Cincom L16/L20

(5M7)

(5M3)

(5M1)

Programmer's Manual

Precision Machinery Division
Citizen Watch Co., Ltd.

NOTICE

1. Although every precaution has been taken in the preparation of this document, it may contain inaccurate descriptions, typographical errors, or omissions. If you find any question, error or omission, please report it to your Citizen representative.
2. Information in this document is subject to change without notice. It is a Citizen's policy to improve products whenever new technology, components, software, or firmware become available. Therefore, changes are made periodically to the information contained herein and the updated information is included in the latest edition of the document. Citizen also reserves the right to change specifications without prior notice.
3. All features, functions, and operations described herein are not common in all parts of the world. Some illustrations in this document may not match the actual machine.
4.
5. The software programs described in this manual are copyrighted by Citizen Co., Ltd.
6. This product contains technology relating to strategic products controlled under the Foreign Exchange and Foreign Trade Control Law of Japan. The product should not be exported without obtaining the export license from the Ministry of International Trade and Industry in accordance with the above law.

Preface

This Programmer's Manual is one of a three-volume set that is provided with your Cincom L16/L20.

This manual contains information you need to program the L16/L20 for various lathing functions. The manual describes programming commands and program sequences with examples.

The other two volumes in the set are the Operator's Manual and the Maintenance Manual.

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

Maintenance Manual provides detailed information on L16/L20 repair and maintenance.

Information for Obtaining Customer Support

Before contacting Citizen for an inquiry, please confirm as many items listed below as possible to allow for a prompt and definite response.

No.	Item	Example	Check point
1	Machine type	L16	Aluminum nameplate on the back of the machine, the operation panel, the manual, etc.
2	Model No.	5M7	Red seal on the front of the machine, the cover of each manual, etc.
3	Machine No.	Q1234	Aluminum nameplate on the back of the machine, etc.
4	NC unit	Cincom SYSTEM M6	Chapter 3 "Machine Specifications" in the Operator's Manual, the operation panel, etc.
5	Delivery date	February 1997	
6	Machine paint color	Purplish two-tone color Your specified color	Appearance
7	Customization	Special workpiece unloading device	
8	Name of the manual and the number of the page containing the information concerning your inquiry	Page 10 in Chapter 5 of the Maintenance Manual	Manual: Operator's Manual, Programmer's Manual, or Maintenance Manual
9	Software version information	TYPE FCA635LC-AC11 SERIAL M6585460262 MAIN BND-358W000-A*0 PLCu 001-000 BOOT BND-362W001-A*D MMIs BND-359W000-A*0 MMIu 001-000 API BND-660W300-A*A MACRO 001-000 PARAMET 001-000 OS 001-000	On-screen display on the operation panel

For an inquiry about a problem, please check it with Chapter 5 "Troubleshooting" in the Maintenance Manual in advance.

(Blank page)

Programer's Manual

Chapter 0 Contents

Chapter 0 Contents 0-01

Chapter 1 Overview 1-01
1.1 About This Manual 1-1
1.2 Who Should Read This Manual 1-2

Chapter 2 Safety Precautions 2-01
2.1 Safety Signs 2-1
2.2 Emergency Stop Button 2-4
2.3 Safety Devices 2-5
2.4 General Precautions During Operation 2-10
2.5 Before Starting the Machine 2-12
2.6 Safety During Setup 2-13
2.7 Safety During Automatic Operation 2-14
2.8 Safety During Maintenance 2-15
2.9 Safety After Everyday Operation 2-15
2.10 Fire Prevention Practices 2-16
2.11 Handling Emergency Situations 2-18
2.11.1 Emergency Situations Requiring Evacuation 2-18
2.11.2 Power Failure 2-18
2.11.3 Resuming Work 2-18

Code No.	C-L51620 I III VII-570 2E1-0002 2E2-0001	Serial No.	M0135 ~, Q0008 ~ M0136 ~, Q0078 ~	Issue Date	1998.6

Chapter 3 Introduction to Programming 3-01
3.1 The NC Program 3-1
3.1.1 Difference between Cam Control and NC 3-2
3.2 Process of Program Creation 3-3
3.3 Basic Program Configuration 3-5
3.3.1 Machining Data (Screen Display) 3-8
3.4 Tool Layout Pattern 3-10
3.5 Drive Axis and Multi-Axis Control Group 3-11
3.5.1 Drive Axis 3-11
3.5.2 Fixed Points 3-12
3.5.3 Multi-Axis Control Group 3-13
3.5.4 Superimpose Control 3-14
3.6 Coordinates 3-15
3.6.1 Coordinate System 3-15
3.6.2 Diameter Designation 3-15
3.6.3 Signs and Coordinate Values 3-16
3.6.4 Coordinate System Setting 3-17
3.6.5 Absolute and Incremental Commands 3-18
3.7 Program and Axis Control Groups 3-19
3.7.1 Number of Axis Control Groups 3-19
3.7.2 Program-Instructed Axis Control Groups (Type VII) 3-19

Chapter 4 Command Codes 4-01
4.1 Summary of Addresses Used in NC Programs 4-1
4.2 T Functions (Tool Selection) 4-3
4.2.1 Tool Mounting and Machining Positions 4-4
4.2.2 T Code Commands and T Code Arguments 4-5
4.2.3 Drilling by Opposite Three-spindle Tool Post during Superimpose Control (Type III, VII) 4-10
4.2.4 Tool Selection for Back Machining (Type VII) 4-11
4.2.5 A Series of Pick-off Operations (Type VII) 4-12
4.3 G Functions (G Codes) 4-13
4.3.1 G Codes Table 4-13
4.3.2 G00 - Rapid Feed Positioning 4-14
4.3.3 Linear Interpolation (G01 for Cutting Feed on a Straight Line) 4-16
4.3.4 Circular Interpolation (G02, G03 in Cutting Feed) 4-18
4.3.5 G04 - Dwell Command 4-20

4.4 Special G Codes 1 and Special G Codes 2 4-21
4.4.1 Special G Code 1 Table 4-21
4.4.2 Special G Code 2 Table 4-21
4.4.3 Superimpose Control (G811, G810 - Outer/Inner Diameter Simultaneous Machining) (Type III, VII) 4-22
4.4.4 Synchronization Control (G821, G820 - Single Machining) (Type III, VII) 4-24
4.4.5 G999 - Last Program Execution 4-26
4.4.6 Phasing Profile Material (G899) 4-26
4.4.7 Superimpose Single Axis Control Group (G851, G850) (Type III, VII) 4-27
4.4.8 Spindle Synchronization Control (G114.1, G814, G113) (Optional) (Type III, VII) 4-28
4.4.9 Start Position Queuing (Type 1) (G115) 4-29
4.4.10 Start Position Queuing (Type 2) (G116) 4-29
4.4.11 Auxiliary Function Output during Axis Feed (G117) 4-30
4.4.12 End Position Specified Queuing (G149) 4-31
4.5 F Functions (F Codes) (Cutting Feed Rate) 4-32
4.5.1 Cutting Feed Rate (F) 4-32
4.5.2 Cutting Feed Rate for Back Machining (F) 4-33
4.6 M Functions (M Codes) 4-34
4.6.1 M Codes Table 4-34
4.6.2 Main Spindle Rotation and Stop (M03, M04, and M05) 4-37
4.6.3 Back Spindle Rotation and Stop (M23, M24, and M25) (Type VII) 4-38
4.6.4 Tool Spindle Rotation and Stop (M58, M59, and M60) 4-39
4.6.5 Front/Back Tool Spindle Rotation and Stop (M80, M81, M82) (Type III, VII) (Optional) 4-40
4.6.6 Back Spindle Product Separation (M32, M33, M34) (Type VII) 4-41
4.6.7 Spindle Chuck Open and Close (M06, M07) 4-47
4.6.8 Back Chuck Open and Close (M15, M16) (Type VII) 4-47
4.6.9 Opposite Tool Post Advance and Return (M140 and M141) (Type III, VII) 4-48
4.6.10 Y90Z Cut-Off Tool Breakage Detector (M51) (Optional) 4-50
4.6.11 Knock-out (M10, M11) (Type VII) 4-51
4.6.12 U10C Support (M10, M11) (Type I) 4-52
4.6.13 U11C Support (M10, M11) (Type III) 4-54
4.6.14 U51Z Stock Gripper (M13, M14) (Type I, III) 4-55
4.6.15 U31J Work Separator (M32, M33) (Type I, III) 4-56
4.6.16 U33J Workpiece Separator (M32, M33) (Type III) 4-57
4.6.17 Bar Feed (M54, M55) 4-58
4.6.18 Bar Feed Program Enable/Terminate (M08, M09) 4-59
4.7 S Functions (S Codes) 4-60
4.8 !(Exclamation) Function (Queuing Code) 4-61

Chapter 5 Programming Practice 5-01
5.1 Program Structure 5-1
5.1.1 Program Structure 5-3
5.1.2 Guidelines for Designing a Machining Program 5-4
5.2 Preparation Process 5-5
5.3 Cut-off and Ending Processes 5-7

5.4 Machining Processes 5-8
5.4.1 Front Turning Process 5-9
5.4.2 Back Turning Process....................5-10
5.4.3 Grooving Process 5-11
5.5 Center Hole Drilling 5-12
5.5.1 Determining Spindle Speed 5-12
5.5.2 Determining Program Point, Cutting Point, and Center Point....................5-12
5.5.3 Center Drilling 5-13
5.5.4 Drilling....................5-14
5.5.5 Drilling a Deep Hole 5-15
5.5.6 Thread Cutting with Tap and Die 5-16
5.5.7 Tapping Process Example....................5-17
5.6 Thread Cutting with the Threading Tool 5-18
5.6.1 Determining the Program Point and Cutting Point....................5-18
5.6.2 G92 - Thread Cutting Canned Cycle....................5-19
5.6.3 G92 - Thread Cutting 5-20
5.6.4 G92 - Taper Thread Cutting Canned Cycle 5-22
5.7 Boring (Coordinate System Shift)....................5-23
5.7.1 Determining the Program Point and Cutting Point....................5-23
5.7.2 Boring Tool Signs....................5-24
5.7.3 Adjusting for Diameter Direction Coordinate Shift and Canceling the Coordinate Shift Amount 5-25
5.7.4 Adjusting for Longitudinal Direction Coordinate Shift and Canceling the Coordinate Shift Amount 5-26
5.7.5 Adjusting for Diameter and Longitudinal Direction Coordinate Shift and Canceling the Coordinate Shift Amount... 5-27
5.8 Tool Offset (T Codes)....................5-29
5.8.1 Compensation Example....................5-30
5.9 Simultaneous Machining 5-32
5.9.1 Simultaneous Machining by Outer Diameter Cutting and Drilling....................5-35
5.9.2 Outer/Inner Diameter Simultaneous Machining (Boring) 5-36
5.10 Free Tool Layout Pattern (Holder Name = Free Tool)....................5-37
5.11 Using Subprograms....................5-38
5.11.1 A Subprogram Call Instruction....................5-38
5.11.2 Calling a subprogram from the main program 5-38
5.11.3 Example of Using a Subprogram....................5-39
5.12 Long Material Lathing (Optional) (Type I)....................5-40

Chapter 6 Programming Practice (Secondary Process)....................6-01
6.1 Secondary Process Guidelines 6-1
6.1.1 Turning Process and Secondary Process 6-1
6.1.2 Machining Process Order 6-1
6.1.3 The Coordinate System for Secondary Process....................6-2

6.2 Main Spindle Indexing 6-3
6.2.1 Main Spindle Indexing (M28, M20) 6-3
6.2.2 Back Spindle Indexing (M78, M79) (Optional) 6-4
6.2.3 Main Spindle C-Axis Indexing (M18) (Optional) 6-5
6.2.4 Back Spindle C Axis Indexing (M48) (Optional) 6-7
6.3 Calculating the Command Speed 6-8
6.3.1 Calculating the Spindle Speed Value 6-8
6.3.2 Calculating the Tool Feed Rate 6-9
6.4 Two-Surface Width Machining (Including Indexing) Program 6-10
6.5 Through-Hole Machining 6-12
6.6 Synchronous Tap Machining (G88, G80) (Optional) 6-13
6.7 Circular Interpolation 6-14
6.7.1 Plane Selection 6-14
6.7.2 Tool Diameter Compensation 6-15
6.8 Front Face Tool Spindle (Optional) 6-18
6.8.1 Front/Back Face Tool Spindle Rotation and Stop (M80, M81, and M82) 6-18
6.8.2 Synchronous Tapping with Front/Back Face Tool Spindle (G84, G80) 6-19
6.9 Differential Speed Rotary Tool Function (Optional) 6-20

Chapter 7 Programming Practice (Back Machining and Others) 7-01
7.1 Back Machining Program 7-1
7.2 Back Machining Coordinates 7-2
7.2.1 Back Spindle Rotation and Stop (M23, M24, M25) 7-3
7.3 Pick-Off and Ending Process 7-4
7.3.1 Pick-Off Process 7-6
7.4 Back Machining Process 7-7
7.4.1 Back Machining Product Separation (M33 and M34) 7-11
7.4.2 Cut-off and Product Separation (M33 and M34) 7-18
7.4.3 Back Spindle Indexing Command (M78, M79) (Optional) 7-20
7.5 Automatic Bar Loader (Optional) 7-21
7.6 Commands for Detecting Spindle Speed Changes 7-24
7.7 Constant Surface Speed Control Command 7-25
.8 Tool Nose Radius Compensation Function 7-27
7.8.1 Tools Subject to Tool Nose Radius Compensation and Virtual Tool Nose Numbers 7-28
.8.2 Basic Pattern of Tool Nose Radius Compensation G Code 7-29
Spindle Synchronization Control (G814, G114.1, G113) (Optional) 7-32
Synchronization Tap Machining (G88, G80) (Optional) 7-36
Interference Check Function 7-38
ack Face Tool Spindle (Optional) 7-39
Front/Back Face Tool Spindle Rotation and Stop (M80, M81, and M82) 7-39

Chapter 8 Program Example 8-01
8.1 Program Creation 8-1

Chapter 9 Cutting Conditions 9-01
9.1 Cutting Conditions 9-1
9.1.1 Determining Cutting Speed and Feed Rate 9-1
9.1.2 Thread Cutting Count with Tool 9-4
9.1.3 Secondary Process Cutting Conditions Table 9-5
9.2 Cutting Feed Rate Quick Reference Table 9-6

Chapter 10 Tooling 10-01

(Blank page)

Product code	C	–	L	5	1	6	2	0	I	III	VII	/	/	/	–	5	7	0

Document code	2	E	1	–	0	0	0	2
	2	E	2	–	0	0	0	1

Chapter 1 Overview

1.1 About This Manual 1-1
1.2 Who Should Read This Manual 1-1

(Blank page)

1.1 About This Manual

The purpose of this manual is to provide programmers with the information required for programming the Cincom L16 and L20 machines to perform a variety of basic lathing functions.

This manual describes how to handle, or program the machine to get the most out of it. Before operating the machine, be sure to understand the contents of this manual to create NC programs for safe and correct machine operation.

This manual contains the following chapters:

Chapter 1 Overview

This chapter describes the purpose and organization of this manual.

Chapter 2 Safety Precautions

This chapter describes safety precautions to be heeded while programming, setting up, operating, and maintaining the machine.

This chapter also includes information on machine safety devices and procedures for handling an emergency situation such as a fire at your factory.

Chapter 3 Introduction to Programming

This chapter provides basic knowledge of NC programming.

Chapter 4 Command Codes

This chapter details the instruction commands required for NC programming.

Chapter 5 Programming Practice

This chapter provides simple programming examples to help you learn basic NC programming.

Chapter 6 Programming Practice (Secondary Process)

This chapter provides advanced application examples to help you learn secondary-process programming.

Chapter 7 Programming Practice (Back Machining)

This chapter provides advanced application examples to help you learn back-machining programming.

Chapter 8 Program Example

This chapter provides sample NC programs.

Chapter 9 Cutting Conditions

This chapter describes a variety of cutting conditions required for machining.

Chapter 10 Tooling

This chapter lists the various tooling systems used for the L16 and L20 machines, showing their specifications and dimensions.

1.2 Who Should Read This Manual

This manual is intended for all programmers for producing the component on the L16/20 machine. Programmers must thoroughly read and understand this manual in order to create machining programs for safe and efficient operation.

(Blank page)

Product code	C	–	L	5	1	6	2	0	I	III	VII	/	/	/	–	5	7	0
Document code	2	E	1	–	0	1	0	2										
	2	E	2	–	0	1	0	1										

Chapter 2 Safety Precautions

2.1 Safety Signs 2-1
2.2 Emergency Stop Button 2-4
2.3 Safety Devices 2-5
2.4 General Precautions During Operation 2-10
2.5 Before Starting the Machine 2-12
2.6 Safety During Setup 2-13
2.7 Safety During Automatic Operation 2-14
2.8 Safety During Maintenance 2-15
2.9 Safety After Everyday Operation 2-15
2.10 Fire Prevention Practices 2-16
2.11 Handling Emergency Situations 2-18
2.11.1 Emergency Situations Requiring Evacuation 2-18
2.11.2 Power Failure 2-18
2.11.3 Resuming Work 2-18

(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.

Safety signs appear in this manual and on the machine. All safety signs are identified by the words DANGER, WARNING, or CAUTION. These words signify the following:

DANGER ; alerts you to an imminently hazardous situation which, if not avoided, will result in death or serious personal injury.

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

CAUTION ; indicates a potentially hazardous situation which, if not avoided, may result in minor or moderate personal injury and/or possible damage to the machine and its components.

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

⚠ WARNING

Do not remove or hide any safety sign (warning label). If it is peeling, contact that division of Citizen Watch Co., Ltd. which is specified on the cover of this manual.

Location of Safety Signs

Figure 2.1-1 shows the location of the safety signs and Figure 2.1-2 shows the contents of the safety signs.

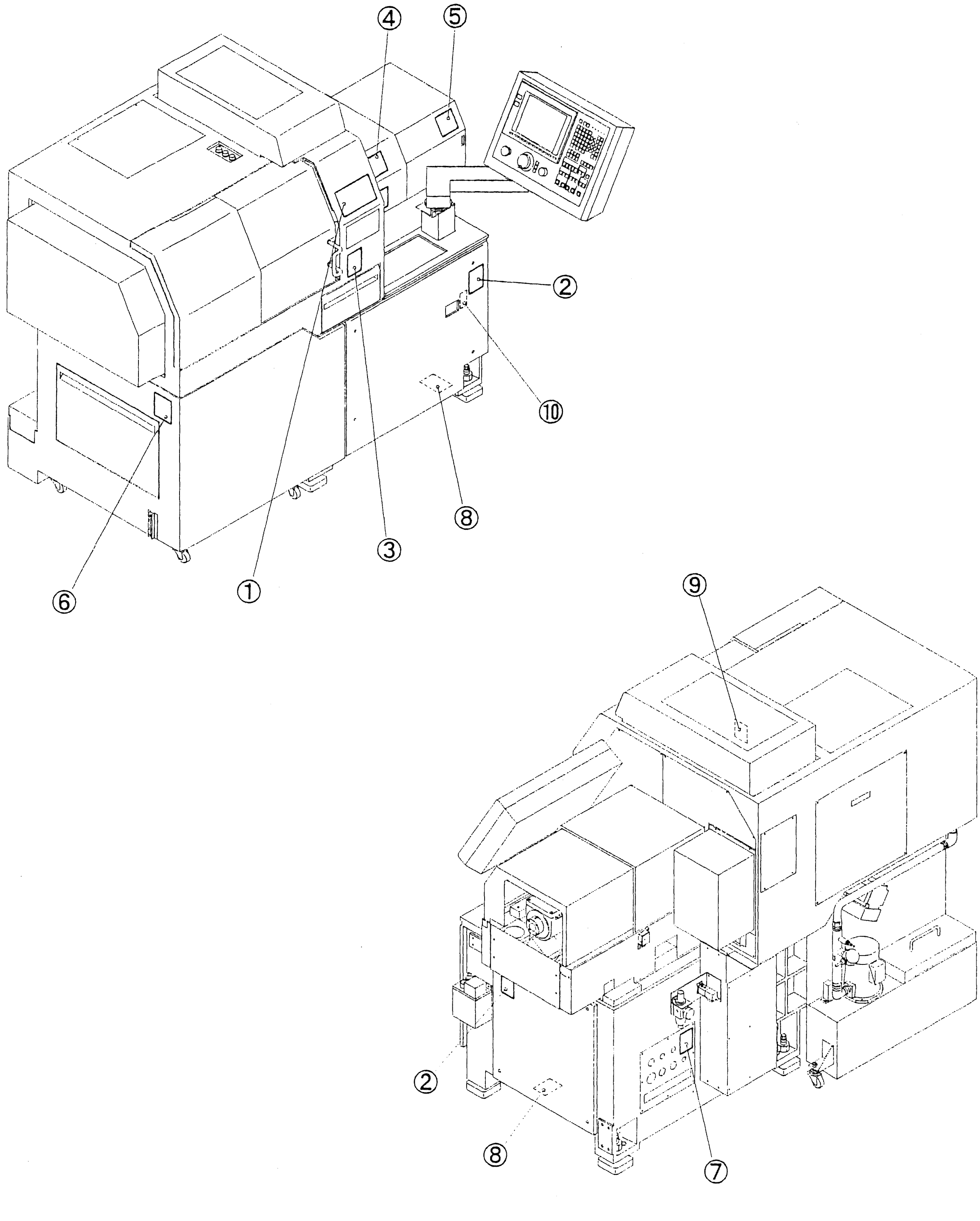


Figure 2.1-1 Location of Safety Signs

Figure 2.1-2 Contents of Safety Signs

2.2 Emergency Stop Button

The red emergency stop button is located on the operation panel. When there is an emergency situation such as fire, power failure, earthquake, or lightning or if you need to evacuate at once, press this button to stop the operation immediately before you leave the work site. Note, however, that pressing these emergency stop buttons during machine operation might damage a tool as well as the product being processed. The machine will display the alarm message EX052 "Emergency button alarm. Emergency Button is pressed." To reset the emergency stop status, turn the pressed button clockwise. The procedure for gradually stopping the machine operation is given in "2.11 Handling Emergency Situations".

Figure 2.2-1 shows the location of emergency stop button.

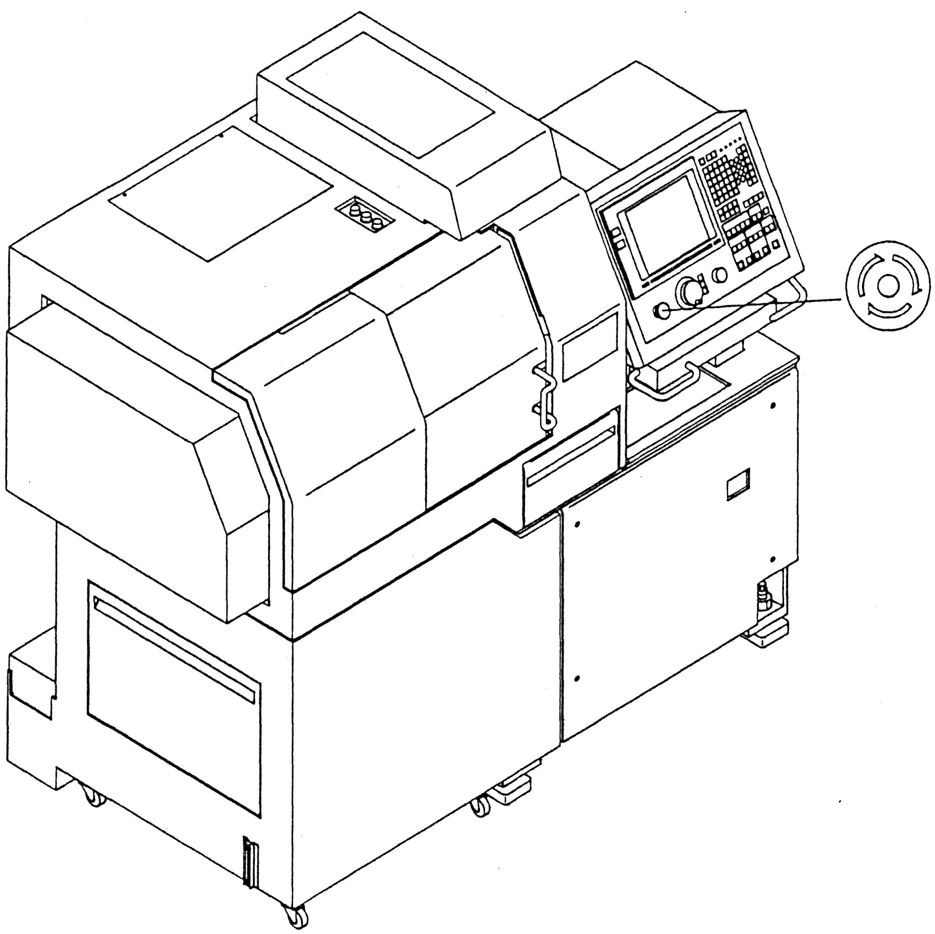


Figure 2.2-1. Location of Emergency Stop Button

2.3 Safety Devices

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 when there is 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 you operate the machine continuously and there is possibility of reduced operator attention.
- When the duties of the operator extend beyond this machine.
- When you operate the machine during the night shift.
- To reduce the possibility of accidents.

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

IMPORTANT:

No safety devices provide complete safety against accidents and hazards.

Be sure to follow the precautions described in sections after 2.4.

Figures 2.3-1 and 2.3-2 show the locations of the safety devices on the machine.

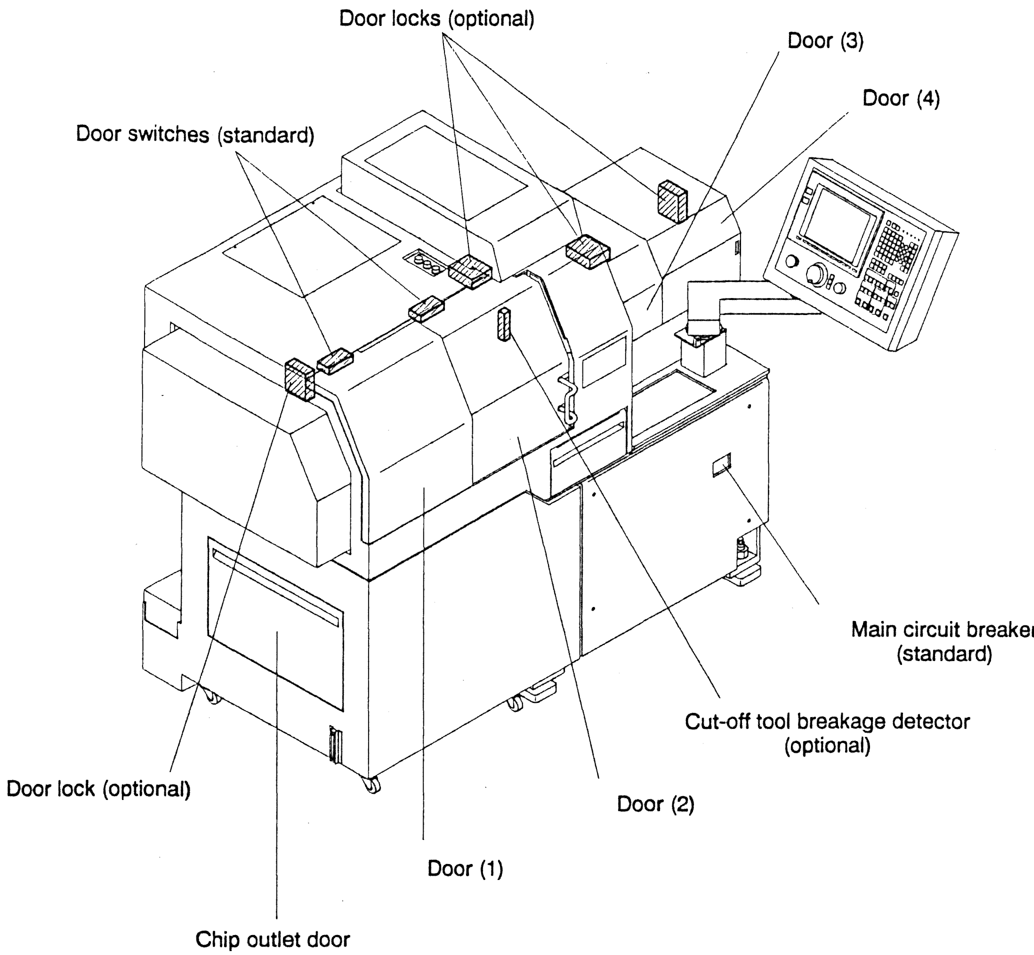


Figure 2.3-1. Location of Safety Devices (Front side)

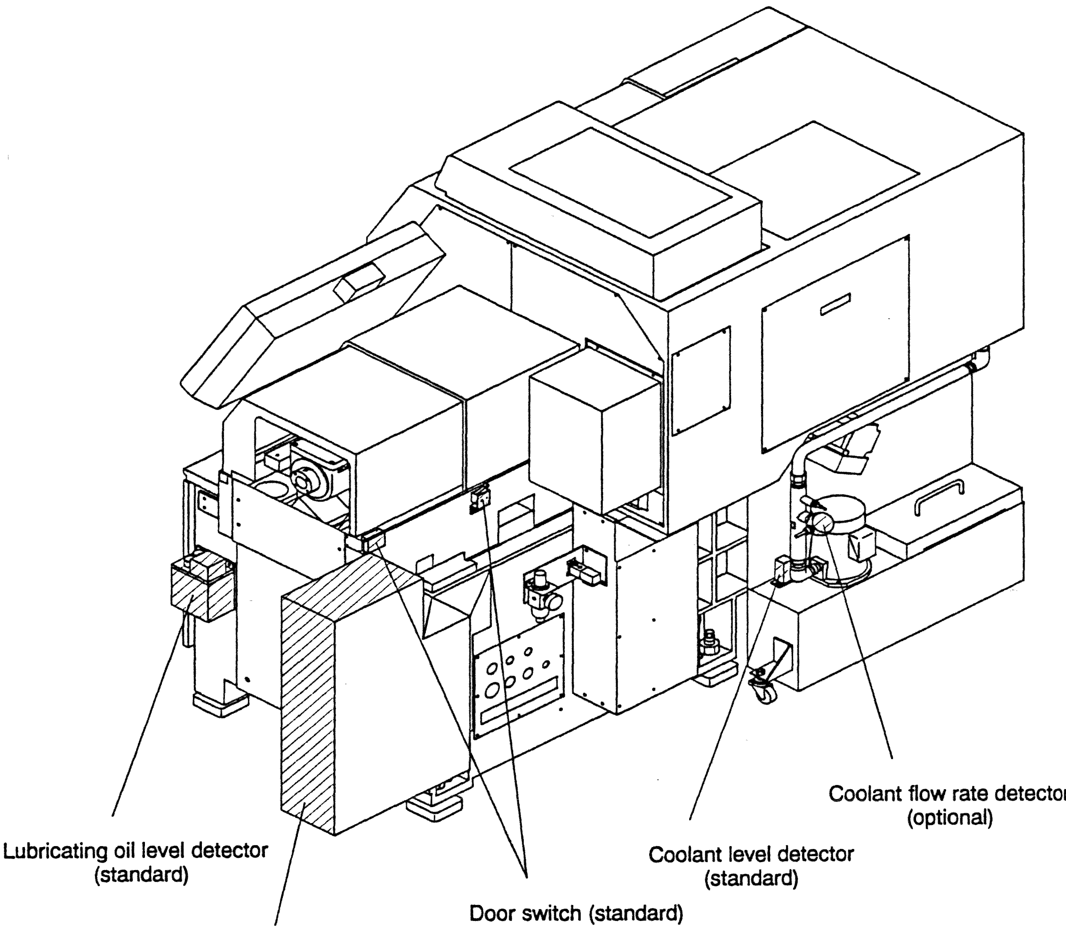


Figure 2.3-2 Location of Safety Devices (Rear side)

The following are detailed descriptions of the safety devices:

Door locks (optional)

Door locks prevent any person from opening the front left (cutting side) doors (1) and (2) and front right (main spindle side) doors (3) and (4) during machine operation. You will only be able to open these doors when the machine is stopped. (See Figure 2.3-1.)

Door locks also serve as door switches. Therefore, if the door locks are installed, the standard door switches are not used.

Door switches (standard)

These door switches disable machine operation while the front left (cutting side) doors (1) and (2) and the front right (main spindle side) doors (3) and (4) are open. (See Figure 2.3-1and 2.3-2.) Unless all the doors are closed, you cannot operate the machine

In the Handle Feed or Preparation mode, however, operation (except cut-off machining) is 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.

Fire extinguisher (optional)

The automatic fire extinguisher starts putting out a fire when the sensor installed in the machine detects a condition matching or exceeding the prescribed setting.

> ⚠ WARNING
>
> To prevent fires from being caused, be sure to follow the precautions below:
> - Before operating the L16/L20 machine, be sure to read the separately prepared "Fire Extinguisher Instruction Manual" for operation, maintenance, and inspection.
> - If the machine is operated with the fire extinguisher removed, always follow the fire prevention practices listed in Section 2.10 and take appropriate measures for fire prevention to replace the fire extinguisher.
> - The carbon dioxide fire extinguisher standard with the machine works to put out a fire with the doors (1) and (2) on the front left side (cutting side) and the chip door both closed (sealed). Therefore, be sure to keep the cutting room doors and the chip door both closed during operation of the machine.
> - The carbon dioxide fire extinguisher standard with the machine cannot put out the combustion (fire) of certain materials such as magnesium, titanium, zirconium, and sodium alloys. If the machine is used to cut these materials, strictly observe the following precautions:
>   - Avoid unattended operation.
>   - Prepare a fire extinguisher dedicated to metal fires near the machine.
>
>   Never use water to fight such fires.

Main circuit breaker (standard)

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

Cut-off tool breakage detector (optional)

A cut-off tool is very easily damaged. If you continue to operate the machine with a damaged cut-off tool, 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 EX 116 "Tool bit breakage alarm. Cut-off tool is broken." is displayed.

Coolant flow rate detector (optional)

This device is installed in the midsection of the coolant supply pipe and detects the change of the coolant flow rate. When the flow rate gets lower than the specified value, the alarm message EX114 "Coolant alarm. Coolant oil is empty or nozzle is jammed." is displayed and the operation of the machine is automatically stopped to prevent a fire hazard.

Coolant level detector (standard)

This device is installed in the coolant tank and detects the height of the coolant level. When the coolant level gets lower than the limit, the alarm message EX203 "Coolant oil empty. Coolant oil is empty. Supply the oil." is displayed and the operation of the machine is automatically stopped after completing one cycle to prevent a fire hazard.

Lubricating oil level detector (standard)

This device is installed in the central lubricating oil unit and detects the level of the lubricating oil. When the oil level gets lower than the limit, the alarm message EX202 "Lubrication oil alarm. Lubrication oil is empty. Supply the oil" is displayed and the operation of the machine is automatically stopped after completing one cycle to prevent machine damage.

The following software functions are also provided for safety:

Spindle speed change detection (standard)

This function detects excessive changes in spindle speed to prevent machine damage. The alarm message EX119 "Main spindle speed fluctuation alarm." or EX127 "Back spindle speed fluctuation alarm." is displayed when it detects a change of more than 10% from the specified speed.

Note, however, that this function must be turned off when the constant surface speed control function is used and during tap and die machining (see "7.6 Commands for Detecting Spindle Speed Changes" and "7.7 Constant Surface Speed Control Command" in the Programmer's Manual).

Interference check (standard)

This function checks for interference between spindles, guide bushing, and tool posts of the machine. When the function determines the possibility of interference during machine operation, the NC alarm message M03 "COLLISION ALARM" 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

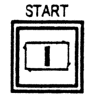
(see "5.4.1 Recovery from Interference Check Alarm" in the Maintenance Manual).

2.4 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 put your fingers or hands into any moving part of the machine during machine operation. When the 

key LED on the operation panel is on, 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 high voltage parts 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. Turn off 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, return all machine moving parts to their return positions, and remove all materials in progress from the machine.
- 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.

⚠ 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.

⚠ 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 Appendix A Relocating the NC Machine in the 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.
- Clean the machine regularly to remove any chips and debris from the cutting area and the chip receiving area.
- Remove stray chips from the coolant tank as required, depending on cutting condition and type of material being machined.

2.5 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 disconnect the main power to the machine.
- Fire prevention procedures (see "2.10 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 rings, watches, jewelry, neckties, and all loose items in your shirt pockets.
- 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 on the machine 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.

2.6 Safety During Setup

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 Cincom 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.

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.

2.7 Safety During Automatic Operation

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 stray chips that get caught in the machine, 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.

2.8 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, especially when any safety device is disengaged.
- 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 main circuit breaker and 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.

2.9 Safety After Everyday Operation

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

2.10 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 stray 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.
- Check to see that there are no coolant mist leaks.
- 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 can mist or vaporize or 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 coolant filter is blocked, the coolant discharge may be insufficient.
- When the coolant recommended by the manufacturer is not used and a flammable coolant is used instead; or when the cooling effect of the coolant is inferior.
- When the room temperature is abnormally high, the coolant may vaporize.

The following situation is especially hazardous. Stop the machine operation immediately, and contact the Citizen Service Center.

- 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.

2.11 Handling Emergency Situations

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

2.11.1 Emergency Situations Requiring Evacuation

When evacuation is required in the case of emergency situations, such as fire, earthquake, or lightning, perform these procedures.

- 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.

2.11.2 Power Failure

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

2.11.3 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 Operator's Manual.

(Blank page)

Product code	C	-	L	5	1	6	2	0	I	III	VII	/	/	/	-	5	7	0
Document code	0	E	1	-	0	2	0	2										

Chapter 3 Introduction to Programming

3.1 The NC Program 3-1
3.1.1 Difference between Cam Control and NC 3-2
3.2 Process of Program Creation 3-3
3.3 Basic Program Configuration 3-5
3.3.1 Machining Data (Screen Display) 3-8
3.4 Tool Layout Pattern 3-10
3.5 Drive Axis and Multi-Axis Control Group 3-11
3.5.1 Drive Axis 3-11
3.5.2 Fixed Points 3-12
3.5.3 Multi-Axis Control Group 3-13
3.5.4 Superimpose Control 3-14
3.6 Coordinates 3-15
3.6.1 Coordinate System 3-15
3.6.2 Diameter Designation 3-15
3.6.3 Signs and Coordinate Values 3-16
3.6.4 Coordinate System Setting 3-17
3.6.5 Absolute and Incremental Commands 3-18
3.7 Program and Axis Control Groups 3-19
3.7.1 Number of Axis Control Groups 3-19
3.7.2 Program-Instructed Axis Control Groups (Type VII) 3-19

(Blank page)

3.1 The NC Program

The Numerical Control (NC) program is a group of commands designed to operate the machine using the NC unit. These commands are simple alphanumeric codes designed to be readable by the NC unit as well as easy to use by the programmer.

NUMERICAL CONTROL

What is NC (Numerical control)?

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

PROGRAM

What is program?

Instructions for controlling the machine according to the predetermined procedures.

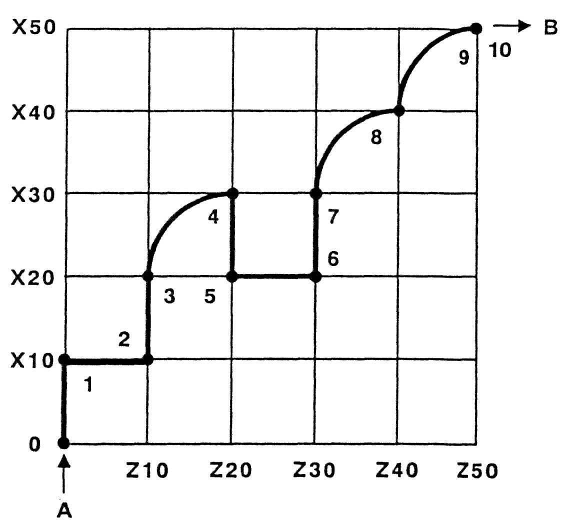
Before reading ahead for details on NC programs, try the following question as an example showing that NC programming is easy.

Q: Use the following four commands to complete the code to move from A to B in the following figure. (See the next page for the answer of this question.)

1)	**G01**	.... Move straight to OO.
2)	**G02**	.... Move circularly to OO.
3)	**M03**	.... Turn the start switch ON.
4)	**M02**	.... Turn the goal switch OFF.

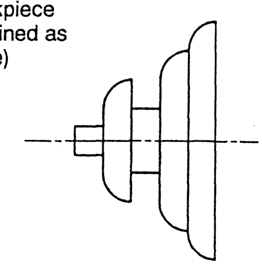
Codes

0 **M03** (Turn the start switch ON)
1 **G01 X10 Z0** (Straight move to X = 10, Z = 0)
2 **G01 X10 Z10** (Straight move to X = 10, Z = 10)
3 ( ) **X( ) Z10** (Straight move to X = 20, Z = 10)
4 **G02 X( ) Z20** (Circular move to X = 30, Z = 20)
5 ( ) **X( ) Z( )** (Straight move to X = 20, Z = 20)
6 ( ) **X( ) Z( )** (**from X = **, Z = **)
7 ( ) **X( ) Z( )** (**from X = **, Z = **)
8 ( ) **X( ) Z( )** (**from X = **, Z = **)
9 ( ) **X( ) Z( )** (**from X = **, Z = **)
10 ( )



The NC program is almost the same as codes above.
(The answer to this above question is given on page 3-18.)

(Workpiece machined as above)

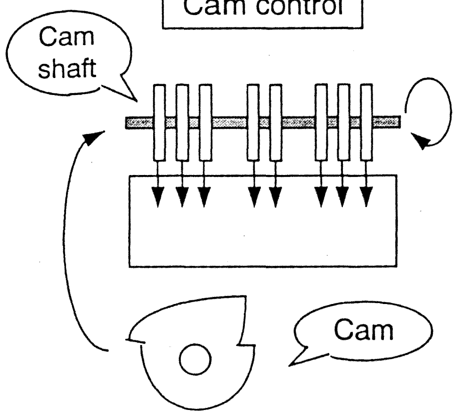


3.1.1 Difference between Cam Control and NC

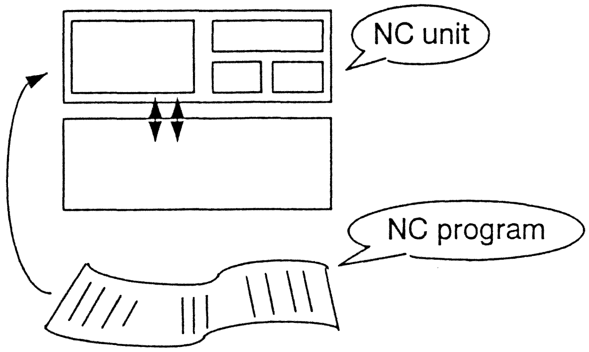
What are the differences between cam control and NC?

Where is the brain of the machine ?

Cam control



Numerical control (NC)



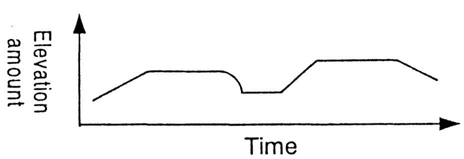
How do you operate the machine ?

Cam control — Mechanical

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

Cam control — Analog

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



NC — Electric

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

NC — Digital

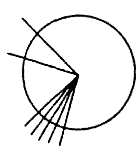
The NC program is created and the operation commands are constructed step by step according to the process.

Use chuck
↓
Rotate the spindle (in 2,000r.p.m.)
↓
Select front turning tool and perform ...
↓
⋮

What is a program like ?

Cam control — 1 cycle is angular

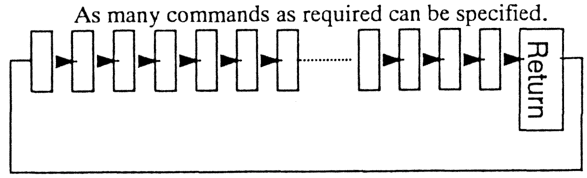
The entire process must be completed within 360°.



- Limitation to complex and long-time machining.

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.

NC — Software

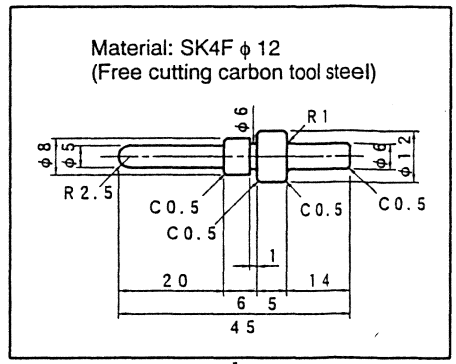
Just type the commands according to the designated manner. The program can easily be rewritten.

- 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

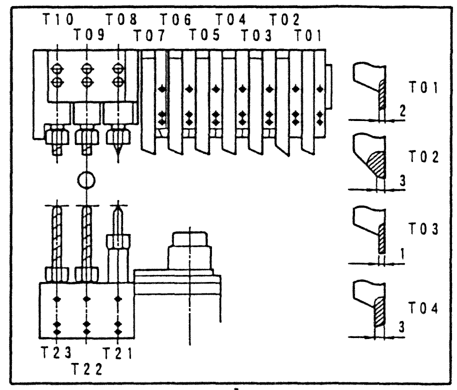
① Workpiece Drawing

- The drawing is the basis for an NC program.



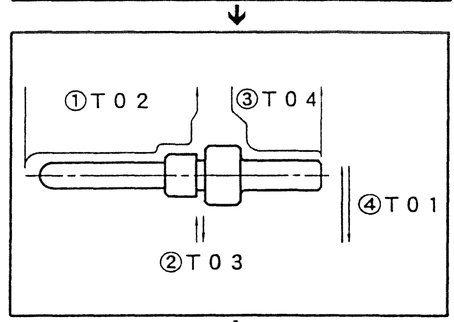
② Creating the Tool Layout

- Select the holder to be used.
- Determine the tool layout.
- Describe the shape of the tool, the dimensions of tool width, and the tool number.



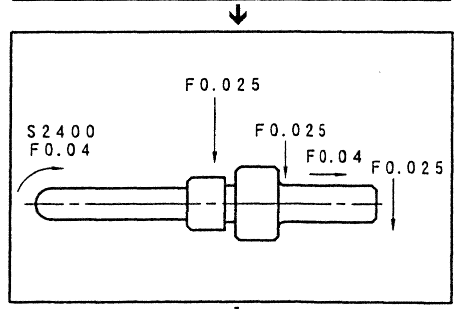
③ Creating the Machining Layout

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



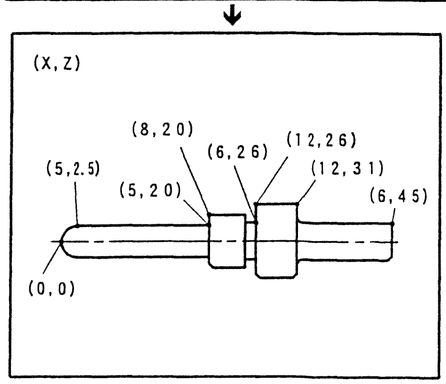
④ Setting the Spindle Speed and Feed Rate

- Select the proper cutting conditions.



⑤ Reading the Coordinate Values

- Read the in-feed amount (X) and longitudinal feed amount (Z) from the drawing.



⑥ Coding the Program

- Specify the type of the tool to be used, range of cutting, and machining conditions for cutting on paper according to the rules.

```
O0002
$1
G50 Z-0.1
(M52)
M06
G00 Z-0.6 S1 = 2400 M03
G821
N0112 T1200
X13.0 Z0
```

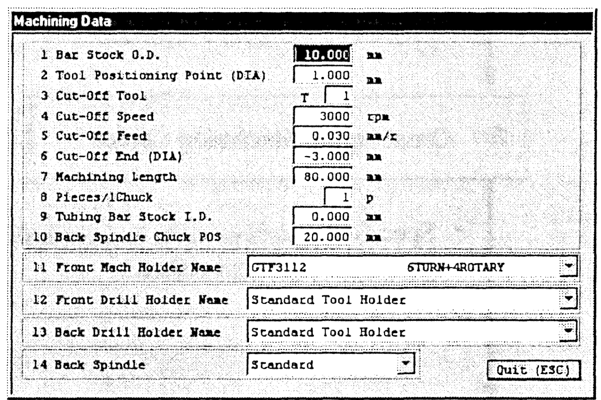
```
$2
G821
G820
G999
N999
M02
M99
%
```

⑦ Entering and Saving the Program

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

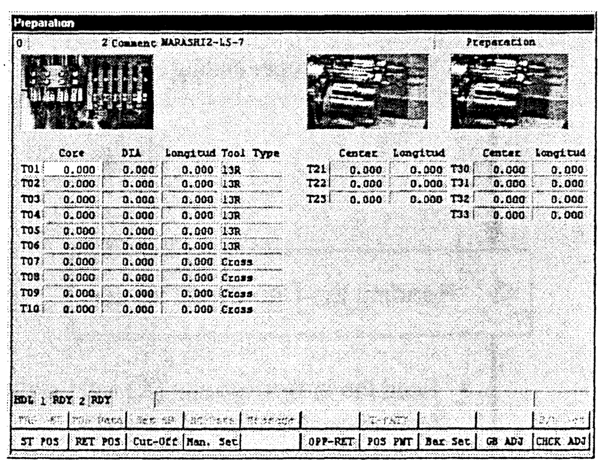
⑧ Entering the Machining Data

- Enter the holder name to be used, material diameter, and machining length as data into the NC in advance.



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 Basic Program Configuration

The program consists of a main program body and program data.In the main program body, operation commands for a machine are arranged in the order of each machining process.

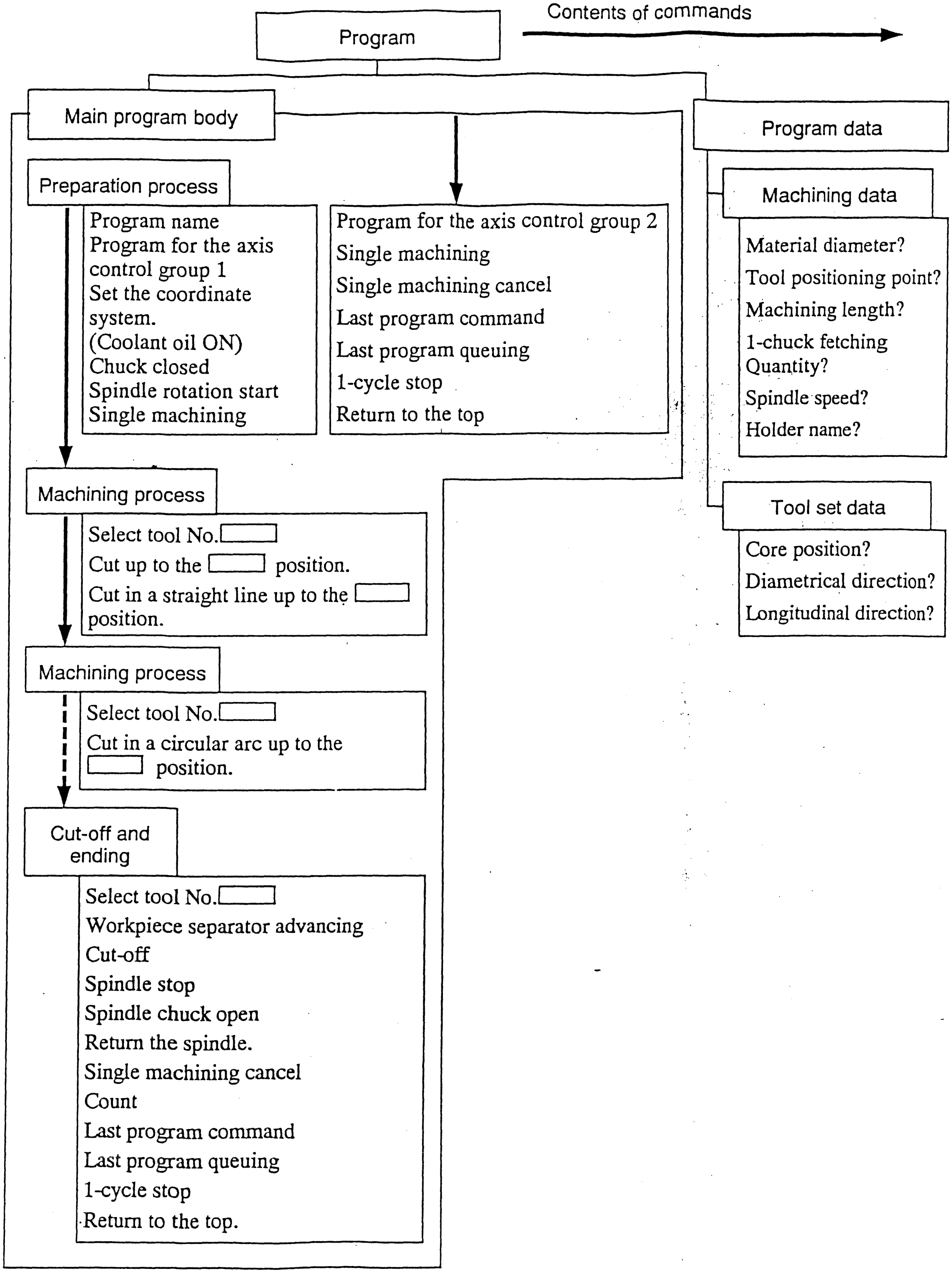
This is usually called a program. In the program data, the preconditions under which a program operates are arranged.

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

The diagram on the next page summarizes the contents of commands used in the program body and for program data.

The diagram on the page that follows lists the actually used command strings and screen displays. The machine operates under control of the program configured with the program body and program data as illustrated in the diagrams.

Determine the holder name to be used as one of machining data items.


Program
Contents of commands
Main program body
Program data
Preparation process
Program name
Program for the axis control group 1
Set the coordinate system.
(Coolant oil ON)
Chuck closed
Spindle rotation start
Single machining
Program for the axis control group 2
Single machining
Single machining cancel
Last program command
Last program queuing
1-cycle stop
Return to the top
Machining data
Material diameter?
Tool positioning point?
Machining length?
1-chuck fetching Quantity?
Spindle speed?
Holder name?
Machining process
Select tool No.
Cut up to the position.
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
Select tool No.
Workpiece separator advancing
Cut-off
Spindle stop
Spindle chuck open
Return the spindle.
Single machining cancel
Count
Last program command
Last program queuing
1-cycle stop
Return to the top.

Screen display → Program

Main program body

Preparation process

```
Oxxxx
$1
G50 Z0
(M52)
M06
S1 = 2000 M03
G821
```

```
$2
G821
G820
G999
N999
M02
M99
%
```

Machining process

```
T □
G01 X □ F □
```

Machining process

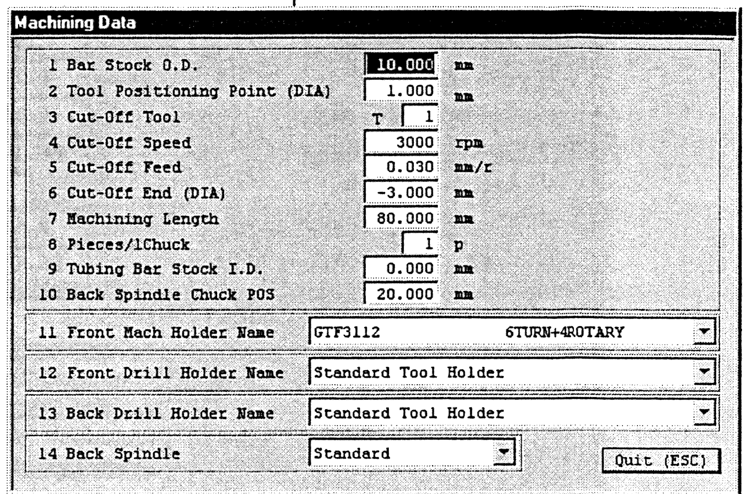
```
T □
G02 X □ Z □ R □ F
```

Cut-off and ending processes

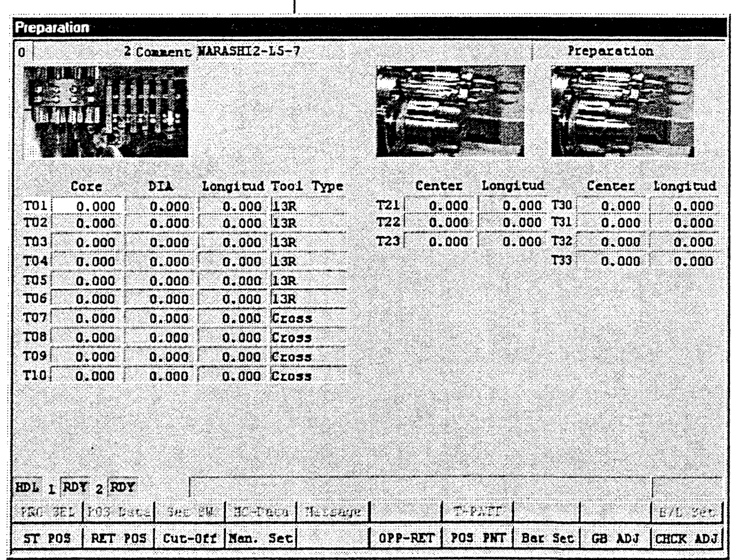
```
T100
M32
G01 X-3.0 F □
M05
M07
G00 Z0 T00
G820
M56
G999
N999
M02
M99
%
```

Program data

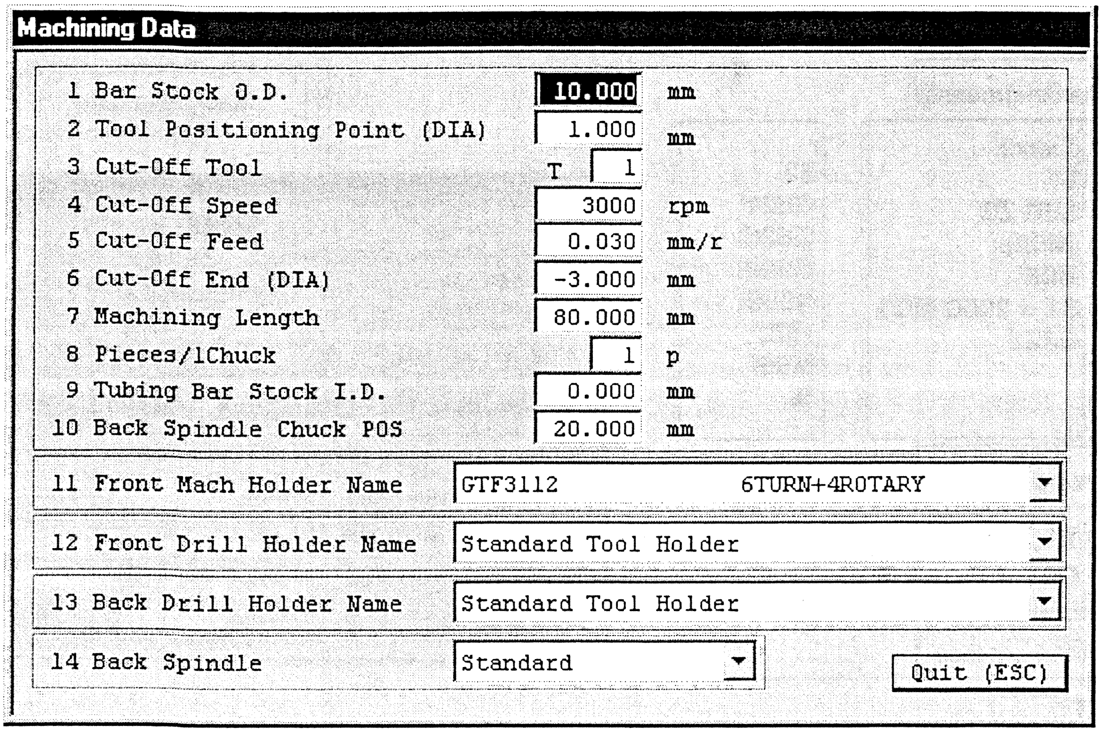
Machining data



Tool set data



3.3.1 Machining Data (Screen Display)



1 Bar Stock O.D.

Enter the outside diameter of materials to be machined.

2 Tool Positioning Point (DIA)

The tool positioning point is the point at which a tool is positioned when selected from among tools T01 to T10. 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 T10 and to which the tool escapes when another tool is selected.

3 Cut-Off Tool

In this machine, a cut-off tool is mounted in T0100. Therefore, enter T1.

4 Cut-Off Speed

The tip of materials is cut at this speed during tool setting.

5 Cut-Off Feed

The tip of materials is cut at this feed rate during tool setting.

6 Cut-Off End (DIA)

The tip of materials is cut to the end position of this X-axis position during tool setting.

This position is also used as the start position of an X-axis when the program starts.

7 Machining Length

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 step① or② described above (as required in the program).

8 Pieces / 1Chuck

Enter the number of product to be machined in the program.

9 Tubing Bar Stock I.D.

If the machined is a pipe material, enter the inner diameter.

If cut-off machining is executed in the preparation mode with this value entered, the tool advances to the position "the pipe material inner diameter - 3.0 mm" in cutting feed, then advances to the cut-off end position in rapid feed.

Example: If the pipe material inner diameter is 10.0 mm and the cut-off end position is -3.0, the tool is fed to X7.0 in cutting feed and then to X-3.0 in rapid feed.

10 Back Spindle Chuck POS

Enter the value of workpiece protruding from the back spindle.

11 Front Mach Holder Name

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

12 Front Drill Holder Name

Enter the name of the front drilling holder attached to the machine.

13 Back Drill Holder Name

Enter the name of the back drilling holder

14 Back Spindle

Enter the type of the back spindle attached to the machine by selecting Standard, With Basket or With Support.

3.4 Tool Layout Pattern

A more rational layout can be selected according to the number of tools to be used.

For more information, see Chapter 10 "Tooling."

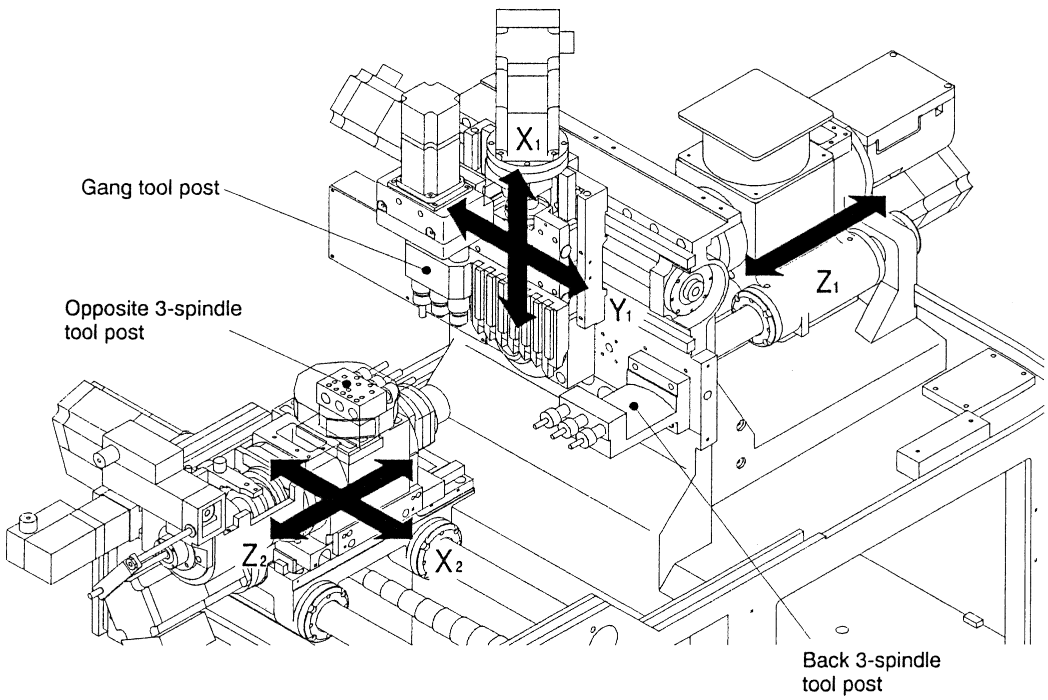
The type of the tool holder you have selected is displayed on the Machining Data screen. For information on holder name selection, see the relevant chapter for operation.

3.5 Drive Axis and Multi-Axis Control Group

3.5.1 Drive Axis

■ Five-axes control specification (Type VII)

- The gang tool post (vertical tools T01 to T10 or T01 to T11) moves back and forth. (Y1 axis)
- The gang tool post moves up and down. (X1 axis)
- The headstook horizontally moves left or right. (Z1 axis)
- The opposite tool post (inner diameter machining tools T21 to T23 and back spindle T30) moves back and forth. (X2 axis)
- The opposite tool post moves left or right. (Z2 axis)

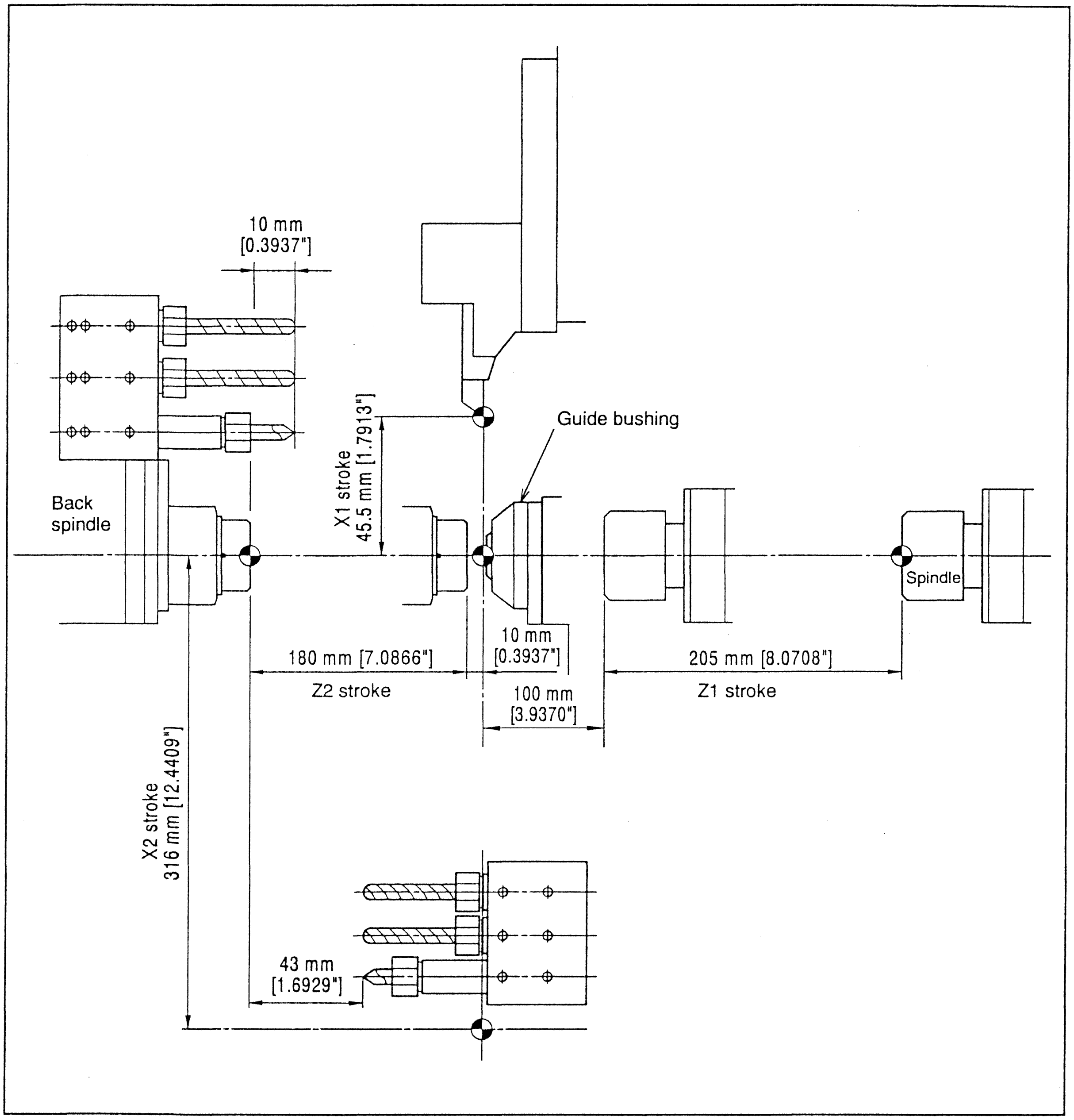


Type / Axis	Type I	Type III	Type VII
X1			
Z1			
Y1			
C1			
X2			
Z2			
C2			
A1	Main chuck	Main chuck	Main chuck
A2			Back chuck
A3			Knock-out
A4	Separator	Separator	
A5	Stock gripper	Stock gripper	
A6	Long-workpiece device		
A7	Bar loader	Bar loader	Bar loader

S1 = Main spindle
S2 = Back spindle
S3 = Gang tool rotary tool
S4 = Opposite rotary tool (option)
S5 = Rotary guide bush

3.5.2 Fixed Points

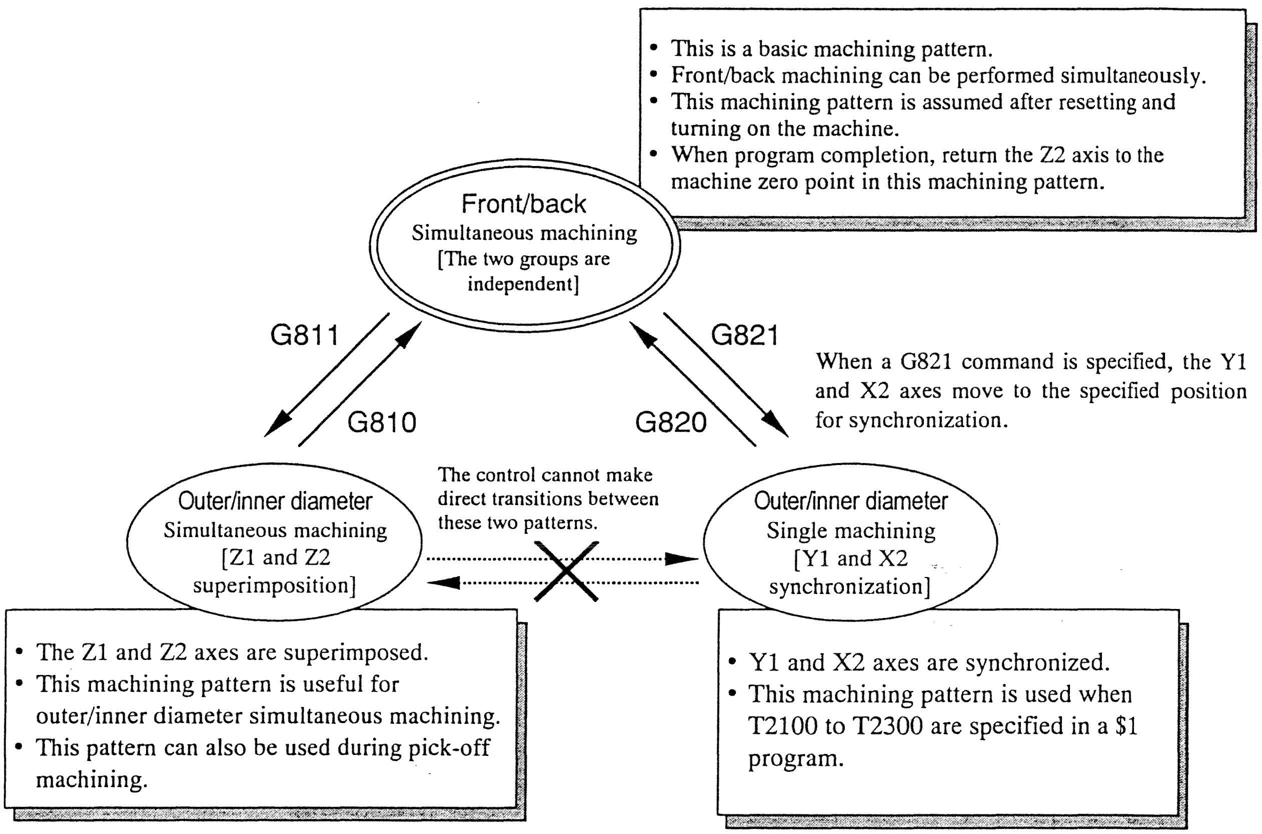
The fixed points shown in the figure below are set by each machining pattern. These fixed points vary according to a change in the pattern.



*: Machine type of the fugure shown above is 5M7.

3.5.3 Multi-Axis Control Group

Five axes move independently. These axes are functionally classified into two groups.
As a basic function, the X1, Y1, and Z1 axes are called the axis control group 1. It is symbolized in the program as "$1". As an additional function, X2 and Z2 axes. It is symbolized in the program as "$2". In general, the two groups move independently. However, the Z1 and Z2 axes can be superimposed by the simultaneous machining of outer and inner diameters or the X1 and X2 axes can be synchronized by the Single machining command. (See the figure below.)



3.5.4 Superimpose Control

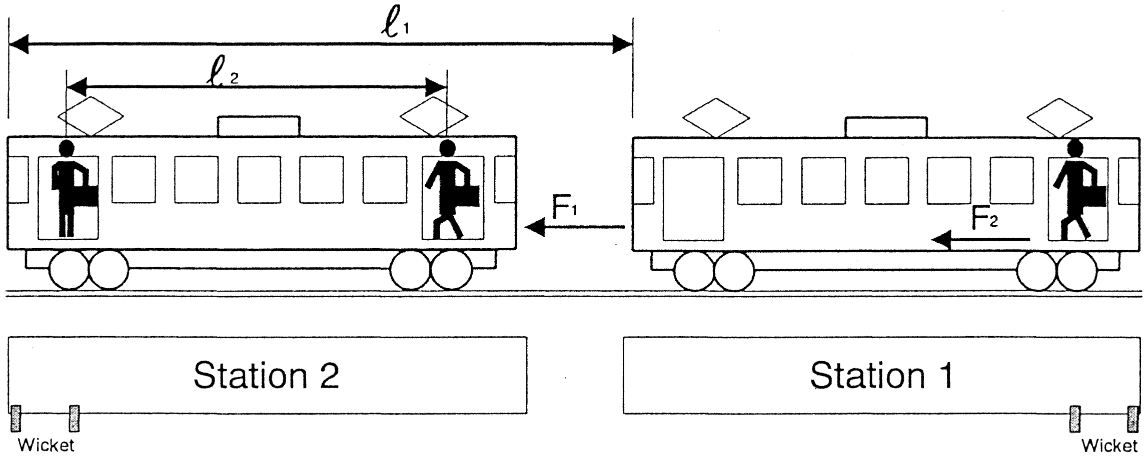
This section describes superimpose control using a precise example.

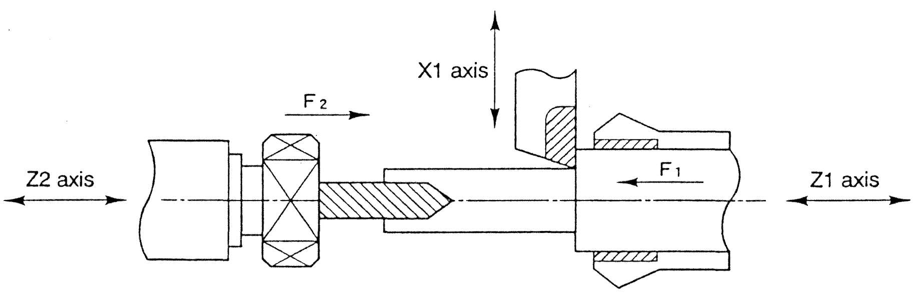
Mr. A got on an electric car in the rear. He got off at the next station. However, the wicket of the next station is located in front. He walked in the car and got off at the next station in front.

The relation between the car and Mr. A's task is described below.

Mr. A did a task of distance "ℓ_2" at a speed of "F_2" while he moved for distance of car task "ℓ_1" at a speed of "F_1". As a result, he could accomplish a task of "$\ell_1 + \ell_2$." The car corresponds to the Z1 axis as the reference axis.

Mr. A (Z2 axis) did a task while he gets on a car (Z1 axis). This is called the superimpose control.





Superimpose control (G811):

- The Z2 axis moves in synchronism with the Z1 axis when a move command for the Z1 axis is specified without a move command for the Z2 axis.
- Only the Z2 axis moves when a move command for the Z2 axis is specified without a move command for the Z1 axis.
- The Z2 axis is superimposed (put) on the Z1 axis for movement when move commands for both the Z2 and Z1 axes are specified.

In the figure above,

if F1 = F2, the Z2 axis looks as if it is in the stop state.

For F1 < F2, the Z2 axis advances in F2 - F1.

For F1 > F2, the Z2 axis retreats.

In the program, a queuing command is used to establish the timing of the Z1 and Z2 axes.

3.6 Coordinates

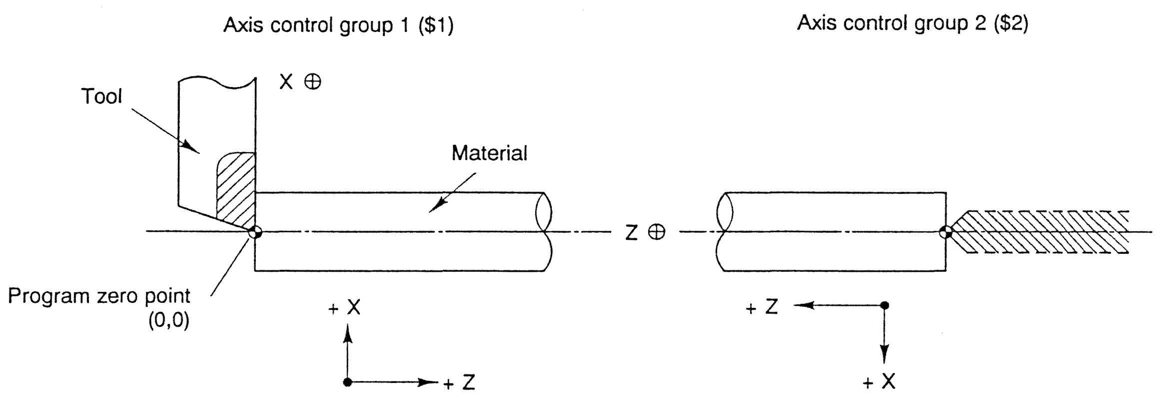
3.6.1 Coordinate System

It is necessary to distinguish the coordinate axis and drive axis. For example, the X axis in a coordinate axis indicates the diametrical direction of the selected tool regardless of a front X1 axis or back X2 axis.

For the Z axis in a coordinate axis, the direction in which the tool moves to the right with the materials fixed, is assumed to be +Z. However, as a matter of fact, it is the spindle that moves. The opposite direction (left direction) is equivalent to +Z. Conceptually, it is same for the axis control group 2 as described above.

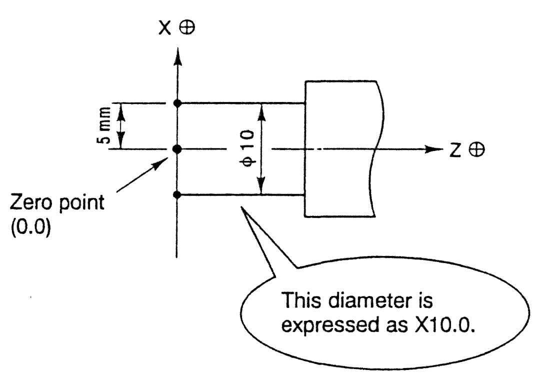
Using coordinate axes X and Z makes easy to understand the machining. Therefore, it is used for programming.

X axis	:	Diameter direction	The X axis is for up and down movement of the tool in the direction of the diameter.
Z axis	:	Longitudinal direction	The Z axis is for moving horizontally left or right (longitudinal direction) with the spindle.



3.6.2 Diameter Designation

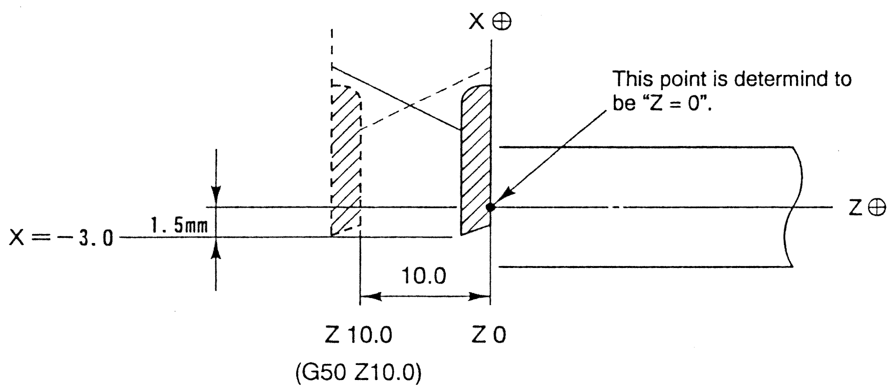
The X coordinate value must be expressed by the diameter designation.

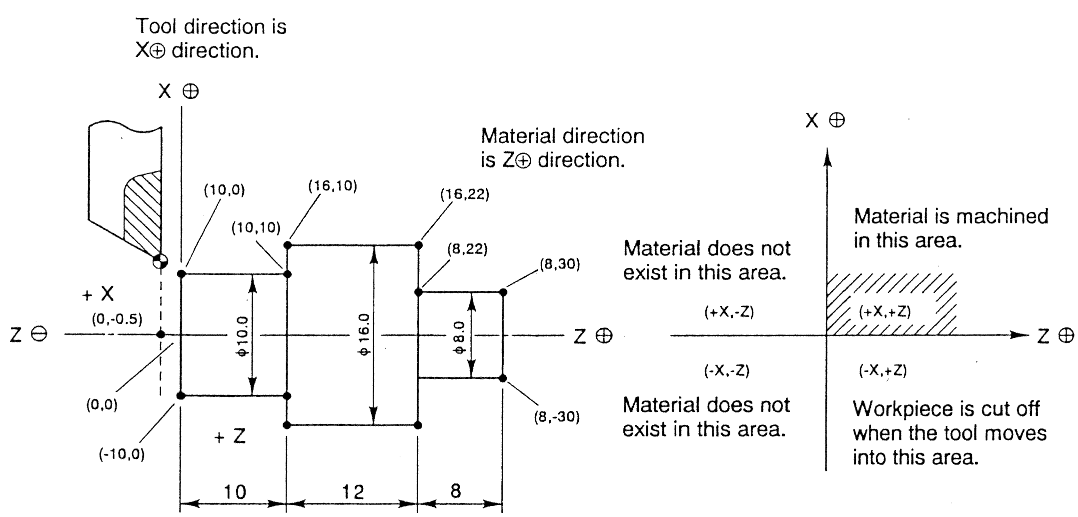


3.6.3 Signs and Coordinate Values

The tool direction is the X + direction. The material direction is the Z + direction.

For the coordinate value, the X is usually the numerical value from the center of materials expressed by diameter, and the Z is the numerical value from the end face of materials.





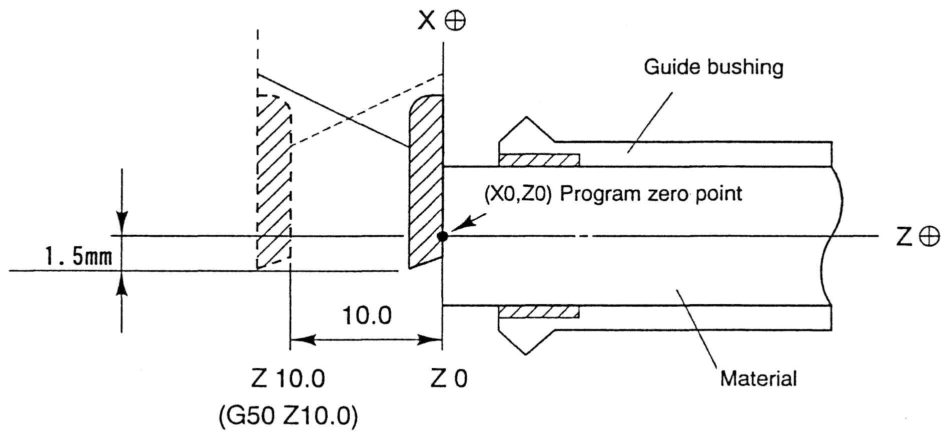
3.6.4 Coordinate System Setting

Specify the reference position for the longitudial direction.

Usually, material end face which touches the cut-off tool is set to the reference position (Z = 0).

Symbol such as "G50 Z0" is used for this declaration. This symbol means "define the current position as Z = 0."

Since the NC unit automatically sets the coordinate system using a selection tool, the X axis requires no setting.

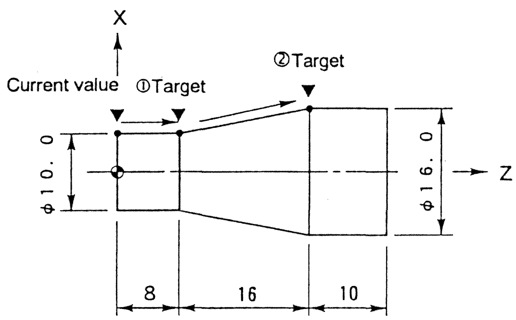


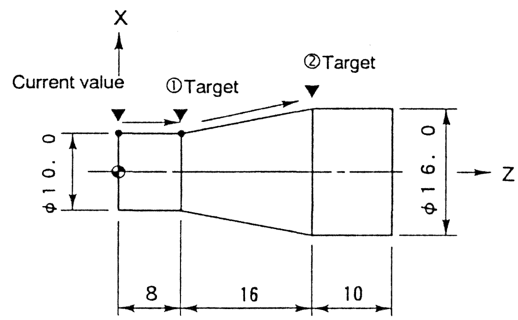
3.6.5 Absolute and Incremental Commands

The absolute command is used for the (X, Z) coordinate value of the target position and the incremental command is used to specify the move distances of X and Z axis directions.

Absolute command : Specifies the absolute coordinate value of the target position.
(X□, Z□)

Incremental command : Specifies the X axis direction and Z axis direction (relative) move distances.
(U□, W□)
The values of X and U are specified by the diameter.





Regard Z0 as the start.

Absolute	Incremental
X10.0 Z0	**U0 W0**
↓	↓
① **X10.0 Z8.0**	① **U0 W8.0**
↓	↓
② **X16.0 Z24.0**	② **U6.0 W16.0**

Answer to the question on page 3-1.

3.G01 X20	4.X30	5.G01 X20 Z20	6.G01 X20 Z30
7.G01 X30Z30	8.G02 X40 Z40	9.G02 X50 Z50	10.M02

3.7 Program and Axis Control Groups

3.7.1 Number of Axis Control Groups

You can change the number of axis control groups for a program during editing. Select [Edit] by pressing the corresponding software key to select the number of axis control groups between 1 (one axis control group) and 2 (two axis control groups). Although the default number of axis control groups is predetermined for each type of machine, you can change it freely for each program according to the machining state.

Type	Standard number of axis control groups
I	1
III	2
VII	2

3.7.2 Program-Instructed Axis Control Groups (Type VII)

For product separation, programming is possible using only axis control group 1 without issuing any G code macro as an internal command.

(Program using a single axis control group)

The program using a single axis control group is effective when:

- The back spindle defined as machining data has a workpiece receiver box and M34 is specified for $1:

  In this case, the current mode is recognized as the workpiece receiver box collection mode, in which the X2 and Z2 axes are used in $1 (Single machining ON).

- The back spindle defined as machining data is a standard or with support and M34 is specified for $1:

  In this case, the current mode is recognized as the pick-off collection mode, in which the X2 and Z2 axes are used in $1 (Single machining ON).

- T3000 is specified with an argument (A1 C_) for $1:

  In this case, the current mode is recognized as the pick-off collection mode with superimpose control ON (O.D. - I.D. simultaneous machining ON).

(For details, see Example 2 for a single axis control group program in Chapter 8.)

- Single axis control group programs can not be executed unless the setting switch of the $2 cycle start are OFF.

Product code C – L 5 1 6 2 0 I III VII – 5 7 0

Document code 2 E 1 – 0 3 0 2

2 E 2 – 0 3 0 1

Chapter 4 Command Codes

4.1 Summary of Addresses Used in NC Programs 4-1
4.2 T Functions (Tool Selection) 4-3
4.2.1 Tool Mounting and Machining Positions 4-4
4.2.2 T Code Commands and T Code Arguments 4-5
4.2.3 Drilling by Opposite Three-spindle Tool Post during Superimpose Control (Type III, VII) 4-10
4.2.4 Tool Selection for Back Machining (Type VII) 4-11
4.2.5 A Series of Pick-off Operations (Type VII) 4-12
4.3 G Functions (G Codes) 4-13
4.3.1 G Codes Table 4-13
4.3.2 G00 - Rapid Feed Positioning 4-14
4.3.3 Linear Interpolation (G01 for Cutting Feed on a Straight Line) 4-16
4.3.4 Circular Interpolation (G02, G03 in Cutting Feed) 4-18
4.3.5 G04 - Dwell Command 4-20
4.4 Special G Codes 1 and Special G Codes 2 4-21
4.4.1 Special G Code 1 Table 4-21
4.4.2 Special G Code 2 Table 4-21
4.4.3 Superimpose Control (G811, G810 - Outer/Inner Diameter Simultaneous Machining) (Type III, VII) 4-22
4.4.4 Synchronization Control (G821, G820 - Single Machining) (Type III, VII) 4-24
4.4.5 G999 - Last Program Execution 4-26
4.4.6 Phasing Profile Material (G899) 4-26
4.4.7 Superimpose Single Axis Control Group (G851, G850) (Type III, VII) 4-27
4.4.8 Spindle Synchronization Control (G114.1, G814, G113) (Optional) (Type III, VII) 4-28
4.4.9 Start Position Queuing (Type 1) (G115) 4-29
4.4.10 Start Position Queuing (Type 2) (G116) 4-29
4.4.11 Auxiliary Function Output during Axis Feed (G117) 4-30
4.4.12 End Position Specified Queuing (G149) 4-31
4.5 F Functions (F Codes) (Cutting Feed Rate) 4-32
4.5.1 Cutting Feed Rate (F) 4-32
4.5.2 Cutting Feed Rate for Back Machining (F) 4-33

4.6 M Functions (M Codes) 4-34
4.6.1 M Codes Table 4-34
4.6.2 Main Spindle Rotation and Stop (M03, M04, and M05) 4-37
4.6.3 Back Spindle Rotation and Stop (M23, M24, and M25) (Type VII) 4-38
4.6.4 Tool Spindle Rotation and Stop (M58, M59, and M60) 4-39
4.6.5 Front/Back Tool Spindle Rotation and Stop (M80, M81, M82) (Type III, VII) (Optional) 4-40
4.6.6 Back Spindle Product Separation (M32, M33, M34) (Type VII) 4-41
4.6.7 Spindle Chuck Open and Close (M06, M07) 4-47
4.6.8 Back Chuck Open and Close (M15, M16) (Type VII) 4-47
4.6.9 Opposite Tool Post Advance and Return (M140 and M141) (Type III, VII) 4-48
4.6.10 Y90Z Cut-Off Tool Breakage Detector (M51) (Optional) 4-50
4.6.11 Knock-out (M10, M11) (Type VII) 4-51
4.6.12 U10C Support (M10, M11) (Type I) 4-52
4.6.13 U11C Support (M10, M11) (Type III) 4-54
4.6.14 U51Z Stock Gripper (M13, M14) (Type I, III) 4-55
4.6.15 U31J Work Separator (M32, M33) (Type I, III) 4-56
4.6.16 U33J Workpiece Separator (M32, M33) (Type III) 4-57
4.6.17 Bar Feed (M54, M55) 4-58
4.6.18 Bar Feed Program Enable/Terminate (M08, M09) 4-59
4.7 S Functions (S Codes) 4-60
4.8 !(Exclamation) Function (Queuing Code) 4-61

4.1 Summary of Addresses Used in NC Programs

As described in previous sections, commands used in NC programs are alphanumeric codes (alphabetic character and number) associated with some special symbols.The alphabetic character in each command is referred to as an address.

This chapter describes the meanings and functions of addresses, $, and ! and explains how to specify their respective commands.

Command address

Preparation function

G□□

Specified code

G00 Rapid feed

G01 Linear cutting

G02 Circular arc cutting

Miscellaneous function

M□□

Specified code

M03 Spindle forward rotation

M06 Close chuck

M52 Turn on coolant supply

Tool selection command

T□□ΔΔ

Tool No. Offset No.

Preparation function

G□□□

Specified code

G811 Superimpose machining command

G821 Synchronization machining command

Coordinate value address

Move distance of diametrical direction axis (X axis)

X□ mm

(U)

Move distance of longitudinal direction axis (Z axis)

Z□ mm

(W)

Radius

R□ mm

Time

U□ sec

(P)

Address for specifying cutting condition

Main spindle speed

$\underbrace{\text{S}\square}_{\text{Specified code}} = \underbrace{\square\square\square\square}_{\text{From cutting conditions}}$ min^{-1}

Feed rate

$\underbrace{\text{F}\boxed{\quad\quad}}_{\text{From cutting conditions}}$ mm/min(mm/rev)

Program configuration address

Program No.

