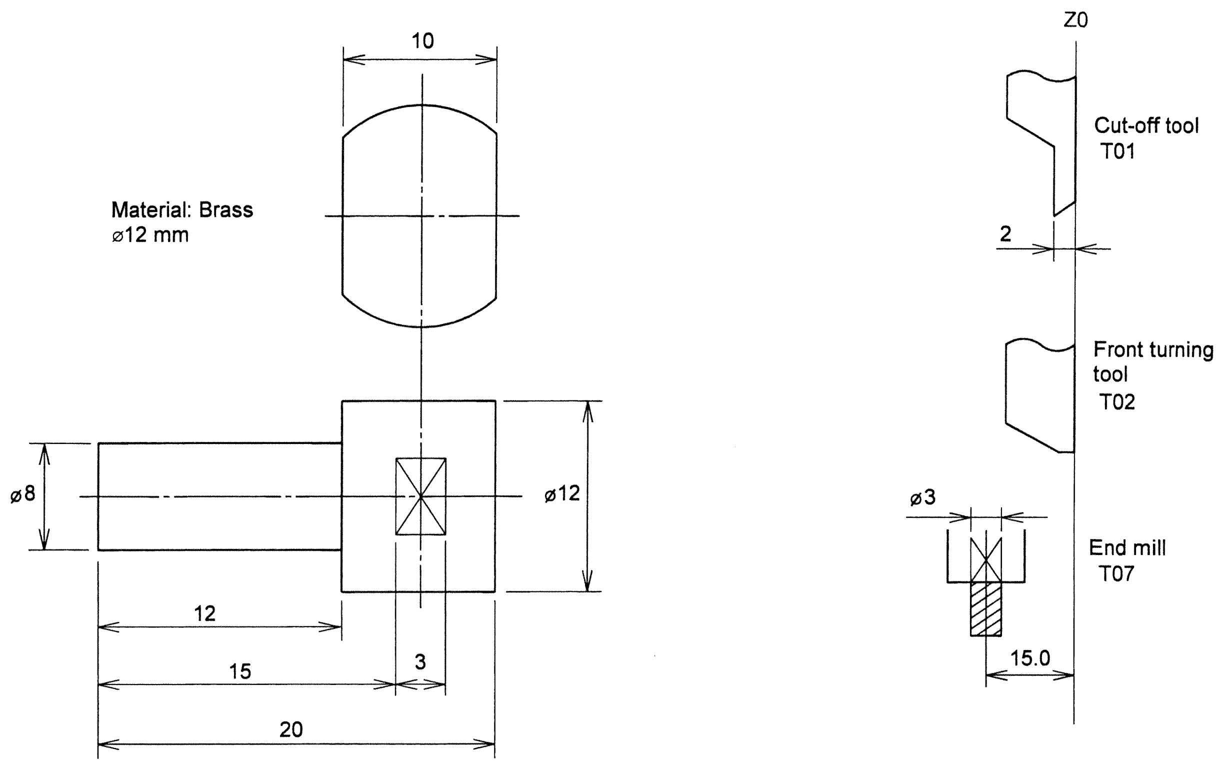
Using formula (B), the feedrate is calculated as follows.

$F = 2000 \times 0.065 \times 2 = 260$ (mm/min)

7.4 Partial Program for 2-side Width Machining (Including Indexing)

The sample program described below shows an extracted secondary machining.

Machining layout



Sample program

```
T0700 ..........................................  Selects the tool.
G50 W-15.0 .....................................  Shifts the tool position on the longitudinal coordinate.
M05
M28 S0 .........................................  Main spindle indexing
G98
M80 S3=[    ] ..................................  Tool spindle forward rotation
G00 Y-10.6 Z16.5 T[    ] .......................  Feeds the Y and Z axes rapidly for positioning
X10.0 ..........................................  Positions the X axis
G01 Y10.6 F[    ] ..............................  Machines one side of the 2-side width
M28 S180 .......................................  Conducts 180° indexing
Y-10.6 .........................................  Machines the other side of the 2-side width
G00 X13.0 M60 ..................................  Moves the tool in radial direction
M82 ............................................  Stops tool spindle
M20 ............................................  Main spindle indexing cancel
G50 W15.0 ......................................  Cancels shifting the tool position on the longitudinal coordinate
M03 S1=[    ] G99
```

Necessary calculation process and philosophy are described in the next page.

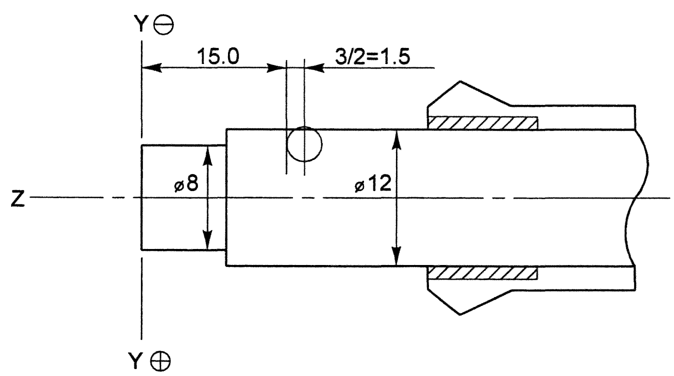
Description

The spindle speed N is calculated as follows.

$$N = \frac{25}{3.14 \times 3} \times 1000 = 2653 \approx 2700\ (\text{min}^{-1})$$

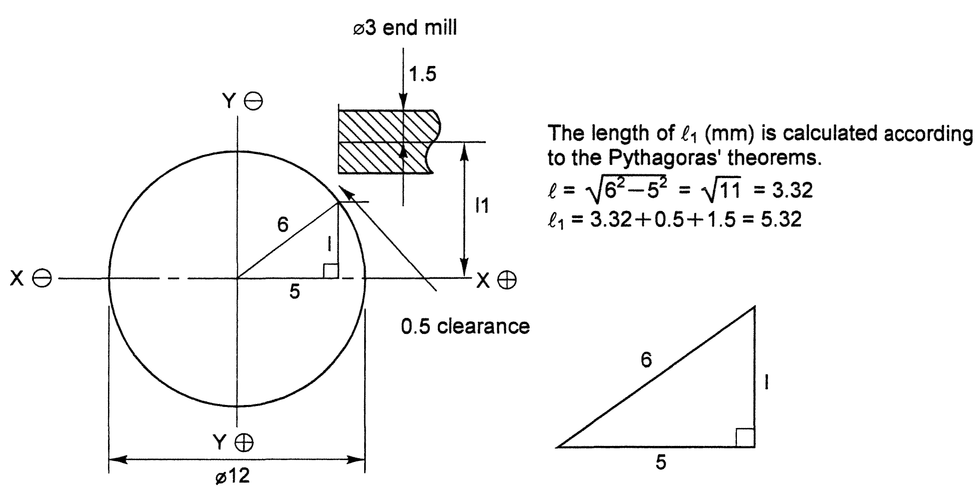
Therefore, the feedrate F of the end mill is calculated as follows.
$F = 2700 \times 0.05 \times 2 = 270$ (mm/min)

The tool is positioned in the longitudinal direction as illustrated below.



Therefore, the Z coordinate of the position is calculated as follows.
$15+3$ (diameter of end mill) $\div 2 = 15+1.5 = 16.5$

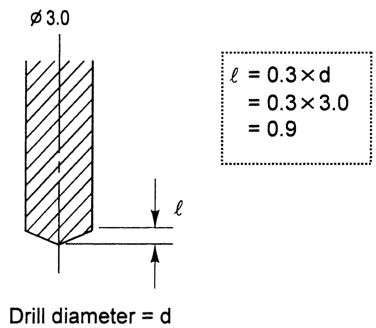
From the illustration below, the Y coordinate (diameter is specified) is calculated as follows.
$Y = (\ell+0.5+1.5) \times 2 = (3.32+0.5+1.5) \times 2 = 10.64 \approx 10.6$



7.5 Through-hole Machining

This section describes the example of drilling a ⌀3.0 cross through-hole on a ⌀12.0 workpiece.

The stroke of the X1 axis is up to X – 5.0 and the normal tool set cannot drill a through-hole on the workpiece. This means that the drill on the tool set must be shifted toward the negative direction long.



If the shift is not considered, the X value on the program is "– 12.0– (1.05 × 2) = – 14.1".

X – 18.1 will cause an X axis overrun.

Since the stroke can be shifted up to X – 5.0, the value to be shifted will be "18.1 – 5.0 = 13.1". Therefore, if this amount is specified in "DIA" field on the Preparation screen, specifying X – 5.0 in the program can drill the through-hole on the workpiece. See <6.2.3.5 Diameter (Rotary tool)> in Operator's Manual for setting procedure.

Although it is enough to shift the diameter by 9.1 mm, we recommend shifting the tool by 11.0 mm considering the margin.

Notes

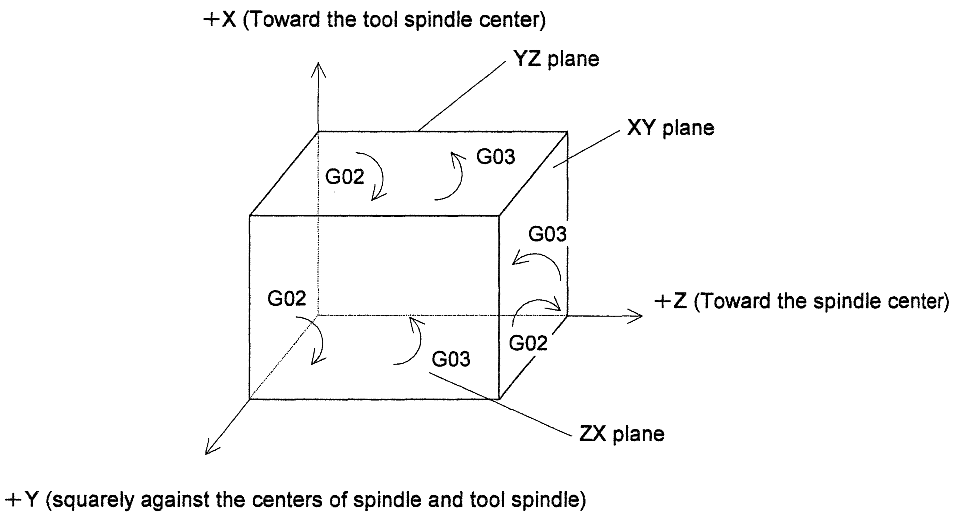
- Enter and set the shift amount on the Tool Set screen.
  (See <6.2.3.5 Diameter (Rotary tool)> in Operator's Manual for setting procedure.)
- Select a tool in consideration with the maximum diameter specified in "DIA" for gang tool set. You do not need to move the tool far away in the program, or increase the value for tool positioning point in the machining data.

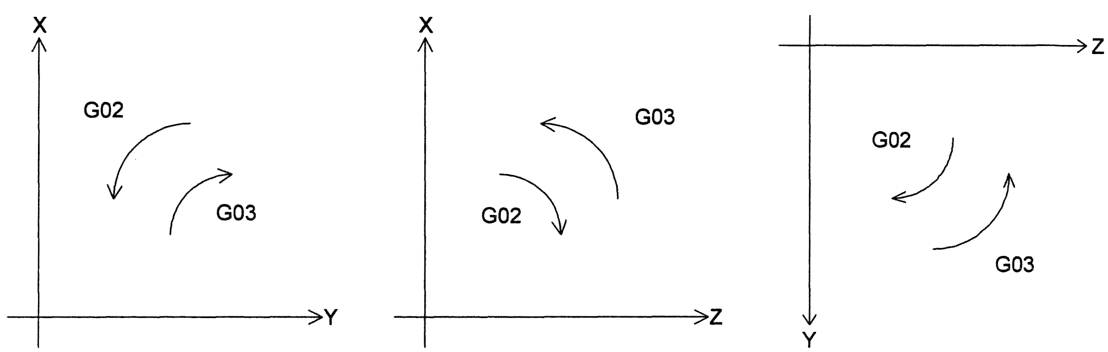
7.6 Circular Interpolation

In the circular interpolation of the secondary machining, the circular interpolation feed is available on the following three planes.

Command format

G17	XY plane
G18	ZX plane (plane selection mode in normal turning)
G19	YZ plane





G17 (X−Y) plane G18 (Z−X) plane G19 (Y−Z) plane

7.6.1 Selecting the Plane

After the machine is turned on or reset, the ZX plane (G18) is normally selected as default. The circular interpolation (G02, G03) is performed on this plane in turning.

If you want to conduct the circular interpolation on other plane in the secondary machining, the plane selection command (G17 or G19) must be issued.

To return to turning, be sure to reselect the ZX plane (G18).

7.6.2 Offsetting the Tool Diameter

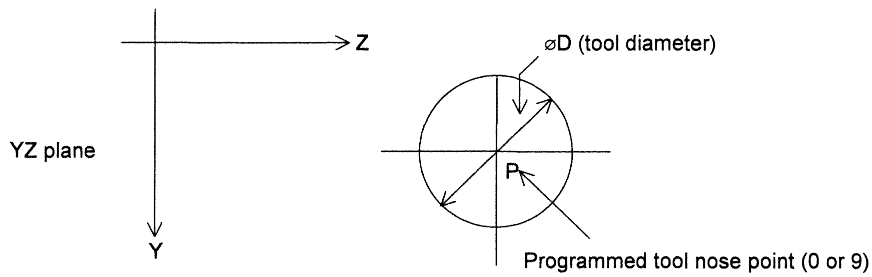
To use the tool diameter offset, program it with the tool nose R offset function of the turning process.

Command format

G40	Cancels the tool diameter offset (tool nose R offset) mode
G41	Enables the tool diameter left offset (tool nose R left offset) mode
G42	Enables the tool diameter right offset (tool nose R right offset) mode

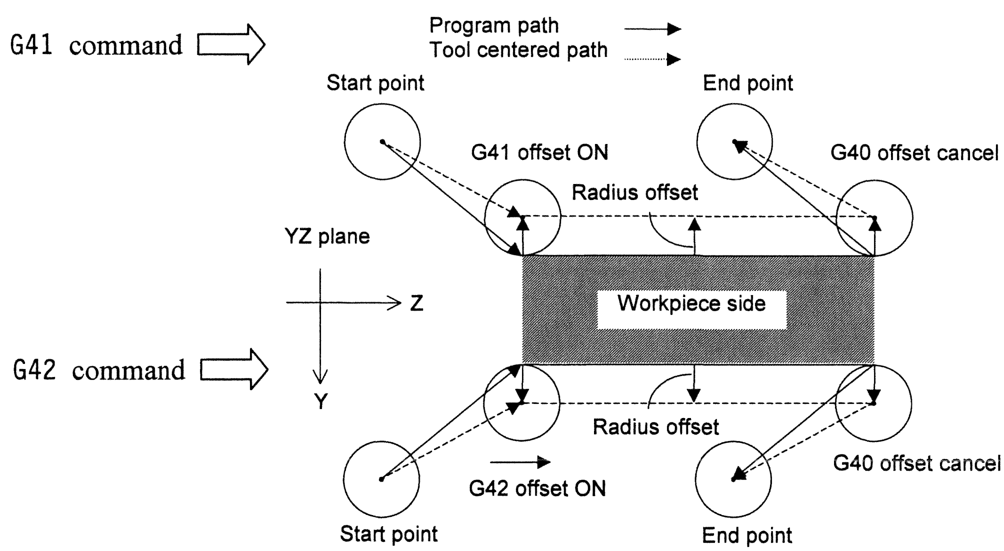
The illustration below shows the relationship between the tool diameter and the programmed cutting position.

Set the virtual tool nose number 0 or 9 to the tool diameter offset registration field beforehand.

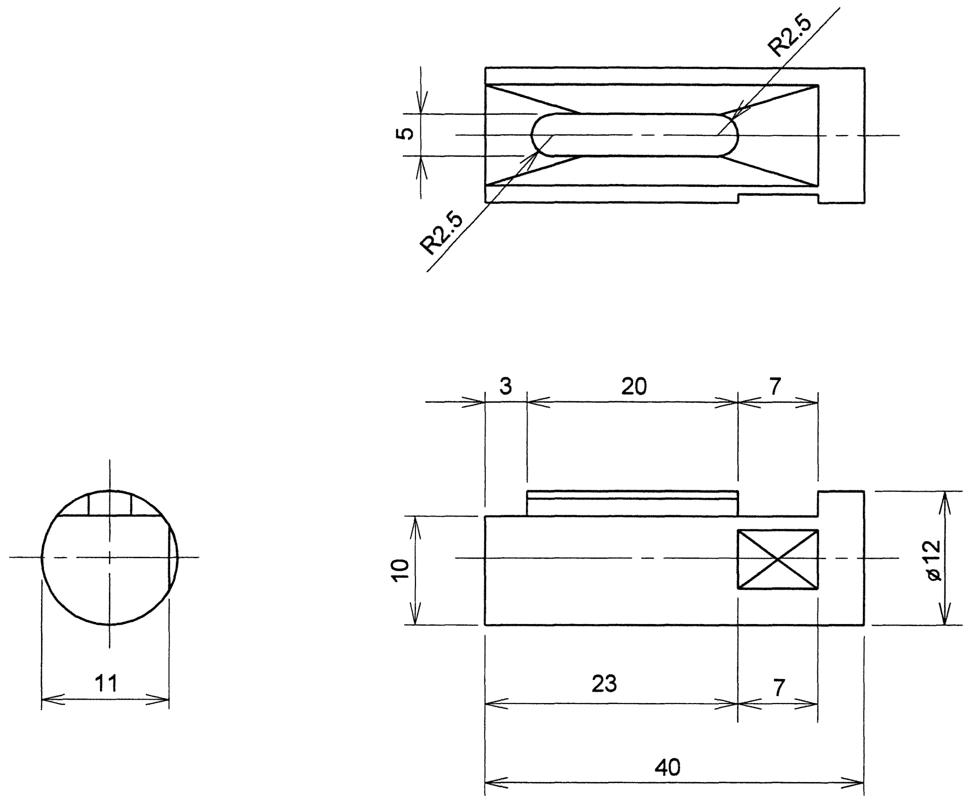


The relationship between tool cutting point and offset sequence is as illustrated below.

G41: Offsets the tool diameter to the left against the tool moving direction.
G42: Offsets the tool diameter to the right against the tool moving direction.



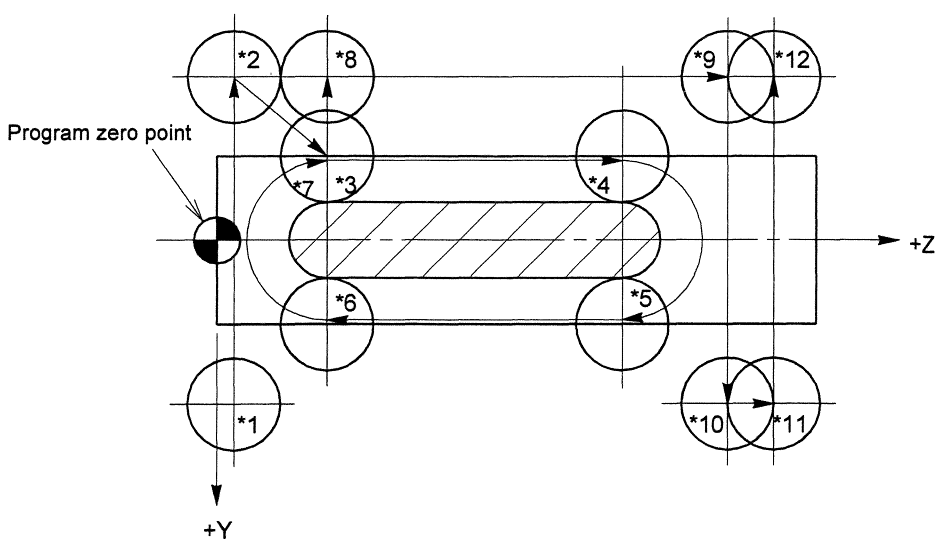
Machining layout



Tool data

Tool diameter	ø4 (end mill)
Tool No.	T07
Offset No.	08
Nose R radius (tool radius value)	2.0
Virtual tool nose No.	0

Numbers 1 through 12 in the figure above indicate the programmed end mill movement.



Sample program

Specify a program for $1.

$1	
T0700	Selects and offsets the tool spindle
M05	Main spindle stop
G50 W−15.0	Sets the coordinate system for the tool spindle position
M28 S0	Indexes the spindle to 0°
M80 S3=□	Run the tool spindle in forward direction
G98	
G19 G00 Y22.0 Z1.0	Specifies the plane to YZ and positions the tool at the cutting start point on this plane
X10.0	Positions the infeed direction (X axis)
G01 Y−22.0 F□	
G41 Y−5.0 Z5.5	Offsets the tool diameter
Z20.5	
G02 Y5.0 R2.5 F□	
G01 Z5.5 F□	
G02 Y−5.0 R2.5 F□	
G40 G1 Y−22.0 F□	Cancels the tool diameter offset
G00 Z25.0	
G01 Y22.0 F□	
G00 Z28.0	
G01 Y−22.0 F□	
M28 S90	Indexes the spindle to 90°
G00 X11.0	
Y−11.8 Z25.0	
G01 Y11.8 F□	
G00 Z28.0	
G01 Y−11.8 F□	
G18 G00 X14.0 T00	Returns the plane designation to the turning mode, cancels the offset, and returns the tool to the positioning point in the radial direction
M82	Stops the tool spindle
G50 W15.0	Returns the coordinate shift to the original state

7.7 Cross Rigid Tapping (G88, 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 rotary tool 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 tool spindle speed S☐ in consideration of deceleration ratio (twice as standard value in case of 1/2 deceleration).

7.7.1 Rigid tapping for outer circumference with a rotary tool (G88, G80)

This function performs tapping while synchronously controlling the tool spindle 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 (U)☐ R☐ F☐ D±☐ S☐ Q☐ T☐ H1` Cross rigid tapping mode

`G80` Rigid tapping mode cancel

`X (U)`: Specify the tapping end position. The value must be specified for the diameter. (U is an incremental command.)

`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 thread pitch.

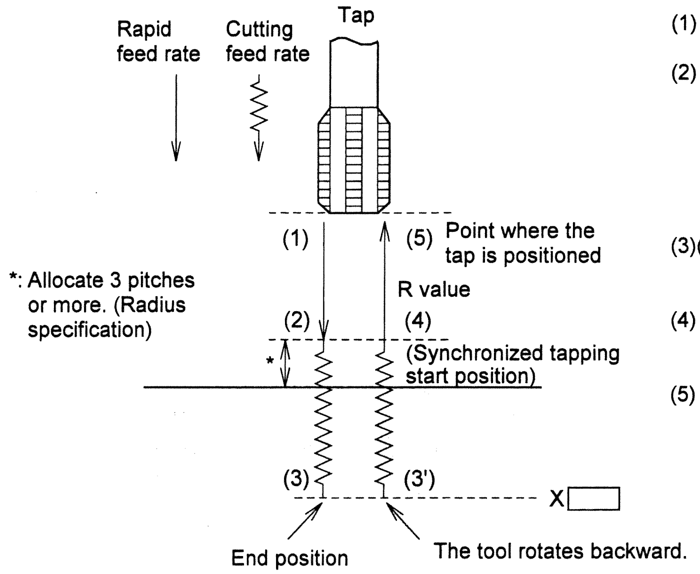
`D±`: Specify the tool spindle and the rotation in which the spindle rotates. When 3 is specified, the cross drilling tool spindle rotates forward. When –3 is specified, the cross drilling tool rotates backward.

`S`: Specify the tool spindle speed.

`Q`: Tap pull-out override specification (%). Specify this argument if you want to make the tap pull-out rate faster. The available values range from 100 to 200 in 10%. The default value is 100.
To specify the Q argument, an option other than the rigid tap function (high-speed rigid tap function) is required.

`T`: Specify the dwell time at the end of tapping in unit of 1/1000 sec. Do not omit the decimal point. If the argument is omitted, the program does not stop (no dwell).

`H1`: Argument for step machining. Does not specify G80 (rigid tapping mode cancel) command at the end of G88 (cross rigid tapping) in macro program. Be sure to specify G80 (rigid tapping mode cancel) when the tapping cycle is finished.



(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 1

When the material diameter is 4.0, and the thread pitch is 0.4:

```
  :
M5
M28 S0
M82
N0711 T0800
G50 W-15.0
G00 X8.4 Z10.0 T08 ................ X8.4 = ⌀4.0 + {(1 + 1.2) × 2} = 8.4
G88 X0.0 R1.0 F0.4 D3 S500
G80
G00 X8.4
G50 W15.0
M82
M20
  :
```

Clearance between the workpiece and tap: (1)

Pitch × 3: 1.2

Doubled because of diameter specification: × 2

Sample program 2

```
  :
M5
M28 S0
M82
T0800 .......................................... Selects a tool for tapping.
  :
G88 X6.0 R1.0 F0.4 D3 S500 H1 .................. First step of rigid tapping command
G88 X5.0 R1.0 F0.4 D3 S500 H1 .................. Second step of rigid tapping command
  ⋮                                                ⋮
G88 X1.0 R1.0 F0.4 D3 S500 H1 .................. (n–1)-th step of rigid tapping command
G88 X0.0 R1.0 F0.4 D3 S500 ..................... n-th step of rigid tapping command (with H1
                                                 argument not specified)
G80
  :
```

Notes

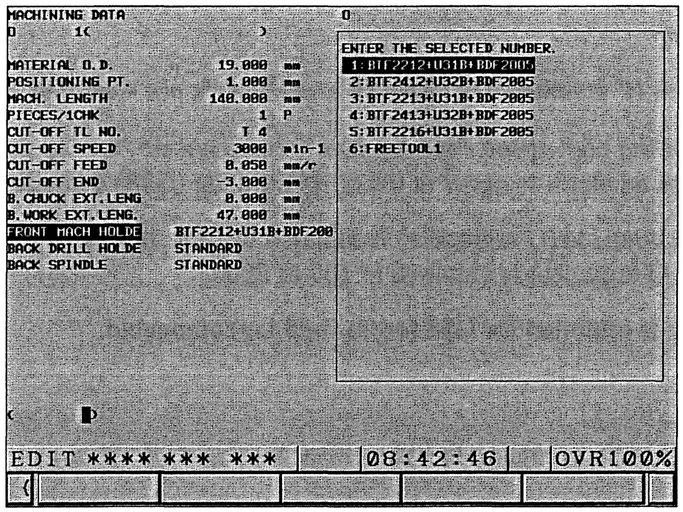
- Stop the tool spindle before specifying G88 (cross rigid tapping).
- Phase adjustment for rigid tapping with tool spindle is not available.
- If no option for synchronous tapping function is specified, specifying G88 command (cross rigid tapping) will cause an alarm to occur.
- If the G88 command is specified in axis control group 2 ($2), an alarm will occur.
- If the end point of tapping (argument X or U) is not specified, an alarm will occur.
- If any other value than ±3 is specified for the rotation direction of tool spindle (D argument), an alarm will occur.
- If the thread pitch (F argument) or speed (S argument) is not specified, an alarm will occur.
- If the options for the high-speed rigid tap function are not set, the tap pull-out override (Q argument) causes an alarm to occur at the cross rigid tap (G88) command.
- The tap pull-out override (Q argument) is rounded in 10%. If a value out of the available range is specified, the Q argument is clumped to 100 or 200.
- Reverse running is inhibited for G88 (cross rigid tap) command.

7.8 End Face Drilling Spindle (BSE107)

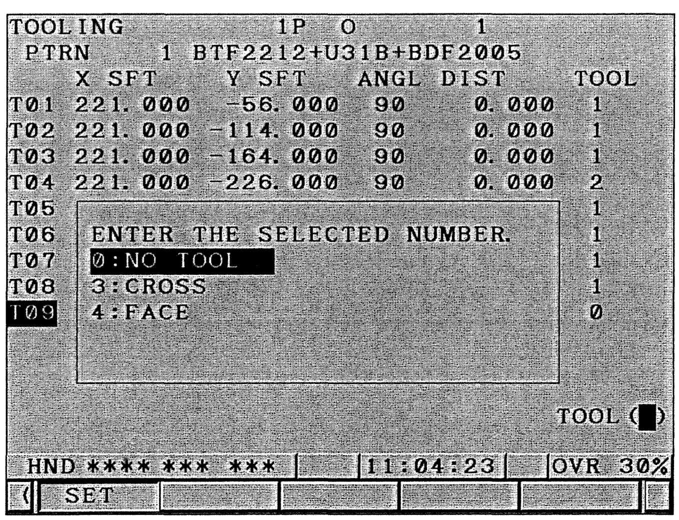
The following describes the procedure of mounting end face drilling spindle BSE107 for the front workpiece on front machining tool holder.

Procedure

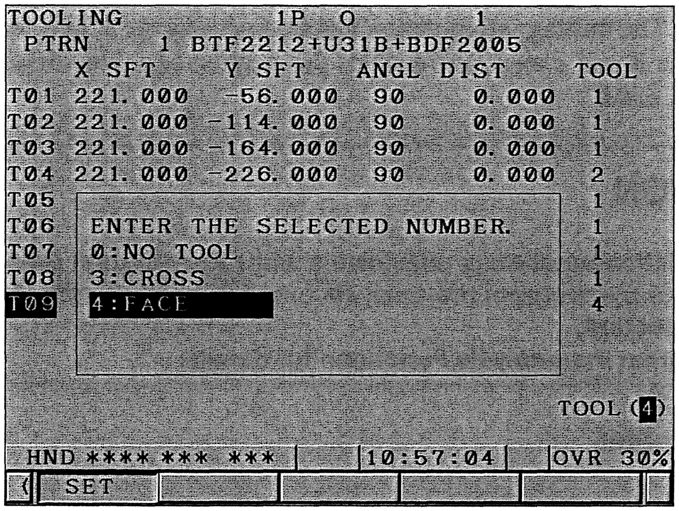
1. Call or newly create the program to be performed on the Edit screen.
2. Select an arbitrary holder out of the front machining tool holders on the Machining Data screen.

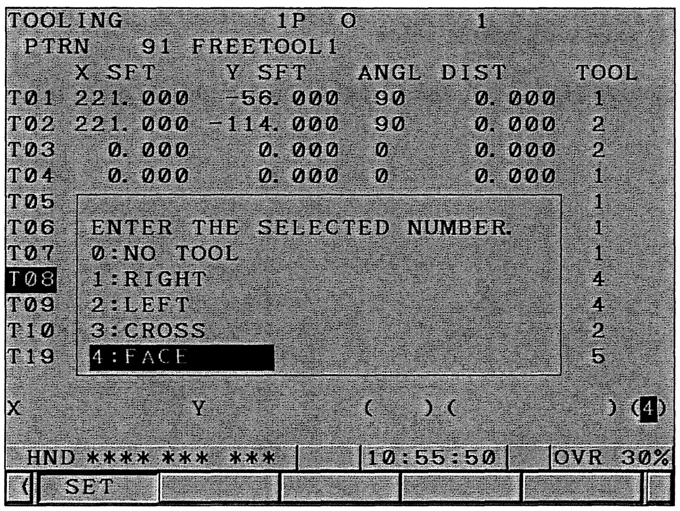


3. Select a program to be performed on the Program Selection screen.
4. Make the Preparation screen appear. Use the menu selection key [►] until the menu key [T-PATT] appears.
   Press the menu key [T-PATT] to display the Tool Layout Pattern screen.
5. Press the menu key [SET] and select a tool on which the end face drilling spindle is mounted.



6. Move the cursor to the position of "TOOL" and enter "4. FACE" according to the guide window.





Note

If a free tool is set as the "Front machining tool holder" on the Machining Data screen in step 2, provide proper specification for other than the tool. See <6.20 Setting the Free Tool Layout Pattern> for details. After specifying the tool layout pattern in the above procedure, be sure to confirm that all tool coordinates are set appropriately.

7.9 Milling Interpolation

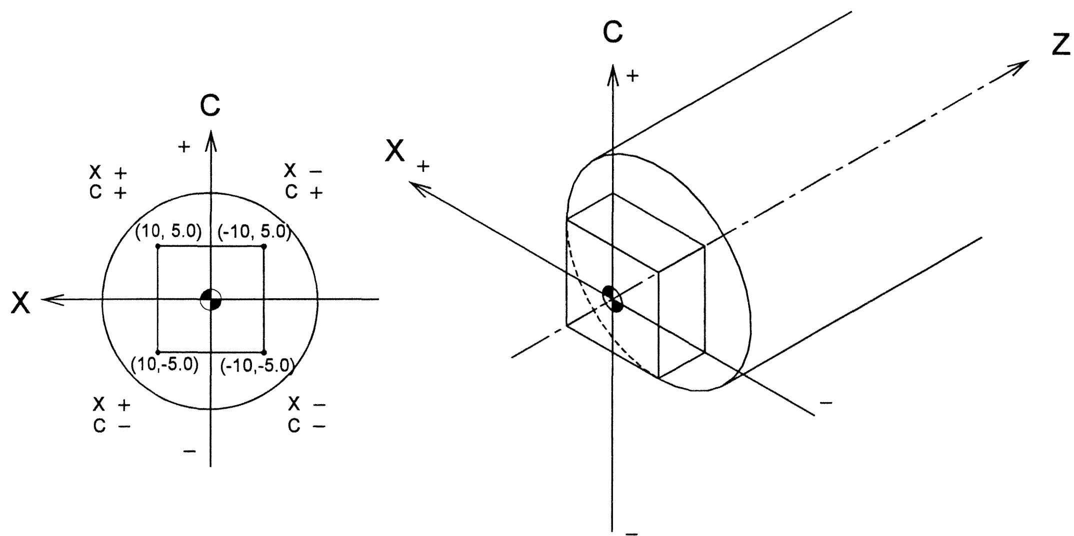
Generally, the X and Z axes are controlled as linear axes and the C axis is controlled as the rotary axis of the spindle. The milling interpolation function permits the C axis to be used as a linear axis at right angles to the X and Z axes, and enables the creation of a three-dimensional program with the three axes X, Z, and C.

> **CAUTION**
>
> **Milling interpolation function is unavailable while the program pre-analysis function is enabled. Disable the program pre-analysis function before using the milling interpolation function.**
> **See <6.23 Program Pre-analysis Function> for details.**

7.9.1 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 with diameter value, and (Z,) and C axes with radius value.

The positive and negative signs of X and C are as shown in the figure below.



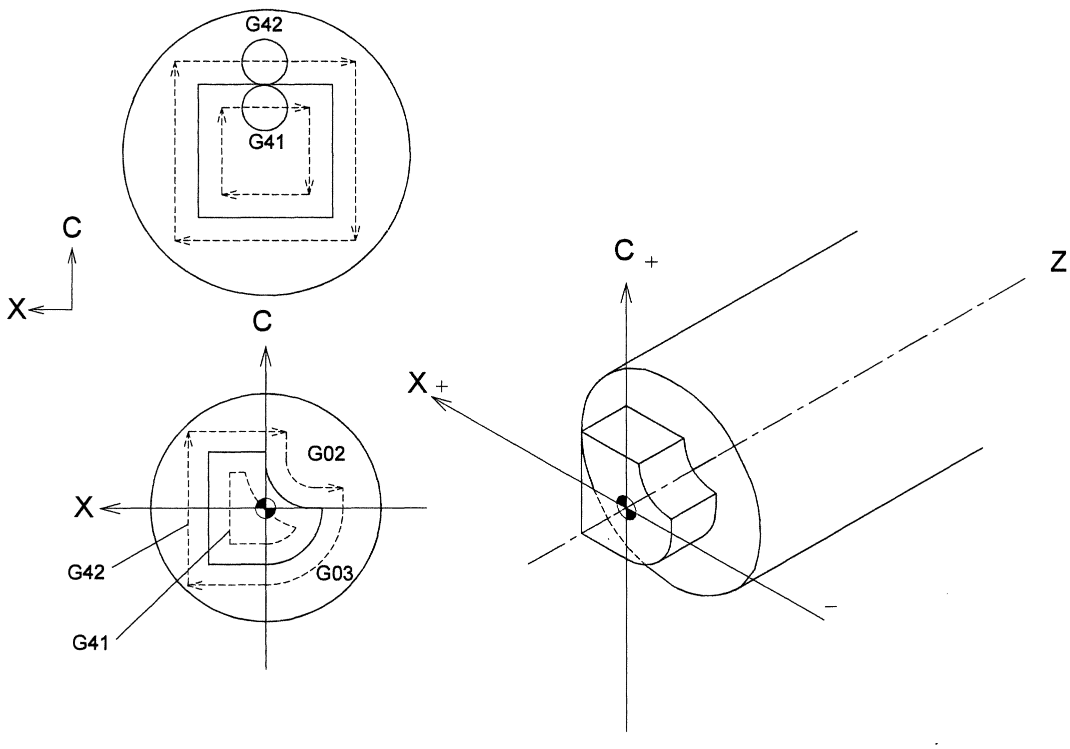
Command format

G12.1	Milling interpolation ON
G13.1	Milling interpolation cancel

7.9.2 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
Arc interpolation	G02	CCW (counterclockwise)
	G03	CW (clockwise)



7.9.3 Programming of a milling process

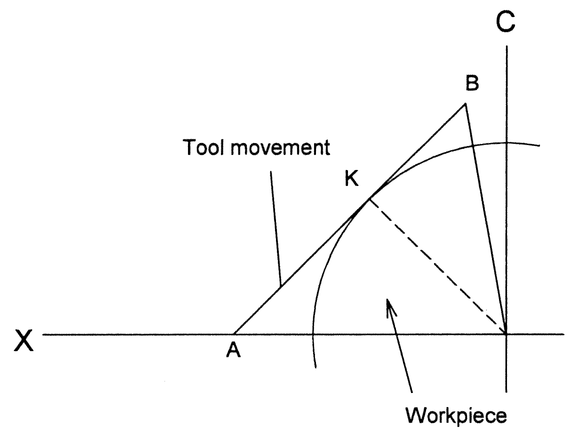
Consider the following points in programming a milling process:

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

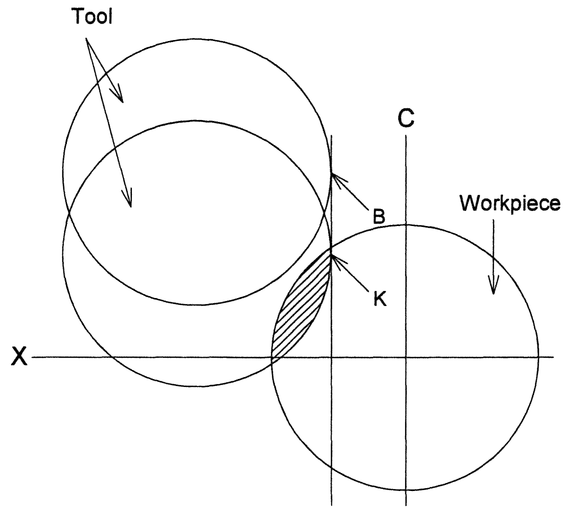


Determine the milling start position in consideration of the diameter of the rotary tool to be used and the diameter of the workpiece to be machined. If the start position is not suitable, the tool and workpiece may interfere with each other.

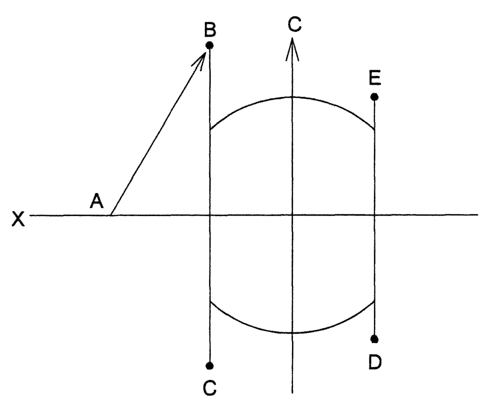
If the machining start position is point K for making the D cut as shown in the following figure, the tool and workpiece interfere with each other in the shaded area when the X axis makes an approach to the workpiece.

To prevent the interference, obtain the machining start position (point B) by calculation.

Point B = Pa position

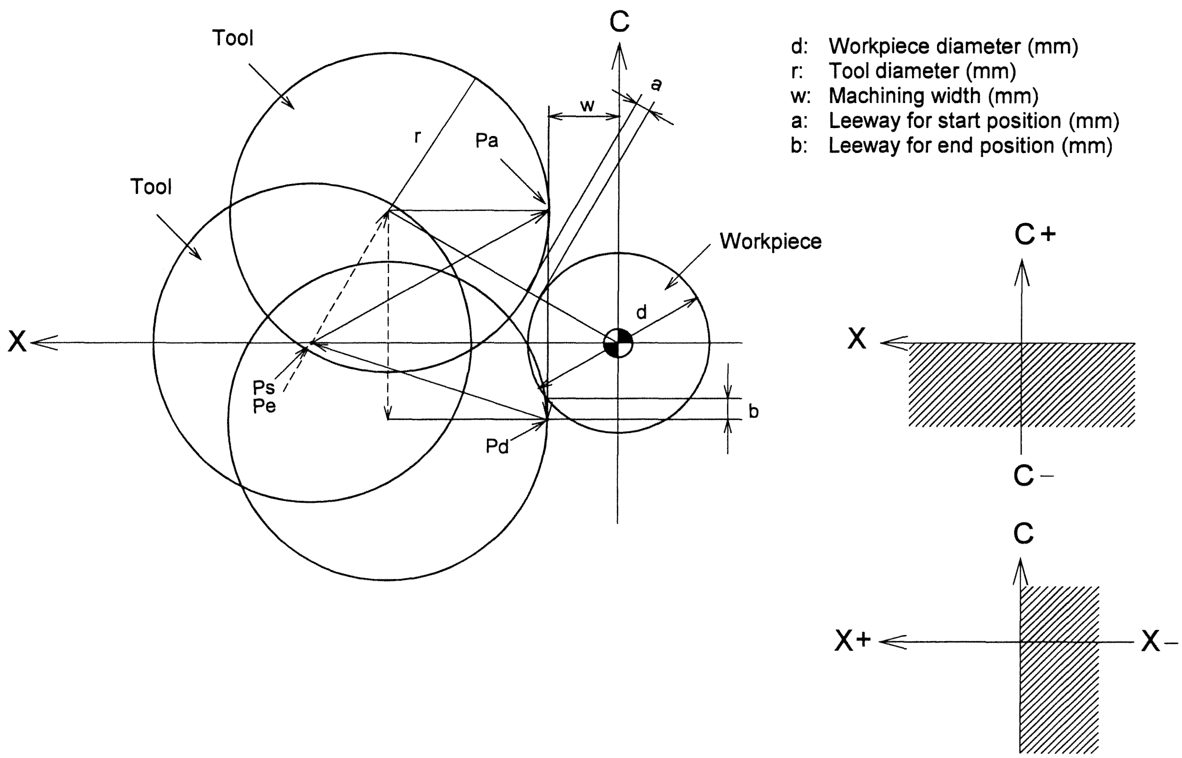


When performing machining both sides of a workpiece (e.g., 2-face width machining), cancel milling interpolation at completion of machining one side, rotate the C axis 180 degrees, then start machining the other side. If both sides are machined continuously without milling interpolation cancel, the tool and workpiece may interfere with each other.



7.9.4 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). Using this calculation, the tool can approach the workpiece from farther point, thus the possibilities of interference between the workpiece and the tool can be reduced.



Example: When d = 12 mm, r = 15 mm, w = 4 mm, a = 2 mm, and b = 2 mm:

Ps = (55.684, 0) Pa = (8.0, 12.961)

Pd = (8.0, –6.472) Pe = (55.684, 0)

Obtain machining positions suitable for the milling process from the formulas shown in the following table:

Machining position	X axis coordinate (diameter)	C axis coordinate (radius)
Milling start positions Ps = (Xs, Cs)	$Xs > ((d/2 + r + a)^2 / (w + r)) \times 2$	$Cs = 0$
Machining start position Pa = (Xa, Ca)	$Xa = w \times 2$	$Ca > \sqrt{(d/2 + r + a)^2 - (w + r)^2}$
Machining end position Pd = (Xd, Cd)	$Xd = w \times 2$	$Cd > \sqrt{(d/2)^2 - w^2} + b$
Milling end position Pe = Ps = (Xe, Ce)	$Xe = Xs > ((d/2 + r + a)^2 / (w + r)) \times 2$	$Ce = Cs = 0$

Notes

- The results of the above formulas are free from the positive and negative sign. When changing the results to coordinates, assign signs to them in consideration of the tool position.
- Specify the coordinates of C axis with values calculated for the radius.
- Specify the coordinates of X axis with values calculated for the diameter.
- 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:

1) Determine the leeway for start position (a), and calculate the machining start position (Pa).
2) Based on the machining start position (Pa), calculate the milling start position (Ps), the first positioning point of milling.
3) Determine the leeway for end position (b), and calculate the machining end position (Pd).
4) Based on the machining end position (Pd), calculate the milling end position (Pe).

1. Determine the leeway for start position as a = 2.0 mm.

Workpiece diameter d :	12.0
Tool radius r :	15.0
Machining width w :	4.0
Leeway for start position a :	2.0

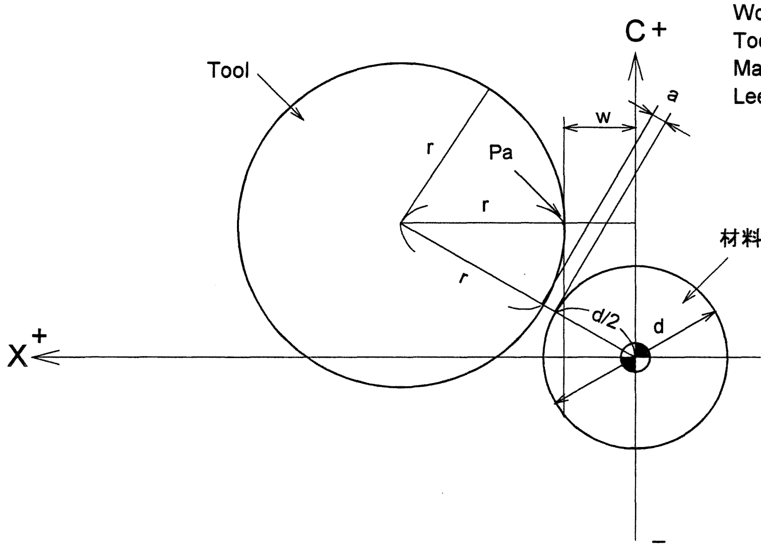
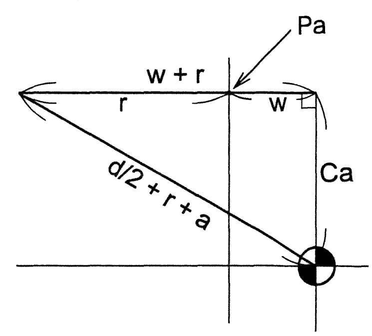


Figure 1



Triangle in Figure 1

$Pa = (Xa, Ca)$
$Xa = w \times 2 = 8.0$ (diameter)
$Ca = \sqrt{(d/2 + r + a)^2 - (w + r)^2}$

≈ 12.961 (radius)
$Pa = (8.0, 12.961)$.

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).

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.

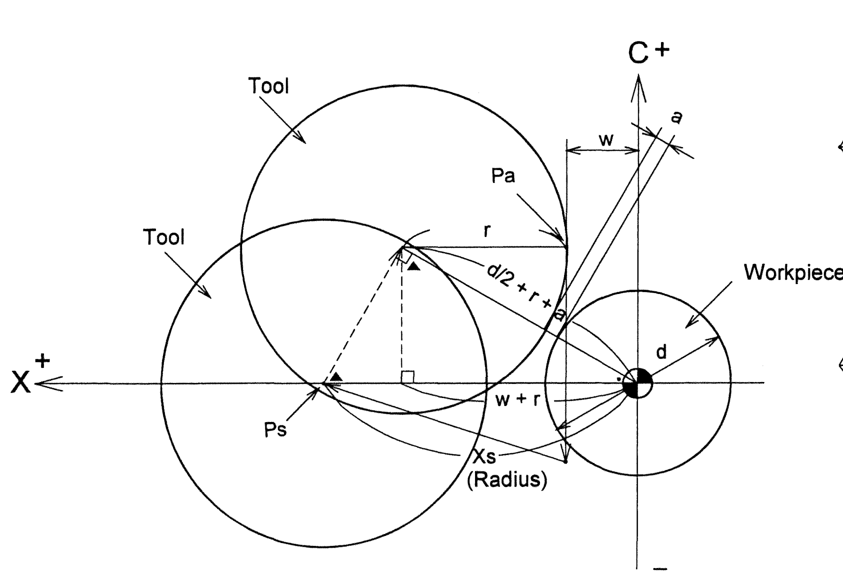
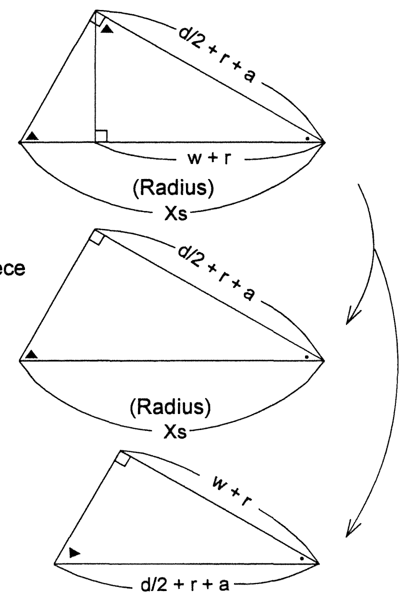


Figure 2



Triangle in Figure 2

Ps = (Xs, Cs)

d/2 + r + a : w + r = Xs : d/2 + r + a

$(w + r)Xs = (d/2 + r + a)^2$

$$\text{Xs (Radius)} = \frac{(d/2 + r + a)^2}{(w + r)} \approx 27.842 \text{ (Radius)}$$

Diameter is: $Xs = 27.842 \times 2$
$= 55.684$

Cs = 0

Ps = (55.684, 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 for end position as b = 2.0 mm.

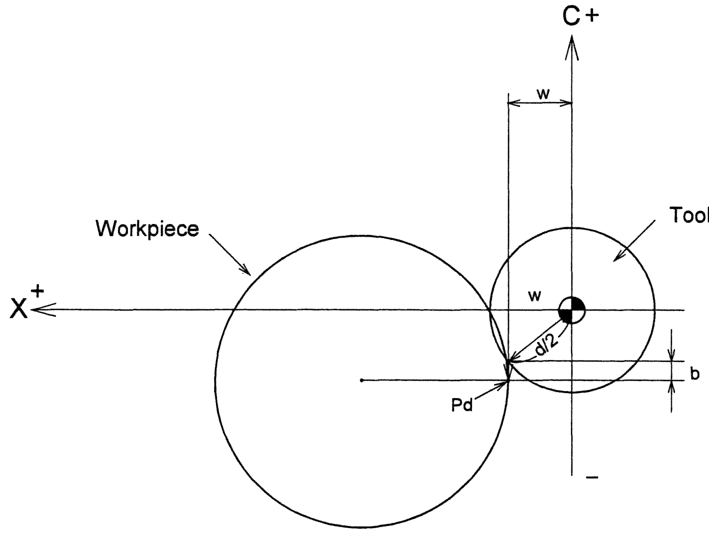
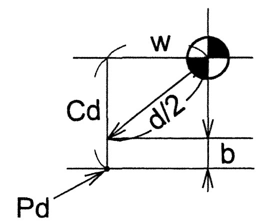


Figure 3



Triangle in Figure 3

Pd = (Xd, Cd)
Xd = w × 2 = 8.0 (diameter)
Cd = $\sqrt{(d/2)^2 - w^2} + b$
= 6.472 (calculated value)

* The actual tool is placed in negative area, accordingly the coordinate value becomes Cd = – 6.472.

Pd = (8.0, – 6.472)

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 = Ps = (55.684, 0)
   ↑
   Diameter value

Sample program

The sample program below uses the values calculated from the milling coordinates.

```
Previous process
  :
M5 ........................................ Stop the main spindle
G98 ....................................... Feed per minute
M80 S3=[    ] ............................. Rotate the tool spindle
M18 C0
G50 C0 .................................... Set C axis coordinate system

Milling process
T[    ] ..................................... Select milling tool
G0 X55.684 Z1.0 C0 T[    ] ................ Go to milling start position (Ps)
G12.1 ..................................... Milling interpolation ON
G17
G42 G1 X8.0 C12.961 F[    ] ............... Tool diameter compensation ON, go to machining start
                                            position (Pa)
                                            (See <7.9.2 Milling plane> and <7.6.2 Offsetting the Tool
                                            Diameter> of tool diameter compensation.)
G1 X8.0 C-6.472 F[    ] ................... Go to machining end position (Pd)
G40 G1 X55.684 C0 F[    ] ................. Go to milling end position (Pe)
G13.1 ..................................... Milling interpolation OFF
M82 ....................................... Tool spindle stop
M20 ....................................... C axis release
G18
  :
```

7.9.5 Program format

Sample program

Previous process

:	
G98	Feed per minute
M80 S3=□	Tool spindle rotation
M18 C0	
G50 C0	C axis coordinate system setting

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	Select end face machining
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	Tool spindle stop
M20	C axis release
G18	Select Z-X plane

Note

G00 command is disabled while milling interpolation is set to on (G12.1).
Use the G01 command to move an axis.

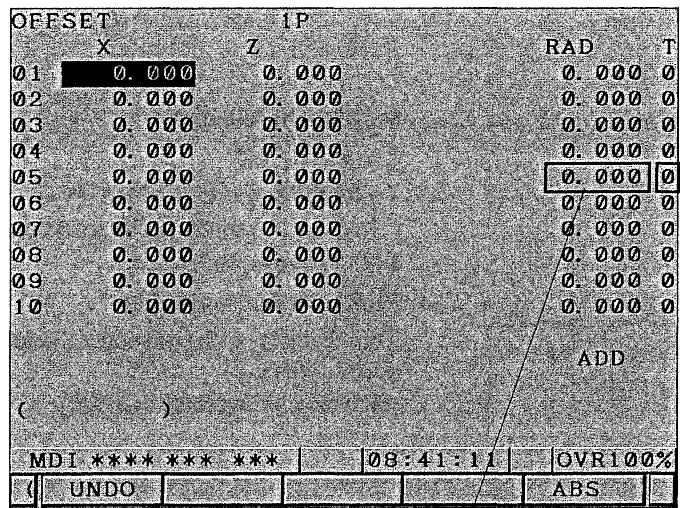
7.9.6 Milling compensation

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 <User's Manual> issued by FANUC, Ltd.

The following shows the setting of nose R compensation:

Example: When T05 is used:

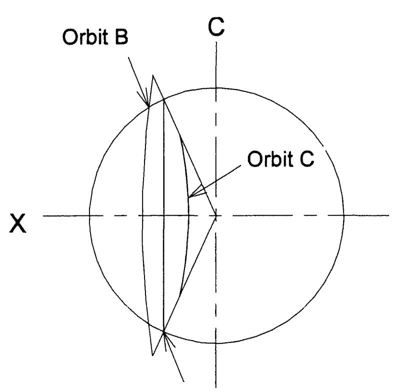


0 must be set for the virtual tool nose number.

Set the radius of the tool to be used.
Example: With an end mill having a diameter of 3.0: 1.500

If the nose R compensation 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 nose R compensation is too large. Conversely, the workpiece has the internally round shape like orbit C when the nose R compensation is too small. If linear interpolation causes a round shape, the nose R compensation is invalid. Correct the nose R compensation.



Orbit specified in the program

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.

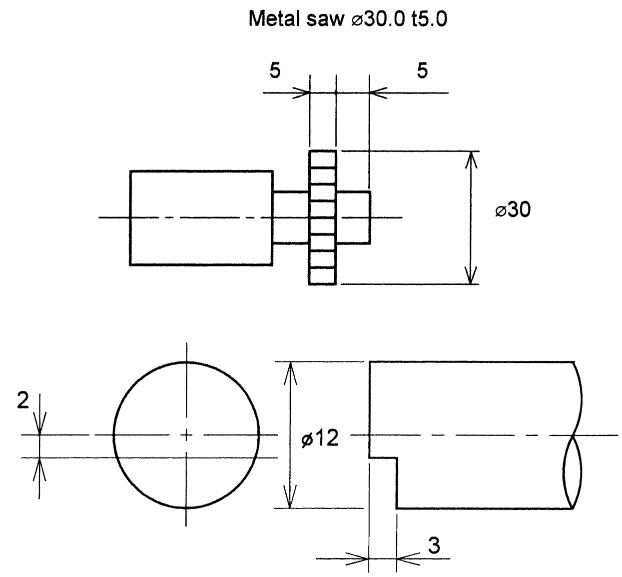
7.9.7 List of NC unit errors during milling interpolation

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.

No.	Message	Explanation
PS0145	ILLEGAL USE OF G12.1/G13.1	The axis No. of plane selection parameter No.5460 (linear axis) and No. 5461 (axis of rotation) in the polar coordinate interpolation mode is out of range (1 to number of controlled axes.)
PS0146	ILLEGAL USE OF G-CODE	The modal G code group contains an illegal G code in the polar coordinate interpolation mode or when a mode was canceled. Only the following G codes are allowed: G40, G50.1, G69.1 An illegal G code was specified while in the polar coordinate interpolation mode. The following G codes are not allowed: G27, G28, G30, G30.1, G31 to G31.4, G37 to G37.3, G52, G92, G17 to G19, G81 to G89, G68 In the 01 group, G codes other than G01, G02, G03, G02.2, and G03.2 cannot be specified.
DS1512	EXCESS VELOCITY	The feedrate of the linear axis during polar coordinate interpolation exceeded the maximum cutting feedrate.
DS1514	ILLEGAL MOTION IN G12.1 MODE	In a hypothetical axis direction compensation during the polar coordinate interpolation mode, an attempt is made to travel to the area in which the travel cannot be made.

Milling interpolation function is disabled while the program pre-analysis function is enabled.

7.9.8 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) $\times 2 = 4$

$Ca > \sqrt{(12/2 + 15 + 2)^2 - (2 + 15)^2} = 18.97 \approx 19.0$

Pd: $Xd = 2 \times 2 = 4$

$Cd > \sqrt{(12/2)^2 - 2^2} + 2 = 7.65 \approx 7.7$

Pe: $Xe > ((2 + 15) + (7.7^2 / (2 + 15)) \times 2 = 4.0975 \approx 41.0$

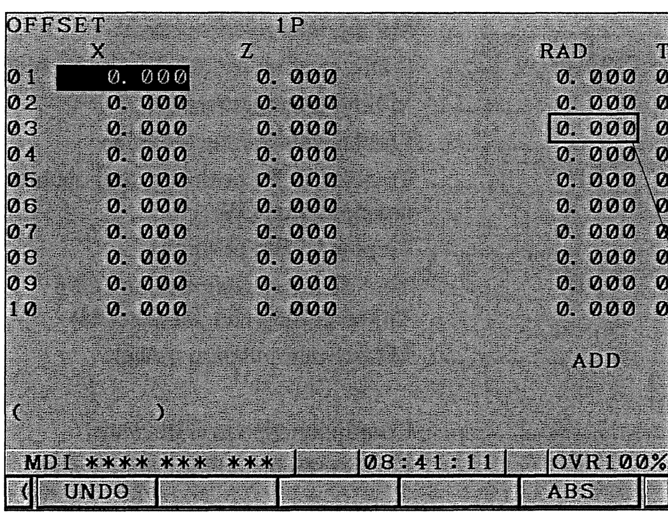
Set about 45.0 with leeway added.

Sample program

$1

```
  :
M5
G98
M80 S1260
M18 C0
G50 C0

T□
G50 W-5.0
G00 X63.0 Z3.0 T03 ........................ Ps: milling start position
G12.1
G17
G41 G01 X4.0 C19.0 F□ .......... Pa: machining start position
G01 C-7.7 F80 .................................. Pd: machining end position
G40 G01 X45.0 C0 F□ .............. Pe: milling end position
G13.1
G50 W5.0
M82
G18
M03 S1=3000 G99
  :
  :
```



Tool diameter compensation setting 15.0

7.9.9 Example of using the milling interpolation function 2

This section shows an example of 2-face width machining. Note that milling interpolation is canceled before the spindle rotates 180°. Shown below is a program sample for milling interpolation.

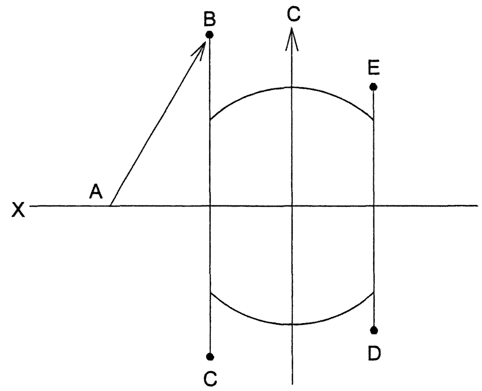
Sample program

$1		
:		
M5	Main spindle stop	
G98	Feed per minute	
M80 S□	Tool spindle rotate	
M18 C0	Sequential operation of main spindle C axis ON (optional)	
G50 C0	C axis coordinate system setting	
Milling machining process		
N□		
T□	Milling tool selection	
G50 W□	Coordinate system shift	
G00 X□ Z□ T□	To point A (Ps position)	
G12.1	Milling interpolation ON	
G17	End face machining selection	
G41 G01 X□ C□ F□	To point B (Pa position), tool diameter compensation ON.	Milling
G01 C-□ F□	To point C (Pd position)	Milling
G40 G01 X□ C□ F□	To the milling end position (Pe position), tool diameter compensation OFF	Milling
G13.1	Milling interpolation cancel	Milling
G00 X□ Z□ C180.0	C axis rotation by 180.0° at point A (Ps position)	
G12.1	Milling interpolation ON	
G17	End face machining selection	
G41 G01 X□ C□ F□	To point D (Pa position), tool diameter compensation ON	Milling
G01 C-□	To point E (Pd position)	Milling
G40 G01 X□ C□ F□	Tool diameter compensation OFF To the milling end position (Pe)	Milling
G13.1	Milling interpolation cancel	Milling
G00 X□ Z□		
G50 W□	Coordinate system shift cancel	
M82	C axis release and tool spindle stop	
G18		
:		

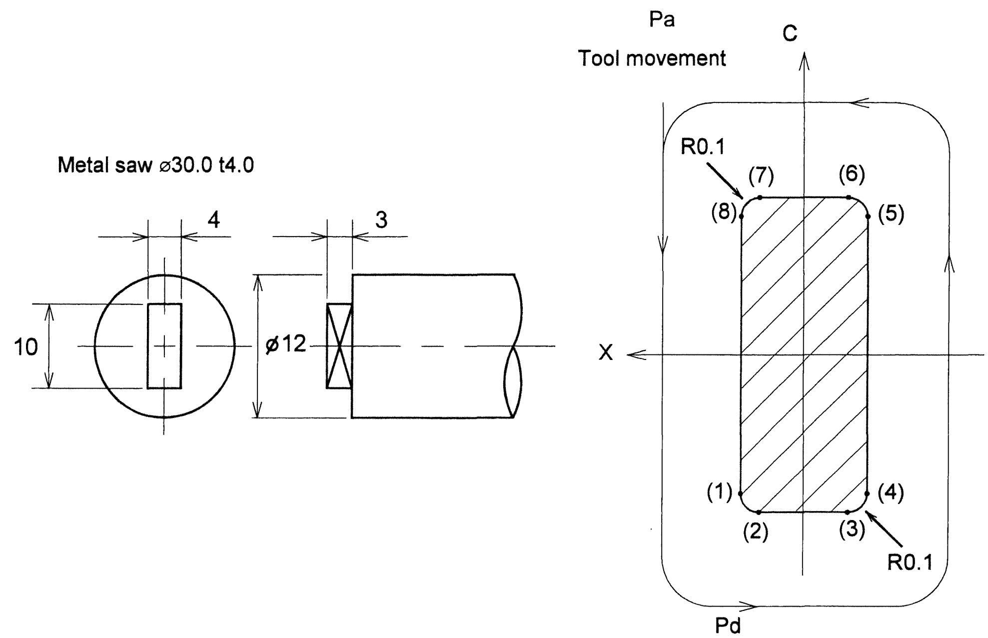
Note

Machining from point D to Point E is performed after the workpiece is rotated by 180°. Be careful of the coordinates of X and C.

B = D and C = E



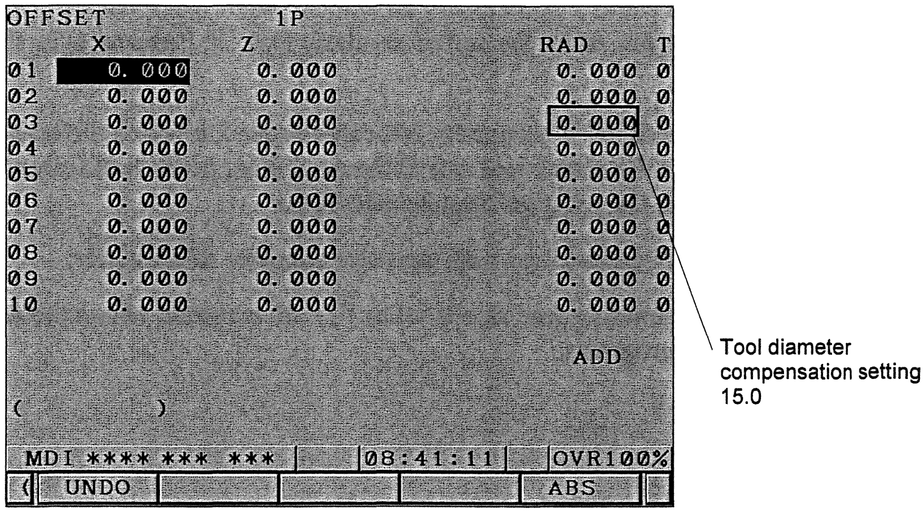
7.9.10 Example of using the milling interpolation function 3



Sample program

$1

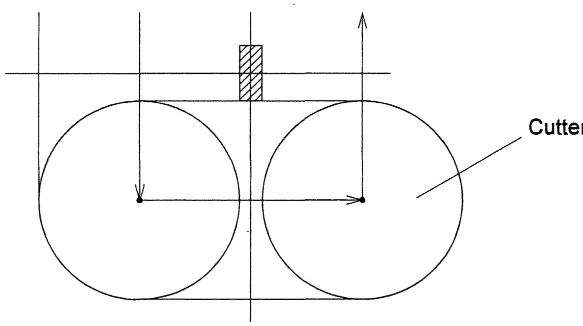
```
  :
G98
M80 S1260
M18 C0
G50 C0
T[    ]
G50 W–5.0
G00 X57.0 Z3.0 T03 ................................ Ps: milling start position
G12.1
G17
G41 G01 X4.0 C23.0 F[    ] .................. Pa: machining start position
G01 X4.0 C–4.9 F80 ................................ (1)
G02 X3.8 C–5.0 R0.1 F1200 .................. (2)
G01 X–3.8 F80 ......................................... (3)
G02 X–4.0 C–4.9 R0.1 F300 .................. (4)
G01 C4.9 F80 ........................................... (5)
G02 X–3.8 C5.0 R0.1 F–1200 ................ (6)
G01 X3.8 F80 ........................................... (7)
G02 X4.0 C4.9 R0.1 F–300 .................... (8)
G01 X4.0 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
G18
M03 S1=3000 G99
  :
```



Note

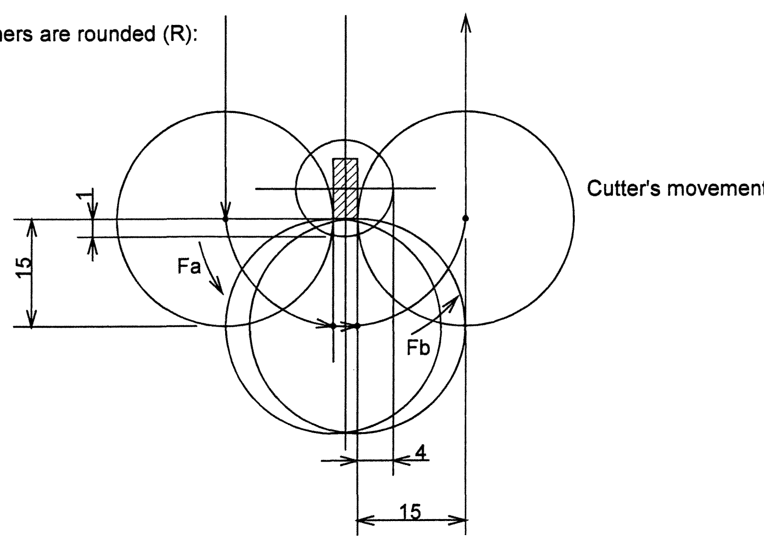
If the R setting value of tool diameter compensation is too large, the cutter makes unnecessary movement when cutting each corner of the workpiece. As a result, machining takes more time than it should. To minimize the machining time and have corners well finished, specify such a value for R that the corners are finely rounded.

Specification for the cutter to move at right angles



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):



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 $F_2 = 80$ mm/min, obtain the feed rates Fa and Fb to be specified in programs.

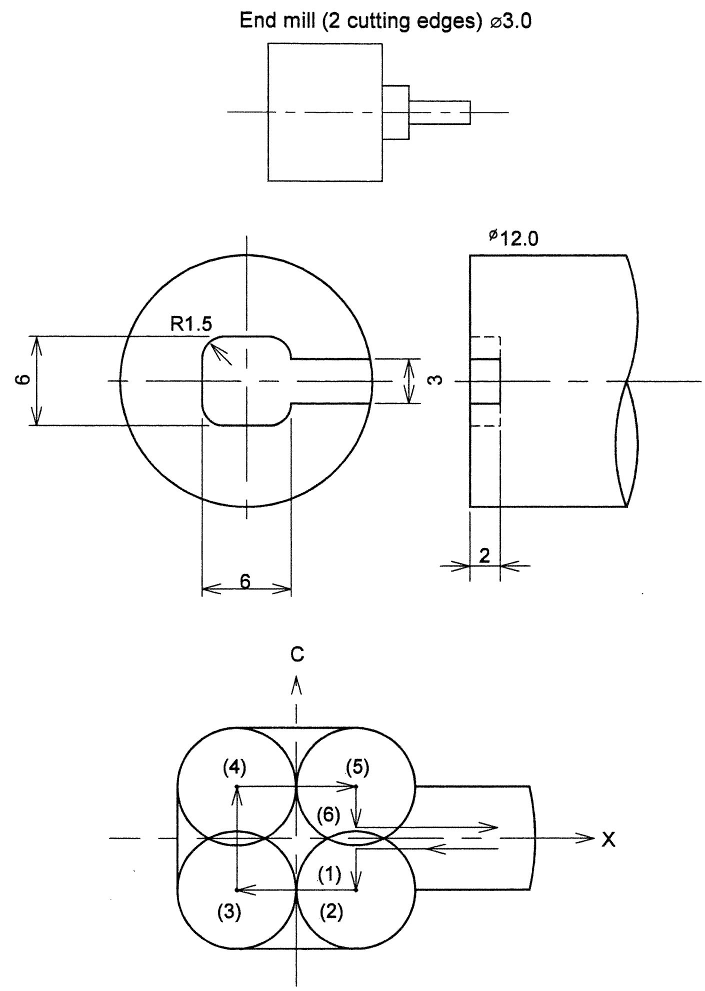
  $$Fa = 80 \times \frac{15}{1} = 1200$$

  $$Fb = 80 \times \frac{15}{4} = 300$$

- 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.

7.9.11 Example of using the milling interpolation function 4

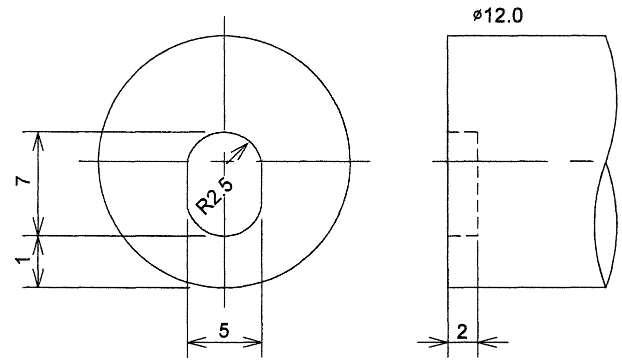


Sample program

$1	
:	
G98	
M80 S2700	
M18 C0	
G50 C0	
N424	
T☐	
G00 X30.0 Z2.0 T04	Ps: milling start position
G12.1	
G17	
G42 G01 X21.0 C–1.5 F☐	Pa: machining start position
G01 X6.0 F200	(1)
C–3.0	(2)
X–6.0	(3)
C3.0	(4)
X6.0	(5)
C1.5	(6) The value, instead of C0, is set in consideration of tool nose R compensation.
X14.0 F500	
Z–1.0	
G40 G01 X14.0 C0 F☐	C0 is acceptable because the tool nose R compensation is canceled.
G13.1	
G0 U0 V0 W0 T0	
M82	
G18	
M03 S1=3000 G99	
:	

7.9.12 Example of using the milling interpolation function 5

End mill (2 cutting edges) ⌀3.0

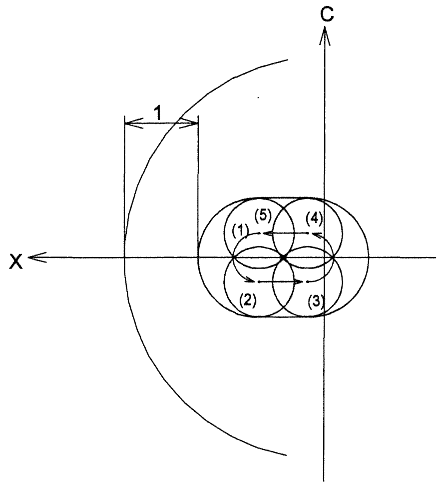


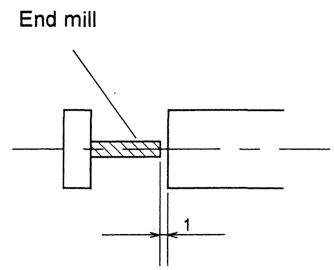
Sample program

$1

```
 :
G98
M80 S2700
M18 C0
G50 C0

T
G00 X16.0 Z-1.0 T02 ......................... Ps: milling start position
G12.1
G17
G42 G01 X5.0 C2.5 F .............. (1)
G01 Z2.0 F125 .................................... (a)
G02 C-2.5 R2.5 F160 ......................... (2)
G01 X-1.0 ........................................... (3)
G02 C2.5 R2.5 .................................... (4)
G01 X3.0 ............................................. (5)
Z-1.0 F1000 ........................................ (b)
G40 G1 U0 W0 F
G13.1
G0 U0 V0 W0 T0
M82
G18
M03 S1=3000 G99
 :
 :
```





(a) The tool cuts into the workpiece until Z2.0.
(b) The tool returns to Z–1.0.

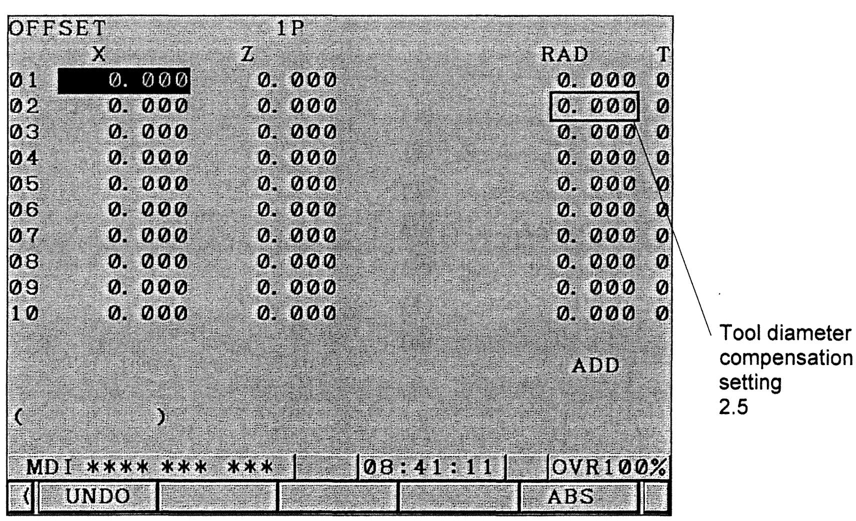
- With an arc inscribed with the inner diameter of the cutter:

d1	Cutter diameter	r1	Cutter radius
d3	Machining diameter	r3	Machining radius

F1 Feed rate specified in the program
F2 Feed rate at which you want to machine workpieces

- Obtaining the feed rate for the machining diameter:

$$F_1 = F_2 \times \frac{(d_3 + d_1)}{d_3} = F_2 \times \frac{(r_3 + r_1)}{r_3}$$

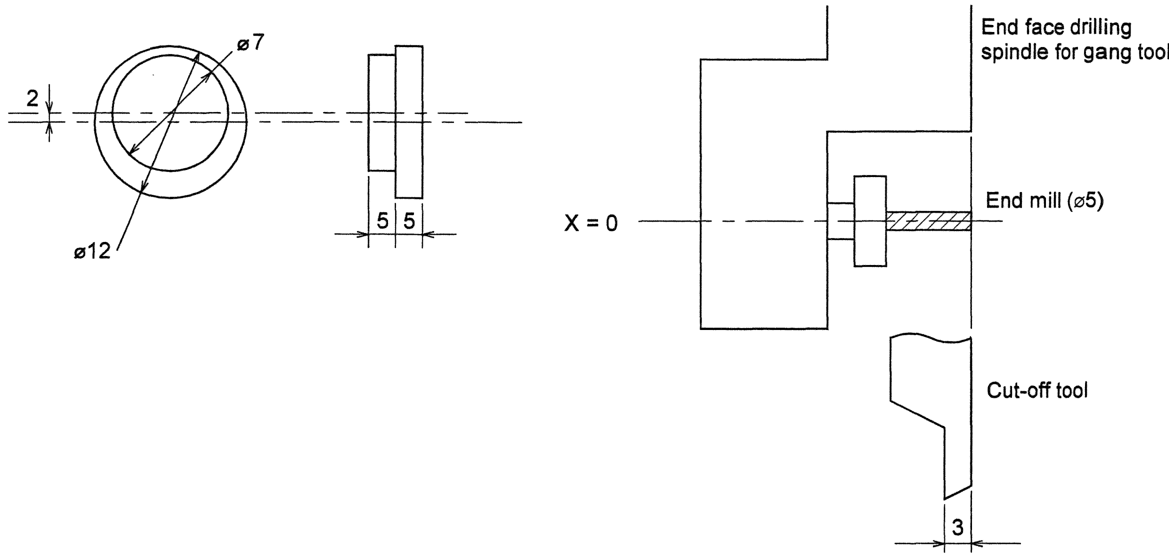


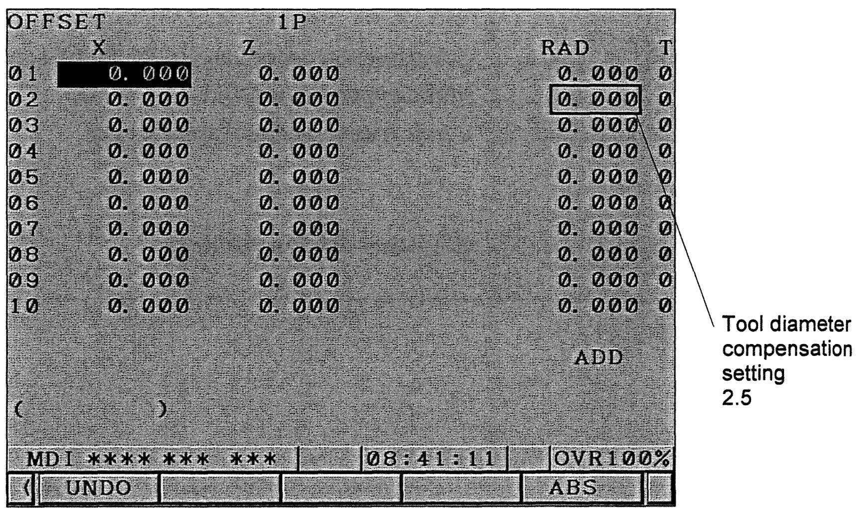
The feed rate for the machining diameter (F2) is set as 100 mm/min.

$$F_1 = 100 \times \frac{2.5 + 1.5}{2.5} = 160$$

Note: The machining feed rate (100 mm/min) is one example.

7.9.13 Example of using the milling interpolation function 6





Ps $X_s > (d/2 + r + a)^2 / (w + r)$
$= (12/2 + 2.5 + 1)^2 / (5.5 + 2.5) = 11.281$
11.281×2 (diameter value specification) $= 22.562 \fallingdotseq 22.6$
$C_s = 0$ (C coordinate)

Pa $X_a = w = 5.5 \times 2$ (diameter value specification)
$C_a > \{(d/2 + r + a)^2 - (w + r)^2\}^{1/2}$
$= \{(12/2 + 2.5 + 1)^2 - (5.5 + 2.5)^2\}^{1/2} = 5.12 \fallingdotseq 5.2$ (C coordinate)

Pd $X_d = w = 5.5 \times 2$ (diameter value specification)
$C_d > \{(d/2)^2 + w^2\}^{1/2} + b$
$= \{(12/2)^2 - 5.5^2\}^{1/2} + 1 = 3.39 \fallingdotseq 3.4$ (C coordinate)

Pe $X_e > (w + r) + C_d^2 / (w + r)$
$= (5.5 + 2.5) + 3.4^2 / (5.5 + 2.5) = 9.44 \fallingdotseq 9.5 \times 2$ (diameter value specification)
$C_e = 0$ (C coordinate)

Sample program

$1	
:	
G0 X26.0 Z–0.5	
M18 C0	Sequential operation with spindle C axis ON (optional)
G50 C0	Coordinate system setting
G98	
M80 S2000	
G17	XY plane selection (end face machining)
:	
N0217 T□	
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 G01 X11.0 C5.2 F□	Right compensation, and to the machining start position (Pa)
G1 C0 F60	Milling interpolation machining, machining end position (Pd)
G2 X11.0 C0 I–3.5 F60	
G1 C–3.4 F300	
G40 G01 X19.0 C0 F□	Compensation cancel, and to the milling end position (Pe)
G13.1	Milling interpolation OFF
M82	C axis release, and tool spindle stop
M3 S1=3000 G99	
:	
G18	XZ plane selection (turning)
:	

(Blank page)

Product Code	C	–	A	2	2	0	P	L	VII					
Document Code	2	E	1	–	0	7	0	1						

8. Sample Programming [Back Machining]

8.1 Back Machining Program

The back machining is performed by combining a front machining program ($1) and back machining program ($2).

Program configuration example

$1	$2	
Preparation process	Preparation process	The back spindle chucks the workpiece machined in the previous cycle.
Front machining process (Center drilling)	Back machining process (Center drilling)	Front/back simultaneous machining
Front machining process (Drilling, outer diameter)	Back machining process (Drilling)	Front/back parallel machining
Front machining process (Outer diameter)	Workpiece separation	Free layout pattern (Machining pattern cancel) Front machining Ejects the completed workpiece.
Front machining process (Outer diameter)		
Cut-off process	Pick-off	Picks off the workpiece by the back spindle.
Ending process	Ending process	Completes the program with the workpiece chucked by the back spindle.

As shown above, the workpiece is finished after two cycles of program execution.