$\text{O}\underbrace{\square\square\square\square\square\square\square\square}_{\text{Eight-digit command}}$

This command can specify any program number within eight digits, except 8000 to 9999. Note, however, that O00000000 cannot be set.

Sequence No.

$\text{N}\underbrace{\square\square\square\square\square}_{\text{Five-digit command}}$

This command can specify any sequence number within five digits. Note, however, that N999 cannot be set.

Axis control group command

$\underbrace{\$\square}_{\text{Specified code}}$

Queuing command

Single queuing **!**

Queuing No. for multi-axis control group

L$\boxed{\quad\quad}$

4.2 T Functions (Tool Selection)

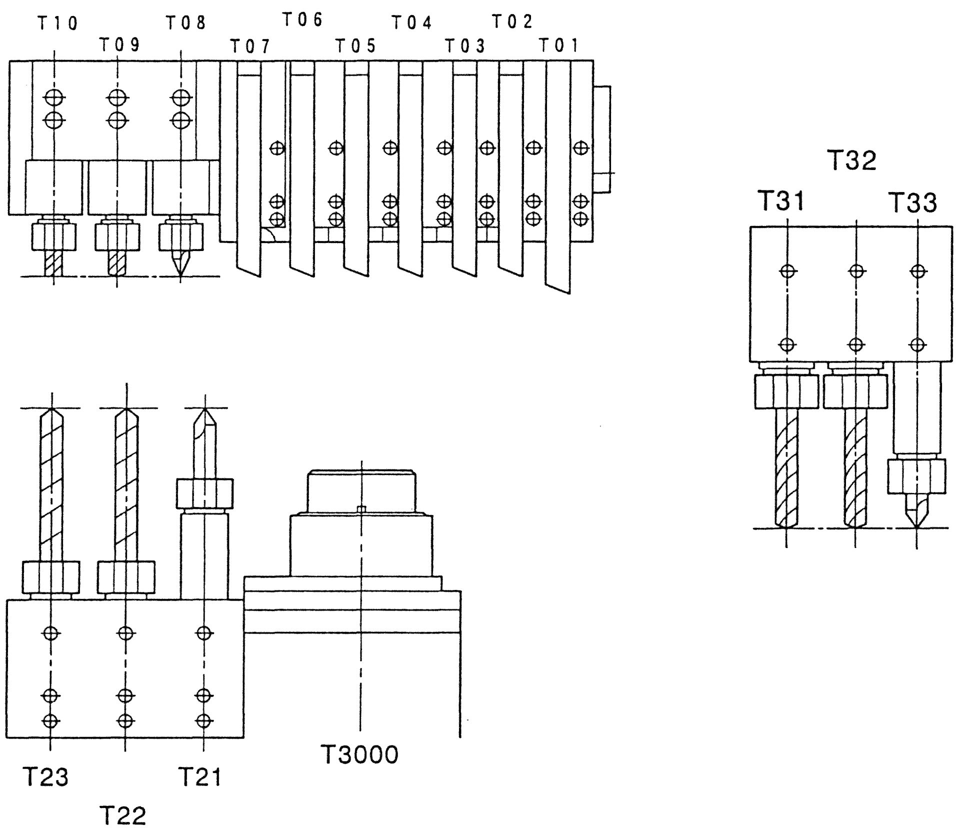
These functions place the specified tool in the machining location.

The tool is positioned to a given point by a "T□□ΔΔ" command. Positioning point "2a" shows below can be freely set as the "Tool Positioning Point" of machining data.

Set 2a to at least 0.2 even when you want to set the positioning point near the material to be machined, for example, when the material is a small diameter. The "T□□ΔΔ" command consists of four-digit numerals. The first two digits correspond to the tool number, and the last two digits to the offset number. In the cancel state, "00" is used.

Command format

T□□ΔΔ Tool selection

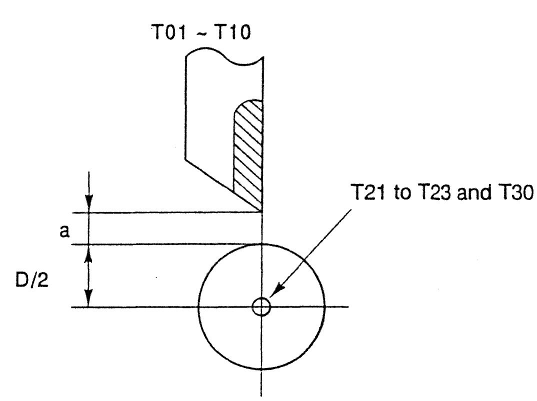


Outer diameter tool (T01 to T10);
Diameter X = Position of "Material diameter D + Positioning point 2a"

Inner diameter tool (T21 to T23), Back spindle (T30);
Center position of materials

Back tool (T31 to T33);
Center of back spindle (Actually, the back spindle moves.)

Note: The above settings apply to type VII.



4.2.1 Tool Mounting and Machining Positions

A "T□□ΔΔ" command consists of four-digit numerals. The first two digits correspond to the tool number, and the last two digits to the offset number. In the cancel state, "00" is used.

Usually, use T01 as a cut-off tool.

The following list applies to GTF3213 (standard tool holder).

Mounting position	Tool No.	Outer diameter machining	Tool spindle	Inner diameter machining	Pick-off	Back machining
Gang tool post (Type I,III,VII)	01	T0100				
	02	T0200				
	03	T0300				
	04	T0400				
	05	T0500				
	06	T0600				
	07	T0700				
	08		T0800			
	09		T0900			
	10		T1000			
	11		Free tool			
	12		Free tool			
	13		Free tool			
	14		Free tool			
Opposite tool post (Type III)	21			T2100		
	22			T2200		
	23			T2300		
	24			T2400		
	25			T2500		
	26			T2600		
	27			T2700		
	28			Free tool		
	29			Free tool		
Opposite tool post (Type VII)	21			T2100		
	22			T2200		
	23			T2300		
	24			Free tool		
	25			Free tool		
Back spindle (Type VII)	30				T3000	
Back tool post (Type VII)	31					T3100
	32					T3200
	33					T3300
	34					Free tool
	35					Free tool

4.2.2 T Code Commands and T Code Arguments

The desired tool can be easily selected, positioned, and set in coordinates by specifying a T code command. Therefore, this enables the tool selection and continuous machining without specifying many commands.

● Four-digit T code:

T□□ΔΔ

□□ : The tool number can be set to 01 to 14 for outer diameter, 21 to 25 (type VII) or 21 to 29 (type III) for front drilling, 31 to 35 (type VII) for back drilling, or 30 (type VII) for back spindle. The axis control method is parameter-set for each tool number.

ΔΔ : Offset number of tool nose wearing

● Four-digit T code with argument:

If no argument is specified for a T□□00 command, the tool being used in a machining process moves to the positioning point in rapid feed and the next tool is selected at the positioning point.

The positioning point determined at this time is "the material outer diameter + tool positioning point" set as machining data.

Note, however, that the positioning point of the tool currently being used for machining is different in the following cases:

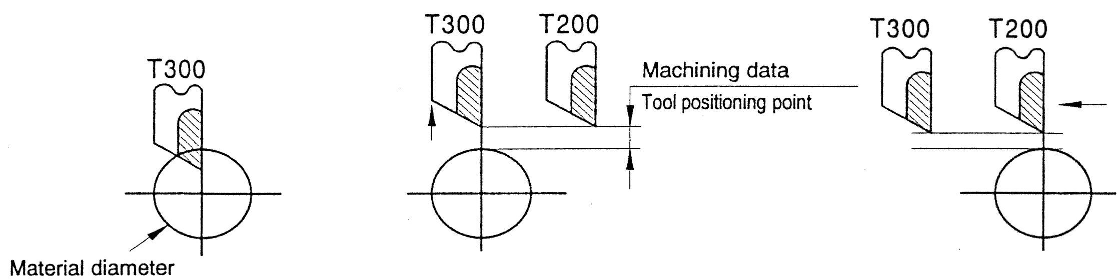
1) When the next tool is T0100
   X : Material diameter (machining data) + tool positioning point (machining data) + machine coordinate difference (20 mm) from T0100
   Y : Current position
2) When the tool currently being used for machining is T0100
   X : Material diameter (machining data) + positioning point (machining data) + 20 mm (machine coordinate difference from T0100)
   Y: Reference Y position + tool set value for T□□ 00
   In this case, the tool goes up in the X direction and moves in the Y direction.
   The tool then moves to the T□□00 positioning point (determined considering tool setting).
3) After execution of M51 (cut-off tool breakage detection ON)
   Same as 2) above.
4) When $1 tool setting is performed manually (excluding some conditions such as the start position and positioning point)
   Same as 2) above.
5) When the next tool is an end face drilling tool
   X : 0
   Y : 0
   In this case, the F, Q2, and H arguments are invalid.

A) **T□□□□** with no argument

If a T□□□□ command is issued with no argument specified, the tool currently being used for machining moves to the positioning point and the next tool is selected at the positioning point.

The current tool moves to the positioning point in rapid feed.

The positioning point determined at this time varies depending on machining data "material diameter + tool positioning point".



Explanation

① Current machining tool

② The tool moves automatically to the positioning point in response to the tool selection command.

③ The next tool is selected at the positioning point.

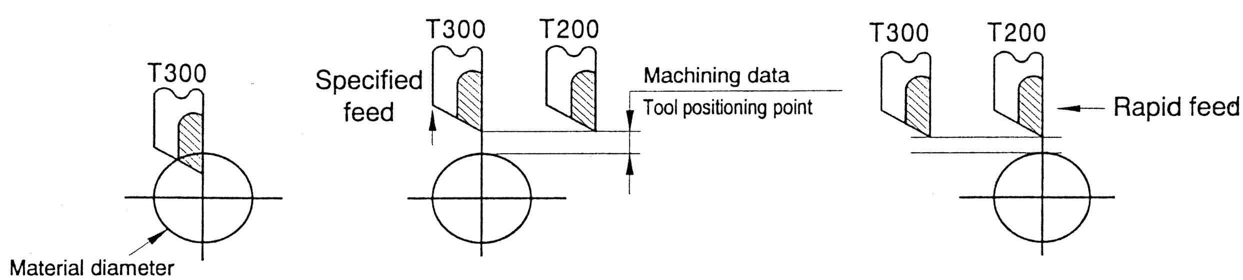
B) **T□□□□F◊◊** (T code with F argument)

When the T0000F◊◊ command is issued, the tool currently being used for machining moves to the positioning point in the feed mode specified by F<><> and the next tool is selected at the positioning point in rapid feed.

Example

T200 F0.03 (Assuming the spindle rotating)

T200 F100



Explanation

① Current machining tool

② The tool moves to the positioning point in the F-specified feed mode.

③ The next tool is selected in rapid feed at the positioning point.

Note

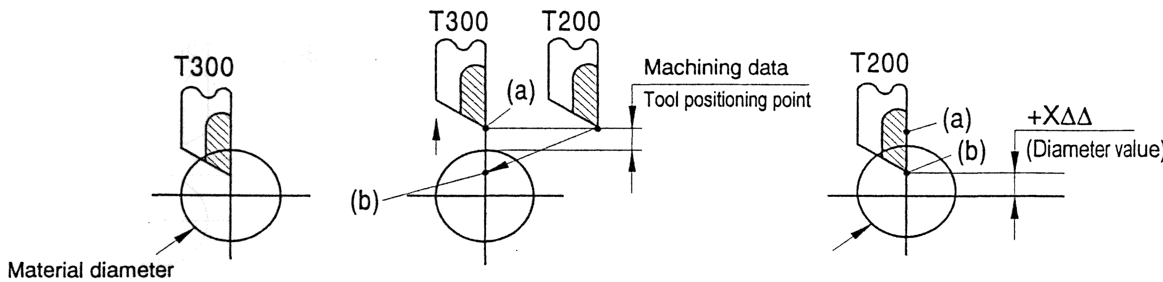
The above example illustrates tool selection from T300 to T200.

C) **T□□□□XΔΔ** (T code with X argument)

The tool currently being used for machining moves to the positioning point and the next tool is positioned at the +XΔΔ position in rapid feed.

Example

T□□□□X5.0



Explanation

① Current machining tool

② The tool moves automatically to the positioning point in response to the tool selection command.

③ The next tool moves straight (the minimum distance) to from point (a) to (b) by the value specified with the X argument.

Note

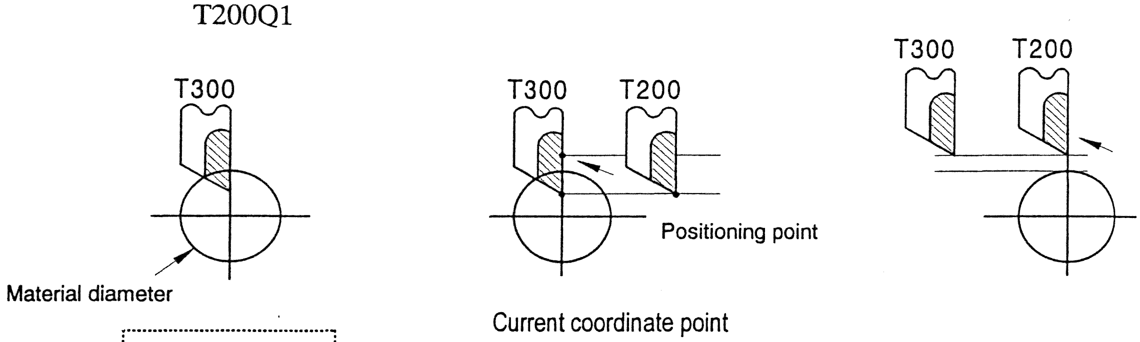
The Z argument can be used in the same way as above.

D) **T□□□□Q1** (T code with Q1 argument)

When the T□□□□Q1 command is issued, the tool currently being used for machining moves to the positioning point of the next tool and that next tool is selected at the positioning point in rapid feed.

Example

T200Q1



Explanation

① Current machining tool

② The tool moves straight (the minimum distance) to the positioning point of the next tool from the current coordinate point.

③ The new tool is selected at the positioning point.

Note

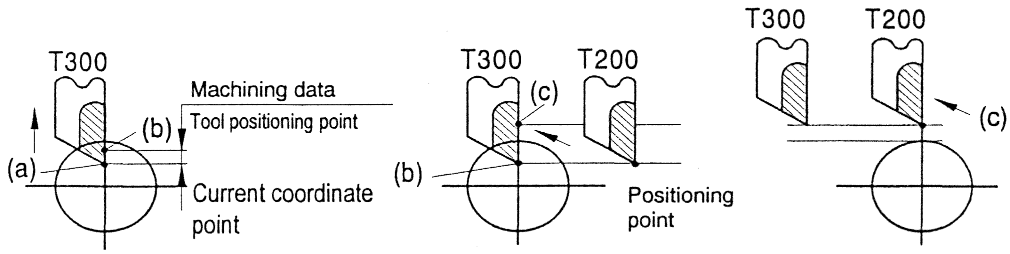
The positioning point determined at this time is "the material diameter + tool positioning point" set as machining data.

E) **T□□□□Q2** (T code with Q2 argument)

The tool currently being used for machining moves the distance of machining data (the tool positioning point) in rapid feed and the next tool is selected. To avoid positioning the tool in rapid feed, use a T□□□□F◇◇Q2 command.

Example

T200Q2



Explanation

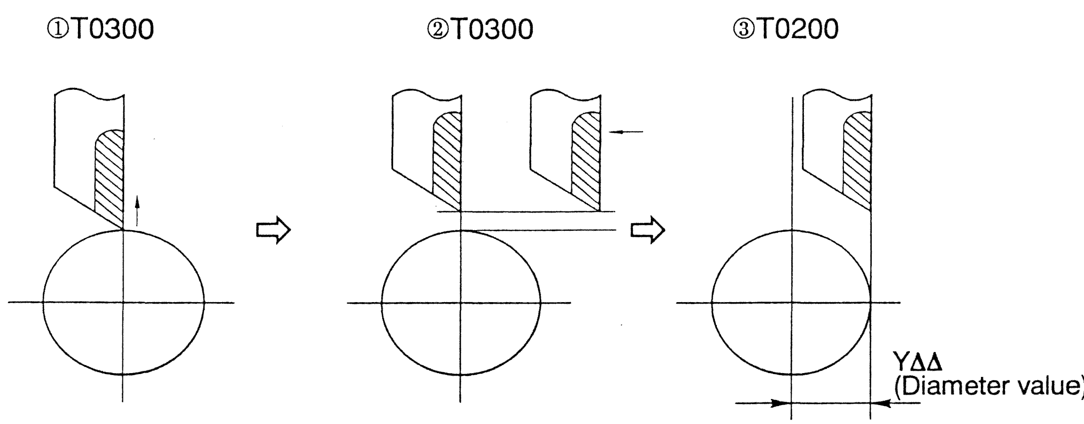
① The current machining tool moves from point (a) to (b) in rapid feed. (Specifying F◇◇ moves the tool in the specified feed mode.)
Point (a) is the current coordinate point.
Point (b) is "the current coordinate point + tool positioning point".
Point (c) is "the material diameter + tool positioning point".

② After moving to point (b), the tool moves straight (the minimum distance) to the positioning point (c) of the next tool. (The tool moves in rapid feed, ignoring F◇◇.)

③ The tool is selected at the positioning point.

F) **T0000YΔΔ** (T code with Y argument)

The tool currently being used for machining moves to the positioning point and the next tool moves to the +YΔΔ position in rapid feed.



Explanation

① Current machining tool

② The tool moves to the positioning point.

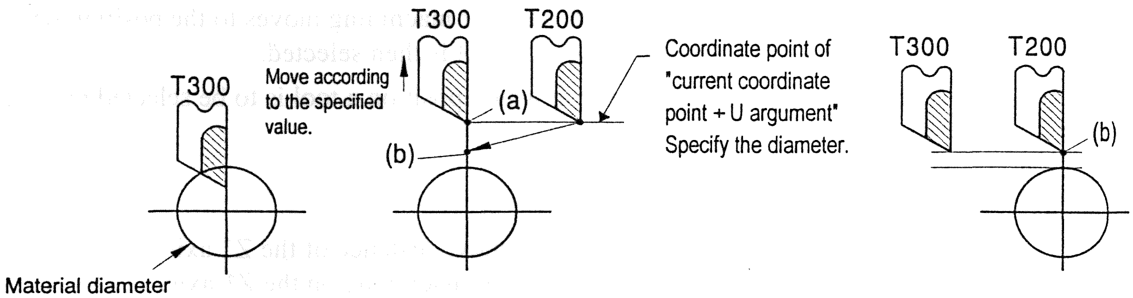
③ The specified tool is selected at the positioning point (X axis) and YΔΔ away from the center.

G) **T□□□□H▽▽** (T code with H argument)

Example

G00X8.0

T200H12.0....... Moves the current tool to the X20.0 position and switches to the next tool.



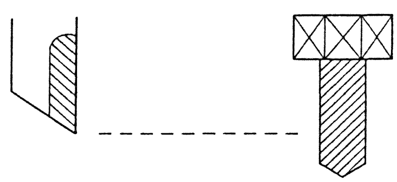
Explanation

① Current machining tool

② The tool moves the distance specified by the U argument from the current coordinate point in the X direction (a), then the next tool moves straight (the minimum distance) to the positioning point (b) set as machining data.

③ The tool is selected at the positioning point.

Use this command, for example, when a drill is set to be protruding longer than usual.



Note

The argument takes effect only when specified. When the argument is omitted, the tool moves based on the values of machining data.
(Single shot)

4.2.3 Drilling by Opposite Three-spindle Tool Post during Superimpose Control (Type III, VII)

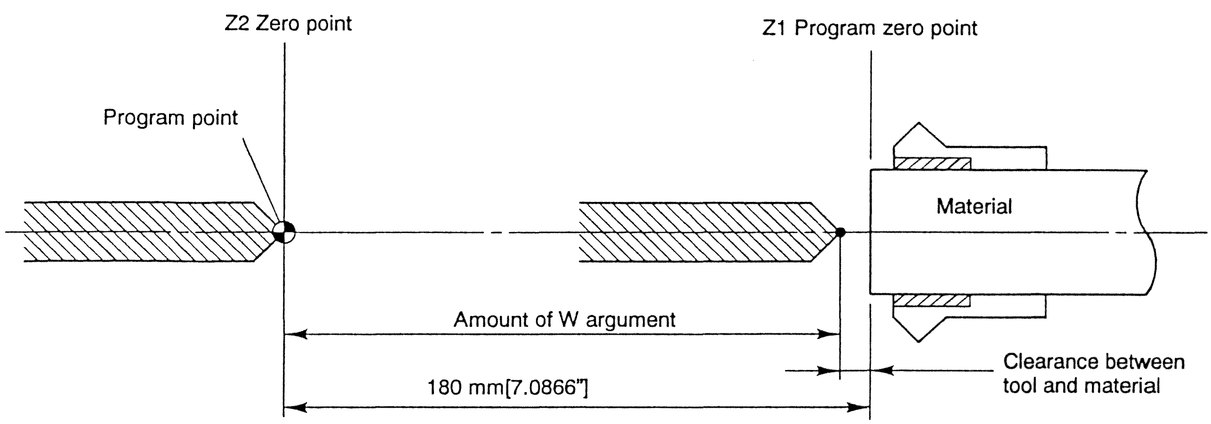
●T□□□□W□□ (T code W argument)

- The Z2 axis returns to the zero point before selecting a tool when a command for tool selection of the opposite 3-spindle tool post is specified for the outer/inner diameter simultaneous machining in the G811 mode.
- When W argument is used, the current tool used for machining moves to the position where an argument command was specified. The next tool is then selected.
- Specify a W argument command defining in which position a tool is to be selected relative to the zero point.

Note

- Interference may occur depending on the move distance of the Z1 axis during tool selection because a tool used for machining on the Z1 axis. Be careful to specify the W□□ argument.
- Specifying the W argument sets the Z2 machine zero point to 0 (zero), resulting in a positive value.

For opposite 3-spindle tool post.



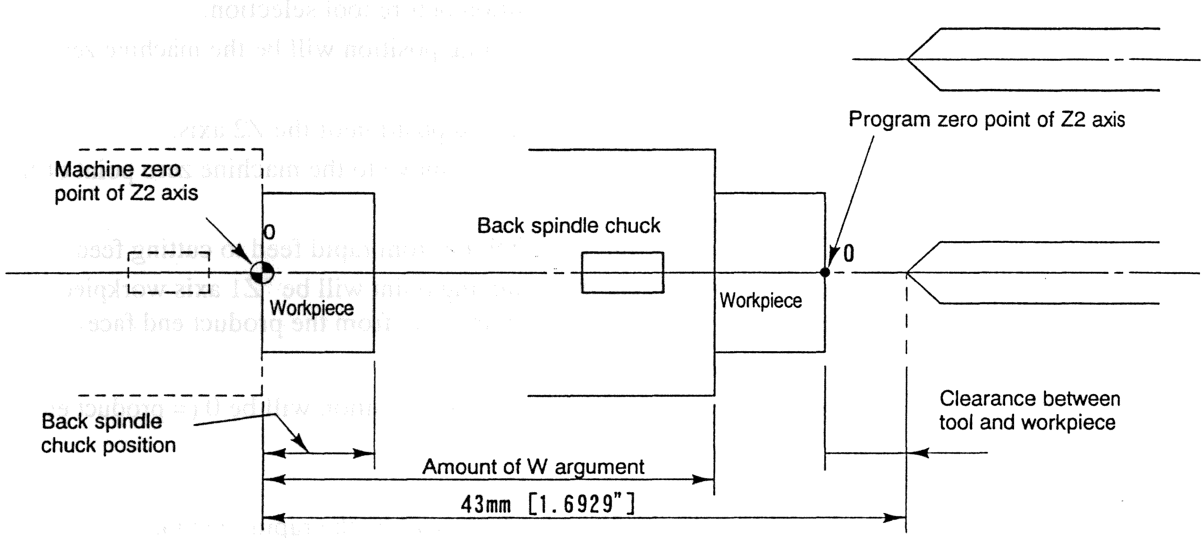
4.2.4 Tool Selection for Back Machining (Type VII)

●T□□□□W□□

This command is used to select a tool for back machining.

If a tool selection command for back machining is specified with no W argument, the Z2 axis of the back spindle returns to the machine zero point and moves to the next tool position (usually, it is the tool center).

If this command is specified with W argument, the back spindle returns to the specified position (with respect to the zero-point reference) and moves to the next tool position.



NOTE

- Specifying the W argument sets the Z2 axis machine zero point to 0 (zero), resulting in a positive value.

4.2.5 A Series of Pick-off Operations (Type VII)

Command format

T3000 A1 W☐ Z☐ B☐ C☐ E☐ S☐ K☐

Use this command only for programming using a single axis control group.

The command is for a series of pick-off operations using the primary axis control group. Program the operation for $1.

A1 represents a series of pick-off operations. Be sure to specify the A1 argument.

- **W☐**: Specify the Z2 axis position before tool selection.
  If the argument is omitted, the returning position will be the machine zero point of the Z2 axis.
- **Z☐**: Specify the tool selection end position of the Z2 axis.
  If the argument is omitted, the tool will move to the machine zero point of the Z2 axis.
- **B☐**: Specify the point of switching from rapid feed to cutting feed.
  If the argument is omitted, the switching point will be "Z1 axis workpiece coordinate point - 1.0". (Point 1.0 mm away from the product end face)
- **C☐**: Specify the back chuck position.
  If the argument is omitted, the back chuck position will be 0 (= product end face).
- **E☐**: Specify the cutting feed rate.
  If the argument is omitted, the feed rate will be the rapid feed rate.
- **S☐**: Specify the back spindle speed.
  Omitting the argument results in an alarm. (The same speed as the main spindle speed is specified.)
- **K☐**: Specify opening or closing the back chuck.
  If the K code is omitted, the back chuck will be closed. Specifying K1, back chuck will not close.
  (Use the K code, e.g., for support or program check).

Program example

MS3S1☐	.........Rotate the main spindle.
T0100	.........Select the cut-off tool.
G00 X13.0 Z40.0	.........Position the cut-off tool in rapid feed.
T3000 A1 C10.0 F1000 S3000	.........Execute a series of pick-off operations.
G01 X-3.0	.........Cut off the workpiece.
G850	.........Cancel superimpose control.
M25	.........Stop the back spindle.

Notes

- The superimpose control ON command (G851) is automatically issued during a series of pick-off operations. After workpiece cut-off operation, therefore, be sure to issue the superimpose control OFF command (G850).
- When using this command for a series of pick-off operations, be sure to specify the A1 and S☐ arguments.

4.3 G Functions (G Codes)

The G functions, specified by 2-digit numerals after the G, are preparatory functions. They also send the axis (X1, Y2, Z1, X2, Z2) control command to the NC.

Command format

G□□

Example: **G00** Rapid feed
G01 Cutting feed

4.3.1 G Codes Table

Once specified, each G code belonging to groups A, C, D, E, F, and H in the following table remains valid until another G code in the same group is specified. That is, the G code is a modal code.

●G codes table

G Code	Function	Group
⊙G00	Rapid feed positioning	A
G01	Linear interpolation	A
G02	Circular interpolation (clockwise)	A
G03	Circular interpolation (counterclockwise)	A
G04	Dwell	
G28	Reference point return	
G32	Single point thread cutting	A
⊙G40	Tool nose radius compensation cancel	C
G41	Tool nose radius compensation left ON	C
G42	Tool nose radius compensation right ON	C
⊙G43	Back spindle feed per revolution OFF	E
G44	Back spindle feed per revolution ON	E
G50	Coordinate system setting / Main spindle speed clamp setting	
G76	Multiple repetitive threading cycle	
G79	Face drilling cycle (optional)	H
G80	Synchronized cross tapping mode OFF (optional)	H
G88	Synchronized cross tapping mode ON (optional)	H
G90	Longitudinal (horizontal) turinig canned cycle	H
G92	Thread cycle canned cycle	H
G96	Constant surface speed control ON	F
⊙G97	Constant surface speed control OFF	F
G98	Millimeter per minute feed	D
⊙G99	Millimeter per revolution feed	D

Notes

- Multiple G codes can be specified in the same block if they belong to different groups.
- G codes cannot be specified with T□□□□in the same block.
- A three-digit G codes (except G899) cannot be executed in MDI (manual data input) mode.
- While the G32, G76, G79, G88, G90, and G92 codes are effective, single block operation is disabled.
  Even if single block operation is being executed, once the NC reads any one of the above G code commands, that G code will be continuously executed until canceled. If either one of the axis control groups ($1 or

$2) operates in this condition, the other axis control group will also continue to operate in the same manner.

- G codes with ⊙ mark are selected when the power is turned on.

4.3.2 G00 - Rapid Feed Positioning

G00 is used to move the tool rapidly to a specified X, Z coordinate and position it there. G00 is also used to return the spindle.

The G00 code is used for moving the tool or the spindle from the current position to the specified position in rapid feed.

The G00 code maintains the current state (modal) until a command (G01, G02, or G03) (see the G functions table) in the same A group and H group is specified.

The rapid feed rate has been defined in advance. This rate does not require specification in a program.

Rapid feed rate	X axis:	X1 = 20m/min, X2 = 20m/min
	Y axis:	Y1 = 20m/min
	Z axis	Z1 = 20m/min, Z2 = 20m/min

The rapid feed rate during Z1 and Z2 superimposition is 24 m/min (maximum).

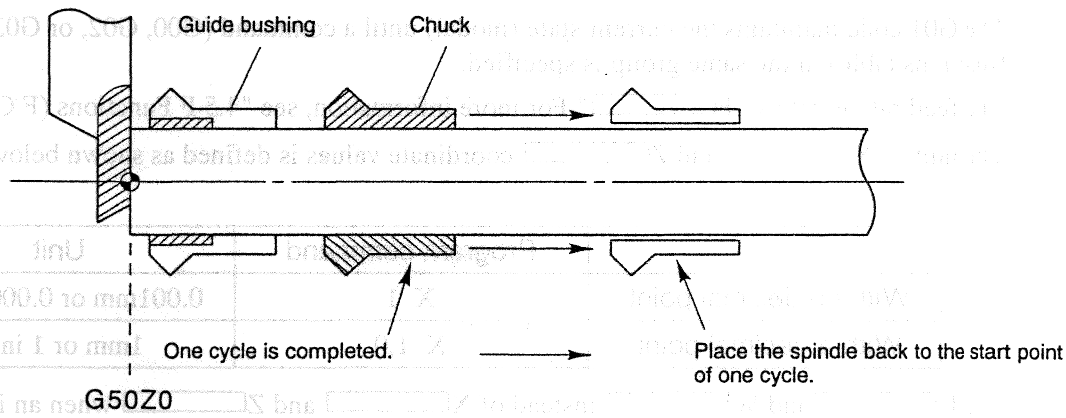
The rapid feed rate during Y1 and X2 synchronization is 20 m/min (maximum).

Use U☐, W☐ and V☐ instead of X☐, Y ☐ and Z☐ when an incremental command is specified.

Command format 1

G00 Z☐ Rapid feed in longitudinal (Z-axis) direction (X is fixed.)

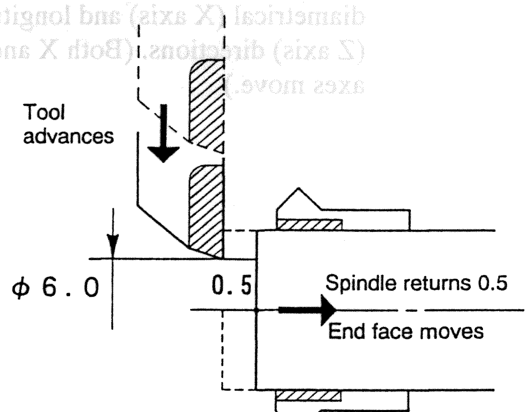
To complete one cycle and to proceed execution to the next cycle, use a G00Z0 command to return the spindle to the start position.



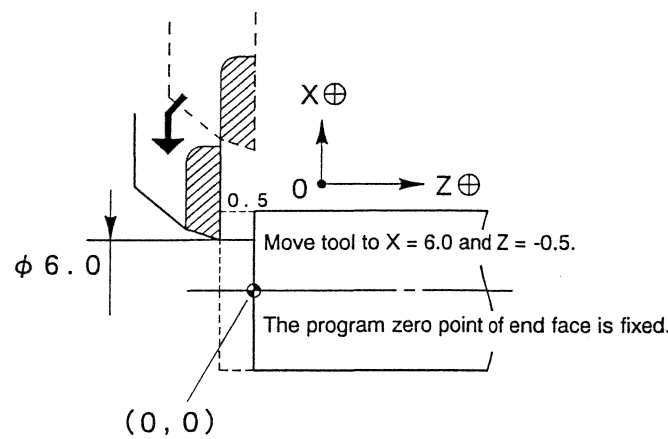
Command format 2

G00 X☐ **Z**☐ Simultaneous rapid feed in diametrical (X-axis) and longitudinal (Z-axis) directions (Both X and Z axes move.)

If you want to position the tool in a diameter of 6.0 mm and set the clearance between the tool and material to 0.5 mm, specify a G00X6.0Z-0.5 command.



Actual movement



Conception in program

4.3.3 Linear Interpolation (G01 for Cutting Feed on a Straight Line)

G01 is used to perform a cutting operation while moving the tool on a line to the desired X and Z coordinates at the specified feed rate.

This function is called linear interpolation in the NC terminology.

The G01 code allows the tool to cut from the current position to the specified position in a straight line.

The G01 code maintains the current state (modal) until a command (G00, G02, or G03) (see the G functions table) in the same group is specified.

The feed rate is set by "F□" For more information, see "4.5 F Functions (F Codes)".

The unit of X□ and Z□ coordinate values is defined as shown below.

	Program command	Unit
With no decimal point.	X 1	0.001mm or 0.0001 inch
With a decimal point.	X 1.0	1mm or 1 inch

Use U□ and W□ instead of X□ and Z□ when an incremental command is specified.

Command format

G01 X□ F□	Feed in diametrical (X axis) direction.(Z axis is fixed.)
G01 Z□ F□	Feed in longitudinal (Z axis) direction. (X axis is fixed.)
G01 X□ Z□ F□	Simultaneous feed (tapering) in diametrical (X axis) and longitudinal (Z axis) directions. (Both X and Z axes move.)

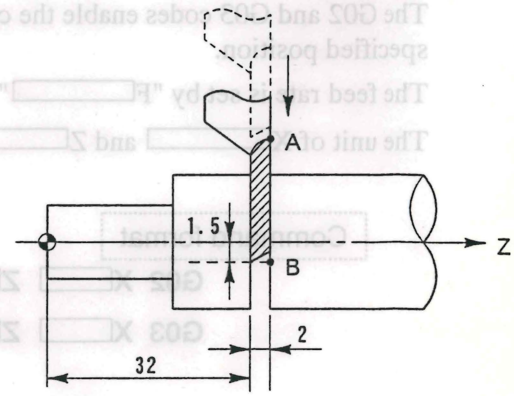
- Tool feed in direction of diameter

Command format

G01 X▭ F▭ Feed in the diametrical (X axis) direction (with the Z axis fixed)

Cutting from point A to point B (-3.0, 32.0)

G01 X-3.0 F▭



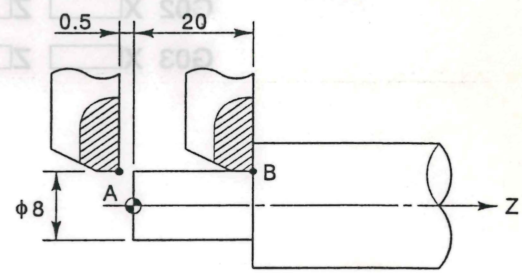
- Tool feed in longitudinal direction

Command format

G01 Z▭ F▭ Feed in the longitudinal (Z axis) direction (with the X axis fixed)

Cutting from point A to point B (8.0, 20.0)

G01 Z20.0 F▭



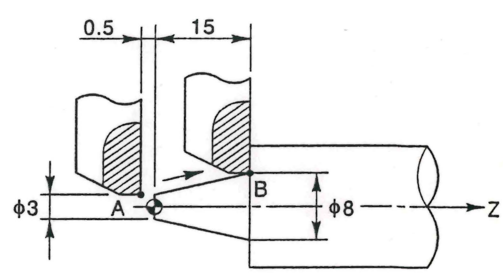
- Simultaneous tool feed in diametrical and longitudinal directions (tapering)

Command format

G01 X▭ Z▭ F▭ Tapering (X and Z axes) feed (moving both the X and Z axes).

Cutting from point A to point B (8.0, 15.0)

G01 X8.0 Z15.0 F▭



Note

- Pay attention to the spindle rotation, coolant, chuck, and guide bushing states when performing the cutting feed.

4.3.4 Circular Interpolation (G02, G03 in Cutting Feed)

G02 and G03 are used for cutting while moving the tool on an arc with radius R from the current position to the specified position at the specified feed rate. This function is called circular interpolation in the NC terminology.

The G02 and G03 codes enable the cutting on an arc with radius R from the current position to the specified position.

The feed rate is set by "F☐"

The unit of X☐ and Z☐ coordinate values is the same as for the G01 code.

Command format

G02 X☐ Z☐ R☐ F☐ (Clockwise)

G03 X☐ Z☐ R☐ F☐ (Counterclockwise)

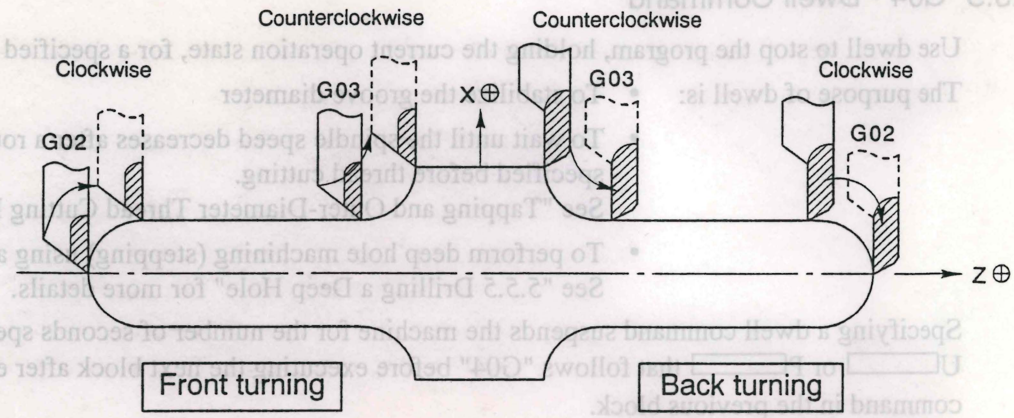
The G02 and G03 codes also can specify coordinates as shown below. "I" and "K" indicate the distance (with signs) to the center of an arc from the start position. In this case, the radius of a circle is not required.

Command format

G02 X☐ Z☐ I☐ K☐ F☐ (Clockwise)

G03 X☐ Z☐ I☐ K☐ F☐ (Counterclockwise)

The coordinate system of G02 (clockwise) and G03 (counterclockwise) is used as shown below.



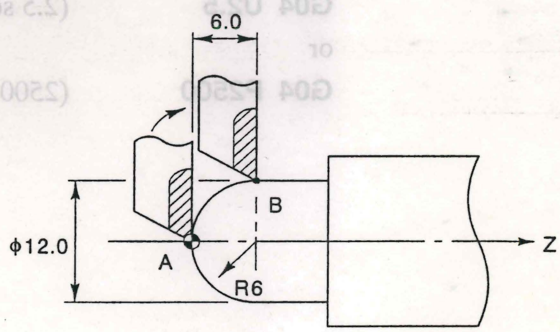
Tips: To move the tool in the + (plus) direction on the Z axis:
Specify G02 to cut the workpiece into a raised shape.
Specify G03 to cut the workpiece into a dented shape.

● **G02** Clockwise (CW)

Arc machining from point A to point B (12.0, 6.0)

G02 X12.0 Z6.0 R6.0 F□

(G02 X12.0 Z6.0 K6.0 F□)

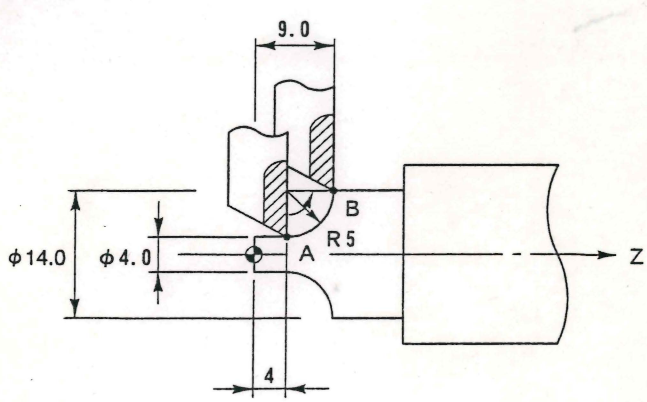


● **G03** Counterclockwise (CCW)

Arc machining from point A to point B (14.0, 9.0)

G03 X14.0 Z9.0 R5.0 F□

(G03 X14.0 Z9.0 I5.0 K0 F□)



4.3.5 G04 - Dwell Command

Use dwell to stop the program, holding the current operation state, for a specified period of time.

The purpose of dwell is:

- To stabilize the groove diameter
- To wait until the spindle speed decreases after a rotation command is specified before thread cutting.
  See "Tapping and Outer-Diameter Thread Cutting Processes".
- To perform deep hole machining (stepping) using a drill.
  See "5.5.5 Drilling a Deep Hole" for more details.

Specifying a dwell command suspends the machine for the number of seconds specified by U☐ or P☐ that follows "G04" before executing the next block after executing the command in the previous block.

Command format

G04 U☐ Pause dwell (indicates the unit of seconds.)

or

G04 P☐ (For address P, no decimal point can be used.)

The maximum command time is 9999.999 seconds.

Program example

For a dwell command of 2.5 seconds, the command format is as follows:

G04 U2.5 (2.5 sec)

or

G04 P2500 (2500 msec)

4.4 Special G Codes 1 and Special G Codes 2

These G code commands are unique to the L16 and L20 machines for machining using multiple axis control groups.
They allow multi-axis control group machining and can simplify the machining program and improve productivity. As a result, these codes reduce the machining time.

Command format

G□□□ Three-digit G code command

4.4.1 Special G Code 1 Table

These G codes apply to both of the $1 and $2 axis control groups.

The G codes are used to perform automatic queuing in $1 and $2 and proceed to the next block.

G code	Function
G810	O.D. - I.D. simultaneous machining OFF
G811	O.D. - I.D. simultaneous machining ON (Z_1-Z_2 axes superimpose)
G820	Single machining OFF
G821	Single machining ON (Y_1-X_2 axes synchronization)
G999	Last program execution

4.4.2 Special G Code 2 Table

These G codes apply to one of the $1 or $2 axis control groups.

G code	Function
G899	Phasing for profile material ($1 only)
G851	Superimpose single line control ($1 only)
G850	Superimpose single line control cancel ($1 only)
G113	Spindle synchronization control cancel
G114	Spindle synchronization control
G814	Spindle synchronization control
G115	Start position queuing (Type 1)
G116	Start position queuing (Type 2)
G117	Auxiliary function output during axis feed
G149	End position specification queuing

Note

- The above G codes (except G117) cannot be used along with the M, S, or T code in the same block.
- Three-digit G codes (except G899) cannot be executed in MDI mode.

4.4.3 Superimpose Control (G811, G810 - Outer/Inner Diameter Simultaneous Machining) (Type III, VII)

The outer/inner diameter simultaneous machining is called superimpose control in the NC terminology.

The outer/inner diameter simultaneous machining can be turned on and off using these commands.

Use these commands when creating a program which uses two axis control groups.

These commands should be independently specified for each axis control group.

Command format

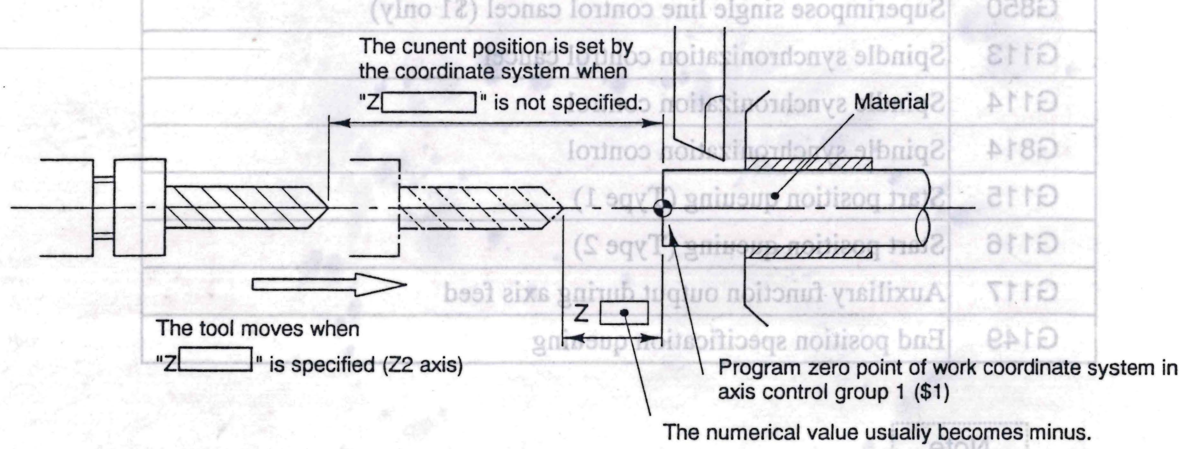
$1	$2	
G811	**G811** Z☐	Superimpose control ON Z☐ can be omitted.

●G811

- Axis control group 1 ($1) Performs no special operation and setting.
- Axis control group 2 ($2) Sets the work coordinates (Z2 axis) of $2 by the work coordinate system of $1 and turns on the superimpose control.
  When a Z code is specified, a tool is positioned to the specified position of a newly set coordinate system.
  When a Z code is not specified, a tool does not move until a Z code of $2 is specified.

The coordinate system is set at the standard set position of the opposite 3-spindle tool post. Therefore, if the tool has been shifted, enter the shift amount in the Tool Set screen or set coordinate again by G50 in the $2 (axis control group 2) program after a G811 command is specified.

Use G851 when creating a program which uses a single axis control group.



● G810

Command format

$1	$2	
G810	**G810 Z☐**	Superimpose control OFF
or	**G810 W☐**	
		Z☐, W☐ can be omitted.

- Axis control group 1 ($1): Performs no special operation and setting.
- Axis control group 2 ($2): When neither a Z nor a W code is specified, Z2 axis is moved to the machine zero point (return position), and the superimpose control is canceled.
  When a Z code is specified, Z2 axis is moved to the position of the value specified by Z of the work coordinate, and the superimpose control is canceled.
  When a W code is specified, Z2 axis is moved to the position of the value specified by W (from the zero point reference), and the superimpose control is canceled.

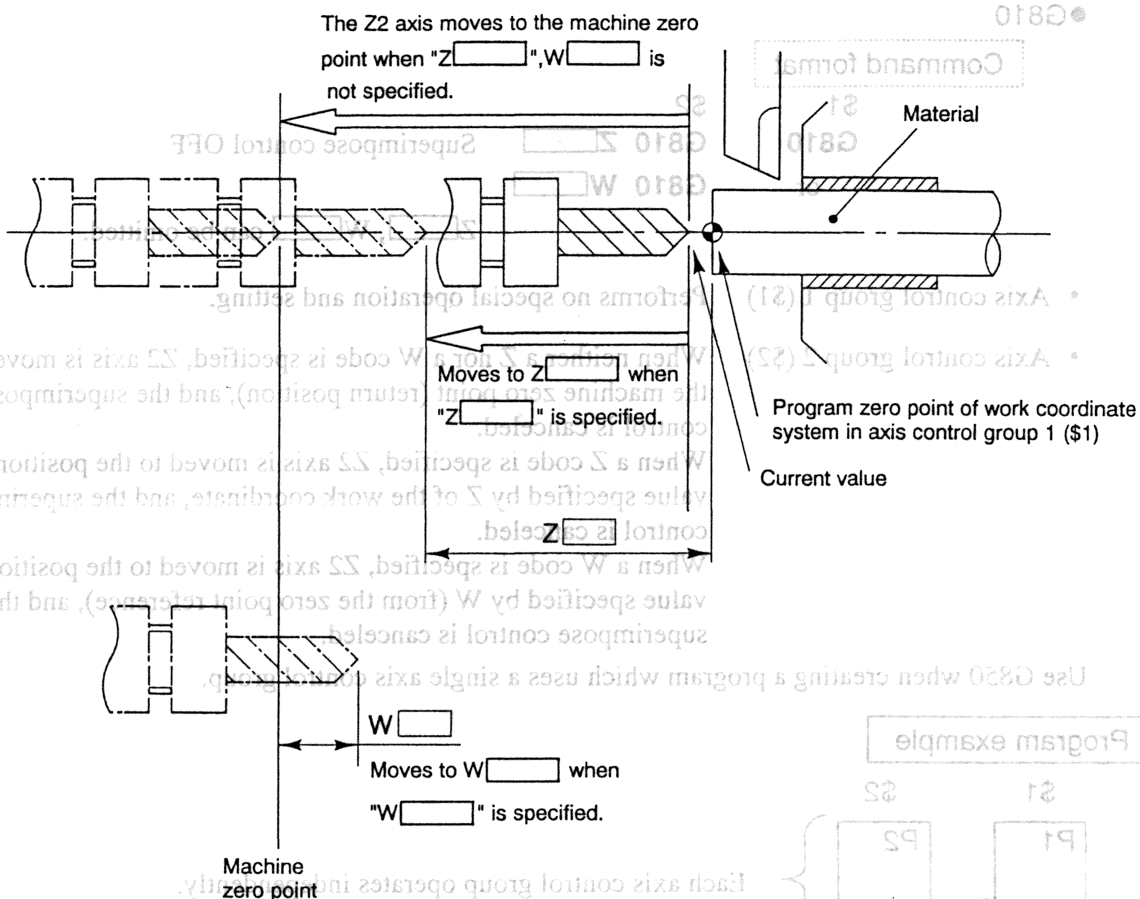
Use G850 when creating a program which uses a single axis control group.

Program example

$1	$2	
P1	P2	Each axis control group operates independently.
G811	**G811**	← $1 and $2 queue for execution.
P3	P4	The Z axis of $2 is superimposed here.
G810	**G810**	← $1 and $2 queue for execution.
P5	P6	Each axis control group operates independently.

Notes

- As the axis control group 1($1) work coordinate program zero point is 0, arguments of Z have minus values.
- As the Z2 machine zero point is 0, arguments of W have plus values.



4.4.4 Synchronization Control (G821, G820 - Single Machining) (Type III, VII)

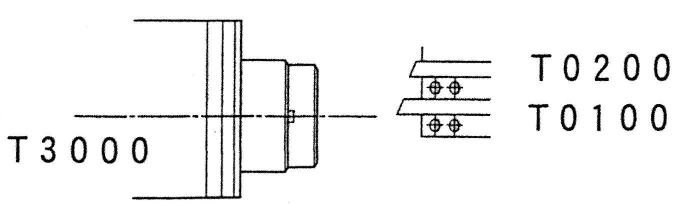
The single machining is called synchronization control, in the NC terminology.
The single machining can be turned on and off using these commands.
Use these commands when creating a program which uses two axis control groups.
These commands should be independently specified for each axis control group.

Command format

$1	$2	
G821 R1	**G821**	Synchronization control ON
G820	**G820**	Synchronization control OFF. R1 can be omitted

- The G821 command enables queuing between axis control groups 1 and 2. The T21ΔΔ to T25ΔΔ commands for axis control group 1 that appear first after the G821 command return the X1 axis a full stroke, position the X2 axis at the prescribed position, and move the Y1 axis to the position corresponding to the position of the X2 axis. From then on, synchronization control is turned on.

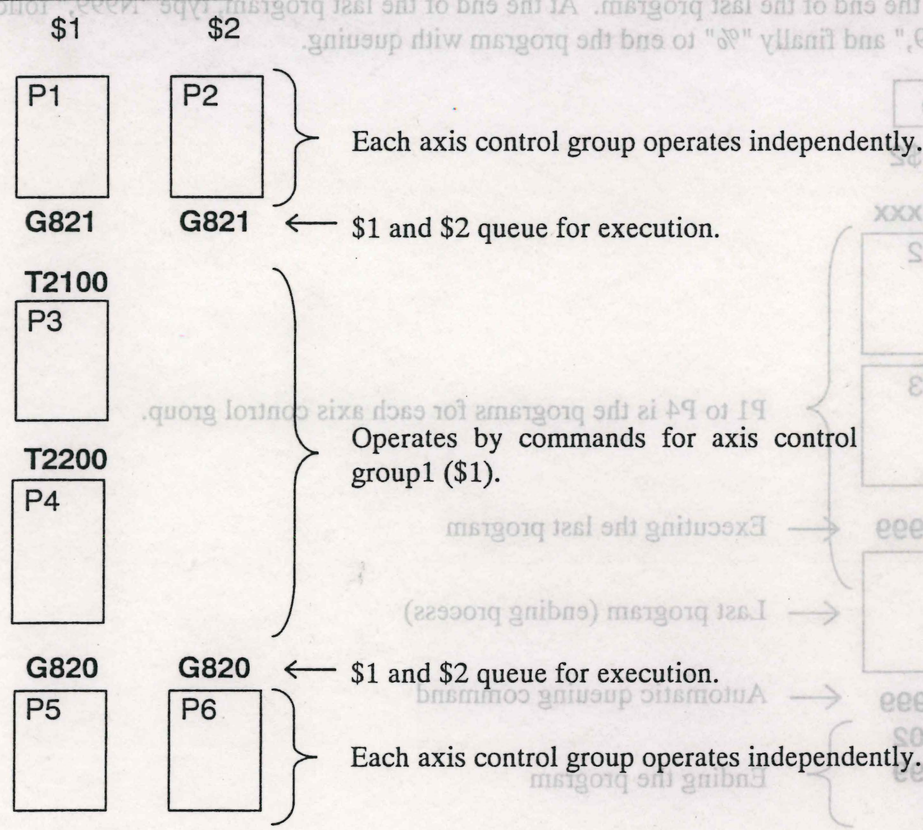
The prescribed position is the position at which T3000 (the back spindle) and T0100 (usually the cutting tool) face each other.



If a G821 R1 command is specified, the T21ΔΔ to T25ΔΔ commands for axis control group 1 that appear first after that command return the X1 axis a full stroke without synchronization control of the Y1 with X2 axes.

- Use the G820 command to cancel the synchronization control after returning the opposite tool post (Z2 axis) to the machine zero point.
  These programs are neither used nor queued in programs which use a single axis control group.

Program example



4.4.5 G999 - Last Program Execution

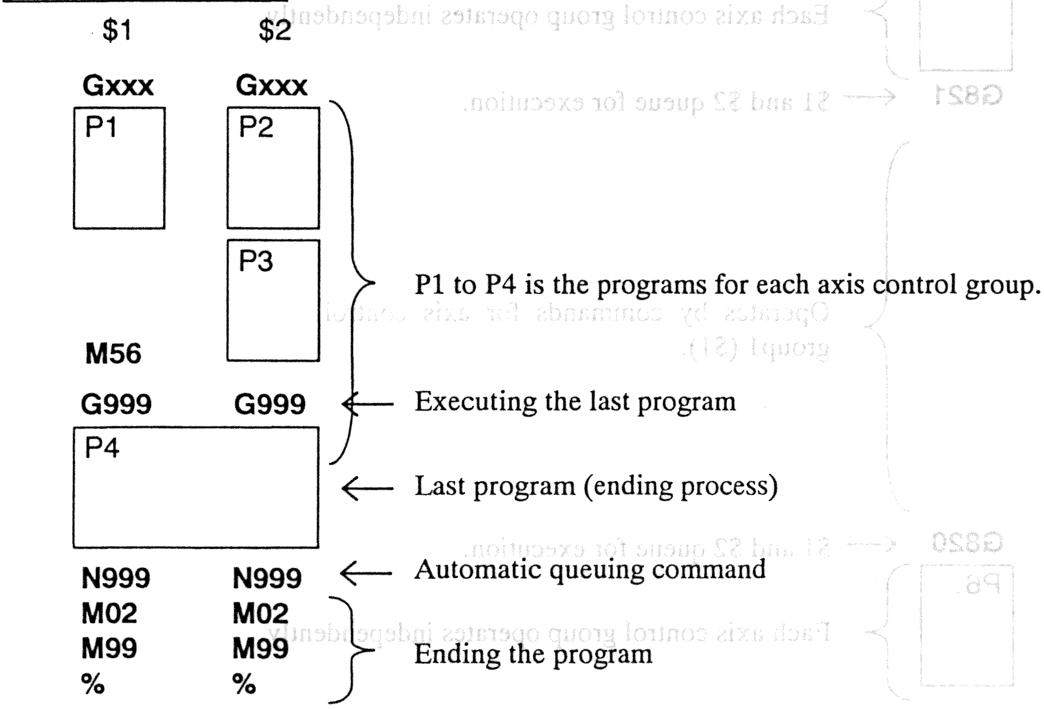
G999 must be inserted at the end of each axis control group program: $1 or $2 to perform the automatic queuing.

Command format

$1	$2	
G999	**G999**	Execution of the last program

The N999 command must be specified for each axis control group so that queuing is necessarily performed after the end of the last program. At the end of the last program, type "N999," followed by "M02," "M99," and finally "%" to end the program with queuing.

Program example



4.4.6 Phasing Profile Material (G899)

This command is used for the NC unit to store the phase shift between the spindle and guide bushing and the phase shift between the spindle and back spindle during profile material machining.

Be sure to specify the command when a profile material chuck and guide bushing are mounted.

Procedure

1. Pass the material (profile) through the spindle and guide bushing.
2. Execute G899 under "All Block" is depressed in MDI($1) for automatic chuck close operation and phase adjustment.

In a material replacing program, specify M26 (Guide bushing phase adjustment) before inserting a material into the guide bushing. Specify G814 (main spindle synchronous control) before chucking the material with the main spindle and back spindle. Specifying the two codes adjusts the stored phase shifts.

Command format

$1

G899 R1 K1

- The G899 R1 command causes the NC unit to store the phase shift between the spindle and back spindle.
  When the command is executed, the back spindle also rotates.
- When G899 R1 K1 is specified, the machine does not close the back chuck.

Note

- Specify G899 to adjust only the phase between the main spindle and guide bushing.
- Specify G899 R1 K1 to adjust both of the phase between the main spindle and guide bushing and the phase between the main spindle and back spindle.

4.4.7 Superimpose Single Axis Control Group (G851, G850) (Type III, VII)

The G851 and G850 commands are used to superimpose the Z2 axis on the Z1 axis.
These commands are used for pick-off and center support purposes.
Use these commands when creating a program which uses a single axis control group.

●G851

Command format

G851 Z☐ Superimpose control ON

Z☐ can be omitted.

- Axis control group 1 ($1): Performs no special operation or setting.

  When a Z code is specified, the tool is positioned according to the value specified on the work coordinate system for the Z1 axis in axis control group 1 ($1).

  When no Z code is specified, the material does not move.

The coordinate system is set at the standard set position of the opposite tool post.

When creating a program which uses two axis control groups, use G811 instead.

●G850

Command format

G850 Z☐ Superimpose control OFF

Z☐ can be omitted.

- Axis control group 1 ($1): Performs no special operation or setting.

  When a Z code is specified, the command cancels superimpose control after moving the Z2 axis to the position of the Z value specified as a work coordinate under superimpose control.

  When a W code is specified, the command cancels superimpose control after moving the Z2 axis to the position of the specified W value (from the zero point reference).

  When creating a program which uses two axis control groups, use G810 instead.

Note

- A setting switch of $2 Cycle Start must be set to OFF before a single-axis control group program runs.

4.4.8 Spindle Synchronization Control (G114.1, G814, G113) (Optional) (Type III, VII)

The G114 and G113 commands are used to control the speed and phase of one spindle in synchronization with the rotation of the other spindle.
These commands are effective to synchronization the two spindles in speed, for example, when chucking a workpiece from the main spindle to the back spindle or when machining a workpiece being held on both of two spindles. They are also effective when chucking a profile material from one spindle to the other by synchronization them in phase.

Command format

G814 or

G114.1 H1 D-2 R□□□

Synchronization spindle phase-shift amount command (0~359.999deg)

Synchronization spindle selection

Reference spindle selection

G113 Cancels spindle synchronization.

Explanation

The G814 command controls the reference spindle as S1 and another spindle to be synchronized as S2, synchronizing the two spindles.

The G114 command specifies the reference spindle and the spindle to be synchronization to synchronization the two specified spindles.

The G113 command cancels synchronization between the two spindles rotating synchronization according to the spindle synchronization command.

For details and examples, see Section 7.9.

4.4.9 Start Position Queuing (Type 1) (G115)

Command format

!nLℓ G115 X☐ Z☐ C☐

This command specifies a start position to provide a queuing point halfway through a block.

(1) When queuing is specified in the program for an axis control group (local axis control group), the other axis control group (remote axis control group) starts moving first.

(2) The local axis control group starts moving when the remote axis control group reaches the specified start position.

(3) If the start position specified by G115 does not exist on the next-block moving path of the remote axis control group, the local axis control group starts moving when the remote axis control group reaches all axis coordinate values of the commanded position.

(4) The start position check applies only to the axis specified by G115.

(5) If the start position is not found when the remote axis control group moves to the next block, the local axis control group waits until the remote axis control group reaches the start position by moving beyond the next block.

(6) If the two axis control groups overlap by the G115 command, they remain queued.

(7) When specifying the start position, use the work coordinate values of the remote axis control group.

(8) Specifying G115 for more than two axis control groups results in the program error P33.

(9) The G115 block is not subject to single-block stop.

(10)If two or more G115 blocks are specified consecutively, the block specified last is valid.

(11)The addresses that follow G115 are the X-axis, Z-axis, and C-axis work coordinates.

4.4.10 Start Position Queuing (Type 2) (G116)

Command format

!nLℓ G116 X☐ Z☐ C☐

(1) When queuing is specified in the program for an axis control group (local axis control group), the local axis control group starts moving first.

(2) The other axis control group (remote axis control group) starts moving when the local axis control group reaches the specified start position.

(3) If the start position specified by G116 does not exist on the next-block moving path of the local axis control group, the remote axis control group starts moving when the local axis control group reaches all axis coordinate values of the commanded position.

(4) The start position check applies only to the axis specified by G116.

(5) If the start position is not found when the local axis control group moves to the next block, the program error P33 occurs before the local axis control group starts moving.

(6) If the two axis control groups overlap by the G116 command, they remain stopped.

(7) When specifying the start position, use the work coordinate values of the remote axis control group.

(8) If G116 is specified for more than two axis control groups, multiple remote axis control groups start moving simultaneously.

(9) The G116 block is not subject to single-block stop.

(10)If two or more G116 blocks are specified consecutively, the block specified last is valid.

(11)The addresses that follow G116 are the X-axis, Z-axis, and C-axis work coordinates.

4.4.11 Auxiliary Function Output during Axis Feed (G117)

Command format

G117 X☐☐ Z☐☐ C☐☐ ☐☐☐☐

Auxiliary function

This command specifies an intermediate point and the auxiliary function to be output at that point, allowing the auxiliary function to be executed during movement.

(1) This command is placed independently immediately before the moving command block in which you want the auxiliary function be executed.

(2) This command is not subject to single-block stop.

(3) The following codes can be specified as the auxiliary function in the G117 block:

M codes	Up to 4 sets (Macro M code case, only one M code can be used.)
S codes	1 set each
T codes	1 set

(4) This command can be specified for up to two consecutive blocks.
If three blocks or more are specified consecutively, the last two blocks are valid.

(5) If the operation start position specified by G117 does not exist on the moving path, the auxiliary function is output when all axis coordinate values of the operation start position are reached.
Only the specified axes are checked.

(6) At the operation start position, after checking that the previous auxiliary function has been output, the next auxiliary function is output. The PC interface can be used as usually without modification.

(7) The auxiliary function specified along with a moving command block is output before movement is started. Movement does not stop at the operation start position. Note that, at the block end position, after checking that all auxiliary functions have been output, the next block is executed.

(8) G117 should be specified in the order of operation start positions. The program error P33 occurs if the order of operation start positions is opposite to the moving sequence.
The operation start positions match the moving sequence, auxiliary functions are output in the order in which they are specified.

(9) If the operation start position for the next block is not found, the program error P33 occurs before movement to the next block.

The descriptions (7) to (9) above can be summarized as in the following table:

First block / Second block	Intermediate point found during movement	No intermediate point during movement
Intermediate point found during movement	The order in (8) is followed.	Program error occur as in (8).
No intermediate point during movement	The order in (9) is followed for the second block.	The order in (9) is followed. Auxiliary functions are output in the order of the first block, then the second block regardless of the order of the specified points.

(11)The addresses that follow G117 are the X-axis, Z-axis, and C-axis work coordinates.

4.4.12 End Position Specified Queuing (G149)

This command enables queuing between an arbitrary position in an arbitrary block in the local axis control group and an arbitrary block position in the remote axis control group.

While the standard queuing function adjusts the start timing between the queuing axis control groups, this function adjusts the end timing of the end block specified by the block ID number.

Command format

!nLℓ G149 Q☐ X☐ Z☐ C☐ End position specified queuing

- Q represents the end position block number. It is specified by BN☐ .

Program example

$1	$2	
!2 G149 Q100 X1.0	**!1 Q120**	Queuing block specification
G1 X5.0 F50		
BN100 G1 X-1.0 F10	**BN120 G1 Z50.0 F100**	

With this example, block BN120 in $2 is completed when the X axis in $1 passes a point of 1.0 in block BN100.

(1) The end position queuing position must be specified in the work coordinate system.

(2) Only the axis on which a queuing position has been specified is checked for its passing the end position.

(3) If the queuing position is specified on multiple axes, the queuing time is the time by which all the specified axes have passed the specified point.

(4) If the specified axis does not pass the point specified in the block of the specified end position block number, a program error (P33 format error) occurs when the G149 command is issued.

(5) If the end position block number of the G149-specified axis control group is not found within 10 blocks from the end position specified queuing command or if no P code has been specified, the required time is calculated assuming the moving block in which all the specified axes has passed the specified point as the end position specification block. Note, however, that a program error (P700BN No number) occurs if the moving block is not found within 10 blocks from the end position queuing command.)

(6) The program error (P700BN No number) also occurs if the end position block number of an axis control group not specified by G149 is not found within 10 blocks from the end-pint queuing command.

(7) If a macro call or subprogram call is included within the range specified for end position specified queuing, the macro call or subprogram call and M99 are calculated as one block.

(8) Whichever axis control group that requires shorter run time for the range of program specified for end position specified queuing waits for processing to be started. ($2 may move first.)

(9) The end position queuing position is not aligned if the range of program specified for end position specified queuing requires run time exceeding two hours.

(10)Do not place an end position specified queuing command within 10 blocks from another end position specified queuing command.
Doing so causes a program error (P700BN No number).

(11)Within the range specified for end position specified queuing, do not include any command for changing the workpiece coordinate system, shifting the local coordinate system, presetting the counter, or for milling. Otherwise, the end position specified queuing position is mis-aligned because the specified position is calculated on the workpiece coordinate system effective in the block in which end position queuing is specified.

(12)The end position may not be able to be obtained if any of the following commands is placed, in the range of program specified for end position specified queuing, for controlling the axis having the specified end position. Do not include the following commands concerning the end position specified axis.
•Arbitrary axis exchange control, Direct axis control •Arbitrary axis superimpose control

(13)Address Q that follows G149 specifies a queued block; X1, Z1, and C specify their respective work coordinates.

4.5 F Functions (F Codes) (Cutting Feed Rate)

4.5.1 Cutting Feed Rate (F)

The F code specifies the feed rate for linear or circular cutting (interpolation).

Command format

F☐ Feed rate command

The cutting feed cannot be executed unless these F codes are specified.

The feed rate given by the F command function maintains the current state (modal) until the next command is specified. (The cutting feed is executed by the F code used in the previous machining.)

The feed rate is specified according to the cutting shape of "G01", "G02", and "G03" codes.

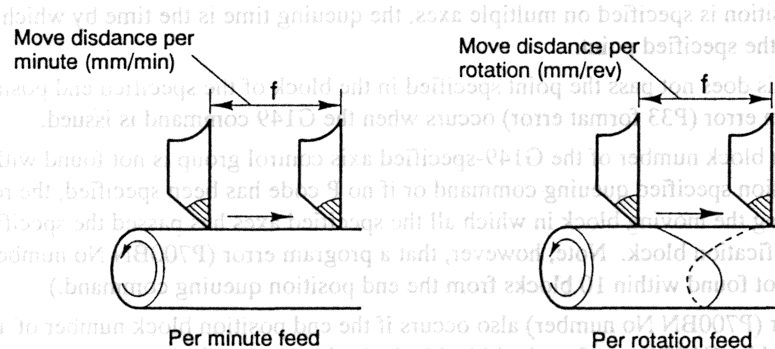
There are 2 types of the feed rate.

- **G98** : mm/min. (per minute feed : distance of the tool in mm per minute)
- **G99** : mm/rev (per revolution feed : distance of the tool in mm per spindle revolution)

The G99 state is assumed when the power is turned on.

When switching the feed method from G99 to G98 in a block of G01 cutting block, the G code must come before the F code.

<Example> **G98 G01 X☐ Z☐ F☐**



	Per Minute Feed	Per Revolution Feed
Description	The feed distance of the tool in mm per minute.	The feed distance of the tool in mm per spindle revolution.
Designation and Address	F	F
Designated G Code	G98	G99
Input Feed Rate Range	1mm/min ~ 8000mm/min	0.001mm/rev ~ 999.999 mm/rev
Override	Both per minute feed and per rotation feed are fixed in 100%. Feed rate changes by 10% in this case.	

Estimation from G98 and G99

The maximum value for F in the per rotation feed is: $F(\text{mm/rev}) \leq \dfrac{8000\text{mm/min}}{\text{spindle speed } (\text{min}^{-1})}$

Determine the proper feed rate by tables of the cutting conditions.

4.5.2 Cutting Feed Rate for Back Machining (F)

A per revolution feed command can be used.

G43: Back spindle control OFF

G44: Back spindle control ON

The per rotation feed command (G99) can be used in the G44 mode. Upon completion of back machining, be sure to specify the G43 command to cancel back spindle control.

4.6 M Functions (M Codes)

4.6.1 M Codes Table

M Code	Function	Description
M00	Program stop	Stops the automatic operation.
M01	Optional stop	Performs automatic operation stop. The operation is restarted by pressing the start switch.
M02*	Cycle stop	Stops the automatic operation in the 1 cycle mode.
M03	Main spindle forward rotation	Starts the main spindle forward rotation.
M04	Main spindle reverse rotation	Starts the main spindle reverse rotation.
M05	Main spindle stop	Stops the main spindle rotation.
M06*	Main spindle chuck close	Closes the main spindle chuck.
M07*	Main spindle chuck open	Opens the main spindle chuck.
M08	Enable bar stock exchange program	Starts the automatic material replacement program.
M09	Completed bar stock exchange program	Completes the automatic material replacement program.
M10*	Material knock-out advance (type VII)	Advances the back spindle knock-out bar.
	Support advance (type I)	Advances the support. (Optional)
	Support advance (type III)	Advances the support (on the setting table). (Optional)
M11*	Material knock-out return (type VII)	Returns the back spindle knock-out bar.
	Support return (type I)	Returns the support. (Optional)
	Support return (type III)	Returns the support (on the setting table). (Optional)
M13*	Stock gripper close (type I or III)	Closes the stock gripper (Optional)
M14*	Stock gripper open (type I,III)	Opens the stock gripper (Optional)
M15*	Back spindle chuck close	Closes back spindle chuck.
M16*	Back spindle chuck open	Opens back spindle chuck.
M18*	Enable main spindle C axis	A series of C-axis ON operations (M18C***) (Optional)
M20	Main spindle indexing, and C axis cancel	Turns the main spindle indexing and C-axis off.
M23	Back spindle forward rotation	Starts the back spindle forward rotation.
M24	Back spindle reverse rotation	Starts the back spindle reverse rotation.
M25	Back spindle stop	Stops the back spindle rotation.
M26	Execute guide bushing phasing	Matches the phases of spindle and guide bushing.
M28*	Main spindle indexing	A series of spindle indexing operations (M28S***)
M31	Work conveyor ON	Workpiece conveyor start ON (Optional)
M32*	Workpiece receiver box advance (type VII)	Advances the workpiece receiver box.
	Workpiece separator advance (type I,III)	Advances the workpiece separator. (Optional)
	Workpiece separator advance (type III)	Advances the workpiece separator (on the setting table). (Optional)
M33*	Series of operations 1 for back-machined product separation (type VII)	Performs a series of axis feed operations (with no knock-out).
	Workpiece separator return (type I,III)	Returns the workpiece separator. (Optional)
	Series of operation for workpiece separator collection (type III)	Returns the workpiece separator (on the setting table) and performs a series of operations for collection. (Optional)
M34*	Series of operations 2 for back-machined product separation (type VII)	Performs a series of axis feed, chuck open, and knock-out operations.
M38	Bar loader synchronization ON	Turns synchronization on between the Z1 axis and the bar loader. (Optional)

M Code	Function	Description
M39	Bar loader synchronization OFF	Turns synchronization off between the Z1 axis and the bar loader. (Optional)
M48*	Enable back spindle C axis (Type VII)	Turns on the back spindle C axis for a series of operations. (M48C***)
M50	Pick-off failure detect (Type VII)	After a workpiece is cut off, the main spindle rotates at low speed and the M50 command is executed.
M51*	Tool breakage detection ON	Starts cut-off tool break detection. (Optional)
M52	Coolant ON	Starts the coolant supply.
M53	Coolant OFF	Stops the coolant supply.
M54	Bar loader torque OFF	Stops the feeding of the pushrod.
M55	Bar loader start	Starts the bar loader material replacement.
M56	Product count	Starts the product count.
M57	1 cycle stop (for sub-program)	Stops the automatic operation in 1 cycle mode (for sub-program).
M58	Tool spindle forward rotation	Starts the tool spindle forward rotation.
M59	Tool spindle reverse rotation	Starts the tool spindle reverse rotation.
M60	Tool spindle stop	Stops the tool spindle rotation.
M61 ~ M65	External M code	Used when optional function is added.
M72	Back spindle air blow ON (Type VII)	
M73	Back spindle air blow OFF (Type VII)	
M77	Spindle synchronization completion queuing (Type VII)	Waits for completion of synchronization of the main and the back spindle.
M78	Back spindle indexing (Type VII)	A series of back spindle indexing operation (M78S***)
M79	Back spindle indexing cancel, C-Axis cancel (Type VII)	Turns the back spindle indexing and C-Axis off.
M80	Front/back tool spindle forward rotation (type III,VII)	Starts forward rotation of the front/back tool spindle. (Optional)
M81	Front/back tool spindle reverse rotation (type III,VII)	Starts reverse rotation of the front/back tool spindle. (Optional)
M82	Front/back tool spindle stop (type III,VII)	Stops the front/back tool spindle. (Optional)
M84	Cutting start interlock enabled	Cutting is not starting until spindle speed reaches the specified value.
M85	Cutting start interlock disabled	Starts the cutting regardless of spindle speed.
M86	Back spindle speed fluctuation detect ON (Type VII)	Enables abnormal rotation detection of the back spindle.
M87	Back spindle speed fluctuation detection OFF (Type VII)	Disables abnormal rotation detection of the back spindle.
M88	Back spindle torque limit L selection (Type VII)	Sets the back spindle torque to 25%.
M89	Back spindle torque limit H selection (Type VII)	Sets the back spindle torque to 50%.
M90	Back spindle torque limit OFF (Type VII)	Sets the back spindle torque to 100%.
M92	Error detect ON	Turns error detection on (for higher precision in edge processing).
M93	Error detect OFF	Turns error detection off.
M96	Main spindle speed fluctuation detection ON	Enables abnomal rotation detection of the main spindle.
M97	Main spindle speed fluctuation detection OFF	Disables abnomal rotation detection of the main spindle.
M98	Call sub-program	Calls a subprogram.
M99	Return to main program	Be sure to insert this code at the ends of the main program and subprograms.
M100	Grip bar stock	Clamps a material on the bar loader. (Optional)
M101	Release bar stock	Releases a material on the bar loader. (Optional)

M Code	Function	Description
M102	Release bar stock quickly	Quickly releases a material on the bar loader. (Optional)
M103	Release bar stock, slide channel forward.	Release a material on the bar loader and move the rail backward. (Optional)
M104	Slide channel forward	Moves the bar loader slide channel forward. (Optional)
M105	Slide channel backward	Moves the bar loader slide channel backward. (Optional)
M106	Stabilizer close	Closes the bar loader stabilizer. (Optional)
M107	Stabilizer open	Open the bar loader stabilizer. (Optional)
M112	Guide bushing phase memory	Automatically controlled by G899 macro.
M117	Back spindle phase memory (Type VII)	Automatically controlled by G899 macro.
M118	Interference check disabled	Disables interference check.
M119	Interference check enabled	Enables interference check.
M125	NC reset and rewind	Used as a macro command. (Not used, usually.)
M126	Unconditional optional stop	Used as a macro command. (Not used, usually.)
M127	Request for recalculation	Used as a macro command. (Not used, usually.)
M140	Opposite tool post advance (Type III,VII)	Can be used only in the G821 command.
M141	Opposite tool post retract (Type III,VII)	Can be used only in the G821 command.

Notes

- The M codes marked an asterisk (*) used macros.
- Specify these macro M codes alone. Do not specify these codes along with any other code in the same block.

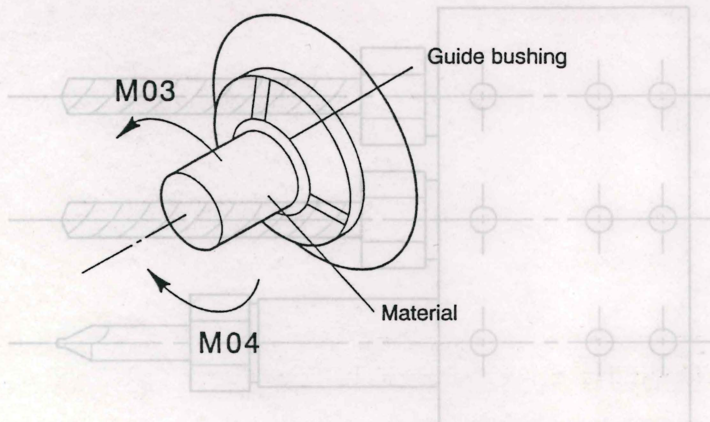
4.6.2 Main Spindle Rotation and Stop (M03, M04, and M05)

These codes are used for rotating (forward and reverse) and stopping of the main spindle.
The forward rotation in this case means the counterclockwise rotation toward the spindle.
To stop the rotation of the spindle, you can also use the Spindle stop key on the operation panel.

Command format

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

The spindle speed changes when only "S1 = ☐" is specified during the rotation of the main spindle.



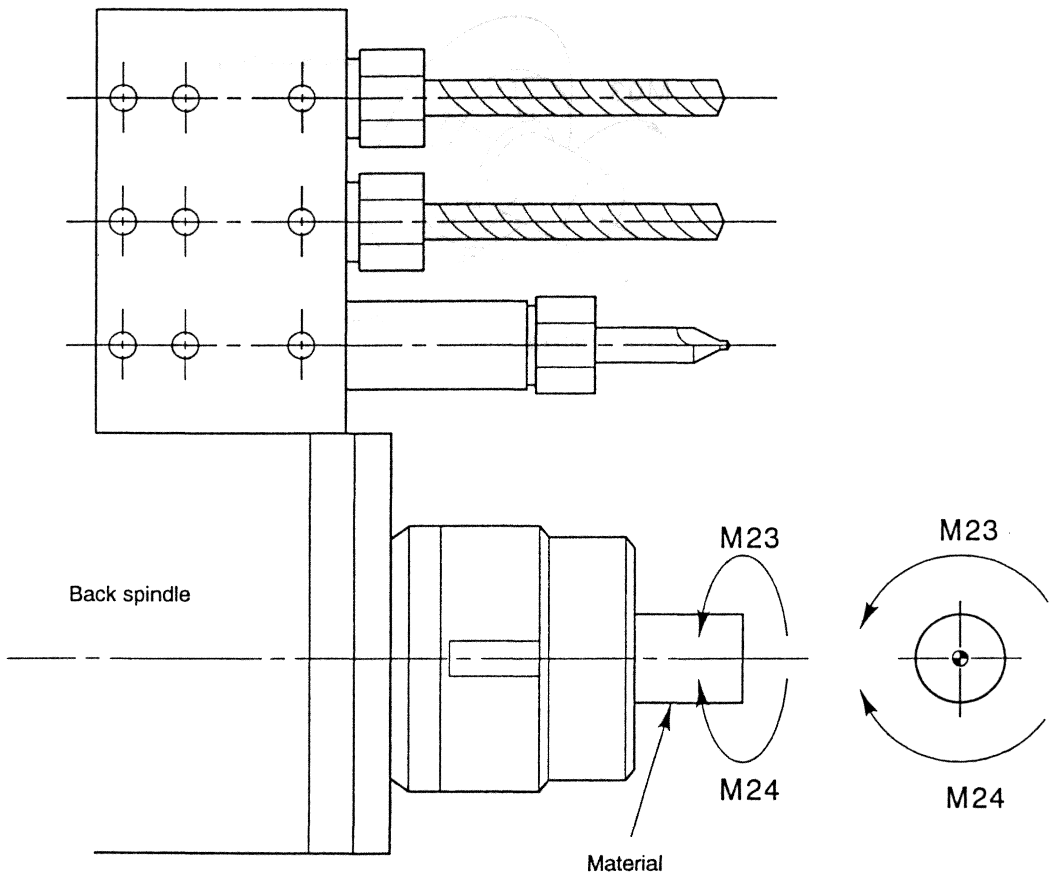
4.6.3 Back Spindle Rotation and Stop (M23, M24, and M25) (Type VII)

These codes are used for rotating (forward and reverse) and stopping of the back spindle.
The forward rotation in this case means the counterclockwise rotation toward the back spindle.
To stop the rotation of the back spindle, you can also use the Back Spindle stop key on the operation panel.

Command format

M23	Starts the back spindle forward rotation.
M24	Starts the back spindle reverse rotation.
M25	Stops the back spindle rotation.

The spindle speed changes when only an "S2 = ☐" is specified during the rotation of the back spindle.



Note

- A workpiece receiver box may have been mounted when the back spindle chuck is opened. Therefore, the back spindle speed is limited to 100 min^{-1} (maximum). The back spindle rotates only at 100 min^{-1} even if M23S2 = 2000 is specified.

  If any of tool selecting commands of T3000, T3100 to T3300 has been specified, however, the spindle speed limit is released, regarding that the workpiece receiver box has been rarely mounted.

  To enable the limitation again, press the reset [RST] key or specify the M2 code.

4.6.4 Tool Spindle Rotation and Stop (M58, M59, and M60)

These codes are used for rotating (forward and reverse) and stopping of the tool spindle in a secondary process.

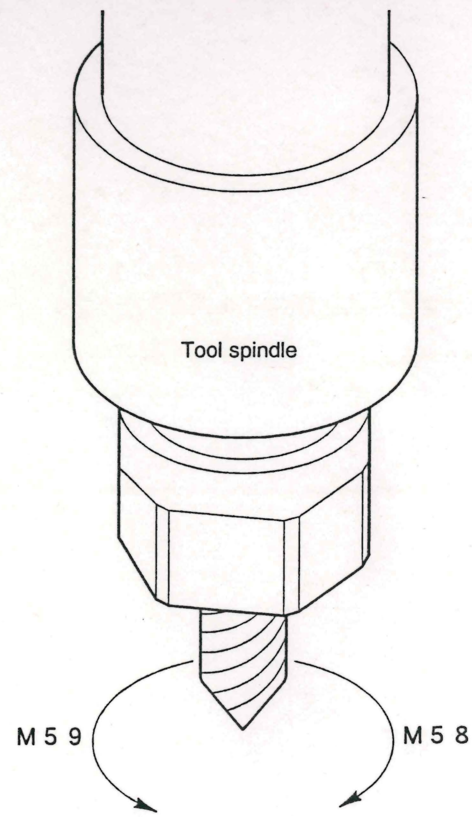
The forward rotation in this case means the counterclockwise rotation toward the tool spindle.

To stop the rotation of the tool spindle, you can also use the Spindle stop key on the operation panel.

Command format

M58	Starts the tool spindle forward rotation.
M59	Starts the tool spindle reverse rotation.
M60	Stops the tool spindle rotation

The spindle speed changes when only an "S3 = ☐" is specified during the rotation of the tool spindle.



4.6.5 Front/Back Tool Spindle Rotation and Stop (M80, M81, M82) (Type III, VII) (Optional)

These codes are used for rotating (forward and reverse) and stopping of the front/back tool spindle during a secondary process or back machining.

The forward rotation in this case means the counterclockwise rotation toward the front/back tool spindle.

To stop the rotation of the front/back tool spindle, you can also use the Spindle stop key on the operation panel.

Command format

M80	Starts forward rotation of the front/back tool spindle.
M81	Starts reverse rotation of the front/back tool spindle.
M82	Stops rotation of the front/back tool spindle.

Specifying only "S4=☐" during rotation of the front/back tool spindle changes the spindle speed.

4.6.6 Back Spindle Product Separation (M32, M33, M34) (Type VII)

This machine provides the following two methods for collecting the product.

1. Picking off the product by the back spindle to collect it in the product box. (Pick-off collection mode)
2. Putting the product in the workpiece receiver box mounted on the back spindle to collect it in the product box. (Workpiece receiver box collection mode)

For a back machining product, the product is collected in the method 1.

For a simple product without back machining, the product is collected using the method 2.

The program of method 1 is included in the program of $2 ($1 if the program is for a single axis control group) and the program of methods 2 is included in the program of $1. In both methods 1 and 2, the M code for product collection is M34 (M33). However, the actual operations of method 1 and 2 differ slightly. In the machine, methods 1 and 2 are identified as described below.

- When the back spindle defined as machining data has a workpiece receiver box and the M34 code has been specified for $1 or $2:

  The machine recognizes the current collection mode as workpiece receiver box collection mode and operates in that mode.

- When the back spindle defined as machine data is a standard or support-provided type and the M34 code has been specified for $1 or $2:

  The machine recognizes the current collection mode as pick-off collection mode and operates in that mode.

Note

Even if the workpiece receiver box is attached to the back spindle, the pick-off collection mode is recognized and used as the collection method during operation when the M34 code is specified after T3000, T3100 to T3300, G811, or G851. The pick-off collection mode is canceled by pressing the Reset button or by specifying the M2 code.

When the back spindle chuck has been opened, a workpiece receiver box may have been mounted. The back spindle speed is therefore limited to a maximum of 100 min^{-1}. Even if M23S2 = 2000 is specified, for example, the back spindle rotates only at up to 100 min^{-1}.
If any of tool selecting commands of T3000, T3100 to T3300 has been specified, however, the spindle speed limit is released, regarding that the workpiece receiver box has been rarely mounted.

To enable the limitation again, press the reset RST key or specify the M2 code.

1. Method for picking off a workpiece by the back spindle and collecting it in a product box (pick-off collection mode)

a) Back spindle product separation 1

Command format

M33 Back spindle product separation 1 A series of operations without product knock-out.

Command format

M33 W0.0 D20000 X4.0 E20000

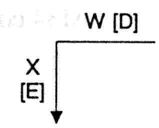
E: Feed rate (per minute) when the X2 axis separation position moves. Moves the X2 axis in rapid feed when omitted.

X: Move position (mm) when the X2 axis separation position moves. Moves the X2 axis to the zero point when omitted.

D: Feed rate (per minute) when the Z2 axis returns. Moves the Z2 axis in rapid feed when omitted.

W: Move position (distance from the zero point of the Z2 axis in mm) when the Z2 axis is retracted. Moves the Z2 axis to the zero point when omitted.

W☐ D☐ X☐ and E☐ can be omitted.



Program example

M33 .... A series of operations is performed for collecting the finished workpiece (without knock-out for the product).

- If M33 has no argument specified, the X1 and Z2 axes move to the zero point and terminates operation with the back spindle chucking the workpiece.
- After the X2 and Z2 axes movement is completed, the coordinate system setting of the Z2 axis is executed. Normally, the G50Z0 command is executed. If WΔΔ is specified, G50ZΔΔ is executed. Available with macro version 002-001 or later.
- Specify a program that puts the finished workpiece in the product box in the next line of the M33 code.

G00 Z32.5 .... Move the back spindle to the workpiece separation position in rapid feed.

M16 .... Open the back spindle chuck.

M10 .... Advance the back spindle knock-out bar.

G04 U0.5 .... Dwell.

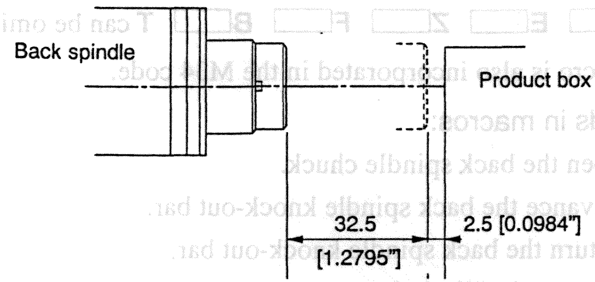
G00 Z0 .... Return to the zero point of the back spindle.

M72 .... Turn on the air blow of the back spindle.

M11 .... Return the back spindle knock-out bar.

T3000 .... Select the back spindle at the center of the guide bushing.

M73 .... Turn off the air blow of the back spindle.



Notes

- The above program example selects T3000 after product separation.
- Compared to M34 for back spindle product separation 2, M33 allows individual operations for product separation to be programmed. M33 can therefore save product separation time by efficient programming.

b) Back spindle product separation 2

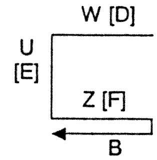
Command format

M34 Back spindle product separation 2
With knock-out for the product of a series of operations

Command format

M34 W0.0 D20000 U0.0 E20000 Z30.0 F20000 B0.0 T2.0

- T: Air blow time after product knock-out. Blows air for two seconds when omitted.
- B: Move position (mm) during the Z2 axis return after product knock-out. Moves the Z2 axis to the zero point when omitted.
- F: Feed rate (per minute) during the Z2 axis advance. Moves the Z2 axis in rapid feed when omitted.
- Z: Move position (mm) during the Z2 axis advance. Moves the Z2 axis to the position of 32.5 mm when omitted.
- E: Feed rate (per minute) when the X2 axis separation position moves. Moves the X2 axis in rapid feed when omitted.
- U: Move position when the X2 axis separation position moves.
- D: Feed rate (per minute) during the Z2 axis return. Moves the Z2 axis in rapid feed when omitted
- W: Move position (distance form the zero point of the Z2 axis in mm) during the Z2 axis return. Moves the Z2 axis to the zero point when omitted.



W□ **D**□ **U**□ **E**□ **Z**□ **F**□ **B**□ **T** can be omitted.

Product knock-out macro is also incorporated in the M34 code.

Actions of commands in macros:

M16 .... Open the back spindle chuck.
M10 .... Advance the back spindle knock-out bar.
M11 .... Return the back spindle knock-out bar.
M31 .... Turn on the Workpiece conveyor.
Z0 .... Return the back spindle to the zero point.
M72 .... Turn on the air blow of the back spindle.
M10 .... Advance the back spindle knock-out bar. Advance to blow chips.
G04 U2.0 ....Air blowing time. The argument is T2.0 seconds.
M11 .... Return the back spindle knock-out bar.
M73 .... Turns off the air blow of the back spindle.

For M34, the operation described above is completed by only the M34 code.

Program example

M34 .... Complete a series of operations for the workpiece collection and product knock-out.

T3000 .... Select the back spindle at the center of the guide bushing.

Note

- The position to which the Z2 axis advances during knock-out operation is normally set to Z32.5 unless the Z argument has been specified. The position can be changed by specifying the Z argument.

2. Method for putting a product in the workpiece receiver box mounted on the back spindle to collect it in the product box (workpiece receiver box collection mode)

The program is described as part of the $1 program.

Command format

M32 Workpiece separator (workpiece receiver box) advance operation

Command format

M34 Back spindle (workpiece receiver box) product separation operations

a) Workpiece separator (workpiece receiver box) advance operation

Command format

M32 X□ W0.0 U632.0 Z145.0 R□ F20000

F: Feed rate (per minute) during Z2 axis advance. Feeds rapidly when omitted.

R: Set "1" when a shift tool is used for the cut-off tool. Set nothing when the shift tool is not used.
The Z2 axis advance position is 130 mm when "1" is set. It is 145 mm when omitted.

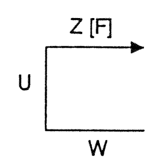
Z: Move position (mm) during Z2 axis advance.
Moves to the position of 145 mm when omitted.
An R1 command is ignored when this is specified.

U: Move position during X2 axis advance. Moves the X2 axis to the position of 632 mm when omitted.

W: Move position (distance from the zero point of the Z2 axis in mm) during Z2-axis return. Moves to the zero point when omitted.

X: End position queuing X position. The workpiece separator (workpiece receiver box) on the Z2 axis finishes advancing when the X axis reaches the specified position during cut-off machining. End position queuing is not performed when not specified.

X□ W□ U□ Z□ R□ F□ can be omitted.