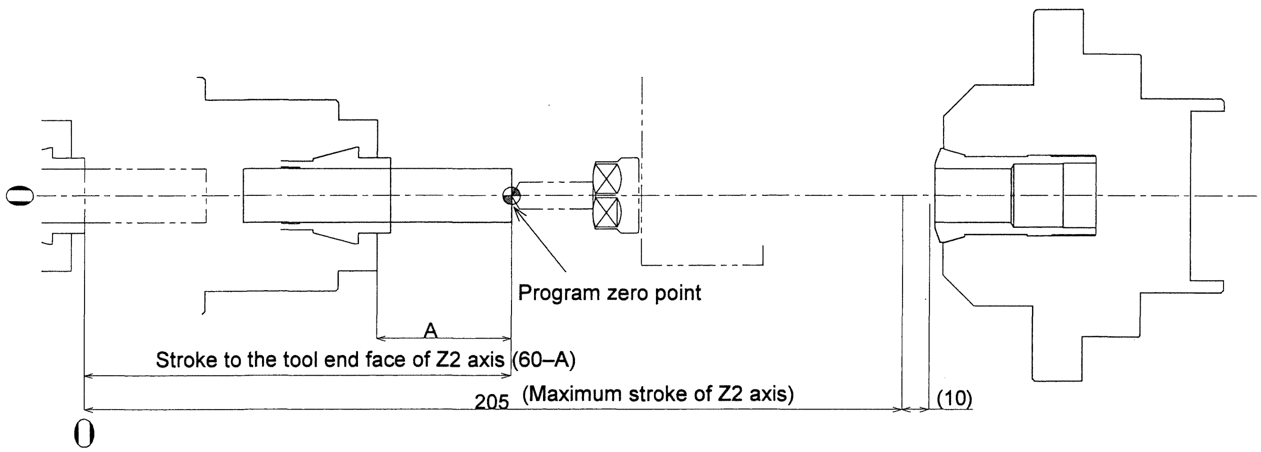
Note that inserting the back machining program at the last program execution (G999) can execute workpiece back machining after cut-off machining both in the first cycle, thereby finishing the workpiece in one cycle. The last program is not executed when No.16 "LAST PART VALID" is set to 0 (disabled) on Operator's panel screen or the menu key [LAS PR] is disabled on Check Screen.

8.2 Back Machining Coordinates

The coordinates in programming must be understood according to the back spindle movement during program creation.

The positional relation between the back spindle, back machining tool, and the workpiece chucked by the back spindle is shown in the figure below.

The same conception as in <Section 3.5 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 50 mm (maximum).

The length of the workpiece protruding from the end face of the back spindle chuck must be entered in the B.WORK EXT.LENG. in machining data.

(Protruding workpiece length = entire workpiece length – programmed back chucking position)

If the setting is "0", 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.

8.3 Back Machining Process

- The back machining is performed by tools T31ΔΔ to T34ΔΔ and T51ΔΔ to T54ΔΔ. This machining is basically the same as for front machining.
- Specify the length of the workpiece protruding from the end face of the back spindle chuck in the B.WORK EXT.LENG. in machining data.
  (Protruding workpiece length = entire workpiece length - programmed back chucking position)
- For "back long-neck chuck protrusion length" in the machining data, enter the protrusion length of the long-neck chuck from the end face of the back spindle cap nut while the long-neck chuck is used. This allows the position "back long-neck chuck protrusion + back workpiece protrusion" to be the program zero point.
- If "Z–1.0" is specified after tool selection of T31ΔΔ to T34ΔΔ or T51ΔΔ to T54ΔΔ, 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.
- Set the tool T31 to T34 to the position within 70 mm from the end face of sleeve holder. (Interference may occur when the back machining tool protrudes more than 70 mm during setting. Pay attention when setting the tool.)
- Set the tool T51 to T54 to the position within 48 mm from the end face of sleeve holder. (Interference may occur when the back machining tool protrudes more than 48 mm during setting. Pay attention when setting the tool.)
- Write a program in $2.

Sample program

(Center hole and drilling)

$2	
G44	
G99 M23 S2=☐	Specify per rotation feed and forward rotation of back spindle.
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.
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.
M241	Move the back spindle to the waiting point.
M34	Perform sequential operation for product collection after back machining is completed.
:	

8.4 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
`M73`	Back spindle air blow OFF

8.5 Move Back Spindle to Waiting Point (M241)

The M241 command moves the back headstock (Z2 axis) to the waiting point (Z2 machine coordinate 0.0 mm) in rapid feed after back machining is completed. The back headstock also returns to the waiting point by changing the machining pattern (with W0 argument not specified in \$2).

Command format

`M241`	Move Back Spindle to Waiting Point The back headstock (Z2 axis) moves to the waiting point (Z2 machine coordinate 0.0 mm).

Note

While Z2 axis is in \$1 (e.g., T51's tool is selected in \$1, or K2 argument for T code is specified in \$1), specifying M241 in \$2 causes an alarm to occur. Be sure to specify M241 in \$1. Or, specify G600 (machining pattern cancel) before specifying M241 in \$2.

8.6 Knock-out Advance/Retract (M10, M11)

Outline

Move the knock-out device on back spindle forward or backward.

Command format

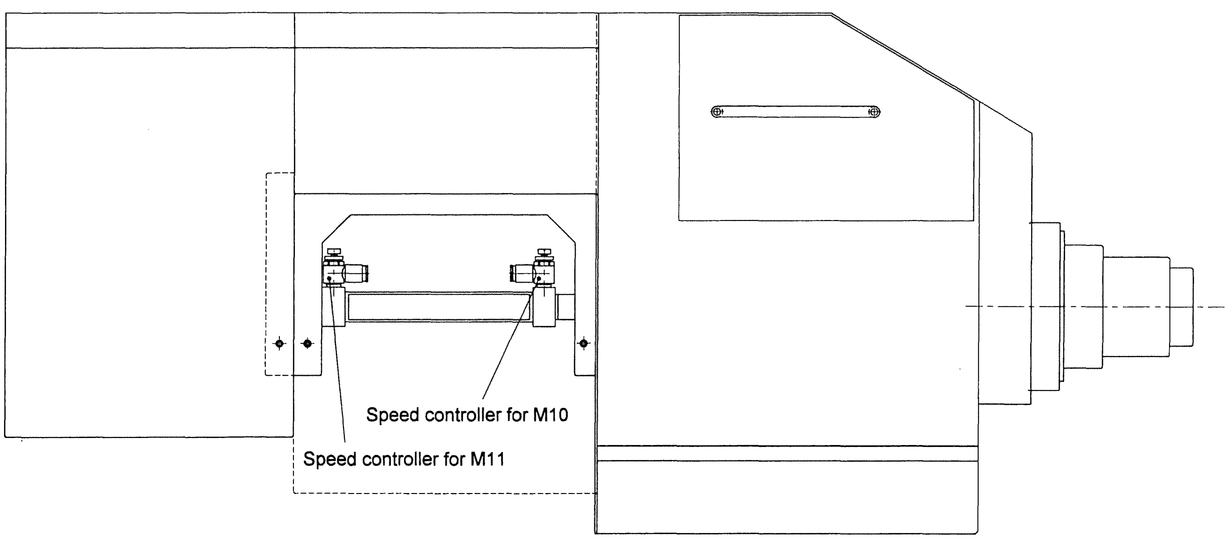
M10	Knock-out advance
M11	Knock-out retract

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.

Notes

- While the back chuck is closed or the back spindle is rotating, the knock-out advance/retract (M10/M11) command causes an alarm to occur.
- Adjusting the speed controller of the air cylinder allows the speed of the knock-out advance/retract to be adjusted.



- If the knock-out pipe fails to work correctly during knock-out process due to jammed workpiece or others, the "Knock-out Overload" alarm occurs. For restting the alarm, see <5.5.11 Restoration from Other Alarms> in the Maintenance Manual.

8.7 Product Collection (M33, M34)

8.7.1 Pick-off collection mode

The back spindle picks off the machined workpiece and carries it into the product receiver box.

a) Move product collection position

Command format

`M33` Move product collection position

`M33 W□ D□ U□ E□`

Argument

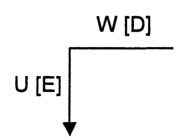
W□ : Specify the position to retract the Z2 axis (distance from zero point of Z2 axis). Specify the value with decimal point. When this argument is omitted, the Z2 axis moves to 0.0 mm.

D□ : Specify the feed rate (per minute) to retract the Z2 axis. Specify the value with decimal point. When this argument is omitted, the Z2 axis moves in rapid feed.

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 –497.0 mm.

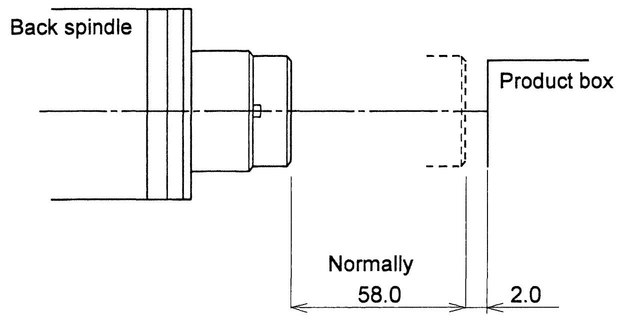
E□ : Specify the feed rate (per minute) when the separator on X2 axis moves. Specify the value with decimal point. When this argument is omitted, the X2 axis moves in rapid feed.

W□ D□ U□ E□ : Can be omitted.



Sample program

$2	
`M33`	Move the product collection position. • If M33 is specified without argument specified, X2 and Z2 axes move to the zero point point and terminates operation with the back spindle chucking the workpiece. • 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 Z58.0`	Move the back spindle to the workpiece collection position in rapid feed.
`M16`	Open the back spindle chuck.
`M10`	Advance the back spindle knock-out device.
`G04 U0.5`	Dwell
`G00 Z0`	Return to the zero point of the back spindle.
`M72`	Turn on the air blow of the back spindle.
`M11`	Return the back spindle knock-out device.
`T3000`	Move the back spindle at the center of main spindle.
`M73`	Turn off the air blow of the back spindle.



Notes

- 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.
- When creating a program after issuing M33 (Move product collection position), consider the following specifications as for the advance position of back spindle (Z2 axis).

"BACK SPINDLE" of the machining data	Advance position of back spindle (Z2 axis)
STANDARD	58.0 – (Protrusion length of back long-neck chuck)
SUPPORT	58.0 – (Stabilizer (=10.0 mm))

Specify M33 (Move product collection position) in $2. Specifying M33 in $1 causes an alarm to occur.

b) sequential operation of product collection

Command format

M34 X□ W□ D□ U□ E□ Z□ F□ T□ B□

Argument

X□: Specify the position to move the X2 axis after product collection. When this argument is omitted, the X2 axis moves to the position 0.0 mm.

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.

U□: Specify the position where the separator on X2 axis collect products. When this argument is omitted, the X2 axis moves to –497.0 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.

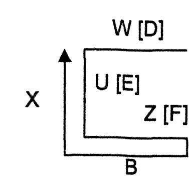
Z□: Specify the position to advance Z2 axis. When this argument is omitted, the Z2 axis moves to 58.0 mm in usual case.

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 blow an air after the product collection. Specify the value with decimal point. When this argument is omitted, the air is blown for two seconds.

B□: Specify the position to move the Z2 axis after product collection. When this argument is omitted, the Z2 axis moves to the position 0.0 mm.

X□ W□ D□ U□ E□ Z□ F□
T□ B□: Can be omitted.



Sample program

$1		$2	
G630	Front/back parallel machining	G630	Front/back parallel machining
T0200		T3100	
:		:	
T0300		M25	Stop back spindle.
:		M34	sequential operation of product collection
:		T3000	Select tool.

Notes

- Specify "STANDARD" in "BACK SPINDLE" of the machining data.
- Specify M34 (sequential operation of product collection) in $2.
  Specifying M34 in $1 causes an alarm to occur.
- The position to advance Z2 axis changes from the standard value (58.0 mm) depending on the specification. Pay attention when specifying the Z argument.

"BACK SPINDLE" of the machining data	Advance position of back spindle (Z2 axis)
STANDARD	58.0 – (Protrusion length of back long-neck chuck)
SUPPORT	58.0 – (Stabilizer (=10.0 mm))

8.7.2 Cut-off and product collection

The product is collected on the back spindle side.

Main spindle and back spindle rotation synchronization command (G814 or G114.1) is optional.

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 sequential operation of product collection.
!L1	Queuing	M16	Open the back spindle chuck.
G814	Rotation synchronization of spindle and back spindle (Option)	M24 S2=1500	Specify 1500 for the back spindle reverse rotation speed.
N11 T0100	Select cut-off tool	!L1	
G00 X26.Z [*1] T01	Position cut-off tool		
G650	Automatic queuing	G650	Pick-off
!L2	Queuing	M72	Turn on the air blow of the back spindle.
		G00 Z−1.0	Position the back spindle 1.0 mm before the workpiece by rapid feed.
G99 G01 X−1.0 F0.03	Cut-off the workpiece	G98 G01 Z [*2] F1000	Advance the back spindle to the chuck position.
G600	Automatic queuing	G04 U1.0	Specify a dwell pause command of one second in the chuck position.
X−3.0 F0.03		M77	Wait until spindle synchronization is completed.
G113	Cancel spindle synchronization mode	M15	Close the back spindle chuck.
M05		M73	Turn off the air blow of the back spindle.
M07		!L2	
G00 Z [*3] T00	Return to the start point.	G600	Cancel the superimpose control.
M56		M25	Stop the back spindle rotation.
G999	Automatic queuing	G999	
N999	Automatic queuing	N999	
M02		M02	
M99		M99	
%		%	

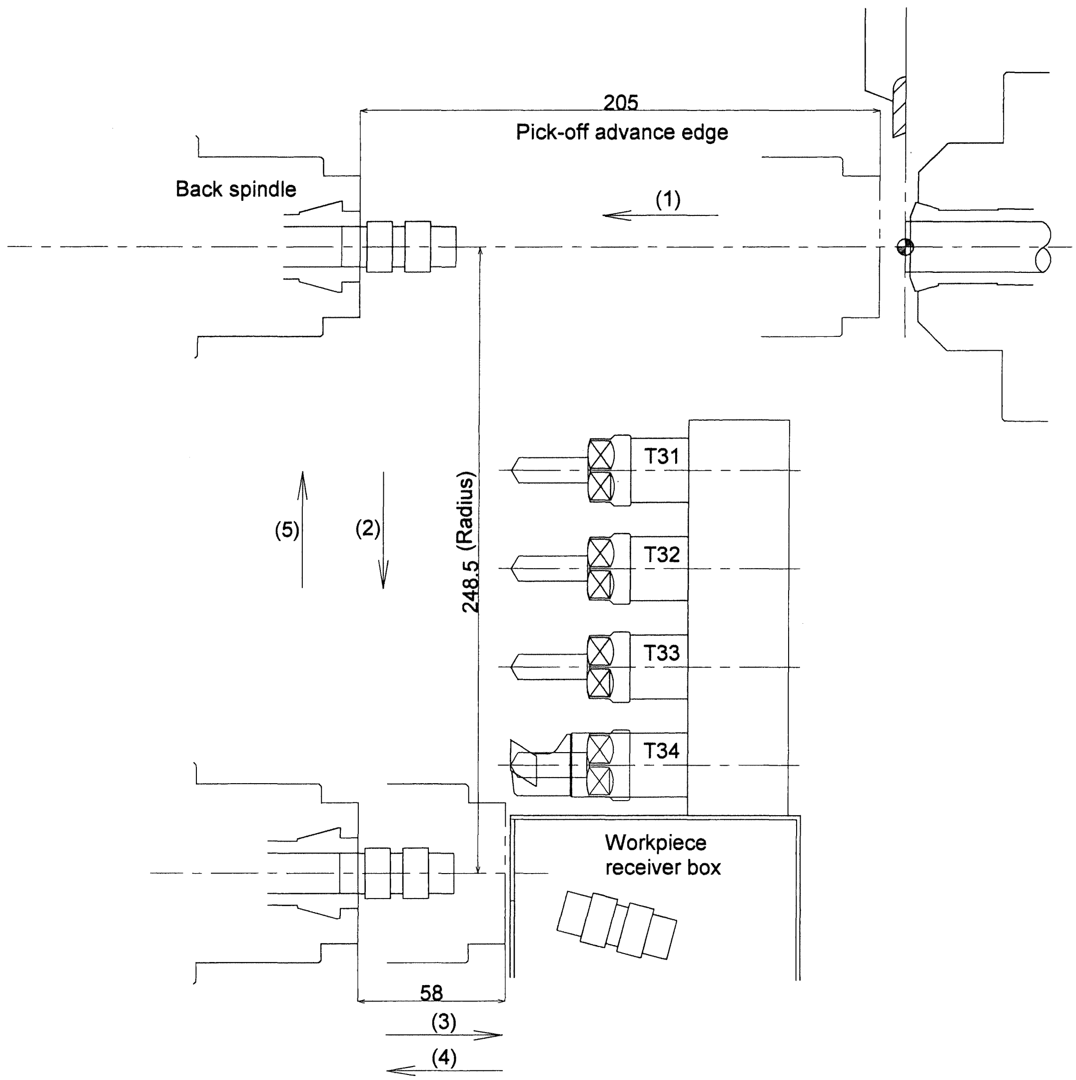
*1: L (product length) + ℓ_3 (tool shift amount) + (tool width)

*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.

Position and sequential operation for product collection

The product after cut-off machining or back machining is collected to a workpiece receiver box at this stroke during this operation.



Description

- The following operations are executed when product collection 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 to the product collection position.
  4. The Z2 axis returns to the zero point after product knock-out.
  5. The X2 axis moves to the center of the guide bushing.

- When an M33 command is issued, the X2 axis in step 2) returns to the zero point to complete the operation for product collection. After that, perform product collection by creating a program.

8.7.3 Basket-collection mode

The finished workpiece is collected in the basket mounted on the back spindle and carried to the product receiver box.

a) Move product collection position

Command format

M33 Move product collection position

M33 W☐ D☐ U☐ E☐

Argument

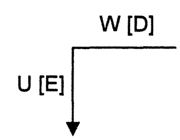
W☐: Specify the position to retract the Z2 axis (distance from zero point of Z2 axis). Specify the value with decimal point. When this argument is omitted, the Z2 axis moves to 0.0 mm.

D☐: Specify the feed rate (per minute) to retract the Z2 axis. Specify the value with decimal point. When this argument is omitted, the Z2 axis moves in rapid feed.

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 –497.0 mm.

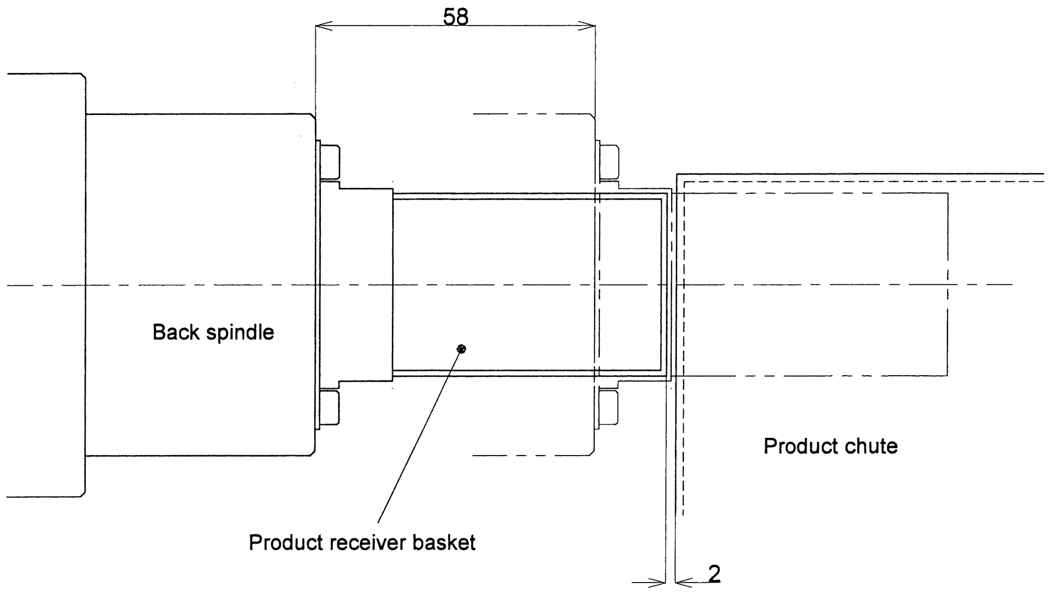
E☐: Specify the feed rate (per minute) when the separator on X2 axis moves. Specify the value with decimal point. When this argument is omitted, the X2 axis moves in rapid feed.

W☐ D☐ U☐ E☐: Can be omitted.



Sample program

$2	
M33	Move the product collection position. • If M33 is specified without argument specified, X2 and Z2 axes move to the zero point point and terminates operation with the back spindle chucking the workpiece. • 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 Z58.0	Move the back spindle to the basket 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.
M31	Turn on the workpiece conveyor.
T3000	Move the back spindle at the center of main spindle.
:	



Notes

- Specify "BASKET" 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.
- Specify M33 (Move product collection position) in $2.
  Specifying M33 in $1 causes an alarm to occur.

b) sequential operation of product collection

Command format

M34 X□ W□ D□ U□ E□ Z□ F□ T□ S□ B□

Argument

X□: Specify the position to move the X2 axis after product collection. Specify the value with decimal point. When this argument is omitted, the X2 axis moves to the position 0.0 mm.

W□: Specify the position to retract the Z2 axis (distance from zero point of Z2 axis). Specify the value with decimal point. 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. Specify the value with decimal point. When this argument is omitted, the Z2 axis moves in rapid feed.

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 –497.0 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. Specify the value with decimal point. When this argument is omitted, the Z2 axis moves to 58.0 mm in usual case.

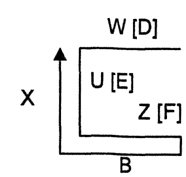
F□: Specify the feed rate (per minute) when Z2 axis advances. Specify the value with decimal point. When this argument is omitted, the Z2 axis moves in rapid feed.

T□: Specify the time to blow an air after the product collection. Specify the value with decimal point. When this argument is omitted, the air is blown 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 move the Z2 axis after product collection. When this argument is omitted, the Z2 axis moves to the position 0.0 mm.

X□ W□ D□ U□ E□ Z□ F□
T□ S□ B□: Can be omitted.



Sample program

$1		$2	
G630	Front/back parallel machining	G630	Front/back parallel machining
T0200		M34	
:			
T0300			
:			

Notes

- Specify "BASKET" in "BACK SPINDLE" of the machining data.
- Specify M34 (sequential operation of product collection) in $2.
  Specifying M34 in $1 causes an alarm to occur.

8.8 Spindle Synchronization Control (G814, G813, G114.1, G113, M77)

When the back spindle performs cut-off process by pick-off, this command permits the cut-off speed and phase of the main spindle to be synchronous/asynchronous with the pick-off speed and phase of the back spindle. See <8.14 Phase Adjustment of Non-conform Material> for phase adjustment.

> **CAUTION**
>
> **Before specifying G814 (spindle synchronization control), be sure to specify M3 S1=[ ] (main spindle rotates forward) and M24 S2=[ ] (back spindle rotates reverse) to adjust the rotation direction.**

Command format

`G814 R[ ]` or `G114.1 R[ ]`

Spindle synchronization

- The back spindle rotates reversibly so that its speed and phase matches with those of the main spindle. (Phase adjustment select mode)
- If R argument is omitted, phase adjustment is not performed, only the spindle rotation synchronization is performed.

`G813` or `G113`

Spindle synchronization cancel
Both spindles rotate at the previous speed.

`M77` Confirm completion of spindle synchronization
Confirms the completion of synchronization between main and back spindles.

Argument

`R[ ]`: Specify the amount of the synchronous spindle phase shift (angle). Specify the value to be added to the value measured or stored automatically in the non-conformed material phase adjustment (preparation or G899 command) as the R argument. The clockwise direction (forward direction) against the spindle is assumed to be positive. Specify R0 if the measured or stored value above is not added. The available values range from 0 to 359.999 (deg). If the argument is omitted, the phase adjustment is not provided.

Sample program

$1		$2	
!2 L1	Queuing	!1 L1	Queuing
M3 S1=1000	Back spindle reverse rotation	M24 S2=1000	Back spindle reverse rotation
!2 L2	Queuing	!1 L2	Queuing
G25	Main and back spindle speed fluctuation detection OFF	G25	Main and back spindle speed fluctuation detection OFF
G50 S2500	Maximum rotation clamp command		
G96 S1=150	Constant surface speed control		
G814	Spindle synchronization command	T3000	
T0100	Cut-off tool select		
G0 X[ ] Z[ ] T01	Cut-off tool positioning		
G650	Pick-off, center support	G650	Pick-off, center support

$1		$2	
		`G00 Z-1.0`	
		`M77`	Waiting for complete spindle synchronization
		`G98 G01 Z□ F1000`	
		`G4 U0.5`	
		`M123`	Back spindle torque limit to 50%
		`M15`	Back spindle chuck close
`!2 L3`	Queuing	`!1 L3`	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		
`!2 L4`	Queuing	`!1 L4`	Queuing
`G26`	Main and back spindle speed fluctuation detection ON	`G26`	Main and back spindle speed fluctuation detection ON
`M124`	Back spindle torque limit cancel		
`!2 L5`	Queuing	`!1 L5`	Queuing
`G813`	Spindle synchronization cancel		
`M5`	Back spindle stop		
`M25`			

Notes

- If the spindle synchronization is used, set the back spindle speed to be equal to or less than 8,000 min^{-1}. It is because the maximum back spindle speed is set to 8,000 min^{-1}.
- Specify a spindle synchronization command (G814 or G114.1) after the back spindle rotation is adjusted to the rotation direction of the main spindle (or M23 or M24 is specified).
- If the workpiece is to be picked off by the back spindle after the G814 or G114.1 command, specify the M77 (complete spindle synchronization) command before the M15 (back spindle chuck close) command.
- Specifying the spindle synchronization (G814 or G114.1) with both the main and back spindles rotated forward (M3 or M23) or backward (M4 or M24) causes an alarm to occur.

8.9 Z1-Z2 Axis Sumperimposition (M130, M131)

The M130 and M131 commands are used to superimpose the Z2 axis (back headstock) on the Z1 axis (front headstock).

Command format

M130	Z1-Z2 axis superimpose control ON The back headstock (Z2 axis) moves along with the move command for front headstock (Z1 axis).
M131	Z1-Z2 axis superimpose control OFF Cancels superimposition of the back headstock (Z2 axis) from the front headstock (Z1 axis).

Sample program

$1		$2	
M03 S1=4000	The main spindle rotates forward and the back spindle rotates reverse.	M24 S2=4000	The main spindle rotates forward and the back spindle rotates reverse.
!2 L10	Queuing	!1 L10	Queuing
G814	Spindle superimpose command	G98 G1 W120.0 F4000	
T0500	Select a rear turning tool.	....	Back spindle advances.
G00 X7.0 Z129.6 T05			
....	Position the turning tool to the position 127.6 mm + rear turning tool width 2 mm.		
G600	Free pattern	G600 W0	Free pattern
		M15	Back spindle chuck closes.
		G04 U0.5	Dwell time
!2 L20	Queuing	!1 L20	Queuing
M130	Z1-Z2 axis superimposition ON		
G50 W–2.0	Specify the coordinate system shift for rear turning tool width –2.0.		
G0 X7.0 Z127.6			
G99 G1 X5.8 Z130.0 F0.02			
X5.0	Machining process		
Z134.8 F0.04			
U–0.8 W0.4 F0.02			
G0 X7.0			
G50 W2.0	Cancel the coordinate system shift for rear turning tool width.		
T0100	Call up a cut-off tool.		
G0 X7.0 Z136.0 T1			
....	Position the cut-off tool at the position 35 mm (whole workpiece length) + 1 mm (cut-off tool width).		
G1 X–1.0 F0.03			
M131	Z1-Z2 axis superimpose control OFF		

$1		$2	
G600	Free pattern (Cancel the machining pattern)	G600	Free pattern (Cancel the machining pattern)
G1 X-3.0 F0.05			
G813	Cancel spindle superimpose command		
M5	The main spindle stops		
M25	The back spindle stops		
G999	Execute the last program	G999	Execute the last program
N999		N999	
M5	The main spindle stops		
M7	The chuck opens		
G0 X-3.0 Z[***] T0			
....	[***] represents the workpiece coordinate set at start time.		
M2		M2	
%		%	

Note

Z1-Z2 axis superimpose control command (M130, M131) can be performed only in automatic operation mode. It cannot be performed in manual operation mode.

8.10 C1-C2 Axis Sumperimposition (M190, M191)

The M190 and M191 commands are used to superimpose the C2 axis (back spindle) on the C1 axis (main spindle).

Command format

M190 C1-C2 axis superimpose control ON
The back spindle (C2 axis) moves along with the move command for 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	
T700	
G0 X21.0 Z77.0 T7	
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
M130 .... Z1-Z2 axis superimpose control ON	
M190 .... C1-C2 axis superimpose control ON	
G50 W–15.0	
G0 X21.0 Z62.0 T7	
:	
:	
G0 X21.0 T0	
G50 W15.0	
M82	
M16	
M131 .... Z1-Z2 axis superimpose control OFF	
M191 .... C1-C2 axis superimpose control OFF	
M20 .... Cancel main spindle C axis function.	
M79 .... Cancel back spindle C axis function.	
:	:
:	:

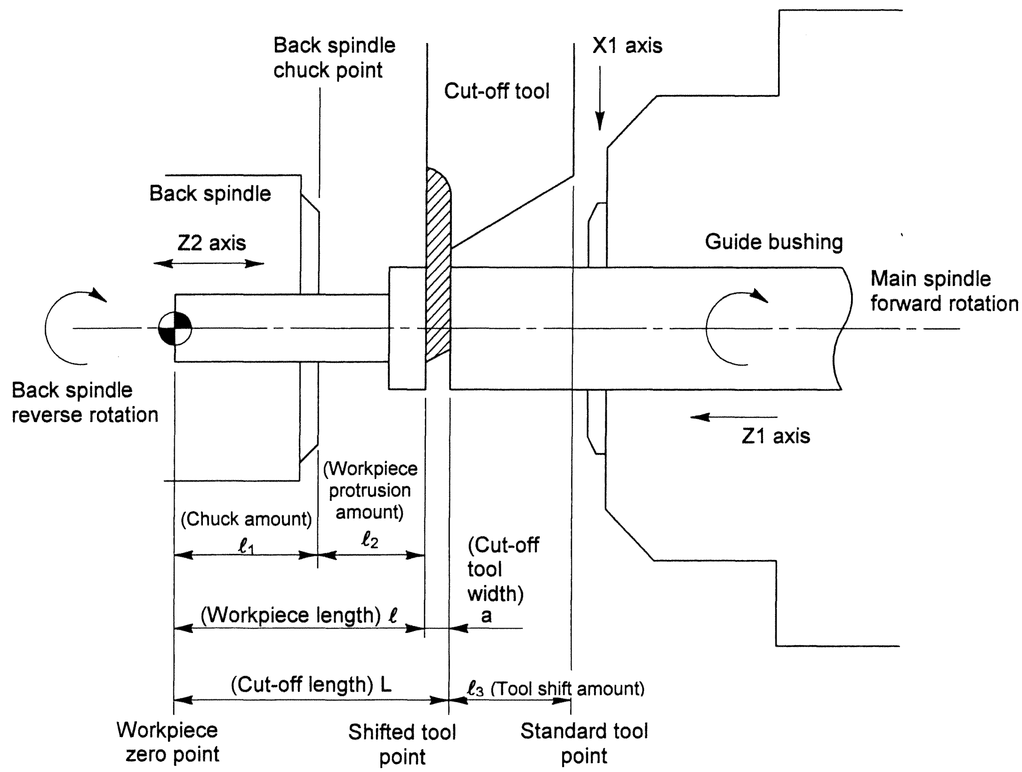
Notes

- To use M190 and M191, options for main spindle C axis and back spindle C axis are required.
- Specify M18 (main spindle C axis function) in $1 and M48 (back spindle C axis function) before 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.

8.11 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 waiting point (Z2-axis: 0.0 mm) to execute the ending process.



Description

1. Select the cut-off tool.
2. Rotate the main spindle in the forward direction and the back spindle in the reverse direction at the same speed.
3. Open the back spindle chuck.
4. 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 in this case. As a result, the following expression is obtained.
   $Z = \ell$ (workpiece length) + a (cut-off tool width) + $\ell 3$ (tool shift amount)
5. Move the back spindle to the back spindle chuck point. (Z2 axis)
6. Make the back spindle chuck the workpiece.
7. Cut off the workpiece by the cut-off tool.
8. Return the back spindle to the waiting point.
9. Execute the ending process.

Notes

- Advance the back spindle up to the point 12 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.
- 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).

8.11.1 Pick-off process

Sample program

	$1		$2
(1)	M03 S1=3000		M24 S2=3000
(2)	G814		
(3)	T0100		
(4)	G00 X7.0 Z32.0 T1		
(5)	G650	(5)	G650
		(6)	G00 Z–1.0 (a)
			M77
		(7)	G98 G01 Z20.0 F1000
		(8)	G04 U0.5
		(9)	M15
		(10)	G04 U0.5
(11)	!2 L1	(11)	!1 L1
(12)	G01 X–0.5 F0.015		
(13)	G600	(13)	G600 (b)
	X–3.0 F0.03		M25
	M05		
	M07		
	G0 Z0 T0		
	M56		
	G999		G999
	N999		N999
	M02		M02
	%		%

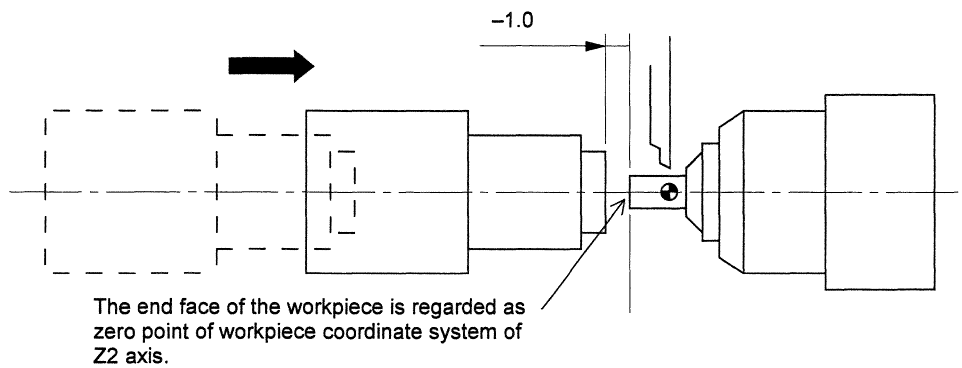
Description

(1) Adjust the main spindle speed to the back spindle speed.
(2) Spindle synchronization command
(3) Cut-off tool selection
(4) Cut-off tool positioning
(5) Set the back spindle workpiece coordinate system so that the workpiece end face on the front spindle becomes zero.
(6) The back spindle is positioned 1 mm before the end face of the workpiece in rapid feed.
(7) The back spindle advances 20 mm from the end face of the workpiece in cutting feed.
(8) Dwell for 0.5 second
(9) Back spindle chuck close
(10) Dwell for 0.5 second
(11) Queuing
(12) Cut-off machining
(13) Machining pattern cancel. The back headstock (Z2 axis) moves to the waiting point in rapid feed.

Operation

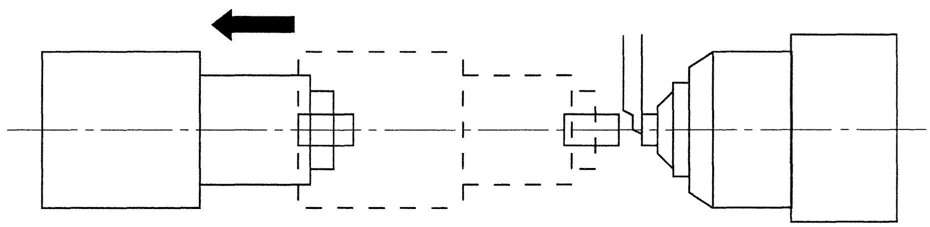
(a)

The back spindle advances to the position Z–1.0 in rapid feed.



(b)

The back headstock (Z2 axis) moves to the retract point (machine coordinate 0.0 mm) in rapid feed.

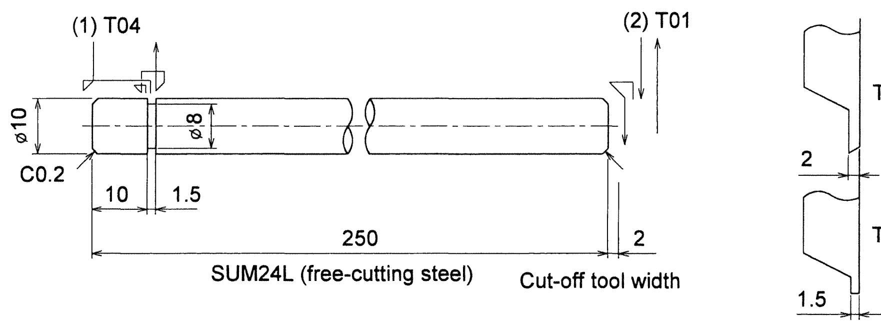


8.11.2 Re-chucking of a Long Workpiece

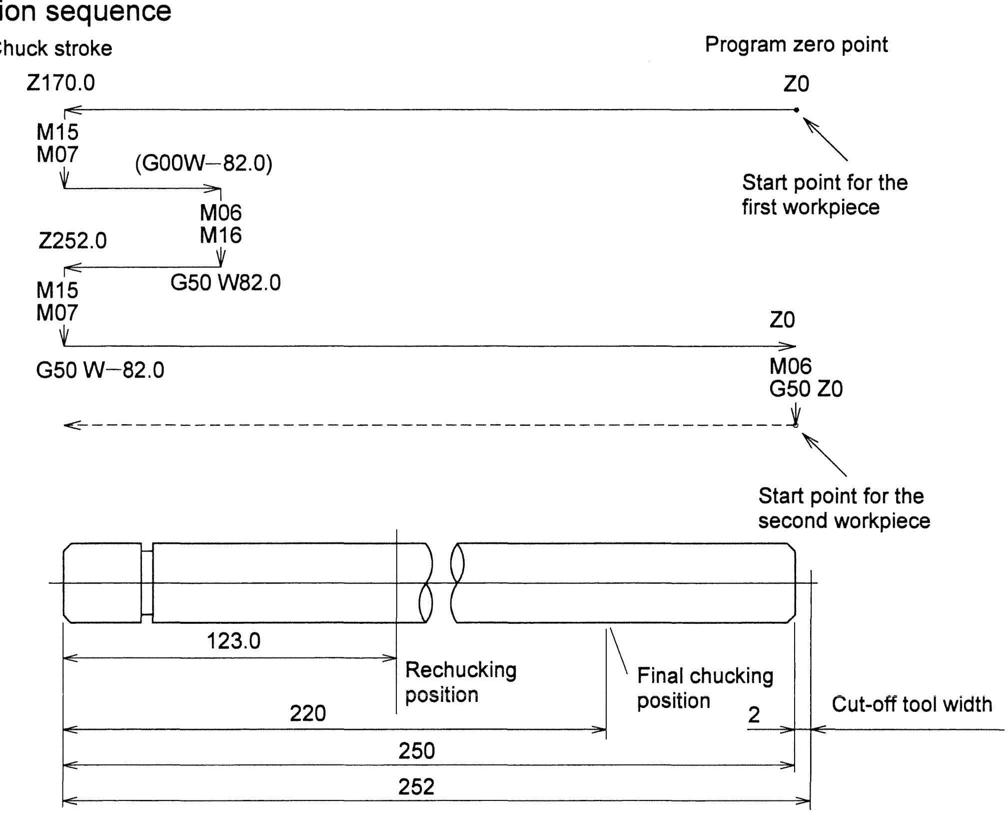
In general, the machining length of a workpiece must be within the Z1 axis stroke. If the machining length of a workpiece exceeds the Z1 axis stroke, the workpiece needs to be re-chucked by the main spindle and the back spindle. In this case, create the program including the re-chucking process as shown below.

The machine without guide bushing device provides machining for a material protruded by the total workpiece length. Accordingly, if the material is protruded by a value larger than the material outer diameter × 2.5, the front end of the material may swing during front spindle rotation, which may disable proper machining. If this occurs, re-checking makes it possible to perform machining with the minimum swing at each point.

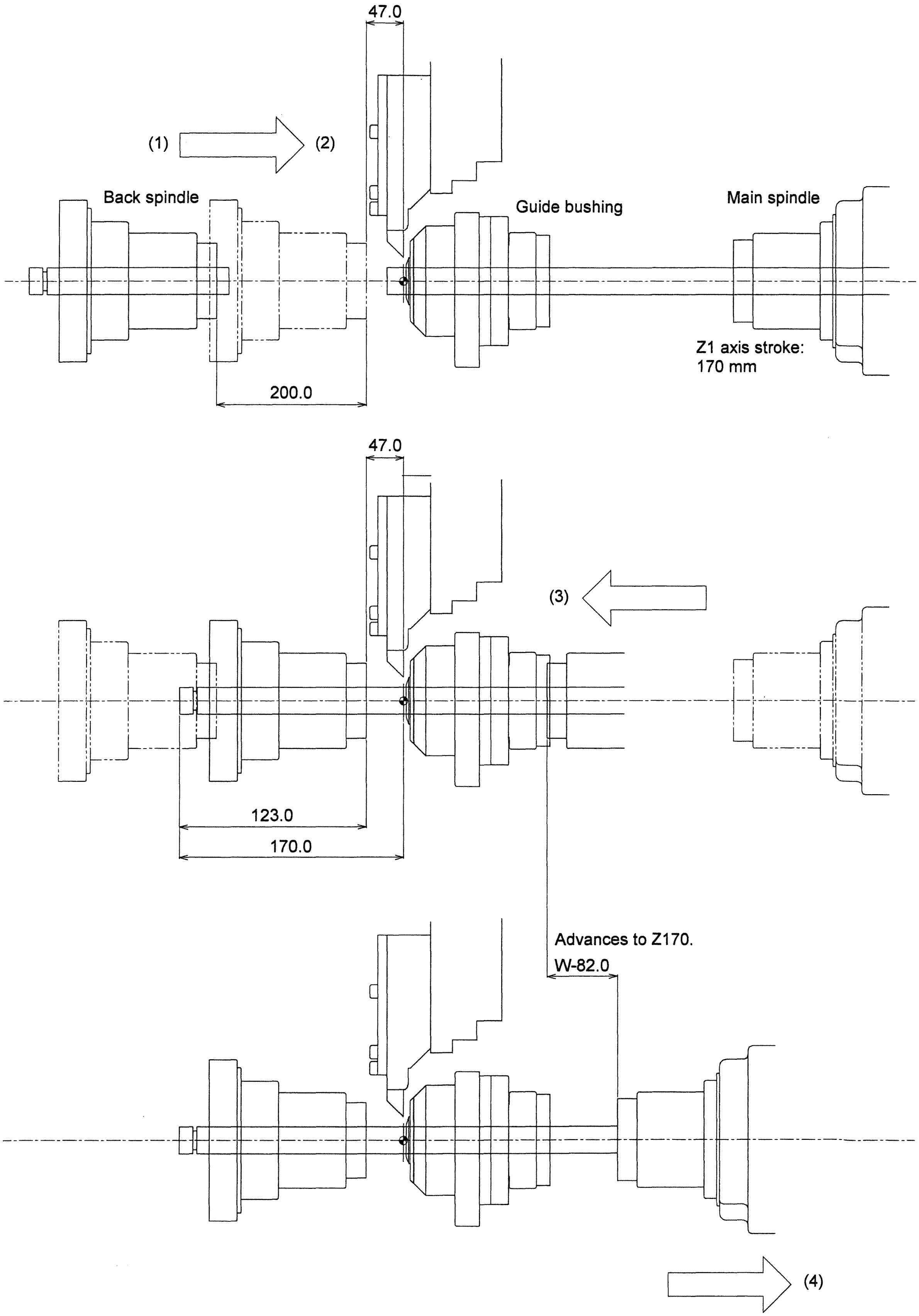
Machining layout



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.