Program example

$1	$2		
G821	G821	.... Single axis control group command ON	(Note 1)
T0100		.... Selects the cut-off tool.	
M32		.... Position the workpiece receiver box to the cut-off position.	
G1 X-3.0 F0.02		.... Cut-off machining	
M141		.... Return the workpiece receiver box.	
G820	G820	.... Single axis control group command OFF	(Note 1)

(Note 1) Omitting these codes does not affect the program.

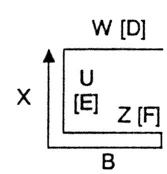
b) Back spindle (workpiece receiver box) product separation operation

Command format

M34 X632.0 W0.0 D20000 U0.0 E20000 Z54.0 F18000 T2.0 S30 B0.

- B0.: Move position during Z2 axis return after product carry-out. Moves the Z2 axis to the zero point when omitted.
- S30: Back spindle speed during product carry-out. Rotates at 30 min^{-1} when omitted.
- T2.0: Back spindle rotation time during product carry-out. Rotates for two seconds when omitted.
- F18000: Feed rate (per minute) during Z2 axis advance. Moves the Z2 axis in rapid feed when omitted (mm/min).
- Z54.0: Move position during Z2 axis advance. Moves the Z2 axis to the position of 54 mm when omitted.
- E20000: Feed rate (per minute) when the X2 axis separation position moves. Moves the X2 axis in rapid feed when omitted.
- U0.0: Move position (mm) when the X2 axis separation position moves. Move position during X2 axis return. Moves the X2 axis to the zero point when omitted.
- D20000: Feed rate (per minute) during Z2 axis return. Moves the Z2 axis in rapid feed when omitted.
- W0.0: Move position (distance from the zero point of the Z2 axis in mm) during Z2 axis return. Moves to the zero point when omitted.
- X632.0: Move position (mm) during X2 axis rise after workpiece separation. Moves the X2 axis to the position of 632 mm when omitted.

X☐ W☐ D☐ U☐ E☐ Z☐ F☐
T☐ S☐ B☐ can be omitted.



Program example

$1	$2	
G821	G821	.... Single axis control group command ON (Note 1)
T0200		
M34		.... Enter M34 after specifying the T0200 (first gang tool selection) after three-spindle hole machining.
:		
T0100		.... Select the cut-off tool.
M32		.... Position the workpiece receiver box at the cut-off position.
G1 X-3.0 F0.02		.... Cut-off machining
M141		.... Return the workpiece receiver box.
G820	G820	.... Single axis control group command OFF (Note 1)

Note

- If a command in the T3000s in the workpiece receiver box collection mode, the current mode is recognized as the pick-off collection mode and thus the basket in the workpiece receiver box will not swing, disabling product separation.

(Note 1) Omitting these codes does not affect the program.

4.6.7 Spindle Chuck Open and Close (M06, M07)

These codes open and close the chuck built in the spindle.
Using the codes can allows the chuck to clamp the material to transfer the spindle rotation to the material.

Command format

Spindle chuck

M06 Chuck close.

M07 Chuck open.

Actions of commands in macros:

M06

Closes the spindle chuck.

Starts synchronization control between the Z1 axis and the A7 axis (bar loader). (The command works this way for a synchronous bar loader.)

M07

Checks the spindle status and opens the spindle chuck when the spindle speed is lowered to the specified speed or less.

The specified spindle speed is set by the parameter. (standard value 2000 min^{-1})

Stops synchronization control between the Z1 axis and the A7 axis (bar loader) and starts constant torque control. The torque value can be set on the bar loader setup screen. (The command works this way for a synchronous bar loader.)

4.6.8 Back Chuck Open and Close (M15, M16) (Type VII)

These codes open and close the chuck built in the back spindle.

Using the codes allows the chuck to clamp the material to transfer the back spindle rotation to the material.

Command format

Back spindle chuck

M15 Chuck close

M16 Check open

Actions of commands in macros:

M15

Closes the back spindle chuck.

M16

Opens the back spindle chuck.

4.6.9 Opposite Tool Post Advance and Return (M140 and M141) (Type III, VII)

These codes can advance and return the inner diameter tool (T21ΔΔ to T25ΔΔ) of the opposite tool post. The M141 code can also be used to return the workpiece separator in the workpiece receiver box collection mode.

Command format

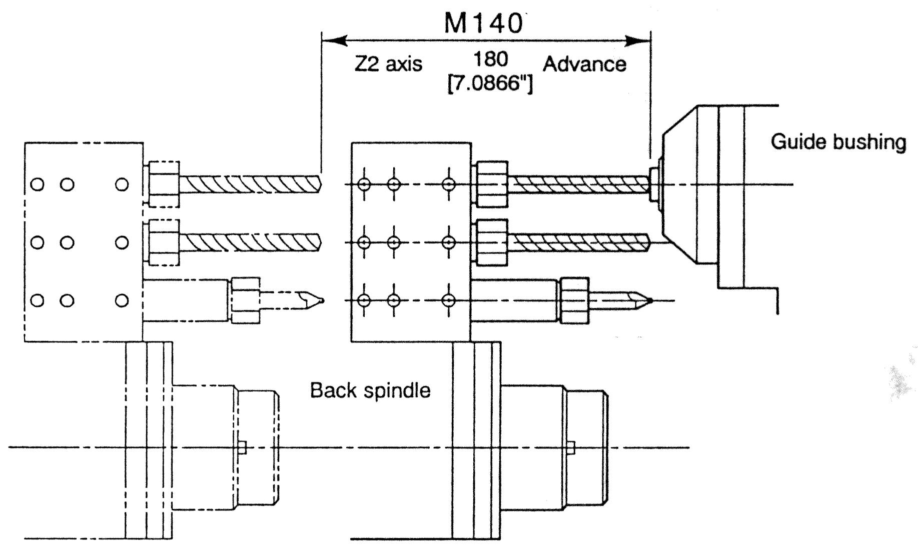
M140 Z□ Opposite tool post advance

M141 Z□ Opposite tool post return

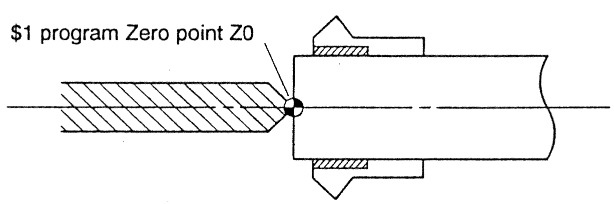
Z□ Z can be omitted.

These codes are enabled for axis control group 1 when a T21ΔΔ to T25ΔΔ command is executed.

The tool moves to the position of the Z2-axis machine coordinates (Z180.0) by the M140 code when a Z□ code is not specified.



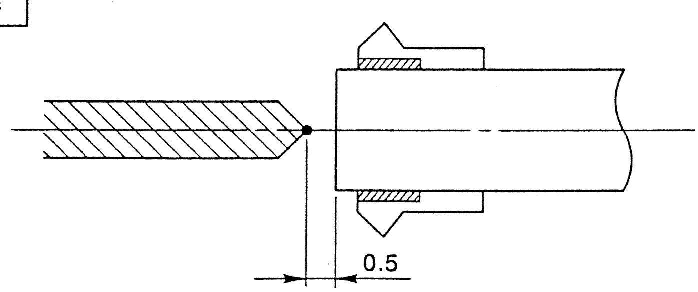
The tool set to the standard position is positioned as shown below by this code.



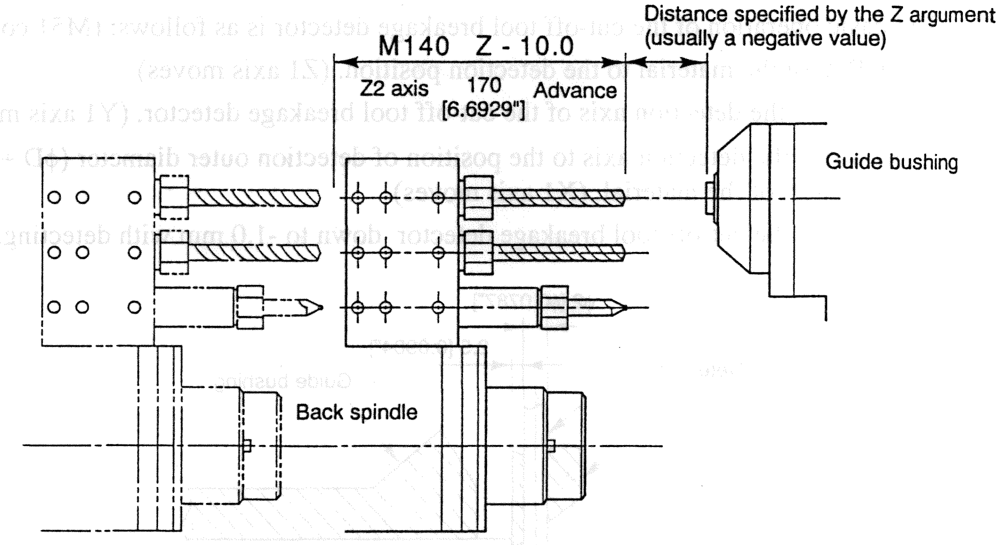
As shown in the figure above, the drill that was set without being shifted strikes against the material end face of Z = 0. Therefore, the end face is moved to Z = -0.5 so as to advance the tool as shown below.

Program example

($1)
G821
T2200
Z-0.5
M140



The M140 Z▭ command positions the opposite tool post on the Z2 axis in the $1 work coordinate system.



The tool returns to the machine zero point of the Z2 axis by the M141 code.

Notes

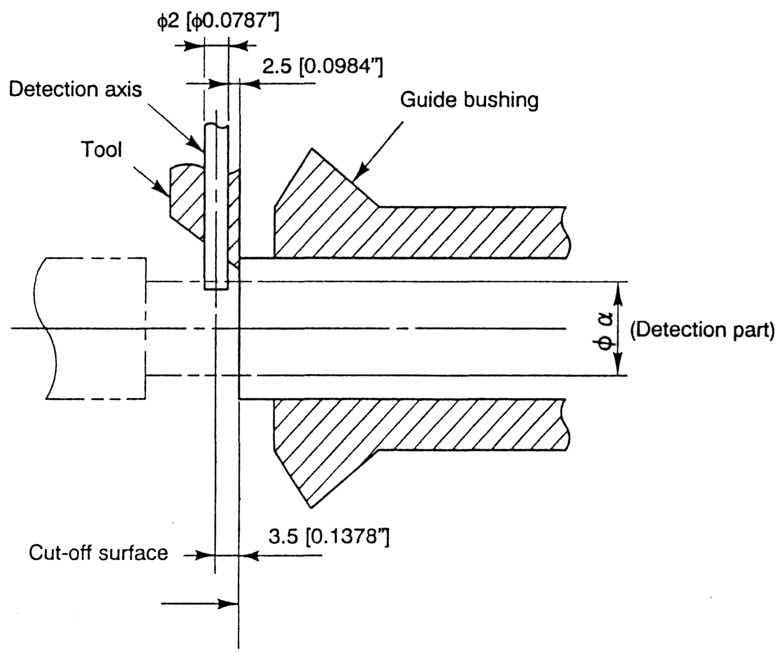
- Z▭ positioning is performed at the standard set position of the opposite tool post. Therefore, the positioning point of shifted tool must be set in axis control group 1 again by the G50 code.
- If T21ΔΔ to T25ΔΔ code is specified after M140 code, the tool selection performed without retracting the opposite tool post. Pay attention to the interference with the material.
- If M140 code is specified after T31ΔΔ to T35ΔΔ code, the M140 code causes no operation to be performed (resulting in an alarm).
- The M141 code can also be used to return the workpiece separator advanced by the M32 command (in the workpiece receiver box collection mode).

4.6.10 Y90Z Cut-Off Tool Breakage Detector (M51) (Optional)

(1) Detection Method

The basic operation of the cut-off tool breakage detector is as follows: (M51 code)

1) Return the material to the detection position. (Z1 axis moves)
2) Select the detection axis of the cut-off tool breakage detector. (Y1 axis moves)
3) Move the detection axis to the position of detection outer diameter (ϕD + 1 (diameter value)) of the material. (X1 axis moves)
4) Move the cut-off tool breakage detector down to -1.0 mm with detecting.



(2) Program example

The example given below assumes that the standard tool holder GTF3213 or GTF3113 is used.

T[a]00 ; ....①

M51 Z[ ] X[ ] F[ ] ; ....②

① Select the cut-off tool.

(a = 01 when the holder is GTF3213 or GTF3113)

② Execute a series of operations for detection.

- The Z argument specifies the amount of move for returning the material. The command does not return the material if this argument is omitted.
- The X argument specifies the position to which the detection axis is fed. The X axis moves to the point of -1.0 if this argument is omitted.
- The F argument specifies the feed rate (per minute) of the detection axis from the positioning point. The detection axis moves at 2000 mm/minute if this argument is omitted.

4.6.11 Knock-out (M10, M11) (Type VII)

These codes advance and return the back spindle knock-out

Command format

Knock-out

M10 Advance the knock-out.

M11 Return the knock-out.

Command format

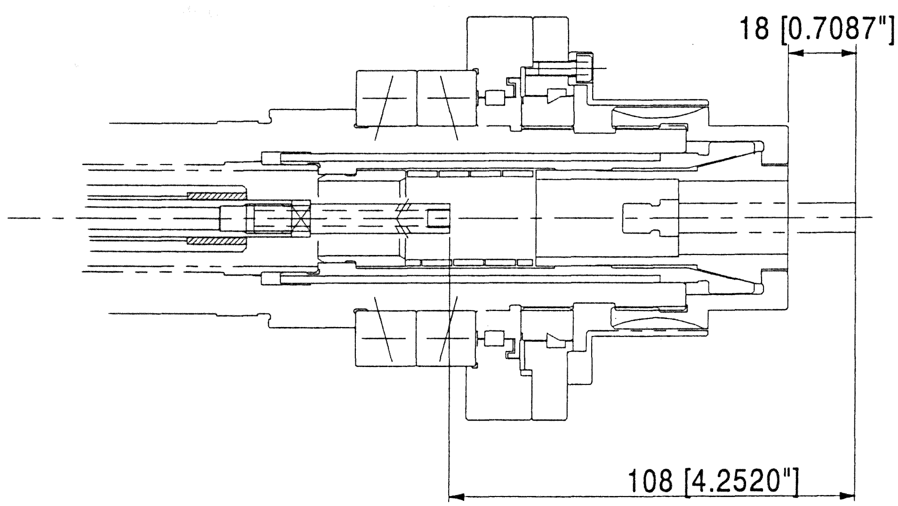
M10 Z☐ F☐ T☐

M11 Z☐ F☐ T☐

- The Z argument specifies the knock-out axis move position (distance). The knock-out moves 108 mm when the argument is omitted.
- The F argument specifies the knock-out axis feed rate (per minute). The knock-out moves in rapid feed when the argument is omitted. The rapid feed rate is 15000 (mm/min).
- The T argument specifies the knock-out axis torque. Specify the ratio (%) to the maximum torque. It is 200% when the argument is omitted.

Note

- If the back chuck is closed or the back spindle is rotating, the command causes no operation to be performed (resulting in an alarm).



4.6.12 U10C Support (M10, M11) (Type I)

These codes advance and return the support.

Command format

Support

M10 Advance the support.

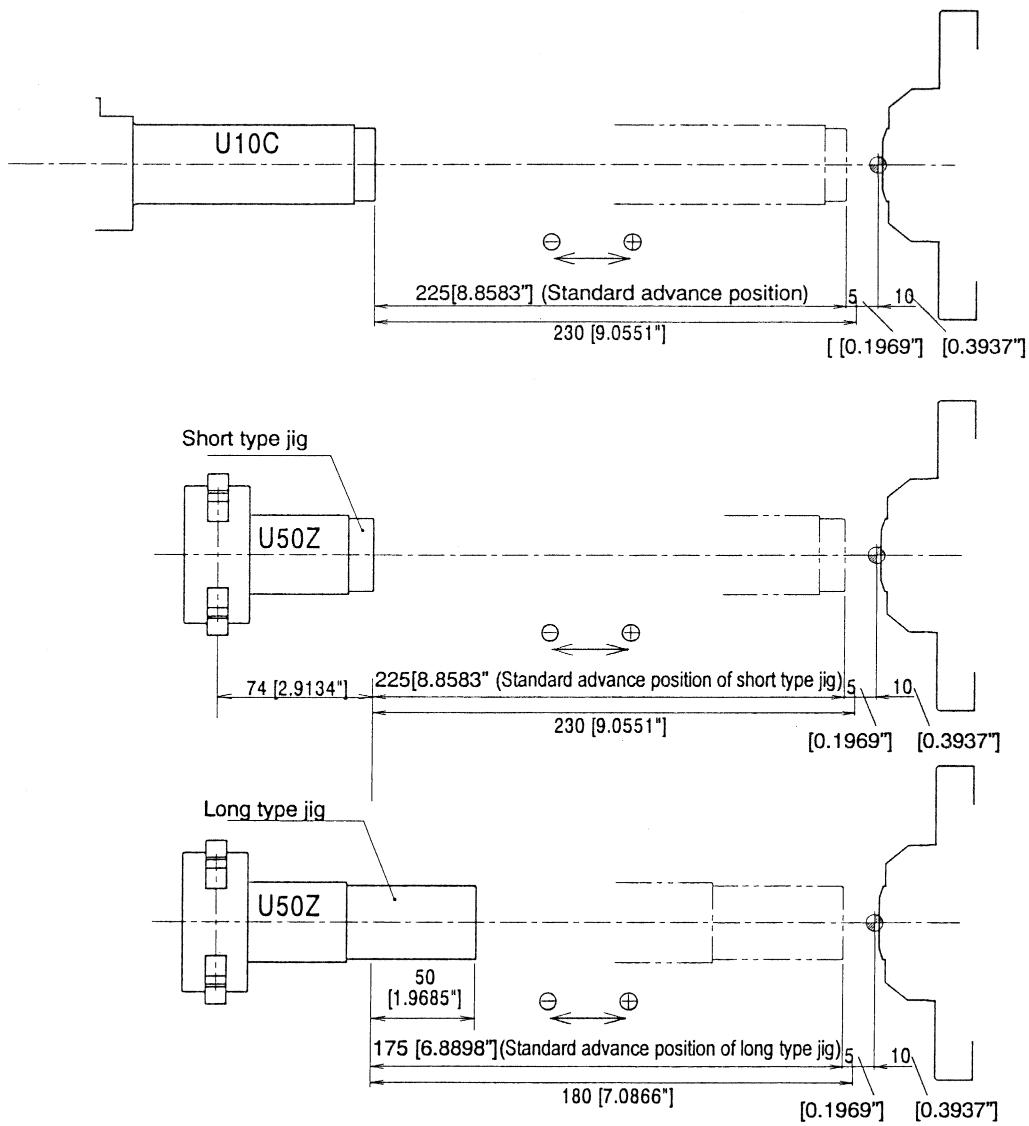
M11 Return the support.

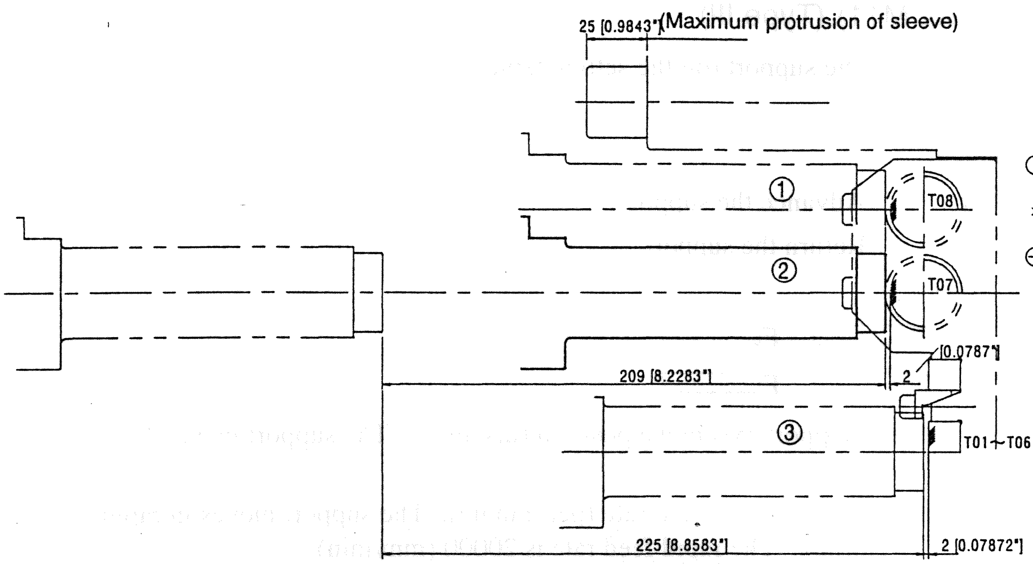
Command format

M10 Z□ F□

M11 Z□ F□

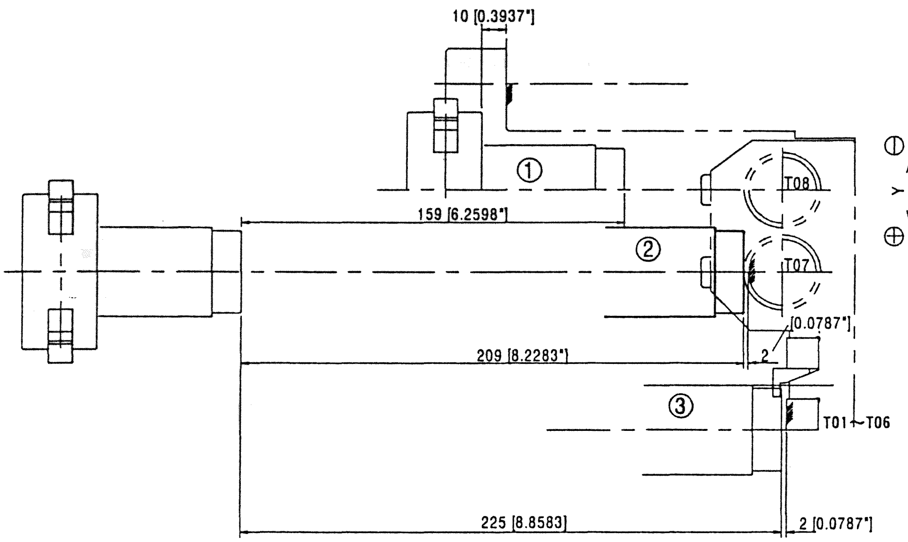
- The Z argument specifies the support axis move position (distance). The support moves 225 mm (175 mm if it is a long type) when the argument is omitted.
- The F argument specifies the support axis feed rate (per minute). The support moves in rapid feed when the argument is omitted. The rapid feed rate is 15000 (mm/min).





U10C

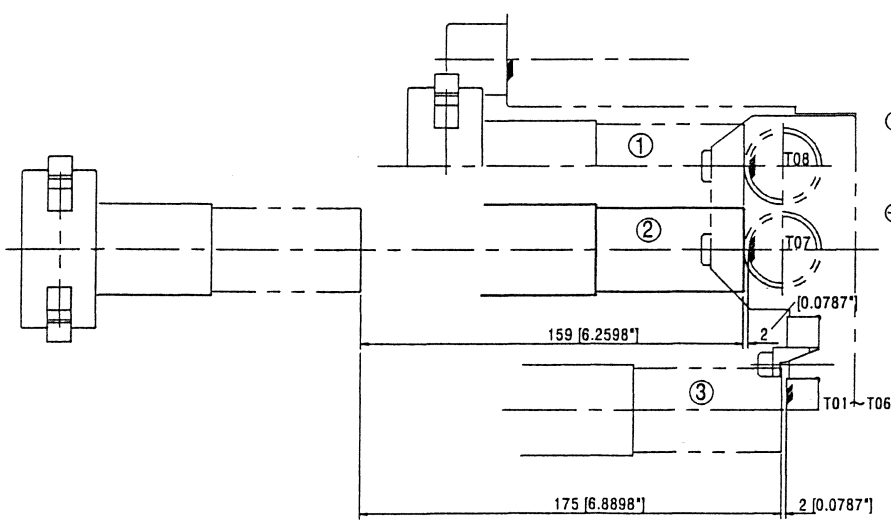
1. If the support advances exceeding Z = 209 at position T08, it may interfere with the rotary tool nut. From the advance position, the Y axis cannot move in the positive direction.
2. If the support advances exceeding Z = 209 at position T07, it may interfere with the rotary tool nut. From the advance position, the Y axis can move 35 mm (70 mm specified in diameter) in the positive direction.
3. If the support advances exceeding Z = 225 (U10C standard advance position) between positions T01 and T06, it may interfere with the tool.



U51Z

(Specifications for short type jig)

1. If the support advances exceeding Z = 159 at position T08, it may interfere with the 3-vertical sleeve holder (GDF306). From the advance position, the Y axis cannot move in the positive direction.
2. If the support advances exceeding Z = 209 at position T07, it may interfere with the rotary tool nut. From the advance position, the Y axis can move 15 mm (30 mm specified in diameter) in the positive direction.
3. If the support advances exceeding Z = 225 (U51Z standard advance position of short type jig) between positions T01 and T06, it may interfere with the tool.



U51Z

(Specifications for long type jig)

1. If the support advances exceeding Z = 159 at position T08, it may interfere with the rotary tool nut. From the advance position, the Y axis cannot move in the positive direction.
2. If the support advances exceeding Z = 159 at position T07, it may interfere with the rotary tool nut. From the advance position, the Y axis can move 35 mm (70 mm specified in diameter) in the positive direction.
3. If the support advances exceeding Z = 175 (U51Z standard advance position of long type jig) between positions T01 and T06, it may interfere with the tool.

4.6.13 U11C Support (M10, M11) (Type III)

These codes advance and return the support (on the setting table).

Command format

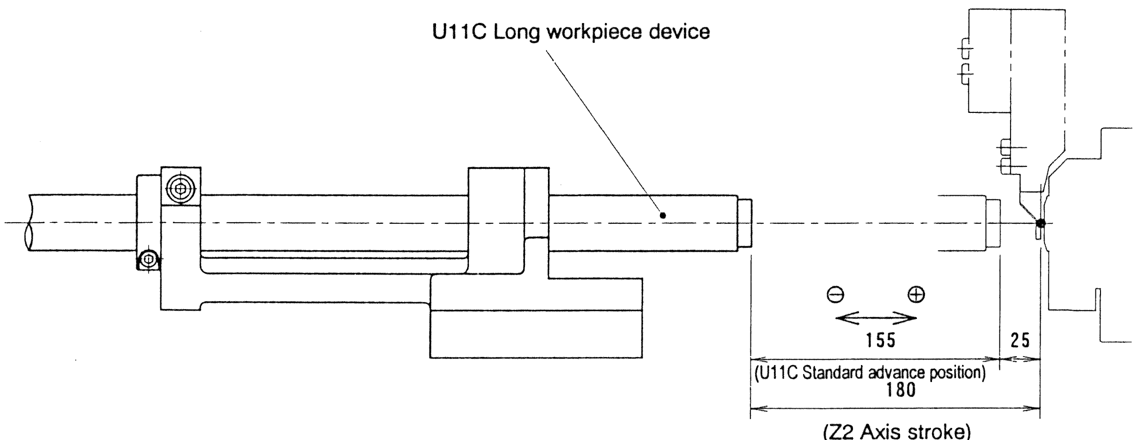
Support	
M10	Advance the support.
M11	Return the support.

Command format

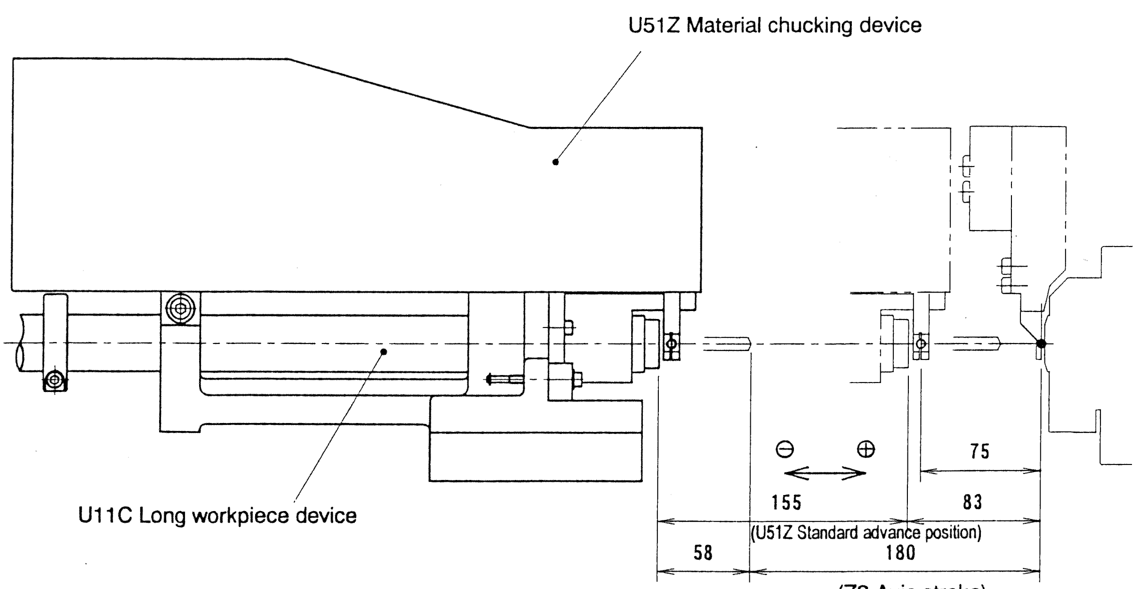
M10 Z☐ F☐

M11 Z☐ F☐

- The Z argument specifies the support axis move position (distance). The support moves 155 mm when the argument is omitted.
- The F argument specifies the support axis feed rate (per minute). The support moves in rapid feed when the argument is omitted. The rapid feed rate is 20000 (mm/min).
- The M10 command returns the Z2 axis once, moves the X2 axis to the support front position, then advances the Z2 axis.



Support and guide pipe protrude amount (U11C)



Support and guide pipe protrude amount (with U51Z)

4.6.14 U51Z Stock Gripper (M13, M14) (Type I, III)

These codes open and close the stock gripper.

Command format

Stock gripper

M13 Open the stock gripper.

M14 Close the stock gripper.

Command format

M13 R☐

M14

- The R argument specifies the position at which the stock gripper is chucked during adjustment. The stock gripper axis moves to the position specified by the R argument. The acceptable range is 0 to 7.32 mm (stroke on each side). Usually, specify 1.0 for round materials. For profile materials, specify the difference between the maximum and minimum diameters.

4.6.15 U31J Work Separator (M32, M33) (Type I, III)

These codes advance and return the work separator.

Command format

Work separator

M32 Advance the work separator.

M33 Return the work separator.

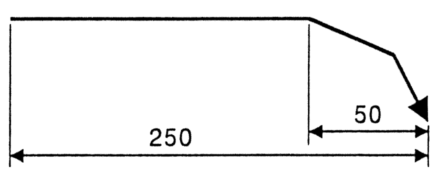
Command format

M32 X☐ Z☐

- The X argument specifies the end position queuing position. The work separator completes advancing when the X1 axis reaches the specified position during cut-off machining. End point queuing is not performed when the argument is omitted.
- The Z argument specifies the separator axis moving distance. The separator axis moves 250 mm when the argument is omitted.

M33 Z☐ W☐ F☐ E☐ K☐

- The Z argument specifies the rapid feed switching position. The separator axis moves to the specified position in rapid feed. The separator axis moves 50 mm in rapid feed when the argument is omitted.
- The W argument specifies the cutting feed switching position. The separator axis changes the speed at the specified position. The separator axis changes the speed at the 0 mm position when the argument is omitted.
- The F argument specifies feed rate 1. The separator axis moves to the cutting feed switching position at the specified feed rate. The feed rate is 5000 (mm/min) when the argument is omitted.
- The E argument specifies feed rate 2. The separator axis moves to the zero point at the specified feed rate. The feed rate is 5000 (mm/min) when the argument is omitted.
- The K argument specifies the number of repetitions. The separator axis repeats moving between the cutting feed switching position and the zero point for the specified number of times. The separator axis does not repeat it when the argument is omitted.



4.6.16 U33J Workpiece Separator (M32, M33) (Type III)

These codes advance and return the workpiece separator (on the setting table).

Command format

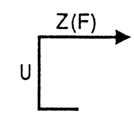
Workpiece separator

M32	Advance the workpiece separator.
M33	Return the workpiece separator.

Command format

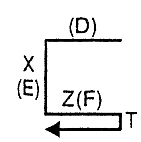
M32 X□ U□ Z□ F□

- The X argument specifies the end position queuing position. The work separator completes advancing when the X1 axis reaches the specified position during cut-off machining. End point queuing is not performed when the argument is omitted.
- The U argument specifies X2 axis advance distance. The X2 axis moves 632 mm when the argument is omitted.
- The Z argument specifies Z2 axis advance distance. The X2 axis moves 160 mm when the argument is omitted.



M33 D□ X□ E□ Z□ F□ T□

- The D argument specifies the Z2 axis return feed rate (per minute). The Z2 axis moves in rapid feed when the argument is omitted.
- The X argument specifies the X2 axis unloading position. The X2 axis moves to the zero point when the argument is omitted.
- The E argument specifies the X2 axis unloading feed rate (per minute). The X2 axis moves in rapid rate when the argument is omitted.
- The Z argument specifies the Z2 axis advance distance. The Z2 axis moves 80 mm when the argument is omitted.
- The F argument specifies the Z2 axis advance feed rate (per minute). The Z2 axis moves in rapid rate when the argument is omitted.
- The T argument specifies the product unloading stop time. Product unloading stops for two seconds when the argument is omitted.



4.6.17 Bar Feed (M54, M55)

These codes are used to operate the bar loader.

Command format

Material feed

M54 Turn off the bar loader torque.

M55 Start the bar loader.

- The bar loader performs the following operations:

Actions of M55 (Synchronous bar loader)

Open the stabilizer.		Extract a. remaining material
Move the pushrod to the material extract position.		
Turn on the material clamp.		
Move the pushrod to the material extract position (= return position).		
Turn off the material clamp.		
Turn on the material clamp.	Detect any material	
Turn off the material clamp.		
Turn on the shelf material motor to move the rail backward.		Feed a shelf material.
Move the pushrod to the primary feed position.		Insert a material.
Move the pushrod to the return position.		
Turn on the shelf material motor to move the rail forward.		
Turn on the material clamp.		
Move the pushrod to the material insert position.		
Turn off the material clamp.		
Move the pushrod to the shortcut position.		Feed a material.
Close the stabilizer.		

4.6.18 Bar Feed Program Enable/Terminate (M08, M09)

These codes enable and terminate a bar changing program. The commands in the block (= bar changing program) enclosed between M08 and M09 are executed when the material shortage signal is received. They are otherwise skipped. (A block skip symbol "/" is used.)

The bar changing program is inserted between the cut-off and end processes.

Command format

Bar feed program enable/terminate

M08 Enable a bar changing program.

M09 Terminate the bar changing program.

Program example

M08	.... Enable the bar changing program.
M08	
/ ( )	.... Program for removing burr from the outer diameter of the material.
/ ( )	....
/ ( )	....
/ G01 X21.0 W-25.0 F0.2	.... Move the cut-off tool by the material outer diameter + 1 mm upon extracting the material from the guide bushing.
/ M53	.... Turn off coolant supply.
/ M05	.... Stop rotating the main spindle.
/ M54	.... Stop the machining torque of the bar loader.
/ M07	.... Open the spindle chuck.
/ M55	.... Issue the material replace command (to start replacing the material on the bar loader).
/ M06	.... Close the spindle chuck.
/ G4 U2.0	.... Prevent bar loader torque switch time delay.
/ M52	.... Turn on coolant supply.
/ M03 S1 = ☐	.... Rotate the main spindle forward.
/ M26	.... Execute guide bushing phase adjustment (for profile material).
/ W25.0 F0.2	.... Insert a material in the guide bushing.
/ X-3.0 F0.02	.... Shortcut the tip of material.
/ M09	.... Terminate the bar changing program.

Note

- The M08 and M09 codes can be specified for both axis control groups.

4.7 S Functions (S Codes)

The S functions specified in the following format are also called rotation functions. These are used to specify the main spindle, back spindle, rotary tool spindle speeds.

Command format

S1 = □□□□	Main spindle	(Five-digit function command)
S2 = □□□□	Back spindle	(Four-digit function command)
S3 = □□□□	Gang tool rotary spindle	
S4 = □□□□	Front/back tool spindle	

The spindle speed is calculated from the following formula and rounded up to the nearest whole number.

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

N : Speed (min^{-1})

V : Cutting speed (m/min.)

D : Material diameter (mm), or hole diameter for drilling

π : Circular constant (approx. 3.14)

Specification Ranges

Axis control group	Spindle	Speed
S1	Main spindle	200 to 10000min^{-1}
S2	Back spindle	200 to 8000min^{-1}
S3	Gang tool rotary spindle	200 to 5000min^{-1}
S4	Front/back tool spindle	200 to 5000min^{-1}

4.8 !(Exclamation) Function (Queuing Code)

Command format

!L□ Multi-axis control group queuing)

→ Queuing number (0 to 9999)

The command format "!L□" allows queuing with other parts in axis control groups $1 and $2 that need queuing.
If the number is entered incorrectly, the program cannot advance in the queuing state.

Program example

Example of multi-axis control group queuing command

```
O□□□□
$1          $2
.           .
.           !L1
!L1         .
.           .
!L2         !L2
.           .
.           .
.           !L3
!L3         .
.           .
.           .
.           .
!L4         !L4
M56
G999        G999
N999        N999
M02         M02
M99         M99
%           %
```

Product code	C	–	L	5	1	6	2	0	I	III	VII	/	/	/	–	5	7	0

Document code	2	E	1	–	0	4	0	2
	2	E	2	–	0	4	0	1

Chapter 5 Programming Practice

5.1 Program Structure 5-1
5.1.1 Program Structure 5-3
5.1.2 Guidelines for Designing a Machining Program 5-4
5.2 Preparation Process 5-5
5.3 Cut-off and Ending Processes 5-7
5.4 Machining Processes 5-8
5.4.1 Front Turning Process 5-9
5.4.2 Back Turning Process 5-10
5.4.3 Grooving Process 5-11
5.5 Center Hole Drilling 5-12
5.5.1 Determining Spindle Speed 5-12
5.5.2 Determining Program Point, Cutting Point, and Center Point 5-12
5.5.3 Center Drilling 5-13
5.5.4 Drilling 5-14
5.5.5 Drilling a Deep Hole 5-15
5.5.6 Thread Cutting with Tap and Die 5-16
5.5.7 Tapping Process Example 5-17
5.6 Thread Cutting with the Threading Tool 5-18
5.6.1 Determining the Program Point and Cutting Point 5-18
5.6.2 G92 - Thread Cutting Canned Cycle 5-19
5.6.3 G92 - Thread Cutting 5-20
5.6.4 G92 - Taper Thread Cutting Canned Cycle 5-22
5.7 Boring (Coordinate System Shift) 5-23
5.7.1 Determining the Program Point and Cutting Point 5-23
5.7.2 Boring Tool Signs 5-24
5.7.3 Adjusting for Diameter Direction Coordinate Shift and Canceling the Coordinate Shift Amount 5-25
5.7.4 Adjusting for Longitudinal Direction Coordinate Shift and Canceling the Coordinate Shift Amount 5-26
5.7.5 Adjusting for Diameter and Longitudinal Direction Coordinate Shift and Canceling the Coordinate Shift Amount ... 5-27
5.8 Tool Offset (T Codes) 5-29
5.8.1 Compensation Example 5-30
5.8.1.1 Outer Diameter Cutting 5-30
5.8.1.2 Face Drilling (Drill, Tap Machining) 5-30
5.8.1.3 Cut-off 5-31

5.9 Simultaneous Machining 5-32
5.9.1 Simultaneous Machining by Outer Diameter Cutting and Drilling 5-35
5.9.2 Outer/Inner Diameter Simultaneous Machining (Boring) 5-36
5.10 Free Tool Layout Pattern (Holder Name = Free Tool) 5-37
5.11 Using Subprograms 5-38
5.11.1 A Subprogram Call Instruction 5-38
5.11.2 Calling a subprogram from the main program 5-38
5.11.3 Example of Using a Subprogram 5-39
5.12 Long Material Lathing (Optional) (Type I) 5-40

This section explains basic programming using the commands (codes) described in Chapter 4.

5.1 Program Structure

Command format

• Program No.	**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**
• Return to main program	**M99**	**M99**
• Stop code	**%**	**%**

- The program number is given on the first program line of a program to identify it and distinguish it from other programs.
  The program number consists of the letter "O" followed by a desired number of up to 8 digits between "00000001" and "99999999."
  In the L16/L20 machine, the system software macros use the program numbers between "8000" and "9999."
  Do not assign these numbers to your programs.
- A single program for the L16/L20 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.

  N□□□□□

  Tool number

  Process sequence

  To specify the first process using tool No. 3, 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.

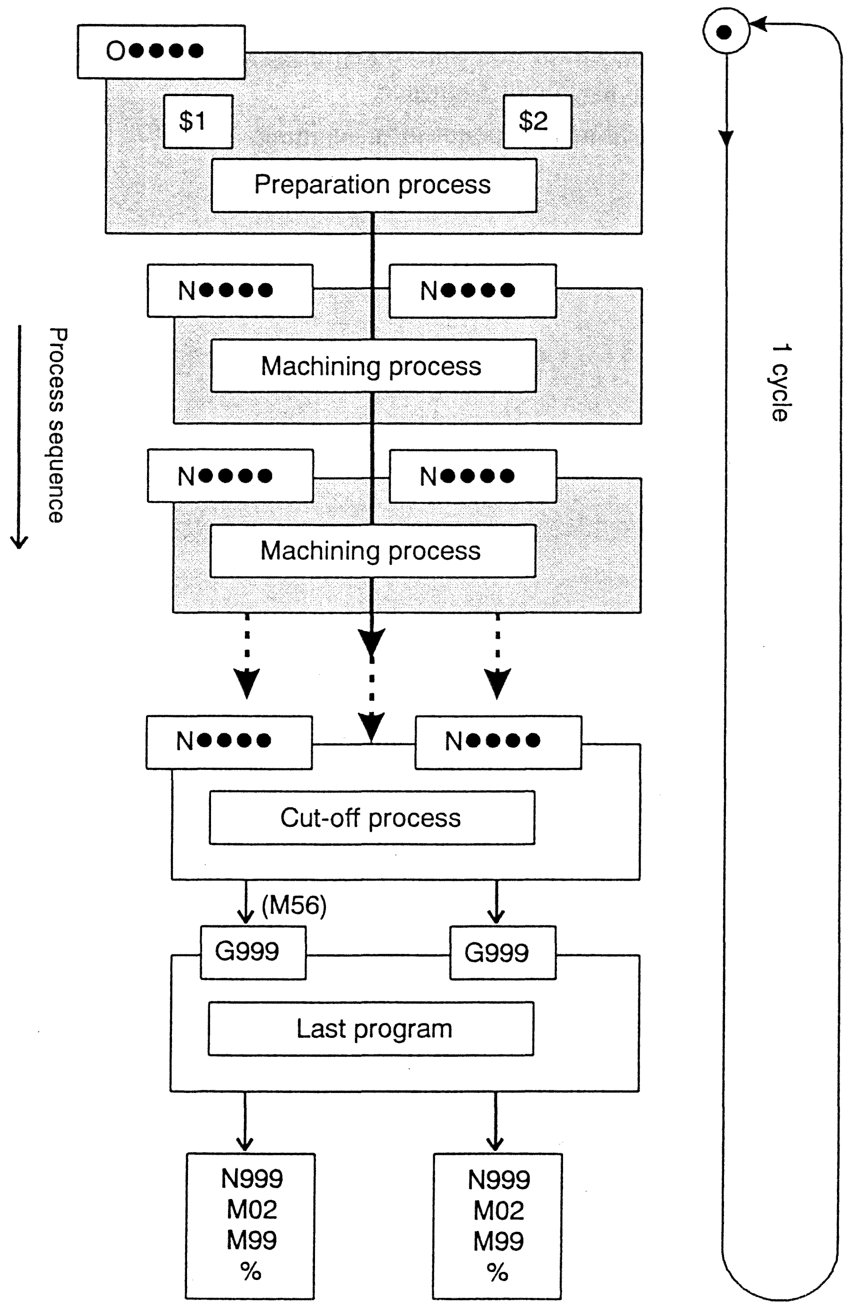
  Note that if the M56 command is included in both of the $1 and $2 programs, two products are counted in one cycle.

- 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), the return to main program command (M99), and the stop code (%).

  (Stop code is automatically entered after normal program entering.)

5.1.1 Program Structure

- 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.

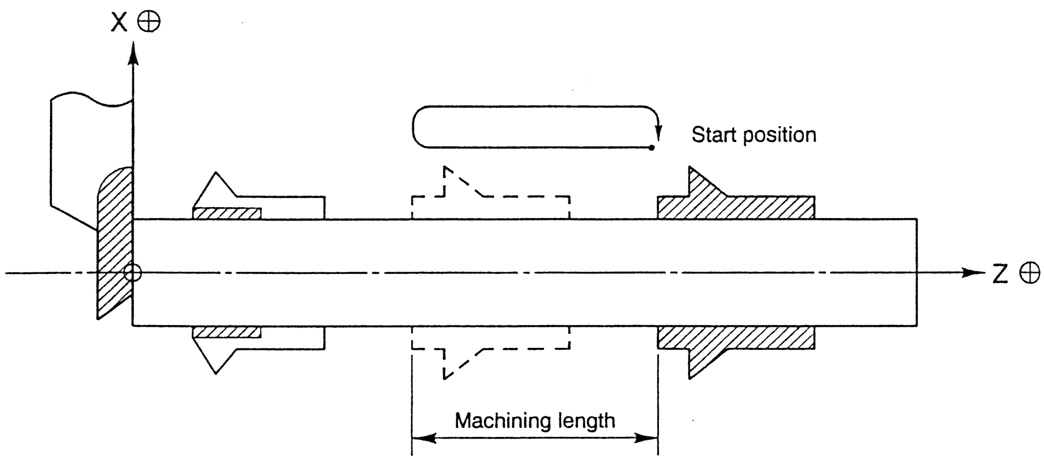


5.1.2 Guidelines for Designing a Machining Program

- 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.
- If your workpiece requires cutting operations in high precision and cutting with large in-feed amount, use the front turning tool as much as possible.
- 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 Process

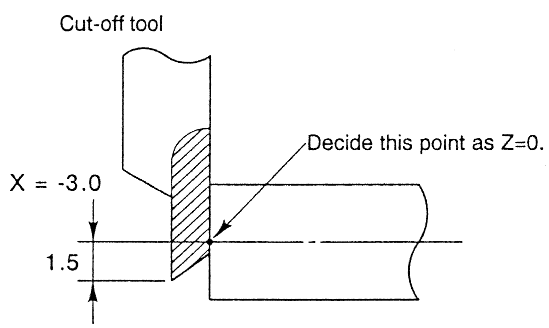
In the preparation process, you will set the required conditions for a cycle operation and start the operation.



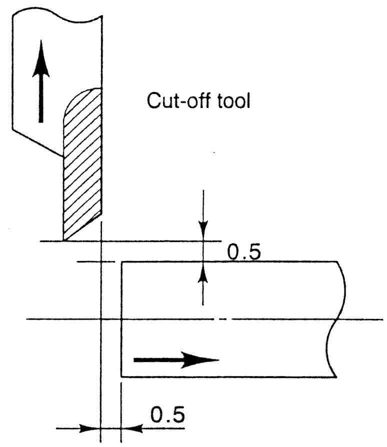
1. If a program is running, wait until the cut-off operation is completed and the Z axis returns to the machining start position.
2. Enter the program number at the beginning of a program.
3. Set the respective axis control group number ($1 or $2).
4. Set the machining conditions required for each axis control group.
   1) Set the feed rate.
      G98: Per minute feed (mm/min)
      G99: Per rotation feed (mm/rev)
   2) Set the reference point in the longitudinal direction.
   The longitudinal direction (along the Z axis) is usually Z0.
5. Start operation.
   1) Close the chuck.
   2) Move the material in the Z axis (longitudinal) direction -0.5 mm.
   3) Rotate the main spindle. (If back machining is required, rotate the back spindle also.)

Program example

G50 Z0



G0 X☐ Z-0.5



1. 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.)
2. 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 Z-axis position depends on the machining length set as machining data.)
3. After positioning the X and Z axes, bring the material in contact with the cut-off tool and start the program.
4. 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.)

Program example

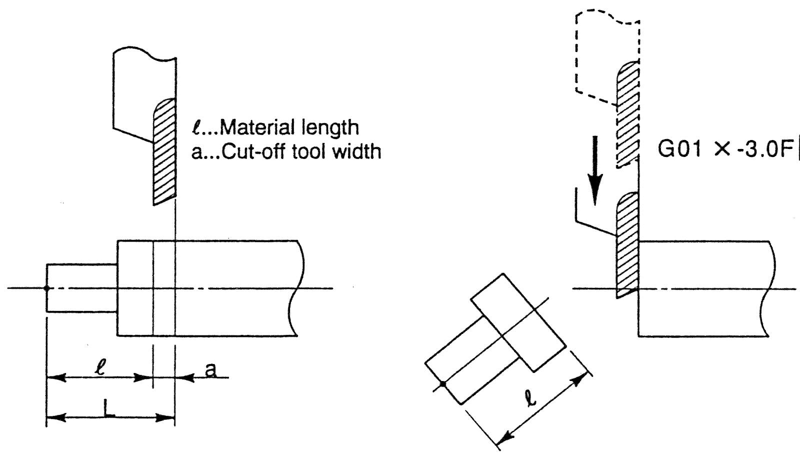
$1	$2	
O☐☐☐☐		.... Specify the program number.
$1	$2	.... Specify each axis control group number.
G50 Z☐		.... Set the coordinate system of the machining start position for each axis control group.
M06		.... Close the chuck.
G00 X☐ Z-0.5		.... Move the cut-off tool. Generally, specify -0.5 mm in the 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.
G821	**G821**	.... Turn on the single machining command.

↓ Machining process ↓

5.3 Cut-off and Ending Processes

Mount the cut-off tool on T01 and perform cut-off machining.

Machining



Program example

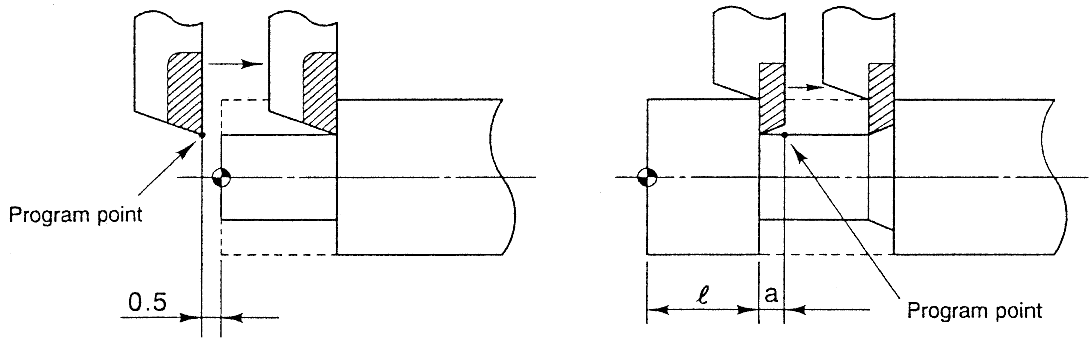
This program example does not contain product separation.
For details, see program examples Product Separation.

Machining process

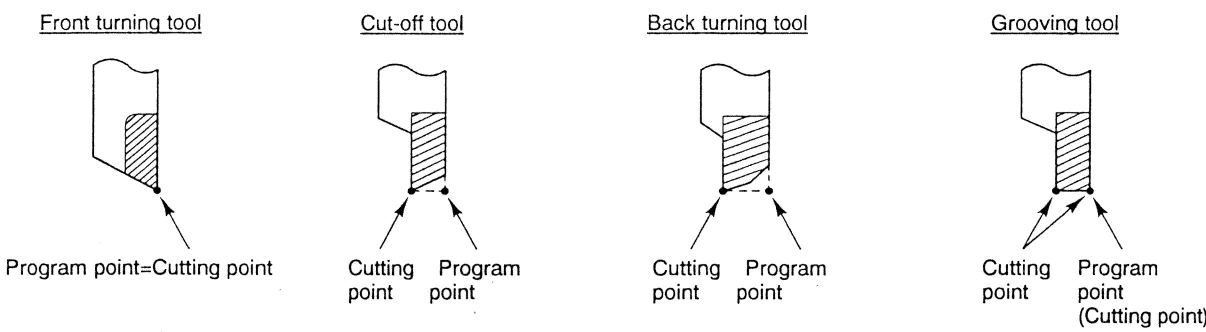
$1	$2	
N☐☐☐☐	**N☐☐☐☐**	.........Enter the sequence number.
T0100		.........Select T0100.
G00 X☐ Z☐ T☐		.........Move the tool to the material cut-off point (Total material length + cut-off tool width) and specify the number of tool offset.
G01 X-3.0 F☐		.........This command cuts off the material by moving the cut-off tool to the cut-off end position in the specified feed rate.
M05		.........M05 stops the rotation of the main spindle.
M07		.........M07 opens the chuck of the main spindle.
G00 Z☐ T00		.........Move the Z axis to the machining start position in rapid feed. (Cancel offset.)
M56		.........Count the number of products.
G999	**G999**	.........G999 is the last program execution command. The command between the axis control groups are queued automatically.
N999	**N999**	.........Be sure to enter N999 after the last program execution command.
M02	**M02**	.........Enter the 1 cycle stop command.
M99	**M99**	.........Enter the return to top of program command.
%	**%**	.........Enter the stop code % (end of program).

5.4 Machining Processes

Outer diameter cutting is performed in several processes such as the front turning, back turning, and grooving processes.



- Move the front turning tool to the position shown above. At this time, move the tool for the length of the machining distance.
- Move the back turning tool into position by adding the length of the tool width to the machining distance. The actual cutting point is on the left side of the program point. Therefore, the move distance will be short unless you specify the Z coordinate value which is the sum of "ℓ" (the machining distance) and "a" (the tool width). The program point in Z axis shall be the sum of "ℓ" and "a" in relation to the cutting point "ℓ" in the Z axis direction. (Z = ℓ + a)
- For also the grooving tool, pay attention to the tool width.

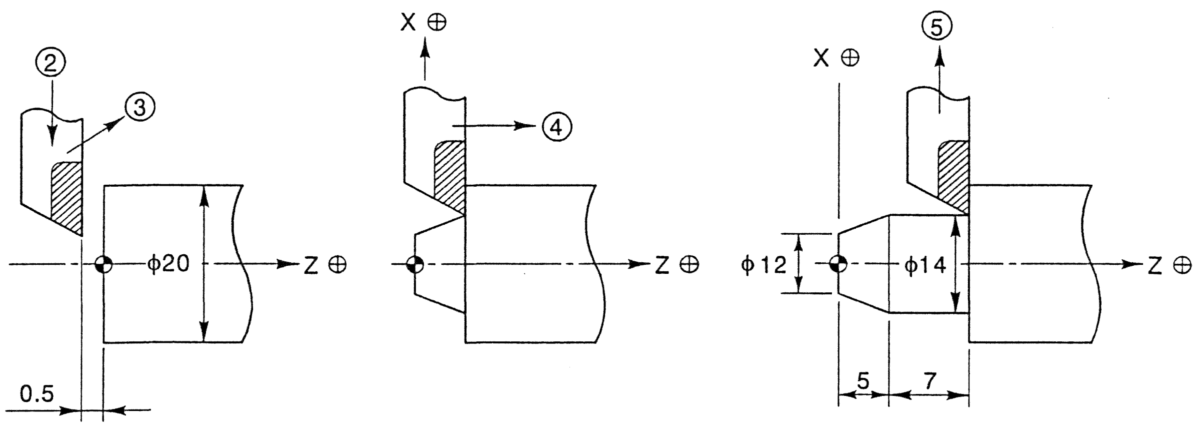


Notes

- To use the back turning or grooving tool, add the tool width to the longitudinal coordinate value. For your convenience, it is recommended to write down the width of a tool on the tool layout diagram.
- Be sure to move the tool to the positioning point at the end of the outer diameter cutting process.

5.4.1 Front Turning Process

Machining

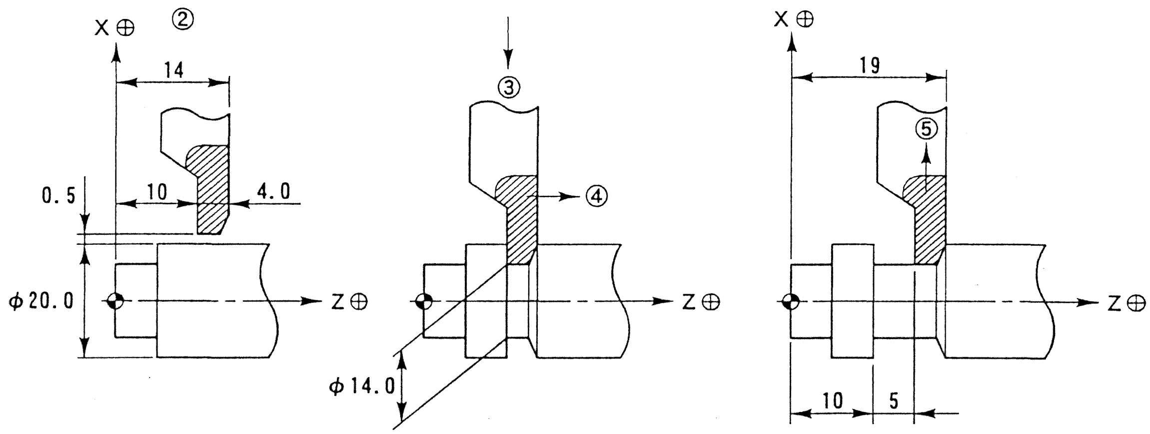


Program example

T□	.... Select tool for front turning T□.	
G00 X11.8 Z-0.5	.... Position the tool at the specified position.	②
G01 X14.0 Z5.0 F□	.... Perform taper cutting for 5.0 mm in the longitudinal direction.	③
Z12.0	.... Perform cutting for 12.0 mm in the longitudinal direction.	④
X21.0 F□	.... Move the tool to the positioning point by cutting feed.	⑤

5.4.2 Back Turning Process

Machining



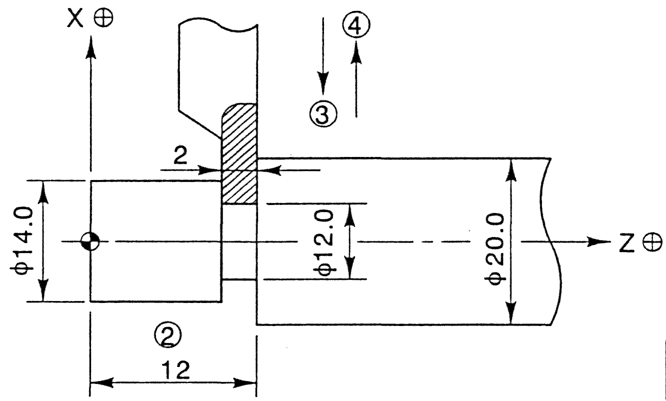
Program example

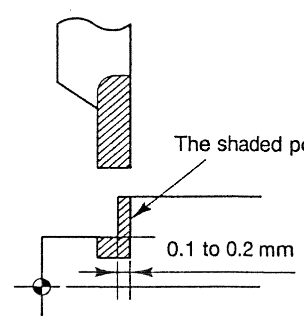
T☐	.... Select the back turning tool T☐.
G00 X21.0 Z14.0	.... Add a tool width of 4 mm to a longitudinal dimension of 10 mm to position the tool at the resulting position of 14 mm. ②
G01 X14.0 F☐	.... Cut the material to the point of 14.0 mm in diameter. ③
Z19.0 F☐	.... Perform cutting for 19.0 mm in the longitudinal direction. ④
X21.0 F☐	.... Move the tool to the positioning point by cutting feed. ⑤

5.4.3 Grooving Process

The grooving tool is used to make grooves on the material. It is usually done after the outer diameter cutting.

Machining





- If grooving is to be performed after front turning, cut the material until 0.1 to 0.2 mm shorter in the longitudinal direction during the front turning process, and then determine the length in the longitudinal direction and perform the finishing work by the grooving tool. This is an easier method to determine the length in the longitudinal direction.

Program example

T☐	.........Select the grooving tool T☐.
G00 X21.0 Z12.0	.........Add a tool width of 2 mm to a longitudinal dimension of 10 mm to position the tool at the resulting position of 12 mm.
G01 X12.0 F☐	.........Cut the material in the diametrical direction until the material diameter becomes ϕ 12.0 mm. ③
X21.0 F☐	.........Move the tool to the positioning point by cutting feed. ④

5.5 Center Hole Drilling

5.5.1 Determining Spindle Speed

The spindle speed N (min^{-1}) for operating the center drill and drill is calculated using the following formula:

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

N: Spindle Speed (min^{-1})

V: Cutting speed (m/min) (circumferential speed)

d: Drill diameter (mm) (maximum diameter of machined taper using a center drill)

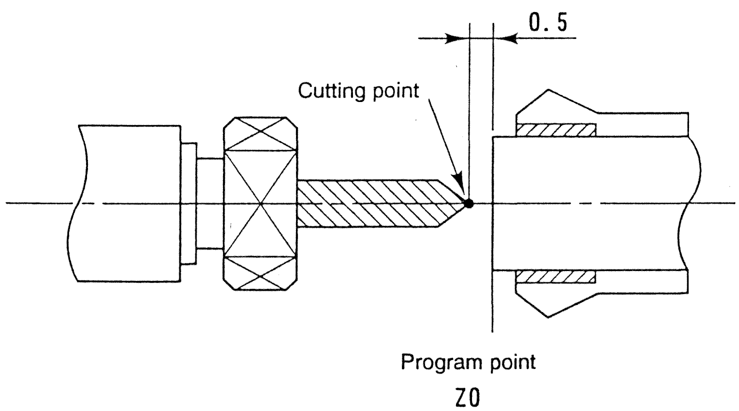
π: Circular constant (approx. 3.14)

5.5.2 Determining Program Point, Cutting Point, and Center Point

The program point (coordinates to be specified in a program) for center drill and drill is located in the following position with relation to the tool holder.

For example, to position the drill at the center of a material in rapid feed as shown in the diagram below.

Machining



Program example

G821 .... Turn the single machining on.

T☐ .... T☐ tool select command

G00 Z-0.5 .... Move the material end face 0.5 mm.

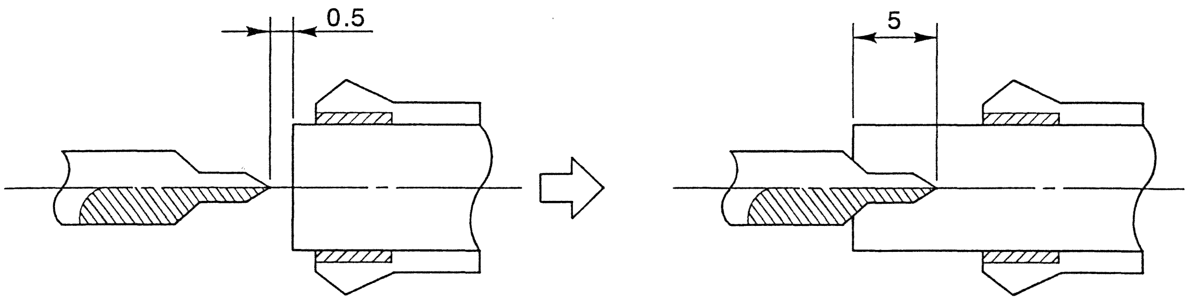
M140 .... Advance the opposite tool post.

↓

Machining Process

5.5.3 Center Drilling

Machining



Program example

$1	$2	
M3 S1 = ☐		..........Rotate the spindle in the forward direction.
G821	**G821**	..........Single machining command ON
T☐		..........T☐ tool selection command
G00 Z-0.5		..........Move the end face of the material by 0.5 mm.
M140		..........Advance the opposite tool post.
G01 Z5.0 F☐ T☐		..........Perform the center machining of 5 mm in depth, and the offset input.
G00 Z-0.5 T00		..........Return to the former position in rapid feed, and the offset cancel.
M141		..........Return the opposite tool post.
G820	**G820**	..........Single machining command OFF

5.5.4 Drilling

Before performing drilling, perform the center drilling.
When the hole to be drilled is too deep, enough coolant cannot be applied on the drill and chips accumulate inside a machined hole.
The drill may be damaged due to galling of material and drill.
Therefore, the in-feed must be performed in several sessions. This is called "stepping."
See the table below for the number of in-feeds and in-feed amount applied for each material type and tool.

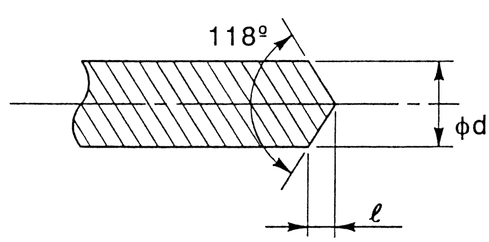
■ In-feed count and in-feed amount

Material	In-feed Amount / Tool	ℓ_n (mm) 1st in-feed	2nd and subsequent in-feeds
Brass	Woodruff drill Flat drill	Entire length	
Aluminum Free cutting steel	Twisted drill	$\ell_1 = \phi\, d \times 3.0$	$\ell_n = \ell_{n-1} \times 0.6$ $\sim \phi\, d \times 0.9$ (mm)
Tool steel Carbonate steel Free cutting stainless steel	Twisted drill	$\ell_1 = \phi\, d \times 2.5$	$\ell_n = \ell_{n-1} \times 0.5$ $\sim \phi\, d \times 0.9$ (mm)
Stainless steel Hard material Twisted drill	Twisted drill	$\ell_1 = \phi\, d \times 2.0$	$\ell_n = \ell_{n-1} \times 0.4$ $\sim \phi\, d \times 0.9$ (mm)

Reference

The drill machining depth is the sum of the effective length and the tool bit length at the tip of a tool.

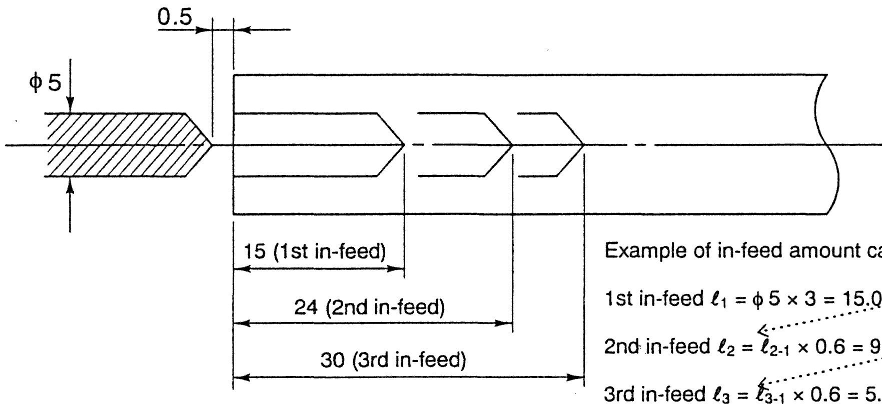
Tool bit length $\ell \fallingdotseq 0.3 \times \phi\, d$



5.5.5 Drilling a Deep Hole

Machining

Material : Free cutting steel



Example of in-feed amount calculation:

1st in-feed $\ell_1 = \phi 5 \times 3 = 15.0$

2nd in-feed $\ell_2 = \ell_{2-1} \times 0.6 = 9.0$ $(\ell_1 + \ell_2 = 24)$

3rd in-feed $\ell_3 = \ell_{3-1} \times 0.6 = 5.4 \fallingdotseq 6.0$ $(\ell_1 + \ell_2 + \ell_3 = 30)$

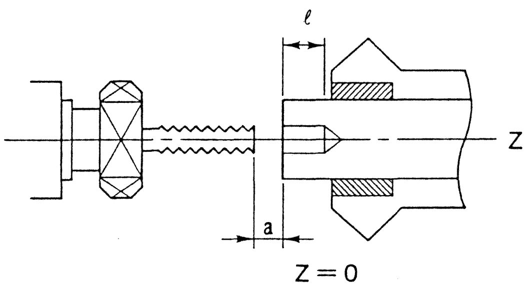
Program example

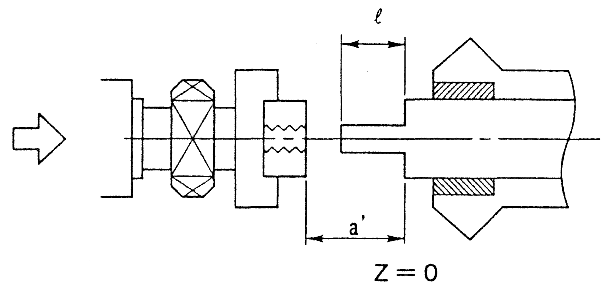
Machining process

$1	$2	
M3 S1 = □		.... Rotate the spindle in the forward direction
G821	**G821**	.... Single machining command ON
T□		.... T□ tool select command.
G00 Z-0.5		.... Move the end face of the material by 0.5 mm.
M140		.... Advance the opposite tool post.
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		.... Second in-feed positioning
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		.... Third in-feed positioning
G01 Z30.0 F0.1		.... Third in-feed
G00 Z-0.5 T00		.... Return the drill to the former position. And cancel the offset.
M141		.... Return the opposite tool post.
G820	**G820**	.... Single machining command OFF

5.5.6 Thread Cutting with Tap and Die

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





The shift amount " a' " for the die is set at "ℓ+1mm".

Tap

Dice

- 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 ~ M3 M4 ~ M6 M7 ~ M10	600 ~ 1000 400 ~ 800 200 ~ 600
Ferrous materials	M1 ~ M3 M4 ~ M6 M7 ~	400 ~ 800 200 ~ 600 ~ 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:

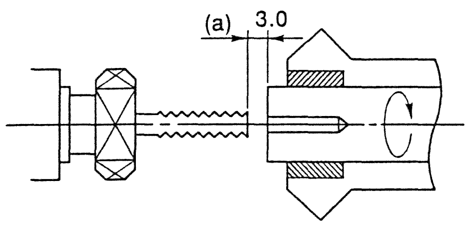
$$t = \frac{0.2 \times \ell + b}{L \times N/60} \times 1000$$

b : Return amount (mm)
L: Lead (mm)
N: Spindle speed (min^{-1})
ℓ : Effective length of thread (mm)

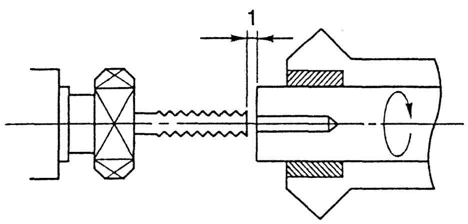
5.5.7 Tapping Process Example

Example of machining with M6 P = 1.0

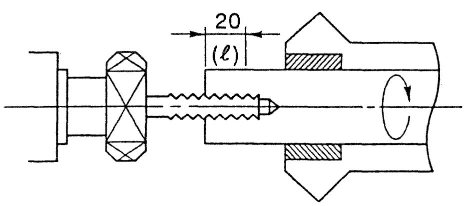
Machining



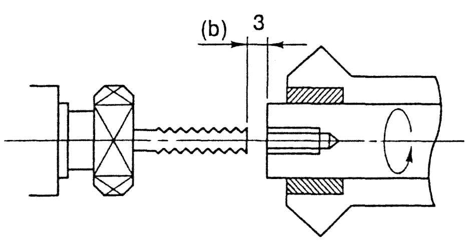
M03 S1 = 300 .....Spindle speed command.
G04 U0.5 ....Dwell in order to stabilize spindle speed (0.5 seconds).
T2300..................Select "23" for tool number.
M140...................Move the opposite tool post forward.



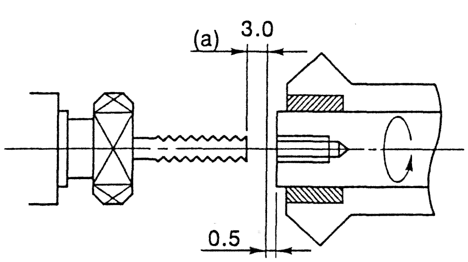
G00 Z2.0.............Move tap to 1 mm before the material end face in rapid feed.
(Z = a - 1.0)



G32 Z19.0 F0.8 T□□
............................Move tap forward in pitch (P × 0.8) feed rate (the feed amount is thread length ℓ. × 80% and specify the compensation number.)
($Z = \ell \times 0.8 + a$, $F = P \times 0.8$)



Z0 F1.0 M04 T00
............................Retrieve tap to the position of 3 mm from the material end face (b) by pitch feed reverse command. Specify the reverse rotation at the same time.
(Offset Cancel)
(Z = a - 3.0)



G04 U2.0 ....Wait till tap is extracted.
(Generally U2.0)
G00 Z-0.5............Return tap to machining start position for next process in rapid feed.
M141....................Retrieve opposite tool post
M03 S1 = ▭
............................Spindle speed command and forward rotation of next process.

5.6 Thread Cutting with the Threading Tool

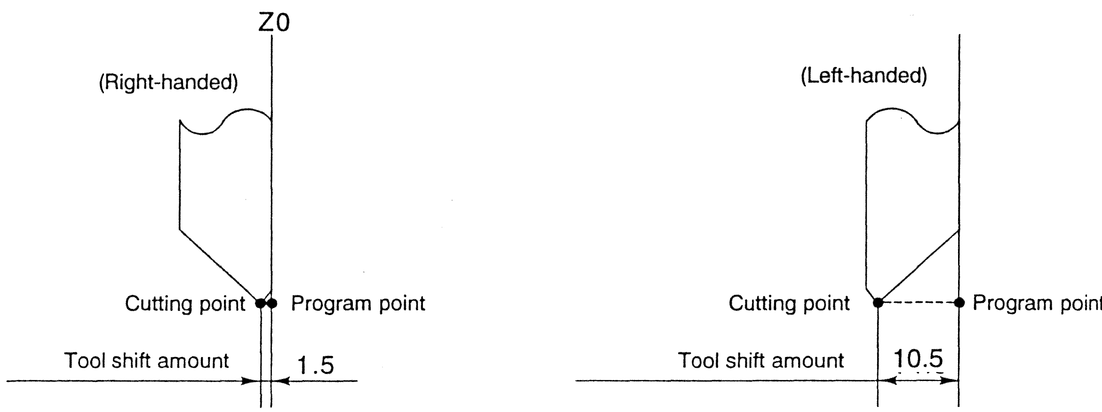
The thread cutting command is generally performed by the G92 canned cycle command, but special thread cutting can be done using the G32 command. The following instructions cover only the G92 commands.

5.6.1 Determining the Program Point and Cutting Point

Threading tools are of two types: the right-handed and the left-handed. They are selected according to applications and their shift amounts vary.

Sometimes the holder itself is shifted. If that happens, the shift amount of "a" equals the sum of the holder shift value and the tool shift value.

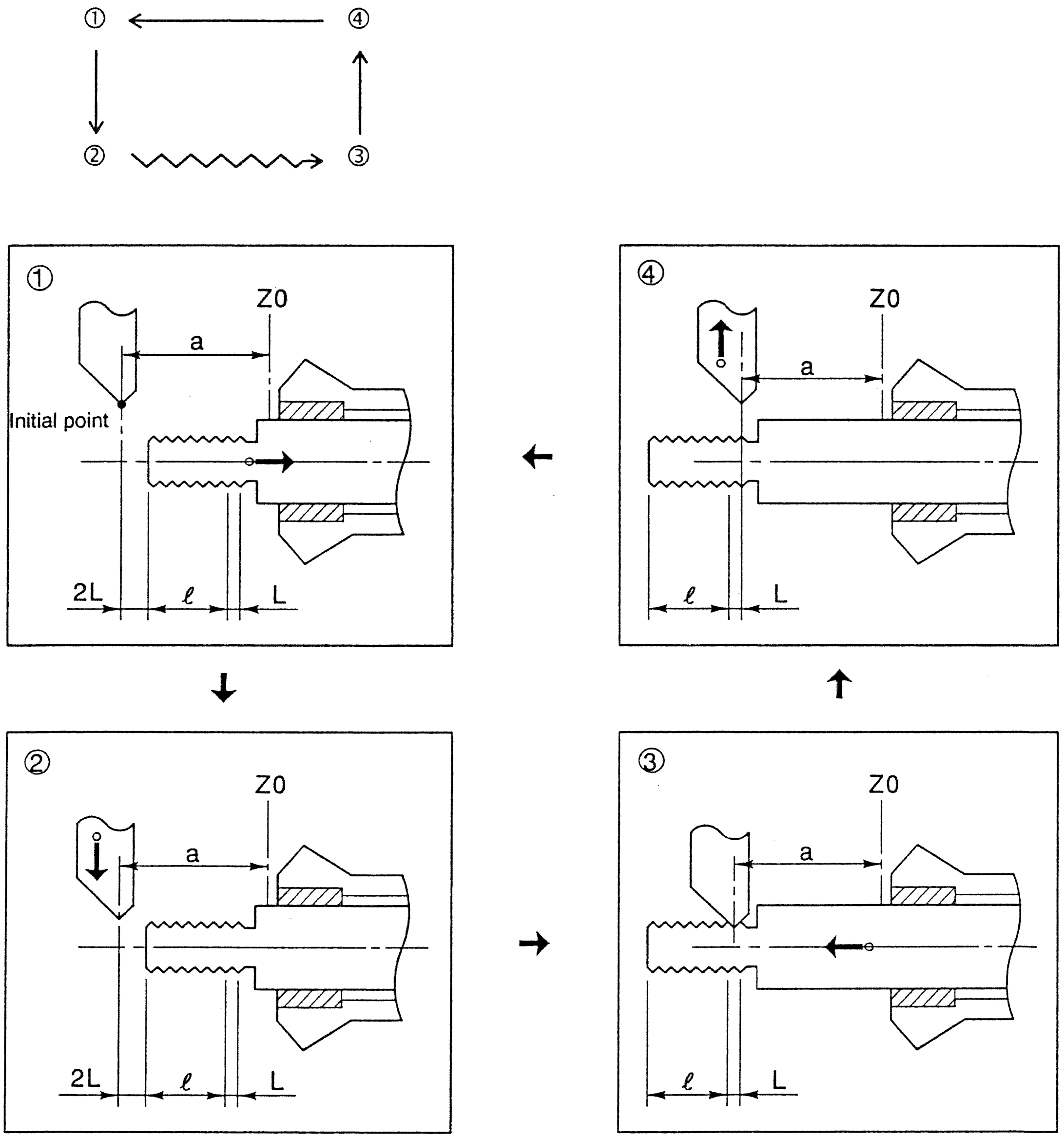
Generally, the right-handed tools are used for the front turning portion and the left-handed tools are used for the back turning portion.



5.6.2 G92 - Thread Cutting Canned Cycle

The Cincom lathe provides the following thread cutting canned cycle.

Machining



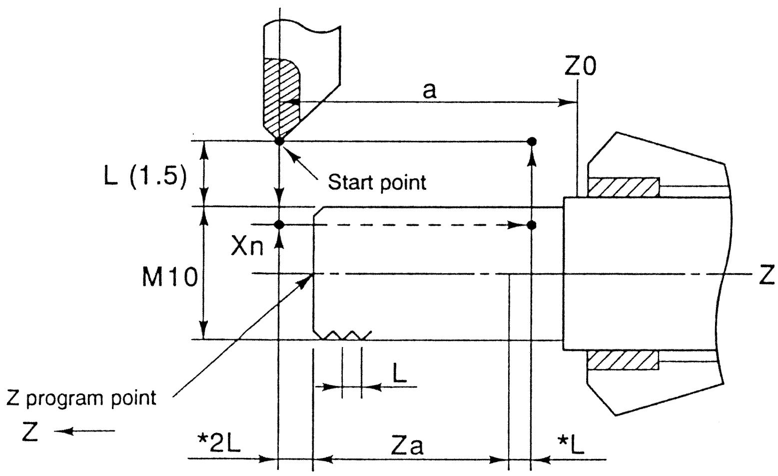
5.6.3 G92 - Thread Cutting

Example of machining with M10 P = 1.5

Machining a non-ferrous material (right-hand thread)

Machining

G92 X□ Z□ F□



L (1.5mm) : Thread lead
Za (12mm) : Effective length
a (10.5mm) : Tool shift
Xn : In-feed
* Uneven thread ridge portion
See "9.1.2 Thread Cutting Count with Tool" in Chapter 9

Program example

T0600	.... Select T06
M03 S1 = 1500	.... Spindle speed command (1.500 min^{-1})
G4 U1.0	.... 1.0 second dwell in order to stabilize rotation
G00 X13.0 Z7.5 T☐	.... Positioning tool at initial point. (Z= a-2L)
G92 X9.3 Z25.5 F1.5	.... 1st in-feed (X1 0.35) (Z=a+Za+2L) (F=L)
X8.8	.... 2nd in-feed (X2 0.25)
X8.48	.... 3rd in-feed (X3 0.16)
X8.28	.... 4th in-feed (X4 0.1)
X8.16	.... 5th in-feed (X5 0.06)
X8.06	.... 6th in-feed (X6 0.05)
X8.06	.... 0 cut (same as last in-feed)
G00 X☐ Z☐	.... To initial point of next process

- The spindle speed is limited as specified by the following equation:

$$N\ (min^{-1}) \leq \frac{8000\ (mm/min)}{L\ (mm)}$$

N : Spindle speed
L : Thread lead
8000 : Maximum feed rate

- The standard spindle speed for thread cutting is 500 to 2000 min^{-1}.
- Uneven thread ridges are produced at the threading start and end points due to the delay of the servo system. The length of uneven thread portions at a spindle speed of 1,500 min^{-1} is approximately two times of the thread lead length (2L) at the entry area, and the thread lead length (L) at the exit area. This uneven thread ridges, length becomes shorter as the spindle speed lowers.
- When selecting right-handed or left-handed threading, reverse the rotation of the main spindle either by changing the Z-axis start position or by using the reverse holder.
- For chamfering (round-up on the screw trailing end), 0 to 89 degrees can be set using a parameter. (See the operator's manual of the NC manufacturer.)

5.6.4 G92 - Taper Thread Cutting Canned Cycle

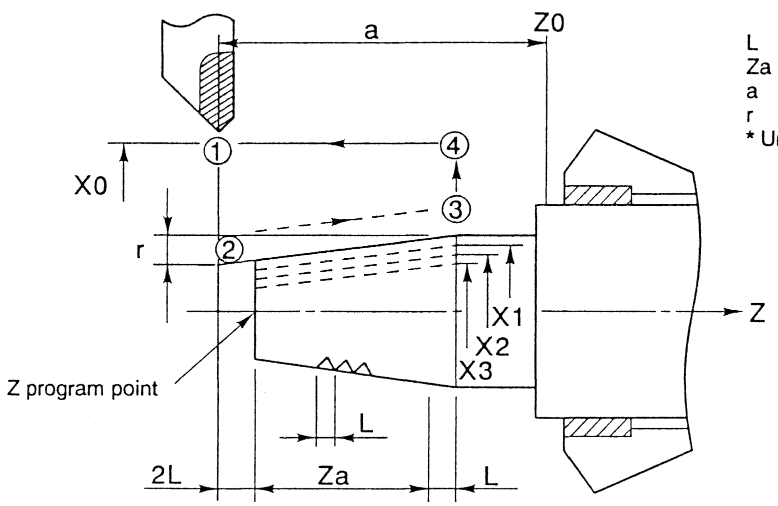
G92 can be used for taper thread cutting.

Taper thread cutting is performed in a canned cycle from ① to ②, ③ to ④, then back to ①. The coordinate reference point is ③.

The sign (+ or -) of the slope "r" indicates the direction of the location of the start point from the end point.

Machining

G92 X□ Z□ R r F□



L : Thread lead
Za : Effective length
a : Tool shift
r : Slope
* Uneven thread ridge portion

Program example

T□	.... Select threading tool
M□ S1 = □	.... Command for spindle forward or reverse rotation at a spindle speed of □ min^{-1}
G4 U1.0	.... 1.0 second dwell in order to stabilize rotation
G00 X X0 Z□ T□	.... Positioning tool at initial point. X(thread diameter +2L), Z(a-2L)
G92 X X1 Z□ R -r F□	.... 1st in-feed X(thread diameter-2 x in-feed 1), Z(a+Za+L), R(-slope), F(L)
X (X2)	.... 2nd in-feed X(X1-2 x in-feed 2)
X (X3)	.... 3rd in-feed X(X2-2 x in-feed 3)
X (X3)	.... Zero cut at diameter X3
G00 X□ Z□	.... Move to initial point of next process. (Threading cycle OFF)

5.7 Boring (Coordinate System Shift)

The center line of the boring tool and that of the nominal cutting diameter are aligned. This displaces the actual cutting point by the amount of the tool radius. This shift amount must be calculated in the program.

5.7.1 Determining the Program Point and Cutting Point

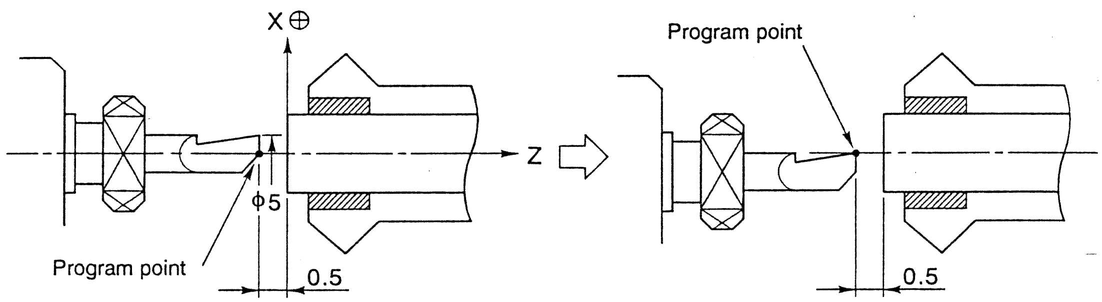
The boring tool is first positioned as shown in the figure below. The tool diameter is then shifted. The command compensates for the shift corresponding to the radius of the tool and facilitates the calculation of the coordinate value. This positioning program is

T□□□□

G50 U▭

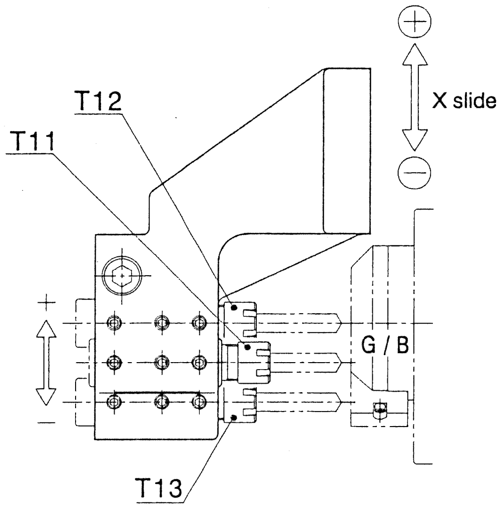
G00 X□ Z□ T□□

This command shifts the program zero point as shown below.

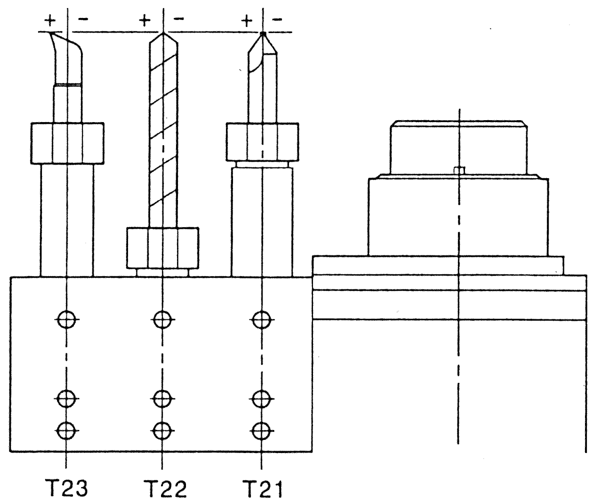


5.7.2 Boring Tool Signs

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



[Type I: Sleeve holder viewed from operator side]



[Type VII: Top view of opposite tool post]

Notes

- Move the tool away from the end face of the material after completing the boring. It is dangerous to specify a tool selecting command without moving the tool away from the end face.
- The boring and inner diameter thread cutting by T23 of type VII may be limited in stroke depending on the back tool post and back 3-spindle tool holder. Use T21 and T22 for boring.
- The back spindle and T31 sleeve may interfere when the center is shifted more than +5.0 mm with the boring and inner diameter thread cutting mounted in T23 of type VII. Use T21 and T22 for boring.
- The boring and inner diameter thread cutting by T13 of type I may cause an overrun to the + side. Use T11 for boring

5.7.3 Adjusting for Diameter Direction Coordinate Shift and Canceling the Coordinate Shift Amount

When machining by the boring tool, the coordinate system shift command can be entered to make it less complicated to calculate the coordinate. Use the G50 coordinate system setting command as in the following example.

[Coordinate system shift] **G50 U5.0** (when the tool diameter is ϕ 5)

The "5.0" after address "U" is determined to the boring tool diameter (ϕ 5). If a ϕ 8 boring tool is used, the coordinate system shift program should be codes such as "G50 U8.0"

The coordinate system shift cancel command must always be entered when the boring tool moves to the material end face after machining.

To cancel the sample shift command, use the following example:

[Coordinate system shift cancel] **G50 U-5.0** (when the tool diameter is ϕ 5)

The "-5.0" is used to return the shifted coordinate system to the former position, such as specified by "5.0".

The coordinate system shift and coordinate system cancel in the diametrical direction used for boring are as described above.

In addition, the coordinate system shift and coordinate shift cancel in the longitudinal direction used for a back turning tool and cut-off tool are available.

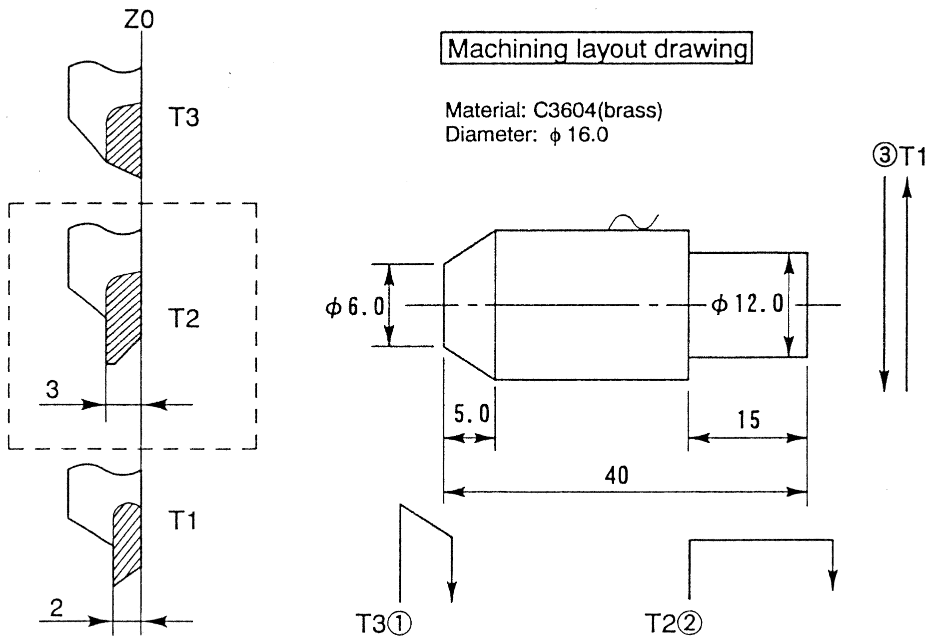
For more information, see the back turning in the next section.

5.7.4 Adjusting for Longitudinal Direction Coordinate Shift and Canceling the Coordinate Shift Amount

The cutting point and program point are shifted during the back turning operation.

To compensate for this, considering the tool width in the longitudinal direction (Z value) in the programming is required.

The coordinate system shift command can be used to specify a tool width as a shift amount, allowing the dimension on the drawing to be directly entered in the machining program.



The sign (+ or -) is assigned depending on whether the program point is at a plus or minus position based on the cutting point.

[Coordinate system shift] **G50 W-3.0** (when tool width is 3 mm)

[Coordinate system shift cancel] **G50 W3.0** (when tool width is 3 mm)

Program example

G50 W-3.0 .... Back turning tool longitudinal coordinate system shift command

G00 X17.0 Z25.0 .... Position tool in rapid feed.
The tool is moved for a distance adding the shift amount.

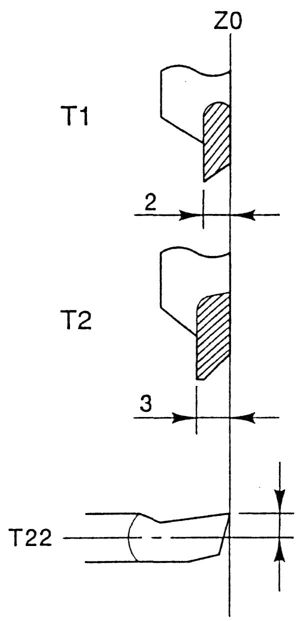
G01 X12.0 F☐ .... Perform diameter direction cutting.

Z40.5 F☐ .... Cut material for 40 mm plus 0.5 mm in longitudinal direction.

X17.0 F0.2 .... Move tool to specified position on material diameter.

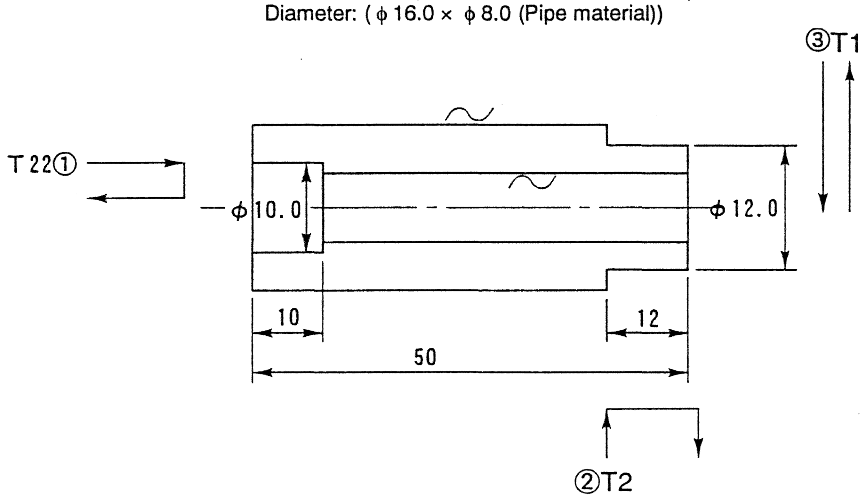
G50 W3.0 .... Coordinate system shift cancel command.
The tool is moved in the longitudinal direction when the next Z code is specified.

5.7.5 Adjusting for Diameter and Longitudinal Direction Coordinate Shift and Canceling the Coordinate Shift Amount



Machining layout drawing

Material: S45C (carbonate steel for structure).
Diameter: (φ 16.0 × φ 8.0 (Pipe material))



Program example

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□	G821
M06	G820
G00 X17.0 Z-0.5	G999
M03 S1 = □ G99	N999
G821	M02
	M99
	%

Boring process

```
N0122 T2200
M140
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)
M141
```

Program example

Back turnning process

```
N0202 T0200
[G50 W-3.0]             ......... Back 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]              ......... Back turning coordinate system shift cancel (longitudinal
                                  direction).
```

Cut-off process

```
N0301 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
G820
M56
G999
N999
M02
M99
%
```

5.8 Tool Offset (T Codes)

The offset function is designed to correct the difference between the drawing dimensions and the dimensions of the actual machined workpiece. The dimensions are corrected automatically by entering the amount of difference without changing the coordinate values in a program.

Note, however, that compensation numbers must be programmed.

Command format

- When compensation is added simultaneously with tool selection T□□□□ (4 digits)
- When only compensation value is given after tool selection T□□ (2 digits)

Explanation

1. The "T□" code for compensation consists of a two-digit numeral.
   - For T□□□□ (4 digits):

     The tool selection and compensation are performed simultaneously (in the diametrical direction move).
     The first two digits indicate the tool number, and the last two digits indicate the compensation number.
   - For T□□ (2 digits):

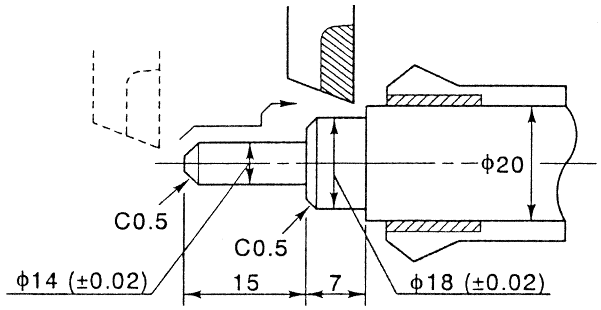
     The tool has been already selected in this case. This command is used to give or change the compensation value without changing the tool. (Usually, T□□ (2 digits) is used.)
2. The machine operator compensates for the tool position. The programmer specifies the compensation number, in the case that compensation is assumed to be required in the program beforehand. The compensation number is prepared in the following cases usually.
   - At the time of tool selection and positioning
   - For each different diameter
   - At the time of longitudinal feed start of face drilling and tap machining
3. The current compensation value is valid until it is canceled by the next tool selection command or another tool compensation command.
4. Compensation can be canceled by T00. It must be canceled in the following cases.
   - When the cut-off machining is completed and the tool returns to the start position
   - Returning of the face drilling and tap machining in the longitudinal direction
5. The compensation value is modal.
6. The numeral that can be used as the compensation number is 01 to 40.
7. The "T□□□□ (four-digit)" command executes the movement that already contains the compensation value (in the diametrical direction only).

   The "T□□ (two-digit)" command executes the compensation when X□ and Z□ commands are specified.

5.8.1 Compensation Example

5.8.1.1 Outer Diameter Cutting

Machining



Program example

N0103 T0300

G00 X12.0 Z-0.5 $\boxed{\text{T01}}$ .... Specify the offset command in the block which moves the tool to the material in rapid feed.

G01 X14.0 Z0.5 F0.03 .... (Compensate for the φ 14)

Z15.0

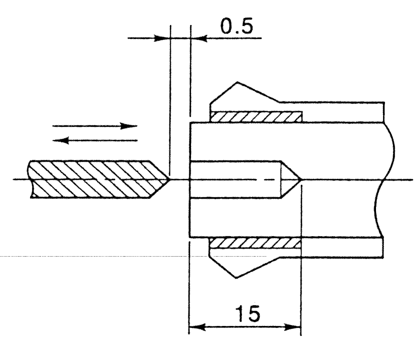
X17.0 $\boxed{\text{T02}}$ .... Specify the offset command each time when there is diameter difference in the workpiece shape as in this program example.

X18.0 Z15.5 .... (Compensate for the φ 18)

Z22.0

5.8.1.2 Face Drilling (Drill, Tap Machining)

Machining



Program example

N0323 T2300

G00 Z-0.5

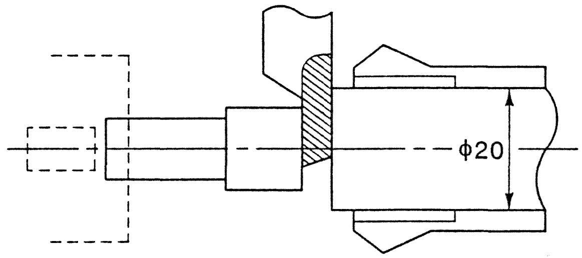
G01 Z15.0 F0.08 $\boxed{\text{T03}}$ .... Specify the offset command to adjust the drill depth.

G00 Z-0.5 $\boxed{\text{T00}}$ .... Offset cancel. Be sure to specify this command when the tool returns.

M141

5.8.1.3 Cut-off

Machining



Program example

N0401 T1100

G00 X21.0 Z22.0 $\boxed{\text{T04}}$ .... The product length is decided by the Z axis direction offset.

G811

!L1

G01 X-3.0 F0.02

G810

M05

M07

G00 Z0 $\boxed{\text{T00}}$ .... Offset cancel
(Always specify this command returning the spindle to the start position.
If it is not specified, the position in the longitudinal direction will shift until it finally causes a Z axis overrun.)

M56

G999

N999

M02

M99

%

5.9 Simultaneous Machining

The Z1 and Z2 axes can be specified independently. This enables free simultaneous machining. The way to machine with superimposition and the way to machine without superimposition are available for simultaneous machining.

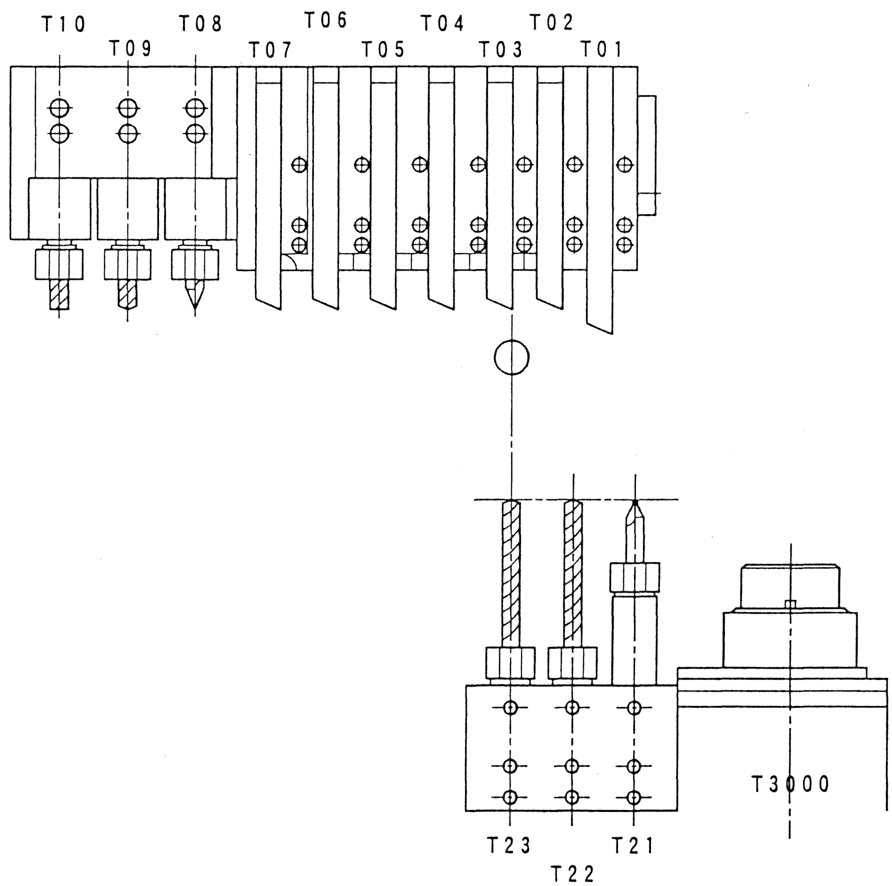
For the simultaneous machining, perform the outer diameter machining by T02 to T03 tools and the hole machining by a T23 tool.

The simultaneous machining in the combination T21, T02 with T22, T03 is possible, however, some restrictions for stroke exist depending on the relation between the tool shape and the diameter of front drilling tool. Interference may occur with another combination of simultaneous machining.

Follow these guidelines to perform the simultaneous machining:

- A tool is selected for each axis control group ($1 (gang tool) and $2 (opposite tool post)) by the ordinary T□□□□ command.
- After machining is completed, move the tool to the positioning point.
- The machining is performed at the feed rate (F) specified in each axis control group.

As shown in the figure below, the machining is performed by combining the vertical tools and the tools on the opposite tool post

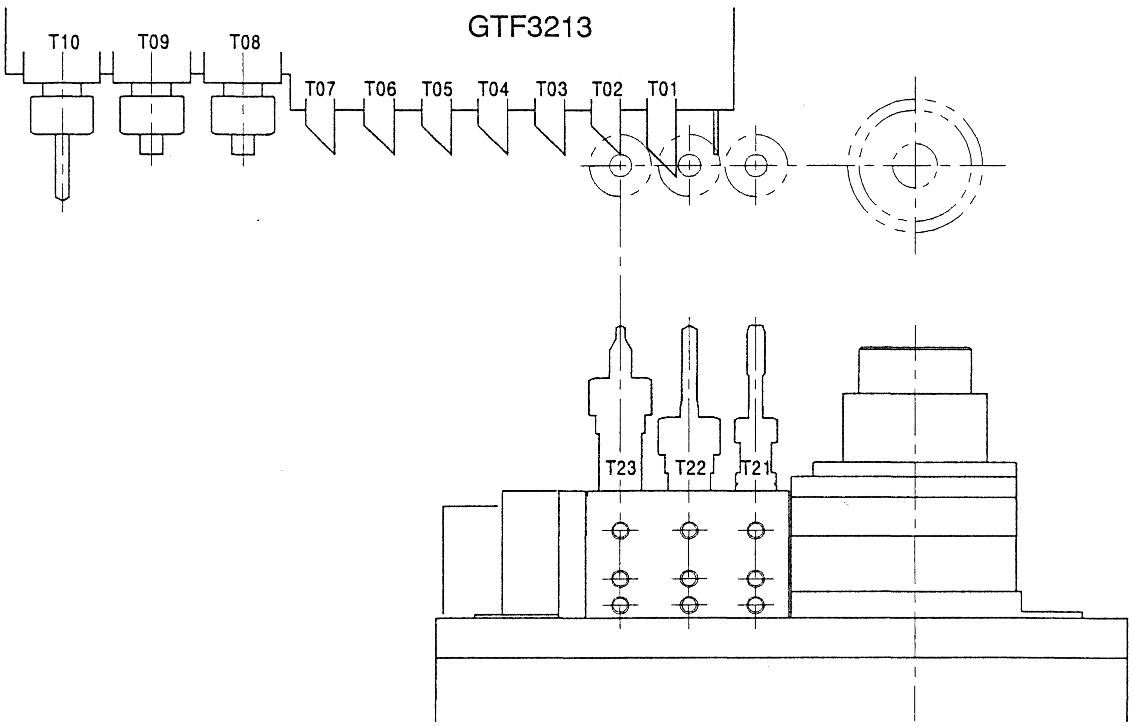


Notes

- The opposite tool post assumes tools whose maximum drilling diameter is 10 mm. Be careful when using tools exceeding the diameter or odd-shaped tools because they have potential risk of interference.
- Be careful in simultaneous boring with the opposite tool post, when the gang tool and the tool on the opposite tool post are off-centered, causing some tools to interfere with each other.

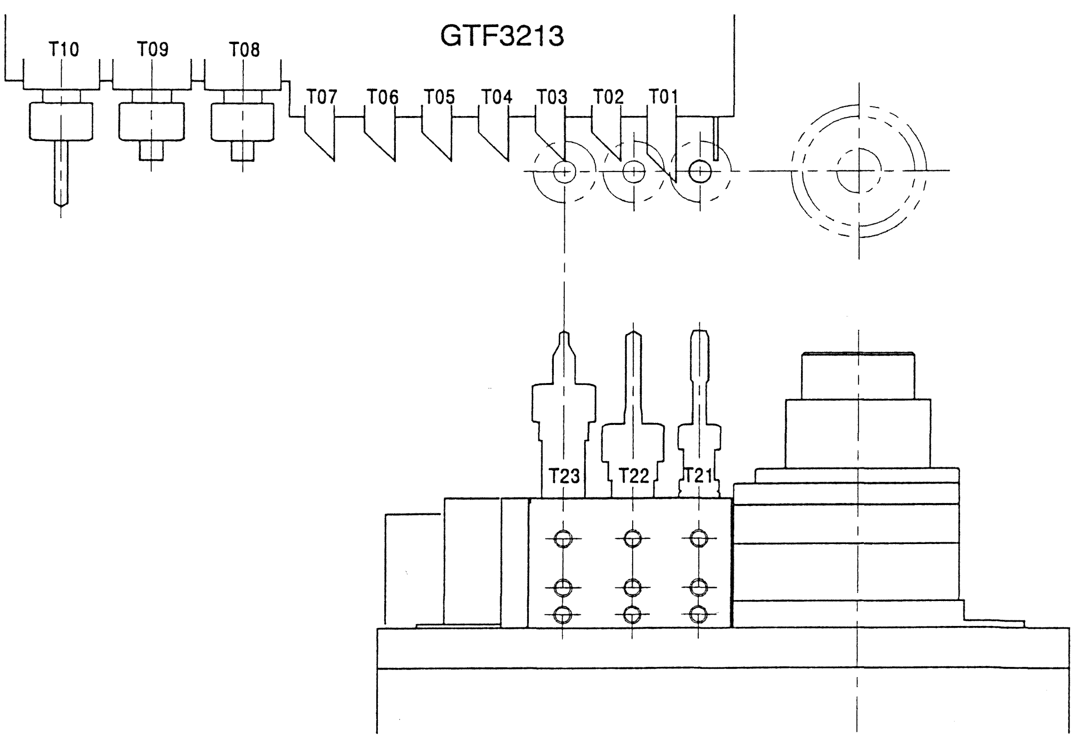
Simultaneous machining pattern (1)

Positioning relation between T02 and T23 is shown below.



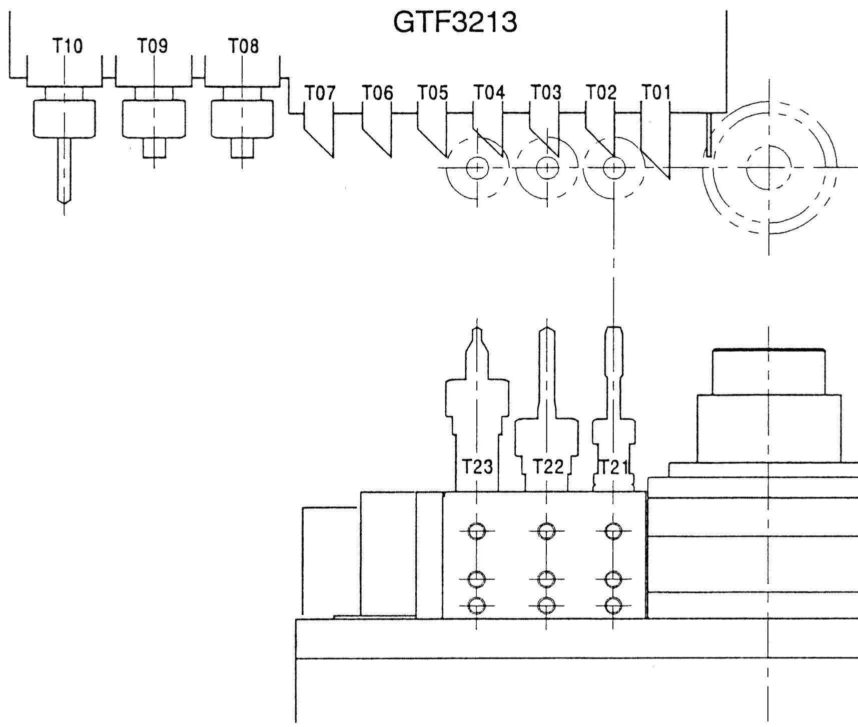
Simultaneous machining pattern (2)

Positioning relation between T03 and T23 is shown below.



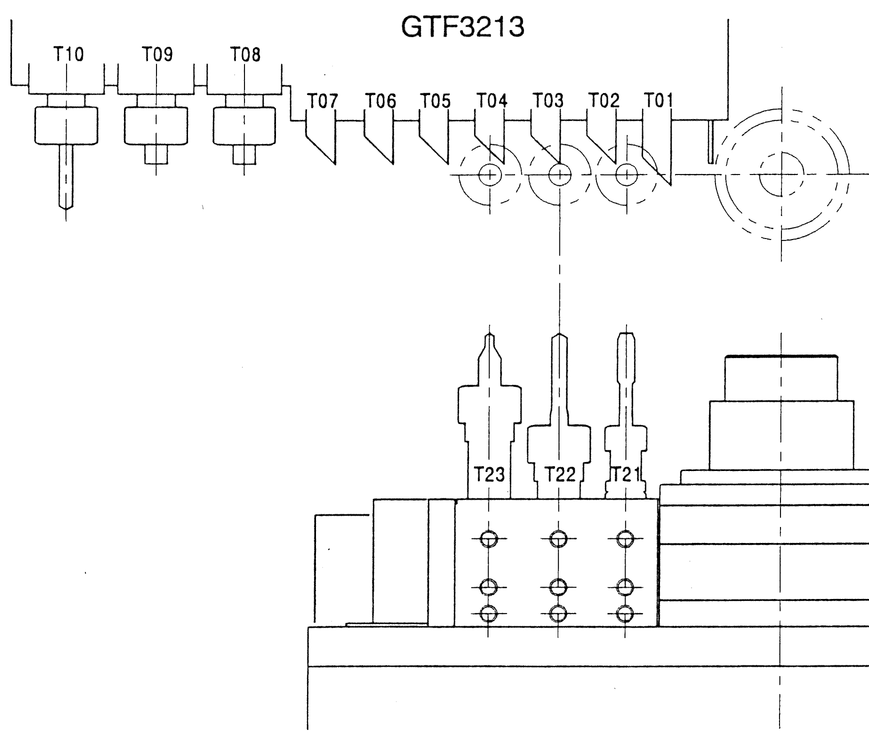
Simultaneous machining pattern (3)

During simultaneous machining by T02 and T21, T02 can cut up to the point of 10 mm in diameter. Cutting further requires caution because it involves a potential risk of interference depending on the tool shapes of T03 and T04 and the front drilling tool diameter.



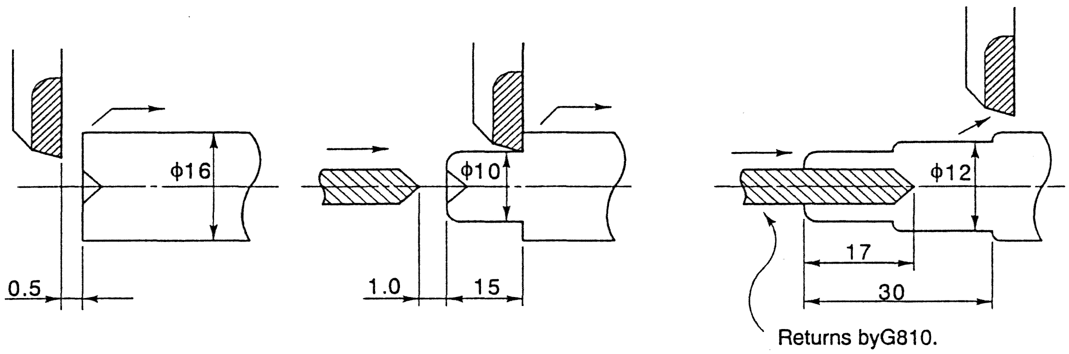
Simultaneous machining pattern (4)

During simultaneous machining by T03 and T22, T03 can cut up to the point of 10 mm in diameter. Cutting further requires caution because it involves a potential risk of interference depending on the tool shape of T04 and the front drilling tool diameter.



5.9.1 Simultaneous Machining by Outer Diameter Cutting and Drilling

Machining



Program example

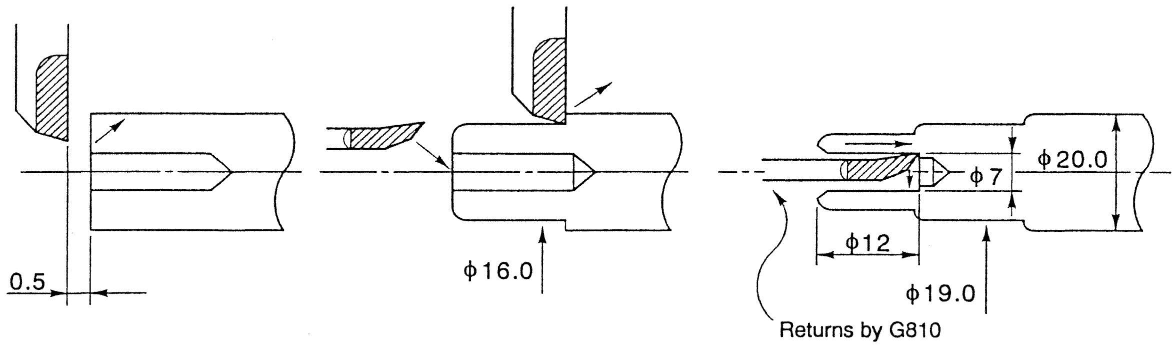
Machining process

$1	$2	
T0300	**T2300**	..........Tool selection (in both axis control groups 1 and 2)
G00 Z-0.5		..........Rapid feed positioning (in longitudinal direction) — For axis control group 1 only
X8.0		..........Rapid feed positioning (in diametrical direction) — For axis control group 1 only
G01 X10.0 Z0.5 F□		..........Chamfering (C0.5) — For axis control group 1 only
Z15.0		..........Cut to the position of 15.0 in the longitudinal direction. — For axis control group 1 only
G811	**G811 Z-1.0**	..........Z1 - Z2 superimpose (Opposite tool post positioning in longitudinal)
X11.0	**G01 Z17.0 F□**	.... Axis control group 2 drilling / Axis control group 1 outer diameter cutting
X12.0 Z15.5	**G00 Z-1.0**	.... Axis control group 2 drilling / Axis control group 1 outer diameter cutting
Z30.0		.... Axis control group 1 outer diameter cutting
X15.0		.... Axis control group 1 outer diameter cutting
X17.0 Z31.0		.... Axis control group 1 outer diameter cutting
G810	**G810**	..........Z1 - Z2 superimpose command OFF (Axis control group 1) Return to the opposite tool post machine zero point. (Axis control group 2)

Machining process

5.9.2 Outer/Inner Diameter Simultaneous Machining (Boring)

Machining



Program example

Machining process

$1	$2		
T0300	T2300	..........Tool selection (in both axis control groups 1 and 2)	
G00 Z-0.5		..........Rapid feed positioning (in longitudinal direction)	Operates in axis control group 1 only.
X14.0		..........Rapid feed positioning (in diametrical direction)	
G01 X16.0 Z0.5 F□		..........Chamfering (C0.5)	
Z11.0		..........Cut to the position of 12.0 in the longitudinal direction.	
G811	G811 Z-0.5	..........Z1 - Z2 superimpose (opposite tool post positioning in longitudinal direction)	
X17.0	G50 U□	..........Cut to ϕ17.0	Coordinate system shift ON
X19.0 Z12.0	G00 X9.0	..........Chamfering (C1.0)	Rapid feed positioning (in diametrical direction)
Z30.0	G01 X7.0 Z0.5 F□	Cut to the position of 30.0 in the longitudinal direction	Inner diameter chamfering (C0.5)
X21.0 Z31.0	Z12.0 F□	..........Move to the outer diameter while chamfering and cutting	Cut to the position of 12.0 in inner diameter in the longitudinal direction
	X5.8 F□	..........Cut to ϕ 5.8 in diameter. (Move the tool bit away.)	
	G00 Z-0.5	..........Return in rapid feed.	
	G50 U-□	..........Coordinate system shift OFF	
G810	G810	..........Z1 - Z2 superimpose command OFF (Axis control group 1) Return to the opposite tool post machine zero point. (Axis control group 2)	

Machining process

5.10 Free Tool Layout Pattern (Holder Name = Free Tool)

This command can change the interval of the gang tools and the tools on the drilling tool post and increase the number of tools. Usually, fourteen gang tools is available. Specify "Free Tool" as holder names 1 to 3 on the Machining Data screen. However, you will need to manufacture a special holder in accordance with your requirement.

Command format

T□□□□ (Four-digit T code)

This command is much the same as an ordinary tool selection command.
Usually, T01 to T14, T21 to T25, and T31 to T35 can be used.
Enter the tool position data on the Tool Pattern screen shown below.
For tool position data Y1, enter the diameter value the distance from the guide bushing center as the reference point (zero point) with the Y1 axis positioned at the machine zero point.
For tool position data X1, enter the diameter value the distance from the bottom face of the gang tool holder to the tool tip.
For tool position data X2, enter the diameter value the distance from the machine zero point of the X2 axis to the tool selection position.
Like the ordinary machining, an ordinary command can be issued in the X-Y coordinates after tool selection. Each program can be created as before.

Tool layout pattern screen

Tool Pattern

	Front Tool Holder			Front Drill Holder		Back Drill Holder
	Free Tool			Standard Tool		Standard Tool
	Y1	X1		X2		X2
T01	552.000	60.000	T21	492.000	T30	632.000
T02	502.000	40.000	T22	432.000	T31	242.000
T03	452.000	40.000	T23	370.000	T32	162.000
T04	402.000	40.000			T33	82.000
T05	352.000	40.000				
T06	300.000	40.000				
T07	218.000	40.000				
T08	148.000	40.000				
T09	78.000	40.000				
T10	8.000	40.000				
T11	0.000	60.000				
T12	0.000	60.000				
T13	0.000	60.000				
T14	0.000	60.000				

Quit (ESC)

Notes

- Notice that the tool moves using the T□□□□ code.
- Confirm the contents of the following before machining when the free tool layout is used.
  1. Check whether "Free Tool" has been specified as holder names 1 to 3 on the Machining Data screen.
  2. Check the position of each tool from the guide bushing center as the reference point.
- Sufficient T codes are provided in the machine software. Therefore, the tool size and other mechanical factors are limited during use.

5.11 Using Subprograms

Once you store a coding sequence (that is going to be repeatedly used within a program) in advance as a subprogram, you can call this subprogram from the main program each time that sequence is needed in the program. And you can make the main program structure simple.

5.11.1 A Subprogram Call Instruction

Subprogram call pattern

The subprogram call has the following regulations.

M98 P□ L□

- L□ — Number of times the subprogram is repeated (1 to 9,999)
- P□ — Subprogram number (1 to 99,999,999)

5.11.2 Calling a subprogram from the main program

- The subprogram is executed only once if the number of repetition is not specified.
- The subprogram can be repeated up to 9,999 times when it is called from a main program once.
- Subprograms can be nested to eight levels.
- The subprogram call command can be specified in either of the axis control groups ($1 and $2) in the main program.
- The main program and subprogram are called for each axis control group.

(Main program)		(Subprogram)
$1	←——→	$1
$2	←——→	$2

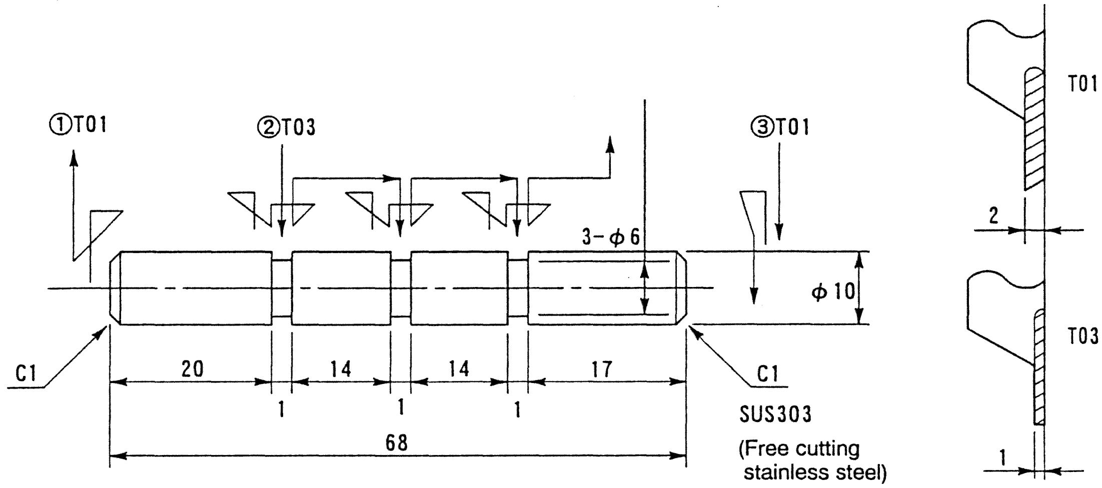
Example

Main program		Subprogram
O0001		O0002
$1	$2	$1
M98 P0002		
G00 X21.0 Z15.0		**M99**
		%
M02	**M02**	
M99	**M99**	

- The M99 command must always be placed at the end.
- The subprogram does not require M02.

5.11.3 Example of Using a Subprogram

Dimensions and machining layout



Program example

Main program

```
O0001
$1
G50 Z0
M06
G00 Z-0.5 M03 S1=1800 G99
G821
N0101
G00 X11.0 T01
Z1.5
G01 X7.0 Z-0.5 F0.015
G00 X11.0
N0203 T0300
G00 X11.0 Z21.0 T02
M98 P0002 L3
N0311 T0100
G00 X11.0 Z70.0 T03
G01 X8.0 F0.015
X11.0 F0.2
W-1.5
X8.0 W1.5 F0.015
X-3.0
M05
M07
G00 Z0 T00
G820
M56
G999
N999
M02
M99
%
```

```
$2
G821
G820
G999
N999
M02

M99
%
```

(Repeated 3 times)

Subprogram

```
O0002
$1
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
%
```

```
$2
```

5.12 Long Material Lathing (Optional) (Type I)

Lathing of long material is used by optional devices to machine long material requiring more than single chucking, to prevent the tips of long products from being vibrated, or to support the material during cut-off machining.

There are two optional devices available for long material lathing:

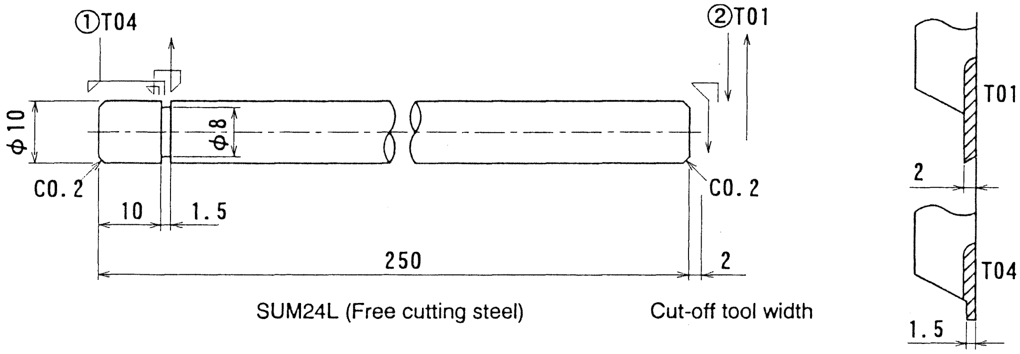
U10C (long workpiece device)

U51Z (stock gripper device)

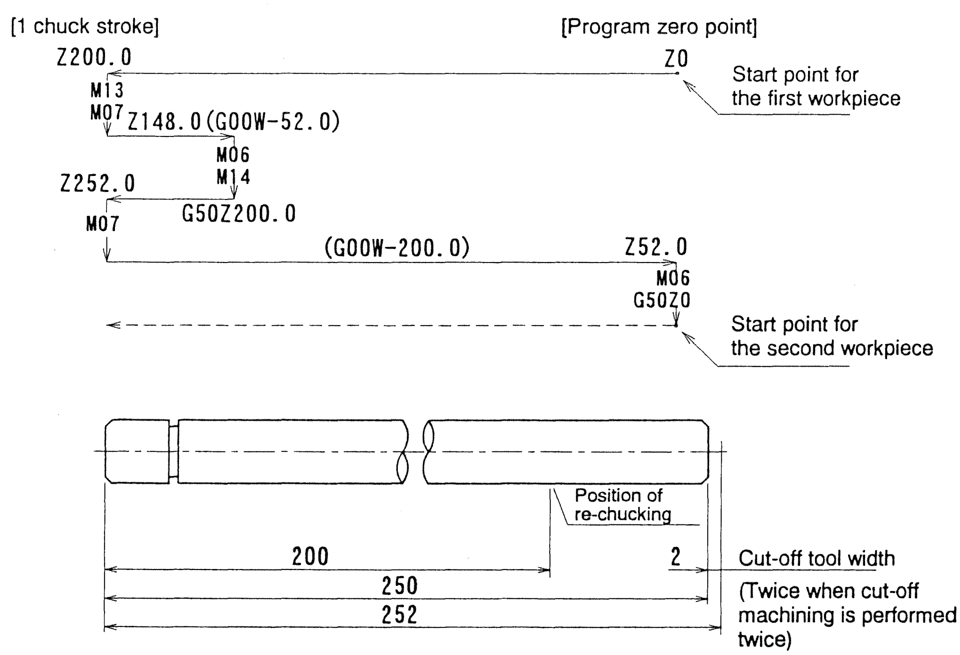
The U10C is used alone to support the material. The U10C and U51Z are used in combination to support the material and to machine long products requiring more than single chucking.

A sample program is given below.

Machining drawing and tool layout



Operation sequence



Follow these guidelines when programming long material lathing:

1. Set "the entire workpiece length + cut-off tool width" as the maximum amount of move. (Including end face turning and cut-off machining twice)
2. Determine the amount of move by single chucking within the range of the maximum machining length free of interference with the machining position when re-chucking.
3. Be sure to return the workpiece machining end point to the program zero point as the start point.

U10C (long workpiece device) command

Support advance	M10
Support return	M11

Note

- Be careful in using this command not to cause interference.

■Long material lathing program

O0001	
M11	..........Support (U10C) return
G50 Z 0	
M06	
G00 X11.0 Z-0.5 S3300 M03 G99	
(M96)	..........(Spindle speed change detection ON)
N0104 T0400	
G00 X11.0 Z0.7 T01	
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	

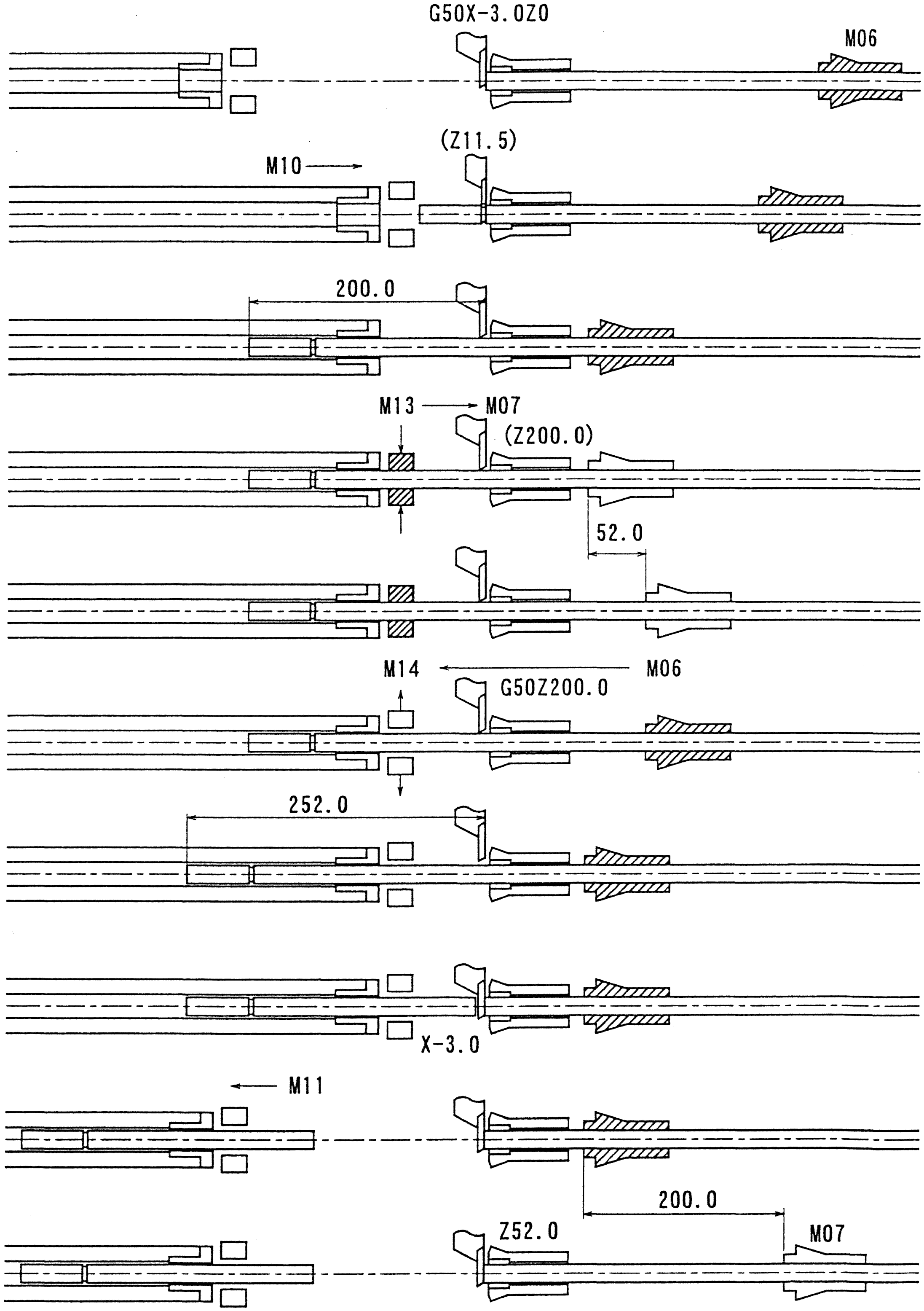
N0201 T0100	..........Select the cut-off tool mounting tool.
M05 (M97)	..........Stop the spindle (turn spindle speed change detection off).
M10	..........Advance the support.
G00 Z200.0 T00	..........Position for the re-chucking and cancel tool offset.
M13	..........Close the stock gripper.
M07	..........Open the spindle chuck.
G00 Z148.0 (W-52.0)	..........Return to Z148.0.
M06	..........Close the spindle chuck.
M14	..........Open the stock gripper.
G50 Z200.0	..........Set the Z-axis coordinate system.
S3300 M03	..........Rotate the spindle at 3300 min^{-1}.
(M96)	..........(Turn spindle speed change detection on.)

```
G00 X11.0 Z252.0 T02
G01 X9.6 F0.03
X11.0 F0.2
W-0.7
X9.6 W0.7 F0.03
X-0.5
X-3.0
M11                          ..........Support return
M05 (M97)
M07
G00 Z52.0 (W-200.0) T00
M02
M99
%
```

The enclosed portions ☐ of this program are related to long material lathing.

Example of using the long workpiece device (U10C) and stock gripper device (U51Z)

■Operation drawing



Notes

- The oblique-lined areas in the above drawing represent the closed main spindle chuck and closed material chuck device.
- "Dwell" may be used to ensure operations of opening and closing the main spindle chuck and stock gripper device.

Product code	C	–	L	5	1	6	2	0	I	III	VII	/	/	/	–	5	7	0

Document code	2	E	1	–	0	5	0	2
	2	E	2	–	0	5	0	1

Chapter 6 Programming Practice (Secondary Process)

6.1 Secondary Process Guidelines 6-1

6.1.1 Turning Process and Secondary Process 6-1

6.1.2 Machining Process Order 6-1

6.1.3 The Coordinate System for Secondary Process 6-2

6.2 Main Spindle Indexing 6-3

6.2.1 Main Spindle Indexing (M28, M20) 6-3

6.2.2 Back Spindle Indexing (M78, M79) (Optional) 6-4

6.2.3 Main Spindle C-Axis Indexing (M18) (Optional) 6-5

6.2.4 Back Spindle C Axis Indexing (M48) (Optional) 6-7

6.3 Calculating the Command Speed 6-8

6.3.1 Calculating the Spindle Speed Value 6-8

6.3.2 Calculating the Tool Feed Rate 6-9

6.4 Two-Surface Width Machining (Including Indexing) Program 6-10

6.5 Through-Hole Machining 6-12

6.6 Synchronous Tap Machining (G88, G80) (Optional) 6-13

6.7 Circular Interpolation 6-14

6.7.1 Plane Selection 6-14

6.7.2 Tool Diameter Compensation 6-15

6.8 Front Face Tool Spindle (Optional) 6-18

6.8.1 Front/Back Face Tool Spindle Rotation and Stop (M80, M81, and M82) 6-18

6.8.2 Synchronous Tapping with Front/Back Face Tool Spindle (G84, G80) 6-19

6.9 Differential Speed Rotary Tool Function (Optional) 6-20

(Blank page)

This chapter describes complex machining.

6.1 Secondary Process Guidelines

6.1.1 Turning Process and Secondary Process

Machining processes such as outer diameter cutting, center hole drilling, and outer diameter threading explained in the previous chapter are the so called "turning" process which means cutting performed by rotating the material (spindle) while the cutting tool remains in place. In the secondary process described in this chapter, the cutting tool rotates while the material remains static.

Turning:	Machining is performed by rotating the material.
Secondary process:	Machining is performed by rotating the cutting tools.

6.1.2 Machining Process Order

Machining of a product that requires a secondary process is all done in a cycle in the same manner as the machining of a turning process.
The secondary machining procedure is basically decided according to the following rule.

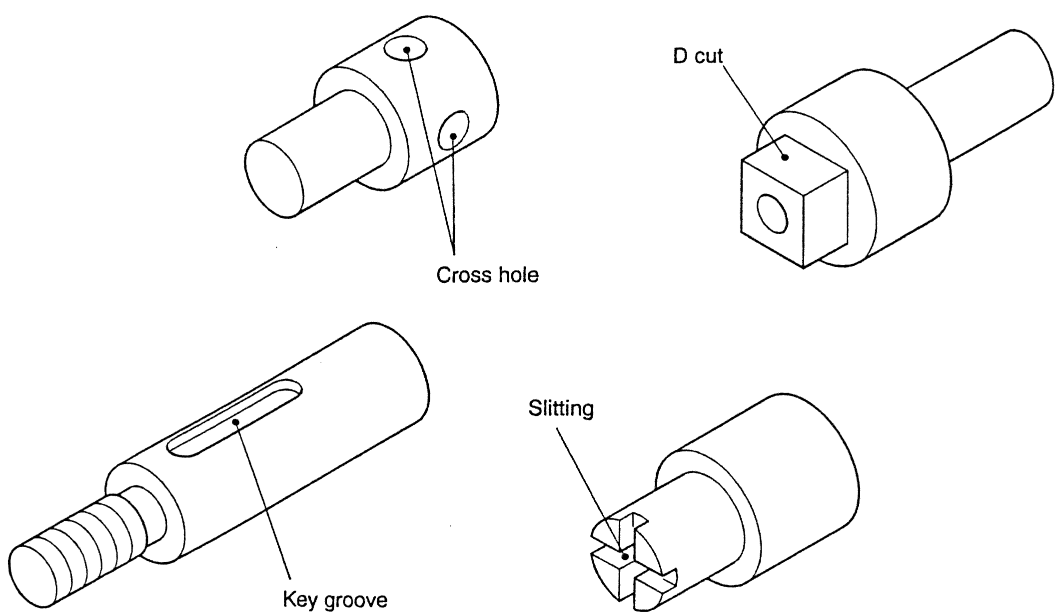
[Basic rules for the secondary machining procedure]

- The secondary process performs the machining tasks from left to right in order on the material.
- The center hole drilling and secondary process on the end face of a material are done before other processes.
- If turning and secondary process are to be performed on the same coordinate, the "turning" is done first.

The tooling layout is created based on the conception described above.

The number of tools to be mounted in a secondary process varies depending on the holder to be used. (For further details, see Chapter 10 "Tooling")

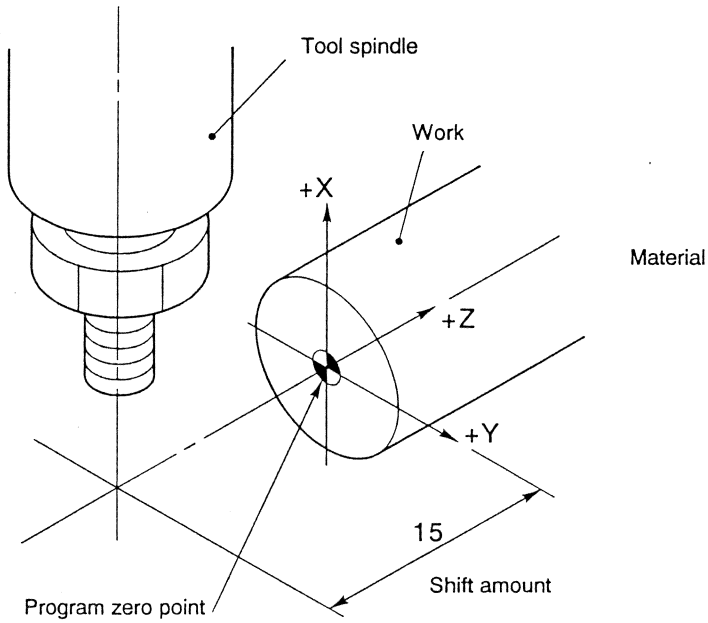
Machining example



6.1.3 The Coordinate System for Secondary Process

Before creating a secondary process program, it is advisable to have an understanding of the coordinate system.

The diagram below shows the tool spindle in its position relative to the material.



[Coordinate values and signs]

Program coordinates for the secondary process are set in the same way as the coordinates for turning. The only difference is that the secondary process can perform machining using three axes (X, Z, and Y) since the Y axis coordinate is movable.

Values for the X and Y axis coordinate must be programmed in diameter. The program zero points for the X and Z axes are the same as the ones for turning.

The Y axis zero point is set at the center of the material.

Signs (+ and -) are determined by considering the material to be fixed in place and assigning the "+" sign to the direction of tool move.

T□□□□ X▭ Y▭ Z▭ ;

[Tool position and coordinate system shift]

The tool spindle is shifted 15 mm toward the Z axis. If a program is created without a consideration for the shift, machining will be done to a point 15 mm away from the intended position.

To avoid this displacement, shift the length between the program zero point in the Z-axis direction and tool spindle center.

This coordinate system shift must always be canceled after the secondary process is complete. Also note that this program must be included in the axis control group 1 program ($1).

Command format

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

6.2 Main Spindle Indexing

6.2.1 Main Spindle Indexing (M28, M20)

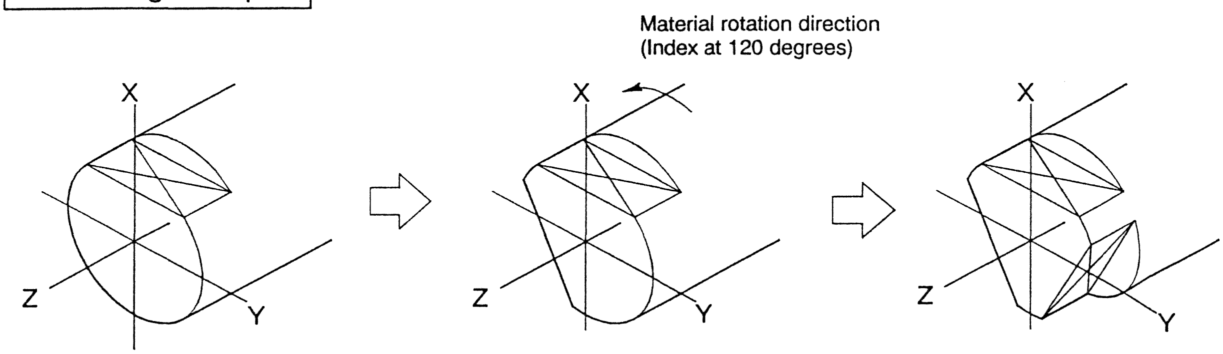
In the secondary process, the spindle can be indexed in 1-degree steps.
The following is an example of the spindle indexing.

Command format

M28 S□	Main spindle indexing
M20	Spindle indexing cancel

- Use these commands in the $1 axis control group program.
- The number specified in the S□ argument must be an absolute value between 0 and 359 divisible by 1.
- The spindle is indexed in the direction of forward rotation of the spindle.
- If the spindle is rotating, stop the spindle, then use M28 S0 to index the spindle.
- To return to turning, use M20 to cancel indexing, then use M03 S1 = □ to start rotating the spindle.

Machining example



Indexing at 0 degree Indexing at 120 degrees Indexing at 240 degrees

Program example

M05	.... Main spindle stop
G04 U0.5	.... Dwell
M28 S0	.... Spindle indexing command
G98 M58 S3 = □	.... Tool spindle forward rotation, per minute feed
⋮	Machining program
M60	.... Tool spindle stop
M20	.... Tool spindle indexing cancel
G4 U0.5	.... Dwell
M03 S1 = □ G99	.... Spindle forward rotation, per rotation feed

6.2.2 Back Spindle Indexing (M78, M79) (Optional)

The back spindle can be indexed in 1-degree steps.

Command format

M78 S☐	Back spindle indexing cancel
M79	Back spindle indexing cancel

- Use these commands in the $2 axis control group program.
- The number specified in the S☐ argument must be an absolute value between 0 and 359 divisible by 1.
- The spindle is indexed in the direction of forward rotation of the back spindle.
- If the spindle is rotating, stop the spindle, then use M78 S0 to index the spindle.
- To return to turning, use M79 to cancel indexing, then use M23 S2 = ☐ to start rotating the spindle.

Program example

```
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 rotate forward
```

6.2.3 Main Spindle C-Axis Indexing (M18) (Optional)

The main spindle C axis command can be specified in the secondary process. (In 0.001° units)
The command turns the spindle C axis on , indexes zero point C0, set that position as C0 in the coordinate system.

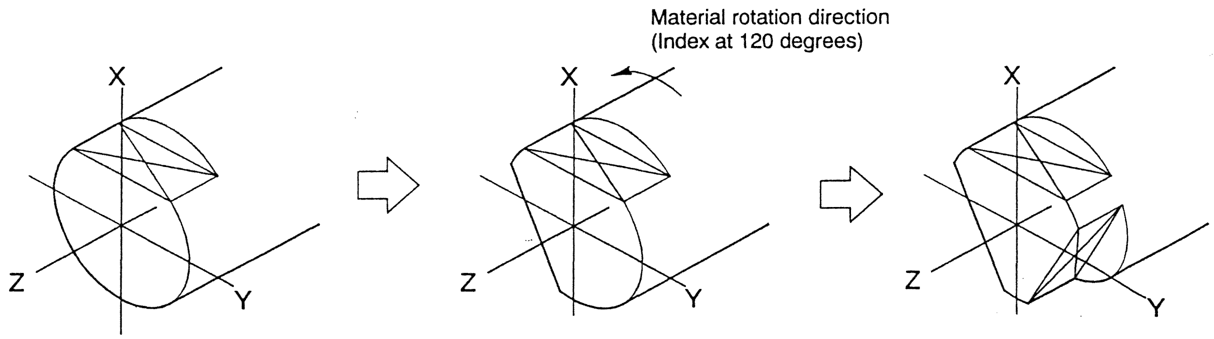
Command format

M18 C□ (Command for a series of main spindle C axis operations)

- Use these command in the $1 axis control group program.
- 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 +(plus) sign, and the spindle reverse rotation as "reverse rotation" by a -(minus) 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 +(plus) sign is omitted.
- After zero point "0" is indexed, any indexing command can be specified.
  The indexing can be specified in the range of 0.001 to 99999.999 degrees.
- After the M18C□ 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



Indexing at 0 degree   Indexing at 120 degrees   Indexing at 240 degrees

Program example

M05	.... Main spindle stop
M18 C0	.... Indexing command
G98 M58 S3 = ☐	.... Tool spindle forward rotation, per minute feed
⋮	Machining program
M60	.... Tool spindle stop
M20	.... Main spindle C-Axis cancel
M03 S1 = ☐ G99	.... Spindle forward rotation, per minute feed

- Indexing can be specified in the range of 0.001 to 99999.999 degrees.
- To return from the secondary process to turning, use M20 to cancel the C axis, then use M03 S1 = ☐ to start rotating the spindle.

[Commands for indexing]

●Absolute commands

M18 C0

↓

M18 C120.001

↓

M18 C240.0

●Incremental command

M18 C0

↓

G0 H120.0

↓

G1 H20.0 F☐

6.2.4 Back Spindle C Axis Indexing (M48) (Optional)

In the secondary process, the back spindle C axis can be indexed.
(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□ (Command for a series of back spindle C axis operations)

- Use this command in the $2 axis control group program.
- 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 + (plus) sign, and the spindle reverse rotation as "reverse rotation" by a - (minus) 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 +(plus) sign is omitted.
- After zero point "0" is indexed, any indexing command can be specified.
  The indexing can be specified in the range of 0.001 to 99999.999 degrees.
- After the M48 C□ command is specified, the usual C axis command becomes effective.

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

G00 C□ (H□)

G01 C□ (H□) F□

6.3 Calculating the Command Speed

6.3.1 Calculating the Spindle Speed Value

The spindle speed value is calculated using the following formula (A).

$$N = \frac{V}{\pi D} \times 1000 \;\ldots\ldots\ldots\ldots\; (A)$$

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

[Example]

Secondary process spindle

Use an 4-mm diameter end mill to machine a material under the following conditions;

Material: Brass

Cutting speed: 25 (m/min)

Tool diameter: ϕ 4 (mm)

The spindle speed is obtained from formula (A) as follows:

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

- The result of the calculation is rounded up to a whole number value.
- If the result of the speed calculation is more than the maximum speed, set the maximum speed (5,000 min^{-1}).

6.3.2 Calculating the Tool Feed Rate

The tool feed rate (mm/min) is calculated using either of the following formulas based on the calculation results of formula (A). Note, however, that the formula to be used depends on the tool used.

Tool feed rate formula:

■End mill and slitting cutter

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

F : Feed rate in mm/min
N : Spindle speed
fz : Feed amount per cutter
Z : Cutters (cutter/rev)

■Drill

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

F : Feed rate in mm/min
N : Spindle speed
fr : Feed amount per revolution

[Example]

Secondary process spindle

Use an 4-mm diameter end mill to machine a material under the following conditions;

Material	: Brass
Spindle speed	: 2000 (min^{-1})
Feed rate per tool	: 0.065 (mm/tool)
Number of tools	: 2 (teeth)

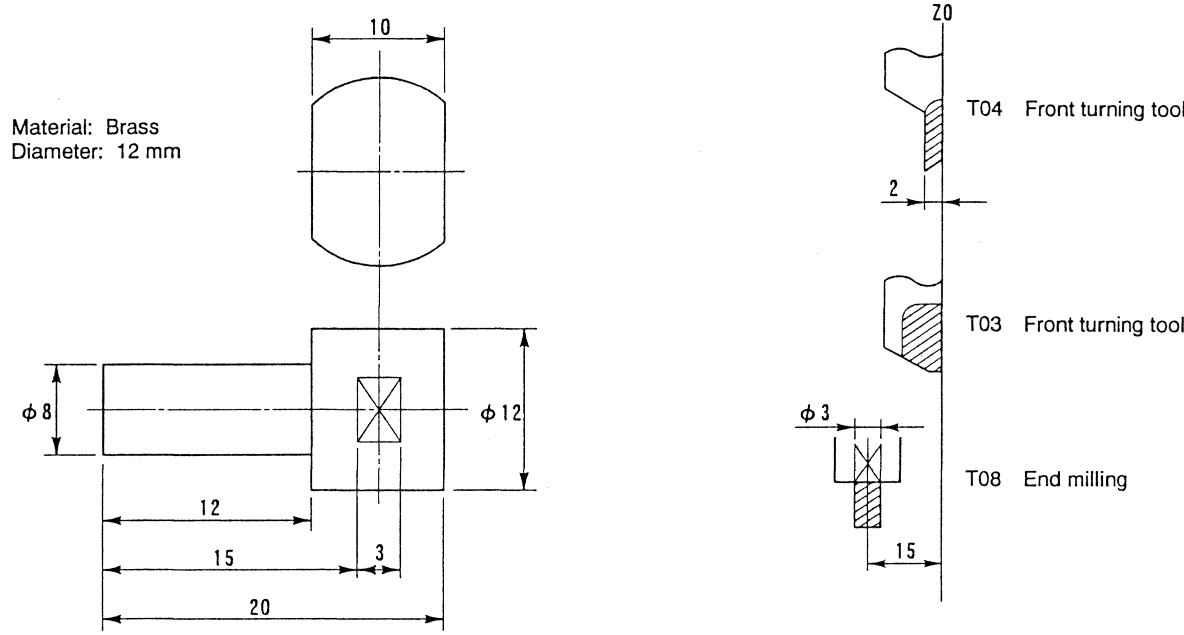
The tool feed rate is obtained from formula (B) as follows:

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

6.4 Two-Surface Width Machining (Including Indexing) Program

This example is a portion of one program relating to the secondary process.

Machining layout drawing



Program example

T0800	.... Tool selection
G50 W-15.0	.... Longitudinal coordinate system shift
M05	
G04 U0.5	
M28 S0	.... Main spindle indexing command
M58 S3 = □ G98	.... Tool spindle rotation command
G00 Y10.6 Z16.5 T□	.... Y and Z axis rapid feed positioning
X10.0	.... X axis positioning
G01 Y-10.6 F□	.... Two-surface width single machining
M28 S180	.... 180-degrees indexing
Y10.6	.... Remaining single machining
G00 X13.0 M60	.... Move in the diametrical direction. (Tool spindle rotation OFF)
M20	.... Main spindle indexing cancel command
G04 U0.5	
G50 W15.0	.... Longitudinal coordinate system shift cancel
M03 S1 = □ G99	

Calculation and explanation are shown in the next page.

[Calculation and explanation]

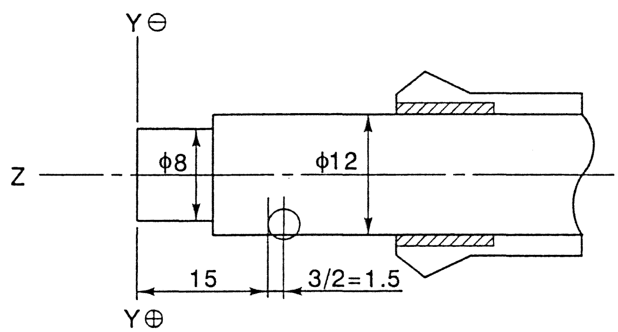
Spindle rotation "N" is given by the following expression.

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

Therefore, end milling feed rate "F" is given by

$F = 2700 \times 0.05 \times 2 = 270$ (mm/min)

The longitudinal direction position is set as shown in the figure below.

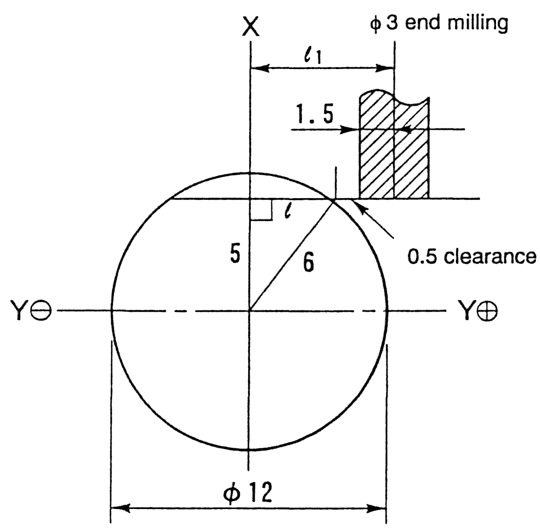


Therefore, the Z coordinates for positioning are as follows:

15+3 (end milling diametet) ÷ 2 = 15+1.5 = 16.5

As shown below, the Y coordinates (diameter specification) are given by the following expression.

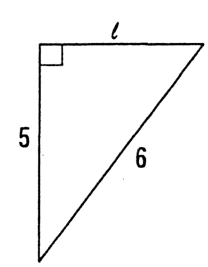
$Y = (\ell+0.5+1.5) \times 2 = (3.32+0.5+1.5) \times 2 = 10.64 \cong 10.6$



ℓ_1(mm) in the figure is given according to the Pythagoras theorem.

$\ell = \sqrt{6^2 - 5^2} = \sqrt{11} = 3.32$

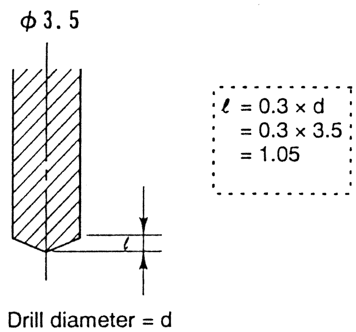
$\ell_1 = 3.32 + 0.5 + 1.5 = 5.32$



6.5 Through-Hole Machining

Explanation of this section is an example of machining which pierce the material of ϕ 16.0 with a cross drill of ϕ 3.5.

The X1 axis stroke of a gang tool is a maximum of X-5.0. The ordinary tool set cannot drill the through-hole of the material. Therefore, the tool set must shift the drill in the minus (-) direction.



The X value of a program that considers no shift is -16.0 - (1.05 × 2) = -18.1.

If X-18.1 is specified, an X axis overrun occurs.

Since there is a stroke in X-5.0, the X value becomes 18.1 - 5.0 = 13.1. The drill is shifted so that the through-hole machining can be performed when X-5.0 is specified to the coordinate system shift. In this case, the drill is shifted 15.0 mm in diameter (satisfactory if shifted 13.1 mm).

Notes

- Enter the shift amount on the Tool Set screen, then set. Return the entered data to "0" when tool setting is completed.
- Before executing [POS PNT], remove the material. or it may break the tool currently being set.
- Set the program by "G50U-□□ (amount to be lengthened <diameter value>) W-15.0" when it is shifted.
- Cancel the coordinate system shift after the shifted tool machining is completed.
- To avoid potential risk of interference between the material and tools when a tool is selected, either specify an argument after the tool selection T code (see "4.2.2 T Code Commands and T Code Arguments") or change the tool positioning point defined as machining data.

6.6 Synchronous Tap Machining (G88, G80) (Optional)

This function enables tapping of the outer circumference of materials with high tap depth precision. This function is synchronized with the tool spindle and X1 axis for tap machining (at the cutting and feed command for the X1 axis) in a tap cycle.

Command format

G88 X□ R□ F□ D□ S□ ,R1 (Tap cycle)

G80 (Tap cycle cancel)

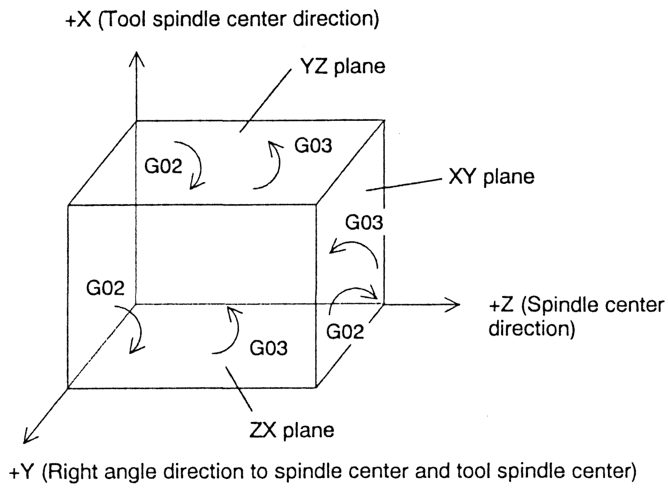
For details and examples, see Section 7.10.

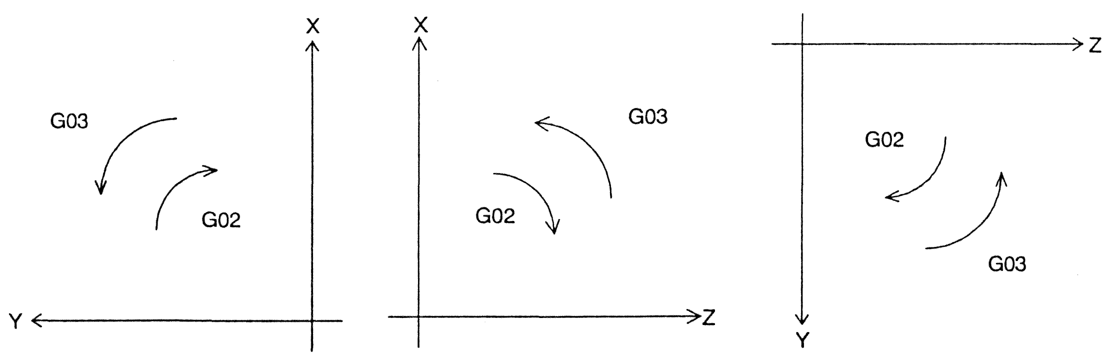
6.7 Circular Interpolation

Circular interpolation feed for the secondary process is available on three planes:

Command format

G17	XY plane
G18	ZX plane (Plane selection mode for normal turning)
G19	YZ plane





G17 (X-Y) plane G18 (Z-X) plane G19 (Y-Z) plane

6.7.1 Plane Selection

When the machine is turned on or after resetting, the ZX plane is selected. (G18)

The circular interpolations (G02 and G03) in turning are also performed on this plane.

If another plane is required for circular interpolation in the secondary process, enter the plane selection command G17 (XY plane) or G19 (YZ plane).

Select ZX plane (G18) again to return to turning.

6.7.2 Tool Diameter Compensation

The tool diameter compensation is programmed using a tool nose R compensation function.

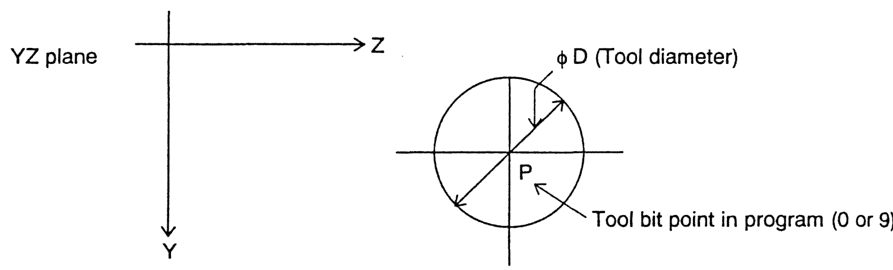
Command format

G40 Tool diameter compensation (tool nose R compensation) mode cancel

G41 Tool diameter compensation left (tool nose R compensation left) mode ON

G42 Tool diameter compensation right (tool nose R compensation right) mode ON

The relation between the tool diameter and the cutting position in a program is described below.
In the registration item of the tool diameter compensation, preset to "0" or "9" to the virtual tool nose number.

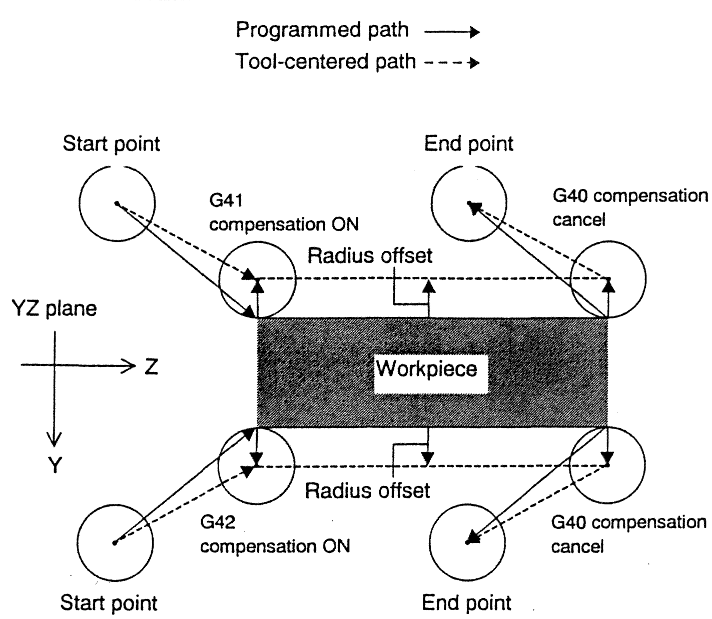


The tool cutting point and compensation are as shown below.

G41 : Offset to the left for the tool advance direction.

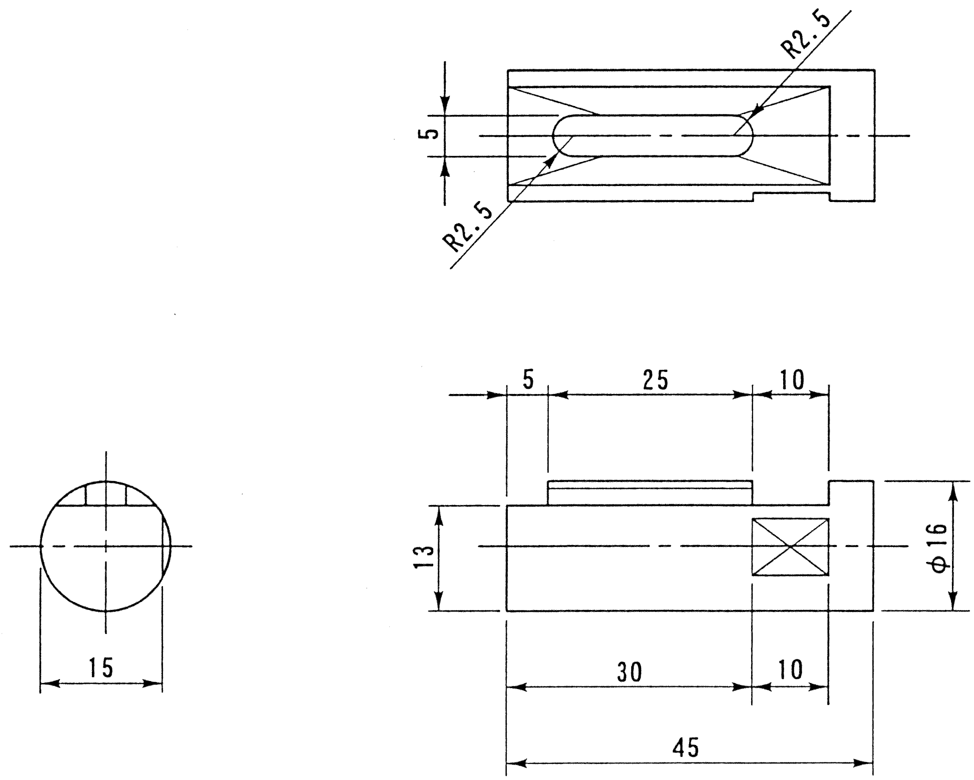
G42 : Offset to the right for the tool advance direction.

G41 command

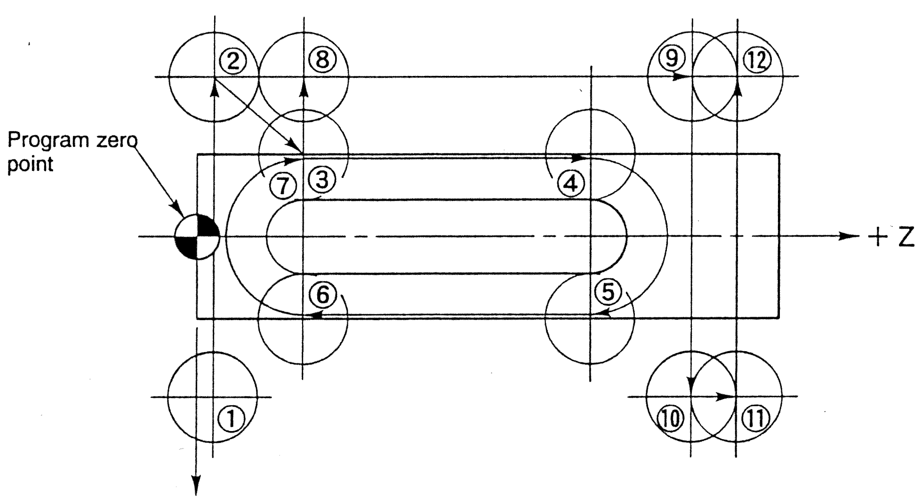


G42 command

Machining layout



Tool data
Tool diameter φ 6 (End milling)
Tool No. T08
Compensation No. 18
Nose radius (Tool radius value) 3.0
Virtual tool nose No. 0
Numbers ① to ⑫ in the figure below indicate the end milling movement by a program



Program example

This program is described in the $1 program.

```
$1
T0808                   ..........Select and compensate the tool spindle.
M05                     ..........Main spindle stop
G50 W-15.0              ..........Longitudinal coordinate system shift.
M28 S0                  ..........Index the spindle by 0°.
S3 = □ M58 G98          ..........Rotate the tool spindle in the forward direction.
G19 G00 Y22.0 Z1.0      ..........Set the plane to YZ and position the tool to the cutting start point in the
                                  YZ plane.
X10.0                   ..........Position in the cutting direction (X axis).
G01 Y-22.0 F□
G41 Y-5.0 Z7.5          ..........Compensate the tool diameter.
Z27.5
G02 Y5.0 R2.5 F□
G01 Z7.5 F□
G02 Y-5.0 R2.5 F□
G40 G1 Y-22.0 F□        ..........Cancel the tool diameter compensation.
G00 Z34.0
G01 Y22.0 F□
G00 Z37.0
G01 Y-22.0 F□
M28 S90                 ..........Index the spindle by 90°.
G00 X14.0
Y-18.0 Z33.0
G01 Y18.0 F□
G00 Z37.0
G01 Y-18.0 F□
G18 G00 X18.0 T00       ..........Return the plane command to the turning mode, cancel the
                                  compensation, and return the tool to the positioning point in the
                                  diametrical direction.
M60 G99                 ..........Stop the tool spindle.
G50 W15.0               ..........Longitudinal coordinate system shift cancel.
```

6.8 Front Face Tool Spindle (Optional)

The front tool spindle is a device that can rotate T22 and T23 among the three tools on the opposite tool post. The device can serve for eccentric drilling with a drill and end mill. The device can accept a standard drill sleeve (ϕ19.05) using a sleeve adapter.

6.8.1 Front/Back Face Tool Spindle Rotation and Stop (M80, M81, and M82)

These codes are used for rotating (forward and reverse) and stopping of the front/back face tool spindle in a secondary process.

The forward rotation in this case means the counterclockwise rotation toward the front/back tool spindle.

The front/back tool spindle stop can be manually specified on the panel.

Command format

M80	Starts the front/back face tool spindle forward rotation.
M81	Starts the front/back face tool spindle reverse rotation.
M82	Stops the front/back face tool spindle rotation

The spindle speed changes when only an "S4 = $\boxed{}$" is specified after the M80 and M81 codes.

Maximum spindle speed is 5000 min^{-1}.

Note

To use the front/back tool spindle detached, use the sleeve adapter or cap to block the hole not to let oil in.

6.8.2 Synchronous Tapping with Front/Back Face Tool Spindle (G84, G80)

The synchronous tapping functions for workpiece end faces (centered or eccentric) enables tapping with high tap depth precision. This function performs tapping by controlling synchronization between the front/back tool spindle and Z1 axis while stopping the rotation of the main/back spindle.

Command format

G80 Tap cycle cancel

G84 Z□ R□ F□ D4 S□ ,R1 Tap cycle

- ,R1 — Synchronous tapping
- S□ — Spindle speed
- D4 — Specifies the front/back tool spindle. (D-4 specifies left-handed threading.)
- F□ — Pitch
- R□ — Distance from Z-axis positioning point to the synchronous tapping start position
- Z□ — Tapping end position

Program example

```
↓
M5
S4=3000 M80 G98
M28 S0
G821

N122 T2200
M140
G01 X-3.0 Z-1.0
G1 Z7.0 F150 T22
G0 Z-2.0 T00 S4=500

N223 T2300
G0X-3.0 Z-2.0
G88 Z5.0 R1.0 F0.5 D4 S500,
R1
G80
M82
M20
M141
G820
M3 S1□
↓
```

6.9 Differential Speed Rotary Tool Function (Optional)

This function controls differential rotation by superimposing the speed of one spindle on that of the other spindle.

Use this function to superimpose the rotation of a spindle on that of a face tool spindle, for example, when you want to tap the center of a workpiece using the face tool spindle while rotating the workpiece clamped on the main spindle.

Command format

G164 H1 D4

D4 — Select the spindle to be superimposed. (D-4 specifies left-handed threading.)

H1 — Select the reference spindle.

G113 Cancel the differential rotary tool function.

Explanation

The G164 command specifies a reference spindle and the spindle to be superimposed on the reference spindle and superimposes the two spindles. The G113 command releases the two spindles from being superimposed, which are rotating under superimpose control by the differential speed rotary tool command.

Note

- If emergency stop occurs, the machine stops rotating the spindles under control of the differential speed rotary tool function and cancels the differential speed rotary tool mode.
- When using the command for this function, pay attention to spindle speed clamping. If the spindle speed of the superimposed spindle has been clamped, it cannot keep the difference in spindle speed from the reference spindle.
- The reference spindle cannot be indexed in the differential rotary tool mode. Cancel the differential rotary tool mode before indexing.
- An alarm occurs if the speed of the spindle is clamped when a synchronous tap command is issued in the differential rotary tool mode.

```
M3 S1 = 4000 ;
G164 H1 D-4 ;                           Differential speed rotary tool
G0 Z-2. ;
G84 X0. Z10. R1. F1. D4 S2000, R1 ;     Synchronous tapping
G113 ;
```

The above example clamps the superimposed spindle to a clamp speed of 5000 mm^{-1} because the spindle speed changes from 2000 mm^{-1} (tap cutting) to 6000 mm^{-1} (tap return operation), resulting in an alarm before tap cutting is started.

(Blank page)

Product code	C	–	L	5	1	6	2	0	I	III	VII	/	/	/	–	5	7	0
Document code	2	E	1	–	0	6	0	2										
	2	E	2	–	0	6	0	1										

Chapter 7 Programming Practice (Back Machining and Others)

7.1 Back Machining Program 7-1
7.2 Back Machining Coordinates 7-2
7.2.1 Back Spindle Rotation and Stop (M23, M24, M25) 7-3
7.3 Pick-Off and Ending Process 7-4
7.3.1 Pick-Off Process 7-6
7.4 Back Machining Process 7-7
7.4.1 Back Machining Product Separation (M33 and M34) 7-11
7.4.2 Cut-off and Product Separation (M33 and M34) 7-18
7.4.3 Back Spindle Indexing Command (M78, M79) (Optional) 7-20
7.5 Automatic Bar Loader (Optional) 7-21
7.6 Commands for Detecting Spindle Speed Changes 7-24
7.7 Constant Surface Speed Control Command 7-25
7.8 Tool Nose Radius Compensation Function 7-27
7.8.1 Tools Subject to Tool Nose Radius Compensation and Virtual Tool Nose Numbers 7-28
7.8.2 Basic Pattern of Tool Nose Radius Compensation G Code 7-29
7.9 Spindle Synchronization Control (G814, G114.1, G113) (Optional) 7-32
7.10 Synchronization Tap Machining (G88, G80) (Optional) 7-36
7.11 Interference Check Function 7-38
7.12 Back Face Tool Spindle (Optional) 7-39
7.12.1 Front/Back Face Tool Spindle Rotation and Stop (M80, M81, and M82) 7-39
7.13 Milling Interpolation — Optional 7-40
7.13.1 Milling coordinates 7-40
7.13.2 Milling plane 7-41
7.3.13 Programming of a milling process 7-42
7.13.4 Calculation of milling coordinates 7-44
7.13.5 Program format 7-46
7.13.6 Milling compensation 7-47
7.13.7 List of NC unit errors during milling interpolation 7-48
7.13.8 Changing the value of a parameter for milling interpolation 7-49
7.13.9 Example of using the milling interpolation function 1 (D cut) 7-51
7.13.10 Example of using the milling interpolation function 2 7-53
7.13.11 Example of using the milling interpolation function 3 7-55
7.13.12 Example of using the milling interpolation function 4 7-59
7.13.13 Example of using the milling interpolation function 5 7-62
7.13.14 Example of using the milling interpolation function 6 7-65

7.1 Back Machining Program

The back machining is performed by combining a front machining program ($1) and back machining program ($2).

Program configuration example

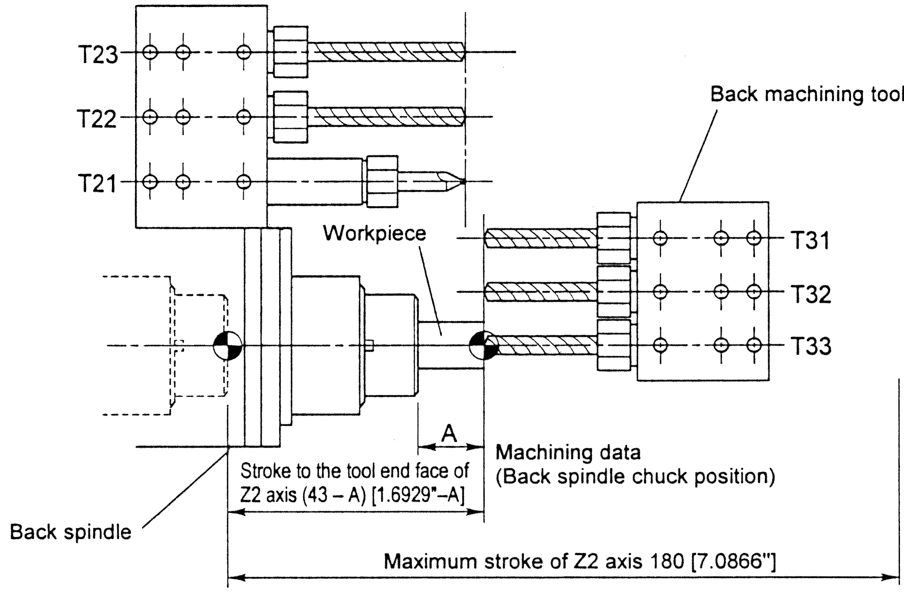
$1	$2	
Preparation process	Preparation process	The back spindle chucks the workpiece machined in the previous cycle.
Front machining process (Inner diameter)		Single machining (Synchronization control))
Front machining process (Outer diameter)	Front machining process (Inner diameter)	Outer/inner diameter simultaneous machining (Superimpose control)
Front machining process (Outer diameter)	Back machining	Independent machining in each axis control group.
	Workpiece separation	Ejects the completed workpiece.
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 program is not however be executed unless the software key [Last PRT] at the bottom of the screen is set to ON. The program is executed when [Last PRT] in the first cycle has been ON.

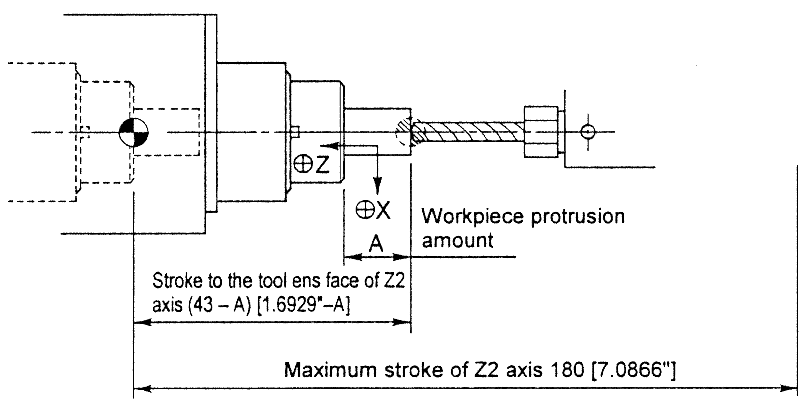
7.2 Back Machining Coordinates

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 "3.6 Coordinates" can be applied as the coordinate axis. Programming is performed on the assumption that the material is fixed.



Notes

- The length of the workpiece that protrudes from the end face of the back spindle is 30 mm (1.18") (maximum). For more than 30 mm (1.18"), the workpiece interferes with the cover of the work separator (work chute) during workpiece separation.
- The length of the workpiece protruding from the back spindle must be entered in the back spindle chuck position in machining data.
  (Protruding workpiece length = entire workpiece length - programmed back chuck 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.

7.2.1 Back Spindle Rotation and Stop (M23, M24, M25)

Following codes are used for rotation (forward rotation and reverse rotation) and stop of the back spindle.

Forward rotation means counterclockwise rotation relative to the spindle.

To stop the rotation of the back spindle, you can also use the Back Spindle stop key on the operation panel.

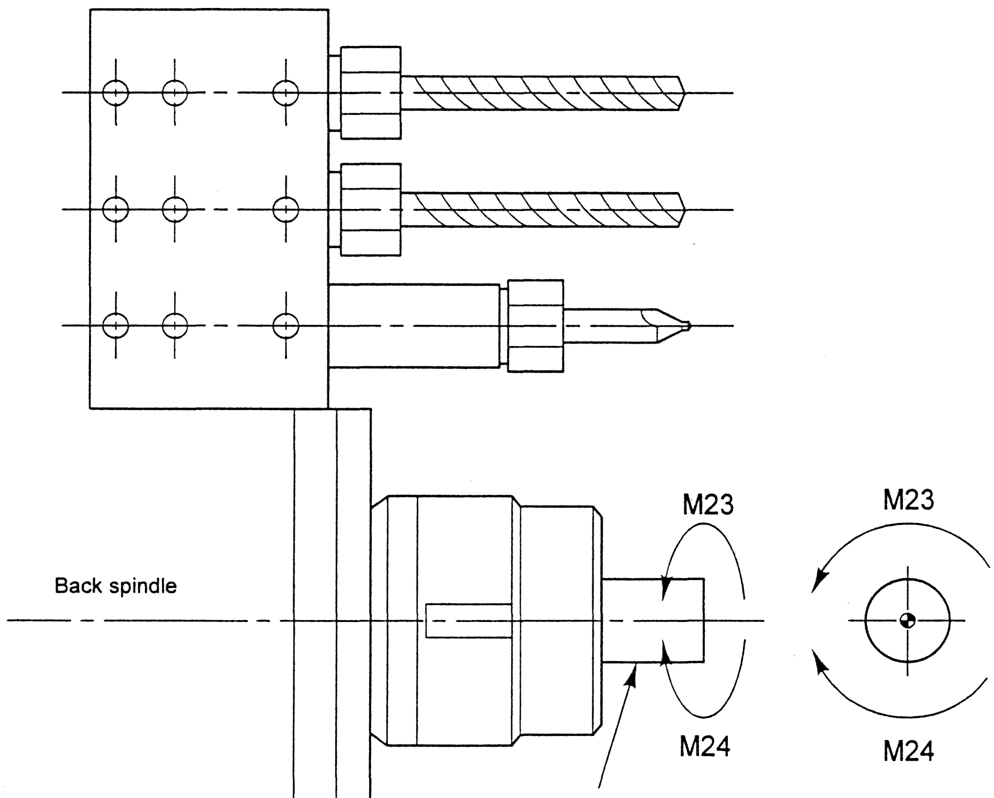
Command format

M23 Back spindle forward rotation

M24 Back spindle reverse rotation

M25 Back spindle rotation stop

The rotation count changes when "S2 = □□□□□□□□□□□□□" is specified during rotation of the back spindle.



Note

- A workpiece receiver box may have been mounted when the back spindle chuck is opened. Therefore, the back spindle speed is limited to 100 min^{-1} The back spindle rotates only at 100 min^{-1} even if M23S2 = 2000 is specified.
  However, the workpiece receiver box is rarely mounted when one of the codes of T3000, T3100 to T3300 is specified. In this case, the limitation is canceled.

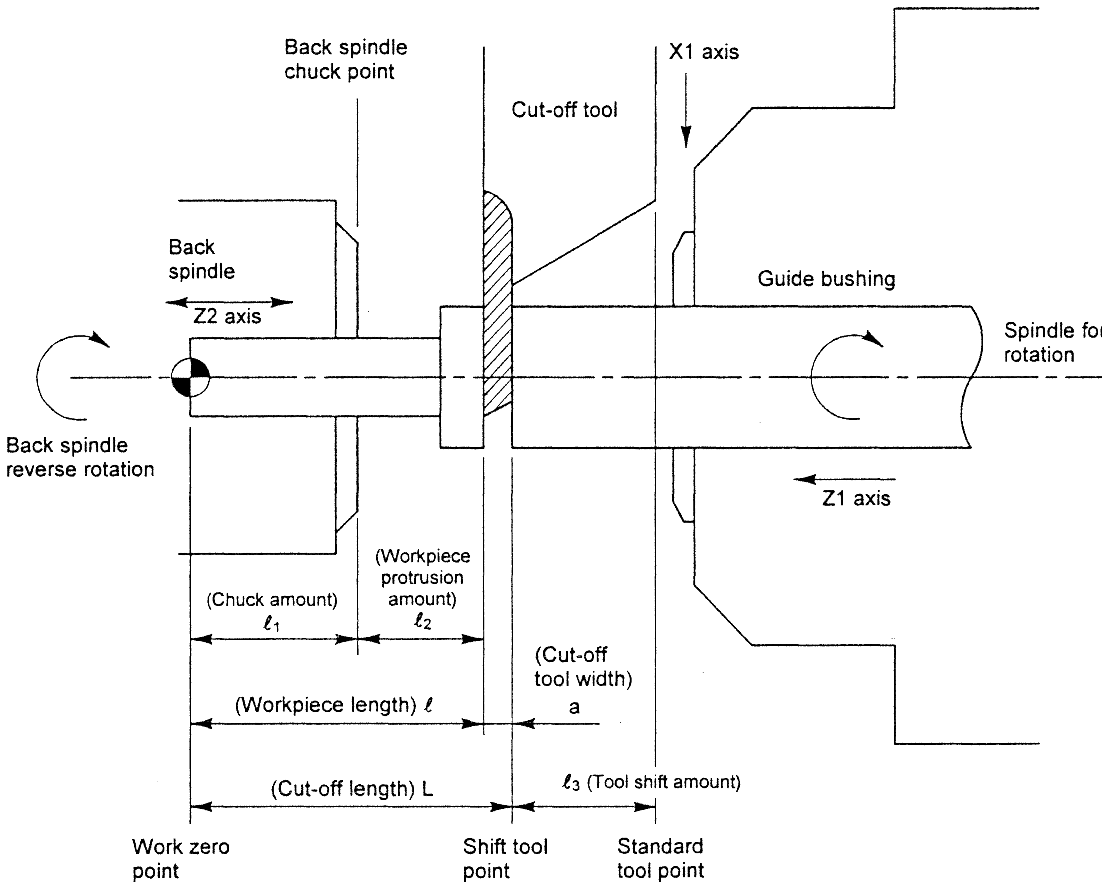
  To enable the limitation again, select, press the reset 

 key or specify the M2 code.

7.3 Pick-Off and Ending Process

The back spindle picks off the workpiece in a cut-off process.

After that, the back spindle returns to the return position (Z2-axis machine zero point) to execute the ending process.