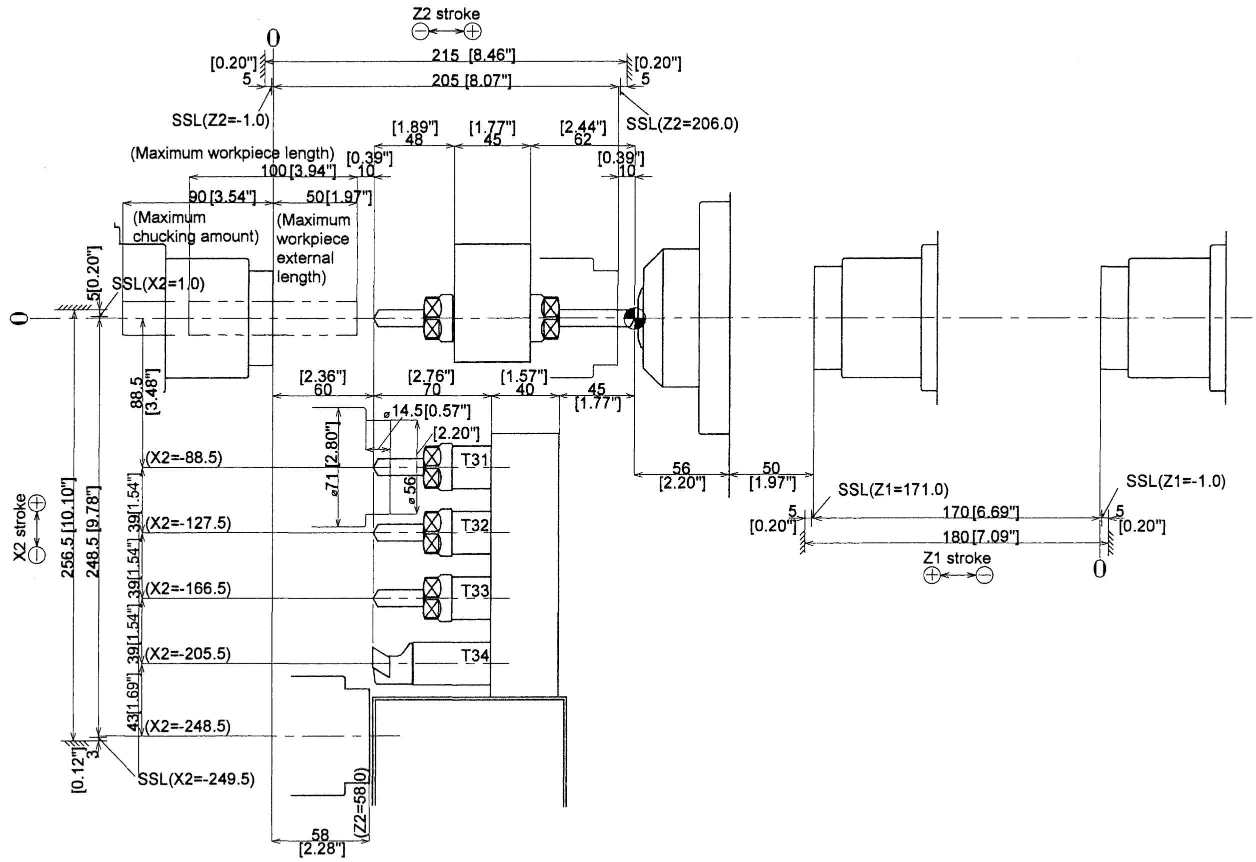
Sample program

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	Cut-off tool selection		
M05	The main spindle stops.	G50 Z0	
!2 L1	Waiting for !1L1	!1 L1	Waiting for !2L1 (1)
		M16	The back spindle chuck opens.
		G0 Z200.0	The back spindle advances 200 mm from the Z2 axis zero point. (2)
		G04 U0.5	
!2 L2	Waiting for !1L2	!1 L2	Waiting for !2L2
G00 Z170.0 T00	Determines rechucking position, and cancels offset (3)		
G04 U0.5			
M15	The back spindle chuck closes.		
G04 U0.5			
M07	The main spindle chuck opens.		
G04 U0.5			
G00 W–82.0	The Z1 axis retracts to –82.0 (4)		
M06	The main spindle chuck closes.		
G04 U0.5			
M16	The back spindle chucks opens.		
G04 U0.5			
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}.		

$1	$2
G00 X11.0 Z252.0 T01	
G650	G650 W0
	G98 G1 Z220.0 F3000
	G04 U0.5
	M15
!2 L3	!1 L3
G01 X9.6 F0.03	
M130 .... Z1-Z2 Superimposition ON	
X11.0 F0.2	
W-0.7	
X9.6 W0.7 F0.03	
X-0.5	
M131 .... Z1-Z2 Superimposition OFF	
G600	G600
X-3.0	
M05	
M25	
M07	
G50 W-82.0	
G0 Z0 T00	
M56	
G999	G999
N999	N999
M02	M2
%	%

: Program portion for long workpieces



* 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

Put a checkmark on "LONG WORK" on Machine structure screen to enable the long workpiece device. In addition, Specify "SUPPORT" for "BACK SPINDLE (#64)" in machining data.

8.12 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 the commands following N999 at the end of the end process. The commands must be specified in the sequence of M02 and then %.

Command format

`G999` Execute last program

Axis control groups

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	
		:	
		M241	
		T3100	
		:	
		M25	
		M34	
		T3000	

$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
%		%

Notes

- The last program is not performed when No.16 "LAST PART VALID" is set to 0 (disabled) on Operator's panel screen.
- 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 material. If the T51's holder (sleeve holder) is not specified in the last program and only the T31's holder (back drilling 4-tool holder) is specified, you need not move the cut-off tool. Accordingly, the processes (1) and (2) can be omitted.
- While the program pre-analysis function is enabled, be sure to coincide the machine coordinates of X1, Y1, Z1, X2, and Z2 axes before calling the last program and after it was called when terminating the last program. If not, an alarm may occur.
- Cancel the coordinate system shift command and compensation command before executing the G999 command.
- The M56 (product count) command must be specified before the G999 command.

8.13 Back Pick-Off Failure Detect (M50)

Operation flow of the Back Pick-Off Failure Detect (M50) function is as follows:

1. Perform cut-off machining.
2. Upon completion of cut-off machining, stop the main spindle and the back spindle, and cancel spindle synchronization.
3. Rotate the main spindle, and execute the M50 (Back Pick-Off Failure Detect) command.
4. If the workpiece has not bee cut off completely, the main spindle rotation involves back spindle rotation (50 min^{-1} or more), and an alarm is issued.

Sample program

$1		$2	
:			
:			
T0100	Cut-off tool selection		
G01 X☐ F☐	Cut-off machining		
M5	Main spindle stop		
M25	Back spindle stop		
G813	Spindle synchronization cancel		
M03 S1=300	Main spindle rotates forward		
G4 U5.0	Dwell		
M50	Back Pick-Off Failure Detect		
G600	Machining pattern cancel	G600	Machining pattern cancel (Back spindle moves to the waiting point.)
:			

8.14 Phase Adjustment of Non-conform Material

A phase shift occurs between the spindle and guide bushing and between the main spindle and back spindle when machining non-conform materials (hexagonal material, etc). Use this command to store the values of phase shifts in the memory of the NC unit to adjust the phase.

Be sure to specify this command to adjust the phase when mounting the non-conform material chuck and guide bushing.

8.14.1 Non-conformed material phase adjustment on Preparation screen

- When a non-conformed material is chucked on the main spindle with another non-conformed material chucked on the back spindle:

Procedure

1. Adjust the phase between the main spindle and guide bushing. See <Section 7.7 Phase Adjustment between Spindle and Guide Bushing (Non-conformed Material)> in Operator's Manual.

2. Store the phase shift between the main spindle and the back spindle.

   2-1 Press the Preparation key to display the Preparation screen.

   Press the Menu selection key, the menu key [ADJUST], [PHASE], and [M.-B.].

   2-2 Press the Start key to lock the door. Press the menu key [M.-B.] and the Start key at the same time.

   2-3 When the Start key starts flashing, put the material (non-conformed material) through the main spindle and the back spindle.

   2-4 Press the Start key again to lock the door. Press the menu key [M.-B.] and the Start key at the same time.

   The main spindle and back spindle chucks automatically close, then the phase shift between the main spindle and the phase spindle is stored.

3. Adjust the phases of the main spindle and the back spindle.
   Specify a spindle synchronization control command (G814 or G114.1) with phase adjustment (R argument) in the machining program to chuck the material on the main and back spindles. (See <8.8 Spindle Synchronization Control (G814, G813, G114.1, G113, M77)>.) This will cause the phase shift stored in step 2 to be automatically canceled for phase adjustment.

- When a round bar is chucked on the main spindle with a non-conformed material chucked on the back spindle:

Procedure

1. Adjust the phase between the main spindle and guide bushing. See <Section 7.7 Phase Adjustment between Spindle and Guide Bushing (Non-conformed Material)> in Operator's Manual.
2. Store the phase shift between the main spindle and the back spindle.

   2-1 Press the Preparation key to display the Preparation screen.

   Press the Menu selection key, the menu key [ADJUST], [PHASE], [M.-B.], and [NO CHK].

   2-2 Press the Start key to lock the door. Press the menu key [M.-B.] and the Start key at the same time.

   (The spindle chuck does not open.)

   2-3 When the Start key starts flashing, press the menu key [Z1 FED] or [Z2 FED] and put the material through the back spindle by using the handle in the manual mode.

   2-4 Press the Start key again to lock the door. Press the menu key [M.-B.] and the Start key at the same time.

   The main spindle and back spindle chucks automatically close, then the phase shift between the main spindle and the phase spindle is stored.
3. Adjust the phases of the main spindle and the back spindle.
   Specify a spindle synchronization control command (G814 or G114.1) with phase adjustment (R argument) in the machining program to chuck the material on the main and back spindles. (See <8.8 Spindle Synchronization Control (G814, G813, G114.1, G113, M77)>.) This will cause the phase shift stored in step 2 to be automatically canceled for phase adjustment.

8.14.2 Non-conformed material phase adjustment in MDI mode

- When a non-conformed material is chucked on the main spindle with another non-conformed material chucked on the back spindle:

Command format

```
$1
G899        Store phase shifts between spindles
```

Procedure

1. Adjust the phase between the main spindle and guide bushing. See <Section 7.7 Phase Adjustment between Spindle and Guide Bushing (Non-conformed Material)> in Operator's Manual.
2. Store the phase shift between the main spindle and the back spindle.

   2-1 Specify G899 for $1 in the MDI mode.

   2-2 Press the Start key START [I] to lock the door, then press the Start key START [I] again.

   2-3 When the Start key START [I] starts flashing, put the material through the back spindle by using the handle in the manual mode.

   2-4 Press the Start key START [I] to lock the door, then press the Start key START [I] again.
   The main spindle and back spindle chucks automatically close, then the phase shift between the main spindle and the phase spindle is stored.
3. Adjust the phases of the main spindle and the back spindle.
   Specify a spindle synchronization control command (G814 or G114.1) with phase adjustment (R argument) in the machining program to chuck the material on the main and back spindles. (See <8.8 Spindle Synchronization Control (G814, G813, G114.1, G113, M77)>.) This will cause the phase shift stored in step 2 to be automatically canceled for phase adjustment.

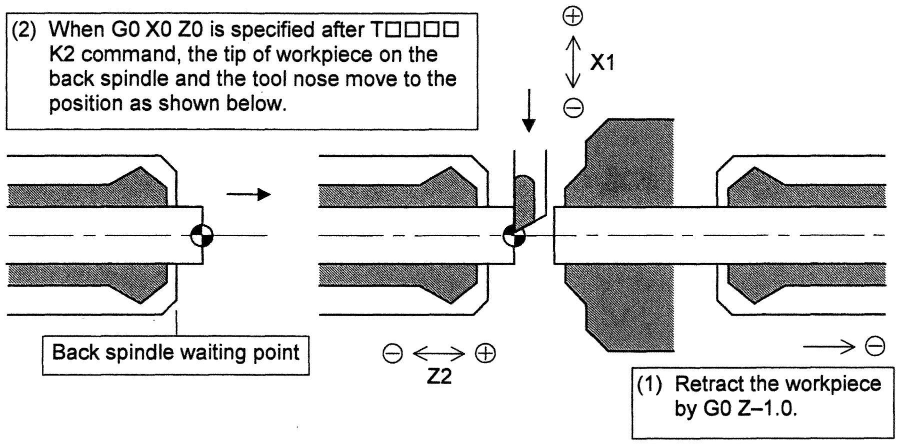
Notes

- In the MDI mode, the phase shifts between the round bar on the main spindle and the non-conformed material on the back spindle cannot be stored. Make the phase adjustment according to <8.14.1 Non-conformed material phase adjustment on Preparation screen>.
- The G899 command is intended for adjustment. Specifying G899 in other than the MDI mode causes an alarm to occur.
- Be sure to specify G899 for $1. If specified for $2, an alarm will occur.

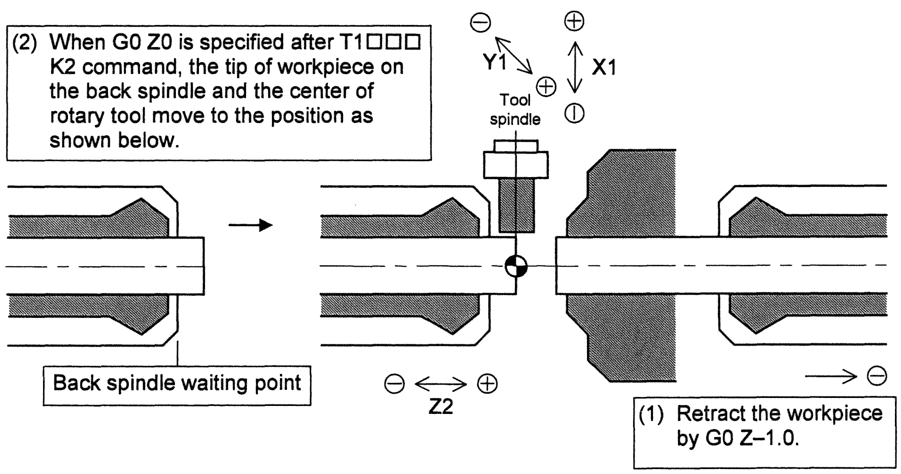
8.15 Machining with Tool or Rotary Tool in Back Machining

Operation example

Machining with tool



Machining with rotary tool



Sample program

$1	$2	
G600	G600	Free pattern (Machining pattern cancel)
:	:	
G0 Z–1.0 T0	:	Retract the workpiece at front side.
!2 L1	!1 L1	Queuing (or G600)
		Specify the back spindle rotation by tool set command.
	M2□ S2=□□□□	
	T300 K2	The axes of axis control group $2 are changed to X1, Y1, Z2 and C2 by the K2 argument.
	G0 X21.0 Z0 T3	□ can be used in the program for either of $1 and $2. If used in $1, the axes of axis control group $1 are changed to X1, Y1, Z2, and C2.
	G1 X0 F0.1	
	:	Machining process
	:	
	G0 X21.0	
	G0 Z–10.0 T0	
	T□□□□ K2	If the K2 argument is used for machining by the next tool, specify K2 following the tool number.
	:	
	:	
G600	G600	Cancel machining pattern (free pattern) after finishing machining with K2 argument.
(X1, Y1, Z1, C1)	(Z2, C2)	(By selecting free pattern again, the axes of axis control group $2 are changed to Z2 and C2.

Notes

- The positioning point of Z2 axis is changed depending on the squares of tool (□12,□13, or □16) selected in tool layout pattern. (The position of the rotary tool Z2 axis is not changed.) In the free tool layout, □16 is used. (To use a □12 tool, coordinate system command G50 W–4.0 must be specified.)
- Specify the command to stop rotation of rotary tool in $1. (See <4.6.4 Tool spindle rotation (M80, M81, M82)>.
- To perform machining with rotary tool in $1, specify M78 and M48 after issuing T□□□□ K2 command.
  See also <4.3 T Code Arguments>.
- When machining the back workpiece by using the rotary tool, the back spindle may interfere with the drill sleeve holder. If "Interference Check" function is disabled (see <4.6.12 Interference Check Function (M88, M89)>, pay strict attention to interference between the sleeve holder and the back spindle, or 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 for back and front machining.

  When the end-face machining rotary tool BSE107 is mounted on holder □12 (BTF2212) of tool spindle drive unit U31B, an interference alarm occurs if T07 is called by the command with K2 argument specified. To machine the workpiece with the interference check function disabled, allowable operation range is restricted because the end-face drilling rotary tool approaches near the back spindle. Therefore, pay strict attention to interference when machining the workpiece in this state.

8.16 Back Spindle Feed Per Rotation ON/OFF (G44/G43)

Outline

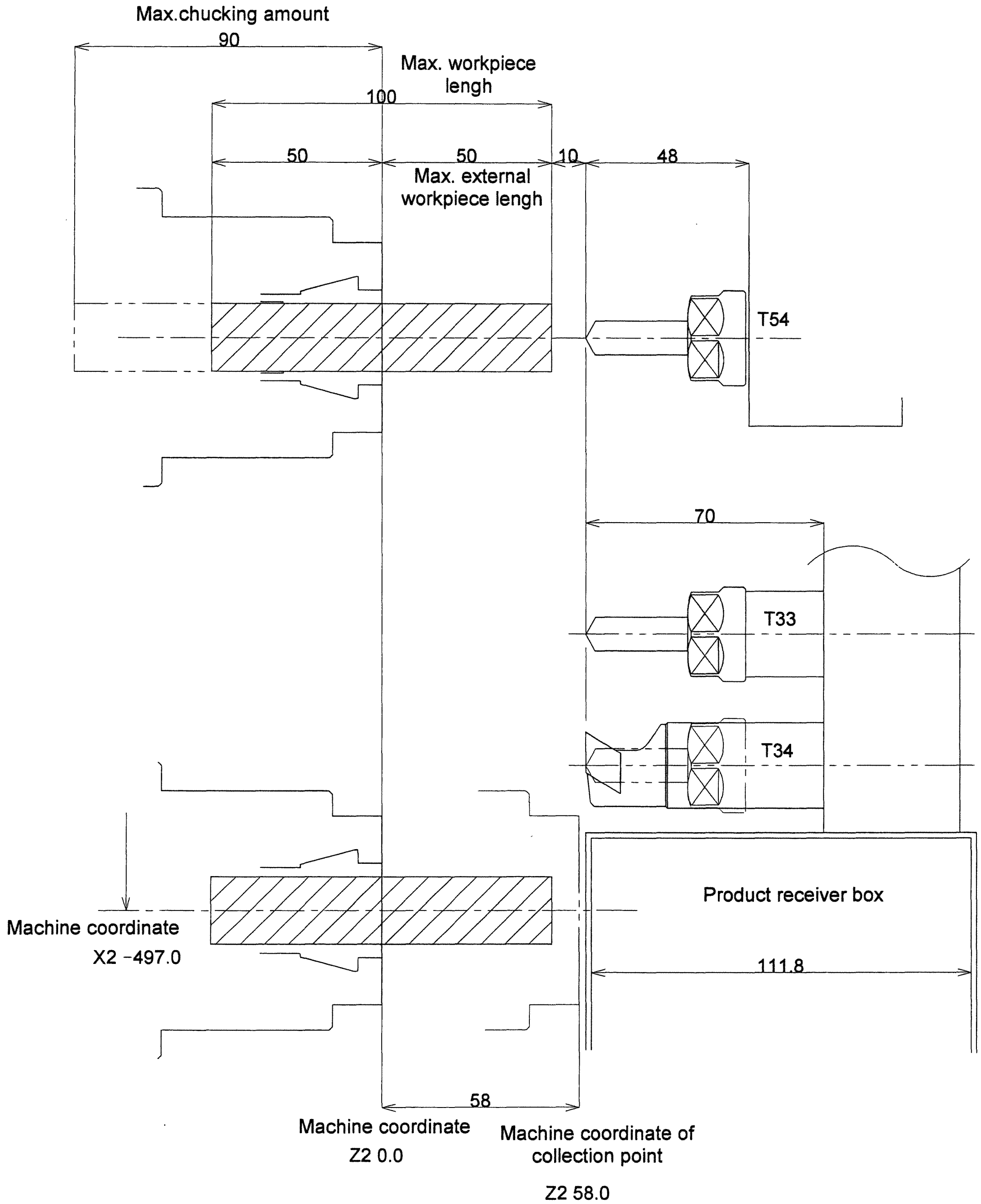
Feed per rotation commands are enabled for the back spindle.

The back spindle feed per rotation ON command (G44) in each axis control group provides feed per rotation at the speed of the back spindle (S2). The back spindle feed per rotation OFF command (G43) in each axis control group provides feed per rotation at the speed of the main spindle (S1). The initial values are the main spindle feed per rotation command in axis control group 1 ($1) and the back spindle feed per rotation command in axis control group 2 ($2).

Command format

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

8.17 Stroke and Notes on Machining



- Maximum workpiece length

  When a workpiece is collected in product receiver mode in the knock-out mode, the maximum workpiece length can be 100.0 mm.

- Maximum back workpiece protrusion length

  The maximum back workpiece protrusion length from the back spindle cap nut can be 50.0 mm. If the protrusion length is set to longer than 50.0 mm, interference between the separator for back machining tool product separation and the back workpiece may possibly occur.

  Note that, if the back machining tool is protruded to be longer than the length in the normal tool set due to a certain machining problem (by entering a negative value to "Longitudinal" on the Preparation screen), the maximum back workpiece protrusion length is shorter by the protrusion length.

- Maximum chuck amount

  The maximum length by which a workpiece can be intruded into the back spindle from the back spindle cap nut (the position of the knock-out front end after the M11 command) is 90.0 mm.

  However, the position of the knock-out front end can be adjusted a little. Accordingly, note interference between the knock-out and the workpiece if the front end position is changed.

- A tool of T51's is normally located at the position separated from the holder end face by 48.0 mm.

- A tool of T31's is normally located at the position separated from the holder end face by 70.0 mm.

- When M34 (sequential operation of product separation) is specified, the machine coordinate at the separation point is X2 –497.0 Z2 58.0.

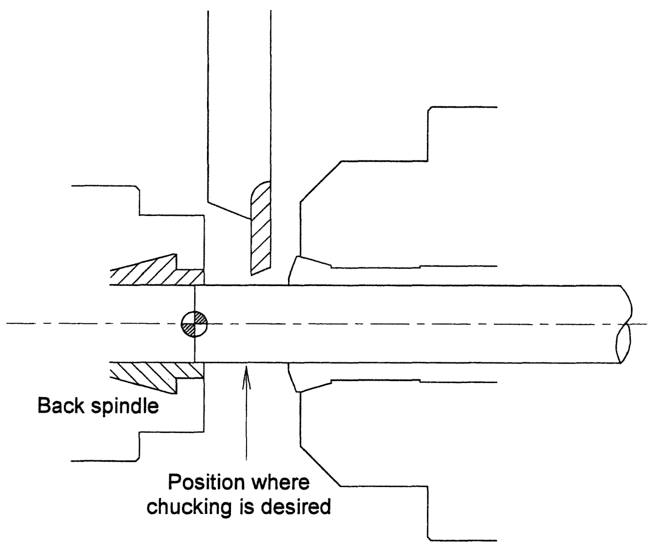
Maximum workpiece length (knock-out mode)	100.0mm
Maximum back workpiece protrusion length	50.0mm
Maximum chucking length (knock-out mode)	90.0mm

* All the values are defined for normal setting.

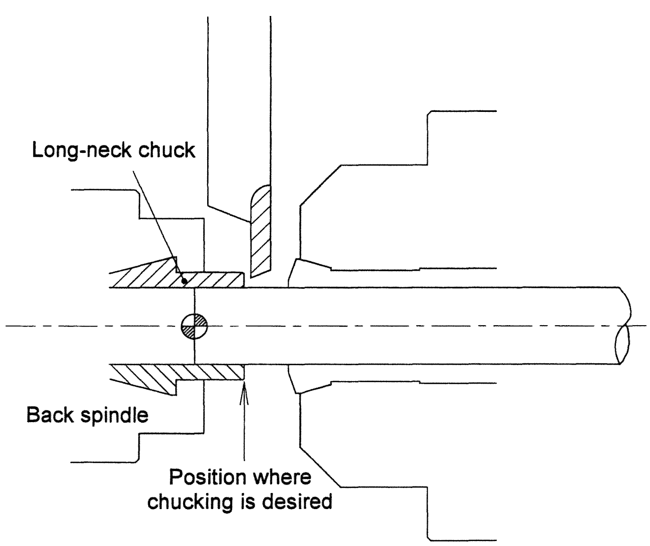
8.18 Use of Machining Data "Back Long-neck Chuck Protrusion Length"

A long-neck chuck may be used if right-handed cut-off must be used for machining and the back chucking position is close to the cut-off position.

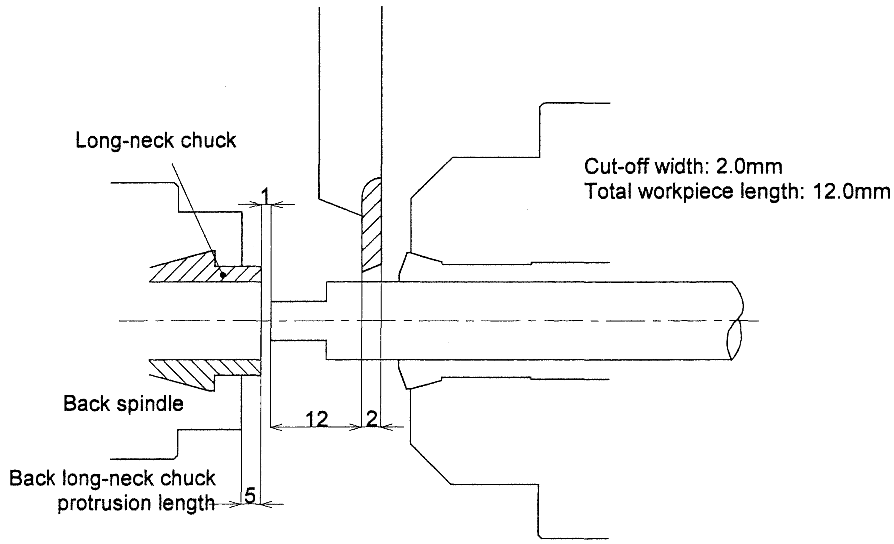
- Use of normal chuck



- 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 as machining data "back long-neck chuck protrusion."



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 workpiece protrusion length." In this example, enter "3.0."

Sample program

$1	$2	
:		
T100		
G0 X21.0 Z14.0 T1		
G650	G650	
	M72	
	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	
:	:	

Notes

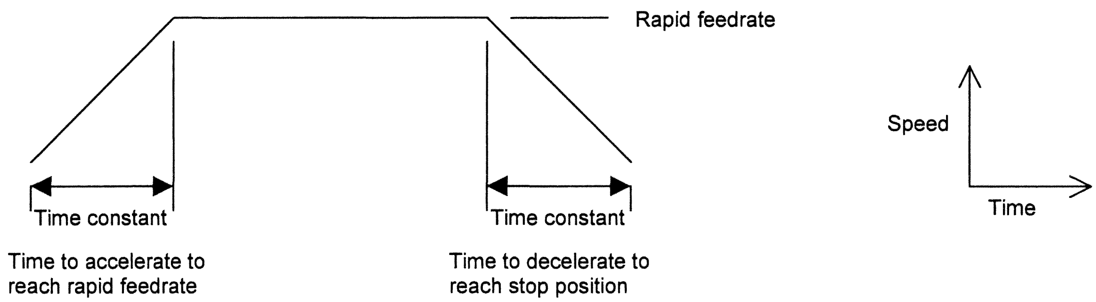
- Select the long-neck chuck and the cut-off tool properly by calculating their shapes which may not cause any interference between them to occur.
- To use a program for a machine prior to the A220VII-type machines as a program using one or more long-neck chucks, change machining data "back long-neck chuck protrusion length" and the pick-off part in the program.
- The product collection point is changed according to the value specified for "back workpiece protrusion length" in the machining data. See <8.7 Product Collection (M33, M34)> for details.

8.19 Setting Acceleration/Deceleration Time Constant in Rapid Feed ON/OFF (M360/M361)

You can change an acceleration/deceleration time constant in rapid feedrate of X1, Z1, and Y1 axis while running the program.
This function is effective to machine the workpiece in which the finish accuracy is required. Such accuracy can be obtained by changing the time constant for the gang tool post (X1 and Y1 axes) during executing G630 command in front/back parallel machining.
Note that rapid feedrate at the tool selection for the gang tool post may become slower depending on the changed time constant value, and the cycle time may be affected.

Acceleration/deceleration time constant of rapid feedrate

The axis specified to feed rapidly by G0 code (rapid feed) accelerates to the rapid feedrate with a constant gradient, then decelerates to the stop position with a constant gradient. This time period is called "acceleration/deceleration time constant". The time required for acceleration is equal to that for deceleration.



Initial value of time constant for each axis

Axis	Acceleration/deceleration time constant
X1	42 msec
Z1	74 msec
Y1	74 msec

Command format

`M360 X□ Z□ Y□` Acceleration/deceleration time constant ON

`X:` Specify a multiple of initial value of acceleration/deceleration time constant for X1 axis with decimal point. Available range is between 1 and 15.

`Z:` Specify a multiple of initial value of acceleration/deceleration time constant for Z1 axis with decimal point. Available range is between 1 and 15.

`Y:` Specify a multiple of initial value of acceleration/deceleration time constant for Y1 axis with decimal point. Available range is between 1 and 15.

`M361` Acceleration/deceleration time constant OFF

Restore the initial value of acceleration/deceleration time constant for each axis.
The initial value is also restored if the running of program is aborted and the Reset key is pressed.

Sample program

In the example below, when the finish machining by back machining tool in $2 and tool selection in $1 overlaps, specify M361 and M360 commands before and after the tool selection command. This supresses the tool selection in $1 to precede the finish accuracy of back machining.

$1		$2
:		
M360 X10.0 Y10.0	10 times as for acceleration/deceleration time constant of X1 and Y1 axes	:
T300	Tool selection	Finish machining by back machining tool
M361	Initial value of acceleration/deceleration time constant of X1 and Y1 axes	:
:		

Notes

- Specify M360 and M361 in $1. Specifying them in $2 causes an alarm to occur.
- The acceleration/deceleration time constant for X1 and Y1 axes of gang tool post and Z1 axis of front headstock can be changed.
- The cycle time may become slower by changing the value of acceleration/deceleration time constant.
- Determine the magnification (argument) for acceleration/deceleration time constant according to the finish accuracy of back machining.

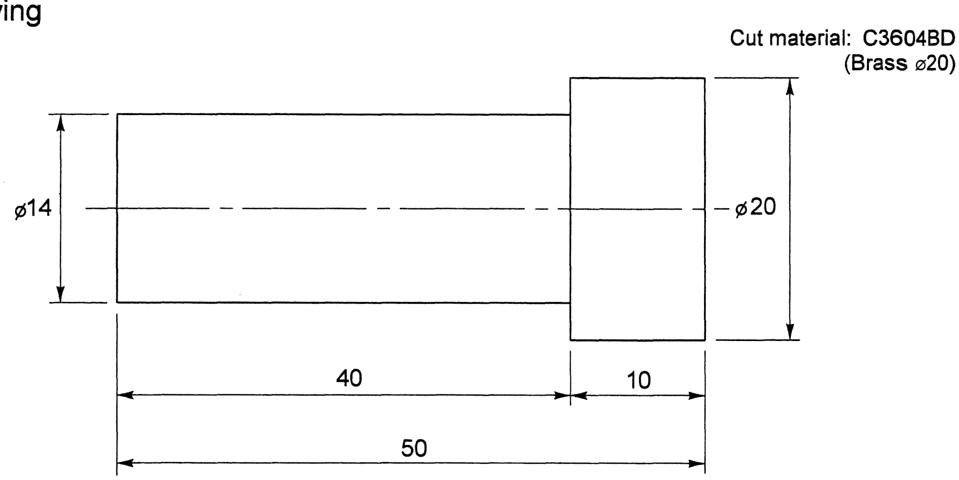
Product Code	C	–	A	2	2	0	P	L	VII					
Document Code	2	E	1	–	0	8	0	1						

9. Programming Practices

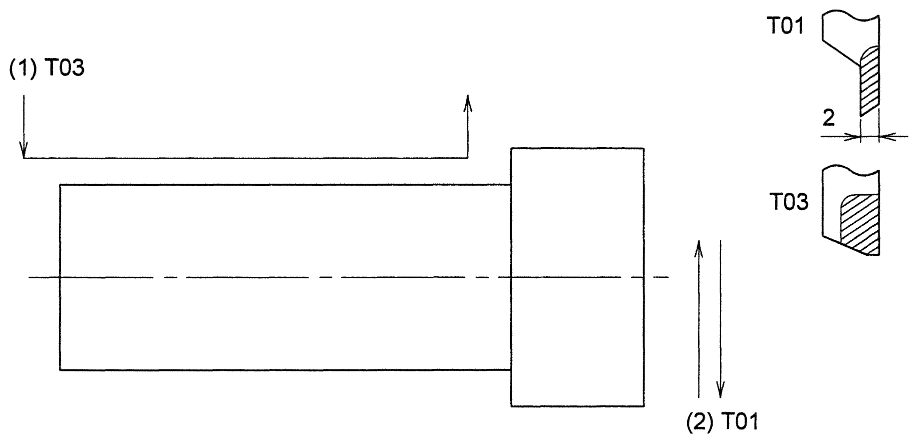
(Blank page)

9.1 Programming Practice 1 (Front Machining)

Machining drawing



Tool layout



Sample program

$1	$2	
O0001		.... Program number
$1	$2	.... Axis control group commands
G50 Z0		.... Coordinate system setting
G600	G600	.... Free pattern (Machining pattern cancel) (Specifies the end point of M320 (basket advance) command)
M06	M34	.... Closes the chuck. Sequential operation for product collection
G00 X21.0 Z–0.5		.... Moves the workpiece away from the tool.
M03 S1=2500 G99		.... Main spindle forward rotation (2,500 min^{-1}) Per rotation feed (mm/rev)
N0103 T0300		.... Gives a sequence number and selects a T03 tool.
G00 X14.0 Z–0.5 T01		.... Positioning to ø14 in rapid feed., and issue tool offset No. 1
G01 Z40.0 F0.05		.... Cuts to 40 mm in longitudinal direction (⌀14).
X21.0		.... Cuts to the material outer diameter + 0.5 mm (radius value).
!2 L1	!1 L1	
N0204 T0100		.... Gives a sequence number and selects a T01 tool.
G00 X21.0 Z52.0 T02		.... Positioning to cut-off point in rapid feed, and issue tool offset No. 2
M320 X3.0		.... Advances to the workpiece separator position of the back spindle.
G99 G01 X–1.0 F0.03		.... Cuts off the workpiece.
X–3.0 F0.05		
G600	G600	.... Free pattern (Machining pattern cancel) (The back spindle workpiece separator retracts.)
M05		.... Main spindle rotation stop
M07		.... Opens the chuck.
G00 X–3.0 Z0 T00		.... Returns to the start point.
M56		.... Products count
G999	G999	.... Last program execution command
	M34	.... Sequential operation for product collection
G600	G600	.... Free pattern (Machining pattern cancel)
N999	N999	.... Last program queuing
M02	M02	.... One cycle stop
%	%	.... Stop code

Spindle speed

$$N = \frac{V}{\pi D} \times 1{,}000$$

$$= \frac{160}{3.14 \times 20} \times 1{,}000$$

$$\approx 2546$$

$$\approx 2500 \text{min}^{-1}$$

V: 160 m/min
D: ⌀20 mm
π: 3.14

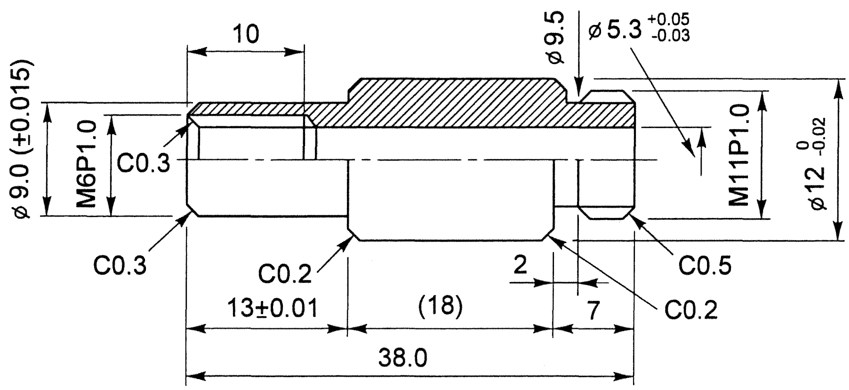
According to the cutting condition table
$F_X = 0.03$ mm/rev
$F_Z = 0.05$ mm/rev

For conversion into per minute feed
F (mm/min) = Speed × F (mm/rev)
$F_X = 2500 \times 0.03 =$ F75
$F_Z = 2500 \times 0.05 =$ F125

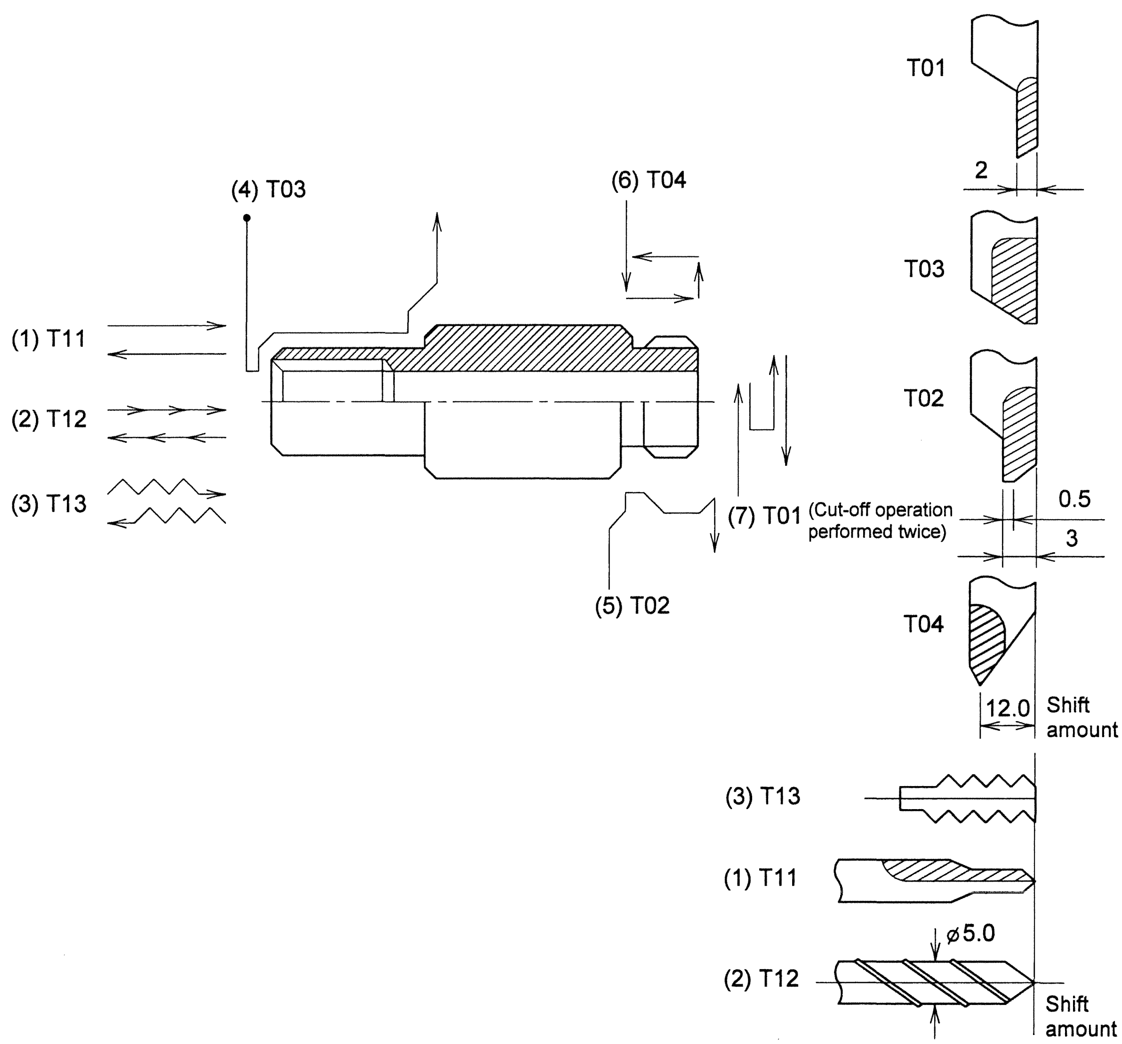
9.2 Programming Practice 2

Machining drawing

Cut material: SUM24L (free cutting steel) (Diameter 12 mm)