Explanation

1. Select the cut-off tool.
2. Select the back spindle.
3. Rotate the main spindle in the forward direction and the back spindle in the reverse direction at the same speed.
4. Open the back spindle chuck.
5. Move the Z1 axis to the cut-off position. The shift amount (ℓ_3) and cut-off tool width (a) of the cut-off tool must be considered in this case. As a result, the following expression is obtained.

   Z =ℓ (workpiece length) + a (cut-off tool width) + ℓ_3 (tool shift amount)
6. Move the back spindle to the back spindle chuck point. (Z2 axis)
7. Make the back spindle chuck.
8. Cut off the workpiece by the cut-off tool.
9. Return the back spindle to the return position.
10. Execute the ending process.

Notes

- Advance the back spindle up to the point 10 mm from the guide bushing. The most advanced position of the back spindle depends on the size of the tool and on the tool holder.
  The most advanced position of the back spindle is 10 mm (0.39") from the standard tool point.
  Therefore, the tool shift amount (ℓ_3) is determined in consideration of the above limitation.
- When cutting off the workpiece with chucking the back spindle, perform the reverse rotation command (M24) if the main spindle rotates in the forward direction (M03). Also use a forward rotation command (M23) if the main spindle rotates in the reverse direction (M04).

7.3.1 Pick-Off Process

Program example

	$1		$2
			T3000
①	**M03 S1 = 2950**		**M24 S2 = 3000 G98**
②	**T0100**		
③	**G00 X7.0 Z32.0 T11**		
④	**G811**	④	**G811**
⑩	**!L1**	⑤	**G00 Z-1.0**
⑪	**G01 X-0.5 F0.015**	⑥	**G01 Z20.0 F1000**
⑫	**G810**	⑦	**G04 U0.5**
	X-3.0 F0.03	⑧	**M15**
	M05	⑨	**G04 U0.5**
	M07	⑩	**!L1**
	G0 Z0 T0	⑫	**G810**
	M56		**M25**
	G999		**G999**
	N999		**N999**
	M02		**M02**
	M99		**M99**
	%		**%**

(Explanation of each process)

①Adjust the main spindle speed to the back spindle speed. (Examine in advance by MDI.)
②Cut-off tool selection
③Cut-off tool positioning
④The Z2 axis is superimposed on the Z1 axis.
⑤The back spindle is positioned 1 mm before the end face of the workpiece in rapid feed.
⑥The back spindle advances 20 mm from the end face of the workpiece in cutting feed.
⑦Pause for 0.5 seconds
⑧Back spindle chuck closed
⑨Dwell for 0.5 second
⑩Queuing
⑪Cut-off machining
⑫Superimpose cancel. The Z2 axis moves to the zero point in rapid feed.

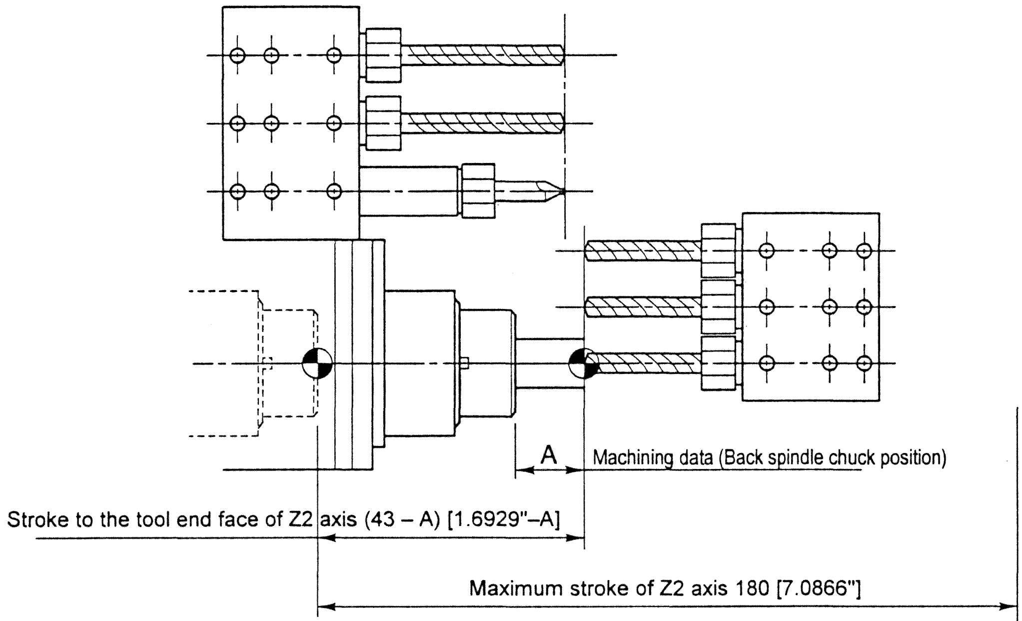
7.4 Back Machining Process

- The back inner diameter machining is performed by tools T31ΔΔ to T35ΔΔ. This machining is basically the same as for front machining.
- For Z2 program zero point after tool selection of T31ΔΔ to T35ΔΔ, the end face of the workpiece in the back spindle chuck position of machining data is prescribed as program zero point.
- The back machining tool is set within 70 mm (2.76") from the end face of a sleeve mounting block.
- If "Z-1.0" is specified after tool selection of T31ΔΔ to T35ΔΔ, 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.
  Interference may occur when the back machining sleeve protrudes more than 70 mm (2.76") during setting. Pay attention when setting the tool.
- The amount of the workpiece that protrudes from the back spindle must be within 30 mm (1.18").
  For more than 30 mm (1.18"), the workpiece interferes with the cover when the opposite 3-spindle tool post advances in the G821 mode using an M140 command.
- The end face of the workpiece is assumed as the program zero point in the machining. Therefore, enter the amount protruding between the end face of the back spindle and the end face of the workpiece in the back spindle chuck position of machining data.
- The contents of the actual machining are the same as for the inner diameter machining of the opposite tool post.
- When using a per rotation feed command (G99) during back machining, the machining is performed by back spindle control ON command (G44) and is completed by back spindle control OFF command (G43).
- After machining is completed, return the back spindle to the return position (machine zero point of Z2 axis).
- The program is described in $2 programs.

Notes

- For back tapping, use G01.
  Die and tapping on the back machining using G32 requires an optional function for back spindle chasing.
- During back machining, tools T31and T32 can be moved into the back spindle up to the position 20 mm from the back spindle end face, 10 mm for the tool T33. Machining with the tool moved further into the back spindle results in interference between the cap nut of the adjacent tool sleeve and the cap nut of the back spindle.
- Boring by T33 cannot be performed because tools T21 , T31 and the back spindle and the workpiece separator (workpiece chute) interference each other.
- Boring up to the inside of back spindle may be limited with the stroke depending on the relation between the cap nut position and the cap nut outer diameter.
  ER11 and AR11 are recommended for the tools T31 to T33.

■ Back machining tool and back spindle positioning



<Back machining process>

- The end face of the workpiece is assumed as the program zero point in the machining.
- For program zero point, enter the amount protruding between the end face of the back spindle and the workpiece to the back spindle chuck position of machining data.
  (Back chucking position) = (whole workpiece length) – (programmed back chuck position)

Program example

(Center hole and drilling)

$2	
G44	.... Back spindle per rotation feed ON
G99 M23 S2 = □	.... Rotate the back spindle and the forward direction and per rotation feed.
T3□	.... Select the back machining tool.
G00 Z-1.0	.... Position the workpiece 1.0 mm away from the tool.
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 return position in rapid feed.
T3□ W20.0	.... Select the back machining tool in the position 20 mm away from the zero point of the Z2 axis.
G00 Z-1.0	.... Position the workpiece 1.0 mm away from the tool.
G01 Z15.0 F0.08 T□□	.... Cut to the specified depth in cutting feed.
G00 Z-1.0 T00	.... Cancel the compensation while returning to the return position in rapid feed.
G43	.... Back spindle per rotation feed OFF
M25	.... Stop the spindle after back machining is completed.
M34	.... A series of operations of product separation is done after back machining is completed.
T3000	.... Select the back spindle for the next machining.

Note

- Product separation is specified by M34 after back machining is completed.
  Each axis operates as follows during execution of M34.

  The Z2 axis returns to the zero point.

  ↓

  The X2 axis returns to the zero point.

  ↓

  The Z2 axis advances and moves to the workpiece separation position.

  ↓

  The Z2 axis returns to the zero point after product knock-out.

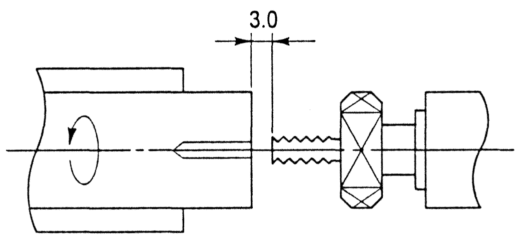
<Tap process> (Machining example of M6 P1.0 and 20 mm deep)

For back tapping, use G01 to specify tap machining.
Threading or tapping using G92 or G32 requires an optional function for back spindle chasing. This function enables you to specify back tap machining programs in the same way as front tap machining.

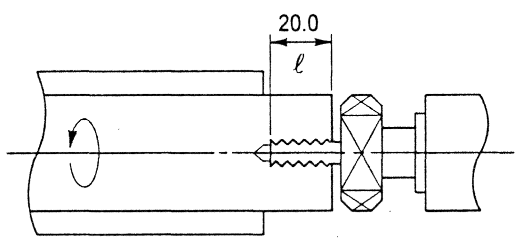
Note

For tapping using G01, set the override to 100%.

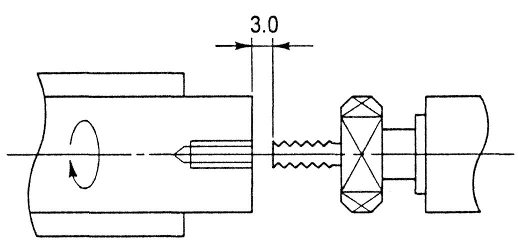
Machining



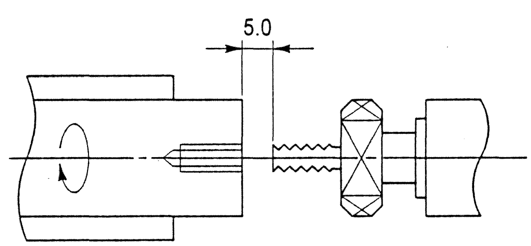
Program example	Explanation
M23 S2 = 300	Specify a back spindle forward rotation command.
G04 U0.5	Specify a dwell command of 0.5 seconds to stabilize the rotation.
G44	Back spindle per-rotation feed ON
T3300	Select tool No. 33.
G99 G00 Z-3.0	Position the workpiece 3 mm away from the tap end face in rapid feed.



Program example	Explanation
G01 Z16.0 F0.8 T□□	Advance the workpiece at a feed rate of pitch P × 0.8. (The feed rate is specified by the 80% and compensation number of screw length "ℓ".) (Z = ℓ × 0.8, F = P × 0.8)



Program example	Explanation
Z-3.0 F1.0 M24 T00	Return the workpiece from the tap end face by 3 mm at the feed rate of a pitch. Also specify a reverse rotation command simultaneously. (Compensation cancel)



Program example	Explanation
G04 U2.0	Wait until the tap comes out.
G00 Z-5.0	Keep the back spindle apart from the tap.
M23 S2 = ☐	Rotate the reverse back spindlein the forward direction.
G43	Back spindle per-rotation feed OFF

7.4.1 Back Machining Product Separation (M33 and M34)

This machine provides the following two modes for collecting (separating) machined workpieces as products.

1. Picking off the product by the back spindle to collect it in the product box.
   (Pick-off collection mode)
2. Putting the product in the workpiece receiver box mounted on the back spindle to collect it in the product box. (Workpiece receiver box collection mode)

Back-machined products are collected in mode 1; relatively simple products finished without back machining are collected in mode 2.

In general, the program using mode 1 is included in the program of \$2 (\$1 if the program is for a single axis control group) and the program using mode 2 is included in the program of \$1.

Although the M code for product separation in both of modes 1 and 2 is M34 (M33), the actual operations in the two modes are slightly different. The NC unit identifies modes 1 and 2 as described below.

- When the back spindle defined as machining data has a workpiece receiver box and the M34 code has been specified for \$1 or \$2:

  The machine recognizes the current collection mode as workpiece receiver box collection mode and operates in that mode.

- When the back spindle defined as machine data is a standard or support-provided type and the M34 code has been specified for \$1 or \$2:

  The machine recognizes the current collection mode as pick-off collection mode and operates in that mode.

Note

Even when the back spindle has been set as a spindle with workpiece receiver box in machining data, the current collection mode is recognized as the pick-off collection mode if M34 is specified after T3000 or T3100 to T3300. Pressing the reset button or specifying M2 cancels that mode.

When the back spindle chuck has been opened, a workpiece receiver box may have been mounted. The back spindle speed is therefore limited to a maximum of 100 min^{-1}. Even if M23S2 = 2000 is specified, for example, the back spindle rotates only at up to 100 min^{-1}.

If any of the tool selecting codes of T3000 or T3100 to T3300 has been specified, however, the spindle speed limit is released, regarding that the workpiece receiver box has been rarely mounted.

The spindle speed limit is enabled again either by pressing the reset button, or by specifying M2.

1. Method for picking off a workpiece by the back spindle and collecting it in a product box (pick-off collection mode)

   a) Back spindle product separation 1

Command format

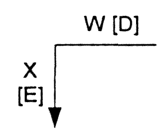
M33 Back spindle product separation 1 A series of operations without product knock-out.

Command format

M33 W0.0 D20000 X4.0 E20000

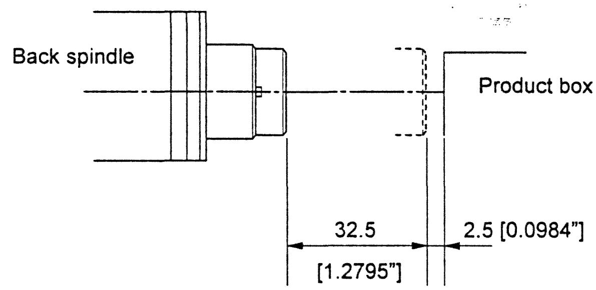
- E20000: Feed rate (per minute) when the X2 axis separation position moves. Moves the X2 axis in rapid feed when omitted.
- X4.0: Move position (mm) when the X2 axis separation position moves. Moves the X2 axis to the zero point when omitted.
- D20000: Feed rate (per minute) when the Z2 axis returns. Moves the Z2 axis in rapid feed when omitted.
- W0.0: Move position (distance from the zero point of the Z2 axis in mm) when the Z2 axis is retracted. Moves the Z2 axis to the zero point when omitted.

W☐ **D**☐ **X**☐ and **E**☐ can be omitted.



Program example

M33	.... A series of operations is performed for collecting the finished workpiece (without knock-out for the product). • If M33 has no argument specified, the X1 and Z2 axes move to the zero point and terminates operation with the back spindle chucking the workpiece. • After the X2 and Z2 axes movement is completed, the coordinate system setting of the Z2 axis is executed. Normally, the G50Z0 command is executed. If W∆∆ is specified, G50Z∆∆ is executed. Available with macro version 002-001 or later. • Specify a program that puts the finished workpiece in the product box in the next line of the M33 code.
G00 Z32.5	.... Move the back spindle to the workpiece separation position in rapid feed.
M16	.... Open the back spindle chuck.
M10	.... Advance the back spindle knock-out bar.
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 bar.
T3000	.... Select the back spindle at the center of the guide bushing.
M73	.... Turn off the air blow of the back spindle.



Notes

- The above program example selects T3000 after product separation.
- Compared to M34 for back spindle product separation 2, M33 allows individual operations for product separation to be programmed. M33 can therefore save product separation time by efficient programming.

b) Back spindle product separation 2

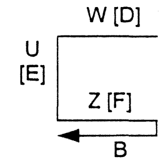
Command format

M34 Back spindle product separation 2
With knock-out for the product of a series of operations

Command format

M34 W0.0 D20000 U0.0 E20000 Z30.0 F20000 B0.0 T2.0

- T2.0: Air blow time after product knock-out. Blows air for two seconds when omitted.
- B0.0: Move position (mm) during the Z2 axis return after product knock-out. Moves the Z2 axis to the zero point when omitted.
- F20000: Feed rate (per minute) during the Z2 axis advance. Moves the Z2 axis in rapid feed when omitted.
- Z30.0: Move position (mm) during the Z2 axis advance. Moves the Z2 axis to the position of 32.5 mm when omitted.
- E20000: Feed rate (per minute) when the X2 axis separation position moves. Moves the X2 axis in rapid feed when omitted.
- U0.0: Move position when the X2 axis separation position moves.
- D20000: Feed rate (per minute) during the Z2 axis return. Moves the Z2 axis in rapid feed when omitted
- W0.0: Move position (distance form the zero point of the Z2 axis in mm) during the Z2 axis return. Moves the Z2 axis to the zero point when omitted.



W□ **D**□ **U**□ **E**□ **Z**□ **F**□ **B**□ **T** can be omitted.

Product knock-out macro is also incorporated in the M34 code.

Actions of commands in macros:

M16 .... Open the back spindle chuck.
M10 .... Advance the back spindle knock-out bar.
M11 .... Return the back spindle knock-out bar.
M31 .... Turn on the Workpiece conveyor.
Z0 .... Return the back spindle to the zero point.
M72 .... Turn on the air blow of the back spindle.
M10 .... Advance the back spindle knock-out bar. Advance to blow chips.
G04 U2.0 ....Air blowing time. The argument is T2.0 seconds.
M11 .... Return the back spindle knock-out bar.
M73 .... Turns off the air blow of the back spindle.

For M34, the operation described above is completed by only the M34 code.

Program example

M34 .... Complete a series of operations for the workpiece collection and product knock-out.

T3000 .... Select the back spindle at the center of the guide bushing.

Note

- The position to which the Z2 axis advances during knock-out operation is normally set to Z32.5 unless the Z argument has been specified. The position can be changed by specifying the Z argument.

2. Method for putting a product in the workpiece receiver box mounted on the back spindle to collect it in the product box (workpiece receiver box collection mode)

The program is described as part of the $1 program.

Command format

M32 Workpiece separator (workpiece receiver box) advance operation

Command format

M34 Back spindle (workpiece receiver box) product separation operations

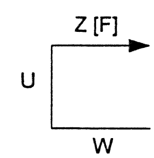
a) Workpiece separator (workpiece receiver box) advance operation

Command format

M32 X☐ W0.0 U632.0 Z145.0 R☐ F20000

- F: Feed rate (per minute) during Z2 axis advance. Feeds rapidly when omitted.
- R: Set "1" when a shift tool is used for the cut-off tool. Set nothing when the shift tool is not used. The Z2 axis advance position is 130 mm when "1" is set. It is 145 mm when omitted.
- Z: Move position (mm) during Z2 axis advance. Moves to the position of 145 mm when omitted. An R1 command is ignored when this is specified.
- U: Move position during X2 axis advance. Moves the X2 axis to the position of 632 mm when omitted.
- W: Move position (distance from the zero point of the Z2 axis in mm) during Z2-axis return. Moves to the zero point when omitted.
- X: End position queuing X position. The workpiece separator (workpiece receiver box) on the Z2 axis finishes advancing when the X axis reaches the specified position during cut-off machining. End position queuing is not performed when not specified.

X☐ W☐ U☐ Z☐ R☐ F☐ can be omitted.



Program example

$1	$2		
G821	**G821**	.... Single axis control group command ON	(Note 1)
T0100		.... Selects the cut-off tool.	
M32		.... Position the workpiece receiver box to the cut-off position.	
G1 X-3.0 F0.02		.... Cut-off machining	
M141		.... Return the workpiece receiver box.	
G820	**G820**	.... Single axis control group command OFF	(Note 1)

(Note 1) Omitting these codes does not affect the program.

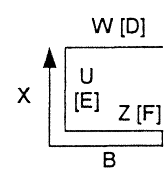
b) Back spindle (workpiece receiver box) product separation operation

Command format

M34 X632.0 W0.0 D20000 U0.0 E20000 Z54.0 F18000 T2.0 S30 B0.

- B: Move position during Z2 axis return after product carry-out. Moves the Z2 axis to the zero point when omitted.
- S: Back spindle speed during product carry-out. Rotates at 30 min^{-1} when omitted.
- T: Back spindle rotation time during product carry-out. Rotates for two seconds when omitted.
- F: Feed rate (per minute) during Z2 axis advance. Moves the Z2 axis in rapid feed when omitted (mm/min).
- Z: Move position during Z2 axis advance. Moves the Z2 axis to the position of 54 mm when omitted.
- E: Feed rate (per minute) when the X2 axis separation position moves. Moves the X2 axis in rapid feed when omitted.
- U: Move position (mm) when the X2 axis separation position moves. Move position during X2 axis return. Moves the X2 axis to the zero point when omitted.
- D: Feed rate (per minute) during Z2 axis return. Moves the Z2 axis in rapid feed when omitted.
- W: Move position (distance from the zero point of the Z2 axis in mm) during Z2 axis return. Moves to the zero point when omitted.
- X: Move position (mm) during X2 axis rise after workpiece separation. Moves the X2 axis to the position of 632 mm when omitted.

X☐ W☐ D☐ U☐ E☐ Z☐ F☐
T☐ S☐ B☐ can be omitted.



Program example

$1	$2	
G821	G821	.... Single axis control group command ON (Note 1)
T0200		
M34		.... Enter M34 after specifying the T0200 (first gang tool selection) after three-spindle hole machining.
:		
T0100		.... Select the cut-off tool.
M32		.... Position the workpiece receiver box at the cut-off position.
G1 X-3.0 F0.02		.... Cut-off machining
M141		.... Return the workpiece receiver box.
G820	G820	.... Single axis control group command OFF (Note 1)

Note

- If a command in the T3000s in the workpiece receiver box collection mode, the current mode is recognized as the pick-off collection mode and thus the basket in the workpiece receiver box will not swing, disabling product separation.

(Note 1) Omitting these codes does not affect the program.

7.4.2 Cut-off and Product Separation (M33 and M34)

The product is separated on the back spindle side.

Main spindle and back spindle rotation synchronization command (G814, 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.

Program example

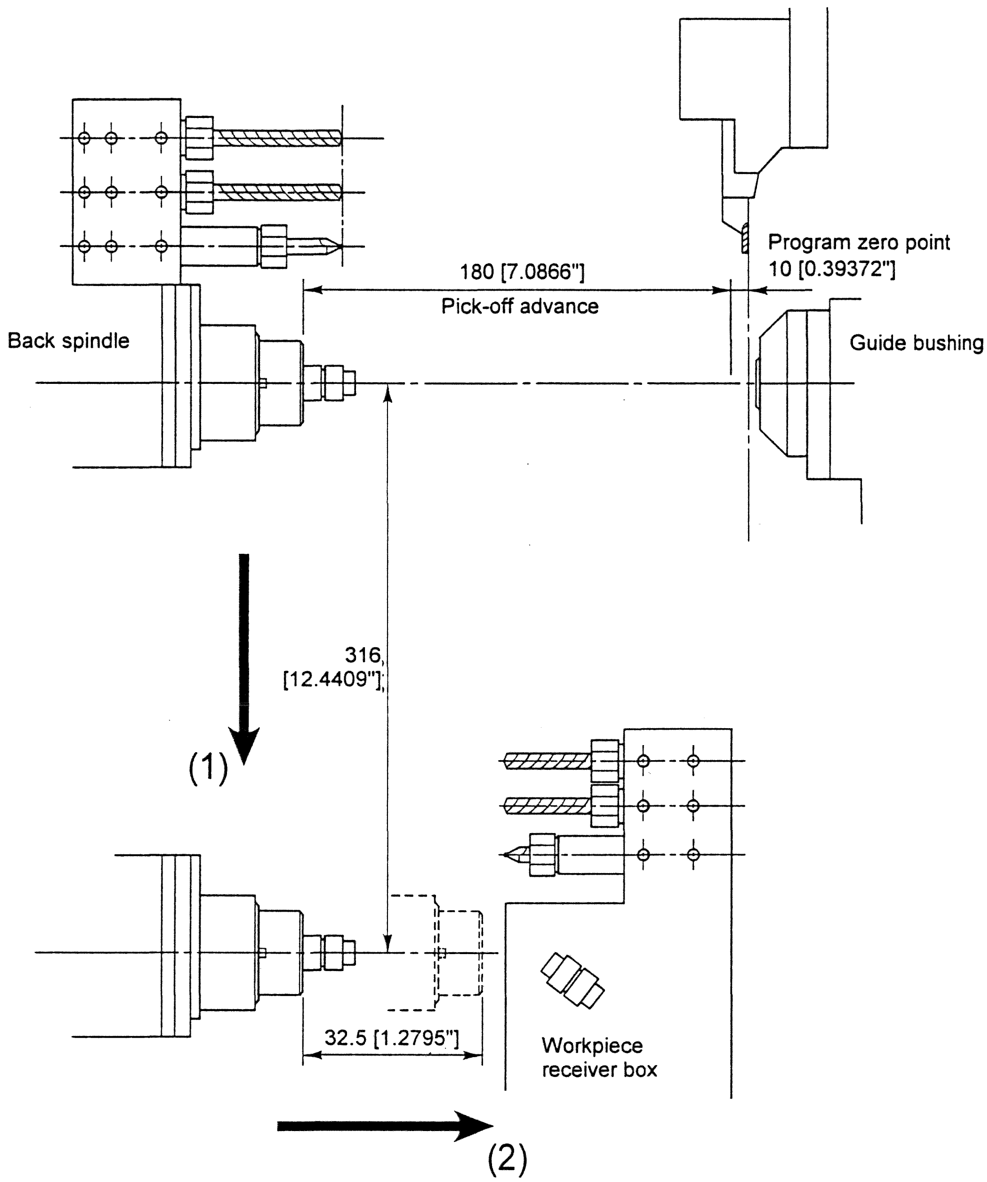
```
$1
M3 S1=1500
!L1 ............................ Queuing command
G814 ........................... Rotation synchronization of
                                 spindle and back spindle
                                 (Option)
N11 T0100 ...................... Cut-off tool selection
G00 X26.Z [*1] T01
................................ Cut-off tool positioning
G811 ........................... Automatic queuing
!L2 ............................ Queuing command
G99 G01 X-1.0 F0.03 Workpiece cut-off
G810 ........................... Automatic queuing
X-3.0 F0.03
G113 ........................... Spindle synchronization
                                 mode cancel
M05
M07
G00 Z [*3] T00 ................. Return to the start point.
M56
G999 ........................... Automatic queuing
N999 ........................... Automatic queuing
M02
M99
%
```

```
$2
M34 ............................ Specify a series of operations for
                                 the product separation.
T3000 .......................... Select the back spindle.
M16 ............................ Open the back spindle chuck.
M24 S2 = 1500 .................. Specify 1500 for the back
                                 spindle reverse rotation speed.
!L1
G811 ........................... Specify the superimpose control
                                 of outer/inner diameter
                                 simultaneous machining.
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.
G98 G01 Z [*2] F1000
................................ Advance the back spindle to the
                                 chuck position.
G04 U1.0 ....................... Specify a dwell pause command of
                                 one second in the chuck position.
M77 ............................ Wait until spindle
                                 synchronization is completed.
M15 ............................ Close the back spindle chuck.
M73 ............................ Turn off the air blow of the back
                                 spindle.
!L2
G810 ........................... Cancel the superimpose control
                                 of outer/inner diameter
                                 simultaneous machining.
M25 ............................ Stop the back spindle rotation.
G999
N999
M02
M99
%
```

- *1 : L (product length) + ℓ_3 (tool shift amount) + (tool width)
- *2 : ℓ_1 (chuck amount) Move distance required until the workpiece is chucked by the back spindle chuck
- *3 : ℓ_3 (tool shift amount) Return to the program start point.

■ Home position of product separation, and serial operation

The product after cut-off machining or back machining is collected to a workpiece receiver box at this stroke during this operation.



Explanation

- The following operations are executed when product separation is specified by M34.
  1) The Z2 axis returns to the zero point.
  2) The X2 axis returns to the zero point.
  3) The Z2 axis advances to the product separation position.
  4) The Z2 axis returns to the zero point after product knock-out.
- When an M33 command is issued, the X2 axis in step 2) returns to the zero point to complete the operation for product separation. After that, perform product separation by creating a program.
- The program is described in the $2 program.

7.4.3 Back Spindle Indexing Command (M78, M79) (Optional)

The back spindle can be indexed in 1-degree steps.

Command format

M78 S□	Back spindle indexing
M79	Back spindle indexing cancel

- Use these commands in the $2 axis control group program.
- The number specified in the S□ argument must be an absolute value between 0 and 359 divisible by 1.
- The spindle is indexed in the direction of forward rotation of the back spindle.
- If the spindle is rotating, stop the spindle, then use M78 S0 to index the spindle.
- To return to turning, use M79 to cancel indexing, then use M23 S2 = □ to start rotating the spindle.

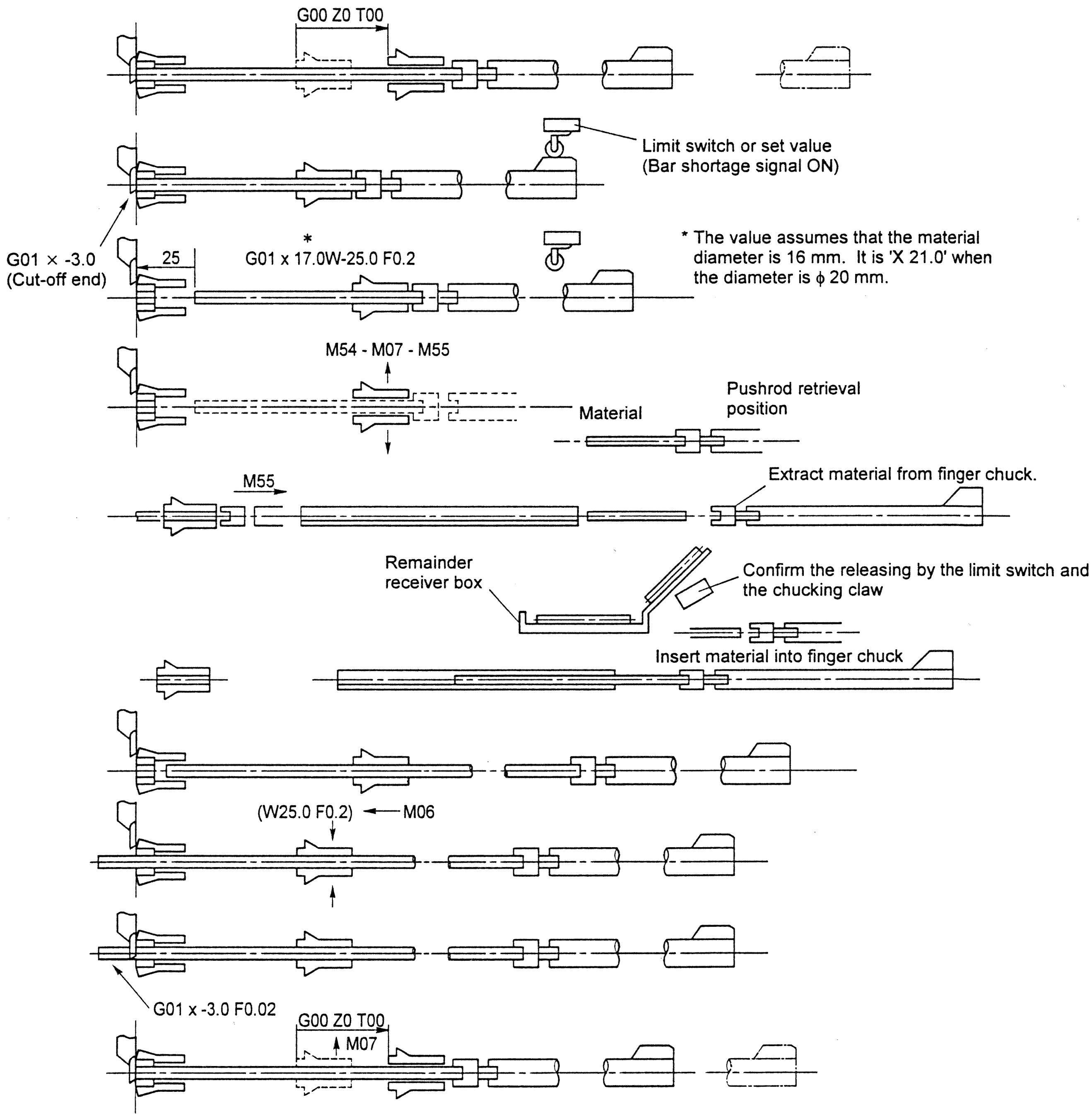
Program example

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 rotate forward

7.5 Automatic Bar Loader (Optional)

When automatically loading the material, the following automatic bar loading process must be specified between the cut-off process and the ending process in your program.

Operation drawing



Automatic Bar Loading Program code

O●●●●

$1	$2
Preparation process	Preparation process

Machining process

M09

Automatic bar loading program and explanation

N901 T0100

G00 X17.0 Z50.0 T□□

G01 X-3.0 F0.02

Code	Explanation
M08	Enable material replacing program.
M08	
/( )	Material outer diameter deburring program
/( )	
/( )	
/G01 X*17.0 W-25.0 F0.2	Extract material from guide bushing, and move cut-off tool to specified position (+ 1.0 mm) on material diameter. (See Note below.)
/M53	Stop coolant supply.
/M05	Stop main spindle rotation (Slow speed for profile materials).
/M54	Turn off the bar loader machining torque.
/M07	Open spindle chuck.
/M55	Material change command (to start replacing the material on the bar loader).
/M06	Close spindle chuck.
/G4 U2.0	Prevent time delay of bar loader torque.
/M52	Turn on coolant.
/M03 S1 = □	Rotate main spindle forward.
/M26	Execute guide bushing phase adjustment (for profile material).
/W25.0 F0.2	Insert material into guide bushing.
/X-3.0 F0.02	Cut end of material short.
M09	Terminate material replacing program. (Hereafter, blocks with a slash ("/") are ignored.)

M05

M07

T □

G00Z □ T00

$1	$2
G820	G820
M56	
G999	G999
N999	N999
M02	M02
M99	M99
%	%

Notes

- *X17.0 is the numerical value when the material diameter is ϕ 16.
  It is "X21.0" when the material diameter is ϕ 20.
- A program using machining data values (machine variables) can only be used with the macro option.
- Leave coolant OFF during material replacement. When a new material is loaded, set coolant to ON.
- Leave coolant OFF when the guide bushing has no material inside (removed for material replacement)
- Program a material tip deburr process as required.
- For machining profile materials with dedicated bar loader, set the machine structure parameter "profile material chuck sleeve/square material chuck sleeve" to ON (for inching of the bar insertion).

7.6 Commands for Detecting Spindle Speed Changes

The spindle speed change detection function monitors the spindle speed. If the speed change exceeds a preset rate, this function automatically stops the machine. This prevents operation overload and damage to the guide bushing baking.

Both main spindle and back spindle have this spindle speed change detection function.

Command Format

M97	Main spindle speed change detection OFF.
M96	Main spindle speed change detection ON
M87	Back spindle speed change detection OFF.
M86	Back spindle speed change detection ON.

Notes

- The main spindle speed change detection function is on by default when the power supply is turned on. To turn the detection function off, include the M97 or M87 command in the program.
  Normally, the program should contain M96 or M86 in the beginning to turn on the detection function for safety.
- To perform tap and die machining or constant surface speed control function, be sure to turn off the speed change detection.

Standard values

• Time before check start	4 seconds (Parameter PLC constant #6305)
• Alarm fluctuation rate	±10% (Parameter Spindle parameter #3024 <1> Main) (Parameter Spindle parameter #3024 <2> Back)

7.7 Constant Surface Speed Control Command

While the material is cut in a diametrical direction or while cutting off the workpiece after finishing a machining process, the diameter of the material may vary depending on the portion of a workpiece, and the surface speed (the relative speed of the workpiece and tool) may also change. The control unit can detect the tool position, calculate the spindle speed, and change the spindle speed accordingly by using this command to instruct a relative speed.

Command format

G50 S□ Q□ : Specifies the spindle speed limitation.

The speed is controlled so that the spindle speed does not exceed the limitation during constant surface speed control.

S : Maximum spindle speed (min^{-1}) clamp value

Q : Minimum spindle speed (min^{-1}) clamp value

G96 S□ : Starts constant surface speed control.

S : Constant surface speed value (m/min.)

G97 : Ends constant surface speed control.

Notes

- If no S code is given when switching G96 to G97, the S value previously specified in the respective mode will be effective.
- The spindle speed fluctuation detection cannot be turned on during constant surface speed control.
- G50 S□ is effective only when the constant surface speed control command is effective by entering the G96 command.
- Specify Q, when the minimum spindle speed may change too lower.
- The tool selection command (T□□□□) cannot be used during constant surface speed control.
- When selecting the tool, specify the tool selection command after a cancel command (G97) is specified.

Example

$1		$2	
Preparation process ↓ Front turning ↓ Thread cutting ↓ Other processes ↓			
(Constant surface speed control of spindle)		(Constant surface speed control of back spindle)	
		M87	Back spindle speed fluctuation detection OFF
		M23 S2 = 1000	
		G44	Back spindle per rotation feed ON
		T3200	
		G00 X16.0	
		G50 S3500	Maximum spindle rotation clamp (3,500 min^{-1})
		G96 S40	Constant surface speed control ON
		G01 X-1.0 F0.03	
		G97	Constant surface speed control OFF
		M25	Back spindle rotation stop
		G43	Back spindle per rotation feed OFF
N501 T0101	Cut-off tool selection		
G00 X21.0 Z50.0	Cut-off positioning by rapid feed		
M97	Main spindle speed change detection OFF		
G50 S5000	Maximum clamp set command of spindle (5,000 min^{-1})		
G96 S100	Constant surface speed control ON (Surface speed: 100 m/min)		
G01 X-3.0 F0.02	Cut-off		
G97 (S1 = 500)	Constant surface speed control cancel (Normal rotation command of 500 min^{-1})		
M96	Main spindle speed change detection ON		
M05			
M07			
G00 Z□ T00			
G820		**G820**	
M56			
G999		**G999**	
N999		**N999**	
M02		**M02**	
M99		**M99**	
%		**%**	

Note

- The above program does not cover product separation.

7.8 Tool Nose Radius Compensation Function

If a rounded tool-bit is used, the rounded tool nose can cause an error between the programmed form and the cutting form during taper cutting or circular cutting.

The tool nose R compensation function automatically calculates the error and compensates. The command code can fix the compensation direction.

Command format

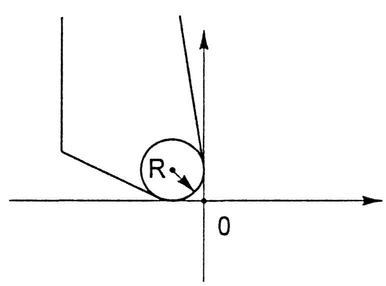
G40	Tool nose radius compensation mode Cancel.
G41	Tool nose radius compensation left mode ON.
G42	Tool nose radius compensation right mode ON.

Enter tool nose R data in advance in <R> (tool nose radius value) and <P> (virtual tool nose No.) on the Tool Data screen.

Note that the meaning of <R> on the Offset screen is different from <R> on the Tool Data screen.

Virtual tool nose

- The virtual tool nose is the point of a nonexistent tip of the tool, corresponding to the zero point shown below.
- Be sure to set the tool bit in the holder as shown in the diagram.

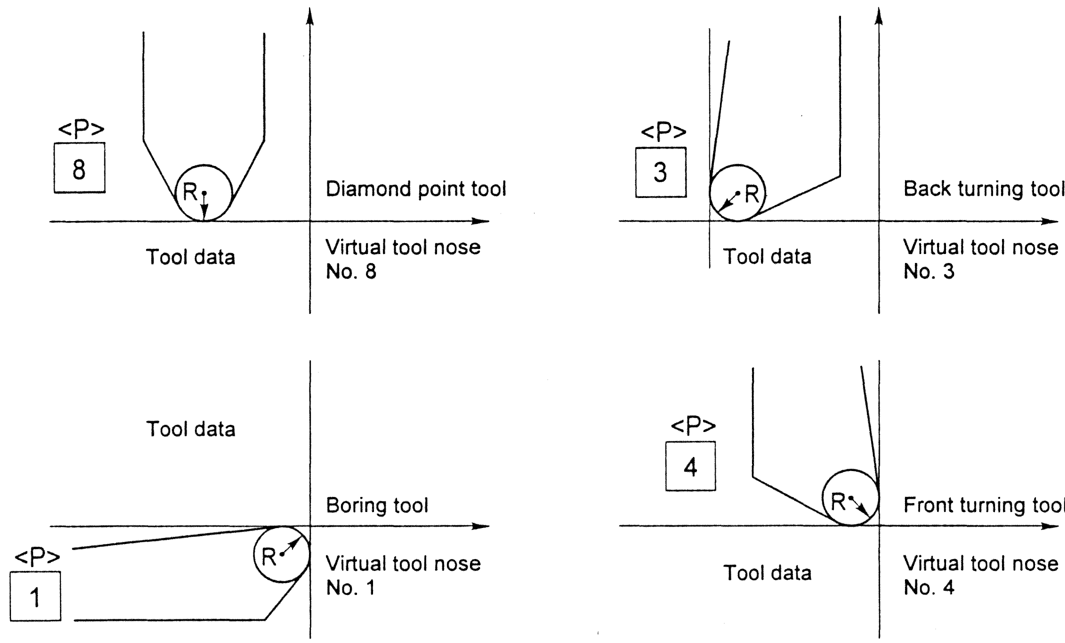


Virtual tool nose number

- The direction of the virtual tool nose viewed from the tool nose radius center is determined as the virtual tool nose number.
- Tool nose between 0 and 9 is determined according to tool noses.

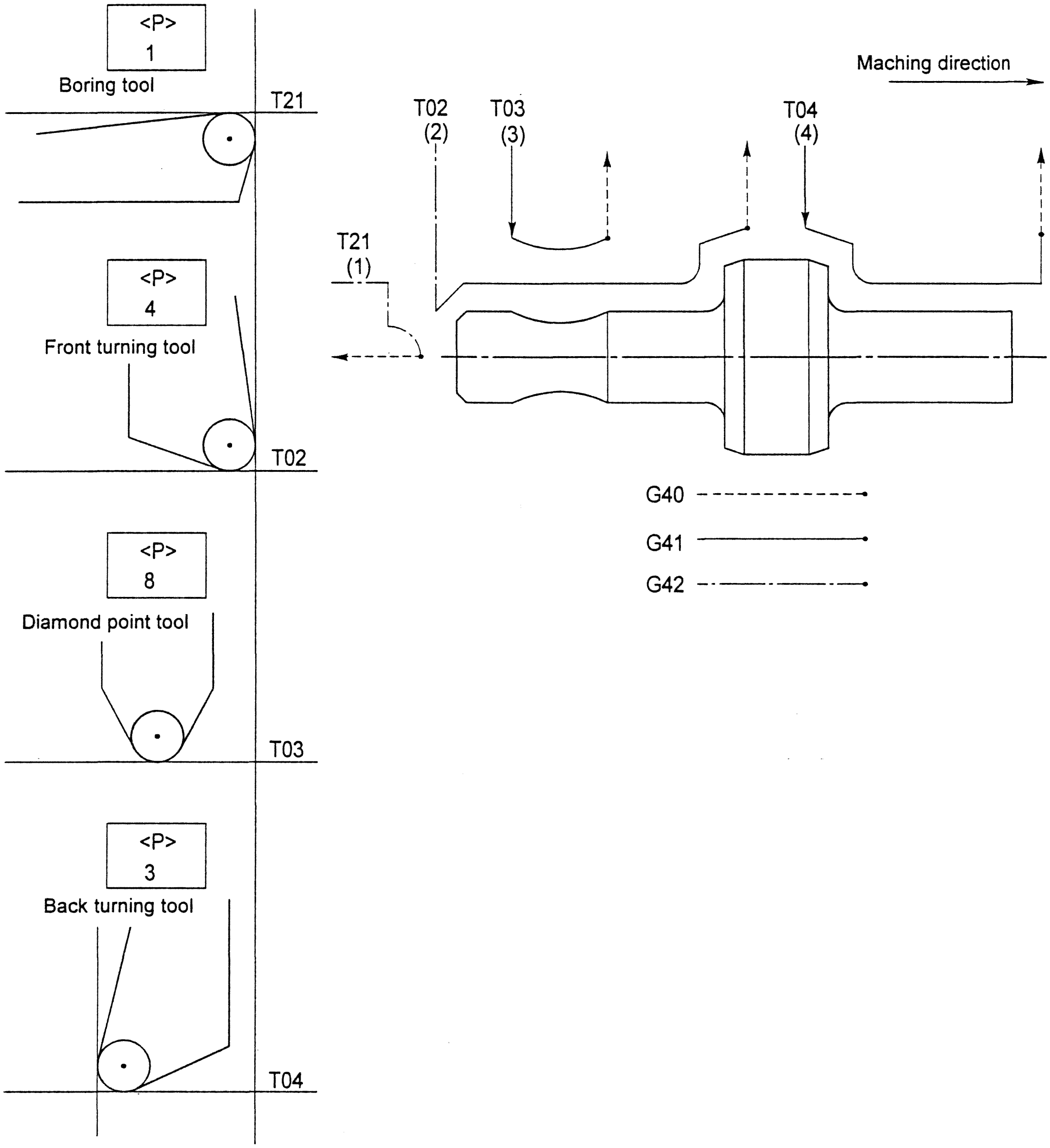
7.8.1 Tools Subject to Tool Nose Radius Compensation and Virtual Tool Nose Numbers

These diagrams illustrate some commonly used tools with the associated virtual tool nose numbers.



7.8.2 Basic Pattern of Tool Nose Radius Compensation G Code

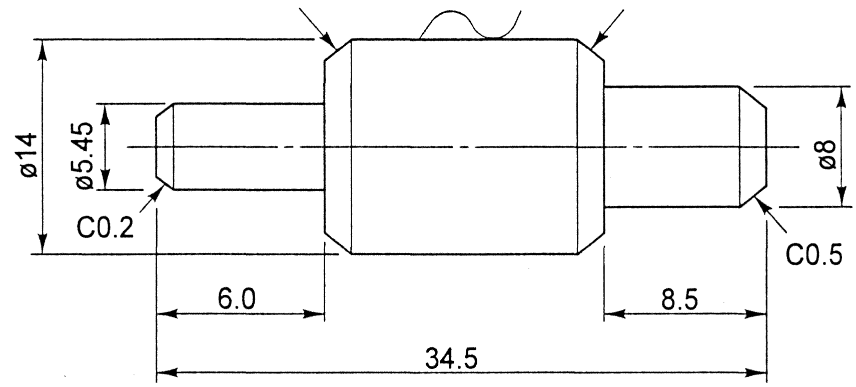
The conception of the tool nose radius compensation.



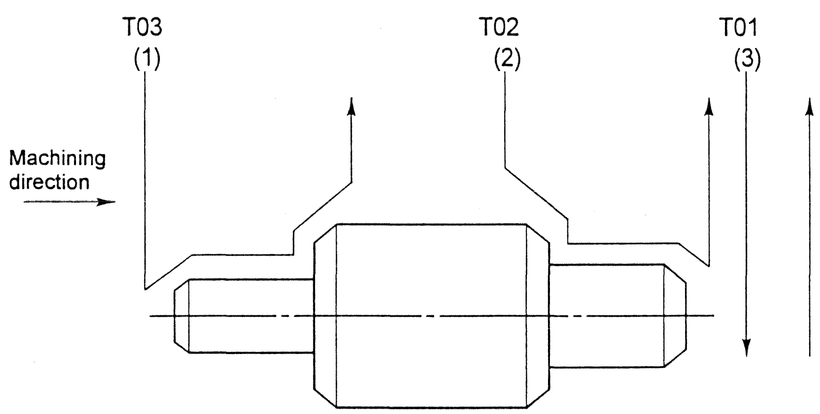
Notes

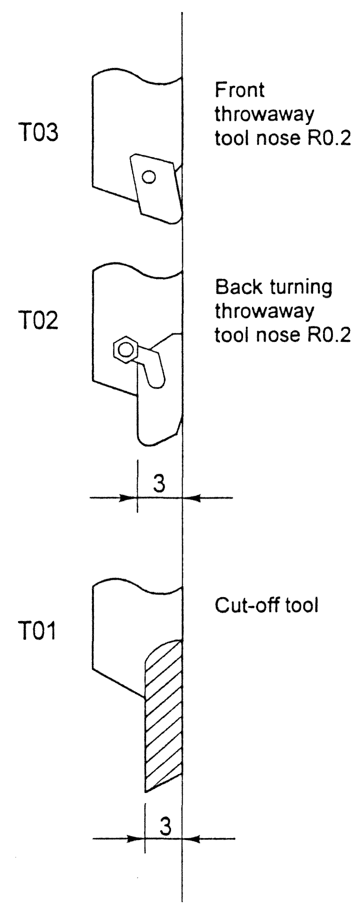
- When positioning in rapid feed or canceling the tool nose radius compensation, pay attention to the interference with the material. The tool nose must be kept apart from the material by more than nose radius.
- If some improper G commands are specified during tool nose radius compensation execution, an alarm occurs.
  *For the contents of the alarm, see the Instruction's manual of the NC manufacturer.

Machining drawing



Machining layout





Program example

$1	$2	
O0300		
G50 Z0		
M06		
G99 G00 X15.0 Z-0.5 S1=2000 M03		
G821	**G821**	.... Axis control group 1 single machining command ON
N103 T0300		
G00 X15.0 Z-0.5 T01		
$\boxed{\text{G42}}$ G01 X4.05 F0.2		.... Tool nose radius compensation right mode ON
$\boxed{\text{G41}}$ X5.45 Z0.2 F0.03		.... Tool nose radius compensation left mode ON
Z6.0 F0.04		
G04 U0.3		
X10.0 F0.2		
X15.0 Z8.5 F0.03		
$\boxed{\text{G40}}$ G00 X16.0 T00		.... Tool nose radius compensation OFF
N202 T0200		
G50 W-3.0		
$\boxed{\text{G41}}$ G00 X15.0 Z23.5 T02		.... Tool nose radius compensation left mode ON
G01 X10.0 Z26.0 F0.03		
X8.0 F0.015		
Z34.0 F0.03		
X6.8 Z34.6 F0.02		
X15.0 F0.2		
$\boxed{\text{G40}}$ G00 X16.0 T00		.... Tool nose radius compensation OFF
G50 W3.0		
N301 T0100		
G50 W-3.0		
G00 X15.0 Z34.5 T03		
G01 X9.0 F0.2		
X-3.0 F0.05		
G50 W3.0		
M05		
M07		
G00 Z0 T00		
G820	**G820**	.... Single machining command OFF
M56		
G999	**G999**	
N999	**N999**	
M02	**M02**	
M99	**M99**	
%	**%**	

Note

- This program does not contain a product separation program.

7.9 Spindle Synchronization Control (G814, G114.1, G113) (Optional)

The G114 and G113 commands are used to control the speed and phase of one spindle in synchronization with the rotation of the other spindle.

These commands are effective to synchronize the two spindles in speed, for example, when chucking a workpiece from the main spindle to the back spindle or when machining a workpiece being held on both of two two spindles. They are also effective when chucking a profile material from one spindle to the other by synchronization them in phase.

Command format

G814 R☐ or

G114.1 H1 D-2 R☐

- R: Synchronization spindle phase-shift amount command (0 to 359.999deg)
- D-2: Synchronization spindle selection
- H1: Reference spindle selection

G113 Cancels spindle synchronization.

Explanation

The G814 command controls the reference spindle as S1 and another spindle to be synchronized as S2, synchronizing the two spindles.

The G114.1 command specifies the reference spindle and the spindle to be synchronization to synchronization the two specified spindles.

The G113 command cancels synchronization between the two spindles rotating synchronization according to the spindle synchronization command.

Notes

- The spindle rotation under spindle synchronization control stops when the Emergency Stop button is pressed. The spindle synchronization control mode is canceled at that time.
- The rotation clamp during spindle synchronization depends on the less value of clamp of synchronization reference spindle or synchronization spindle.
- The synchronization reference spindle and synchronization spindle cannot be indexed in the spindle synchronization control mode. Cancel the spindle synchronization control mode first before indexing.
- The spindle speed must be less than 2,500 min^{-1} when the workpiece is chucked in the back spindle by spindle synchronization control (Range of the stabilized spindle speed).
- The R argument specifies the amount of synchronized spindle phase shift. The synchronized spindle rotates in synchronization with the reference spindle, with the phase (angle) shifted by the R-specified shift amount.
  When the G814 command is specified without the R code, the value obtained by measurement of profile material phase adjustment (G899) in MDI mode is stored and used as the shift amount.
  When the G114.1 command has no R code, no phase shift is performed.

- Spindle Synchronization Control in Relative Position

This section explains the case that the reference spindle and synchronization spindle are under the spindle synchronization control mode in the relative position and chuck the same workpiece during spindle rotation.

Program example

	$1		$2
!L1	Queuing	**!L1**	Queuing
M3 S1 = 1000	Spindle forward rotation	**M24 S2 = 1000**	Back spindle reverse rotation
!L2	Queuing	**!L2**	Queuing
M97	Main spindle speed change detection OFF		
M87	Back spindle speed change detection OFF		
G95 S150	Constant surface speed control		
G50 S2500	Maximum rotation clamp command		
G814 R□	Spindle synchronization control		
T0100	Cut-off tool select		
G0 X□ Z□	Cut-off positioning		
T01			
G811	Z1-Z2 axis superimpose ON	**G811**	Z1-Z2 axis superimpose ON
		G00 Z-1.0	
		G98 G01 Z□ F1000	
M88	Back spindle torque limit	**G4 U0.5**	
		M77	Waiting for complete spindle synchronization
		M15	Back spindle chuck close
!L3	Queuing	**!L3**	Queuing
G1 X-1.0 F0.03			
G810	Z1-Z2 axis superimpose OFF	**G810**	Z1-Z2 axis superimpose OFF
X-3.0 F0.05	Cut-off machining		
G97	Constant surface speed control cancel		
M96	Main spindle speed fluctuation detection ON		
M86	Back spindle speed fluctuation detection ON		
M90	Back spindle torque limit cancel		
!L4	Queuing	**!L4**	Queuing
G113	Spindle synchronization control cancel		
M5	Spindle stop	**M25**	Back spindle stop

Notes

- This program does not contain a product separation program.
- Specify an M77 command one block before the back spindle chuck is closed.

- The block in ⬚ is constant surface speed control
- M89 limits the torque of the back spindle to 50%.
  The block in ☐ limits the torque of the back spindle to 25%.
  In principle, the back spindle torque limit is not required.
  Use the torque limit if a over-torque problem arises, for example, if products are scratched or alarms occur.
- No tool can be selected during constant surface speed control. (For more details, see "7.7 Constant Surface Speed Control Command".)

- Profile Material Phase Adjustment (G899)

This command is used for the NC unit to store the phase shift between the main spindle and guide bushing and the phase shift between the main spindle and back spindle during profile material machining.

Be sure to specify the command when a profile material chuck and guide bushing are mounted.

Procedure

1. Pass the material (profile) through the spindle and guide bushing.
2. Execute G899 under "All Block" is depressed in MDI($1) for automatic chuck close operation and phase adjustment.

In a material replacing program, specify M26 (Guide bushing phase adjustment) before inserting a material into the guide bushing. Specify G814 (main spindle synchronous control) before chucking the material with the main spindle and back spindle. Specifying the two codes adjusts the stored phase shifts.

Command format

$1

G899 R1 K1

- The G899 R1 command causes the NC unit to store the phase shift between the spindle and back spindle.
  When the command is executed, the back spindle also rotates.
- When G899 R1 K1 is specified, the machine does not close the back chuck.

Note

- Specify G899 to adjust only the phase between the main spindle and guide bushing.
- Specify G899 R1 K1 to adjust both of the phase between the main spindle and guide bushing and the phase between the main spindle and back spindle.

- R (phase shift) value adjustment procedure (with no G code macro)

(1) Open the front and back chucks with no material present.

(2) Use [1 block] in MDI mode to run the following program:

M03 S1=1000 M24 S2=1000

G4 U1.0

G114.1 H1 D-2 R☐ (G814R☐)

G4 U1.0

M77

S1=0 (Stop with R shift value kept)

G113

(3) Insert a profile material, check the R (phase shift) value, then remove the material.

(4) Press the spindle stop and back spindle stop buttons.

(5) Enter the R value and repeat steps (2) to (5).

Notes

- Be sure to adjust the R value so that the material and chuck are in phase with each other. Failure to adjust the R value prevents the material from being fit into the chuck, resulting in trouble.
- After replacing or adjusting the guide bush unit driven belt, adjust the R value.
- A higher spindle speed during chucking results in much dispersion in synchronization.

  When chucking a workpiece, keep the spindle speed at about 2500 min^{-1} or less.

7.10 Synchronization Tap Machining (G88, G80) (Optional)

This function requires no tap sleeve and enables the tap machining with high tap depth precision. This function is synchronization with the tool spindle and X1 axis for tap machining (at the cutting and feed command for the X1 axis) in a tap cycle.

Command format

G88 X□ R□ F□ D□ S□ ,R1 Tap cycle

G80 Tap cycle cancel

Note that the R□ code moves the tool from the positioning point in the material direction in rapid feed according to the specified R value.

Program example

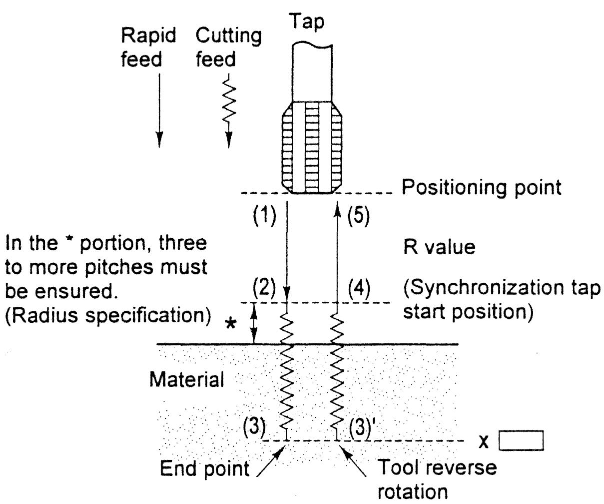
G88 X□ R2.0 F0.7 D3 S500 ,R1 ....Synchronization tap cycle

- ,R1 — Synchronization tap
- S500 — Spindle speed
- D3 — D-3 when the secondary process spindle is specified as a left screw (tap)
- F0.7 — Pitch
- R2.0 — R value (Specify a radius value that is a distance from the positioning point to the position where the synchronization tap is started.) Synchronization starts from the positioning point with no specification.
- X□ — Positioning (Tap machining end point)

G80 ....Tap cycle cancel

If ",R1" is not specified, the feed is executed in the same state of G98 specified immediately before.

In the block in which only "G88 ,R1" is specified, the mode is selected. However, no tap machining is performed in this case.



(1) Tap spindle positioning point (Diameter command)

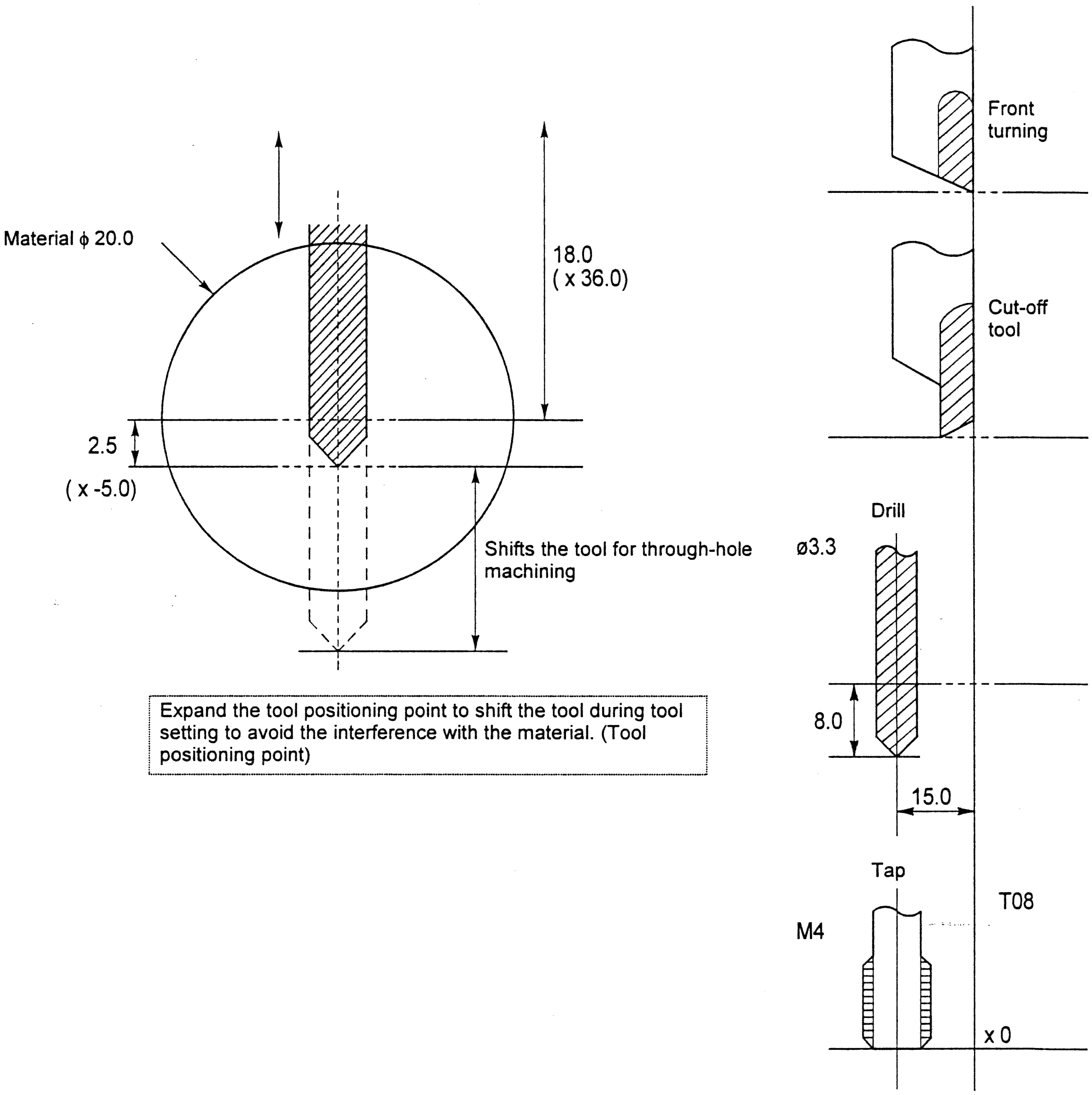
(2) Positioning by R value (synchronization tap start position) in rapid feed. (Radius specification) Three or more pitches must be ensured from position (2).

(3)(3)' Tap machining end point (Tap end point + Tool reverse rotation)

(4) Return to the point positioned by R value (position where the synchronization tap starts).

(5) Return to the tap spindle positioning point in rapid feed.

Machining



Program example

M28 S0

S3 = 1000 M58 G98

N0718 T0800

G00 X26.2 Z25.0 T18

Clearance between material and tap

Pitch x 3

As specifying in the diameter

X26.2 = ϕ 20.0 + {(1 + 2.1) × 2} = 26.2

G88 X $\boxed{0.0}$ **R** $\boxed{1.0}$ **F** $\boxed{0.7}$ **D3 S** $\boxed{500}$ **,R1**

G80

G00 X $\boxed{26.2}$

M60

M03 S1 = $\boxed{3000}$ **G99**

7.11 Interference Check Function

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 usually when the power supply is turned on.

The interference check function does not work if you either set the "Interfere Check Off" setting switch to ON or including 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 meticulous care. You should enable the interference check function in all the processes to which the function is available.

Command format

M118 Disable interference check.

M119 Enable interference check.

1. Parts checked for interference <Type I>
   - Guide bushing
   - Gang tool post ....Data varies depending on the "front machining holder name" setting in machining data.
   - Long workpiece device .....Data varies depending on the machine structure settings for "long workpiece device", "stock gripper device", and "supports (long and short) for stock gripper device".
   - Workpiece separator ....Data varies depending on the "workpiece separator" machine structure setting.

2. Parts checked for interference <Type VII>
   - Guide bushing
   - Gang tool post ....Data varies depending on the "front machining holder name" setting in machining data.
   - Opposite tool post including the back spindle......Data varies depending on the "back spindle" and "back spindle chuck position" settings in machining data and the tool number set between T3000 and T3300.
   - Back tool post including the product separator ....Data varies depending on the "back spindle" setting in machining data.

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 wrong holder or settings have been selected.
- If you run the machine with this function disabled by the program, check all the relevant parts in advance and use meticulous care.
- Even when the back spindle is being used in the workpiece receiver box collection mode, the interference check applies to the workpiece receiver box.

7.12 Back Face Tool Spindle (Optional)

The back tool spindle is a device that can rotate T32 and T33 among the three tools on the back tool post. The device can serve for eccentric drilling with a drill and end mill. The device can accept a standard drill sleeve (19.05 mm diameter) using a sleeve adapter.

7.12.1 Front/Back Face Tool Spindle Rotation and Stop (M80, M81, and M82)

These codes are used for rotating (forward and reverse) and stopping of the front/back face tool spindle in a secondary process.

The forward rotation in this case means the counterclockwise rotation toward the front/back tool spindle.

The front/back tool spindle stop can be manually specified on the panel.

Command format

M80 Starts the front/back face tool spindle forward rotation.

M81 Starts the front/back face tool spindle reverse rotation.

M82 Stops the front/back face tool spindle rotation

The spindle speed changes when only an "S4 = ☐" is specified after the M80 and M81 codes.

Maximum spindle speed is 5000 min^{-1}.

Note

To use the front/back tool spindle detached, use the sleeve adapter or cap to block the hole not to let oil in.

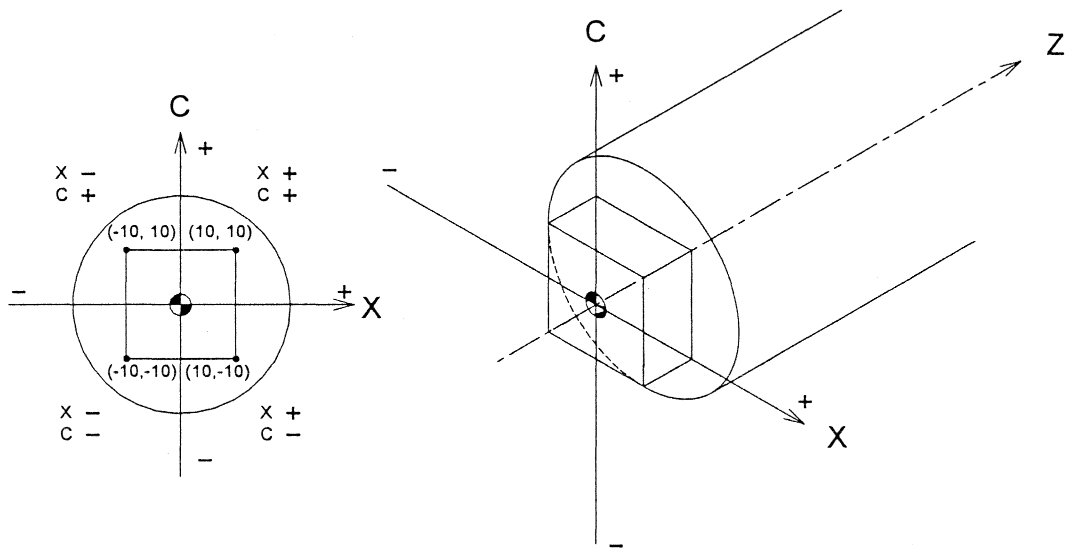
7.13 Milling Interpolation — Optional

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.

7.13.1 Milling coordinates

Milling interpolation is performed between when G12.1 (milling interpolation ON) is executed and when G13.1 (milling interpolation cancel) is executed. Specify the coordinates of X, (Z,) and C axes with values calculated for the radius.

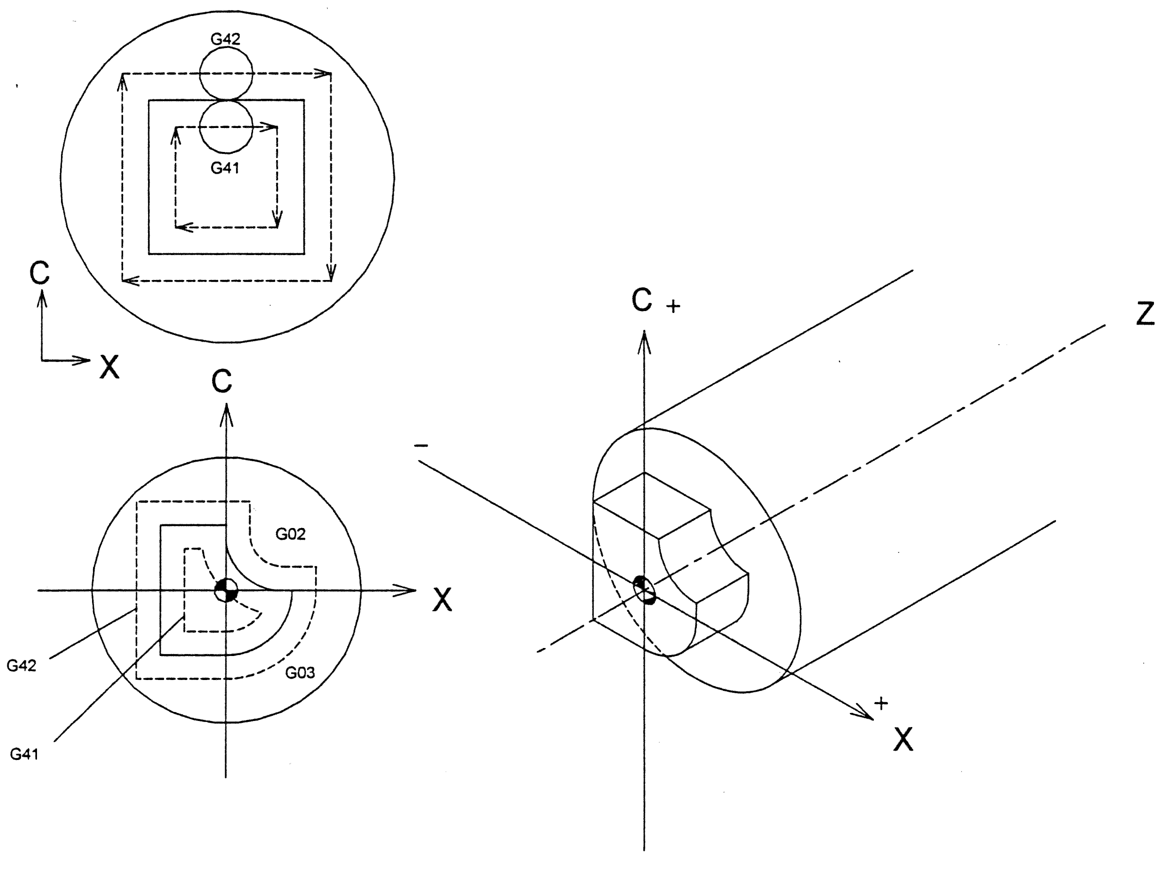
The plus and minus signs of X and C are as shown in the figure below.



7.13.2 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)



7.3.13 Programming of a milling process

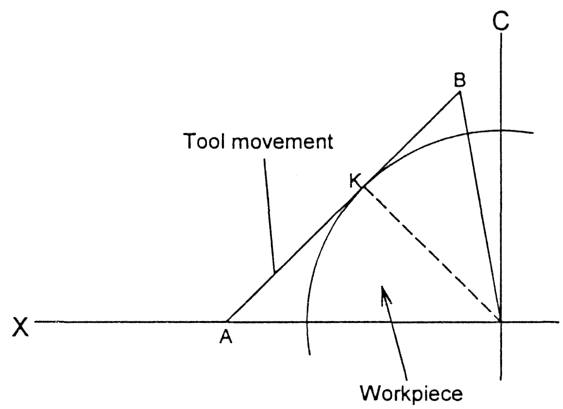
Consider the following points in programming a milling process:

When starting a milling process, move the X axis away from the workpiece to obtain an enough approach distance. The X axis makes an approach to the workpiece, and then moves to the specified position. Therefore, if the approach distance is not enough as shown in the following figure, the tool and workpiece interfere with each other.

During the positioning of the X axis from point A to point B in the following figure, the X axis makes an approach to the workpiece in the range from point A to point K, and moves away from the workpiece in the range from point K to point B.

The milling start position (point A) for avoiding interference with the workpiece must be obtained by calculation.

Point A = Ps position

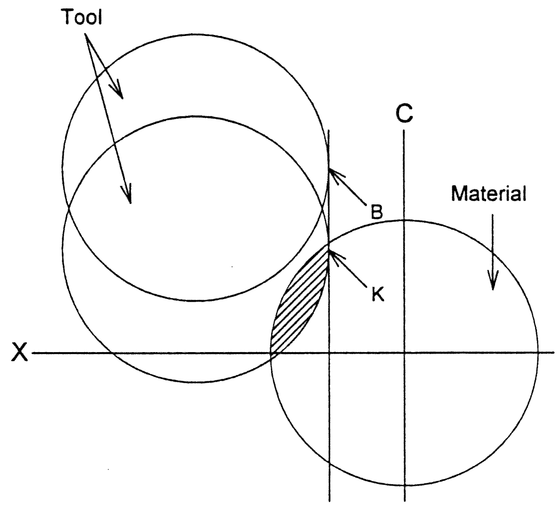


Determine the milling start position in consideration of the diameter of the rotary tool to be used and the diameter of the workpiece to be machined. If the start position is not suitable, the tool and workpiece may interfere with each other.

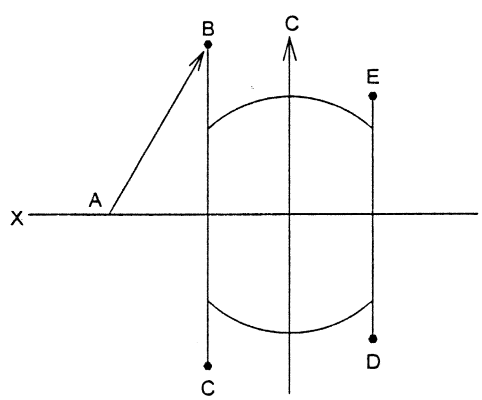
If the machining start position is point K for making the D cut as shown in the following figure, the tool and material interfere with each other in the shaded area when the X axis makes an approach to the workpiece.

To prevent the interference, obtain the machining start position (point B) by calculation.

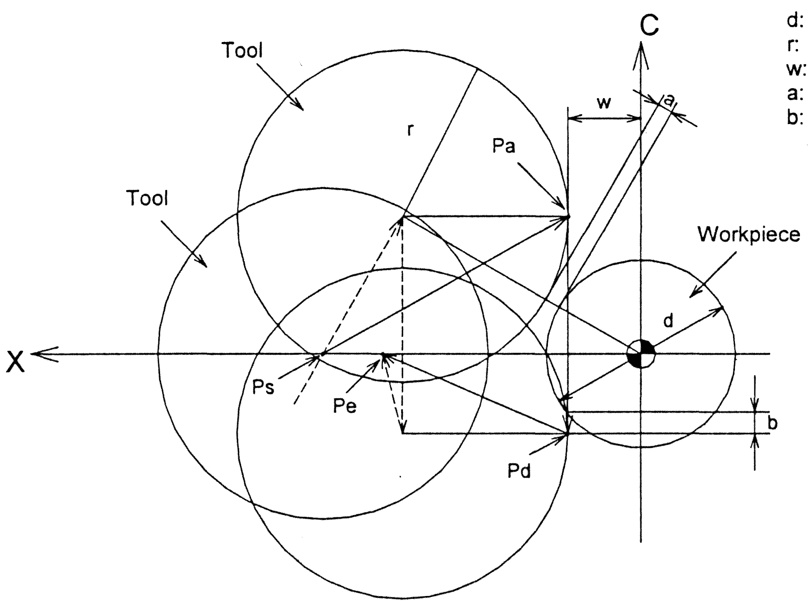
Point B = Pa position



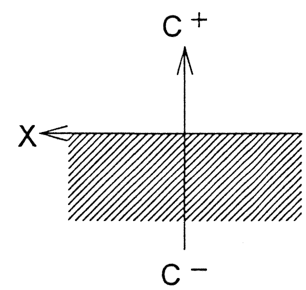
When performing machining both sides of a workpiece (e.g., 2-face width machining), cancel milling interpolation at completion of machining one side, rotate the C axis 180 degrees, then start machining the other side. If both sides are machined continuously without milling interpolation cancel, the tool and workpiece may interfere with each other.

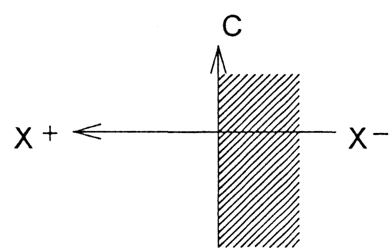


7.13.4 Calculation of milling coordinates



d: Workpiece diameter (mm)
r: Tool diameter (mm)
w: Machining width (mm)
a: Leeway for start position (mm)
b: Leeway for end position (mm)





Example: When d = 20 mm, r = 30 mm, w = 12 mm, a = 2 mm, and b = 2 mm:

Ps = (42.0, 0) Pa = (12.0, 0)

Pd = (12.0, – 8.63) Pe = (42.51, 0)

Obtain machining positions suitable for the milling process from the formulas shown in the following table:

Machining position	X axis coordinate	C axis coordinate
Milling start positions Ps = (Xs, Cs)	$Xs > (d/2 + r + a)^2 / (w + r)$	$Cs = 0$
Machining start position Pa = (Xa, Ca)	$Xa = w$	$Ca > \sqrt{(d/2 + r + a)^2 - (w + r)^2}$
Machining end position Pd = (Xd, Cd)	$Xd = w$	$Cd > \sqrt{(d/2)^2 - w^2} + b$
Milling end position Pe = (Xe, Ce)	$Xe > (w + r) + (Cd^2 / (w + r))$	$Ce = 0$

Notes

- The results of the above formulas are free from the plus and minus signs. When changing the results to coordinates, assign signs to them in consideration of the tool position.
- The coordinate Xs of Ps is moved before milling interpolation ON is specified. Therefore, be sure to change the coordinate to a value calculated for the diameter when specifying it.

  The other values of Xa, Xd, and Xe are calculated for the radius. The calculated values can be used as they are.

7.13.5 Program format

This section shows a program sample for performing milling interpolation with a front rotary tool (option).

Previous process

:	
:	
M05 G98 M80 S4=	Spindle stop, feed per minute, and turret tool spindle rotation
M18 C0	Sequential operation of spindle C axis ON (optional)
G00 H180.0	Note that the machining position is 180o different between when a gang tool is used for the machining and when a turret tool is used for the machining.
G50 C0	C axis coordinate system setting
G821	Line 1 single machining mode ON

Milling process

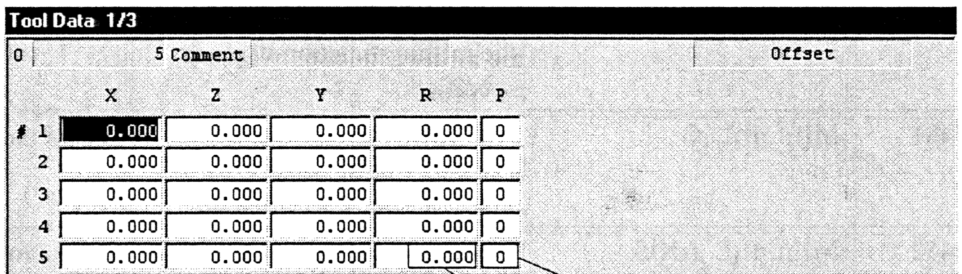
N□ T□	Milling tool selection
G50□ W□	Coordinate shift
G00 X□ Z□ T□	To the milling start position (Ps) (The Xs value of the milling start position Ps is calculated for the radius. Change Xs to a value calculated for the diameter when specifying it. The other values of Xa, Xd, and Xe are calculated for the radius. They can be specified as they are.)
G12.1	Milling interpolation ON
G17	End-face machining selection
G41□ G00 X□ C□	Tool diameter compensation ON, and to the machining start position (Pa) (G42 may be executed instead.)
:	
:	
G01 X□ C□	Milling
:	
:	
G40 G00 X□ C□	Tool diameter compensation OFF, and to the milling end position (Pe)
G13.1	Milling interpolation cancel
G50 W□	Coordinate system shift cancel
M20 M82	C axis release, and tool spindle stop
G820	Line 1 single machining mode OFF
:	

7.13.6 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 <Programming Guide> issued by Mitsubishi Electric Co., Ltd. The following shows the setting of nose R compensation:

Example: When T05 is used:


Tool Data 1/3

0 | 5 Comment | Offset

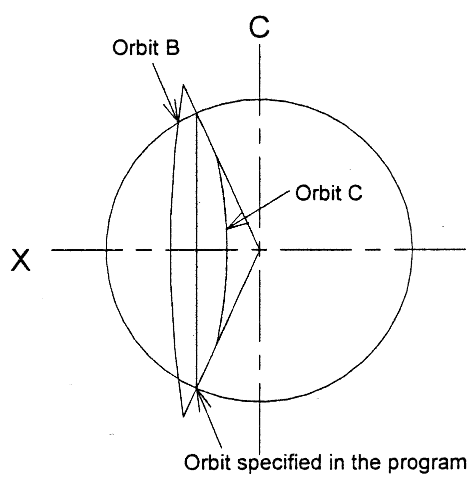
	X	Z	Y	R	P
# 1	0.000	0.000	0.000	0.000	0
2	0.000	0.000	0.000	0.000	0
3	0.000	0.000	0.000	0.000	0
4	0.000	0.000	0.000	0.000	0
5	0.000	0.000	0.000	0.000	0

0 must be set for the virtual tool nose number.

The radius of the tool to be used is set.
Examples
With a side cutter having a diameter of 60.0: R30.000
With an end mill having a diameter of 3.0: R1.500

If the coordinate system shift in the diametrical direction of the tool is different from the actual shift, the workpiece is cut in an invalid shape.

For example, if linear interpolation is performed for the end face as shown in the following figure, the workpiece has the externally round shape like orbit B when the coordinate system shift is too large. Conversely, the workpiece has the internally round shape like orbit C when the coordinate system shift is too small. If linear interpolation causes a round shape, the coordinate system shift is invalid. Correct the coordinate system shift.



To compensate the dimensions of a finished workpiece, change the tool data R.

When a dimension of an actually machined workpiece is greater than the specified value, decrease the value specified in the tool data by the error. When a dimension of an actually machined workpiece is smaller than the specified value, increase the value specified in the tool data by the error.

7.13.7 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.

Error no.	Message displayed on the screen	Description	Procedure
P480	NO. MILL SPEC	The milling command (G12.1 or G13.1) was executed when the milling function was not specified.	Check the specification.
P481	MILL ILL. G	A G code, which must not be executed during milling interpolation, was executed.	Delete the invalid G code.
P482	MILL ILL. AXIS	The command was executed for the rotary axis during milling interpolation.	Delete the command for the rotary axis.
P484	INCOMPLETE RETURN AXIS (MILL)	When the milling interpolation process starts, there is an axis which has not returned to the reference point.	Return the axis to the reference point in manual or automatic mode.
P485	INVALID MODAL (MILL)	• When the milling interpolation process starts, tool diameter compensation, constant surface speed control, or the boring cycle is in progress. • A T command was executed during milling interpolation.	• Execute the cancel command (G40, G97, or G80) before the G12.1 command. • Execute the T command before the G12.1 command.

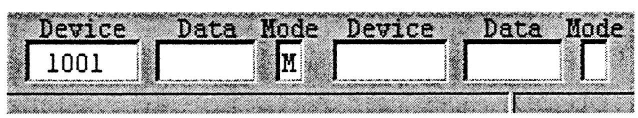
7.13.8 Changing the value of a parameter for milling interpolation

The setup value of "1125 mill_ax" for milling interpolation is different when a gang tool is used and when a front rotary tool is used. Change the setup value on the Basic Specification Parameter <$1> screen depending on the tool as shown below.

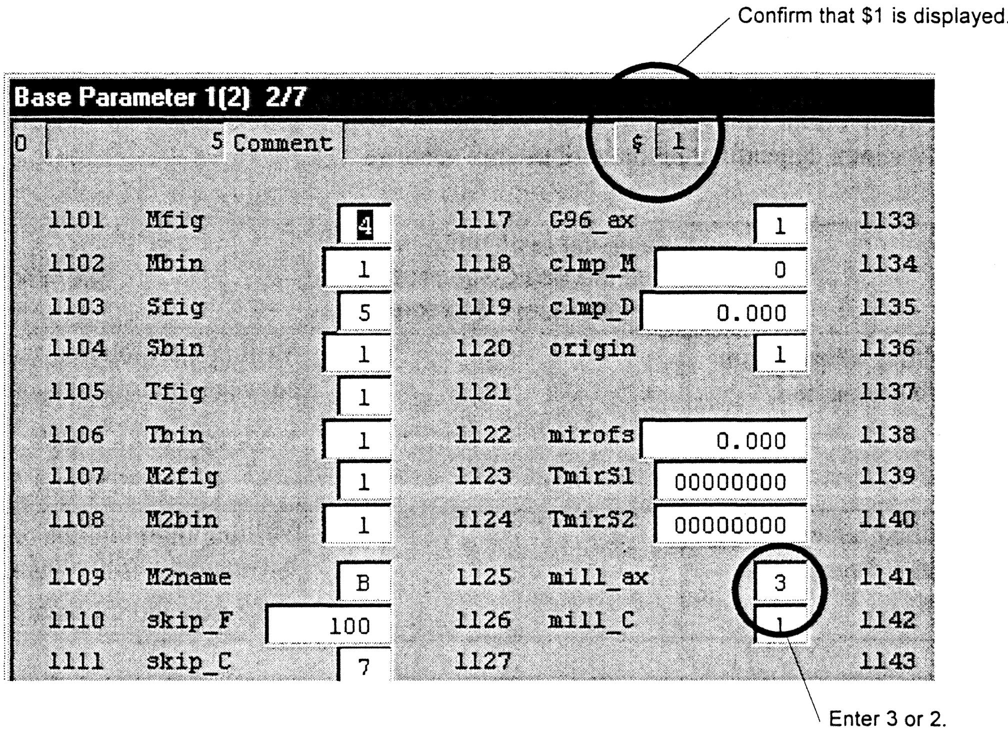
	Setup value of "1125 mill_ax" on the Basic Specification Parameter <$1> screen	Remarks
Milling interpolation with a gang tool	3	Milling interpolation can be performed between the following axes: • X1 axis and C1 axis • X2 axis and C2 axis
Milling interpolation with a front rotary tool	2	Milling interpolation can be performed between the following axes: • X2 axis and C1 axis

Procedure

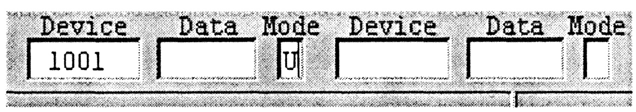
- Press the [MAINT.] button on the operation panel.
- Press the menu key [I/F Diag] to display the [I/F Diagnosis] screen.
- Enter "(1001) ( ) (M)," then press the [INPUT] button.



- Press the [PRM.] button on the operation panel.
- Press the menu key [Basic]. If [Basic] is not displayed, press [Menu SEL] to display [Basic].
- Press the [PAGE ↓] button on the operation panel once. If the "Basic Specification Parameter 1 (2)" <$1> screen does not appear, keep pressing the menu key [$-SELECT] until the Basic Specification Parameter 1 (2)" <$1> screen appears.
- Move the cursor to the item "1125," enter "3" or "2," then press the [INPUT] button.

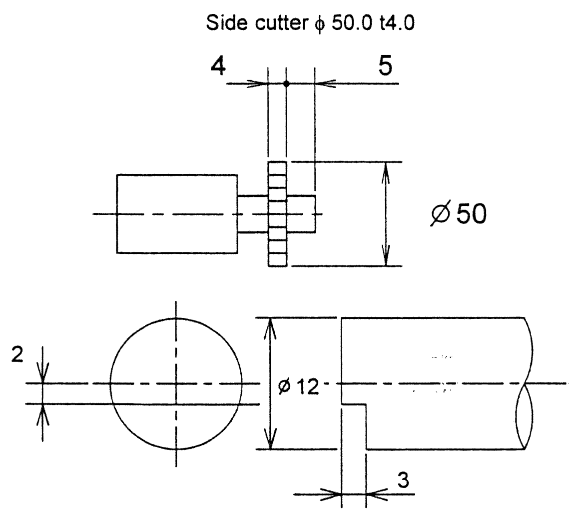


- Press the [MAINT.] button on the operation panel.
- Press the menu key [I/F Diag] to display the [I/F Diagnosis] screen.
- Enter "(1001) ( ) (M)," then press the [INPUT] button.



7.13.9 Example of using the milling interpolation function 1 (D cut)

This section shows a program sample for performing milling interpolation with a front rotary tool (option).



Ps: $Xs > (12/2 + 25 + 2)^2 / (25 + 2) = 40.33$

The following is obtained since Xs is specified with a value calculated for the diameter.

$40.33 \times 2 \approx 81.0$

Pa: $Xa = 2$ (from the above figure)

$Ca > \sqrt{(12/2 + 25 + 2)^2 - (2 + 25)^2} = 18.97 \approx 19.0$

Pd: $Xd = 2$

$Cd > \sqrt{(12/2\,)^2 - 2^2} = 7.65 \approx 7.7$

Pe: $Xe > (2 + 25) + \{7.72 / (2 + 25)\} \approx 29.19 \approx 30.0$

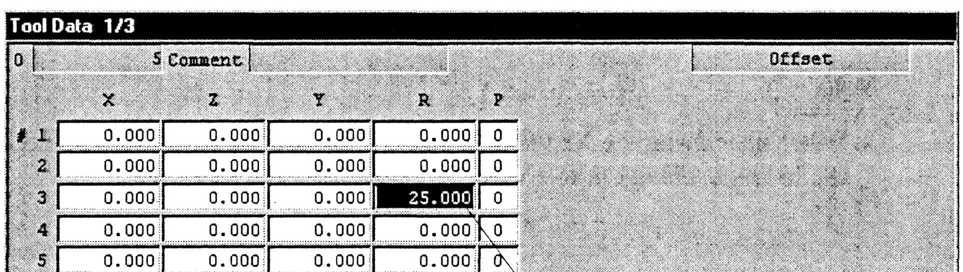
Set about 34.0 with leeway added.

Note

When specifying the Xs value of the milling start position Ps obtained from the formula, change it to a value calculated for the diameter.

Program sample

$1	$2
:	
G821 ....Line 1 single machining mode ON	**G821**
:	
M05 G98 M80 S4=1260	
M18 C0	
G50 C0	
N322	
T2200	
G50 W–5.0	
G00 X81.0 Z3.0 T03 ....Ps milling start position	
G12.1	
G17	
G41 G00 X2.0 C19.0 ....Pa machining start position	
G01 C–7.7 F80 ....Pd machining end position	
G40 G00 X34.0 C0.......................Pe milling end position	
G13.1	
G50 W5.0	
M20 M82	
M03 S1=3000 G99	
:	
G820..Line 1 single machining mode OFF	**G820**
:	


Tool Data 1/3

0 | 5 Comment | Offset

	X	Z	Y	R	P
# 1	0.000	0.000	0.000	0.000	0
2	0.000	0.000	0.000	0.000	0
3	0.000	0.000	0.000	25.000	0
4	0.000	0.000	0.000	0.000	0
5	0.000	0.000	0.000	0.000	0

Tool diameter compensation setting

7.13.10 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°. This section shows a program sample for performing milling interpolation with a front rotary tool (option).

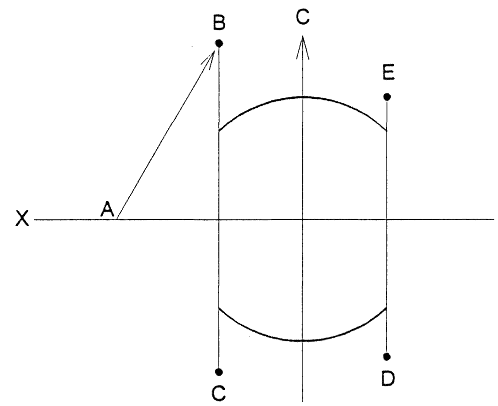
Program sample

$1			$2
Previous process			
G821	Line 1 single machining mode ON		**G821**
:			
:			
M05 G98 M80 S4=□	Spindle stop, feed per minute, and turret tool spindle rotation		
M18 C0	Sequential operation of spindle C axis ON (optional)		
G50 C0	C axis coordinate system setting		
Milling 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 G00 X□ C□	To point B (Pa position), tool diameter compensation ON.	Milling	
G01 C–□ F□	To point C (Pd position)		
G40 G00 X□ C□	To the milling end position (Pe position) Tool diameter compensation OFF		
G13.1	Milling interpolation cancel		
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 G00 X□ C□	To point D (Pa position), tool diameter compensation ON	Milling	
G01 C–□	To point E (Pd position)		
G40 G00 X□ C□	Tool diameter compensation OFF To the milling end position (Pe)		
G13.1	Milling interpolation cancel		
G00 X□ Z□			
G50 W□	Coordinate system shift cancel		
M20 M82	C axis release and tool spindle stop		
G820	Line 1 single machining mode OFF		**G820**

Note

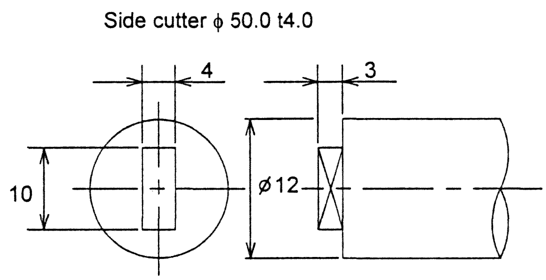
- Machining from point D to Point E is performed after the workpiece is rotated by 180°. Be careful of the coordinates of X and C.

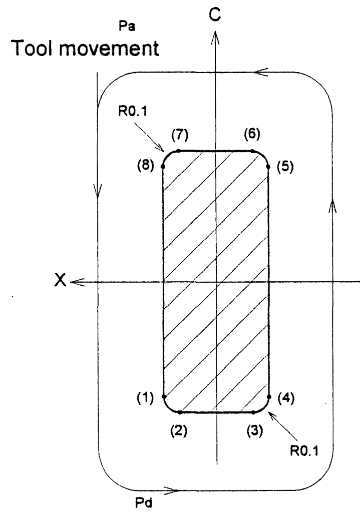
  B = D and C = E



7.13.11 Example of using the milling interpolation function 3

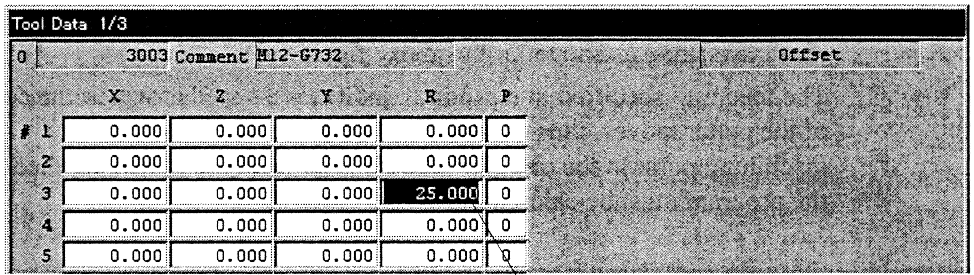
This section shows a program sample for performing milling interpolation with a front rotary tool (option).





Program sample

$1	$2
:	
G821 ....Line 1 single machining mode ON	**G821**
M05 G98 M80 S4=1260	
M18 C0	
G50 C0	
N0322	
T2200	
G50 W–5.0	
G00 X81.0 Z3.0 T03 ....Ps milling start position	
G12.1	
G17	
G41 G00 X2.0 C23.0 ....Pa machining start position	
G01 X2.0 C–4.9 F80....................(1)	
G02 X1.9 C–5.0 R0.1 F750........(2)	
G01 X–1.9 F80 ....(3)	
G02 X–2.0 C–4.9 R0.1 F410......(4)	
G01 C4.9 F80 ....(5)	
G02 X–1.9 C5.0 R0.1 F750........(6)	
G01 X1.9 F80................................(7)	
G02 X2.0 C4.9 R0.1 F410 ....(8)	
G01 X2.0 C–23.0 F500 ....Pd machining end position	
G40 G00 X34.0 C–23.0...............Pe milling end position	
G13.1	
G50 W5.0	
M20 M82	
M03 S1=3000 G99	
G820..Line 1 single machining mode OFF	**G820**
:	



Tool Data 1/3

0 3003 Comment M12-G732 Offset

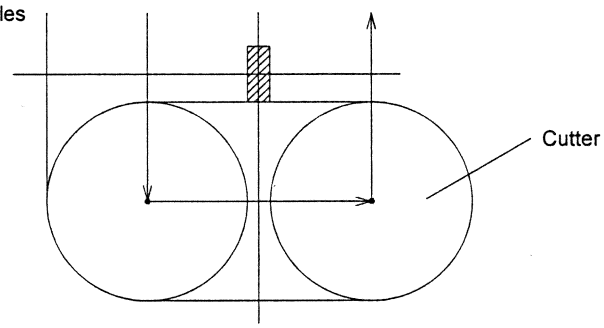
	X	Z	Y	R	P
# 1	0.000	0.000	0.000	0.000	0
2	0.000	0.000	0.000	0.000	0
3	0.000	0.000	0.000	25.000	0
4	0.000	0.000	0.000	0.000	0
5	0.000	0.000	0.000	0.000	0

Tool diameter compensation setting

Note

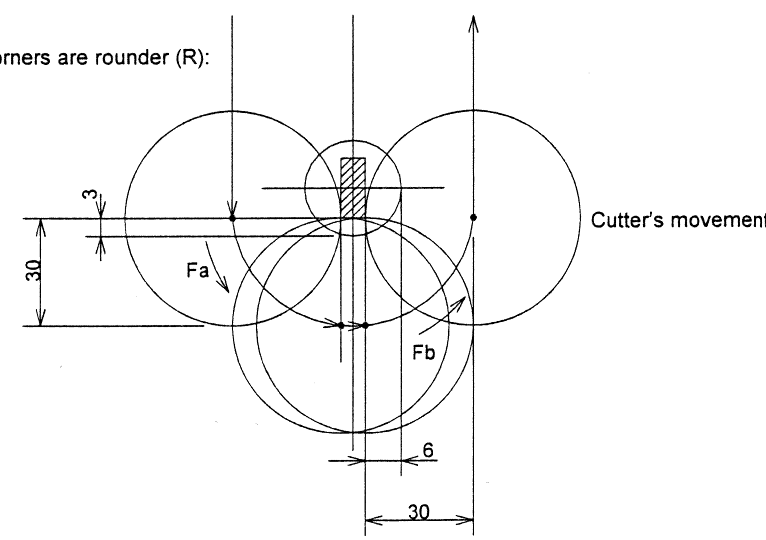
- If the R setting value of tool diameter compensation is too large, the cutter makes unnecessary movement when cutting each corner of the workpiece. As a result, machining takes more time than it should. To minimize the machining time and have corners well finished, specify such a value for R that the 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 rounder (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 F2 = 80 mm/min, obtain the feed rates Fa and Fb to be specified in programs.

$$Fa = 80 \times \frac{(3 + 25)}{3} = 750$$

$$Fb = 80 \times \frac{(6 + 25)}{6} = 750$$

- Obtaining the feed rate for the machining diameter:

d_1	Cutter diameter	r_1	Cutter radius
d_3	Machining diameter	r_3	Machining radius
ℓ	Eccentricity		
F_1	Feed rate specified in the program		
F_2	Feed rate at which you want to machine workpieces		

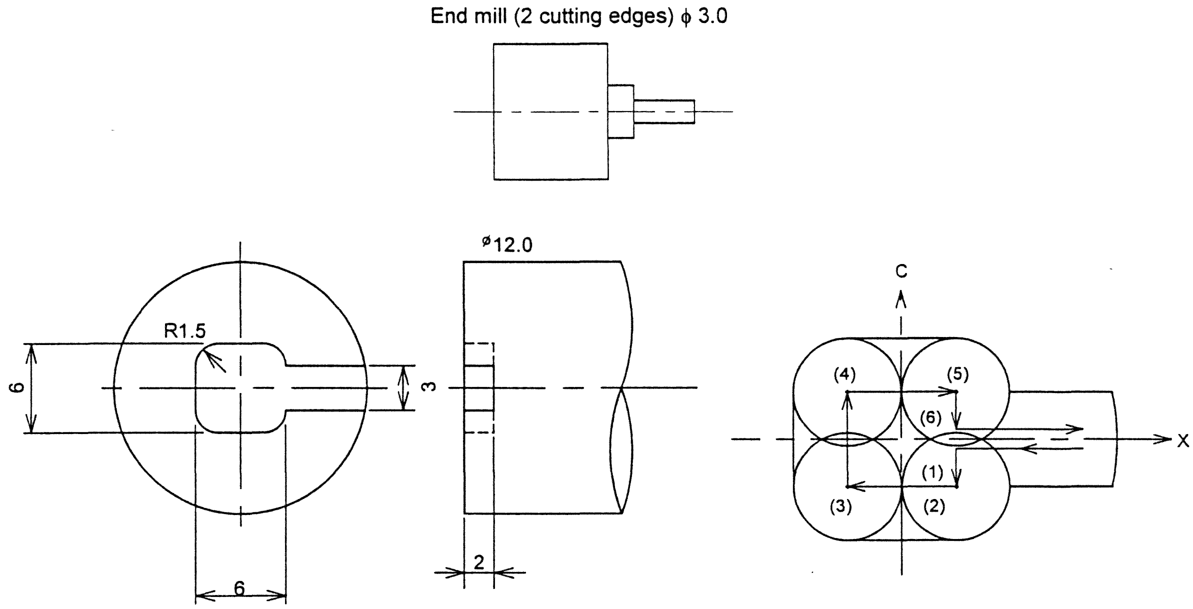
$$F_1 = F_2 \times \frac{(d_3 + d_1)}{d_3} = F_2 \times \frac{(r_3 + r_1)}{r_3}$$

- 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.

7.13.12 Example of using the milling interpolation function 4

This section shows a program sample for performing milling interpolation with a front rotary tool (option).



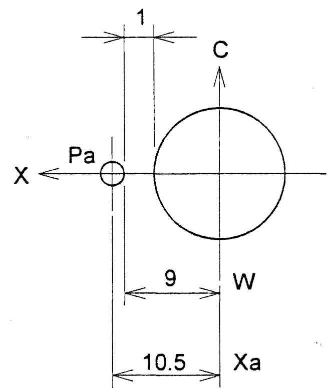
Program sample

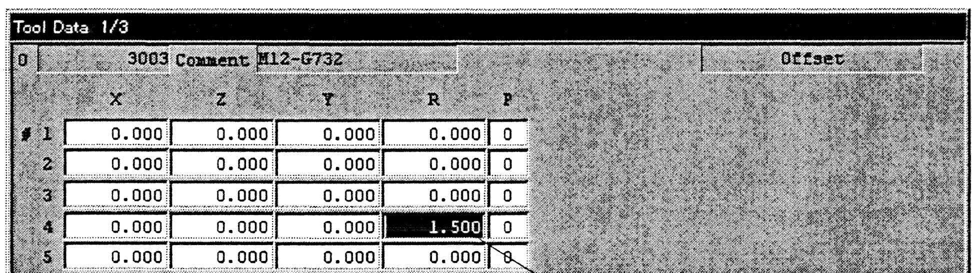
$1		$2
:		
G821	......Line 1 single machining mode ON	**G821**
M05 G98 M80 S4=2700		
M18 C0		
G50 C0		
N424		
T2400		
G00 X17.0 Z2.0 T04	......Ps milling start position	
G12.1		
G17		
G42 G00 X7.0 C–1.5	......Pa machining start position	
G01 X3.0 F200	......(1)	
C–3.0	......(2)	
X–3.0	......(3)	
C3.0	......(4)	
X3.0	......(5)	
C1.5	......(6) The value, instead of C0, is set in consideration of tool diameter compensation.	
X7.0 F500		
G40 G00 X7.0 C0	......C0 is acceptable because the tool diameter compensation is canceled.	
G13.1		
M20 M82		
M03 S1=3000 G99		
G820	......Line 1 single machining mode OFF	**G820**
:		

Ps: $Xs > (6 + 1.5 + 1)^2 / (9 + 1.5) \approx 7.0$

Pa: Xa (See the following figure.)

Ca is 0 according to the following figure, but -1.5 should be set in consideration of tool diameter compensation.





Tool Data 1/3

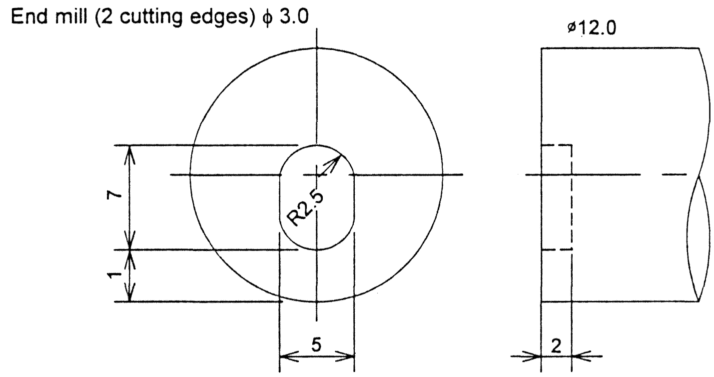
0 | 3003 Comment M12-G732 | Offset

	X	Z	Y	R	P
# 1	0.000	0.000	0.000	0.000	0
2	0.000	0.000	0.000	0.000	0
3	0.000	0.000	0.000	0.000	0
4	0.000	0.000	0.000	1.500	0
5	0.000	0.000	0.000	0.000	0

Tool diameter compensation setting

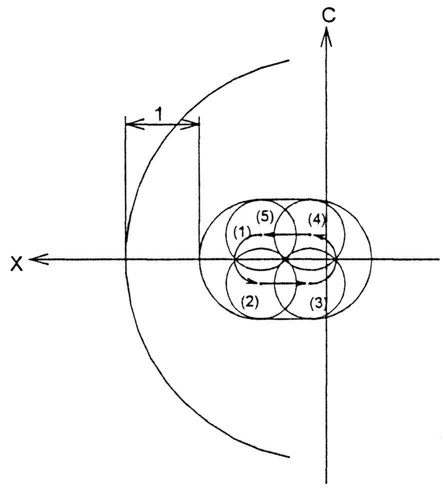
7.13.13 Example of using the milling interpolation function 5

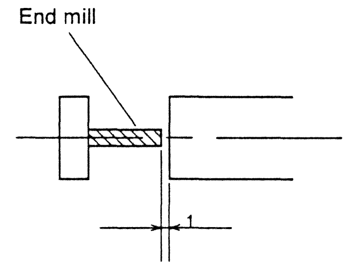
This section shows a program sample for performing milling interpolation with a front rotary tool (option).



Program sample

$1	$2
:	
G821 ....Line 1 single machining mode ON	**G821**
M05 G98 M80 S4=2700	
M18 C0	
G50 C0	
N0224	
T2400	
G00 X16.0 Z–1.0 T02 ....Ps milling start position	
G12.1	
G17	
G42 G00 X2.5 C2.5 ....(1)	
G01 Z2.0 F125..............................(a)	
G02 C–2.5 R2.5 F160 ....(2)	
G01 X–0.5 ....(3)	
G02 C2.5 R2.5..............................(4)	
G01 X1.5.......................................(5)	
Z–1.0 F1000.................................(b)	
G40	
G13.1	
M20 M82	
M03 S1=3000 G99	
:	
G820..Line 1 single machining mode OFF	**G820**
:	





(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:

d_1	Cutter diameter	r_1	Cutter radius
d_3	Machining diameter	r_3	Machining radius

F_1 Feed rate specified in the program
F_2 Feed rate at which you want to machine workpieces

- Obtaining the feed rate for the machining diameter:

$$F_1 = F_2 \times \frac{(d_3 + d_1)}{d_3} = F_2 \times \frac{(r_3 + r_1)}{r_3}$$

Tool Data 1/3

0 3003 Comment M12-G732 Offset

	X	Z	Y	R	P
# 1	0.000	0.000	0.000	0.000	0
2	0.000	0.000	0.000	1.500	0
3	0.000	0.000	0.000	0.000	0
4	0.000	0.000	0.000	0.000	0
5	0.000	0.000	0.000	0.000	0

Tool diameter compensation setting

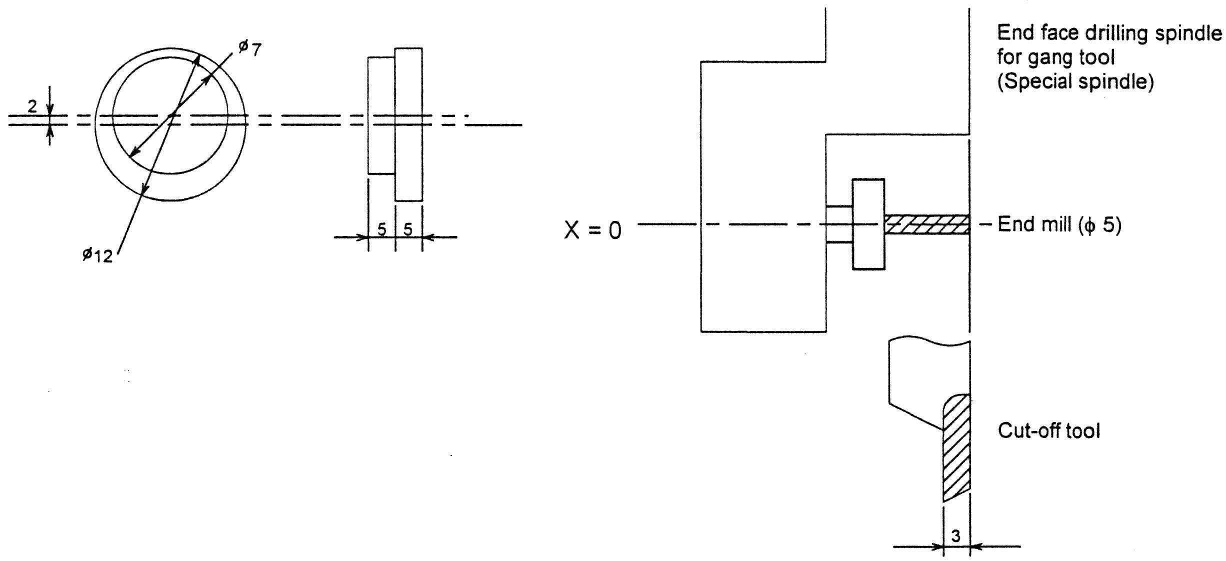
The feed rate for the machining diameter (F_2) is set as 100 mm/min.

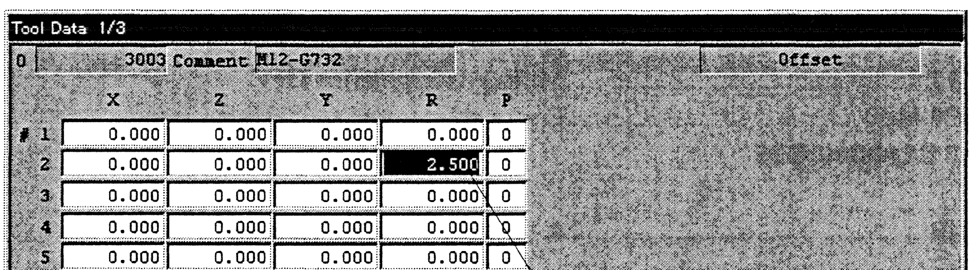
$$F_1 = 100 \times \frac{2.5 + 1.5}{2.5} = 160$$

Note: The machining feed rate (100 mm/min) is a standard value.

7.13.14 Example of using the milling interpolation function 6

This section shows a program sample for performing milling interpolation with a gang tool. A special spindle is necessary in this case.





#	X	Z	Y	R	P
1	0.000	0.000	0.000	0.000	0
2	0.000	0.000	0.000	2.500	0
3	0.000	0.000	0.000	0.000	0
4	0.000	0.000	0.000	0.000	0
5	0.000	0.000	0.000	0.000	0

Enter the radius of the tool.

Ps $Xs > (d/2 + r + a)^2 / (w + r)$

$= (12/2 + 2.5 + 1)^2 / (5.5 + 2.5) = 11.281$

11.281×2 (diameter value specification) $= 22.562 \approx 22.6$

$Cs = 0$ (C coordinate)

Pa $Xa = w = 5.5$

$Ca > \{(d/2 + r + a)^2 - (w + r)^2\}^{1/2}$

$= \{(12/2 + 2.5 + 1)^2 - (5.5 + 2.5)^2\}^{1/2} = 5.12 \approx 5.2$ (C coordinate)

Pd $Xd = w = 5.5$

$Cd > \{(d/2)^2 + w^2\}^{1/2} + b$

$= \{(12/2)^2 - 5.5^2\}^{1/2} + 1 = 3.39 \approx 3.4$ (C coordinate)

Pe $Xe > (w + r) + Cd^2 / (w + r)$

$= (5.5 + 2.5) + 3.4^2 / (5.5 + 2.5) = 9.44 \approx 9.5$

$Ce = 0$ (C coordinate)

Program sample

$1		$2
:		
G0 X26.0 Z–0.5		
M18 C0	Sequential operation with spindle C axis ON (optional)	
G50 C0	Coordinate system setting	
G98 M58 S3=2000		
G17	X-Y plane selection (end face machining)	
:		
N0217 T0800		
G0 X22.6 Z5.0 T02	To the milling start position (Ps), and tool data (tool diameter compensation) call	
G12.1	Milling interpolation ON	
G41 G0 X5.5 C5.2	Right compensation, and to the machining start position (Pa)	
G1 C0 F60 **G2 X5.5 C1 R–3.5 F60** **G1 C–3.4 F300**	Milling interpolation machining, machining end position (Pd)	
G40 G0 X9.5 C0	Compensation cancel, and to the milling end position (Pe)	
G13.1	Milling interpolation OFF	
M20 M60	C axis release and tool spindle stop	
M3 S1=3000 G99		
:		
G18	X-Z plane selection (turning)	
:		

(Blank page)

Product code	C	–	L	5	1	6	2	0	I	III	VII	/	/	/	–	5	7	0

Document code	2	E	1	–	0	7	0	3
	2	E	2	–	0	7	0	2

Chapter 8 Program Example

8.1 Program Creation 8-1

(Blank page)

8.1 Program Creation

Following programs are only examples. During actual machining, the optimum spindle speed and feed must be selected in consideration of various conditions.

Following examples describe when you program by machining patterns (see "3.5.3 Multi-Axis Control Group") (2 axis control group) and by non-machining (single axis control group: Front machining, Front machining with opposite tool post)

Example 1 .... Front machining

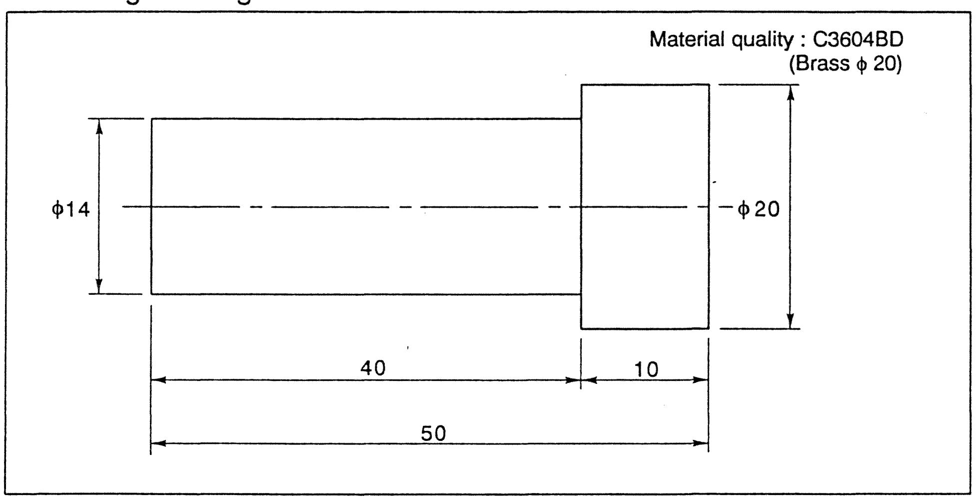
Example 2 .... Front machining with opposite tool post

Example 3 .... Back machining

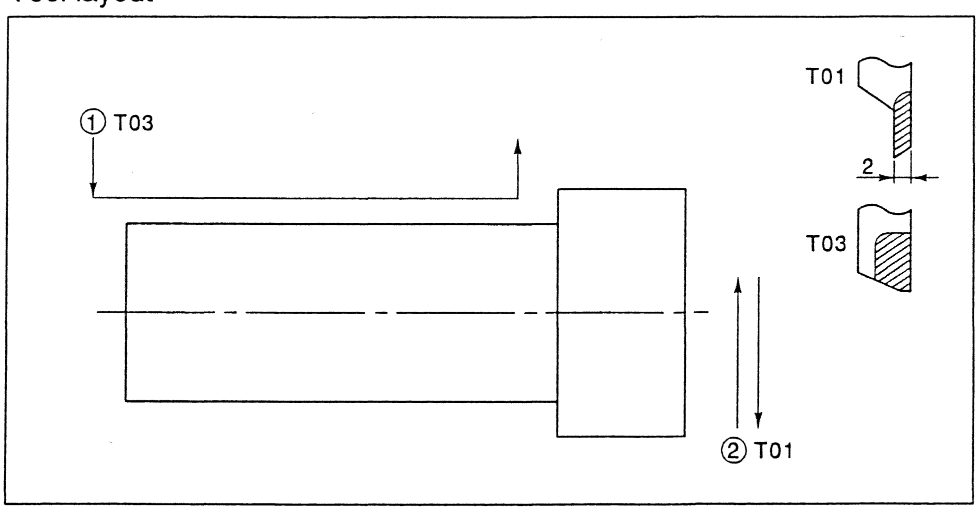
[Example 1] Front machining

This example assumes the workpiece receiver box collection mode.

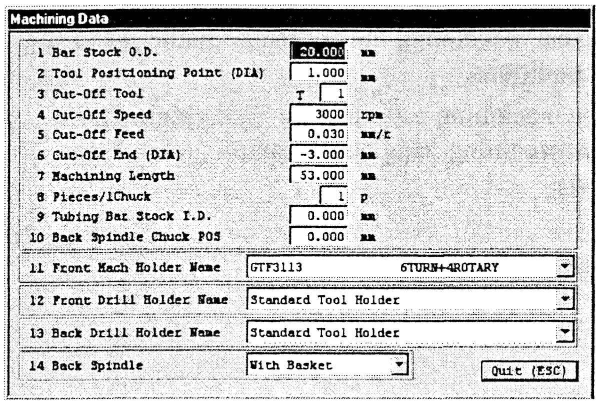
Machining drawing



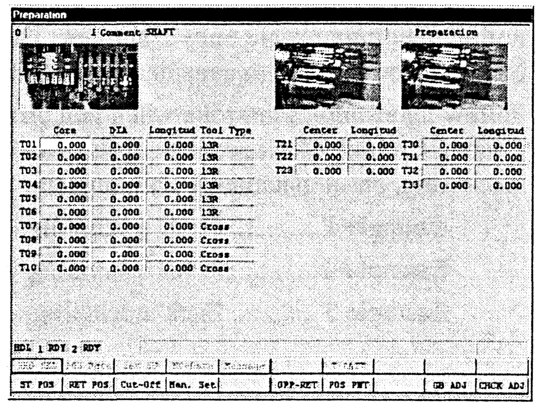
Tool layout



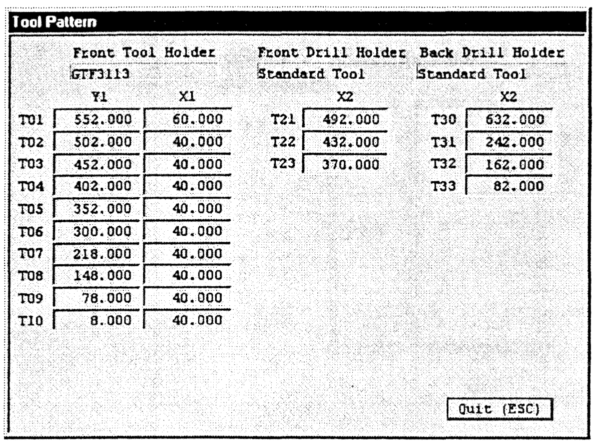
Machining Data



Tool Set



Tool Pattern



Program

[Example 1] (Machining pattern provided, two axis control groups used)

O0001 .... Program number

$1	$2	
		.... Axis control group commands
G50 Z0		.... Coordinate system setting
M06		.... Chuck close
G00 X21.0 Z-0.5		.... Move the material from the tool.
M03 S1 = 2500 G99		.... Main spindle forward rotation (3,600 min^{-1}) Per rotation feed (mm/rev)
N0103T0300		.... Sequence number and T03 tool selection
M34		.... A series of operations for product separation
G00 X14.0 Z-0.5 T01		.... Positioning to ϕ 14 in rapid feed. Compensation No. 1 command
G01 Z40.0 F0.05		.... Cut to 40 mm in longitudinal direction (ϕ 14).
X21.0		.... Cut to the material outer diameter + 0.5 mm (radius value).
N0201 T0100		.... Sequence number and T01 tool selection
G00 X21.0 Z52.0 T02		.... Cut-off positioning compensation No. 2 command in rapid feed
M32		.... Advance to the workpiece separator position of the back spindle.
G99 G01 X-1.0 F0.03		.... Workpiece cut-off
X-3.0 F0.05		
M141		.... Return the back spindle workpiece separator to Z2 axis zero point.
M05		.... Main spindle rotation stop
M07		.... Chuck open
G00 Z0 T00		.... Return to the start point.
M56		.... Product count
G999	G999	.... Last program execution command
G821	G821	.... Single machining command ON
M34		.... A series of operations for product separation
G820	G820	.... Single machining command OFF
N999	N999	.... Last program queuing
M02	M02	.... One cycle stop
M99	M99	.... Return to the top of a program.
%	%	.... Stop code

Program

[Example 1] (No machining pattern, single axis control group used)

O0001

```
$1
G50 Z0
M06
G00 X21.0 Z-0.5
M03 S1 = 2500 G99

N0103 T0300
M34
G00 X14.0 Z-0.5 T01
G01 Z40.0 F0.05
X21.0

N0201 T0100
G00 X21.0 Z52.0 T02
M32
G99 G01 X-1.0 F0.03
X-3.0 F0.03
M141
M05
M07
G00 Z0 T0
M56
G999
M34
N999
M02
M99
%
```

- A parameter setting switch of $2 Cycle Start must be set to OFF before the above program can be run.

■Spindle speed

$$N = \frac{V}{\pi D} \times 1000$$
$$= \frac{160}{3.14 \times 20} \times 1000$$
$$\cong 2546$$
$$\cong 2500\text{min}^{-1}$$

V : 160m/min

D : ϕ 20

π : 3.14

From cutting condition table in Chapter 9

FX = 0.03mm/rev

FZ = 0.05mm/rev

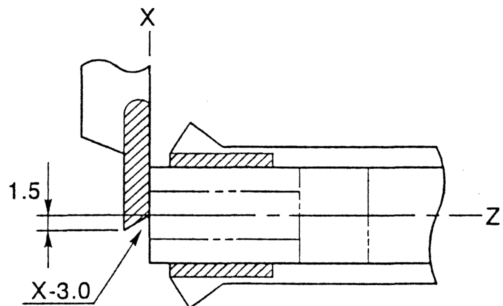
For conversion into per minute feed

F (mm/min) = Spindle speed × F (mm/rev)

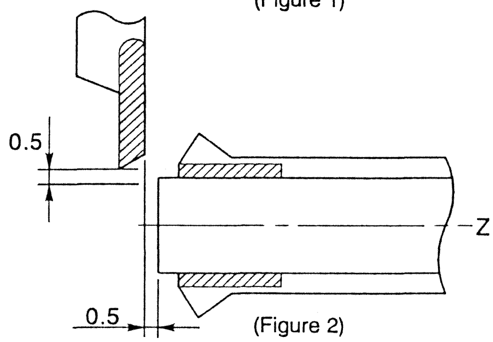
FX = 2500 × 0.03 = F75

FZ = 2500 × 0.05 = F125

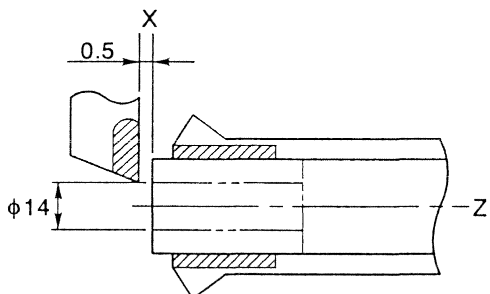
Operation diagram



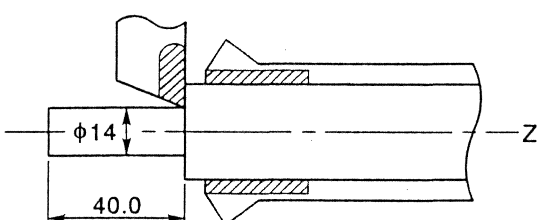
(Figure 1)



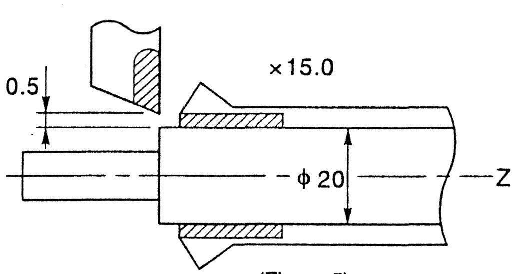
(Figure 2)



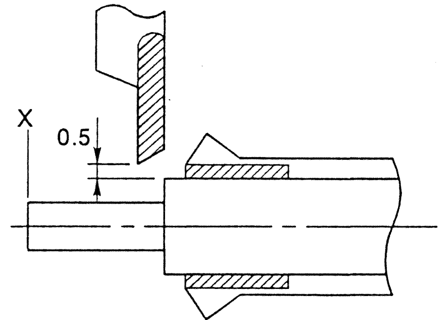
(Figure 3)



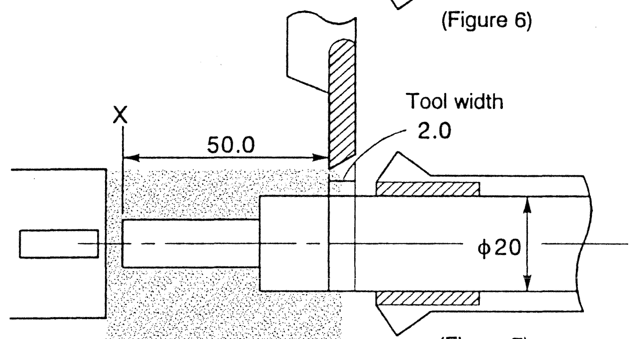
(Figure 4)



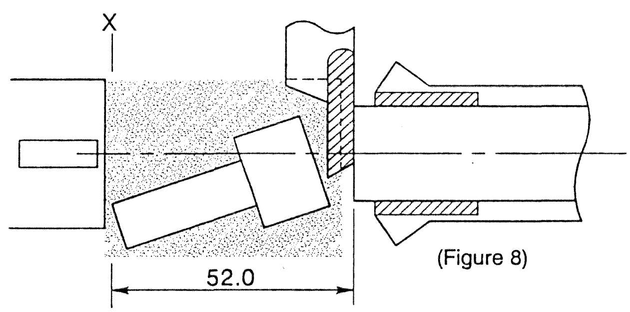
(Figure 5)



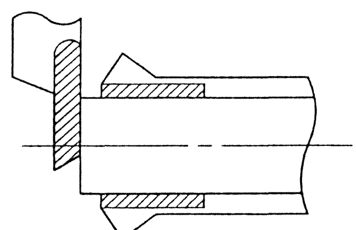
(Figure 6)



(Figure 7)



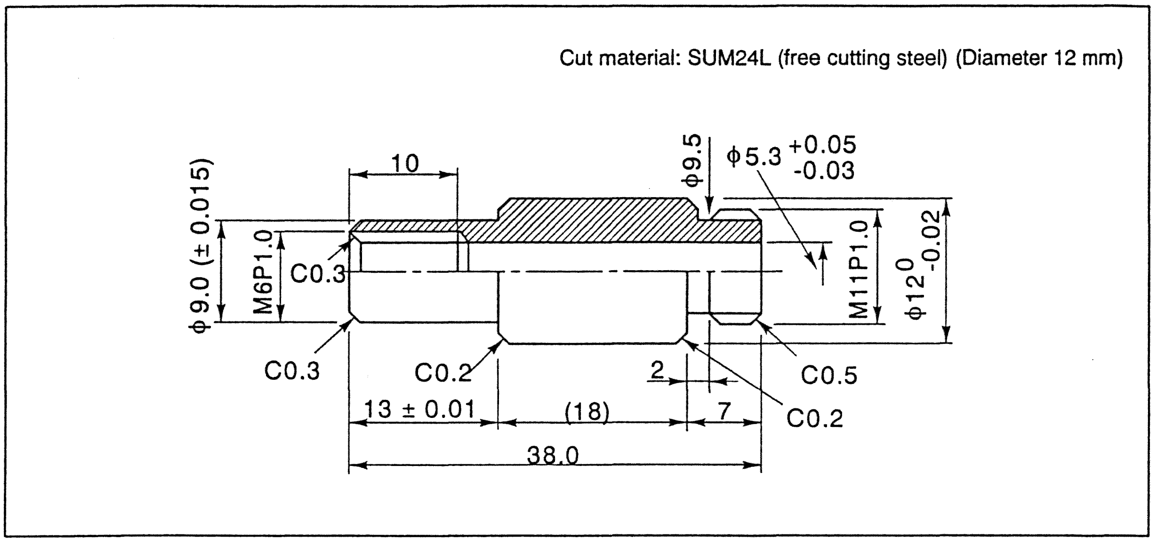
(Figure 8)



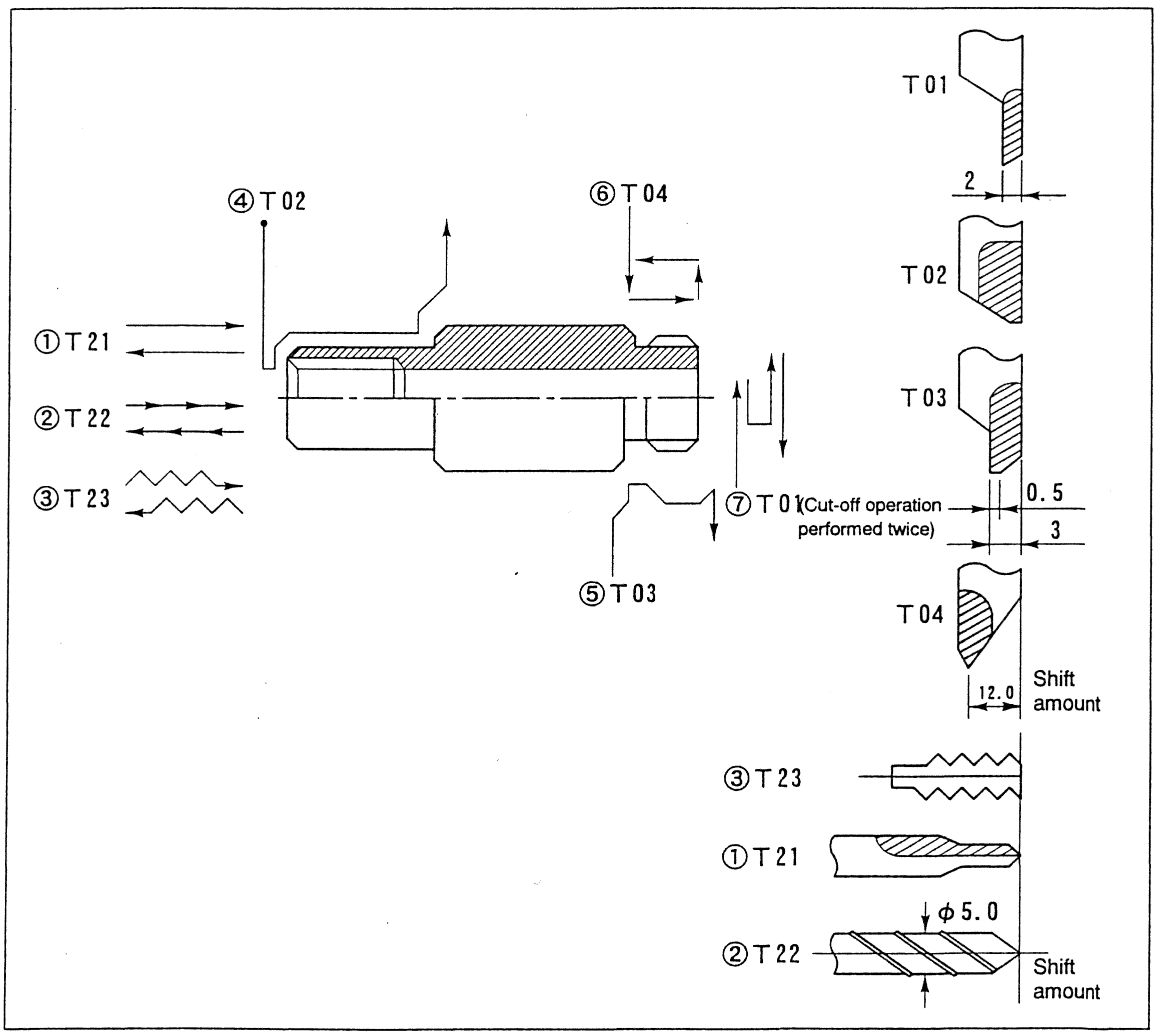
(Figure 9)

[Example 2] Front machining with opposite tool post

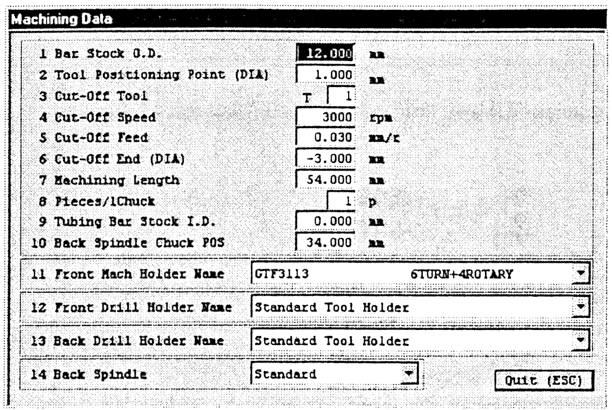
Machining drawing



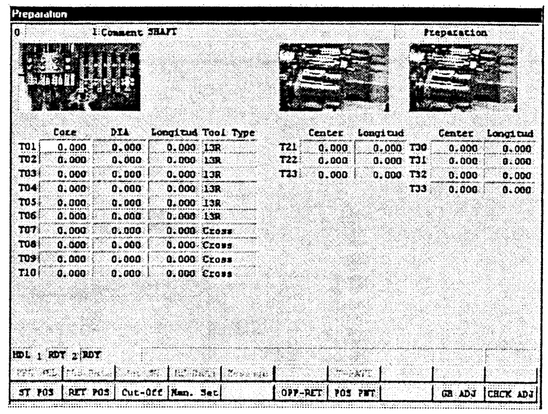
Tool layout



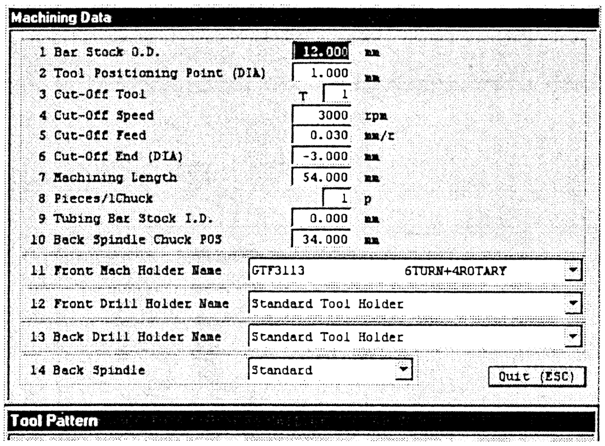
Machining Data



Tool Set



Tool Pattern



Program

[Example 2] (Machining pattern provided, two axis control groups used)

O0002

$1	$2	
G50 Z-0.1		
M06		
G00 X13.0 Z-0.6		
M03 S1 = 1800 G99		
G821	G821	..........Single machining
N121 T2100		
M140		..........Advance the opposite tool post. (Enabled with G821)
G00 Z-0.6		
G01 Z6.0 F0.08 T01		
G00 Z-0.6 T00		
N222 T2200		
G01 Z16.0 F0.1 T02		..........Start 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 T2300		
M97		
S1 = 400		
G04 U1.0		
G32 Z8.0 F0.8 T03		..........Forward in-feed operation for M6 thread cutting
Z-3.0 F1.0 M04 T00		..........Reversing
G04 U2.0		
M141		..........Return the opposite tool post. (Enabled with G821)
G00 Z-0.6		
M03 S1 = 2800		

M96		
G820	G820	.... Single machining
N402 T0200	M34	.... Product separation
G00 X5.0 Z-0.6 T04	T3000	
G01 Z0 F0.06	M16	
X8.4	M24 S2 = 2800	
X9.0 Z0.3		
Z13.0		
X11.6		
X13.0 Z13.7		
N503 T0300		
G00 X13.0 Z33.3 T05		
G01 X11.6 Z34.0 F0.03		
X9.5		
Z35.5 F0.06		
X11.0 Z36.25		
Z40.5		
X10.0 Z41.0 F0.03		
Z42.0 F0.06		
X13.0 F0.2		
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		

G811	G811
	M72
	G00 Z-1.0
	G98 G01 Z12.0 F1000
	G04 U1.0
	M15
	M73
!L1	!L1
G01 X5.0 F0.03	
G810	G810
G00 X13.0 W-0.5	M25
Z42.0	
G01 X-3.0	
M05	
M07	
G00 Z-0.1 T00	
M56	
G999	G999
	M34
	T3000
N999	N999
M02	M02
M99	M99
%	%

Program

[Example 2] (No machining pattern, single axis control group used)

O0002

```
$1
G50 Z-0.1
M06
G00 X13.0 Z-0.6
M03 S1 = 1800 G99

N121 T2100
M140
G00 Z-0.6
G01 Z6.0 F0.08 T01
G00 Z-0.6 T00

N222 T2200
G01 Z16.0 F0.1 T02
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
G00 Z-0.6 T00

N323 T2300
M97
S1 = 400
G04 U1.0
G32 Z8.0 F0.8 T03
Z-3.0 F1.0 M04 T00
G04 U2.0
M141
G00 Z-0.6
M03 S1 = 2800
M96
```

```
N402 T0200
M34
G00 X5.0 Z-0.6 T04
G01 Z0 F0.06
X8.4
X9.0 Z0.3
Z13.0
X11.6
X13.0 Z13.7

N503 T0300
G00 X13.0 Z33.3 T05
G01 X11.6 Z34.0 F0.03
X9.5
Z35.5 F0.06
X11.0 Z36.25
Z40.5
X10.0 Z41.0 F0.03
Z42.0 F0.06
X13.0 F0.2
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

T3000 A1 B-1.0 C12.0 E1000 S2800 K0
```

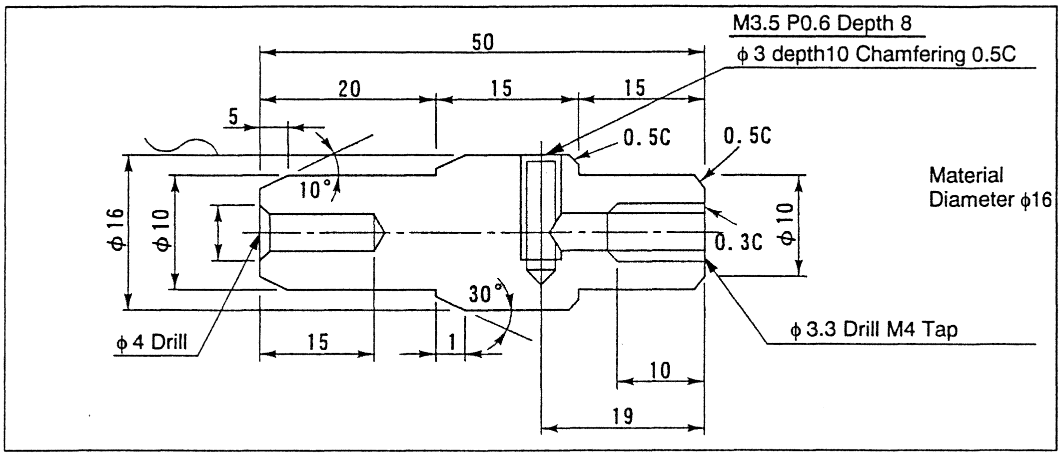
```
G01 X5.0 F0.03
G850
M25
G00 X13.0 W-0.5
Z42.0
G01 X-3.0

M05
M07
G00 Z-0.1 T00
M56
G999
M34
T3000
N999
M02
M99
%
```

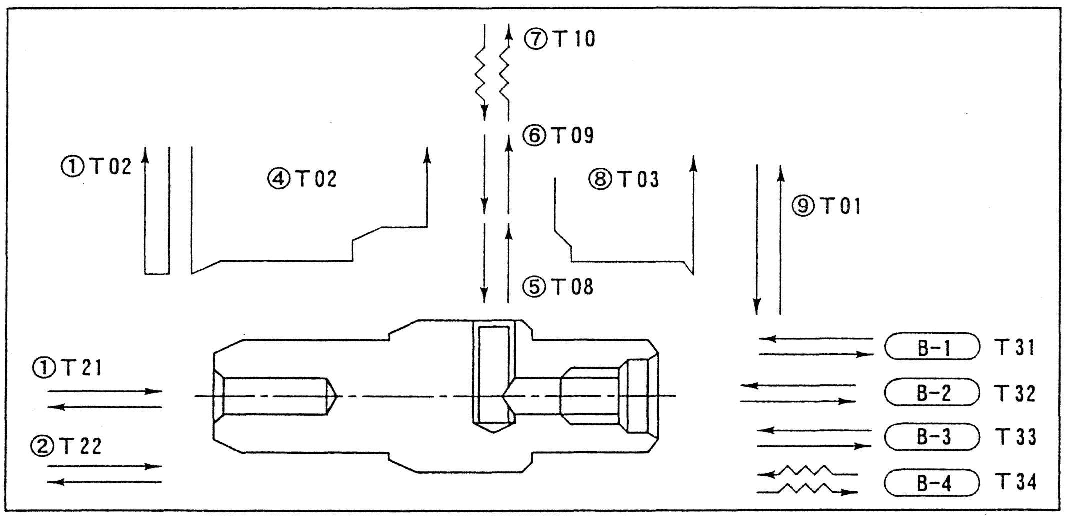
- A parameter setting switch of $2 Cycle Start must be set to OFF before the above program can be run.

[Example 3] Back machining

Machining drawing



Tool layout



(Refer to the figures related to the cutting tools)

Machine condition settings

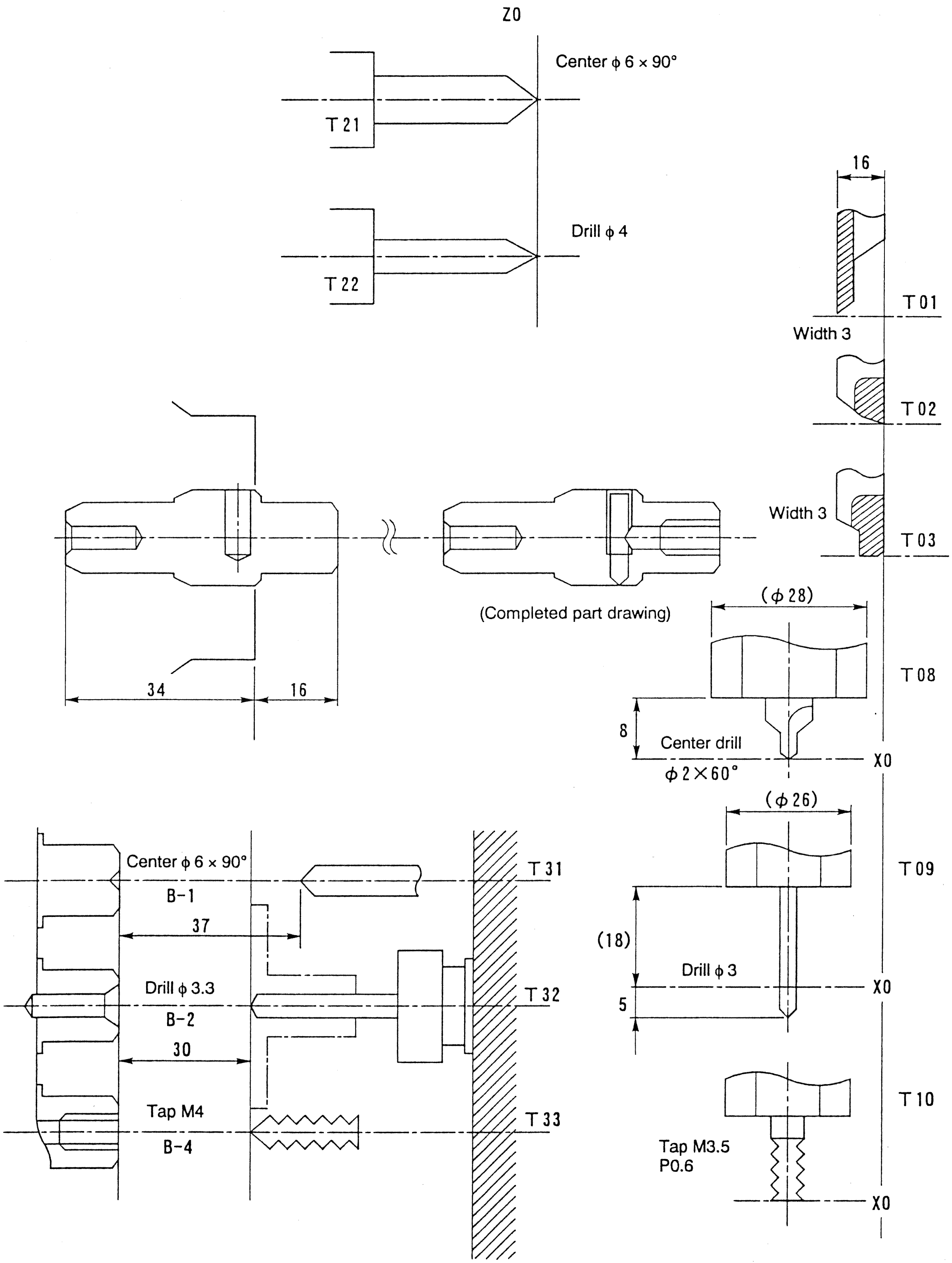
- Machining data settings

Material outer diameter:	16.000mm
Machining length:	70.000mm
Product count per chuck:	1
Tool positioning point:	11.0mm
Back spindle chuck position:	16.0mm

Note

- The tool positioning point is 11.0 mm, shifted 5 mm for setting the T09 drill.

Cutting tool drawings



Program

[Example 3] (Machining pattern provided, two axis control groups used)

O0003

$1	$2	
G50 Z12.9		
G99		
M6		
G0 X17.0 Z-0.5 M3 S1 = 2400		
G821	G821	..........Single machining
N1 T0200		
G0 X17.0 Z0 T02		
G1 X-0.5 F0.1		
Z-0.5		
G0 X17.0 S1 = 1061		
N2 T2100		
M140		
G0 Z-1.0		
G1 Z2.7 F0.08 T21		
G0 Z-1.0 T0 S1 = 1600		
N3 T2200		
G1 Z12.0 F0.06 T22		
G0 Z-1.0		
G4 U0.5		
Z11.5		
G1 Z16.15 F0.06		
G0 Z-1.0		
M141		
G820	G820	..........Front/back simultaneous machining
S1 = 2400	M98 H1	.........."H" calls the sequence number; "H1" calls N1.
	G99 M24 S2 = 1600	
N4 T0200	T3000	
G0 X8.06 Z-0.5 T12		
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		

L16 / L20 Program Example

G98 S3 = 1300 M58		
N5 T0800		
M28 S0		
G50 W-15.0		
G0 X27.0 Z31.0 T08		
X17.0		
G1 X4.8 F50		.........X4.8 = ϕ 16 - 5.6 × 2 = 4.8
G0 X27.0 S3 = 1600		
N6 T0900		
G0 X27.0 Z31.0 T09		
G1 X3.7 F80		.........X3.7 = ϕ 26 - 11.15 × 2 = 3.7
		‖
		ϕ 16 + (shift amount 5mm × 2)
G0 X27.0		
N7 T1000		
G00 X21.6 Z31.0 T10		.........X21.6 = ϕ 16 + {(1 + 1.8) × 2} = 21.6
		‖
		Pitch × 3
G88 X0 R1.0 F0.6 D3 S500 ,R1		
G80		
G0 X22.0		
G50 W15.0		
M60 G99		
M20		
M3 S1 = 2000		
N8 T0300		
G50 W-3.0		
G0 X17.0 Z34.0 T03		
G1 X15.0 Z35.0 F0.015		
X10.0		
Z49.5 F0.03		
X8.8 Z50.1 F0.015		
X17.0 F0.1		
G50 W3.0		
S1 = 1600		
N9 T0100		
G0 X17.0 Z63.0 T01		
G811	G811	
	M72	
	G0 Z-1.0	

	G98 G1 Z34.0 F1000	
	G4 U0.5	
	M15	
	M73	
!L1	!L1	
G1 X-1.0 F0.02		
G810	G810	
X-3.0 F0.05	M25	
M5		
M7		
G0 Z12.9 T0		
M56		
G999	G999	
	M98 H1	
N999	N999	
M2	M2	
M99	M99	
	N1	
	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 face of the workpiece on 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	
	G01 Z13.0 F0.56	
	Z-1.5 F0.7 M24	
	G4 U2.0	
	G0 Z-5.0 M25	
	G43	
	M34	..........A series of operations for product separation
	T3000	
	M99	
%	%	

(Blank page)

Product code	C	–	L	5	1	6	2	0	I	III	VII				–	5	7	0

Document code	2	E	1	–	0	8	0	2
	2	E	2	–	0	8	0	1

Chapter 9 Cutting Conditions

9.1 Cutting Conditions 9-1
9.1.1 Determining Cutting Speed and Feed Rate 9-1
9.1.2 Thread Cutting Count with Tool 9-3
9.1.3 Secondary Process Cutting Conditions Table 9-4
9.2 Cutting Feed Rate Quick Reference Table 9-6

(Blank page)

9.1 Cutting Conditions

This section provides various cutting conditions only as reference values. When using the L16/L20 machine, set these conditions optimized for the materials of workpieces to be cut and the shapes and materials of tools to be used.
The cutting speeds listed in the following tables assume use of a rotary guide bushing. If you use a fixed guide bushing, set the cutting speed 50 to 70% of the corresponding value in the table.

9.1.1 Determining Cutting Speed and Feed Rate

The cutting feed rate is generally based on the outer diameter of the material. However, if the diameters of the material and desired product diameter are very different (for example, when a product requires a considerable in-feed amount), it is necessary to set a higher cutting feed rate for cutting at the outer diameter of the material.

■Outer Diameter Cutting Speed (Super Hard Tool)

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		
Carbon Steel for Structure	80 to 180	Hard Cutting Stainless Steel (SUS304F equivalent or more)	30 to 80

■Outer Diameter Cutting Feed Rate (Speed Hard Tool)

Table 2

mm/rev

Material	Diametrical Direction Feed (X)			Longitudinal Direction 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
Carbon Steel for Structure	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

Cutting feed rate changes depending on a in-feed amount, a required surface roughness, precision and a tool life other than the quality of the material.

A A has a dimensional tolerance of up to ±5 microns, a surface roughness of up to 3S, and an in-feed amount (radius) of 4 mm or more.

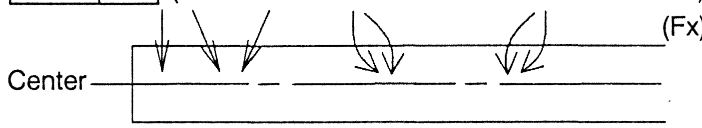
B B has a dimensional tolerance of up to ±10 microns, a surface roughness of up to 6S, and an in-feed amount (radius) of 2 to 4 mm.

C C has a dimensional tolerance of more than ±10 microns, a surface roughness of more than 6S, and an in-feed amount (radius) of 2 mm or less.

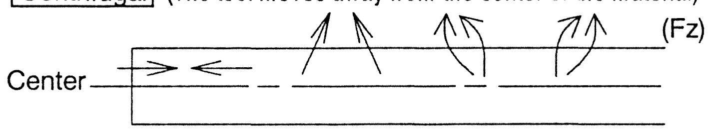
Note

The meaning of FX ("centripetal") and FZ ("centrifugal") is shown in the figure below.

Centripetal (The tool moves toward the center of the material)



Centrifugal (The tool moves away from the center of the material)



■Inner Diameter Cutting Speed (Super Hard Tool)

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		
Carbon Steel for Structure	40 to 90	Hard Cutting Stainless Steel (SUS304F equivalent or more)	15 to 40

■Inner Diameter Cutting Feed Rate (Super Hard Tool)

Cutting conditions for boring largely depend on the rigidity of the tool and tool holder. Diametrical direction feed rate is about half of the longitudinal direction feed rate.

Table 4 mm/rev

Material	Longitudinal Direction 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
Carbon Steel for Structure	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)	Feed Rate (mm/rev)		
		ϕ 1 to ϕ 2	ϕ 2 to ϕ 5	ϕ 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
Carbon Steel for Structure	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 feed rate of the material varies considerably depending on the tools to be used.

■Drilling Speed (High Speed Tool)

Table 6 m/min

Material	ϕ 1 to 2mm	ϕ 2 to 5mm	ϕ 5 to 12mm
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
Carbon Steel for Structure	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

■Drilling Feed Rate (High Speed Tool)

Table 7 mm/rev

Material	ϕ 1 to 2mm	ϕ 2 to 5mm	ϕ 5 to 12mm
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
Carbon Steel for Structure	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

9.1.2 Thread Cutting Count with Tool

■Non-Ferrous Materials

Table 8 (mm)

Pitch		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 Materials Count

Table 9 (mm)

Pitch		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

9.1.3 Secondary Process Cutting Conditions Table

■End Mill (Super Hard)

Table 10

Material	Diameter (mm)	Cutting Speed (m/min)	In-feed Amount (mm)	*Feed Rate (mm/cutter)
Non-ferrous Materials Brass, Aluminum etc.	ϕ 2	30 to 100	2.0	0.02 to 0.03
	ϕ 3	30 to 100	3.0	0.03 to 0.04
	ϕ 4	30 to 100	4.0	0.04 to 0.05
	ϕ 5, ϕ 6	30 to 100	5.0 to 6.0	0.05 to 0.08
Ferrous Free Cutting Materials like Free Cutting Steel	ϕ 2	30 to 80	2.0	0.015
	ϕ 3	30 to 80	3.0	0.02
	ϕ 4	30 to 80	4.0	0.03
	ϕ 5, ϕ 6	30 to 80	5.0 to 6.0	0.03 to 0.06
Ferrous Materials like Carbon Tool Steel, Carbon Steel for Structure etc.	ϕ 2	20 to 60	2.0	0.015 to 0.02
	ϕ 3	20 to 60	3.0	0.015 to 0.02
	ϕ 4	20 to 60	4.0	0.025 to 0.03
	ϕ 5, ϕ 6	20 to 60	5.0 to 6.0	0.04 to 0.06
Alloy Steel, Stainless Steel etc.	ϕ 2	15 to 50	2.0	0.008
	ϕ 3	15 to 50	3.0	0.015
	ϕ 4	15 to 50	4.0	0.025
	ϕ 5, ϕ 6	15 to 50	5.0 to 6.0	0.03 to 0.04

* The above cutting conditions for end mills are 1-D in-feed guidelines. For in-feed operation exceeding 1 D, adjust the settings.

■Slitting Cutter (90 Cutters, High Speed Tool Steel)

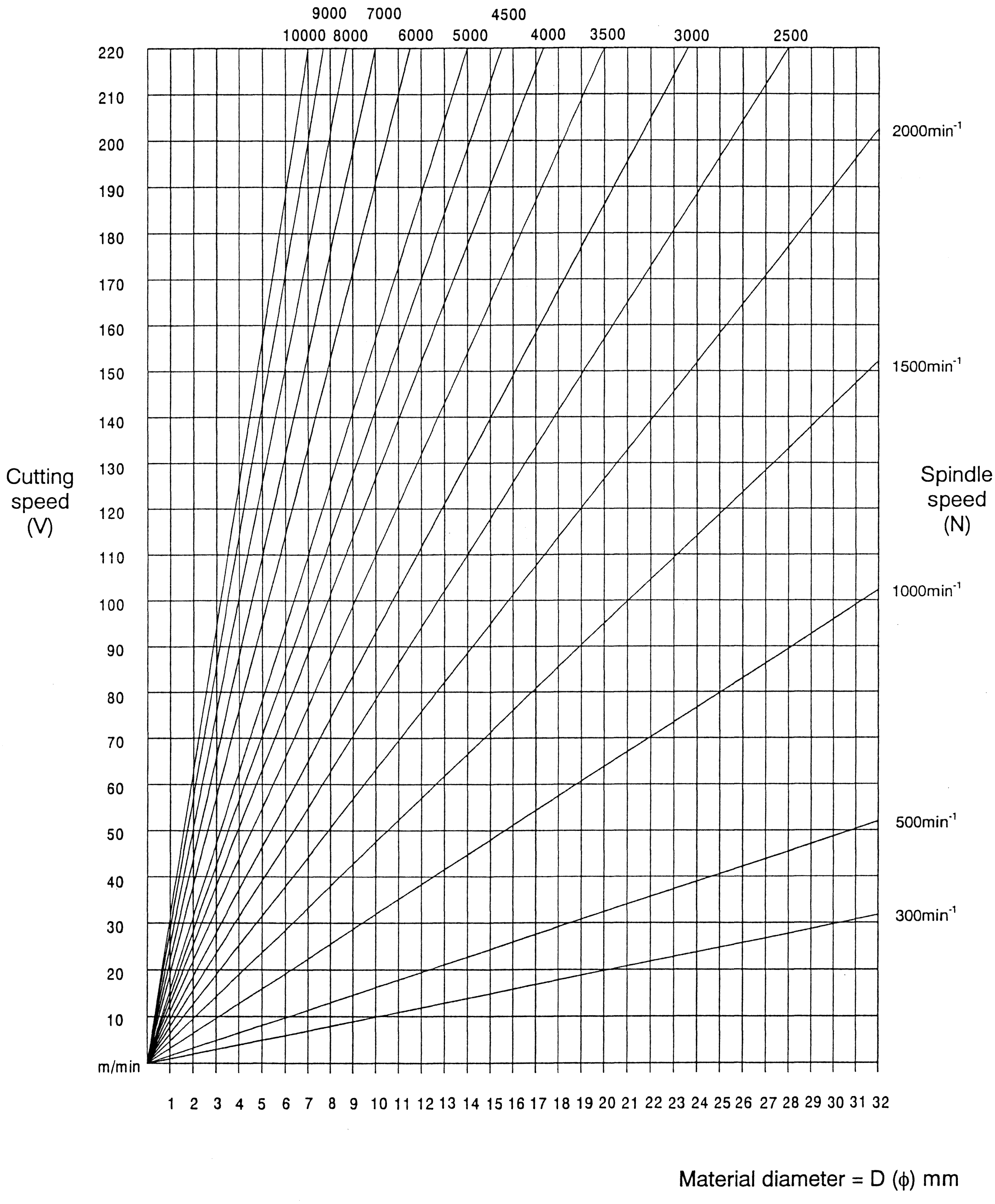
Table 11 *Indicates the mm/rev of 90 cutters.

Material	Diameter (mm)	Cutting Speed (m/min)	In-feed Amount (mm)	*Feed Rate (μ/cutter)
Non-ferrous Materials 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 Materials like 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 Materials like Carbon Tool Steel, Carbon Steel for Structure 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 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 seriously depending on the power and the speed reduction rate of the drive motor to be used.

9.2 Cutting Feed Rate Quick Reference Table

Table 12



(Blank page)

Product code	C	–	L	5	1	6	2	0	I	III	VII				–	5	7	0
Document code	2	E	1	–	0	9	0	2										
	2	E	2	–	0	9	0	1										

10. TOOLING

	Name		Specification	Model	page
¦ Tool Holder					
11-tool vertical holder	4-rotary tool		[□10]	GTF3110	10-3
11-tool vertical holder	4-rotary tool		[□3/8"]	GTF3110L	10-4
10-tool vertical holder	4-rotary tool		[□12]	GTF3112	10-5
10-tool vertical holder	4-rotary tool		[□13][□1/2"]	GTF3113	10-6
11-tool vertical holder	3-rotary tool		[□10]	GTF3210	10-7
11-tool vertical holder	3-rotary tool		[□3/8"]	GTF3210L	10-8
10-tool vertical holder	3-rotary tool		[□12]	GTF3212	10-9
10-tool vertical holder	3-rotary tool		[□13][□1/2"]	GTF3213	10-10
9-tool vertical holder	2-rotary tool		[□10]	GTF3410	10-11
9-tool vertical holder	2-rotary tool		[□3/8"]	GTF3410L	10-12
8-tool vertical holder	2-rotary tool		[□12]	GTF3412	10-13
8-tool vertical holder	2-rotary tool		[□13][□1/2"]	GTF3413	10-14
Tool holder	For opposite tool post(Type III)		[□12]	GTA112	10-15
Tool holder	For opposite tool post(Type III)		[□12]	GTA212	10-16
Shift tool holder	15mm-shift, Adapter type		[□10]	GTF3310	10-17
Shift tool holder	5/8"mm-shift, Adapter type		[□3/8"]	GTF3310L	10-18
Shift tool holder	15mm-shift, Adapter type		[□12]	GTF3312	10-19
Shift tool holder	15mm-shift, Adapter type		[□13]	GTF3313	10-20
Shift tool holder	5/8"mm-shift, Adapter type		[□1/2"]	GTF3313L	10-21
Wedge for □12 tool	For □13 vertical holder		[□12]	GTJ212	10-22
¦ Sleeve Holder					
3-tool vertical sleeve holder	For 8 or 9- vertical holder		[Sleeve diameter £25.4]	GDF306	10-23
1-tool vertical sleeve holder	For rotary tool vertical holder		[Sleeve diameter £19.05]	GDF506	10-24
3-tool vertical sleeve holder	For 3-tool rotary tool vertical holder		[Sleeve diameter £19.05]	GDF606	10-25
3-tool vertical sleeve holder	For 4-tool rotary tool vertical holder		[Sleeve diameter £19.05]	GDF607	10-26
1-tool sleeve holder	For opposite tool post(Type III)		[Sleeve diameter £19.05]	GDF102	10-27
1-tool sleeve holder	For opposite tool post(Type III)		[Sleeve diameter £19.05]	GDF103	10-28
¦ Rotary tool					
[Outer Circumference Milling]					
Rego type chuck (~£7)			[ER11][AR11]	GSC507	10-29
Rego type chuck (~£10)			[ER16][AR16]	GSC510	10-30

Name	Specification	Model	page
[Face Drilling]			
Rego type chuck (~£7)	For U121B,U151B [ER11][AR11]	GSE307	10-31
Rego type chuck (~£7)	For U131B [ER11][AR11]	GSE407	10-32
Rego type chuck (~£10)	For U131B [ER16][AR16]	GSE410	10-33

¦ Sleeve

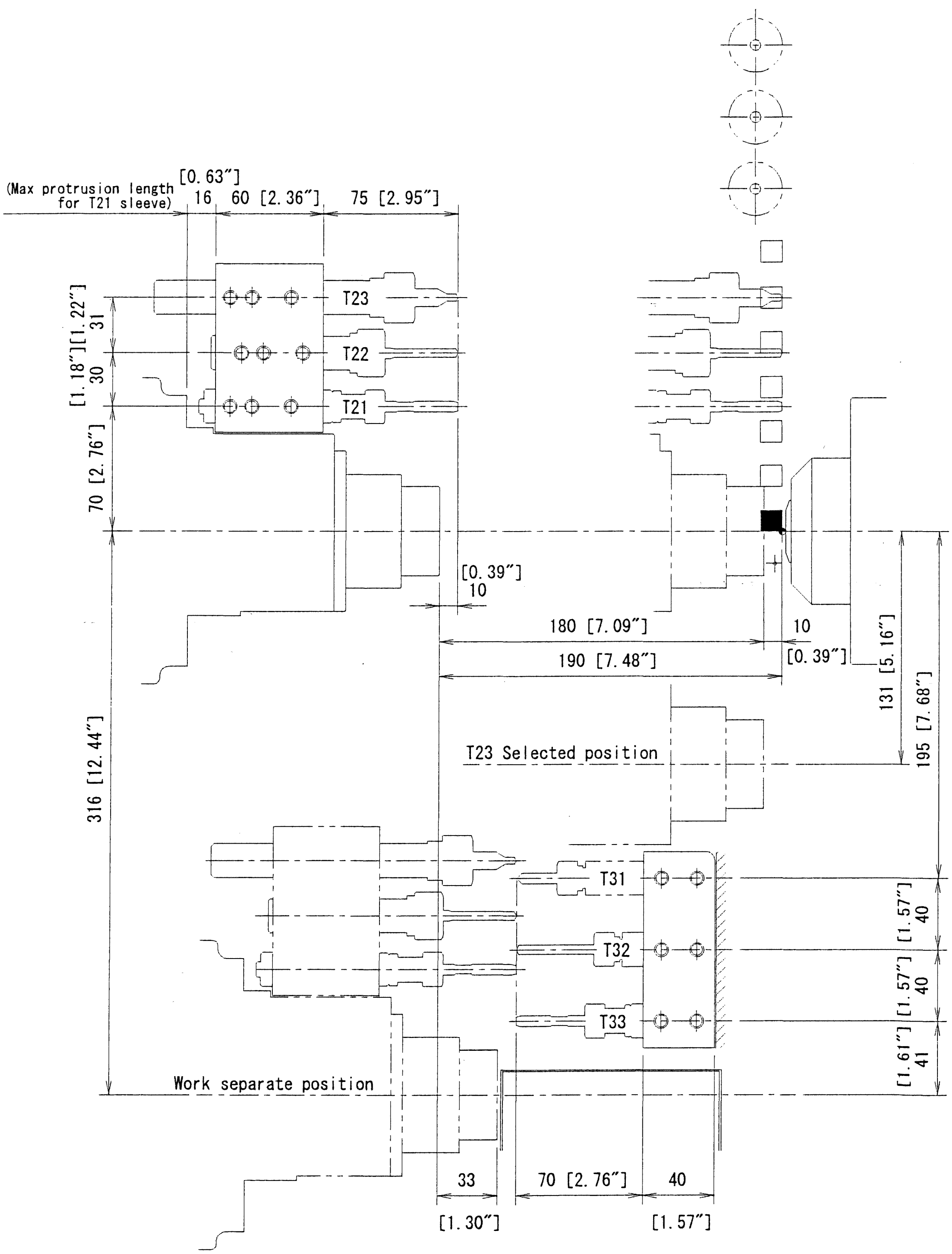
Name	Specification	Model	page
[Drill and Boring]			
Rego type chuck (~£7,Short type)	Shank dia£19.05 [ER11][AR11]	BDS507	10-34
Rego type chuck (~£10, Short type)	Shank dia £19.05 [ER16][AR16]	BDS508	10-35
Rego type chuck (~£7, Semi short type)	Shank dia £19.05 [ER11][AR11]	BDS607	10-36
Rego type chuck (~£10, Semi short type)	Shank dia £19.05 [ER16][AR16]	BDS610	10-37
Rego type chuck (~£7)	Shank dia £19.05 [ER11][AR11]	VDS506	10-38
Rego type chuck (~£10)	Shank dia £19.05 [ER16][AR16]	VDS110	10-39
Rego type chuck (~£7,Long type)	Shank dia £19.05 [ER11][AR11]	LDS107	10-40
Rego type chuck (~£10,Long type)	Shank dia £19.05 [ER16][AR16]	LDS110	10-41
Rego type chuck (~£7)	Shank dia £25.4 [ER11][AR11]	ADS707	10-42
Rego type chuck (~£10)	Shank dia £25.4 [ER16][AR16]	ADS710	10-43
Rego type chuck (~£7,Both ends of short type)	Shank dia £19.05 [ER11][AR11]	HDS5406	10-44
Rego type chuck (~£7,Both ends)	Shank dia £19.05 [ER11][AR11]	HDS5506	10-45
[Tap and Die]			
Rego type chuck (~£7,Short type)	Shank dia £19.05 [ER11][AR11] Die £16,£20	BNS407	10-46
Rego type chuck (~£7,Short type)	Shank dia £19.05 [ER11][AR11] Die £5/8",£13/16"	BNS507	10-47
Rego type chuck (~£7)	Shank dia £19.05 [ER11][AR11] Die £16,£20	VNS406	10-48
Rego type chuck (~£7)	Shank dia £19.05 [ER11][AR11] Die £5/8",£13/16"	VNS506	10-49
Rego type chuck (~£7)	Shank dia £25.4 [ER11][AR11]	ATS307	10-50

¦ Adapter

Name	Specification	Model	page
Sleeve adapter	Shank dia £25 [Sleeve diameter£19.05]	SAU119	10-51
Sleeve adapter	Shank dia £32 [Sleeve diameter£19.05]	SAU219	10-52

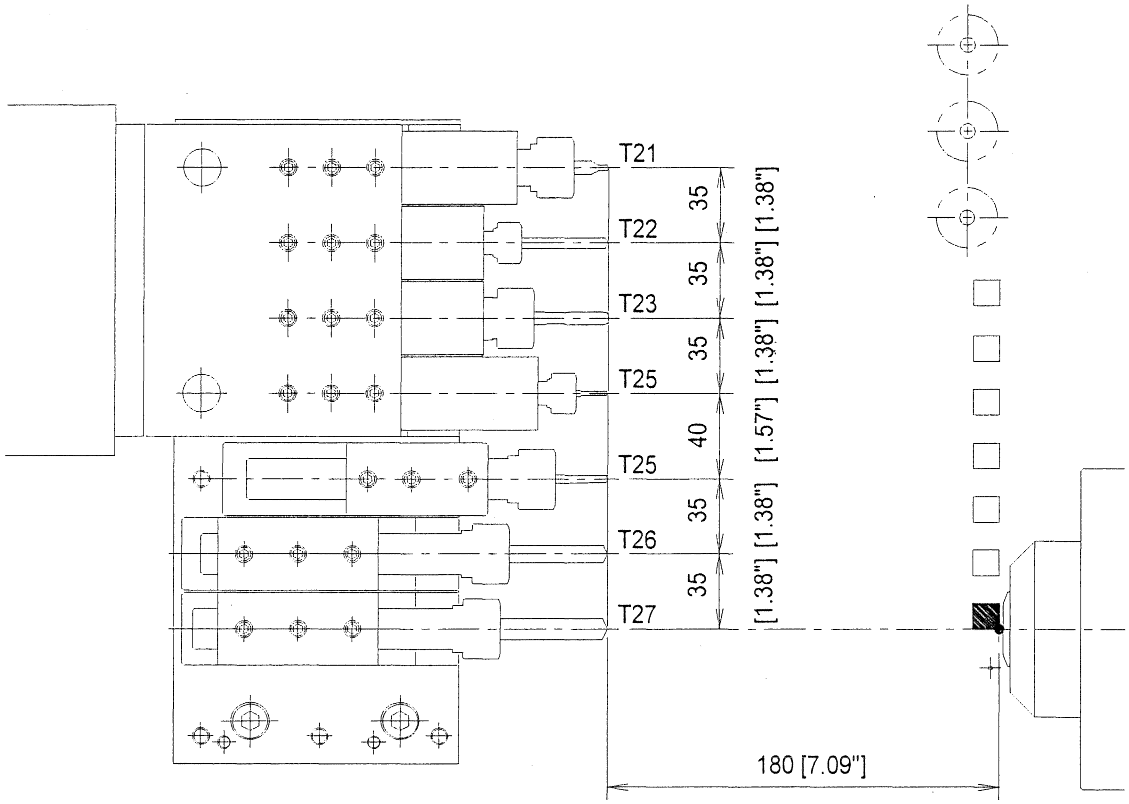
Appendix Restrictions on Tooling 10-53

L16(Type VII) / L20(Type VII) Standard tool layout



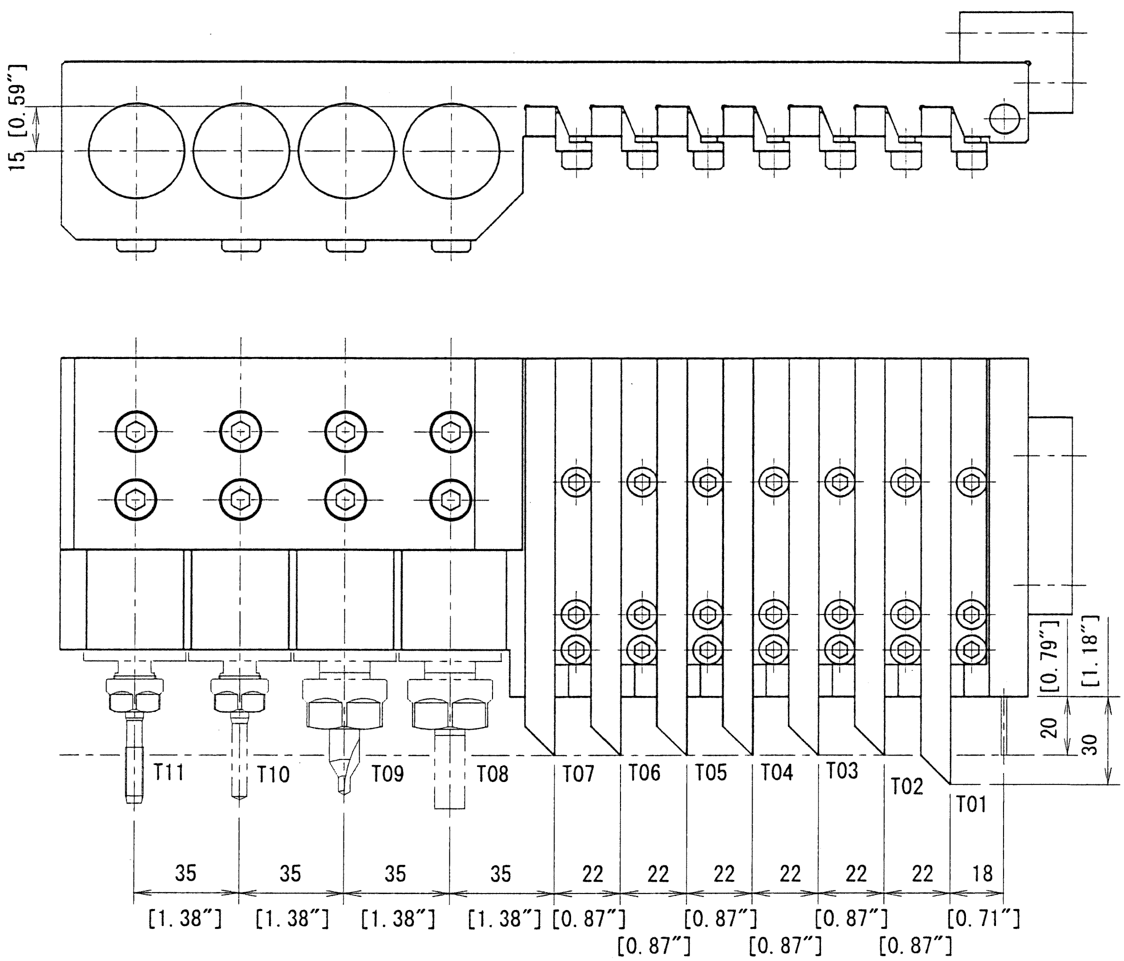
The standard tool layout and stroke of opposite tool post device of L16(Type VII) / L20(Type VII) shown above.
It is not always necessary to follow the above form.
Machining can be done by optimum layout for the workpiece.

L16(Type III) / L20(Type III) Standard tool layout



The standard tool layout and stroke of opposite tool post device of L16(Type III) / L20(Type III) shown above.
It is not always necessary to follow the above form.
Machining can be done by optimum layout for the workpiece.

GTF3110 11-Tool Vertical Holder(4-Rotary tool and 7-tool)

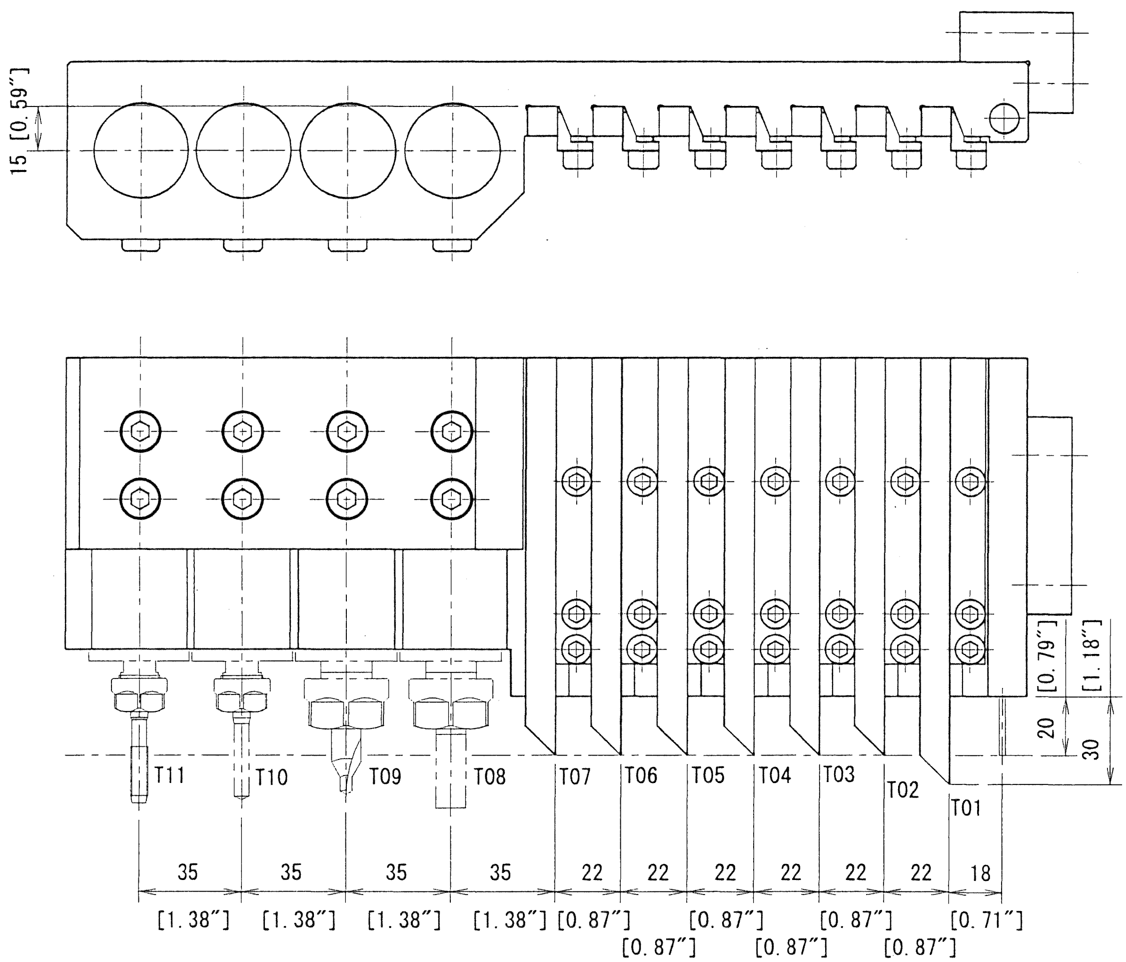


GTF3110 is provided for outer diameter cutting.
4 rotary tools can be mounted on the 15mm [0.59"] shifted position.

NOTE
- Cut-off tool shall be mounted on the right most position (T01)
- When the end milling operation is used at the left most position(T11), D cutting can not be operated to exceed the center of the workpiece for the stroke limit. Endmill operation shall be done at T08 to T10.

Tool holder name	GTF3110
Usage	All types of tools
Tool size	10 x 10 x 120mm
Rotary tool	GSC507,GSC510
Face drilling sleeve holder	GDF506,GDF607
Shift tool holder	GTF3310

GTF3110L 11-Tool Vertical Holder(4-Rotary tool and 7-tool)

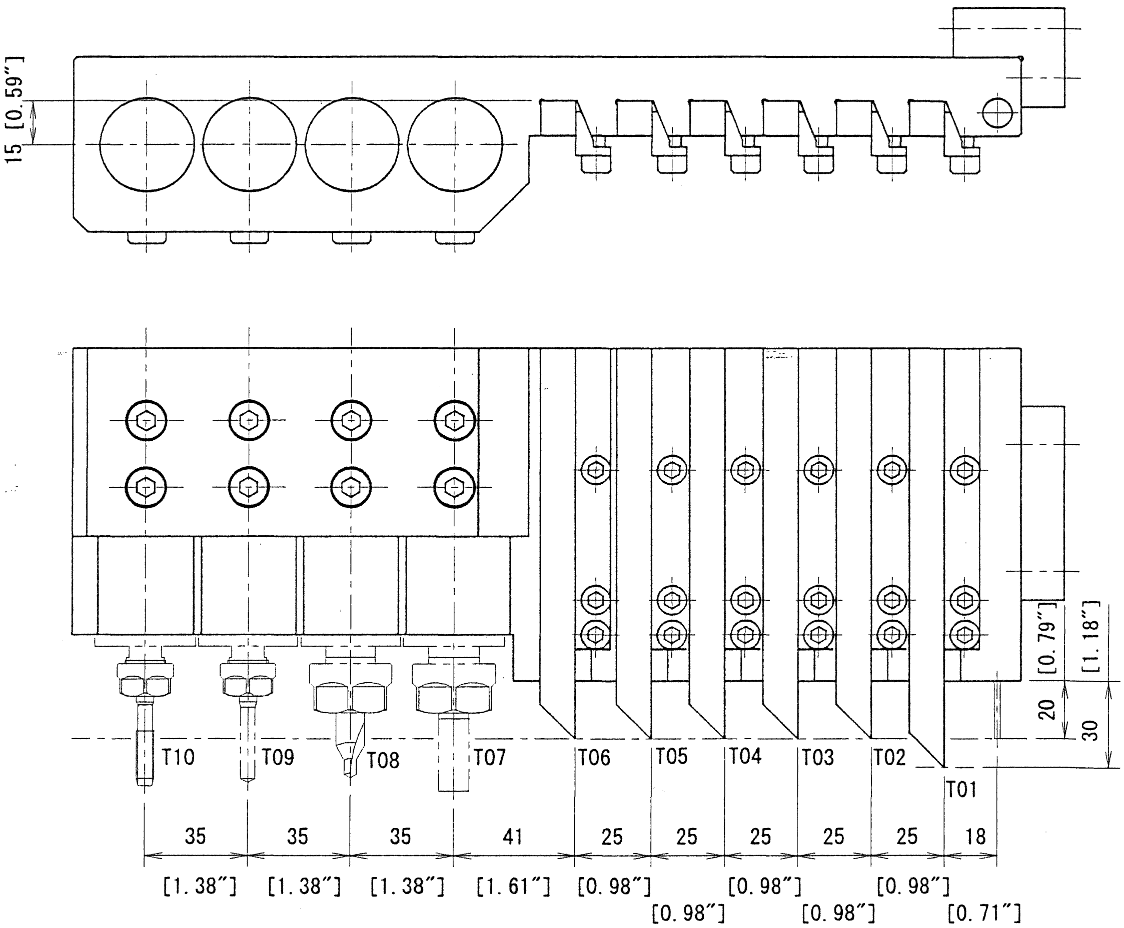


GTF3110L is provided for outer diameter cutting.
4 rotary tools can be mounted on the 15mm [0.59"] shifted position.

NOTE
- Cut-off tool shall be mounted on the right most position (T01)
- When the end milling operation is used at the left most position(T11), D cutting can not be operated to exceed the center of the workpiece for the stroke limit. Endmill operation shall be done at T08 to T10.