Tool layout



Sample program

O0002

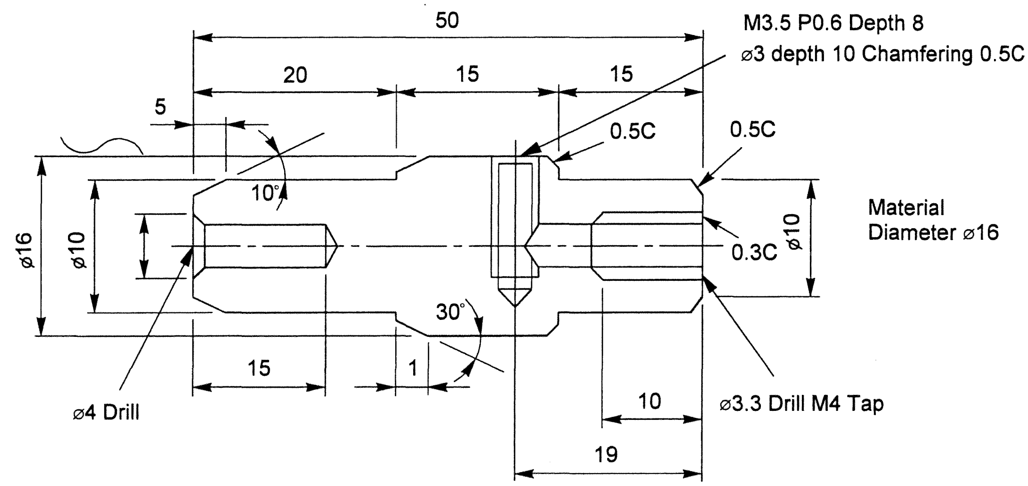
$1		$2	
G50 Z–0.1			
M06			
G00 X13.0 Z–0.6			
M03 S1=1800 G99			
G630	Front/back parallel machining	G630	Front/back parallel machining
N121 T1100			
G00 Z–0.6		M34	Product collection
G01 Z6.0 F0.08 T01			
G00 Z–0.6 T00			
N222 T1200			
G01 Z16.0 F0.1 T02	Starts drill in-feed operation		
G00 Z–0.6			
G04 U0.5			
Z15.5			
G01 Z27.0 F0.09			
G00 Z–0.6			
G04 U0.5			
Z26.5			
G01 Z34.0 F0.08			
G00 Z–0.6			
G04 U0.5			
Z33.5			
G01 Z40.0 F0.07	Last drill in-feed operation		
G00 Z–0.6 T00			
N323 T1300			
G25			
S1=400			
G04 U1.0			
T03			
G32 Z8.0 F0.8	Forward in-feed operation for M6 thread cutting		
Z–3.0 F1.0 M04	Reversing		
T00			
G04 U2.0			
G00 Z–0.6			
M03 S1=2800			

$1	$2
N402 T0300	
G00 X5.0 Z-0.6 T04	
G01 Z0 F0.06	M16
X8.4	M24 S2=2800
X9.0 Z0.3	
Z13.0	
X11.6	
X13.0 Z13.7	
N503 T0200	
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	
S1=1000	
G4 U1.0	
N604 T0400	
G00 X13.0 Z44.0 T06	
G92 X10.56 Z51.0 F1.0	
X10.26	
X10.04	
X9.86	
X9.76	
X9.7	
X9.7	
G00 X13.0	
S1=2800	
N0401 T0100	
G00 X13.0 Z40.0 T07	

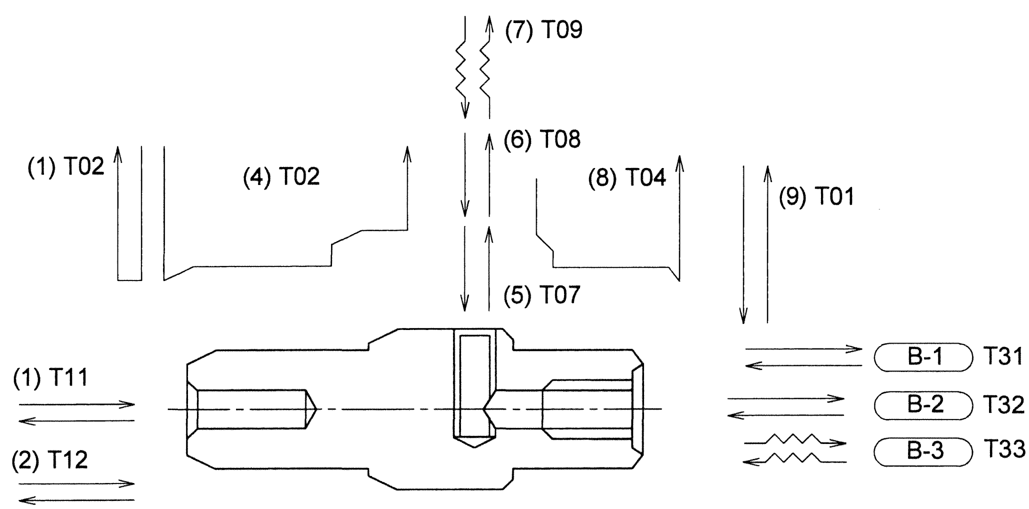
$1	$2
G650	G650
	M123
	M72
	G00 Z–1.0
	G98 G01 Z12.0 F1000
	G04 U1.0
	M15
	M73
!L1	!L1
G01 X5.0 F0.03	
G600	G600
G00 X13.0 W–0.5	M124
W2.5	M25
G01 X–3.0 F0.05	
M05	
M07	
G00 Z–0.1 T00	
M56	
G999	G999
	M34
N999	N999
M02	M02
%	%

9.3 Programming Practice 3 (Back Machining)

Machining drawing



Tool layout

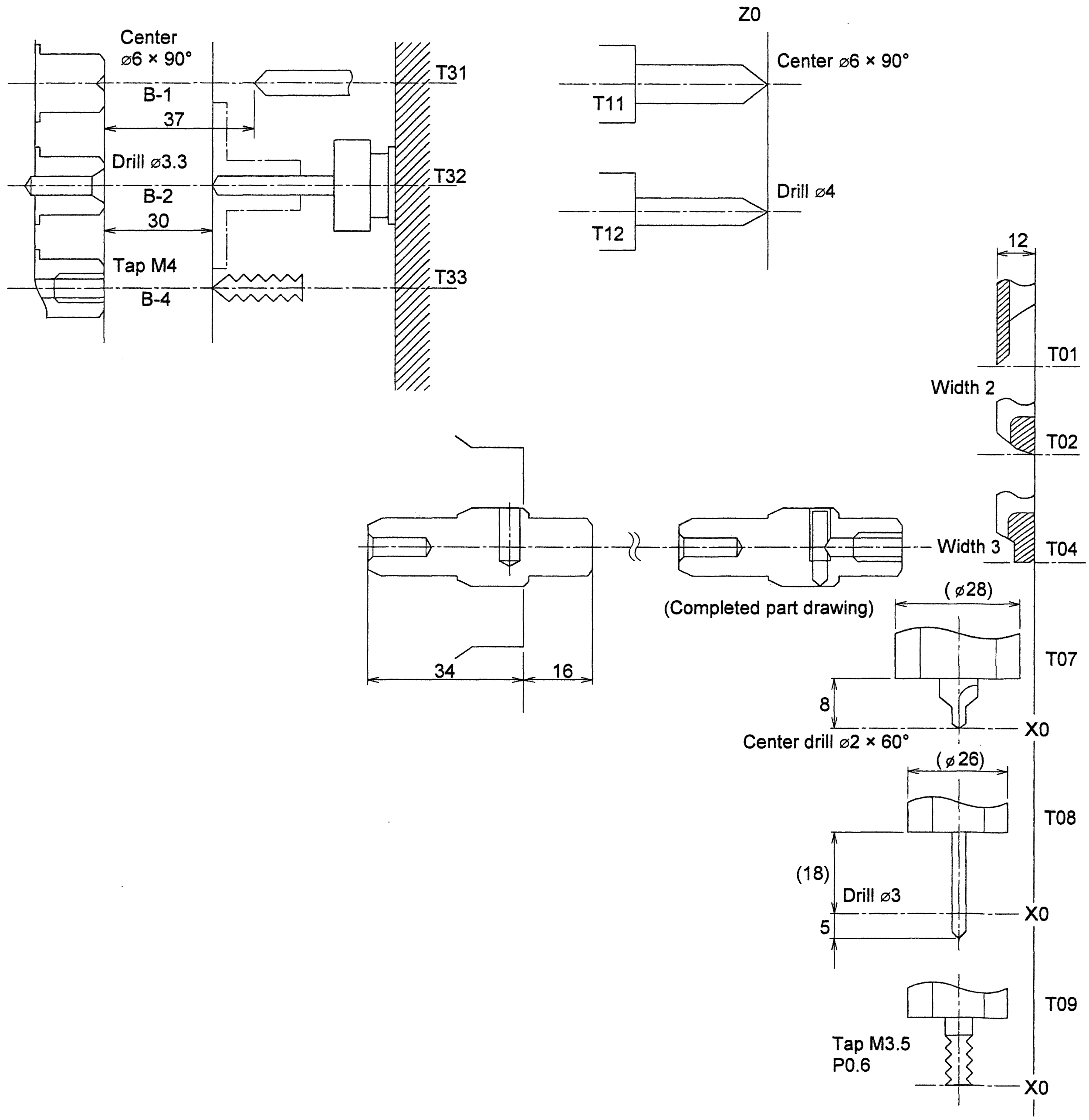


See <Cutting tool drawings> described later.

Setting on Machining data screen

MATERIAL O.D.	16.000 mm
MACH. LENGTH	70.000 mm
PIECES/1CHK	1
POSITIONING PT.	11.0 mm
B.CHUCK EXT.LENG.	16.0 mm

Cutting tool drawings



Sample program

O0003

$1		$2	
G50 Z9.9			
G99			
M6			
G0 X17.0 Z–0.5			
M3 S1=2400			
G630	Front/back parallel machining	G630	Front/back parallel machining
		M98 P0004	Calls subprogram O0004.
N1 T0200		G99 M24 S2=1600	
G0 X17.0 Z0 T02			
G1 X–0.5 F0.1			
Z–0.5			
G0 X17.0 S1=1061			
N2 T1100			
G0 Z–1.0			
G1 Z2.7 F0.08 T11			
G0 Z–1.0 T0 S1=1600			
N3 T1200			
G1 Z12.0 F0.06 T12			
G0 Z–1.0			
G4 U0.5			
Z11.5			
G1 Z16.15 F0.06			
G0 Z–1.0			
S1=2400			
N4 T0200			
G0 X8.06 Z–0.5 T22			
G1 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			

$1	$2
G98	
M80 S3=1300	
N5 T0700	
M28 S0	
G50 W–15.0	
G0 X17.0 Z31.0 T07	
G1 X4.8 F50 .... X4.8 =⌀16 – 5.6 × 2 = 4.8	
G0 X27.0	
M80 S3=1600	
N6 T1200	
G0 X17.0 Z31.0 T12	
G1 X3.7 F80 .... X3.7 =⌀26 – 11.15 × 2 = 3.7	
‖	
⌀16 + (shift amount 5 mm × 2)	
G0 X17.0	
N7 T0800	
G00 X21.6 Z31.0 T08 .... X21.6 =⌀16 + {(1 + 1.8) × 2}	
= 21.6	
‖	
Pitch × 3	
G88 X0 R1.0 F0.6 D3 S500	
G80	
G0 X22.0	
G50 W15.0	
M82	
G99	
M20	
M3 S1=2000	
N8 T0400	
G50 W–3.0	
G0 X17.0 Z34.0 T04	
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	
N9 T0100	
G0 X17.0 Z63.0 T01	
G650	G650
	M72
	G0 Z–1.0

$1	$2
	G98 G1 Z34.0 F1000
	G4 U0.5
	M15
	M73
!2 L2	!1 L2
G1 X–1.0 F0.02	
G600	G600
X–3.0 F0.05	M25
M5	
M56	
G999	G999
	M98 P0004
N999	N999
M7	
G0 Z9.9 T0	
M2	M2

O0004 (Subprogram)

$1	$2
	G44 G99 M23 S2=1061 T3100 G0 Z–1.0 T31 G1 Z2.8 F0.05 G0 Z–1.0 S2=1900 T3200 G0 Z–1.0 T32 .... Distance between the end faces of the workpiece chucked by the back spindle and cutting tool G1 Z10.0 F0.05 Z–1.0 F1.0 G4 U0.5 Z9.5 Z13.3 F0.05 Z–1.0 F1.0 G4 U0.5 Z12.8 Z16.6 F0.05 Z–1.0 F1.0 G4 U0.5 Z16.1 Z19.6 F0.05 Z–1.0 F1.0 G4 U0.5 Z19.1 Z21.0 F0.05 Z–1.0 F1.0 S2=500 G4 U1.0 T3300 G0 Z–1.5 G84 Z13.0 F0.7 D2 S500 G80 G0 Z–1.5 G0 Z–5.0 M25 M34 .... Sequential operation for product collection M99 %

Product Code	C	–	A	2	2	0	P	L	VII					
Document Code	2	E	1	–	0	9	0	1						

10. Cutting Conditions

The cutting conditions presented in this chapter provide a guideline. Actually, choose the best cutting condition considering the workpiece material, cutting tool form, and cutting tool material.

The cutting speeds are determined on the condition of using the rotary guide bushing. The cutting speeds when using the fixed guide bushing are required to setup 50 to 70 % per preceding values.

(Blank page)

10.1 Cutting Speeds and Feedrates

Cutting speeds are generally determined as those on the periphery of the workpiece. However, if the finish diameter is greatly different from the workpiece diameter (much in-feed is required), the cutting speed on the workpiece periphery should be slightly higher than the typical value.

Outer diameter cutting speed (carbide tools)

Table 1

m/min

Material	Cutting speed	Material	Cutting speed
Brass	200 to 350	Alloy tool steel	60 to 150
Phosphor bronze	150 to 250	Free-cutting stainless steel (SUS303F equivalent or less)	100 to 180
Aluminum	200 to 400		
Free-cutting steel	150 to 250	Stainless steel (SUS304F equivalent or less)	60 to 150
Carbon tool steel	120 to 200		
Structural carbon steel	80 to 180	Hard cutting stainless steel (SUS304F equivalent or more)	30 to 80

Outer diameter cutting feedrate (carbide tools)

Table 2

mm/rev

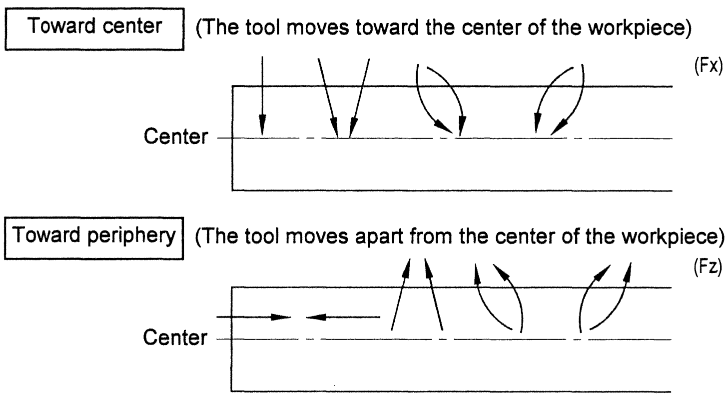
	Diametrical feed (X)			Longitudinal feed (Z)		
	A	B	C	A	B	C
Brass	0.03	0.05	0.08	0.03	0.08	0.15
Phosphor bronze	0.015	0.03	0.06	0.03	0.08	0.15
Aluminum	0.03	0.05	0.08	0.03	0.08	0.15
Free-cutting steel	0.015	0.03	0.06	0.03	0.08	0.15
Carbon tool steel	0.015	0.03	0.06	0.03	0.05	0.1
Structural carbon steel	0.015	0.025	0.05	0.03	0.05	0.1
Alloy tool steel	0.01	0.02	0.03	0.03	0.05	0.1
Free-cutting stainless steel (SUS303F equivalent or less)	0.015	0.025	0.05	0.03	0.05	0.1
Stainless steel (SUS304F equivalent or less)	0.01	0.02	0.03	0.03	0.05	0.1
Hard cutting stainless steel (SUS304F equivalent or more)	0.008	0.015	0.02	0.02	0.04	0.08

The feedrate varies according to a degree of in-feed, required surface roughness, accuracy, and tool life as well as the workpiece material.

	Tolerance of dimension	Surface roughness	In-feed (radius)
A	within ± 5 μ	within 3S	4 mm or more
B	within ± 10 μ	within 6S	2 to 4 mm
C	± 10 μ or more	6S or more	2 mm or less

Note

Fx (toward center) and Fz (toward periphery) mean the following:



Inner diameter cutting speed (carbide tools)

Table 3

m/min

Material	Cutting speed	Material	Cutting speed
Brass	100 to 170	Alloy tool steel	30 to 80
Phosphor bronze	80 to 120	Free-cutting stainless steel (SUS303F equivalent or less)	50 to 90
Aluminum	100 to 200		
Free-cutting steel	80 to 120	Stainless steel (SUS304F equivalent or less)	30 to 80
Carbon tool steel	60 to 100		
Structural carbon steel	40 to 90	Hard cutting stainless steel (SUS304F equivalent or more)	15 to 40

Inner diameter cutting feedrate (carbide tools)

Cutting conditions for boring largely depend on the rigidity of the tool and tool holder. Diametrical direction feed rate should be about half of the longitudinal direction feed rate.

Table 4

mm/rev

Material	Longitudinal feed (Z)		
	A	B	C
Brass	0.015	0.04	0.1
Phosphor bronze	0.015	0.04	0.1
Aluminum	0.015	0.04	0.1
Free-cutting steel	0.015	0.04	0.1
Carbon tool steel	0.015	0.03	0.05
Structural carbon tool steel	0.015	0.03	0.05
Alloy tool steel	0.015	0.03	0.05
Free-cutting stainless steel (SUS303F equivalent or less)	0.015	0.03	0.05
Stainless steel (SUS304F equivalent or less)	0.015	0.025	0.04
Hard cutting stainless steel (SUS304F equivalent or more)	0.01	0.02	0.03

Reaming

Table 5

Material	Cutting speed (m/min)	Feedrate (mm/rev)		
		⌀1 to ⌀2	⌀2 to ⌀5	⌀5 to
Brass	10 to 20	0.07 to 0.25	0.15 to 0.5	0.25 to 1.0
Phosphor bronze	10 to 20	0.03 to 0.15	0.1 to 0.3	0.3 to 0.5
Aluminum	20 to 30	0.08 to 0.25	0.15 to 0.5	0.25 to 1.0
Free-cutting steel	10 to 20	0.06 to 0.25	0.15 to 0.5	0.25 to 1.0
Carbon tool steel	10 to 20	0.05 to 0.15	0.1 to 0.3	0.1 to 0.5
Structural carbon tool steel	10 to 20	0.03 to 0.15	0.05 to 0.3	0.1 to 0.5
Alloy tool steel	6 to 12	0.03 to 0.1	0.05 to 0.2	0.1 to 0.3
Free-cutting stainless steel (SUS303F equivalent or less)	10 to 20	0.03 to 0.12	0.05 to 0.2	0.1 to 0.4
Stainless steel (SUS304F equivalent or less)	10 to 20	0.02 to 0.1	0.05 to 0.2	0.1 to 0.3
Hard cutting stainless steel (SUS304F equivalent or more)	6 to 12	0.02 to 0.08	0.04 to 0.15	0.08 to 0.25

* The feedrate of the material varies considerably depending on the tools to be used.

Hole machining (high-speed tool steel)

Table 6

m/min

Material	⌀1 to ⌀2 mm	⌀2 to ⌀5 mm	⌀5 to 12 mm
Brass	40 to 60	60 to 80	80 to 100
Aluminum	40 to 60	60 to 80	80 to 100
Free-cutting steel	30 to 40	40 to 60	60 to 80
Carbon tool steel	20 to 30	30 to 50	50 to 60
Structural carbon tool steel	15 to 20	20 to 30	30 to 35
Alloy tool steel	8 to 12	12 to 15	15 to 20
Free-cutting stainless steel (SUS303F equivalent or less)	20 to 30	30 to 40	40 to 50
Stainless steel (SUS304F equivalent or less)	to 10	10 to 15	15 to 20
Hard cutting stainless steel (SUS304F equivalent or more)	to 8	8 to 10	10 to 15

Hole machining speed feedrate (high-speed tool steel)

Table 7

m/rev

Material	⌀1 to ⌀2 mm	⌀2 to ⌀5 mm	⌀5 to 12 mm
Brass	0.03 to 0.06	0.06 to 0.12	0.12 to 0.25
Aluminum	0.03 to 0.06	0.06 to 0.12	0.12 to 0.25
Free-cutting steel	0.025 to 0.05	0.05 to 0.10	0.1 to 0.2
Carbon tool steel	0.02 to 0.04	0.04 to 0.08	0.08 to 0.16
Structural carbon tool steel	0.04 to 0.05	0.04 to 0.08	0.08 to 0.16
Alloy tool steel	0.015 to 0.03	0.03 to 0.06	0.06 to 0.12
Free-cutting stainless steel (SUS303F equivalent or less)	0.02 to 0.04	0.04 to 0.08	0.08 to 0.16
Stainless steel (SUS304F equivalent or less)	0.015 to 0.03	0.03 to 0.06	0.06 to 0.12
Hard cutting stainless steel (SUS304F equivalent or more)	0.01 to 0.02	0.02 to 0.04	0.04 to 0.08

10.2 Thread Cutting Count with Tool

Nonferrous material

Table 8

mm

P		0.5	0.7	0.8	1.0	1.25	1.5	1.75	2.0	2.5	3.0
Thread height		0.32	0.45	0.52	0.65	0.81	0.97	1.13	1.3	1.62	1.95
Count	1st in-feed	0.22	0.22	0.25	0.25	0.32	0.35	0.35	0.35	0.40	0.40
	2nd in-feed	0.07	0.18	0.20	0.22	0.22	0.25	0.23	0.27	0.35	0.36
	3rd in-feed	0.03	0.05	0.07	0.13	0.14	0.16	0.20	0.22	0.25	0.30
	4th in-feed				0.05	0.08	0.10	0.15	0.15	0.20	0.25
	5th in-feed					0.05	0.06	0.09	0.12	0.15	0.20
	6th in-feed						0.05	0.06	0.08	0.12	0.15
	7th in-feed							0.05	0.06	0.07	0.12
	8th in-feed								0.05	0.05	0.07
	9th in-feed									0.03	0.05
	10th in-feed										0.03
	11th in-feed										0.02

Ferrous material

Table 9

mm

P		0.5	0.7	0.8	1.0	1.25	1.5	1.75	2.0	2.5	3.0
Thread height		0.32	0.45	0.52	0.65	0.81	0.97	1.13	1.3	1.62	1.95
Count	1st in-feed	0.18	0.22	0.22	0.22	0.22	0.25	0.28	0.30	0.32	0.35
	2nd in-feed	0.08	0.13	0.14	0.15	0.16	0.18	0.18	0.21	0.22	0.23
	3rd in-feed	0.04	0.07	0.10	0.11	0.12	0.14	0.12	0.16	0.18	0.19
	4th in-feed	0.02	0.03	0.04	0.09	0.08	0.09	0.09	0.11	0.13	0.16
	5th in-feed			0.02	0.05	0.08	0.08	0.08	0.09	0.12	0.15
	6th in-feed				0.03	0.07	0.07	0.07	0.08	0.11	0.13
	7th in-feed					0.05	0.06	0.07	0.08	0.10	0.11
	8th in-feed					0.03	0.05	0.06	0.07	0.08	0.11
	9th in-feed						0.03	0.06	0.05	0.08	0.09
	10th in-feed						0.02	0.05	0.05	0.07	0.08
	11th in-feed							0.04	0.05	0.06	0.07
	12th in-feed							0.03	0.03	0.06	0.06
	13th in-feed								0.02	0.04	0.06
	14th in-feed									0.03	0.05
	15th in-feed									0.02	0.05
	16th in-feed										0.04
	17th in-feed										0.02

10.3 Cutting Condition Tables for the Secondary Machining Process

End milling (carbide tools)

Table 10

Material	Diameter mm	Cutting speed m/min	In-feed amount mm	Feed rate* mm/cutting edge
Non-ferrous material Brass Aluminum, etc.	ø2	30 to 100	0.4	0.02 to 0.03
	ø3	30 to 100	0.6	0.03 to 0.04
	ø4	30 to 100	0.8	0.04 to 0.05
	ø5, ø6	30 to 100	1 to 1.2	0.05 to 0.08
Ferrous free-cutting material (e.g., free cutting steel)	ø2	30 to 80	0.4	0.015
	ø3	30 to 80	0.6	0.02
	ø4	30 to 80	0.8	0.03
	ø5, ø6	30 to 80	1 to 1.2	0.03 to 0.06
Ferrous material Carbon tool steel Structural carbon steel, etc.	ø2	20 to 60	0.4	0.015 to 0.02
	ø3	20 to 60	0.6	0.015 to 0.02
	ø4	20 to 60	0.8	0.025 to 0.03
	ø5, ø6	20 to 60	1 to 1.2	0.04 to 0.06
Alloy tool steel Stainless steel, etc.	ø2	15 to 50	0.4	0.008
	ø3	15 to 50	0.6	0.015
	ø4	15 to 50	0.8	0.025
	ø5, ø6	15 to 50	1 to 1.2	0.03 to 0.04

* The cutting conditions for end milling on the table above is a criterion as the 0.2-dimensional in-feed. If the in-feed is greater than the standard 0.2-dimensional in-feed, adjustment is required.

Slitting cutter (high-speed tool steel: 90 cutting edges)

Table 11

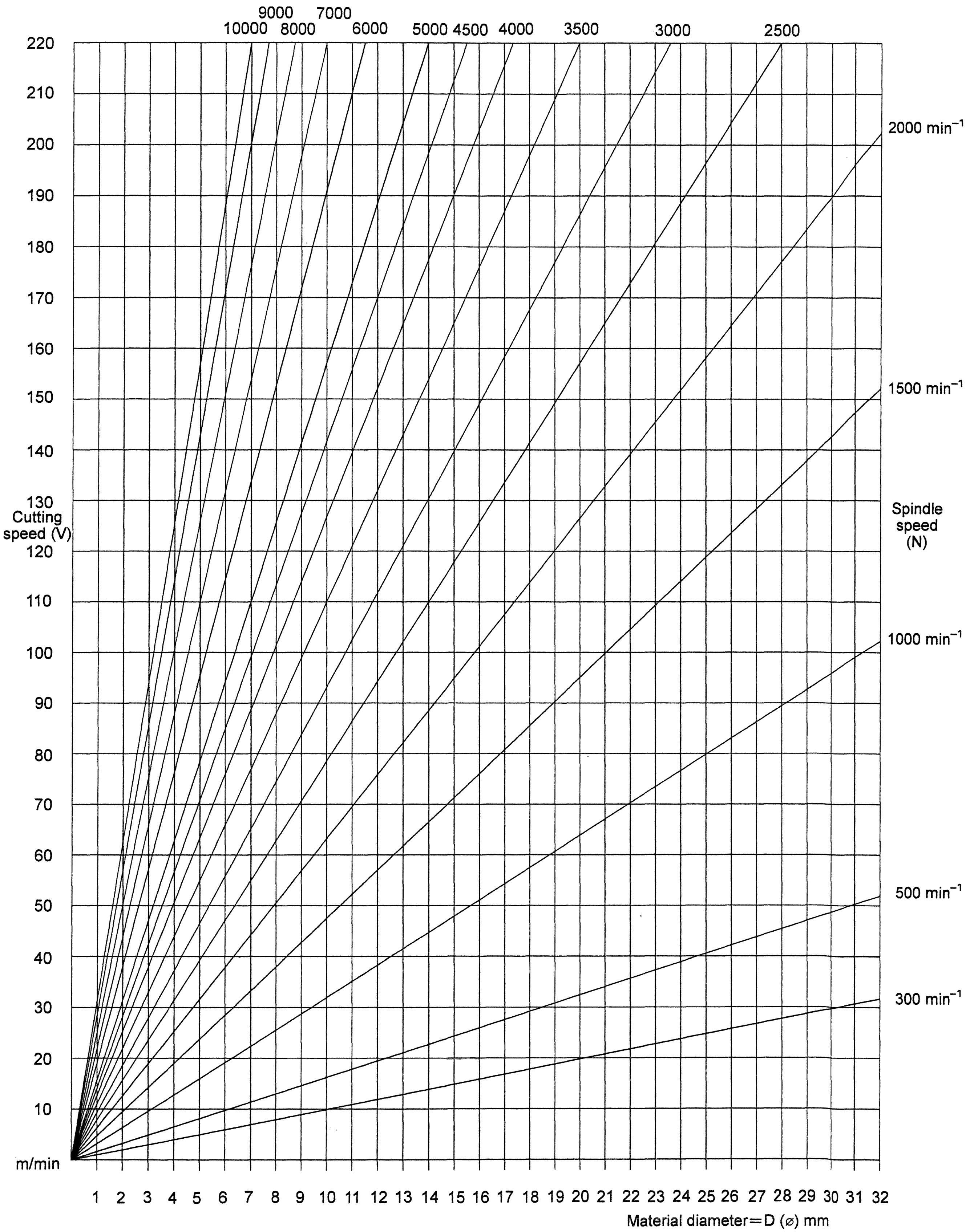
Values in parentheses: mm/rev for a slitting cutter having 90 cutting edges

Material	Width mm	Cutting speed m/min	In-feed amount mm	* Feed rate µm/cutting edge
Non-ferrous material Brass Aluminum, etc.	0.5	40 to 100	to 6.0	8 (0.70)*
	1	40 to 100	to 6.0	6 (0.55)
	2	40 to 100	to 6.0	5 (0.45)
	3	40 to 100	to 6.0	4 (0.35)
Ferrous free-cutting material (e.g., free cutting steel)	0.5	40 to 80	to 4.0	3.5 (0.32)
	1	40 to 80	to 4.0	3 (0.30)
	2	40 to 80	to 4.0	3 (0.25)
	3	40 to 80	to 4.0	2 (0.20)
Ferrous material Carbon tool steel Structural carbon steel, etc.	0.5	30 to 70	to 3.0	2.5 (0.23)
	1	30 to 70	to 3.0	2 (0.18)
	2	30 to 70	to 3.0	2 (0.15)
	3	30 to 70	to 3.0	1.5 (0.13)
Alloy tool steel Stainless steel, etc.	0.5	20 to 50	to 2.5	2 (0.18)
	1	20 to 50	to 2.5	2 (0.15)
	2	20 to 50	to 2.5	1 (0.09)
	3	20 to 50	to 2.5	0.5 (0.05)

* The slitting conditions vary significantly depending on the power and the speed reduction rate of the drive motor to be used.

10.4 Quick Reference Table for Cutting Speed

Table 12

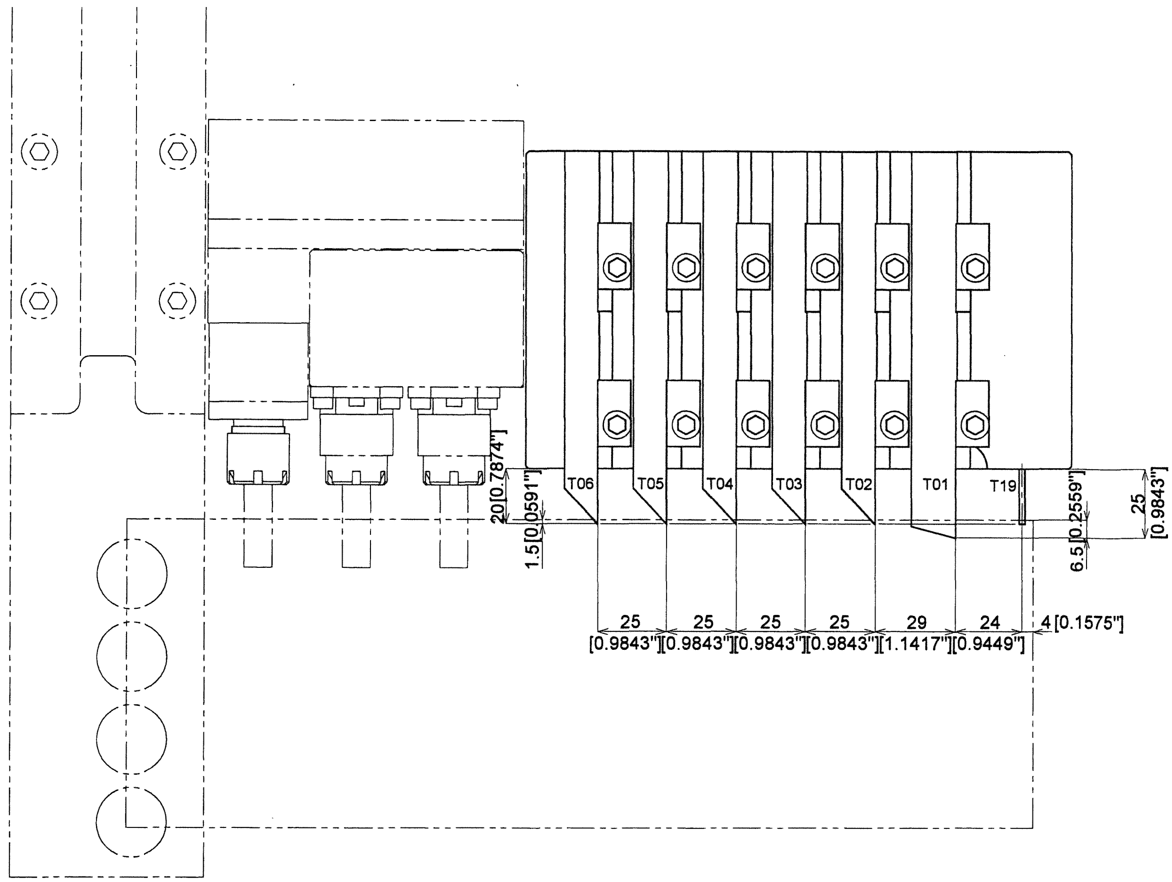


Product Code	C	–	A	2	2	0	P	L	Ⅶ						
Document Code	2	E	1	–	1	0	0	1							

11. Tooling

BTF2212 6-Tool Vertical Holder (□12)

A20L



This tool holder is provided for outer diameter cutting.

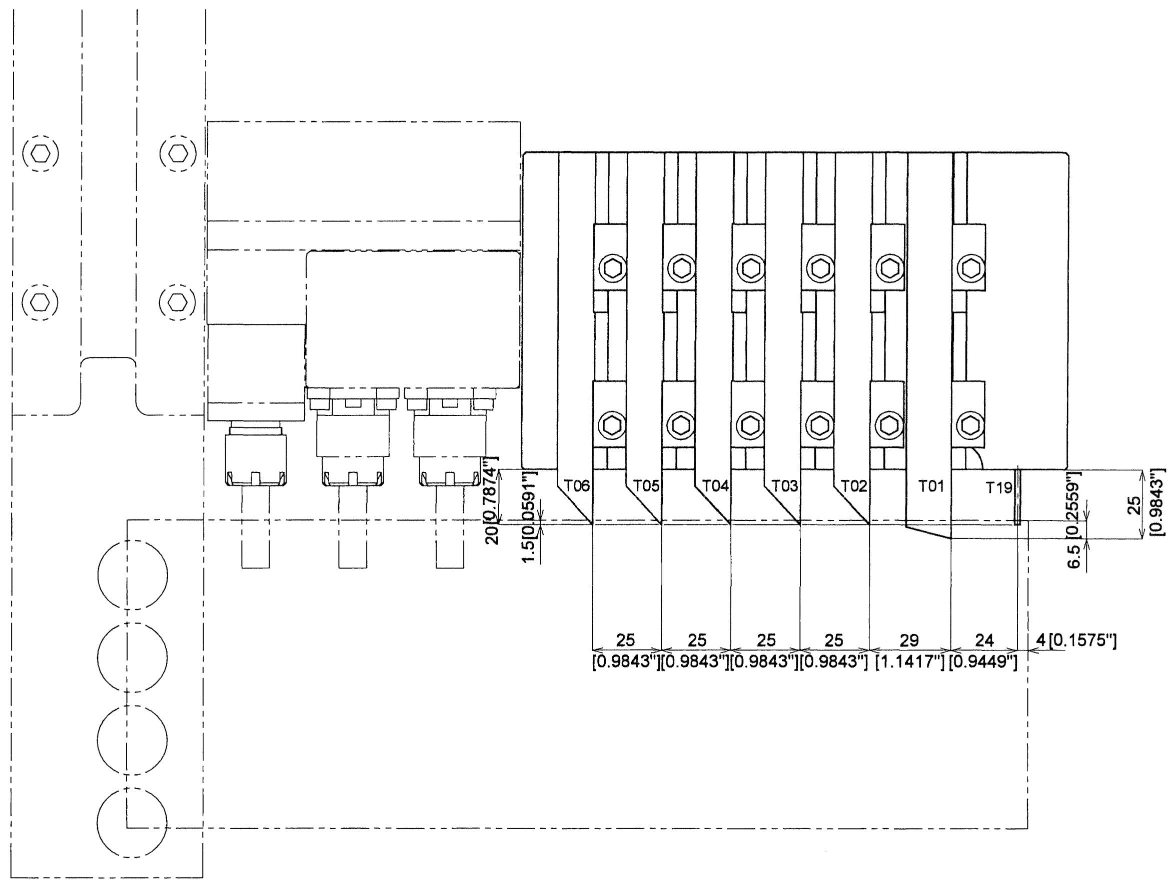
Note

Cut-off tool shall be mounted on T01. A □16 tool can be mounted on T01.

Tool holder name	BTF2212
Usage	All types of tools
Tool size	12×12×120 mm, 16×16×120 mm (T01)
Quick wedge	TJU312
Sleeve holder	BDF2005
Tool spindle drive unit	A220L U31B

BTF2213 6-Tool Vertical Holder (□13, □1/2")

A20L



This holder is provided for outer diameter cutting.

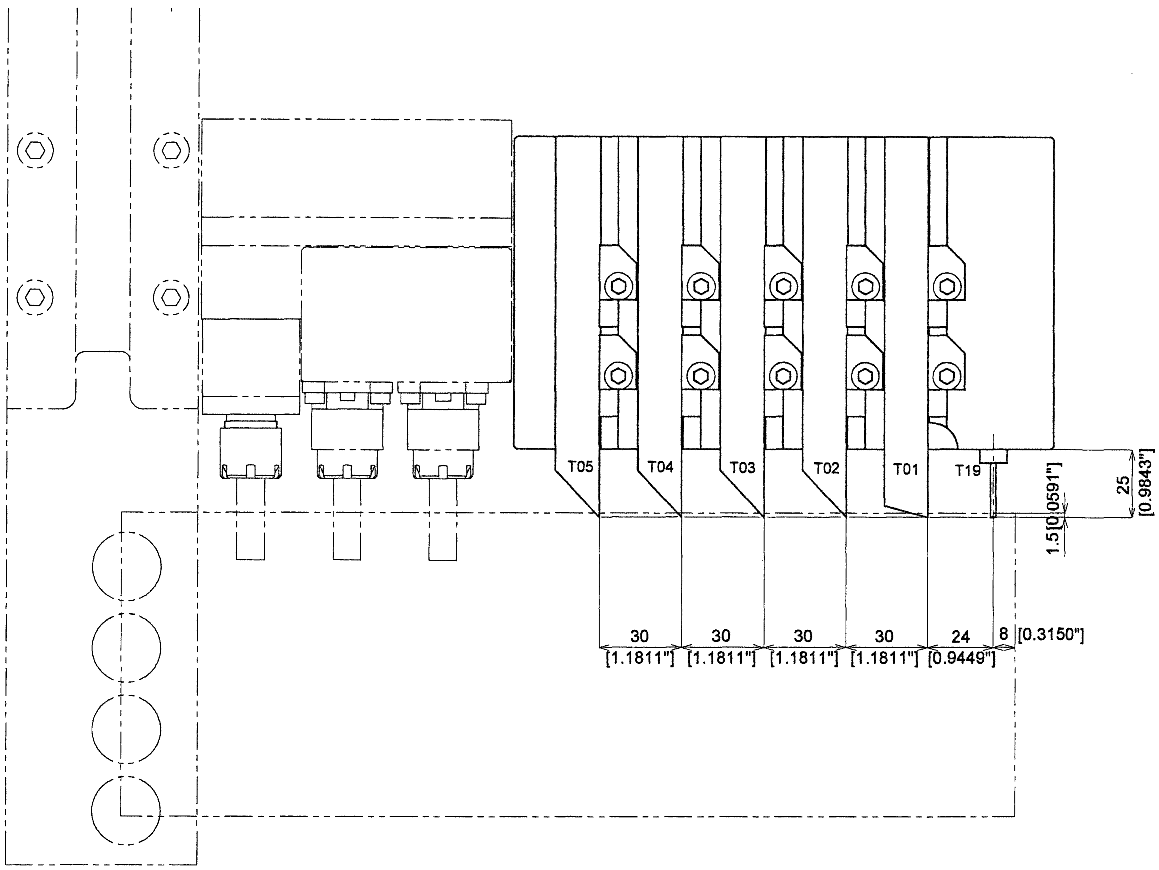
Note

Cut-off tool shall be mounted on T01. A □16 tool can be mounted on T01.

Tool holder name	BTF2213
Usage	All types of tools
Tool size	13×13×120 mm, 16×16×120 mm, 1/2"×1/2"×4"
Quick wedge	TJU312
Sleeve holder	BDF2005
Tool spindle drive unit	A220L U31B

BTF2216 5-Tool Vertical Holder (□16)

A20L



This holder is provided for outer diameter cutting.