Tool holder name	GTF3110L
Usage	All types of tools
Tool size	3/8" x 3/8" x 4-3/4"
Rotary tool	GSC507,GSC510
Face drilling sleeve holder	GDF506,GDF607
Shift tool holder	GTF3310L

GTF3112 10-Tool Vertical Holder(4-Rotary tool and 6-tool)

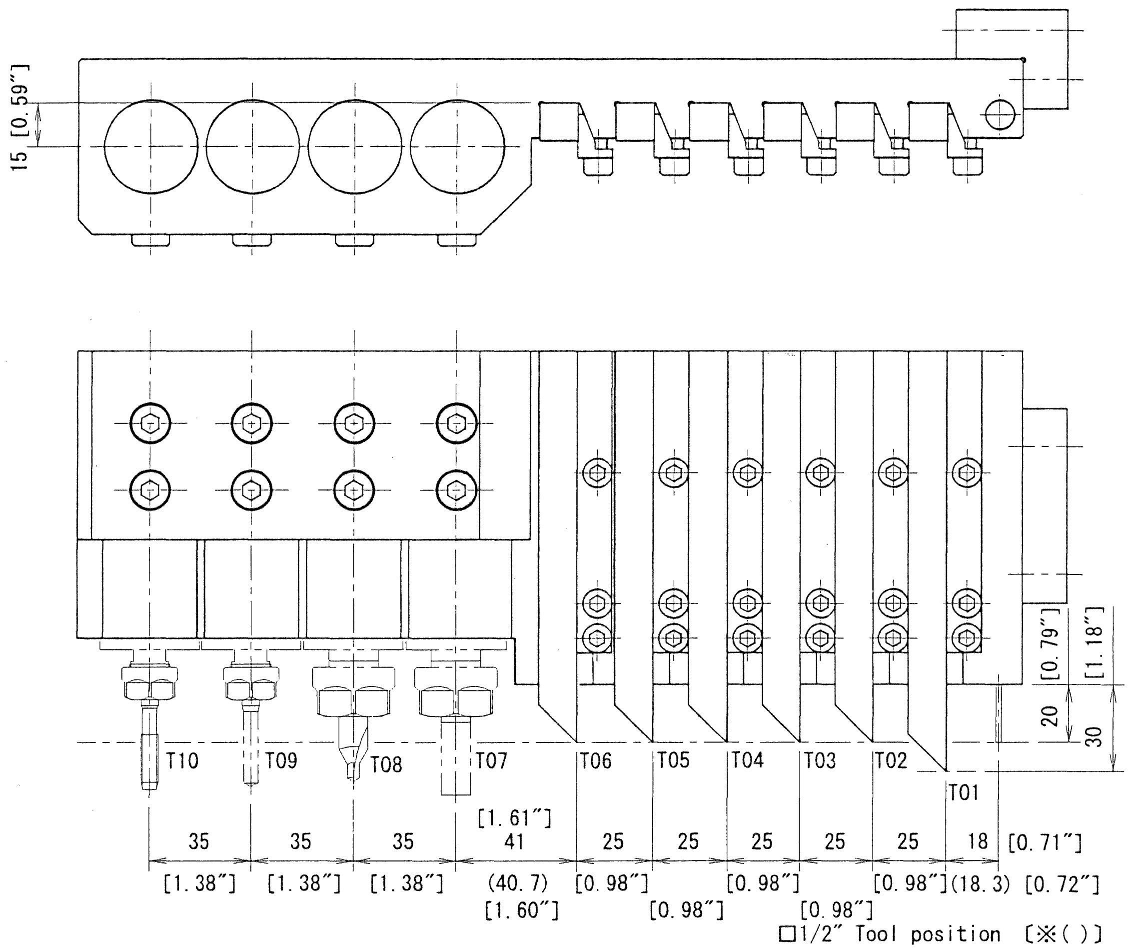


GTF3112 is provided for outer diameter cutting.
4 rotary tools can be mounted on the 15mm [0.59"] shifted position.

NOTE
- Cut-off tool shall be mounted on the right most position (T01)
- When the end milling operation is used at the left most position(T10), D cutting can not be operated to exceed the center of the workpiece for the stroke limit. Endmill operation shall be done at T07 to T09.

Tool holder name	GTF3112
Usage	All types of tools
Tool size	12 x 12 x 120mm
Rotary tool	GSC507,GSC510
Face drilling sleeve holder	GDF506,GDF607
Shift tool holder	GTF3312

GTF3113 10-Tool Vertical Holder(4-Rotary tool and 6-tool)

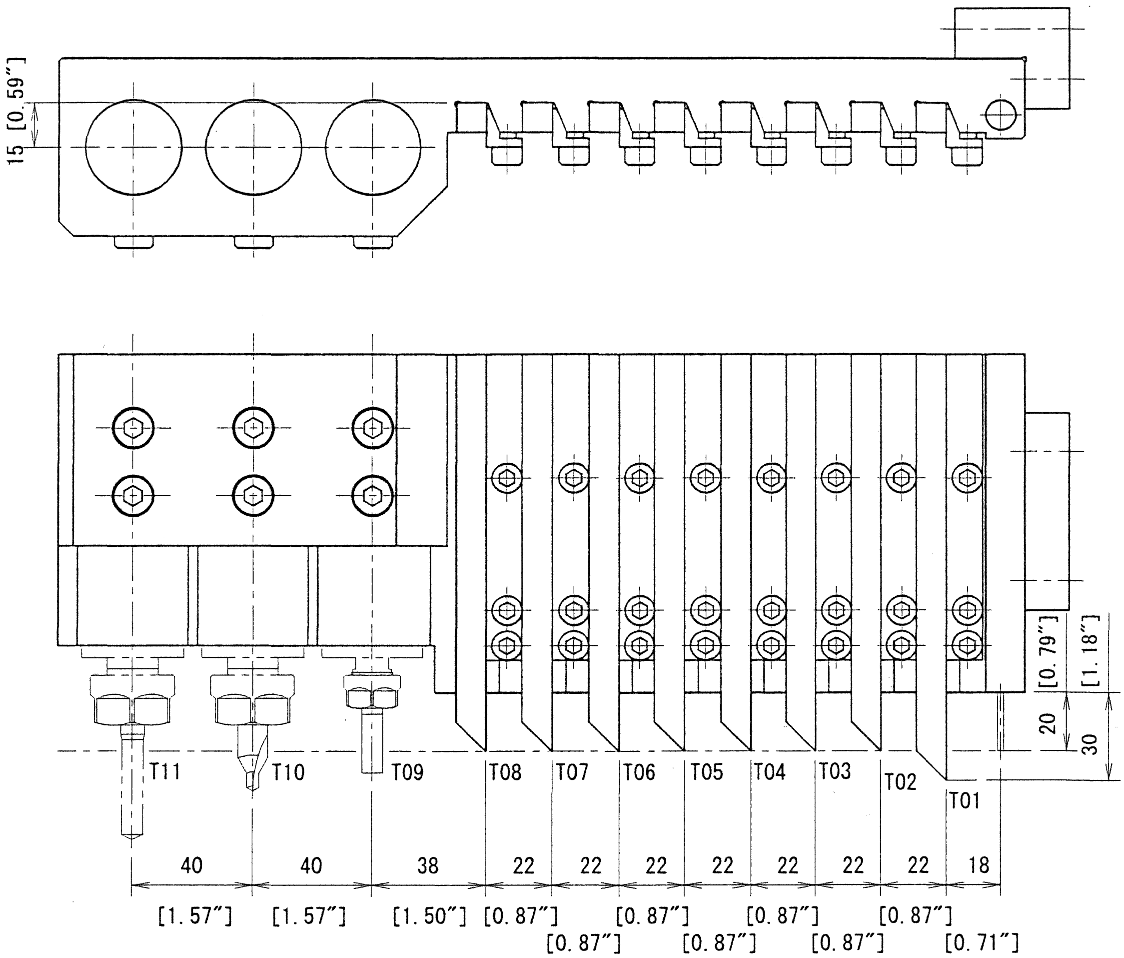


GTF3113 is provided for outer diameter cutting.
4 rotary tools can be mounted on the 15mm [0.59"] shifted position.

NOTE · Cut-off tool shall be mounted on the right most position (T01)
· When the end milling operation is used at the left most position(T10), D cutting can not be operated to exceed the center of the workpiece for the stroke limit. Endmill operation shall be done at T07 to T09.

Tool holder name	GTF3113
Usage	All types of tools
Tool size	13 x 13 x 120mm [1/2" x 1/2" x 4-3/4"]
Rotary tool	GSC507,GSC510
Face drilling sleeve holder	GDF506,GDF607
Shift tool holder	GTF3313,GTF3313L

GTF3210 11-Tool Vertical Holder(3-Rotary tool and 8-tool)

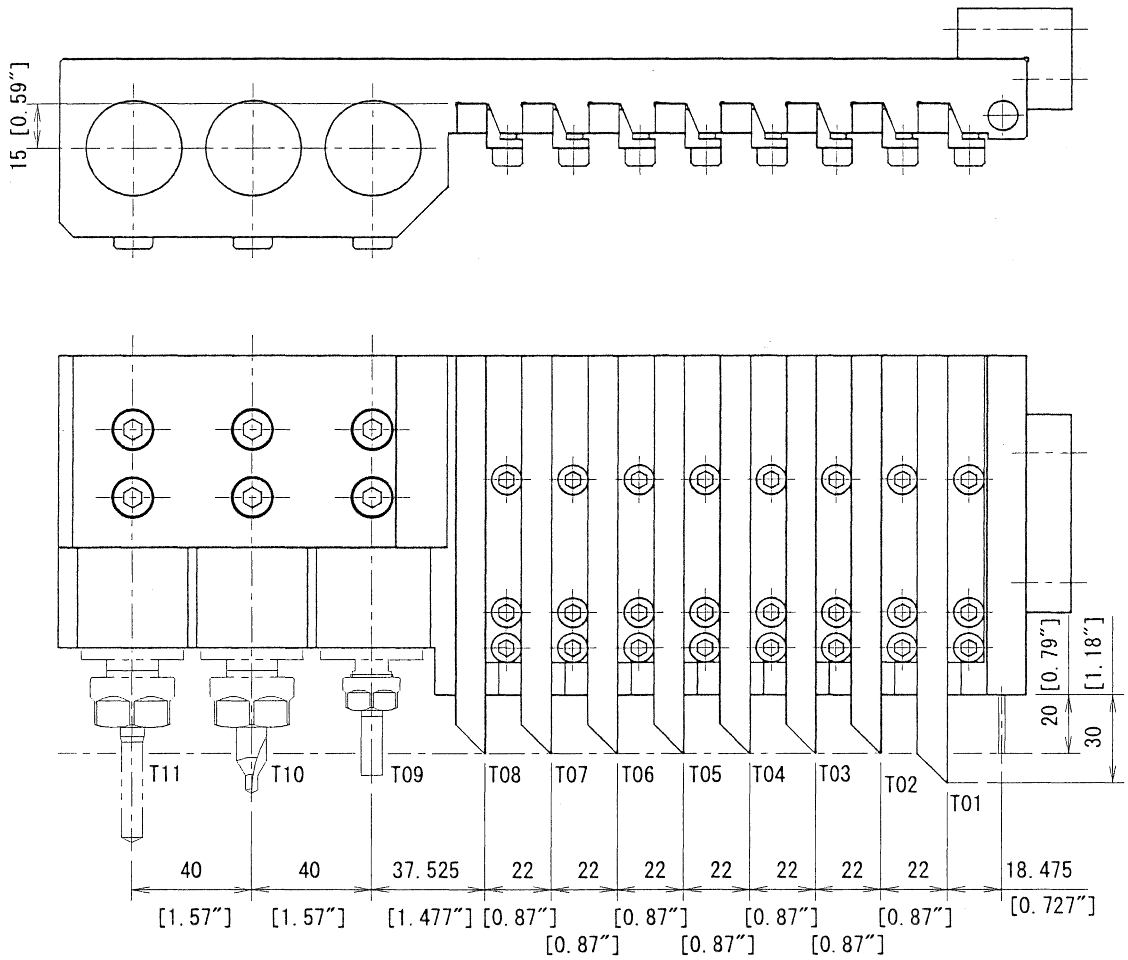


GTF3210 is provided for outer diameter cutting.
3 rotary tools can be mounted on the 15mm [0.59"] shifted position.

NOTE
- Cut-off tool shall be mounted on the right most position (T01)
- When the end milling operation is used at the left most position(T11), D cutting can not be operated to exceed the center of the workpiece for the stroke limit. Endmill operation shall be done at T09 or T10.

Tool holder name	GTF3210
Usage	All types of tools
Tool size	10 x 10 x 120mm
Rotary tool	GSC507,GSC510
Face drilling sleeve holder	GDF506,GDF606
Shift tool holder	GTF3310

GTF3210L 11-Tool Vertical Holder(3-Rotary tool and 8-tool)

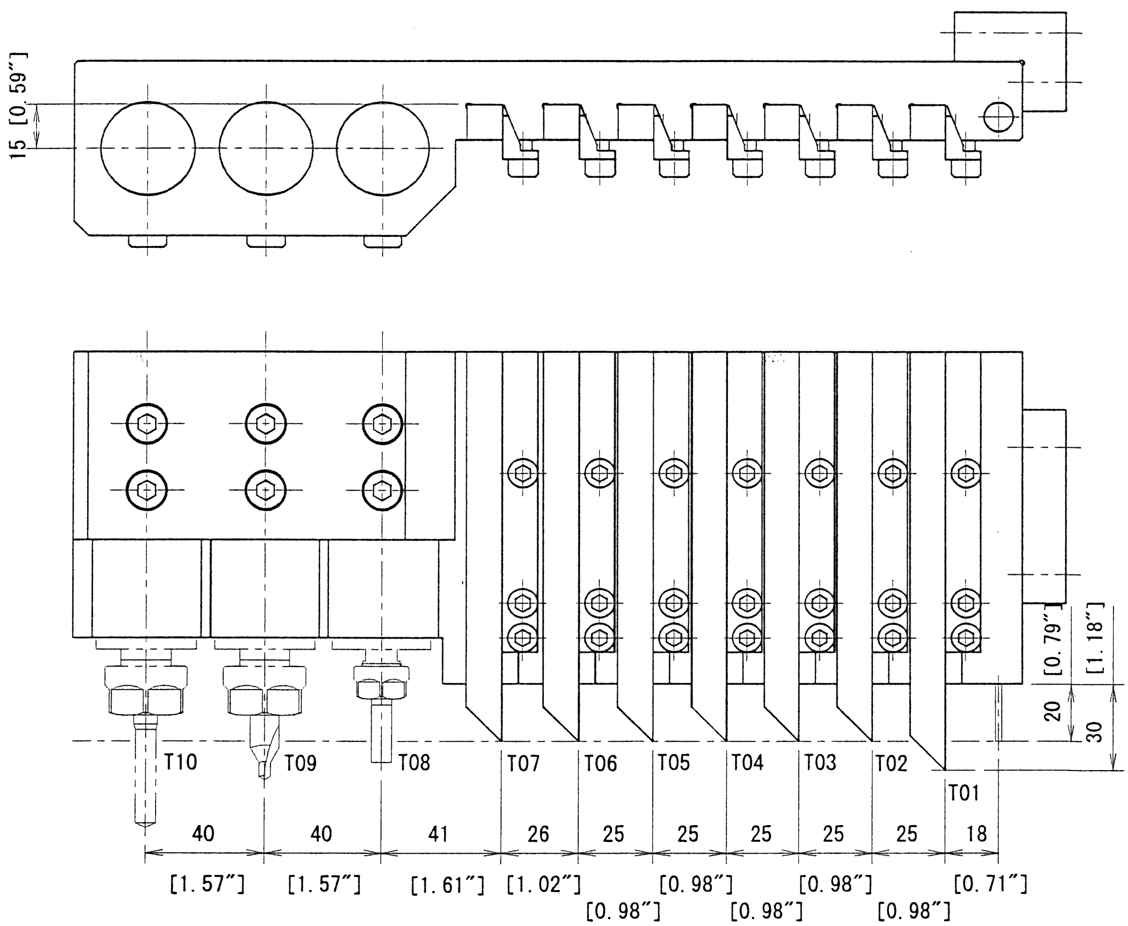


GTF3210L is provided for outer diameter cutting.
3 rotary tools can be mounted on the 15mm [0.59"] shifted position.

NOTE
- Cut-off tool shall be mounted on the right most position (T01)
- When the end milling operation is used at the left most position(T11), D cutting can not be operated to exceed the center of the workpiece for the stroke limit. Endmill operation shall be done at T09 or T10.

Tool holder name	GTF3210L
Usage	All types of tools
Tool size	3/8" x 3/8" x 4-3/4"
Rotary tool	GSC507,GSC510
Face drilling sleeve holder	GDF506,GDF606
Shift tool holder	GTF3310L

GTF3212 10-Tool Vertical Holder(3-Rotary tool and 7-tool)

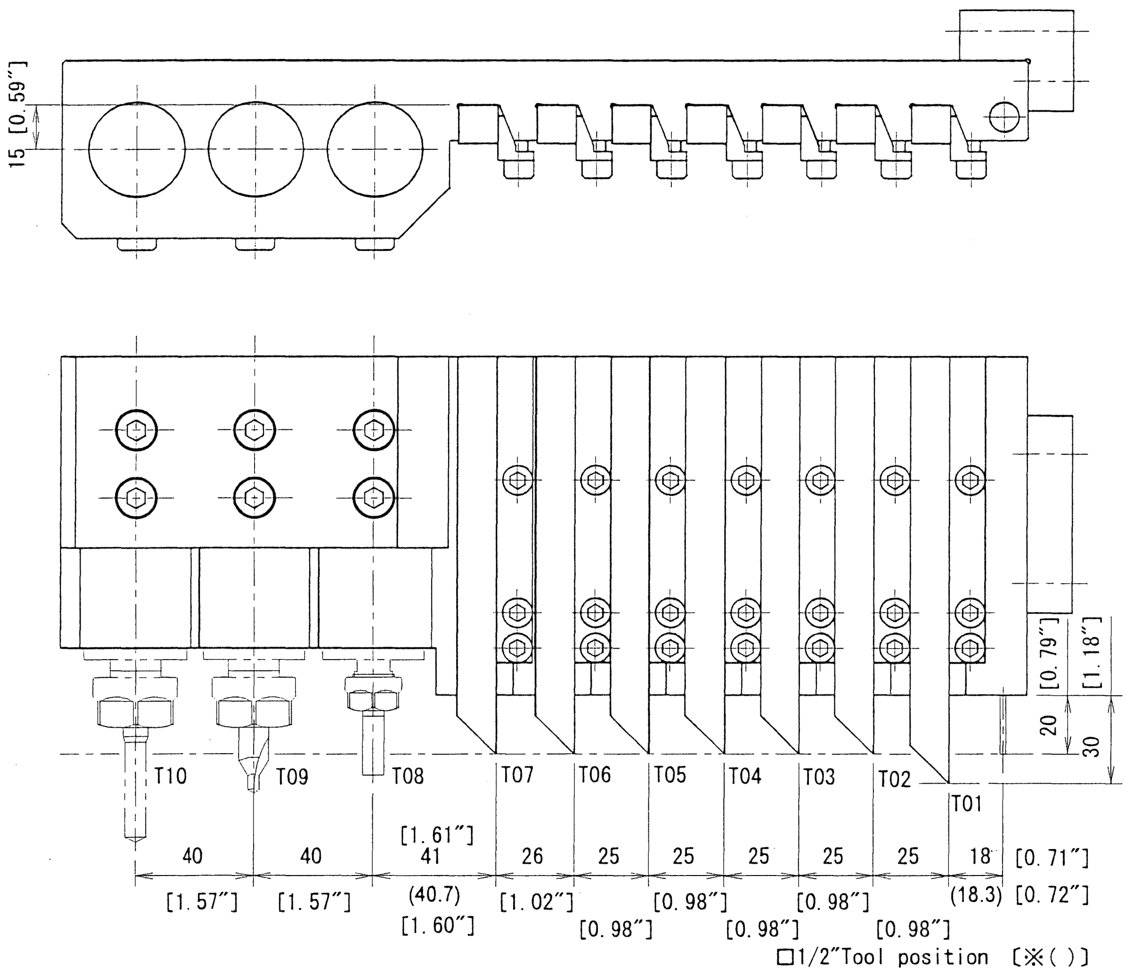


GTF3212 is provided for outer diameter cutting.
3 rotary tools can be mounted on the 15mm [0.59"] shifted position.

NOTE
- Cut-off tool shall be mounted on the right most position (T01)
- When the end milling operation is used at the left most position(T10), D cutting can not be operated to exceed the center of the workpiece for the stroke limit. Endmill operation shall be done at T08 or T09.

Tool holder name	GTF3212
Usage	All types of tools
Tool size	12 x 12 x 120mm
Rotary tool	GSC507,GSC510
Face drilling sleeve holder	GDF506,GDF606
Shift tool holder	GTF3312

GTF3213 10-Tool Vertical Holder(3-Rotary tool and 7-tool)

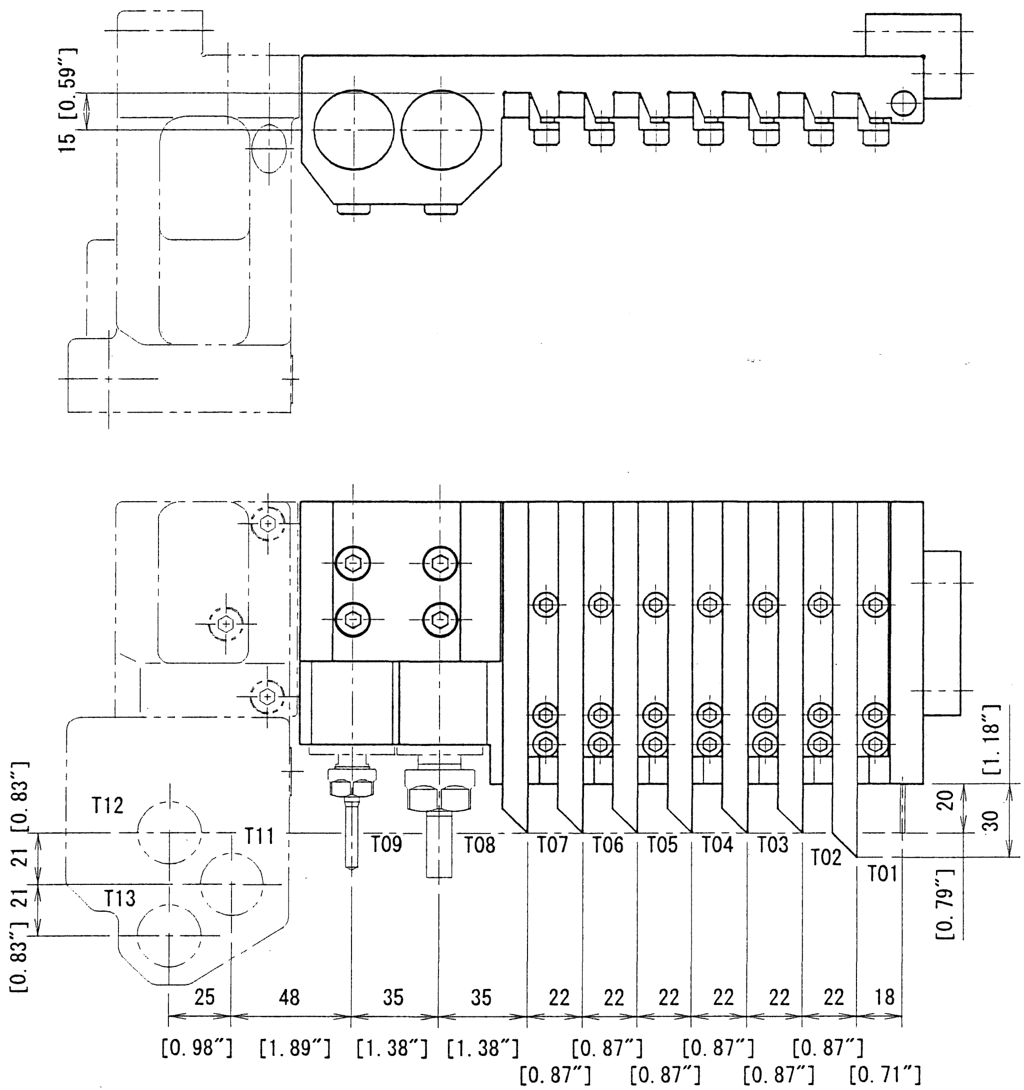


GTF3213 is provided for outer diameter cutting.
3 rotary tools can be mounted on the 15mm [0.59"] shifted position.

NOTE
- Cut-off tool shall be mounted on the right most position (T01)
- When the end milling operation is used at the left most position(T10), D cutting can not be operated to exceed the center of the workpiece for the stroke limit. Endmill operation shall be done at T08 or T09.

Tool holder name	GTF3213
Usage	All types of tools
Tool size	13 x 13 x 120mm [1/2" x 1/2" x 4-3/4"]
Rotary tool	GSC507,GSC510
Face drilling sleeve holder	GDF506,GDF606
Shift tool holder	GTF3313,GTF3313L

GTF3410 9-Tool Vertical Holder(2-Rotary tool and 7-tool)

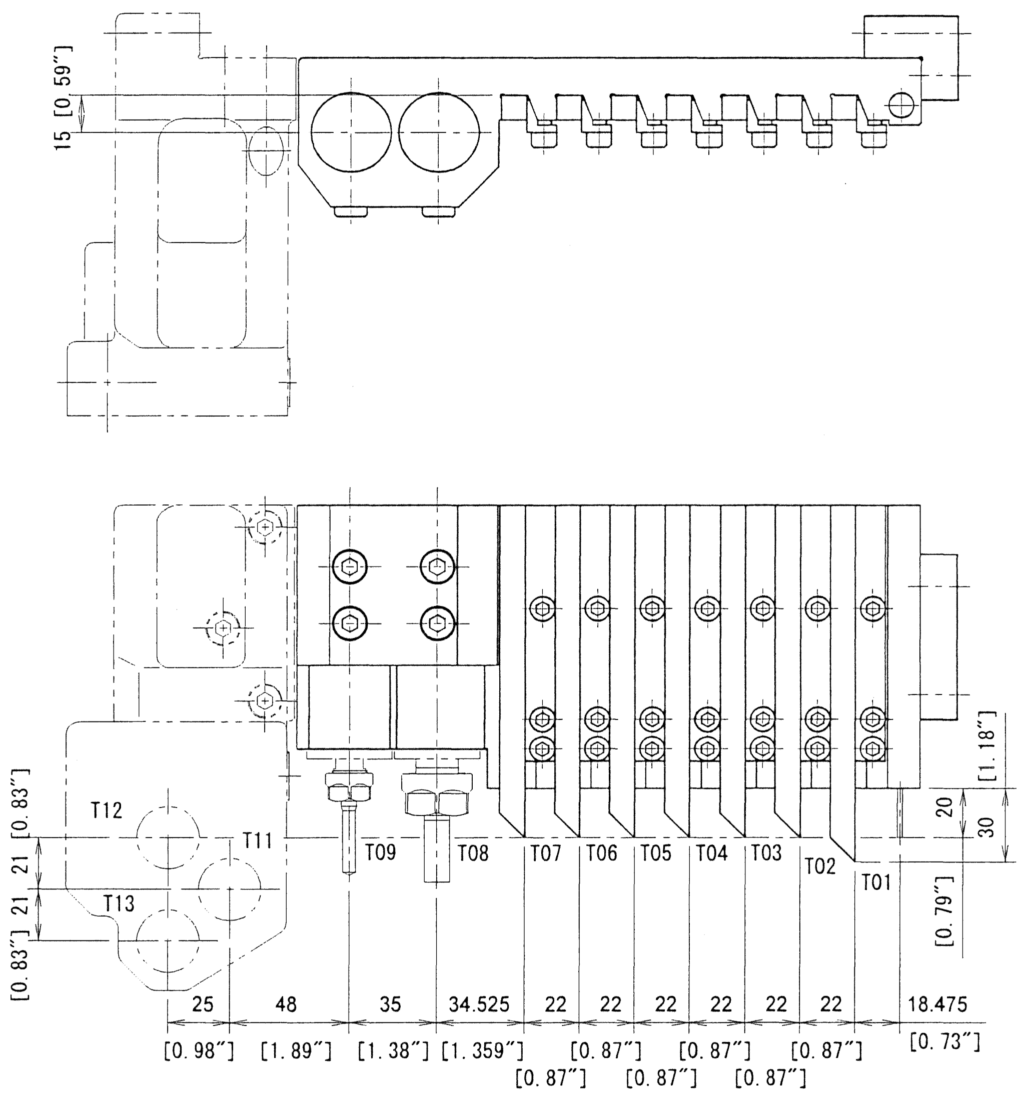


GTF3410 is provided for outer diameter cutting.
2 rotary tools can be mounted on the 15mm [0.59"] shifted position.

NOTE ・Cut-off tool shall be mounted on the right most position (T01)

Tool holder name	GTF3410
Usage	All types of tools
Tool size	10 x 10 x 120mm
Rotary tool	GSC507,GSC510
Face drilling sleeve holder	GDF306,GDF506
Shift tool holder	GTF3310

GTF3410L 9-Tool Vertical Holder(2-Rotary tool and 7-tool)

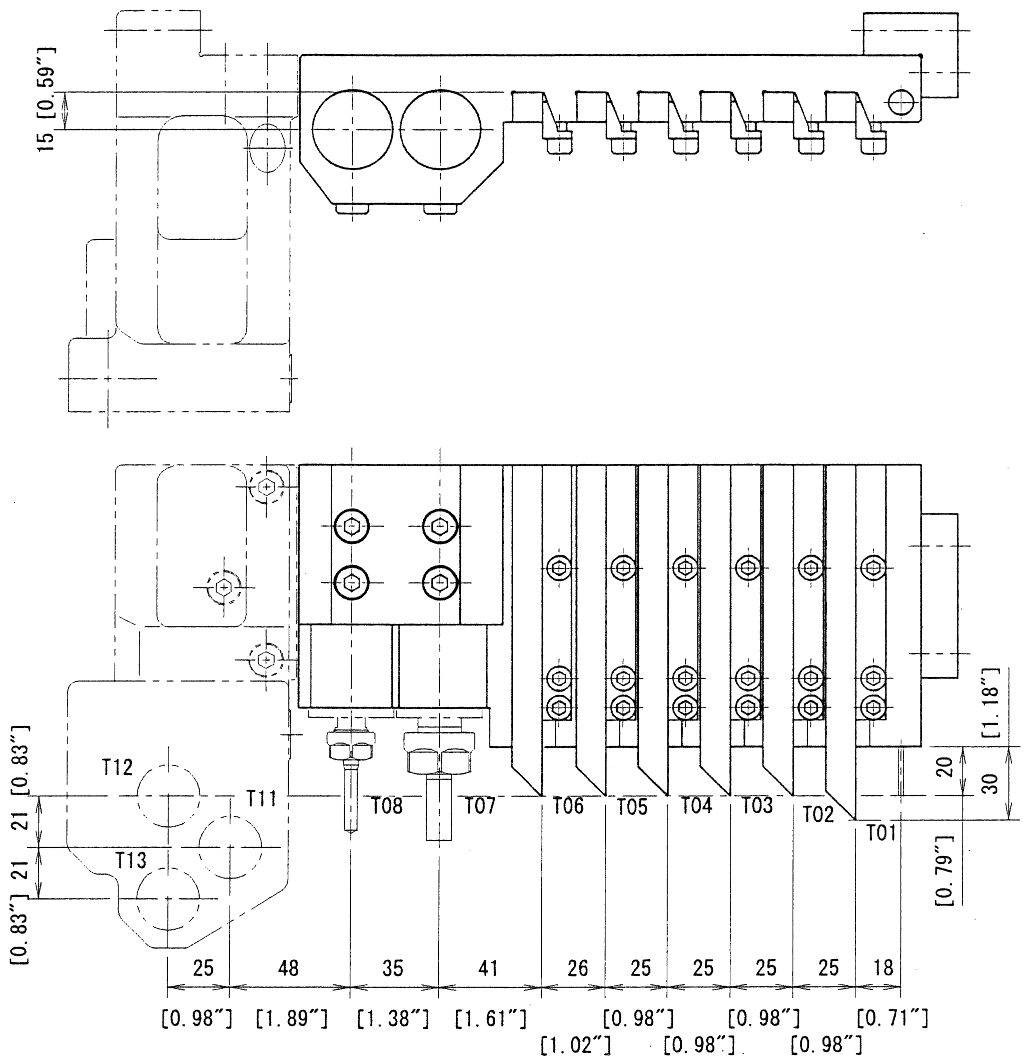


GTF3410L is provided for outer diameter cutting.
2 rotary tools can be mounted on the 15mm [0.59"] shifted position.

NOTE ・Cut-off tool shall be mounted on the right most position (T01)

Tool holder name	GTF3410L
Usage	All types of tools
Tool size	3/8" x 3/8" x 4-3/4"
Rotary tool	GSC507,GSC510
Face drilling sleeve holder	GDF306,GDF506
Shift tool holder	GTF3310L

GTF3412 8-Tool Vertical Holder(2-Rotary tool and 6-tool)

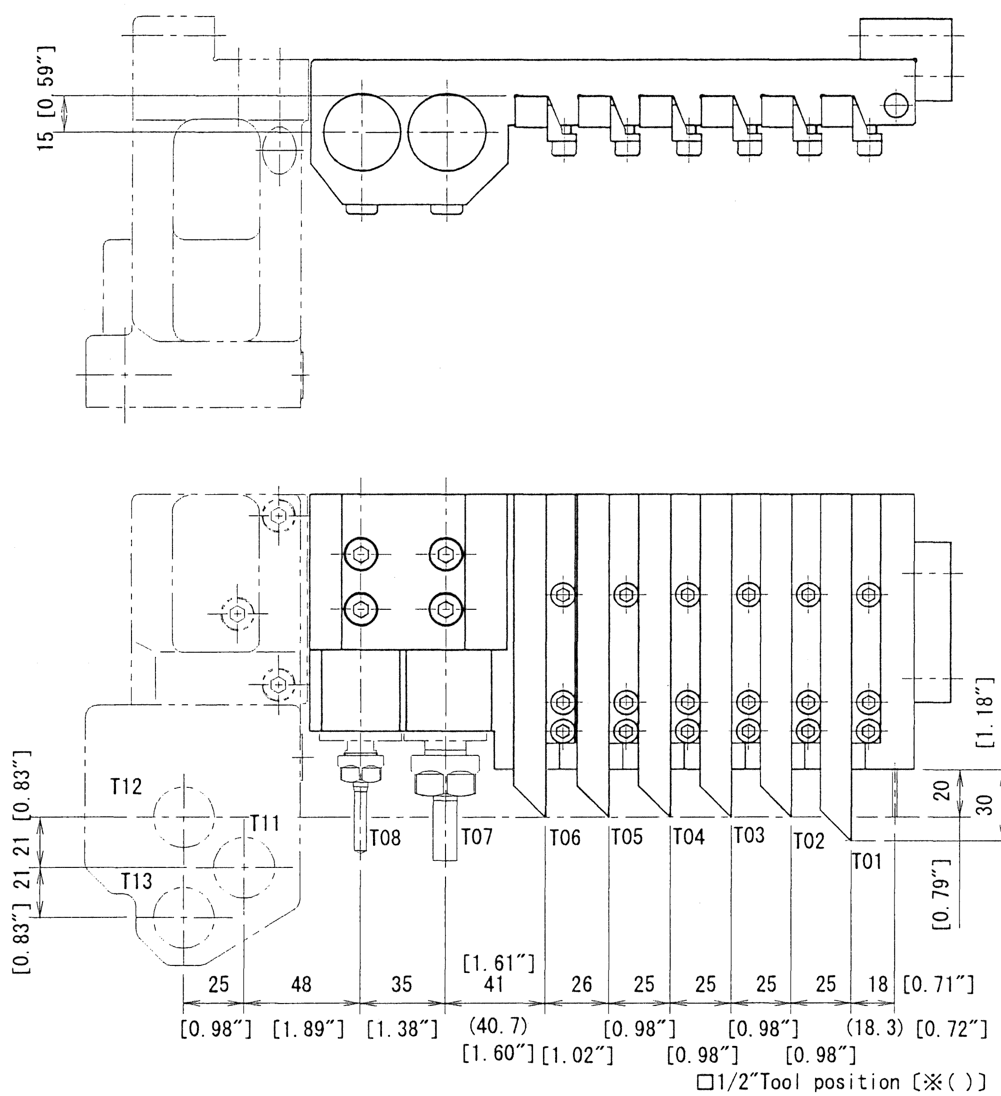


GTF3412 is provided for outer diameter cutting.
2 rotary tools can be mounted on the 15mm [0.59"] shifted position.

NOTE ·Cut-off tool shall be mounted on the right most position (T01)

Tool holder name	GTF3412
Usage	All types of tools
Tool size	12 x 12 x 120mm
Rotary tool	GSC507,GSC510
Face drilling sleeve holder	GDF306,GDF506
Shift tool holder	GTF3312

GTF3413 8-Tool Vertical Holder(2-Rotary tool and 6-tool)

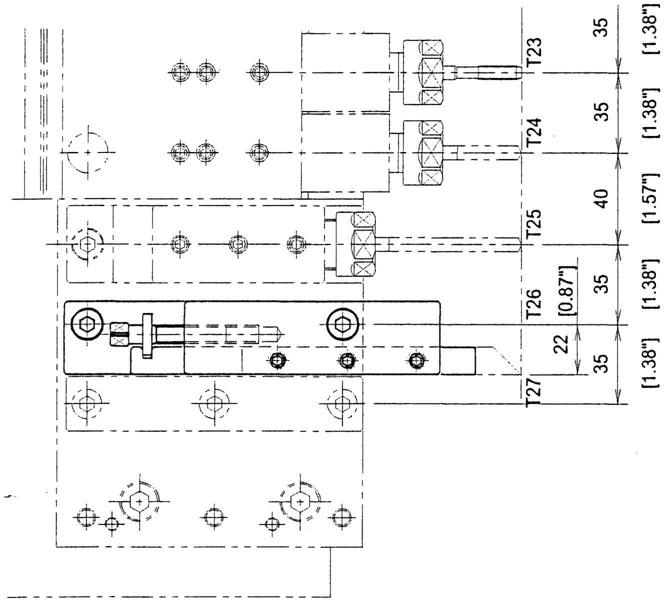


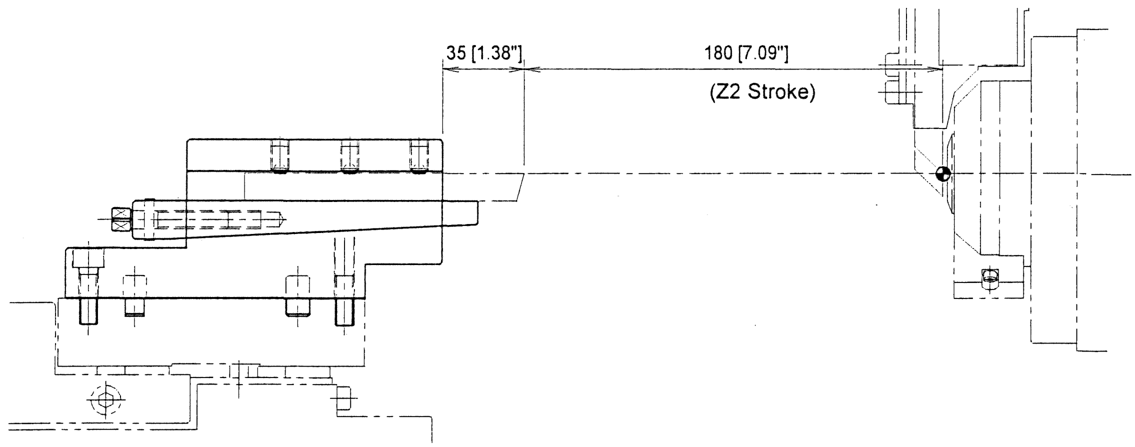
GTF3413 is provided for outer diameter cutting.
2 rotary tools can be mounted on the 15mm [0.59"] shifted position.

NOTE · Cut-off tool shall be mounted on the right most position (T01)

Tool holder name	GTF3413
Usage	All types of tools
Tool size	13 x 13 x 120mm [1/2" x 1/2" x 4-3/4"]
Rotary tool	GSC507,GSC510
Face drilling sleeve holder	GDF306,GDF506
Shift tool holder	GTF3313,GTF3313L

GTA112 Tool Holder(Type III for opposite tool post)



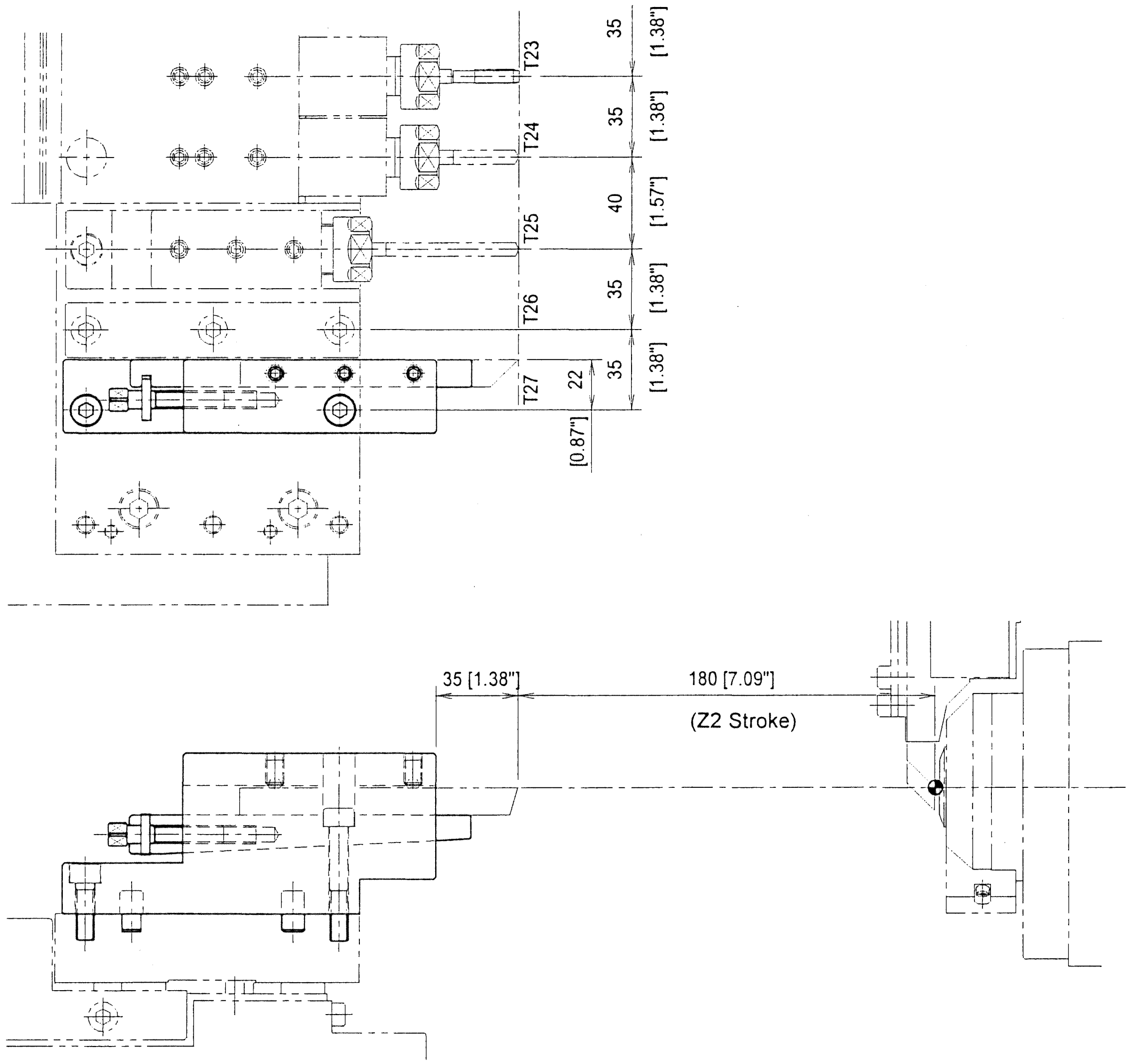


This tool holder is provided for outer diameter cutting. It is adjustable by the hight control.

NOTE ·The depth of cut shall be within 0.1mm because of heavy duty cutting cannot be done with this holder.

Tool holder name	GTA112
Usage	All types of tools
Tool size	12 x 12 x 120mm
Feature	It is adjustable to the stroke 0.0625mm/one rotation (Max±0.25mm)

GTA212 Tool Holder(Type III for opposite tool post)

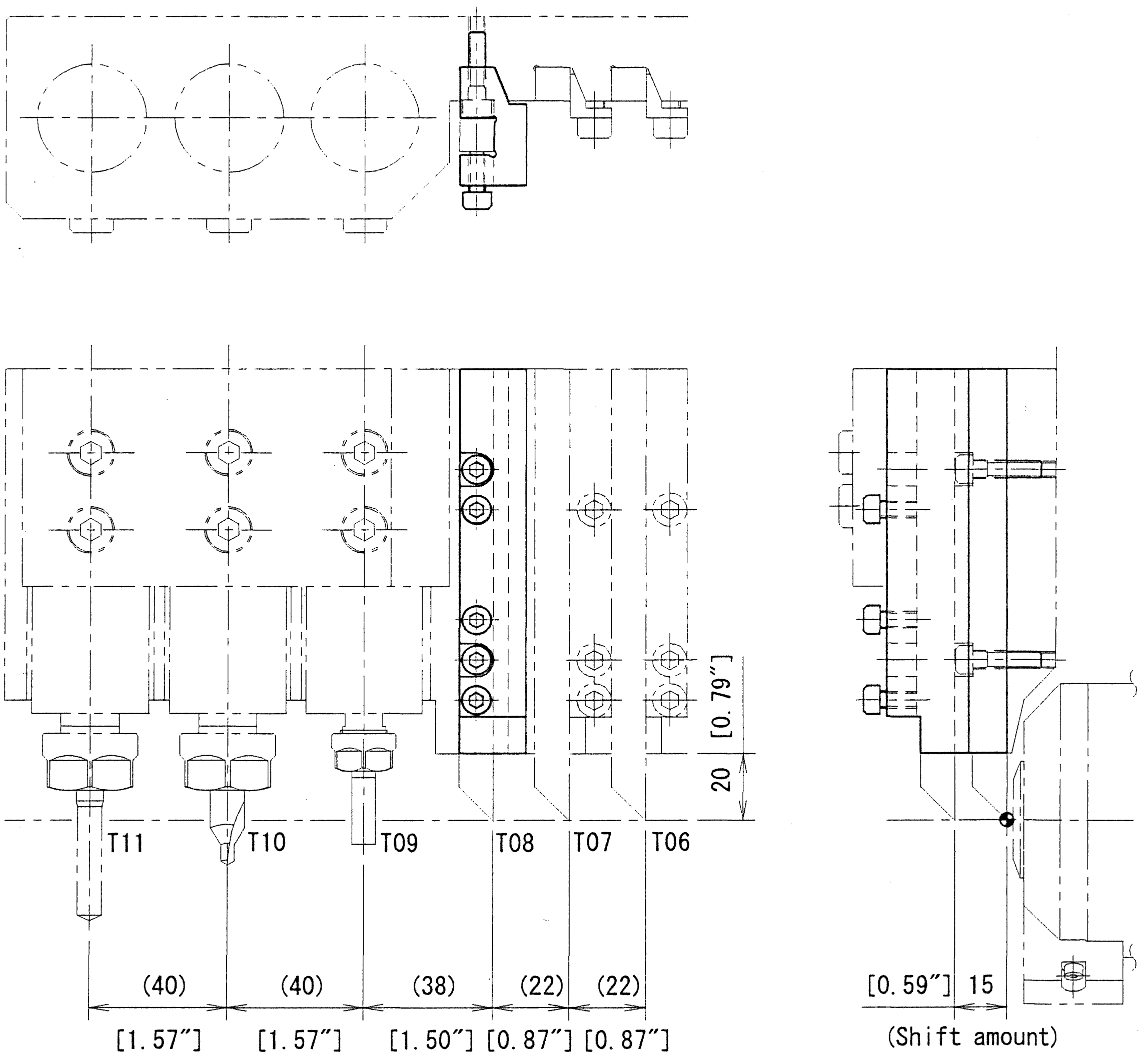


This tool holder is provided for outer diameter cutting. It is adjustable by the hight control.

NOTE · The depth of cut shall be within 0.1mm because of heavy duty cutting cannot be done with this holder.

Tool holder name	GTA212
Usage	All types of tools
Tool size	12 x 12 x 120mm
Feature	It is adjustable to the stroke 0.0625mm/one rotation (Max±0.25mm)

GTF3310 Shift Tool Holder (Adapter type)

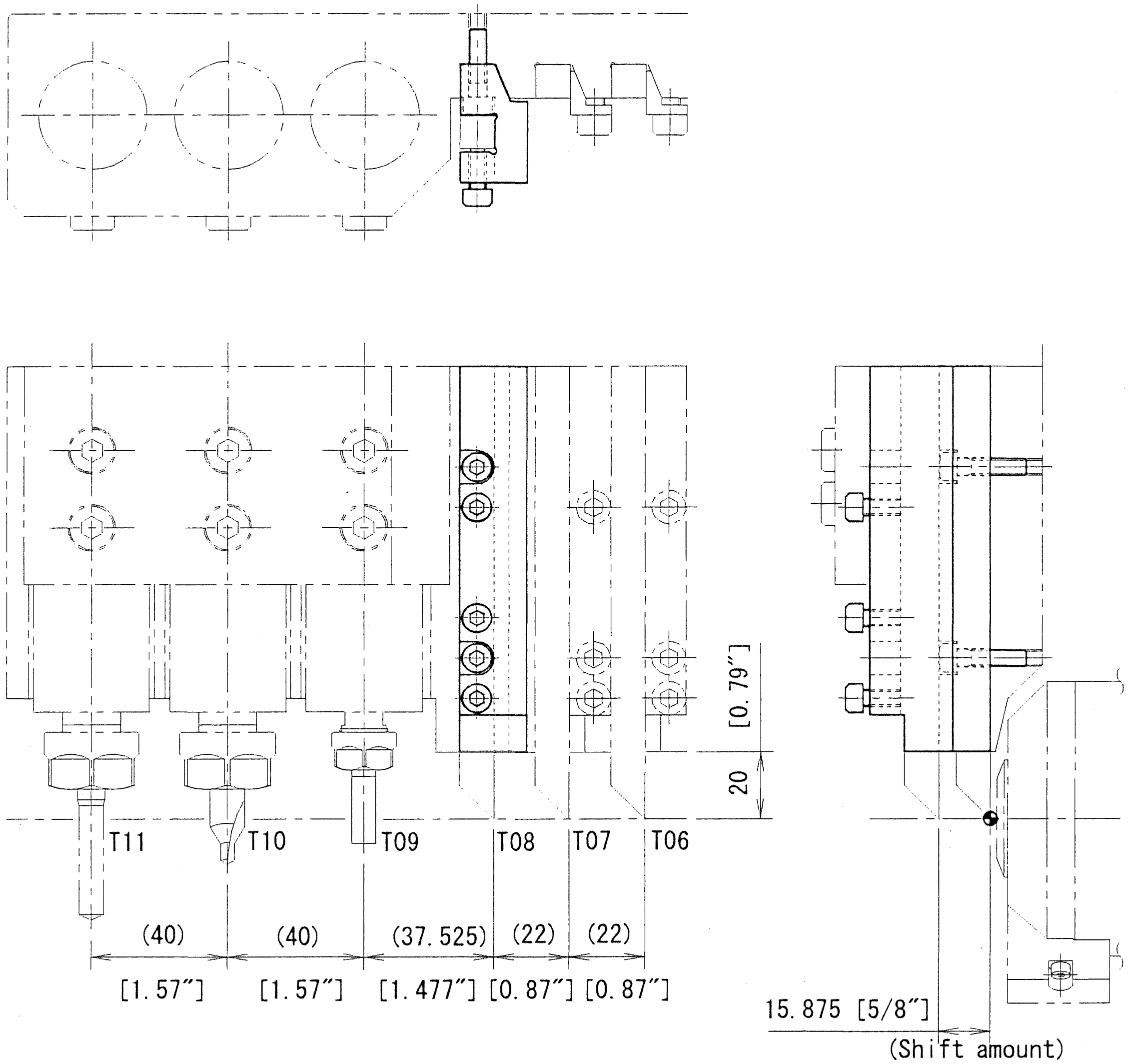


GTF3310 is provided for the turning tool. The tool can be mounted with 15mm [0.59"] longitudinal shift from standard fitting point. This tool holder can be mounted at the turning tool station positions except T01 on the vertical holder-GTF3110,GTF3210 and GTF3410.

NOTE · Take care to the interference between the tool holder and the back spindle, the tools mounting on the opposite tool post.

Tool holder name	GTF3310
Usage	Threading etc.
Tool size	10 x 10 x 120mm(Shifted by 15mm [0.59"])

GTF3310L Shift Tool Holder (Adapter type)

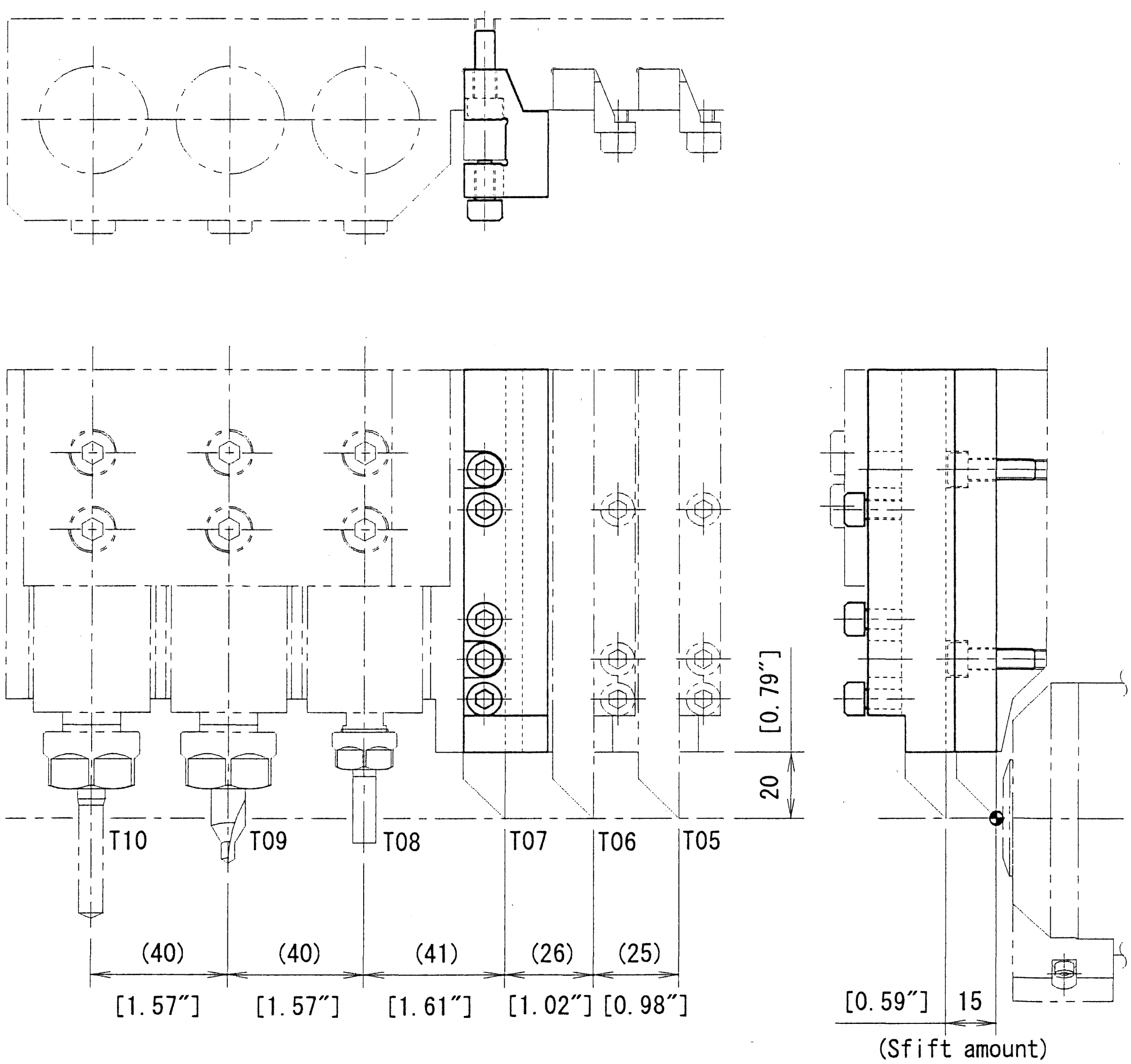


GTF3310L is provided for the turning tool. The tool can be mounted with 15.875mm [5/8"] longitudinal shift from standard fitting point. This tool holder can be mounted at the turning tool station positions except T01 on the vertical holder-GTF3110L,GTF3210L and GTF3410L.

NOTE · Take care to the interference between the tool holder and the back spindle, the tools mounting on the opposite tool post.

Tool holder name	GTF3310L
Usage	Threading etc.
Tool size	3/8" x 3/8" x 4-3/4"(Shifted by 5/8")

GTF3312 Shift Tool Holder (Adapter type)

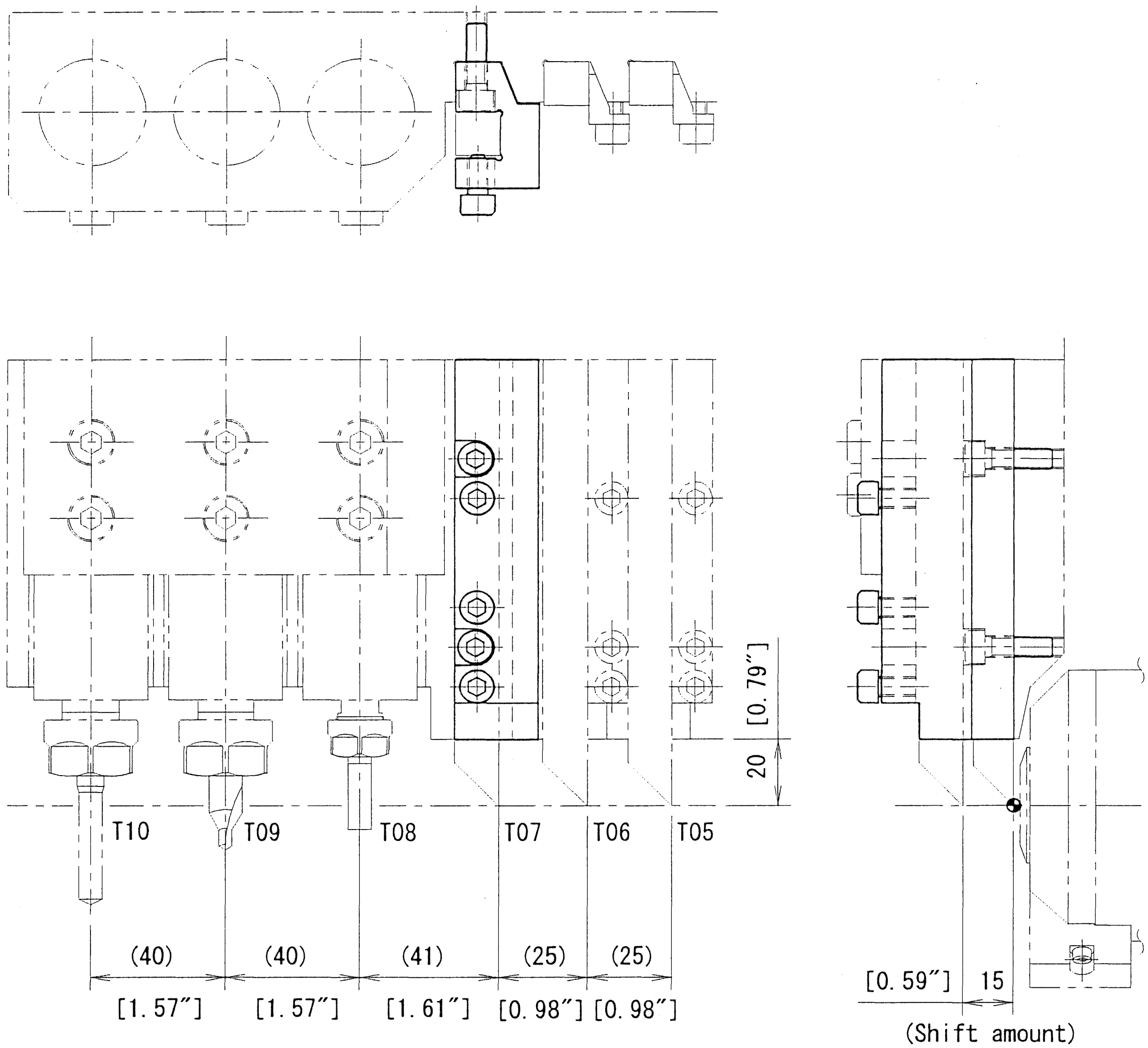


GTF3312 is provided for the turning tool. The tool can be mounted with 15mm [0.59"] longitudinal shift from standard fitting point. This tool holder can be mounted at the turning tool station positions except T01 on the vertical holder-GTF3112,GTF3212 and GTF3412.

NOTE · Take care to the interference between the tool holder and the back spindle, the tools mounting on the opposite tool post.

Tool holder name	GTF3312
Usage	Threading etc.
Tool size	12 x 12 x 120mm(Shifted by 15mm [0.59"])

GTF3313 Shift Tool Holder (Adapter type)

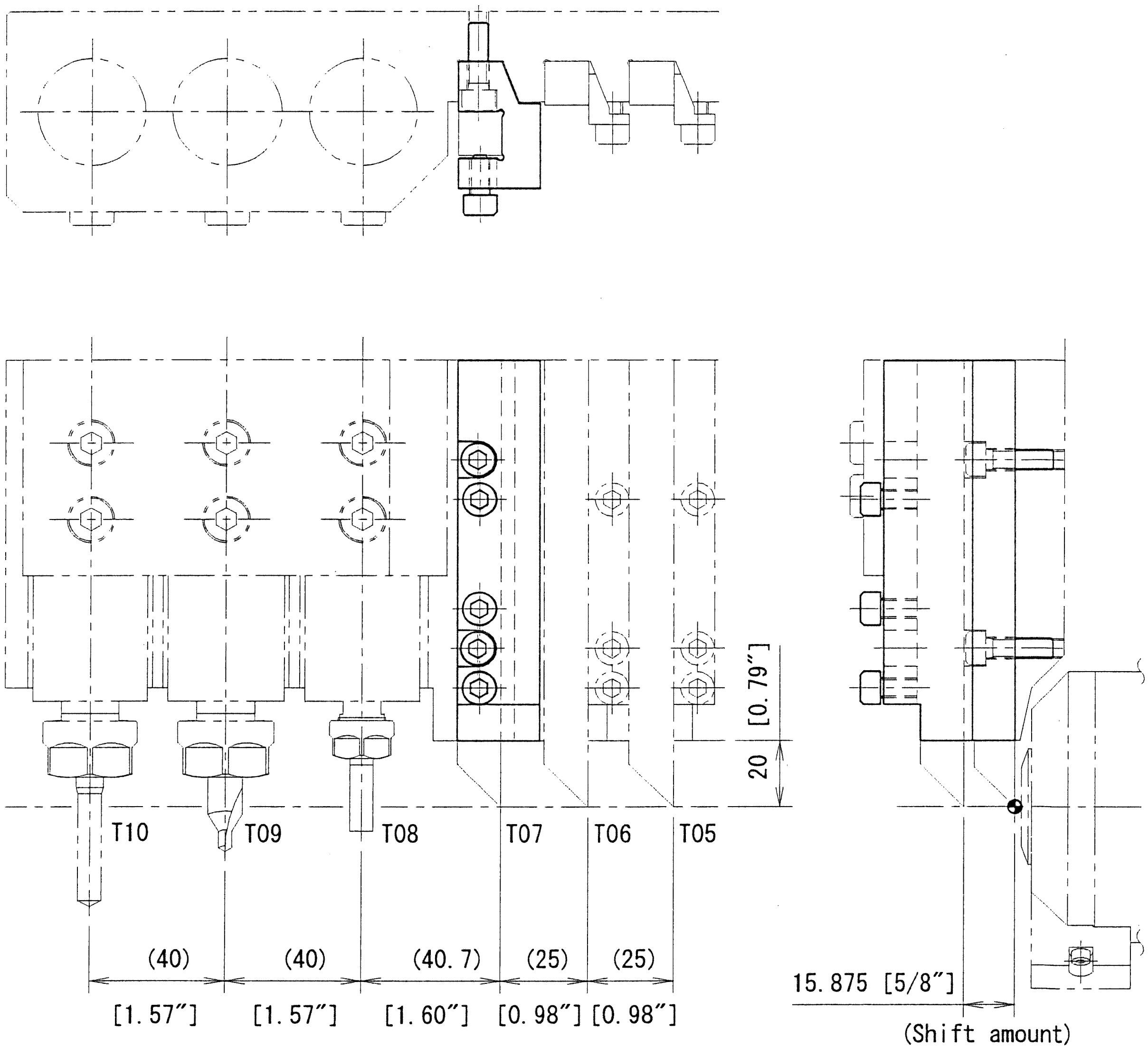


GTF3313 is provided for the turning tool. The tool can be mounted with 15mm [0.59"] longitudinal shift from standard fitting point. This tool holder can be mounted at the turning tool station positions except T01 on the vertical holder-GTF3113,GTF3213 and GTF3413.

NOTE · Take care to the interference between the tool holder and the back spindle, the tools mounting on the opposite tool post.

Tool holder name	GTF3313
Usage	Threading etc.
Tool size	13 x 13 x 120mm(Shifted by 15mm [0.59"])

GTF3313L Shift Tool Holder (Adapter type)

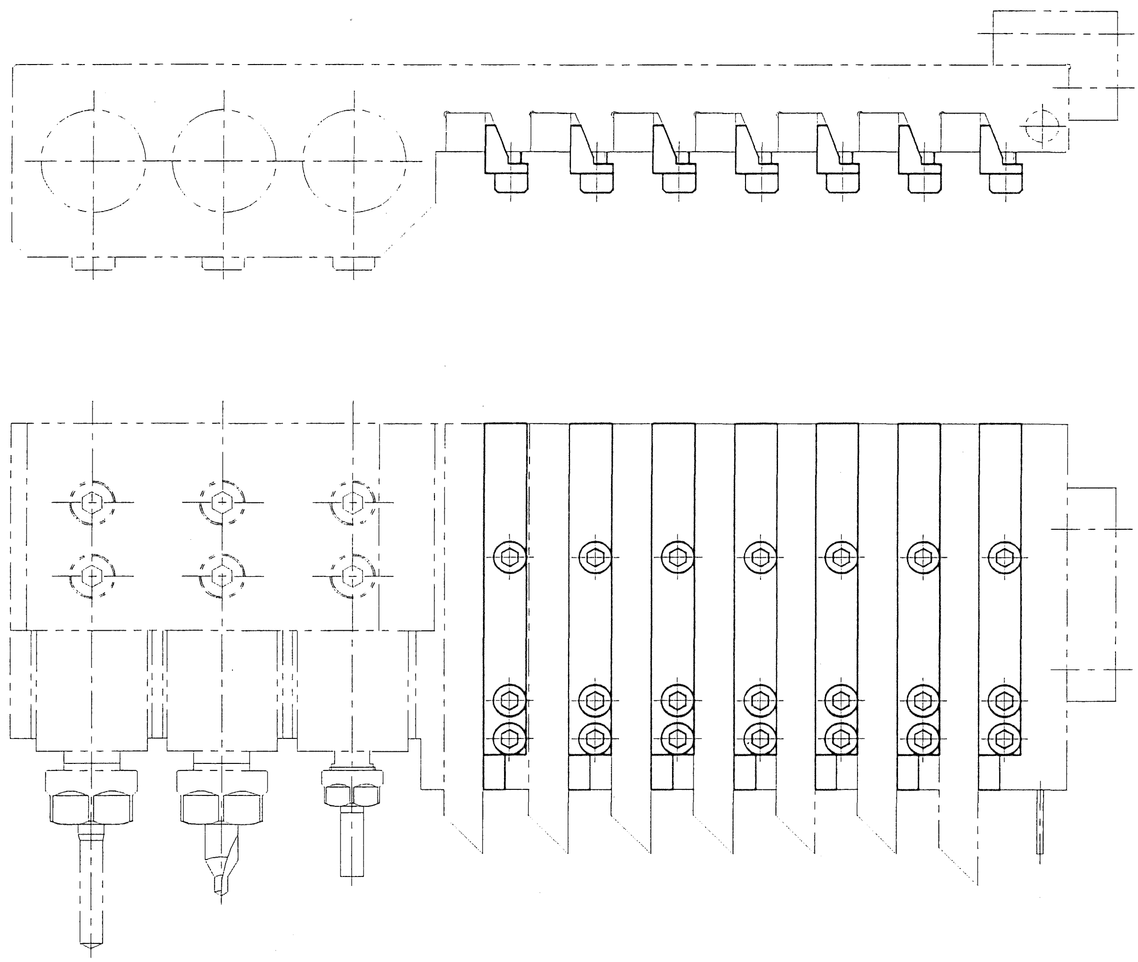


GTF3313L is provided for the turning tool. The tool can be mounted with 15.875mm [5/8"] longitudinal shift from standard fitting point. This tool holder can be mounted at the turning tool station positions except T01 on the vertical holder-GTF3113,GTF3213 and GTF3413.

NOTE · Take care to the interference between the tool holder and the back spindle, the tools mounting on the opposite tool post.

Tool holder name	GTF3313L
Usage	Threading etc.
Tool size	1/2" x 1/2" x 4-3/4"(Shifted by 5/8")

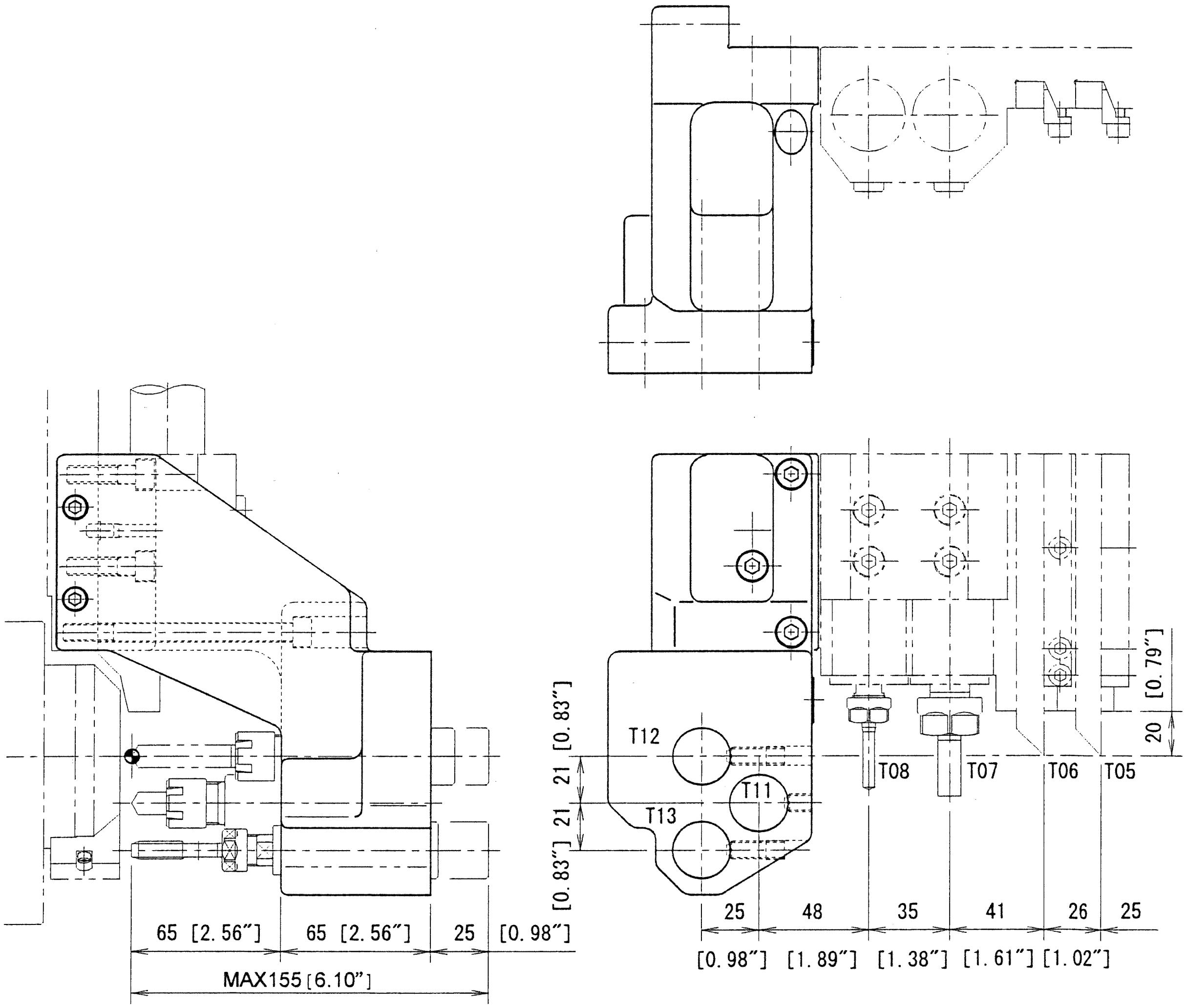
GTJ212 Wedge for 12 tool



This wedge mounted on the □13 tool holder accept □12 tool.
The tooling is helpful to use both □12 tool and □13 tool .

Tool name	GTJ212
Usage	GTF3113,GTF3213,GTF3413
Tool size	12 x 12 x 120mm

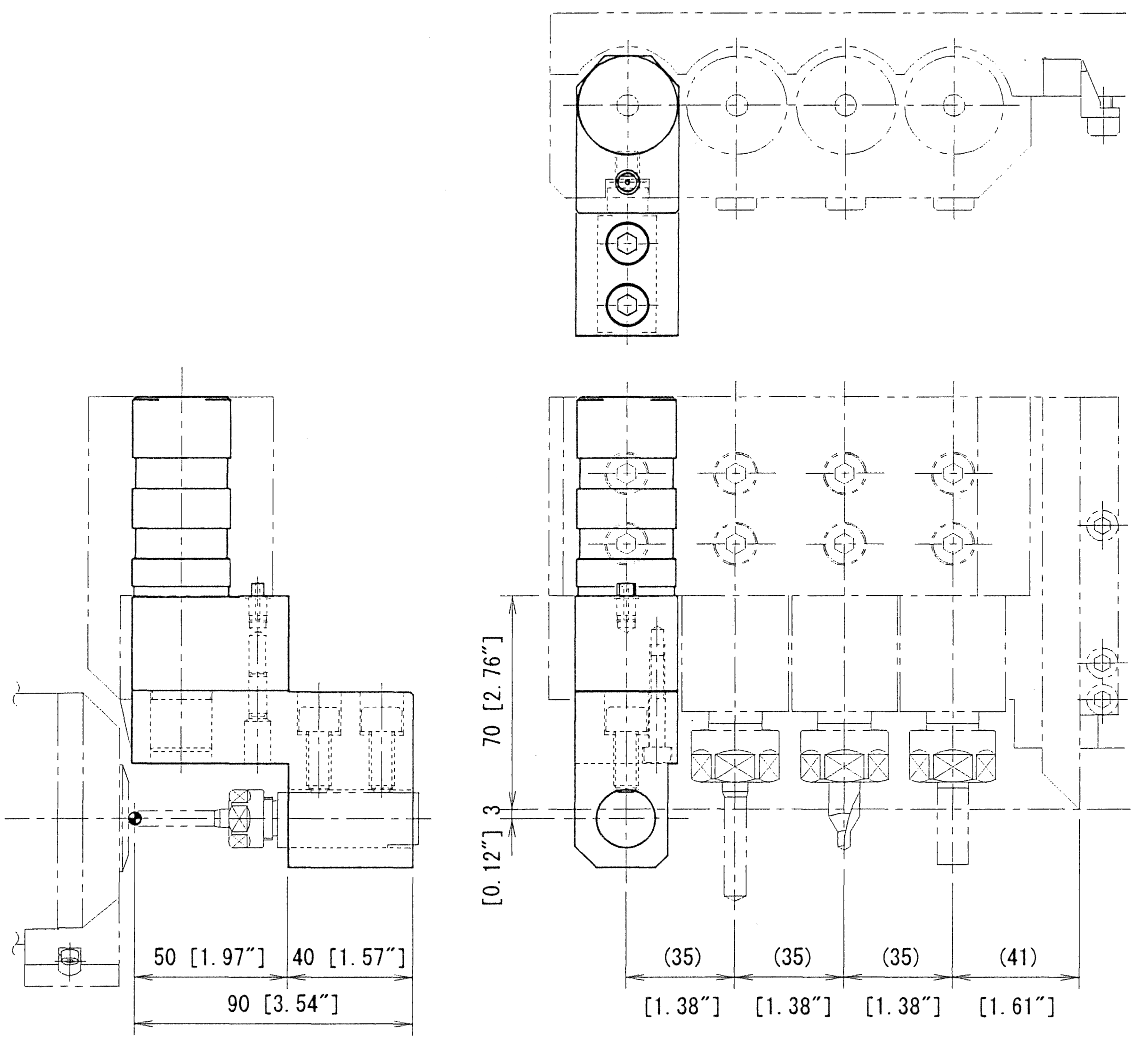
GDF306 3-tool Sleeve Holder (£25.4)



GDF306 is provided for drill work by maximum 3 arbitrary sleeves attached.
It is used combined with the tool holder GTF3410,GTF3410L,GTF3412 and GTF3413.

Tool holder name	GDF306
Usage	ADS707,ADS710,ATS307
Mounting hole diameter	£25.4mm [£1"]

GDF506 Face Drilling Sleeve Holder (1-tool)

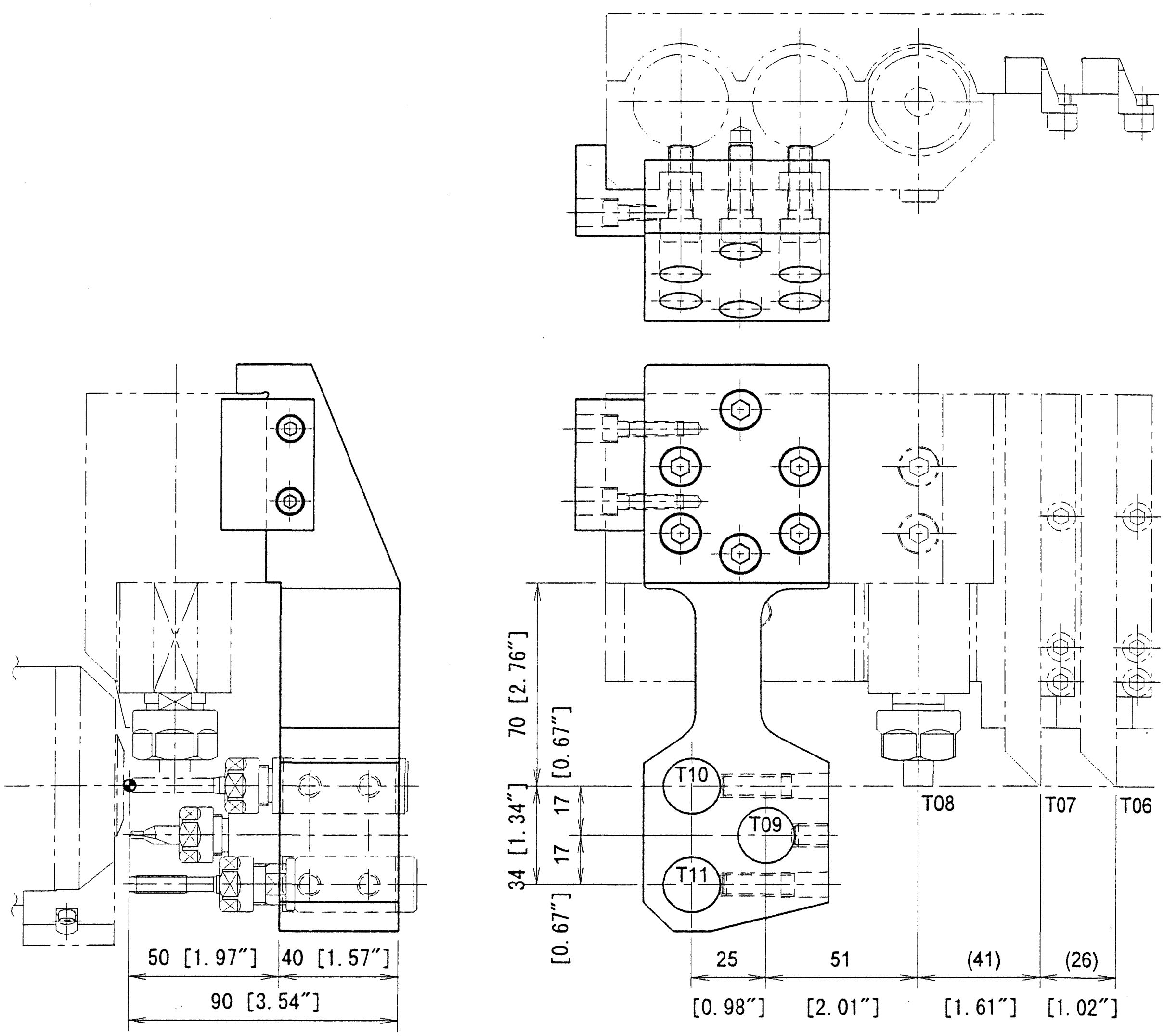


GDF506 is used to perform drilling on the face, by installing on the vertical holder.

NOTE
- When cross milling operation using the rotary tool installed next position of this holder, take care to the interference between the workpiece and this holder.
- Take care to the interference between the back spindle including the opposite sleeves, when mounting this holder.

Tool holder name	GDF506
Usage	BDS507,BDS607,VDS506,HDS5406,HDS5506 BNS407,BNS507,VNS406,VNS506
Mounting hole diameter	£19.05mm [£3/4"]

GDF606 Face Drilling Sleeve Holder (3-tool)

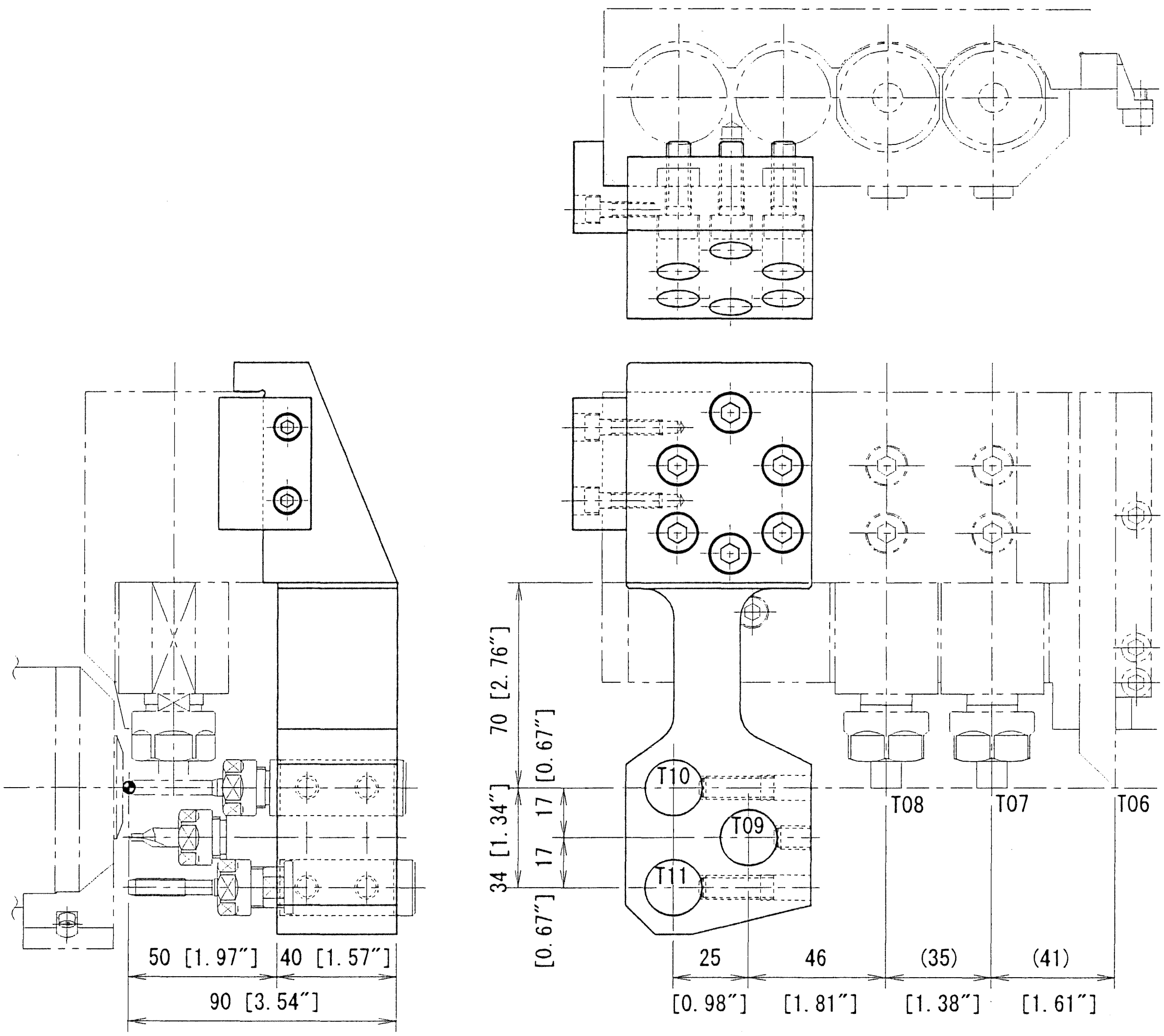


GDF606 is used to perform drilling on the face by installing the vertical holder for 3 rotary tools. It can be mounted up to 3 sleeves.

NOTE · Take care to the interference between the back spindle including the opposite sleeves, when mounting this holder.

Tool holder name	GDF606
Usage	BDS507,BDS607,VDS506,HDS5406,HDS5506 BNS407,BNS507,VNS406,VNS506
Mounting hole diameter	ƒ19.05mm [ƒ3/4"]

GDF607 Face Drilling Sleeve Holder (3-tool)

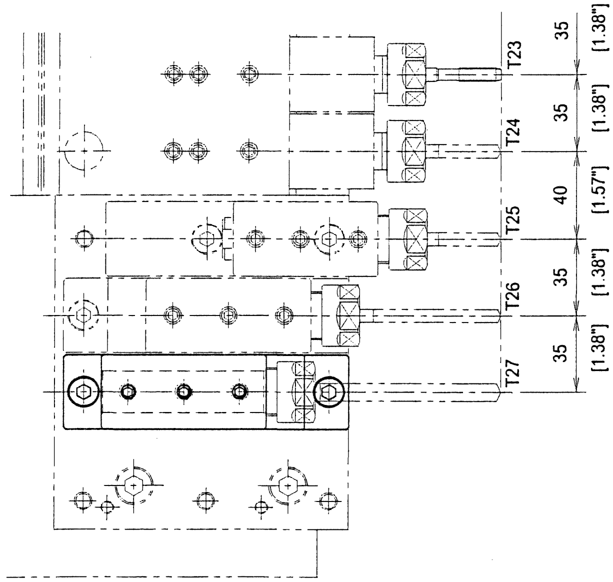


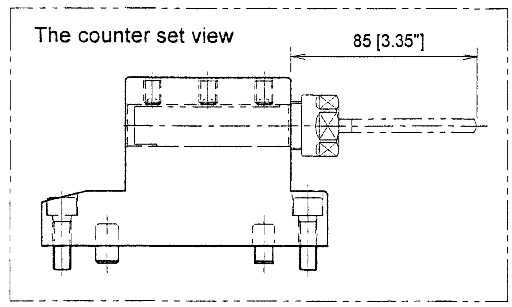
GDF606 is used to perform drilling on the face by installing the vertical holder for 4 rotary tools. It can be mounted up to 3 sleeves.

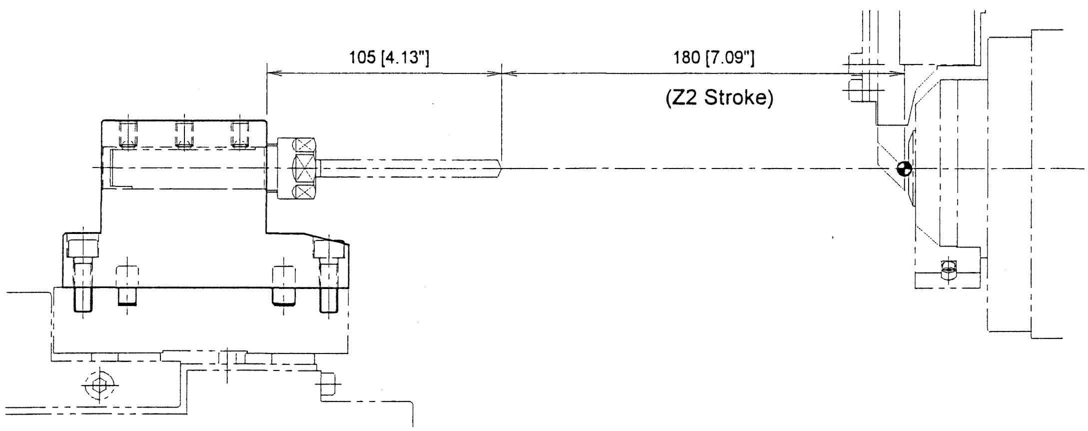
NOTE · Take care to the interference between the back spindle including the opposite sleeves, when mounting this holder.

Tool holder name	GDF607
Usage	BDS507,BDS607,VDS506,HDS5406,HDS5506 BNS407,BNS507,VNS406,VNS506
Mounting hole diameter	£19.05mm [£3/4”]

GDF102 Fixed Type Sleeve Holder (£19.05)



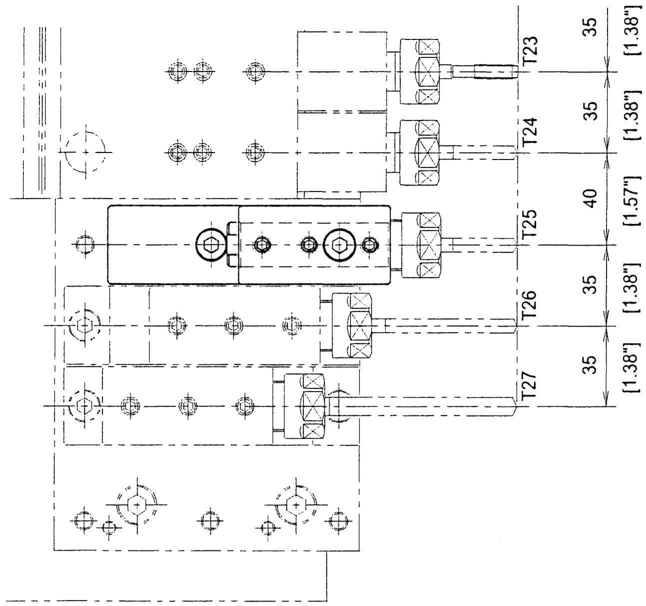


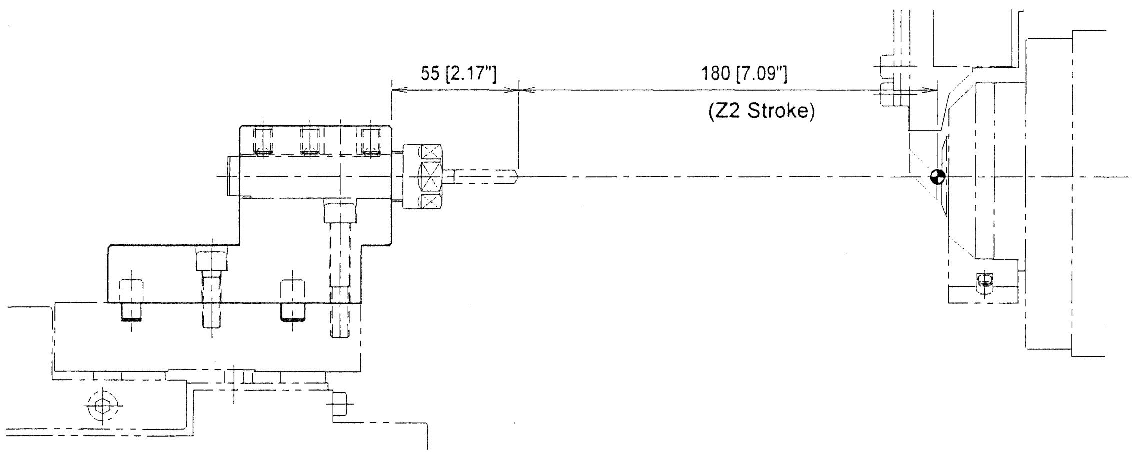


This holder is provided for drill work by arbittary sleeves attached. It can be set on the 3-spindle tool post (T25-T27) for type III machine.
The protrusion amount of the sleeve can be changed by setting the holder frontside back.