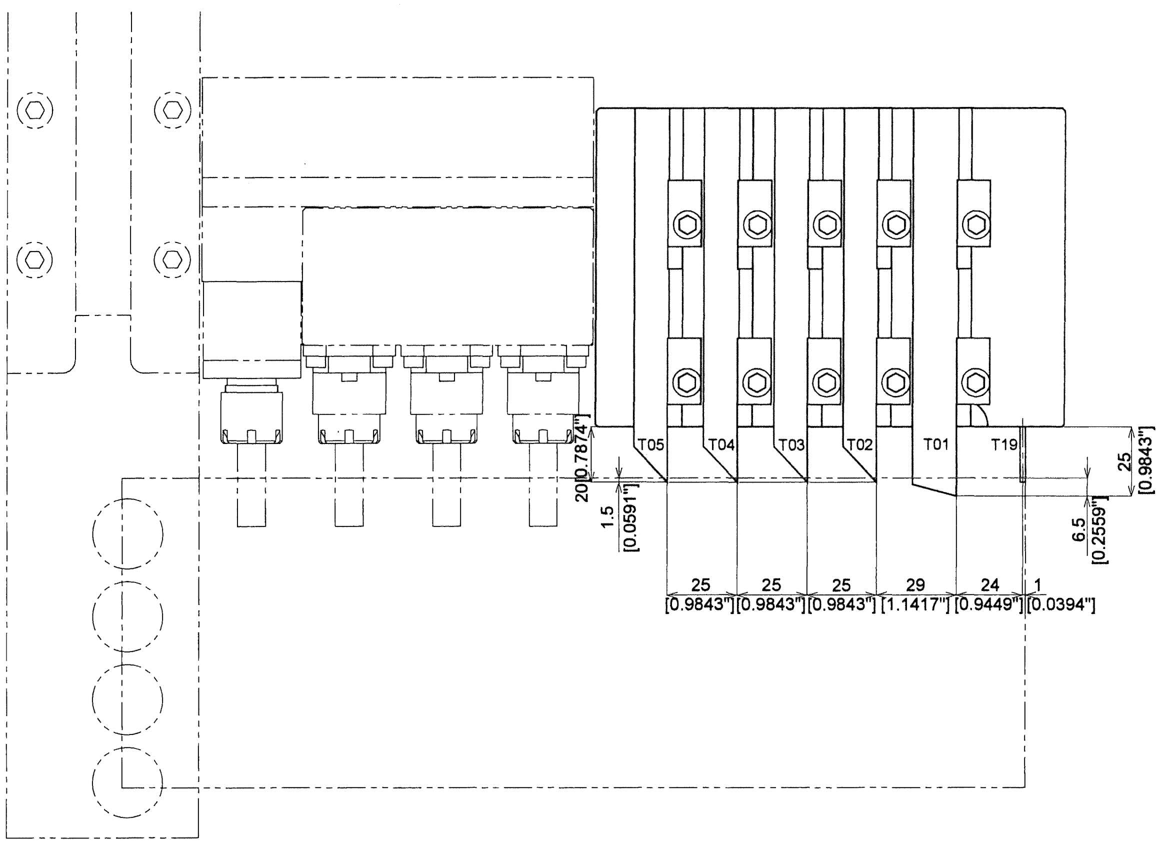
Note

Cut-off tool shall be mounted on T01.

Tool holder name	BTF2216
Usage	All types of tools
Tool size	16×16×120 mm
Quick wedge	TJU116
Sleeve holder	BDF2005
Tool spindle drive unit	A220L U31B

BTF2412 5-Tool Vertical Holder (□12)

A20L



This holder is provided for outer diameter cutting.

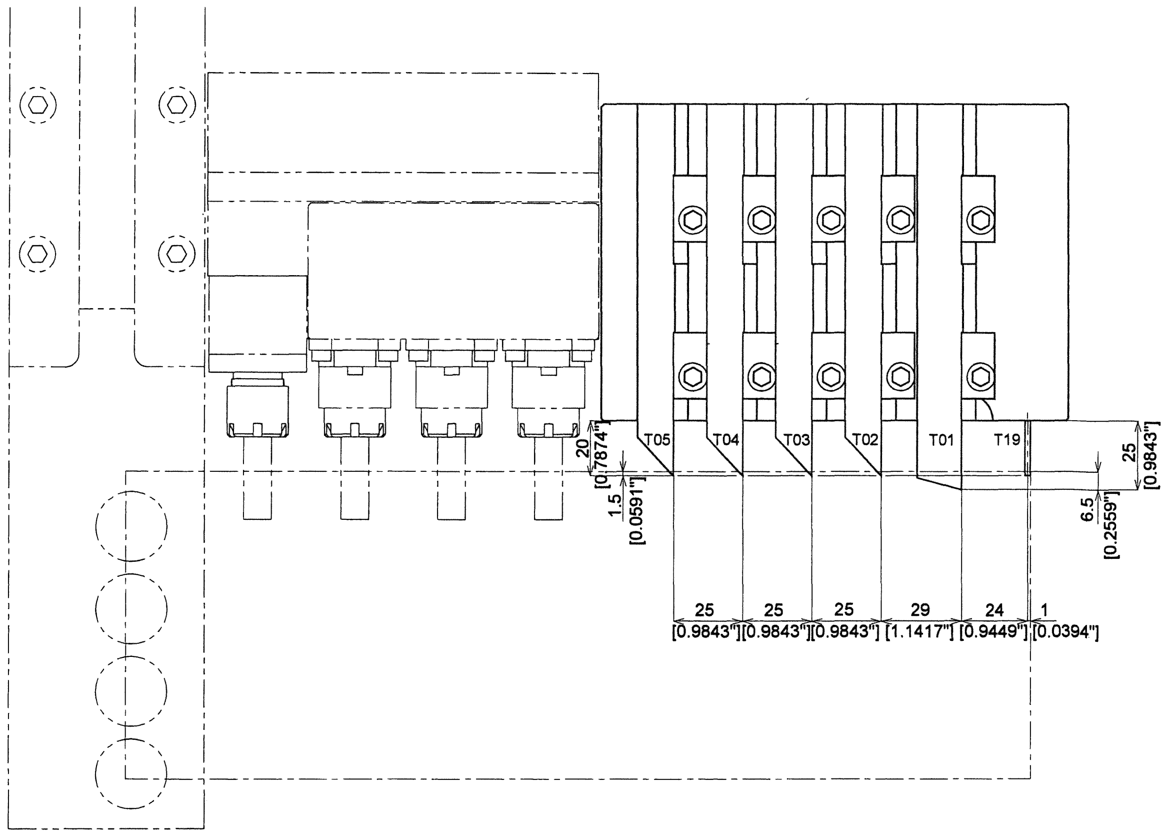
Note

Cut-off tool shall be mounted on T01. A □16 tool can be mounted on T01.

Tool holder name	BTF2412
Usage	All types of tools
Tool size	12×12×120 mm, 16×16×120 mm (T01)
Quick wedge	TJU312
Sleeve holder	BDF2005
Tool spindle drive unit	A220L U32B

BTF2413 5-Tool Vertical Holder (□13)

A20L



This holder is provided for outer diameter cutting.

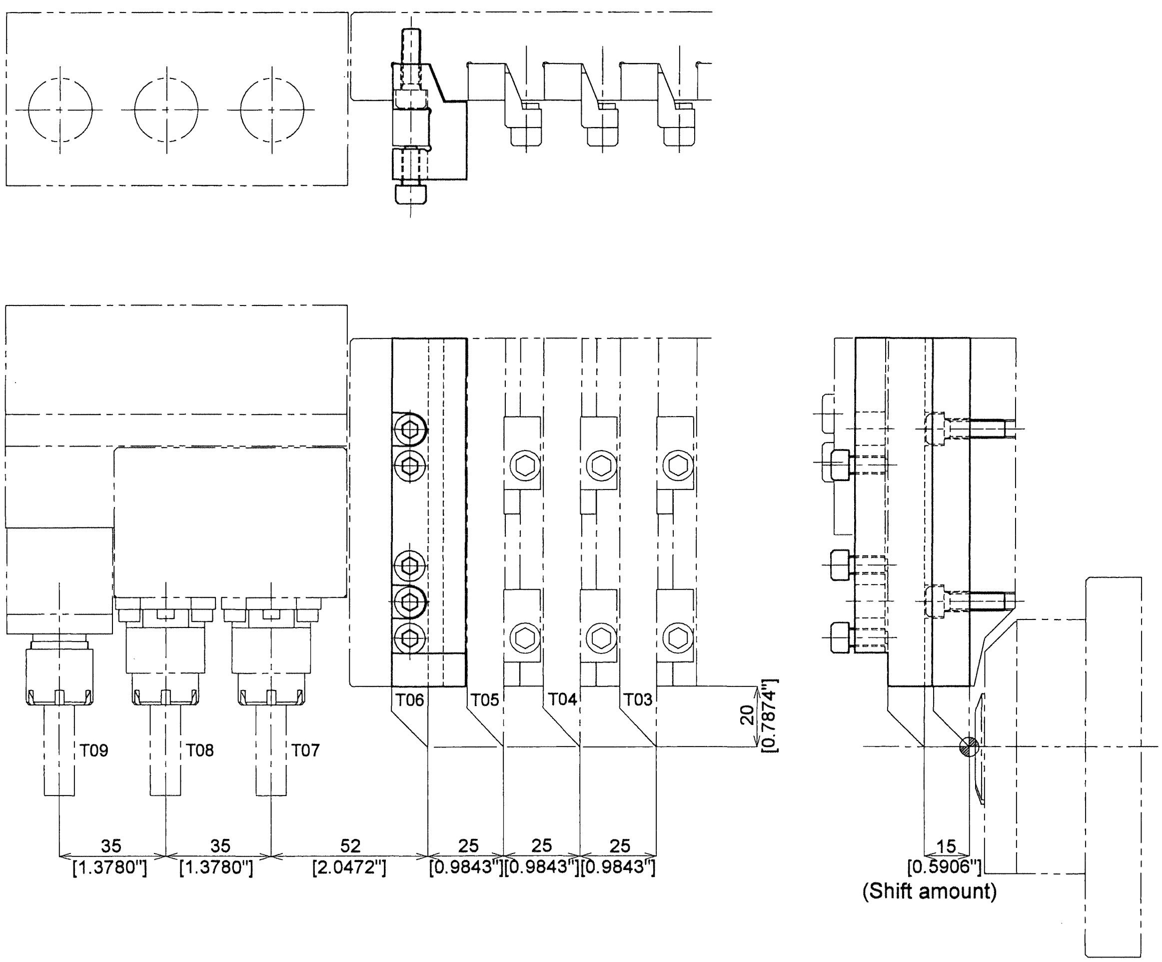
Note

Cut-off tool shall be mounted on T01. A □16 tool can be mounted on T01.

Tool holder name	BTF2413
Usage	All types of tools
Tool size	13×13×120 mm, 16×16×120 mm (T01)
Quick wedge	TJU312
Sleeve holder	BDF2005
Tool spindle drive unit	A220L U32B

GTF3312 Shift Tool Holder (Adapter Type)

A20L | A20R

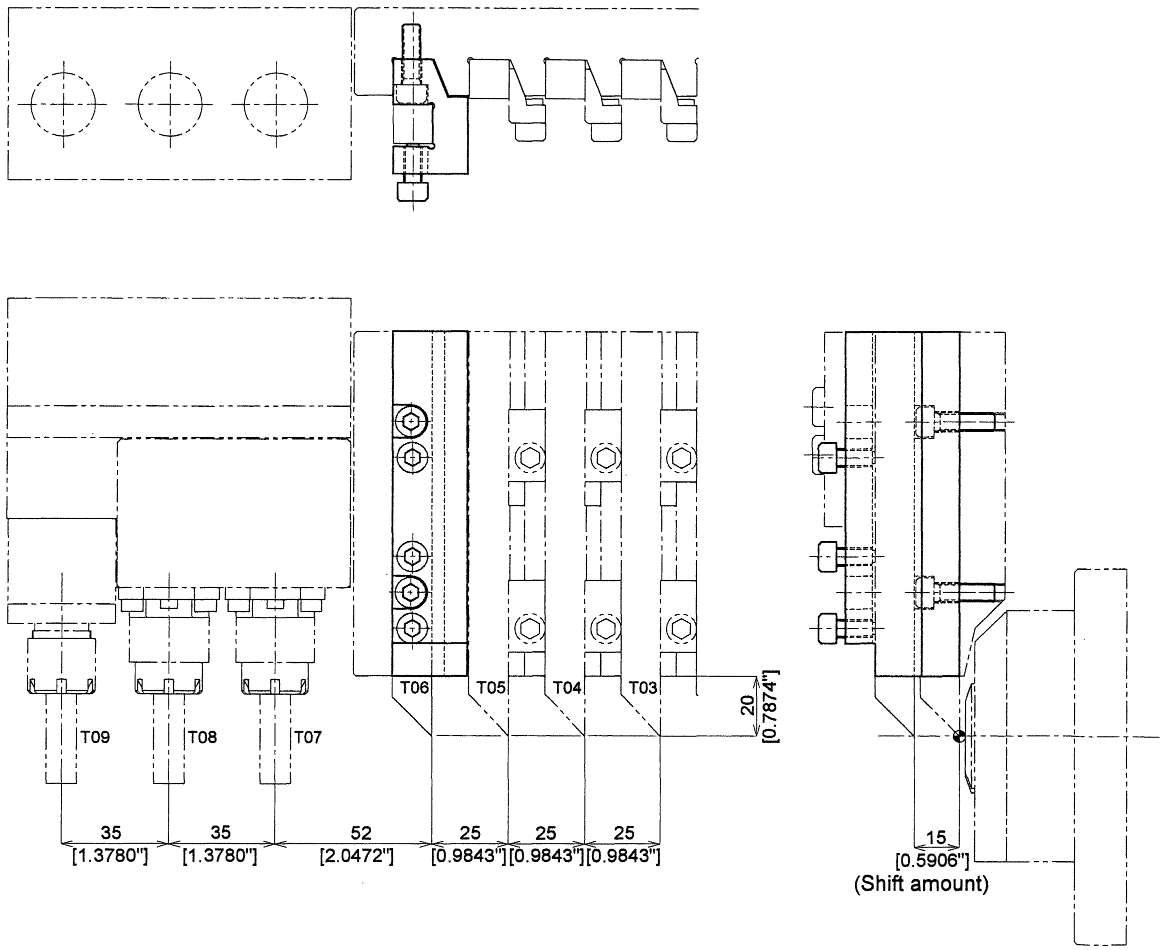


This tool holder is provided for the turning tool. The tool can be mounted with 15 mm [0.59"] longitudinal shift from standard mouting point.

Tool holder name	GTF3312
Usage	Thread cutting and others
Tool size	12×12×120 mm (Shifted by 15 mm [0.59"])

GTF3313 Shift Tool Holder (Adapter Type)

A20L A20R

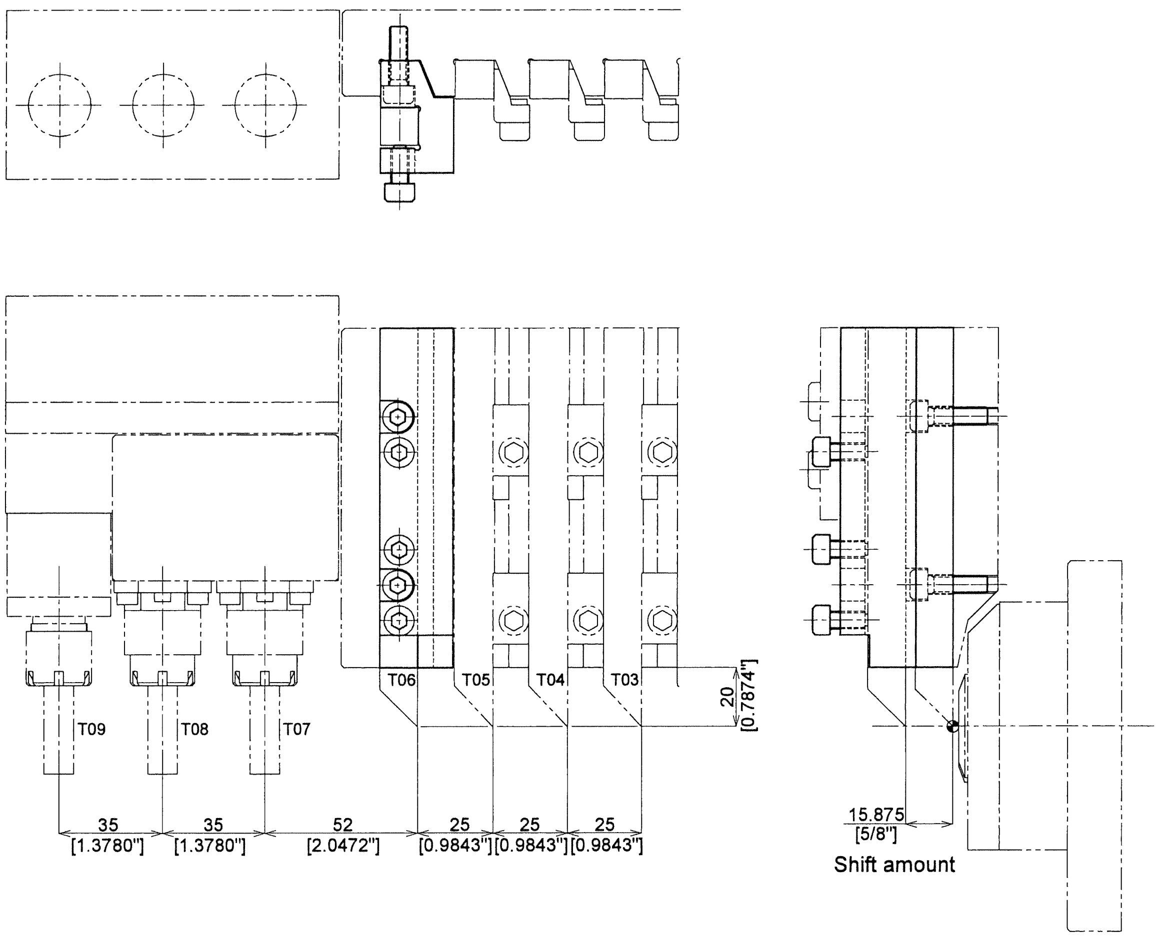


This tool holder is provided for the turning tool. The tool can be mounted with 15 mm [0.59"] longitudinal shift from standard mouting point.

Tool holder name	GTF3313
Usage	Thread cutting and others
Tool size	13×13×120 mm (Shifted by 15 mm [0.59"])

GTF3313L Shift Tool Holder (Adapter Type)

A20L A20R

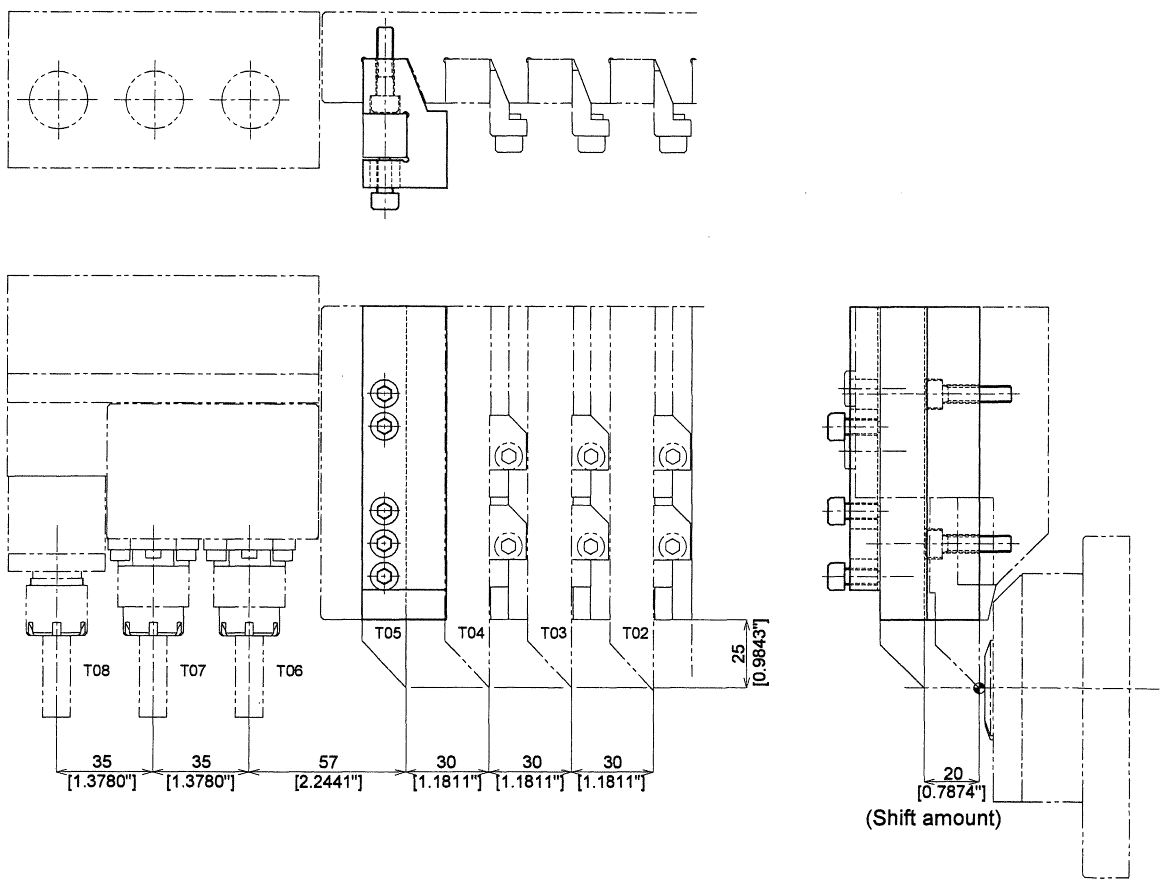


This tool holder is provided for the turning tool. The tool can be mounted with 15.875 mm [5/8"] longitudinal shift from standard mouting point.

Tool holder name	GTF3313L
Usage	Thread cutting and others
Tool size	1/2"×1/2"×4-3/4" (Shifted by 15.875 mm [5/8"])

GTF5116 Shift Tool Holder (Adapter Type)

A20L | A20R

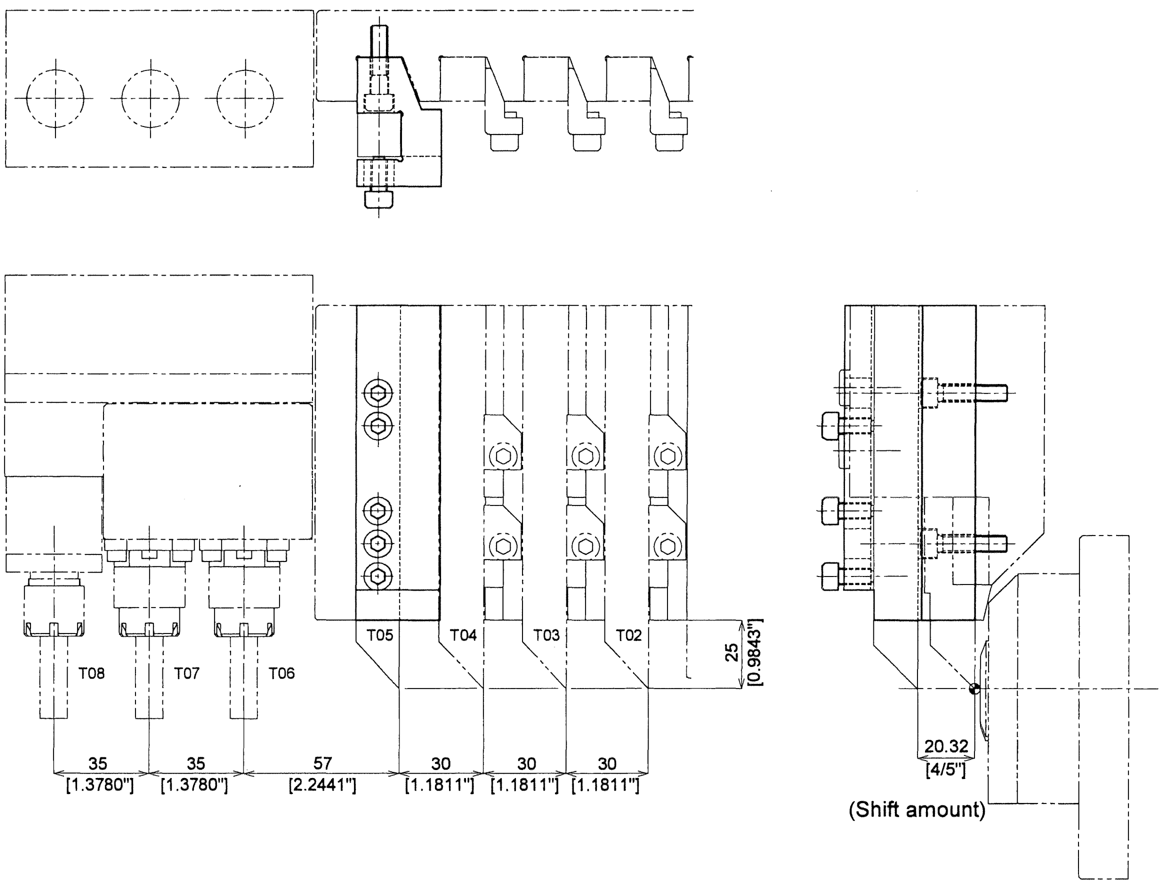


This tool holder is provided for the turning tool. The tool can be mounted with 20 mm [0.79"] longitudinal shift from standard mouting point.

Tool holder name	GTF5116
Usage	Thread cutting and others
Tool size	16×16×120 mm (Shifted by 20 mm [0.79"])

GTF5116L Shift Tool Holder (Adapter Type)

A20L A20R

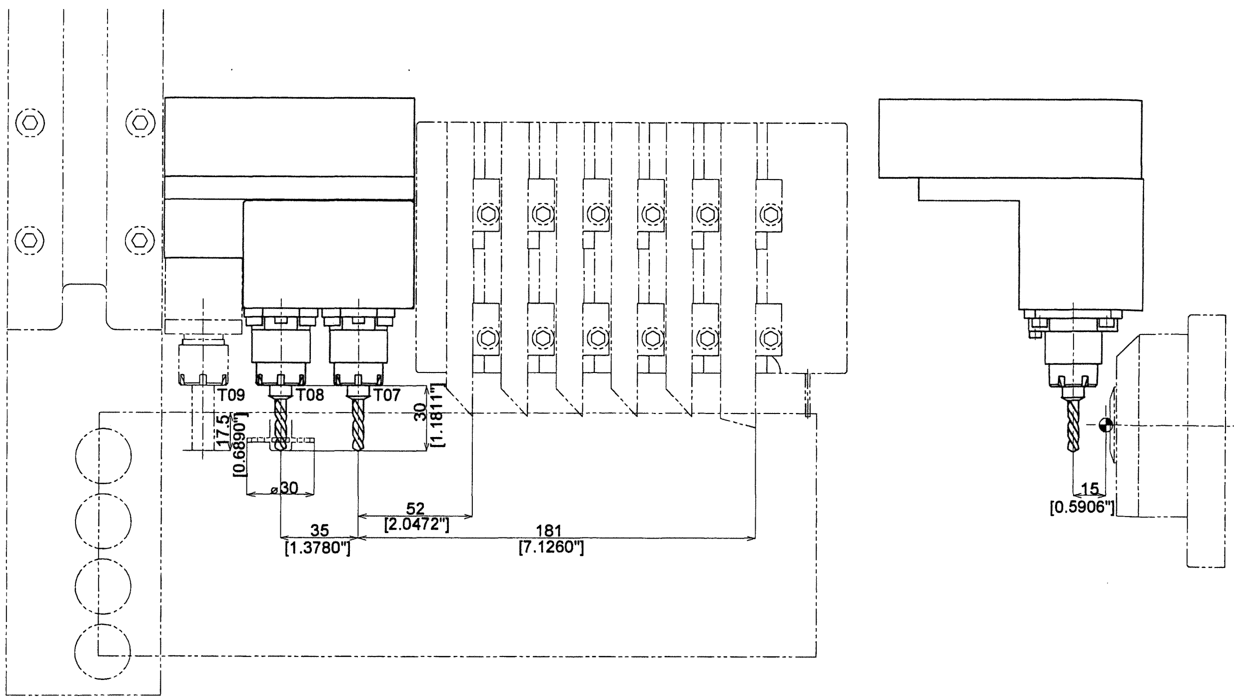


This tool holder is provided for the turning tool. The tool can be mounted with 20.32 mm [4/5"] longitudinal shift from standard mouting point.

Tool holder name	GTF5116L
Usage	Thread cutting and others
Tool size	5/8"×5/8"×4-3/4" (Shifted by 20.32 mm [4/5"])

A220L U31B 3-Rotary Tool Driving Device

A20L

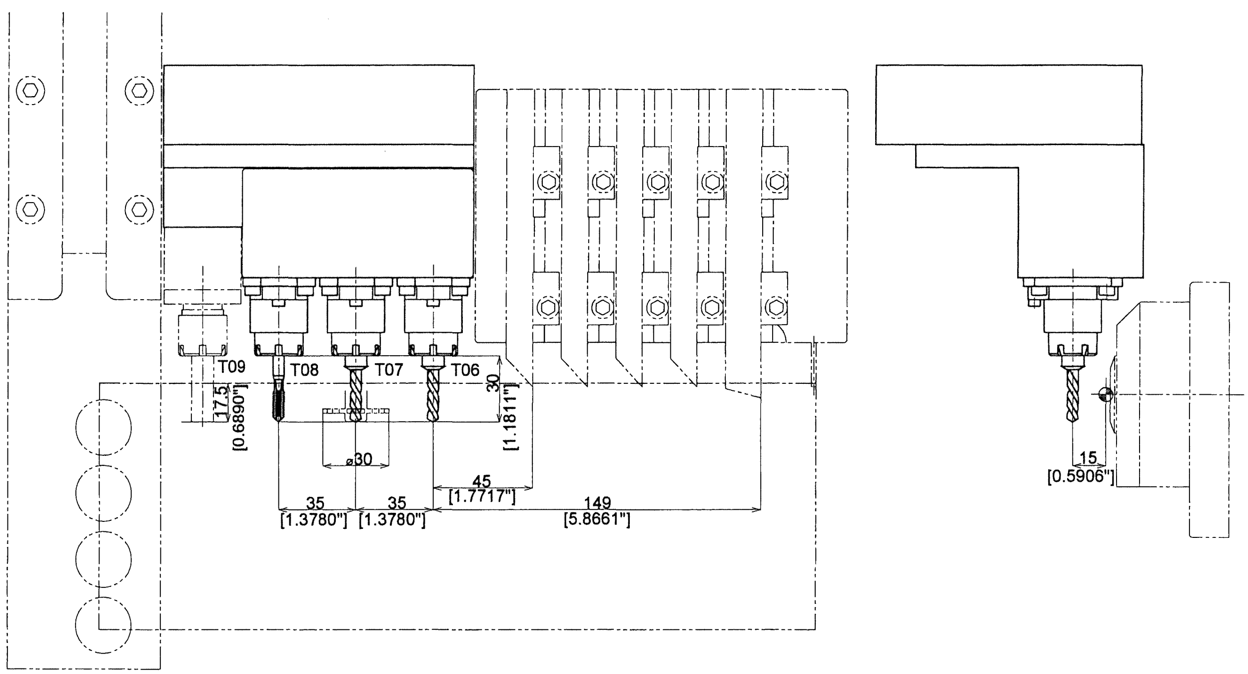


This device is used to perform drilling on the outer circumference of the workpiece, or to perform milling by the end mill, after stopping the spindle.
Two rotary tools (T08 and T07) are mounted at the position shifted by 15 mm [0.59"].
Only a tool mounted on T09 can be replaced with an end-face machining tool.

Tool holder name	U31B
Max. chuck dia.	ø10 mm [ø3/8"]
Spindle speed	Stationary speed: 100 to 6,000 min^{-1} (Rating) Max. instantaneous speed: 8,000 min^{-1}
Chuck type	ER16, AR16

A220L U32B 4-Rotary Tool Driving Device

A20L



This device is used to perform drilling on the outer circumference of the workpiece, or to perform milling by the drill, after stopping the spindle.

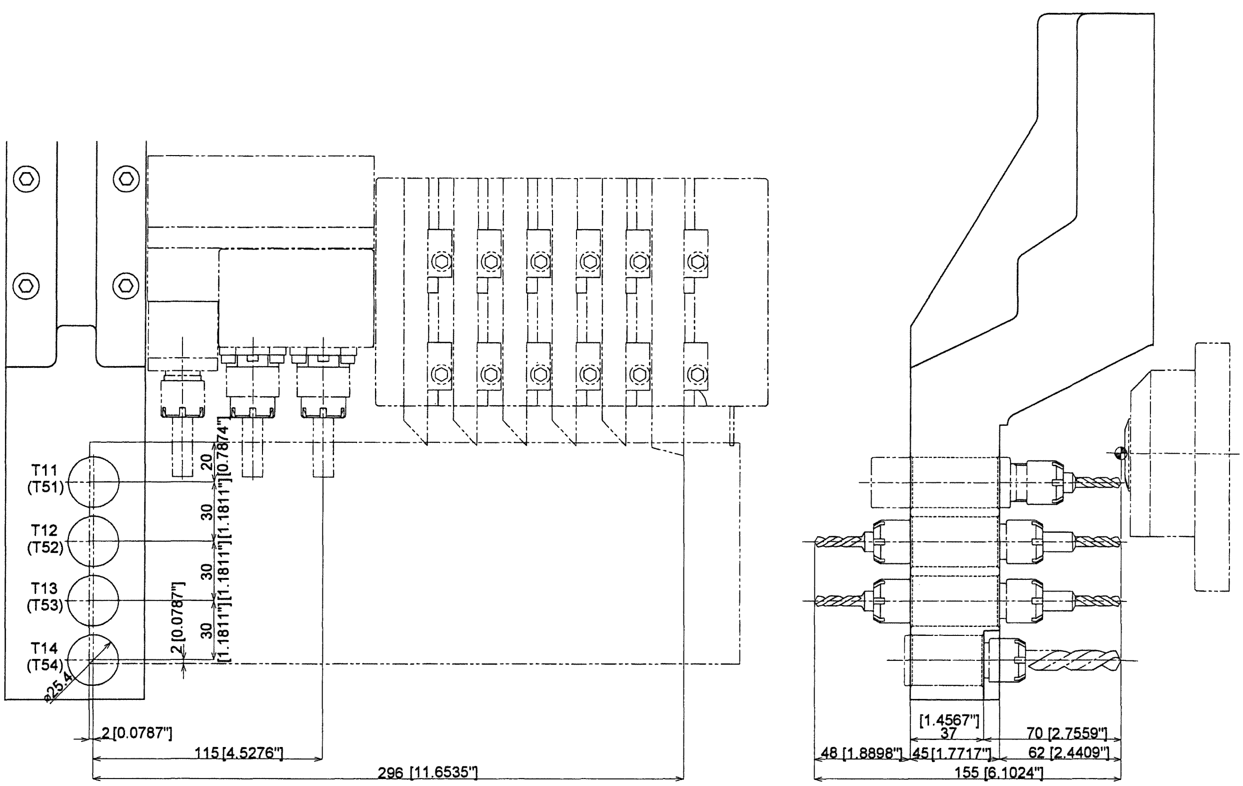
Three rotary tools are mounted at the position shifted by 15 mm [0.59"].

Only a tool mounted on T09 can be replaced with an end-face machining tool.

Tool holder name	U32B
Max. chuck dia.	ø10 mm [ø3/8"]
Spindle speed	Stationary speed: 100 to 6,000 min^{-1} (Rating) Max. instantaneous speed: 8,000 min^{-1}
Chuck type	ER16, AR16

BDF2005 4-Tool Sleeve Holder

A20L



This sleeve holder is provided for performing front and back drilling with up to four arbitrary sleeves mounted. The available tool holders are listed below.

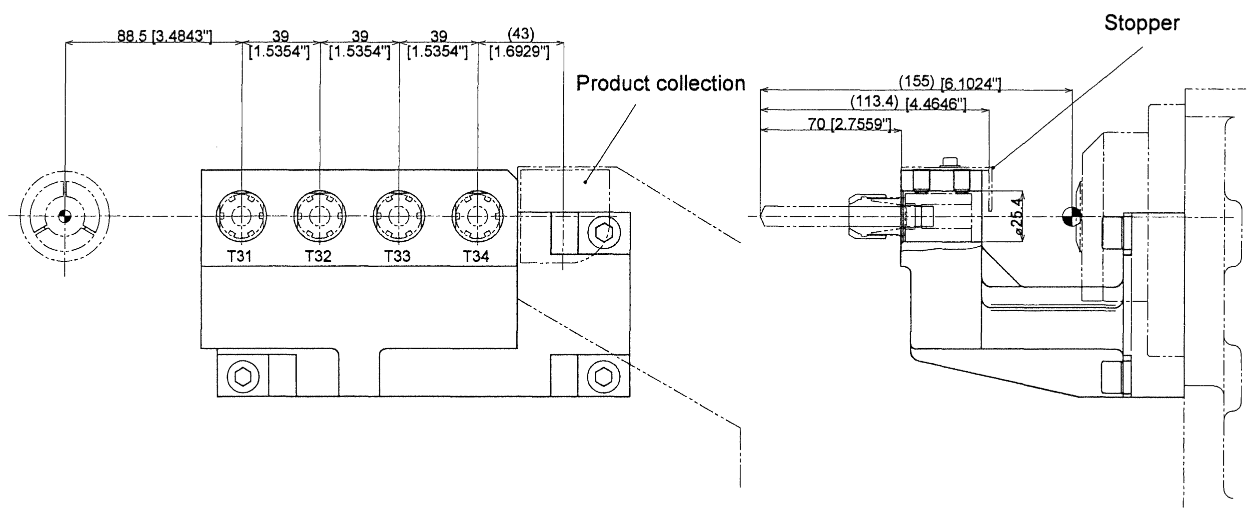
Some restrictions are imposed for outer diameter machining and boring when:

– the cutting face of front tool is placed downward
– the cutting face of back tool is placed upward.

Tool holder name	BDF2005
Tool holder	BTF2212, BTF2213, BTF2216, BTF2412, BTF2413
Sleeves used	ADS110, ADS410, ADS707, ADS710, ADS810, ADS1110, SAU619, SAU620
Sleeve mounting hole dia.	ø25.4 mm [ø1"]

A20L U150B Back 4-Tool Holder

A20L



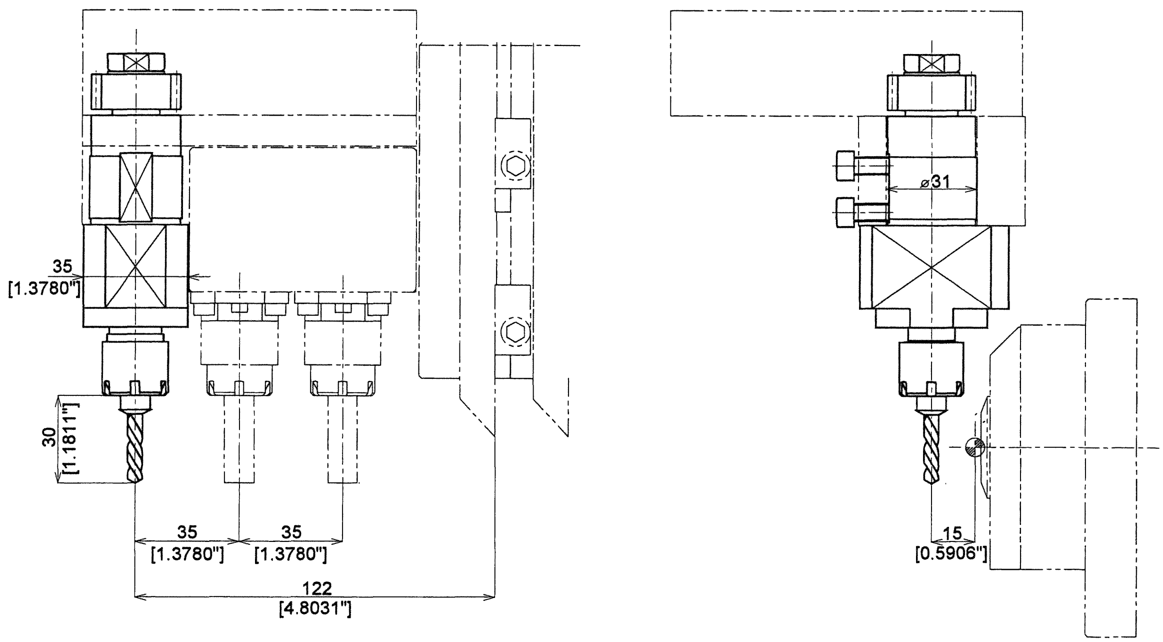
The stopper is mounted because the back end protrusion length of each sleeve is limited. Do not use the machine with the stoppers removed.

Mount up to four sleeves to be used for back drilling.

Tool holder name	A220L U150B
Sleeves used	ADS810, SAU619, SAU620
Sleeve mounting hole dia.	ø25.4 mm [ø1"]

BSC210 Outer Circumference Milling Spindle (1)

A20L | A20R

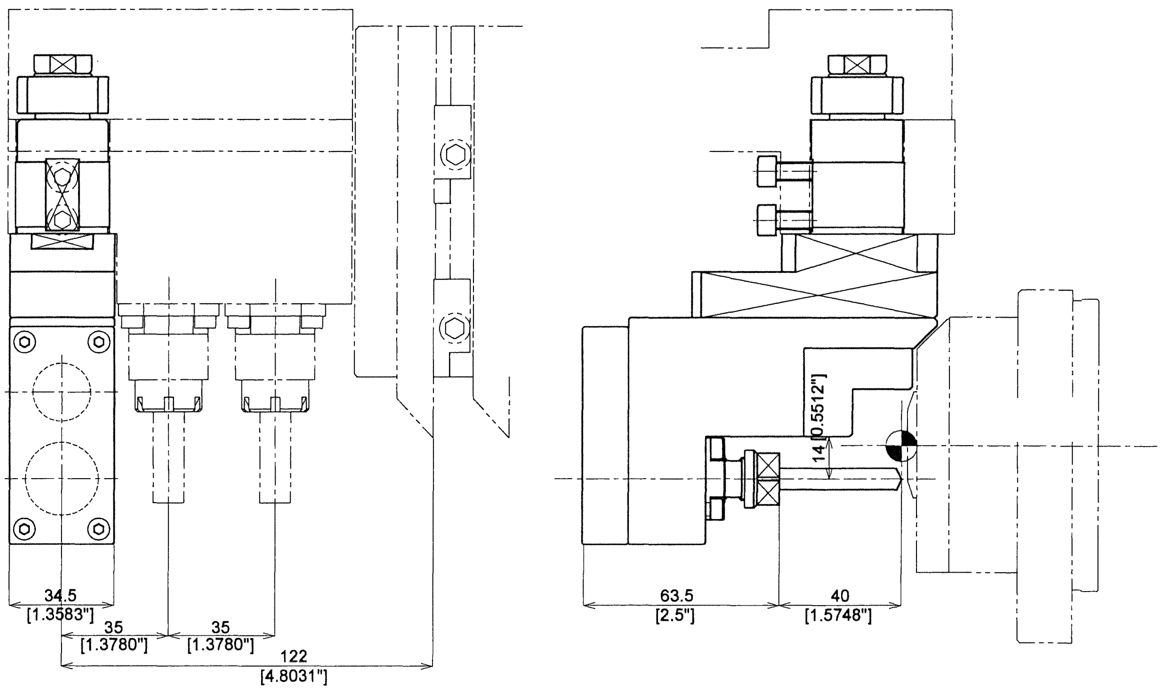


This sleeve holder is mounted on rotary tool drive device, and used for cross drilling.

Tool holder name	BSC210
Max. chuck dia.	ø10 mm [ø3/8"]
Chuck type	ER16, AR16
Rotary tool drive device	U31B, U32B

BSE107 End-face Drilling Sleeve Holder (1)

A20L A20R

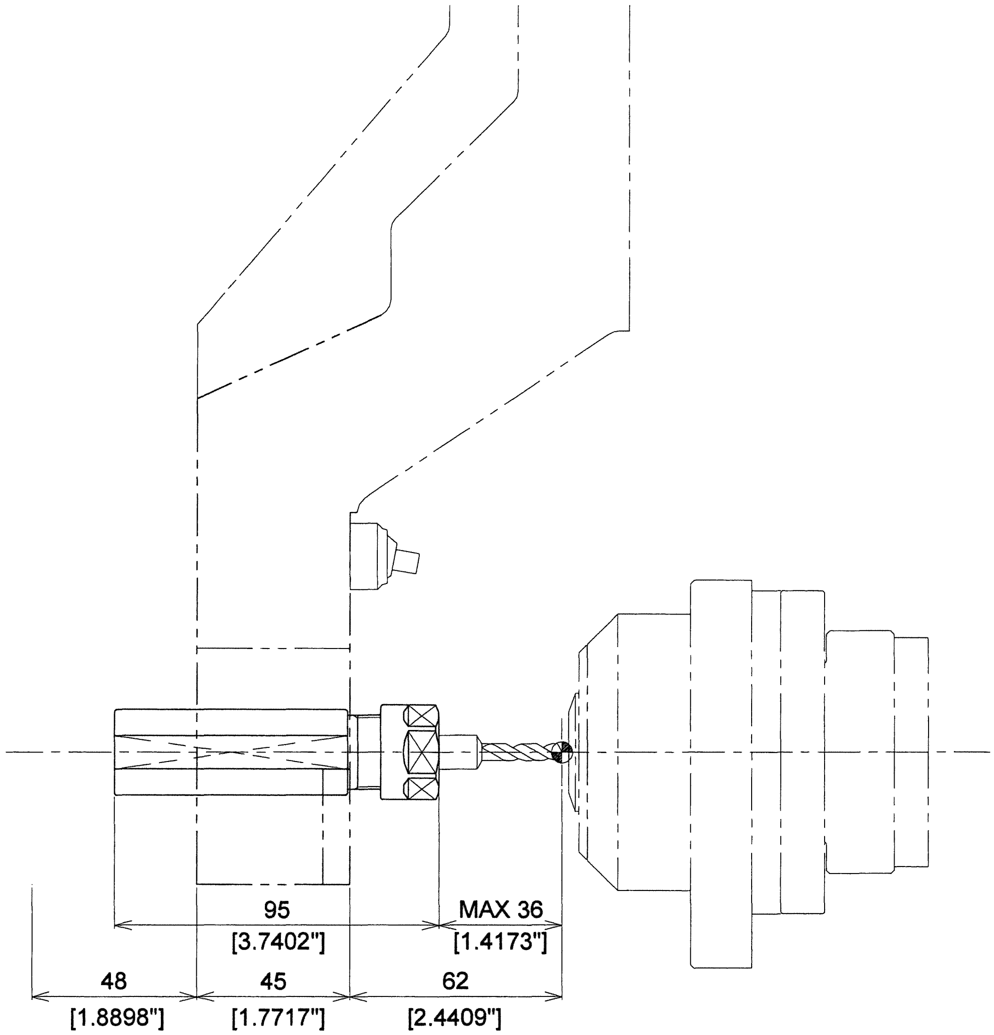


This sleeve holder is mounted on rotary tool drive device, and used for end-face drilling.

Tool holder name	BSE107
Max. chuck dia.	⌀7 mm [⌀0.2756]
Chuck type	ER11, AR11
Rotary tool drive device	U31B, U32B

ADS110 Drill Sleeve (⌀10)

A20L | A20R

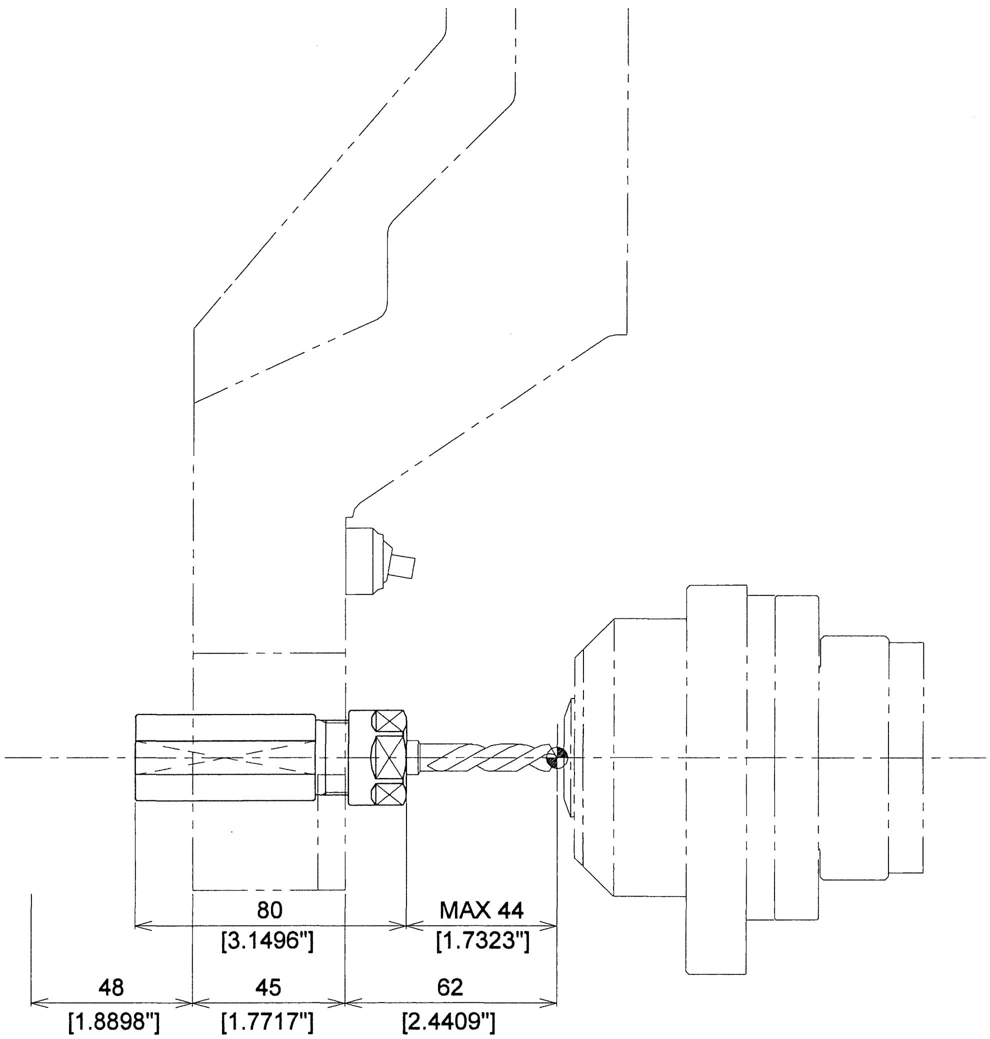


This sleeve chucks the drills having straight shanks with the collet chuck.

Tool holder name	ADS110
Max. chuck dia.	⌀10 mm [⌀3/8"]
Chuck type	ER16, AR16
Sleeve outer dia.	⌀25.4 mm [⌀1"]

ADS410 Drill Sleeve (⌀10)

A20L A20R

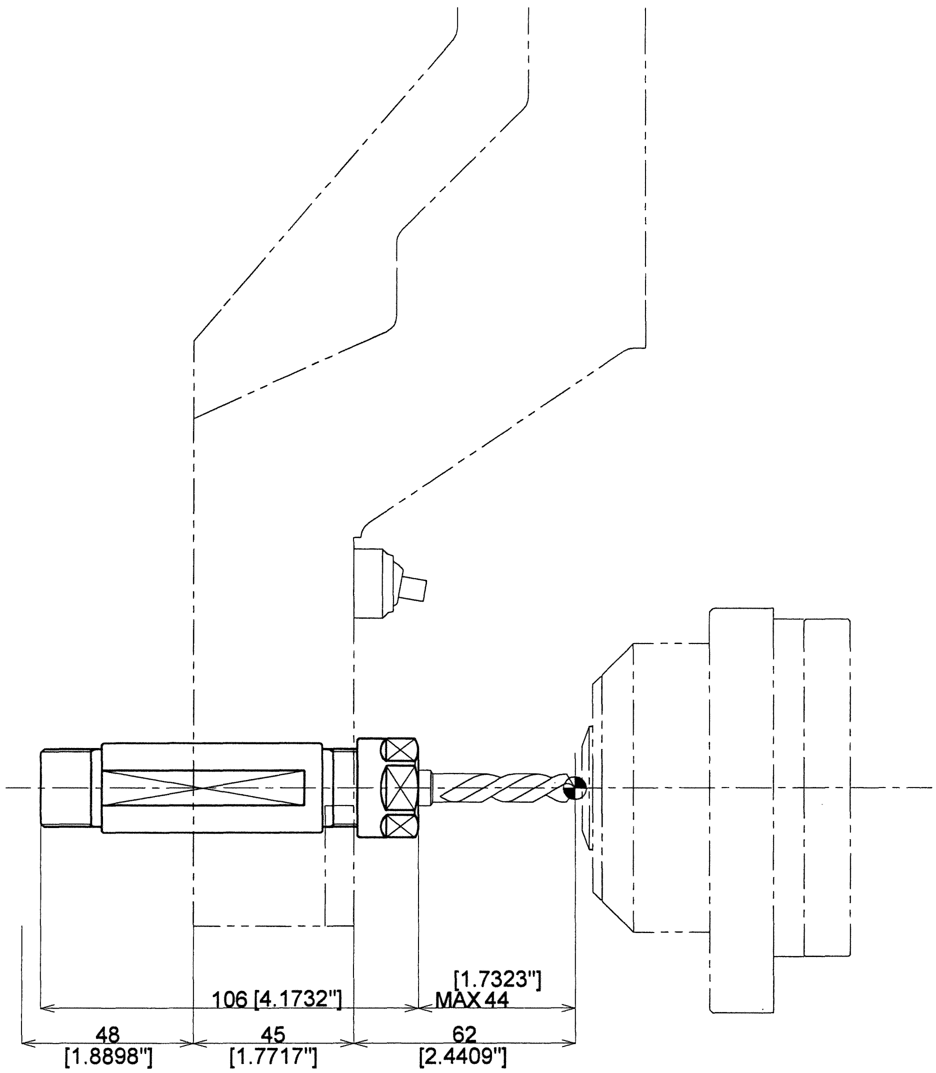


This sleeve chucks the drills having straight shanks with the collet chuck.

Tool holder name	ADS410
Max. chuck dia.	⌀10 mm [⌀3/8"]
Chuck type	ER16, AR16
Sleeve outer dia.	⌀25.4 mm [⌀1"]

ADS310 Drill Sleeve (⌀10)

A20L | A20R

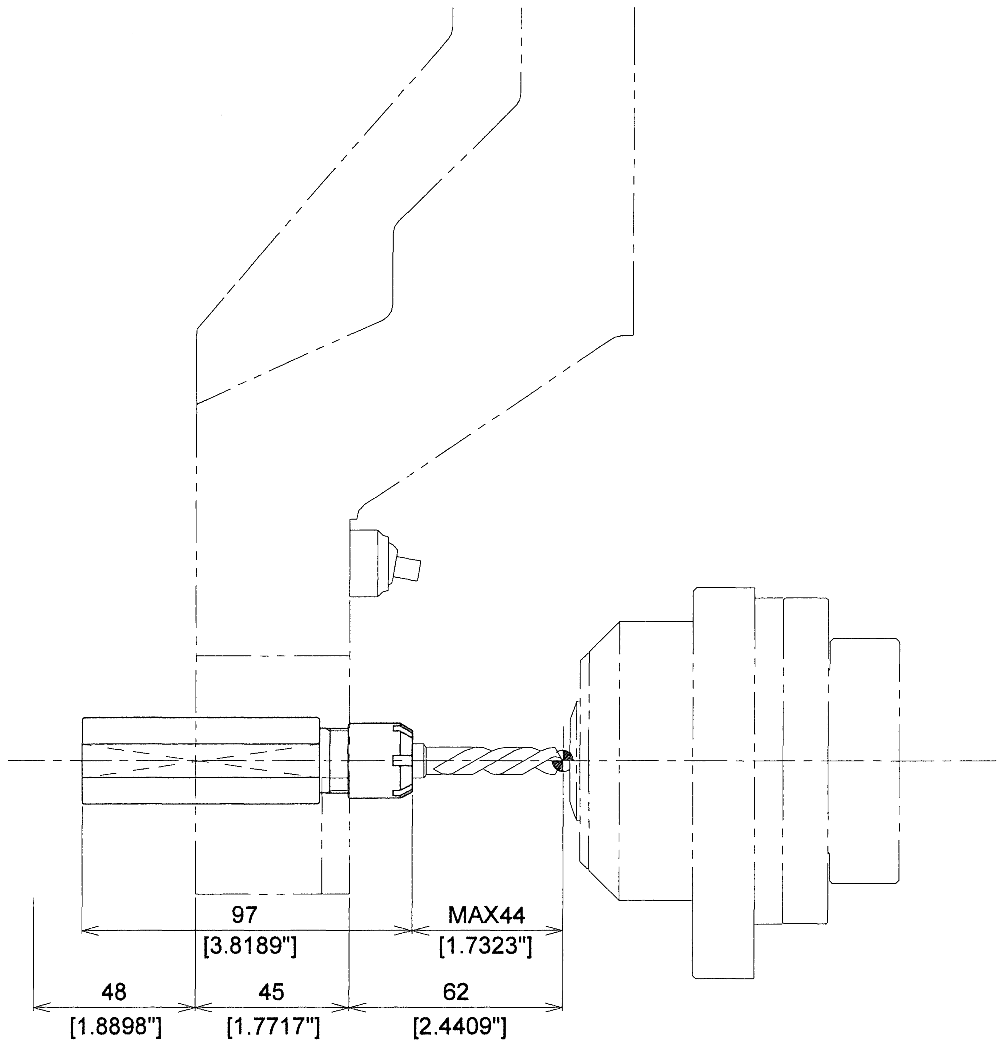


This sleeve chucks the drills having straight shanks with the collet chuck.

Tool holder name	ADS310
Max. chuck dia.	⌀10 mm [⌀3/8"]
Chuck type	ER16, AR16
Sleeve outer dia.	⌀25.4 mm [⌀1"]

ADS710 Drill Sleeve (⌀10)

A20L A20R

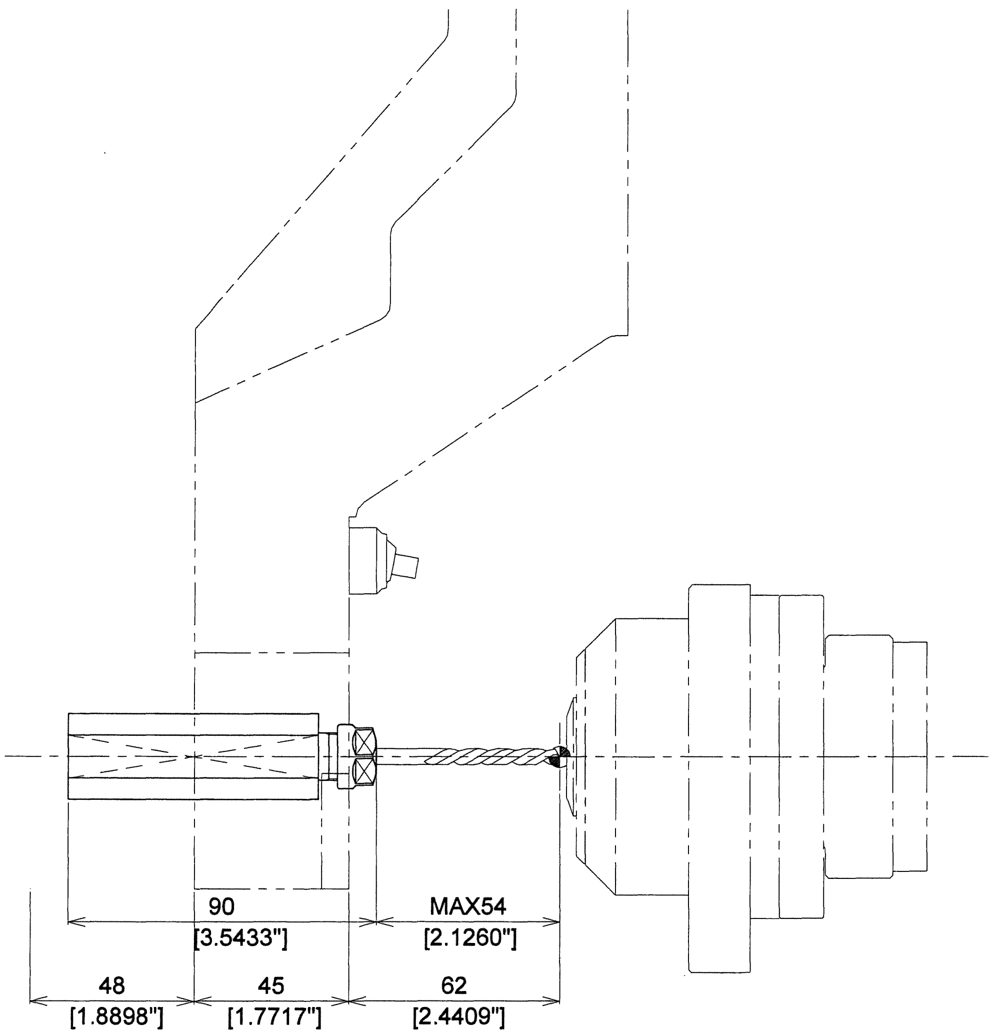


This sleeve chucks the drills having straight shanks with the collet chuck.

Tool holder name	ADS710
Max. chuck dia.	⌀10 mm [⌀3/8"]
Chuck type	ER16, AR16
Sleeve outer dia.	⌀25.4 mm [⌀1"]

ADS707 Drill Sleeve (⌀7)

A20L | A20R

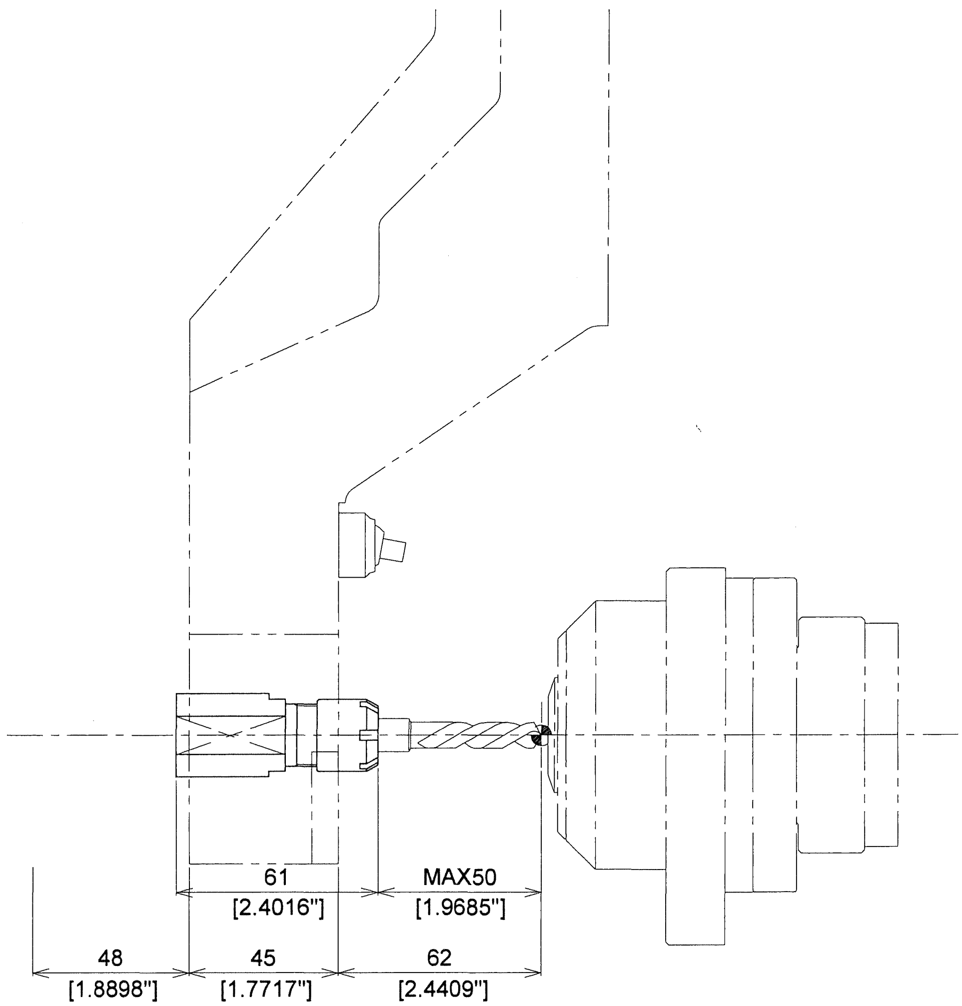


This sleeve chucks the drills having straight shanks with the collet chuck.

Tool holder name	ADS707
Max. chuck dia.	⌀7 mm [⌀0.2756"]
Chuck type	ER11, AR11
Sleeve outer dia.	⌀25.4 mm [⌀1"]

ADS810 Drill Sleeve (⌀10)

A20L | A20R

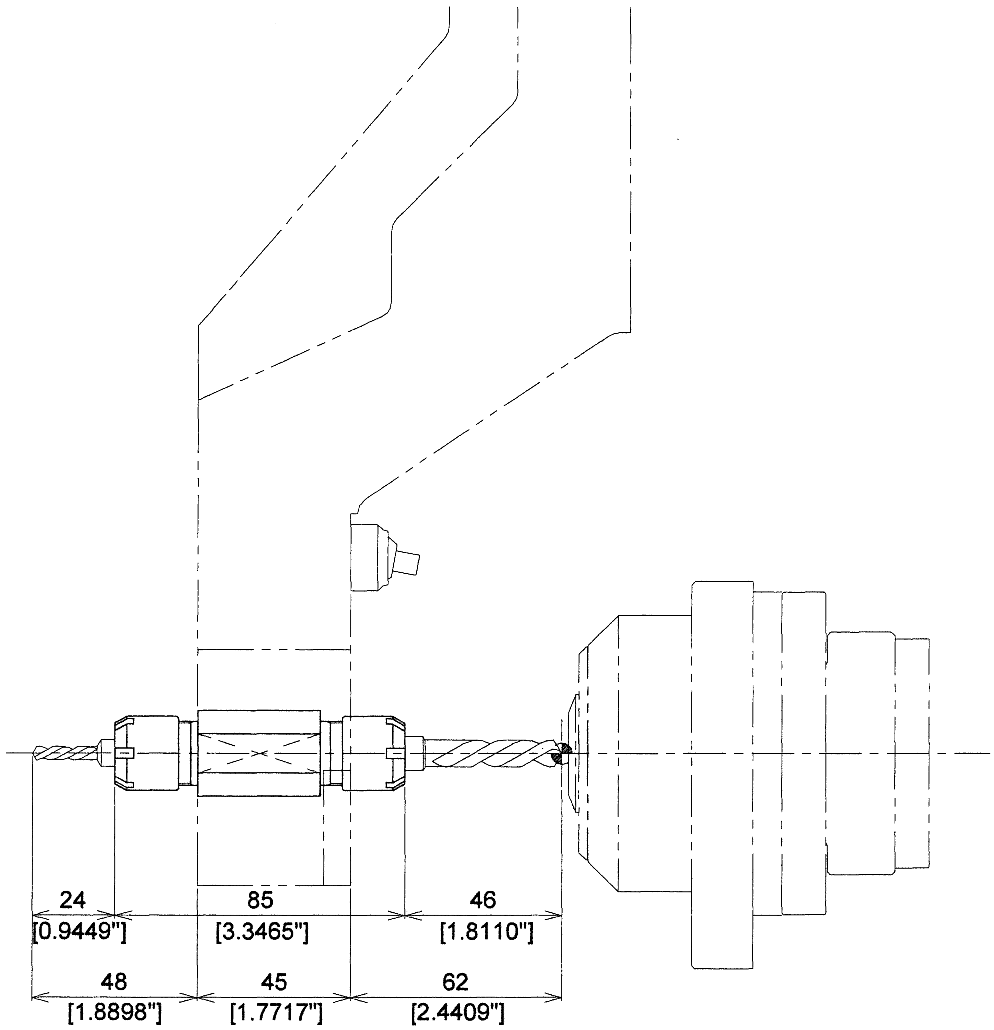


This sleeve chucks the drills having straight shanks with the collet chuck.

Tool holder name	ADS810
Max. chuck dia.	⌀10 mm [⌀3/8"]
Chuck type	ER16, AR16
Sleeve outer dia.	⌀25.4 mm [⌀1"]

ADS1110 Drill Sleeve (⌀10)

A20L A20R

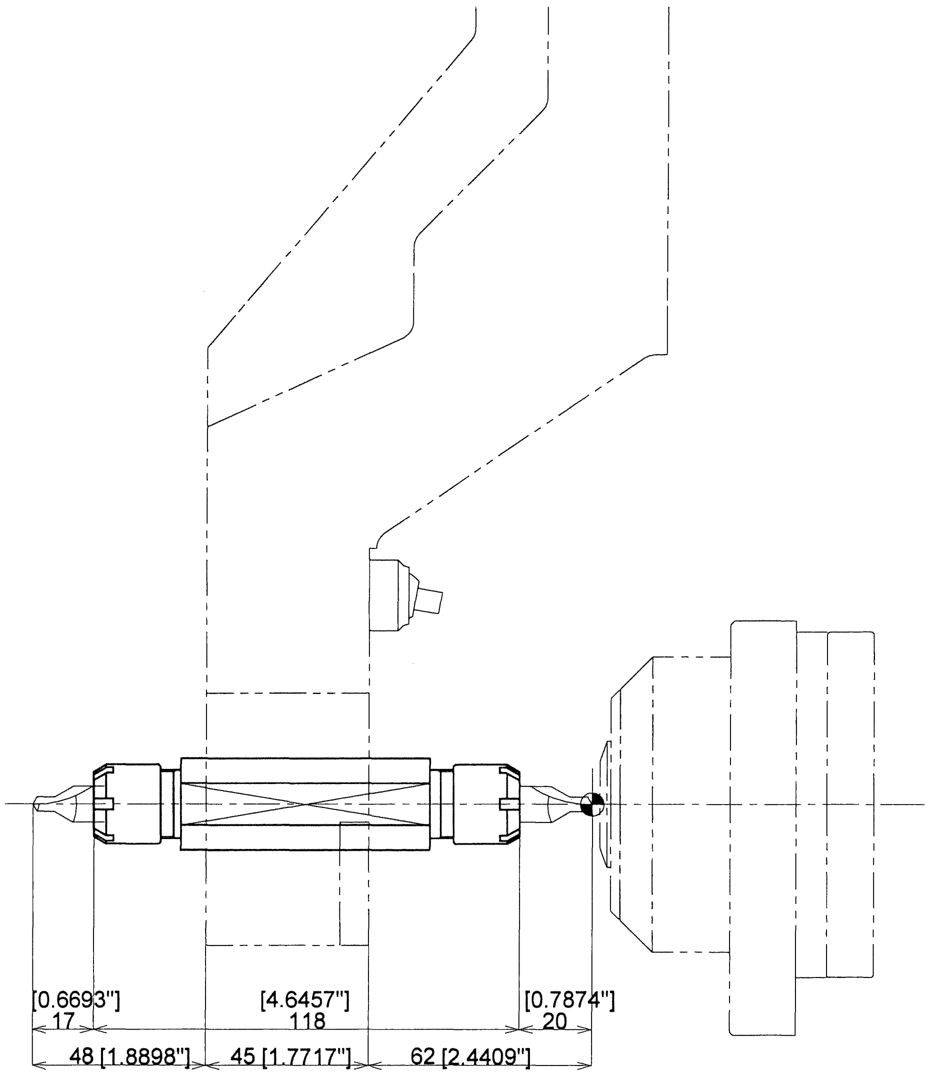


This sleeve chucks the drills having straight shanks with the collet chuck.

Tool holder name	ADS1110
Max. chuck dia.	⌀10 mm [⌀3/8"]
Chuck type	ER16, AR16
Sleeve outer dia.	⌀25.4 mm [⌀1"]

ADS1410 Drill Sleeve (⌀10)

A20L A20R

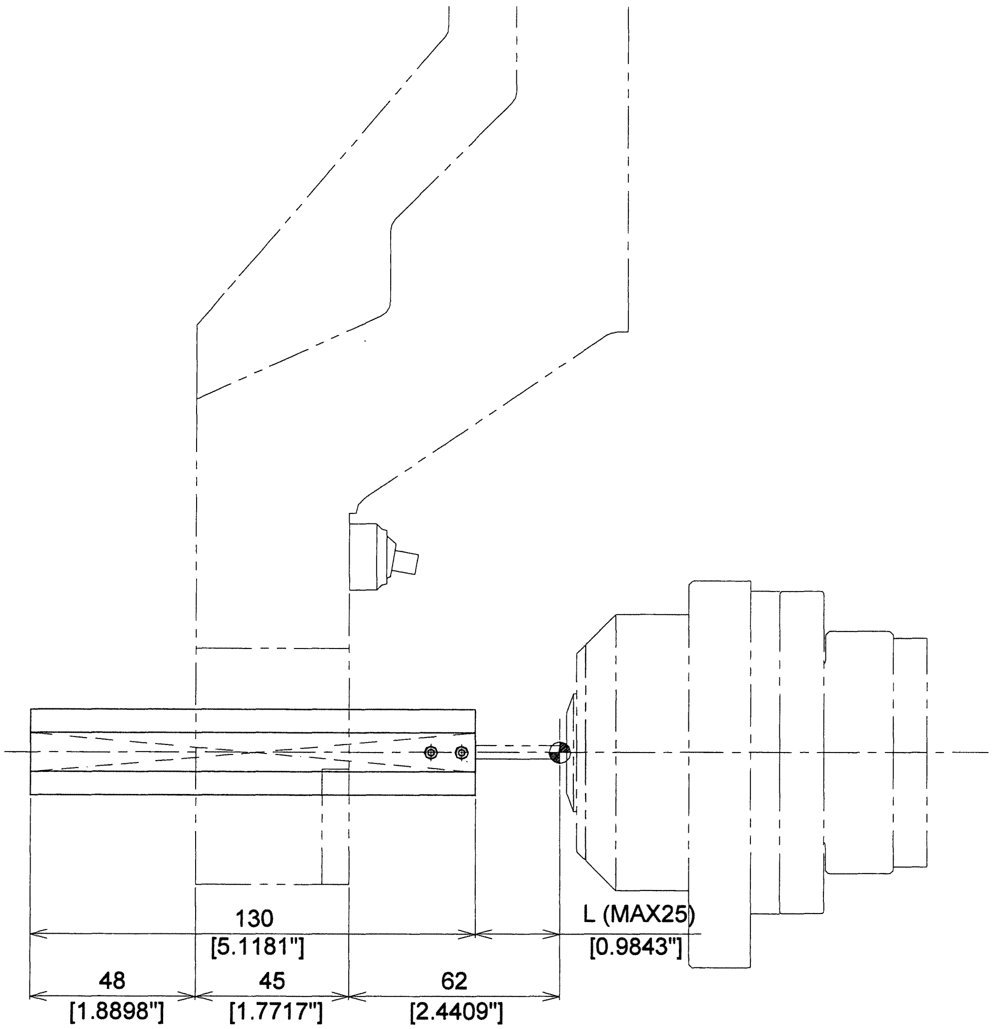


This sleeve chucks the drills having straight shanks with the collet chuck.

Tool holder name	ADS1410
Max. chuck dia.	⌀10 mm [⌀3/8"]
Chuck type	ER16, AR16
Sleeve outer dia.	⌀25.4 mm [⌀1"]

ABS104 Sleeve (Boring)

A20L A20R

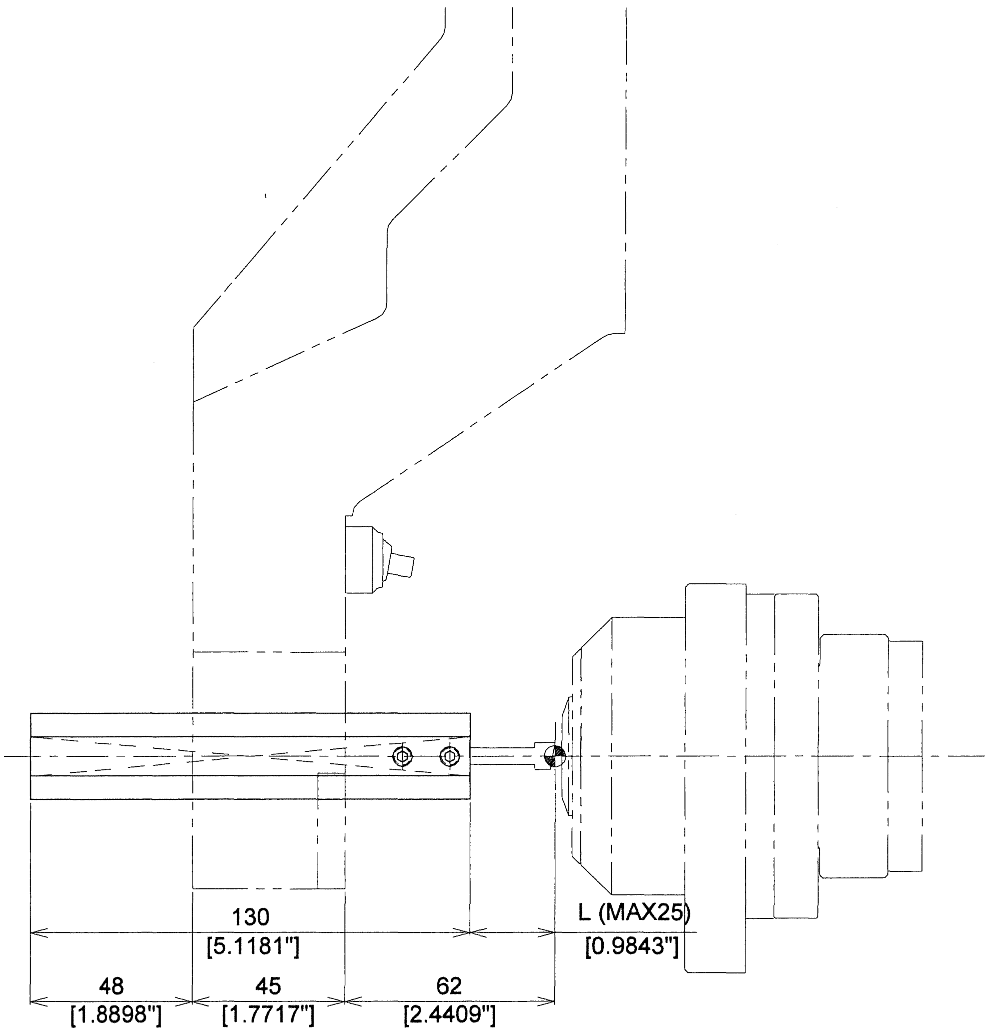


This sleeve is provided for boring. The dimension L can be adjusted by moving the sleeve.

Sleeve name	ABS104
Tool No.	IE6140, IE6240
Tool size	⌀4 mm [⌀5/32"]
Sleeve outer dia.	⌀25.4 mm [⌀1"]

ABS108 Sleeve (Boring)

A20L A20R

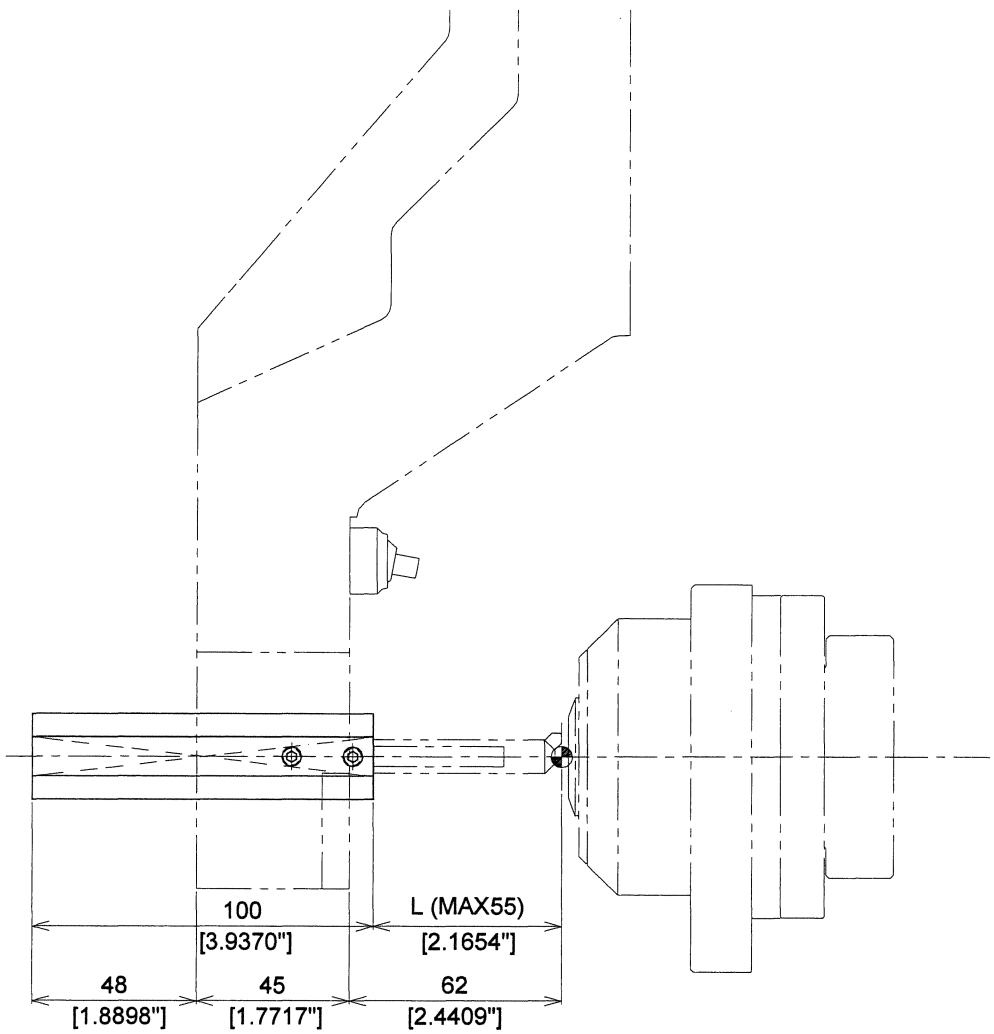


This sleeve is provided for boring. The dimension L can be adjusted by moving the sleeve.

Sleeve name	ABS108
Tool No.	IE6180, IE6280
Tool size	⌀8 mm [⌀5/16"]
Sleeve outer dia.	⌀25.4 mm [⌀1"]

ABS110 Sleeve (Boring)

A20L | A20R

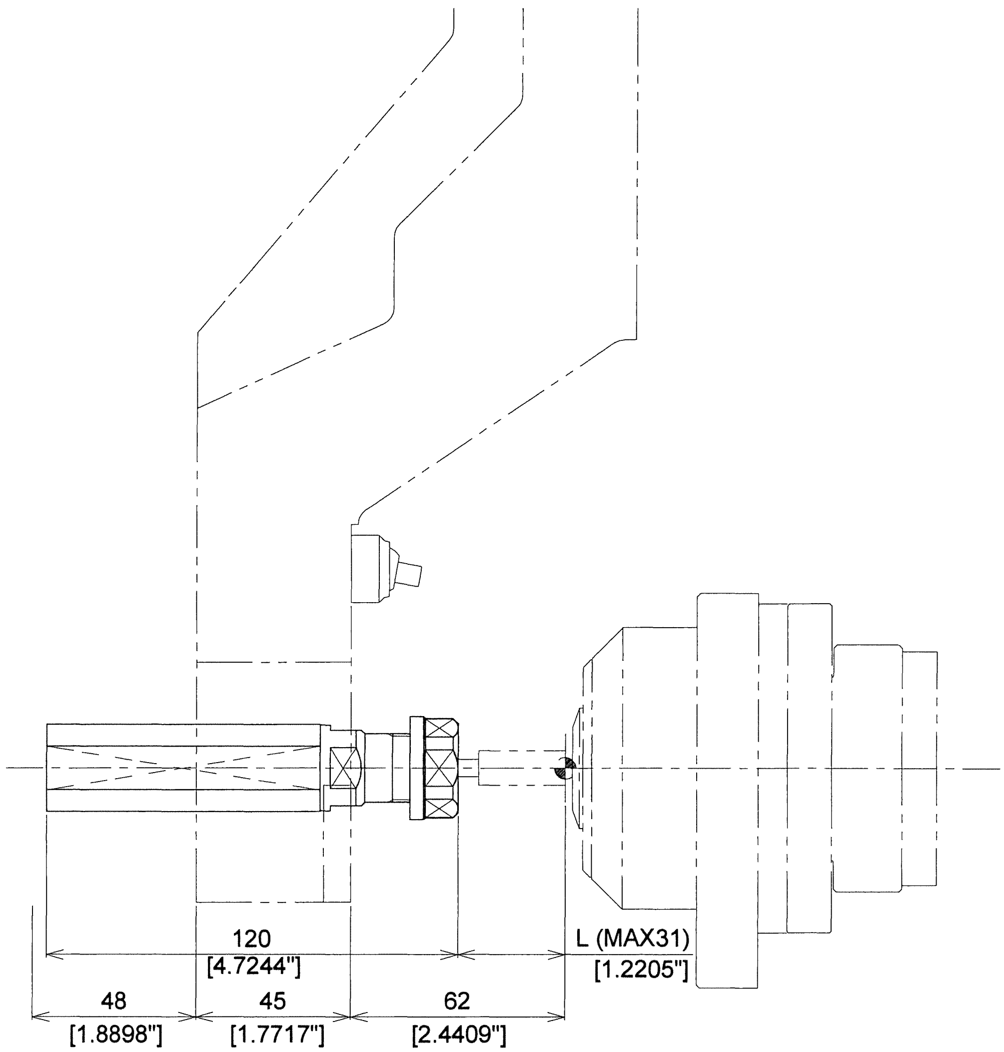


This sleeve is provided for boring with boring bar. This sleeve has a larger diameter than others and suitable for deep hole. The dimension L can be adjusted by moving the sleeve.