Tool holder name	GDF102
Usage	VDS506,VDS110,LDS107,LDS110,VNS406 VNS506
Mounting hole diameter	£19.05mm [£3/4"]

GDF103 Fixed Type Sleeve Holder (£19.05)

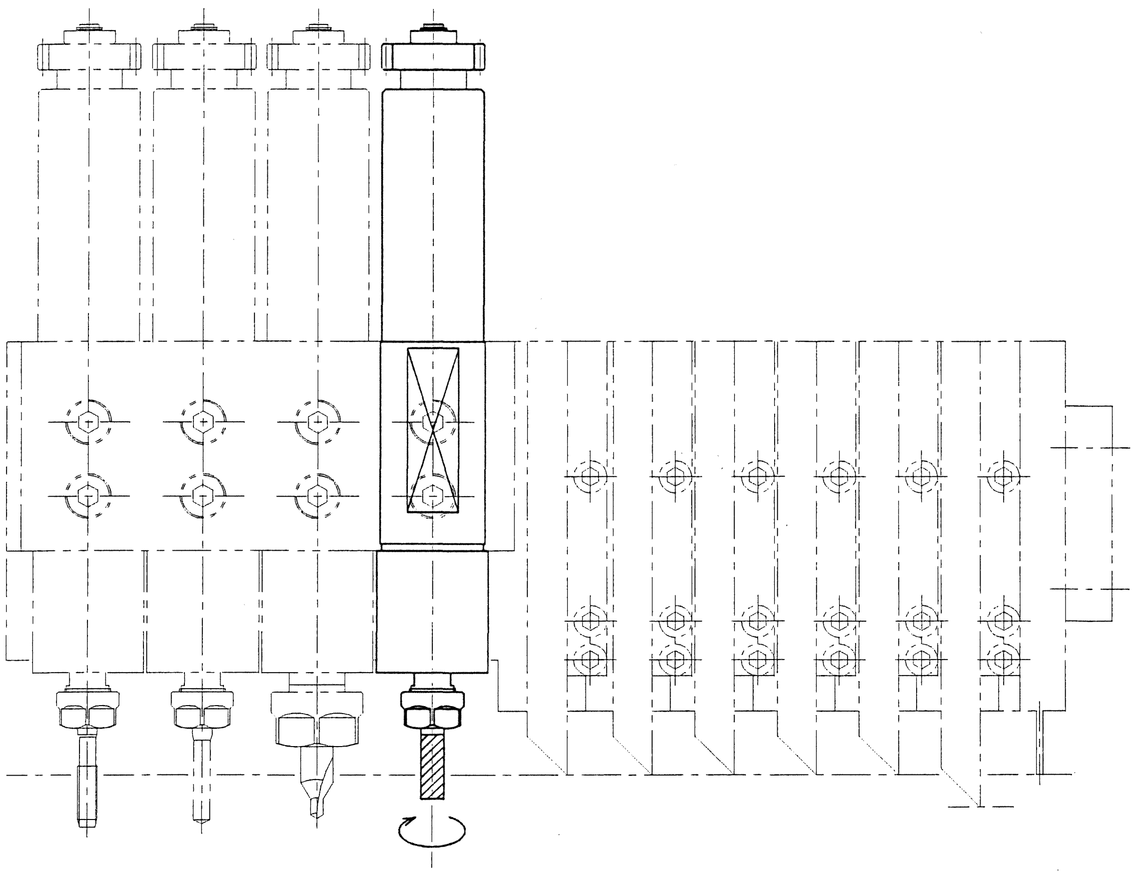




This holder is provided for drill work by arbittary sleeves attached. It can be set on the 3-spindle tool post (T25-T27)for type III machine.

Tool holder name	GDF103
Usage	BDS507,BDS508,BDS607,BDS610,VDS506 VDS110,LDS107,LDS110,BNS407,BNS507 VNS406,VNS506
Mounting hole diameter	£19.05mm [£3/4"]

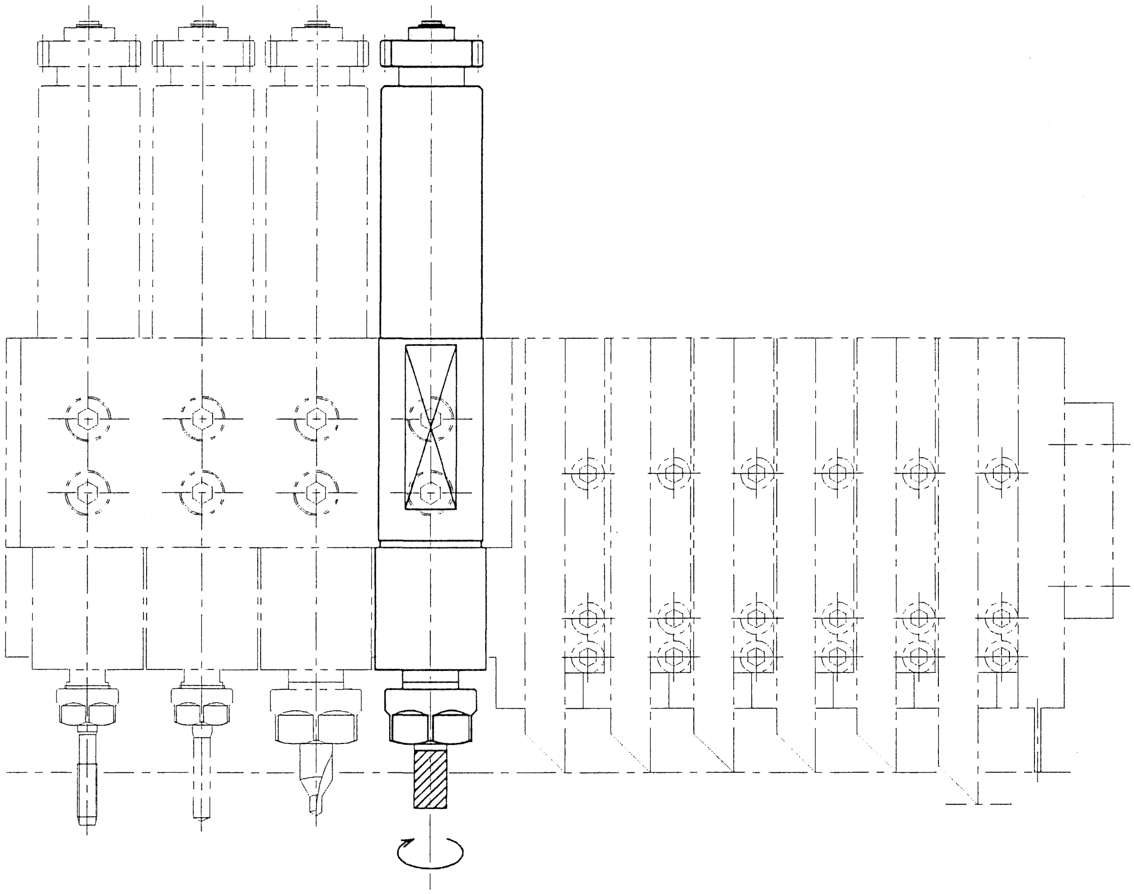
GSC507 Outer Circumference Milling Spindle (Rego Type Chuck)



GSC507 is used to perform drilling on the outer diameter of the workpiece, or to perform key grooving by the endmill etc., after stopping the spindle.

Tool holder name	GSC507
Max. chuck dia.	£7mm [£0.28"]
Spindle speed	200 to 5000r.p.m
Chuck type	ER11 , AR11
Spindle rotation direction	Direction shown by the arrow in the figure above for a forward rotation command

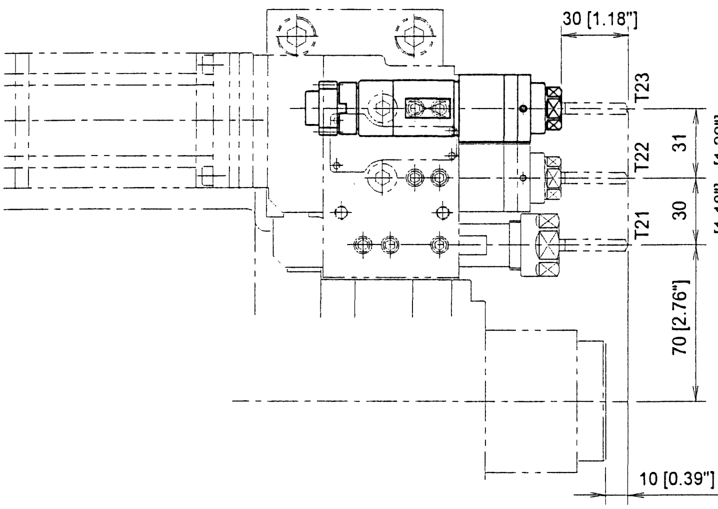
GSC510 Outer Circumference Milling Spindle (Rego Type Chuck)

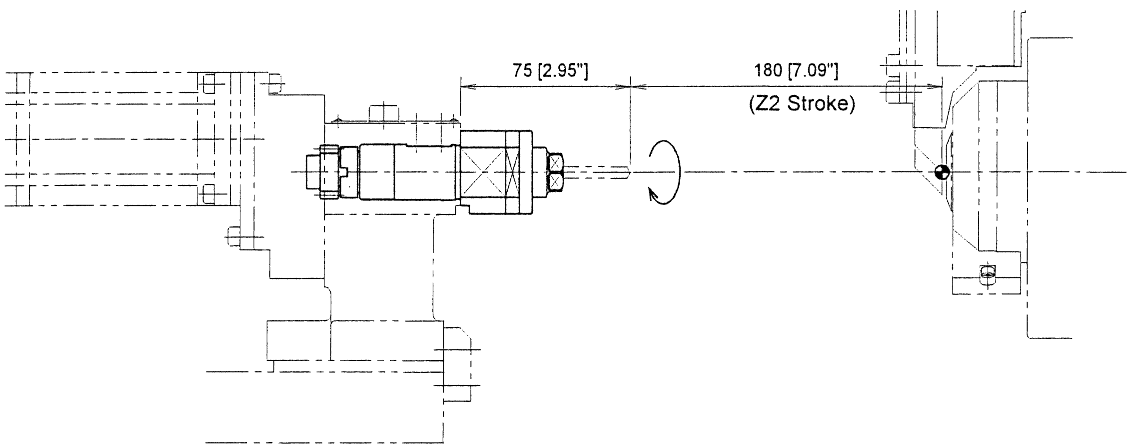


GSC510 is used to perform drilling on the outer diameter of the workpiece, or to perform key grooving by the endmill etc., after stopping the spindle.

Tool holder name	GSC510
Max. chuck dia.	f10mm [f0.39"]
Spindle speed	200 to 5000r.p.m
Chuck type	ER16 , AR16
Spindle rotation direction	Direction shown by the arrow in the figure above for a forward rotation command

GSE307 Face Drilling Spindle (Rego Type Chuck)



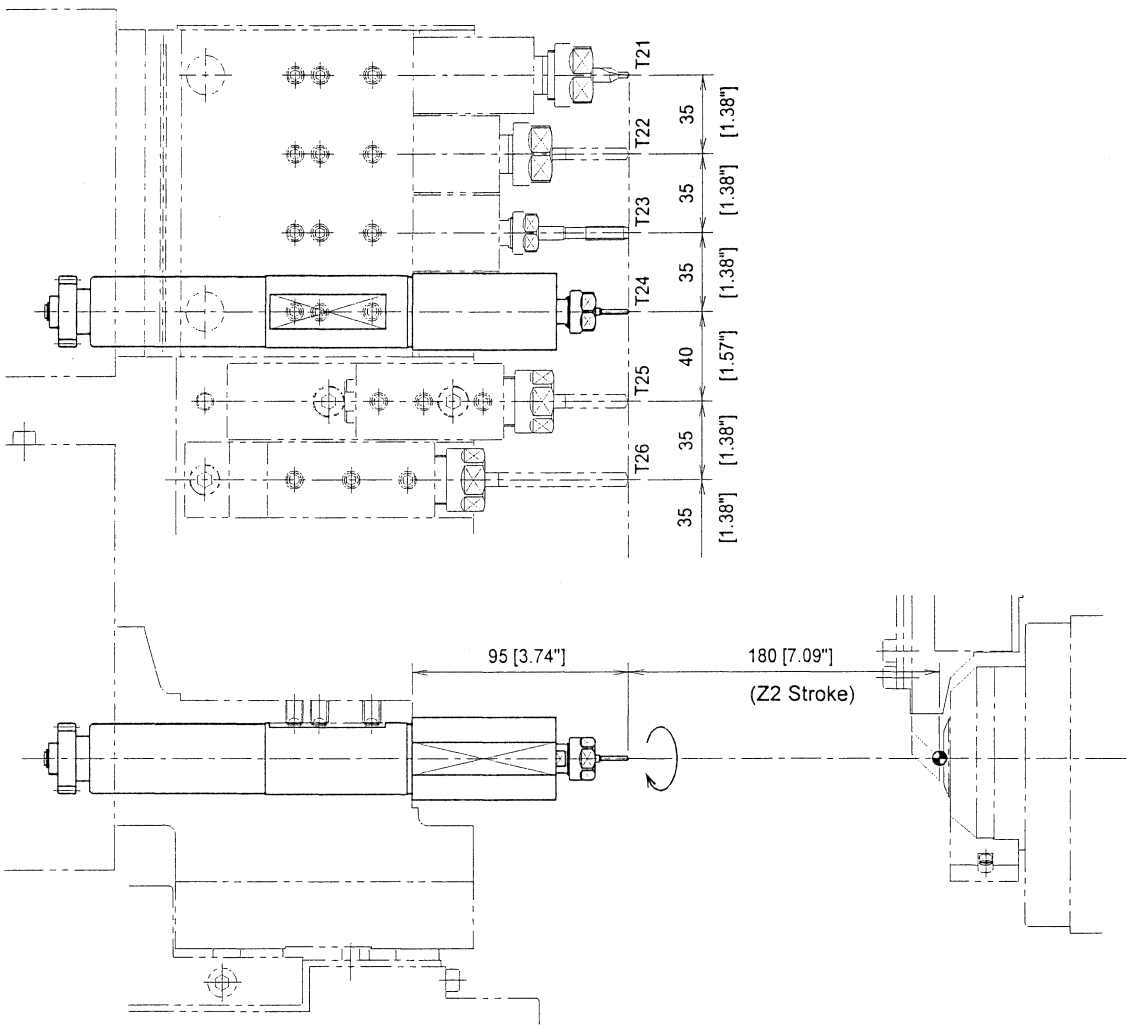


This spindle is provided for the drilling, the tapping and end milling etc. on the end face or outer circumference of the workpiece.

NOTE · This spindle is used only for the front face drilling device U121B and the back face drilling device U151B.

Tool holder name	GSE307
Max. chuck dia.	£7mm [£0.28"]
Spindle speed	200 to 5000r.p.m
Chuck type	ER11 , AR11
Spindle rotation direction	Direction shown by the arrow in the figure above for a forward rotation command

GSE407 Face Drilling Spindle (Rego Type Chuck)

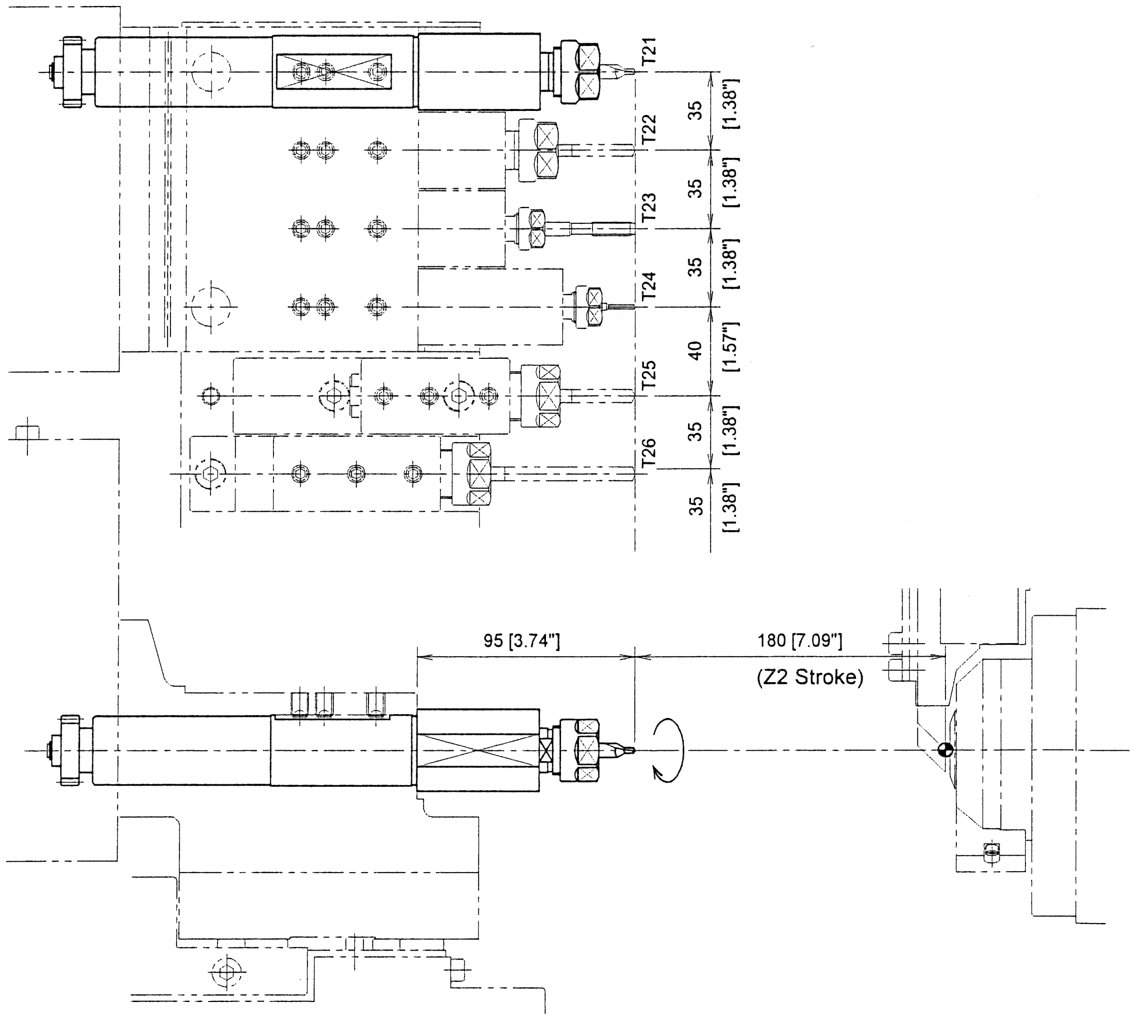


This spindle is provided for the drilling, the tapping and end milling etc. on the end face or outer circumference of the workpiece.

NOTE · This spindle is used only for the front face drilling device U131B , it can not be mounted on the vertical tool holder.

Spindle name	GSE407
Max. chuck dia.	f7mm [f0.28"]
Spindle speed	200 to 5000r.p.m
Chuck type	ER11 , AR11
Spindle rotation direction	Direction shown by the arrow in the figure above for a forward rotation command

GSE410 Face Drilling Spindle (Rego Type Chuck)

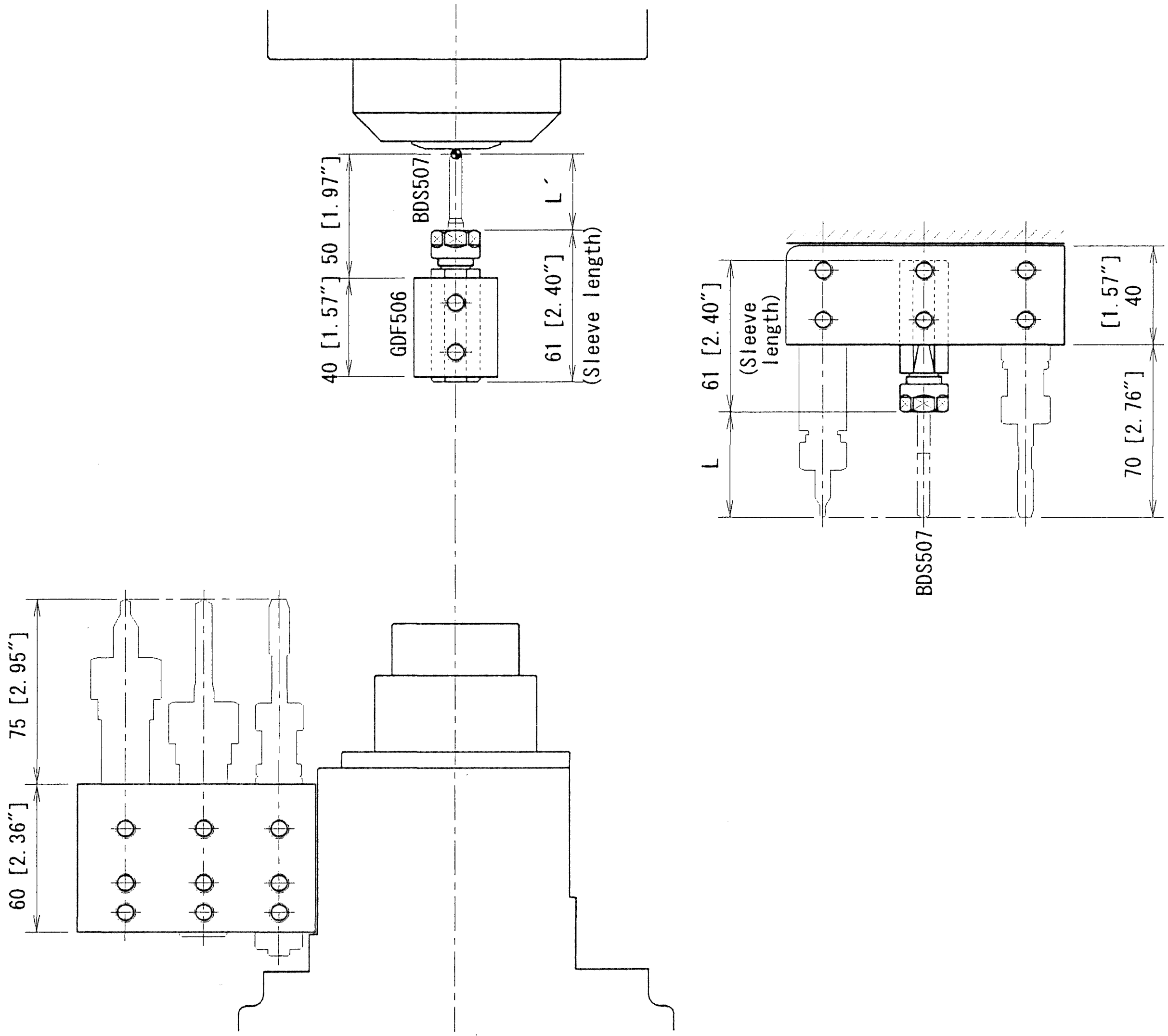


This spindle is provided for the drilling, the tapping and end milling etc. on the end face or outer circumference of the workpiece.

NOTE · This spindle is used only for the front face drilling device U131B , it can not be mounted on the vertical tool holder.

Spindle name	GSE410
Max. chuck dia.	£10mm [£0.39"]
Spindle speed	200 to 5000r.p.m
Chuck type	ER16 , AR16
Spindle rotation direction	Direction shown by the arrow in the figure above for a forward rotation command

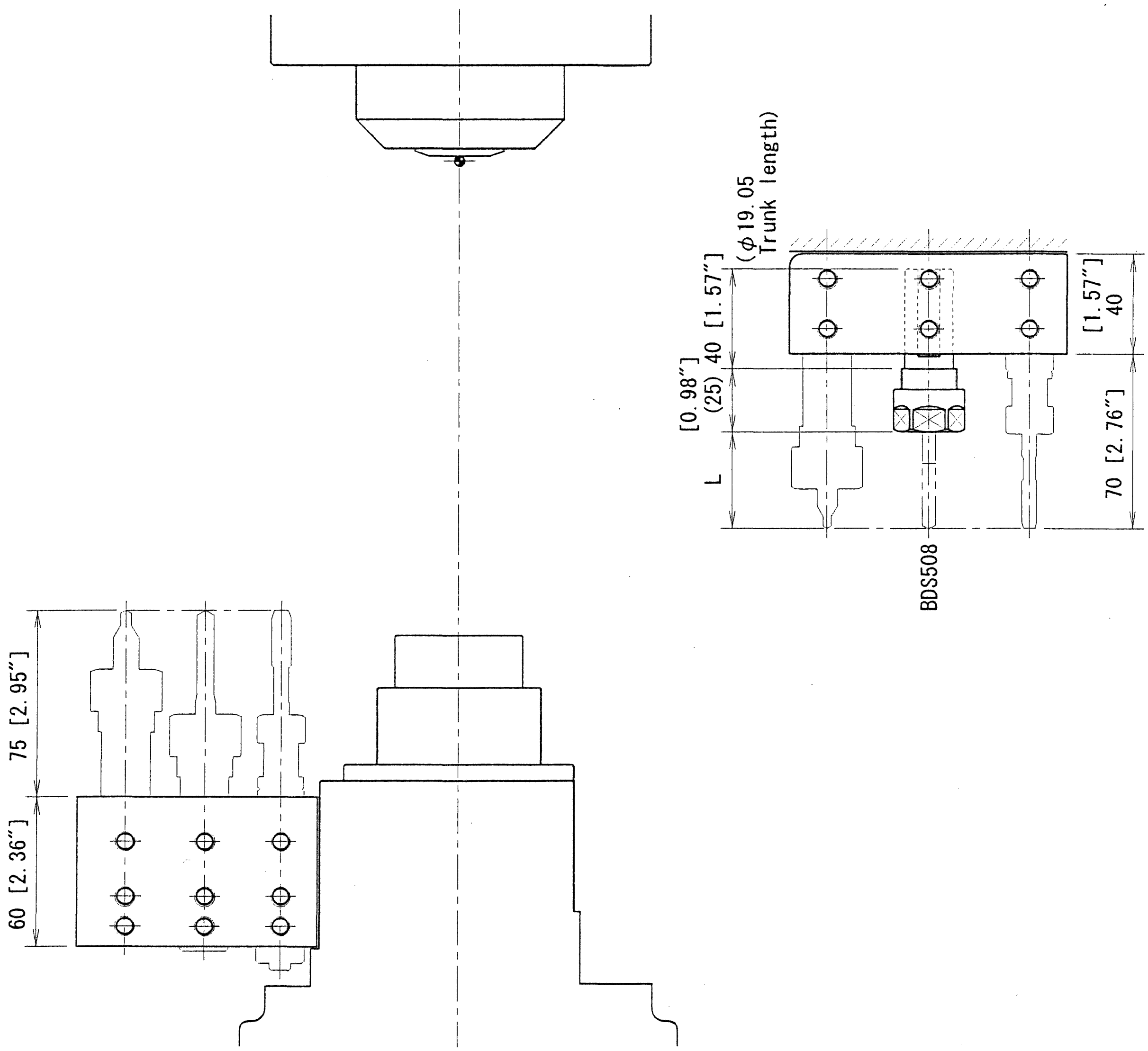
BDS507 Boring and Drill Sleeve (~f7)



BDS507 chucks the drill, etc. with the straight shank used for the front and back machining by the rego type chuck. L or L' dimension is adjustable by putting sleeve in and out.

Sleeve name	BDS507
Max. chuck dia.	f7mm [f0.28"]
Chuck type	ER11 , AR11
Sleeve outer dia.	f19.05mm [f3/4"]

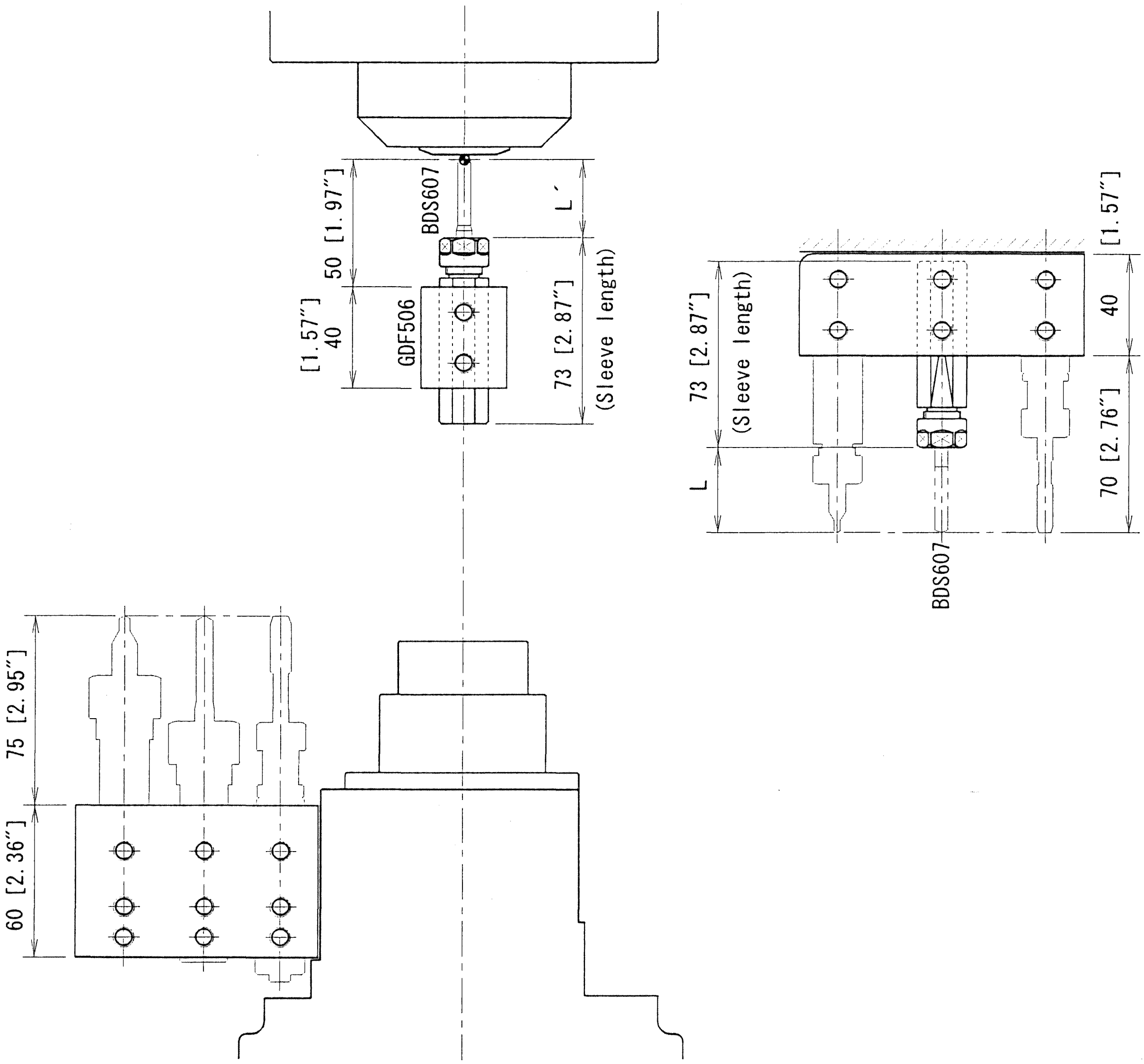
BDS508 Boring and Drill Sleeve (~£10)



BDS508 chucks the drill, etc. with the straight shank used for the back machining by the rego type chuck. L dimension is adjustable by putting sleeve in and out.

Sleeve name	BDS508
Max. chuck dia.	£10mm [£0.39"]
Chuck type	ER16 , AR16
Sleeve outer dia.	£19.05mm [£3/4"]

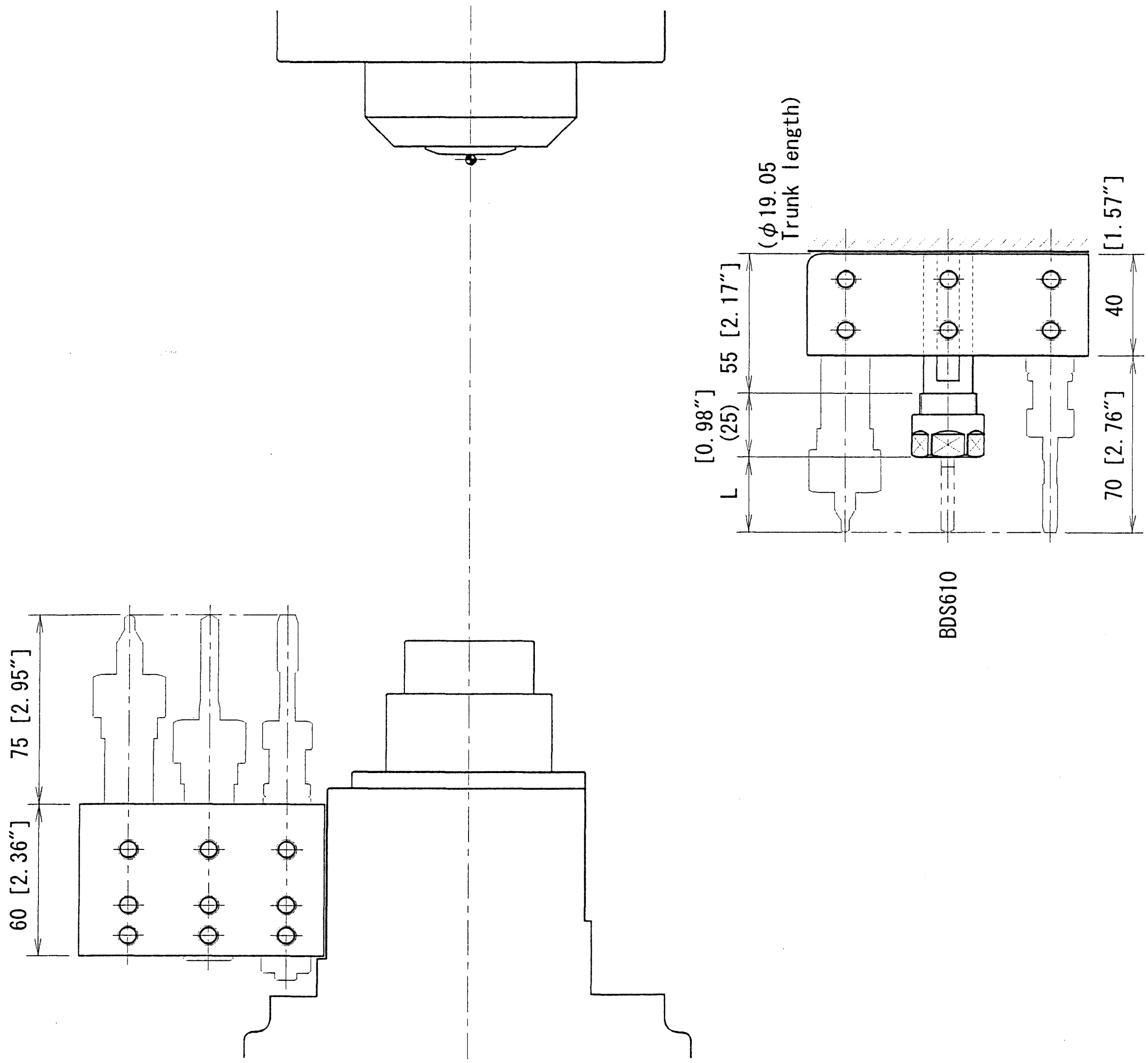
BDS607 Boring and Drill Sleeve (~f7)



BDS607 chucks the drill, etc. with the straight shank used for the front and back machining by the rego type chuck. L or L' dimension is adjustable by putting sleeve in and out.

Sleeve name	BDS607
Max. chuck dia.	f7mm [f0.28"]
Chuck type	ER11 , AR11
Sleeve outer dia.	f19.05mm [f3/4"]

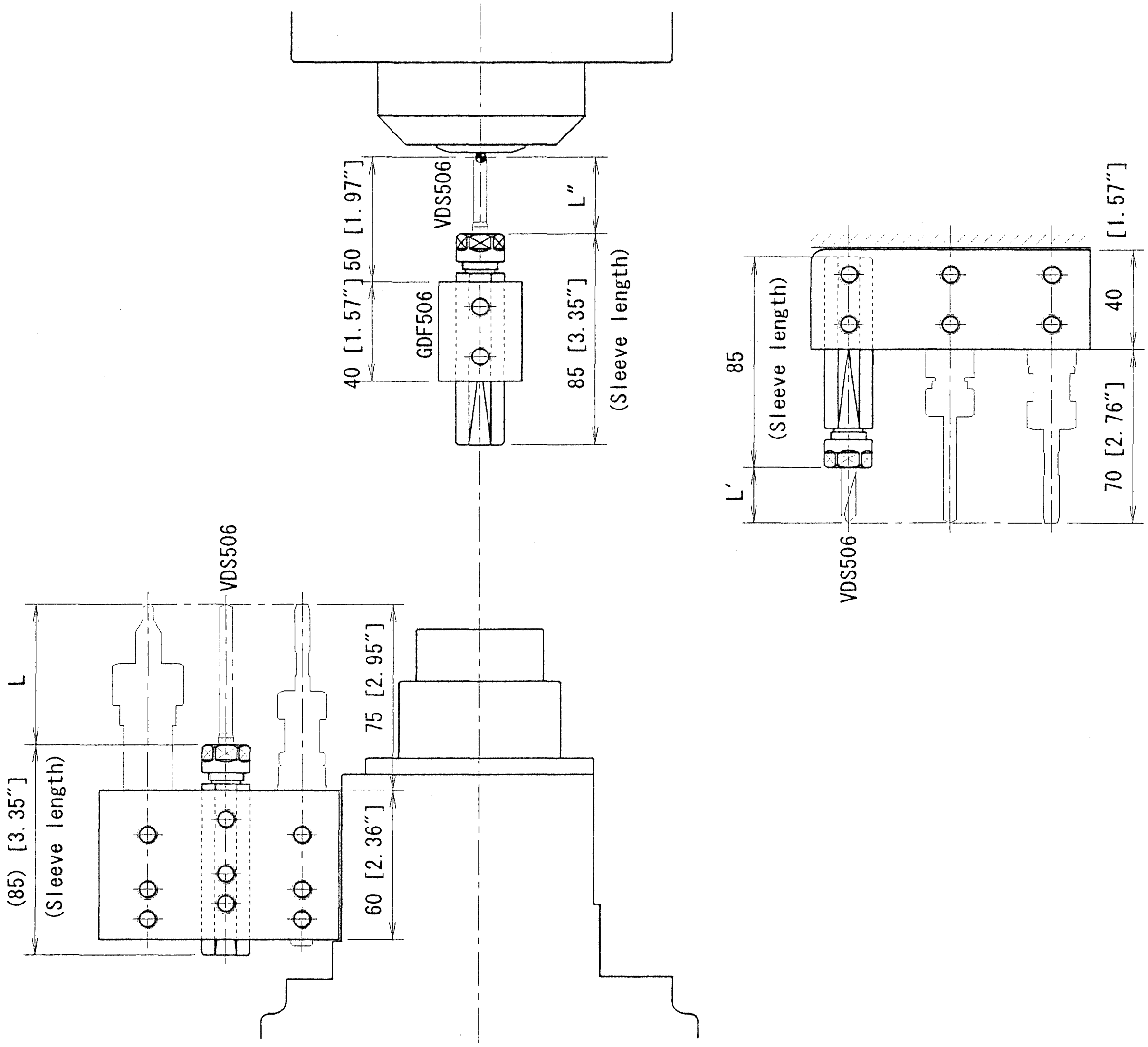
BDS610 Boring and Drill Sleeve (~£10)



BDS610 chucks the drill, etc. with the straight shank used for the back machining by the rego type chuck. L dimension is adjustable by putting sleeve in and out.

Sleeve name	BDS610
Max. chuck dia.	£10mm [£0.39"]
Chuck type	ER16 , AR16
Sleeve outer dia.	£19.05mm [£3/4"]

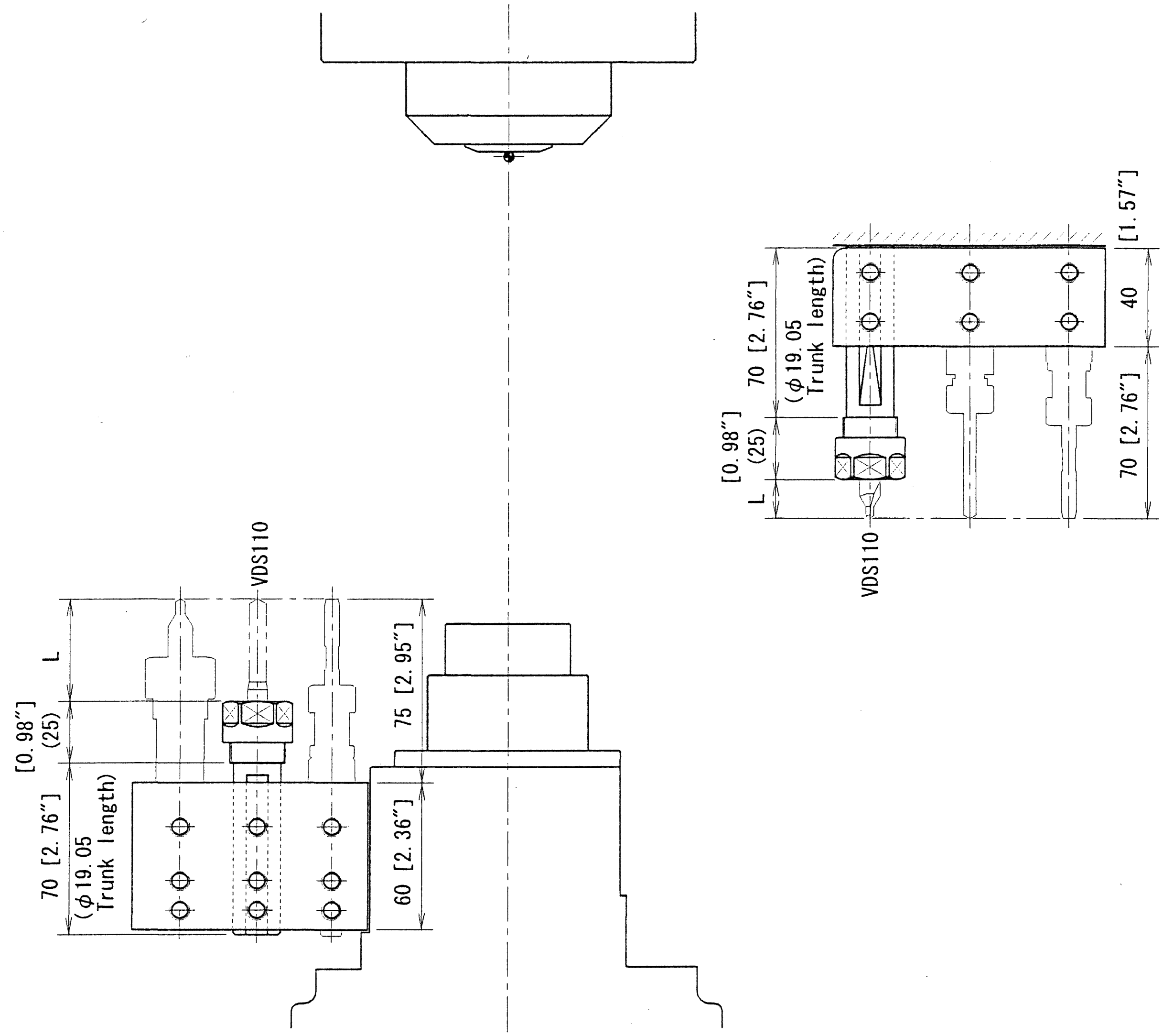
VDS506 Boring and Drill Sleeve (~ƒ7)



VDS506 chucks the drill, etc. with the straight shank used for the front and back machining by the rego type chuck. L and L' or L"dimension is adjustable by putting sleeve in and out.

Sleeve name	VDS506
Max. chuck dia.	ƒ7mm [ƒ0.28"]
Chuck type	ER11 , AR11
Sleeve outer dia.	ƒ19.05mm [ƒ3/4"]

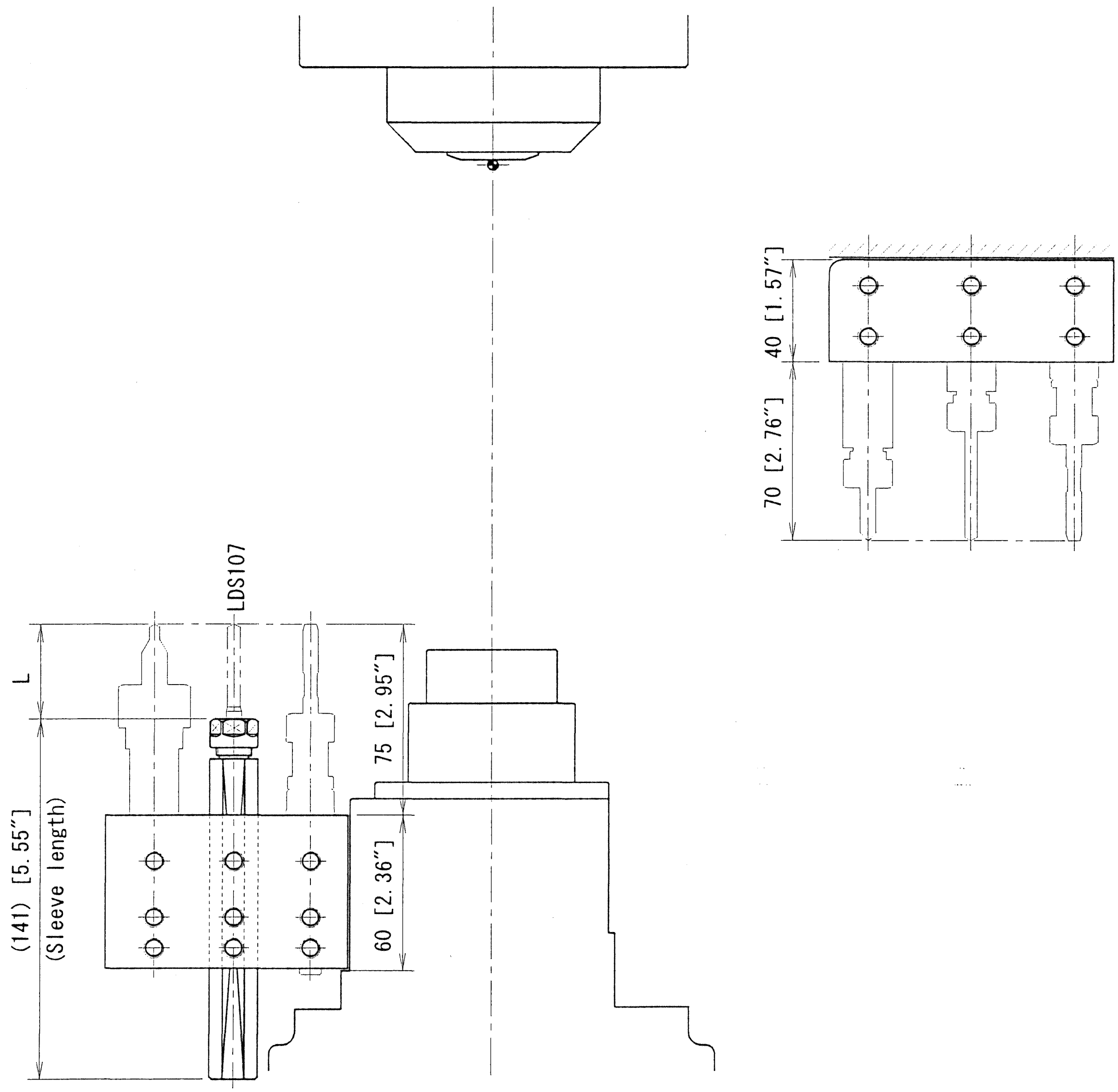
VDS110 Boring and Drill Sleeve (~f10)



VDS110 chucks the drill, etc. with the straight shank used for the front and back machining by the rego type chuck. L or L'dimension is adjustable by putting sleeve in and out.

Sleeve name	VDS110
Max. chuck dia.	f10mm [f0.39"]
Chuck type	ER16 , AR16
Sleeve outer dia.	f19.05mm [f3/4"]

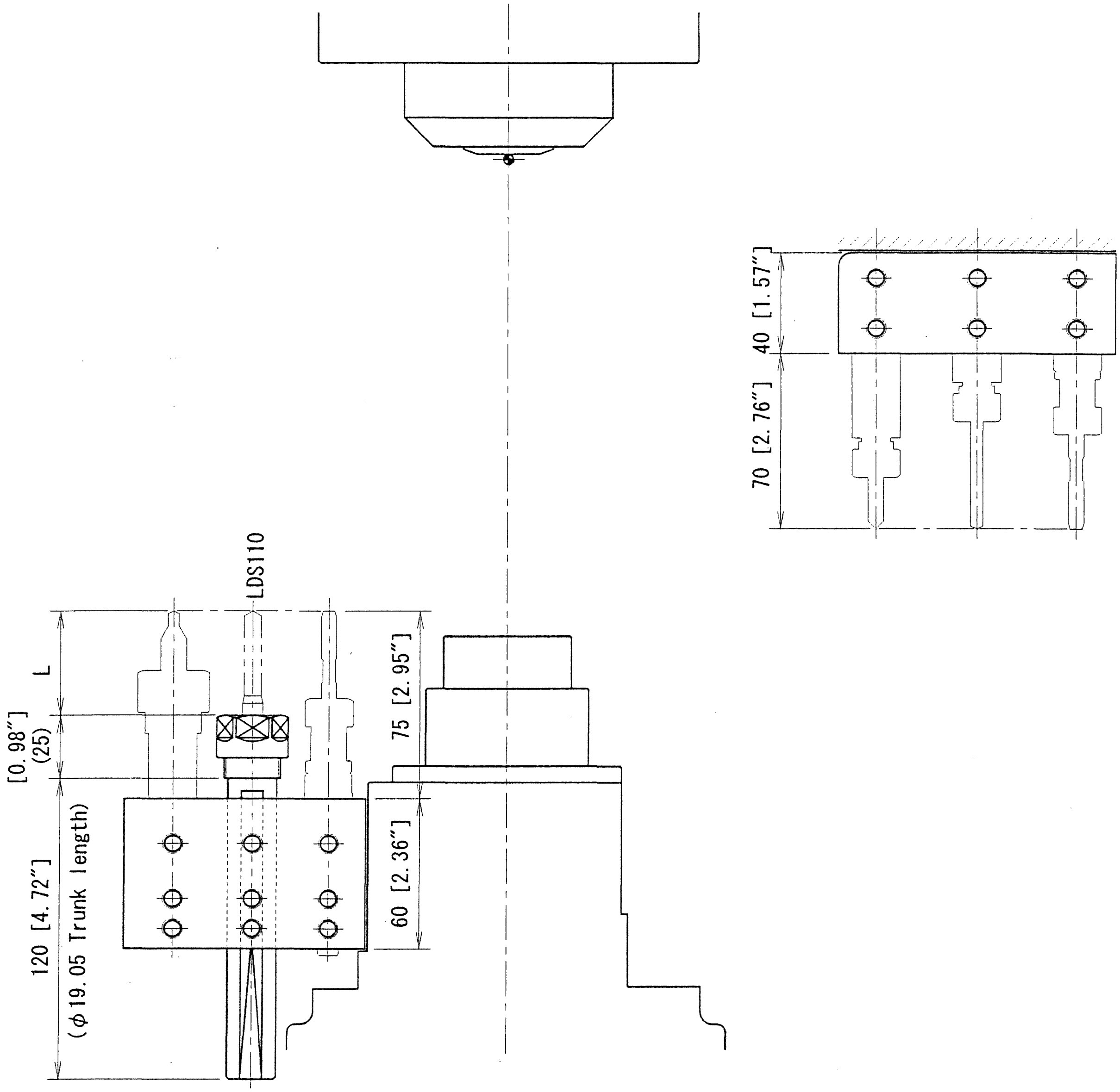
LDS107 Boring and Drill Sleeve (~£7)



LDS107 chucks the drill, etc. with the straight shank used for the front machining by the rego type chuck. L dimension is adjustable by putting sleeve in and out.

Sleeve name	LDS107
Max. chuck dia.	£7mm [£0.28"]
Chuck type	ER11 , AR11
Sleeve outer dia.	£19.05mm [£3/4"]

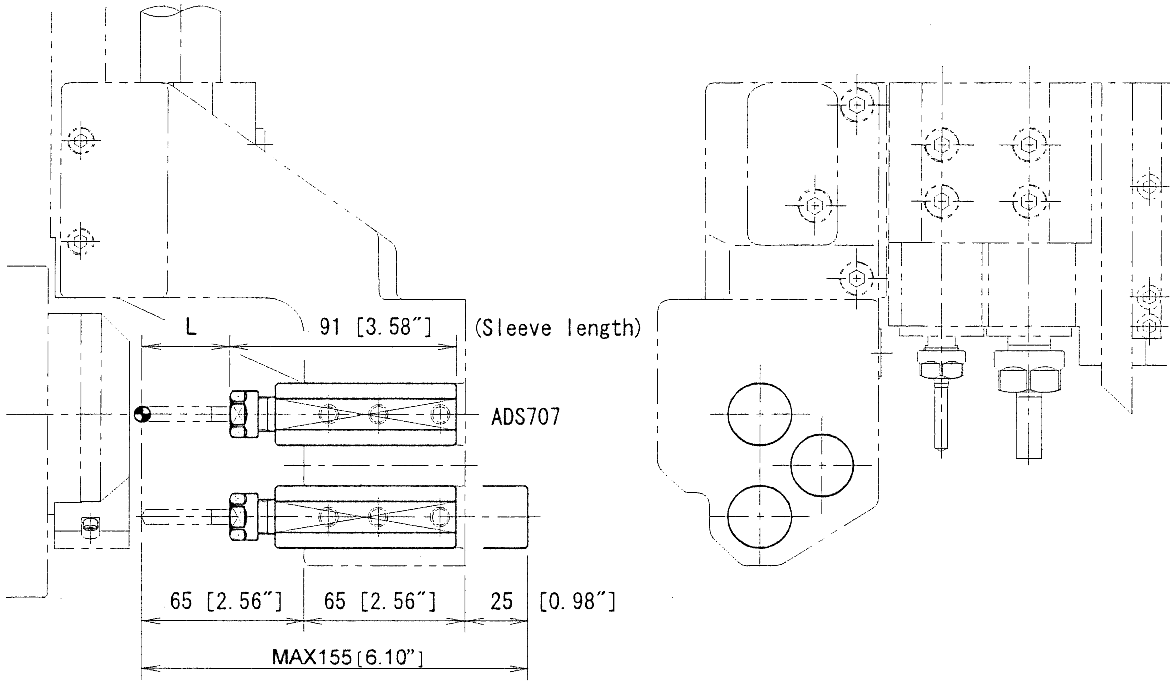
LDS110 Boring and Drill Sleeve (~£10)



LDS110 chucks the drill, etc. with the straight shank used for the front machining by the rego type chuck. L dimension is adjustable by putting sleeve in and out.

Sleeve name	LDS110
Max. chuck dia.	£10mm [£0.39"]
Chuck type	ER16 , AR16
Sleeve outer dia.	£19.05mm [£3/4"]

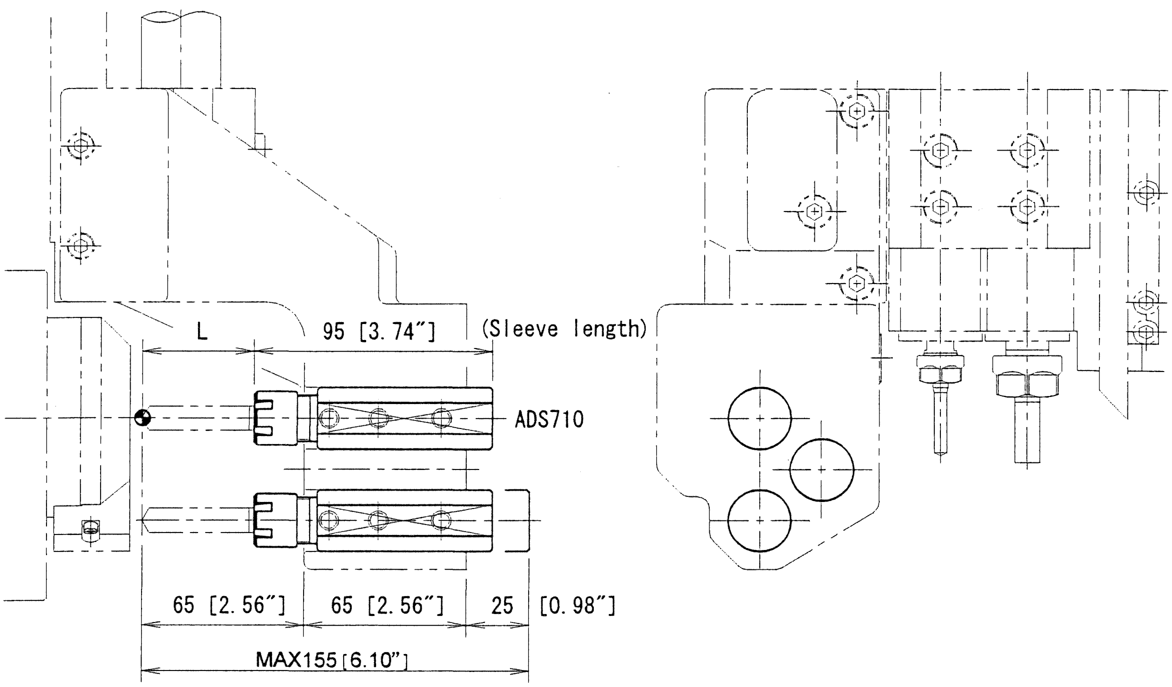
ADS707 Boring and Drill Sleeve (~ƒ7)



ADS707 chucks the drill, etc. with the straight shank used for the front machining by the rego type chuck. L dimension is adjustable by putting sleeve in and out.

Sleeve name	ADS707
Max. chuck dia.	ƒ7mm [ƒ0.28"]
Chuck type	ER11 , AR11
Sleeve outer dia.	ƒ25.4mm [ƒ1"]
Sleeve holder	GDF306

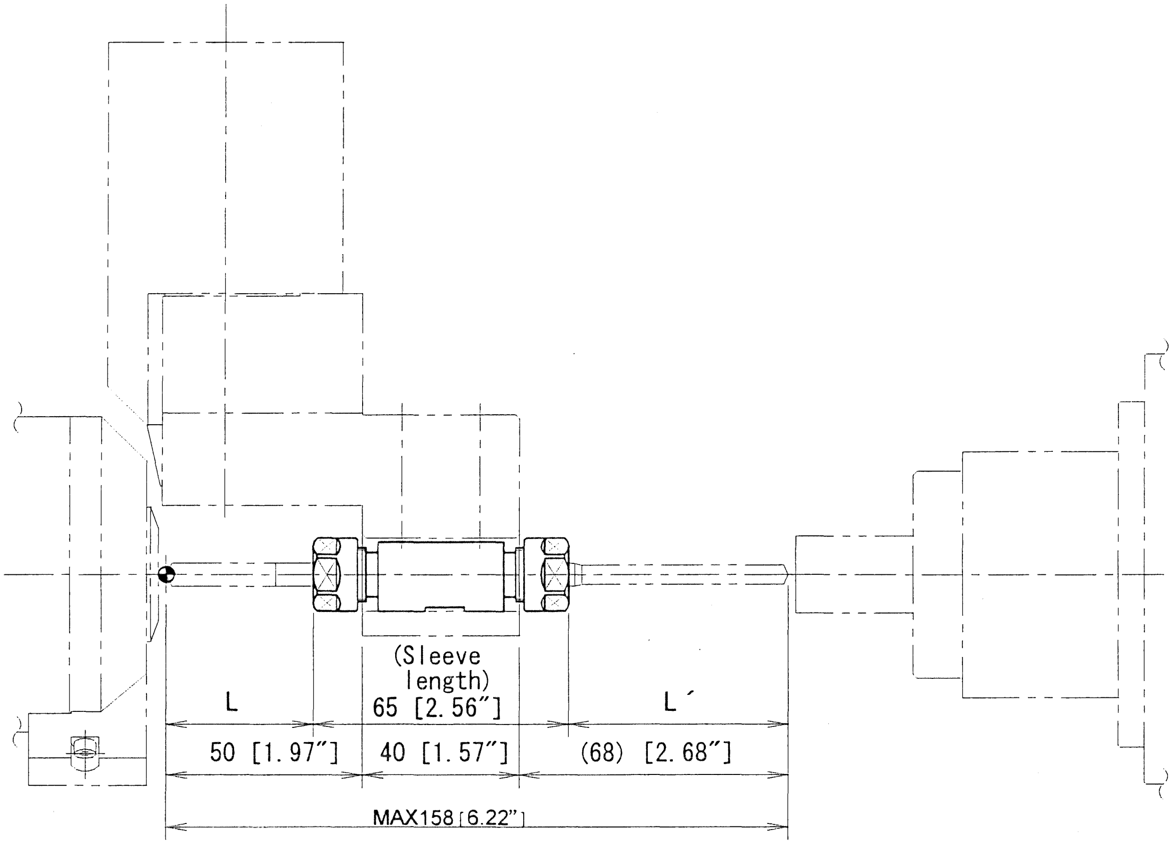
ADS710 Boring and Drill Sleeve (~£10)



ADS710 chucks the drill, etc. with the straight shank used for the front machining by the rego type chuck. L dimension is adjustable by putting sleeve in and out.

Sleeve name	ADS710
Max. chuck dia.	£10mm [£0.39"]
Chuck type	ER16 , AR16
Sleeve outer dia.	£25.4mm [£1"]
Sleeve holder	GDF306

HDS5406 Boring and Drill Sleeve (Both ends, ~f7)

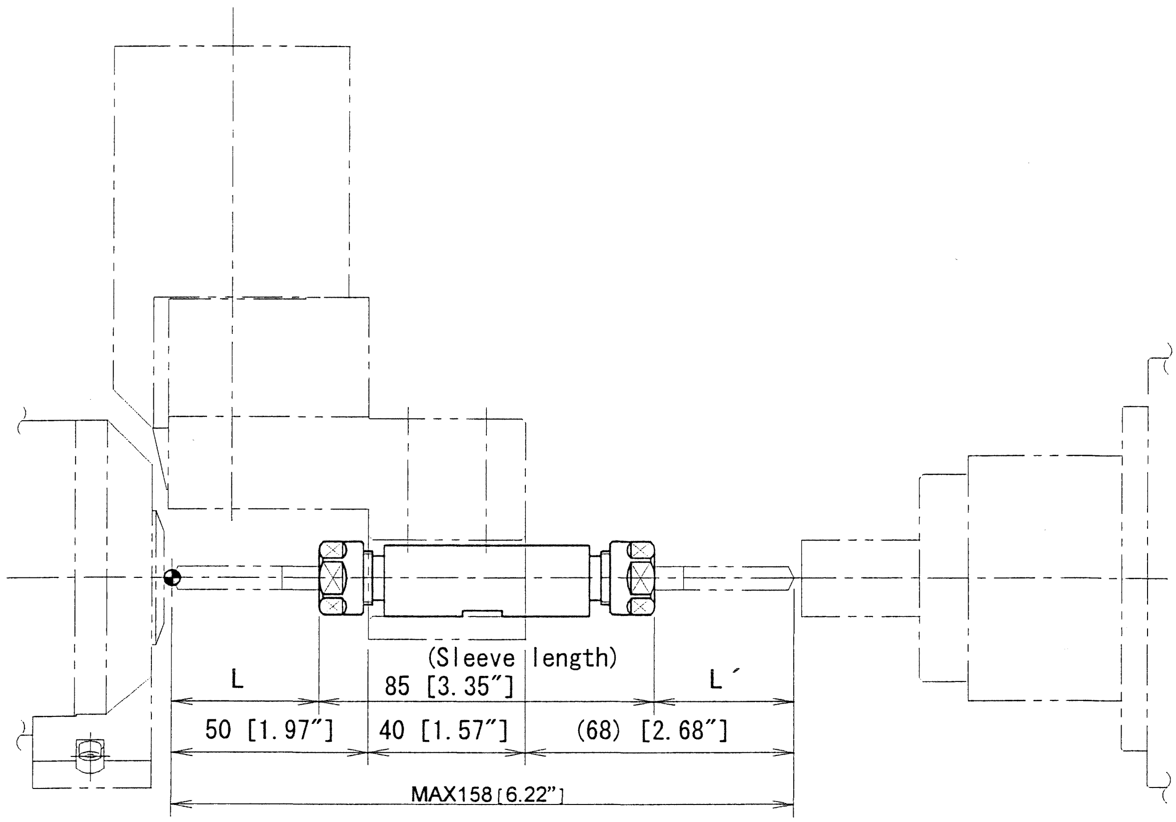


HDS5406 chucks the drill, etc. with the straight shank used for the front and back machining by the rego type chuck. L or L'dimension is adjustable by putting sleeve in and out.

NOTE · Take care to the interference between the back spindle including the opposite sleeves, when mounting this sleeve.

Sleeve name	HDS5406
Max. chuck dia.	f7mm [f0.28"]
Chuck type	ER11 , AR11
Sleeve outer dia.	f19.05mm [f3/4"]
Sleeve holder	GDF506,GDF606,GDF607

HDS5506 Boring and Drill Sleeve (Both ends, ~£7)

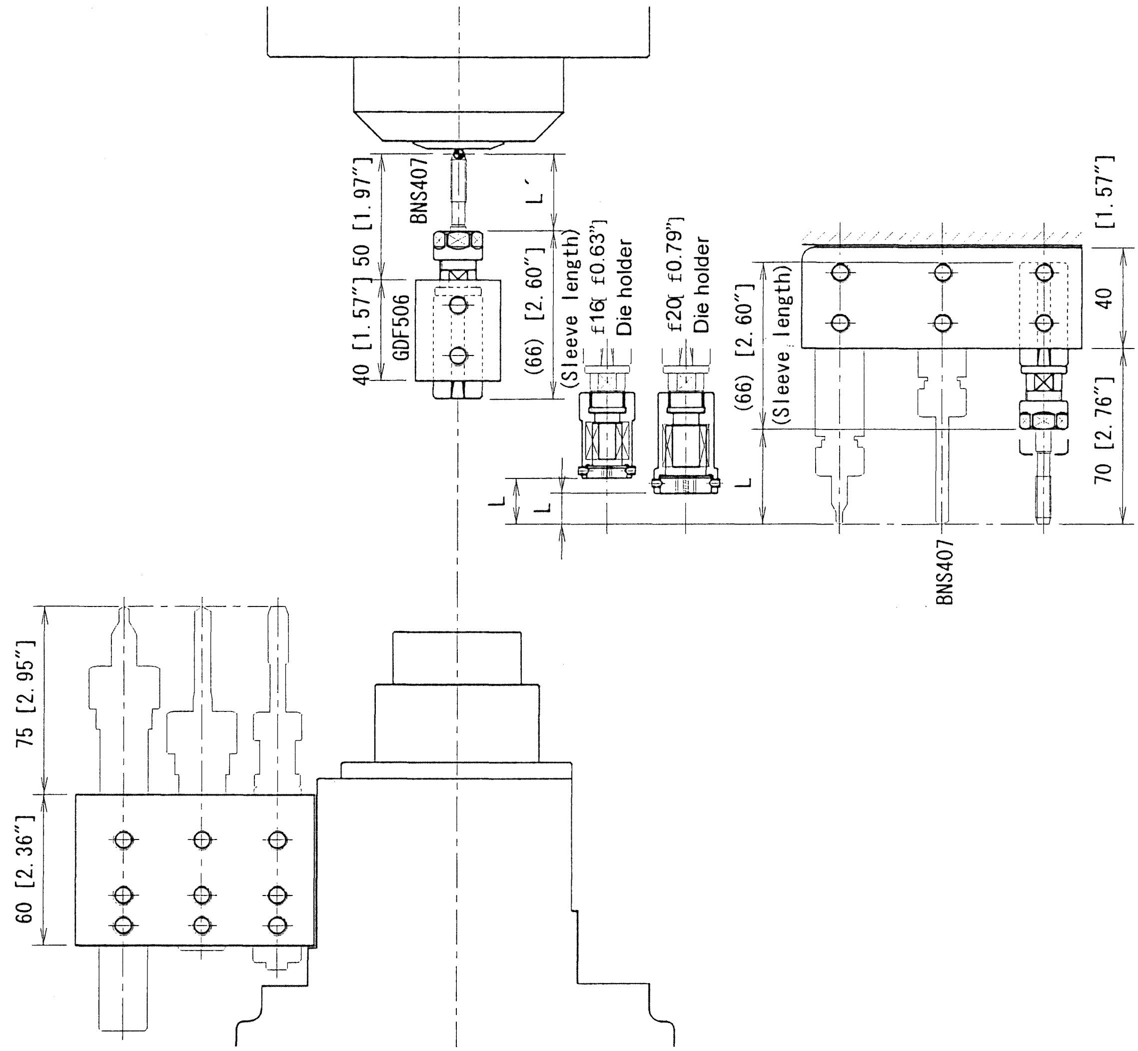


HDS5406 chucks the drill, etc. with the straight shank used for the front and back machining by the rego type chuck. L or L'dimension is adjustable by putting sleeve in and out.

NOTE · Take care to the interference between the back spindle including the opposite sleeves, when mounting this sleeve.

Sleeve name	HDS5506
Max. chuck dia.	£7mm [£0.28"]
Chuck type	ER11 , AR11
Sleeve outer dia.	£19.05mm [£3/4"]
Sleeve holder	GDF506,GDF606,GDF607

BNS407 Tapping and Die Sleeve (~£7)(Die£16,£20)

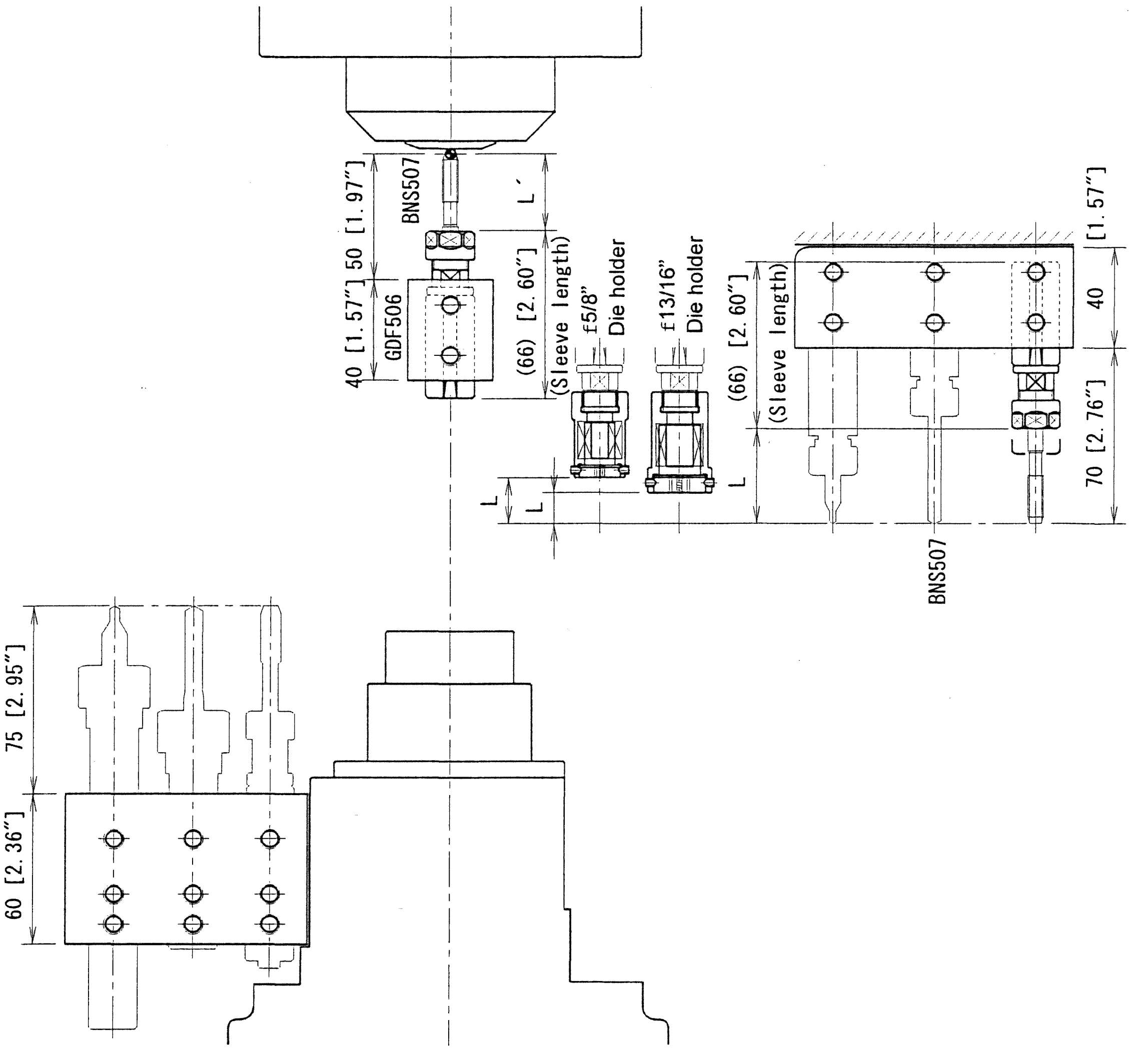


BNS407 is used to hold the tapping and die. Tap can be hold by rego type chuck. The die is hold by mounting die holder in place of cap nut of rego type chuck. L or L' dimension is adjustable by putting sleeve in and out.

NOTE · In case of installing the sleeve to the sleeve holder GDF506,GDF606 and GDF607, the die holder is not available for use.

Sleeve name	BNS407
Max. chuck dia.	£7mm [£0.28"]
Chuck type	ER11 , AR11
Die size	£16 x 5mm , £20 x 7 mm
Sleeve outer dia.	£19.05mm [£3/4"]

BNS507 Tapping and Die Sleeve (~£7)(Die £5/8",£13/16")

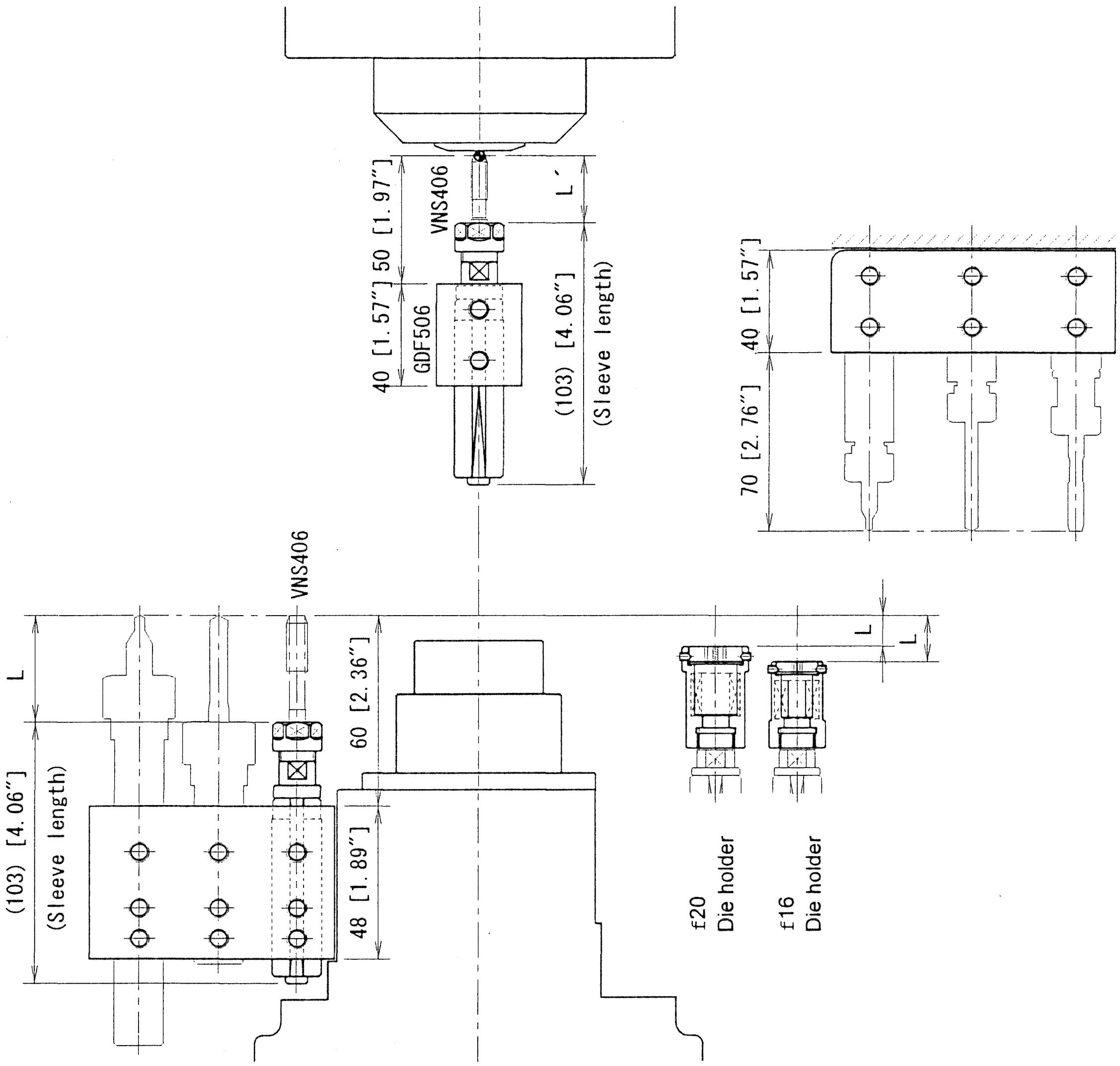


BNS507 is used to hold the tapping and die. Tap can be hold by rego type chuck. The die is hold by mounting die holder in place of cap nut of rego type chuck. L or L' dimension is adjustable by putting sleeve in and out.

NOTE · In case of installing the sleeve to the sleeve holder GDF506,GDF606 and GDF607, the die holder is not available for use.

Sleeve name	BNS507
Max. chuck dia.	£7mm [£0.28"]
Chuck type	ER11 , AR11
Die size	£5/8"x1/4" , £13/16" x 1/4"
Sleeve outer dia.	£19.05mm [£3/4"]

VNS406 Tapping and Die Sleeve (~f7)(Dief16,f20)

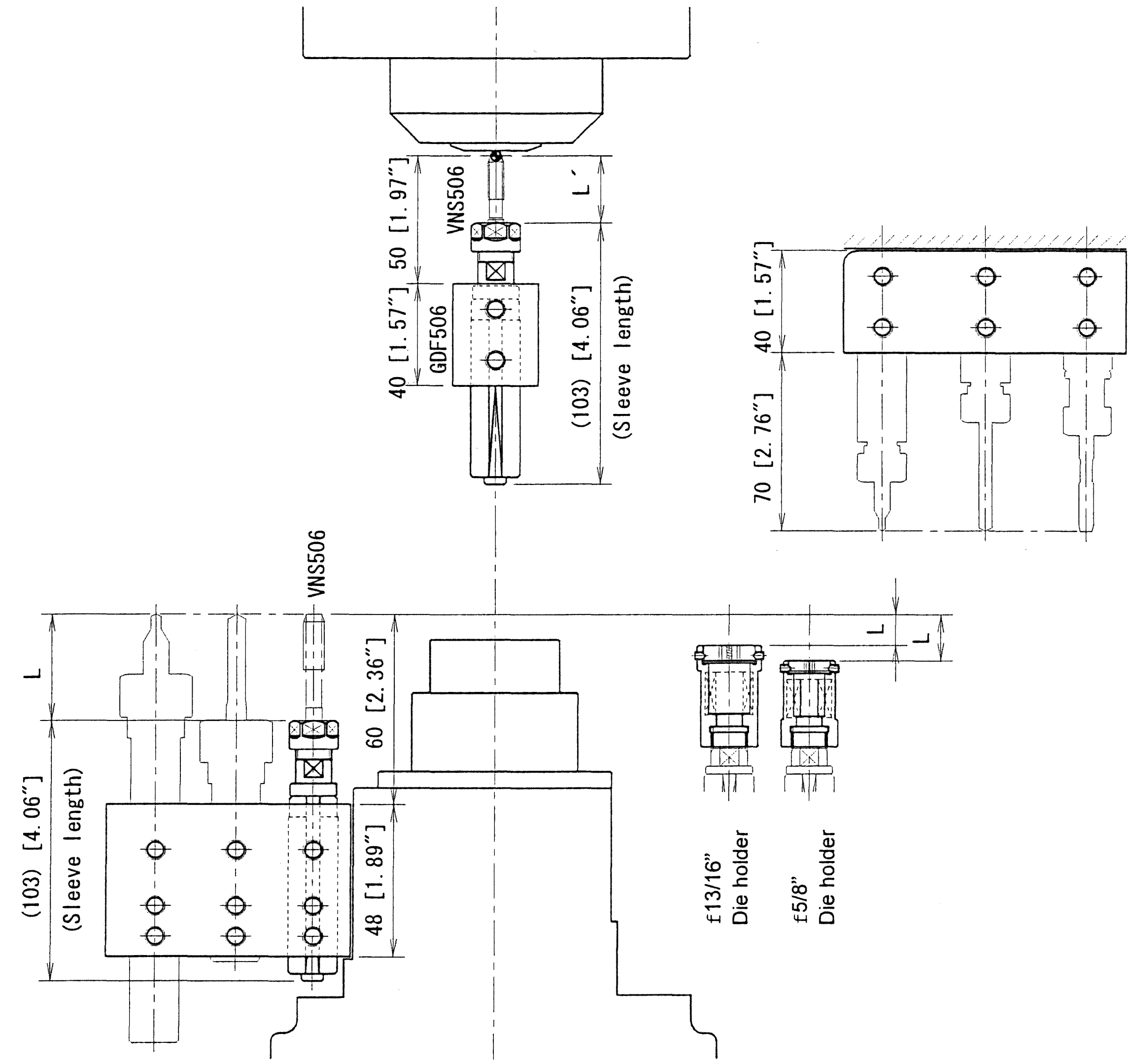


VNS406 is used to hold the tapping and die. Tap can be hold by rego type chuck. The die is hold by mounting die holder in place of cap nut of rego type chuck. L or L' dimension is adjustable by putting sleeve in and out.

NOTE · In case of installing the sleeve to the sleeve holder GDF506,GDF606 and GDF607, the die holder is not available for use.

Sleeve name	VNS406
Max. chuck dia.	f7mm [f0.28"]
Chuck type	ER11 , AR11
Die size	f16 x 5mm , f20 x 7 mm
Sleeve outer dia.	f19.05mm [f3/4"]

VNS506 Tapping and Die Sleeve (~£7)(Die £5/8“,£13/16”)

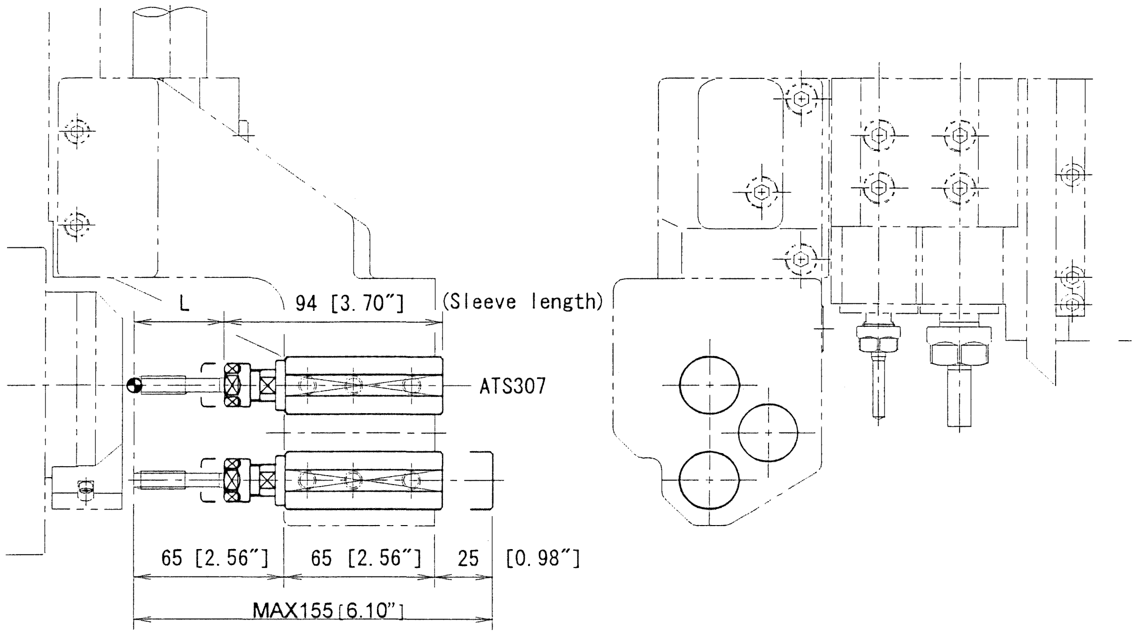


VNS506 is used to hold the tapping and die. Tap can be hold by rego type chuck. The die is hold by mounting die holder in place of cap nut of rego type chuck. L or L’ dimension is adjustable by putting sleeve in and out.

NOTE · In case of installing the sleeve to the sleeve holder GDF506,GDF606 and GDF607, the die holder is not available for use.

Sleeve name	VNS506
Max. chuck dia.	£7mm [£0.28”]
Chuck type	ER11 , AR11
Die size	£5/8“ x 1/4” , £13/16“ x 1/4”
Sleeve outer dia.	£19.05mm [£3/4”]

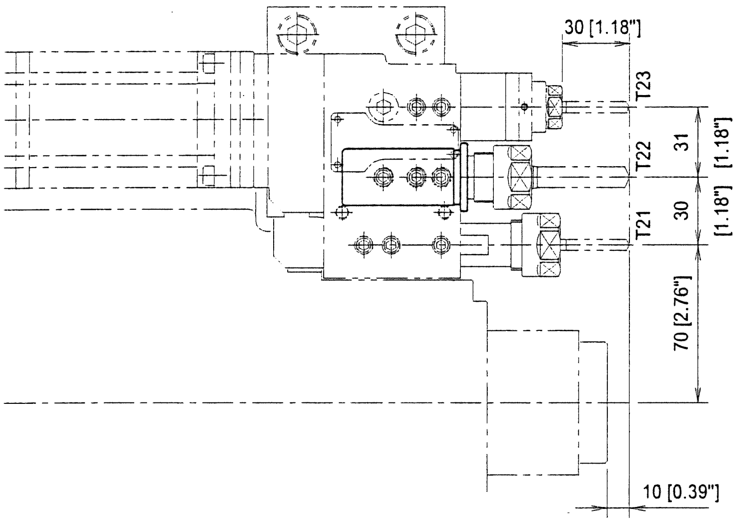
ATS307 Tap Sleeve (~f7)

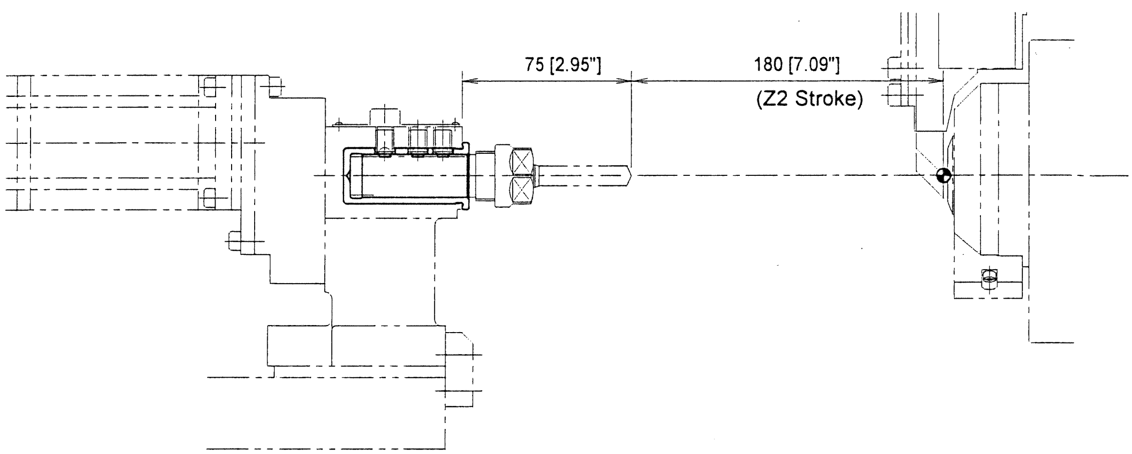


ATS307 is prepared for tapping. It can chuck by rego type chuck irrespecpective of right hand threads or left hand threads. L dimension is adjustable by putting sleeve in and out.

Sleeve name	ATS307
Max. chuck dia.	f7mm [f0.28"]
Chuck type	ER11 , AR11
Sleeve outer dia.	f25.4mm [f1"]
Sleeve holder	GDF306

SAU119 Sleeve Adapter (£19.05)

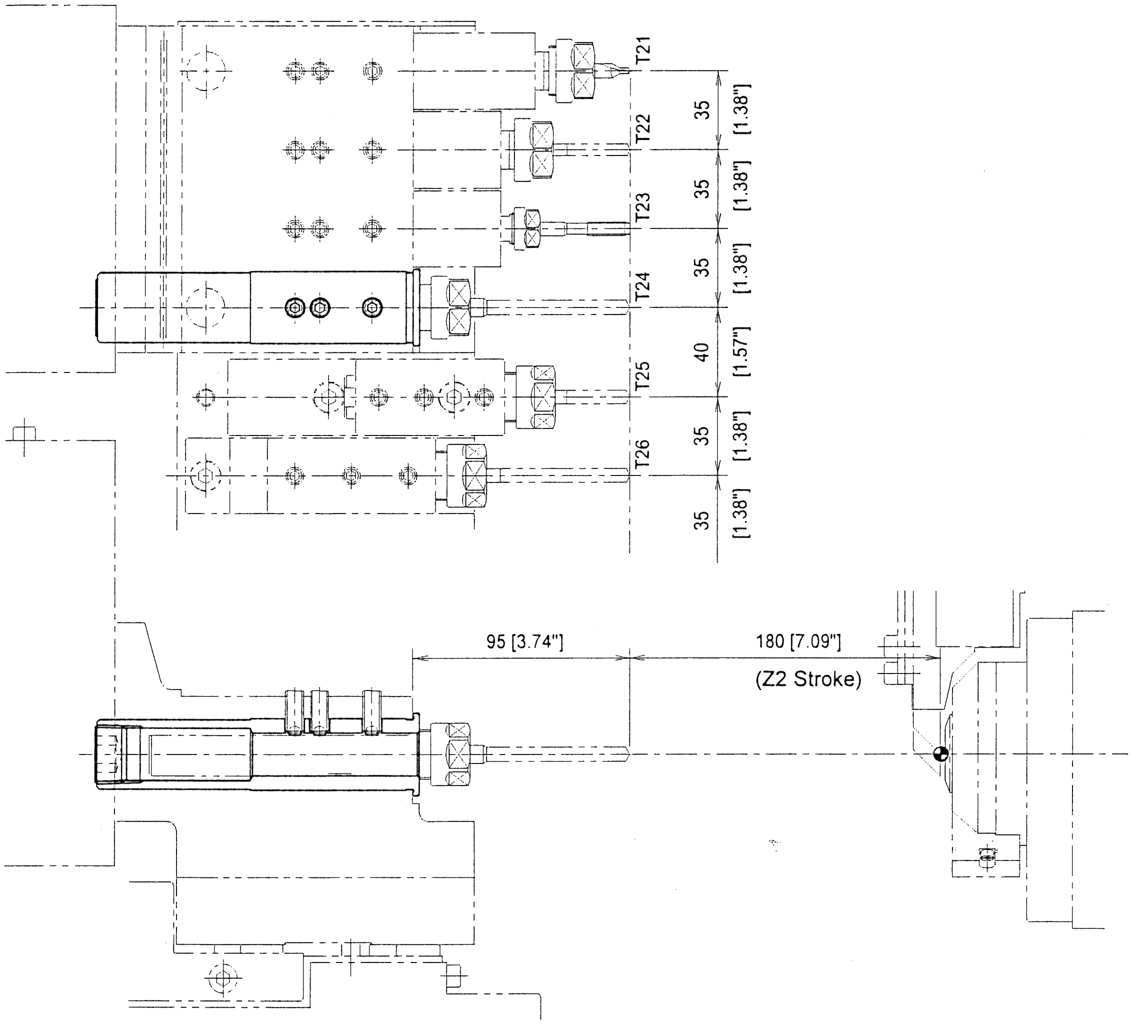




This adapter is prepared for using the shank dia £19.05mm sleeve with the front face drilling device U121B and the back face drilling device U151B.

Adapter name	SAU119
Adapter inner dia	£19.05mm [£3/4"]
Adapter outer dia.	£25.0mm [£0.98"]

SAU219 Sleeve Adapter (ƒ19.05)



This adapter is prepared for using the shank dia ƒ19.05mm sleeve with the front face drilling device U131B.

Adapter name	SAU219
Adapter inner dia	ƒ19.05mm [ƒ3/4"]
Adapter outer dia.	ƒ32.0mm [ƒ1.26"]

Appendix Restrictions On Tooling

1. Restrictions on the rotary drilling tool projecting amount

When the drilling or tapping of a cross through-hole is performed using a rotary tool on a workpiece, it is necessary to set the drilling tool longer. Under this condition the tools may interfere with the tools mounting on T20s when you perform machining with the T20s tools . To use the tools T20s, first select a vertical station which does not have longer projecting drilling tool and then perform the machining.

2. Restrictions on using the end face drilling sleeve holder to be mounted on the vertical holder

1) It is possible to be mounted for GDF506 on following tool station. Take cake to interference GDF506 and the opposite 3-spindles tool post by setting on another station, when cutting off.

Vertical holder name	GDF506 Setting station
GTF3110 , GTF3110L GTF3210 , GTF3210L	T10 , T11
GTF3112 , GTF3113 GTF3212 , GTF3213	T09 , T10

2) In case of machining with the T20s with the sleeve holder GDF506, GDF606 and GDF607, the sleeve holder may interfere with the opposite 3-spindles or the tools mounting on T20s. Please machine with T20s, after the sleeve holder is escaped to -Y1 direction.

Product code	C	–	L	5	1	6	2	0	I	III	VII	/	/	/	–	5	7	0
Document code	2	E	1	–	1	0	0	2										
	2	E	2	–	1	0	0	1										