Sleeve name	ABS110
Tool No.	IE6340-BR-404
Tool size	⌀10 mm [⌀3/8"]
Sleeve outer dia.	⌀25.4 mm [⌀1"]

ATS110 Sleeve (Tap)

A20L A20R

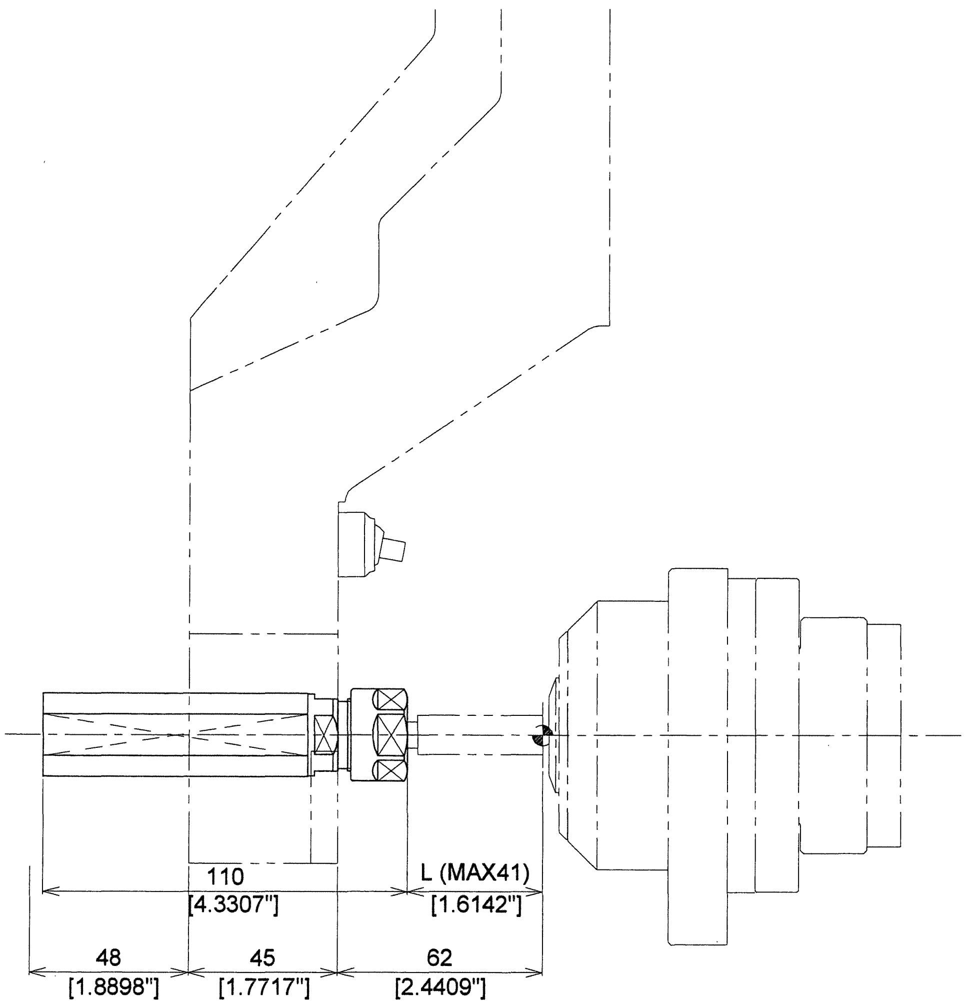


This sleeve is provided for tapping and designed for mounting a right-handed or left-handed tool. A tap is chucked by the rego type chuck. The dimension L can be adjusted by moving the sleeve.

Sleeve name	ATS110
Max. tapping dia.	M10
Chuck type	FC090
Max. chuck dia.	⌀10 mm [⌀3/8"]
Sleeve outer dia.	⌀25.4 mm [⌀1"]

ATS112 Sleeve (Tap)

A20L A20R

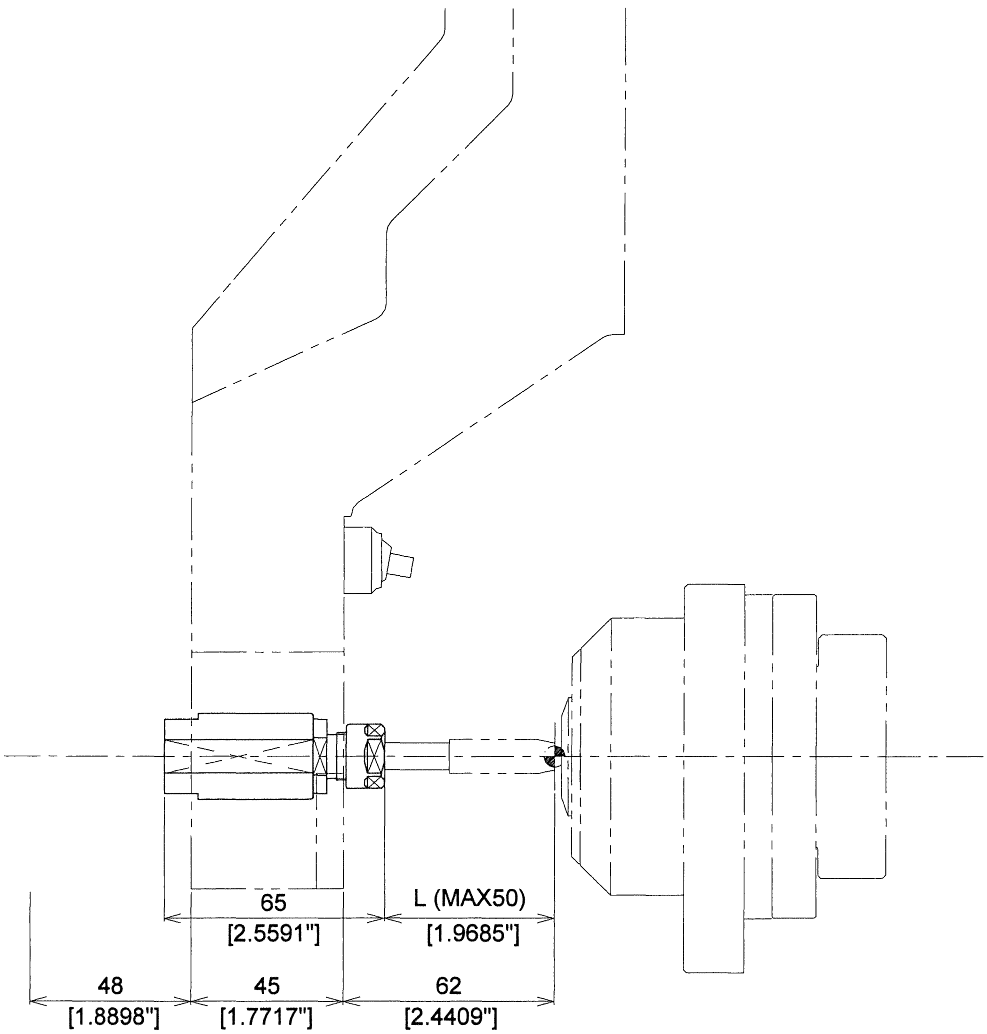


This sleeve is provided for tapping and designed for mounting a right-handed or left-handed tool. A tap is chucked by the rego type chuck. The dimension L can be adjusted by moving the sleeve.

Sleeve name	ATS112
Max. tapping dia.	M10
Chuck type	AR16, ER16
Max. chuck dia.	⌀10 mm [⌀3/8"]
Sleeve outer dia.	⌀25.4 mm [⌀1"]

ATS210 Sleeve (Tap)

A20L | A20R

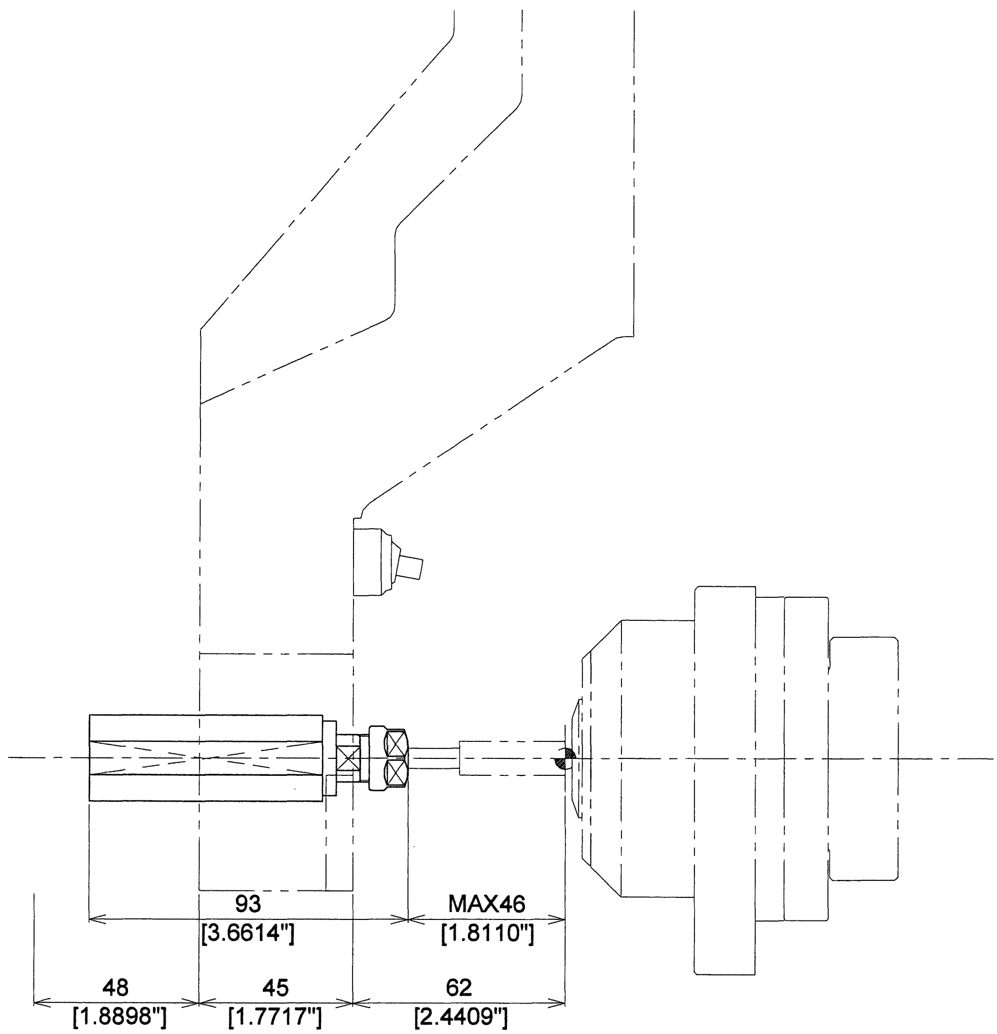


This floating sleeve is provided for holding taps. A tap is chucked by the rego chuck. The dimension L can be adjusted by moving the sleeve.

Sleeve name	ATS210
Max. chuck dia.	⌀7 mm [⌀0.2756"]
Chuck type	AR11, ER11
Sleeve outer dia.	⌀25.4 mm [⌀1"]

ATS307 Sleeve (Tap)

A20L | A20R

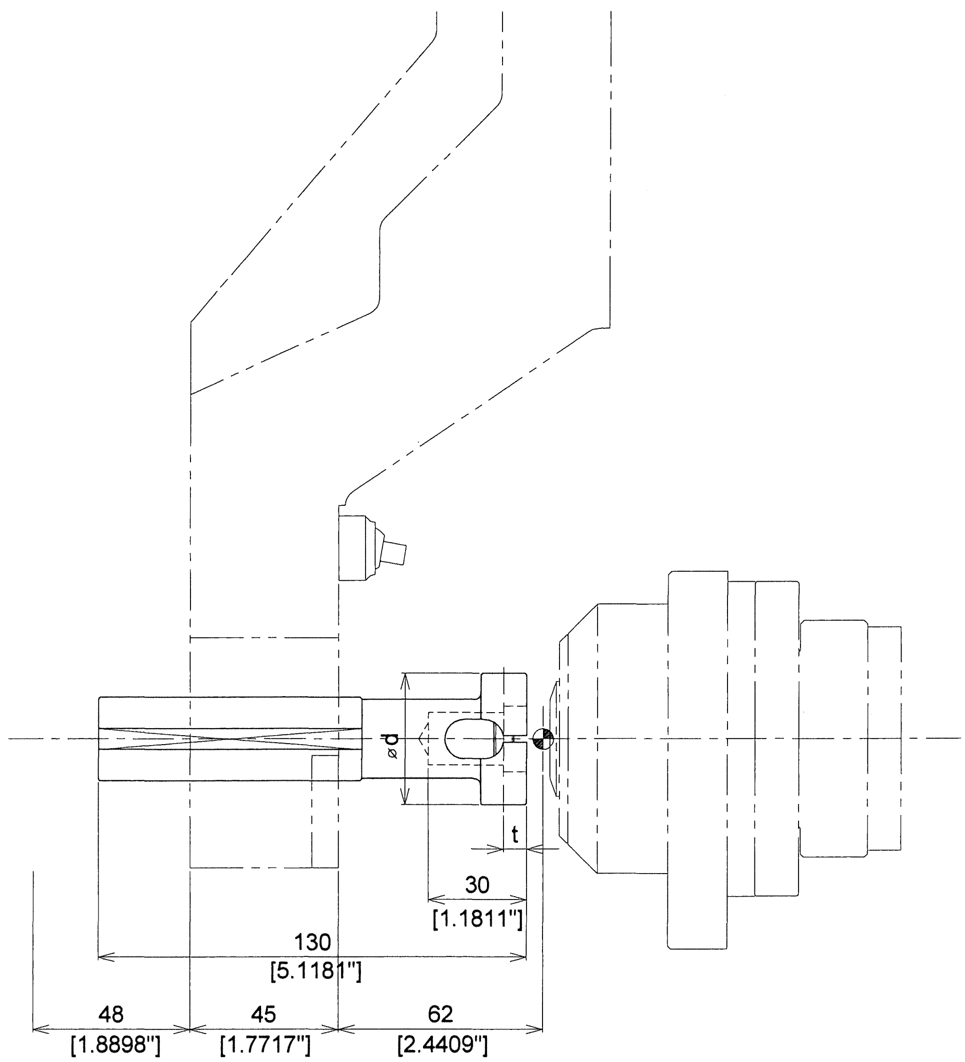


This floating sleeve is provided for holding taps. A tap is chucked by the rego chuck. The dimension L can be adjusted by moving the sleeve.

Sleeve name	ATS307
Max. chuck dia.	⌀7 mm [⌀0.2756"]
Chuck type	AR11, ER11
Sleeve outer dia.	⌀25.4 mm [⌀1"]

AUS106 Sleeve (Die)

A20L | A20R

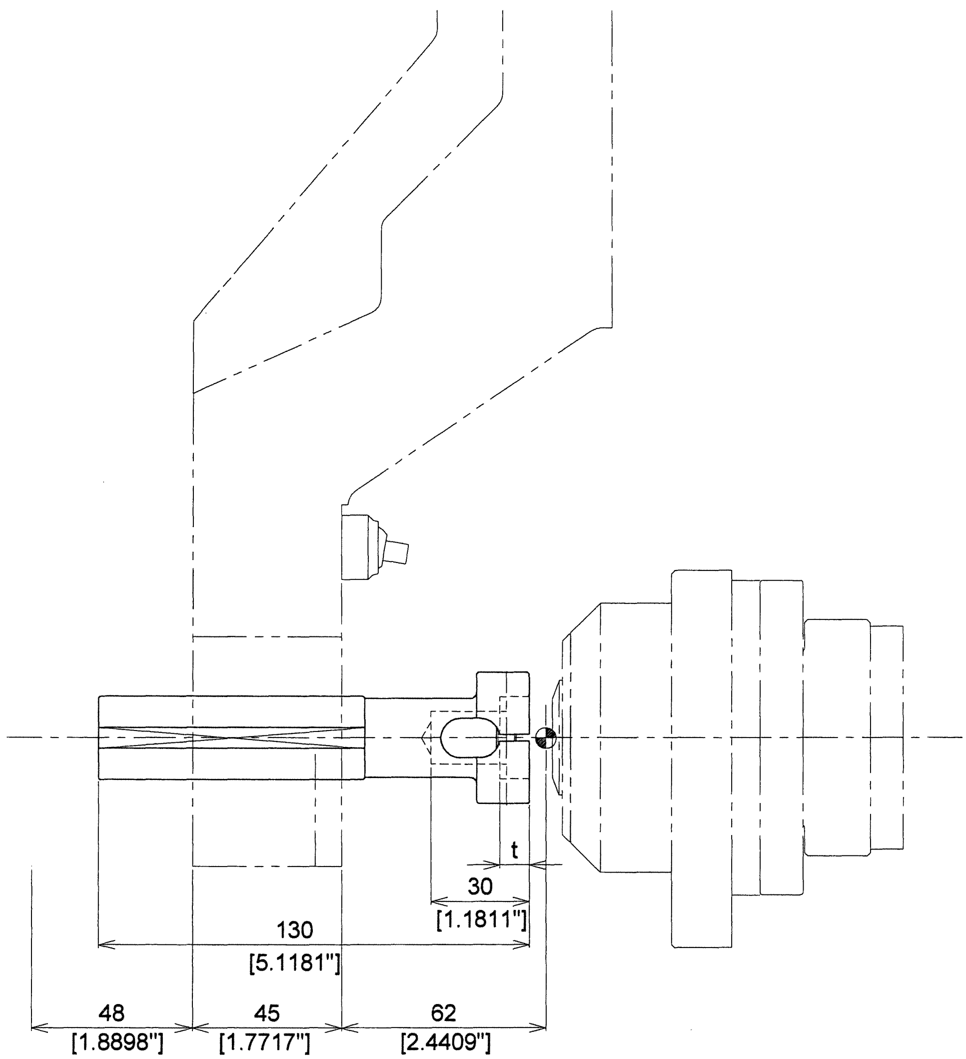


This sleeve is provided for machining a die of M2 to M6 diameter.

Sleeve name	AUS106
Range of tapping dia.	M2 ~ M6×30
Size (ød×t)	ø20×7 mm
Sleeve outer dia.	ø25.4 mm [ø1"]

AUS110 Sleeve (Die)

A20L | A20R

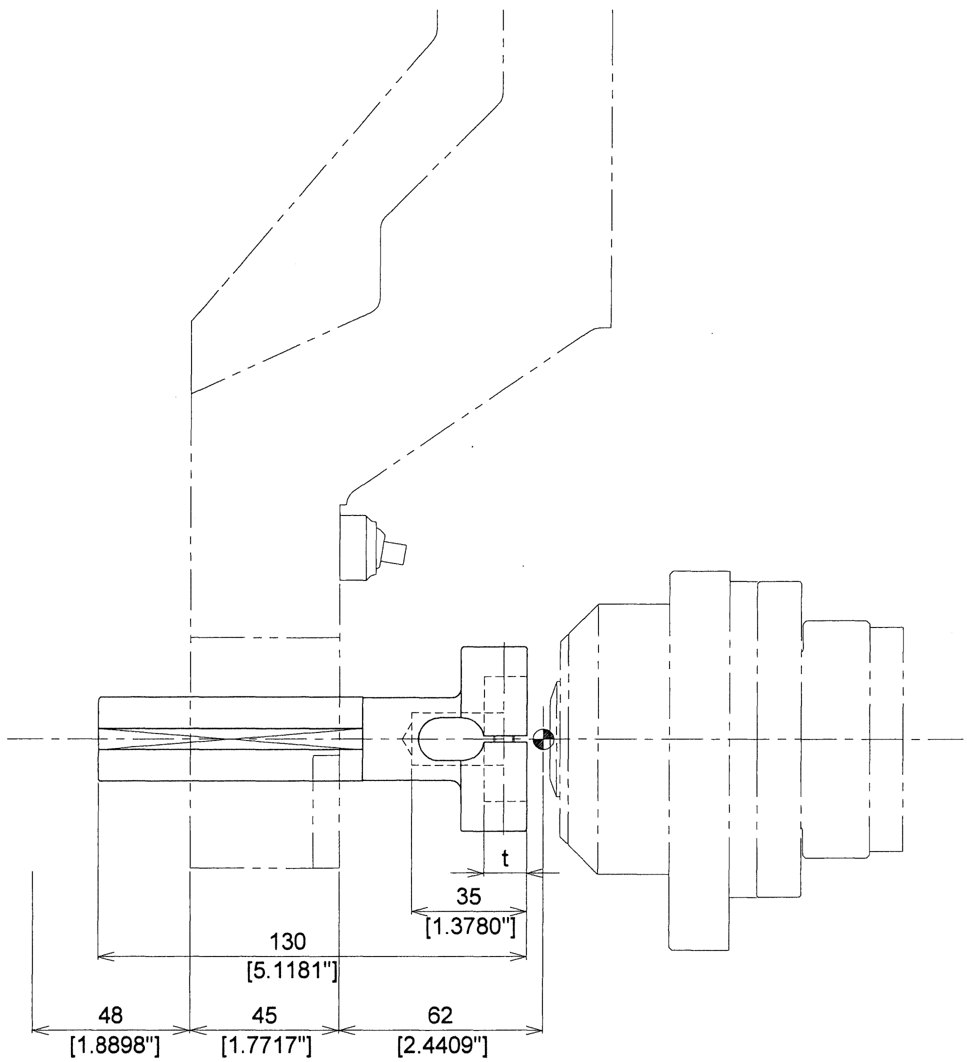


This sleeve is provided for machining a die of M6 to M8 diameter.

Sleeve name	AUS110
Range of tapping dia.	M6 ~ M8×30
Size (⌀d×t)	⌀25×9 mm
Sleeve outer dia.	⌀25.4 mm [⌀1"]

AUS112 Sleeve (Die)

A20L | A20R

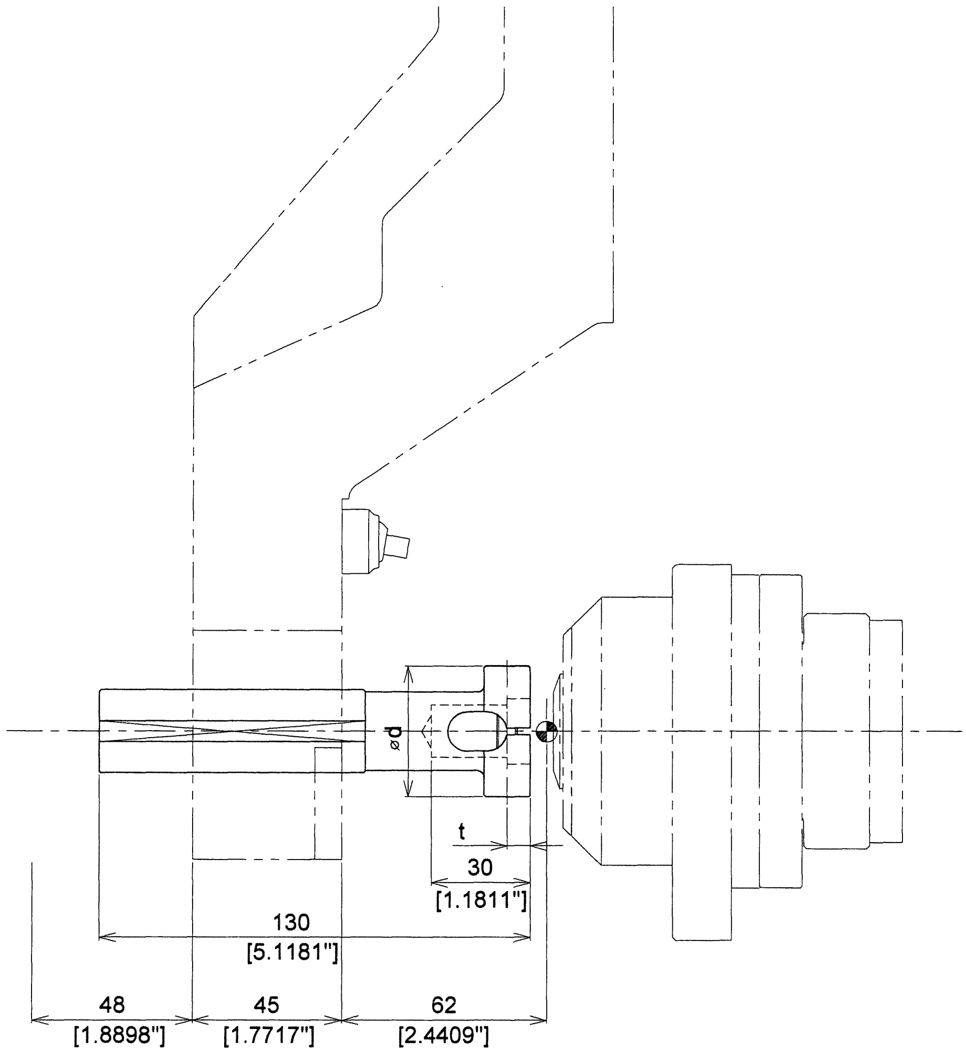


This sleeve is provided for machining a die of M8 to M10 diameter.
No other sleeve can be mounted next to this sleeve.

Sleeve name	AUS112
Range of tapping dia.	M8 ~ M10×35
Size (⌀d×t)	⌀38×13 mm
Sleeve outer dia.	⌀25.4 mm [⌀1"]

AUS106L Sleeve (Die)

A20L | A20R

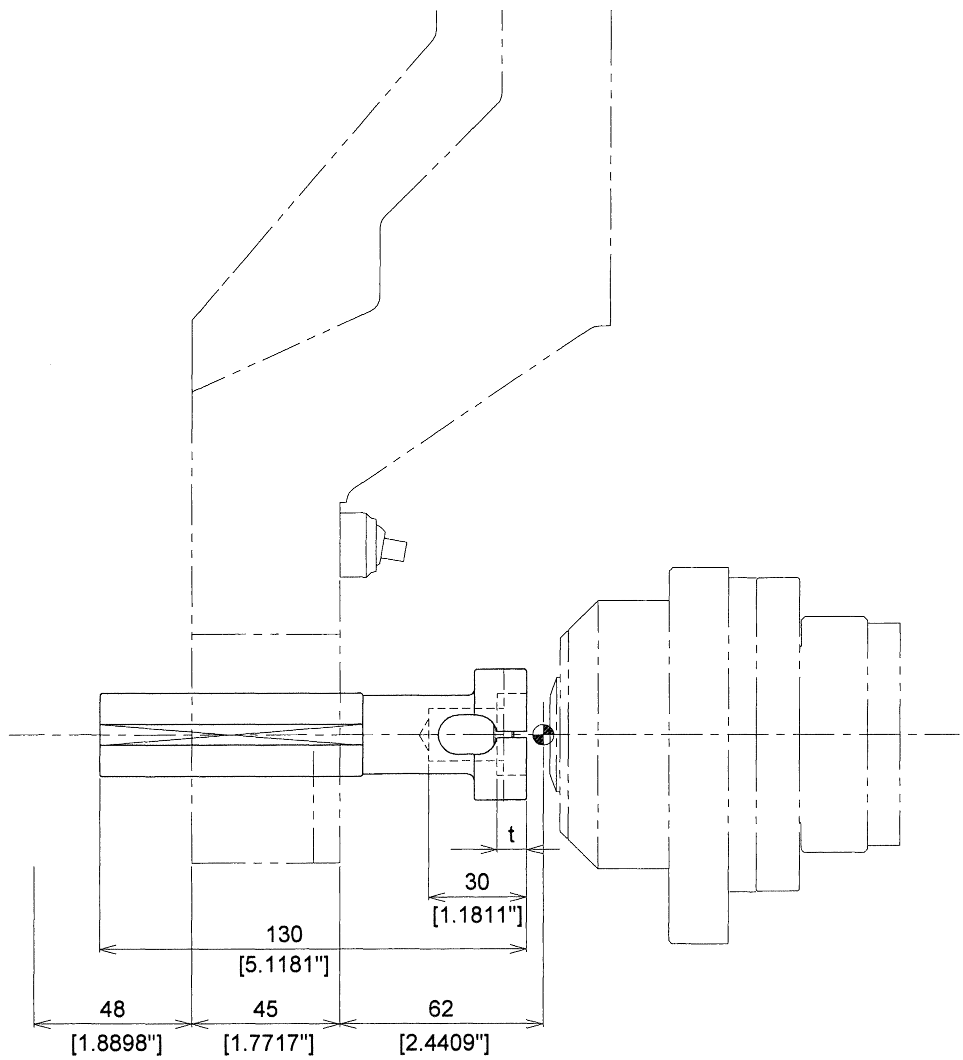


This sleeve is provided for machining a die of M2 to M6 diameter.

Sleeve name	AUS106L
Range of tapping dia.	M2 ~ M6×1-3/16"
Size (ød×t)	ø 3/4"×1/4"
Sleeve outer dia.	ø25.4 mm [ø1"]

AUS110L Sleeve (Die)

A20L | A20R

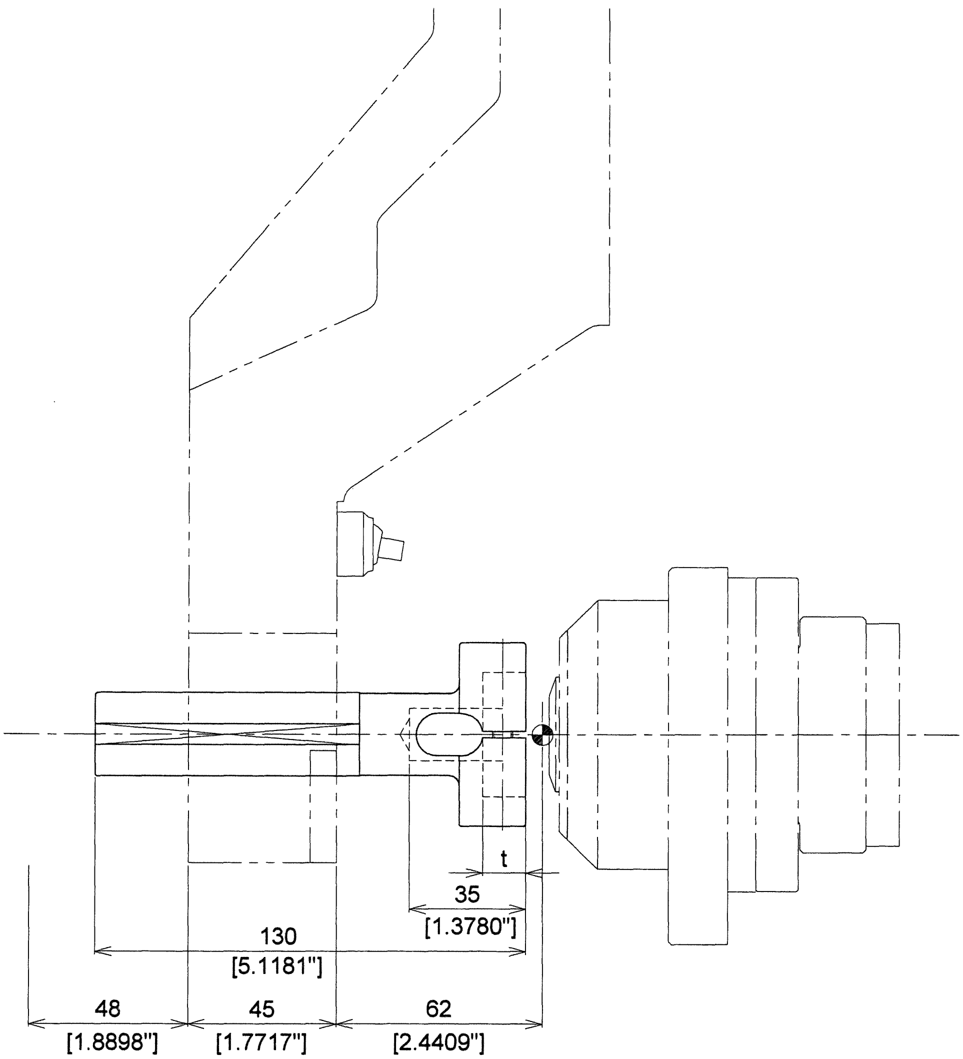


This sleeve is provided for machining a die of M6 to M8 diameter.

Sleeve name	AUS110L
Range of tapping dia.	M6 ~ M8×1-3/16"
Size (⌀d×t)	⌀1"×11/32"
Sleeve outer dia.	⌀25.4 mm [⌀1"]

AUS112L Sleeve (Die)

A20L | A20R

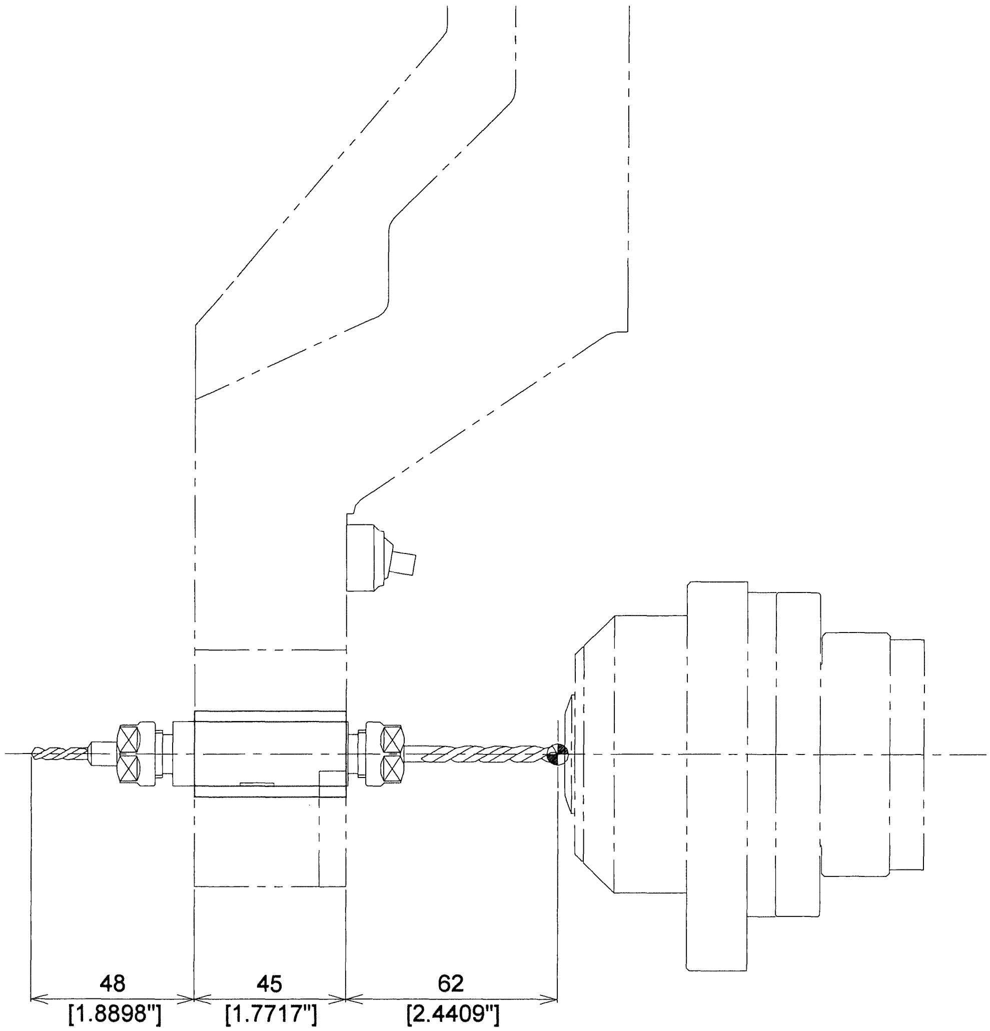


This sleeve is provided for machining a die of M6 to M8 diameter.

Sleeve name	AUS112L
Range of tapping dia.	M8 ~ M10×1-3/8"
Size (⌀d×t)	⌀1-1/2"×1/2"
Sleeve outer dia.	⌀25.4 mm [⌀1"]

SAU619 Sleeve Adapter (⌀19.05)

A20L A20R

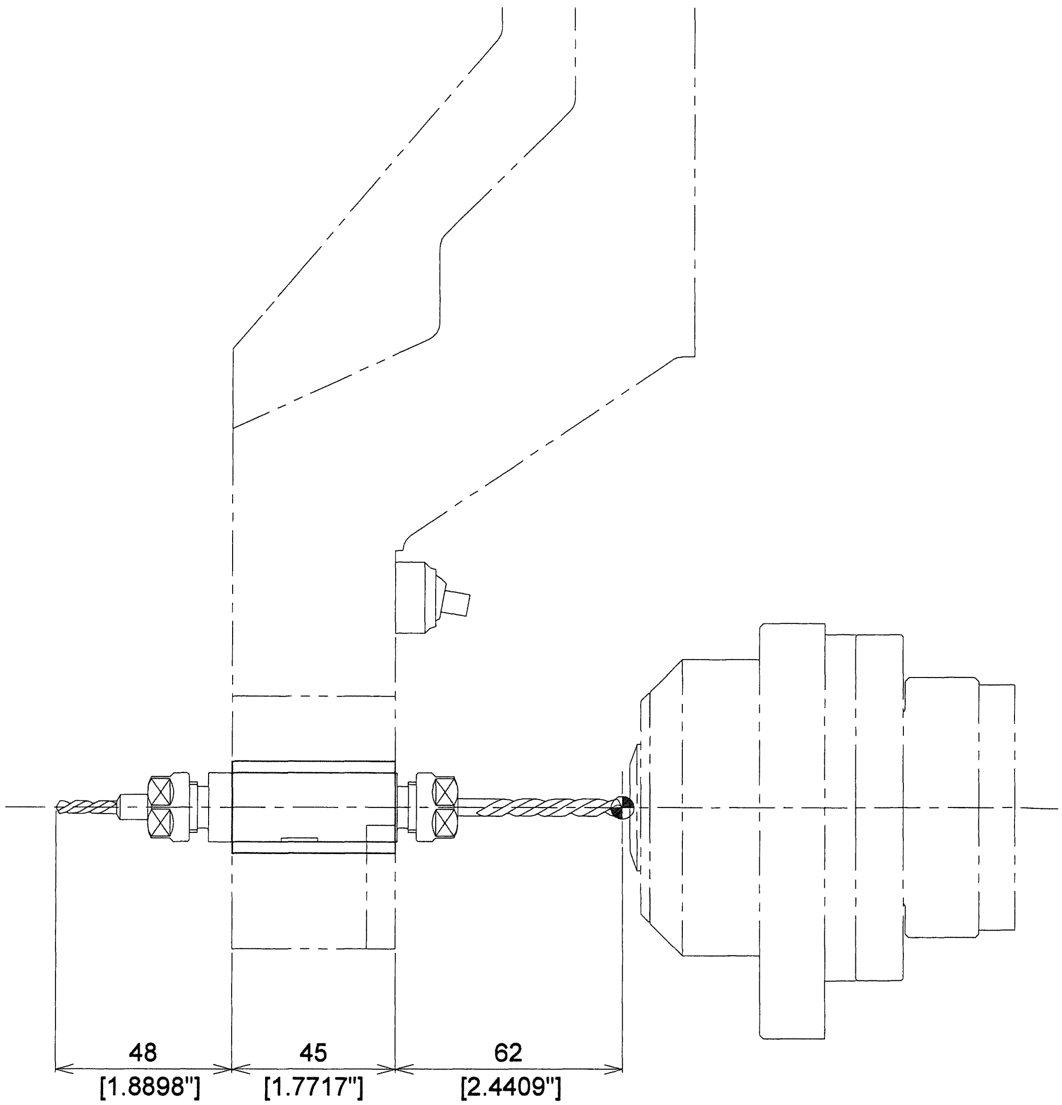


This sleeve is provided to use the commercial drill sleeve of 19.05 mm diameter with the sleeve holder.

Tool holder name	SAU619
Sleeve inner dia.	⌀19.05 mm [⌀3/4"]
Sleeve outer dia.	⌀25.4 mm [⌀1"]
Sleeve holder	BDF2005, BDF2006, U150B

SAU620 Sleeve Adapter (⌀20)

A20L | A20R



This sleeve is provided to use the commercial drill sleeve of 20 mm diameter with the sleeve holder.

Tool holder name	SAU620
Sleeve inner dia.	⌀20 mm [⌀0.7874"]
Sleeve outer dia.	⌀25.4 mm [⌀1"]
Sleeve holder	BDF2005, BDF2006, U150B

Product Code	C	–	A	2	2	0	P	L	VII					
Document Code	2	E	1	–	1	1	0	1						